# Autodesk® Robot™ Structural Analysis Professional 2010 Training Manual - Metric Version

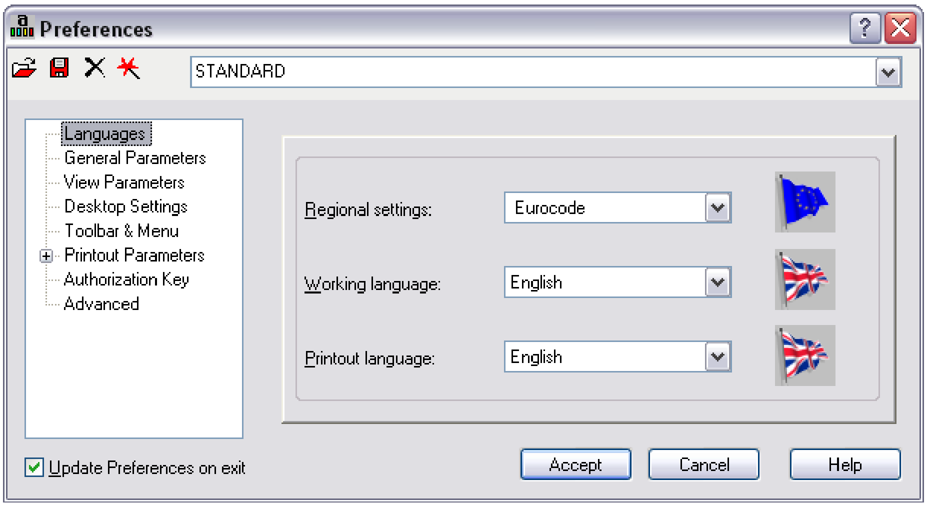
***NOVEMBER 2009***

# General Information

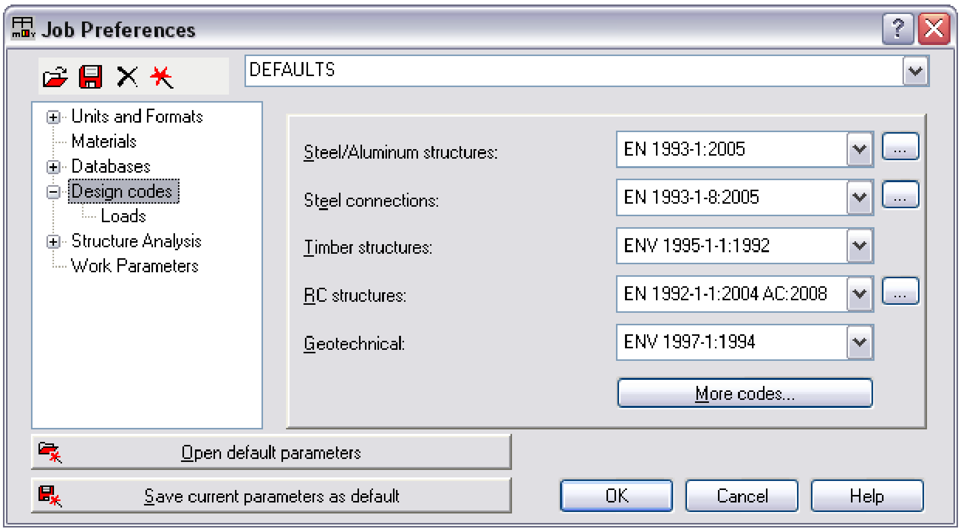
## Setup and Preferences

***Preferences*** are available from text menu *Tools* > *Preferences*. Here are groups of general settings to customize the look of the user interface and define how the program works. Here you can choose working language (language of interface), regional settings (codes, databases) and printout language. All of these are set independently, so you can work with one language (chosen from one of ten available) according to different regional codes and print documentation in another.
Also within ***Preferences,*** you can change look of every particular element of the desktop by using predefined templates or by creating your own.

Before commencing structure definition, one should set the working language and codes to be applied in the project as shown in the picture below:



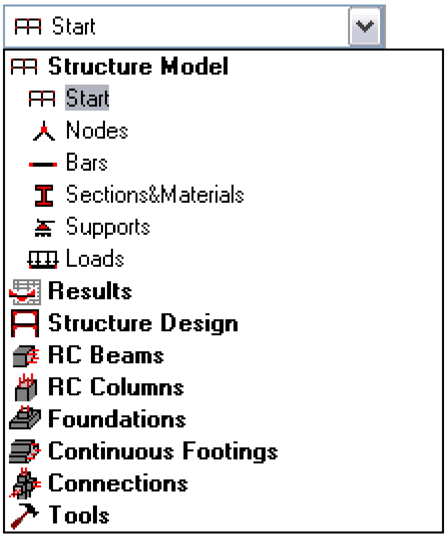

Confirm the operation by pressing the Accept button, and then select from the main menu Tools / Job Preferences option. Set the codes and actions as shown below:




***Job Preferences*** are grouped in six categories: units, materials, databases, design codes, structure analysis and work parameters.
Autodesk® Robot™ Structural Analysis Professional includes more than 60 sections and materials databases from around the world. With an array of 70 built-in design codes, structural engineers can work with country-specific section shapes, imperial or metric units, and country-specific building codes within the same integrated mode.

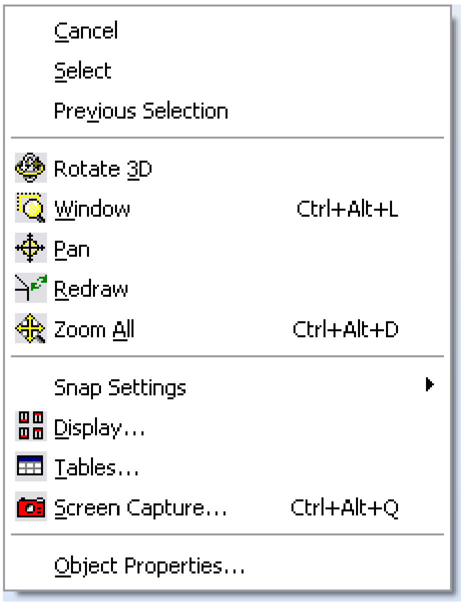
## Layout selection

It is necessary to select appropriate layouts in the process of structure definition. The layouts are accessible by clicking the list box in the top right corner of the main window which opens the layout list shown in the figure below:




## Context menu

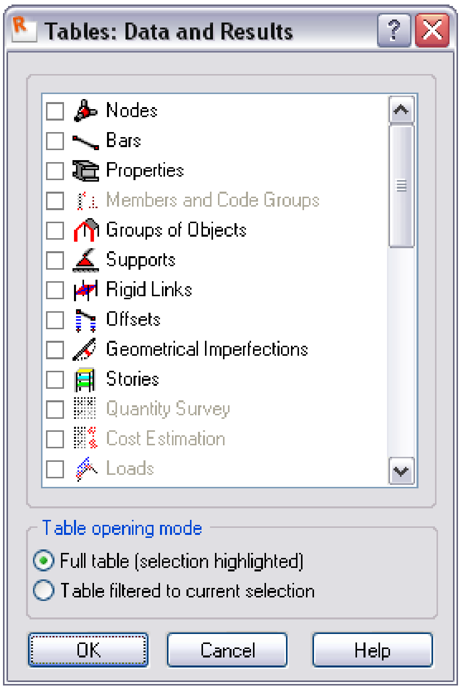
While working in the graphical viewer, one may activate the context menu (shown below) by pressing the right-hand mouse button.




The menu allows one to perform many useful (and frequently used) operations while the program is carrying out the formerly issued commands.

## Data and Results Tables

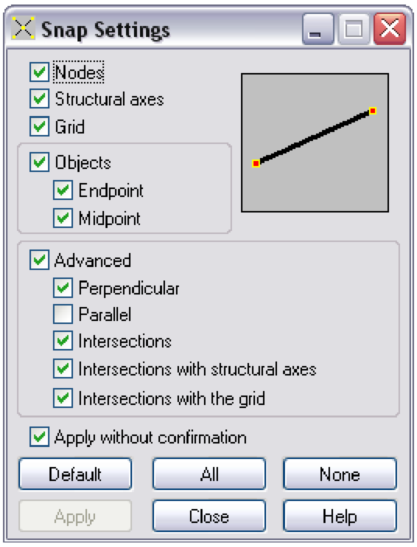
Structure parameters may be modified by means of the relevant tables. The tables relevant to the current layout become visible when one enters the layout. In order to be able to perform global edit operations, one should use the *View menu / Tables* option from the main menu. There will appear the ***Tables: Data and Results*** dialog box.




In this dialog box, one should indicate the required items and press the **OK** button. A table containing data will be generated for each of the indicated items. Once the *Edit* tab is activated in the bottom left corner of a given table, one may perform the operation of modifying structure parameters.

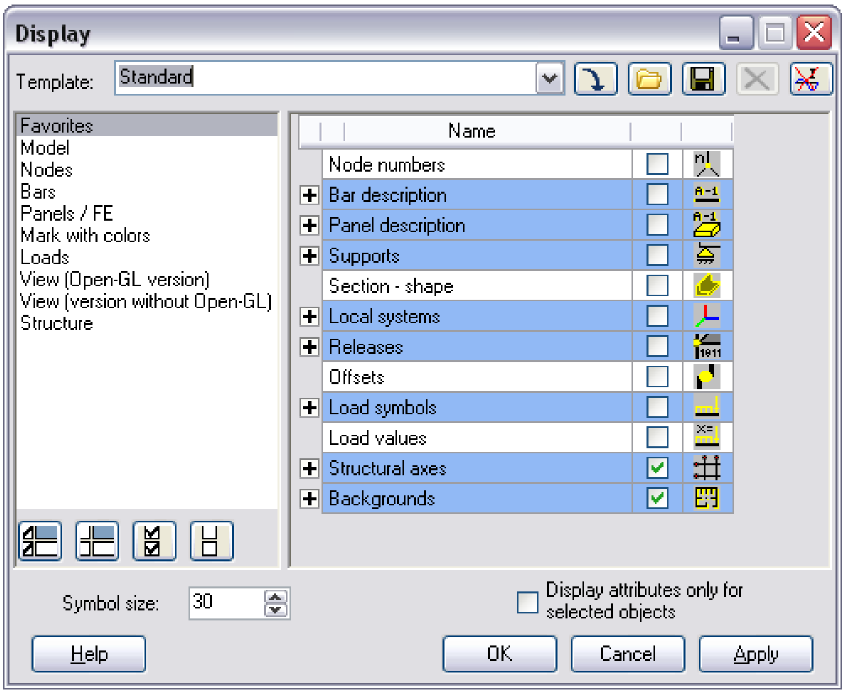
## Snap settings

The ***Snap Settings*** dialog box becomes available once the icon is pressed (the first one icon located in the bottom left corner of the screen).



## Display of Structural Parameters

The ***Display*** dialog box becomes accessible once the icon (the third one icon in the bottom left corner of the screen) is pressed as shown below.




The available tabs allow one to get access to the data on structure parameters. This option is also available from the main menu by means of selecting the *View menu / Display* command.

## Object Inspector

The Object Inspector is located along the left-hand side of the interface. Using this tool user can

- Presents the project contents in an organized manner
- Selects elements that should be acted upon by a selected command
- Presents and modifies properties of project elements (both single elements and whole objects)
- Filters model elements
- Creates and manages documentation of a project

The Object Inspector consists of several topic-specific elements. Tabs to select these topics are along the bottom of the dialog.

Object Inspector (tabs: Geometry and Groups)

Steel Connection Inspector

RC Component Inspector

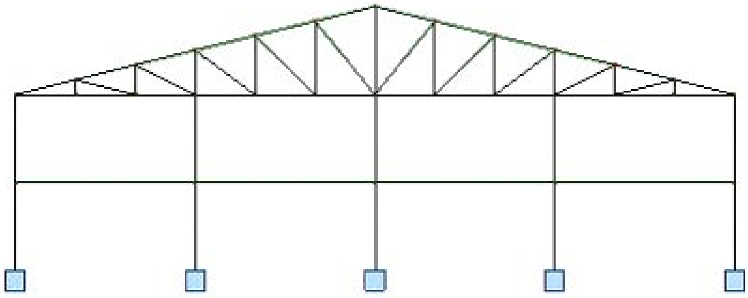
Inspector - Preparation of Results

NOTE: In the examples below the following rule has been assumed to indicate definition of the beginning and end of a structure bar:
for example, (0,0,6) (8,0,6) means that a bar beginning is positioned at a node with the coordinates as follows x = 0.0, y = 0.0 and z = 6.0 and a bar end - at a node with the coordinates as follows x = 8.0, y = 0.0 and z = 6.0. The separator (set in the Windows operating system) separates the successive coordinates by using a comma ‘,‘ between the values..

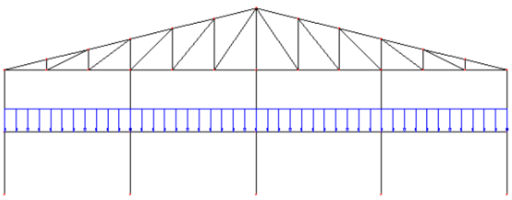
# 1. Reinforced Concrete Design – 2D Frame

This example is used to show the definition, analysis and design of a simple 2D frame illustrated in the figure below. The frame is made of the RC frame and the truss generated by using the library of typical structures available in the ***RSAP*** program.

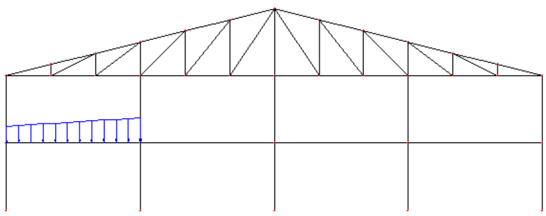
Data units: (m) and (kN).



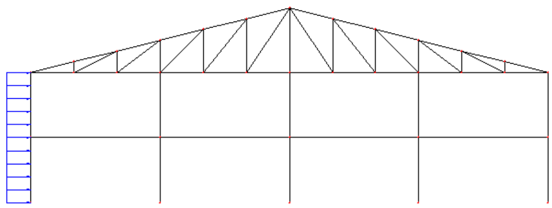
Four out of five load cases applied to the structure are displayed in the drawing below.



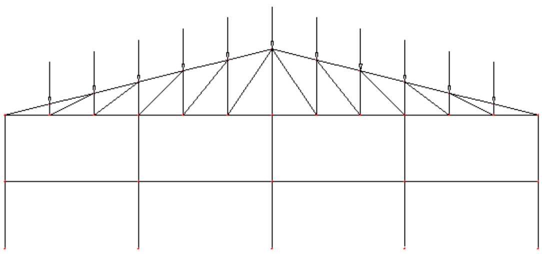
LOAD CASE 2



LOAD CASE 3



LOAD CASE 4

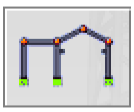


LOAD CASE 5

The following rules will apply during structure definition:

- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the ‘x’ option in the dialog box or entering the ‘x’ value,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

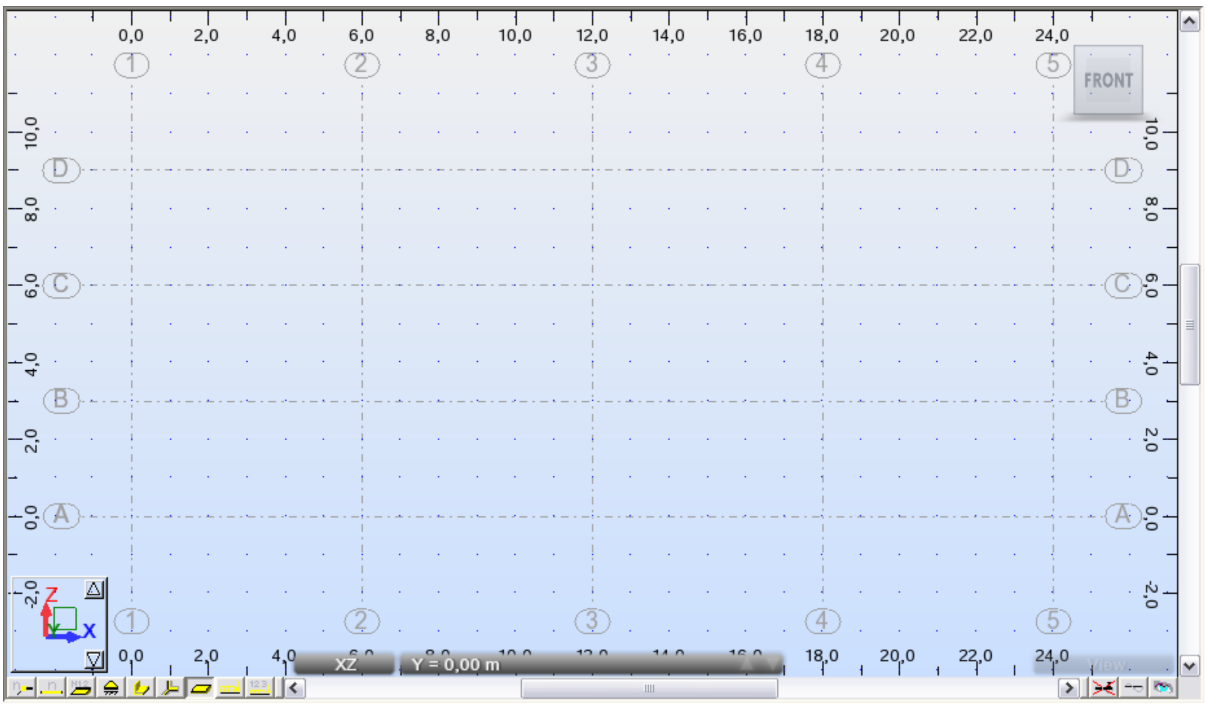
To run structure definition start the ***RSAP*** program (press the appropriate icon or select the command from the taskbar). The vignette window will be displayed.

Select icon  in the first row ***2D Frame Design***).

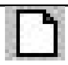
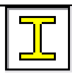
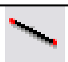
*NOTE: The European Section Database (EURO) has been used in this example.*

## 1.1 Model Definition

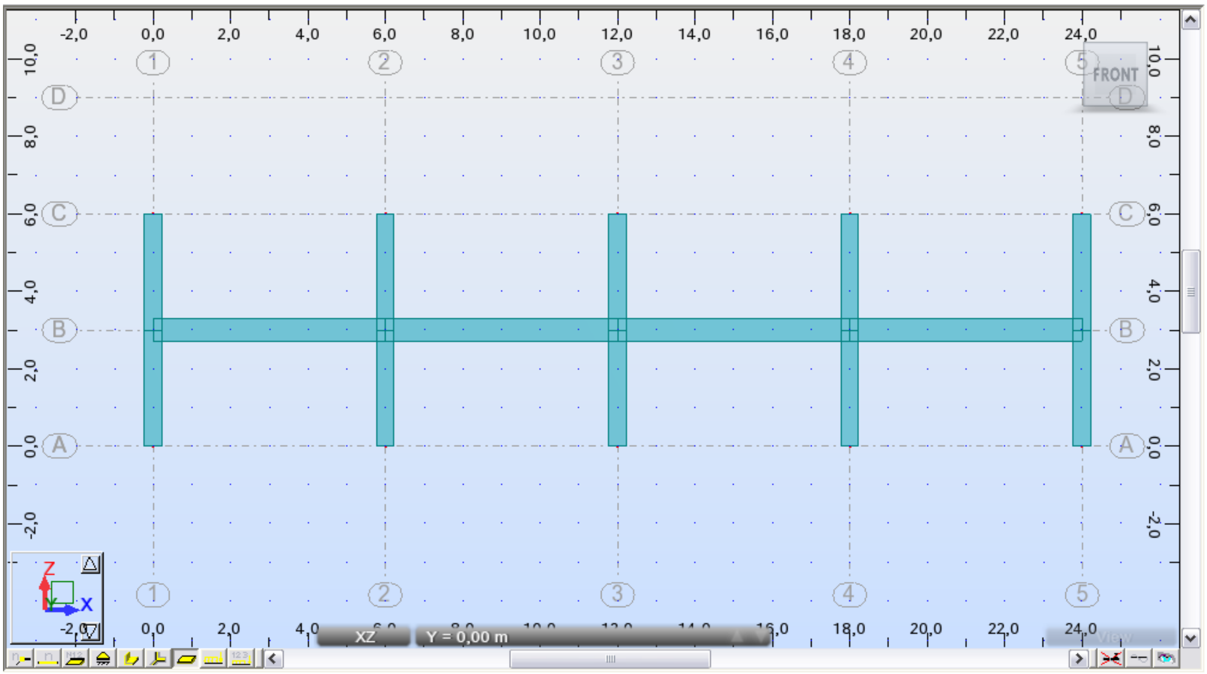
| PERFORMED OPERATION | DESCRIPTION |
|---|---|
| Select the *Axis Definition* icon from the Structural Model toolbar. | Starts definition of structural axes. The ***Structural Axis*** dialog box appears on the screen. |
| On the ***X*** tab:<br>Position: (0)<br>Number of repetitions: (4)<br>Distance: (6)<br>Numbering: (1, 2, 3 ...) | Defines vertical axis parameters. |
| LMC on the **Insert** button | Vertical axes have been defined and will be presented in the *Set of Created Axis* field. |
| LMC on the ***Z*** tab | Starts definition of horizontal axis parameters. |
| On the ***Z*** tab:<br>Position: (0)<br>Number of Repetitions: (3)<br>Distance: (3)<br>Numbering: (A, B, C ...) | Defines horizontal axis parameters. |
| LMC on the **Insert** button | Horizontal axes have been defined and will be presented in the *Set Of Created Axes* field. |
| **Apply**, **Close** | Creates defined structural axes and closes the ***Structural Axes*** dialog box. Structural axes will be displayed on the screen as shown in the figure below. |



### *1.1.1 Member Definition*

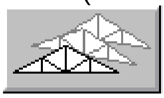
| | |
|---|---|
|  Select the *Bar Section* icon from the Structural Model toolbar. | Opens the ***Sections*** dialog box. |
|  Select the *New Section Definition* icon. | Opens the ***New Section*** dialog box. |
| LMC the "I" family  icon, pick (HEB) from the Family List, and select (HED 240) from the Section list.<br>**Add** | Defines a new section. The section from the European section database (EURO) has been used. |
| LMC in the *Section Type* field (lower right corner of dialog box) and select the *RC beam* option.<br>In the *Label* field enter B 45x60.<br>Under Basic Dimensions, type<br>in fields b = (45) cm, h = (60) cm<br>**Add**, **Close** | Defines an RC beam section. |
| **Close** | Closes the ***Sections*** dialog box. |
|  Select the *Bars* icon from the Structral Model toolbar | Opens the ***Bars*** dialog box. |
| LMC on the *Bar type* field and select ***RC column***<br>LMC on the *Section* field and select the type: (C 45x45) | Selects bar properties. |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of bars in the structure (structure columns). |
| Enter the following points in the *Beginning* and *End* fields.<br>Beginning: (0,0) End: (0,3), **Add**<br>Beginning; (0,3) End: (0,6), **Add** | Defines the first two bars located on structural axis number 1. |
| RMC within the graphics view area and choose *Select* command from the context menu | Opens context menu and switches to selection mode.<br>The mouse cursor changes its shape to "hand". |
| CTRL+A | Selects all bars. (Remember to activate the View window first.) |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the field dX,dZ=: (6,0)<br>LMC on the fields:<br>*Numbering Increment Nodes: (1)*<br>*Numbering Increment Elements:*<br>(1) | Defines the translation vector and numbering increment for nodes and bars. |

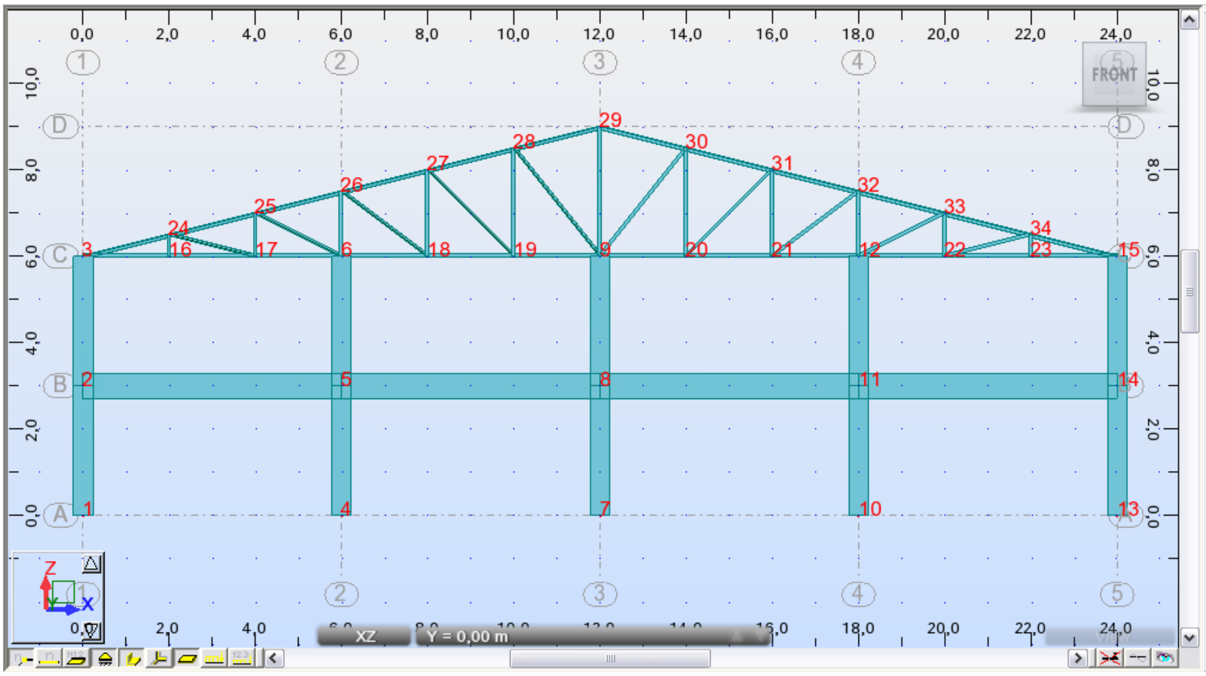
| | |
|---|---|
| LMC on the *Number of repetitions* field: (4) | Defines the number of repetitions for performed translation operations. |
| **Execute**, **Close** | Column translation; closes the ***Translation*** dialog box. |
| LMC on the *Bar type* field in the ***Bars*** dialog box and select *RC beam*<br>LMC on the *Section* field and select (B 45x60) | Starts definition of beams in the structure and selects their properties. |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of bars in the structure. |
| Beginning: (0,3) End: (6,3), **Add**<br>Beginning: (6,3) End: (12,3), **Add**<br>Beginning: (12,3) End: (18,3), **Add**<br>Beginning: (18,3) End: (24,3), **Add** | Defines the RC beam located on the structural axis B. |
| **Close** | Closes the ***Bars*** dialog box. |
| *View menu / Display* | Opens the ***Display*** dialog box. |
| *LMC Bars* tab<br>Turn on the *Section-Shape* option, **Apply**, **OK** | This option allows for the display of section shapes for the defined structure bars. Bars will be displayed on the screen as shown in the figure below. |




### 1.1.2 Library Structure Definition

| | |
|---|---|
| *View menu / Display* | Opens the ***Display*** dialog box |
| LMC *Nodes* tab<br>Turn on the *Node numbers* option, **Apply**, **OK** | This options allows for the display of node numbers located at the ends of the bars. |

| | |
|---|---|
|  Select the *Library Structure* icon located on the Structure Model toolbar. | Opens the ***Typical Structures*** dialog box and starts definition of a library structure. |
| LMC (double-click) on the icon  | Selects a triangular truss of type 1. The ***Merge Structure*** dialog box appears and truss parameters can be defined. |
| LMC on the *Length L* field on the *Dimensions* tab: (24) | Defines the truss length (it can also be defined graphically in the graphic viewer). |
| LMC on the *Height H* field: (3) | Defines the truss height (it can also be defined graphically in the graphic viewer). |
| LMC on the *Number of Fields* field: (12) | Defines the number of fields into which the truss will be divided. |
| LMC on the *Sections* tab;<br>To all truss chords (upper and lower) assign (DCED 90x10) and to diagonals, posts asign (CAE 70x7) | Assigns the section to the truss bars. |
| LMC on the *Insert* tab | |
| LMC on the *Insertion point* field, select the node number 3 of the following coordinates: (0,0,6) | Defines the truss beginning node. |
| **Apply**, **OK** | Locates the defined structure in the appropriate place and closes the ***Merge structure*** dialog box. The defined structure is presented on the drawing below. |




| | |
|---|---|
| *View menu / Display* | Opens the ***Display*** dialog box. |

| | |
|---|---|
| *LMC Nodes* tab<br>Turn off the *Node numbers* option<br>*LMC Structure* tab<br>Turn off the *Structural axis* option,<br>**Apply**, **OK** | |
| *Geometry menu / Releases* | Opens the ***Releases*** dialog box. |
| LMC on the *Pinned-Fixed* release type | Chooses the release type that will be assigned to a truss bar. |
| LMC on the *Current selection* field, switch to the graphic viewer and indicate (hover curser over) the highest post of the truss (the bar between the nodes 9 and 29) | Selects the truss bar; ATTENTION: take note of the arrows that appear on the highlighted truss bar – while indicating the bar the arrows should be pointed up (the direction of the release is significant: at the first node the pinned connection remains, whereas at the second one – the fixed connection is defined) |
| **Close** | Closes the ***Releases*** dialog box. |

### *1.1.3 Support Definition*

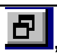
| | |
|---|---|
|  Select the *Supports* icon from the Structral Model toolbar | Opens the ***Supports*** dialog box. |
| LMC on the *Current Selection* field on the *Nodal* tab (the cursor should be blinking in that field) | Selects structure nodes in which supports will be defined. |
| Switch to the graphic viewer by pressing the left mouse button; select all lower column nodes with the window | Selected nodes: 1to13by3 will be entered to the *Current Selection* field. |
| In the ***Supports*** dialog box select the Fixed support icon (the support will be highlighted) | Selects the support type. |
| **Apply**, **Close** | Selected support type will be assigned to selected structure nodes, closes the ***Supports*** dialog box. |

### *1.1.4 Load Case Definition*

| | |
|---|---|
|  Select the *Load Types* icon from the Structural Model toolbar | Opens the ***Load Types*** dialog box. |
| LMC on the **New** button | Defines a *dead load (self-weight)* with a standard name DL1. |
| LMC on the *Nature* field: *(Live1)* | Selects the load nature: *live*. |
| LMC on the **New** button<br>LMC on the **New** button | Defines two *live load* cases with standard names LL1 and LL2. |
| LMC on the *Nature* field: *(Wind)* | Selects the load case nature: *wind*. |

| | |
|---|---|
| LMC on the **New** button | Defines a wind load case with a standard name WIND1. |
| LMC on the *Nature* field: *(Snow)* | Selects the load case nature: *snow*. |
| LMC on the **New** button, **Close** | Defines a snow load case with a standard name SN1 and closes the ***Load Types*** dialog box. |

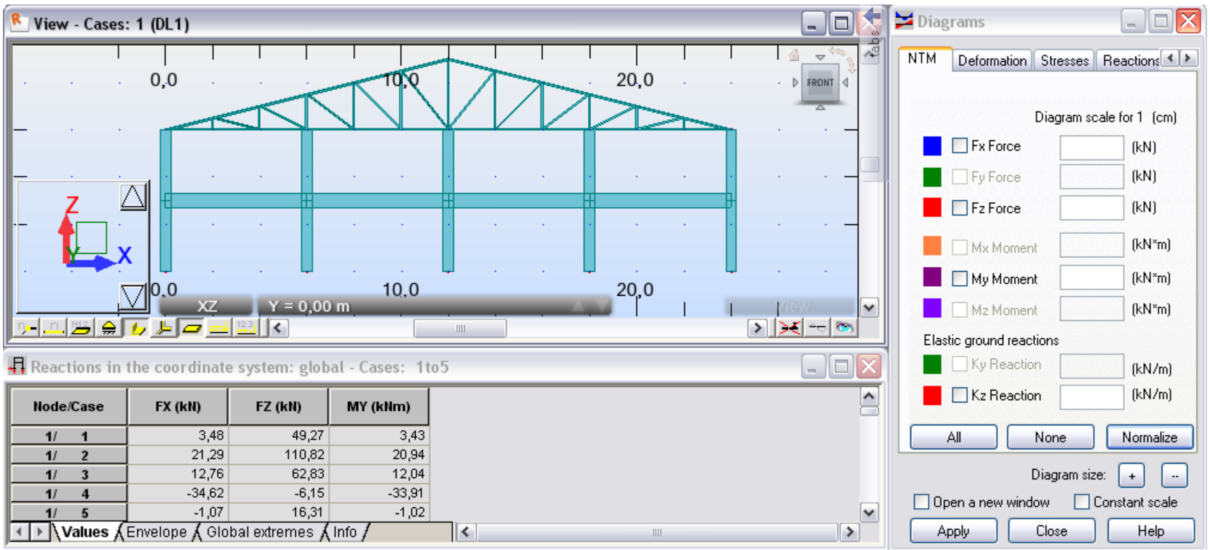
### *1.1.5 Load Definition for Generated Cases*

| | |
|---|---|
| *Loads menu / Load Table* | Opens a table for loads acting in defined load cases. |
| , Select the *Restore Down* icon in the upper right corner of the table view.<br>Place the table in the lower part of the screen in such a way so that it is adjusted to its width and the defined structure model is displayed. | Decreases the table size so that the load graphic definition is possible. (You can use Windows/Align Windows after the loads window is resized.) |
| | Dead Load (direction “-Z”) automatically applied to all structure bars. |
| LMC on the second field in the ***Case*** column, select the $2^{nd}$ load case LL1 from the list | Defines loads for the second load case. |
| LMC on the field in the ***Load Type*** column, select the *uniform load* | Selects the load type. |
| LMC on the field in the ***List*** column, select all the concrete beams in the graphic viewer (bars 11to14) | Selects bars to which the *uniform* load will be applied. |
| LMC on the field in the "PZ=" column and enter the value: (-40) | Selects the direction and sign of the *uniform* load. |
| LMC on the next field in the ***Case*** column, select the $3^{rd}$ load case LL2 from the list | Defines loads for the third load case. |
| LMC on the ***Load Type*** column, select the *trapezoidal load (2p)* | Selects the load type. |
| LMC on the field in the ***List*** column, select graphically in the graphic viewer the first left span of the concrete beam (bar 11) | Selects bars to which the *trapezoidal* load will be applied. |
| LMC on the field in the "PZ1=" column and enter the value: (-20)<br>LMC on the X2 field and enter value: (1.0)<br>LMC on the field in the "PZ2=" column and enter the value: (-25) | Selects the direction and sign of the *trapezoidal* load |
| LMC on the next field in the ***Case*** column, select the $4^{th}$ load case WIND1 from the list | Defines loads for the fourth load case. |

| | |
|---|---|
| LMC on the field in the ***Load Type*** column, select the *uniform load* | Selects the load type. |
| LMC on the field in the ***List*** column, Select graphically in the graphic viewer the left edge column (bars 1 and 2) | Selects bars to which the *uniform* load will be applied. |
| LMC on the field in the "PX=" column and enter the value: (15) | Selects the direction and value of the uniform load. |
| LMC on the field in the ***Case*** column, select the 5th load case SN1 from the list | Defines loads for the fifth load case. |
| LMC on the field in the ***Load Type*** column, select *nodal force* as a load type | Selects the load type. |
| LMC on the field in the ***List*** column, select graphically in the graphic viewer the nodes on the upper truss chords (without the edge nodes) (nodes 24to34) | Selects nodes to which the nodal force load will be applied. |
| LMC on the field in the "FZ=" column and enter the value: (-25) | Selects the direction and the load value. |
| Close the ***Load table*** | |

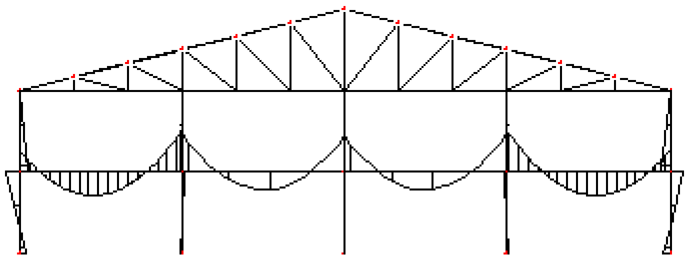
## 1.2 Structural Analysis

| | |
|---|---|
| *Tools menu / Job Preferences* | Opens the ***Job Preferences*** dialog box |
| *Units and Formats / Other* | Selects the option that enables defining a number of decimal places for selected quantities. |
| Increase of the number of decimal places for Displacement to 4 | Increases the number of decimal places for Displacement to 4. |
| **OK** | Accepts assumed parameters and closes the ***Job Preferences*** dialog box |
|  Select the *Calculations* icon from the Standard toolbar | Starts calculations for the defined structure. |
| *Results menu / Diagrams for bars* | |

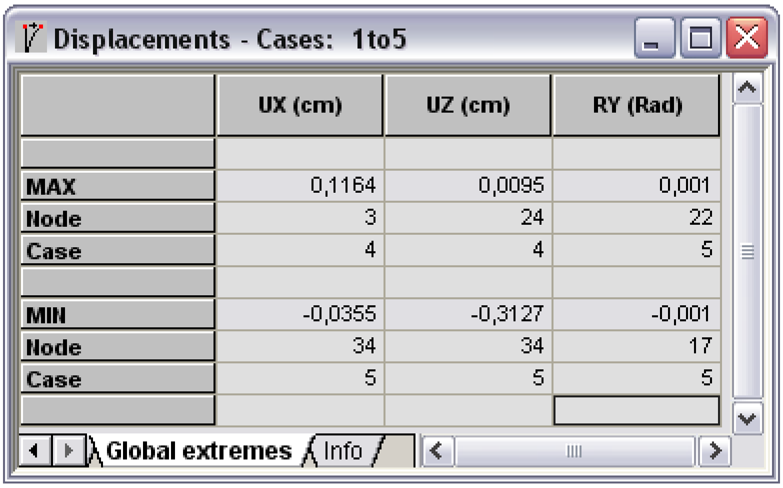



## 1.3 Analysis Results

| | |
|---|---|
| LMC ***Reactions*** table   From the Selection toolbar, select (2: LL1) | Displays the results for the second load case. |
| Turn on the *My Moment* option on the *NTM* tab in the ***Diagrams*** dialog box | Selects the bending moment My for presentation. |
| **Apply** | Displays a diagram of the bending moment for structure bars (see the drawing below). In a similar way, diagrams that exhibit other values available from the ***Diagrams*** dialog box can be displayed. |



| | |
|---|---|
| Turn off the *My Moment* option in the ***Diagrams*** dialog box, **Apply** | |
| Select the *Displacements* icon from the Structure Model toolbar | Opens a table containing structure displacements. |
| LMC on the *Global extremes* tab in the **Displacements** table | Displays the maximum and minimum displacements obtained in structure nodes (see the drawing below). |




| | UX (cm) | UZ (cm) | RY (Rad) |
|---|---|---|---|
| | | | |
| MAX | 0,1164 | 0,0095 | 0,001 |
| Node | 3 | 24 | 22 |
| Case | 4 | 4 | 5 |
| | | | |
| MIN | -0,0355 | -0,3127 | -0,001 |
| Node | 34 | 34 | 17 |
| Case | 5 | 5 | 5 |

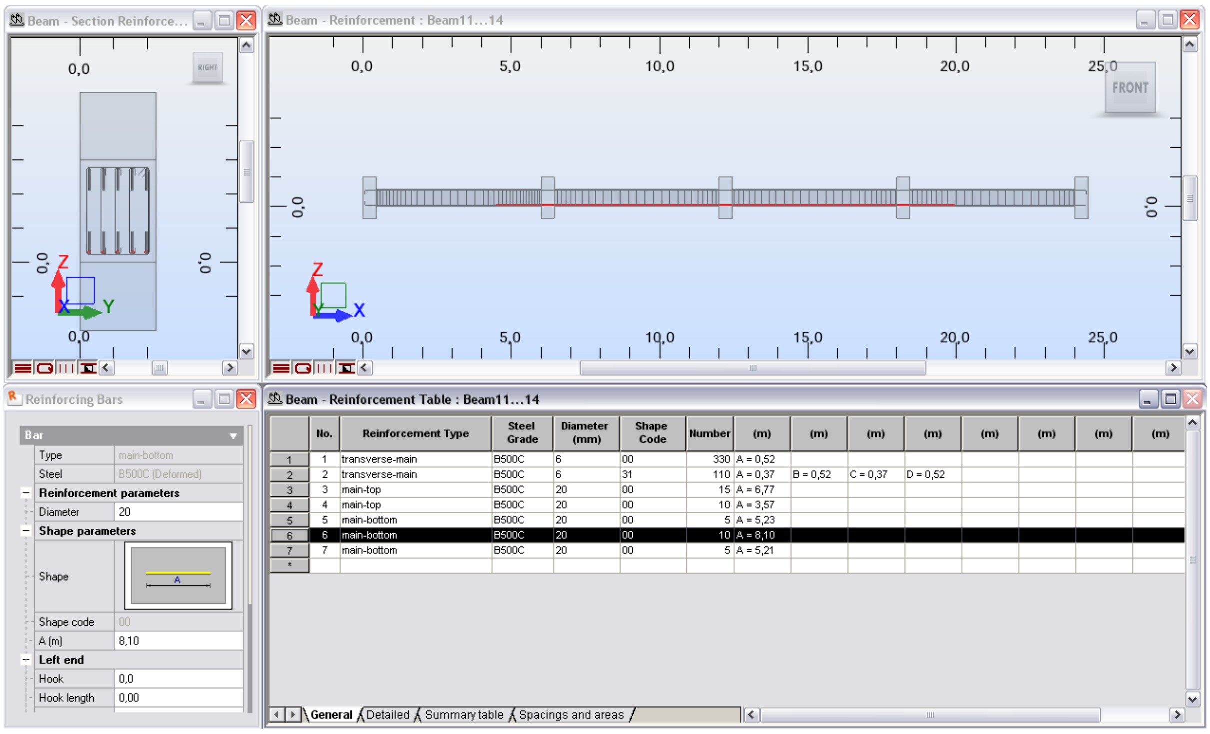
| | |
|---|---|
| LMC on the *Values* tab | |
| RMC on the ***Displacements*** table | Calls up the context menu. |
| *Table Columns* | Selects the *Table Columns* option and opens the dialog box. |
| LMC on the *General* tab, select the *Coordinates* option, **OK** button | Two additional columns containing node coordinates appear. |
| Close the ***Displacements*** table | |

## 1.4 Reinforced Concrete Beam Design

*NOTE: The code calculations are performed according to EN 1992-1-1:2004 AC:2008.*

| | |
|---|---|
| RMC on the graphic viewer and choose the *Select* option from the context menu; select all RC beams from the window | Selects the beams for design. |
| *Analysis menu / Design of RC Structure Elements / RC Beam Design* | Runs a module that allows for concrete beam design. Data on the beam together with the static analysis results will be loaded to this module. |
| *Simple Cases*<br>**OK** | Selects the *Simple Cases* option in the ***Parameters of RC Elements*** dialog box. |
| LMC move to the ***Beam - Section*** viewer | Selects a view presenting the beam section. |
| *Analysis / Calculation Options* | Opens the ***Calculation Options*** dialog box. |
| On the *Concrete* tab select C25/30 from the *Name* field<br>On the *Longitudinal reinf.* unselect dimeter from 6 to 18 mm<br>**OK** | Definition of concrete and steel parameters. Closes the ***Calculation Options*** dialog box. |
| *Analysis / Reinforcement pattern* | Opens the ***Reinforcement patern*** dialog box. |

| | |
|---|---|
| On the *Shapes* tab for *Longitudinal bars – Main* change the *Left and Right hook* value to 90.0<br>**OK** | Definition of reinforcement patern. Closes the ***Reinforcement patern*** dialog box. |
| <br>LMC on the box for selection of the ***RSAP*** program layouts<br>Layout: **RC Beams / Beam - results** | Graphic and tabulated presentation of obtained results (cross section force diagrams for various limit states and diagrams of reinforcement area along the beam's length).<br>*NOTE: Design of an RC beam starts automatically.* |
| <br>**RC Beams / Beam – reinforcement** Layout | Graphic and tabulated presentation of reinforcement in the beam (see the drawing below). |

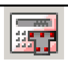


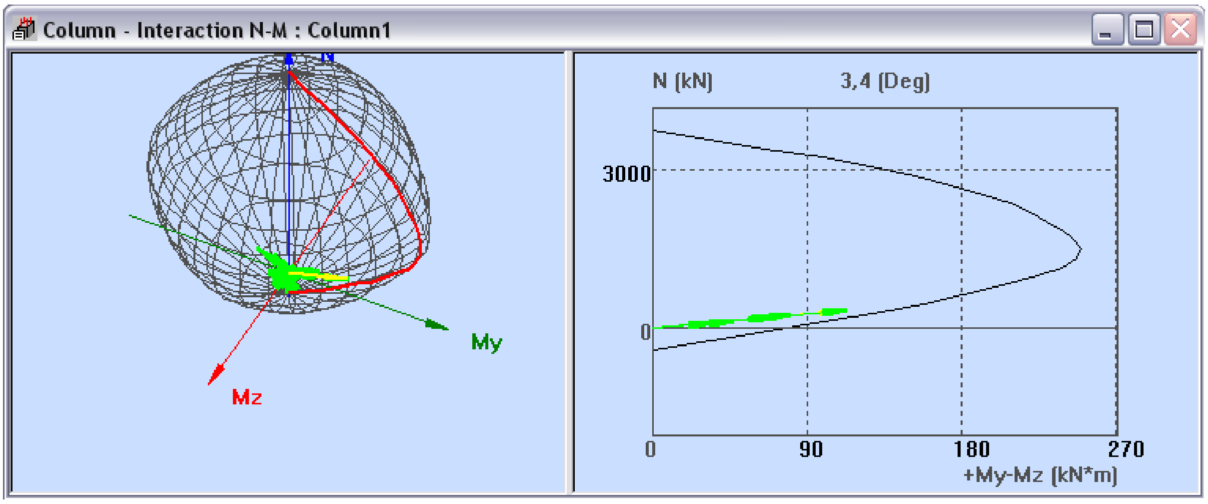

| | |
|---|---|
| *Results menu / Drawings* | Displays a working drawing of the first span of the designed beam. |
| ***RC Beams / Beam - Reinforcement*** | Returns to the **BEAM - REINFORCEMENT** layout |
| *Results menu / Calculation Note*<br>**OK** | Opens the ***Calculation Note*** dialog box where one can select the components of the calculation note and starts the ***RSAP*** program editor for presentation of data and results for the beam. |
| Close the editor with the calculation note | |

## 1.5 Reinforced Concrete Column Design

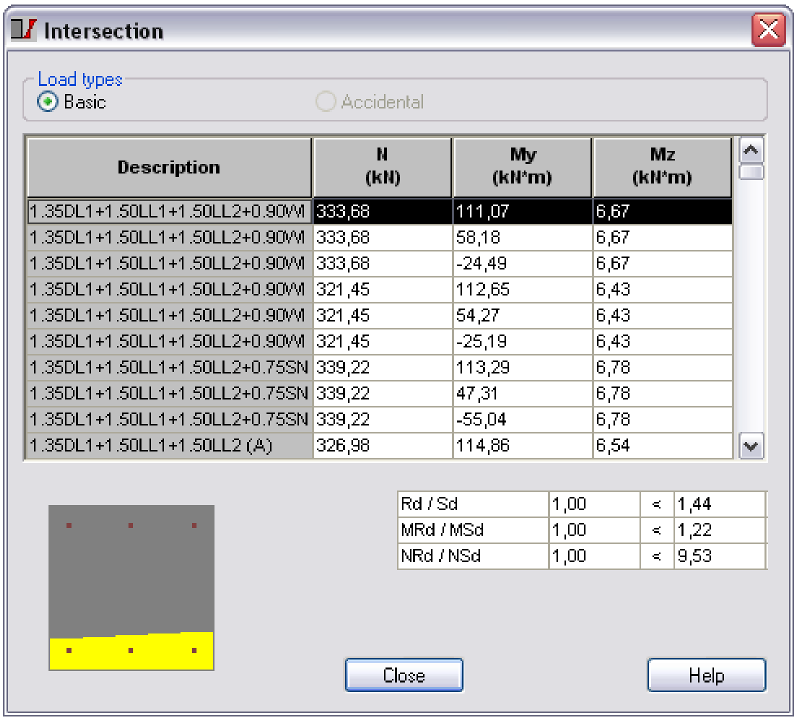
*NOTE: The code calculations are done according to EN 1992-1-1:2004 AC:2008.*

| | |
|---|---|
| <br>**Structure Model / Start** Layout | Selects the **START** layout from the list of available layouts of the ***RSAP*** program |

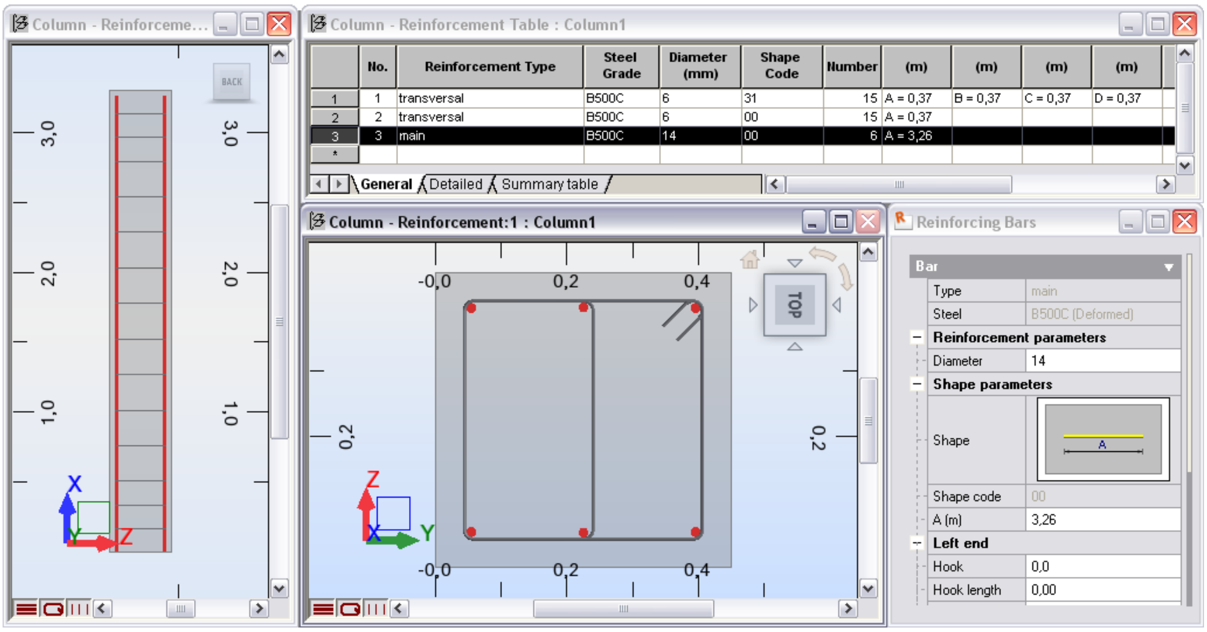
| | |
|---|---|
| While in the graphical viewer RMC and choose the *Select* option; select with the window the outermost bottom left column (bar 1) | Selects the column that will undergo design |
| *Analysis menu / Design of RC Structure Elements / RC Column Design* | Runs module that enables RC column design. Data on the column together with the static analysis results will be loaded to this module. |
| *Simple cases*,<br>**OK** | Selects the *Simple cases* option in the ***Parameters of RC Elements*** dialog box. |
| LMC the ***Column - Section*** viewer | Selects a view presenting the column section. |
| *Analysis menu / Calculation Options* | Opens the ***Calculation Options*** dialog box. |
| On the *Concrete* tab select C25/30 from the *Name* field<br>On the *Longitudinal reinf.* tab unselect dimeter from 6 to 12 mm<br>**OK** | Definition of concrete and steel parameters. Closes the ***Calculation Options*** dialog box. |
|  Select the *Start Calculations* icon from the Standard toolbar | Starts calculations of the reinforcement required according to the adopted parameters. |
| LMC the *Results layout* option in the the ***Calculation Option Set*** *dialog box*<br>**Calculations** | When the calculation are completed the screen presents surfaces (curves) of the interactions N-M, My-Mz. |




| | |
|---|---|
| From the list of available combinations located on the left side of the ***Intersection*** dialog box select the first combination from the top | Presents the column section with the following elements marked on it: neutral axis, compressive and tensile zones together with the appropriate safety factors for the selected combination. |




| | |
|---|---|
| **Close** | Closes the ***Intersection*** dialog box |
|  LMC the field for selection of the ***RSAP*** program layout **RC Columns / Column - reinforcement** | Presents the obtained reinforcement in the column graphically and in the form of a table (see the drawing below) |



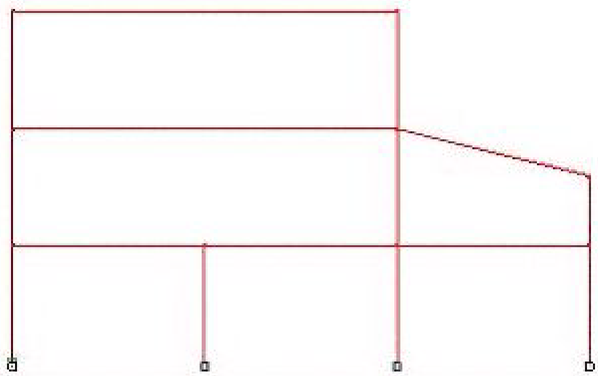

## 1.6 Design of Multiple Reinforced Concrete Members

Code EN 1992-1-1:2004 AC:2008

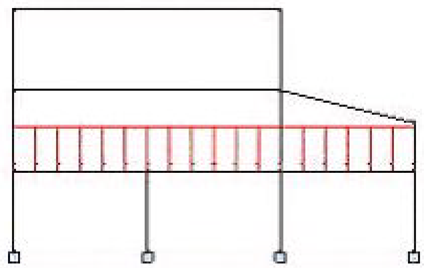
| | |
|---|---|
|   **Structure Model / Start** Layout | The **START** layout is selected from among those provided by the ***RSAP*** program |
| *Analysis menu / Design of RC Structure Elements / RC Member Design / Calculations* | The ***Calculations According to EN 1992-1-1:2004 AC:2008*** dialog box is opened. |
| Introduce the list of bars 1to14 in the *Calculations for:* field (with the *Design* option active) | Selection of members that will undergo the design process |
| Introduce the list of the load cases (1to5) applied to the structure and used during its design into the *Lists of cases* field | Selection of all load cases |
| For the **Calculate** option for beams assume the following parameters:<br>in (11) points | Determination of the parameters of searching for the theoretical (required) area of reinforcement for the selected members of the structure |
| LMC the **Calculate** button | Calculations of the theoretical (required) area of reinforcement for the selected members of the structure and the adopted calculation parameters are started. |
| **Close** in the ***RC Member Calulations: Report*** dialog box | Display of a window containing calculation warnings and errors concerning member theoretical (required) reinforcement |
| Close the ***Calculations According to EN 1992-1-1:2004 AC:2008*** dialog box | |
| *Results menu / Reinforcement / RC Member Reinforcement* | Opens the R*esults* for *required member reinforcement* table in which calculation results of theoretical (required) reinforcement for selected RC member sections will be displayed |
| Close the *Results for required member reinforcement* table | |

# 2. Steel Design – 2D Frame

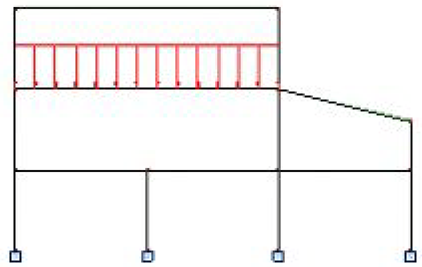
The following is an example of a definition, analysis and design of a simple, 2D steel frame presented in the drawing below.
Data units: (m) and (kN).



Three load cases will be applied to the structure (self-weight and two cases of live loads presented in the drawing below). Moreover, (10) load cases generated automatically for snow/wind loads will be applied to the structure.



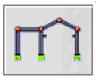
CASE 2



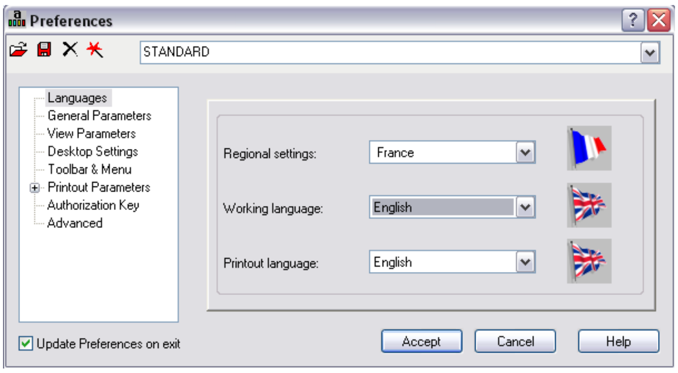
CASE 3

The following rules will be applied during structure definition:

- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

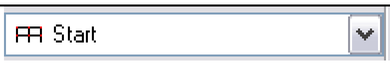
In order to start defining a structure, one should run the ***RSAP*** program (press the relevant icon or select the relevant command from the toolbar). After a while, there appears on screen the dialog box, where one should select the first icon in the first row  (**2D frame**).

NOTE: The French Section Database (RCAT) is used in this example. Set French regional settings in Preferences (*Tools menu / Preferences).*

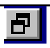


## 2.1 Model Definition

| OPERATION PERFORMED | DESCRIPTION |
|---|---|
| <br>**Structure model / Bars** Layout | The **BARS** layout should be selected from those available in the ***RSAP*** program |
| LMC in the *Bar type* field and select the ***Column*** type<br>LMC in the *Section* field and select the ***HEA 300*** type (if the section is absent from the list of available sections, one should open the ***New section*** dialog box by pressing the  button and select the required section) | Definition of bar properties. The section from the French section database (Catpro) has been used in this example. |
| LMC in the *Beginning* field (the bacground will be highlighted in green) | Beginning of the definition of structure bars (columns of the structure) |
| column 1:<br>Beginning:(0,0) End:(0,5)<br>Beginning:(0,5) End:(0,10)<br>Beginning:(0,10) End:(0,15)<br>column 2:<br>Beginning:(8,0) End:(8,5)<br>column 3:<br>Beginning:(16,0) End:(16,5)<br>Beginning:(16,5) End:(16,10)<br>Beginning:(16,10) End:(16,15)<br>column 4:<br>Beginning:(24,0) End:(24,5)<br>Beginning:(24,5) End:(24,8) | Definition of columns in the frame |
| LMC in the *Bar type* field and select the ***Beam*** type.<br>LMC in the *Section* field and select the type ***HEA 300*** | Beginning of the definition of structure beams and definition of their properties. The section from the French section database (Catpro) has been used in this example. |
| LMC in the *Beginning* field (the background will be highlighted in green) | Beginning of the definition of structure beams |
| beam 1:<br>Beginning:(0,5) End:(8,5)<br>Beginning: (8,5) End:(16,5)<br>Beginning:(16,5) End:(24,5)<br>beam 2:<br>Beginning:(0,10) End:(16,10)<br>beam 3:<br>Beginning:(16,10) End:(24,8)<br>beam 4:<br>Beginning:(0,15) End:(16,15) | Definition of beams in the frame |

| | |
|---|---|
|  LMC in the field for selecting layouts in the ***RSAP*** program and select **Structure model / Start** Layout | Selection of the initial layout of the ***RSAP*** program |
|  Select the *Zoom All* icon from the Standard toolbar | Initial view |
|  Select the *Supports* icon from the Structure Model toolbar | Opening the ***Supports*** dialog box |
| LMC on the *Current selection* field on the *Nodal* tab | Selection of structure nodes where supports will be applied |
| Go to the graphical viewer; while pressing the left mouse button, select all the bottom nodes of columns | The selected nodes 1, 5, 7 and 11 will be introduced into the *Actual selection* field |
| Select the icon denoting a fixed support in the ***Supports*** dialog box (it will get highlighted) | Selection of support type |
| **Apply**, **Close** | The selected support type will be applied to the selected nodes of the structure |

## 2.2 Definition of Load Cases and Loads

| | |
|---|---|
|  Select the *Load Types* icon from the Structure Model toolbar | Opening the ***Load Types*** dialog box |
| LMC on the **New** button | Definition of a case with the dead nature (self-weight) and the standard label DL1 |
| LMC the ***Nature*** field<br>*(Live)* | Selection of the nature of load case: live |
| LMC the **New** button<br>LMC the **New** button | Definition of two load cases with the live nature and standard labels LL1 and LL2 |
| **Close** | Closing the ***Load types*** dialog box |
| *Loads menu / Load Table* | Opening the table for defining loads operating in the defined load cases |
| Press , to place the table in the bottom part of the screen, so that it takes the entire width of the viewer and allows the model of the defined structure to be visible | Reducing the table size in order to make the graphical load definition possible |
| LMC the second cell in the CASE column, select the 2. load case: LL1 | Definition of loads operating in the second load case |

| | |
|---|---|
| Continuing in the same row LMC the cell in the LOAD TYPE column, selection of the uniform load | Selection of load type |
| LMC the cell in the LIST column, graphical selection in the viewer of the beam 1 (bars 10to12) | Selection of bars to which the uniform load will be applied |
| LMC the cell in the "PZ=" column and enter the (-20) value | Selection of the direction and value of the uniform load |
| LMC the third cell in the CASE column, select Load case 3 - LL2 | Definition of loads operating in the third load case |
| LMC the cell in the LOAD TYPE column, select the uniform load | Selection of load type |
| LMC the cell in the LIST column, select graphically the beam 2 (bar 13) | Selection of bars to which the uniform load will be applied |
| LMC the cell in the "PZ=" column and enter the (-14) value | Selection of the direction and value of the uniform load |
| Close the table of loads | |

## 2.3 Definition of Snow/Wind Loads

French code: NV65 Mod99+Carte 96 04/00

| | |
|---|---|
| *Loads menu / Special loads / Wind and Snow 2D/3D* | Opening of the ***Snow and Wind 2D/3D*** dialog box |
| Press the **Auto** button;<br>inactive options:<br>*without parapets*<br>*with base not on ground*<br>*isolated roofs* | Automatic generation of the structure envelope for the generation of snow/wind loads (in the *Envelope* field the program introduces the following node numbers: 1, 2, 3, 4, 10, 9, 13, 12, 11) and definition of basic parameters for the structure envelope |
| Define the following parameters:<br>*Total depth* = (60)<br>*Bay spacing* = (10)<br>active options:<br>w*ind*<br>*snow* | Definition of the basic parameters of snow/wind loads |
| Press the **Parameters** button | Opening the additional dialog box (***Snow/Wind Loads 2D/3D***), where one can define detailed parameters |
| Define the parameters of snow/wind load:<br>*Global parameters* tab:<br>Departament: Alpes-Maritimes<br>altitude above the sea level: (200)<br>structure height: (15) m<br>reference level: (0.8) m<br>rise of roof: automatic | Definition of global parameters |

| | |
|---|---|
| *Wind* tab:<br>Site: Normal<br>Type: Normal<br>Wind pressure: automatic<br>Structure dimension effect: automatic<br>inactive options in the *Specific actions* group | Definition of parameters for wind loads |
| *Snow* tab:<br>Snow pressure: automatic for normal and extreme<br>active option: *Snow redistribution* | Definition of parameters for snow loads |
| **Generate** | Pressing the button results in starting the generation of snow and wind loads with the accepted parameters. The calculation note will appear on screen. It will present the parameters of snow/wind laod cases |
| Close editor with the calculation note | |
| Close the ***Snow and Wind 2D/3D*** dialog box | |

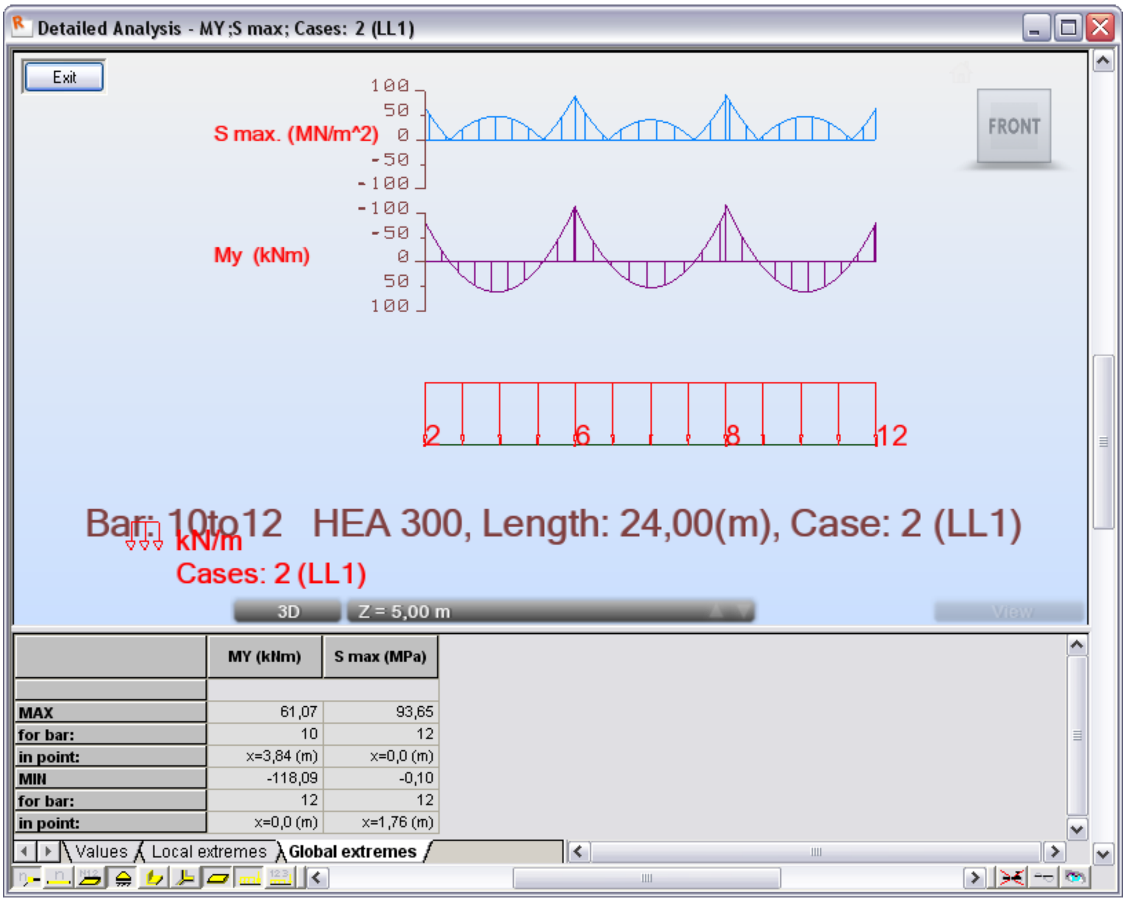
## 2.4 Structural Analysis

| | |
|---|---|
|  Select the *Calculations* icon from the Standard toolbar | Calculations of the defined structure are started. Once they are completed, the upper bar of the ***RSAP*** program will display the message: *Results (FEM) : available*. |

## 2.5 Detailed Analysis

| | |
|---|---|
| Select beam 1 in the graphical viewer (bars 10,11,12) | |
|  <br>LMC the ***RSAP*** program layout selection:<br>**Results / Detailed Analysis** Layout | Detailed analysis of structure bars is commenced. The monitor screen is divided into two parts: the graphical viewer presenting the structure model and the ***Detailed Analysis*** dialog box |
| Select the second load case<br>  | |
| In the ***Detailed Analysis*** dialog box select option *Open a new window* located in the lower left corner,<br>on the *NTM* tab select the *MY Moments* option | Selection of the quantities to be presented for the selected beam |

| | |
|---|---|
| **Apply** | An additional graphical viewer appears on screen. It consists of two parts: a graphical presentation of information (diagrams, loads, bar sections) for selected bars and a table presenting numerical results obtained for selected bars |
| In the ***Detailed analysis*** dialog box select the following options:<br>Select the maximum stress *Smax* on the *Stresses* tab<br>Select Characteristic points on the *Division points* tab,<br>LMC in **Refresh** | Selection of the quantities to be presented for the selected beam |
| **Apply** | Adds new quantities to be presented for the selected beam |
| Select the *Global extremes* tab in the table | Activates presentation of global extremes obtained for the selected beam (see figure below). |

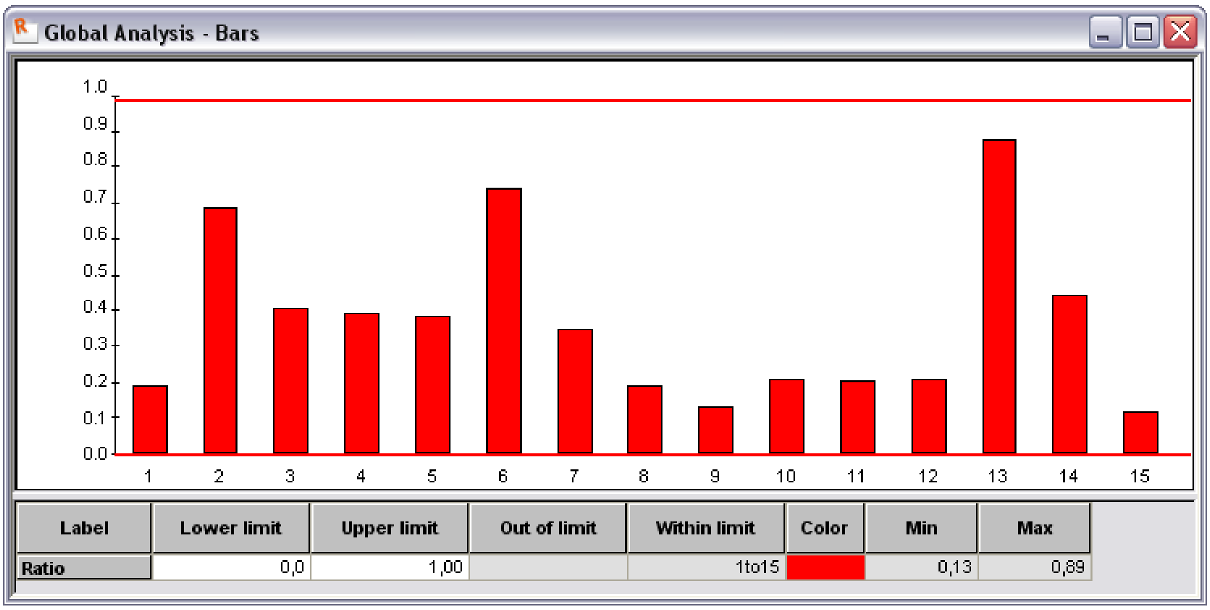



| | |
|---|---|
| **Exit** | Closing the viewer presenting the detailed analysis of the selected beam |

## 2.6 Global Analysis

| | |
|---|---|
| <br>LMC the ***RSAP*** program layout selection:<br>**Structure Model / Start** Layout | Selection of the initial ***RSAP*** layout. |

| | |
|---|---|
| *Results menu / Global Analysis - Bars* | Beginning of the global analysis of all the bars in the structure. An additional graphical viewer appears. It consists of two parts: the graphical presentation of information and the table presenting the numerical results |
| RMC while the cursor is located in the additional graphical viewer | A context menu appears on screen |
| *Table Columns* | Selection of this option in the context menu opens the ***Parameters of presentation windows*** dialog box |
| On *Stresses* tab inactivate all check boxes.<br>*Design* tab: activate the *Ratio* option | Selection of quantites for which global analysis will be presented |
| LMC the **OK** button | The selection is accepted |
| LMC the *Upper limit* in the table and enter the value 1.0 | The upper value of the ratio is determined |
| RMC while the cursor is located in the additional graphical viewer | A context menu appears on screen |
| Select the *Constant display of limit values* option | The values of limits are presented with horizontal lines in the graphical viewer of global analysis (see below). |

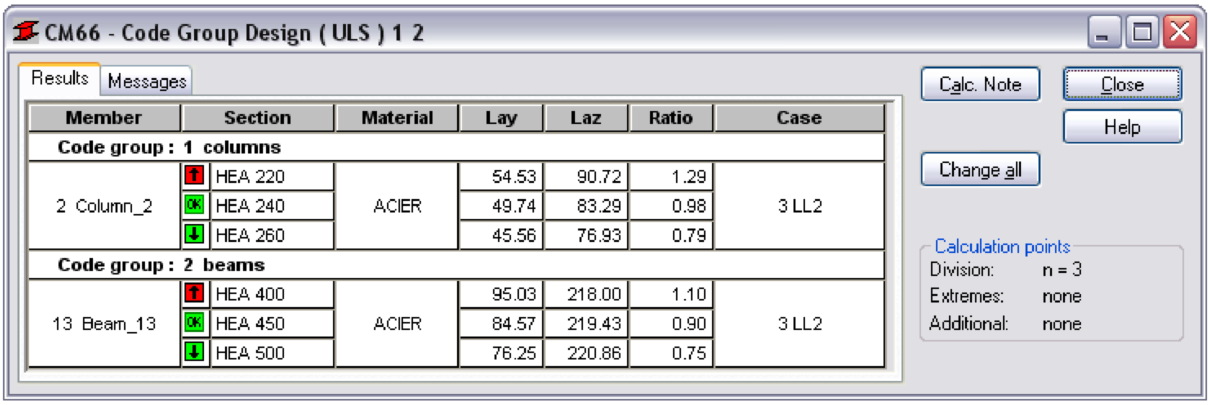



| | |
|---|---|
| **Close** | Close the graphical viewer with global analysis presented |

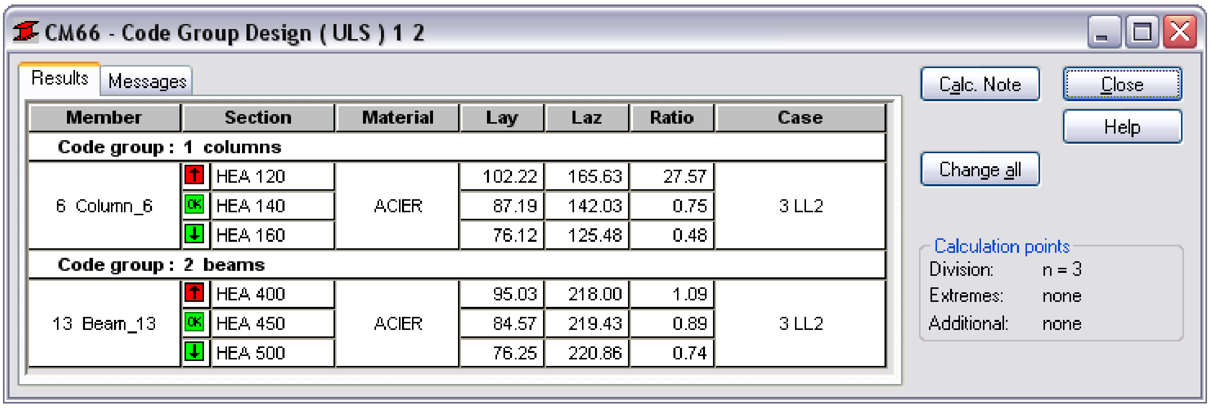
## 2.7 Steel Design

CM66 code

| | |
|---|---|
|   LMC the ***RSAP*** program layout selection:<br>**Structure design / Steel/Aluminum design** Layout | Design of steel structure members is commenced. The monitor screen is divided into three parts: the graphical viewer, the ***Definitions*** dialog box and the ***Calculations*** dialog box |
| LMC the **New** button on the *Groups* tab in the ***Definitions*** dialog box | Definition of member groups is commenced |
| Define the first group with the following parameters:<br>Number: 1<br>Name: columns<br>Member list: 1to9<br>Material: ACIER Defaut | Definition of the first group consisting of all the columns in the structure |
| **Save** | Saving the parameters of the first member group |
| LMC the **New** button on the *Groups* tab in the ***Definitions*** dialog box | Definition of the second group |
| Define the second group with the following parameters:<br>Number: 2<br>Name: beams<br>Member list: 10to15<br>Material: ACIER Defaut | Definition of the first group consisting of all the beams in the structure |
| **Save** | Saving the parameters of the first member group |
| LMC the **List** button in the *Code group design* line in the ***Calculations*** dialog box | Going to the ***Calculations*** dialog box and opening the ***Code Group Selection*** dialog box |
| LMC the **All** button (in the field above the **Previous** button, there will appear the list: 1to2), **Close** | Selection of the member groups to be designed |
| LMC the **List** button in the *Loads* group (***Calculations*** dialog box) | Opening the ***Load Case Selection*** dialog box |
| LMC the field above the **Previous** button; define the list: 1to3, **Close** | Selection of the first three load cases (DL1, LL1, and LL2) |
| Activate the option: *Optimization*<br>and Limit state: *Ultimate*<br>Inactive the option: *Save calculation results* | Group design will use the optimization procedures (appropriate sections with respect to their weight); the ultimate limit state will be checked |
| LMC the **Calculations** button | Design of the selected member groups is commenced; there appears the ***CM66 – Code Group Design*** dialog box on screen |

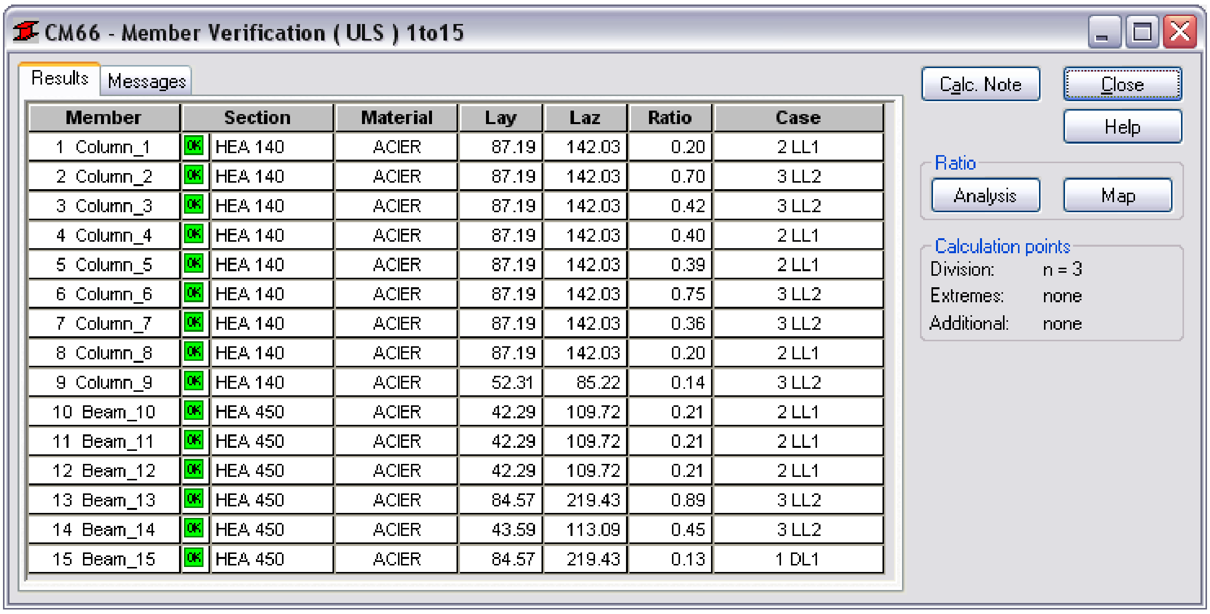



| | |
|---|---|
| LMC the **Change all** button in the ***Code Group Design*** dialog box shown above; accept the warning about the possible change of the result status to 'not available' | Change of the currenly used profiles in the members belonging to both member groups to the calculated sections (for columns: from HEA 300 to HEA 240, for beams: from HEA 300 to HEA 450). Once the sections are changed, the upper bar of ***RSAP*** will display the following message: *Results (FEM) : out of date*. |
| **Close** | Closing the ***Code Group Design*** dialog box |
|  Select the *Calculations* icon from the Standard toolbar | Recalculation of the structure with the changed member sections. Once the sections are changed, the upper bar of ***RSAP*** will display the following message: *Results (FEM) : available*. |
| LMC the **Calculations** button in the ***Calculations*** dialog box | Re-design of the selected member groups in the structure (1,2) with the optimization options active; there will appear the **Short results** viewer |
| LMC the **Change all** button in the ***Code group design*** dialog box; accept the warning about the possible change of the result status to 'not available' | Change of the currenly used profiles in the members belonging to both member groups to the calculated sections. Once the sections are changed, the upper bar of ***RSAP*** will display the following message: *Results (FEM) : out of date*. |
| **Close** | Closing the ***Code Group Design*** dialog box |
|  Select the *Calculations* icon from the Standard toolbar | Recalculation of the structure with the changed member sections. Once the sections are changed, the upper bar of ***RSAP*** will display the following message: *Results (FEM) : available*. |
| LMC the **Calculations** button in the ***Calculations*** dialog box | Re-design of the selected member groups in the structure (1,2) with the optimization options active; there will appear the **Short results** viewer shown below. When the sections do not change during group design one can say the calculated sections are the optimal sections for designing member groups.You have to repeat this re-desing process as long you will see the below results. |




| Member | Section | Material | Lay | Laz | Ratio | Case |
|---|---|---|---|---|---|---|
| Code group : 1 columns | | | | | | |
| 6 Column_6 | HEA 120 | ACIER | 102.22 | 165.63 | 27.57 | 3 LL2 |
| | HEA 140 | | 87.19 | 142.03 | 0.75 | |
| | HEA 160 | | 76.12 | 125.48 | 0.48 | |
| Code group : 2 beams | | | | | | |
| 13 Beam_13 | HEA 400 | ACIER | 95.03 | 218.00 | 1.09 | 3 LL2 |
| | HEA 450 | | 84.57 | 219.43 | 0.89 | |
| | HEA 500 | | 76.25 | 220.86 | 0.74 | |

| | |
|---|---|
| **Close** | Closing the ***Code Group Design*** dialog box |
| LMC in the *Member verification* field in the ***Calculations*** dialog box and enter there: (1to15) | Selection of members to be verified |
| LMC the *Load case list* field in the ***Calculations*** dialog box and enter there: (1to3) | Selection of all load cases |
| LMC the **Calculations** button | Verification of the selected structure members is started (the verification is performed to obtain the results for particular structure members; however, it is not necessary); there will appear the **Short results** viewer |
| **Close** | Closing the ***Member Verification*** dialog box |




| Member | Section | Material | Lay | Laz | Ratio | Case |
|---|---|---|---|---|---|---|
| 1 Column_1 | HEA 140 | ACIER | 87.19 | 142.03 | 0.20 | 2 LL1 |
| 2 Column_2 | HEA 140 | ACIER | 87.19 | 142.03 | 0.70 | 3 LL2 |
| 3 Column_3 | HEA 140 | ACIER | 87.19 | 142.03 | 0.42 | 3 LL2 |
| 4 Column_4 | HEA 140 | ACIER | 87.19 | 142.03 | 0.40 | 2 LL1 |
| 5 Column_5 | HEA 140 | ACIER | 87.19 | 142.03 | 0.39 | 2 LL1 |
| 6 Column_6 | HEA 140 | ACIER | 87.19 | 142.03 | 0.75 | 3 LL2 |
| 7 Column_7 | HEA 140 | ACIER | 87.19 | 142.03 | 0.36 | 3 LL2 |
| 8 Column_8 | HEA 140 | ACIER | 87.19 | 142.03 | 0.20 | 2 LL1 |
| 9 Column_9 | HEA 140 | ACIER | 52.31 | 85.22 | 0.14 | 3 LL2 |
| 10 Beam_10 | HEA 450 | ACIER | 42.29 | 109.72 | 0.21 | 2 LL1 |
| 11 Beam_11 | HEA 450 | ACIER | 42.29 | 109.72 | 0.21 | 2 LL1 |
| 12 Beam_12 | HEA 450 | ACIER | 42.29 | 109.72 | 0.21 | 2 LL1 |
| 13 Beam_13 | HEA 450 | ACIER | 84.57 | 219.43 | 0.89 | 3 LL2 |
| 14 Beam_14 | HEA 450 | ACIER | 43.59 | 113.09 | 0.45 | 3 LL2 |
| 15 Beam_15 | HEA 450 | ACIER | 84.57 | 219.43 | 0.13 | 1 DL1 |

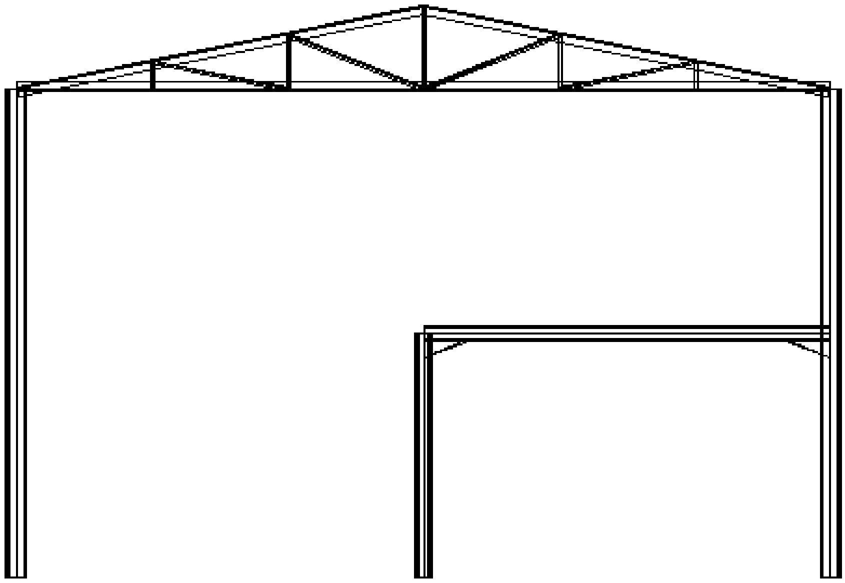
## 2.8 Printout Composition

| | |
|---|---|
| *File Menu / Printout Composition* | Opening the ***Printout Composition - Wizard*** dialog box, where one can define the shape of the printout for the currently designed structure |
| LMC the *Simplified printout* tab | Go to the *Simplified printout* tab |

| | |
|---|---|
| Switch off the options (symbol  disappears):<br>*Quantity survey, Combinations* | Data concerning quantity survey and combinations will not be included in the printout |
| Select the following data from the available lists:<br>Reactions - global extremes<br>Displacements - envelope<br>Forces - values<br>Stresses - envelope | Selection of the data to be presented for the results of structure calculations |
| LMC the **Save template** button | Pressing this button results in going to the *Templates* tab in the ***Printout composition - Wizard*** dialog box and including the selected simplified printout elements in the right panel. |
| LMC the *Standard* tab | Going to the *Standard* tab |
| Highlight the option in the left panel:<br>*Member Group Design* | Selection of elements for prinout composition |
| LMC the **Add** button | Going to the selected option in the right panel |
| LMC the **Preview** button | Presentation of the print preview of the defined printout for the designed structure |
| **Close** | Closing the print preview viewer |
| **Close** | Closing the ***Printout Composition - Wizard*** dialog box |

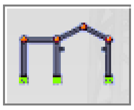
# 3. Elasto-Plastic Analysis

This example presents definition, analysis and design of a simple 2D steel frame shown in the figure below. The definition process involves application of the truss generated by means of the library of typical structures available in the **Autodesk® Robot™ Structural Analysis Professional** program. The model considers the EuroCode code requirements with respect to geometrical imperfections and elasto-plastic material analysis.
Data units: (m) and (kN).



The following rules apply during structure definition:

- any icon symbol means that the relevant icon is pressed with the left mouse button,
- { x } stands for selection of the 'x' option in the dialog box,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

To run structure definition start the ***RSAP*** program (press the appropriate icon or select the command from the taskbar). In the vignette that will be displayed on the screen the first icon  ***(Frame 2D Design***) should be selected.

## 3.1 Model Definition

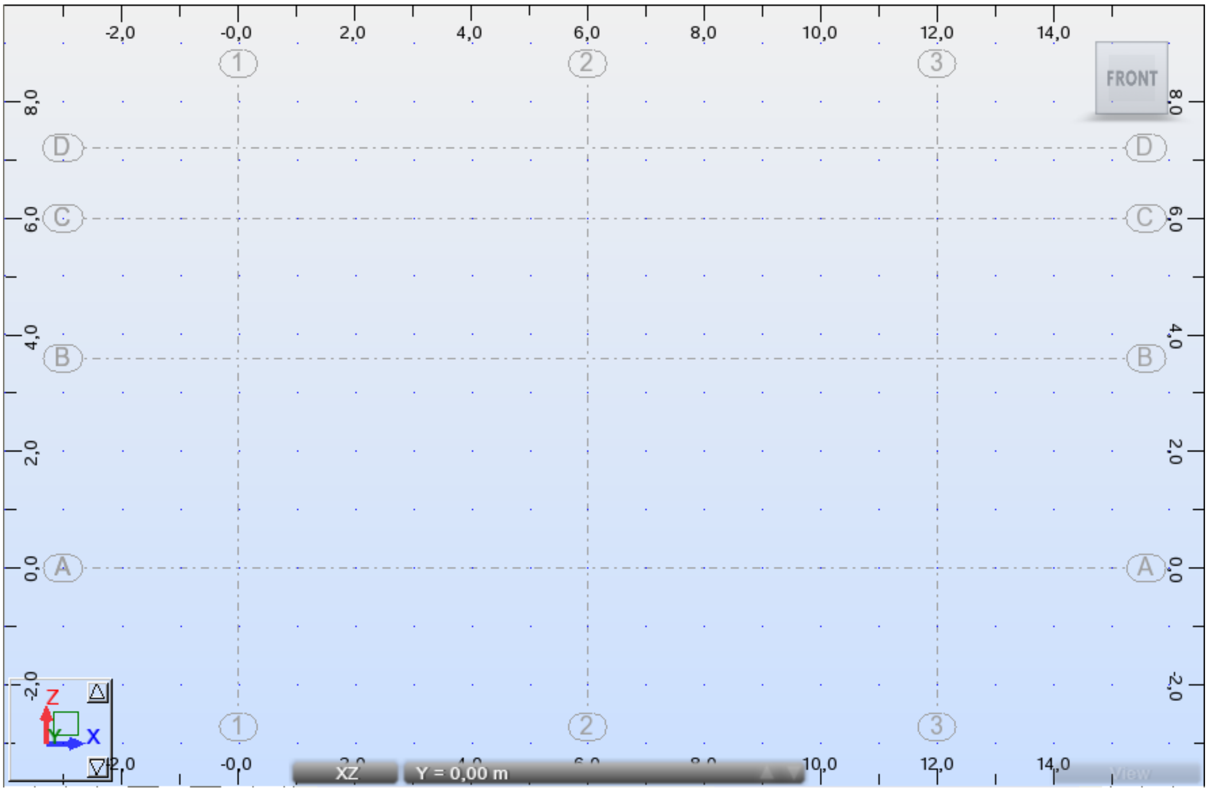
### *3.1.1 Code Selection*

| PERFORMED OPERATION | DESCRIPTION |
|---|---|
| *Tools menu / Job Preferences* | Opens the ***Job Preferences*** dialog box |
| *Materials* | Selects the *Materials* option from the tree in the dialog box |
| Selection from the *Materials* unfolding list: *Eurocode* | Selects the Eurocode material database |
| *Design codes* | Selects the Design *codes* option from the tree in the dialog box |
| *Steel / Aluminum structures: (EN 1993-1:2005)* | Selects EuroCode for steel structure design |
| *Loads* | Selects the *Loads* option from the tree in the dialog box |

| | |
|---|---|
| *Code combinations:*<br>*EN 1990:2002*<br>Accept warnings of the code changes | Selects EuroCode for automatic code combinations |
| **OK**<br>Accept warnings of the code changes | Accepts adopted parameters and closes the ***Job preferences*** dialog box<br>Accept warnings of the code changes. |

### *3.1.2 Structural Axis Definition*

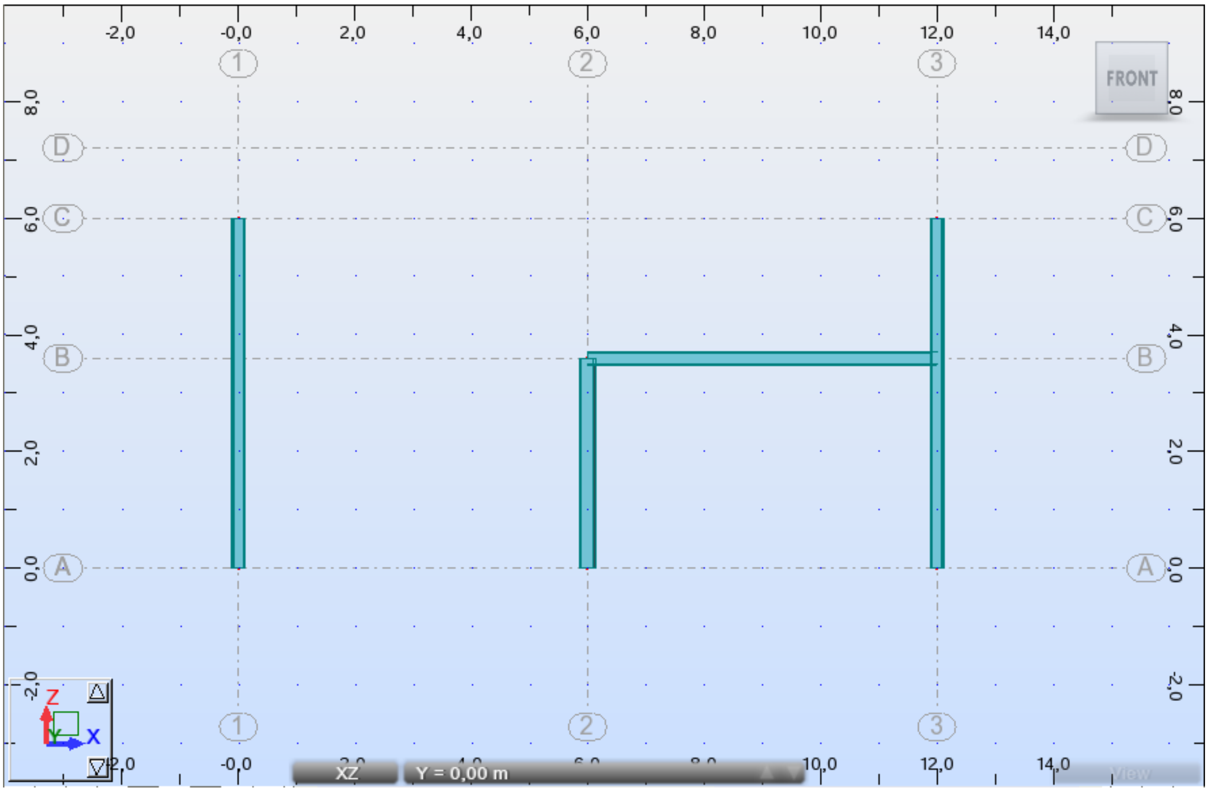
| | |
|---|---|
| *Geometry menu / Axis definition* | Starts definition of structural axes. The ***Structural Axis*** dialog box is displayed on the screen |
| On the X tab:<br>Position: {0}<br>Number of Repetitions: {2}<br>Distance: {6}<br>Numbering: 1, 2, 3 ... | Defines parameters of the vertical structural axes |
| LMC the **Insert** button | Vertical axes have been defined and are entered to the *Defined axes* field |
| LMC the Z tab | Starts defining parameters of the horizontal structural axes |
| On the Z tab:<br>Position: {0.0}<br>Numbering: A, B, C ... | Defines parameters of the horizontal structural axes |
| LMC the **Insert** button | First horizontal axis has been defined and entered to the *Defined axes* field |
| Position: {3.6}, **Insert** | |
| Position: {6.0}, **Insert** | |
| Position: {7.2}, **Insert** | The remaining axes have been defined and entered to the *Defined axes* field |
| **Apply**, **Close** | Generates defined structural axes and closes the ***Structural Axis*** dialog box. The structural axes presented in the figure below are displayed on the screen. |



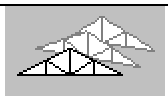

### *3.1.3 Member Definition*

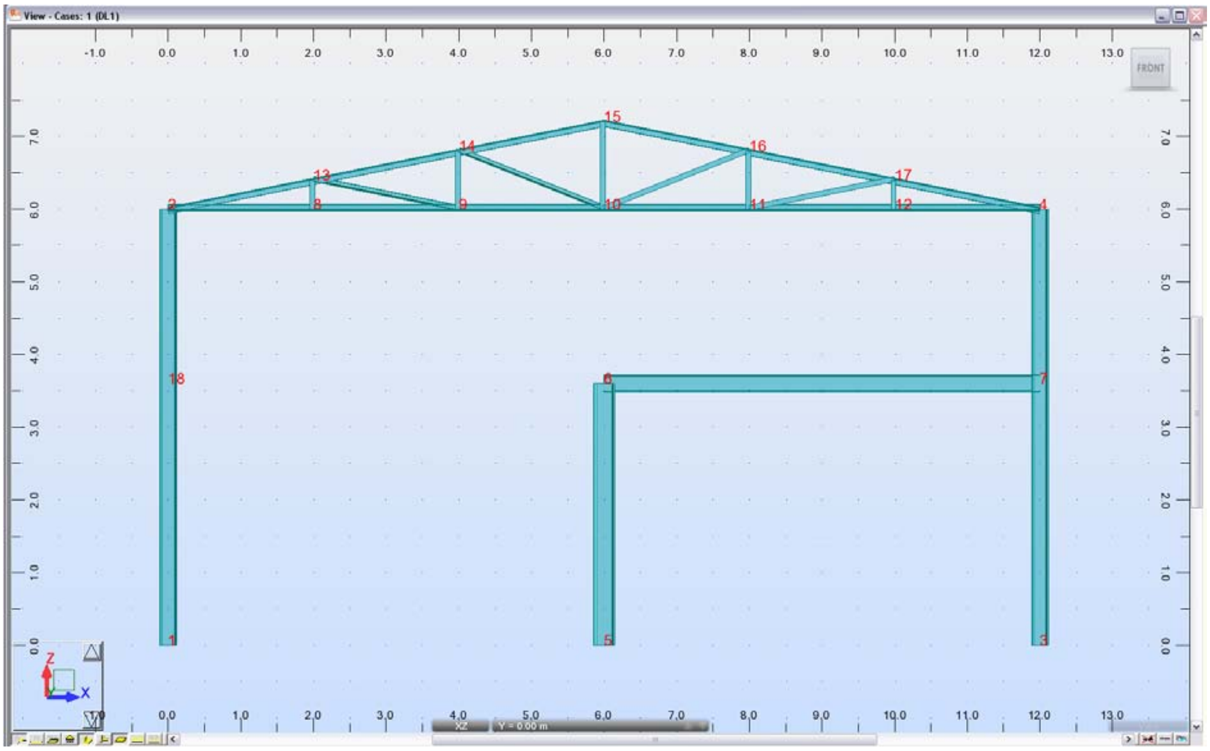
| | |
|---|---|
| *Geometry / Properties / Sections* | Opens the ***Sections*** dialog box |
| Select the *New section definition* icon. | Opens the ***New Section*** dialog box |
| Select the I-section family,<br>in the *Section* field select section<br>IPE 240, **Add**<br>HEA 300, **Add**<br>HEA 240, **Add** | Defines the following sections: *IPE 240*, *HEA 240* and *HEA 300* |
| **Close** (***New Section*** dialog box)<br>**Close** (***Sections*** dialog box) | Closes the ***Sections*** and ***New Section*** dialog boxes |
| Select the *Bars* icon from the Structure Model toolbar. | Opens the ***Bars*** dialog box |
| LMC the *Bar type* field and select type: *Column* | Selects properties of a bar to be designed. The *Section* field should show the recently-defined section *HEA 240* |
| LMC the *Beginning* field (the field background changes to green) | Starts defining bars in the structure (structure columns) |
| Indicate graphically or type the points of the beginning and end of bars<br>(0,0) (0,6), **Add**<br>(12,0) (12,6), **Add** | Defines columns positioned on the structural lines marked with numbers 1 and 3 (in the A-C range) |

| | |
|---|---|
| LMC the *Section* field and select section *HEA 300* | Selects *HEA 300* as the current section |
| LMC the *Beginning* field (the field background changes to green) | Starts defining bars in the structure (middle column) |
| Indicate graphically or type the points of bar beginning and end (6,0) (6,3.6), **Add** | Defines a column positioned on the structural line marked with number 2 (in the A-B range) |
| LMC the *Bar type* field and select the type: *Beam* | Selects properties of a bar to be designed. |
| LMC the *Section* field and select section *IPE 240* | Selects *IPE 240* as the current section |
| LMC the *Beginning* field (the field background changes to green) | Starts defining bars in the structure (a beam between the columns) |
| Indicate graphically or type the points of bar beginning and end (6.0,3.6) (12.0,3.6), **Add** | Defines a beam positioned on the structural line marked with letter B (in the 2-3 range) |
| **Close** | Closes the ***Bars*** dialog box |
| *View menu / Display* | Opens the ***Display*** dialog box |
| *Bars* tab, switch on the *Section - shape* option<br>**Apply** | |



### *3.1.4 Library Structure Definition*

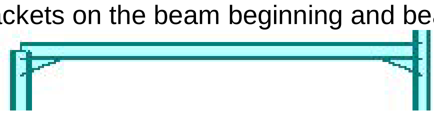
| | |
|---|---|
| *Nodes* tab, switch on the *Node numbers* option<br>*Structure* tab, switch off the *Structural axis* option<br>**Apply**, **OK** | |
|  Select the *Library Structure* icon from the Structure Model toolbar. | Opens the ***Typical Structures*** dialog box and starts defining a library structure |
| LMC (twice) the  icon (first icon in the last row) | Selects the triangular truss of the 1 type. The ***Merge Structure*** dialog box is displayed on the screen in which truss parameters may be defined. |
| On the *Dimensions* tab<br>LMC the *Length L* field<br>{12} | Defines truss length (it may also be defined graphically in the graphical viewer) |
| LMC the *Height H* field<br>{1.2} | Defines truss height (it may also be defined graphically in the graphical viewer) |
| LMC the option:<br>*Moments Released*: *No* | |
| LMC on the *Sections* tab;<br>To all truss chords (upper and lower) assign (DCED 90x10) and to diagonals, posts asign (CAE 70x7) | Assigns the section to the truss bars. |
| LMC the *Insert* tab | |
| LMC the *Insertion point* field<br>Indicate graphically node no. 2 of the coordinates (0, 0, 6) | Defines the beginning node of the truss |
| **Apply** | Considers the data entered, data modification is possible |
| **OK** | Generates the defined truss and closes the ***Merge Structure*** dialog box. The structure defined is shown in the figure below. |




### *3.1.5 Auxiliary Node Addition*

| | |
|---|---|
| *Edit menu / Divide* | Opens the ***Division*** dialog box |
| The *Division* field LMC<br>• *in distance* | Selects the manner of defining the insertion of the division node - through a coordinate on the bar length |
| In the *Distance from the top* field enter the value 3.6 (m) | Determines the point where the auxiliary node is to be inserted |
| Move to the graphical viewer and indicate (LMC) the left column at its base (bar no.1) | Indicates the bar to be divided. Note: if the division through the coordinate on the bar length is defined, take note that the coordinate is calculated from the indicated bar beginning. |
| **Close** | Closes the ***Division*** dialog box |

### *3.1.6 Brackets on Bars Definition*

| | |
|---|---|
| *Geometry menu / Additional Attributes / Brackets* | Opens the ***Brackets*** dialog box |
| LMC the field with the list of defined attributes, select the default one *Bracket_ 0.1x1* | Selects the bracket type (it will be highlighted) |
| Move to the graphical viewer; indicate beginning and end of the beam (no. 4) | Defines brackets on the beam beginning and beam end  |
| **Close** | Closes the ***Brackets*** dialog box |

### *3.1.7 Support Definition*

| | |
|---|---|
|  Select the *Supports* icon from the Structure Model toolbar. | Opens the ***Supports*** dialog box |
| In the ***Supports*** dialog box select *Fixed* support | Select the fixed support type (it will be highlighted) |
| Move to the graphical viewer; indicate node no.1 (the bottom node of the extreme column) | Assigns the support at node no. 1. |
| In the ***Supports*** dialog box select *Pinned* support | Select the pinned support type (it will be highlighted) |
| Move to the graphical viewer; indicate nodes nos. 3 and 5 (bottom nodes of the remaining columns) | Assigns the supports at nodes nos. 3 and 5. |
| **Close** | Closes the ***Supports*** dialog box |

### *3.1.8 Definition of Geometrical Imperfections*

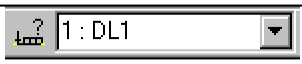
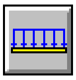
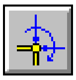
| | |
|---|---|
| *Geometry menu / Additional Attributes / Geometrical Imperfections* | Opens the ***Geometrical Imperfections*** dialog box |
|  Select *Definition of a new type of geometrical imperfection* icon. | Opens the ***Imperfection definition*** dialog box |
| In the *Label* field enter *During_Assembly*<br>switch off the *Automatic* option<br>switch on the *User-defined* option<br>switch on the *Absolute* option<br>enter the value 5 (cm) | Defines parameters of a new imperfection type with the deflection value equal to 5 cm. |
| **Add**, **Close** | Defines the imperfection and closes the ***Imperfection definition*** dialog box |
| LMC in the field with the list of defined attributes, select the default imperfection type (*Automatic*) | Selects the imperfection type (it will be highlighted) |
| Move to the graphical viewer; indicate bar no. 1 (left column) | Defines the imperfection (automatic imperfection according to EC3) for the column |
| LMC the field with the list of defined attributes, select the defined imperfection type *During_Assembly* | Selects the imperfection type (it will be highlighted) |
| Move to the graphical viewer; select the bottom truss chord (bar no. 5) | Defines imperfection (defined by the user) for the bottom truss chord |

| | |
|---|---|
| **Close** | Closes the ***Geometrical Imperfections*** dialog box |

### *3.1.9 Load Case Definition*

| | |
|---|---|
|  in the bottom status bar | Restores a default set of attribute display |
| *Loads menu / Load Types* | Opens the ***Load Types*** dialog box |
| LMC the **New** button | Defines the load case with the nature: dead and standard name DL1 |
| LMC the *Nature* field: *Live (Live 1)* | Selects the load case nature: live |
| LMC the **New** button | Defines the load case with the nature: live and standard name LL1 |
| LMC the **Close** button | Closes the ***Load Types*** dialog box |

### *3.1.10 Load Definition for Generated Cases*

| | |
|---|---|
|   select 1: DL1 | Selects case no. 1 - self-weight load DL1. |
| *Loads menu / Load Definition* | Opens the ***Load Definition*** dialog box. |
| Select the *Bar* tab  | Selects ***Uniform load***. |
| *Values:* <br> *pZ*: {-3} (kN/m), <br> **Add** | Defines the value of the uniform load on the bar |
| LMC the *Apply to* field - enter the bars of the external envelope: <br> 1 2 6 7 | Defines the uniform load on the indicated bars - it models the weight of wall and roof cladding. |
| **Apply** | Defines a load applied to the list of bars |
|   select 2: LL1 | Selects the live load case LL1. |
| Select the *Node* tab  | Selects the ***Nodal Force*** load |
| *Load parameters*, <br> *X*: {10} (kN) <br> *Z*: {-100} (kN) | Defines values of the nodal load. |
| LMC the **Add** button provided in the bottom part of the dialog box | |

| | |
|---|---|
| Move to the graphical viewer presenting the structure view and indicate (LMC) nodes no. 6 and 18 | Defines the nodal load which models the overhead traveling crane load. |
| **Close** | Closes the ***Load Definition*** dialog box. |

### *3.1.11 Snow/Wind Load Generation*

| | |
|---|---|
| *Loads menu / Special loads / Wind and Snow 2D/3D* | Opens the ***Snow and Wind 2D/3D*** dialog box |
| Press the **Auto** button<br>the *Total depth* field: 30,<br>the *Bay spacing* field: 6 (m) | Automatically generates the external structure envelope for generation of snow/wind loads |
| Press the **Parameters** button | Opens the additional dialog box ***(Snow/Wind Loads 2D/3D)*** in which detailed parameters may be defined. The default parameters will be adopted. |
| **Generate, OK** | Pressing this button starts generation of snow/wind loads for the adopted parameters. On the screen calculation notes will be displayed presenting parameters of snow and wind load cases. |
| Close the text editor with the calculation notes | New load cases have been generated (wind and snow loads). |
| **Close** | Closes the ***Snow and Wind 2D/3D*** dialog box |

### *3.1.12 Automatic Code Combinations Generation*

| | |
|---|---|
| *Loads menu / Automatic Combinations* | Opens the ***Load Case Code Combinations*** dialog box according to ***EN 1990:2002***. |
| Select *Full automatic combinations* | Selecting this option and clicking OK generates full code combinations after static structure calculations. |
| **More >**<br>On the Combinations tab switch off *ACC* and *FEU* options | |
| **Generate** | Closes the code combination dialog box and defines combinations. |

## 3.2 Structural Analysis and Result Verification

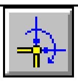
| | |
|---|---|
| *Analysis menu / Calculations* | Runs calculations. |
| *Results menu / Stresses* | Opens the bar stress table. |

| | |
|---|---|
| LMC the *Global extremes* tab located at the bottom of the table area. | Calculates maximal stresses in bars. |
| LMC  in the top right table corner | Closes the table. |
| *File menu / Save* | Opens the dialog box for saving. |
| In the File name field enter the selected name of the example, e.g. *Frame_EC3* | The default saving format: RTD. |
| LMC the **Save** button | Saves the example. |

## 3.3 Elasto-Plastic Analysis

In addition, the analysis of accidental hitting the workshop column by the overhead traveling crane will be performed. In this case, the analysis in the plastic range will be considered.

### *3.3.1 Change of Load Case Definitions*

| | |
|---|---|
| *Loads menu / Load Types* | Opens the ***Load Types*** dialog box |
| LMC the **Delete all** button | Deletes all load cases |
| LMC the **New** button | Defines the load case with the default nature and the standard name DL1 |
| **Close** | Closes the ***Load Types*** dialog box |
| *Loads menu / Load Definition* | Opens the ***Load Definition*** dialog box. |
|  | Selects the ***Nodal Force*** load. |
| *Load parameters*,<br>*X*: {120} (kN)<br>*Z*: {0} | Defines values of the nodal load. |
| LMC the **Add** button provided in the bottom part of the dialog box | |
| Move to the graphical viewer with the structure view and indicate (LMC) node no. 18 | Defines the nodal load - it models the accidental load resulting from the overhead traveling crane. |
| **Close** | Closes the ***Load Definition*** dialog box. |

### *3.3.2 Structural Analysis*

| | |
|---|---|
| *Analysis menu / Calculations* | Runs calculations |
| Locate the mouse cursor on the extreme column (bar 1) so that it becomes highlighted,<br>RMC | Opens the context menu of the structure view. |
| *Object Properties* | Activates the ***Bar properties*** option containing information about bar no.1. |
| The *Code check* tab | Performs the simplified design of the steel bar.<br>As it can be seen, it does not satisfy the conditions of code verification. |
| **Close** | Closes the ***Bar properties*** dialog box. |

### *3.3.3 Change of Bar Sections for Elasto-Plastic Analysis*

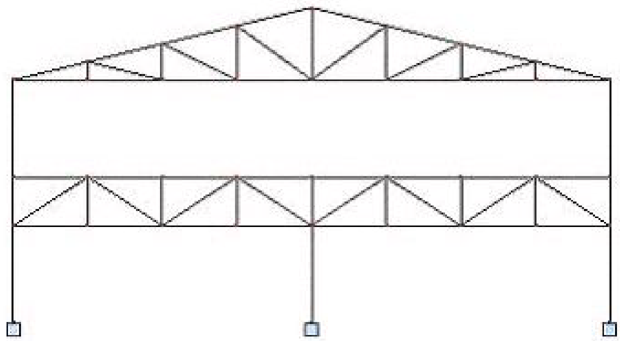
| | |
|---|---|
| Select the *Bar Sections* icon from the Structure Model toolbar. | Opens the ***Sections*** dialog box |
| LMC on HEA 240 on the section list | Selects the current section |
| Select the *New section definition* icon. | Opens the ***New Section*** dialog box with HEA 240 section selected |
| LMC the field next to the **Elasto-plastic analysis** button | Switches on the elasto-plastic analysis for the section selected. A new section name is defined: *HEA 240EP* |
| **Add**, **Close** | Defines the section *HEA 240EP*, closes the ***New Section*** dialog box. |
| Move to the graphical viewer with the structure view and select (LMC) external columns (bars no. 1, 2) | Changes the section of the indicated bars to *HEA 240EP* section.<br>Accept the warning of changing the result status to 'not available'. |
| In the ***Sections*** dialog box LMC on IPE 240 on the section list | Selects the current section |
| Select the *New section definition* icon. | Opens the ***New Section*** dialog box with IPE 240 section selected |
| LMC the field next to the **Elasto-plastic analysis** button | Switches on the elasto-plastic analysis for the section selected. A new section name is defined: *IPE 240EP* |
| **Add**, **Close** | Defines the section *IPE 240EP*, closes the ***New Section*** dialog box. |
| Move to the graphical viewer with the structure view and indicate (LMC) the beam (bar no. 4) | Changes the section of the indicated bar to *IPE 240EP* section. |
| **Close** in the ***Sections*** dialog box | Closes the ***Sections*** dialog box. |

### *3.3.4 Structural Analysis and Result Verification*

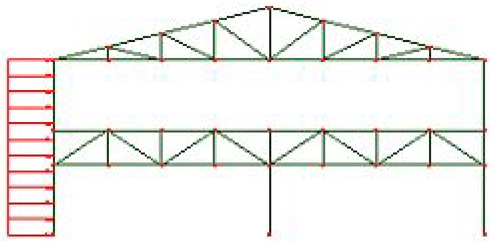
| | |
|---|---|
| *Analysis menu / Calculations* | Runs calculations. |
| *Results menu / Displacements* | Opens the node displacement table. |
| LMC the *Global Extremes* tab | Calculates maximal displacements of nodes (see the figure below). As it can be seen, in spite of the work in the plastic range, the structure retains stability. |
| *File menu / Save as* | Opens the saving dialog box. |
| In the *File name* field enter a selected name of the example e.g. *Frame_EC3_EP* | The default saving format - RTD. |
| LMC the **Save** button | Saves the example. |

# 4. Moving Loads - 2D Frame

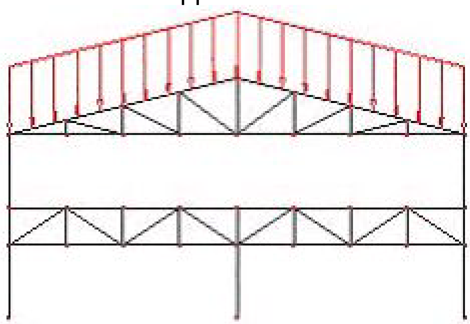
This example presents the definition, analysis and design of a simple 2D frame (see the figure below), for which a moving load case is defined.
Units: (m) and (kN).



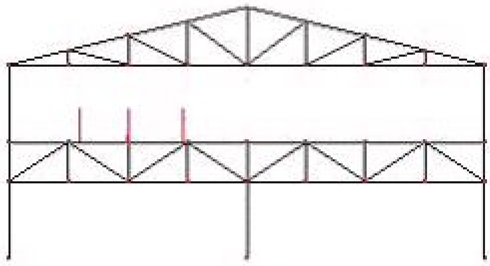
Three load cases will be applied to the structure (self-weight and two load cases: wind and snow, shown in the figure below). Moreover, a moving load case will be applied to the structure.



LOAD CASE 2



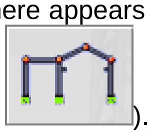
LOAD CASE 3



MOVING LOAD CASE

The following rules will apply during structure definition:

- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

In order to start defining a structure, one should run the ***RSAP*** program (press the relevant icon or select the relevant command from the toolbar). After a while, there appears on screen the dialog box, where one should select the first icon in the first row (**2D frame** ).

*NOTE: The European (French) section database (CATPRO) is used in the example.*

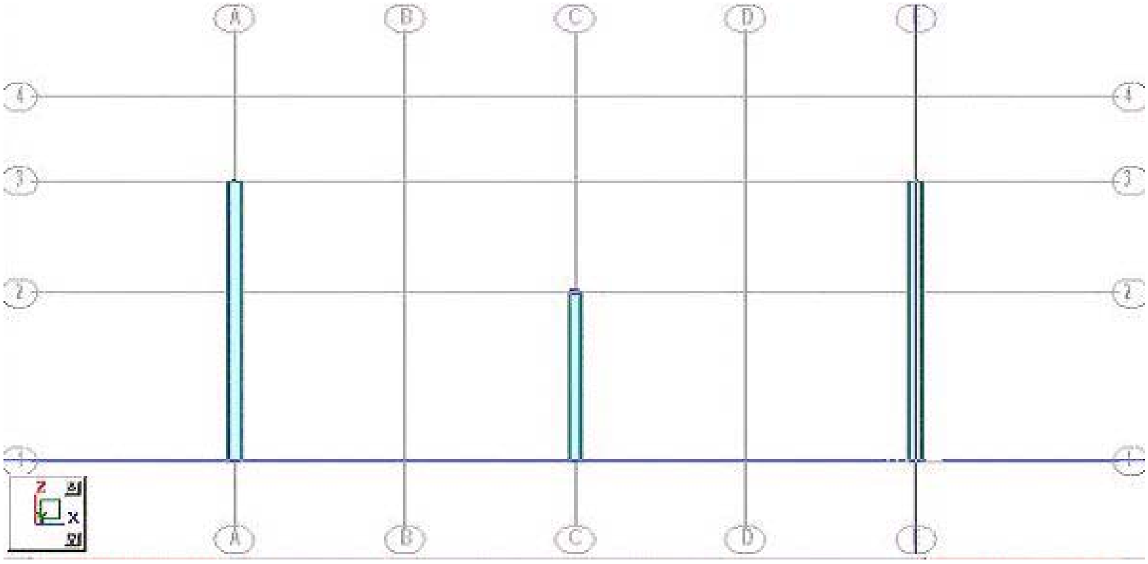
## 4.1 Model Definition

| OPERATION PERFORMED | DESCRIPTION |
|---|---|
| Select the Axis Definition icon from the Structural Model toolbar. | Starts the definition of structure axes. The ***Structural axis*** dialog box appears on screen. |
| In the X tab:<br>Position: {0}<br>Number of repetitions: {4}<br>Distance: {3}<br>Numbering: A, B, C ... | Definition of the parameters of vertical structural axes. |
| LMC the **Insert** button | Vertical axes have been defined and introduced into the *Set of defined axes* field. |
| LMC in the Z tab | Starts the definition of the parameters of horizontal structural axes. |
| In the Z tab:<br>enter the following coordinates of the successive axes:<br>{0}, **Insert**<br>{3}, **Insert**<br>{5}, **Insert**<br>{6.5}, **Insert**<br>Numbering: 1, 2, 3 ... | Defines the parameters of horizontal structural axes. |
| **Apply**, **Close** | Creates the defined structural axes and closes the ***Structural axis*** dialog box. |

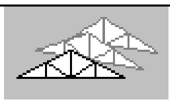
### *4.1.1 Member Definition*

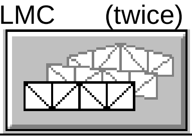
| | |
|---|---|
| Select the Bar Selections icon from the Structural Model toolbar | Opens the ***Sections*** dialog box |
| Select the New Selection Definition icon | Opens the ***New sections*** dialog box |
| Select the I-section group in the *Section* field and select the following sections: HEA 200, HEA 260 and IPE 200<br>**Add**, **Close** | Defines a new section and closes the ***New sections*** dialog box |
| **Close** | Closes the ***Sections*** dialog box |
| Select the Bars icon from the Structural Model toolbar | Opens the ***Bars*** dialog box |
| LMC in the BAR TYPE field: ***Column***<br>LMC in the SECTION field and select the section type: ***HEA 260*** | Selects bar properties |

| | |
|---|---|
| LMC in the *Beginning* field (color of a field background changes to green) | Starts defining bars in the structure (columns of the structure) |
| Column 1 - between Grids A1-A3, with the following coordinates:<br>Beginning: (0,0)<br>End: (0,5)<br><br>Column 2 - between Grids E1-E3, with the following coordinates:<br>Beginning: (12,0)<br>End: (12,5)<br><br>Column 3 – between Grids C1-C2, with the following coordinates:<br>Beginning: (6,0)<br>End: (6,3) | Defines structure columns. The figure below presents the structure created up to this moment. |
| **Close** | Closes the ***Bars*** dialog box |

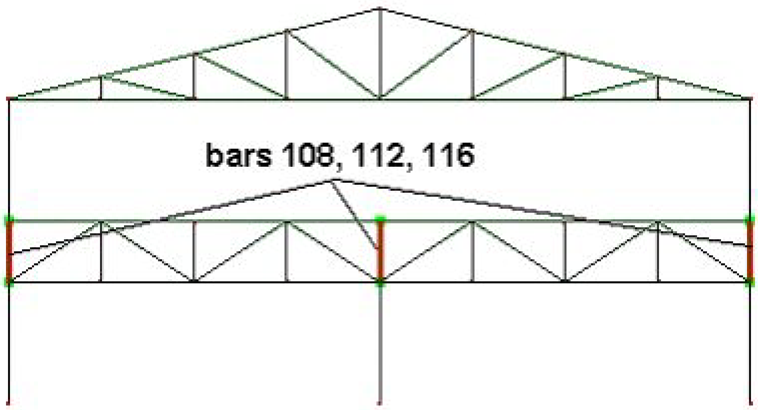



### *4.1.2 Library Structure Definition (a Roof and an Overhead Traveling Crane Beam)*

| | |
|---|---|
|  Select the Library Structure icon from the Structural Model toolbar. | Opens the ***Typical structures*** dialog box and starts defining a library structure (roof). |
| LMC (twice) the icon  (1st icon in the last row) | Selects the triangular truss of type 1. On screen, there appears the ***Merge structure*** dialog box where one may define truss parameters |
| In the *Dimensions* tab<br>LMC the *Length L* field<br>{12} | Defines truss length (one may also define it graphically in the graphical viewer) |
| LMC the *Height H* field<br>{1.5} | Defines truss height (one may also define it graphically in the graphical viewer) |

| | |
|---|---|
| LMC in the *Number of fields* field<br>{8} | Defines the number of fields into which the truss will be divided |
| LMC on the *Sections* tab;<br>To all truss chords (upper and lower) assign (DCED 90x10) and to diagonals, posts asign (CAE 70x7) | Assigns the section to the truss bars. |
| LMC in the *Insert* tab | |
| LMC in the *Insertion point* field select point A3 with the following coordinates (0,0,5) | Defines the insertion node for the truss |
| **Apply**, **OK** | Creates the defined structure in the indicated place within the structure and closes the ***Merge structure*** dialog box |
| Geometry menu / Releases | Opens the ***Releases*** dialog box |
| LMC on the release type: *Pinned-Fixed* | Selects the release type to be assigned to the truss bar |
| LMC on the *Current selection* field, switch to the graphic viewer and indicate the highest truss post (in the roof ridge) by hovering your curser over the element. | Selects the truss bar; ATTENTION: take note of the arrows that appear on the highlighted truss bar – while indicating the bar the arrows should be pointed up (the direction of the release is significant: at the first node the pinned connection remains, whereas at the second one – the fixed connection is defined) |
| **Close** | Closes the ***Releases*** dialog box |
|  Select the Library Structure icon from the Structural Model toolbar. (3rd icon in the second row) | Reopens the ***Typical structures*** dialog box and starts defining a library structure (moving-crane beam). |
| LMC (twice) in the icon  | Selects the rectangular truss of type 3. On screen, there appears the ***Merge structure*** dialog box where one may define truss parameters |
| On the *Dimensions* tab<br>LMC the *Length L* field<br>{12} | Defines truss length (one may also define it graphically in the graphical viewer) |
| LMC the *Height H* field<br>{1.0} | Defines truss height (one may also define it graphically in the graphical viewer) |
| LMC in the *Number of fields* field<br>{8} | Defines the number of fields into which the truss will be divided |
| LMC on the *Sections* tab;<br>To all truss chords (upper and lower) assign (DCED 90x10) and to diagonals, posts asign (CAE 70x7) | Assigns the section to the truss bars. |
| LMC in the *Insert* tab | |
| LMC in the *Insertion point* field select the point with the following coordinates (0,0,2) | Defines the insertion node for the truss |

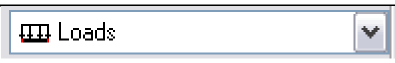
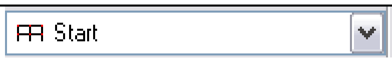
| | |
|---|---|
| **Apply**, **OK** | Creates the defined structure in the indicated place within the structure and closes the ***Merge structure*** dialog box |
| Select the side posts of the moving-crane truss and the central post (see the figure) - the bars become highlighted (bars 108, 112 and 116) | |
| Press the **Delete** button on the keyboard<br>Select *No, delete only the selected bars* and press **Apply** button | Deletes the selected structure bars |




### *4.1.3 Support Definition*

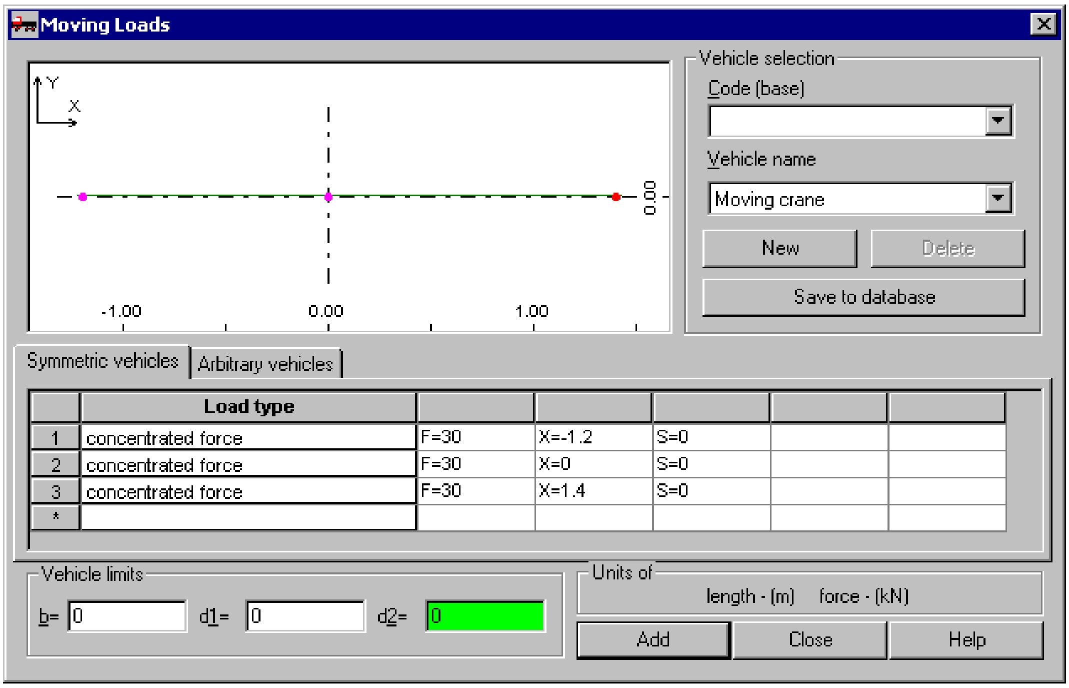
| | |
|---|---|
|  Select the Supports icon from the Structural Model toolbar | Opens the ***Supports*** dialog box |
| LMC in the *Current selection* field on the *Nodal* tab | Selects structure nodes where supports will be applied |
| Go to the graphical viewer; while holding the left mouse button pressed, select with the window all the lower nodes of the columns (the points located at the level of structural axis 1) | The selected nodes (1 3 5) will be inserted in the *Current selection* field. |
| In the ***Supports*** dialog box, select the icon referring to the fixed support (it will be highlighted) | Selects the support type |
| **Apply** | The selected support type will be applied to the selected structure nodes. |
| **Close** | Closes the ***Supports*** dialog box |

### *4.1.4 Structural Loads Definition*

| | |
|---|---|
|   LMC in the ***RSAP*** layout selection field<br>**Structure model / Loads** Layout | Selects the ***RSAP*** layout that allows one to define structure loads |
| LMC in the **New** button in the ***Load types*** dialog box | Defines the following load case:<br>nature: dead (self-weight)<br>standard name: DL1 |
| LMC in the ***Nature*** field:<br>*Wind* | Selects load case nature: wind |
| LMC in the **New** button | Defines the following load case:<br>nature: wind<br>standard name: WIND1 |
| LMC in the ***Nature*** field<br>*Snow* | Selects load case nature: snow |
| LMC in the **New** button | Defines the following load case:<br>nature: snow<br>standard name: SN1 |
| | The self-weight was applied automatically to all structure bars in the first row (direction "-Z”) |
| LMC the second field in the CASE column and select 2nd load case: WIND1 | Defines loads operating for the second load case |
| LMC the field in the LOAD TYPE column and select the uniform load | Selects load type |
| LMC the field in the LIST column and select graphically in the graphical viewer the left structure column | Selects the bar to which the program will apply the load with nodal forces (bar 1) |
| LMC the field in the "PX=" column and type the value 5.0 | Selects the direction and value of the load |
| LMC the third field in the CASE column, select 3rd load case: SN1 | Defines loads operating for the third load case |
| LMC the field in the LOAD TYPE column and select the uniform load | Selects load type |
| LMC the field in the LIST column and select in the graphical viewer the upper chords of the roof truss | Selects the bar to which the program will apply the uniform load (bars 5 and 6) |
| LMC the field in the "PZ=" column and enter the value: -3.0 | Selects the direction and value of the uniform load |
|   LMC in the ***RSAP*** layout selection field<br>**Structure Model / Start** Layout | Selects the initial ***RSAP*** layout |

### *4.1.5 Moving Load Definition Applied to the Structure*

| | |
|---|---|
| *Tools menu / Job Preferences* | Opens the ***Job Preferences*** dialog box |
| LMC the *Databases / Vehicle loads* option | Selects the option from the tree in the left part of the dialog box |
| Select the *Create a new database* icon. | Pressing the *Create a new database* icon results in opening the ***New Moving Load*** dialog box |
| Type:<br>in the *Database* field: USER<br>in the *Database name* field:<br>User-defined database<br>Units: length - (m)<br>force - (kN) | Defines a user database |
| **Create** | Closes the ***New Moving Load*** dialog box |
| **OK** | Closes the ***Job Preferences*** dialog box |
| *Loads menu / Special loads / Moving* | Opens the ***Moving Loads*** dialog box |
| Select *New vehicle* icon. | Opens the ***Moving Loads*** dialog box and starts defining a new vehicle |
| LMC on the **New** button | Defines a new vehicle |
| Type the vehicle name: *Moving crane*, **OK** | Defines the name of the new vehicle and closes the ***New vehicle*** dialog box |
| LMC the first line in the table located in the lower part of the dialog box | Defines the operating forces |
| Select the load type: concentrated force | Selects a load type |
| F = 30, X = -1.2, S = 0 | Defines the value and location of the concentrated force |
| LMC the next line in the table located in the lower part of the dialog box | Defines the operating forces |
| Select the load type: concentrated force | Selects a load type |
| F = 30, X = 0.0, S = 0 | Defines the value and location of the concentrated force |
| LMC the next line in the table located in the lower part of the dialog box | Defines the operating forces |
| Select the load type: concentrated force | Selects a load type |
| F = 30, X = 1.4, S = 0 | Defines the value and location of the concentrated force. The ***Moving Loads*** dialog box is presented below. |



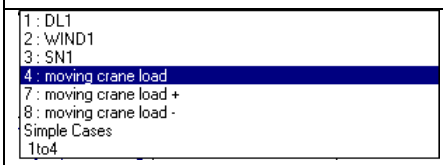

| | |
|---|---|
| LMC the **Save to database** button | Opens the ***Moving Load Databases*** dialog box |
| Select *USER* database and **OK** in the ***Moving Load Databases*** dialog box | Saves the defined vehicle to the user-defined database |
| **Add**, **Close** | Adds the defined vehicle to the list of active vehicles and closes the ***Moving Loads*** dialog box |
| In the Name field, type the name of the moving load (case 4):<br>*moving crane load* | Defines the name of the moving load |
| LMC the **Define** button | Starts the definition of the route of the Moving Crane vehicle: the ***Polyline - Contour*** dialog box is opened, with the *Polyline* option active. |
| Define two points determining the route of the vehicle:<br>beginning (0,3)<br>end (12,3) | Defines the vehicle route |
| **Apply, Close** | Closes the ***Polyline - contour*** dialog box |
| LMC the *Step* field {1}<br>Assume the default value of direction (0,0,-1), which means that the load will operate in the Z direction and its sense will be opposite with respect to the sense of the Z axis | Defines the step of position change of the moving load and the direction of load application. |
| LMC the *Selection* option located in the *Application plane* field | Selects the plane of load application |
| {8} | Selects the upper chord of the moving-crane truss (bar no. 8) |
| LMC the **Parameters** button | Opens the ***Route Parameters*** dialog box |

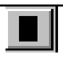
| | |
|---|---|
| LMC the field for factors:<br>Coef. LR and coef. LL<br>and type the value 0.1 | Defines the factors for the forces operating along the vehicle movement route. It generates the forces originating in vehicle braking, whose value equals 0.1*F |
| Switch on the following options:<br>*Vehicle position limit – route beginning*<br>*Vehicle position limit – route end* | Switching these options on assures that the forces defining the moving crane load will not be positioned off the defined structure model. |
| **OK** | Closes the ***Route Parameters*** dialog box |
| **Apply**, **Close** | Generates the moving load case according to the adopted parameters and closes the ***Moving Loads*** dialog box. |

## 4.2 Structural Analysis

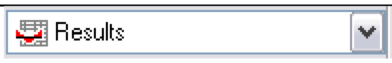
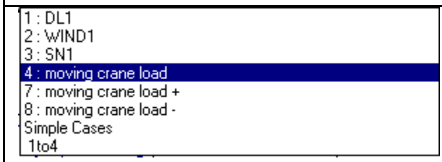
| | |
|---|---|
| *Tools menu / Job Preferences* | Opens the ***Job Preferences*** dialog box |
| *Structure Analysis* | Selects the *Structure Analysis* option from the tree in the dialog box |
| *Method of Solving the System of Equations: Iterative* | Selects the method of solving the equation system for the defined structure |
| **OK** | Accepts assumed parameters and closes the ***Job Preferences*** dialog box |
|  Select the *Calculations* icon from the Standard toolbar. | Starts calculations of the defined structure. Once the calculations are completed, the title bar of the viewer will present the following information: *Results - (FEM): available*. |

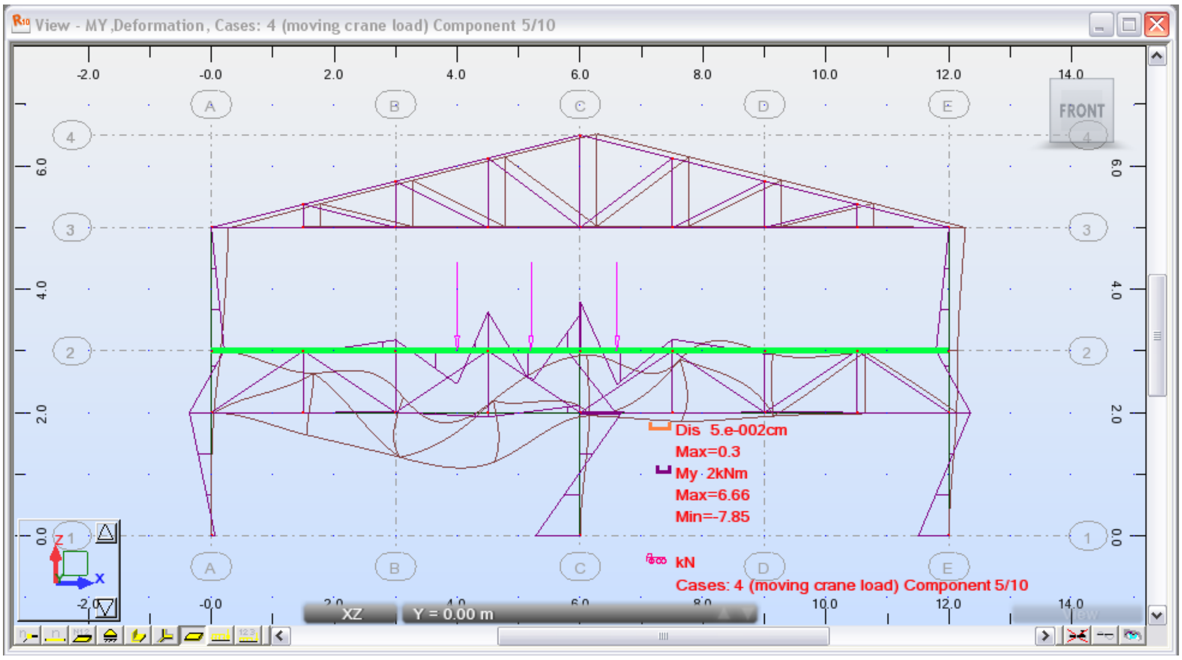
## 4.3 Presentation of the Vehicle and the Moving Load Case

| | |
|---|---|
| *View menu / Display* | Opens the ***Display*** dialog box |
| The *Loads* tab: switch on the *Moving loads / Moving loads - vehicle* option, **OK** | Presents the defined vehicle on the structure |
| <br>Select 4: moving-crane load | Select load case 4 (moving-crane load) |
| *Loads / Select Case Component* | Opens the ***Case component*** dialog box |
| Select: *Current component* 4 | Select component 4 of the moving load case |
| LMC the **Animation** button | Opens the ***Animation*** dialog box |
| LMC the **Start** button | Starts the animation of the moving load over the structure; the vehicle will move along the defined route. |

| | |
|---|---|
| Stop (LMC the  button) and close the animation toolbar | Stops the vehicle animation |
| **Close** | Closes the ***Case component*** dialog box |

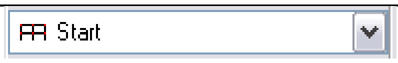
## 4.4 Analysis Results

| | |
|---|---|
|   LMC the field for selecting ***RSAP*** layout<br>**Results / Results** Layout | The **RESULTS *RSAP*** layout opens. The monitor screen will be divided into tree parts: the graphical viewer containing the structure model; the ***Diagrams*** dialog box and the table presenting the values of reactions.<br>*NOTE: The table presents additional moving load cases (marked with symbols "+" and "-") determining the value of the upper and lower envelope, respectively.* |
|   Select: 4 moving load | Selects load case 4 (moving crane load). |
| Switch on the *My Moment* option in the ***Diagrams*** dialog box | Selects the presentation of the bending moment in the structure for the selected moving load case. |
| Select the *Deformation* tab in the ***Diagrams*** dialog box<br>switch on the *Deformation* option | Selects the presentation of the deformation in the structure for the selected moving load case. |
| **Apply** | Presents the bending moment diagram and deformation diagram for the structure. Similarly, one can present the diagrams of other quantities available in the ***Diagrams*** dialog box. |

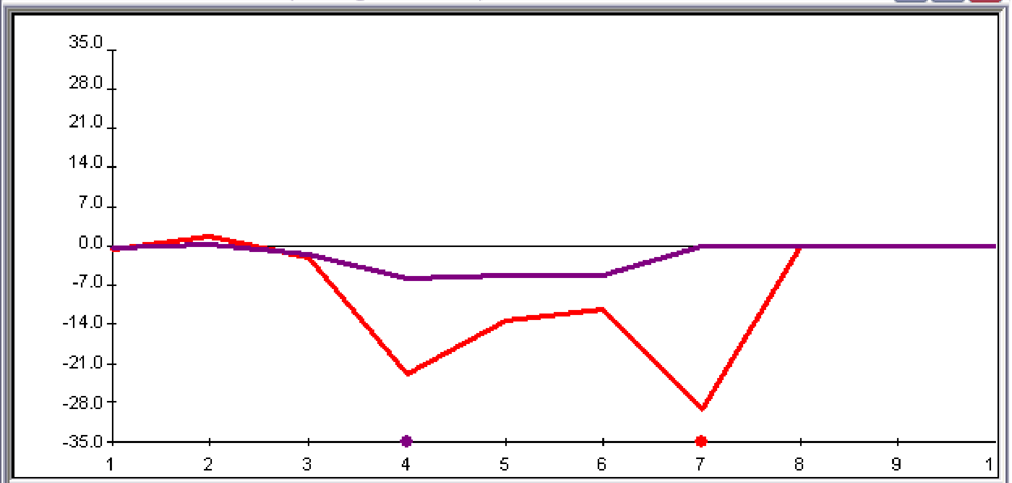



| | |
|---|---|
| *Loads menu / Select Case Component* | Opens the ***Case component*** dialog box |
| LMC the **Animation** button | Opens the ***Animation*** dialog box |
| LMC the **Start** button | Starts recording the animation of the bending moment and deformation for the structure |
| Stop (LMC the button) and close the animation toolbar | Stops recording the animation |
| **Close** | Closes the ***Case component*** dialog box |
| Switch off the *My Moment* option in the ***Diagrams*** dialog box | |
| Select the *Deformation* tab in the ***Diagrams*** dialog box<br>switch off the *Deformation* option,<br>**Apply** | |

## 4.5 Influence Lines

| | |
|---|---|
|   LMC in the ***RSAP*** layout selection field<br>**Structure model / Start** Layout | Goes to the **START** layout of the ***RSAP*** program. |
| *Results menu / Advanced / Influence Line* | Opens the ***Influence Lines*** dialog box |
| On the *NTM* tab of the ***Influence Lines*** dialog box, switch on the two options: My and Fz | Selects for presentation: the bending moment and the shear force for the moving load case |
| LMC in the *Element* field and type {8} | Selects the bar for which the program will present influence lines. The point position (equal 0.5) means that the influence line will be created for the point located in the middle of the bar length. |
| **Apply** | Opens an additional window presenting the influence lines of the selected quantities (see the figure below). |
| In the *Nodes* tab of the ***Influence Lines*** dialog box, switch on the two options: Ux and Uz | Selects the presentation of nodal displacements for the moving load case. |
| LMC in the *Node* field and type {2} | Selects the node for which the program will present influence lines. |
| Switch on the *Open in a new window* option | The diagrams of influence lines for the node no. 2 will be presented in a new window. |
| **Apply** | Opens an additional window where the influence lines of the selected quantities will be presented. |
| RMC in the ***Influence Lines*** dialog box where the influence lines are presented for node 2 | Opens the context menu |
| *Add coordinates* | If the option is selected, the table located under the diagrams of influence lines will display additional columns containing the coordinates of the successive structure points. |

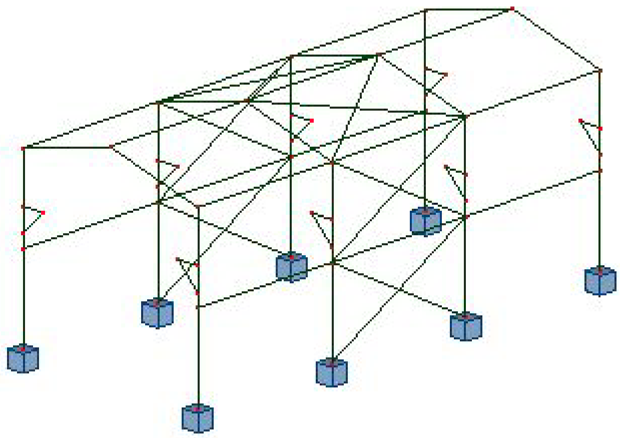
Influence Line - Case: 4 (moving crane load)



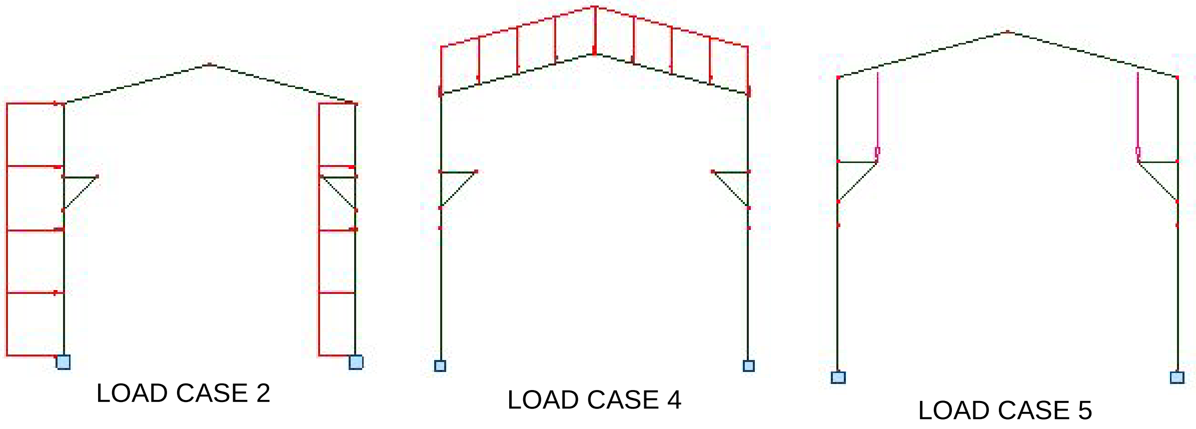

| Bar/Point/Case/Posit./Dist. | FZ (kN) | MY (kNm) |
|---|---|---|
| | | |
| Sum (integral) * (m) | -80,80 | -20,14 |
| Sum (+) * (m) | 1,07 | 0,15 |
| Sum (-) * (m) | -81,87 | -20,29 |
| | | |
| 8/ x=0.50/ moving crane load/ 1/10 1,20 | -0,94 | -0,71 |
| 8/ x=0.50/ moving crane load/ 2/10 2,20 | 1,93 | 0,48 |
| 8/ x=0.50/ moving crane load/ 3/10 3,20 | -2,54 | -1,86 |
| 8/ x=0.50/ moving crane load/ 4/10 4,20 | -23,60 | -6,36 |
| 8/ x=0.50/ moving crane load/ 5/10 5,20 | -13,77 | -5,82 |
| 8/ x=0.50/ moving crane load/ 6/10 6,20 | -11,91 | -5,61 |
| 8/ x=0.50/ moving crane load/ 7/10 7,20 | -30,11 | -0,15 |
| 8/ x=0.50/ moving crane load/ 8/10 8,20 | -0,18 | -0,25 |
| 8/ x=0.50/ moving crane load/ 9/10 9,20 | -0,11 | -0,16 |
| 8/ x=0.50/ moving crane load/ 10/10 10,20 | -0,10 | -0,14 |

# 5. Moving Load – 3D Frame

This example presents definition, analysis and design of a simple steel workshop illustrated in the figure below.
Data units: (m) and (kN).

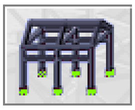


Five load cases have been assigned to structure and three of them are shown in the drawings below.




The following rules apply during structure definition:

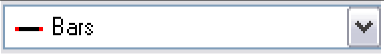
- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

To run structure definition start the ***RSAP*** program (press the appropriate icon or select the command from the taskbar). The vignette window will be displayed on the screen and the icon  (***Frame 3D Design***), the last but one in the first row, should be selected.

*NOTE: The European section database (EURO) has been used in this example.*

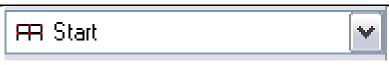
## 5.1 Model Definition

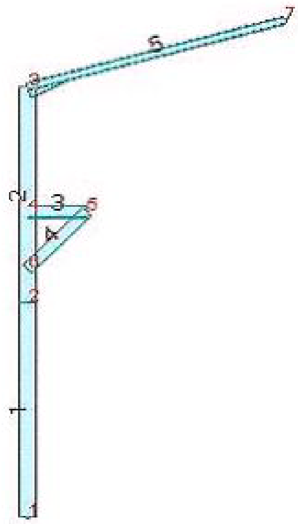
*Definition of Structure Bars*

| PERFORMED OPERATION | DESCRIPTION |
|---|---|
| <br>**Structure Model / Bars** Layout | Selects the **BARS** layout from the list of the available ***RSAP*** layouts. |
| LMC on the *Bar Type* field and select ***Column***<br>LMC on the *Section* field and select (IPE 600) | Selects bar properties. The section from the European section database (EURO) has been used.<br>*Note: If the IPE 600 section is not available on the list, one should select the **STRUCTURE MODEL / SECTION and MATERIALS** layout, press the **New** icon in the **Sections** dialog box and add the section to the list of active sections.* |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of bars in the structure (structure column). |
| Enter the following coordinates in the *Beginning* and *End* field:<br>(-8,0,0) (-8,0,7), **Add**<br>(-8,0,7) (-8,0,14), **Add** | Defines a column of the structure. |
| LMC on the *Bar Type* field in the ***Bars*** dialog box and select *Beam*<br>LMC on the *Section* field and select: (IPE 240) | Starts definition of a beam and selects its properties.<br>The section from the European section database (EURO) has been used.<br>*Note: If the IPE 240 section is not available on the list, one should follow the above procedure.* |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of a beam in the structure. |
| Enter the following points in the *Beginning* and *End* field:<br>(-8,0,10) (-6,0,10), **Add** | Defines a beam. |
| LMC on the *Bar Type* field in the ***Bars*** dialog box and select *Simple bar*<br>LMC on the *Section* field and select UPN 240 | Starts definition of a simple bar and selects its properties. The section from the European section database (EURO) has been used. |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of a simple bar in the structure. |
| Enter the following points in the *Beginning* and *End* field:<br>(-8,0,8) (-6,0,10), **Add** | Defines a simple bar. |
| LMC the *Bar type* field in the ***Bars*** dialog box, select *Simple bar*<br>LMC the *Section* field, select HEB 240 | Starts to define the bar and assign the properties to it.<br>*NOTE: If the section HEB 240 is not present on the list of available sections, then press the*  *button and next follow the steps mentioned above.* |
| LMC the *Beginning* field (background color changes to green) | Starts to define the bar coordinates in a structure. |

| | |
|---|---|
| Enter the bar coordinates in the *Beginning* and *End* fields: (-8,0,14) (0,0,16), **Add** | Defines a bar. |
| Select the *Zoom All* icon from the standard toolbar. | Restores the initial structure view. |

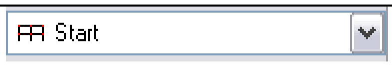
*Bracket Definition*

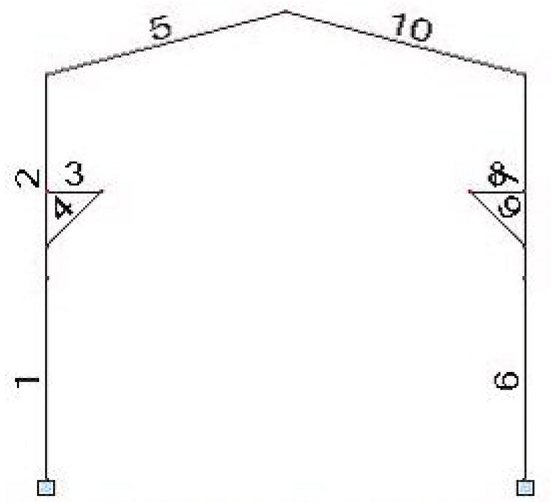
| | |
|---|---|
|  LMC on the field to select the **Structure Model / Start** Layout | Selects the initial layout of the ***RSAP*** program. |
| *Geometry menu / Additional Attributes / Brackets* | Opens the ***Brackets*** dialog box that is used to define nodal brackets for structure bars. |
| Select the *New Bracket* icon. | Opens the ***New Bracket*** dialog box. |
| In the *Length (L)* field enter the value 0.15; leave the remaining parameters unchanged | Defines the bracket length |
| **Add**, **Close** | Defines a new bracket, closes the ***New Bracket*** dialog box |
| LMC the *Bars* field, move to the graphical viewer and select the recently-defined bar (number 5 should be displayed in the *Bars* field) | Selects a bar to which a bracket will be assigned. |
| **Apply**, **Close** | Assigns the bracket to the selected bar, closes the ***Brackets*** dialog box. The structure defined is displayed in the drawing below. |



*Definition of Structure Supports*

| | |
|---|---|
|  LMC on the field to select the **Structure Model / Supports** Layout | Selects the ***RSAP*** layout which allows defining supports. |

| | |
|---|---|
| In the ***Supports*** dialog box, LMC on the *Current Selection* field on the *Nodal* tab (the cursor is blinking in the field) | Selects structure nodes for which supports will be defined. |
| Switch to the graphic viewer; pressing the left mouse button select the lower column node by means of the window | The selected node 1 will be entered to the *Current Selection* field. |
| From the ***Supports*** dialog box select the fixed support icon (the icon will be highlighted) | Selects the support type. |
| **Apply** | The selected support type will be assigned to the chosen structure nodes. |
|   LMC on the field to select the **Structure Model / Start** Layout | Selects the initial ***RSAP*** program layout. |
| CTRL+A | Selects all nodes and bars. |
| *Edit menu / Edit / Vertical Mirror* | Mirrors selected bars. |
| Locate graphically the vertical symmetry axis (x = 0), LMC, **Close** | Performs the axial symmetry of selected bars and closes the ***Vertical Mirror*** dialog box. |
|  Select the *Zoom All* icon from the standard toolbar. | Once this option is selected the initial view of the structure will be presented.<br>The defined structure is shown in the drawing below. |

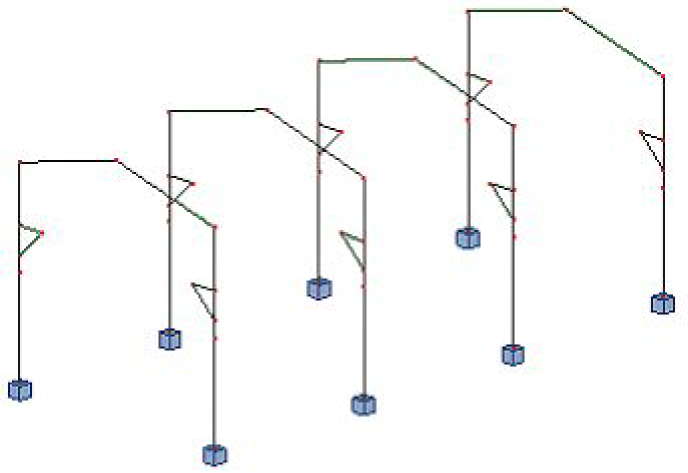



*Definition of Structure Loads*

| | |
|---|---|
|   LMC on the field to select the **Structure Model/Loads** Layout | Selects the ***RSAP*** program layout that allows defining structure loads. |
| LMC on the **New** button in the ***Load Types*** dialog box. | Defines a *dead load (self-weight)* with a standard name DL1. |

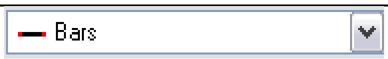
| | |
|---|---|
| LMC on the *Nature* field<br>*(Wind)* | Selects the load case type: *wind*.<br>*NOTE: If the load case number is not changed automatically, enter number (2) manually* |
| LMC on the **New** button<br>LMC on the **New** button | Defines two cases of *wind* load with the standard names: WIND1 and WIND2. |
| LMC on the *Nature* field<br>*(Snow)* | Selects the load case type: *snow*. |
| LMC on the **New** button | Defines a *snow* load with a standard name SN1. |
| | *Note: The self-weight load was automatically applied to all structure bars (in the "Z" direction).* |
| LMC on the second field in the ***Case*** column of the **Loads** table, select the 2nd load case: WIND1 from the list | Defines loads for the second load case. |
| LMC on the field in the ***Load Type*** column, select the (*uniform load*) load type | Selects the load type. |
| LMC on the field in the ***List*** column, select the left column in a graphical way | Selects the column to which the *uniform load* will be applied. |
| LMC on the field in the "PX=" column and enter the value: (2.0) | Selects the direction and value of the uniform load. |
| LMC on the third field in the ***Case*** column, select the 2nd load case WIND1 from the list | Defines another load for the second load case. |
| LMC on the field in the ***Load Type*** column, select the (*uniform load*) load type | Selects the load type. |
| LMC on the field in the ***List*** column, select the right column graphically | Selects bars to which the *uniform load* will be applied. |
| LMC on the field in the "PX=" column and enter the value: (1.5) | Selects the direction and value of the uniform load. |
| LMC on the fourth field in the ***Case*** column, select the 4th load case: SN1 from the list | Defines loads for the third load case. |
| LMC on the field in the ***Load Type*** column, select the (*uniform load*) load type | Selects the load type. |
| LMC on the field in the ***List*** column, select the beams of the steel girder graphically | Selects bars to which the uniform load will be applied. |
| LMC on the field in the " PZ =" column and enter the value: (-1.75) | Selects the direction and value of the uniform load. |
| LMC in the ***View*** viewer | |

| | |
|---|---|
| CTRL + A | Selects all structure bars. |
| While the graphic viewer with the structure model is active, select *Edit / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the field (dX, dY, dZ),<br>(0,12,0) | Defines the translation vector. |
| LMC on the *Number of Repetitions* field<br>(3) | Defines a number of repetitions for the performed translation operations. |
| **Execute**, **Close** | Translates the structure and closes the ***Translation*** dialog box (proceed to the next step to see changes). |
| *View menu / Projection / 3d xyz* | Selects the isometric structure view (see the drawing below). |
|  Select the *Zoom All* icon from the standard toolbar | Once this option is selected the initial view of the structure will be presented.<br>The defined structure is presented in the drawing below. |

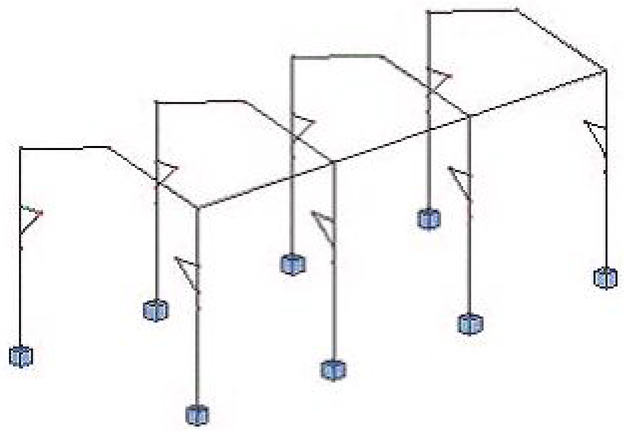


*Definition of Additional Elements of the Structure (Longitudinal Beams, Bracings, Crane Girder)*

*Longitudinal Beams - Definition*

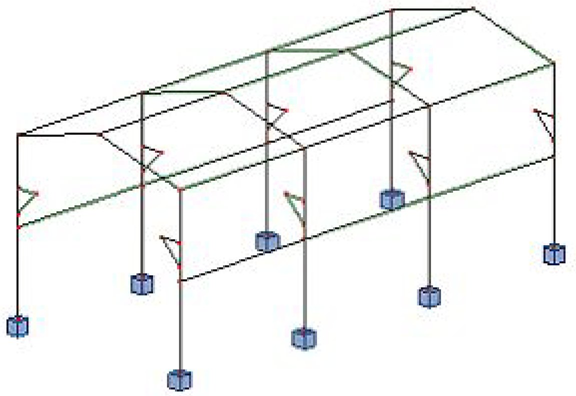
| | |
|---|---|
| <br>LMC on the field to select the **Structure Model/Bars** Layout | Selects the ***RSAP*** layout that allows defining bars. |
| *View /Display* | Opens the ***Display*** dialog box |
| On the *Nodes* tab switch off the option: *Node numbers*<br>On the *Bars* tab switch off the options: Bar description / *Bar numbers* and *Symbols*,<br>**Apply**, **OK** | Switches off display of node numbers, bar numbers and symbols of bar sections, closes the ***Display*** dialog box |

| | |
|---|---|
| LMC on the *Bar Type* field in the ***Bars*** dialog box and select: *Beam* LMC on the *Section* field and select (IPE 200) | Selects bar properties. The section from the American section database (AISC) has been used. |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of bars in the structure. |
| Enter the following coordinates in the *Beginning* and *End* field:<br>(8,0,14) (8,12,14), **Add**<br>(8,12,14) (8,24,14), **Add**<br>(8,24,14) (8,36,14), **Add** | Defines longitudinal beams as shown in the drawing below. |

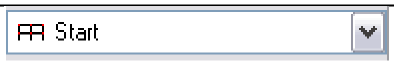


| | |
|---|---|
| Switch to the graphic viewer; RMC in any place in the viewer, which opens the context menu. Chose the *Select* option and sort out three recently defined bars - while the CTRL key is pressed LMC on three beams | |
| While the graphic viewer with the structure model is active, select *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the field (dX, dY, dZ), (0,0,-7) | Defines the translation vector. |
| **Execute** | Translates the structure and highlights translated beams. |
| LMC on the field (dX, dY, dZ), (-16,0,0) | Defines a new translation vector. |
| **Execute** | Translates beams and highlights translated beams. |
| LMC on the field (dX, dY, dZ), (0,0,7) | Defines a new translation vector. |
| **Execute** | Translates the structure and highlights translated beams. |
| LMC on the field (dX, dY, dZ), (8,0,2) | Defines a new translation vector. |

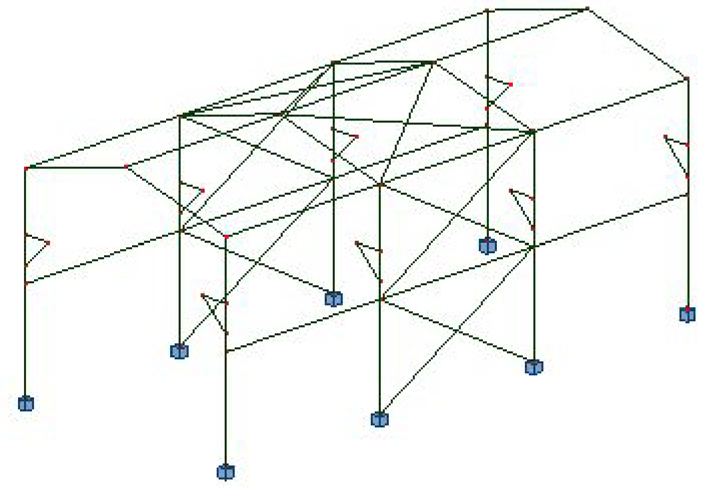
| | |
|---|---|
| **Execute**, **Close** | Translates the structure and closes the ***Translation*** dialog box. The defined structure is presented in the drawing below. |



*Bracing - Definition*

| | |
|---|---|
| LMC in the *Bar Type* field and select: *Simple bar*<br>LMC on the *Section* field and select (CAE 100x12) | Selects bar properties. |
| LMC on the *Beginning* field (background color changes to green)<br>(8,12,0) (8,24,7), **Add**<br>(8,12,7) (8,24,0), **Add** | Defines bracing. |
|  <br>LMC on the field to select the **Structure Model / Start** Layout | Selects the initial layout of the ***RSAP*** program. |
| Select the two recently defined bars - while the CTRL key is pressed LMC on two bars | |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the field (dX, dY, dZ),<br>(0,0,7), **Execute** | Defines the translation vector. |
| LMC on the graphic viewer; open the context menu clicking RMC on any point in the viewer. Choose the *Select* option (the context menu will close then); select all the recently defined bracings – with the CTRL key pressed, LMC on the four bars | |
| LMC on the field (dX, dY, dZ) in the **Translation** dialog box,<br>(-16,0,0) | Defines the translation vector. |
| **Execute**, **Close** | Translates bars and closes the ***Translation*** dialog box. |

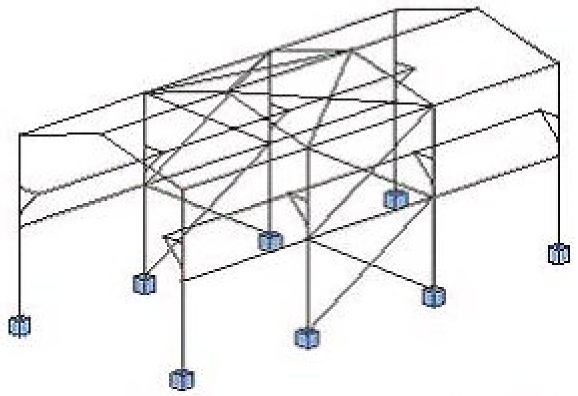
| | |
|---|---|
| Bars<br>LMC on the field to select the **Structure Model/Bars** Layout | Selects the ***RSAP*** layout that allows defining bars. |
| LMC on the *Bar Type* field and select: *Simple bar*<br>LMC on the *Section* field and select (CAE 100x12)) | Selects bar properties. |
| LMC on the *Beginning* field (background color changes to green)<br>(8,12,14) (0,24,16), **Add**<br>(0,12,16) (8,24,14), **Add**<br>(-8,12,14) (0,24,16), **Add**<br>(-8,24,14) (0,12,16), **Add** | Defines bracing. |



*Crane Girder - Definition*

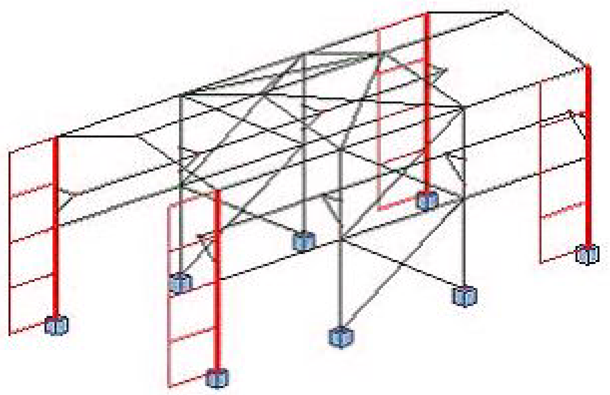
| | |
|---|---|
| Start<br>LMC on the field to select the **Structure Model / Start** Layout | Selects the initial layout of the ***RSAP*** program. |
| Select the *Bar Sections*_icon from the Structure Model toolbar. | Opens the ***Sections*** dialog box. |
| Select the *New section definition* icon. | Opens the ***New Section*** dialog box. |
| Select the icon on the *Parametric* tab | Defines a user section: *I-ASYM_1* |
| On the *Dimension* tab enter:<br>b1 = 40, h = 55, b2 = 25,<br>tw = 1.5, tf1 = 1.5, tf2 = 1.5<br>**Add**, **Close** | Defines dimensions of the user section. |
| **Close** | Closes the ***Sections*** dialog box |
| Select the *Bar*_icon from the Structure Model toolbar. | Opens the ***Bars*** dialog box |

| | |
|---|---|
| LMC on the *Bar Type* field and select: *Beam*<br>LMC on the *Section* field and select (*I-ASYM_1*) | Selects bar properties. |
| LMC on the *Beginning* field (background color changes to green) (6,0,10) (6,36,10), **Add** | Defines a crane girder. |
| LMC on the ***View*** edit viewer;<br>Select the recently defined bar | |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the field (dX, dY, dZ), (-12,0,0) | Defines the translation vector. |
| **Execute**, **Close** | Translates bars and closes the ***Translation*** dialog box. |



*Definition of Additional Loads*

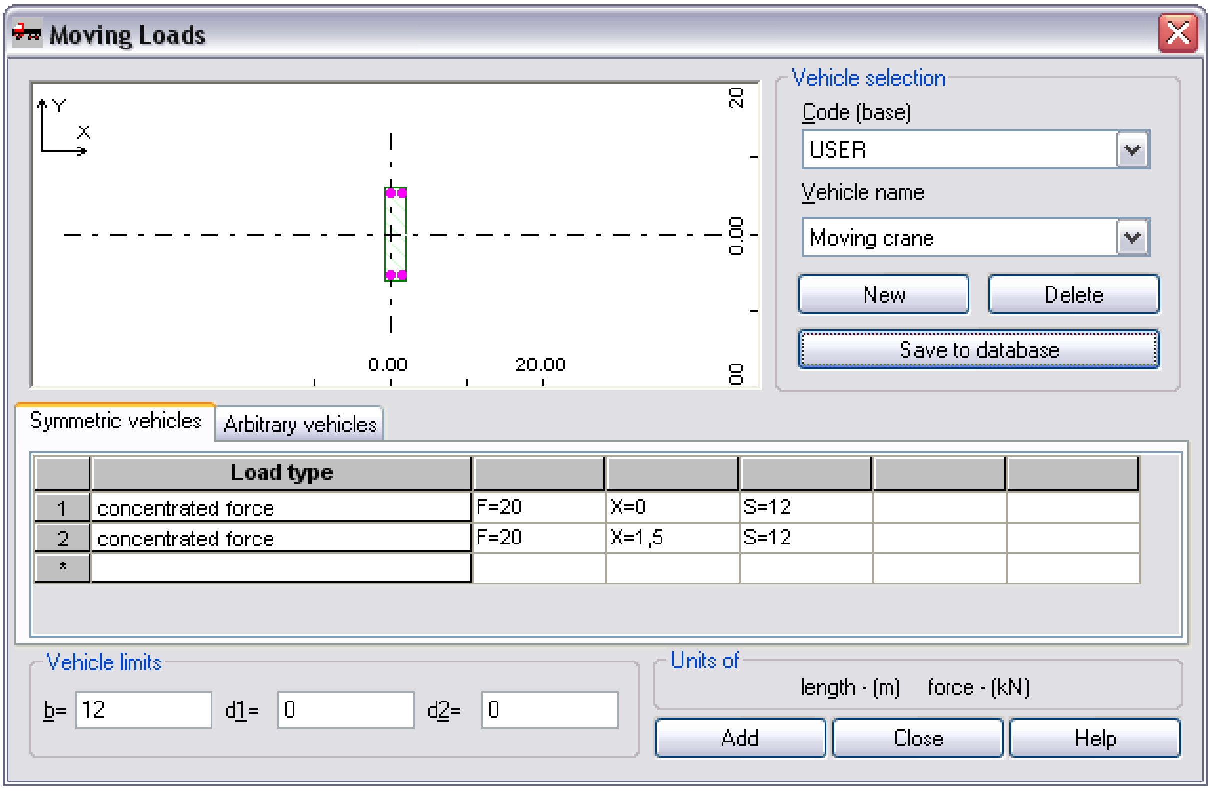
| | |
|---|---|
| <br>LMC on the field to select the **Structure Model / Loads** Layout | Selects the ***RSAP*** program layout that allows defining structure loads. |
| LMC on the fifth field in the ***Case*** column, select the 3rd load case: WIND2 from the list | Defines loads for the fourth load case. |
| LMC on the field in the ***Load Type*** column, select the (*uniform load*) load type | Selects the load type. |
| LMC on the field in the ***List*** column, select the corner columns | Selects columns to which the *uniform load* will be applied. |
| LMC on the field in the "PY=" column and enter the value: (2.0) | Selects the direction and value of the uniform load. |
| LMC in the ***View*** viewer | The defined load is presented in the drawing below. |



*Definition of the Moving Load Applied to the Crane Girder*

| | |
|---|---|
| *Tools menu / Job Preferences / Databases / Vehicle Loads* | Open the ***Job Preferences*** dialog box. |
| Select the *Create new user's database* icon | Pressing the *Create new database icon results* in opening the ***New Moving Load*** dialog box. |
| Enter:<br>in the *Database* field: User<br>in the *Database name* field: User-defined database<br>in the *Database description* field: User-defined vehicles<br>in the *Internal units of the database* select: (kN) as *Force units* and (m) as *Length units* | Note: If you already created this moving load database, you can skip this step. |
| **Create** | Creates a new database and closes the ***New Moving Load*** dialog box. |
| **OK** | Closes the ***Job Preferences*** dialog box. |
| *Loads menu / Special loads / Moving* | Opens the ***Moving Loads*** dialog box. |
| Select the *New vechicle* icon | Opens the **Moving *Loads*** dialog box and starts defining a new vehicle. |
| On the *Symmetric vehicles* tab LMC on the **New** button | Opens the ***New vehicle*** dialog box. |
| Enter the vehicle name: *Moving crane*<br>**OK** | Defines the name of the new vehicle, closes the ***New vehicle*** dialog box. |
| LMC the first line in the table located in the lower part of the dialog box | Defines acting forces. |
| Select the load type: concentrated force | Selects a load type. |
| F = 20, X = 0.0, S = 12 | Defines the value and location of the concentrated force. |
| LMC the second line in the table located in the lower part of the dialog box | Defines operating forces. |

| | |
|---|---|
| Select the load type: concentrated force | Selects a load type. |
| F = 20, X = 1.5, S = 12 | Defines the value and location of the concentrated force. |




| | |
|---|---|
| LMC the **Save to database** button | Opens the ***Moving load databases*** dialog box. |
| **OK** in the ***Moving load databases*** dialog box | Saves the defined vehicle to the user-defined database. |
| **Add**, **Close** | Adds the defined vehicle to the list of active vehicles and closes the ***Moving loads*** dialog box. |
| In the *Name* field, enter the name of the moving load (case number 5) *Moving crane* | Defines the name of the moving load. |
| LMC the **Define** button and active the *Line* option. | Starts defining the route of the Moving Crane vehicle: the ***Polyline - Contour*** dialog box is opened. Activate the *Line* option. |
| In the ***Geometry*** dialog box define two points determining the route of the moving load:<br>Point P1 (0,0,10)<br>Point P2 (0,36,10) | Defines the vehicle route. |
| **Apply**, **Close** | Closes the ***Polyline - Contour*** dialog box. |

| | |
|---|---|
| LMC the *Step* field {1}<br>Assume the default value of load direction (0,0,-1) which means that the load will operate in the Z direction and its sense will be opposite to Z axis sense | Defines the step of a position change for the moving load and the direction of load application. |
| LMC the *Automatic* option located in the *Application Plane* field | Selects the plane of load application. |
| LMC the **Parameters** button | Opens the ***Route Parameters*** dialog box. |
| LMC the fields for the LR and LL factors and enter the value 0.1 | Defines the factors for the forces operating along the vehicle movement route. It generates the forces originating in vehicle braking, whose value equals 0.1*F. |
| Activate the following options:<br>*Vehicle position limit – route beginning* and *Vehicle position limit – route end* | Switching these options on assures that the forces defining the load will not be positioned off the route limits defining the movement of the moving load. |
| **OK** | Closes the ***Route Parameters*** dialog box. |
| **Apply**, **Close** | Generates the moving load case according to the adopted parameters and closes the ***Moving loads*** dialog box. |

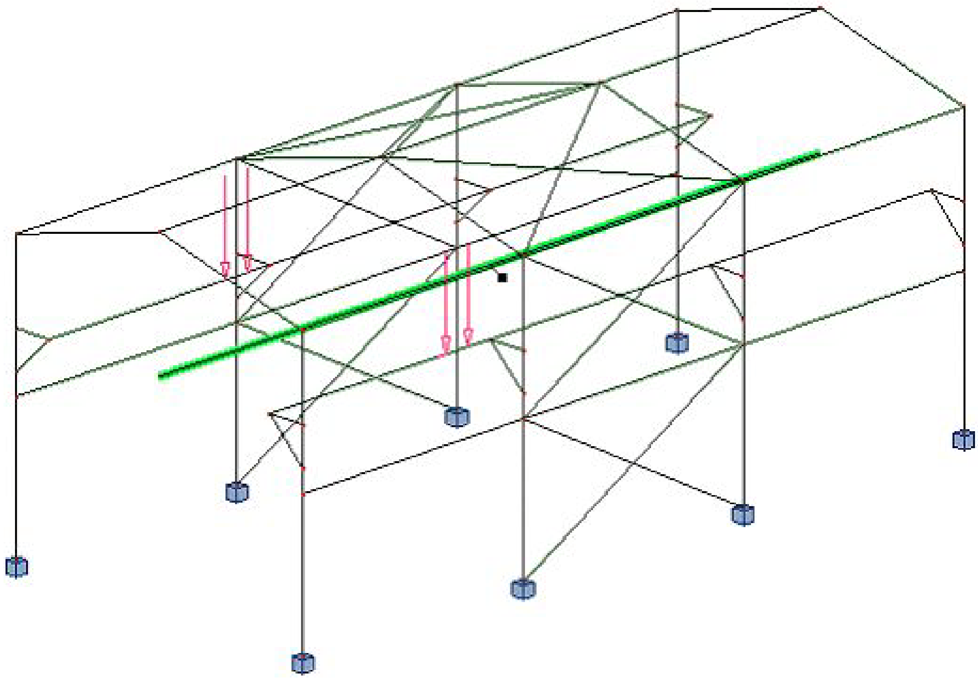
## 5.2 Structural Analysis

| | |
|---|---|
| *Tools menu / Job Preferences* | Opens the ***Job Preferences*** dialog box |
| *Structure Analysis* | Selects the *Structure Analysis* option from the tree in the dialog box |
| *Method of solving the system of equations: Iterative* | Selects the iterative method of solving the equation system for the defined structure |
| Switch off the option *Automatic freezing of results of structure calculations* | Switches off freezing of structure calculation results. |
| **OK** | Accepts assumed parameters and closes the ***Job Preferences*** dialog box |
|  Select the *Calculations* icon from the Standard toolbar. | Starts calculations of the defined structure. Once the calculations are completed, the title bar of the viewer will present the following information: *Results (FEM): available.* |

*Presentation of the Vehicle and the Moving Load Case*

| | |
|---|---|
| *View menu / Display* | Opens the ***Display*** dialog box. |
| In the *Loads* tab: switch on the *Moving loads / Moving loads - vehicle* option, **Apply OK** | Presents the defined vehicle on the structure. |

| | |
|---|---|
| 1 : DL1<br>2 : WIND1<br>3 : WIND2<br>4 : SN1<br>5 : Moving crane<br>8 : Moving crane +<br>9 : Moving crane -<br>Simple Cases | Selects the load case: 5 (Moving crane). |
| *Loads menu / Select Case Component* | Opens the ***Case Component*** dialog box. |
| Select: *Current component 5* | Selects the component 5 of the moving load case. |
| LMC the **Animation** button | Opens the ***Animation*** dialog box. |
| LMC the **Start** button | Starts the animation of the moving load applied to the structure; the vehicle will move along the defined route. |
| Stop the animation pressing the button; close the animation toolbar | Stops presenting the vehicle animation. |
| **Close** | Closes the ***Case Component*** dialog box. |



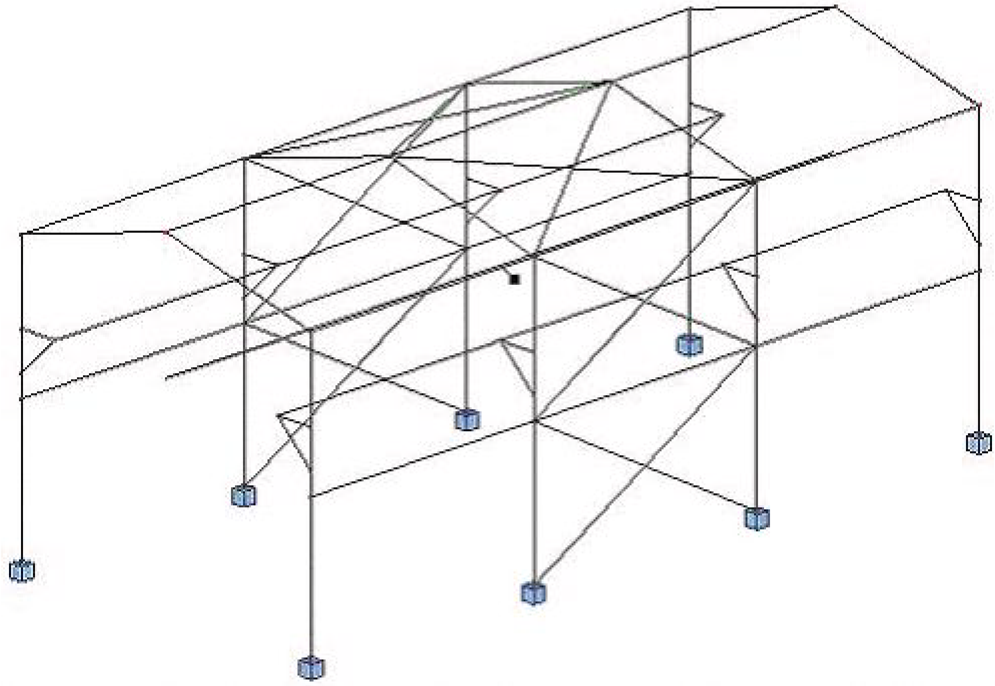
| | |
|---|---|
| Results<br>LMC on the field to select of the **Results/Results** Layout | The **RESULTS** layout of the ***RSAP*** program will be opened. The screen will be divided into three parts: a graphic viewer containing the structure model, the ***Diagrams*** dialog box and a table with reaction values. |
| 1 : DL1<br>2 : WIND1<br>3 : WIND2<br>4 : SN1<br>5 : Moving crane<br>8 : Moving crane +<br>9 : Moving crane -<br>Simple Cases | Selects the load case: 5 (Moving crane). |
| Select the *Deformation* tab in the ***Diagrams*** dialog box, turn on the *Deformation* option | Selects presentation of deformation for the selected moving load case. |

| | |
|---|---|
| **Apply** | Presents the deformation diagram for the structure. Similarly, the diagrams of other quantities available in the ***Diagrams*** dialog box can be presented. |
| *Loads menu / Select Case Component* | Opens the ***Case Component*** dialog box. |
| LMC the **Animation** button | Opens the ***Animation*** dialog box. |
| LMC the **Start** button | Starts animation of deformation for the structure. |
| Stop (LMC the  button) and close the animation toolbar | Stops the animation. |
| **Close** | Closes the ***Animation*** dialog box. |
| Select the *Deformation* tab in the ***Diagrams*** dialog box<br>Turn off the *Deformation* option, **Apply** | |

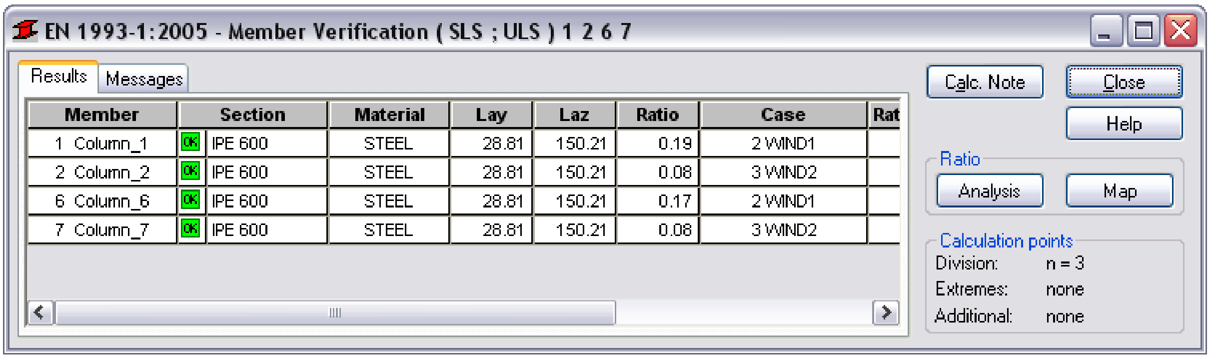
## 5.3 Steel Design

Code: EN 1993-1:2005

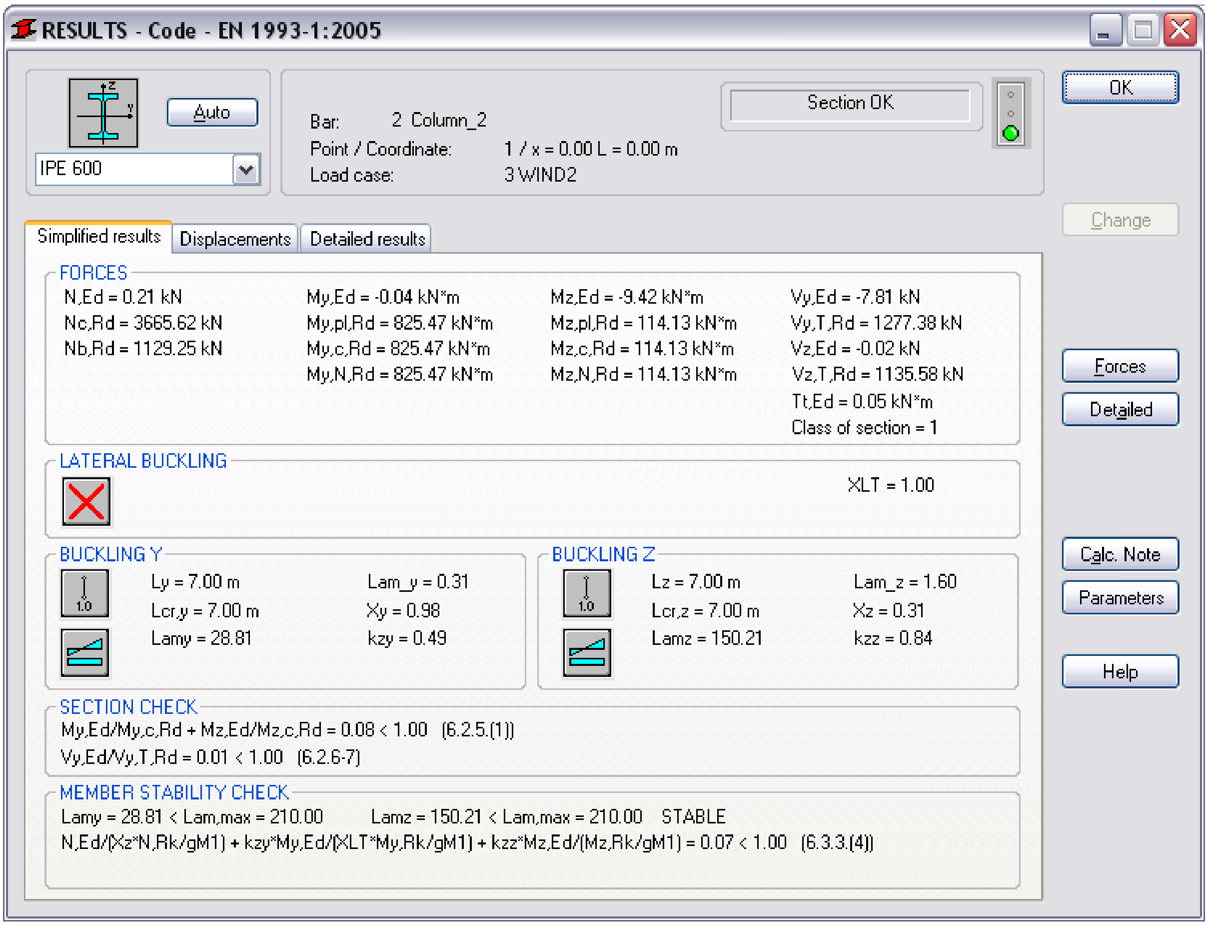
| | |
|---|---|
|   LMC on the field to select the **Structure Design/Steel/Aluminum Design** Layout | Starts steel member design. The screen will be divided into three parts: a graphic viewer containing the structure model, the ***Definitions*** dialog box and the ***Calculations*** dialog box. |
| LMC on the **List** button located beside the *Member Verification* field in the ***Calculations*** dialog box | Opens the ***Member Selection*** dialog box. |
| Enter the member numbers: 1, 2, 6, 7 (columns) in the field located above the **Previous** button, **Close** (see the figure below) | Selects members for verification. |
| LMC on the **List** button in **Loads** group in ***Calculations*** dialog box | Opens the ***Load Case Selection*** dialog box. |
| LMC on the **All** button, **Close** | Selects all load cases. |



| | |
|---|---|
| LMC on the **Calculations** button | Starts verification of the selected structure members; the ***Member Verification*** dialog box shown below will be displayed on the screen. |

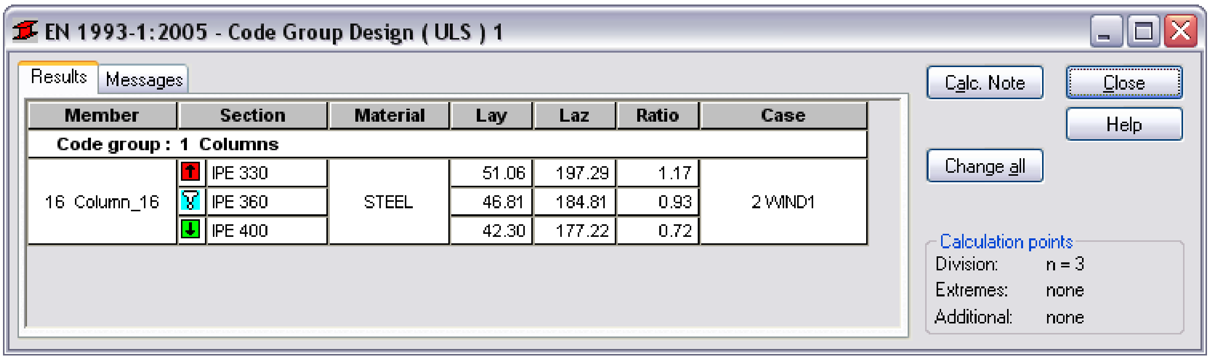



| | |
|---|---|
| LMC on the line containing simplified results for member no. 2 | Opens the ***Results*** dialog box for the selected member. |
| LMC on the *Simplified Results* tab | Displays design results for member no. 2 (see the dialog box presented below). |

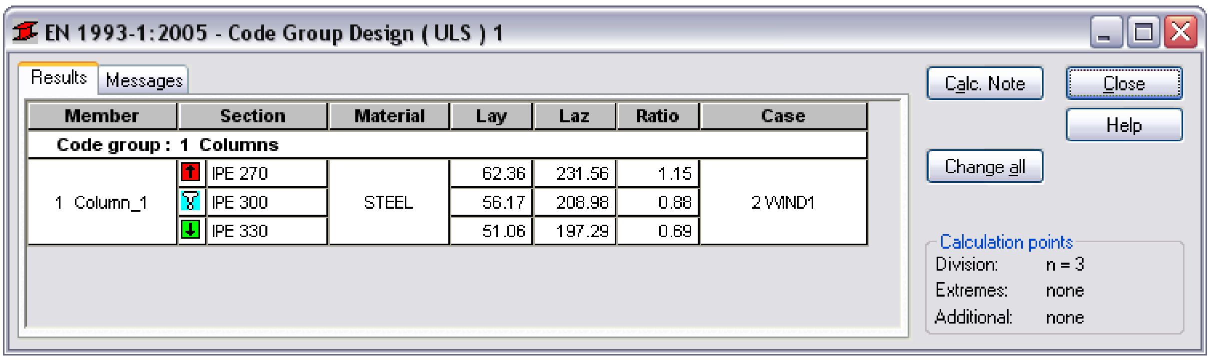



| | |
|---|---|
| **OK** | Closes the ***RESULTS*** dialog box. |
| **Close, Cancel** | Closes the ***Member Verification*** dialog box and ***Calculation Result Archiving*** dialog box |
| LMC the **New** button on the *Groups* tab in the ***Definitions – EN 1993-1:2005*** dialog box | Allows definition of the first member group. |
| Define the first group with the following parameters:<br>Number: 1<br>Name: Columns<br>Member list: LMC on the ***View*** edit viewer; select all columns while the CTRL key is pressed<br>Material: STEEL Steel | Defines the first group consisting of all columns in the structure |
| **Save** | Saves the parameters of the first member group. |
| LMC the **List** button in *Code group design* line in the ***Calculations*** dialog box | Opens the ***Code Group Selection*** dialog box. |
| LMC the **All** button (in the field above the **Previous** button), the list: 1 will appear there, **Close** | Selects member groups to be designed. |
| LMC on the **List** button in **Loads** group in ***Calculations*** dialog box | Opens the ***Load case selection*** dialog box. |

| | |
|---|---|
| LMC the **All** button (in the field above the **Previous** button), **Close** | Selects all load cases, closes the ***Load case selection*** dialog box. |
| Activate the options: *Optimization* and Limit state: *Ultimate* | |
| Press the **Options** button and Activate the *Weight* option | Opens the ***Optimization Options*** dialog box; it will result in finding the section with the smallest weight during the optimization process. |
| **OK** | Closes the ***Optimization Options*** dialog box. |
| LMC the **Calculations** button | Starts design of the selected member groups; the ***Short results*** dialog box appears on the screen (see the drawing below). |

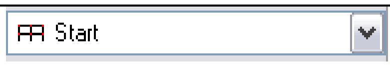
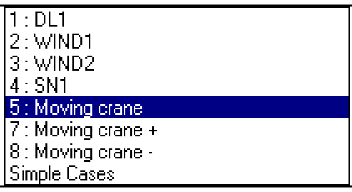



| | |
|---|---|
| LMC the **Change All** button in the ***EN 1993-1:2005 - Code Group Design*** dialog box shown above | Changes the currently used sections in the members belonging to column group to the calculated sections (from IPE 600 to IPE 360) |
| **Close, Cancel** | Closes the ***Code Group Design*** dialog box and ***Calculation Result Archiving*** dialog box. |
| Select the *Calculations*_icon from the Standard toolbar. | Recalculates the structure with the changed member sections. Once calculations are finished, the following information will be displayed in the ***RSAP*** top bar: *Results (FEM): available* |
| LMC the **Calculations** button in the ***Calculations*** dialog box | Starts design of selected member groups; the ***Short results*** dialog box appears on the screen (see the drawing below). Keep on repeating the calculations until the optimal sections are obtained. |

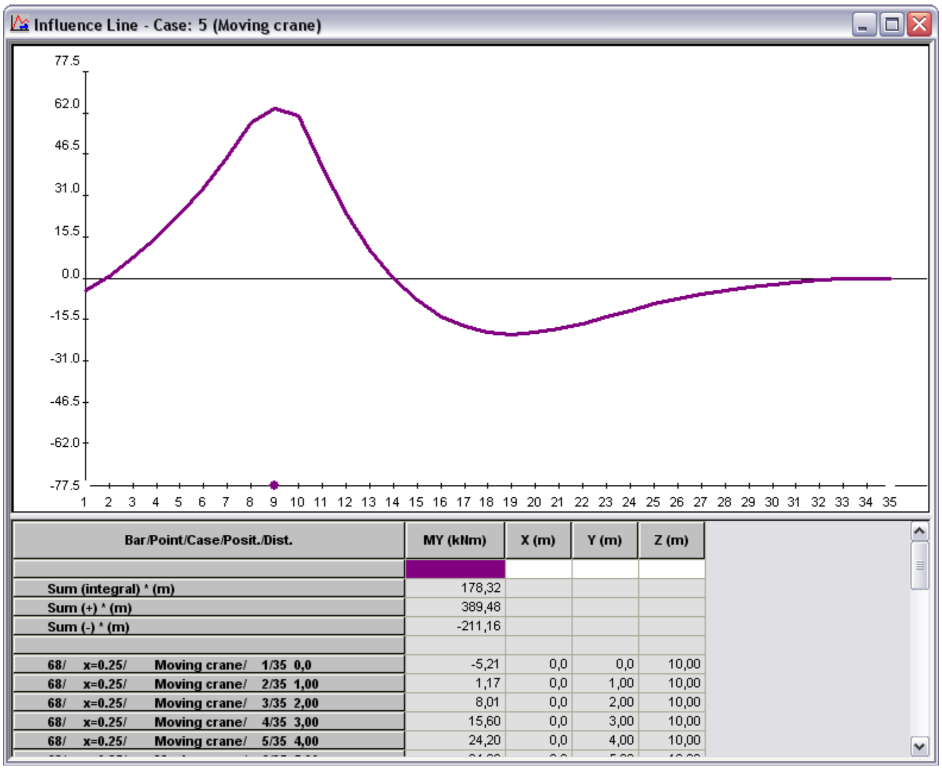


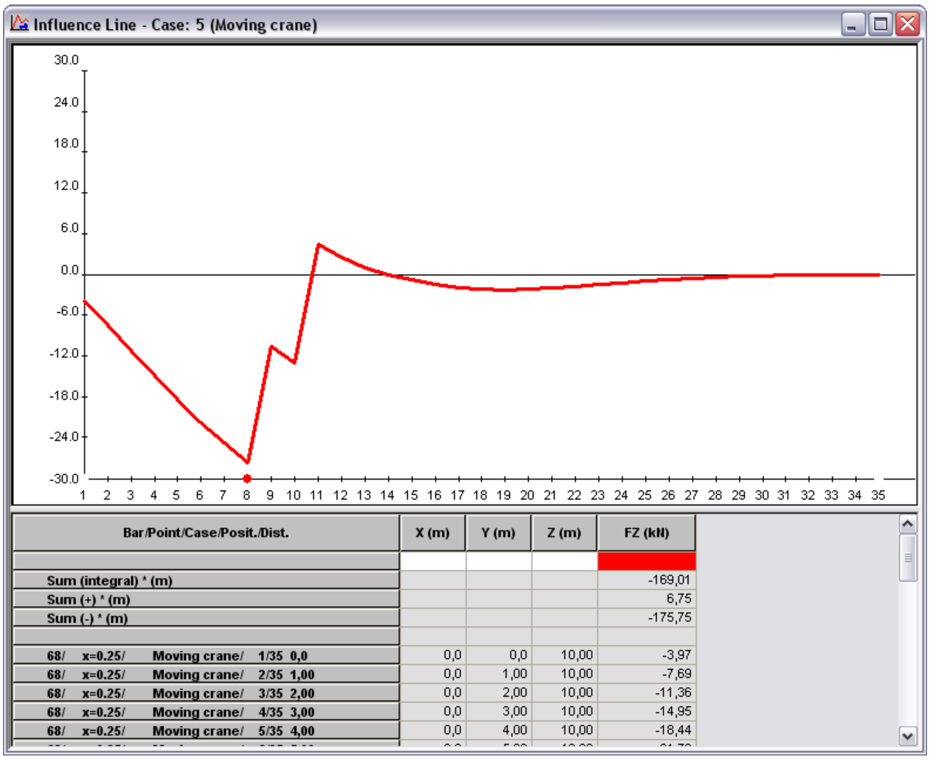
| | |
|---|---|
| LMC the **Change All** button in the ***EN 1993-1:2005 - Code Group Design*** dialog box shown above | Changes the currently used sections in the members belonging to column group to the calculated sections (from IPE 360 to IPE 300). |
| **Close, Cancel** | Closes the ***Code Group Design*** dialog box and ***Calculation Result Archiving*** dialog box. |
|  Select the *Calculations*_icon from the Standard toolbar. | Recalculates the structure with the changed member sections. Once calculations are finished, the following information will be displayed in the ***RSAP*** top bar: *Results (FEM): available* |

## 5.4 Influence Lines

| | |
|---|---|
| <br>LMC on the field to select the **Structure Model / Start** Layout | Activates the **START** layout of the ***RSAP*** program. |
| *Results menu / Advanced / Influence line* | Opens the ***Influence Lines*** dialog box. |
| On the *NTM* tab of the ***Influence Lines*** dialog box, switch on the *My* option | Selects the My bending moment for a moving load case for presentation. |
| LMC in the *Element* field and choose the right crane girder (bar no. 68) | Selects the bar for which the program will present influence lines. |
| In the *Point* field set the *Point position* at 0.25<br>*Range* from 1 to 35<br>Select *Open a new window* | The point position (equal to 0.25) means that the influence line will be created for the point at one fourth of the bar length. |
|  | Selects the 5th load case from the load case list.<br>***Note:*** *The influence lines can be created only for a moving load case.* |
| **Apply** | Opens another window presenting the influence lines for the selected quantities. |
| RMC in the *Influence lines* viewer where the influence lines are presented for the right crane girder | Opens the context menu. |
| *Add coordinates* | If the option is selected, the table located under the diagrams of influence lines will display additional columns containing the coordinates of the successive structure points (see the figure below). |
| On the *NTM* tab of the ***Influence Line*** dialog box, switch off the *My* option; activate the *Fz* option | Selects the shear force for a moving load case for presentation. |
| LMC in the *Open a new window* option, **Apply** | Opens a new window for presentation of influence lines. |

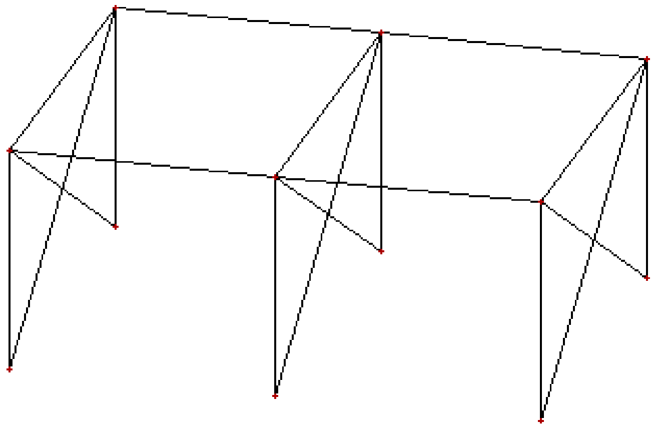
| | |
|---|---|
| RMC in the *Influence lines* viewer where the influence lines are presented for the right crane girder | Opens the context menu. |
| *Add coordinates* | If the option is selected, the table located under the diagrams of influence lines will display additional columns containing the coordinates of the successive structure points (see the figure below). |



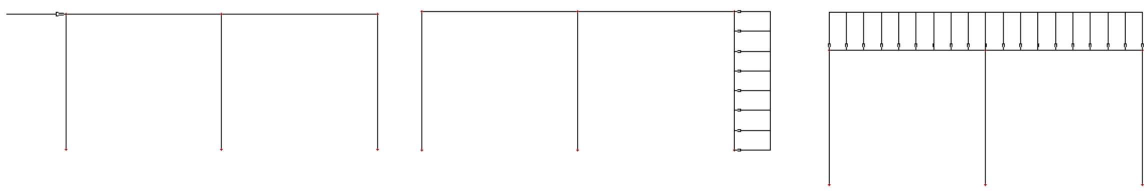



# 6. 3D Steel Structure with Steel Connections

This example presents definition, analysis and design of a simple steel 3D frame illustrated in the figure below.
Data units: (m) and (kN).



Four load cases have been assigned to each of the structure frames and three of them are displayed in the drawings below.

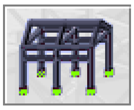


LOAD CASE 2 LOAD CASE 3 LOAD CASE 4

The following rules apply during structure definition:

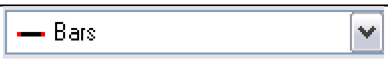
- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick,
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

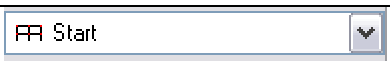
To run structure definition start the **RSAP** program (press the appropriate icon or select the command from the taskbar). The vignette window will be displayed on the screen.

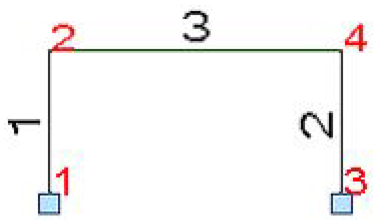
Select icon  in the first row (***Frame 3D Design***).

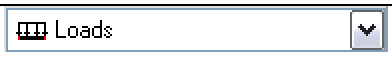
*NOTE: The European Section Database (EURO) has been used in this example.*

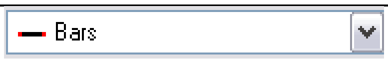
## 6.1 Model Definition

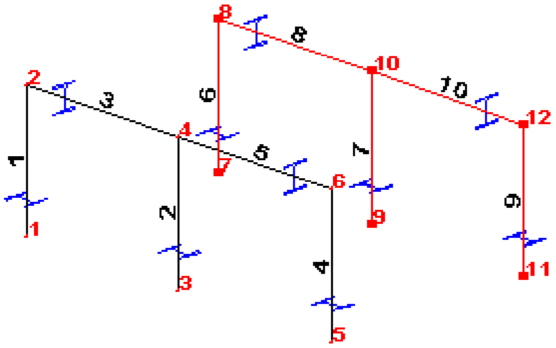
| PERFORMED OPERATION | DESCRIPTION |
| --- | --- |
|   LMC on the field to select the **Structure Model/Bars** Layout | Selects the **BARS** layout from the list of available ***RSAP*** layouts. |

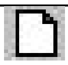
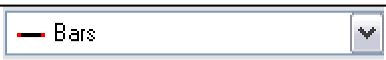
| | |
|---|---|
| LMC on the *Bar Type* field and select *Column*<br>LMC on the *Section* field and select (***HEB 340***) | Selects bar properties. The section from the European section database (*EURO*) has been used.<br>*Note: If the **HEB 340** section is not available on the list, one should press the (…) button located beside the Section field and add this section to the active section list in the **New section** dialog box* |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of bars in the structure (structure columns). |
| Enter the following points in the *Beginning* and *End* field.<br>(0,0,0) (0,0,6) **Add**<br>(8,0,0) (8,0,6) **Add** | Defines two columns of the frame. |
| LMC on the *Bar Type* field in the ***Bars*** dialog box and select *Beam*<br>LMC on the *Section* field and select (***HEB 300***) | Starts definition of a beam and selects its properties. The section from the European section database (EURO) has been used.<br>*Note: If the **HEB 300** section is not available on the list, one should press the (…) button located beside the Section field and add this section to the active section list in the **New section** dialog box* |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of a beam in the structure. |
| Enter the following points in the *Beginning* and *End* field.<br>(0,0,6) (8,0,6) Add<br>**Close** | Defines a beam. |
|  <br>LMC on the field to select the **Structure Model/Supports** Layout | Selects the **SUPPORTS** layout from the list of available ***RSAP*** layouts which allows support definition. |
| In the ***Supports*** dialog box, LMC on the *Current Selection* field (cursor is blinking in the field) | Selects structure nodes for which supports will be defined. |
| Switch to the graphic viewer; pressing the left mouse button select with the window all lower column nodes | Selected nodes 1 and 3 will be entered to the *Current Selection* field. |
| From the ***Supports*** dialog box select the Fixed support icon (the icon will be highlighted) | Selects the Fixed support type. |
| **Apply** | Selected support type will be assigned to chosen structure nodes; the defined structure is displayed on the drawing below. |
|  <br>LMC on the field to select the for the selection of the ***RSAP*** program layout<br>**Structure Model/Start** Layout | Selects the initial ***RSAP*** program layout.<br>*Note: If the structure is not visible in the graphic viewer, press the*  *Zoom All* icon. |
| <br>Make sure that propare buttons are switch on | Those icons can be found on bottom left corner of the viewer. |




| | |
|---|---|
| CTRL+A | Selects all bars. |
| *Edit pull-down menu / Edit / Vertical Mirror* | Mirrors selected bars. |
| Graphically locate the vertical symmetry axis in the place of the right column (x = 8), LMC, **Close** | Performs the axial symmetry of selected bars and closes the ***Vertical Mirror*** dialog box. |
|   LMC on the field to select the **Structure Model/Loads** layout<br>Press  to show the whole structure | Selects the ***RSAP*** program layout allowing for the structure load definition. |
| LMC on the **New** button located in the ***Load Types*** dialog box | Defines a *dead load (self-weight)* with a standard name DL1. |
| LMC on the *Nature* field *(wind)* | Selects the type of load case *wind*. |
| LMC on the **New** button<br>LMC on the **New** button | Defines two cases of *wind* load with the standard names: WIND1 and WIND2 |
| LMC on the *Nature* field *(Live1)* | Selects the type of load case *live*. |
| LMC on the **New** button | Defines a *live load* with a standard name LL1. |
| | The self-weight load was automatically applied in the first row to all structure bars (in the "Z" direction). |
| LMC on the second field in the ***Case*** column, select the 2nd load case WIND1 from the list | Defines loads for the second load case. |
| LMC on the field in the ***Load Type*** column, select (***nodal force***) as a load type | Selects the load type. |
| LMC on the field in the ***List*** column, select the upper node of the left column (no. 2) in a graphic way | Selects nodes to which a nodal force load will be applied. |
| LMC on the field in the "FX=" column and enter the value: (100.0) | Selects the direction and value of the force load. |

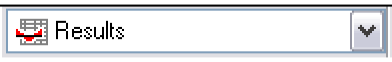
| | |
|---|---|
| LMC on the third field in the ***Case*** column, select the 3rd load case WIND2 from the list | Defines loads for the third load case. |
| LMC on the field in the ***Load Type*** column, select the (***uniform)*** load | Selects the load type. |
| LMC on the field in the ***List*** column, select graphically the right edge column (bar no. 4) | Selects bars to which the *uniform* load will be applied. |
| LMC on the field in the "PX=" column and enter the value: (-15.0) | Selects the direction and value of the uniform load. |
| LMC on the fourth field in the ***Case*** column, select the 4th load case LL1 from the list | Defines loads for the fourth load case. |
| LMC on the field in the ***Load Type*** column, select the (***uniform***) load | Selects the load type. |
| LMC on the field in the ***List*** column, select graphically both beam spans (bars No. 3 and 5) | Selects bars to which the *uniform* load will be applied.<br>*Note: 2 bars can be selected simultaneously by means of window or by indicating successive bars with CTRL button pressed.* |
| LMC on the field in the "PZ=" column and enter the value: (-20.0) | Selects the direction and value of the *uniform* load. |
| LMC in the ***View*** viewer | |
| CTRL + A | Selects all structure bars. |
| While the graphic viewer with the structure model is active, select *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the field (dX, dY, dZ), (0,10,0) | Defines the translation vector. |
| LMC on the *Number of Repetitions* field (1) | Defines the number of repetitions for performed translation operations. |
| **Execute**, **Close** | Translates the column and closes the ***Translation*** dialog box (proceed to the next step to see changes). |
| *View menu / Projection / 3d xyz* | Selects the isometric structure view (see the drawing below). |
|  <br>LMC on the field to select the **Structure Model/Bars** Layout | Selects the ***RSAP*** layout which allows definition of the bars. |



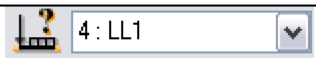
| | |
|---|---|
| LMC on the *Bar Type* field and select: ***Beam***<br>LMC on the *Section* field and select (**HEB 300**) | Selects bar properties. The section from the European section database (EURO) has been used. |
| LMC on the *Beginning* field (background color changes to green) | Starts definition of bars in the structure. |
| Enter the following points in the *Beginning* and *End* field.<br>(16,0,6) (16,10,6), **Add** | Defines a beam between the 6 and 12 nodes in the structure. |
| <br>LMC on the field to select the **Structure Model/**<br>**Sections & Materials** Layout | Selects the **SECTIONS & MATERIALS** layout from the list of available ***RSAP*** layouts. |
|  in the ***Section*** dialog box | Opens the ***New Section*** dialog box. |
| Selection of the angle family,<br>in the *Section* field selection of the (CAE 70x7) section<br>**Add**, **Close** | Defines a new section. The section from the European section database (EURO) has been used. |
| <br>LMC on the field to select the **Structure Model/ Bars** Layout | Selects the **BARS** layout from the list of available ***RSAP*** layouts. |
| LMC in the *Bar Type* field and select: ***Simple bar***<br>LMC on the *Section* field and select (CAE 70x7) | Selects bar properties. |
| LMC on the *Beginning* field (background color changes to green)<br>(16,0,6) (16,10,0), **Add**<br>(16,10,6) (16,0,0), **Add** | Bracing definition. |
| LMC on the box for selection of the ***RSAP*** program layouts<br>**Structure Model / Start** | Selects the initial layout of the ***RSAP*** program. |

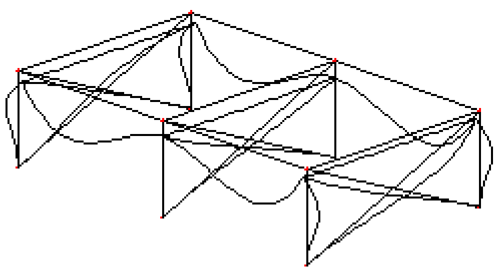
| | |
|---|---|
| LMC on the ***View*** edit viewer;<br>Select three recently defined bars (beam and bracing) - while the CTRL key is pressed LMC on three bars | |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the field (dX, dY, dZ), (-8,0,0) | Defines the translation vector. |
| LMC on the *Number of Repetitions* (2) | Defines the number of repetitions for performed translation operations. |
| **Execute**, **Close** | Column translation; closes the ***Translation*** dialog box. |

## 6.2 Structure Analysis

| | |
|---|---|
|  Select the *Calculations* icon from the Standard toolbar | Starts calculations for the defined structure |
| <br>LMC on the field to select the **Results/Results** Layout | The **RESULTS** layout of the ***RSAP*** program opens. The screen is divided into three parts: a graphic viewer containing the structure model, the ***Diagrams*** dialog box and a table with reaction values. |

## 6.3 Result Analysis

| | |
|---|---|
| <br>Select 4: LL1 from the Cases list located on the Standard toolbar | Displays results for the fourth load case. |
| Select the *Deformation* tab from the ***Diagrams*** dialog box<br>Turn on the *Deformation* option | Displays structure deformation for the selected load case. |
| **Apply** | Displays structure deformation (see the drawing below).<br>In a similar way, diagrams that exhibit other values available from the ***Diagrams*** dialog box can be viewed. |



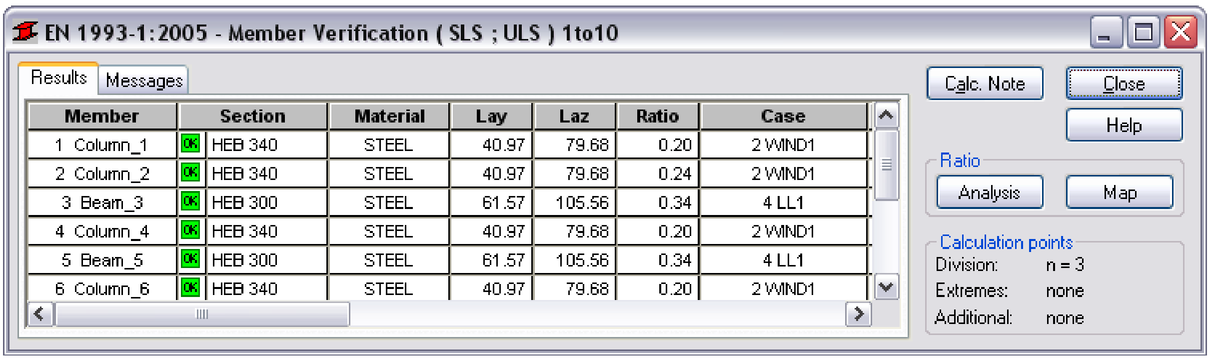
| | |
|---|---|
| Turn off the *Deformation* option in the ***Diagrams*** dialog box,<br>**Apply** | |

| | |
|---|---|
| LMC in the ***Reactions*** table on the field with the name of FZ | Selects the whole column FZ. |
| *Format menu / Alignment / Centered* and *Format menu / Font / Bold* | Edits result presentation for the Fz force. |
| RMC on the ***Reactions*** table | Calls up the context menu. |
| *Table Columns* | Selects the *Table Columns* option and opens the dialog box |
| LMC on the *Supports* tab, select the *Support code* option, **OK** | (Scroll to the left to reach the *Supports* tab). An additional column with codes defined for the structure supports appears. |

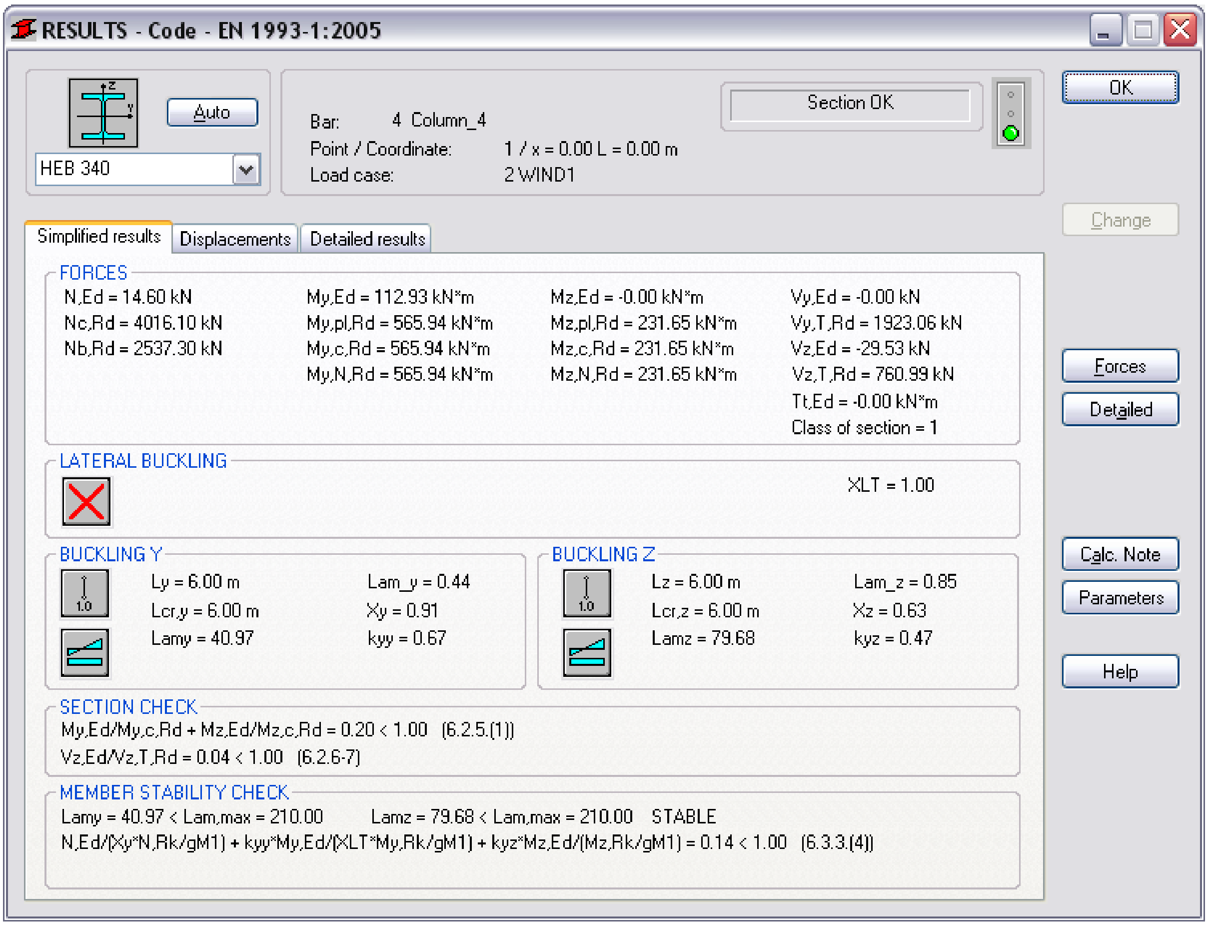
## 6.4 Steel Design

Code: EN 1993-1:2005

| | |
|---|---|
|   LMC on the field to select the **Structure Design / Steel/Aluminum Design** Layout | Starts steel member design. The screen will be divided into three parts: a graphic viewer containing the structure model, the ***Definitions*** dialog box and the ***Calculations*** dialog box. |
| LMC on the **List** button in the *Member Verification* row from the ***Calculations*** dialog box | Opens the ***Member Selection*** dialog box. |
| Enter 1to10 in the field located above the **Previous** button, **Close** | Selects members for verification. |
| LMC on the **List** button in **Loads** group in ***Calculations*** dialog box | Opens the ***Load Case Selection*** dialog box. |
| LMC on the **All** button, **Close** | Selects all load cases. |
| LMC on the **Calculations** button | Starts verification of selected structure members; the ***Member Verification*** dialog box shown below will be displayed on the screen. |




| | |
|---|---|
| LMC on the row containing simplified results for member No. 4 | Opens the ***RESULTS – Code - EN 1993-1:2005*** dialog box for the selected member. |
| LMC on the *Simplified results* tab | Displays design results for member No. 4 (see the dialog box presented below). |

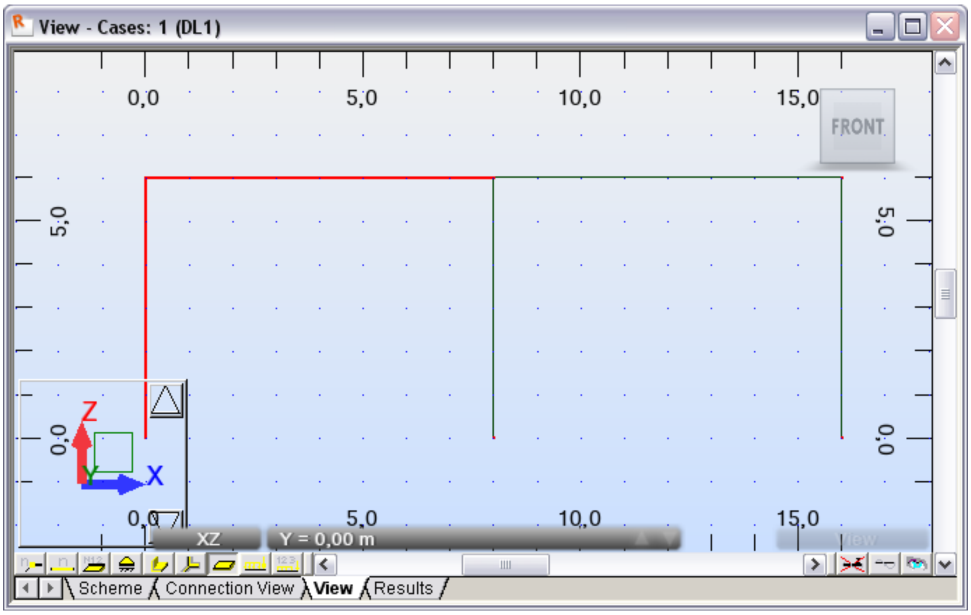



| | |
|---|---|
| Close ***Results*** and ***Member Verification*** dialog boxes | |

## 6.5 Design of Steel Connections

Code: EN 1993-1-8:2005

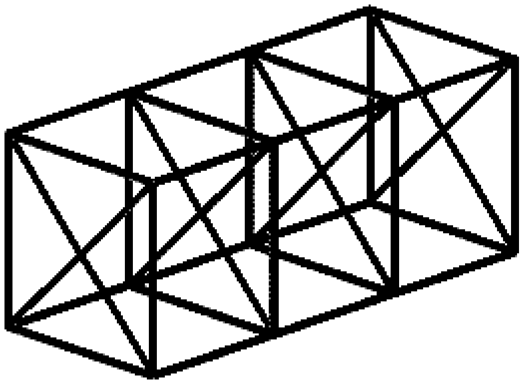
| | |
|---|---|
|   LMC the field of the **Structure design / Connections** Layout | Design of steel connections in a structure starts. The monitor screen will be divided into two parts: the ***Object Inspector*** dialog box (Steel Connections) and the graphical viewer; at the bottom of the graphical viewer there are four tabs: *Scheme, Connection View, View* and *Results*. |
| Move on to the *View* tab and while having the graphical field displaying structure view active (highlighted), select from the menu: *View / Projection / zx* | The structure will be presented as projected on the zx plane (y coordinate is assumed to equal 0). |
| While pressing the CTRL button, select both the left column and the left-side beam using the left mouse button. | Selection of bars for which the connection will be verified. |



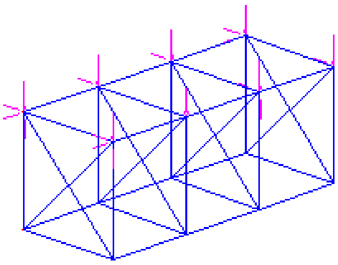

| | |
|---|---|
| *Connections / New Connection for Selected Bars* | A connection is defined between the selected bars. The ***Define a Beam-toColumn (Frame Knee) connection – EN 1993-1-8:2008*** dialog box starts to display several tabs. |
| Select the *Welded connection* option located in the dialog box (the *Geometry* tab),<br>**Apply, OK** | Selection of the type of the defined steel connection |
| *Connections menu / Calculations* | Opening the ***Connection Calculations*** dialog box |
| LMC the *List* field in the **Load cases** field | Definition of load cases considered during the connection verification |
| Enter here (1to4) | Selection of all the load cases |
| LMC the **Calculations** button | Verification of the connection starts; short results are presented in the ***Object Inspector*** dialog box and a detailed calculation note is displayed on the *Results* tab. |

# 7. 3D Steel Frame with Masses

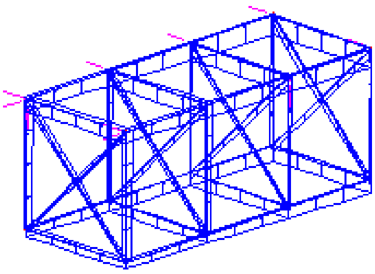
This example presents definition of a 3D steel frame shown in the figure below.
Data units: (m) and (kN).



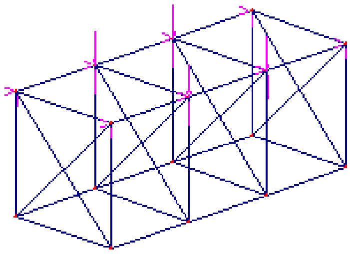
Added masses will be defined for the structure as well. They will participate in static and dynamic loads. The loads will include definition of body forces (inertia loads due to rectilinear acceleration forces) and centrifugal and angular acceleration forces (inertia loads due to rotational motion forces). The example comprises also modal and harmonic analyses.



CASES 1 and 2



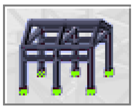
CASE 3



CASE 4

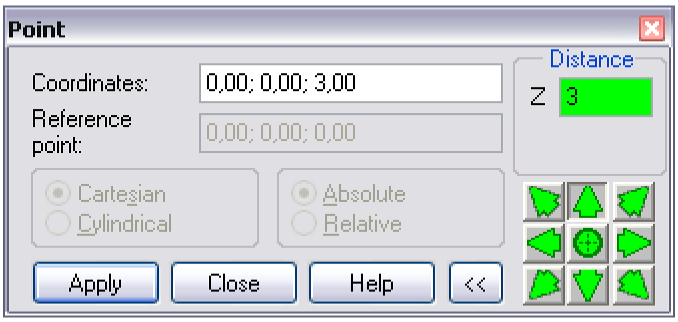
The following rules apply during structure definition:

- any icon symbol means that the relevant icon is pressed with the left mouse button,
- { x } stands for selection of the 'x' option in the dialog box,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

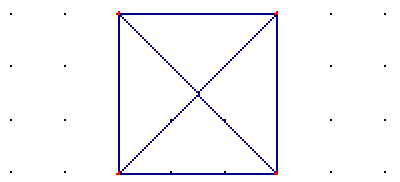
To start structure definition, run the ***RSAP*** system (press the appropriate icon or select the command from the taskbar). In the vignette window that will be displayed on the screen, the last but one icon  (***Frame 3D Design***) in the first row should be selected.

## 7.1 Model Definition

| PERFORMED OPERATION | DESCRIPTION |
|---|---|
| Select the *Bar Selections* icon from the structure model toolbar. | Opens the ***Sections*** dialog box. |
| | Opens the ***New Section*** dialog box. |
| The *Parametric* tab, section type: | Defines a new round, tubular section with determined dimensions. |
| *Label:* O 100x5<br>*d* = 10.0 (cm)<br>*t* = 0.5<br>**Add** | Defines the tubular section 100x5 (mm). |
| *Label:* O 75x3<br>*d* = 7.5 (cm)<br>*t* = 0.3<br>**Add, Close** | Defines the tubular section 75x3 (mm). |
| **Close** | Closes the ***Sections*** dialog box. |
| Select the *Bars* icon from the structure model toolbar. | Opens the ***Bars*** dialog box. |
| LMC on the BAR TYPE field and select type: *Simple bar*<br>LMC on the SECTION field and select type: ***O 100x5*** | Selects bar properties. |
| ☑ *Drag* | Switches on the *Drag* option which enables definition of successive bars in such a way that an end of the previous bar is a beginning of the next bar. |
| LMC on the *Beginning* field (color of the field background changes to green) | Starts definition of structure bars (structure columns). |
| Indicate the point wih coordinates: (0,0,0) in the graphical viewer | Defines a bar beginning. |
| Press any digit key on the keyboard | Displays the ***Point*** dialog box for numeric definition of nodes. |

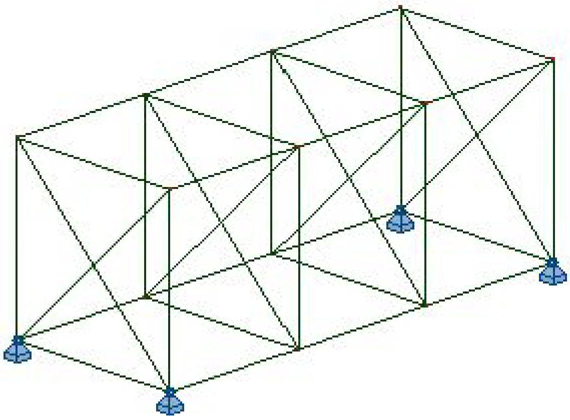


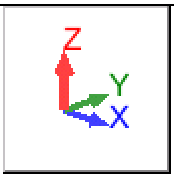
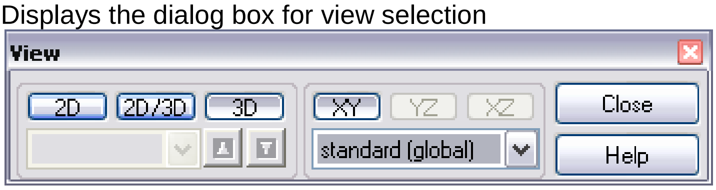
| | |
|---|---|
| Key {Backspace}, {↑}, {3}<br>Key {→}, {3}<br>Key {↓}, {3}<br>Key {←}, {3}, {Enter} | Defines bars that form a square. |
| **Close** in the ***Point*** dialog box | Closes the ***Point*** dialog box. |
| LMC on the SECTION field and select type: ***O 75x3*** | Selects bar properties. |
| ☐ *Drag* | Switches off the *Drag* option. |
| LMC on the *Beginning* field (color of the field background changes to green) | Starts definition of structure bars (structure columns). |
| In the graphical viewer indicate the points with coordinates:<br>(0, 0, 0) – (3, 0, 3)<br>(0, 0, 3) – (3, 0, 0) | Defines two bars being diagonals of the square. |



| | |
|---|---|
| **Close** | Closes the ***Bars*** dialog box. |
| *View menu / Projection / 3D xyz* | Selects structure axonometric view. |
|  | Opens the ***Sections*** dialog box. |
| LMC on the name: ***O 100x5*** on the section list, **Close** | Selects the *O 100x5* section as a default one and closes the ***Sections*** dialog box. |
| **CTRL + A** | Selects all structure bars<br>(they may also be selected with the window). |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the (dX, dY, dZ) field and enter the coordinates (0, 2.5, 0) | Defines the translation vector. |
| LMC on the *Number of repetitions* field<br>{3} | Defines how many times the copying operation is to be repeated. |
| ☑ *Drag* | Switches on the *Drag* option that enables automatic definition of bars between copied nodes. The bars defined automatically are assigned the properties that are currently chosen as default ones. |
| **Execute, Close** | Translates the structure and closes the ***Translation*** dialog box. |
| Click on the graphical viewer on the point out of the structure | Switches off the current selection of bars and nodes. |

| | |
|---|---|
| *View menu / Projection / Xy* | Selects 2D view of the structure in XY plane for Z=0.0. |
| Select the *Supports* icon from structure model toolbar. | Opens the ***Supports*** dialog box. |
| In the ***Supports*** dialog box select the icon which stands for pinned support - *Pinned* (it will be highlighted) | Selects support type. |
| LMC on the *Current selection* field | Selects structure nodes at which structure supports will be defined. |
| Switch to the graphical viewer; keeping the left mouse button pressed – select with the window nodes of the top bar and (with Ctrl key pressed) nodes of the bottom bar | Enters the list of selected nodes: 1 4 13 16 to the *Current selection* field. |
| **Apply, Close** | Assigns the selected support type to the selected structure nodes; closes the ***Supports*** dialog box. |
| *View menu / Projection / 3D xyz* | Selects structure axonometric view. |
| Turn on the *Support Symbols* icon in the lower left corner. | Switches on displaying of supports. |

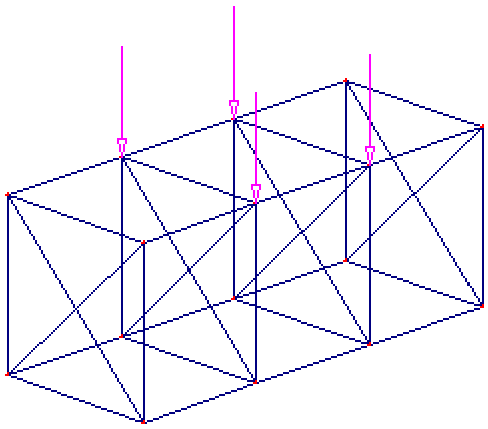


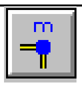
| | |
|---|---|
| Loads<br>LMC on the field to select the **Structure Model/Loads** Layout | Selects the layout of the ***RSAP*** system which facilitates definition of structure loads (there are dialog box and table for load definition). |
| LMC on the **New** button in the ***Load Types*** dialog box | Defines the load case with the nature: self-weight and standard name DL1.<br>In the first load case the self-weight of the whole structure is added automatically, which can be seen in the load table. |
|  | Displays the dialog box for view selection<br> |
| 2D | Selects 2D view of the structure. |
| XY | Selects XY projection plane (initially, for Z=0.0). |

| | |
|---|---|
| 2D 2D/3D<br>3,00 enter {3} {Enter}<br>**Close** | Selects XY projection plane with the coordinate Z=3.0.<br><br>Closes the ***View*** dialog box. |
| *Loads menu / Load Definition* | Opens the dialog box fo load definition. |
| *Self-weight and mass* tab | Goes to the tab for definition of self-weight loads and inertia loads. |
| *Added masses - nodes* | Opens the dialog box for definition of added masses. |
| Enter *weight values (kG):*<br>*X* = 100<br>*Y* = 100<br>*Z* = 100<br>☑ *Apply to all cases*<br>**Add** | Defines nodal masses whose weight is 100 kG for translational degrees of freedom.<br>The masses will participate in all the load cases (static and dynamic ones). |
| LMC on the *Apply to* field | Selects structure nodes at which nodal masses will be defined. |
| Switch to the graphical viewer; keeping the left mouse button pressed select with the window - all the nodes in the presented work plane | Enters the list of selected nodes: 2to14By4 3to15By4 to the *Apply to* field. |
| **Apply, Close** | Applies defined added masses to selected structure nodes; closes the ***Load Definition*** dialog box. |
| *View menu / Projection / 3D xyz* | Selects structure axonometric view. |
| LMC on the **New** button in the ***Load Types*** dialog box | Defines a new load case with the nature: self-weight and the standard name: DL2. |
| *Loads menu / Load Definition* | Opens the dialog box for load definition. |
| *Self-weight and mass* tab | Goes to the tab for definition of self-weight loads and inertia loads. |
| *Body forces* | Opens the dialog box for definition of inertia loads due to rectilinear acceleration forces. |
| ☑ *relative x g*<br>Enter *a:*<br>*Z* = -1<br>☑ *Apply to added masses*<br>**Add** | Defines body forces with acceleration of gravity *g* for nodal masses, i.e. takes account of self-weight of added masses. |
| **Apply, Close** | For the load applied to added masses object selection is not required because action of this load concerns all the masses assigned to a given load case.<br>Closes the ***Load Definition*** dialog box. |

| | |
|---|---|
| In the ***Load Types*** dialog box select the load nature: *Live1*<br>enter the case name: TRANSPORT<br>LMC on the **New** button | Defines a new load case with the nature: live, named: TRANSPORT.<br><br>*This load case is aimed at modeling the action of inertia forces on the frame and on additional masses due to rotational motion forces caused by (ship) rolling during transport.* |
| *Loads menu / Load Definition* | Opens the dialog box for load definition. |
| *Self-weight and mass* tab | Goes to the tab for definition of self-weight loads and inertia loads. |
| *Centrifugal and angular acceleration forces* | Opens the dialog box for definition of inertia loads due to rotational motion forces, i.e. angular acceleration forces (tangential forces) and velocity forces (centrifugal forces). |
| Enter coordinates of the rotation center C: (0.0, 0.0, -5.0)<br>Enter to *Centrifugal velocity and acceleration (Rad/..):*<br>v*X* = 0,5 a*X* = 0,2<br>v*Y* = 0,2 a*Y* = 0,1<br>**Add** | Defines inertia forces due to rotational motion about point C.<br>Rotation about X axis with velocity v= 0.5 (rad/s) and acceleration a= 0.2 (rad/s2).<br>Rotation about Y axis with velocity v= 0.2 (rad/s) and acceleration a= 0.1 (rad/s2). |
| LMC on the *Apply to* field | Selects structure elements for which centrifugal and angular acceleration forces will be defined. |
| Click on the graphical viewer;<br>{ Ctrl + A } | Selects the whole structure. Enters the list of all bars to the *Apply to* field. |
| **Apply** | Defines the load. |
| *Centrifugal and angular acceleration forces* | Opens again the dialog box for definition of inertia loads due to rotational motion forces. |
| ☑ *Apply to added masses*<br>**Add** | For the current load parameters - selects the option enabling definition of load generated by added masses. |
| **Apply**, **Close** | Defines the load; for the load applied to added masses object selection is not required because action of this load concerns all the masses assigned to a given load case.<br>Closes the ***Load Definition*** dialog box. |
| In the ***Load Types*** dialog box,<br>for the load nature: *Live1*<br>enter the case name: ROTOR<br>LMC on the **New** button | Defines the new load case with the nature: live, named: ROTOR.<br><br>*This load case is aimed at modeling operation of the equipment mounted on the frame by considering its weight and vibrating force in hamonic analysis.* |
| *Loads / Load Definition* | Opens the dialog box for load definition. |
| *Node* tab<br>*Nodal force* | Opens the dialog box for definition of loads due to nodal forces. |

| | |
|---|---|
| Enter*:*<br>*FZ* = -0,5 (kN)<br>**Add** | Defines the nodal force. Afterwards, this load will be used in harmonic analysis as an excitation load. |
| LMC on the *Apply to* field | Selects structure nodes at which nodal forces will be applied. |
| Switch to the graphical viewer; keeping the left mouse button pressed select with the window - four middle nodes on the top plane of the frame | Enters the list of selected nodes: 6 7 10 11 to the *Apply to* field. |
| **Apply** | Assigns defined forces to selected structure nodes. |

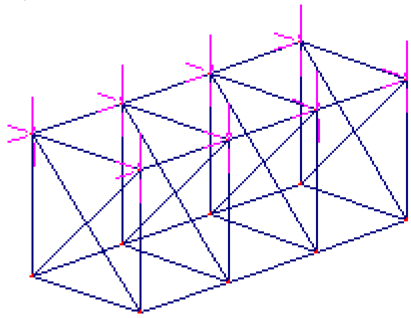
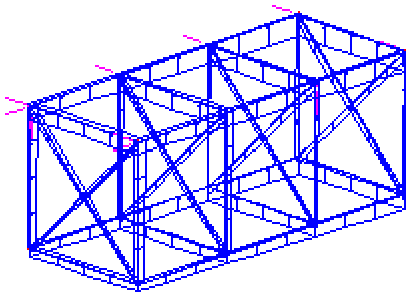


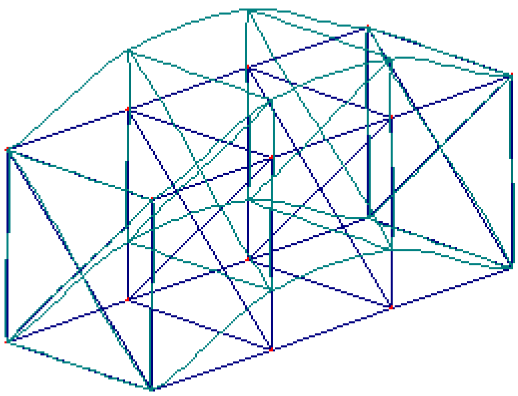
| | |
|---|---|
| *Self-weight and mass* tab | Goes to the tab for definition of self-weight loads and inertia loads. |
|  *Added masses – nodes* | Opens the dialog box for definition of added masses. |
| Enter *Values of weight (kG):*<br>*X* = 0<br>*Y* = 0<br>*Z* = 200<br>☑ *Apply to all cases*<br>**Add** | Defines nodal masses with weight 200 kG for the direction of freedom Z.<br>Masses are defined only for the current load case. |
| LMC on the *Apply* to field | Selects structure nodes at which added masses will be defined. |
| Switch to the graphical viewer; keeping the left mouse button pressed select with the window - four middle nodes on the top plane of the frame | Enters the list of selected nodes: 6 7 10 11 to the *Apply* to field. |
| **Apply**, **Close** | Applies defined added masses to the selected structure nodes;<br>closes the ***Load Definition*** dialog box. |
| *Analysis menu / Analysis Types* | Opens the ***Analysis Type*** dialog box. |
| **New** | Opens the ***New Case Definition*** dialog box. |

| | |
|---|---|
| *Modal*<br>**OK** | Selects modal analysis. |
| **OK** | Accepts default parameters of modal analysis and closes the dialog box. |
| **Close** | Closes the ***Analysis Type*** dialog box. |
| *Loads menu / Mass Table* | Opens the table of added masses. |
| in the mass table | Closes the *Added masses* table. |
| *Loads menu /Manual Combinations* | Opens the ***Combination Definition / Modification*** dialog box. |
| **OK** in the dialog box for definition of combination parameters | Accepts combination parameters. Opens the ***Combinations*** dialog box. |
| Select case 1 from *Case list*, enter the factor to the *Factor* field | Defines combination cases and factors.<br>*Note: if "auto" is selected in the Factor field, then combination factors will be adopted automatically according to the code assumed in Job Preferences.* |
| LMC on for the selected case, next, repeat the selection for cases nos. 2 and 3,<br><br>**Apply** | Defines the combination of cases 1+2+3, as shown below:<br>Factor / No. / Case name<br>1.35 / 1 / DL1<br>1.35 / 2 / DL2<br>1.50 / 3 / TRANSPORT |
| **New** | Defines a new combination. |
| **OK** in the dialog box for definition of combination parameters | Accepts combination parameters; opens the ***Combinations*** dialog box. |
| Select cases and move them to the field with combination definition for cases 1, 2 and 4.<br>**Apply, Close** | Defines the combination of cases 1+2+4; closes the ***Combinations*** dialog box. |

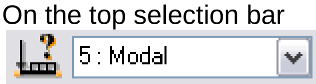
## 7.2 Calculations and Result Analysis

| | |
|---|---|
| Select the *Calculations* icon from the Standard toolbar. | Starts calculations of the defined structure. |
| Results<br>LMC on the field to select of the **Results/Results** Layout | Opens the **RESULTS** layout of the ***RSAP*** system. The monitor screen will then be split into three parts: graphical viewer with a structure model, ***Diagrams*** dialog box and table presenting reaction values. |
| RMC, *Display* | Opens the dialog box for selection of structure attributes to be displayed. |
| LMC on the *Loads* tab | Goes to the tab for selection of structure attributes concerned with loads to be displayed. |

| | |
|---|---|
| ☑ *Load symbols*<br>☑ *Forces generated automatcally*<br>**OK** | Switches on display of forces that are generated automatically for certain types of loads. |
| On the top selection bar<br>2 : DL2<br>select 2: DL2 | Selects the current load case, the program displays nodal forces generated automatically for added masses in the body force load.<br> |
| On the top selection bar:<br>3 : TRANSPORT<br>select 3: TRANSPORT | Selects the current load case, the program displays linear and nodal foces generated automatically for added masses and bars in the rotational motion load.<br> |
| LMC on in the bottom toolbar | Restores the default set of displayed attributes. |
| Select the *Deformation* tab in the ***Diagrams*** dialog box<br>Switch on the *Deformation* option | Selects presentation of structure deformations for the selected load case. |
| LMC on the **Apply** button | Presents structure deformations (see the figure below); similarly, diagrams of other quantities available in the ***Diagrams*** dialog box may be presented. |



| | |
|---|---|
| Switch off the *Deformation* option in the ***Diagrams*** dialog box,<br>**Apply** | |
| *Results menu / Stresses* | Opens the *Stresses* result table. |

| | |
|---|---|
| On the top selection bar   enter 6 and 7 {Enter} | Selects the combination 6 and 7 as the current case in the table. |
| RMC in the table<br>*Table Columns* | Opens the ***Bar value selection*** dialog box from the context menu in the table. |
| Switch off the stress options:<br>*Bending*<br>*Axial*<br>**OK** | Excludes the columns with results for stresses due to axial forces and bending from the table.<br>Closes the dialog box with parameters. |
| LMC on the ***Global extremes*** tab in the table | Goes to the tab where maximum and minimum values are displayed for the quantities and selection set in the table. |
| in the stress table | Closes the *Stresses* table. |
| *Results menu / Advanced / Modal Analysis* | Opens the *Dynamic Analysis Results* table. |
| On the top selection bar   select case 5: Modal | Select the modal analysis case. |
| in the table with the dynamic analysis results | Closes the *Dynamic Analysis Results* table. |

# 8. Defining and Analyzing a Concrete Floor

This example will demonstrate step-by-step how the user can define and analyze a simple slab with an opening.

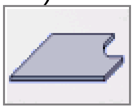
Data units: (m) and (kN).

A slab with an opening will be generated and analyzed. The slab will consist of concrete elements. All the steps required will be presented. Four load cases will be defined (self-weight and three live load cases). Five structure modes will also be found.

The following rules will apply during structure definition:

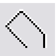
- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- LMC and RMC **-** abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick,
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

To run structure definition start the ***RSAP*** program (press the appropriate icon or select the command from the taskbar). The vignette window will be displayed on the screen.

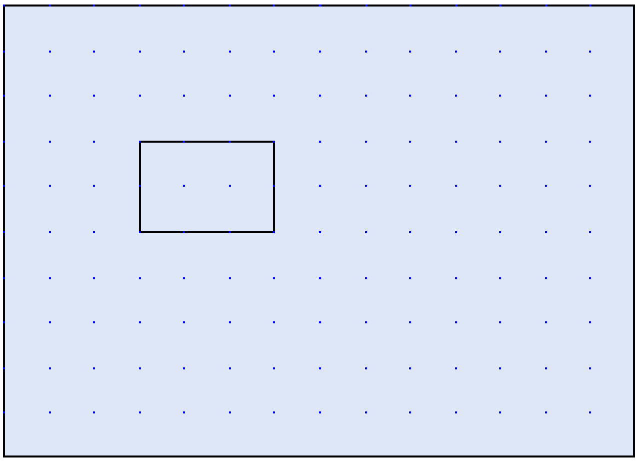
Select icon  in the second row (Plate Design).

## 8.1 Model Definition

### 8.1.1 Contour Definition

| PERFORMED OPERATION | DESCRIPTION |
|---|---|
| *View menu / Grid / Grid Step Definition* | Opens the ***Grid Step Definition*** dialog box. |
| Dx = 1.0<br>Dy =1.0 | Defines grid step on the screen (equal in both directions) |
| **Apply**, **Close** | Accepts the defined parameters and closes the ***Grid Step Definition*** dialog box. |
|  Select the *Objects* icon from the Structure Model toolbar | Selects polyline to define a rectangle. |
| LMC on *Polyline – Contour* option in Definition Method  | Selects polyline to define a slab contour. |
| Using mouse select the following points in the graphical window:<br>(-7, -5)<br>(-7, 5)<br>(7, 5)<br>(7, -5)<br>(-7, -5) | Defines a rectangle contour. |

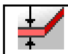
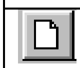
| | |
|---|---|
| (-4, 2)<br>(-4, 0)<br>(-1, 0)<br>(-1, 2)<br>(-4, 2) | Defines a rectangle contour by entering four rectangle vertexes and the fifth point to make a closure. It models dimensions of an opening in the slab. |
| **Close** | Closes ***Polyline - Contour*** dialog box. |



### *8.1.2 Mesh Definition*

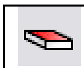
| | |
|---|---|
| *Tools menu / Job Preferences / Work Parameters* | Opens the Work Parameters dialog. |
| In the *Meshing type* field select:<br>User, LMC **Modification** button | Selects user defined meshing type. |
| LMC in *Available Meshing Methods / Delaunay* | Selects Delaunay's option. |
| *Mesh Generation / Element size*: (0,5 m) | Defines the size of the mesh size. |
| **OK** | Accepts changes in the ***Meshing Options*** dialog box. |
| **OK** | Accepts changes in the ***Job Preferences*** dialog box. |

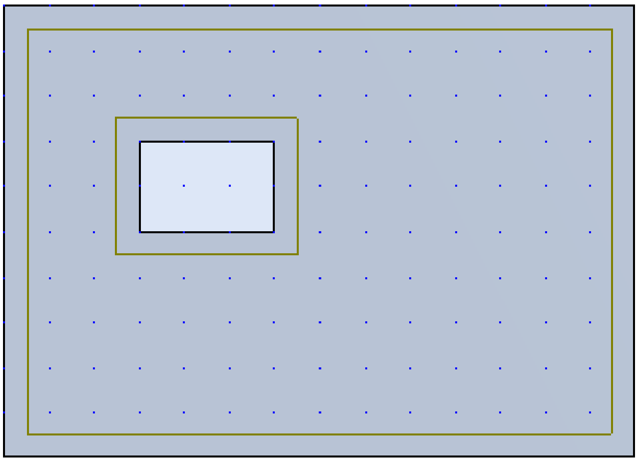
### *8.1.3 Slab Properties*

| | |
|---|---|
|  Select the *Thickness* icon from the Structure Model toolbar | Opens window where the slab thickness will be defined. |
|  Select the *New Thickness Definition* icon from the Structure Model toolbar | Defines a *new* FE thickness. |
| On the *Homogenuous* tab in the *Th*= field type the value (30) | Defines slab thickness; in the *Label* field enter TH30. |

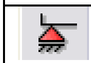
| | |
|---|---|
| In the *Material:* field select (C25/30) | Selects C25/30. |
| **Add**, **Close** | Adds the new thickness: TH30 and closes the ***New thickness*** dialog box. |
| **Close** | Closes the ***FE Thickness*** dialog box. |

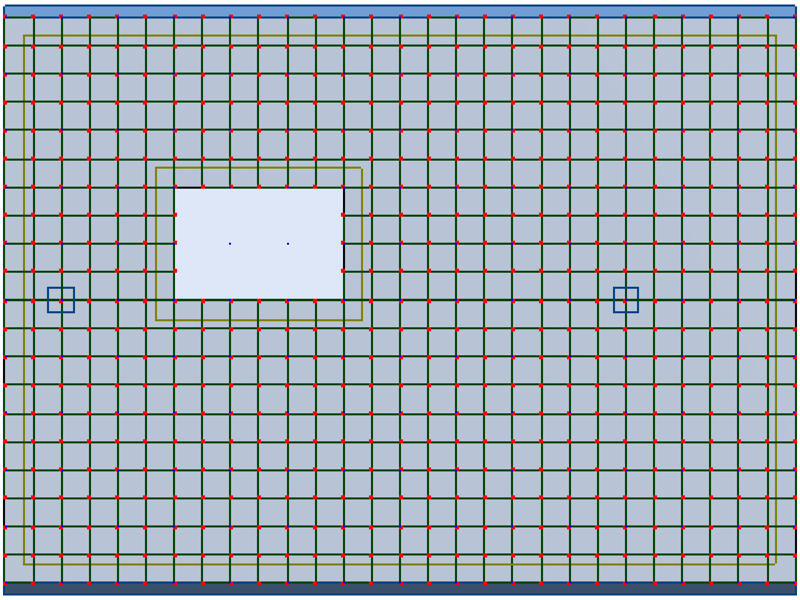
### 8.1.4 Panel and Opening Definition

| | |
|---|---|
|  Select the *Panals* icon from the Structure Model toolbar | Opens the ***Panel*** idialog box to define the plate. |
| LMC *Contour type: Panel* | Defines the panel around the opening. |
| LMC *Properties / Reinforcement*:<br>Select: RC floor<br>LMC *Properties/Thickness*:<br>Select: TH30<br>LMC *Properties/Model*:<br>Select: Shell | Selects thickness type TH30, reinforcement and model type. |
| LMC in *Creation with / Internal Point*:<br>LMC at (0, 0) in the *View* graphical window | Creates a contour for the panel. Select a point inside the panel by clicking outside of the opening defined above but inside the panel rectangle, for example at (0,0) point. And the contour appears around the panel. |
| LMC *Contour type: Opening* | Defines the opening contour. |
| LMC in *Creation with/Internal Point*:<br>LMC at (-3, 1) in the *View* graphical window | Creates a contour for the hole. Select a point inside the opening by clicking inside the opening defined above, for example at (-3,1) point. And the contour appears on the opening. |
| **Close** | Closes panel definition. |



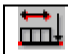
### 8.1.5 Support Definition

| | |
|---|---|
| *Analysis menu / Calculation Model / Generation* | Generates a finite element mesh according to the adopted parameters of mesh generation |
|  Select the *Supports* icon from the Structure Model toolbar | Selects ***Supports*** icon option to define the supports for the slab. |
|  | Defines a new support type |
| **Advanced** on the *Rigid* tab | Opens the ***Support Definition – Advanced*** dialog box to define a support determined by means of dimensions of the column cross-section |
| *column* | Selects the support type - column |
| Rectangular<br>*b = 45, h = 45* | Defines the column type (rectangular) and dimensions of the column cross-section. |
| **OK** | Closes the ***Support Definition – Advanced*** dialog box |
| In the *Label* field enter *Column45x45,*<br>set all the directions (UZ, RX, RY) as fixed | Specifies name of the defined support type |
| **Add** and **Close** | Adds the new support type (column45x45) to the list of available support types and closes the ***Support Definition*** dialog box |
| LMC on *column45x45* | Selects type of the support. |
| LMC on *Current Selection*<br>LMC in the field<br>LMC on points P1 (-6, 0), P2 (4, 0), during selection press down CTRL button | Selects the points at which supports will be defined – see the figure below. Numbering of nodes may differ after completing generation of a finite element mesh. The user should select the corner points P1, P2 as shown in the drawing below. |
| **Apply** | Defines supports in the structure. |
| **LMC** on the *Linear* tab. Select *Pinned* type of support and LMC on upper and lower eges of the slab (1_Edge(2), 1_Edge(4)). | Defines linear pinned support in the structure. |

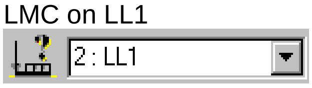
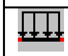
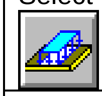


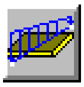
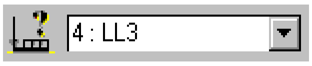
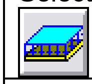
| | |
|---|---|
| **Close** | Closes the ***Supports*** dialog box. |

### *8.1.6 Load Case Definition*

| | |
|---|---|
|  Select the *Load Types* icon from the Structure Model toolbar | Opens the ***Load Types*** dialog box. |
| | After generation of a finite element mesh the first load case a dead load (self-weight) has been generated. |
| LMC on the *Nature* field *(Live1)* | Selects the load nature: *live*. |
| LMC on the **New** button,<br>LMC on the **New** button,<br>LMC on the **New** button, **Close** | Defines three *live load* cases with standard names LL1, LL2 and LL3 and closes ***Load Types*** dialog box. |

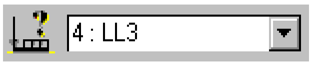
### *8.1.7 Load Definition for Generated Cases*

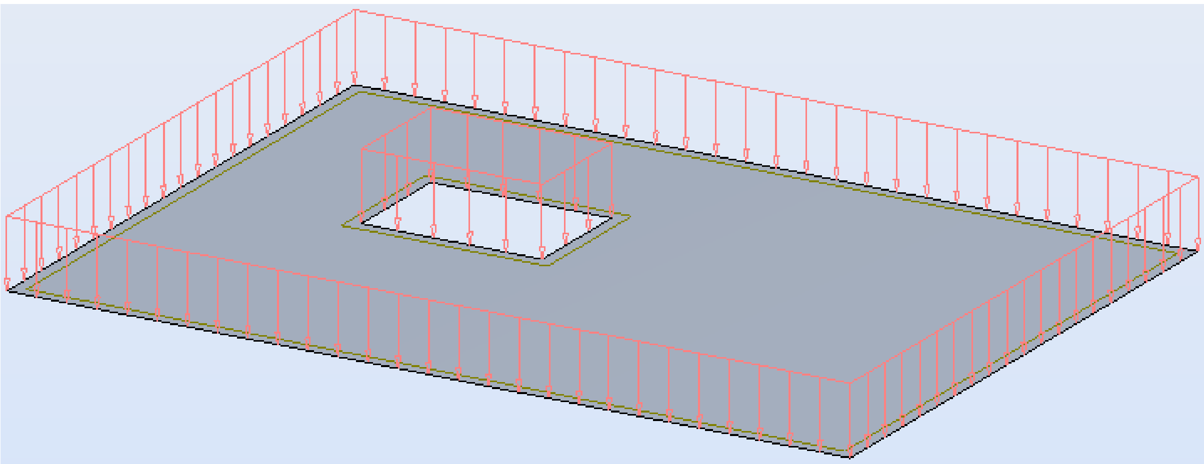
| | |
|---|---|
| LMC on LL1   | Selects load case LL1. |
|  Select the *Load Definition* icon from the Structure Model toolbar | Selects Load Definition. |
| Select the *Surface* Tab  | Selects ***Uniform planar load on contour***. |
| *Load Parameters*, *Z*: (-5 kPa) | Defines the load intensity. |

| | |
|---|---|
| LMC *Contour definition* | Defines a rectangle contour on which the load will be applied. |
| Define the following points<br>(2, 2)<br>(4, 2)<br>(4, 1)<br>(2, 1) | |
| LMC on **Add** button at the very bottom of the ***Uniform Planar Load*** dialog box | |
| LMC on LL2<br> | Selects load case LL2. |
| Select the *Surface* Tab<br> | Selects ***Linear load 2p***. |
| *Values: P1*, *P2*<br>***Z***: (-10, -10) kPa<br><br>Point *Coordinates*<br>***A***: (1, -5)<br>***B***: (1, 5) | Defines the load intensity (P1 and P2) on the two ends of the load line segment and their coordinates (A and B). |
| LMC **Add** | |
| LMC on LL3<br> | Selects load case LL3. |
| Select the *Surface* Tab<br> | Selects ***Uniform planar load*** |
| *Values:*<br>***Z***: (-3) kPa | Defines values of the load for a whole panel. |
| **Add,**<br>*Apply to: 1* **Apply,**<br>**Close** | Closes the ***Load Definition*** dialog box |
| *Loads menu / Automatic Combinations* | Defines combinations. |
| *Combinations according to code: EN 1990:2002*<br>LMC on the *Full automatic combinations type* field<br>**More >** | Selecting this option and clicking generates full code combinations after static structure calculations.<br><br>You do not need to specify parameters for generating combinations; however, if you want to change the parameters for generating combinations (such as, definitions of groups, relations, and so on), click **More**. It opens the **Load Case Code Combinations** on the *Combinations* tab. |

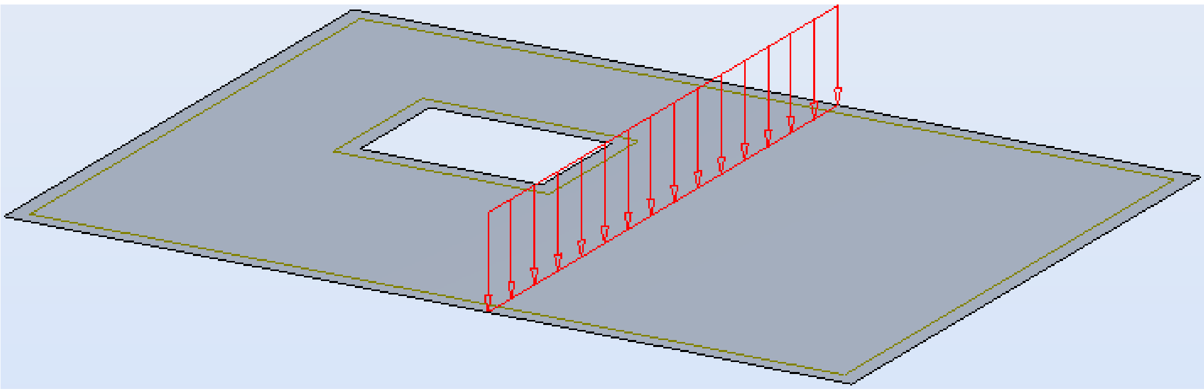
| | |
|---|---|
| Make sure that *ULS* and *SLS* options are selected,<br>Unselect *ACC* and *FEU* options | Numeric procedures let you calculate numerous combination types (rules) described in the code files. Depending on the combination method and coefficient number, these regulations are included in the template used in various codes as follows: requirements for dead load, live loads, accidental load, and seismic load combinations. Which regulations **RSAP** considers is defined by the code file.<br>Similar to the active case number, you can decide before calculating the code combinations, which of the proposed sets to disregard. |
| **Generate** | The combinations will be generated after calculations. |

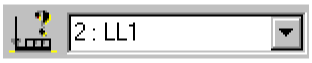
### *8.1.8 Display of Generated Load Cases*

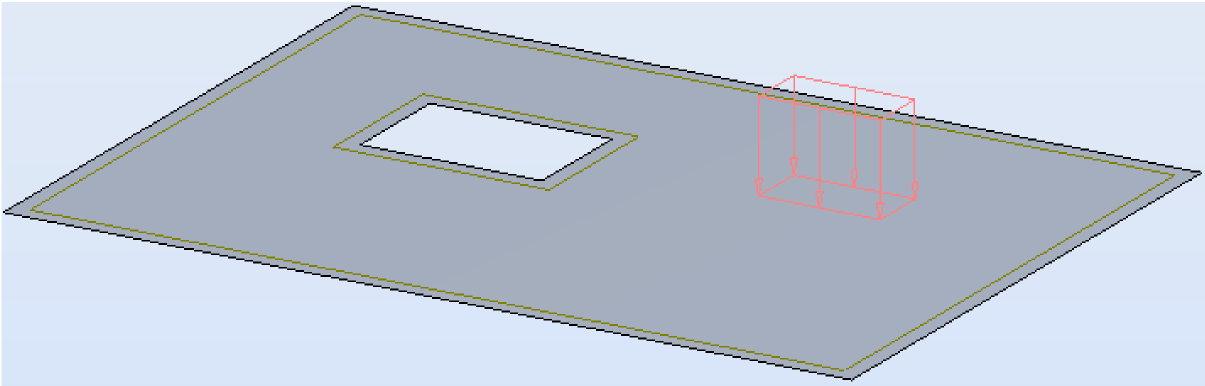
| | |
|---|---|
| *View menu / Projection / 3D xyz* | Selects isometric view. |
| *View menu / Display* | Opens the ***Display*** dialog box |
| LMC *on Loads* tab | |
| LMC on *Load symbols* option | Selects *Symbols* checkbox |
| LMC on *Panels / FE* tab | Moves on to the *Finite Elements* tab in the ***Display*** dialog box |
| LMC on the options: *Panel description* and on the option *Finite elements.* | Switches off the options of structure element display |
| LMC on *Nodes* tab<br>Turn on the *Hide nodes* option | |
| **Apply**, **OK** | |
| LMC on LL3   | Selects load case LL3 |



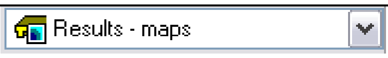
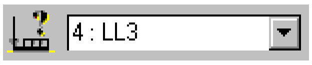
| | |
|---|---|
| LMC on LL2   | Selects load case LL2 |



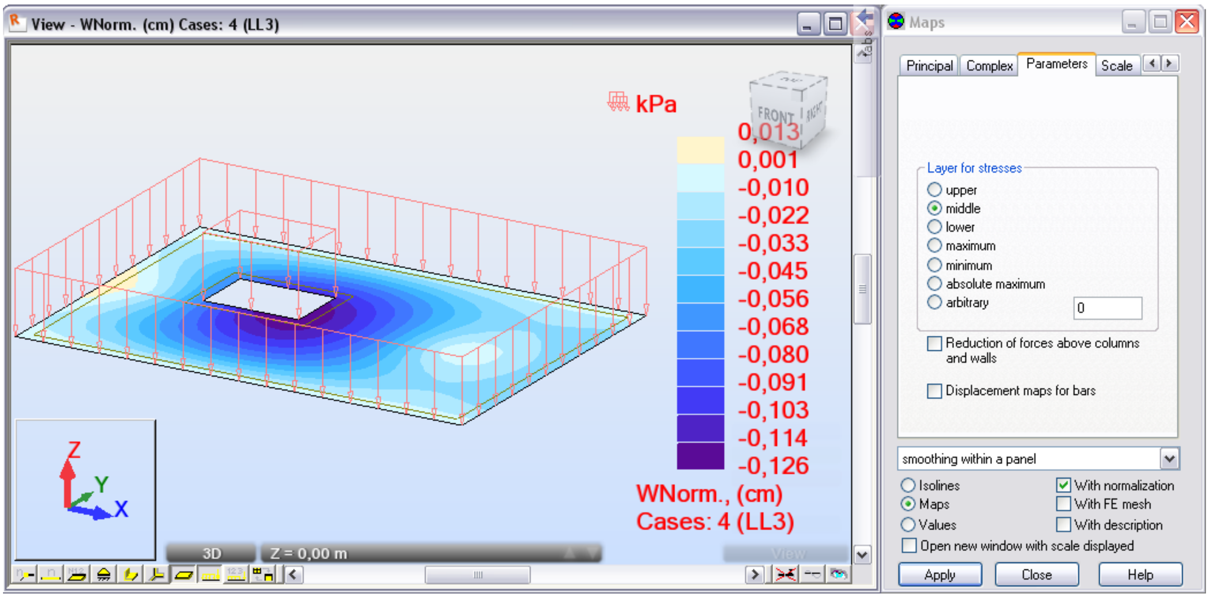
| | |
|---|---|
| LMC on LL1  | Selects load case LL1 |

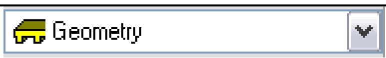


## 8.2 Structural Analysis / Results (Maps on Panels Cuts)

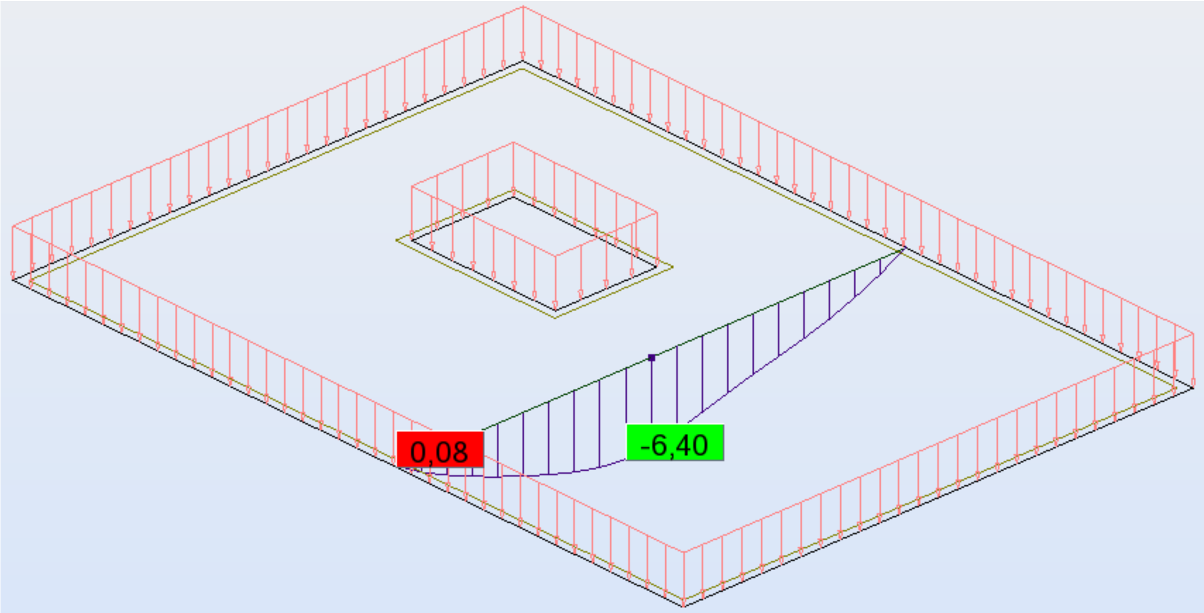
| | |
|---|---|
|  Select the *Calculations* icon from the Standard toolbar | Starts calculations of the defned structure |
|  LMC in the ***RSAP*** program layout selection, ***Results / Results - maps*** | Opens the **RESULTS** / **RESULTS - MAPS** layout of the ***RSAP*** program. |
| *Tools menu / Job Preferences* | Opens the ***Job Preferences*** dialog box |
| *Units and Formats / Other* | Selects the option that enables defining a number of decimal places for selected quantities. |
| Increase of the number of decimal places for Displacement to 3 | Increases the number of decimal places for Displacement to 3. |
| LMC on LL3  | Selects load case LL3. |
| LMC on the *Displacements - u, w* option in the ***Maps*** dialog box | Selection of the displacement to be presented |

| | |
|---|---|
| Go to the *Parameters* tab in the ***Maps*** dialog box and select the *middle* in the *Layer for stresses* field | Selection of the layer for which the determined displacements will be presented |
| **Apply** | |




| | |
|---|---|
| Go to the *Detailed* tab in the ***Maps*** dialog box and switch off the presentation of displacements for the plate, **Apply** | |
|   **Structure Model / Geometry** Layout | Select the initial layout of the ***RSAP*** program |
| *Results menu / Panel Cuts* | Opens the ***Panel Cuts*** dialog box that allows creating diagrams of internal forces and displacements in planar finite elements |
| LMC the *Displacements – u, w* option on the *Detailed* tab | Selects the Mxx moment diagram for presentation |
| On the *Definition* tab in the ***Panel Cuts*** dialog box select the *Parallel to axis -Y* option, enter the coordinates: (1.00, -5.00) into the field below | Selects the method of cut plane definition |
| Move to the *Parameters* tab and afterwards, select the *middle* option in the *Layer for stresses* field | Selects the layer for which the displacements in a given cut will be presented |
| On the *Diagrams* tab select the following options: *labels* in the *Diagram descriptions* field, *fence* in the *Filling field* and *normal* in the *Diagram position* field | Selects the manner of diagram presentation on structure cuts |
| **Apply** | Switches on presentation of displacements on the panel cuts (the drawing below). The drawing below presents the structure as defined so far. |

| | |
|---|---|
|  Select the *Rotate,Zoom,Pan* icon from the Standard toolbar | Using the option rotate the plate to view the diagram (which is initially shown under the plate). |

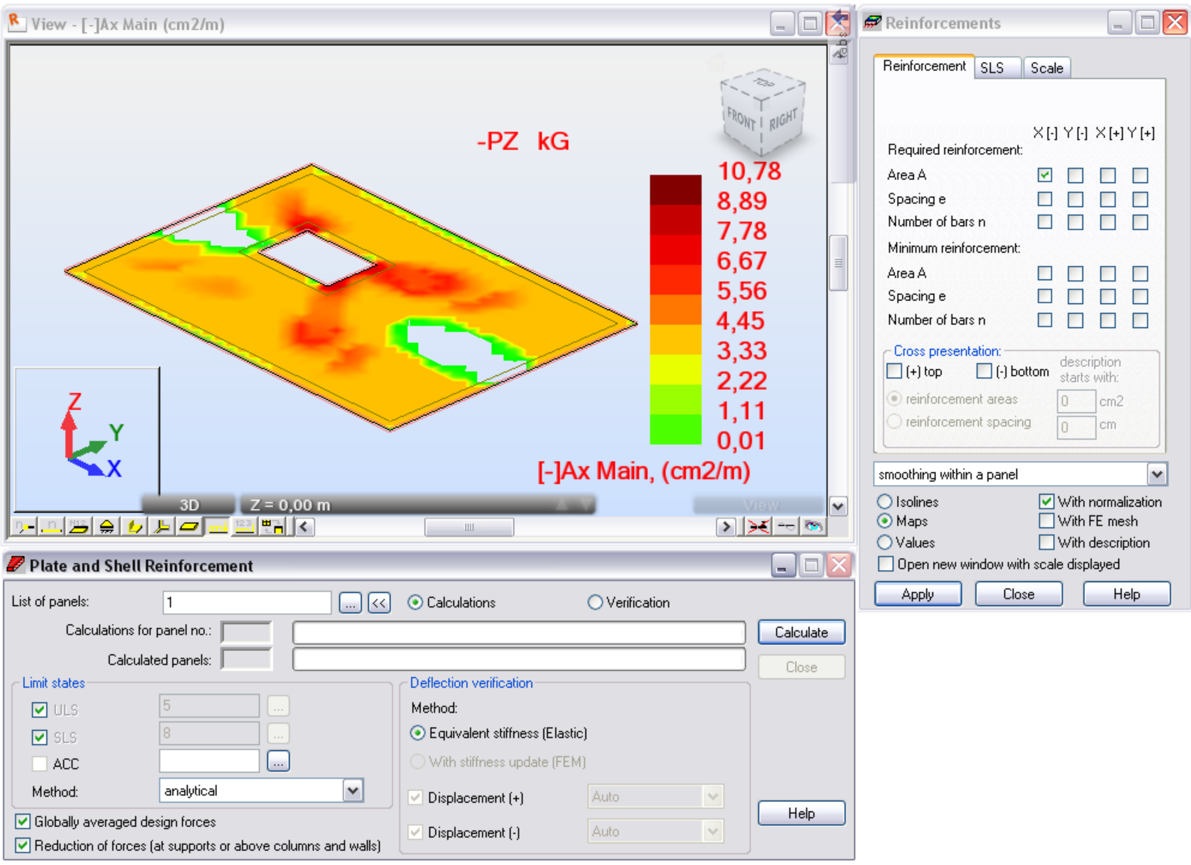



| | |
|---|---|
| Move to the *Cuts* tab and turn off display of the diagram in the defined cut (√ symbol will disappear) | Turns off display of the diagram on the cut through the slab. |
| **Apply**, **Close** | Turns off display of displacements in the panel cut and closes the ***Panel Cuts*** dialog box. |

## 8.3 Calculations of the Required (Theoretical) Reinforcement Area

Code: EN 1992-1-1:2004 AC:2008

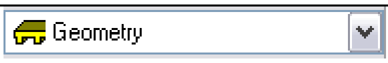
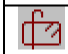
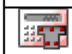
| | |
|---|---|
|  LMC the field allowing one to select ***RSAP*** program layouts and select: *RC Slabs / Slabs - required reinforcement* | The user goes to the layout of the ***RSAP*** program allowing one to determine the theoretical (required) reinforcement area for the defined slab. The screen will be divided into three parts: the graphical viewer with the structure model and two dialog boxes*:* ***Plate and Shell Reinforcement*** and ***Reinforcements***. |
| LMC on the *ULS* field in the *List of cases* panel and introduce 5 in the ***Plate and Shell Reinforcement*** dialog box | Calculation of the theoretical (required) reinforcement area will be carried out for the Ultimate Limit State with all the load cases applied to the slab considered. |
| LMC on the *SLS* field in the *List of cases* panel and introduce 8 in the ***Plate and Shell Reinforcement*** dialog box | Calculation of the theoretical (required) reinforcement area will be carried out for the Serviceability Limit State taking account of the defined combination. |
| LMC the *Method* field and select the *analytical* method | Selection of the analytical method of calculating the reinforcement area |
| Turn on the option: *Reduction of forces (at supports or above columns)* | If this option is turned on, it means that for slab elements supported at point (e.g. by means of the column support), values of moments and stresses near the supported points are substituted for the average value from the vicinity of these supports/columns |
| LMC the **Calculate** button in the ***Plate and Shell Reinforcement*** dialog box | Calculations of the theoretical (required) reinforcement area for the defined slab (panel no. 1) are started |
| Once the calculations are finished, LMC the *Area A [-]* option in the ***Reinforcements*** dialog box | Selection of the quantities to be presented |
| Go to the *Scale* tab and select the *256 colors* option in the *Color palette* field | Selection of the color palette to be used during the presentation of reinforcement maps |
| LMC the **Apply** button in the ***Reinforcements*** dialog box | Presentation of the reinforcement area for the selected area and the selected direction (the map of the reinforcement area is shown in the figure below) |
| Make the *Area A Y[-]* option inactive (the ***Reinforcements*** dialog box) | The reinforcement maps presentation is made inactive |

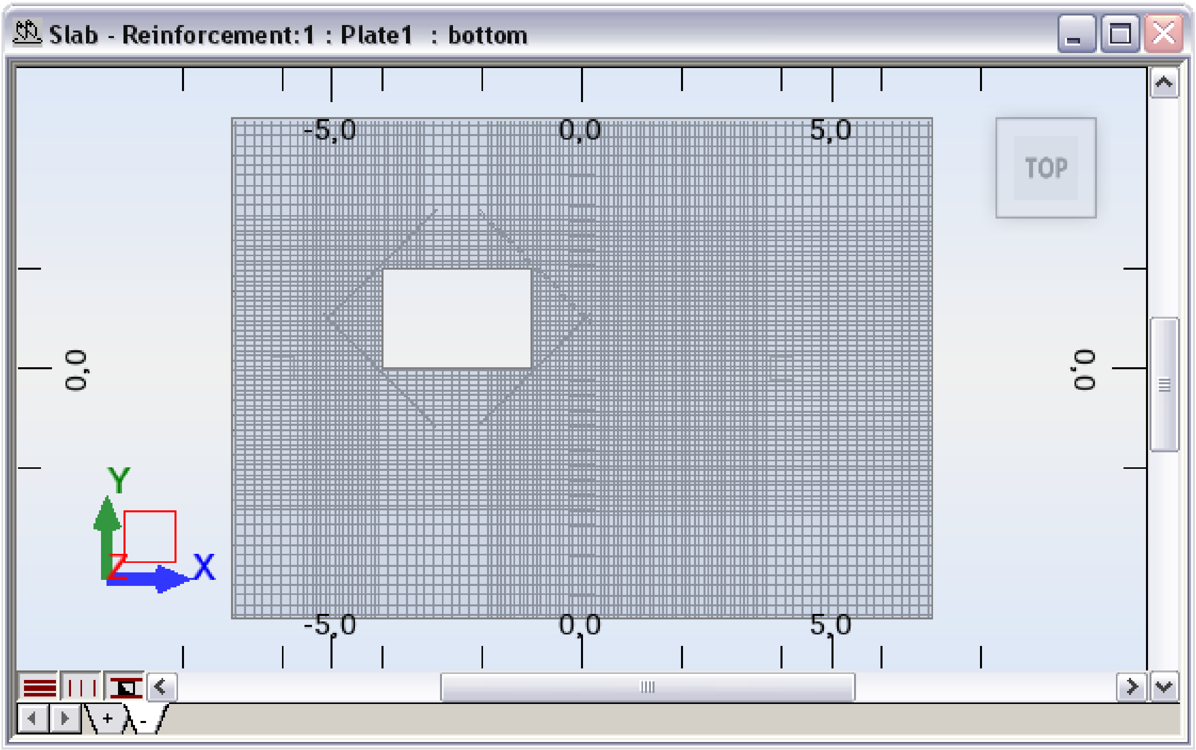



| | |
|---|---|
|  Select the *FE Resuts* icon from the Structure Model toolbar | Opens the table presenting the results obtained from the calculations of the theoretical (required) reinforcement areas for the slab |
| RMC while the cursor is positioned within the *Reinforcement Areas* table | Displays the context menu on the screen |
| *Table Columns* | Opens the ***Reinforcement Areas*** dialog box |
| Switch on two options in the *Required reinforcement* field:<br>*Spacing e X[-]*<br>*Spacing e X[+]* | Selects the quantities to be presented in the table |
| **OK** | Closes the ***Reinforcement Areas*** dialog box |
| Go to the *Global extremes* tab in the *Reinforcement Areas* table | Presentation of the global extremes on the surface and the reinforcement spacings obtained for the designed slab |
| Close the ***Reinforcement Areas*** table | |

## 8.4. Calculations of the Provided (Real) Reinforcement Area

Code: EN 1992-1-1:2004 AC:2008

| | |
|---|---|
|  LMC on the field for selection of layouts in the ***RSAP*** program: **Structure Model / Geometry** | Selects the initial layout of the ***RSAP*** program |
| Select – by window selection – the whole plate (the plate becomes highlighted) | Selects the plate for which provided (real) reinforcement will be calculated. *NOTE: if a model includes more panels, then these panels should be selected for which provided reinforcement is to be calculated.* |
| *Analysis menu / Design of RC Structure Elements / RC Panel Design / Provided Reinforcement* | Starts provided reinforcement calculations of the plate. Accept messages if any are displayed. |
|  | Activates display of the bottom reinforcement for the direction X. |
|  Select the *Reinforcement Parameters* icon from the Slab Parameters toolbar | Opens the ***Reinforcement Pattern*** dialog box |
| Select the *Bars* option | On the General tab – selects the *Bars* option in the *Reinforcement type* field; it means that the generated plate reinforcement will be the reinforcement with the use of reinforcing bars |
| Go on the *Bars* tab and set top and bottom reinforcement in both directions as *12* min | Modification of top and bottom reinforcement parameters. |
| **OK** | Accepts the selection made and closes the ***Reinforcement Pattern*** dialog box |
|  Select the *Calculations* icon from the Standard toolbar | Opens the ***Calculation Option Set*** dialog box |
|  Select the option that allows switching to the ***Reinforcement layout*** after calculations. | Once calculations are completed, the program will open automatically the ***RSAP*** layout: **RC Slabs / Slab – Reinforcement** |
| **Calculations** | Starts calculations of the plate provided reinforcement. |




Slab - Reinforcement Table : Plate1

| | No. | Reinforcement Type | Steel Grade | Diameter (mm) | Shape Code | Number | (m) | (m) |
|---|---|---|---|---|---|---|---|---|
| 1 | 1 | <different value> | B500A | 12 | 00 | 193 | A = 9,94 | |
| 2 | 2 | <different value> | B500A | 12 | 00 | 33 | A = 4,94 | |
| 3 | 3 | <different value> | B500A | 12 | 00 | 82 | A = 2,94 | |
| 4 | 4 | <different value> | B500A | 12 | 00 | 45 | A = 7,94 | |
| 5 | 5 | <different value> | B500A | 12 | 00 | 226 | A = 7,21 | |
| 6 | 6 | construction | B500A | 12 | 00 | 16 | A = 3,19 | |

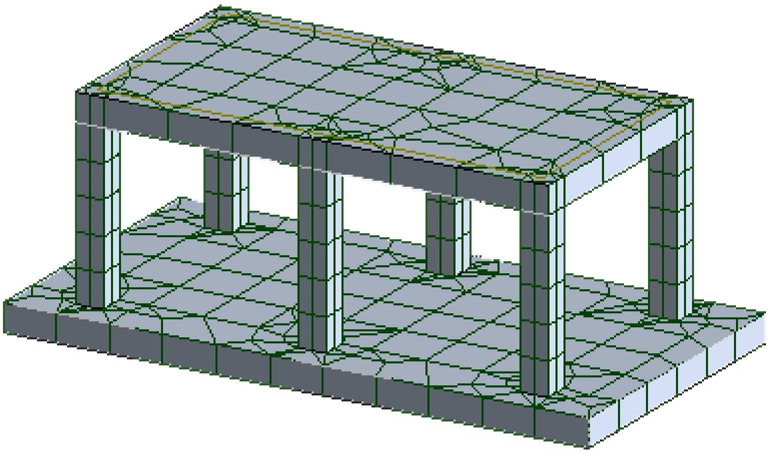
General / Detailed / Summary tal

| No. | Wire fabric name | Steel Grade | D (mm) | d (mm) | Number | S (cm2/m) | s (cm2/m) | E (n |
|---|---|---|---|---|---|---|---|---|
| * | | | | | | | | |

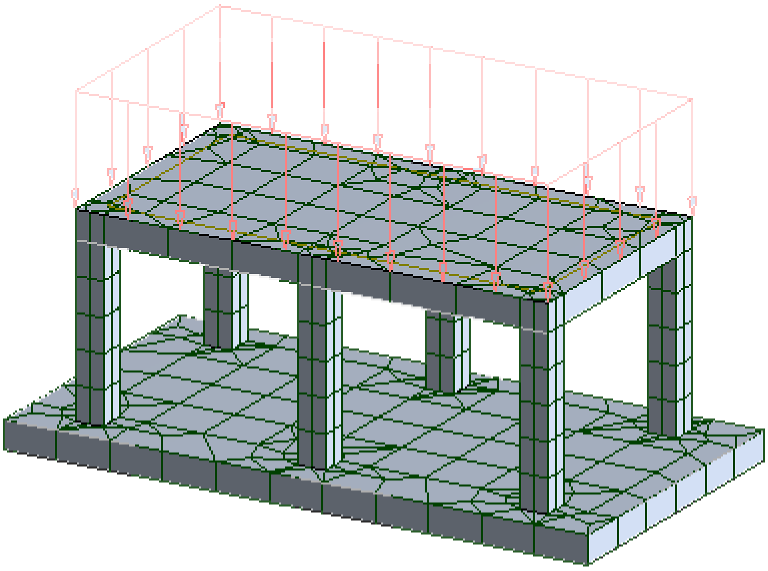
General / Detailed

# 9. 3D Solid Structure

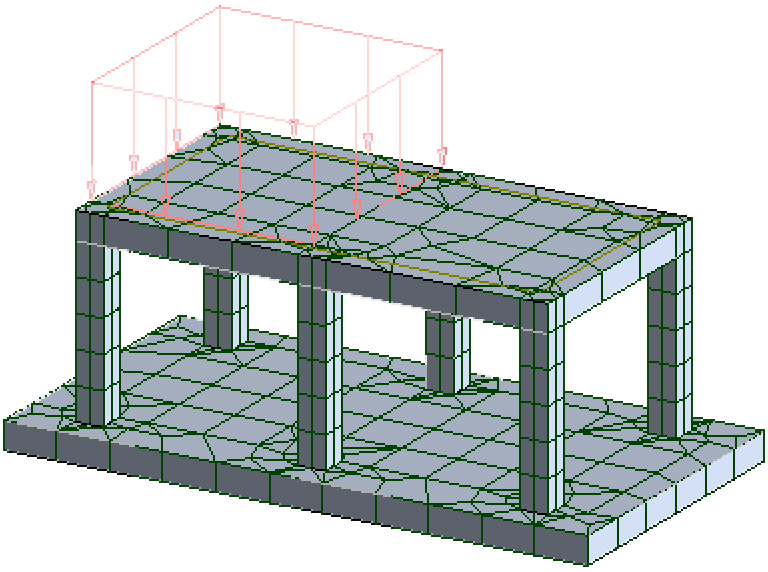
This example presents definition, analysis and design of a machine foundation shown in the figure below.
Data units: (m) and (kN).



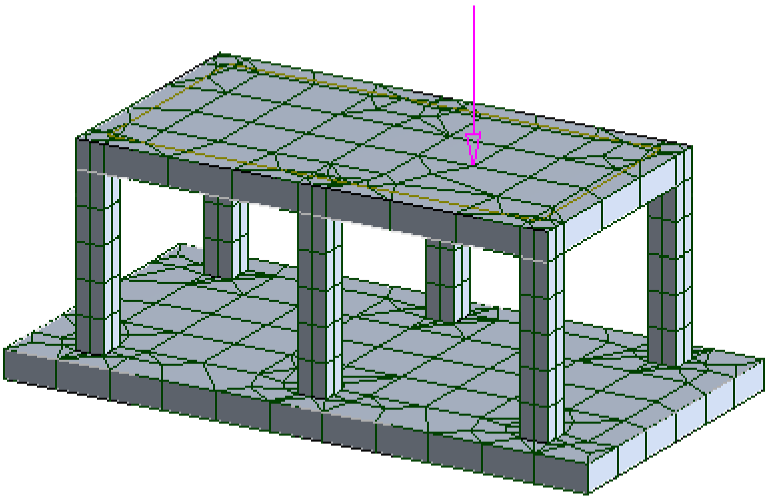
Four load cases have been assigned to the structure and three of them are displayed in the drawings below.



LOAD CASE 2 - LL1



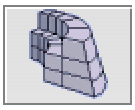
LOAD CASE 3 - LL2



LOAD CASE 4 - LL3

The following rules apply during structure definition:

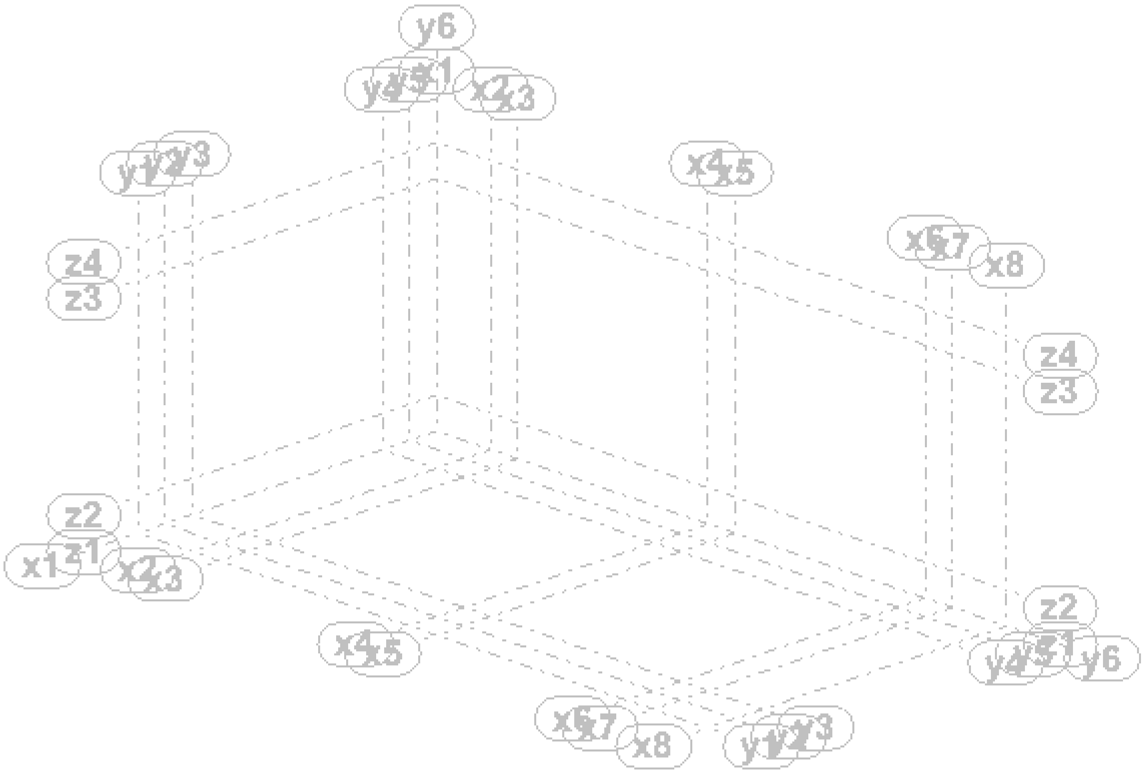
- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- **LMC** and **RMC** - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

To run structure definition start the ***RSAP*** program (press the appropriate icon or select the command from the taskbar). The vignette window will be displayed on the screen and the icon  (***Volumetric Structure Design***), the last but one in the first row, should be selected.

## 9.1 Model Definition

*Definition of Structural Axes*

| PERFORMED OPERATION | DESCRIPTION |
|---|---|
| *Geometry menu / Axis Definition* | Opens the ***Structural Axis*** dialog box which allows defining structural axes. |
| On the X tab chose the *Define* option located in the *Numbering* field and then enter the **x1** number in the edit field.<br>Enter the following values in the *Position* field:<br>(0) **Insert**,<br>(1) **Insert**,<br>(1.5) **Insert**,<br>(5.0) **Insert**,<br>(5.5) **Insert**,<br>(9.0) **Insert**,<br>(9.5) **Insert**,<br>(10.5) **Insert** | Defines the method of axis numbering. Creates the vertical axes designated with consecutive numbers x1, x2, x3, etc. |
| On the Y tab chose the *Define* option located in the *Numbering* field and then enter the **y1** number in the edit field.<br>Enter the following values in the *Position* field:<br>(0) **Insert**,<br>(0.5) **Insert**,<br>(1) **Insert**,<br>(4.5) **Insert**,<br>(5) **Insert**,<br>(5.5) **Insert** | Defines the method of axis numbering. Creates the vertical axes designated with consecutive numbers y1, y2, y3 etc. |
| On the Z tab chose the *Define* option located in the *Numbering* field and then enter the **z1** number in the edit field.<br>Enter the following values in the *Position* field:<br>(0) **Insert**,<br>(0.5) **Insert**,<br>(3.5) **Insert**,<br>(4) **Insert** | Defines the method of axis numbering. Creates the vertical axes designated with consecutive numbers z1, z2, z3 etc. |
| **Apply**, **Close** | Displays the recently defined structural axis on the screen, closes the ***Structural Axis*** dialog box. |
| *View menu / Projection / 3D xyz* | Displays a 3D view of the structure. |
|  Select the *Zoom All* icon from the Standards toolbar. | Presents the initial view of the structure axes (see the picture below). |

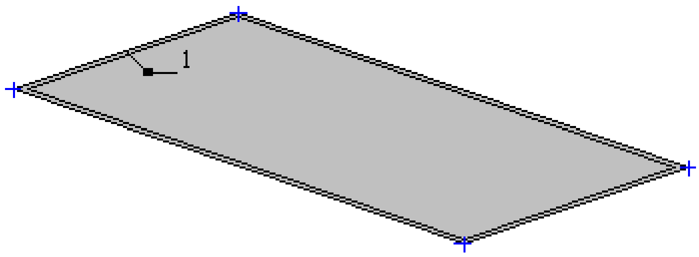



*Definition of the Structure*

*A Base of the Foundation*

| | |
|---|---|
| *View menu / Projection / Xy* | Once this option is selected, the structure is set on the XY plane. |
| *Geometry menu / Objects / Polyline - contour* | Opens the ***Polyline - Contour*** dialog box that allows defining various line types. |
| **LMC** in the **Geometry** button | Opens the dialog box that allows defining a contour. |
| Set the cursor in the green field, then switch to the graphic viewer and select graphically the consecutive points defining the contour (i.e. the intersection points of the appropriate structural axes):<br>**x1 - y1**, (0, 0)<br>**x8 - y1**, (10.5, 0)<br>**x8 - y6**, (10.5, 5.5)<br>**x1 - y6**  (0, 5.5)<br><br>**Apply**, **Close** | Defines a contour, closes the ***Polyline - Contour*** dialog box. |
|  Select the *Zoom All* icon from the Standards toolbar. | Presents the initial view of the structure. |
| *Geometry menu / Panels* | Opens the ***Panel*** dialog box that allows defining structure panels. |

| | |
|---|---|
| Activate the *Face* option in the *Contour type* field | Once this option is selected, the currently generated object will be defined as a face (without assigning properties), which enable using such an object during generation of a volumetric structure. |
| **LMC** in the *Internal point* field and select the point inside the contour by left-clicking on it | Applies current properties to the selected panel. |
| **Close** | Closes the ***Panel*** dialog box. |
| *View menu / Projection / 3D xyz* | Once this option is selected, a 3D view of the structure is displayed. The defined structure (without presentation of the structural axes) is shown in the drawing below. |

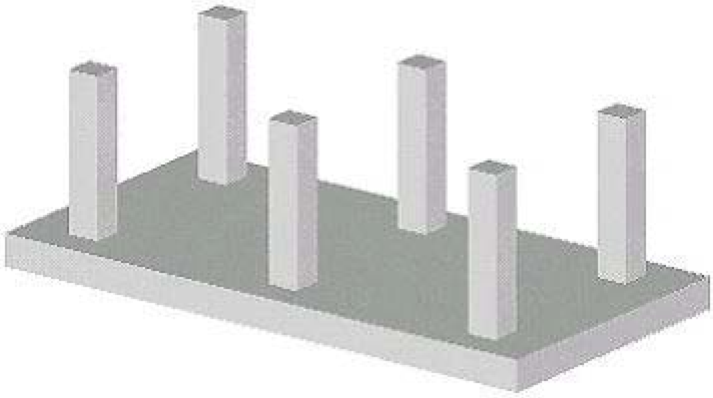



| | |
|---|---|
|  In the selection field enter number **1**, **Enter** | Selects the recently defined panel, whose color changes to red. |
| *Geometry menu / Objects / Extrude* | Opens the ***Extrude*** dialog box which is used to create simple solid-like elements by extruding predefined two-dimensional objects. |
| Activate the *II to Axis* option and select the **Z** axis | Once this option is selected, the object will be extruded along the axis that is parallel to the Z axis of the global coordinate system. |
| In the edit field set the length of the extrusion vector as **0.5** | Defines the length of the extrusion vector. |
| Enter **1** in the *Division Number* field | Defines the number of divisions to be performed while extruding the selected object. |
| **Apply**, **Close** | Extrudes the selected two-dimensional object along the appropriate axis. |

*Columns*

| | |
|---|---|
| *View menu / Work in 3D / Global Work Plane* | Opens the ***Work Plane*** dialog box that allows setting the work plane for structure definition/modification. |
| Switch to the graphic viewer and select graphically the intersection point of the following axes:<br>x1 - y1 - z2 and then press the **Apply** button | Sets a new global work plane for structure definition.<br>The coordinates in the ***Work Plane*** dialog box will change automatically to the selected ones e.g. (0.0, 0.0, 0.5). |

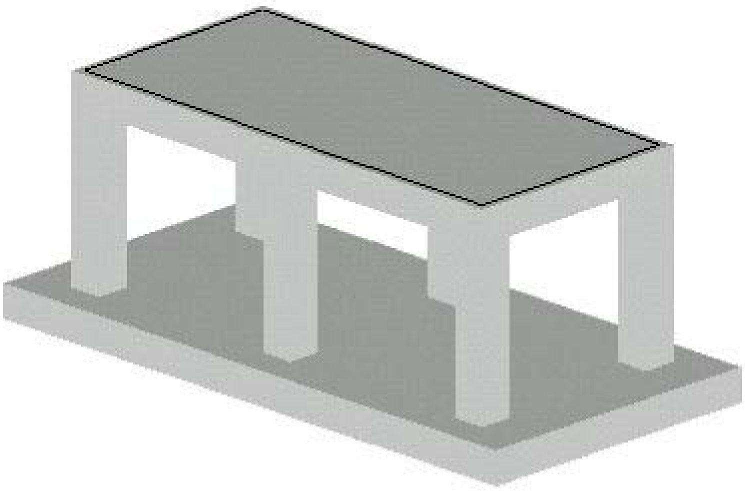
| | |
|---|---|
| Close the ***Work Plane*** dialog box by means of the button | Closes the ***Work Plane*** dialog box. |
| *View menu / Projection / Xy* | Once this option is selected, the structure is set on the XY plane for the Z coordinate value recently defined (i.e. Z = 2.0). Only the structure components located in this plane will remain visible. |
| *Geometry menu / Objects / Cube* | Opens the ***Cube*** dialog box that allows defining cubes. |
| Select the *Three points* option in the *Definition method* field | Selects a rectangle as a base of the cube. The rectangle will be defined by means of the two opposite apexes of the rectangle. |
| Switch to the graphic editor and select two opposite apexes of the rectangle defined by means of the intersection points of the following axes:<br>**x2** - **y2**, (1, 0.5)<br>**x3** - **y2**, (1.5, 0.5)<br>**x3** - **y3**, (1.5, 1)<br><br>then in the *Height* field located in the ***Geometry*** dialog box enter the value **3** and press the **Apply** and **Close** buttons | Defines the cube, closes the ***Cube*** dialog box. |
| Switch to the graphic viewer and enter the number **2** in the selection field next to the , **Enter** | Selects the recently defined cube. |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| In the graphic viewer select the top-right apex of the rectangle, which defines base of the cube. In the *Translation Vector* field located in the ***Translation*** dialog box, enter the following numbers:<br>(**0**, **4**, **0**), **Execute** | Translates the selected cube. |
| **RMC** in the graphic viewer and choose the *Select* option | Opens the context menu |
| Select the recently defined cubes (nos. 2 and 3 appear in the edit field). Switch to the ***Translation*** dialog box and in the *Number of Repetitions* field enter **2**, then define the translation vector:<br>(**4**, **0**, **0**), **Execute**, **Close** | Translate the selected cubes. |
| *View menu / Projection / 3D xyz* | Once this option is selected, a 3D view of the structure is displayed. The defined structure (without presentation of the construction axis) is shown in the drawing below. |



*Top of the Foundation*

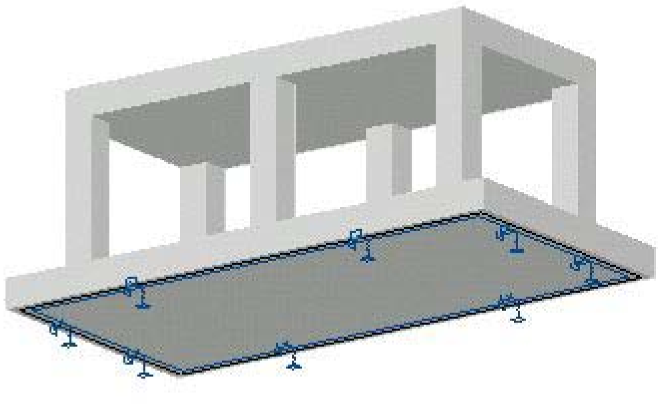
| | |
|---|---|
| *View menu / Work in 3D / Global Work Plane* | Opens the ***Work Plane*** dialog box that allows setting the work plane for structure definition/modyfication. |
| Switch to the graphic viewer and select graphically the intersection point of the following axes:<br>x2 - y2 - z4 and then press the **Apply** button | Sets a new global work plane for structure definition.<br>The coordinates in the ***Work Plane*** dialog box will change automatically to the selected ones e.g. (1.00, 0.50, 4.00). |
| Close the ***Work Plane*** dialog box by means of the button | Closes the ***Work Plane*** dialog box. |
| *View menu / Projection / Xy* | Once this option is selected, the structure is set on the XY plane for the Z coordinate value recently defined<br>(i.e. z = 14.0). Only the structure components located in this plane will remain visible. |
| *Geometry menu / Objects / Polyline - contour* | Opens the ***Polyline - Contour*** dialog box which allows defining various line types. |
| Set the cursor in the green field, then switch to the graphic viewer and select graphically the consecutive points defining the contour (i.e. the intersection points of the appropriate structural axes):<br>**x2 - y2**, (1, 0.5)<br>**x7 - y2**, (9.5, 0.5)<br>**x7 - y5**, (9.5, 5)<br>**x2 - y5**, (1, 5)<br><br>**Apply**, **Close** | Defines contour, closes the ***Polyline - Contour*** dialog box. |
| *Geometry menu / Panels* | Opens the ***Panel*** dialog box that enables definition of panels in a structure |
| Switch on the *Face* option located in the *Contour type* field | If this option is selected, the created object will be defined as a wall (without assigning such properties as a reinforcement type or thickness) which makes it possible to use this object while generating a volumetric structure |

| | |
|---|---|
| LMC on the *Internal point* option located in the *Creation with* field, select any point within the contour | Assigns the selected properties to the chosen panel |
| **Close** | Closes the ***Panel*** dialog box |
| In the selection field, next to the icon, enter **8**, **Enter** | Selects the recently defined contour whose color changes to red. |
| *Geometry menu / Objects / Extrude* | Opens the ***Extrude*** dialog box which is used to create simple solid-like elements by extruding predefined two-dimensional objects. |
| Activate the *II to Axis* option and select the **Z** axis | Once this option is selected, the object will be extruded along the axis that is parallel to the Z axis of the global coordinate system. |
| In the edit field set the length of the extrusion vector as **– 0.5** | Defines the length of the extrusion vector. |
| Enter **1** in the *Division Number* field | Defines the number of divisions to be performed during the extrusion process. |
| **Apply**, **Close** | Extrudes the selected two-dimensional object along the appropriate axis. |
| Select the *Hidden* icon in the lower left corner of screen. | If this option is selected, invisible lines in the structure will not be displayed |
| *View menu / Projection / 3D xyz* | Once this option is selected, a 3D view of the structure is displayed. |
| Select the *Shading* icon in the lower left corner of screen. | If this option is selected, invisible lines in the structure will not be displayed |
| Select the *Zoom All* icon from the Standard toolbar.. | Presents the initial view of the structure (see the picture below). |



*Support Definition*

| | |
|---|---|
| LMC on the field to select the **Structure Model/Supports** Layout | Selects the ***RSAP*** layout that allows defining supports. |
| In the ***Supports*** dialog box press the icon | Opens the ***Support Definition*** dialog box that allows defining a new support. |
| On the *Elastic* tab switch off the *UZ* option and in the *KZ* field that becames available enter 70000 (kN/m) | Defines the support elasticity coefficient for the displacement in the Z direction. |
| In the *Label* field enter the name for a new support: ***Elastic Foundation*** **Add**, **Close** | Assigns the name to the defined support |
| In the ***Supports*** dialog box, **LMC** on the *Current Selection* field on the *Planar* tab | Selects a structure surface for which supports will be defined. |
| Switch to the graphic viewer; pressing the left mouse button select the surface being the base of the foundation - in the *Current Selection* field 1_REF(1) will appear | Selects the surface of the foundation base. |
| From the ***Supports*** dialog box select the recently defined *Elastic foundation* support (the icon will be highlighted) | Selects the support type. |
| **LMC** on the **Apply** button | The selected support type will be assigned to the chosen structure surface. |
| LMC on the field to select the **Structure Model/Geometry** Layout | Selects the initial ***RSAP*** layout. |
| *View menu / Display* | Opens the ***Display*** dialog box that allows selecting structure attributes for presentation. |
| On the *Structure* tab, in the ***Display*** dialog box activate *Supports - symbols*, **Apply**, **OK** | Displays symbols of structure supports on the screen, closes the ***Display*** dialog box.<br>The defined structure is shown in the drawing below. |
| Select the *Rotate, Zoom, Pan* icon from the Standard toolbar. | Using the dynamic zoom option enables structure rotation and pan, so that the bottom structure part with supports can be presented. |



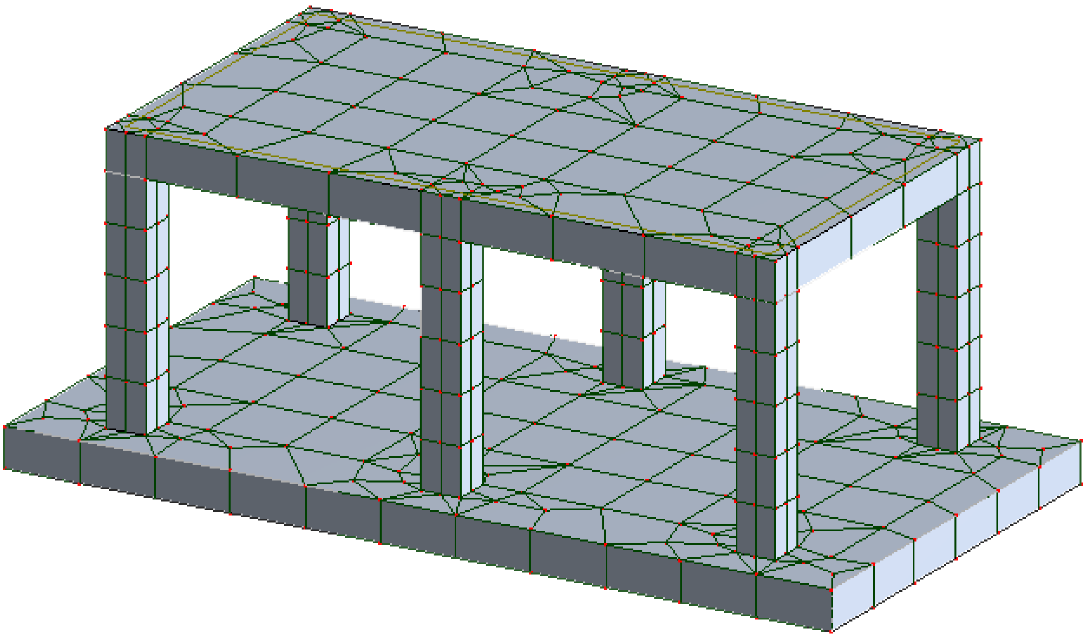
*Mesh Generation*

In order to improve mesh generation define additional nodes.

| | |
|---|---|
| *View menu / Grid / Grid Step Definition* | Opens the ***Grid Step Definition*** dialog box, which is used to change the grid step presented on the screen. |
| In the *Grid Step* field set the grid step Dx and Dy as 0.25, **Apply**, **Close** | Changes the grid step, closes the ***Grid Step Definition*** dialog box |
| *View menu / Work in 3D / Global Work Plane* | Opens the ***Work Plane*** dialog box that allows setting the work plane for structure definition/modification. |
| Switch to the graphic viewer and select graphically the intersection point of the following axes:<br>x1 - y1 - z1 and then press the **Apply** button | Sets a new global work plane for structure definition.<br>The coordinates in the ***Work Plane*** dialog box will change automatically to the selected ones e.g. (0.0, 0.0, 0.0). |
| Close the ***Work Plane*** dialog box by means of the button | Closes the ***Work Plane*** dialog box. |
| *View menu / Projection / Xy* | Once this option is selected, the structure is set on the XY plane for the Z coordinate value recently defined (i.e. z = 0.0). Only the structure components located on this plane will remain visible. |
| *Geometry menu / Nodes* | Opens the ***Nodes*** dialog box that allows defining the structure nodes. |
| Define the additional nodes whose coordinates are the intersection points of the following structure axes:<br>x2 - y2, x3 - y2, x3 - y3, x2 - y3,<br>and the nodes of the following coordinates:<br>(1.25, 0.50, 0.00),<br>(1.00, 0.75, 0.00),<br>(1.25, 1.00, 0.00),<br>(1.50, 0.75, 0.00), | Defines nodes, closes the ***Node*** dialog box. |
| In the edit field located next to the icon enter **all, Enter** | Selects all the nodes defined in the structure. |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |

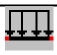
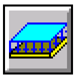
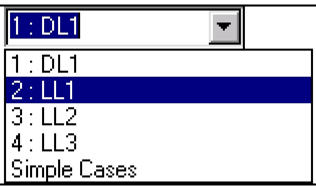
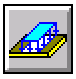
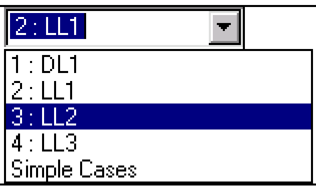
| | |
|---|---|
| In the *Translation vector* field enter: **(4**, **0**, **0)**<br>In the *Number of repetitions* field enter: **2**<br>**Execute**, **Close** | |
| In the edit field located next to the icon enter **all, Enter** | Selects all the nodes defined in the structure. |
| *Edit menu / Edit / Horizontal mirror* | Opens the ***Horizontal Mirror*** dialog box |
| **LMC** on the *Plane Location* field: **2.75** | Defines the position of the horizontal symmetry axis. |
| **Execute**, **Close** | Performs horizontal symmetry of the selected nodes, closes the ***Horizontal Mirror*** dialog box. |
| *View menu / Projection / 3D xyz* | Once this option is selected, a 3D view of the structure is displayed. |
| *View menu / Work in 3D / Global Work Plane* | Opens the ***Work Plane*** dialog box that allows setting the work plane for structure definition/modification. |
| In the graphic viewer select graphically the intersection point of the following axes:<br>x2 - y2 - z4 and then press the **Apply** button | Sets a new global work plane for structure definition.<br>The coordinates in the ***Work Plane*** dialog box will change automatically to the selected ones e.g. (1.0, 1.0, 4.0). |
| Close the ***Work Plane*** dialog box by means of the button | Closes the ***Work Plane*** dialog box. |
| *View menu / Projection / Xy* | Once this option is selected, the structure is set on the XY plane for the Z coordinate value recently defined<br>(i.e. z = 4.0). Only the structure components located on this plane will remain visible. |
| *Geometry menu / Nodes* | Opens the ***Nodes*** dialog box which allows defining the structure nodes. |
| Define additional nodes whose coordinates are the intersection points of the following structure axes:<br>x2 - y3, x3 - y2, x3 - y3, x2 - y2,<br>and the nodes of the following coordinates:<br>(1.25, 1.00, 4.00),<br>(1.50, 0.75, 4.00),<br>(1.25, 0.50, 4.00),<br>(1.00, 0.75, 4.00), | Defines nodes, closes the ***Nodes*** dialog box. |
| In the edit field located next to the icon enter: **49to56,**<br>**Enter** | Selects nodes defined in the current work plane. |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |

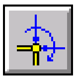
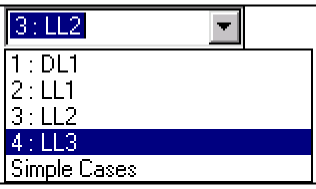
| | |
|---|---|
| In the *Translation vector* field enter: **(4**, **0**, **0)**<br>In the Number of repetitions field: **2**<br>**Execute**, **Close** | |
| In the edit field located next to the icon enter numbers of the recently defined nodes: **49to72**, **Enter** | Selects nodes defined in the current work plane. |
| *Edit menu / Edit / Horizontal mirror* | Opens the ***Horizontal Mirror*** dialog box. |
| In the *Plane Location* edit field enter **2.75** | Defines the coordinate of the horizontal mirror axis. |
| **Execute**, **Close** | Mirrors horizontally the selected nodes, closes the ***Horizontal Mirror*** dialog box. |
| *Geometry menu / Nodes* | Opens the ***Nodes*** dialog box which allows defining structure nodes. |
| In the *Coordinates* field enter the coordinates of the additional node: (**7.25**, **2.75**, **4.0**),<br>**Add**, **Close** | Defines the additional node no. 97 to which a nodal force will be applied, closes the ***Nodes*** dialog box. |
| In the edit field next to the icon enter: **1** and **8 (1 8)**, **Enter** | Selects the base and top of the foundation. |
| *Analysis menu / Calculation Model / Meshing Options* | Opens the ***Meshing Options*** dialog box. |
| In the *Available Meshing Methods* field select the *Delaunay* option, in the *Mesh Generation* field select the *Elemet size* and enter 1 (m) in the field, **OK** | Sets the meshing parameters for the selected structure components. |
| In the edit field next to the icon enter: **2to7**, **Enter** | Selects all columns of the foundation. |
| *Analysis menu / Calculation Model / Meshing Options* | Opens the ***Meshing Options*** dialog box. |
| In the *Available Meshing Methods* field select the *Delaunay* option, in the *Mesh Generation* field select the *Automatic* option and enter 2 in the *Division 1* field, **OK** | Sets the meshing parameters for selected structure components. |
| *Analysis menu / Calculation Model / Generation* | If this option is selected, the program starts to generate the calculation model of the structure (finite elements), see the picture below. |
| *View menu / Projection / 3D xyz* | Once this option is selected, a 3D view of the structure is displayed. The defined structure is shown in the drawing below. |

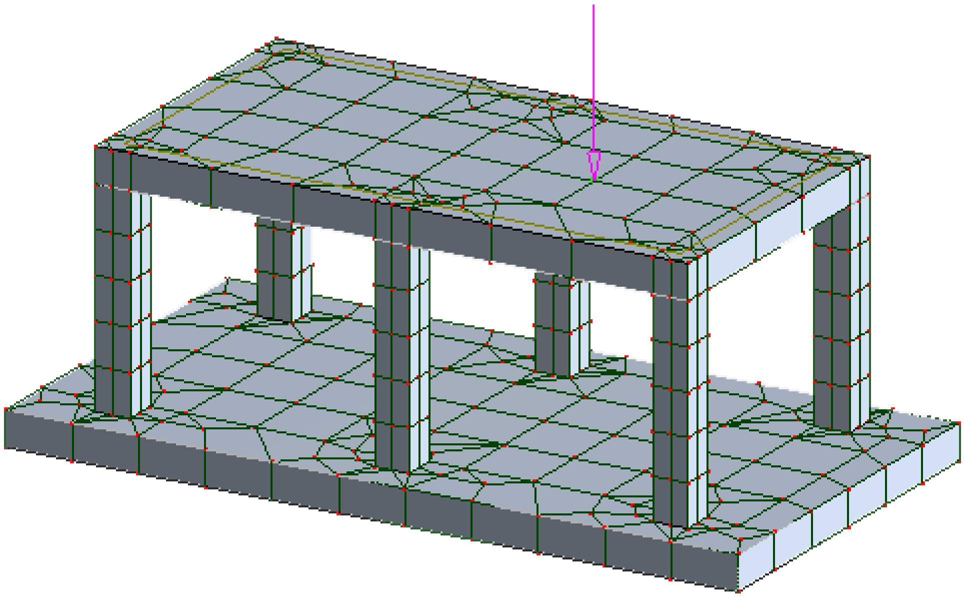


| | |
|---|---|
| *Geometry menu / Properties / Solid Properties* | Opens the ***Solid Properties*** dialog box. |
| LMC on the *Selection* field | Enter all (all structure elements) |
| LMC on the material *Concrete* | Selects material. If the material is not available on the available material list, the user should press the icon *Definition of new solid properties* and add concrete to the list of materials |
| **Apply**, **Close** | Assigns the material to all the structure elements and closes the dialog box |

*Load Definition*

| | |
|---|---|
| *View menu / Projection / Xy* | Once this option is selected, the structure is set on the XY plane for the Z coordinate recently defined (i.e. z = 4.0). Only the structure components located on this plane will remain visible. |
| <br>LMC on the field to select the **Structure Model / Loads** Layout | Selects the ***RSAP*** program layout that allows defining structure loads. |
| **LMC** in the *Nature* field, *(live1)* | Selects the type of a load case: *live*. |
| **LMC** on the **New** button<br>**LMC** on the **New** button<br>**LMC** on the **New** button | Defines two cases of *live* load with the standard names: LL1, LL2 and LL3. |
| **LMC** on the  *Load Definition* icon located in the right toolbar | Opens the ***Load Definition*** dialog box. |
| In the ***Load Definition*** dialog box select the *Surface* tab and press the  icon | Opens the ***Uniform Planar Load*** dialog box |
|  | Selects the load case: Live Load 1 (*2:LL1*). |
| In the *Values Z:* field enter **-20** | Defines the value of the uniform load acting on surface FEs in the direction of the Z axis of the global coordinate system. |
| **Add** | Closes the ***Uniform Planar Load*** dialog box. |
| Set the cursor in the *Apply To* field, switch to the graphic viewer and select the contour 8 defining the top surface of the foundation - 8_REF(1) will appear in the edit field | Displays the currently selected structure panel. |
| **Apply** | Applies the predefined load to the chosen panel contour. |
| In the ***Load Definition*** dialog box select *Surface* tab and press the  icon | Opens the ***Uniform Planar Load (contour)*** dialog box. |
|  | Selects the load case: Live Load 2 (*3:LL2)*. |

| | |
|---|---|
| In the *Values Z:* field enter -40 | Defines the value of the uniform load acting on surface FEs in the direction of the Z axis of the global coordinate system. |
| **LMC** on the **Contour *definition*** button | Opens the dialog box that allows defining the contour to which the load will be applied. It may be performed either in the dialog box or graphically on the screen. |
| In the green field enter the points defining the contour by clicking on the appropriate points of structure axes intersections:<br>**x2 - y3**, (1, 1)<br>**x4 - y3**, (5, 1)<br>**x4 - y5**, (5, 5)<br>**x2 - y5**, (1, 5) | Defines the contour to which the loads will be applied. |
| **LMC** on the **Add** button located in the lower part of the dialog box ***Uniform Planar Load (contour)*** | Closes the ***Uniform Planar Load (contour)*** dialog box. |
| Set the cursor in the *Apply To* field, switch to the graphic viewer and select the contour 8 defining the top surface of the foundation - 8_REF(1) will appear in the edit field | Displays the currently selected structure panel. |
| **Apply** | Applies the defined load to the chosen contour on the panel. |
| In the ***Load Definition*** dialog box select the *Node* tab and press the  (**Nodal force**) icon | Opens the ***Nodal Force*** dialog box. |
|  | Selects the load case: *Live Load 3*. |
| In the *Values Z:* field enter -100, **Add** | Defines the concentrated force loads acting on a selected structure node. |
| Select a node located nearest (10;0;4) | Displays the currently selected structure panel (see the picture below).<br>. |

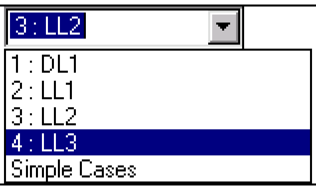


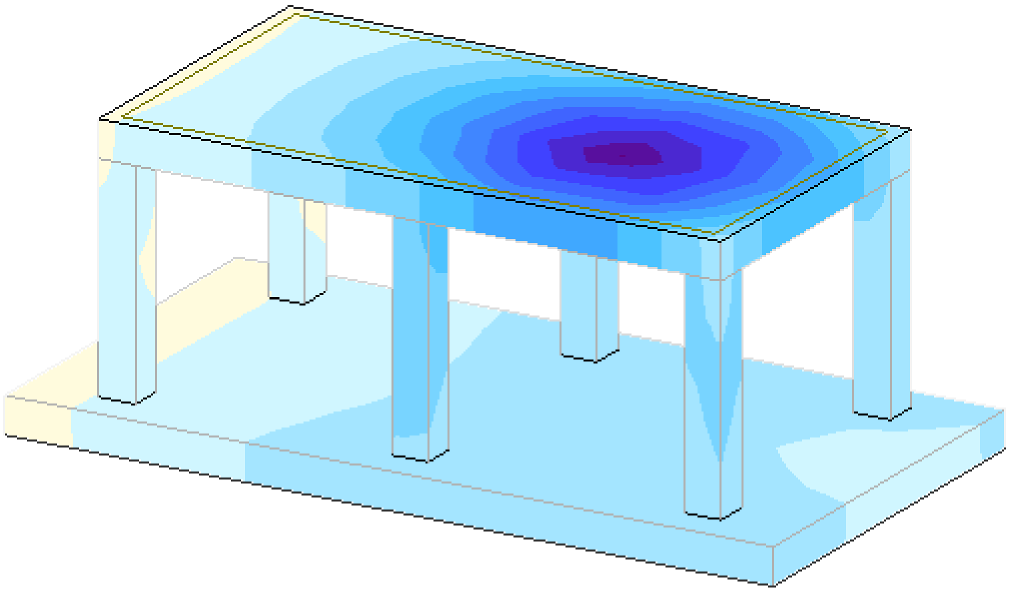
| | |
|---|---|
| **Apply**, **Close** | Applies the defined load to the chosen node, closes the ***Nodal Force*** dialog box. |

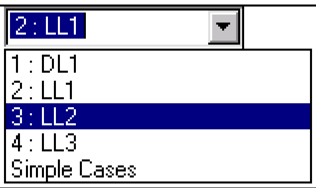
## 9.2 Structural Analysis

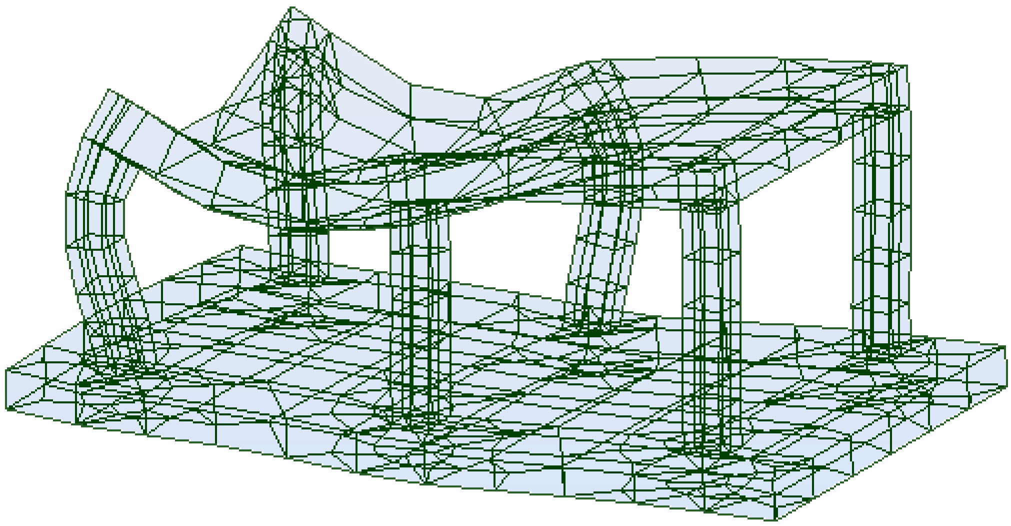
| | |
|---|---|
|  Select the *Calculations* icon from the Standard toolbar. | Starts the calculation of the defined structure. Once the calculations are completed, the viewer title bar will show the following information: *Finite Elements Results - available*. |

## 9.3 Presentation of Results in the Form of Maps

| | |
|---|---|
|  LMC on the field to select the **Structure Model / Results - maps** Layout | The **RESULTS** layout of the ***RSAP*** program will open. The screen will be divided into two parts: a graphical viewer containing the structure model and the ***Maps*** dialog box. |
|  | Select the load case: 4 (LL3). |
| On the *Detailed* tab, in the *Values in the local system* field, activate the third option in the **Displacement** line | Selects the visualization of the displacement for individual FEs in the local coordinate system. |
| Activate *Maps* option | Allows presentation of results obtained for FEs in the form of maps. |
| **LMC** the **Apply** button | Presents the structure displacement (see the picture below). |



| | |
|---|---|
| Switch off the *Displacement* **-** z option. On the *Deformation* tab select the *active* option located in the *Deformations* field, **Apply** | If this option is selected, the program will present deformation of the currently designed structure - see the picture below. |
|   | Select the load case: 3 (LL2). |
| In the *Deformations* field switch on the *Active* option | Activates presentation of deformation for the currently designed structure. |
| **Apply** | Presents the structure displacement. |



# 10. Shell Structures

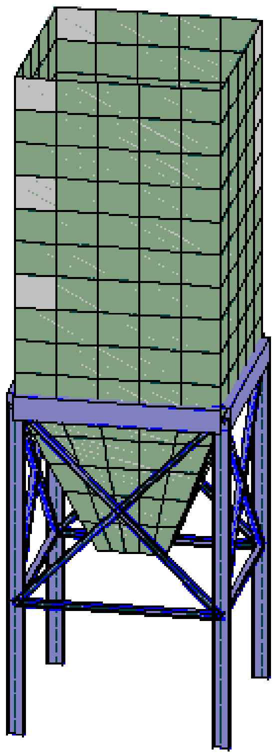
This chapter contains a presentation of several short examples of modeling three-dimensional structures by means of extrude and revolve options. All the presented structures are defined as **shells**. The following rules will be applied during the presentation of these structures:

- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick,
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

In order to start defining a structure, one should run the ***RSAP*** program (press the relevant icon or select the relevant command from the toolbar). After a while, there appears on screen the dialog box, where one should select the second icon in the second row  (**Shell design**).

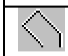
## 10.1 Silo

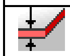
This example provides a definition of a silo, presented schematically on the drawing below.
Data units: (m).



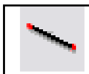
*STRUCTURE DEFINITION*

*Silo Geometry*

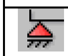
| OPERATION PERFORMED | DESCRIPTION |
|---|---|
| *View menu / Projection / XY* | Selection of work plane |
| *View menu / Grid / Grid Step Definition* | Opens the ***Grid Step Definition*** dialog box. |
| Dx = 1.0<br>Dy =1.0 | Defines grid step on a screen (equal in both directions) |
| **Apply**, **Close** | Accepts the defined parameters and closes the ***Grid Step Definition*** dialog box. |
|  Select the *Polyline Contour* icon from the Structure Model toolbar | Opening the ***Polyline - Contour*** dialog box to define successive components of a contour |
| Select *Contour* option in the *Definition method* part of the dialog box | |
| Define the following square on the graphical viewer: side length: 2m, points:<br>(-1,-1,0),<br>(-1,1,0),<br>(1,1,0),<br>(1,-1,0),<br>(-1,-1,0) | Definition of the square that will serve as the basis for modeling the silo |
| Close the ***Polyline - contour*** dialog box | |
| *View menu / Projection / 3d xyz* | |
| Select the *Edit menu / Substructure modification / Object modification* command from the menu | Opening the ***Objects: operations/modifications*** dialog box |
| LMC in the *Object* field and indicate with the cursor the square defined on the graphical viewer | Selection of the square (the number of the object defined in the *Object* field) |
| Press the **Extrude** button | Beginning of the definition of object modification |
| Press the **Object modification parameters** button | Definition of the parameters of extrusion |
| Define the following extrusion parameters:<br>II to axis Z,<br>Length: (5) m<br>Division number = (5)<br>Inactive options: top, base | Extrusion parameters |
| Press the **Apply** button | Extrusion performed for the square according to the defined parameters |

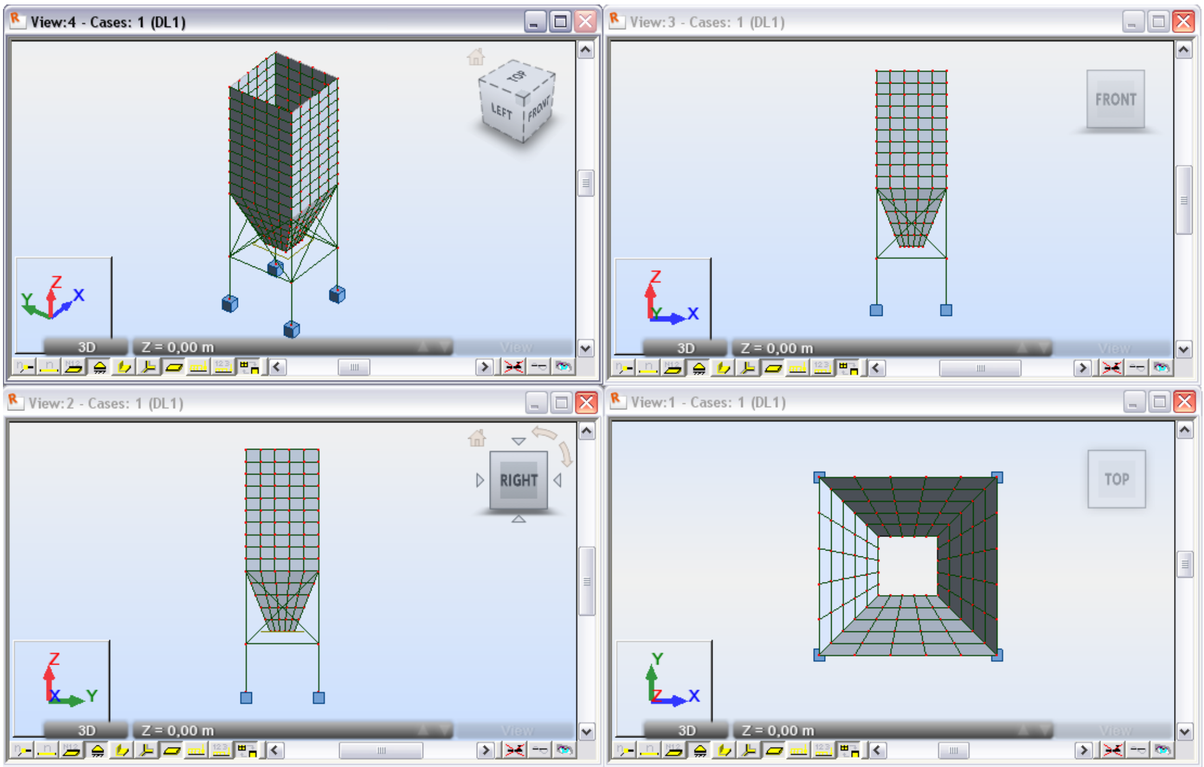
| | |
|---|---|
| Press the **Scaling** button | Beginning of the definition of the operation of modifying the result of extruding the object |
| Press the **Operation parameters** button | Definition of the parameters of modification to be applied to the extruded square |
| Define scaling parameters:<br>Scale x=y= (3)<br>Scale z= (1)<br>Scale center (0,0,0) | Parameters of scaling to be performed on the operation of extrusion |
| Press the **Apply** button | The operation of scaling is applied to the product of extrusion of the square according to the defined parameters |
| Press the **Extrude** button | Beginning of object modification definition |
| Press the **Object modification parameters** button (if necessary to expand) | Definition of the parameters of extrusion |
| Define the following extrusion parameters:<br>II to axis Z,<br>Length: (10) m<br>Number of divisions = (10)<br>Inactive options: top, base | Extrusion parameters |
| Press the **Apply** button | Extrusion performed for the square according to the defined parameters |
| Close the ***Objects: operations / modifications*** dialog box | |
|  Select the *Zoom All* icon from the Standard toolbar | Initial view |
|  Select the *Thickness* icon from the Structure Model toolbar | Opening the dialog box used for defining thickness |
| Select the default panel thickness: TH_30CONCR | Selection of the thickness that will be applied to particular components of the silo |
| Write **all** in the *Panels* field | Selection of all silo elements |
| Press the **Apply** button | Application of the default thickness to all the elements of the silo |
| Close the ***FE Thickness*** dialog box | |

*Support Structure*

| | |
|---|---|
|  Select the *Bars* icon from the Structure Model toolbar | Opening the ***Bars*** dialog box |

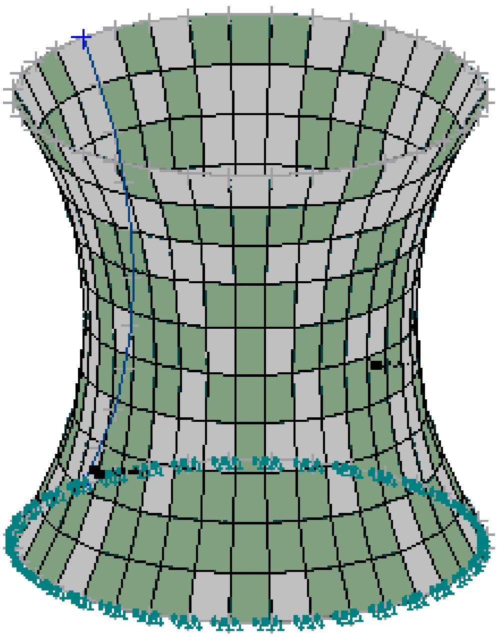
| | |
|---|---|
| LMC in the *Bar type* field and select the type: ***RC Beam***<br>LMC in the *Section* field and select the type ***B50x70*** *(If the **B50x70** section is not available on the list, one should press the (...) button located beside the Section field and define this section to the active section list in the **New section** dialog box.)* | Selection of bar properties<br>*Note:* |
| Define the following four bars:<br>beam 1: begin. (-3,-3,5), end (3,-3,5)<br>beam 2: begin. (3,-3,5), end (3,3,5)<br>beam 3: begin. (3,3,5), end (-3,3,5)<br>beam 4: begin. (-3,3,5), end (-3,-3,5) | Definition of RC beams |
| LMC in the *Bar type* field and select the type: ***Column***<br>LMC in the *Section* field and select the type ***HEB 400*** (if the section is absent on the list of available sections, open the ***New section*** dialog box by pressing the ... button and select the required section) | Selection of bar properties. The section from the European section database (EURO) has been used. |
| Define the following four steel columns of the 10 m length:<br>col.1: begin. (-3,-3,5), end (-3,-3,-5)<br>col.2: begin. (3,-3,5), end (3,-3,-5)<br>col.3: begin. (3,3,5), end (3,3,-5)<br>col.4: begin. (-3,3,5), end (-3,3,-5) | Definition of steel columns |
| LMC in the *Bar type* field and select the type: ***Beam***<br>LMC in the *Section* field and select the type ***HEB 400****.* | Selection of bar properties. The section from the European section database (EURO) has been used. |
| Define the following four beams:<br>beam1:begin. (-3,-3,-1), end (3,-3,-1)<br>beam2:begin. (3,-3,-1), end (3,3,-1)<br>beam3:begin. (3,3,-1), end (-3,3,-1)<br>beam4:begin. (-3,3,-1), end (-3,-3,-1) | Definition of steel spandrel beams |
| LMC in the *Bar type* field and select the type: ***Simple Bar***<br>LMC in the *Section* field and select the type ***CAE 100x12*** (if the section is absent on the list of available sections, open the ***New section*** dialog box by pressing the ... button and select the required section) | Selection of bar properties. The section from the European section database (EURO) has been used. |
| Define the following bracings:<br>1: begin. (-3,-3,5), end (3,-3,-1)<br>2: begin. (3,-3,5), end (-3,-3,-1) | Bracing definition |
| 3: begin. (3,-3,5), end (3,3,-1)<br>4: begin. (3,3,5), end (3,-3,-1) | |

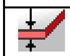
| | |
|---|---|
| Close the ***Bars*** dialog box | |
| Select bracing 1 and 2 | |
| *Edit menu / Edit / Translate* | Opening the dialog box used to define translation |
| Translation vector (0,6,0)<br>Edit mode = Copy<br>Drag = switched off, **Execute** | |
| Select bracing 3 and 4 | |
| Translation vector (-6,0,0)<br>Edit mode = Copy<br>Drag = switched off, **Execute** | |
| Close the ***Translation*** dialog box | |
|  Select the *Supports* icon from the Structure Model toolbar | Opening the ***Supports*** dialog box |
| LMC in the *Current selection* field on the *Nodal* tab | Selection of structure nodes where structure supports will be applied |
| Go to the graphical viewer; while pressing the left mouse button, select all bottom nodes of columns | You should see numbers of support nodes appearing in the *Current selection* field |
| Select the fixed support icon in the ***Supports*** dialog box (it will become highlighted), **Apply** | Selection of support type, the selected support type will be applied to the selected structure nodes |
| **Close** | Closing the ***Supports*** dialog box |
| *Analysis menu / Calculation Model / Generation* | Creation of the structure calculation model (mesh of planar finite elements) |



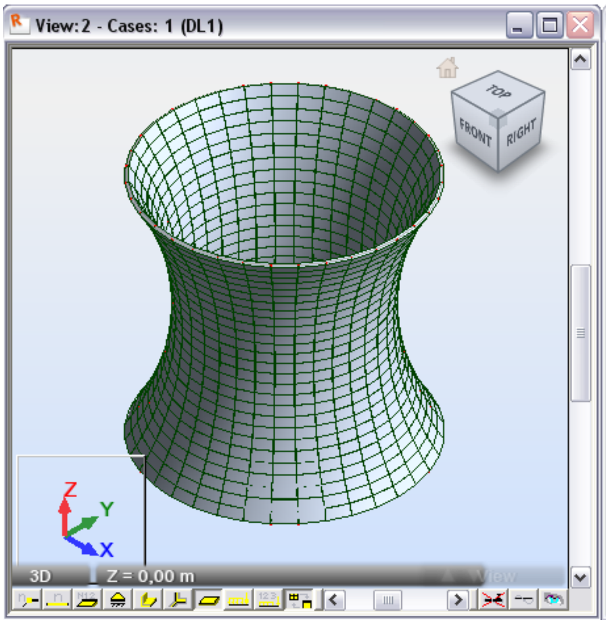

## 10.2 Cooler

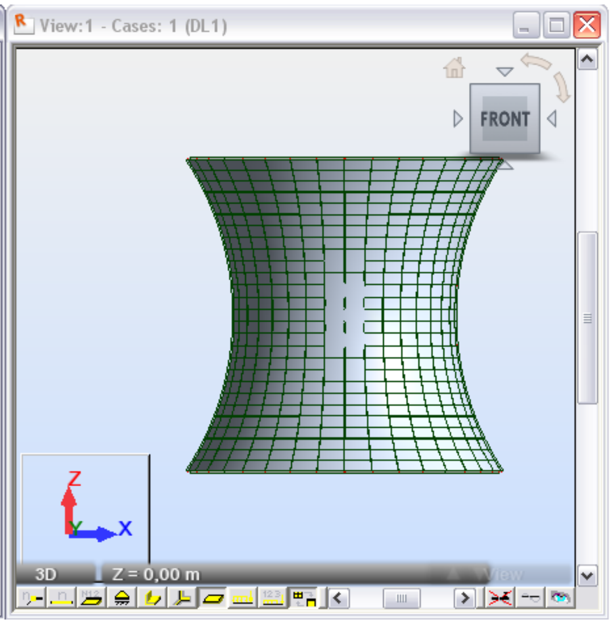
This example provides a definition of a shell structure (chimney cooler), presented schematically in the drawing below.
Data units: (m).



| OPERATION PERFORMED | DESCRIPTION |
| --- | --- |
| *View menu / Projection / ZX* | Selection of a work plane |
| *Geometry menu / Objects / Arc* | Opening the ***Arc*** dialog box to define successive components of a contour |
| Select the following arc definition method: B*eginning - End - Middle* | |
| Define the following arc on the graphical viewer:<br>Begin. (-10,0,10)<br>End (-10,0,-10)<br>Middle (-7,0,0) | Define an arc |
| **Close** | Closing the ***Arc*** dialog box |
| CTRL + A | Selection of the defined arc object |
| *Geometry menu / Objects / Revolve* | Opening the ***Revolve*** dialog box |
| Define the rotation parameters:<br>Axis: beginning (0,0,0)<br>end (0,0,10)<br>rotation angle (360)<br>number of divisions: (36)<br>Inactive options: top, base, new object | Rotation parameters |
| **Apply, Yes** | Rotation of the object is performed, accepting the message about limitations of the *Revolve* function for revolutions by 360-degree angle |
| **Close** | Closing the ***Revolve*** dialog box |
| *View menu / Projection / 3d xyz* | |
|  Select the *Thickness* icon from the Structure Model toolbar | Opening the dialog box used for defining thickness |
| Select the default panel thickness: TH_30CONCR | Selection of the thickness that will be applied to particular components of the structure |
| Write **all** in the *Panels* field | Selection of all structure elements |
| **Apply** | Application of the default thickness to all the elements of the structure |
| Close the ***FE Thickness*** dialog box | |
| *View menu / Display* | Opening the ***Display*** dialog box for visualizing selected attributes |
| Select the *Panel thickness* option on the *Panels / FE* tab | |
| **Apply**, **OK** | Closing the ***Display*** dialog box for visualizing selected attributes |

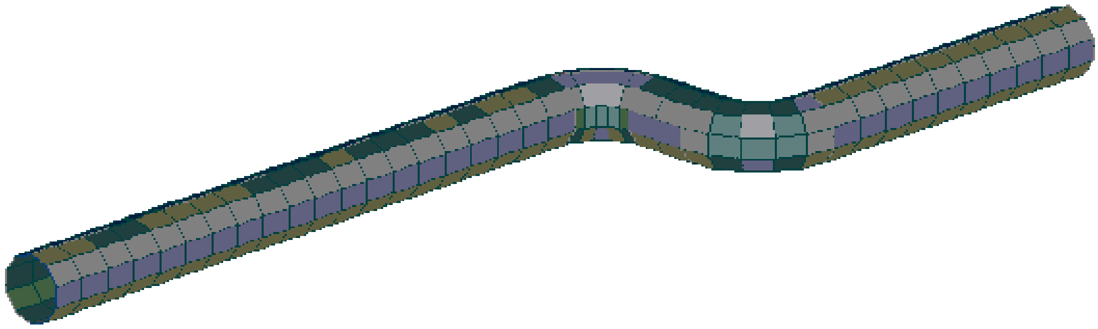
| | |
|---|---|
| Select the *Zoom All* icon from the Standard toolbar | Initial view |
| Select the *Supports* icon from the Structure Model toolbar | Opening the ***Supports*** dialog box |
| Select *Line* tab in the the ***Supports*** dialog box | |
| Select the fixed support icon in the ***Supports*** dialog box (the icon will be highlighted) | Selection of the support type |
| Indicate the bottom line (circle) of the structure | *NOTE: To select the circle you have to find a place along its circumference in such a way it will get highlighted. If you have trouble finding this place, you should add labels by checking the "Numbers and labels of edges" on the Panels / FE tab in the* ***Display*** *dialog box.* |
| **Aplay**, **Close** | Closing the ***Supports*** dialog box |
| *Analysis menu / Calculation Model / Meshing Options* | Opens the ***Meshing Options*** dialog box. |
| In the *Available meshing methods* field select the *Delaunay* option, in the *Mesh generation* field select the *Element size* option and enter 1 in the field, **OK** | Sets the meshing parameters. |
| *Analysis menu / Calculation model / Generation* | Creation of the structure calculation model (mesh of planar finite elements) |



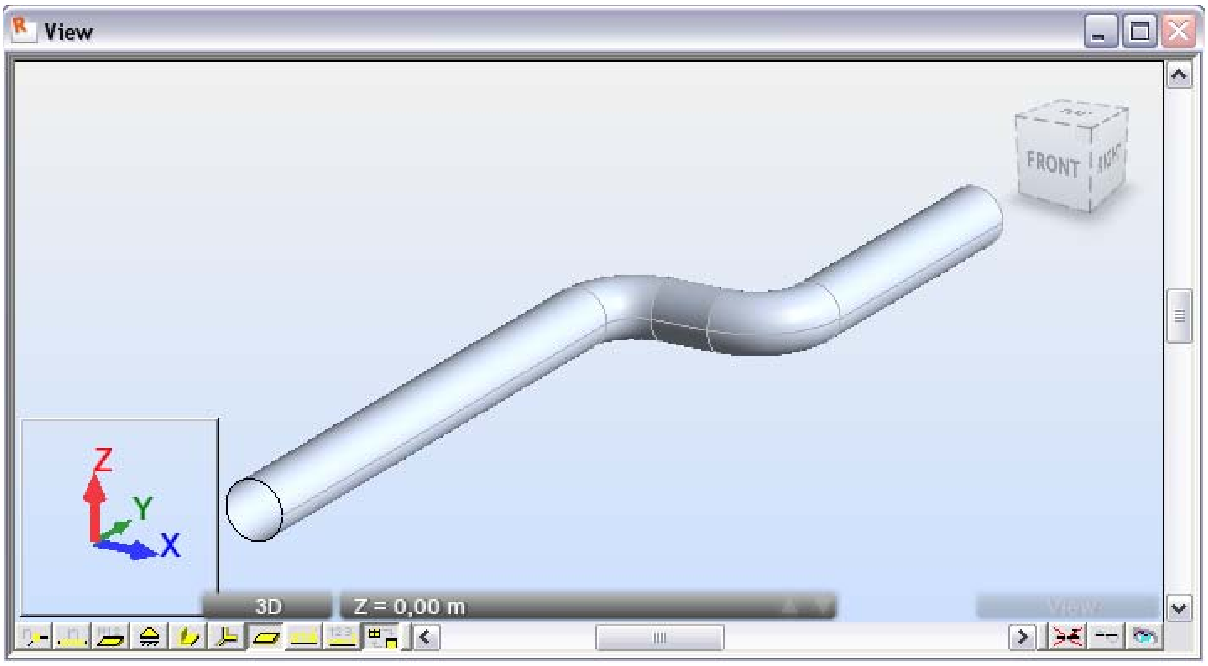

## 10.3 Pipeline

This example provides a definition of shell structure (pipe), presented schematically in the drawing below.
Data units: (m).



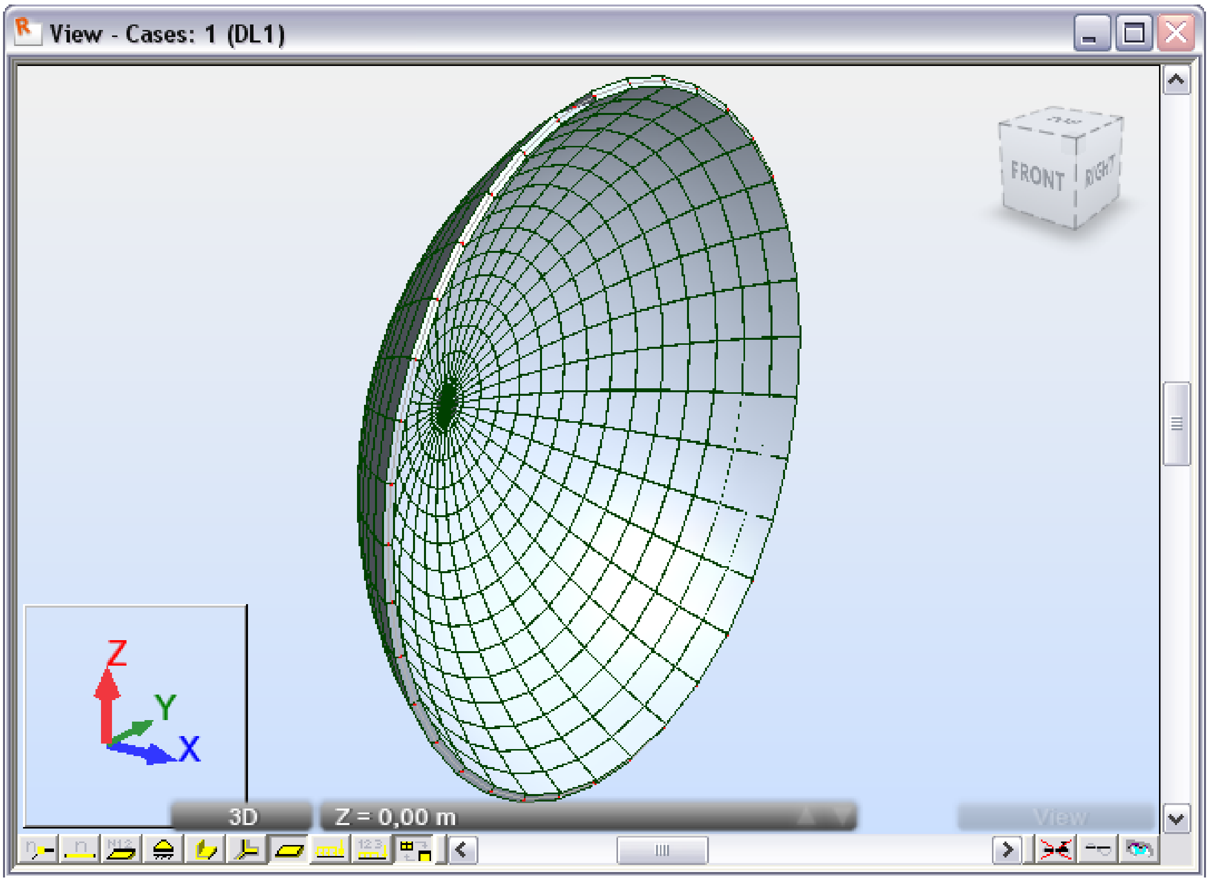
| OPERATION PERFORMED | DESCRIPTION |
|---|---|
| *View menu / Projection / ZX* | Selection of the work plane |
| *Geometry menu / Objects / Circle* | Opening the ***Circle*** dialog box to define contour components |
| Select the *Center - radius* option in the *Definition Method* group of the dialog box | |
| In the graphical viewer, define a circle with the radius of 1 m and the center at the point (0,0,0) | Definition of a circle that will be the basis for creating the pipeline |
| Close the **Circle** dialog box | |
| *View menu / Projection / 3d xyz* | |
| Select the *Edit menu / Substructure Modification / Object Modification* command from the menu | Opening the dialog box ***Objects - operations / modifications*** |
| LMC in in the *Object* field and indicate the defined circle in the graphical viewer | Selection of the circle (the number of the object is introduced into the *Object* field) |
| Press the **Extrude** button | Beginning of the definition of object modification |
| Press the **Object modification parameters** button | Definition of the parameters of extrusion |
| Define the following parameters of extrusion:<br>II to axis Y,<br>length (20) m<br>number of divisions (20)<br>Inactive options: top, base | Extrusion parameters |
| **Apply** | Extrusion performed for the circle according to the defined parameters |
| Press the **Revolve** button | Opening the ***Revolve*** dialog box |

| | |
|---|---|
| Define the following parameters for revolving the object:<br>axis beginning (2,20,0)<br>end (2,20,1)<br>rotation angle (-90)<br>number of divisions (5)<br>Inactive options: top, base | Rotation parameters |
| **Apply** | Rotation of the object is performed |
| Press the **Extrude** button | Beginning to define the parameters of extrusion |
| Define the following parameters of extrusion:<br>II to axis X,<br>length (2) m<br>number of divisions (2)<br>Inactive options: top, base | Extrusion parameters |
| Press the **Apply** button | Extrusion performed for the circle according to the defined parameters |
| Press the **Revolve** button | Opening the ***Revolve*** dialog box |
| Define the following parameters for revolving the object:<br>axis beginning (4,24,0)<br>end (4,24,1)<br>rotation angle (90)<br>number of divisions (5)<br>Inactive options: top, base | Rotation parameters |
| **Apply** | Rotation of the object is performed |
| Press the **Extrude** button | Beginning of the definition of object modification |
| Define the following parameters of extrusion:<br>II to axis Y,<br>length (10) m<br>number of divisions (10)<br>Inactive options: top, base | Extrusion parameters |
| **Apply** | Extrusion performed for the circle according to the defined parameters |
| **Close** | |
|  Select the *Zoom All* icon from the Standard toolbar | Initial view |

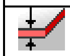

View
FRONT
Z
Y
X
3D
Z = 0,00 m

## 10.4 Axisymmetric Structures

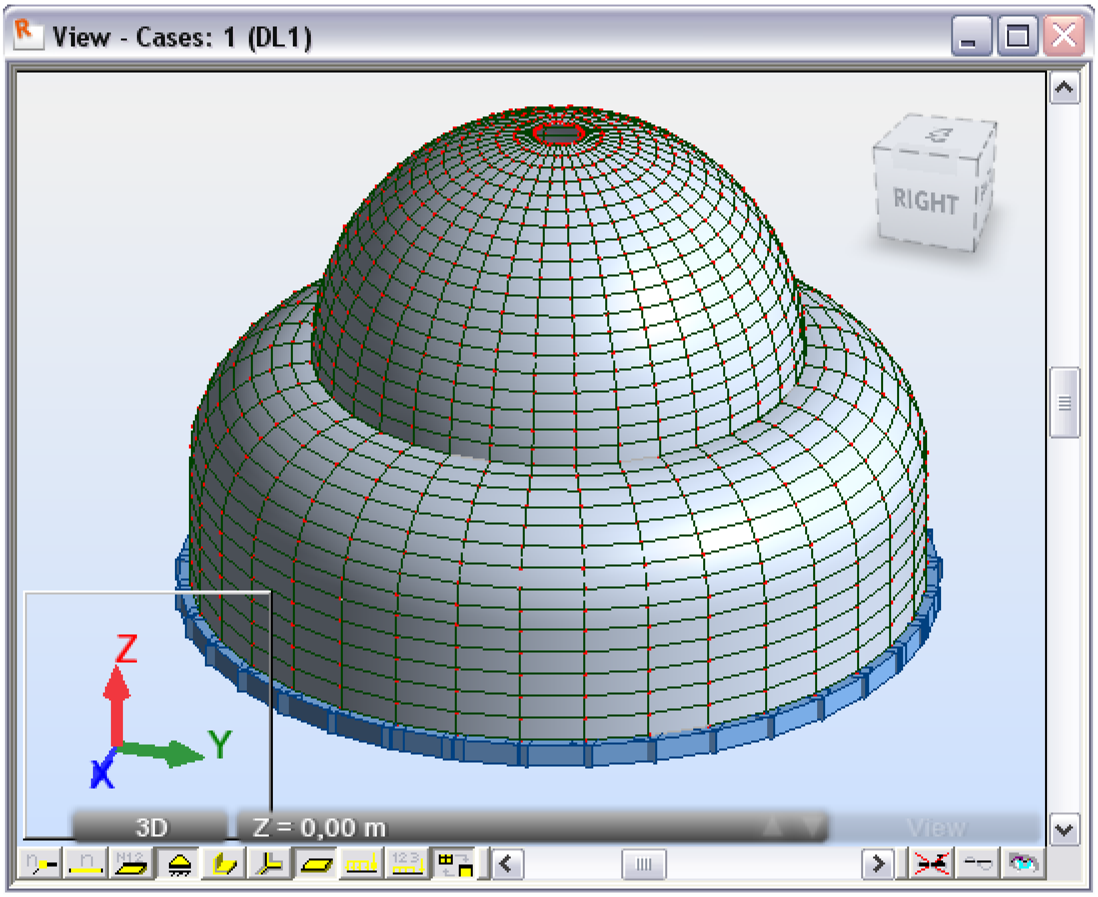
This example provides a definition of shell structure, presented schematically in the drawing below.
Data units: (m).

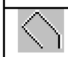


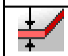
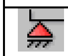

| OPERATION PERFORMED | DESCRIPTION |
|---|---|
| *View menu / Projection / ZX* | Selection of a work plane |
| *Geometry menu / Objects / Arc* | Opening the ***Arc*** dialog box to define successive components of a contour |
| Select the following arc definition method: *Beginning - End - Middle* | |
| Define the following arc in the graphical viewer:<br>Begin (0,0,10)<br>End (0,0,-10)<br>Middle (-5,0,0) | |
| **Close** | Closing the ***Arc*** dialog box |
| CTRL + A | Selection of the defined arc object |
| *Geometry menu / Objects / Revolve* | Opening the ***Revolve*** dialog box |

| | |
|---|---|
| Define the rotation parameters:<br>Axis: beginning (0,0,0)<br>end (-5,0,0)<br>rotation angle (180)<br>number of divisions (18)<br>Inactive options: top, base, new object | Rotation parameters |
| **Apply** | Rotation of the object is performed |
| **Close** | Closing the ***Revolve*** dialog box |
| *View menu / Projection / 3D xyz* | |
|  Select the *Thickness* icon from the Structure Model toolbar | Opening the dialog box used for defining thickness |
| Select the default panel thickness: TH_30CONCR | Selection of the thickness that will be applied to particular components of the structure |
| Write **all** in the *Panels* field | Selection of all structure elements |
| **Apply** | Application of the default thickness to all the elements of the structure |
| Close the ***FE Thickness*** dialog box | |
| *View / Display* | Opening the ***Display*** dialog box for visualizing selected attributes |
| Select the *Panel thickness* option on the *Panels /FE* tab | |
| **Apply**, **OK** | Closing the ***Display*** dialog box for visualizing selected attributes |
| *Analysis menu / Calculation Model / Meshing Options* | Opens the ***Meshing Options*** dialog box. |
| In the *Available meshing methods* field select the *Delaunay* option, in the *Mesh generation* field select the *Element size* option and enter 1 in the field, **OK** | Sets the meshing parameters. |
| *Analysis menu / Calculation model / Generation* | Creation of the structure calculation model (mesh of planar finite elements) |

This example provides a definition of shell structure, presented schematically in the drawing below.
Data units: (m).



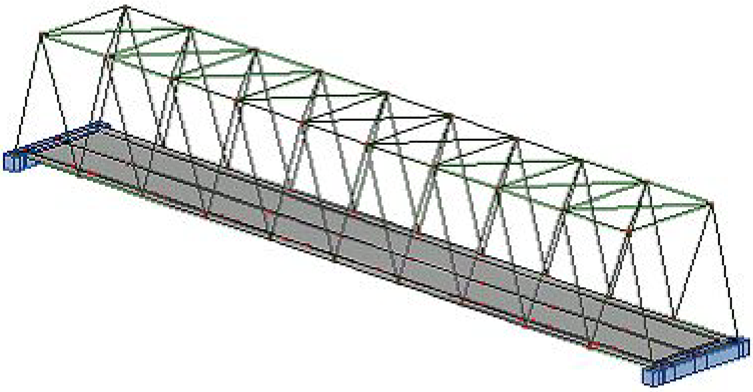

| OPERATION PERFORMED | DESCRIPTION |
|---|---|
| *View menu / Projection / ZX* | Selection of the work plane |
|  Select the *Polyline Contour* icon from the Structure Model toolbar | Opening the ***Polyline - contour*** dialog box to define successive components of a contour |
| Select *Line* option in the *Definition method* part of the dialog box | |
| Define two lines in the graphical viewer:<br>line 1: beginning (-10,0,0)<br>end (-10,0,10)<br>line 2: beginning (-15,0,0)<br>end (-15,0,5) | Definition of two lines |
| **Close** | Closing the ***Polyline - contour*** dialog box |
| *Geometry menu / Objects / Arc* | Opening the ***Arc*** dialog box to define successive components of a contour |
| Select the following arc definition method: *Center - Begin - End* | |

| | |
|---|---|
| Define the following two arcs in the graphical viewer:<br>Arc 1 with radius = 5<br>Center (-10,0,5)<br>Begin. (-15,0,5)<br>End (-10,0,10)<br>Arc 2 with radius = 10<br>Center (0,0,10)<br>Begin. (-10,0,10)<br>End (0,0,20) | |
| **Close** | Closing the ***Arc*** dialog box |
| CTRL + A | Selection of the defined arc object |
| *Geometry menu / Objects / Revolve* | Opening the ***Revolve*** dialog box |
| Define the rotation parameters:<br>Axis: beginning (0,0,10)<br>end (0,0,20)<br>rotation angle (360)<br>number of divisions (36)<br>Inactive options: top, base, new object | Rotation parameters |
| **Apply** | Rotation of the object is performed |
| **Close** | Closing the ***Revolve*** dialog box |
| *View menu / Projection / 3d xyz* | |
|  Select the *Thickness* icon from the Structure Model toolbar | Opening the dialog box used for defining thickness |
| Select the default panel thickness: TH_30CONCR | Selection of the thickness that will be applied to particular components of the structure |
| Write **all** in the *Panels* field | Selection of all structure elements |
| **Apply** | Application of the default thickness to all the elements of the structure |
| Close the ***FE Thickness*** dialog box | |
|  Select the *Zoom All* icon from the Standard toolbar | Initial view |
| *View menu / Display* | Opening the ***Display*** dialog box for visualizing selected attributes |
| Select the *Panel thickness* option on the *Panels /FE* tab | |
| **Apply**, **OK** | Closing the ***Display*** dialog box for visualizing selected attributes |
|  Select the *Supports* icon from the Structure Model toolbar | Opening the ***Supports*** dialog box |

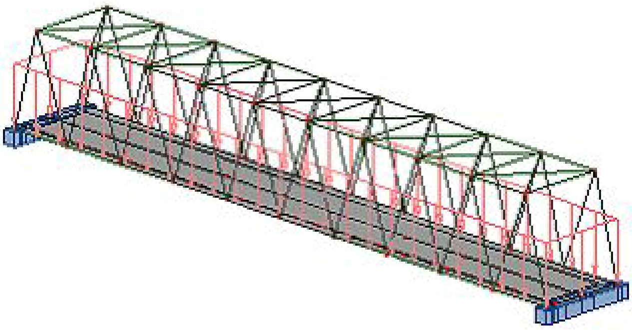
| | |
|---|---|
| Select *Line tab* in the ***Supports*** dialog box | |
| Select the fixed support icon in the **Supports** dialog box (the icon will be highlighted) | Selection of the support type |
| Point to the bottom line (circle) of the structure and select it | *NOTE: To select the circle you have to find a place along its circumference in such a way it will get highlighted. If you have trouble finding this place, you should add labels by checking the "Numbers and labels of edges" on the Panels / FE tab in the* ***Display*** *dialog box.* |
| **Apply**, **Close** | Closing the ***Supports*** dialog box |
| *Analysis menu / Calculation Model / Meshing Options,* **YES** | Opens the ***Meshing Options*** dialog box. |
| In the *Available meshing methods* field select the *Delaunay* option, in the *Mesh generation* field select the *Element size* option and enter 1 in the field, **OK** | Sets the meshing parameters. |
| *Analysis menu / Calculation model / Generation* | Creation of the structure calculation model (mesh of planar finite elements) |

# 11. 3D Single-Span Road Bridge with a Moving Load

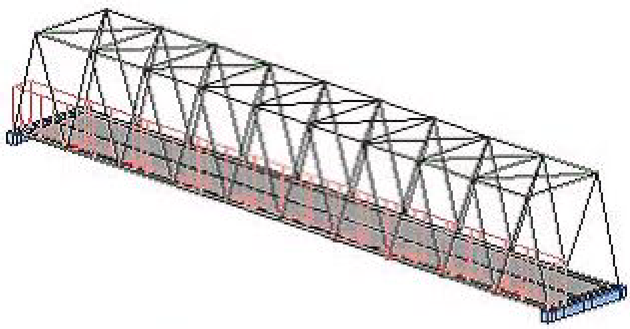
This example presents definition, analysis and design of a single-span bottom-road bridge shown in the figure below.
Data units: (m) and (kN).



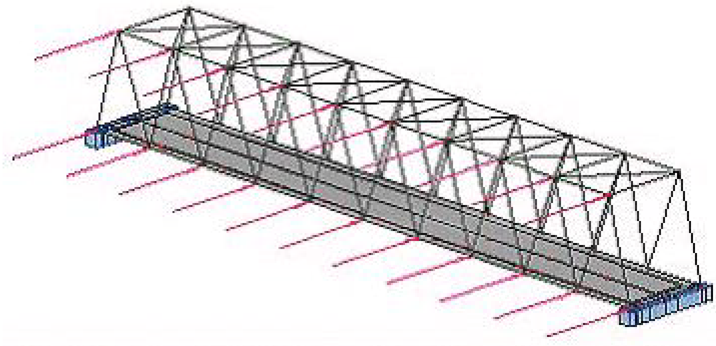
Eight load cases have been assigned to the structure and six of them are displayed in the drawings below.



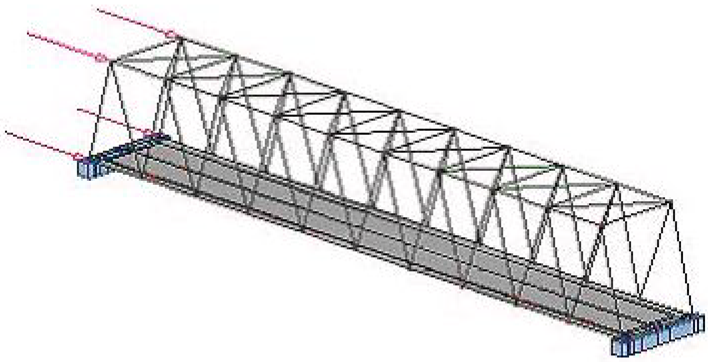
LOAD CASE 2 - LL1



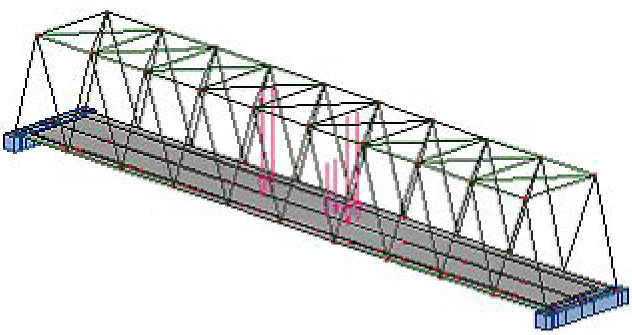
LOAD CASE 3 - LL2
LOAD CASE 4 - LL3 vertical mirror of LOAD CASE 3



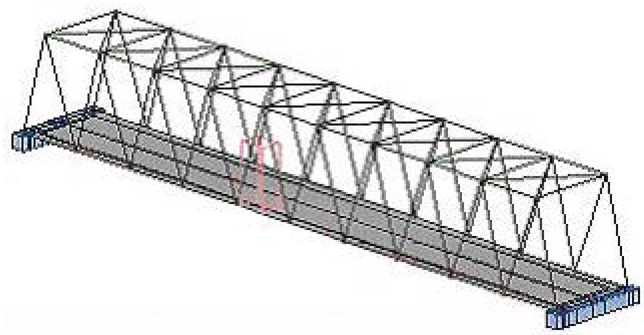
LOAD CASE 5 - WIND1



LOAD CASE 6 - WIND2



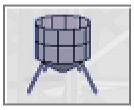
LOAD CASE 7 - Moving vehicle



LOAD CASE 8 - Moving uniform load

The following rules apply during structure definition:

- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

To run structure definition start the ***RSAP*** program (press the appropriate icon or select the command from the taskbar). The vignette window will be displayed on the screen and the icon  in the second row (***Shell Design***) should be selected.

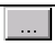
*NOTE: The European section database (EURO) has been used in this example.*

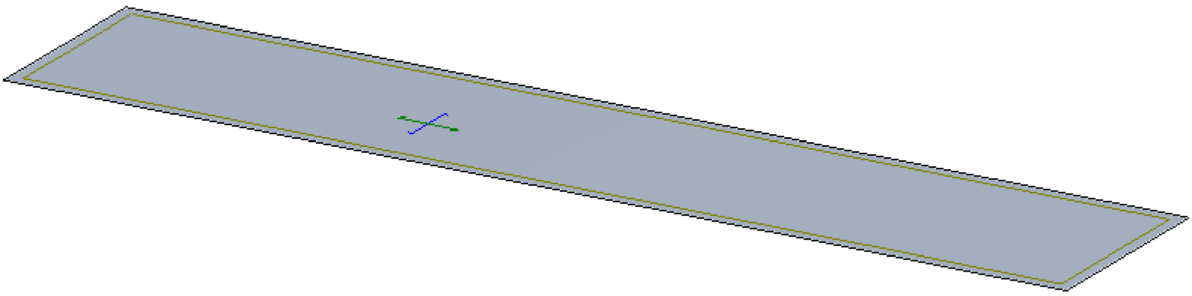
## 11.1 Model Definition

### *11.1.1 Structure Geometry Definition*

*Bridge Floor - Definition*

| **PERFORMED OPERATION** | **DESCRIPTION** |
|---|---|
| *View menu / Projection / Xy* | Once this option is selected the structure is set on the XY plane. |
| *Geometry menu / Objects / Polyline - contour* | Opens the ***Polyline - Contour*** dialog box which allows defining various line types. |
| LMC in the **Geometry** button | Opens the dialog box which allows defining a contour. |
| Enter the following coordinates into the field highlighted in green:<br>(0,0,0) **Add**,<br>(30,0,0) **Add**,<br>(30,6,0) **Add**,<br>(0,6,0) **Add**,<br>**Apply**, **Close** | Defines a contour. |
|  Select *Zoom All* icon from the standard toolbar. | Presentation of the structure initial view. |

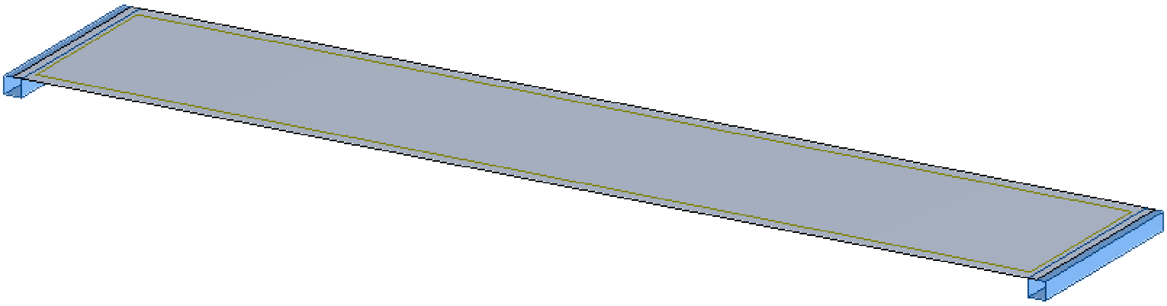
| | |
|---|---|
| *Geometry menu / Panels* | Opens the ***Panel*** dialog box that allows defining panels within structures. |
| LMC in the  button located on the right side of the *Thickness* field | Opens the ***New Thickness*** dialog box. |
| Set the new *thickness* value: 20 cm, *Material* C25/30 and enter the new name TH20, **Add**, **Close** | Defines a new panel and closes the dialog box. |
| LMC in the ***Reinforcement*** field and set *RC floor* reinforcement | Defines reinforcement type that will be applied to the defined panel. |
| LMC in the ***Model*** field and set *Shell* model | Defines a calculation model that will be applied to the defined panel. |
| LMC in the *Internal point* field and select a point inside the panel by left-clicking on it | Applies current properties to the selected panel. |
| **Close** | Closes the ***Panel*** dialog box. |
| *View menu / Projection / 3D xyz* | Once this option is selected, a 3D view of the structure is displayed. The defined structure is displayed in the drawing below. |



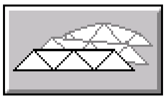
*Support Definition*

| | |
|---|---|
|  LMC on the field to select the **Structure Model/Supports** Layout | Selects the ***RSAP*** layout which allows defining supports. |
| In the ***Supports*** dialog box select the fixed support icon (the icon will be highlighted) | Selects the support type. |
| In the ***Supports*** dialog box on the *Linear* tab LMC on the *Current Selection* field | |
| Switch to the graphic viewer; pressing the left mouse button select two shorter edges of the structure, **Apply** | Assigns fixed supports to two shorter edges of the structure. |

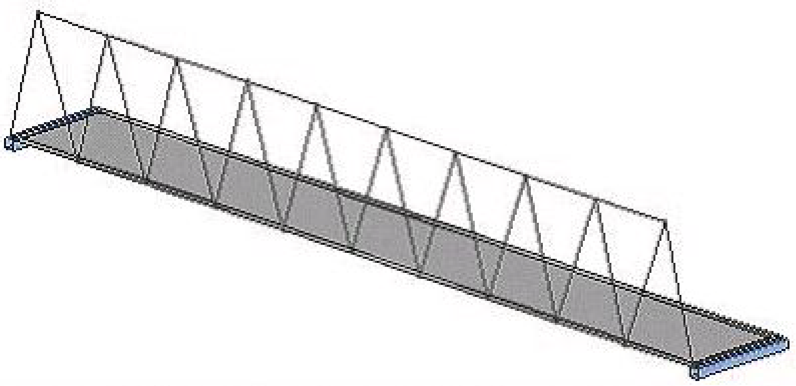
| | |
|---|---|
|   LMC on the field to select the **Structure Model/Geometry** Layout | Selects the initial ***RSAP*** layout. |
| *View menu / Display* | Opens the ***Display*** dialog box, which allows selecting structure attributes for presentation. |
| On the *Model* tab in the ***Display*** dialog box activate Supports/*Supports - symbols*, **Apply**, **OK** | Displays symbols of structure supports on the screen, closes the ***Display*** dialog box.<br>The defined structure is displayed in the drawing below. |



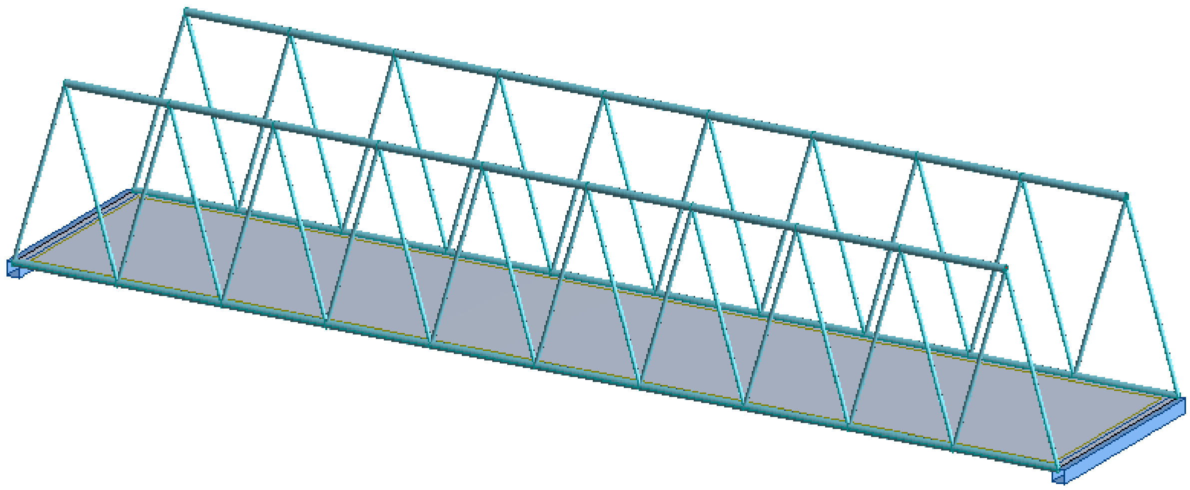
*Definition of Bridge Trusses (Application of Library Structures)*

| | |
|---|---|
|  Select the *Library Structure* from the Structure Model toolbar. | Opens the ***Typical Structures*** dialog box that allows defining typical structures (structure elements). |
| On the *Structure Database Selection* field select the *Library of typical structures - beams, frames, trusses* option.<br><br>LMC (twice) the icon the last one in the third row | In the ***Typical Structures*** dialog box a new *Structure Selection* field appears. Opening of the **Trapezoid Truss Type 3** dialog box. |
| In the *Dimension* tab LMC the *Length L1* field<br>{30} | Defines length of the lower truss chord (it may be defined graphically in the graphical viewer). |
| On the *Dimension* tab LMC the *Length L2* field<br>{27} | Defines length of the upper truss chord (it may be defined graphically in the graphical viewer). |
| LMC the *Height H* field<br>{5} | Defines truss height (it may be defined graphically in the graphical viewer). |
| LMC the *Number of Fields*<br>{10} | Defines a number of fields into which the truss will be divided. |
| On the *Dimensions* tab in the *Continuous chord* field activate option ***No*** | Applies divided chords to the structure. |
| LMC on the *Sections* tab;<br>To all truss chords (upper and lower) assign (TRON 219x6.3) and to diagonals asign (TRON 114x6.3) | Assigns the section to the truss bars. |

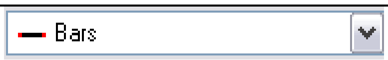
| | |
|---|---|
| On the *Insert* tab LMC the *Insertion Point*, select coordinates: (0,0,0) | Defines the insertion point for the truss; the defined structure is displayed in the drawing below. |
| **Apply**, **OK** | Creates the defined structure at the indicated point within the construction and closes the ***Merge Structure*** dialog box. |



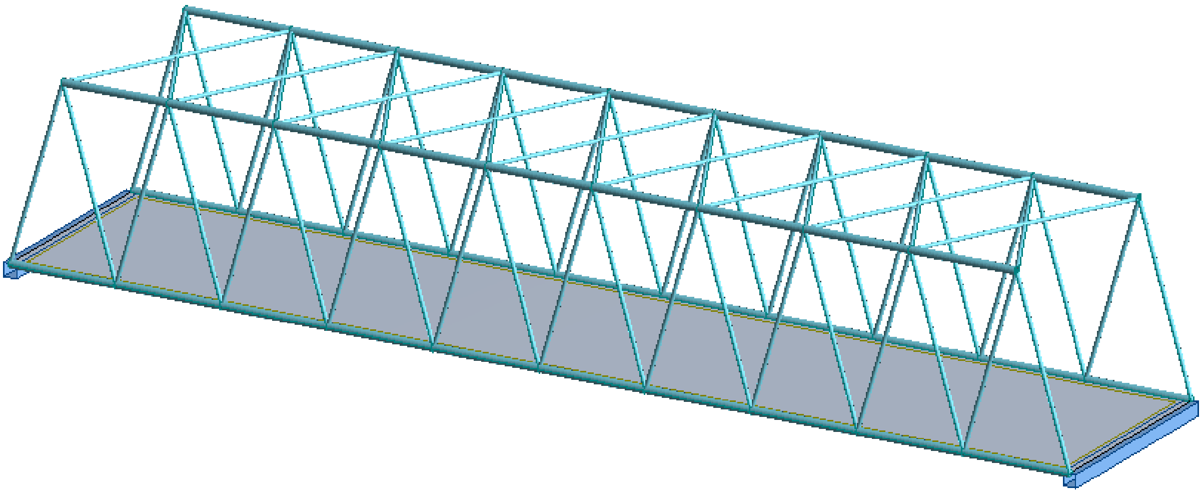
| | |
|---|---|
| Switch to the graphic viewer and select all truss bars | Selects all truss bars. |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC on the field (dX, dY, dZ), (0,6,0) | Defines a new translation vector. |
| **Execute**, **Close** | Translates beams, highlights translated beams and closes the ***Translation*** dialog box. The defined structure is displayed in the drawing below. |



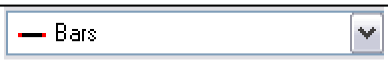
*Bracing Definition*

| | |
|---|---|
|   LMC on the field to select the **Structure Model/Bars** Layout | Selects the **BARS** layout from the list of available ***RSAP*** layouts, which allows defining bars. |

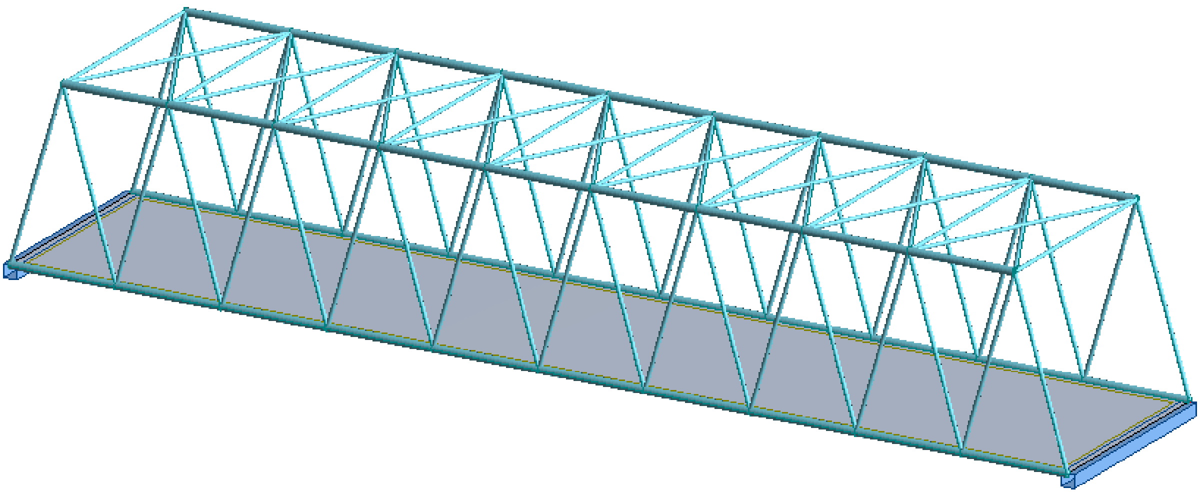
| | |
|---|---|
| LMC in the *Bar Type* field and select: *Simple bar*,<br>LMC in the *Section* field and select (TRON 114x6.3) | Defines bar properties. The section from the European section database (EURO) has been used.<br>*Note: If the TRON 114x6.3 section is not available on the list, one should select **Structure Model / Properties**, press the icon and add the section to the list of active sections.* |
| LMC in the *Beginning* and *End* fields (background color changes to green)<br>(1.5,0,5) (4.5,6,5), **Add**<br>(1.5,6,5) (4.5,0,5), **Add** | Defines bracing. |
| RMC in any point within the graphic viewer which opens the context menu. Choose the *Select* option and indicate two recently defined bars, while the CTRL key is pressed. | Selects two recently defined bars. |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC in the field (dX, dY, dZ)<br>(3,0,0),<br>in the *Number of Repetitions* field<br>{8} | Defines the translation vector and allows defining the number of repetitions. |
| **Execute**, **Close** | Translates the structure, highlights translated bars and closes the ***Translation*** dialog box. |
|  <br>LMC on the field to select the **Structure Model/Geometry** Layout | Selects the initial ***RSAP*** layout. The defined structure is displayed in the drawing below. |



*Cross Beams - Definition*

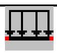
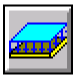
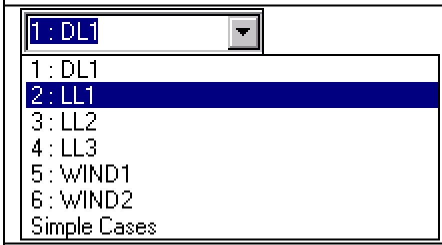
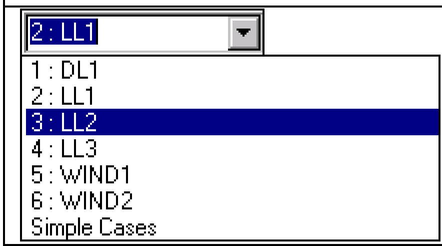
| | |
|---|---|
|  <br>LMC on the field to select the **Structure Model/Bars** Layout | Selects the **BARS** layout from the list of available ***RSAP*** layouts, which allows defining bars. |

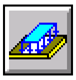
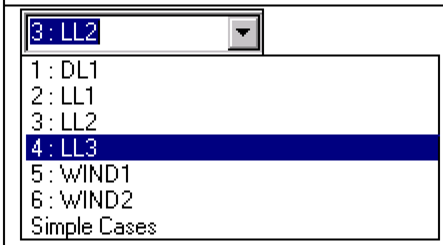
| | |
|---|---|
| LMC in the *Bar Type* field and select: *Simple bar*<br>LMC in the *Section* field and select (TRON 114x6.3). | Defines bar properties. The section from the European section database (EURO) has been used.<br>. |
| LMC in the *Beginning* and *End* fields (background color changes to green)<br>(1.5,0,5) (1.5,6,5), **Add** | Defines cross beams. |
| RMC in any point within the graphic viewer, which opens the context menu. Chose the *Select* option and indicate the recently defined bar. | Selects the recently defined bar. |
| *Edit menu / Edit / Translate* | Opens the ***Translation*** dialog box. |
| LMC in the field (dX, dY, dZ)<br>(3,0,0)<br>in the *Number of Repetitions* field<br>{9} | Defines a translation vector and allows defining a number of repetitions. |
| **Execute**, **Close** | Translates the structure and closes the ***Translation*** dialog box. |
| Geometry<br>LMC on the field to select the **Structure Model/Geometry** Layout | Selects the initial ***RSAP*** layout. The defined structure is displayed in the drawing below. |



### *11.1.2 Load Definition*

| | |
|---|---|
| Loads<br>LMC on the field to select the **Structure Model / Loads** Layout | Selects the ***RSAP*** program layout that allows defining structure loads. |
| LMC in the **New** button in the ***Load Types*** dialog box | Defines a *dead load* with the standard name DL1. |
| LMC in the *Nature* field: *(Live1)* | Selects the type of a load case: *live*. |

| | |
|---|---|
| LMC on the **New** button<br>LMC on the **New** button<br>LMC on the **New** button | Defines two cases of *live* load with the standard names: LL1, LL2 and LL3. |
| LMC on the *Nature* field: *(Wind)* | Selects the type of a load case: *wind*. |
| LMC on the **New** button<br>LMC on the **New** button | Defines two cases of *wind* load with standard names: WIND1 and WIND2. |
| | *Note: The self-weight load has been automatically applied to all structure bars (in the "Z" direction).* |
| LMC on the  icon on the **Bar Loads** toolbar | Opens the ***Load Definition*** dialog box. |
| In the ***Load Definition*** dialog box select *Surface* tab and press the  icon | Opens the ***Uniform Planar Load*** dialog box |
|  | Selects the load case: Live Load 1. |
| In the *Values Z:* field enter -2.5 | Defines a value of the uniform load acting on surface FEs in the direction of the Z axis of the global coordinate system. |
| **Add** | Closes the ***Uniform Planar Load*** dialog box. |
| In the *Apply To* field enter 1 | Displays the current selection of structure panel. |
| **Apply** | Applies predefined load to a chosen panel. |
| In the ***Load Definition*** dialog box select *Surface* tab and press  | Opens the ***Uniform planar load on contour*** dialog box. |
|  | Selects load case: Live Load 2. |
| In the *Values Z:* field enter -2.0 | Defines a value of the uniform load acting on surface FEs in the direction of the Z axis of the global coordinate system. |
| LMC on the **Contour definition** button | Opens the dialog box that allows defining the contour to which the load is applied. It may be performed either in the dialog box or graphically on the screen. |

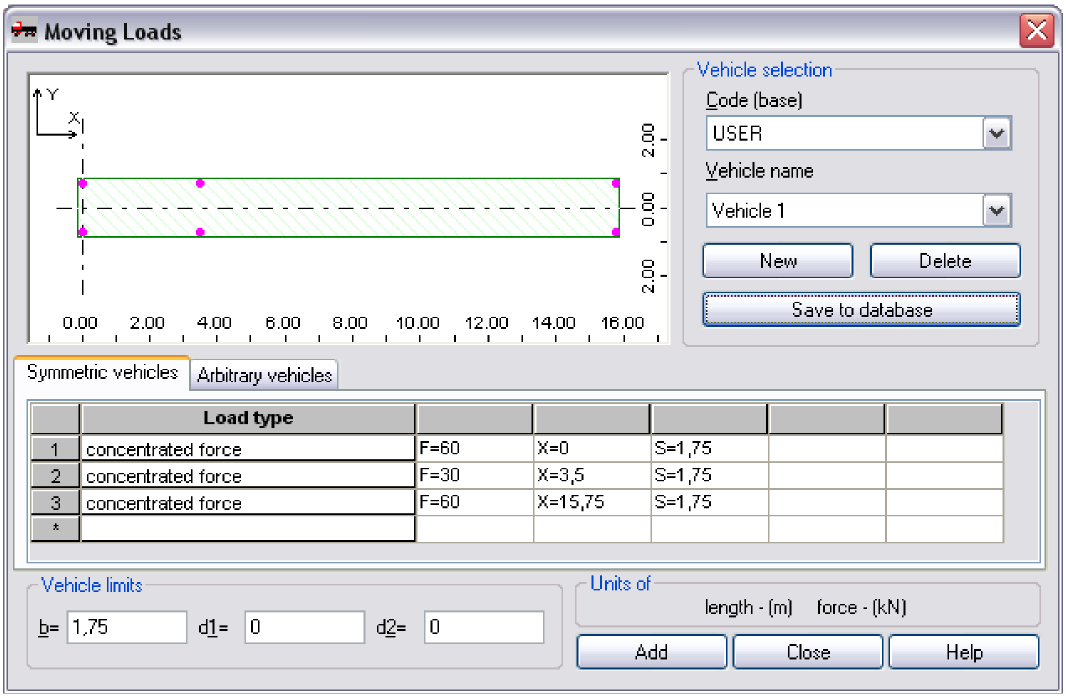
| | |
|---|---|
| In the green field enter coordinates that define a contour:<br>(0,0,0), **Add**<br>(30,0,0), **Add**<br>(30,1.5,0), **Add**<br>(0,1.5,0), **Add** | Defines the contour to which the loads will be applied. |
| **Add** | Closes the ***Uniform planar load on contour*** dialog box. |
| In the *Apply To* field enter 1 | Displays the current selection of structure panel. |
| **Apply** | Applies predefined load to a chosen panel. |
| In the ***Load Definition*** dialog box select the *Surface* tab and press the  icon | Opens the ***Uniform Planar Load (contour)*** dialog box. |
|  | Selects load case: Live Load 3. |
| In the *Values Z:* field enter -2.0 | Defines a value of the uniform load acting on surface FE in the direction of the Z global coordinate system. |
| LMC on the **Contour definition** button | Opens dialog box that allows defining the contour, to which the load will be applied. It may be performed either in the dialog box or graphically on the screen. |
| In the green field enter coordinates that define a contour:<br>(0,4.5,0), **Add**<br>(30,4.5,0), **Add**<br>(30,6,0), **Add**<br>(0,6,0), **Add** | Defines contour to which the loads will be applied. |
| **Add** | Closes the ***Uniform Planar Load (contour)*** dialog box. |
| In the field *Apply To* type 1 | Displays the current selection of structure panel. |
| **Apply**, **Close** | Applies the predefined load to a chosen panel, closes the ***Uniform Planar Load (contour)*** dialog box. |
| *View menu / Projection / Zx* | Once this option is selected, the Zx plane is chosen. |
| LMC on the fifth field in the ***Case*** column, select 5th load case: WIND1 from the list | Defines loads for the fifth load case. |
| LMC on the field in the ***Load Type*** column, select (*nodal force)* from the list as a load type | Selects the load type. |
| LMC on the field in the ***List*** column, select all nodes of the front truss in a graphic way | Selects nodes to which *nodal force* will be applied. |

| | |
|---|---|
| LMC on the field in the "FY=" column and enter the value: (10) | Selects the direction and value of the *nodal force* load. |
| *View menu / Projection / 3D xyz* | Once this option is selected, a 3D view of the structure is displayed. The defined structure is displayed in the drawing below. |
| LMC on the fifth field in the ***Case*** column, select 6th load case: WIND2 from the list | Defines loads for the sixth load case. |
| LMC on the field in the ***Load Type*** column, select (*nodal force)* from the list as a load type | Selects the load type. |
| LMC on the field in the ***List*** column, select four left nodes belonging to both trusses | Selects nodes to which *nodal force* will be applied. |
| LMC on the field in the "FX=" column and enter the value: (6) | Selects the direction and value of the *nodal force* load. |

### *11.1.3 Definition of the Moving Load Applied to the Bridge Floor*

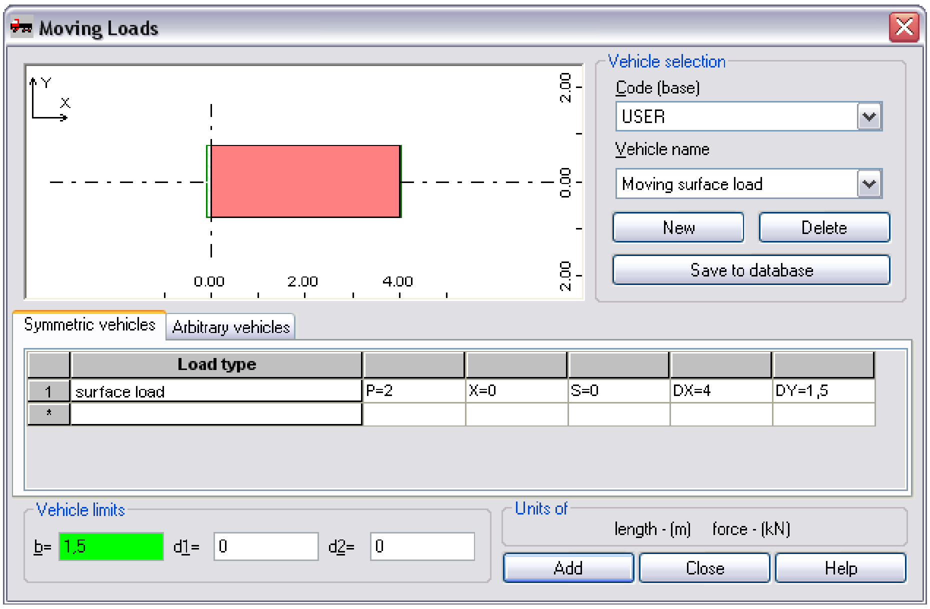
| | |
|---|---|
|  LMC on the field to select the **Structure Model/Geometry** Layout | Selects the initial ***RSAP*** layout. |
| *Tools menu / Job preferences / Databases/ Vehicle loads* | Open the ***Job preferences*** dialog box. |
|  | Pressing the *Create new database icon* results in opening the ***New Moving Load*** dialog box. |
| Enter:<br>in the *Database* field: User<br>in the *Database name* field: User-defined database<br>in the *Database description* field: User-defined vehicles<br>in the *Internal units of the database* chose (m) as *Length units* and (kN) as *Force unit* | |
| **Create** | Closes the ***New Moving Load*** dialog box. |
| **OK** | Closes the ***Job Preferences*** dialog box. |
| *Loads / Special loads / Moving* | Opens the ***Moving Loads*** dialog box. |
|  | Opens the ***Moving Loads*** dialog box and starts defining a new vehicle. |
| On the *Symmetric Vehicles* tab LMC on the **New** button | Defines a new vehicle. |
| Enter the vehicle name: *Vehicle 1*, **OK** | Defines the name of the new vehicle. |
| LMC the first line in the table located in the lower part of the dialog box | Defines operating forces. |
| Select the load type: concentrated force | Selects a load type. |
| F = 60, X = 0.0, S = 1.75 | Defines the value and location of the concentrated force. |
| LMC the second line in the table located in the lower part of the dialog box | Defines operating forces. |
| Select the load type: concentrated force | Selects a load type. |
| F = 30, X = 3.5, S = 1.75 | Defines the value and location of the concentrated force. |
| LMC the third line in the table located in the lower part of the dialog box | Defines operating forces. |
| Select the load type: concentrated force | Selects a load type. |

| | |
|---|---|
| F = 60,X = 15.75, S = 1.75 | Defines the value and location of the concentrated force. |
| LMC the **Save to database** button | Opens the ***Moving Load Databases*** dialog box. |
| Select User database and press the **OK** button in the ***Moving load databases*** dialog box | Saves the defined vehicle to the user-defined database. |
| **Add**, **Close** | Adds the defined vehicle to the list of active vehicles and closes the ***Moving Loads*** dialog box. |



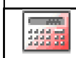

| | |
|---|---|
| In the *Name* field, enter the name of the moving load (case number 7) *Moving vehicle* | Defines a name of the moving load. |
| LMC the **Define** button | Starts defining the route of the Moving Crane vehicle; the ***Polyline - Contour*** dialog box is opened. Activate the *Line* option. |
| In the ***Geometry*** dialog box define two points determining the route of the moving load:<br>Point P1 (0,3,0)<br>Point P2 (30, 3,0) | Defines the vehicle route. |
| **Apply**, **Close** | Closes the ***Polyline - Contour*** dialog box. |
| LMC the *Step* field: {8}<br>Assume the default value of load direction: (0,0,-1) which means that the load will be operating in the Z direction and its sense will be opposite to the Z axis sense | Defines the step of a position change for the moving load and the load application direction. |
| LMC the *Automatic* option located in the *Application Plane* field | Selects the plane of load application. |

| | |
|---|---|
| **Apply** | Generates the moving load case according to the adopted parameters. |
| | Opens the ***Moving Loads*** dialog box and starts defining a new vehicle. |
| On the *Symmetric vehicles* tab LMC on the **New** button | Defines a new vehicle. |
| Type the vehicle name: *Moving surface load*<br>**OK** | Defines the name of a new vehicle. |
| LMC the first line in the table located in the lower part of the dialog box | Defines operating forces. |
| Select the load type: *surface load* | Selects the load type. |
| P = 2.0, X = 0.0, S = 0.0, DX = 4.0, DY = 1.5 | Defines the value and location of the surface load. |
| LMC the **Save to database** button | Opens the ***Moving Load Databases*** dialog box. |
| Select the User database, **OK** in the ***Moving load databases*** | Saves the defined vehicle to the user-defined database. |
| **Add**, **Close** | Adds the defined vehicle to the list of active vehicles and closes the ***Moving Loads*** dialog box. |

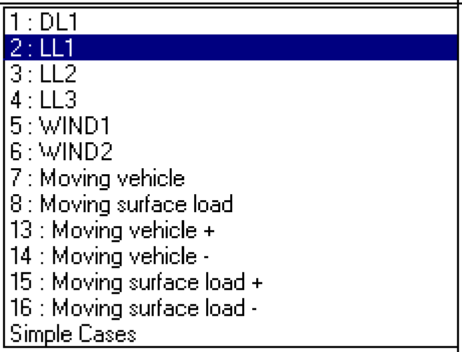
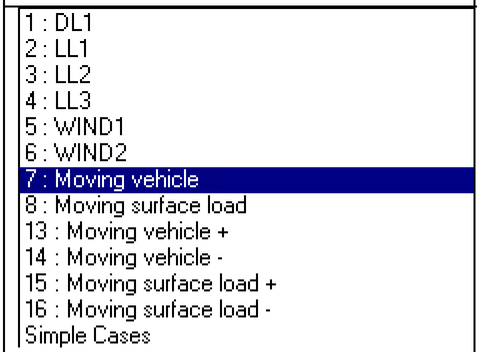


| | |
|---|---|
| In the *Name* field, type the name of the moving load (case number: 8): *Moving surface load* | Defines the name of a moving load. |
| LMC the **Define** button | Starts defining the route of the Moving Crane vehicle: the ***Polyline - Contour*** dialog box is opened. Activate the *Line* option. |
| In the ***Geometry*** dialog box define two points determining the route of the moving load:<br>Point P1(0,1.5,0)<br>Point P2 (30,1.5,0) | Defines the vehicle route. |
| **Apply**, **Close** | Closes the ***Polyline - Contour*** dialog box. |
| LMC the *Step* field<br>{8}<br>Assume the default value of load direction: (0,0,-1) which means that the load will be operating in the Z direction and its sense will be opposite to the Z axis sense | Defines the step of a position change for the moving load and the load application direction. |
| LMC the *Automatic* option located in the *Application Plane* field | Selects the plane of load application. |
| **Apply**, **Close** | Generates a second moving load case according to the adopted parameters and closes the ***Moving Loads*** dialog box. |

## 11.2 Structural Analysis

| | |
|---|---|
| *Tools menu / Job Preferences / Structure Analysis* | Opens the ***Job Preferences*** dialog box |
| Switch off the option: *Automatic freezing of results of structure calculations,* | Switches off freezing of structure calculations results. |
| *Job Preferences / Work Parameters*<br>Select *Fine Meshing type*, and switch on *Automatic mesh adjustment*<br>**OK** | Defines meshing parametres, closes the ***Job Preferences*** dialog box. |
|  | Starts calculations of the defined structure. Once the calculations are completed, the title bar of the viewer will present the following information: *Finite Elements Results - available*. |

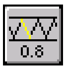
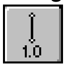
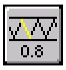
## *11.2.1 Result Presentation in the Form of Maps*

| | |
|---|---|
|   LMC on the field to select the **Structure Model / Results - maps** Layout | The **RESULTS** layout of the ***RSAP*** program will open. The screen will be divided into two parts: the graphic viewer containing the structure model and the ***Maps*** dialog box. |
|   | Selects the load case: 2 (LL1). |
| On the *Detailed* tab activate the **z** option in the *Displacement - u,w* line | Activates visualization of the displacement for individual surface FEs in the local coordinate system. These are the displacements in the direction perpendicular to the element surface. |
| Activate *Maps* option | Allows presentation of results obtained for FE in the form of maps. |
| **Apply** | Presents the structure displacement. |
|   | Selects the load case: 7 (Moving vehicle). |
| On the *Deformation* tab switch on the *Active* option | Activates presenting deformation of the currently designed structure. |
| **Apply** | Presents the structure displacement. |
| *Loads menu / Select Case Component* | Opens the ***Case Component*** dialog box. |
| LMC the **Animation** button | Opens the ***Animation*** dialog box. |
| LMC the **Start** button | Starts performing the displacement animation for the structure. |
| Stop (LMC the  button) and close the animation toolbar | Stops the animation. |
| **Close** | Closes the ***Case component*** dialog box. |

| | |
|---|---|
| Switch off the options *Displacement - u,w* and *active* in the ***Maps*** dialog box<br>**Apply** | |

## 11.3 Structure Member Design

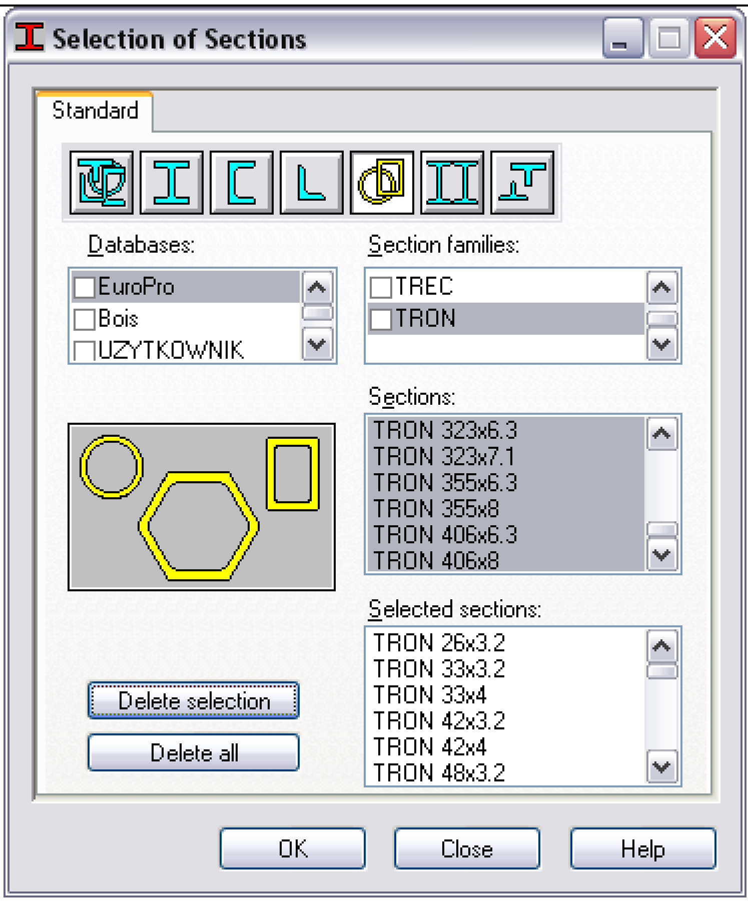
| | |
|---|---|
| LMC on the field to select the **Structure Model/Geometry** Layout | Selects the initial ***RSAP*** layout. |
| Switch to the graphic viewer and select from the upper menu: *Geometry / Code Parameters / Steel/Aluminium Member Type* | Opens the ***Member Type*** dialog box. |
| | Opens the ***Member Definitions - Parameters*** dialog box. |
| In the *Buckling length coefficient Y* press the button | Opens the ***Buckling Diagrams*** dialog box which allows defining buckling length for members. |
| Select the last button in the second row , **OK** | Applies the selected buckling diagram and appropriate buckling length coefficient, closes the ***Buckling Diagrams*** dialog box. |
| In the *Buckling length coefficient Z* press the button | Opens the ***Buckling Diagrams*** dialog box which allows defining buckling length for members. |
| Select the last button in the second row , **OK** | Applies selected buckling diagram and appropriate buckling length coefficient, closes the ***Buckling Diagrams*** dialog box. |
| In the *Member Type* field enter: *Chords* | Applies the name to a new bar type. |
| **Save**, **Close** | Saves current parameters for the *Chord* member type, closes the ***Member Definitions - Parameters*** dialog box. |
| LMC on the *Line/Bars* option located in the ***Member Type*** dialog box, switch to the graphic viewer and select all bars belonging to chords | Selects truss chords. |
| **Apply** | Applies current member type (*Chords*) to the selected truss bars. |
| | Opens the ***Member Definitions - Parameters*** dialog box. |
| In the *Buckling length coefficient Y* press the button | Opens the ***Buckling Diagrams*** dialog box which allows defining buckling length for members. |

| | |
|---|---|
| Select the first button in the third row , **OK** | Applies the selected buckling diagram and appropriate buckling length coefficient, closes the ***Buckling Diagrams*** dialog box. |
| In the *Buckling length coefficient Z* press the  button | Opens the ***Buckling Diagrams*** dialog box which allows defining buckling length for members. |
| Select the first button in the third row , **OK** | Applies the selected buckling diagram and appropriate buckling length coefficient, closes the ***Buckling Diagrams*** dialog box. |
| In the *Member Type* field enter: *Diagonals* | Assigns the name to a new bar type. |
| **Save**, **Close** | Saves current parameters for the *Cross braces* member type, closes the ***Member Definitions - Parameters*** dialog box. |
| LMC on the *Line/Bars* option located in the ***Member Type*** dialog box, switch to the graphic viewer and select all diagonals belonging to trusses | Selects cross braces in the trusses. |
| **Apply**, **Close** | Applies current member type (*Diagonals*) to the selected truss bars, closes the ***Member Type*** dialog box. |

### *11.3.1 Steel Design*

Code EN 1993-1-8:2005

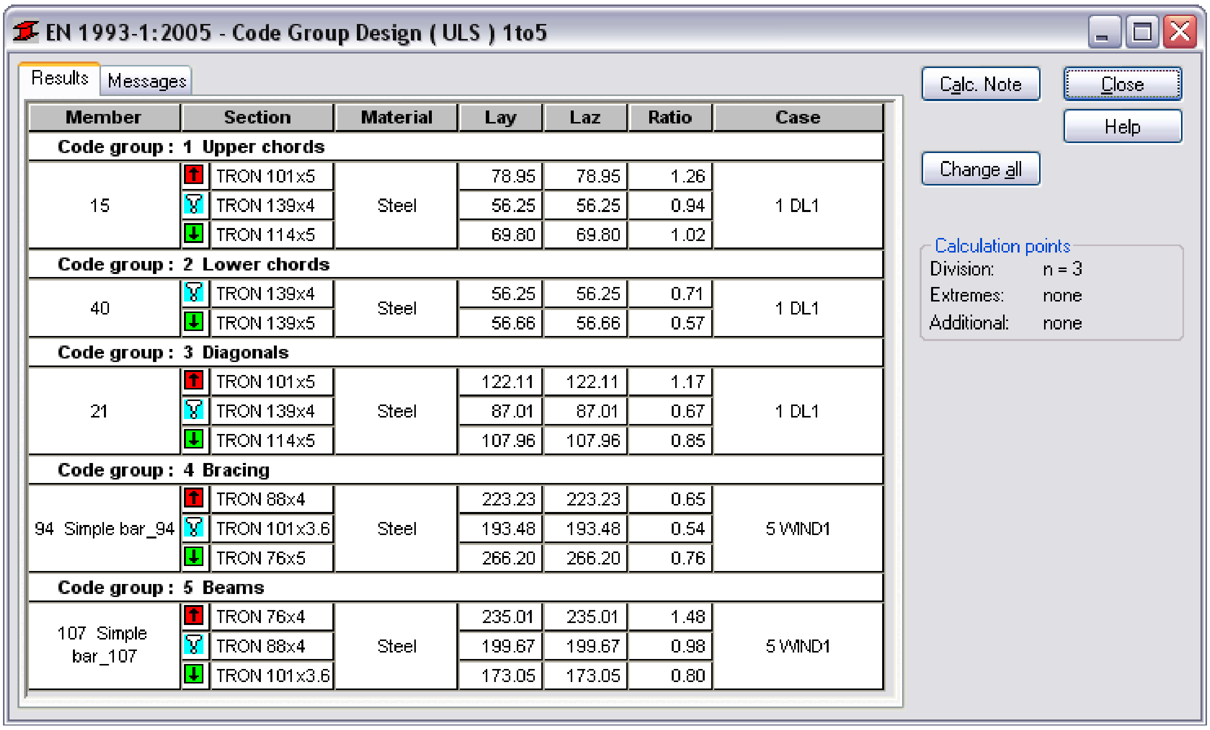
| | |
|---|---|
|  LMC on the field to select the **Structure Design/Steel/Aluminum Design** Layout | Selects the **STEEL/ALUMINUM DESIGN** layout from the list of available ***RSAP*** layouts. |
| In the *Groups* tab located in the ***Definitions*** dialog box press the **New** button | Starts defining a new group. |
| Define the first group with the following parameters:<br>Number: 1<br>Name: Upper chords<br>Member list: LMC on the ***View*** edit viewer; select all the bars of the upper chords while the CTRL key is pressed<br>Material: STEEL EC3 Steel S235 | Defines the first group consisting of all bars belonging to the upper chords in the structure. |
| **Section**<br>In the ***Selection of Sections*** dialog box select all TRON profiles which thikness is bigger than 3 mm<br>**OK** | Opens the ***Selection of Sections*** dialog box. Using this option user can select sections that will be applied during verification and design of steel and aluminum elements. |




| | |
|---|---|
| **Save** | Saves the parameters of the first member group. |
| LMC the **New** button on the *Groups* tab in the ***Definitions*** dialog box | Allows defining a second member group. |
| Define the second group with the following parameters:<br>Number: 2<br>Name: Lower chords<br>Member list: LMC on the ***View*** edit viewer; select all the bars of the lower chords while the CTRL key is pressed<br>Material: STEEL EC3 Steel S235 | Defines the second group consisting of all bars belonging to the lower chords in the structure. |
| **Section**<br>In the ***Selection of Sections*** dialog box select all TRON profiles which thikness is bigger than 3 mm and TRON 114x10<br>**OK** | Opens the ***Selection of Sections*** dialog box. Using this option user can select sections that will be applied during verification and design of steel and aluminum elements. |
| **Save** | Saves the parameters of the second member group. |
| LMC the **New** button on the *Groups* tab in the ***Definitions*** dialog box | Allows defining a third member group. |

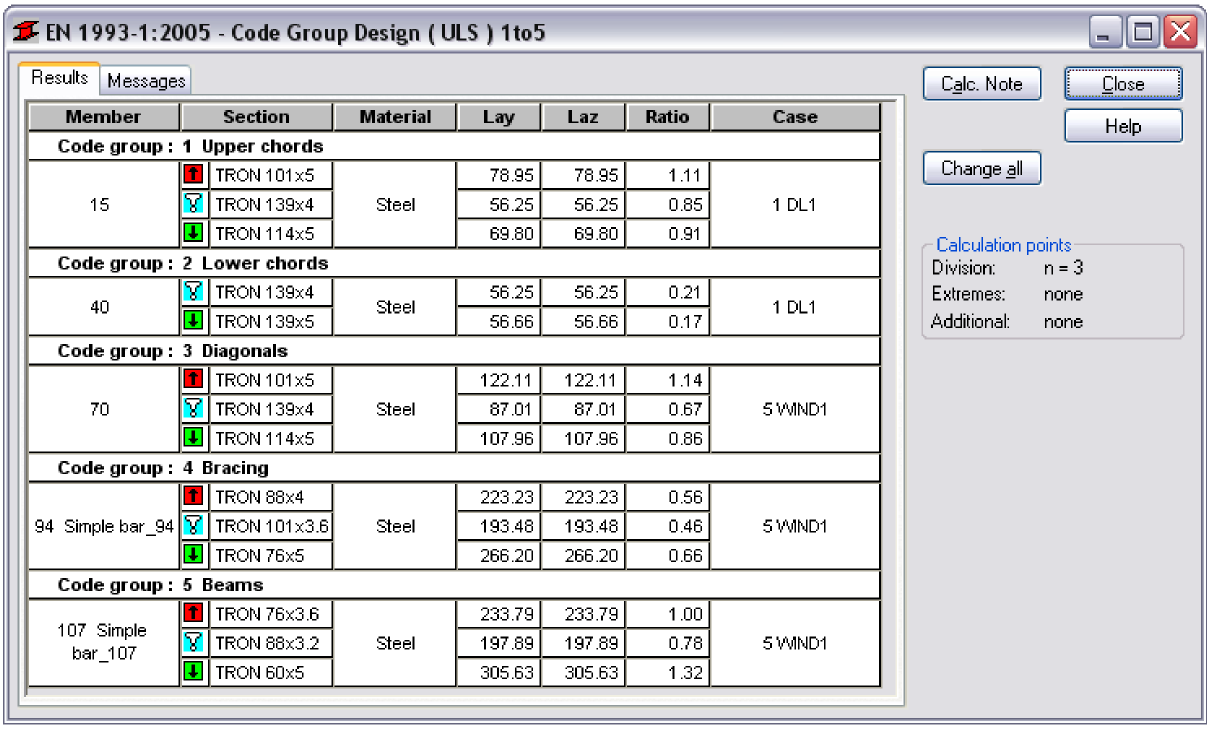
| | |
|---|---|
| Define the third group with the following parameters:<br>Number: 3<br>Name: Diagonals<br>Member list: LMC on the ***View*** edit viewer; select all the diagonals belonging to the trusses while the CTRL key is pressed<br>Material: STEEL EC3 Steel S235 | Defines the third group consisting of all diagonals belonging to trusses in the structure. |
| **Section**<br>In the ***Selection of Sections*** dialog box select all TRON profiles which thikness is bigger than 3 mm<br>**OK** | Opens the ***Selection of Sections*** dialog box. Using this option user can select sections that will be applied during verification and design of steel and aluminum elements. |
| **Save** | Saves the parameters of the third member group. |
| LMC the **New** button on the *Groups* tab in the ***Definitions*** dialog box | Allows defining a fourth member group. |
| Define the fourth group with the following parameters:<br>Number: 4<br>Name: Bracing<br>Member list: LMC on the ***View*** edit viewer; select all the bracings in the structure while the CTRL key is pressed<br>Material: STEEL EC3 Steel S235 | Defines the third group consisting of all bracings in the structure. |
| **Section**<br>In the ***Selection of Sections*** dialog box select all TRON profiles which thikness is bigger than 3 mm and<br>**OK** | Opens the ***Selection of Sections*** dialog box. Using this option user can select sections that will be applied during verification and design of steel and aluminum elements. |
| **Save** | Saves the parameters of the fourth member group. |
| LMC the **New** button on the *Groups* tab in the ***Definitions*** dialog box | Allows defining a fifth member group. |
| Define the fifth group with the following parameters:<br>Number: 5<br>Name: Beams<br>Member list: LMC on the ***View*** edit viewer; select all cross beams while the CTRL key is pressed<br>Material: STEEL EC3 Steel S235 | Defines the fifth group consisting of all cross beams in the structure. |
| **Section**<br>In the ***Selection of Sections*** dialog box select all TRON profiles which thikness is bigger than 3 mm<br>**OK** | Opens the ***Selection of Sections*** dialog box. Using this option user can select sections that will be applied during verification and design of steel and aluminum elements. |
| **Save** | Saves the parameters of the fifth member group. |

| | |
|---|---|
| In the ***Calculations – EN 1993-1-8:2005*** dialog box switch on the *Code Group Design* option | Activates design in groups. |
| LMC on the **List** button in the *Code group design* line in the ***Calculations*** dialog box | Opens the ***Code Group Selection*** dialog box. |
| Press the **All** button located in the upper part of the ***Code Group Selection*** dialog box.<br>In the field below the **All** button the list: 1to5 will appear<br>**Close** | Selects member groups to be designed, closes the ***Code Group Selection*** dialog box. |
| In the ***Calculations – EN 1993-1-8:2005*** dialog box switch on the *Optimization* option | The option allows determining parameters of calculations performed for member groups taking the optimization options into account. |
| LMC the **Options** button | Opens the ***Optimizations Options*** dialog box. |
| In the ***Optimization Options*** dialog box switch on the *Weight* option | Activation of this option will result in searching for the lightest section in the group of sections that meet the code-defined criteria. |
| **OK** | Closes the ***Optimization Options*** dialog box. |
| LMC on the **List** button in **Loads** group in ***Calculations*** dialog box | Opens the ***Load Case Selection*** dialog box. |
| LMC the **All** button (in the field above the **Previous** button), the list: 1to8 13to16 will appear there, **Close** | Selects all load cases. |
| Activate the *Ultimate* option in the *Limit State* field<br>Switch off the *Save calculation results* option in the *Calculation archive* field | |
| LMC the **Calculations** button | Starts design of the selected member groups; the ***Code Group Design*** dialog box appears on the screen (see the drawing below). |



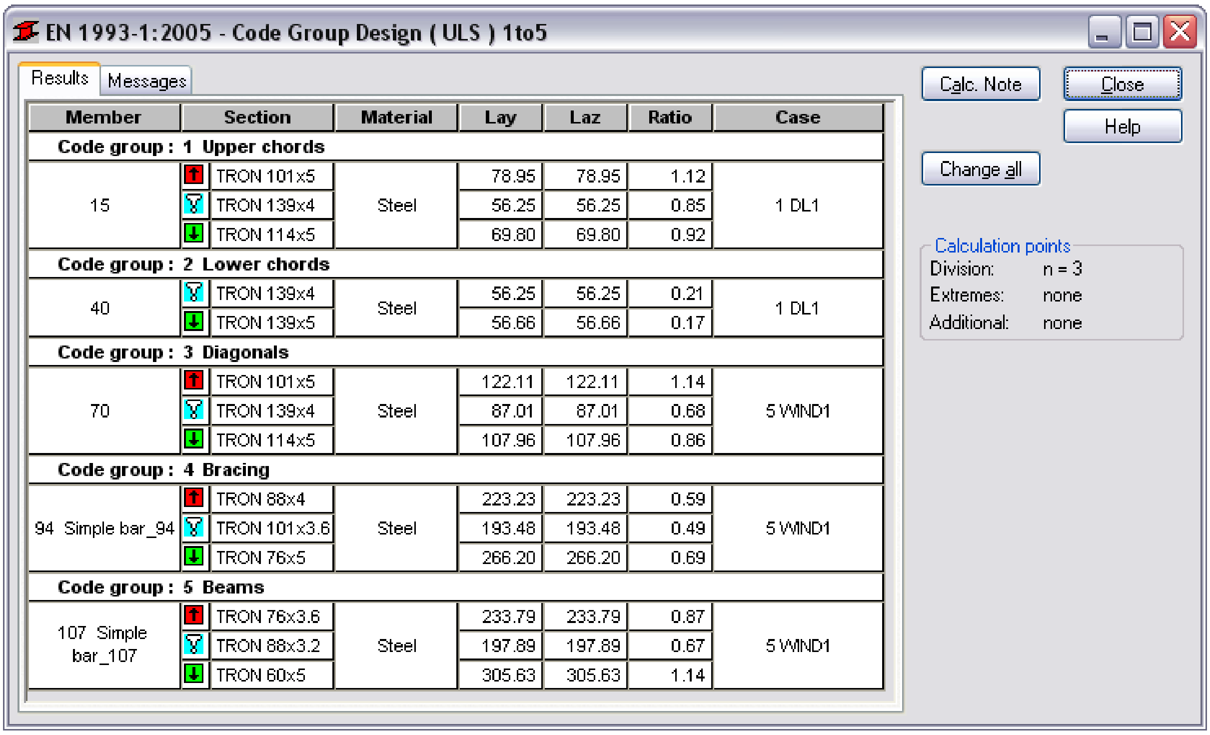

| Member | Section | Material | Lay | Laz | Ratio | Case |
|---|---|---|---|---|---|---|
| Code group : 1 Upper chords | | | | | | |
| 15 | TRON 101x5 | Steel | 78.95 | 78.95 | 1.26 | 1 DL1 |
| | TRON 139x4 | | 56.25 | 56.25 | 0.94 | |
| | TRON 114x5 | | 69.80 | 69.80 | 1.02 | |
| Code group : 2 Lower chords | | | | | | |
| 40 | TRON 139x4 | Steel | 56.25 | 56.25 | 0.71 | 1 DL1 |
| | TRON 139x5 | | 56.66 | 56.66 | 0.57 | |
| Code group : 3 Diagonals | | | | | | |
| 21 | TRON 101x5 | Steel | 122.11 | 122.11 | 1.17 | 1 DL1 |
| | TRON 139x4 | | 87.01 | 87.01 | 0.67 | |
| | TRON 114x5 | | 107.96 | 107.96 | 0.85 | |
| Code group : 4 Bracing | | | | | | |
| 94 Simple bar_94 | TRON 88x4 | Steel | 223.23 | 223.23 | 0.65 | 5 WIND1 |
| | TRON 101x3.6 | | 193.48 | 193.48 | 0.54 | |
| | TRON 76x5 | | 266.20 | 266.20 | 0.76 | |
| Code group : 5 Beams | | | | | | |
| 107 Simple bar_107 | TRON 76x4 | Steel | 235.01 | 235.01 | 1.48 | 5 WIND1 |
| | TRON 88x4 | | 199.67 | 199.67 | 0.98 | |
| | TRON 101x3.6 | | 173.05 | 173.05 | 0.80 | |

| | |
|---|---|
| LMC the **Change All** button in the ***EN 1993-1-8:2005 - Code Group Design*** dialog box shown above | Changes the currently used sections of the members belonging to all member groups to the calculated sections:<br>- for Upper Chords from TRON 219x6.3 to TRON 139x4,<br>- for Lower Chords from TRON 219x6.3 to TRON 139x4,<br>- for Diagonals from TRON 114x6.3 to TRON 139x4,<br>- for Bracings from TRON 114x6.3 to TRON 101x3.6<br>- for Beams from TRON 114x6.3 to TRON 88x4 |
| **Close** | Closes the ***Code Group Design*** dialog box. |
|  Select the *Calculations* icon from the Standard toolbar. | Recalculates the structure with the changed member sections |
| LMC the **Calculations** button in the ***Calculations*** dialog box | Starts design of the selected member groups; the ***Short Results*** dialog box appears on the screen (see the drawing below). |




| Member | Section | Material | Lay | Laz | Ratio | Case |
|---|---|---|---|---|---|---|
| Code group : 1 Upper chords | | | | | | |
| 15 | TRON 101x5 | Steel | 78.95 | 78.95 | 1.11 | 1 DL1 |
| | TRON 139x4 | | 56.25 | 56.25 | 0.85 | |
| | TRON 114x5 | | 69.80 | 69.80 | 0.91 | |
| Code group : 2 Lower chords | | | | | | |
| 40 | TRON 139x4 | Steel | 56.25 | 56.25 | 0.21 | 1 DL1 |
| | TRON 139x5 | | 56.66 | 56.66 | 0.17 | |
| Code group : 3 Diagonals | | | | | | |
| 70 | TRON 101x5 | Steel | 122.11 | 122.11 | 1.14 | 5 WIND1 |
| | TRON 139x4 | | 87.01 | 87.01 | 0.67 | |
| | TRON 114x5 | | 107.96 | 107.96 | 0.86 | |
| Code group : 4 Bracing | | | | | | |
| 94 Simple bar_94 | TRON 88x4 | Steel | 223.23 | 223.23 | 0.56 | 5 WIND1 |
| | TRON 101x3.6 | | 193.48 | 193.48 | 0.46 | |
| | TRON 76x5 | | 266.20 | 266.20 | 0.66 | |
| Code group : 5 Beams | | | | | | |
| 107 Simple bar_107 | TRON 76x3.6 | Steel | 233.79 | 233.79 | 1.00 | 5 WIND1 |
| | TRON 88x3.2 | | 197.89 | 197.89 | 0.78 | |
| | TRON 60x5 | | 305.63 | 305.63 | 1.32 | |

| | |
|---|---|
| LMC the **Change All** button in the ***LRFD:2000 - Code Group Design*** dialog box shown above | Changes the currently used sections of the members belonging to all member groups to the calculated sections:<br>- for Upper Chords - without changes,<br>- for Lower Chords - without changes,<br>- for Diagonals - without changes,<br>- for Bracings - without changes,<br>- for Beams from TRON 88x4 to TRON 88x3.2 |
| **Close** | Closes the ***Code Group Design*** dialog box. |
| Select the *Calculations* icon from the Standard toolbar. | Recalculates the structure with the changed member sections |
| LMC the **Calculations** button in the ***Calculations*** dialog box | Starts design of the selected member groups; the ***Code Group Design*** dialog box appears on the screen (see the drawing below). |



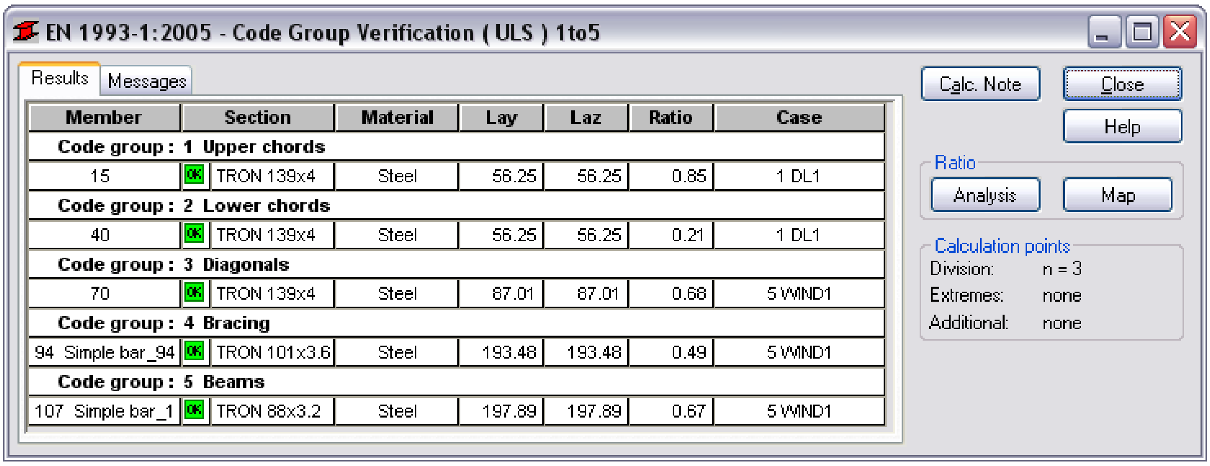


| Member | Section | Material | Lay | Laz | Ratio | Case |
|---|---|---|---|---|---|---|
| Code group : 1 Upper chords | | | | | | |
| 15 | TRON 101x5 | Steel | 78.95 | 78.95 | 1.12 | 1 DL1 |
| | TRON 139x4 | | 56.25 | 56.25 | 0.85 | |
| | TRON 114x5 | | 69.80 | 69.80 | 0.92 | |
| Code group : 2 Lower chords | | | | | | |
| 40 | TRON 139x4 | Steel | 56.25 | 56.25 | 0.21 | 1 DL1 |
| | TRON 139x5 | | 56.66 | 56.66 | 0.17 | |
| Code group : 3 Diagonals | | | | | | |
| 70 | TRON 101x5 | Steel | 122.11 | 122.11 | 1.14 | 5 WIND1 |
| | TRON 139x4 | | 87.01 | 87.01 | 0.68 | |
| | TRON 114x5 | | 107.96 | 107.96 | 0.86 | |
| Code group : 4 Bracing | | | | | | |
| 94 Simple bar_94 | TRON 88x4 | Steel | 223.23 | 223.23 | 0.59 | 5 WIND1 |
| | TRON 101x3.6 | | 193.48 | 193.48 | 0.49 | |
| | TRON 76x5 | | 266.20 | 266.20 | 0.69 | |
| Code group : 5 Beams | | | | | | |
| 107 Simple bar_107 | TRON 76x3.6 | Steel | 233.79 | 233.79 | 0.87 | 5 WIND1 |
| | TRON 88x3.2 | | 197.89 | 197.89 | 0.67 | |
| | TRON 60x5 | | 305.63 | 305.63 | 1.14 | |

| | |
|---|---|
| **Close** | Closes the ***Code Group Design*** dialog box. |

*Member Verification*

| | |
|---|---|
| In the ***Calculations - EN 1993-1-8:2005*** dialog box switch on the *Code Group Verification* option | Activates the code group verification option. |
| LMC in the **List** button in the *Code Group Verification* option | Opens the ***Code Group Selection*** dialog box. |
| Press the **All** button located in the upper part of the ***Code Group Selection*** dialog box.<br>In the field below the **All** button the list: 1to5 will appear, **Close** | Selects member groups to be designed, closes the ***Code Group Selection*** dialog box. |
| LMC on the **List** button in **Loads** group in ***Calculations*** dialog box | Opens the ***Load Case Selection*** dialog box. |
| LMC the **All** button (in the field above the **Previous** button), the list: 1to8 13to16 will appear, **Close** | Selects all load cases. |
| LMC the **Calculations** button in the ***Calculations*** dialog box | Starts code group verification of selected member groups; the ***Code Group Design*** dialog box appears on the screen (see the drawing below). |

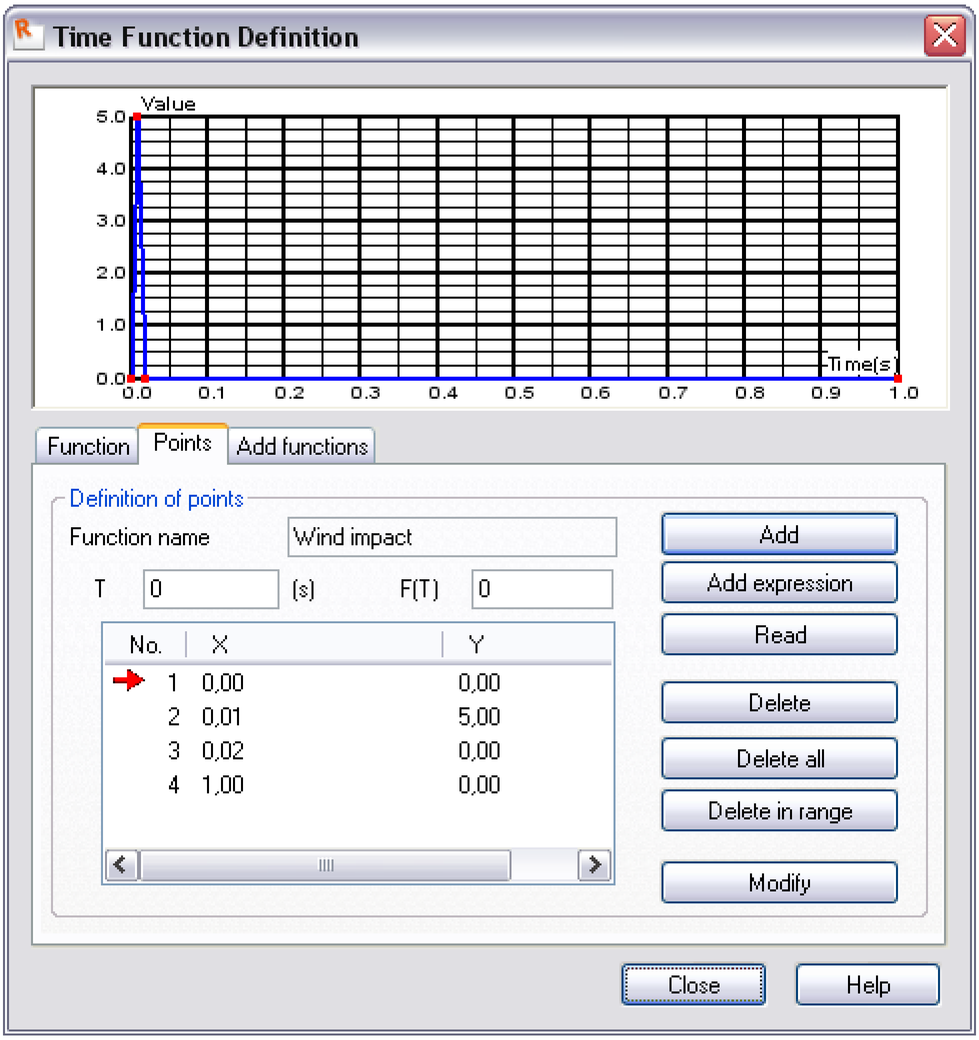



| | |
|---|---|
| **Close** | Closes the ***Code Group Verification*** dialog box. |

## 11.4 Time History Analysis

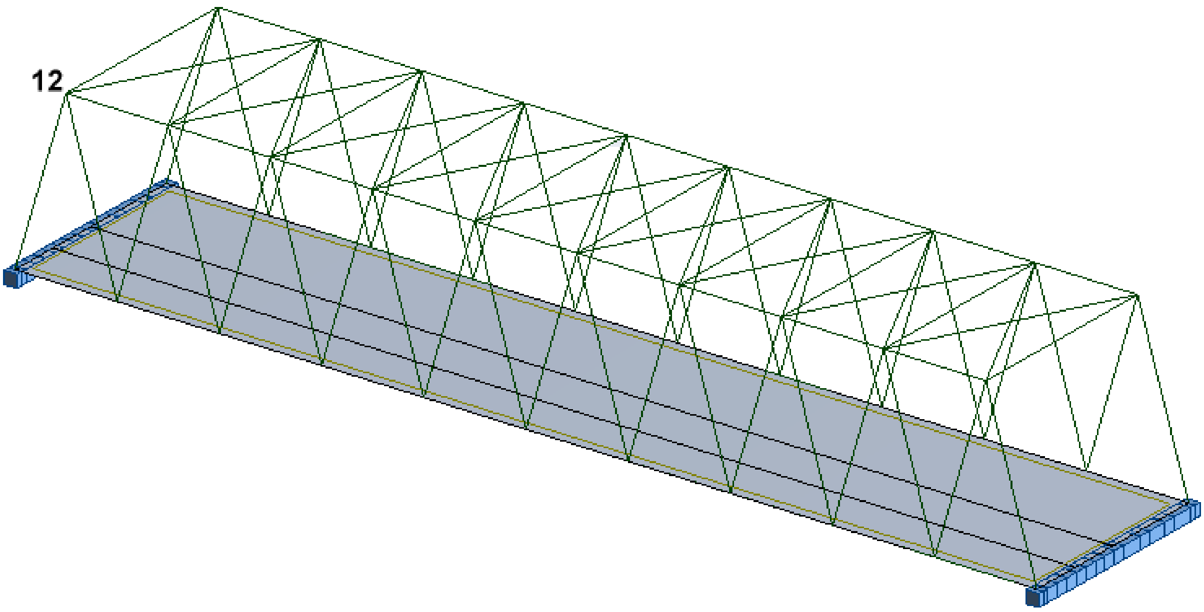
| | |
|---|---|
|  LMC on the field to select the **Structure Model/Geometry** Layout | Selects the initial ***RSAP*** layout. |
| *Analysis / Analysis Types* | Opens the ***Analysis Type*** dialog box which allows defining a new load case (modal analysis, spectral analysis, seismic analysis, etc.), changing the load case type and introducing changes to the parameters of the selected load case. |
| LMC in the **New** button | Opens the ***New Case Definition*** dialog box which allows defining new dynamic cases within the structure. |
| LMC in the **OK** button | Opens the ***Modal Analysis Parameters*** dialog box which allows defining modal analysis parameters for the new dynamic cases in the structure |
| Leave parameters as default. **OK** | Closes the ***Modal Analysis Parameters*** dialog box and adds a new load case to the list of available load cases. |
| LMC in the **New** button | Opens the ***New Case Definition*** dialog box which allows defining new dynamic cases within the structure. |
| Select the *Time history* option, **OK** | Opens the ***Time History Analysis*** dialog box which is used to define time history analysis parameters for a new dynamic load case defined for the structure |
| LMC in the **Function definition** button | Opens the ***Time Function Definition*** dialog box. |
| In the *Defined function* field enter the function name: *Wind impact*, **Add** | Assigns the name to the time function. The new tabs: *Points* and *Add functions* will appear in the dialog box. |

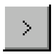
| | |
|---|---|
| On the *Points* tab define consecutive points of the time function:<br>T = 0.00, F(T) = 0.00 **Add**<br>T = 0.01, F(T) = 5.00 **Add**<br>T = 0.02, F(T) = 0.00 **Add**<br>T = 1.00, F(T) = 0.00 **Add**<br>**Close** | Defines the time function, closes the ***Time Function Definition*** dialog box. |

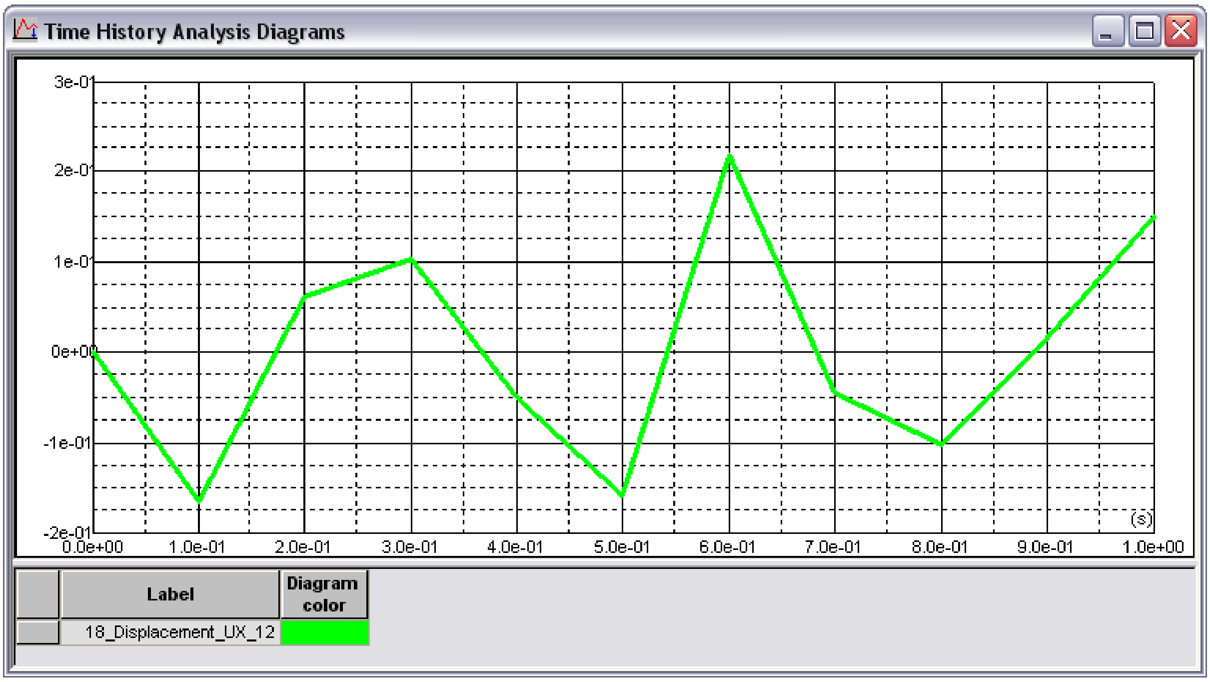



| | |
|---|---|
| In the *Time History Analysis* field select 5th load case: *WIND1* from the available load case list | Selects the number of a selected case. |
| LMC in the buttons **Add**, **OK** | Assigns a static load case which will be used during time analysis, closes the ***Time History Analysis*** dialog box. |
| *Tools / Job Preferences / Structure Analysis* | Opens the ***Job Preferences*** dialog box |
| Select the *DSC Algorithm* option, **OK** | Assumes the DSC algorithm for calculations, closes the ***Job Preferences*** dialog box |
| LMC in the **Calculations** button | Starts calculation of the structure for the defined load cases. Once the calculations are completed, the viewer title bar will show the following information: *Finite Elements Results - available*. |
| **Close** | Closes the ***Analysis Types*** dialog box. |

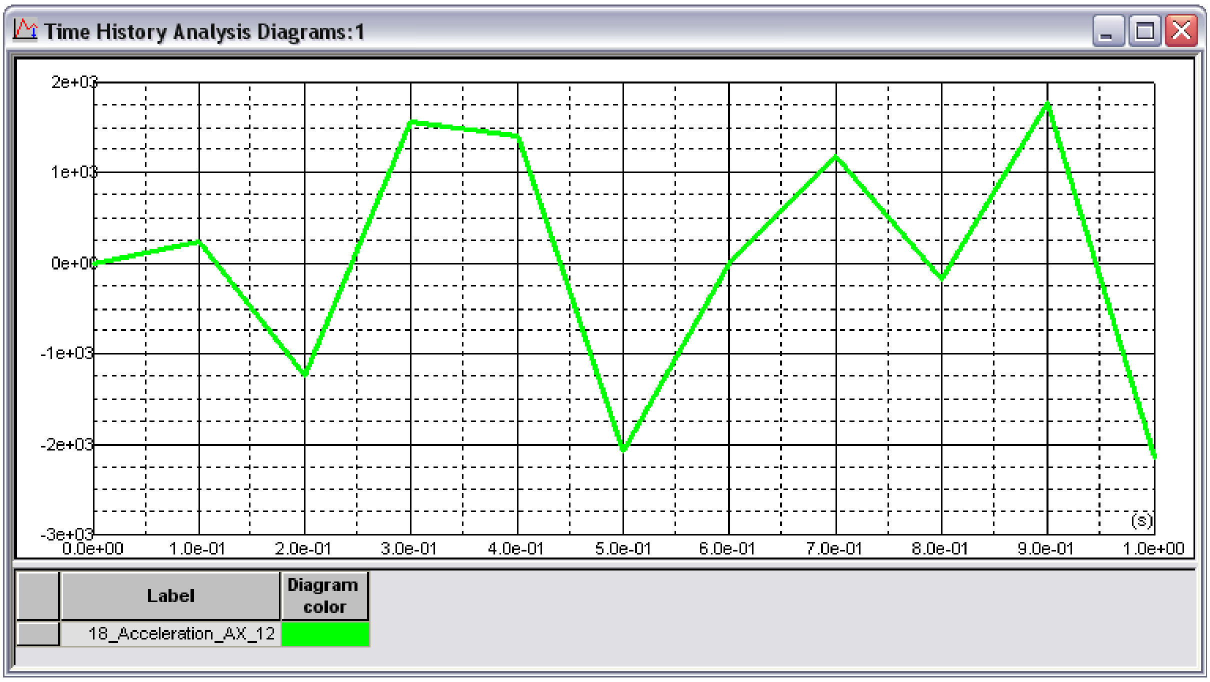
| | |
|---|---|
| Select from the upper menu:<br>*Results / Advanced / Time History Analysis - Diagrams* | Opens the ***Time History Analysis*** dialog box |
| **Add** | Opens the ***Diagram Definition*** dialog box which is used to define a diagram of the quantities calculated during the time history analysis. |
| Select the following option on the *Nodes* tab:<br>*Displacement*, *UX* | Selects displacement in the UX direction |
| In the *Point* field enter the node number:<br>{12} | Selects node no. 12 (see the figure below) for which the diagram will be presented |




| | |
|---|---|
| **Add**, **Close** | In the panel located on the left side of the screen (*Available diagrams*), the defined displacement with the default name: *Displacement_UX_12* appears; closing of the ***Diagram Definition*** dialog box. |
| LMC in the *Displacement_UX_12* (it will be highlighted) and press the  button | Moves the selected diagram to the panel on the right side of the screen (*Presented diagrams*). |
| Switch on the *Open a new window* option and press the **Apply** button | Displays the displacement (*Displacement_UX_12*) diagram on the screen (see the figure below). |




| | |
|---|---|
| Select *Displacement_UX_12* from the panel on the right-hand side of the screen and then, press the < button | Deletes the selected quantity from the panel on the right side of the screen |
| **Add** | Opens the ***Diagram Definition*** dialog box which is used to define diagrams of the quantities calculated during time history analysis. |
| Select the following option on the *Node* tab: *Acceleration*, *UX* | Selects acceleration in the UX direction. |
| In the *Point* field enter the node number: {12} | Selects the node no. 12 (see the picture below) for which the diagram will be prepared |
| **Add**, **Close** | In the panel on the left side of the screen (*Available diagrams*) the defined displacement with default name: *Acceleration_AX_12* appears, closing of the ***Diagram Definition*** dialog box. |
| LMC in the *Acceleration_AX_12* (it will be highlighted) and the press the > button | Moves the selected diagram to the panel on the right side of the screen (*Presented diagrams*). |
| Switch on the *Open a new window* option and press the **Apply** button | Displays the acceleration (*Acceleration_AX_12*) diagram on the screen (see the figure below). |

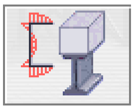

Time History Analysis Diagrams:1
2e+03
1e+03
0e+00
-1e+03
-2e+03
-3e+03
0.0e+00
1.0e-01
2.0e-01
3.0e-01
4.0e-01
5.0e-01
6.0e-01
7.0e-01
8.0e-01
9.0e-01
1.0e+00
(s)
Label
Diagram color
18_Acceleration_AX_12

# 12. Section Definition

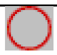
The example presents the definition of solid/thin-walled sections. The results obtained for the sections mentioned are also presented here. The sections are saved to the user's database.
Data units: (m) and (kN).

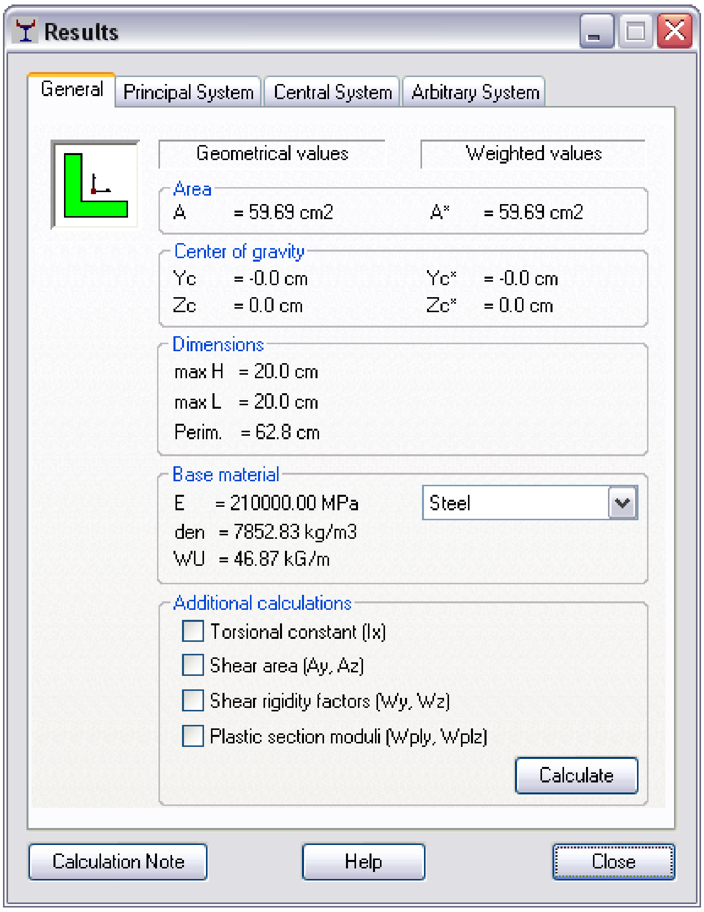
The following rules apply during structure definition:

- any icon symbol means that the relevant icon is pressed with the left mouse button,
- ( x ) stands for selection of the 'x' option in the dialog box or entering the 'x' value,
- LMC and RMC - abbreviations for the **L**eft **M**ouse button **C**lick and the **R**ight **M**ouse button **C**lick.
- **RSAP** - abbreviations for the **Autodesk® Robot™ Structural Analysis Professional**.

To run structure definition start the ***RSAP*** program (press the appropriate icon or select the command from the taskbar). The vignette window will be displayed on the screen and the icon  in the last row (***Section definition***) should be selected.

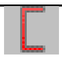
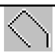
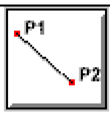
## 12.1 Solid Section

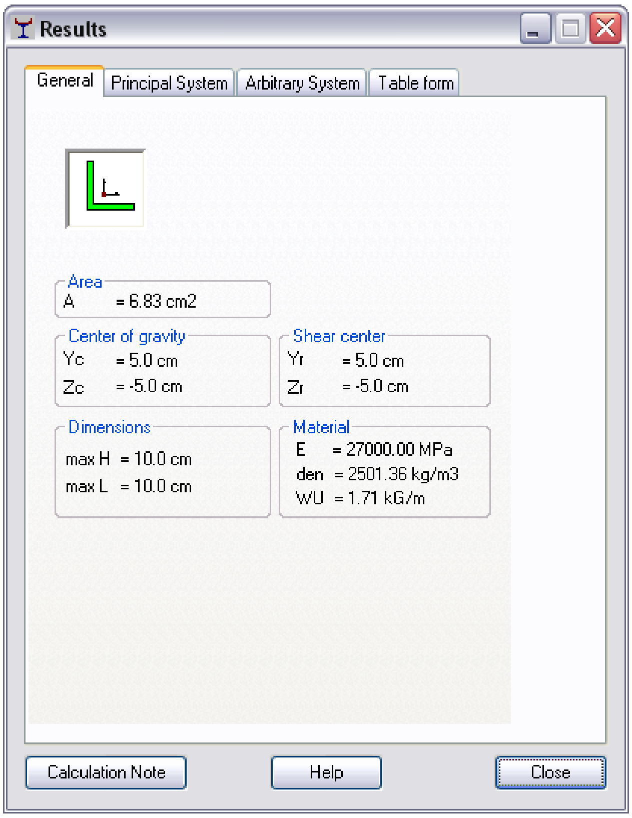
| **PERFORMED OPERATION** | **DESCRIPTION** |
|---|---|
| *File menu / New Section / Solid* | Starts definition of a solid section. |
| *View menu / Grid Step* | Opens the ***Grid step definition*** dialog box. |
| *{Dx}*, *{Dy} = 1,0 (cm)* | Defines a grid step. |
| **Apply**, **Close** | Closes the dialog box. |
|  Select *Circle* icon from the Section Definition toolbar. | Opens the ***Circle*** dialog box. |
| Enter the following points in the *Center* and *Radius* fields:<br>*Center*: (0,0), *Radius*: 10, **Apply** | Defines the external circle. |
| Enter the following points in the *Center* and *Radius* fields:<br>*Center*: (0,0), *Radius*: 9, **Apply** | Defines the internal circle. |
| Select the  in the upper right corner of dialog. | Closes the ***Circle*** dialog box. |
| LMC on the external contour | Selects the external contour. |
| *Contour menu / Properties* | Opens the ***Properties*** dialog box. |
| Steel, **Apply, OK** | Selects the material type and closes the dialog box. |
| *Results menu / Geometric Properties / Results* | Starts calculations of section properties. The dialog box presented below is opened on the screen. |




| | |
|---|---|
| Switch on the *Torsional constant* option, **Calculate** | Starts calculations of the torsional constant. The results are presented on the *Principal* tab. |
| **Calculation Note** | Opens the calculation note with the section data and results. |
| Close the calculation note | |
| LMC on the **Close** button | Closes the ***Results*** dialog box. |
| *File menu / Save to Databases* | Opens the ***Saving section to databases*** dialog box. |
| Enter:<br>*Database*: User<br>*Name*: Circ<br>*Dimension 1*: 20<br>*Dimension 2*: 1<br>*Dimension 3*: 1 | Sets the section properties. |
| *Section Type*: select circle symbol | Selects the section type. |
| Enter: h = 20, t = 1 | Defines section dimensions. |
| **OK** | Saves the section to the database. |

## 12.2 Thin-Walled Section

| | |
|---|---|
|  Select the *New thin-walled section* icon from the Standard toolbar.<br>Select *No* when asked to save the file | Starts definition of the thin-walled section.<br>Note: If asked to save the file, select 'No'. |
|  Select the *Polygon* icon from the Standard toolbar. | Opens the ***Section definition*** dialog box. |
|  | Selects the method of section definition. |
| Enter a thickness value: 0.2 | Defines the thickness of the thin-walled section. |
| Enter the following points:<br>P1 (0.0, 0.0), **Apply**<br>P2 (10.0, 0.0), **Apply**<br>P2 (0.0, -10.0), **Apply**<br>P2 (10.0, -10.0), **Apply** | Defines the characteristic points of a Z-shaped section. |
| Select the  in the upper right corner of dialog. | Closes the ***Section definition*** dialog box. |
| *Results menu / Geometric Properties / Results* | Starts calculations of section properties. The dialog box presented below is opened on the screen. |



| | |
|---|---|
| LMC on the **Close** button | Closes the ***Results*** dialog box. |
| *Results menu / Geometric Properties / Graphical Results* | Opens the ***Diagrams*** dialog box. |
| Turn the *Somega (s)* option on, **Apply**<br>Adjust scale of diagram using **Scale- Scale+** buttons | Selection of section properties for presentation. The diagram shown below will be presented on the Z-shaped section. |
| **Close** | Closes the ***Diagrams*** dialog box. |