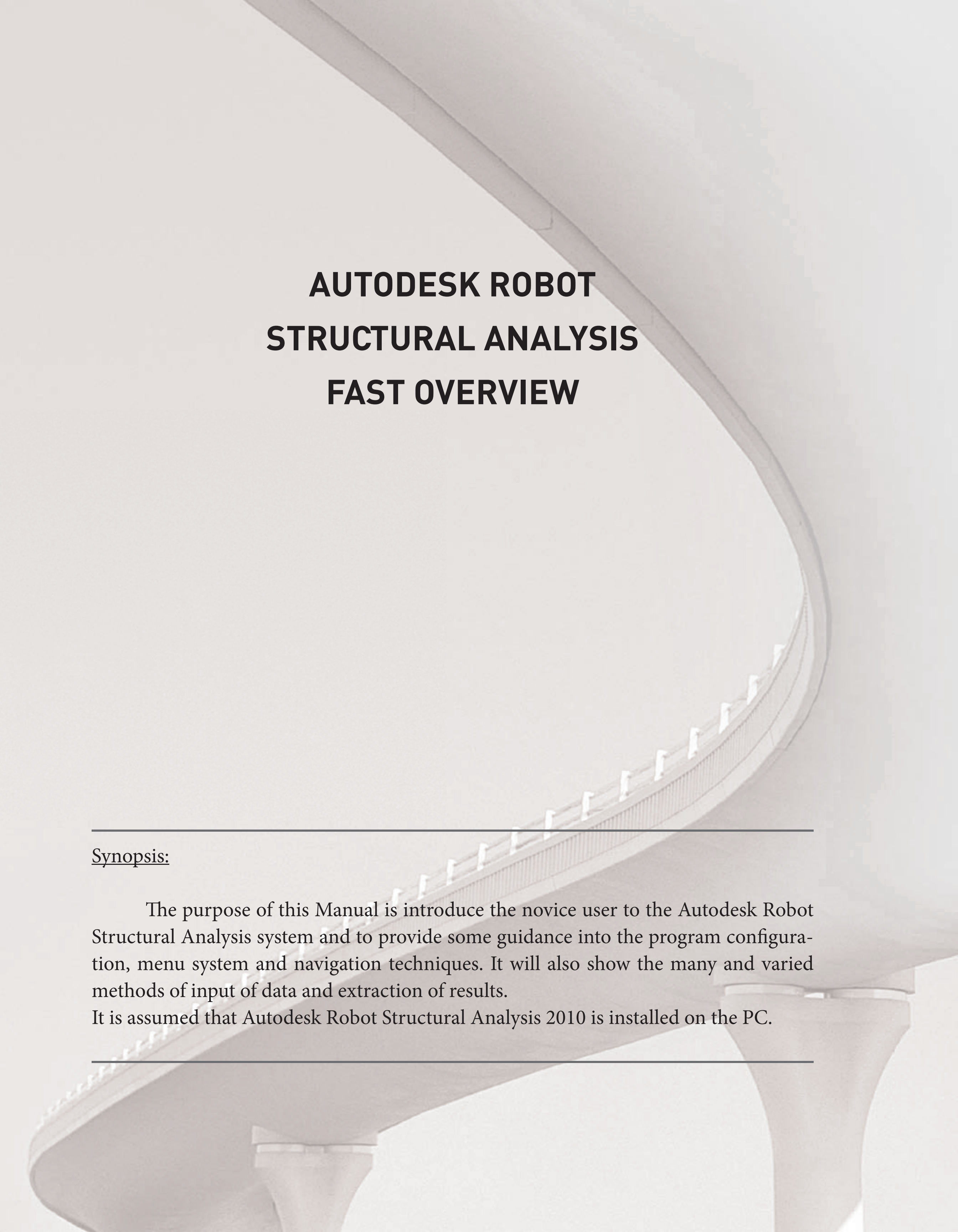
Autodesk Robot Structural Analysis

Metric Getting Started Guide

Autodesk®

This manual text is printed
on 40 percent postconsumer
waste recycled paper

TABLE OF CONTENTS



AUTODESK ROBOT STRUCTURAL ANALYSIS FAST OVERVIEW

Synopsis:

The purpose of this Manual is introduce the novice user to the Autodesk Robot Structural Analysis system and to provide some guidance into the program configuration, menu system and navigation techniques. It will also show the many and varied methods of input of data and extraction of results.
It is assumed that Autodesk Robot Structural Analysis 2010 is installed on the PC.

GENERAL DESCRIPTION OF THE PROGRAM

What is Autodesk Robot Structural Analysis?

Autodesk Robot Structural Analysis (Robot) is a single integrated program used for modeling, analyzing and designing various types of structures. The program allows users to create structures, to carry out structural analysis, to verify obtained results, to perform code check calculations of structural members and to prepare documentation for a calculated and designed structure.

Robot – key features of the Commercial version are shown below:

- Linear, nonlinear and dynamic (modal, spectral, seismic, time history, push over, P-Delta, buckling deformation, plasticity) structure analysis
- working in a multilingual environment (15 languages independently set to user interface, designing and calculation notes)
- working in a multinational environment - designing according to over 50 design codes
- frames, plates and shells, plus a powerful GUI modeler and mesher allows the user to define virtually any shape or configuration of structure – you analyze the true structure geometry
- quality bi-directional integration with Revit® Structure, plus integration through IFC, CIS2 etc.
- an open API to allow the user to interface their own applications for pre and/or post processing

ROBOT MODULES

Robot is a single product with many functions and a common user enviroment.

Once Robot is activated (click on the appropriate icon on the desktop or choose the appropriate command from the taskbar), the window shown below (Figure 2.1) will appear on the screen. The window is used to select the type of structure that will be analyzed or to load an existing structure.

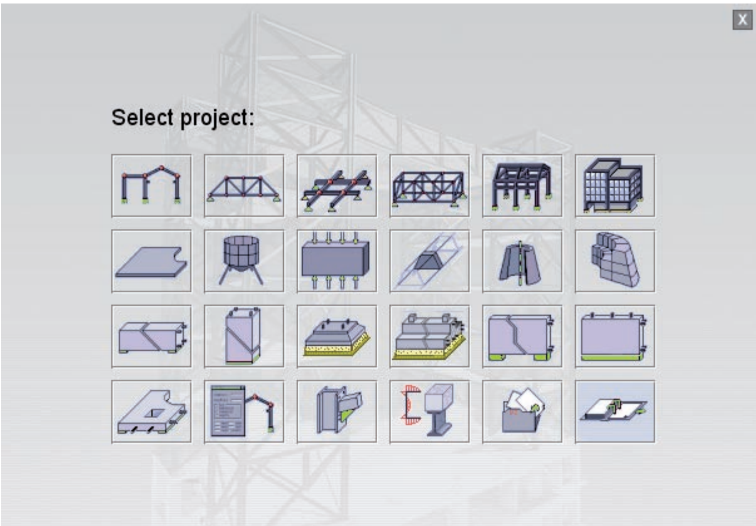


Figure 2.1 - Robot modules window

> **NOTE:**
> When the cursor is positioned on an icon, a short description of its function is displayed.

The most commonly used icons are described below:

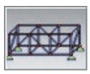
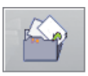
	2D Frame Analysis		2D Truss Analysis
	Grillage Analysis		3D Truss Analysis
	3D Frame Analysis		2D FE Plate Analysis
	Shell Analysis – used to model surfaces of any shape in 3d structures		Bulding Analysis
	Open the previously created project file		Open a new project

Table 2.1 - Robot basic modules

ROBOT SCREEN LAYOUT

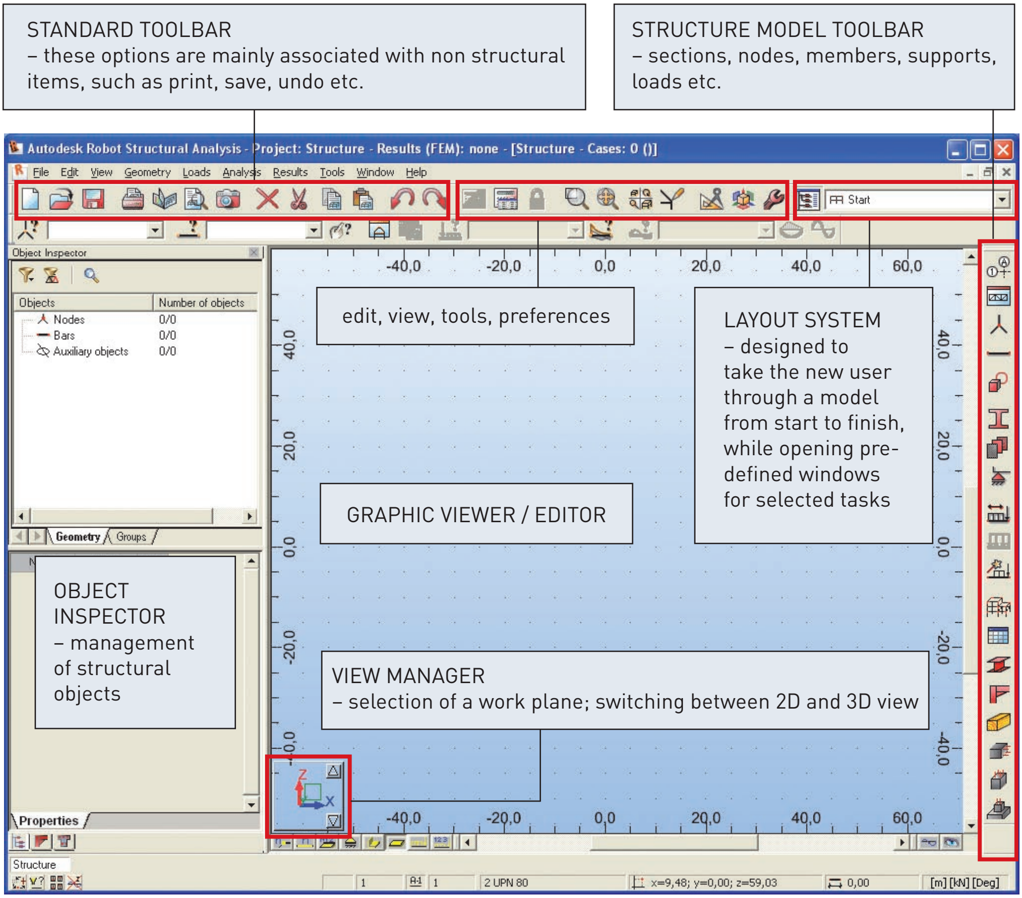


Figure 3.1 - Robot typical screen layout

BASIC CONFIGURATION OF THE PROGRAM

The two options, **Preferences** and **Job Preferences**, allowing the user to set program parameters in the Robot system, are available from the menu by opening toolbar **Tools** and pressing the appropriate icon:

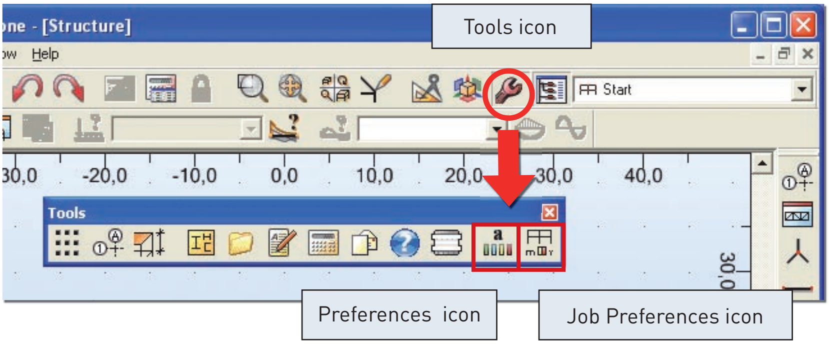


Figure 4.1 - Tools toolbar

Preferences

The **Preferences** dialog box presented below is used to define basic parameters in the program:

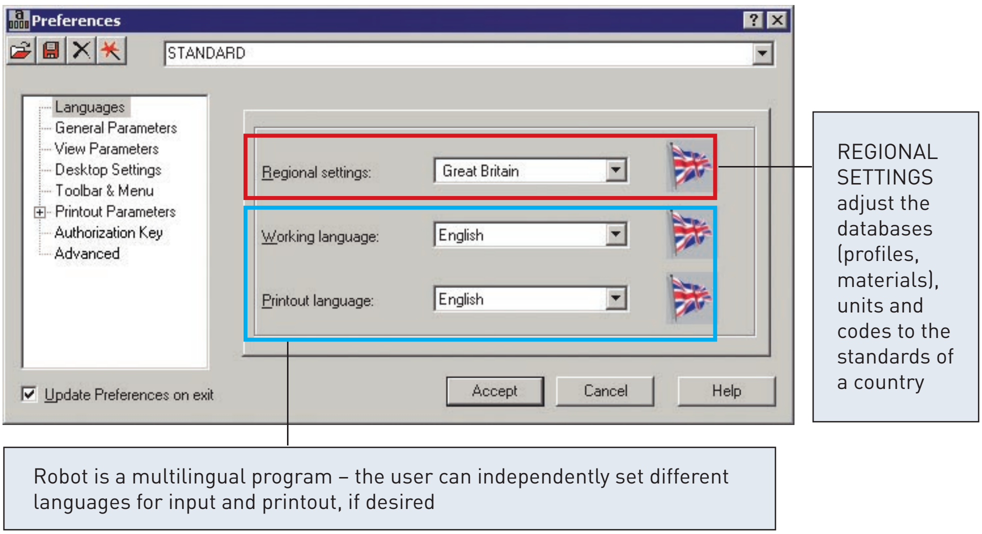


Figure 4.2 - Preferences menu

The most regularly used options are:

- languages - selection of regional settings (definition of the country whose codes materials and regulations - e.g. code combination regulations - will be used during the design process, calculations and structure design) and working and printout language
- general parameters (saving parameters, number of recently used structures, sound on/off etc.)
- display parameters (colors and fonts for screen components)
- toolbar and menu (menu type and the type of toolbars)
- printout parameters (colors and fonts for printouts, scale and symbols, line thickness)
- protection parameters (protection, authorization) - for changing the system protection
- COM interface - presentation of the registered additional programs/modules

Job Preferences

The **Job Preferences** dialog box, presented below, allows you to define general program parameters to be used in a given project:

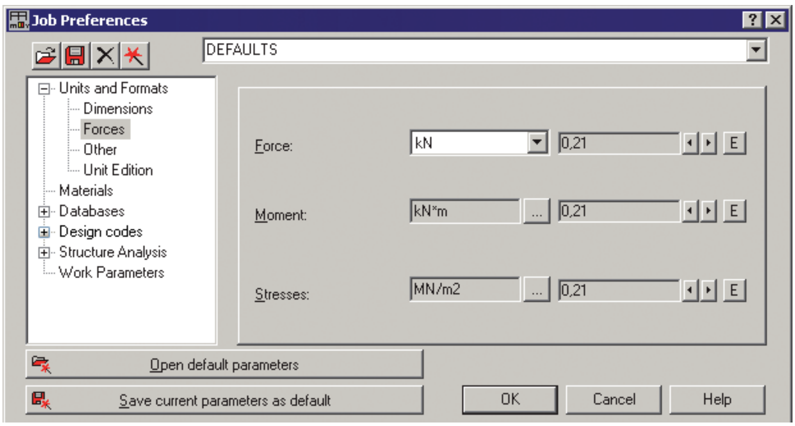


Figure 4.3 - Job Preferences menu

The most important functions are:

- number units and formats (dimensions, forces, possibility of unit edition)
- materials (selection of material set, according to the country and the possibility of creating user-defined material)
- section database (selection of the appropriate database with member sections)
- structure analysis parameters (selection of the static analysis method and definition of basic parameters for dynamic and non-linear analysis; selection of analysis types, possibility of saving results for seismic analysis – combination of seismic cases)
- parameters for generation of surface finite element meshes for plates and shells

NAVIGATION TECHNIQUES

In the Robot software, various mechanisms have been introduced to make structure definition simple and more efficient. According to the type of operation performed, the mouse cursor changes its shape to:

- "hand" – a selection mode for highlighting entities
- "cross pointer" - during node and bar definition, to define precise points e.g. start and end points of members
- shape of the appropriate feature e.g. when adding sections the cursor is in shape of an I section, when adding supports a support icon appears etc.

The cursor operation in a viewer by means of the third mouse button (or wheel) is identical to that in the AutoCAD® program. The following cursor support modes are available:

- wheel rotation – zoom in / out
- wheel rotation + Ctrl key – horizontal pan
- wheel rotation + Shift key – vertical pan
- pressing the third button - pan
- double-click with the third button – initial view

The user should take note of the work capabilities in 3D views when the menu option **Dynamic View** (View → Dynamic View → Dynamic View) is switched on. 3D viewing enables work in one of five modes:

- four simple modes: 3D rotation, 2D rotation, zoom and pan
- one multi-function mode

The user may switch from one work mode to another by selecting an appropriate option in the View / Dynamic View menu, on the View toolbar and in the context menu. After choosing a work mode, the mouse cursor movement (with mouse left button pressed) brings about the relevant change in the 3D view:

- 3D Rotation – rotates a structure in all planes
- 2D Rotation - rotates a structure in the plane parallel to the screen plane
- Zoom – movement down the view – zooming in / zooming out a structure to / from the screen plane
- Pan – movement in the view plane (structure shift with respect to the screen center)

The multi-function mode (Rotation / Zoom / Pan) enables work using all the modes at the same time. The viewer of 3D view is divided into quarters and each of them is ascribed one of the modes:

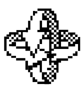
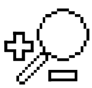
	top left: 3D rotation		top right: pan
	bottom left: zoom		bottom right: 2D rotation

Table 5.1 - Cursor modes

Once the cursor is positioned in the relevant quarter of the screen, the cursor shape changes (see the icons above).

Navigation tool (ViewCube) is also available in the program. ViewCube is a 3D interactive tool that lets you reorient a view and set the current orientation of a structure model. Clicking a predefined face, edge or corner on the ViewCube, you can reorient the view of a model. Moreover, clicking and dragging the ViewCube lets you reorient the model to different directions. Access the ViewCube option by selecting the View > ViewCube - Properties.

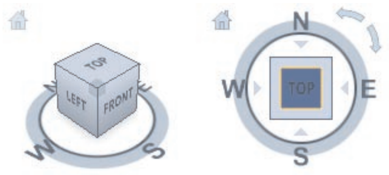


Figure 5.1 - ViewCube

The ViewCube also uses the compass to indicate a direction from which you view a model. To change the viewpoint of a model, click a selected direction on the compass (N, S, E, W). You can show or hide the compass from the ViewCube context menu after you right-click on the ViewCube and select the Show compass option.

METHODS OF WORKING WITH ROBOT INTERFACE

There are two methods to work with Robot - by using System Menu to entering data, or special Layout System.

System menu

The system menu consists of two parts: a text menu and toolbars with appropriate icons. They can be used interchangeably, according to the users' needs and preferences. Both are displayed in the same way - as a horizontal bar at the top of the screen (additionally, for some layouts in the Robot system, another toolbar with most frequently used icons is displayed on the right side of the screen).
Basic options available within the modules are accessible both from the text menu and the toolbar.

Though contents of the text menu and toolbars for successive modules vary, the main options are always available regardless of which module is active.
The figure below illustrates both types of menus (the main menu that appears once the Start layout is selected is shown as an example):

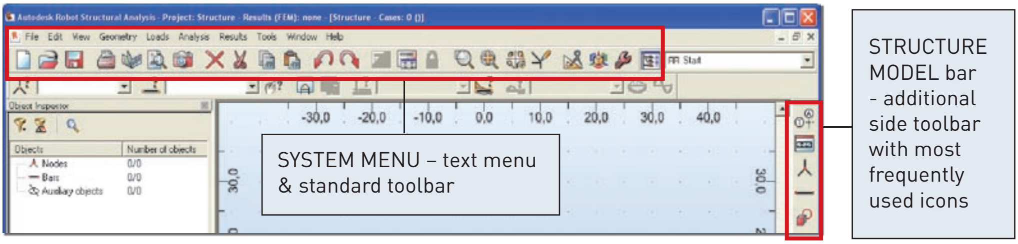


Figure 6.1 - System menu

Options available in the text menu are grouped as follows:

File menu

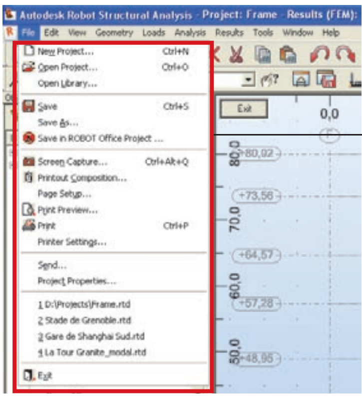


Figure 6.2 - File menu window

FILE MENU – contains options for:
- File management (New, Open, Save etc.)
- Calculation notes and drawing printouts (Screen Capture..., Printout Composition... etc.), a list of information about project and a list of recently open files

Edit menu

EDIT MENU – contains options for:
- Elements edition (Undo, Redo, Cut, Copy etc.)
- Selection (Select...,Select All)
- Selection using filters (Select Special)
- Model modification (Edit, Complex Edit..., Divide, Intersect etc.)

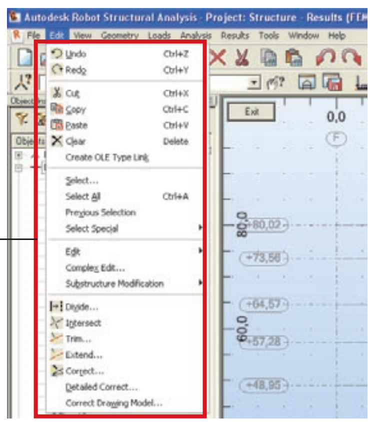
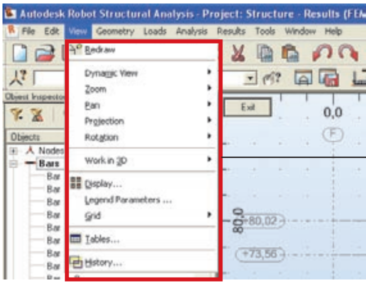


Figure 6.3 - Edit menu window

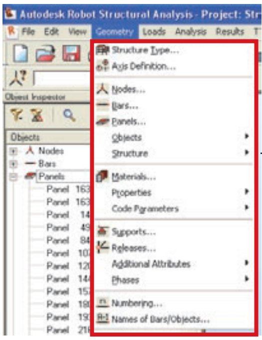
View menu



VIEW MENU – contains options for:
- 2D/3D view management of a structure's model (Dynamic View, Zoom, Pan etc, Work in 3D)
- Structure attributes to be presented on the screen (Display); definition of grid parameters (Grid)
- Selection of tables with data or results (Tables) and saving defined views of a structure (History)

Figure 6.4 - View menu window

Geometry menu



GEOMETRY MENU - contains options to:
- Select or modify a type of a structure
- Define a model construction axis (Axis Definition)
- Define structural data - nodes, bars, panels and auxiliary objects
- Define materials
- Define bar profiles and their orientation and direction of local coordinate system
- Define supports and many other structural items

Figure 6.5 - Geometry menu window

Loads menu

LOADS MENU – contains options to define load cases and combinations

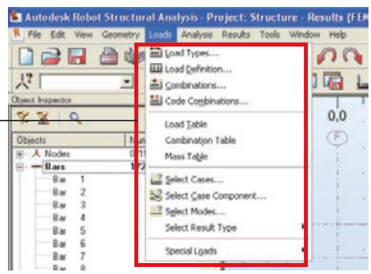
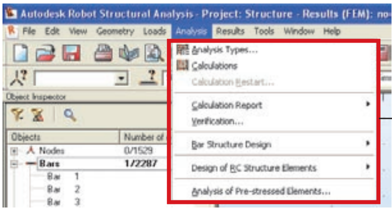


Figure 6.6 - Loads menu window

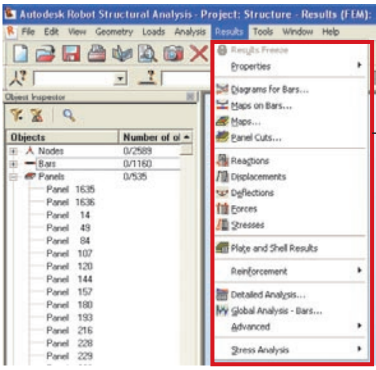
Analysis menu



ANALYSIS MENU – contains options to:
- Start the calculation process
- Change from linear to non linear, P-Delta or buckling
- Set up dynamic analyses
- Design concrete beams, columns and surfaces

Figure 6.7 - Analysis menu window

Results menu



RESULTS MENU – contains options to:
- Display beam results graphically
- Display detailed results for beams graphically
- Display results for surfaces
- Display tables that easily be edited, sorted, filtered, exported to MS Excel® etc.

Figure 6.8 - Results menu window

Tools menu

TOOLS MENU – contains different types of configuration options (Preferences, Job Preferences...) and possibility to define user's interface, menu, shortcuts and form of calculation notes

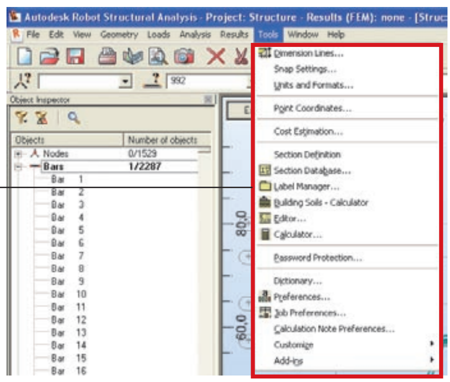
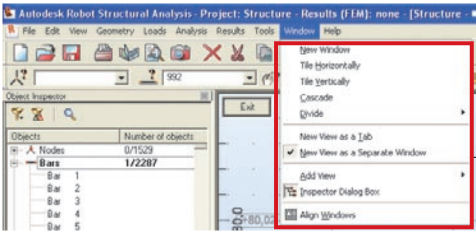


Figure 6.9 - Tools menu window

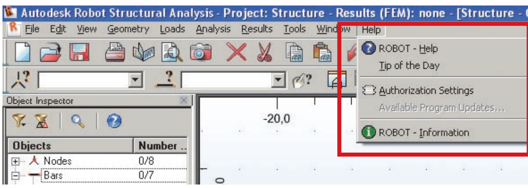
Window menu



WINDOW MENU – offers options to manage and arrange the graphic windows and an option to activate/deactivate the "Object Inspector"

Figure 6.10 - Window menu

Help menu



HELP MENU – contains help options and various product information items

Figure 6.11 - Help menu window

Layout System

The second method of work with Robot is by using the special layout system.

Robot has been equipped with a layout mechanism that simplifies the design process. The layouts in Robot are specially designed systems of dialog boxes, viewers and tables that are used to perform specific defined operations.

Layouts available in Robot were created to make consecutive operations leading to defining, calculating, and designing the structure easier – the layouts guide the user through the process from model generation to results.
In order to make the system as easy to use as possible, each layout has its own predefined set of windows which are automatically opened on entering the layout and closed on exit.
Layouts are available from the selection list found in the right, upper corner of the screen:

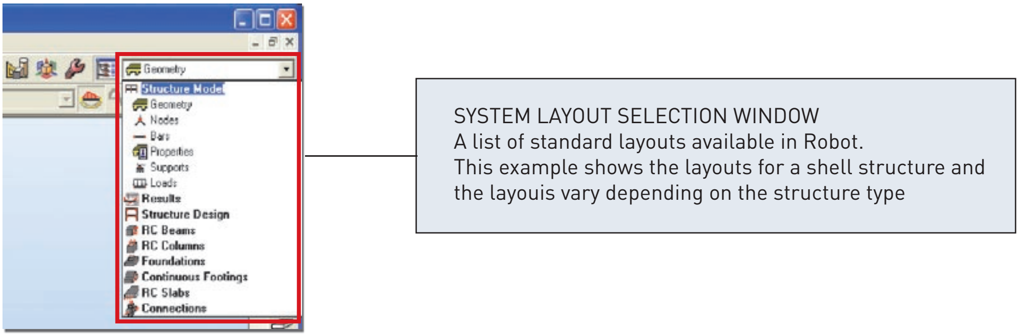


Figure 6.12 - System layout selection window

The layout order and arrangement follows a chronological process, starting from defining nodes, beams, then supports, sections etc.

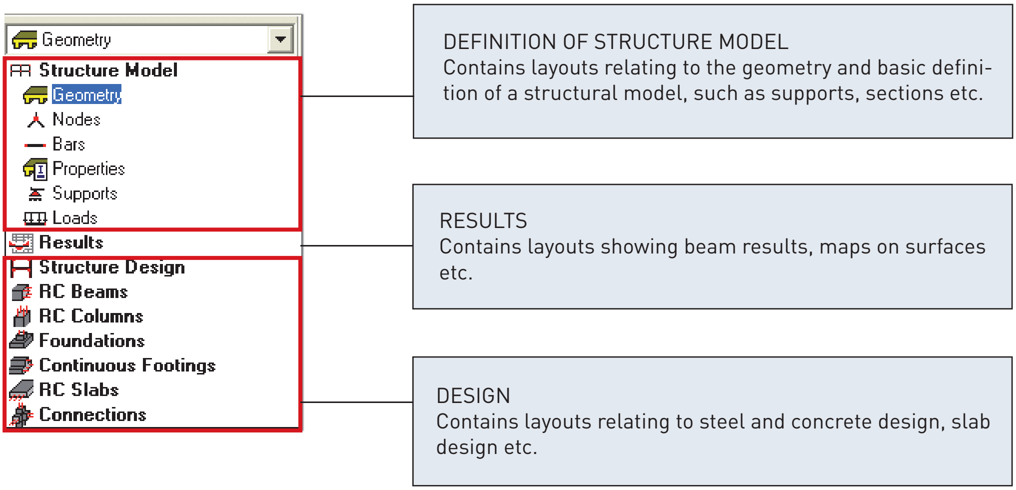


Figure 6.13 - System layout selection window - chronological layout order

A typical layout for nodes is shown – note that each window cannot be closed until a new layout is selected.

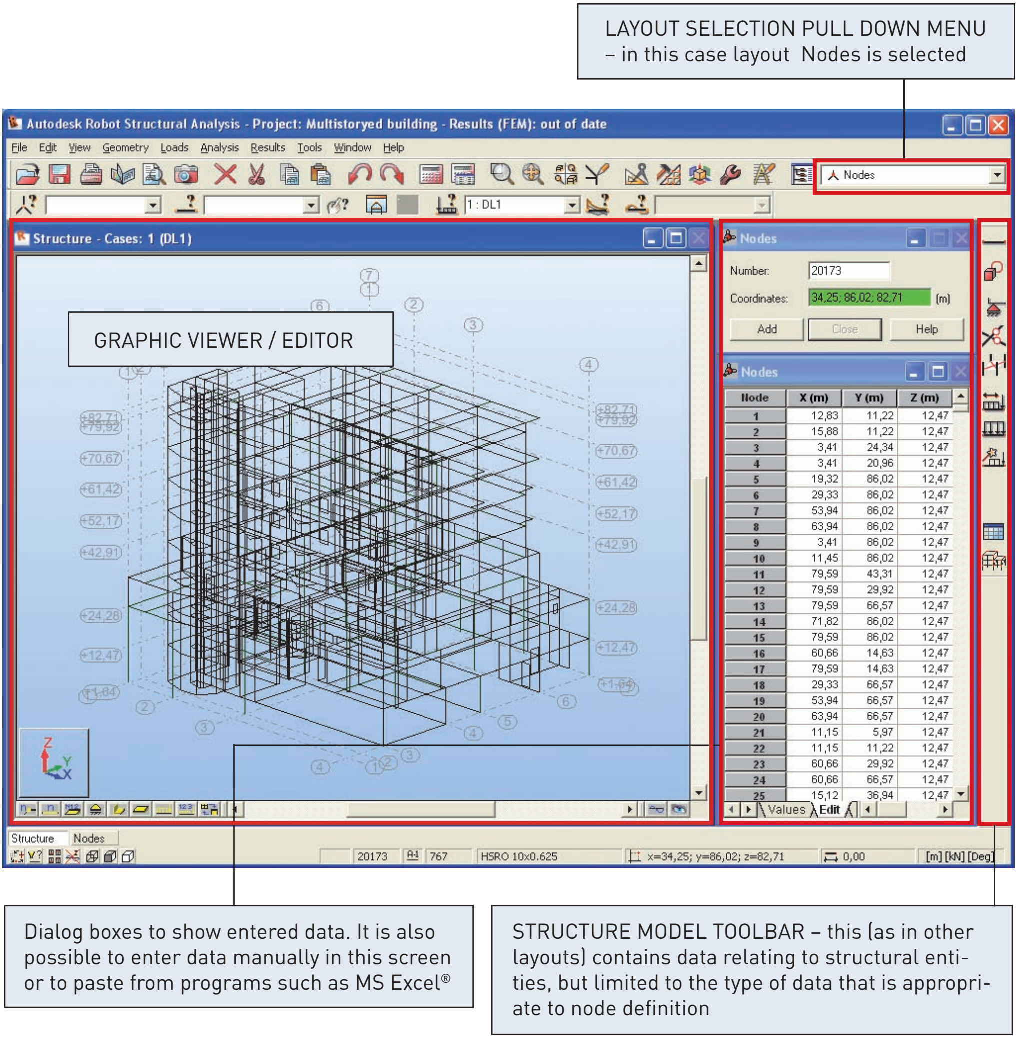


Figure 6.14 - Typical layout for nodes

However it is not necessary to define the structure according to the layout order. This may be done in any order chosen by the user. The layout system was introduced in such a way that Robot structure definition is intuitive and efficient.

All Robot operations may be performed without using the defined layouts but by using system menu instead or also taking advantage of both methods (simultaneously) according to the user's needs and preferences.

ENTERING THE STRUCTURAL ANALYSIS DATA

There are 3 ways to enter data:

1. By entering data using the appropriate text dialog box or direct in a table (or pasted from MS Excel®)

2. By entering data in the Robot GUI – using tools for graphic structure definition - i.e. snap grids or structural axes.

   Examples of data entering using the System Layout mechanism (for Bars and Loads options) are presented below:

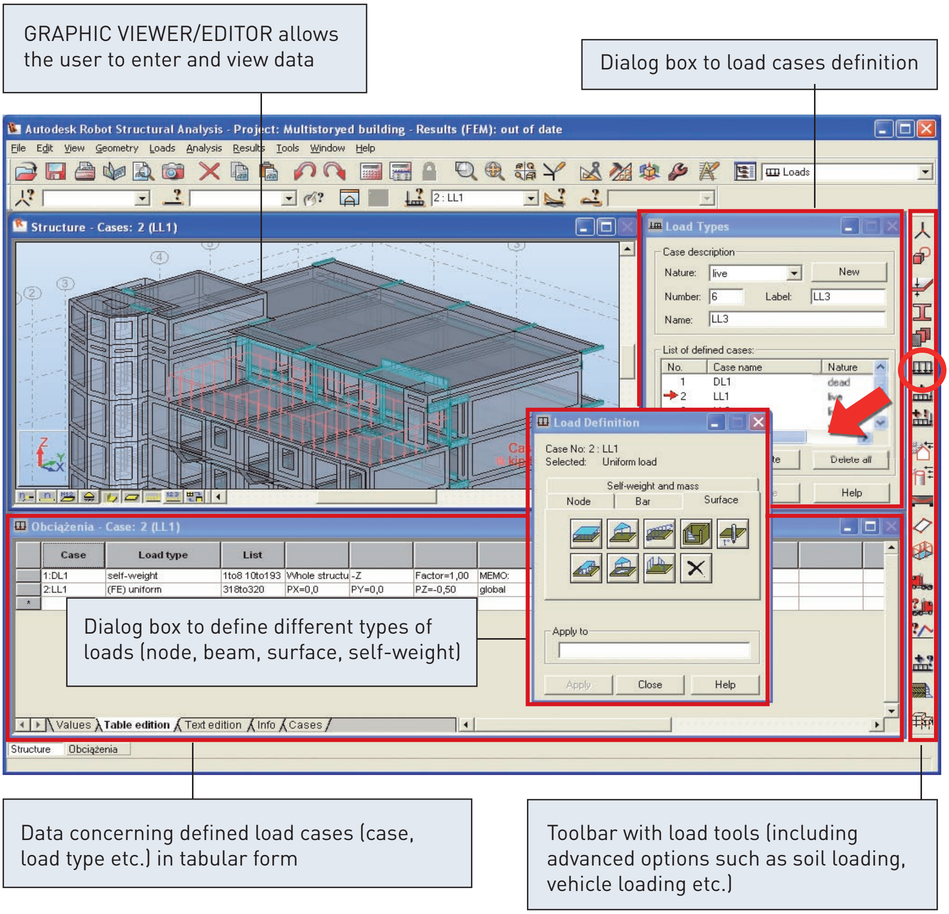


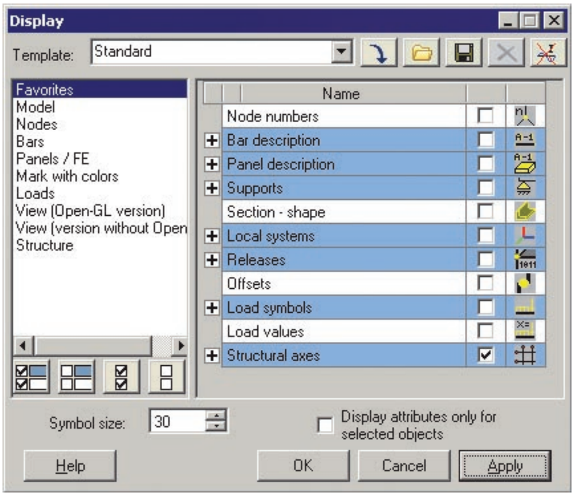
Figure 7.1 - Examples of loads data entering with System Layout mechanism

> **NOTE:**
>
> The updating of data is dynamic – tables reflect graphics and vice versa at all times.

3. By entering data in another application and importing into Robot - several file formats are supported including:

   DXF, DWG, IFC, SDNF (steel detailing neutral file), CIS/2. A dynamic link to and from Revit® Structure also provides bi-directional integration.

   As the preparation of the structural model progresses, the user can control exactly what is seen on the screen by using the **Display** settings (it is available by pressing  icon from context menu or direct from the bottom, right corner on the Robot screen):



DISPLAY allows the user to switch on and off a wide selection of items, including node and members numbers, section shapes, supports, FE mesh and also sets up hidden line and render options.
A wide range of customization options allows to the user to define a view of structure as required exactly to user's preferences.

Figure 7.2 - Display window

After defining the model data, the user now proceeds to the analysis stage. However, prior to this stage, the model must be "discretized" into finite elements. Robot has wide ranging capabilities for "automeshing" the structure and meshing is generally a very fast process for even the largest of models. Some of the "meshing parameters are shown below:

AUTOMESHING PARAMETERS can mesh all or part of the structure and also define mesh density and meshing type

MESH REFINING OPTIONS allow the user to easily refine the mesh in user selected areas.

Figure 7.3 - FE Mesh Generation toolbar

Example of structure with meshed panels is shown below:

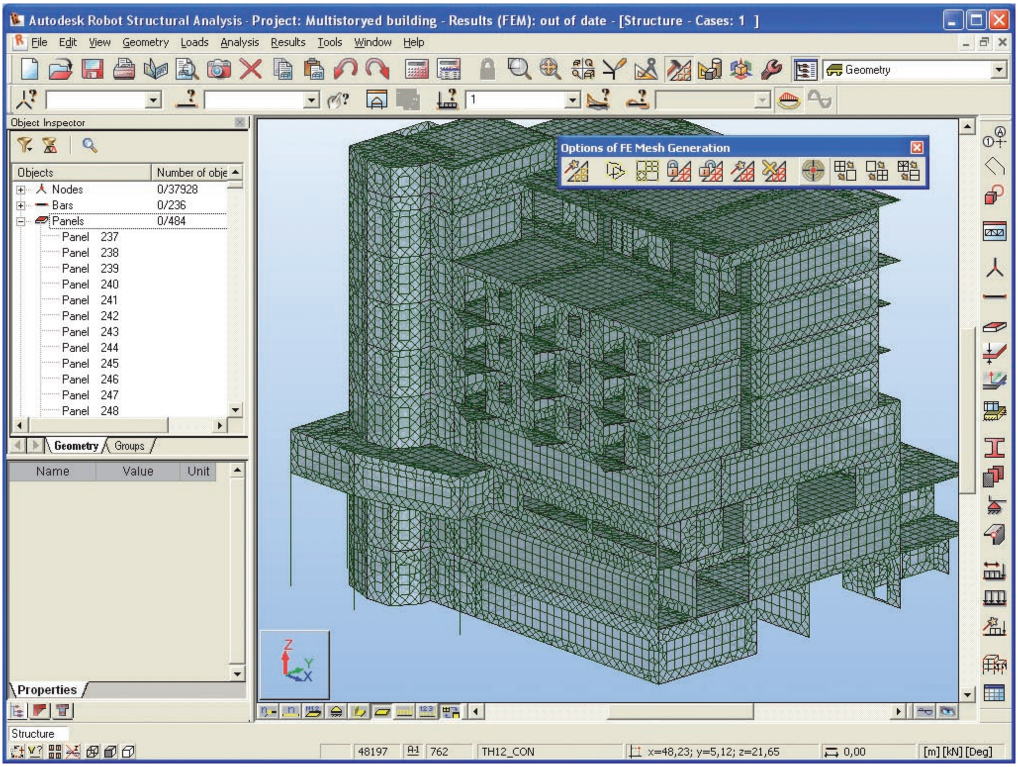


Figure 7.4 - Example of meshing

ANALYZING THE STRUCTURE

Structural analysis can be started by selecting one of the two “calculation” buttons in the horizontal toolbar.

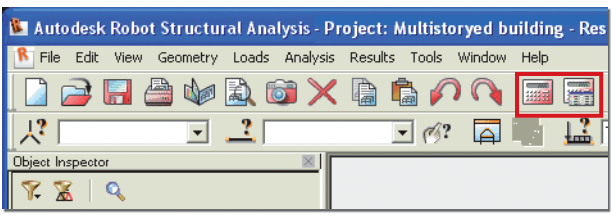


Figure 8.1 - Calculation icons

The icon on the left side starts the calculation process. The second icon is used to set different analysis parameters. This option allows the user to change specific analysis types from linear to non linear or to set up dynamic analysis.

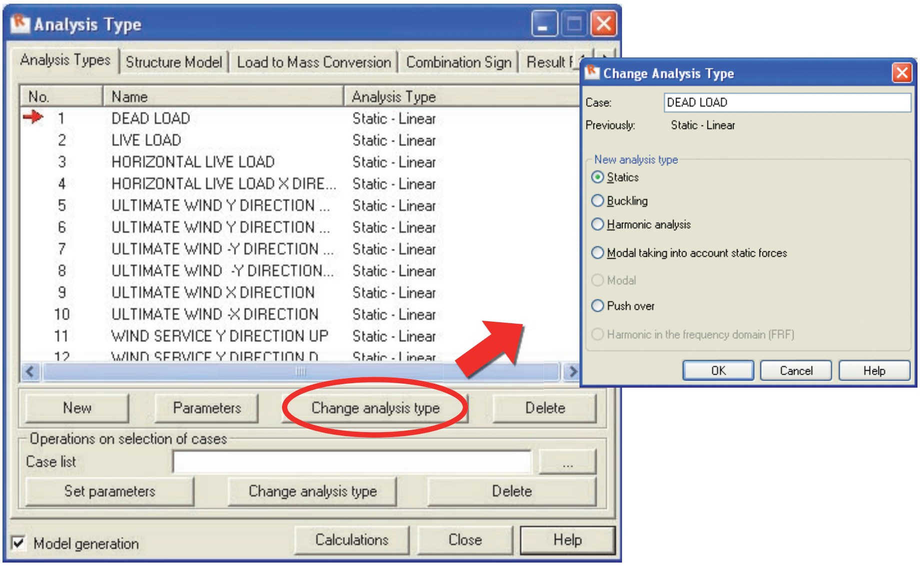


Figure 8.2 - Changing analysis parameters

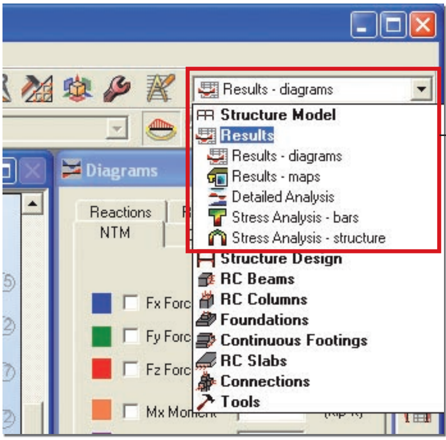
As a default, all analysis is set as "linear static", unless the user has included some members or parameters such as cables, tension only, hinges etc. In such a case the analysis type is automatically changed to "non linear" and Robot will apply the loads incrementally to ensure the true structural equilibrium and a final exact geometry is reached. Robot has many non linear parameters that the user can set in the case of non convergence of analysis data, including the options to set full or modified Newton-Raphson and "step halving". In addition to general non linear calculations, the user can also set the following analysis options:

- P-Delta
- Modal
- Seismic (to codes such as IBC, UBC, EC etc.)
- Spectral response
- Time history
- Buckling

RESULTS PREVIEW

The Results Layout shows a wide range of analysis results presentation.

NOTE:
All of results preview options are also available through the RESULTS pull-down menu.



User can view results of calculations by selecting the results layout, where a set of submenus for particular results then appears

Figure 9.1 - Results type selection from results layout

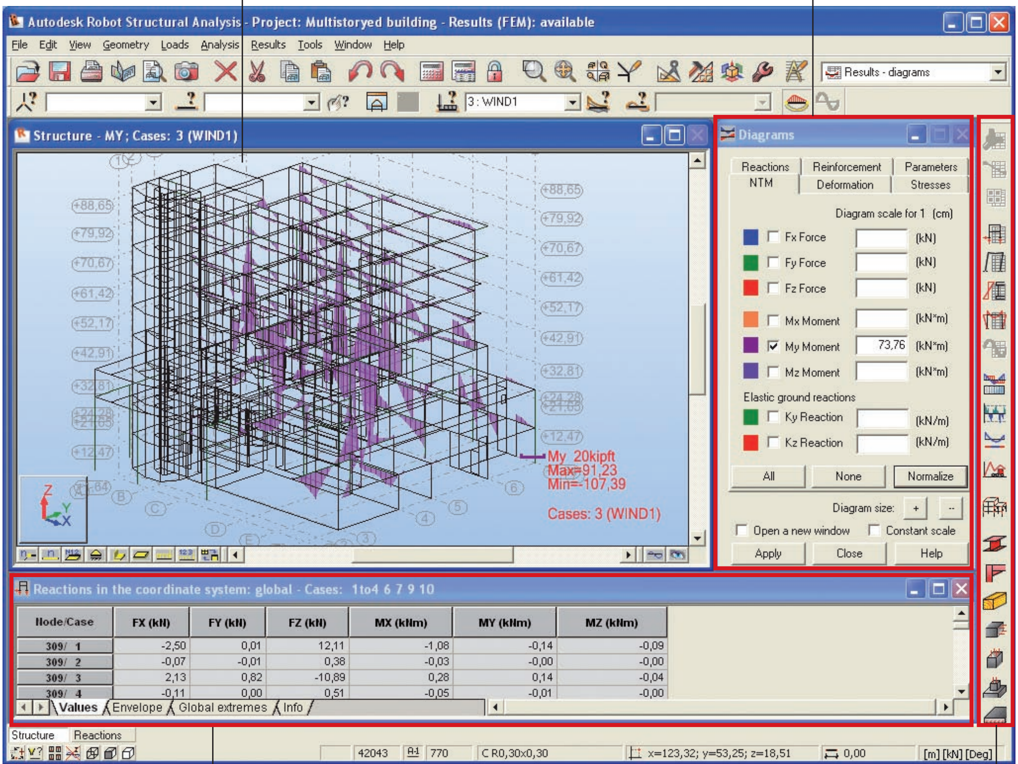
Results can be split into various categories, each having their own characteristics.

Graphical results for beams

(graphical results for individual beams, or selection of beams, or groups of beams presentation - bending moment, shear, stress, deflection animation of results in .avi format etc.)

Graphical presentation of results as diagrams in graphic viewer/editor

Dialog box selects type of result to be displayed



Tables of results – can easily be converted to MS Excel® format by clicking right hand mouse

Side toolbar shows additional results of structural analysis in tabular form, graphical form, plus links to design modules

Figure 9.2 - Results in the diagrams form

There is also easy acces to calculation results for single bars. In context menu of the bar (right mouse click on the element, then select Object Properties option) user can see diagrams and values for selected quantity of internal forces.

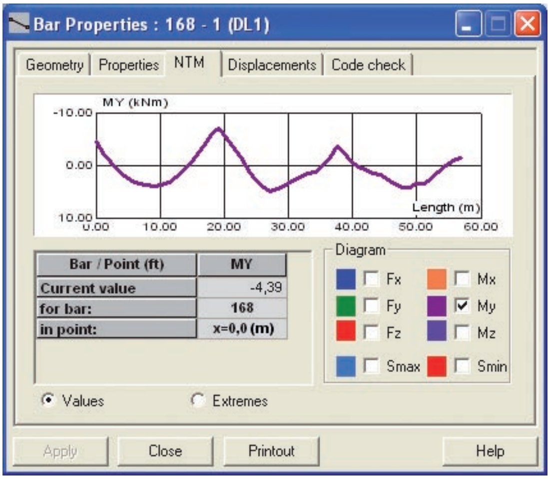


Figure 9.3 - Calculation results for single bar

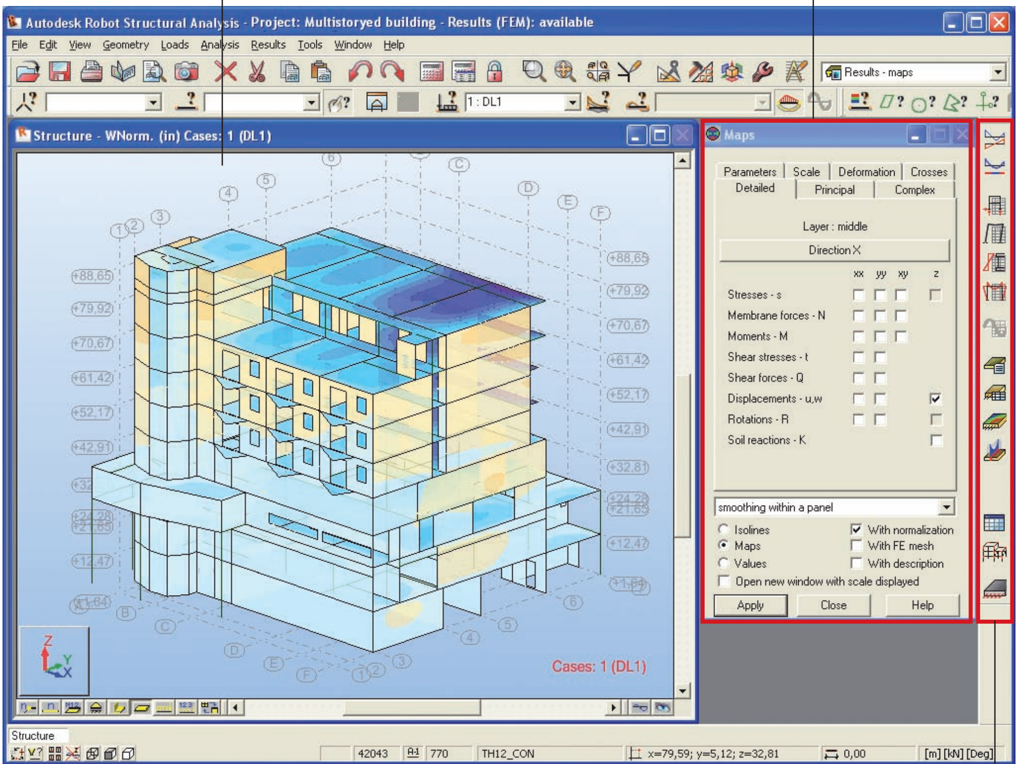
A right mouse click on any bar displays its properties and results (if analysis has been carried out)

Graphical results for surfaces

(graphical results for surfaces, contour maps for bending, deflection, stress, animations; "reduced results" showing global forces in surface cuts, including direct reduced results for cores and stiff diaphragms)

Graphical presentation of result maps in graphic viewer/editor

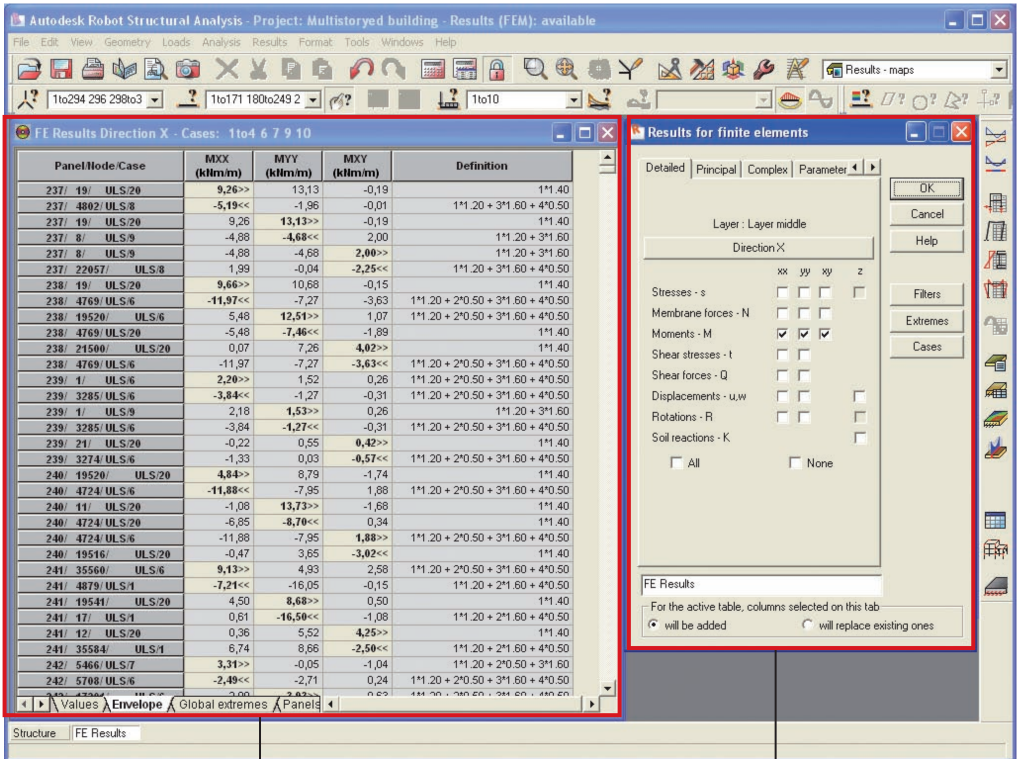
Dialog box to select type of map to display



Side toolbar shows additional results of structural analysis in tabular form, graphical form, plus links to design modules

Figure 9.4 - Results in the maps form

Tabular results



Tables can easily be edited, enveloping results, setting limits, filtering data by load case etc.

Dialog box used to define contents of the table. It is available from the menu activated by pressing the right mouse button when the cursor is located in the table, and selecting TABLE COLUMNS

Figure 9.5 - Tabular results

Robot tables can be easily transferred to MS Excel® by pressing the right hand mouse button.
Inside Robot, the tables can be manipulated in many ways, in the same way as input tables:

- filtering data for specific load cases, members or groups of members
- filtering data inside or outside user defined limits showing global maxima and minima for selected members, nodes or surfaces

REPORTS AND PRINTOUT COMPOSITION

Robot has a built in “report generator” which permits the user to create a user defined printout for a Project.
There are two options on the menu bar - “Printout Composition” and “Screen Capture” which help to prepare the final calculations notes, tables and graphic printouts.

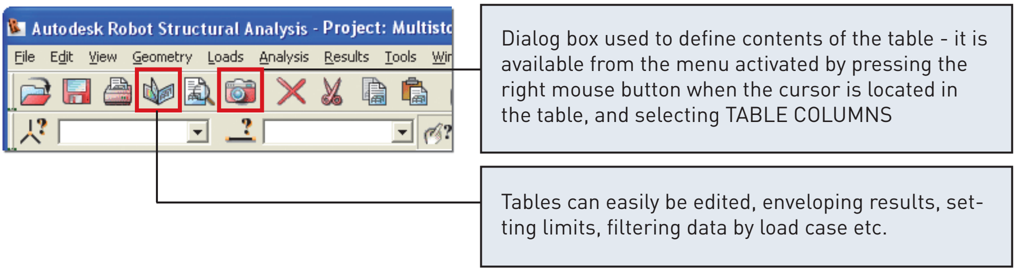


Figure 10.1 - Printout Composition and Screen Capture icons

The **Standard** tab lists a set of predefined options for printout, which can be mixed with screen captures to the user’s requirements.

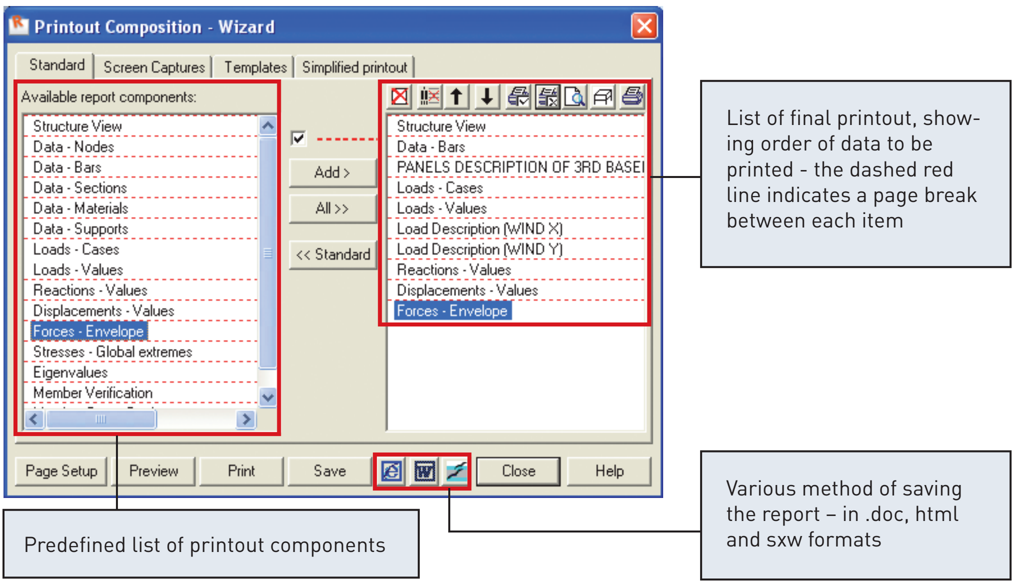


Figure 10.2 - Printout composition

A key feature of Robot is that all of the data in the program is saved in a common binary file – this data includes input, output, design plus also printout data. When structural data is altered or amended the output and printout (tables and graphics) are automatically updated on re-analysis to reflect the new results of the analysis. This is particularly useful when considering a schematic design or “sensitivity study” since the printout is automatically updated at every analysis step to reflect the current structural configuration.

ADVANCED ANALYSIS AND DESIGN

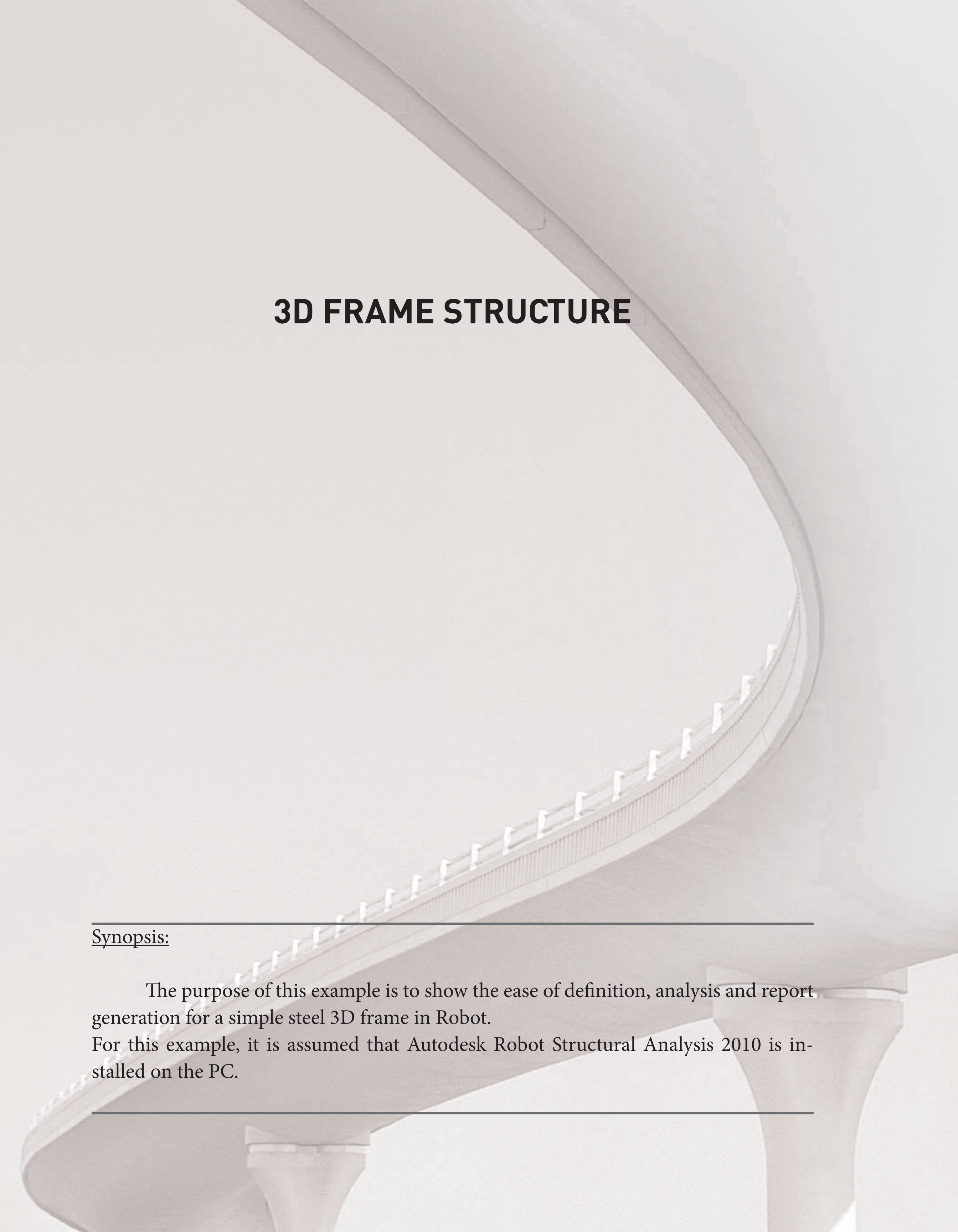
Following analysis of the structure, the user may wish to take advantage of many of the additional features – such as:

- Steel design
- Concrete design for beams, columns, floors
- Advanced analysis – P-Delta, dynamic etc.

These items are beyond the scope of this “Getting Started Manual” and are covered in more depth in the help for Robot.

LIST OF SHORTCUTS

In order to	Press
select all	**Ctrl + A**
copy a text or a drawing	**Ctrl + C**
open a new project	**Ctrl + N**
open an existing project	**Ctrl + O**
start printing	**Ctrl + P**
save the current project	**Ctrl + S**
cut a text or a drawing	**Ctrl + X**
repeat the last operation	**Ctrl + Y**
paste a text or a drawing	**Ctrl + V**
undo the last operation	**Ctrl + Z**
display the 3D view of a structure (3D XYZ)	**Ctrl + Alt + 0**
project a structure on XZ plane	**Ctrl + Alt + 1**
project a structure on XY plane	**Ctrl + Alt + 2**
project a structure on YZ plane	**Ctrl + Alt + 3**
zoom in the structure view on screen	**Ctrl + Alt + A**
display the initial view of the structure (defined by the initial angles and scale)	**Ctrl + Alt + D**
“exploded” view of structure elements (on/off)	**Ctrl + Alt + E**
zoom window	**Ctrl + Alt + L**
turn on/off section drawing display	**Ctrl + Alt + P**
screen capture	**Ctrl + Alt + Q**
zoom out structure view on screen	**Ctrl + Alt + R**
turn on/off section symbol display	**Ctrl + Alt + S**
rotate continuously around the X axis	**Ctrl + Alt + X**
rotate continuously around the Y axis	**Ctrl + Alt + Y**
rotate continuously around the Z axis	**Ctrl + Alt + Z**
delete a text or a drawing	**Del**
call Robot Help system for the active option in the active dialog box	**F1**
call text editor	**F9**
reduce structure attributes (supports, numbers of nodes, bars, loads) presented on screen	**PgDn**
enlarge structure attributes (supports, numbers of nodes, bars, loads) presented on screen	**PgUp**



3D FRAME STRUCTURE

Synopsis:

The purpose of this example is to show the ease of definition, analysis and report generation for a simple steel 3D frame in Robot.
For this example, it is assumed that Autodesk Robot Structural Analysis 2010 is installed on the PC.

CONFIGURATION OF THE PROGRAM

Following installation of Robot, the user may configure the working parameters or "Preferences". To do this:

1. Start the Robot program (click the appropriate icon or select the command from the taskbar).

2. On the opening window, displayed on the screen click the last but one icon  in the first row (**Frame 3D Design**). Other options are for 2d and 3d frames, models with surfaces such as walls and floors, access to stand alone design modules etc.

3. Select **Tools** > **Preferences** from the text menu or click on the  , then  icons on the toolbars to open the "Preferences" dialog box. Preferences allow the user to set up working and printout languages, fonts, colors etc.

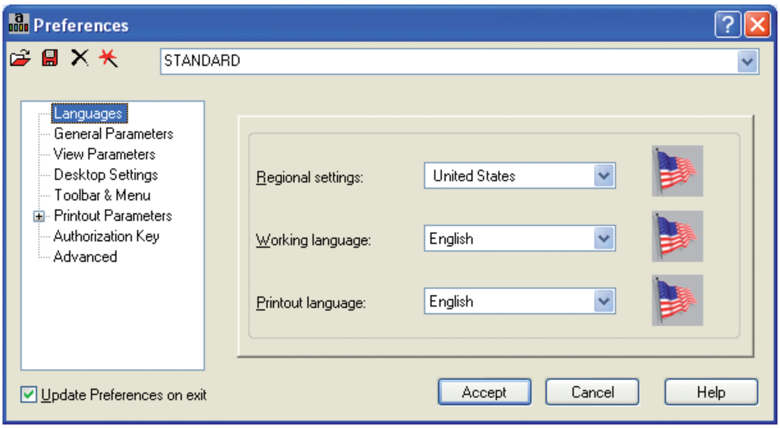


Figure 1 - Regional settings in Preferences dialog box.

4. Select the first preferences option - **Languages** (tree on the left part of the window), then as the **Regional settings** choose **United States**.

NOTE:

Regional settings set the default databases (profiles, materials), units and codes to the standards of a country. In the example above, we have chosen American section database (AISC) and metric data units: [m],[cm], [kN].

5. Click **Accept** to close the window.

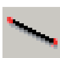
NOTE:

You can check active units in right, bottom corner of the screen. In this example should be the display: [m] [kN] [Deg].

MODEL DEFINITION

Bars definition (frame 2D)

In this step, you create frame members consisting of 2 columns and a beam.

1. Click  icon (right side of the screen) to open the **Bars** dialog box.

2. Set **Bar type**: as **Column** – whatever is selected is not important for analysis, but affects the design parameters for subsequent member design, such a buckling length, position of restraint etc.

3. Define **Section**: as **W14x211**.

NOTE:
If the **W14x211** section is not available on the list, you should click the (...) button located beside the **Section** field and add this section from the database. In opened **New section** dialog box, in **Section selection** field for **Database: AISC** – choose **Family: W** then Section: **W14x211**. Click **Add** and **Close** the box.

INFO

There are many extra options that may be entered for fabricated members, tapering sections and also for beams that the user wants to define as able to exhibit plasticity.

4. Enter the following points in the **Beginning** and **End** field:
   - to define first column of frame: (0;0;0) and (0;0;3) **Add**
   - to define second column: (7;0;0) and (7;0;3) **Add**

5. Set **Bar type**: as **Beam**.

6. Define **Section**: as **W14x211**.

7. To define a beam in the structure, enter the following points in the **Beginning** and **End** field: (0;0;3) and (7;0;3) **Add**.

8. Close the **Section** window.

Supports definition

In this step, you create supports for the frame structure.

1. Click  icon (right side of the screen) to open the **Supports** dialog box.

2. To choose structure nodes, the user can either select a support type from the list and then select a node or nodes by click or window, or the user can directly input

the node number and apply. To make a window selection, press the right hand mouse key and then the **Select** menu option and window around the nodes to be supported and click inside the selection box.

3. From the **Supports** dialog box select the **Fixed** support icon.

4. Click **Apply**.

2-bay frame definition

In this step, we create a 2 bay frame by mirroring the existing frame.

1. Select all bars (by window or **CTRL+A**) and mirror them **Edit** > **Edit** > **Vertical Mirror** by vertical axis of the right column (just click on this axis).

> **INFO**
>
> There are many editing possibilities: copy, move, divide, intersect, rotation etc. to make modeling of structure easier and more effective.

2. Click  icon (top of the screen) to show the whole structure.

3. To display bars and nodes numbers click  icon (bottom left corner of the screen) then in the Display dialog box tick on **Favorites** > **Node numbers** and **Favorites** > **Bar description** > **Bar numbers**.

4. Click **Apply** and **OK**.

5. Click ,  icon at the bottom of the screen to display supports symbol and sections shape.

6. The frame should appear as below:

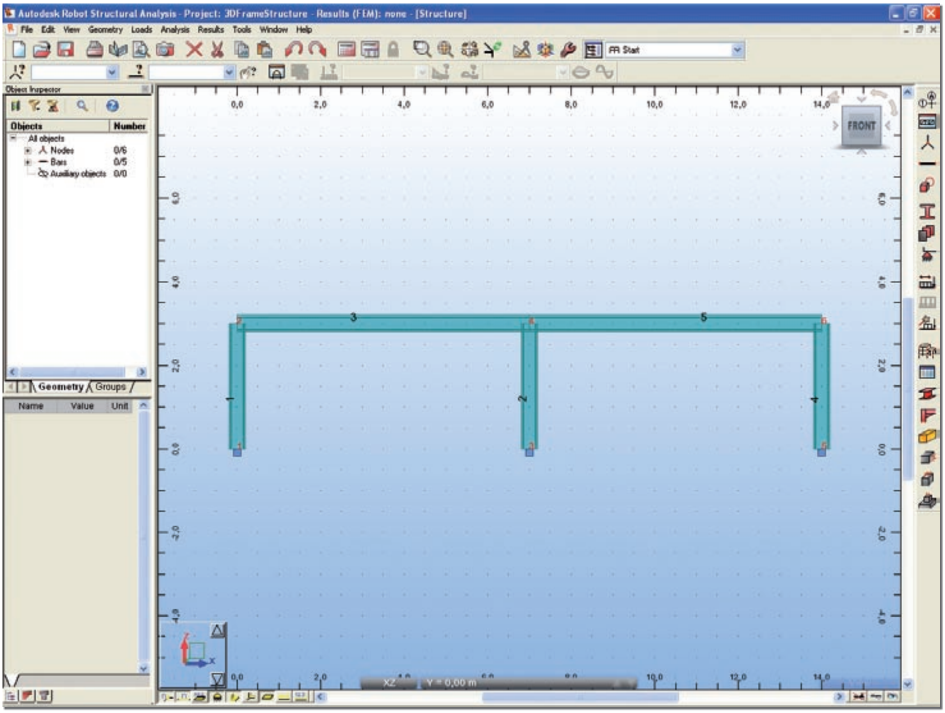
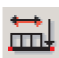


Figure 2 - Frame with supports and sections shape view.

Load case definition

In this step, you define load cases (their number, nature and description).

1. Click  icon (right side of the screen) to open the **Load Types** dialog box.
2. Click **New** to define a **dead load** (self-weight) with a standard name **DL1**.

> **NOTE:**
>
> The self-weight load is automatically applied in the first row to all structure bars (in the "Z" direction).

3. Choose in the **Nature** field **live**, then **New** to define a **live load** with a standard name **LL1**. We have now defined 2 load cases.

4. Choose in the **Nature** field **wind**, then **New** to define a **wind load** with a standard name **WIND1** (in the same way define **WIND2**). We have now defined 4 load cases.

5. **Close** the dialog box.

Loads definition for particular load cases

In this step, you define loads for each created load case.

1. To define loads for **LL1** case select the 2nd load case in the list of defined cases field:

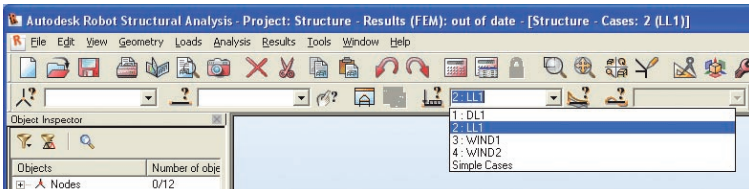
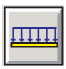


Figure 3 - Load case LL1 selection.

2. Click  icon (right side of the screen) to open the **Load Definition** dialog box.

3. On the **Bar** tab click  icon to open the **Uniform load** dialog box, then type value of load **-45.0** in the **pZ** field, click **Add (Uniform load** dialog box is automatically closed).

4. Select beam span (no.3) – indicate them or just type **3** in field **Apply to**.

5. Click **Apply**.

6. On the **Bar** tab click  icon to open **Trapezoidal load** dialog box, then type values of loads and coordinates as shown below, click **Add** and close the box. This generates a varying load on the beam.

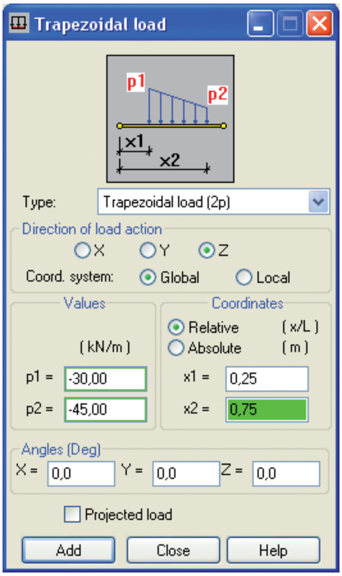
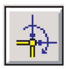


Figure 4 - Trapezoidal load definition.

7. Select beam span (no.5) – indicate the beam or just type **5** in field **Apply to**.

8. Click **Apply** and close **Load Definition** dialog box.

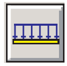
9. To define loads for **WIND1** case select the 3rd load case in the list of defined cases field.

10. On the **Node** tab in **Load Definition** dialog box click  icon to open **Nodal Force** dialog box, then type value of load **9.0** in the **FX** field, click **Add** and close the box.

11. Select the upper node of the left column (no.2) – graphically or by cross-window or just type **2** in field **Apply to**.

12. Click **Apply** and close **Load Definition** dialog box.

13. To define loads for **WIND2** case select the 4th load case in the list of defined cases field.

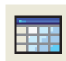
14. On the **Bar** tab in **Load Definition** dialog box click  icon to open **Uniform load** dialog box, then type value of load **-20.0** in the **pX** field (pY=0; pZ=0), click **Add** and close the box.

15. Select the right column (no.4) – indicate it or just type **4** in field **Apply to**.

16. Click **Apply** and close **Load Definition** dialog box.

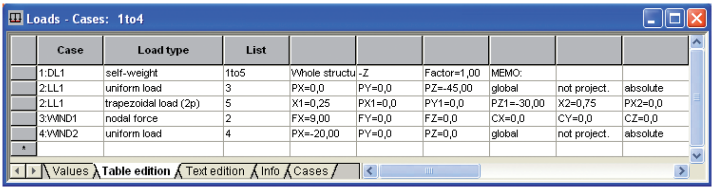
17. Close the **Load Types** dialog box.

18. Click ,  icons at the bottom of the screen to turn displaying loads symbols and loads values description on.

19. The loads can also be seen in tabular form – click  **Tables** icon (right side of the screen) then tick **Loads** on or select **View** > **Tables** > **Loads** command (see below).

INFO

All tables in Robot can be exported to MS Excel® by simply clicking the right hand mouse button in the table.



Case	Load type	List						
1:DL1	self-weight	1to5	Whole structu	-Z	Factor=1,00	MEMO:		
2:LL1	uniform load	3	PX=0,0	PY=0,0	PZ=-45,00	global	not project.	absolute
2:LL1	trapezoidal load (2p)	5	X1=0,25	PX1=0,0	PY1=0,0	PZ1=-30,00	X2=0,75	PX2=0,0
3:WIND1	nodal force	2	FX=9,00	FY=0,0	FZ=0,0	CX=0,0	CY=0,0	CZ=0,0
4:WIND2	uniform load	4	PX=-20,00	PY=0,0	PZ=0,0	global	not project.	absolute

Figure 5 - Load cases data in tabular form.

20. In the list of defined load cases choose the load case for which the loads will be displayed:

21. All defined loads cases will be displayed together (as shown below):

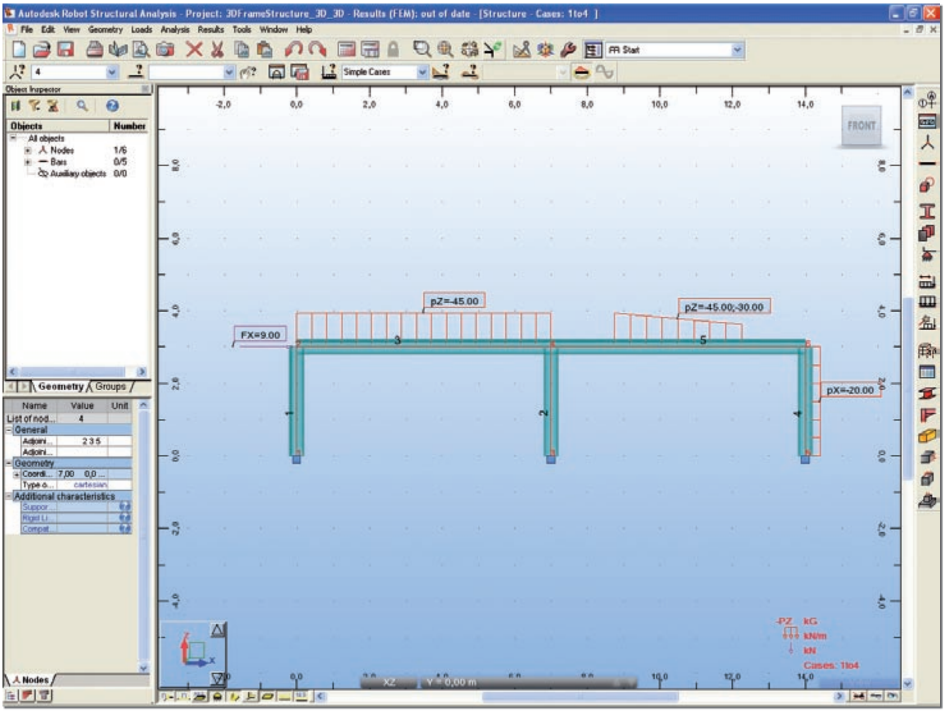
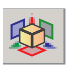
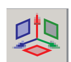


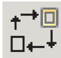
Figure 6 - All defined load cases view.

Copying existing frame

In this step, we copy the 2D frame to generate 3D structure. When we copy that frame, all attributes attached to it (loads, sections, supports etc.) are also copied:

1. Click  icon (top of the screen) to open the **View** dialog box, then  or choose **View** > **Work in 3D** > **3D xyz** from the menu to select the isometric structure view.

2. Click CTRL+A to select all of the structure.

3.  **Edit** then  **Translation** – or select the **Edit** > **Edit** > **Translate** option from the pull-down menu to open the **Translation** dialog box.

4. In **Translation vector** field type coordinates as shown below (be sure that in **Edit mode** option **Copy** instead of **Move** is checked on and **Number of repetitions** is **1**).

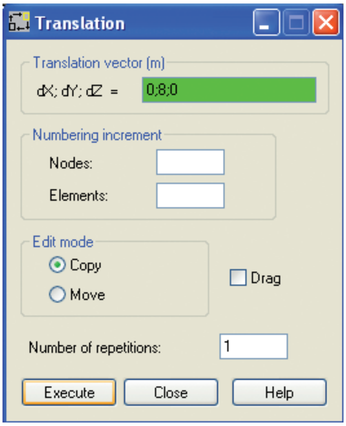


Figure 7 - Copy frame parameters definition.

5. Click **Execute** – second frame should be display on the screen.

6. Close the **Translation dialog** box.

7. Click  icon to show the whole structure. The structure should look as shown below:

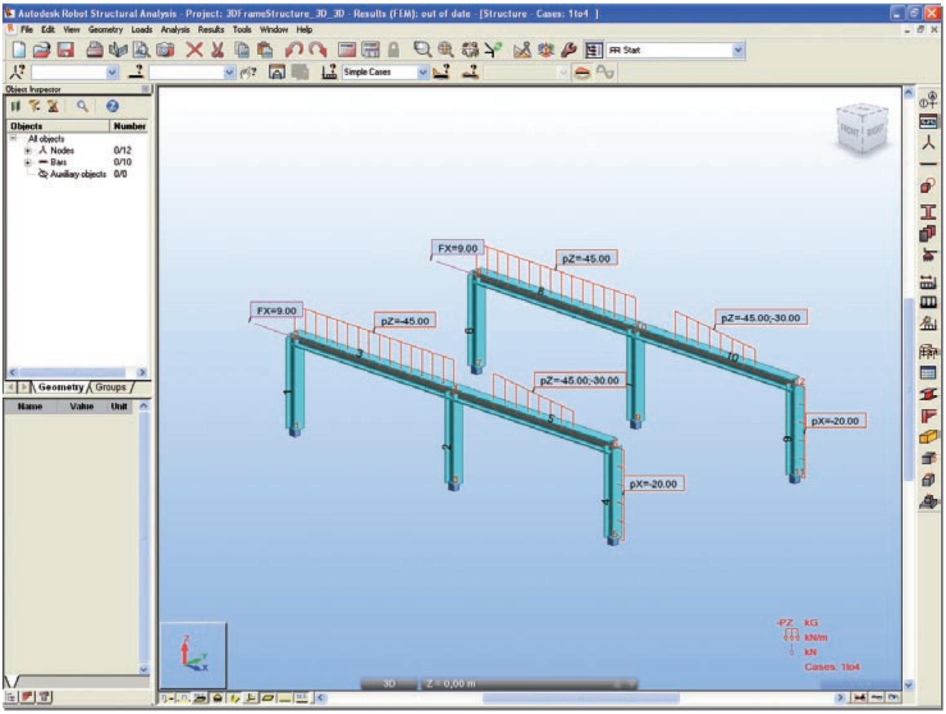
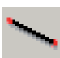


Figure 8 - 3D View of copied frame.

Definition of lateral beam

In this step, you define beams to join the selected frames together:

1. Click  icon (right side of the screen) to open the **Bars** dialog box.
2. In **Bars** dialog box set **Bar type**: as **Beam**.
3. Define Section: as **W12x190**.

NOTE:

If the **W12x190** section is not available on the list, one should click the **(...)** button located beside the **Section** field and add this section to the active section list in the **New section** dialog box.

4. In the fields **Beginning** and **End** type coordinates (as shown below) or simply select beginning field and then graphically draw the start and end points of the member.

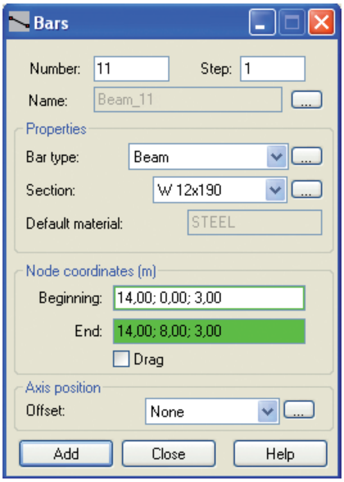
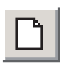


Figure 9 - Bars definition.

5. If entered by coordinates, Click **Add** and close **Bars** dialog box.

Definition of cross bracings

1.  Sections then  **New section definition** – or select the **Geometry > Properties > Sections > New section definition** option from the menu to open the **New Section** dialog box:

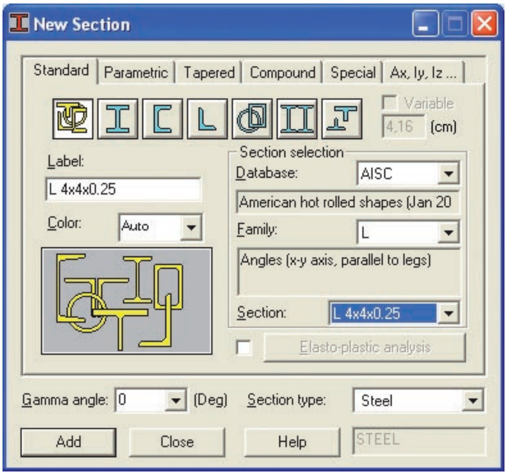
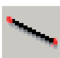


Figure 10 - New section definition.

2. Select **Section** (L4x4x0.25) from American section **Database** (AISC) as shown above.

3. Click **Add** - new section type will be added to the active **Sections** list.

4. Close **New section definition** and **Sections** dialog boxes.

5. Click  icon (right side of the screen) to open the **Bars** dialog box.

6. In **Bars** dialog box set **Bar type**: as **Simple bar**.

7. Define **Section**: as **L4x4x0.25**.

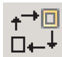
8. In the fields **Beginning** and **End** type coordinates:
   (14;0;3) (14;8;0) click **Add** then **(14;8;3) (14;0;0)** and press **Add**. Or draw in the GUI.

9. Close **Bars** dialog box.

Copying defined bars (lateral beam and bracings)

In this step, you copy the recently defined bars to finish the definition of the frame:

1. Select the three recently defined bars – just indicate them (bars no.11,12,13) - multiple selections can be made by holding down CTRL key.

2.  **Edit** then  **Translation** – or select the **Edit** > **Edit** > **Translate** option from the menu to open the **Translation** dialog box.

3. In **Translation vector** field type coordinates: **(-7;0;0)** or select 2 points on the screen that represent the translation vector such as the bottom of 2 columns (be sure that in **Edit** mode option **Copy** is checked on) and in **Number of repetitions** type **2**.

4. If entered by coordinate, click **Execute** button then close **Translation** and **Edit** dialog boxes.

5. The structure should look as shown below:

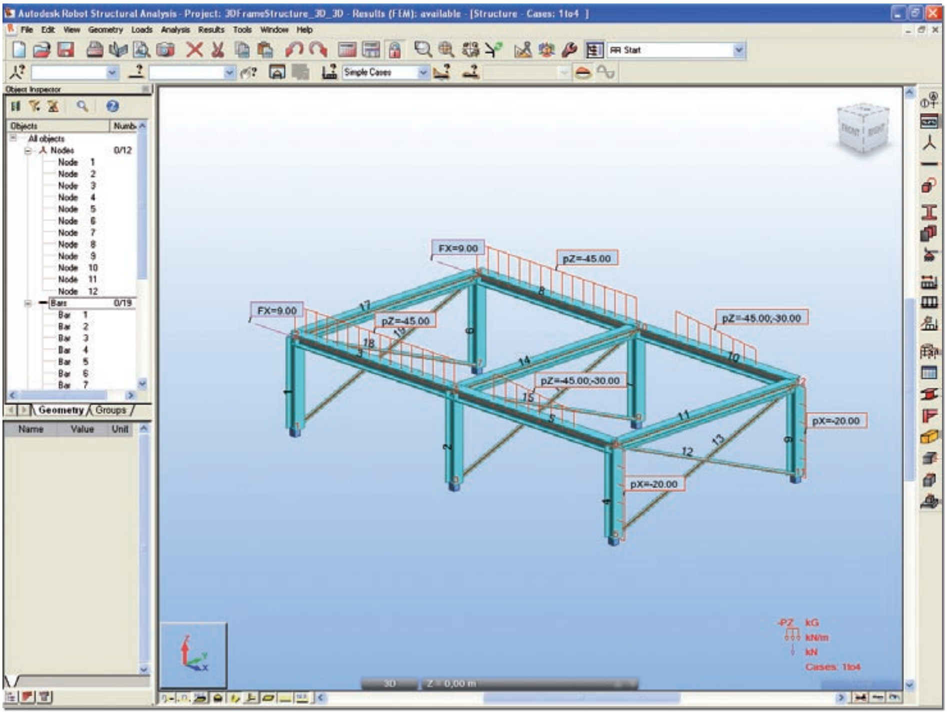


Figure 11 - Complete 3D frame structure view.

STRUCTURE ANALYSIS

Here we start the analysis process, but firstly we will tell Robot to make automatic code combinations (from any one of the list of Codes in Job Preferences):

1. Select **Loads** > **Automatic Combinations…** option from the menu to open the **Load Case Code Combinations** dialog box. Select the **Full automatic combinations** option. The program will now automatically assign a number of combinations to find the most onerous load combination.

2. Click **OK** button – automatic calculation of code combinations will be done.

3.  **Calculations** – click this icon or select the **Analysis** > **Calculations** option from the menu to start calculation process.

4. Once the calculations are completed the information: **Results (FEM): Available** should be displayed at the top of the screen.

> **NOTE:**
>
> A key strength of Robot is the possibility to define a wide range of analysis types (linear static, non-linear geometry and material, buckling, modal analysis, harmonic analysis, seismic analysis, time history analysis etc.)
> If the user wishes to see these possibilities, he/she may look in the **Analysis > Analysis Types > Change analysis type** pull down menu.
>
> However, for this simple example, we will just assume the default linear static type of analysis.

RESULTS PREVIEW

Displaying beam results graphically

In this step, we display **My** bending moment on bars for selected load case:

1. Click  icon at the bottom of the screen to switch off the section shape and simply display a "stick model" for all members.

2. In the list of defined load cases choose the desired load case to display:

3. Select **Results** > **Diagrams for Bars…** 

from the menu to open **Diagrams** dialog box.

4. On the **NTM** tab check **My Moment** (this shows the major axis bending moment on the beams), then click **Normalize** button (to auto scale) and click **Apply** to display bending moments diagrams for all beams:

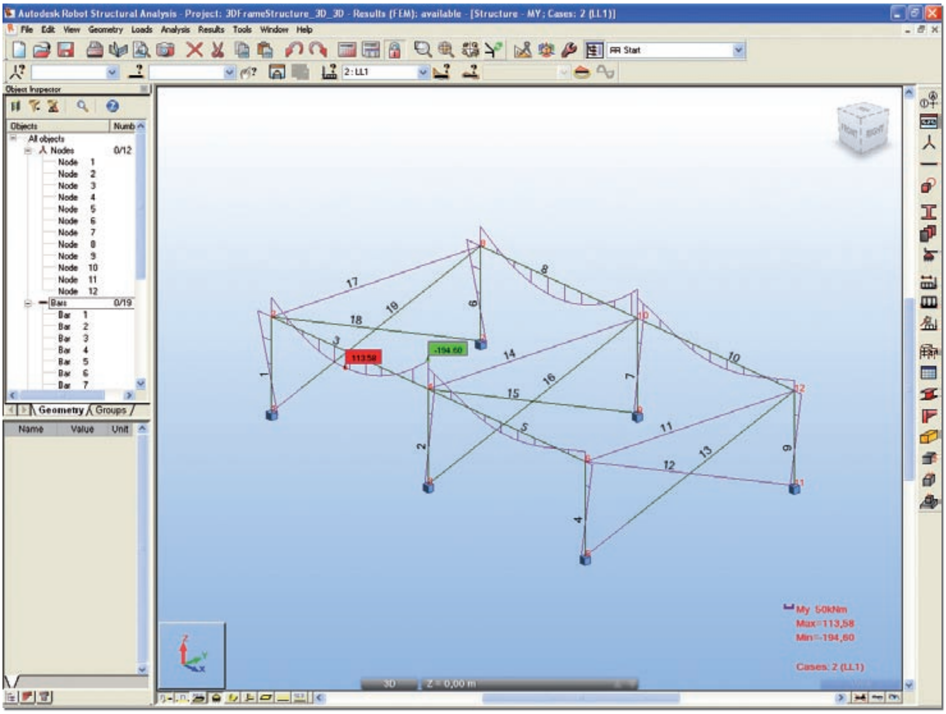


Figure 12 - Diagram of bending moment.

> **NOTE:**
>
> To display numerical values of internal forces in the **Diagrams** dialog box, click on tab **Parameters, Diagram description** tick labels and **Apply**.

5. In a similar way, diagrams that exhibit other values available from the **Diagrams** dialog box can be viewed.

6. Tick off **My Moment** and **Apply**.

Displaying results on bars in tabular form

In this step, you display internal forces for bars for particular load cases and combinations:

1. Select **View** > **Tables** to open **Tables: Data and Results** dialog box, then tick on **Forces** and **OK** to view all information about internal forces (Values, Envelope, Global extremes).

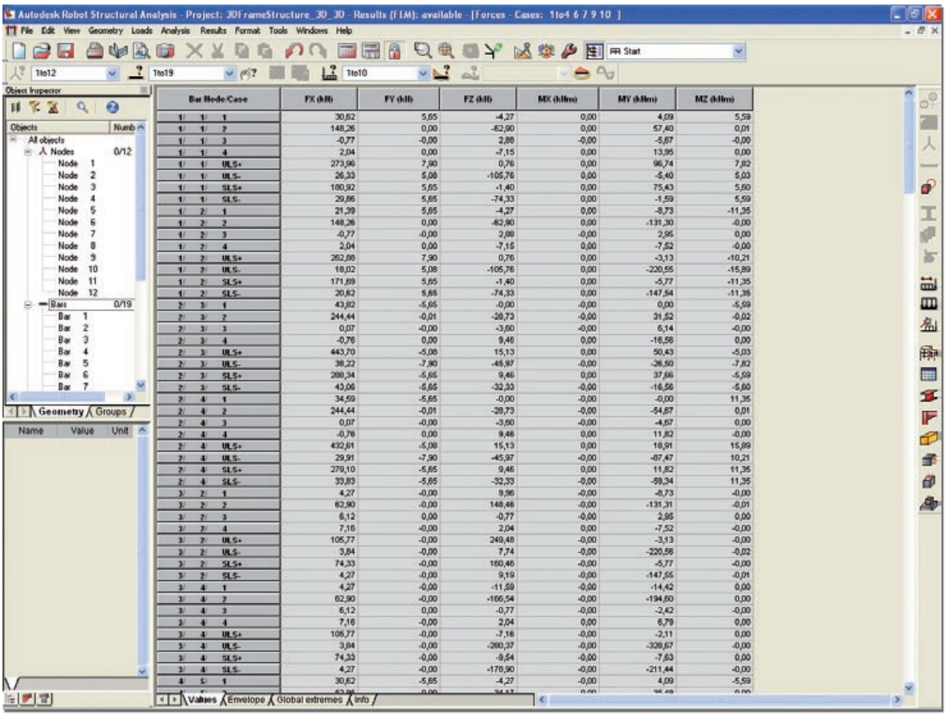


Figure 13 - Example of results presentation in tabular form.

> **NOTE:**
>
> In a similar way, tables that exhibit other values available from the **Tables: Data and Results** dialog box can be viewed.
> The tables can easily be exported to MS Excel® (by right hand mouse click) and tables can also be sorted by choosing certain load cases, member types etc. Advanced users can also define "groups" and display results only for these groups. Column values may be placed in order by simply clicking the appropriate column header, for example **FX**.

2. Close the **Forces** table.

3. Click  icon at the bottom of the screen to turn sections shape back on.

Stress analysis

In this step, we learn how to obtain and analyze stress diagrams and maps for the entire bar structure:

1. Switch layout to **Stress Analysis** - **Structure**. To do this go to the Robot layouts selection combo-box (right top screen corner)

2. Click the **Results** option and later the **Stress Analysis** - **Structure**.

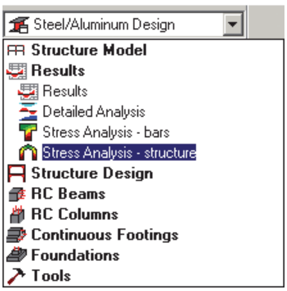


Figure 14 - Layout selection pull-down menu (stress analysis selection).

NOTE:

Now the screen is divided into three main parts:

- the graphic viewer where the structure is displayed,
- the **Stress Analysis -Structure** dialog box – to managing stress analysis. This dialog box is used for selecting stresses and determining the manner of stress graphical presentation
- the **Stress Analysis -Structure** results table to view actual values of stresses for bars

3. In the list of defined load cases choose the load case for which the results will be displayed:

4. On the **Diagrams** tab located in the **Stress Analysis - Structure** dialog box select the **Max** option from the **Mises** field, then **Apply** (the result should look as shown below):

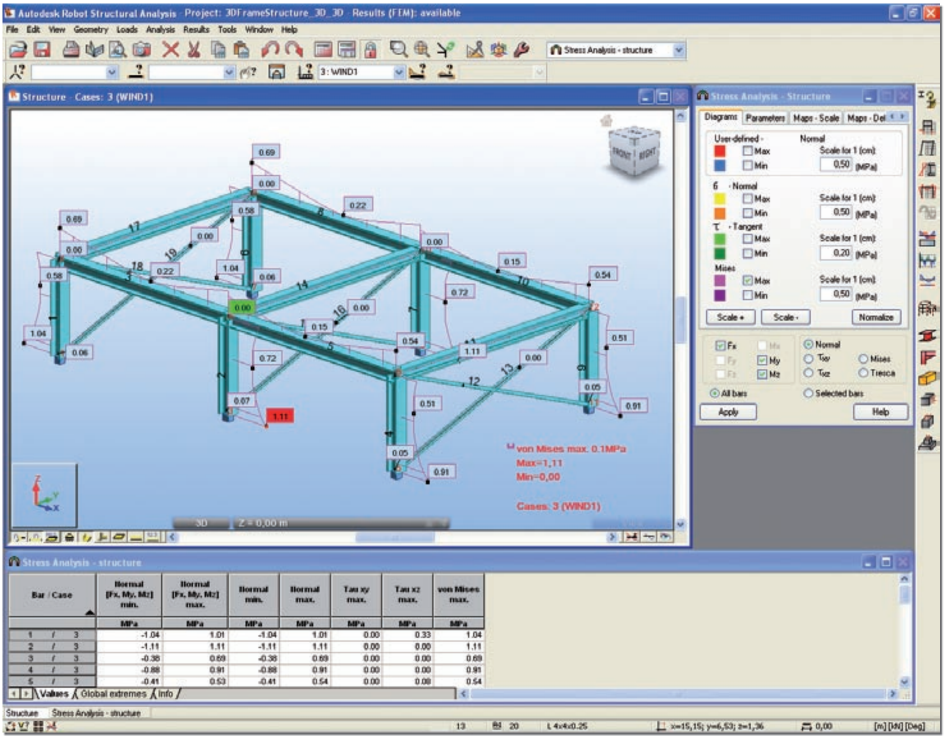


Figure 15 - Stress analysis in diagram form.

NOTE:

This dialog box allows selecting a user-defined stress and a set of basic stress types: normal, tangent, Mises and Tresca.

5. On the **Stress Analysis** - **Structure** dialog box tick off the **Max** option from the **Mises** field, then **Apply**.

6. On the **Maps** - **Deformation** tab located in the **Stress Analysis** - **Structure** dialog box tick the **Deformation** option on, then **Apply**.

7.  **Stress Analysis for structure - maps** - click this icon or select the **Results** > **Stress Analysis** > **Stress Maps** option from the menu to view the structure together with section shapes and accurate detailed stress maps on these sections (as shown in the figure below).

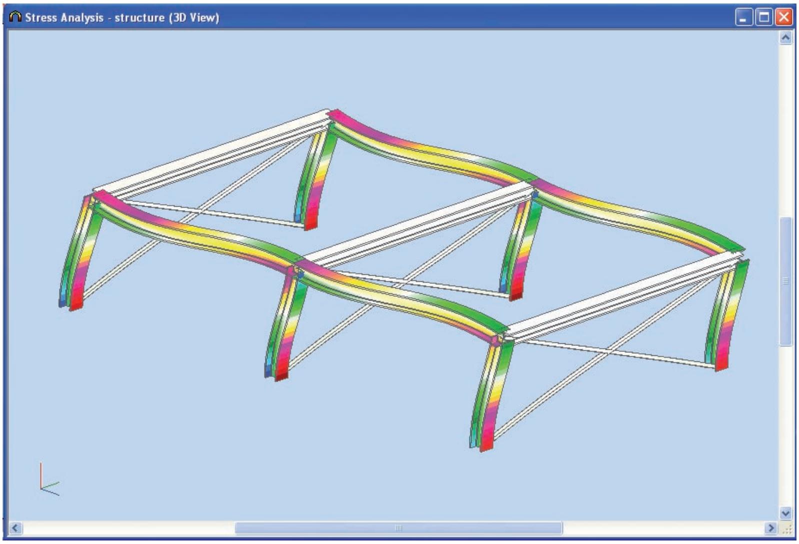


Figure 16 - Stress analysis in maps form.

NOTE:

The size of member sections in stress maps is by default greater than the real size of member sections so that the stress maps presented for these sections are more readable. Real proportions between the member length and dimensions of the member cross section can be obtained after pressing the **Home** key on the keyboard. These proportions can be modified using the following keys on the keyboard: **PgUp** and **PgDn**.

8. Close the **Stress Analysis – structure (3D View)** window.

PREPARATION OF PRINTOUTS

Robot has a powerful built in report generator that allows the user to compile and manipulate standard data, plus screen captures of graphical and tabular data.

"Capturing" views and data for the calculation note

1. In layouts selection combo-box (right top screen corner) switch to **Start** layout. The picture of the model is shown on the screen with no other windows. Let's assume we want to print this image.

2. Click  icon (top of the screen on the standard toolbar) to open the **Screen Capture** dialog box.

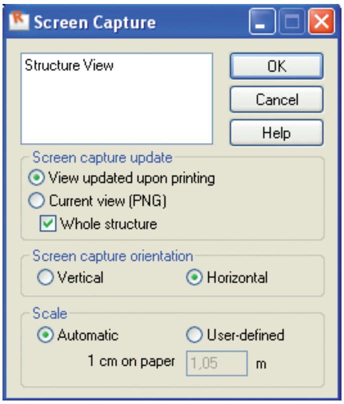


Figure 17 - Screen capture parameters definition.

3. In the above dialog box change **Screen capture orientation** to **Horizontal** and type name of the picture as a **Structure View**.

NOTE:

The **Screen capture update** enables the user to choose automatic update of screenshots – this means that the images and data are automatically updated should the structure be changed. Alternatively, they can be saved in JPG format.

4. Click **OK** to close the box.

5. In the list of defined load cases choose the **2:LL1** load case.

6. Click  icon (top of the screen on the standard toolbar) to open the **Screen Capture** dialog box.

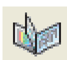
7. In this dialog box change **Screen capture orientation** to **Horizontal** and leave name of the picture as a **Structure – Cases 2: (LL1)**.

8. Click **OK** to close the box.

9. In the same way prepare pictures: **Structure – Cases 3: (WIND1)** and **Structure – Cases 4: (WIND2)**

Preparing printout composition

Here we prepare the layout of the report:

1. Click  icon (top of the screen on the standard toolbar) to open the **Printout Composition - Wizard** dialog box.

2. On the **Standard** tab, select report components (click them while pressing CTRL key) as shown below:

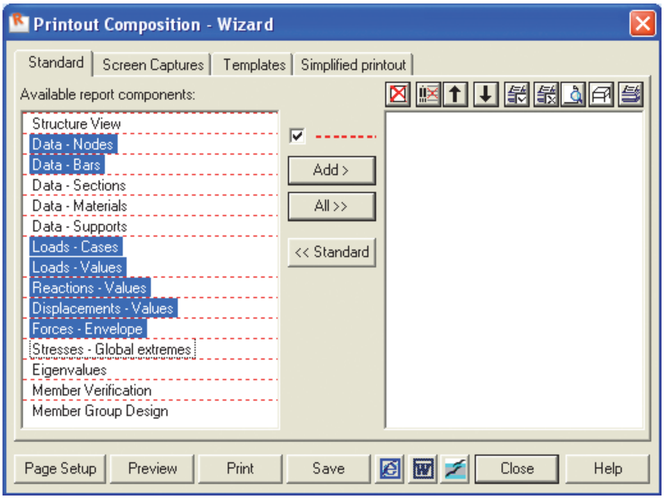


Figure 18 - Report components selection.

3. Click **Add** to transfer the marked components to the right panel which shows the actual layout of the printout.

4. On the **Screen Captures** tab select all screenshots.

5. Click **Add** to transfer marked components to the right panel.

6. Using icons  move particular components to the order shown below:

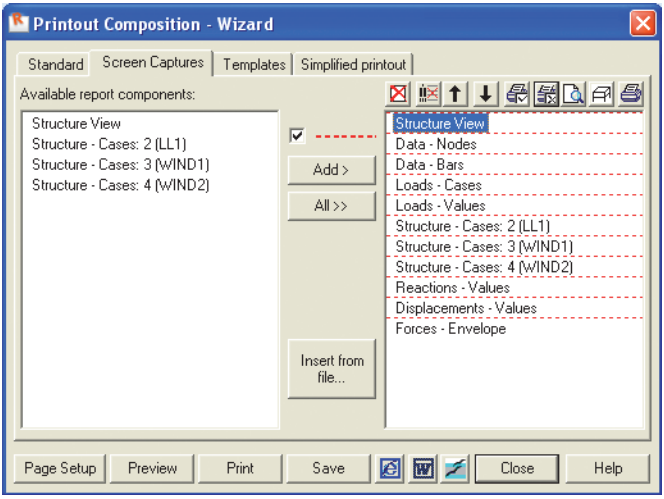


Figure 19 - Screen captures selection.

Printing and exporting the calculation report

In this step, we learn how to print and save the calculation note:

1. Click **Preview** to display the report:

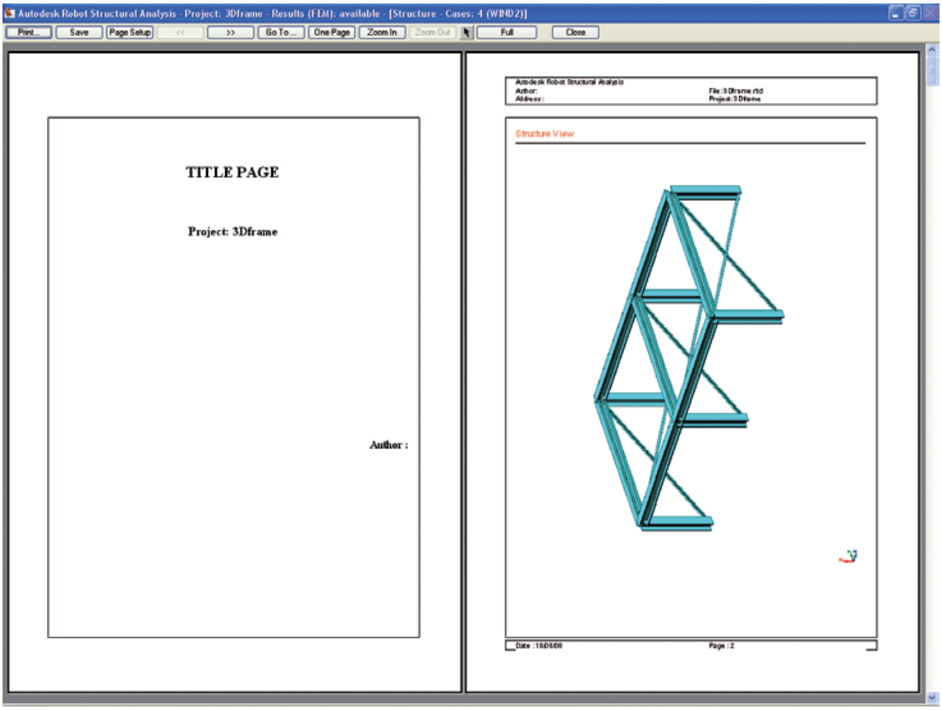


Figure 20 - Report preview.

NOTE:

Before printing of the report, save your project. In this case we save it as "**Frame 3D**". Thanks to this operation actual information about project will be used to generate the documentation (i.e. project name on the title page – as shown above). To change this information (and many more) later we can use **File > Project properties** option from the top menu.

2. Using buttons  browse each page of the Report.

NOTE:

If you need to make some corrections to the layout or tabular content you can edit them directly in this preview window. To do this, you should press icon  (top bar) and double click on the appropriate item to be edited. The general Robot window will be open, which allows the user to make changes or corrections. After that press button **Return to Preview** to continue the preview of the calculation report.

3. Click **Page setup**. The **Page** and **Margins** tab allows the user to set various parameters.

4. Click **Header/footer** tab. In this tab you can define the look of the title page, header and footer.

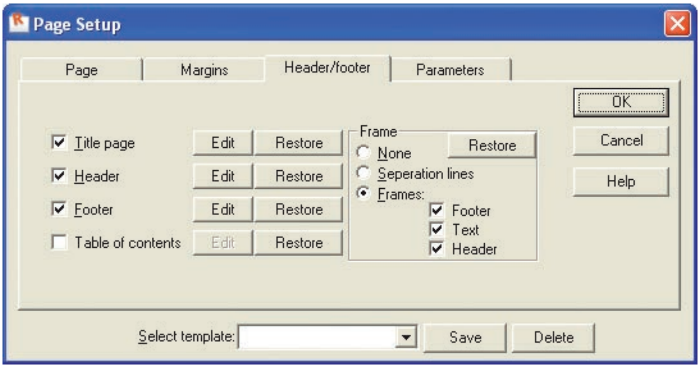


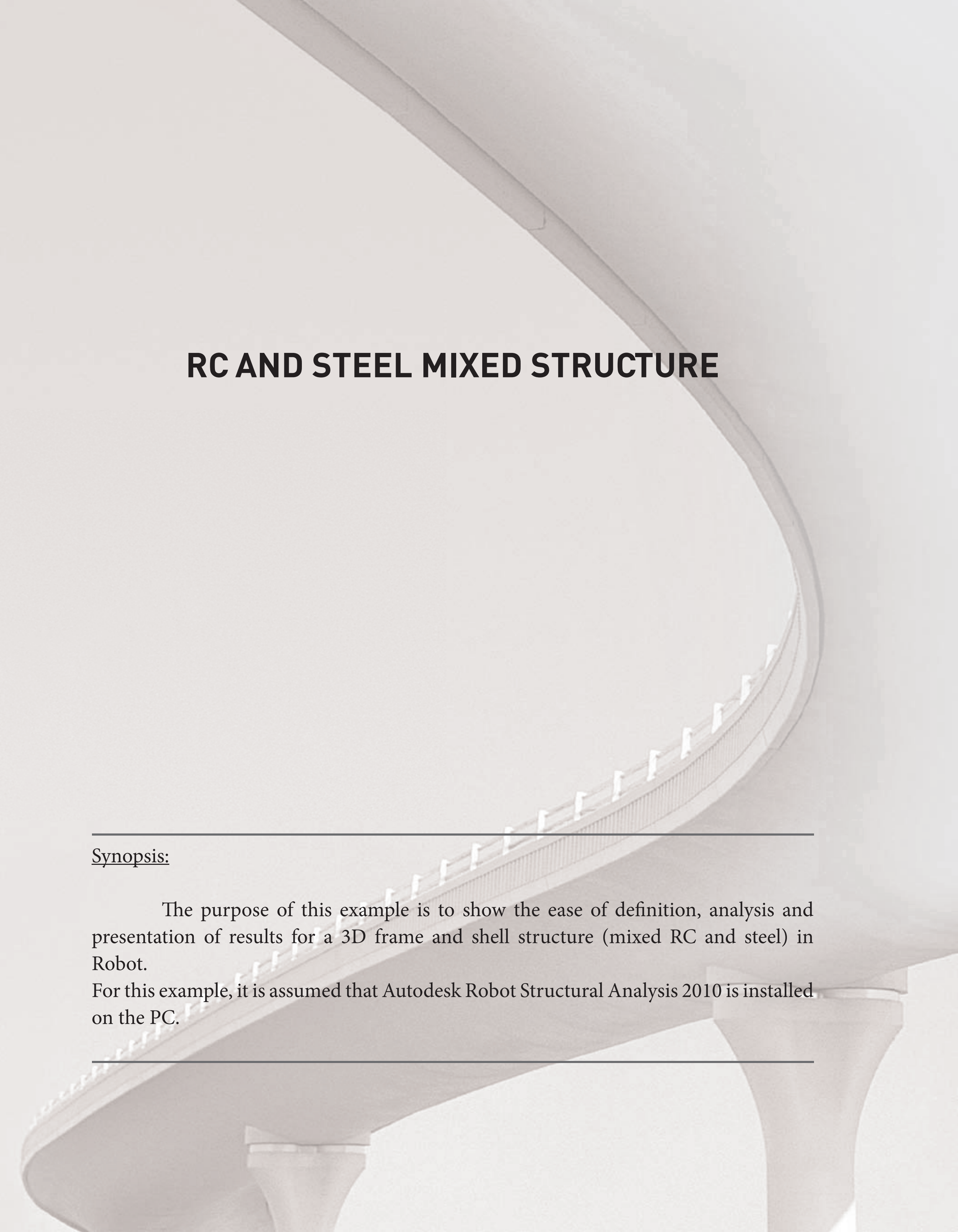
Figure 21 - Page setup settings.

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

6. Close the preview of report printout window.

7. If desired, In **Printout Composition – Wizard** dialog box click on **Print** button to print all reports.

8. To save the calculation note, click on icon  to export the report to .html format,  to export to .rtf (MS Word®) format or  to save in .sxw (Open Office) format.



RC AND STEEL MIXED STRUCTURE

<u>Synopsis:</u>

The purpose of this example is to show the ease of definition, analysis and presentation of results for a 3D frame and shell structure (mixed RC and steel) in Robot.

For this example, it is assumed that Autodesk Robot Structural Analysis 2010 is installed on the PC.

CONFIGURATION OF THE PROGRAM

Following installation of Robot, the user may configure the working parameters or "Preferences". To do this:

1. Start the ROBOT program (click the appropriate icon or select the command from the taskbar).

2. On the opening window, displayed on the screen click the first icon  in the first row (**Frame 2D Design**). Other options are for 3d frames and models with surfaces such as walls and floors, access to stand alone design modules etc

3. Select **Tools** > **Preferences** from the text menu or click on the  , then  icons on the toolbars to open the **Preferences** dialog box. **Preferences** allow the user to set up working and printout languages, fonts, colors etc.

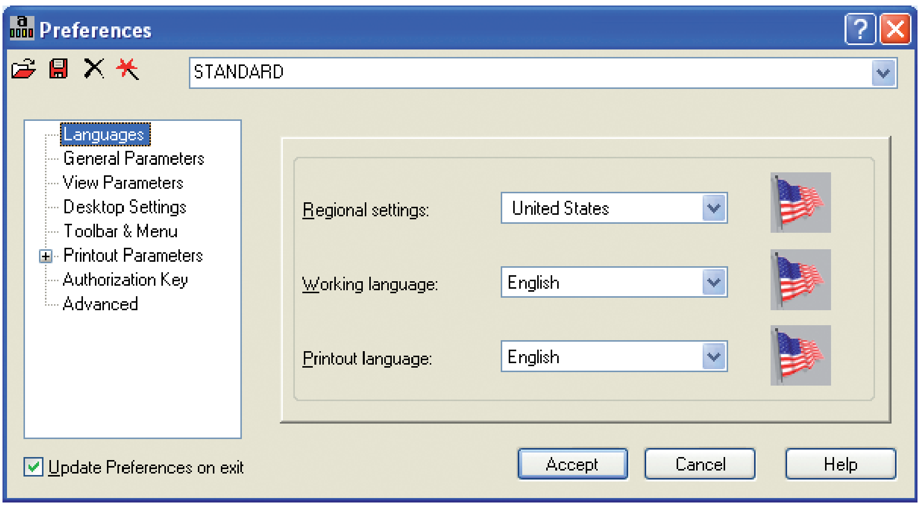


Figure 1 - Regional settings in Preferences dialog box.

4. Select the first preferences option - **Languages** (tree on the left part of the window), then as the **Regional settings** choose **United States**

> **NOTE:**
>
> Regional settings set the default databases (profiles, materials), units and codes to the standards of a country. In the example above, we have chosen American section database (AISC) and metric data units: [m],[cm], [kN].

5. Click **Accept** to close the window

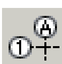
> **NOTE:**
>
> You can check active units in right, bottom corner of the screen. In this example should be the display: [m] [kN] [Deg].

MODEL DEFINITION

Definition of structural axes

In this step, we define a rectangular axis grid in the Cartesian coordinate system.

> **INFO**
>
> The axes of the structure create an additional grid which can be used to define different elements of the structure and select structure components.
> The grid intersections form points that facilitate the users work by guiding cursor movements during graphical structure definition.

1. Press  icon (right side of the Robot screen) or select **Geometry > Axis Definition…** from the menu to open the **Structural Axis** dialog box.

2. On the **X** tab in the **Position** field type **-1** and press **Insert** button, in the same way enter following axis coordinates as shown below (Figure 2):

3. Set **Numbering** as **A B C…**

4. On the **Z** tab in the **Position** field type **0** and press **Insert** button, in the same way enter following axis coordinates as shown below (Figure 3):

5. Set **Numbering** as **1 2 3…**

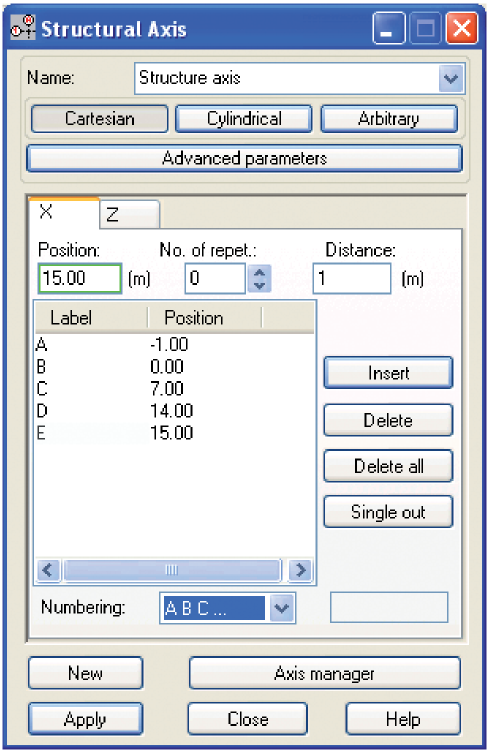


Figure 2 - Definition of structural axis in X direction.

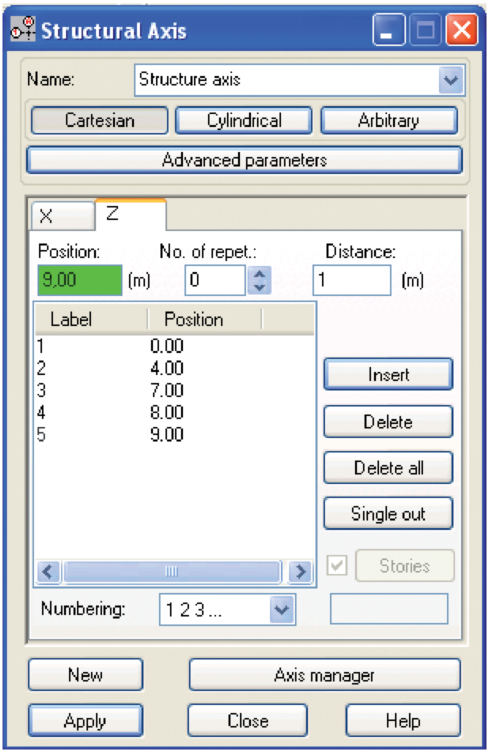


Figure 3 - Definition of structural axis in Z direction.

6. **Apply** and **Close** the **Structural Axis** dialog box.

7. Defined axes should appear as shown below:

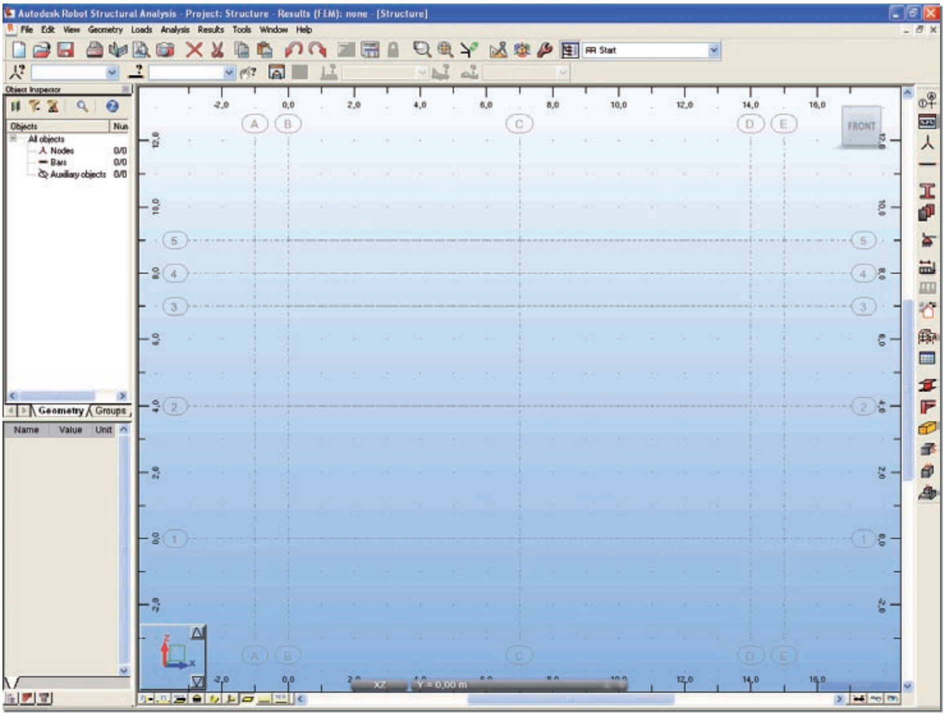


Figure 4 - View of defined structural axis.

Section definition

In this step, we learn how to add new sections to the list of available sections.

1. Press  icon (right side of the screen) or select **Geometry** > **Properties** > **Sections** from the menu to open the Sections dialog box.

2. Check presence of the following sections:

   CR30x30, BR30x60, W8x28, W10x45.

3. If the above sections are not present in the list of available sections press the **New section definition**  icon to open **New Section** dialog box.

4. In the **Section type** pull-down menu choose **RC Column**.

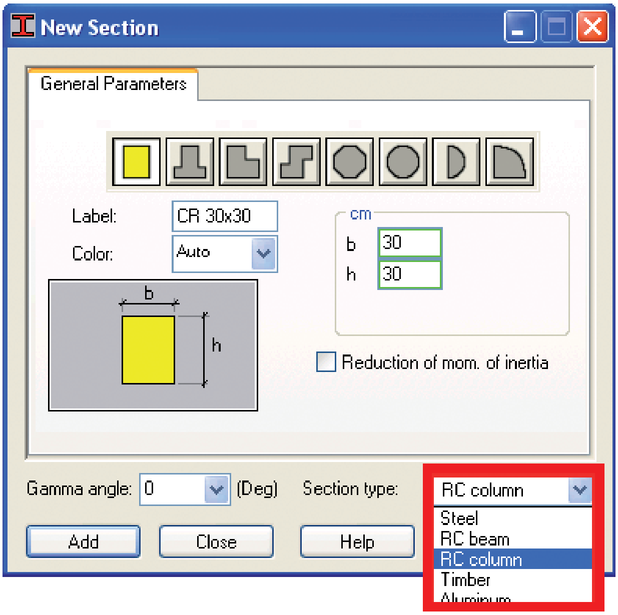


Figure 5 - New RC column definition.

5. Set **30** in the **b** and **h** field, in the **Label** field column name: **CR30x30** will be automatically created and click **Add** (see above).

6. In the **Section type** pull-down menu choose **RC Beam**.

7. Set **30** in the **b** and **60** in the **h** field, in the **Label** field column name: **BR30x60** will be automatically created, then press **Add**.

8. In the **Section type** field select **Steel**.

9. Set **AISC** in the **Database** field, **W** in the **Family** field and **W 8X28** in the **Section** field.

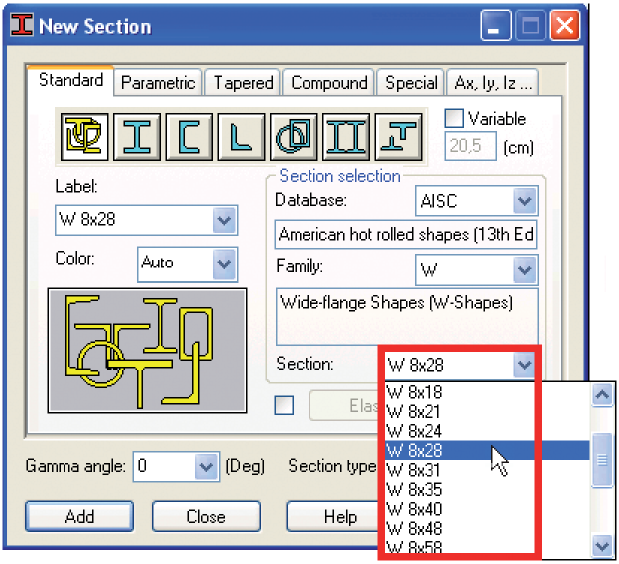


Figure 6 - New steel section definition.

10. Press **Add** to add defined section to the list of the active sections.

11. As before, set **AISC** in the **Database** field, **W** in the **Family** field and **W 10x45** in the **Section** field.

12. Press **Add**, close **New Section** and **Sections** dialog boxes.

Bars definition

In this step, we define a 2D frame using the previously defined sections.

1. Go to the Robot layouts selection box (right top corner of screen) and switch layout to the **Bars**:

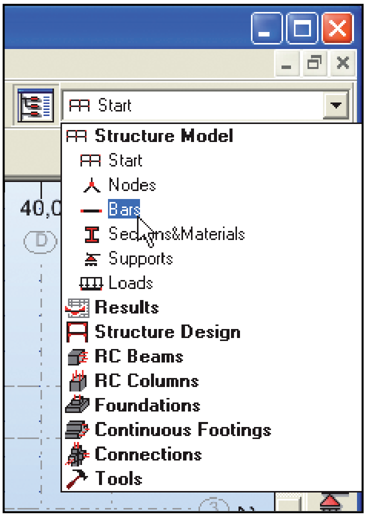


Figure 7 - Bars layout selection.

2. Go to the **Bars** dialog box and set: **RC Column** (Bar Type field), **CR30x30** (Section field).

3. Place the cursor in the **Node Coordinates > Beginning** field, switch to the graphic viewer and select graphically the beginning and end of the columns by means of the coordinates of the intersection point of the defined axes: **B1-B2, C1-C2, D1-D2** (see below):

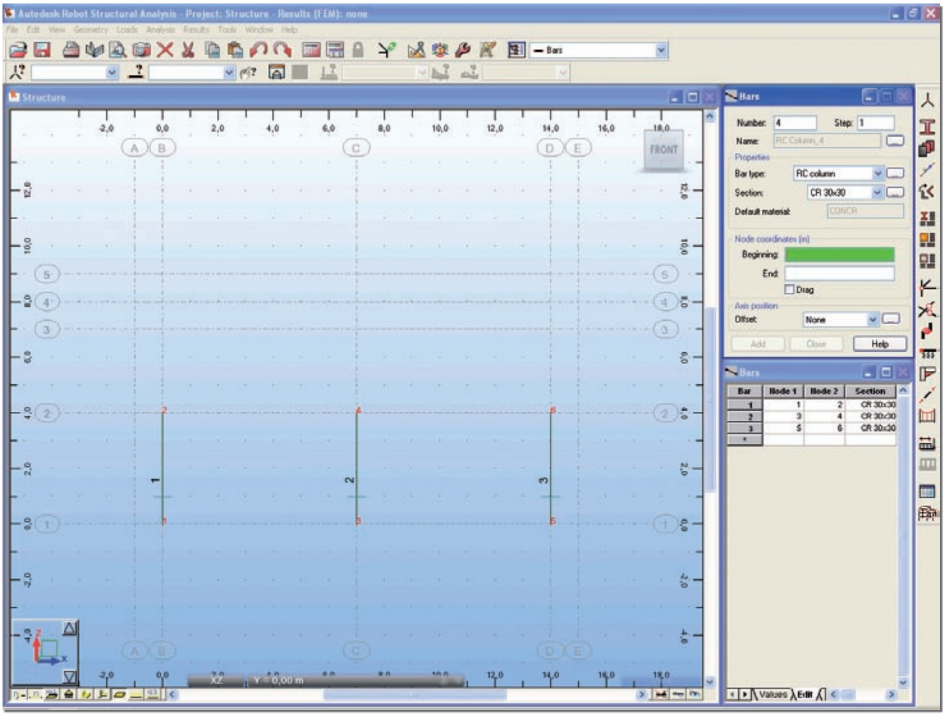


Figure 8 - View of defined columns.

4. Set the following options: **RC Beam** (Bar Type field), **BR30x60** (Section field).

5. Go to the **Node Coordinates > Beginning** and, as before, indicate the beginning and end of the beams: **A2-B2, B2-C2, C2-D2, D2-E2** (see below):

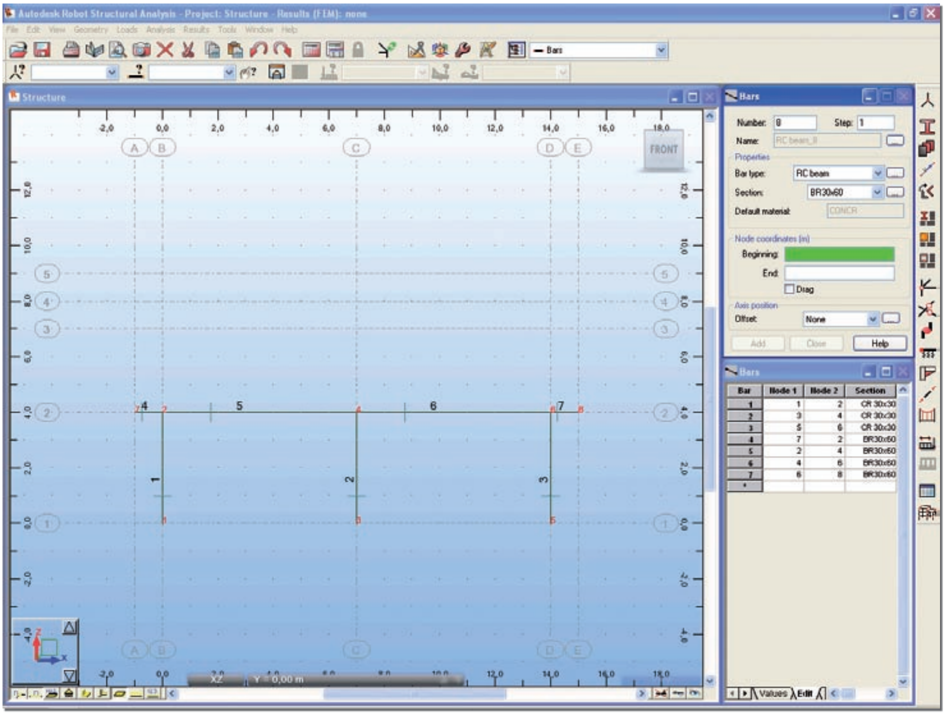


Figure 9 - View of added beams.

6. Set the following options: **Column** (Bar Type field), **W8x28** (Section field).

7. Go to the **Node Coordinates** > **Beginning** and, as before, indicate the beginning and end of the columns: **A2-A3, C2-C5, E2-E3** (see below):

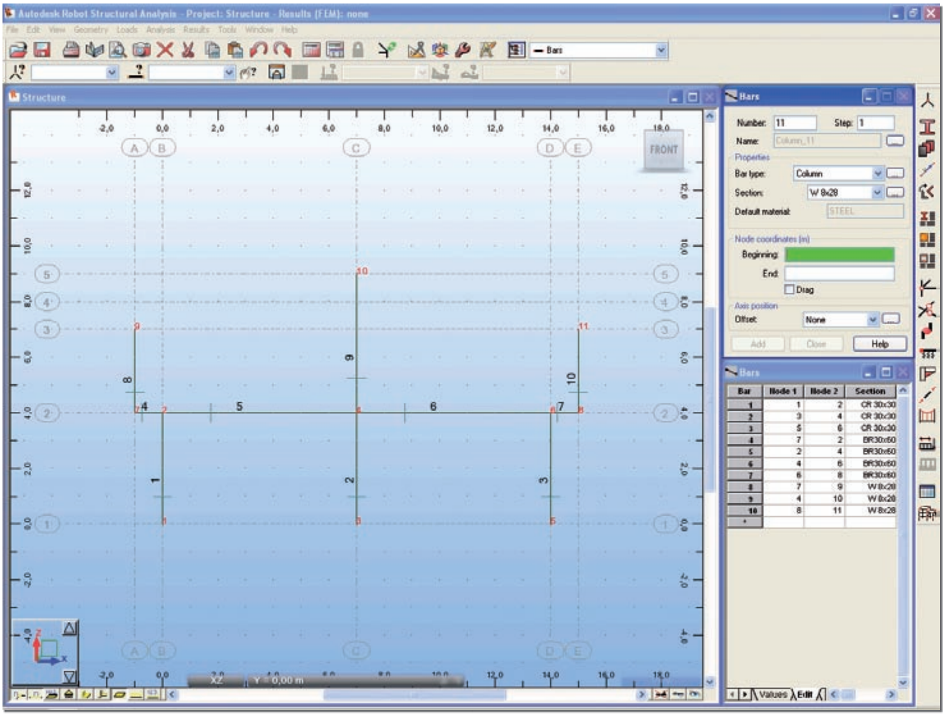


Figure 10 - View of added columns on the 2nd storey.

8. Set the following options: **Beam** (Bar Type field), **W10x45** (Section field).

9. Go to the **Node Coordinates > Beginning** and, as before, indicate the beginning and end of the beams: **A3-C5, E3-C4** (see below):

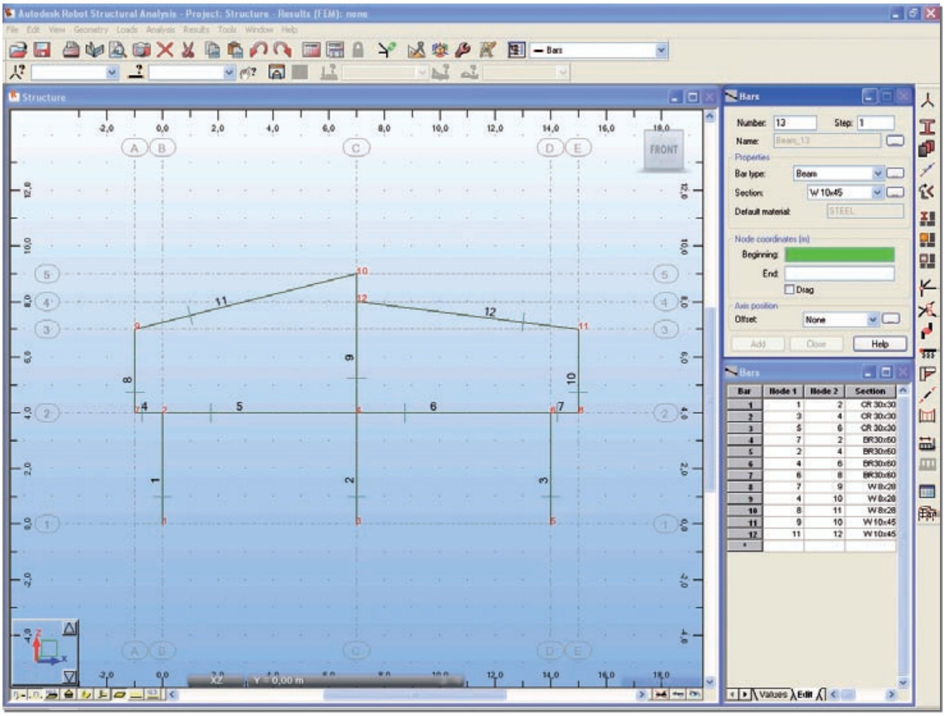


Figure 11 - Finished 2D frame view.

10. Press ,  icons at the bottom of the screen to display supports symbols and sections shapes.

11. Press  icon at the bottom of the screen to open the **Display** dialog box.

12. Select the **Bars** option (left panel) and tick **Symbols** off (right panel) to turn off display of cross sections.

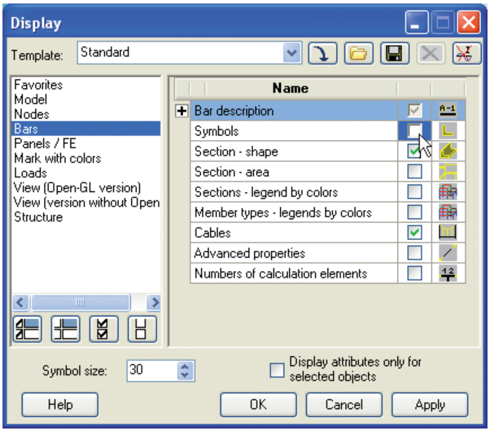


Figure 12 - Switch off bars symbols displaying.

13. Click **Apply** and **OK** to close the box. Frame 2D should look as shown below:

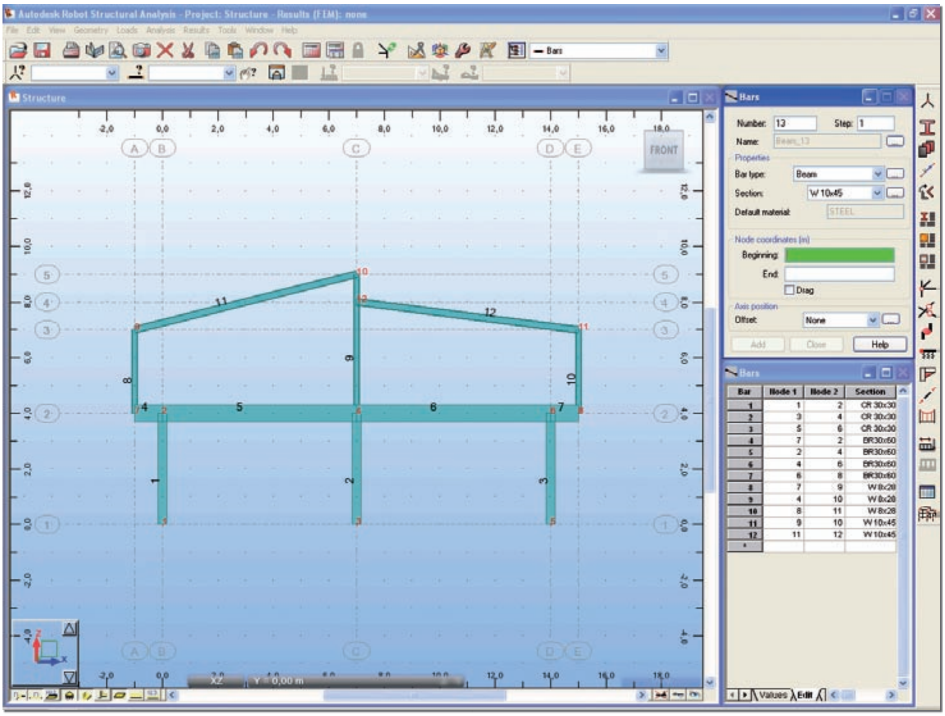


Figure 13 - 2D frame view with additional attributes - section shapes.

Supports definition

In this step, we create supports for the frame structure.

1. Go to the Robot layouts selection box (right top screen corner) and switch layout to **Supports**.

2. Go to the **Supports** dialog box and select **Fixed** in the list of supports.

3. Move the cursor to the graphical view. Note that a support symbol is shown that will allow the user to add supports to discrete nodes. To add to all nodes, press right mouse button and "select" and drag window round the nodes that have supports.

4. Click inside "current selection" box and apply.
   Notice that the supports are also now shown in the tab.

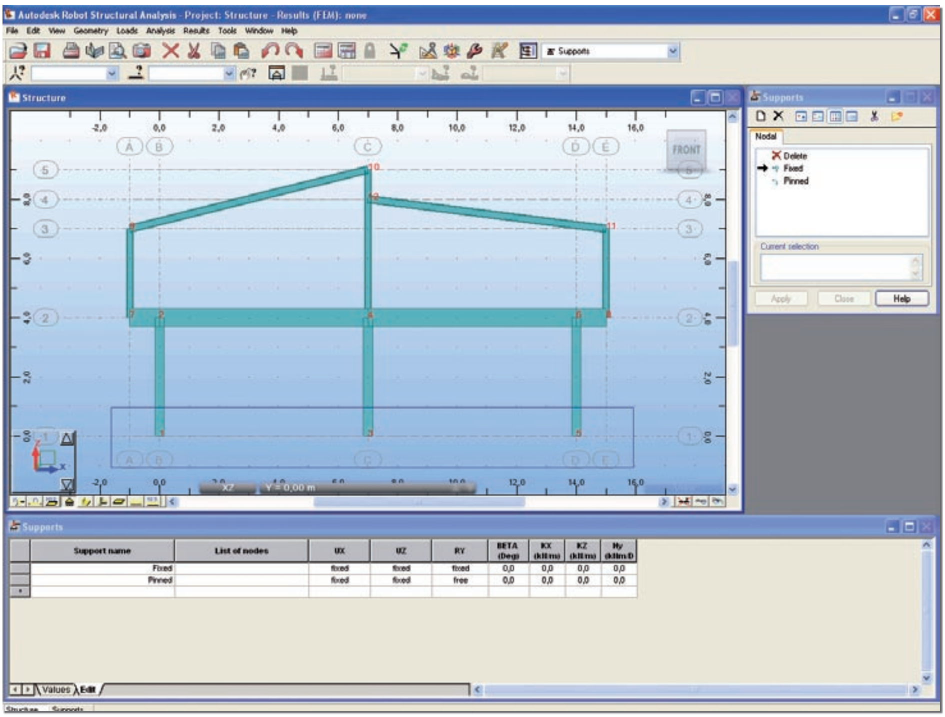


Figure 14 - Nodes selection to supports definition.

NOTE:

You may also type the relevant supports numbers (1 3 5) into the **Current Selection** field and apply.

5. Defined frame with supports should look as shown below:

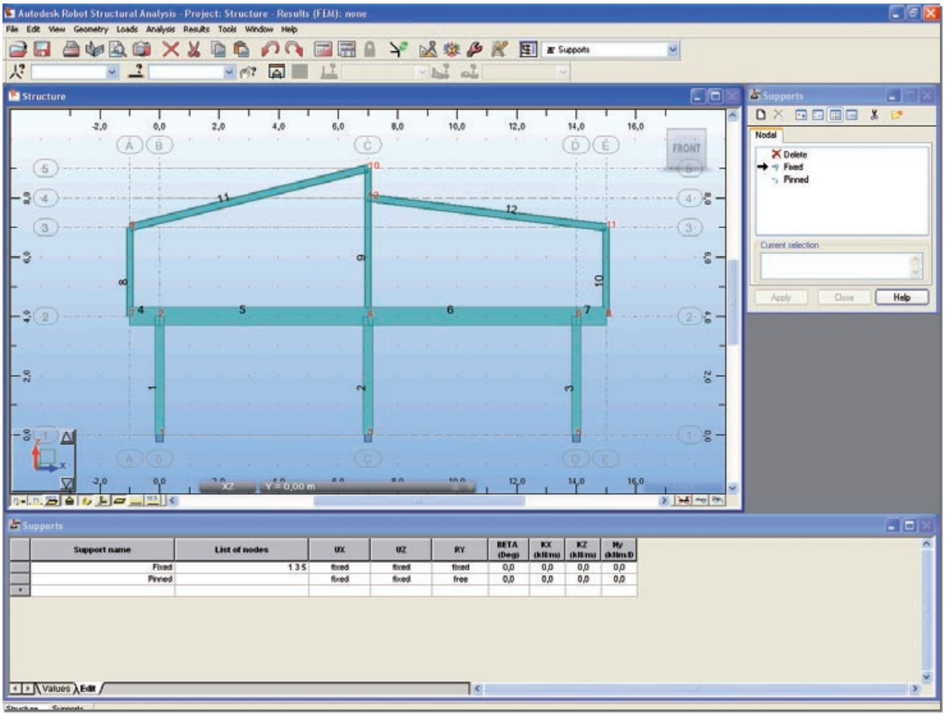


Figure 15 - Frame with defined supports view.

Load cases definition

In this step, we define names and natures of all load cases.

1. Go to the Robot layouts selection box (right top screen corner) and switch layout to the **Loads**.

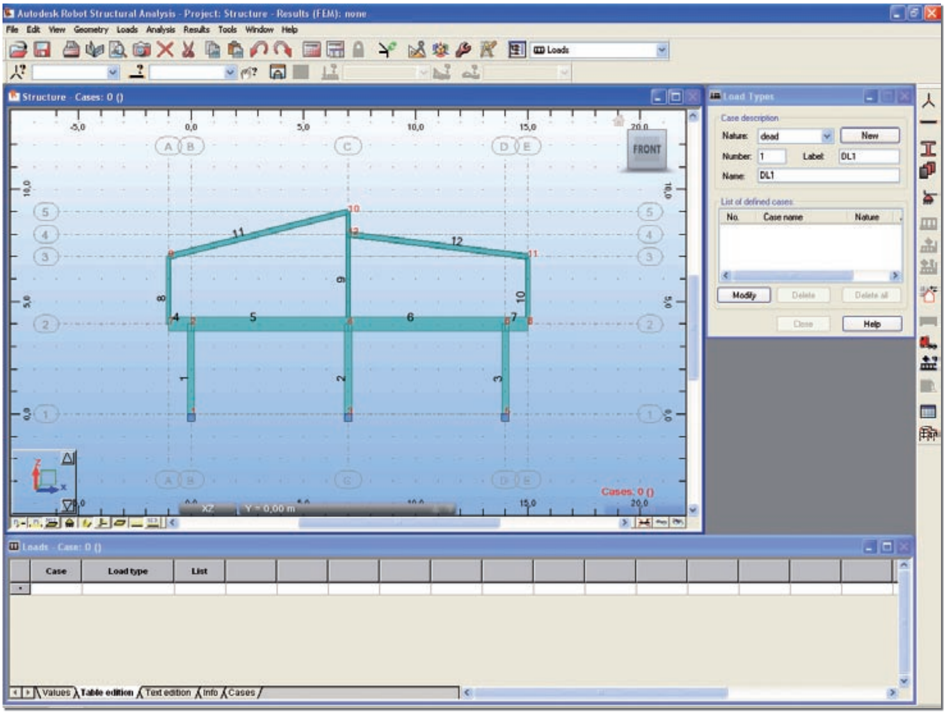


Figure 16 - Loads layout.

2. In the **Load Types** dialog box click **New** to define a dead load (self-weight) with a standard name **DL1**.

> **NOTE:**
>
> The self-weight load is automatically applied – see the table (in the "Z" direction).

3. Click **New** once again to define a dead load with a standard name **DL2**.

4. Choose in the **Nature** field **live**, then **New** to define a live load with a standard name **LL1**.

5. Click **New** once again to define a live load with a standard name **LL2**.

6. Choose in the **Nature** field wind, then **New** to define a wind load with a standard name **WIND1**.

7. Choose in the **Nature** field snow, then **New** to define a snow load with a standard name **SN1**:

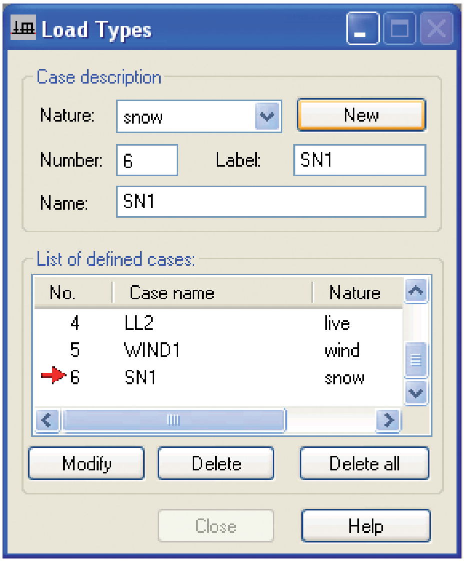


Figure 17 - Load types definition.

Definition of loads for predefined load cases

In this step, we define types and values of loads for particular load cases. Each load case can have several loads applied within it.

1. To define loads for **DL2** case select the 2nd load case in the list of defined load cases field (see below):

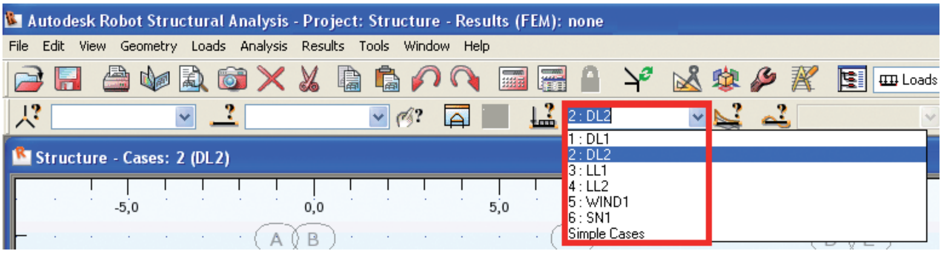
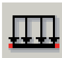
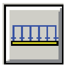


Figure 18 - DL2 load type selection.

2. Press  icon (right side of the screen) to open the **Load Definition** dialog box.

3. On the **Bar** tab press  icon to open **Uniform** load dialog box, then type value of load **–10 [kN/m]** in the **pZ** field, click **Add** and close the box:

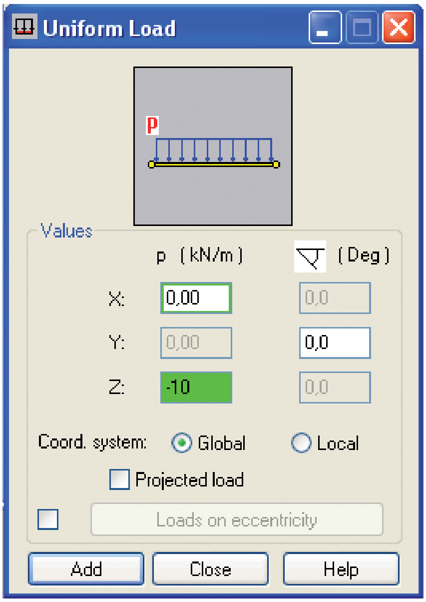


Figure 19 - Uniform load definition for DL2 case.

4. Select beams **(no. 4 5 6 7)** – indicate them or just type **4to7** in field **Apply** to and click **Apply** (as before its possible to window them too)

5. Press  icon at the bottom of the screen to turn display **Load value descriptions** on.

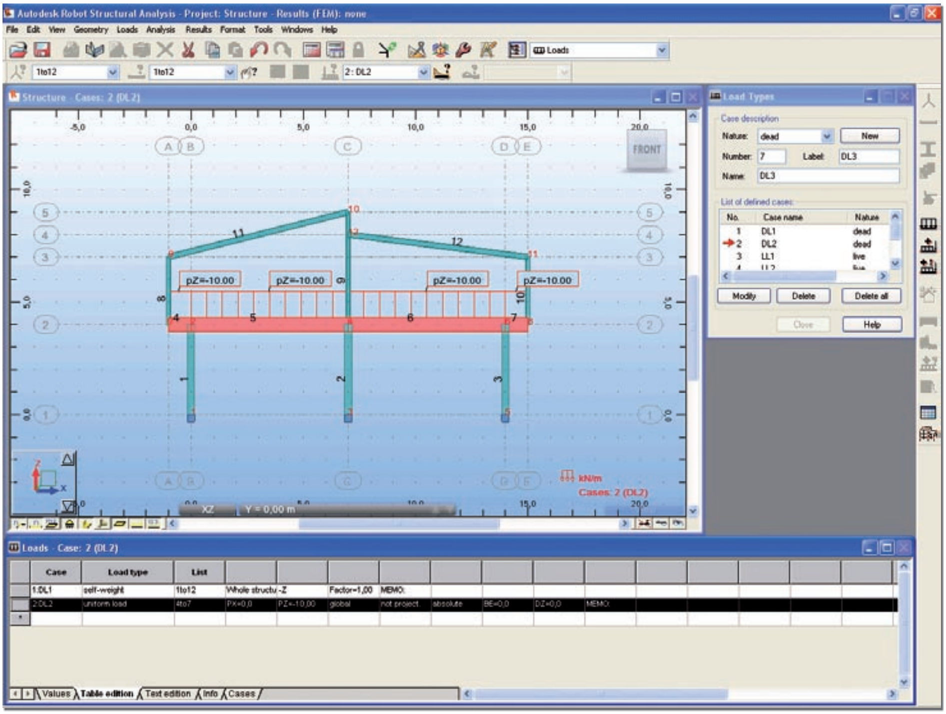
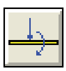


Figure 20 - DL2 load case view.

6. To define loads for **LL1** case select the **3rd load case** in the list of defined load cases field.

7. On the **Bar** tab press  icon to open **Bar Force** dialog box, then type value of load **–20 [kN]** in the **FZ** field.

8. On the **Coordinate** field leave **x=0,50** value (defined force will be applied halfway along the member), click **Add** and **Close** the box.

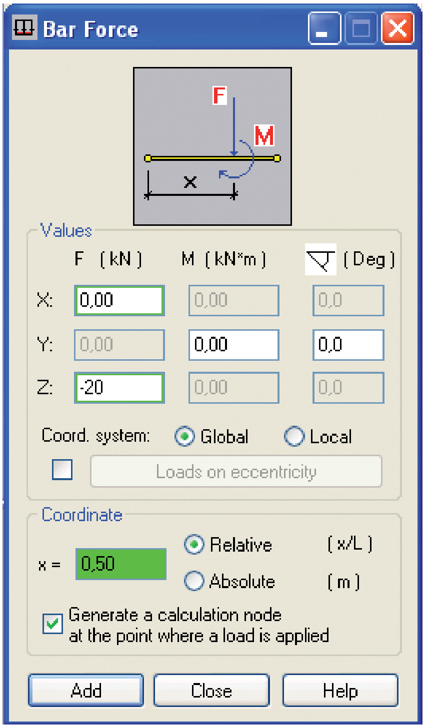


Figure 21 - Bar force definition for LL1 case.

9. Select beams **(no.5)** – indicate them or just type **5** in field **Apply** to and click **Apply**.

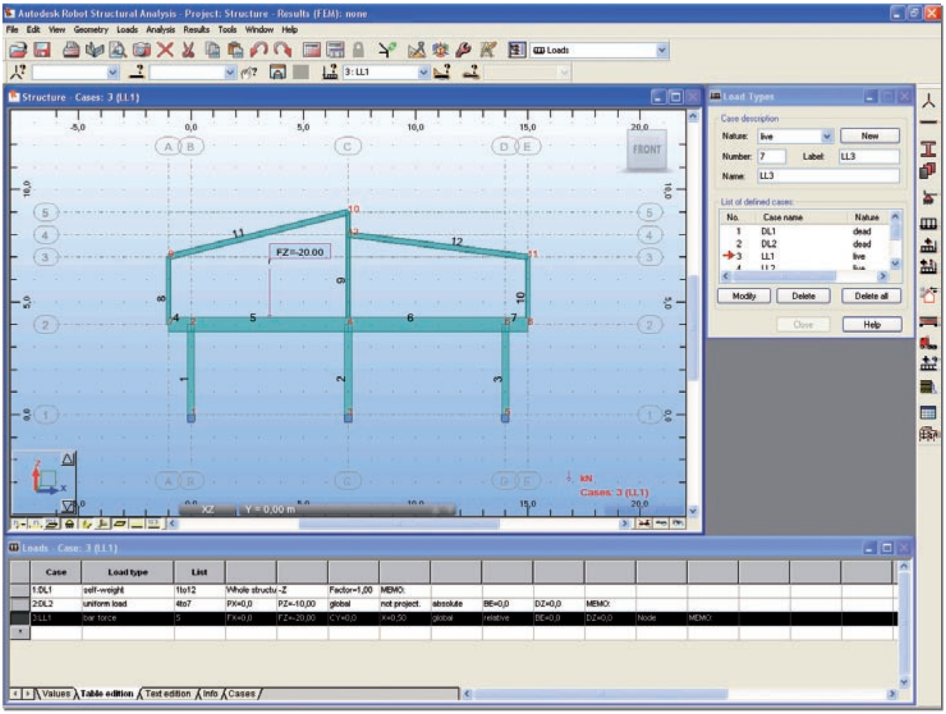


Figure 22 - LL1 load case view.

10. In the same way you may define for **4th:LL2** load case bar force **–30 [kN]** on the half length of the bar **No.6**.

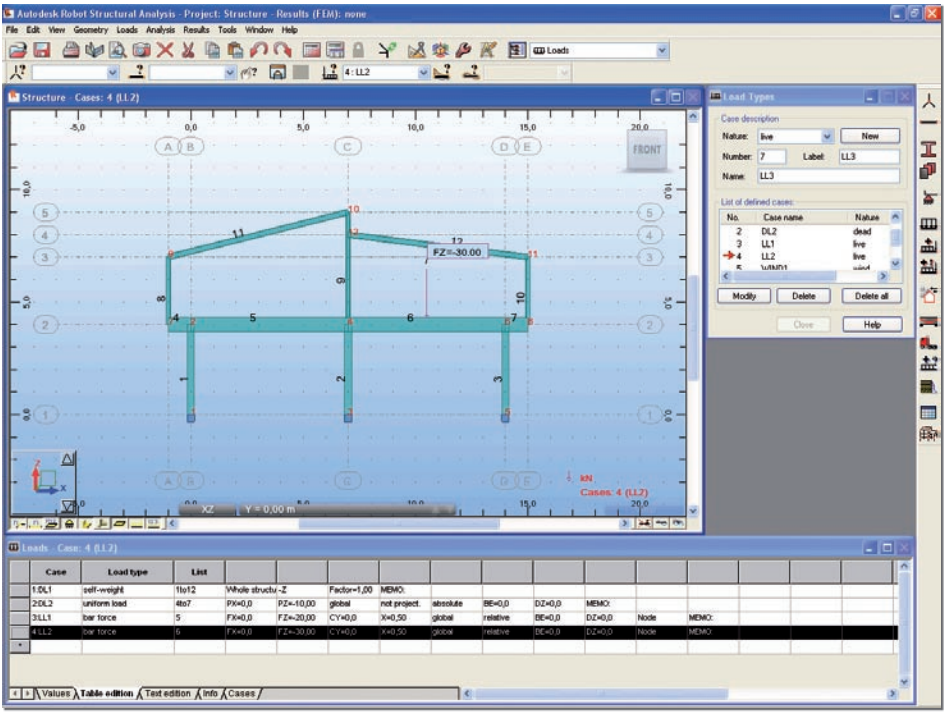
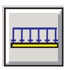


Figure 23 - LL2 load case view.

11. To define loads for **WIND1** case select the **5th** load case in the list of defined load cases field.

12. On the **Bar** tab press  icon to open **Uniform Load** dialog box, then type value of load **–1,5 [kN/m]** in the **pZ** field, tick Coord. system – **Local** on, then click **Add** and close the box.

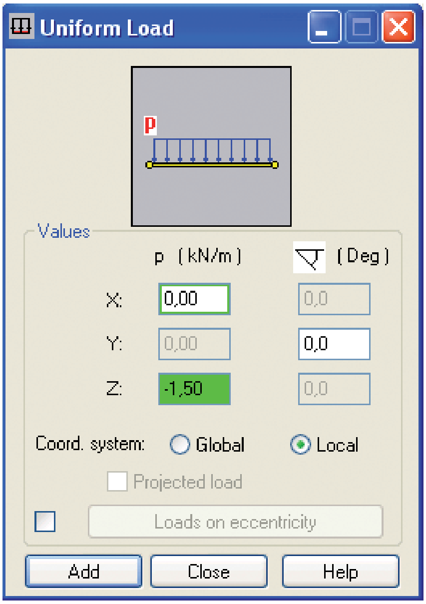


Figure 24 - Uniform load definition for WIND1 case.

13. Select beam **(no.1)** – indicate them or just type **1** in field **Apply** to and click **Apply**.

14. In the same way define uniform loads: **-2,50 [kN/m]** for bar **no. 8** and -**3 [kN/m]** for bar **no. 11**.

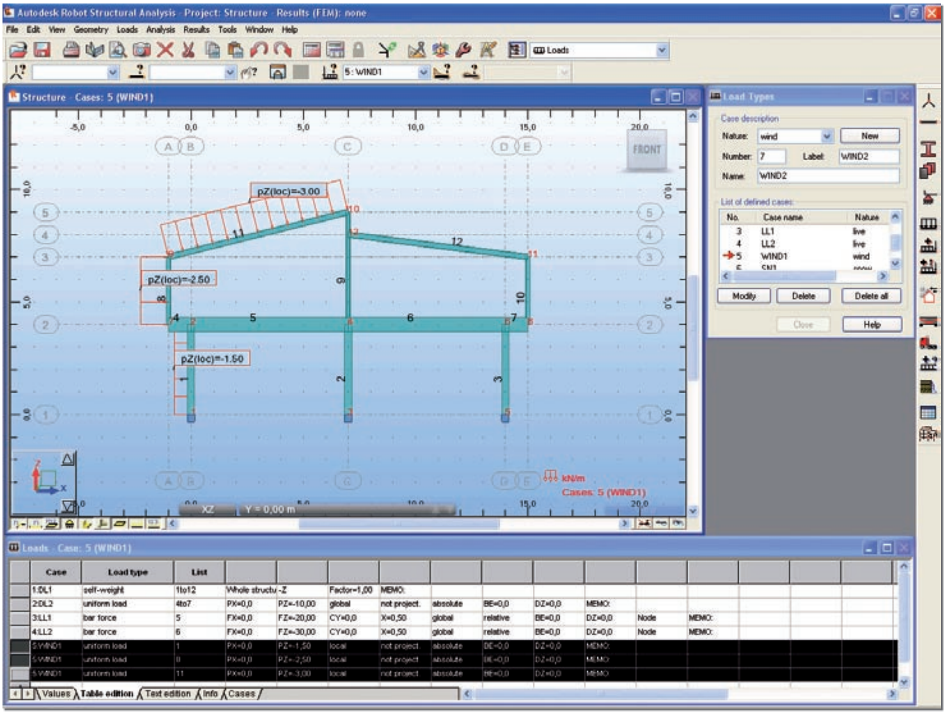
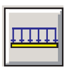


Figure 25 - WIND1 load case view.

15. To define loads for **SN1** case select the **6th** load case in the list of defined load cases field.

16. On the **Bar** tab press  icon to open **Uniform Load** dialog box, then type value of load **–2,50 [kN/m]** in the **pZ** field, tick **Projected load** on, then click **Add** and **close** the box.

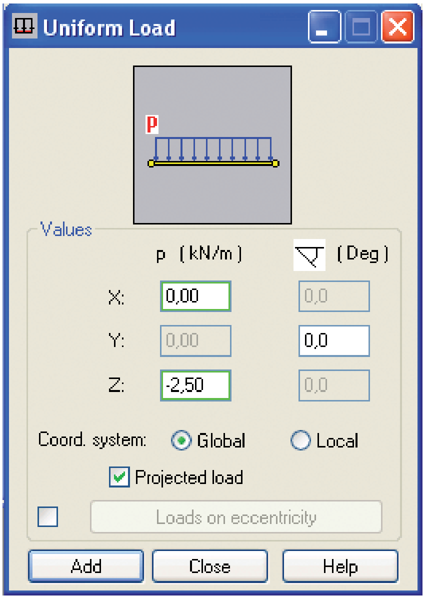


Figure 26 - Uniform load definition for SN1 case.

17. Select beams **(no.11 and 12)** – indicate them or just type **11 12** in field **Apply** to and click **Apply**.

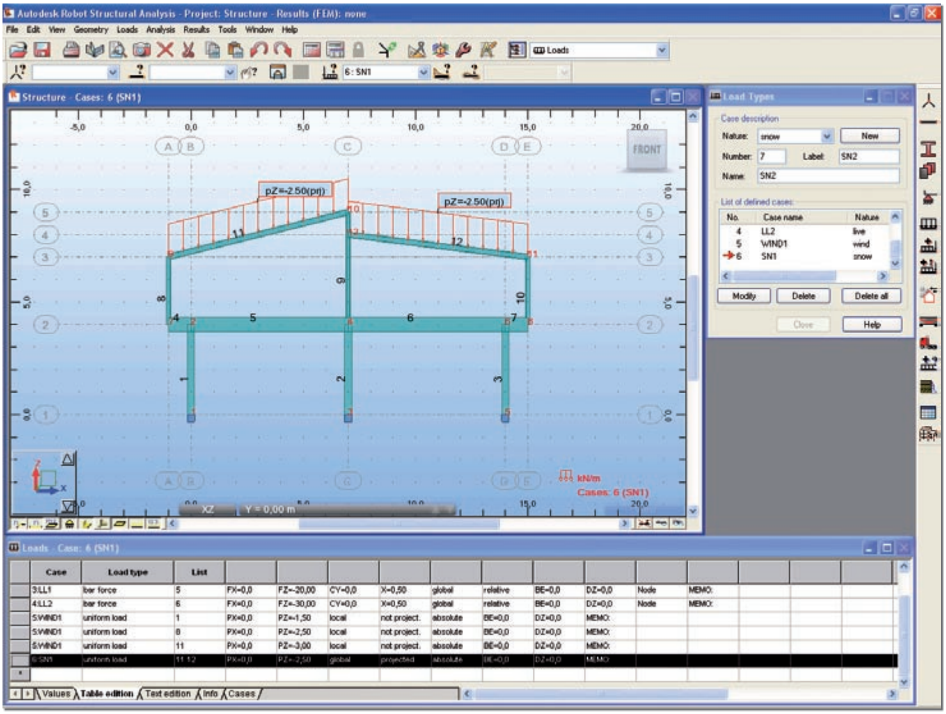


Figure 27 - SN1 load case view.

18. Close the **Load Definition** dialog box.

19. Go to the Robot layouts selection box (right top screen corner) and switch layout to **Start**.

Now, using the existing 2D frame, we define a 3D frame structure with slabs and walls.

Changing the structure type

So far we have worked in 2D. In this step, we open new module **Shell** design to enable wall and slab definition and work in 3D.

1. Select **Geometry** > **Structure type** from the menu, which opens the window presented below:

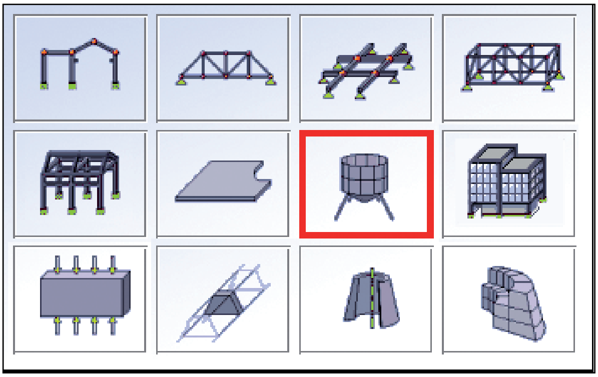
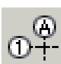


Figure 28 - Structure type selection window.

2. Select **Shell** design icon (the third one from the left in the second row) to change structure type from 2D frame to shell. The program prompts you to save your work so far.

Definition of additional structural axis

In this step, w define additional structural axes on in the third axis - the Y direction.

1. Press  icon (right side of the Robot screen) or select **Geometry** > **Axis Definition…** from the menu to open the **Structural Axis** dialog box.

2. Set **Numbering** as **Define** and type in the next field (on the right side) **L1**.

3. On the **Y** tab in the **Position** field type **0** and press **Insert** button, in the same way enter following axis coordinates as shown below:

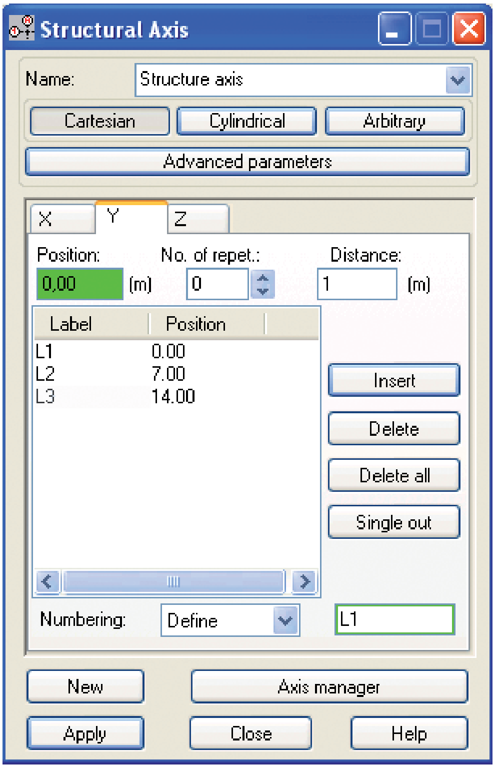
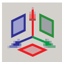


Figure 29 - Definition of structural axis in Y direction.

4. Press **Apply** and close the **Structural Axis** dialog box.

Copying existing frame

In this step, we copy the 2D frame to generate a 3D structure. When we copy that frame, all attributes attached to it (loads, sections, supports etc) are also copied:

1. Press  icon (top of the Robot screen) to open the **View** dialog box, then  or choose **View > Work in 3D > 3D xyz** from the menu to select the isometric structure view (see picture below):

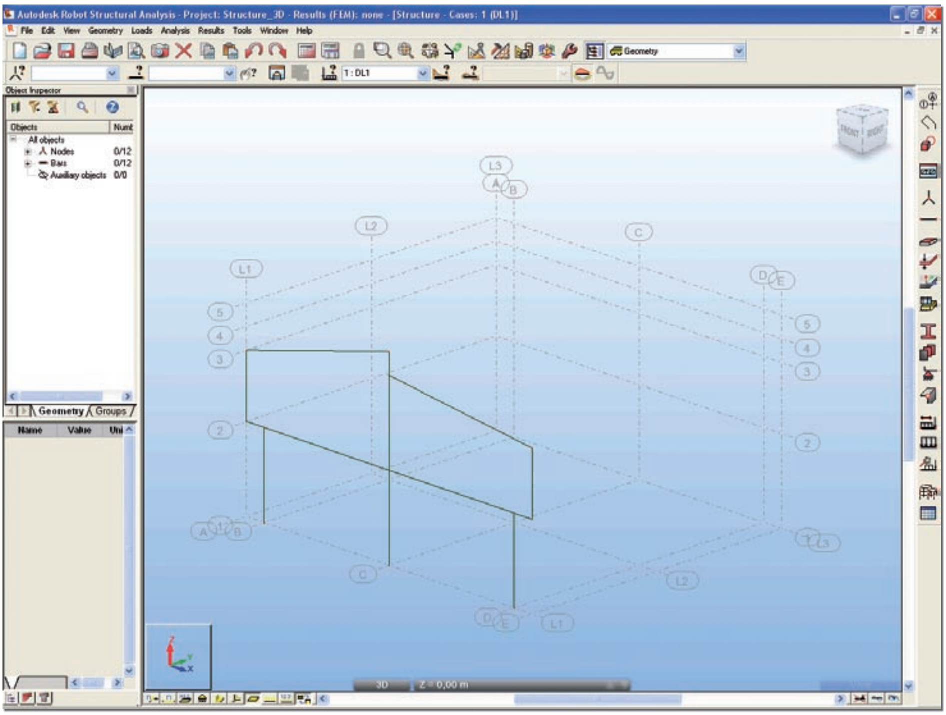
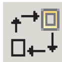


Figure 30 - 3D view of defined frame.

2. Press **CTRL+A** to select all of the structure.

3.  **Edit** then  **Translation** – press this icons or select the **Edit/Edit/Translate** option from the menu to open the **Translation** dialog box.

4. In **Translation** vector type the value to copy in each axis as shown below:

5. In **Edit** mode tick option **Copy** on and **Number of repetitions** set as **2** (see below):

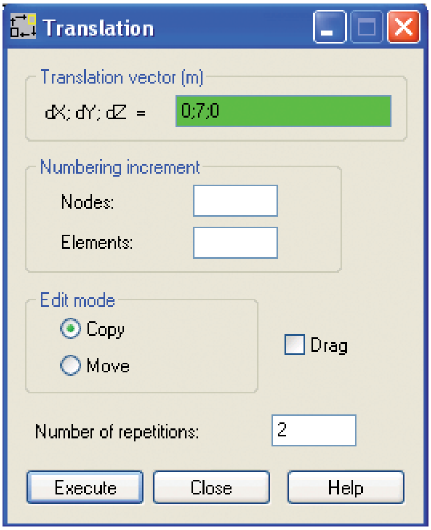


Figure 31 - Copy of the frame - translaction vector definition.

NOTE:

You may define translation vector graphically by indicating the beginning and end of the translation vector (in isometric structure view).

6. Copied frames should look as shown below:

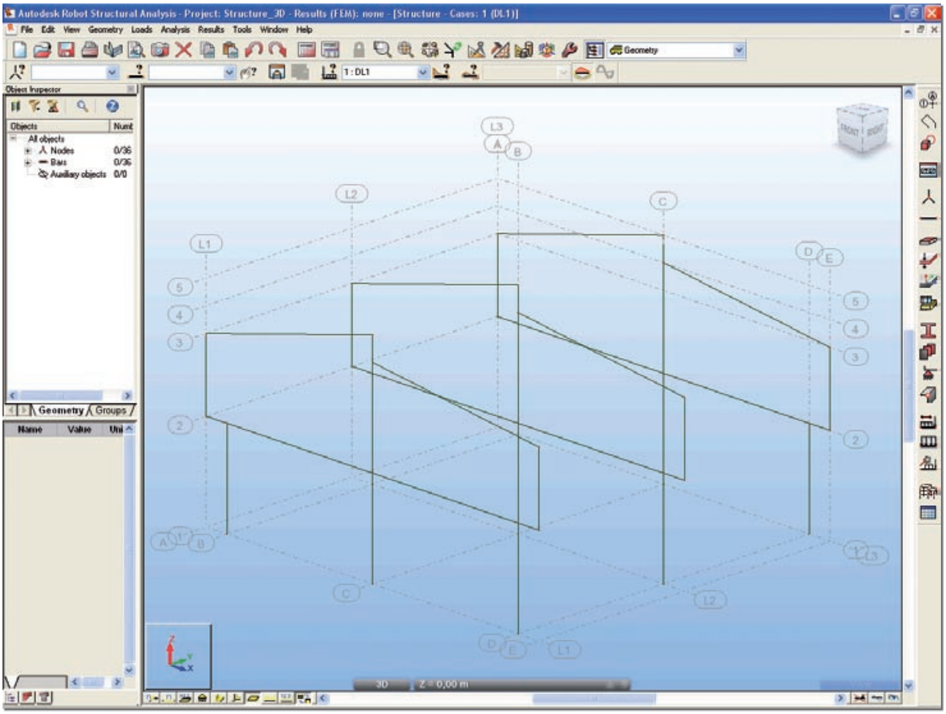


Figure 32 - Copied frame view.

Definition of lateral beams

In this step, we define beams to join the selected frames together:

1. Go to the Robot layouts selection box (right top screen corner) and switch layout to the **Bars**.
2. Click , icons at the bottom of the screen to turn node and bars numbers display off.
3. In **Bars** dialog box set **Bar type**: as **Beam**.
4. Define **Section**: as **W8x15**.

NOTE:

If the **W8x15** section is not available on the list, you should click the **(...)** button located beside the **Section** field and add this section from the database. In opened **New** section dialog box, in **Section** selection field for **Database: AISC** – choose **Family: W** then Section: **W8x15**. Click **Add** and **close** the box (see below)

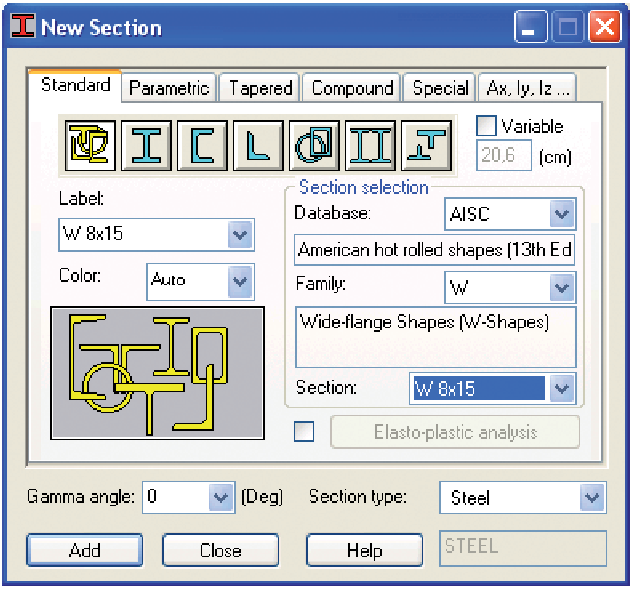


Figure 33 - New steel section definition.

5. Press **Add** - new section type to be added to the active **Sections** list.

6. Close **New Section** dialog box.

7. Place the cursor in the **Node Coordinates** > **Beginning** field, then switch to the graphic viewer and select graphically the beginning and end of the bars by means of the coordinates of the intersection point of defined axes:
   - (E,L1,3) and (E,L2,3)
   - (E,L2,3) and (E,L3,3)
   - (C,L1,5) and (C,L2,5)
   - (C,L2,5) and (C,L3,5)
   - (A,L1,3) and (A,L2,3)
   - (A,L2,3) and (A,L3,3)

8. The structure with lateral tie beams should look as shown below:

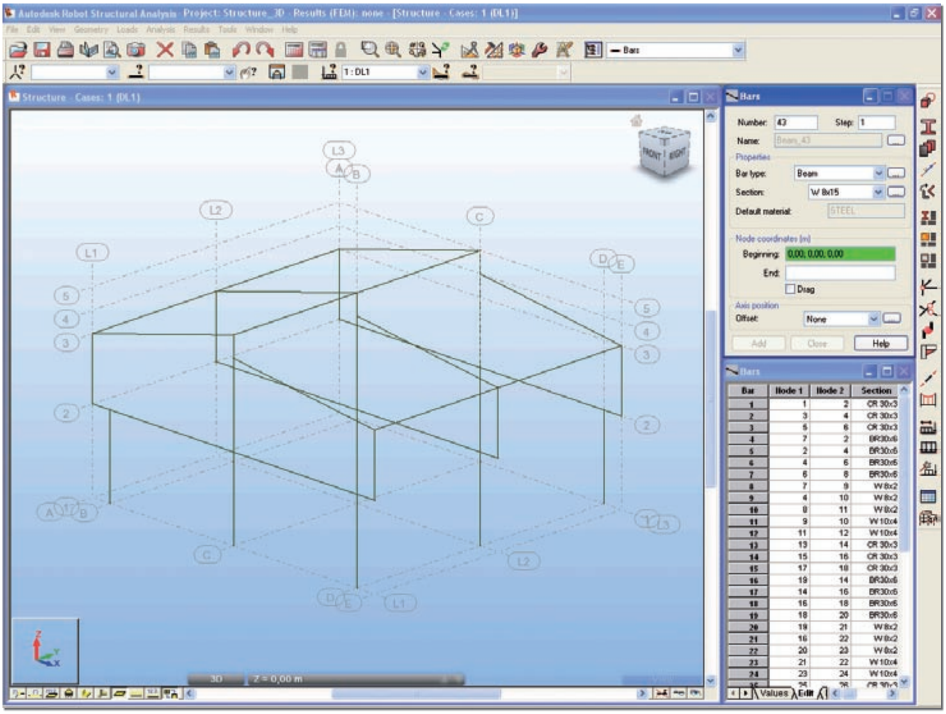


Figure 34 - View of structure with lateral tie beams added.

9. Go to the Robot layouts selection box (right top screen corner) and switch layout to the **Geometry**.

Definition of slab

In this step, we learn how to define a slab, first we define a contour (shape) of the plate then we assign physical properties to it. Firstly, we need to define the elevation of the structure that we wish to work on – we do this with a "work plane".

1. To set new a work plane press the icon  provided in the bottom left corner of the screen which represents the active work plane – a **View** dialog box will be opened.

2. Press **2D** and **XY** buttons and select **"Structure axis 2" 4,00 (m)** – as shown below. Once these options are selected the structure is set on the **XY** plane at the recently defined **Z** coordinate (e.g. Z=12); only structure components from this plane are displayed.

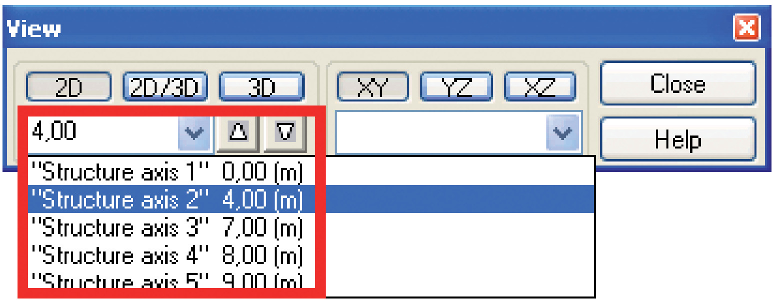


Figure 35 - New work plane definition (in axis no 2).

3. Close the **View** dialog box.

4. Press  icon (right side of the Robot screen) or select **Geometry > Objects > Polyline-contour** from the menu to open the **Polyline-Contour** dialog box.

5. Indicate required points of the contour in the graphic viewer by cursor as follow: (A,L1); (E,L1); (E,L3); (A,L3); (A,L1) – note, always select the start and end point to "close"the contour.

6. Close the **Polyline-Contour** dialog box, the defined contour should look as shown below:

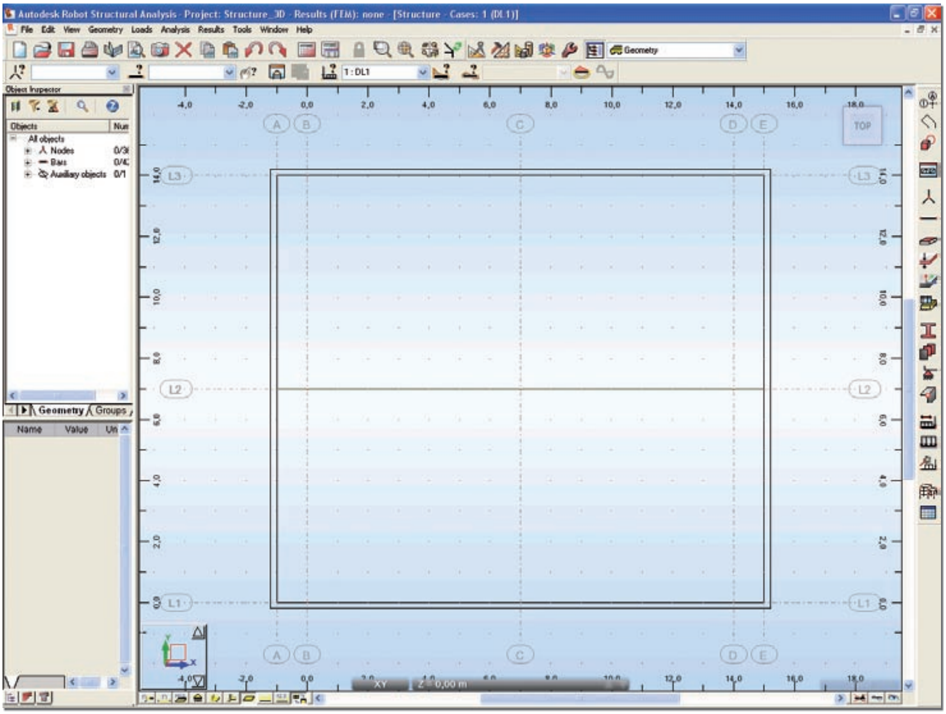


Figure 36 - Defined contour of panel view.

7. Press  icon (right side of the Robot screen) or select **Geometry** > **Panels** from the menu to open the **Panel** dialog box.

> **INFO**
>
> A surface in Robot is called a "panel". So if you want to create a slab or wall, you must define an appropriate panel.

8. In the **Panel** dialog box set parameters: **Panel** in **Contour type** field, **Internal point** (Creation with field), **none** (Reinforcement field), **TH12_CONCR** (Thickness field) and **Shell** (Model field) - see picture below:

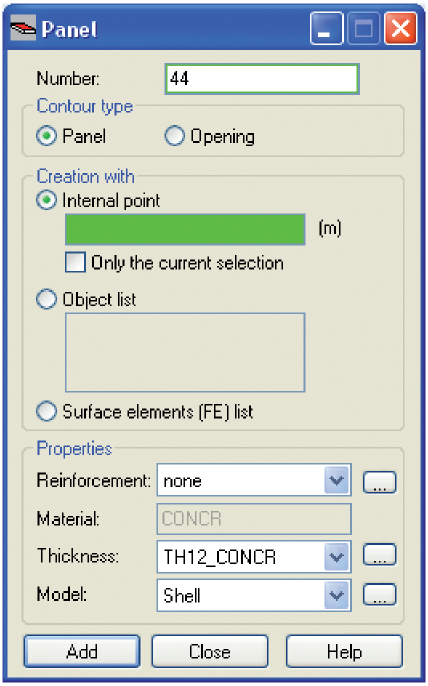


Figure 37 - Panel definition settings.

NOTE:

In the **Reinforcement** option you can define a reinforcement type. However this option is unavailable in Robot Free, in the Slab design module, in the full commercial version of the program, you can perform a complete design process of an RC slab (including calculation of provided reinforcement and deflection verification with reinforcement and cracking taken into consideration).

9. To assign the recently defined panel to the selected contour set the cursor in the **Internal Point** field then move it to the graphical viewer and click on a point located within the area of the slab.

10. Close the **Panel** dialog box.

11. To set the 3D view of the structure, click the icon  (bottom left corner of the screen), then in **View** dialog box press **3D** button and close the box.

12. Press ,  icons at the bottom of the screen to turn on appropriate display of supports and sections shapes.

13. The defined structure with slabs should look as shown below:

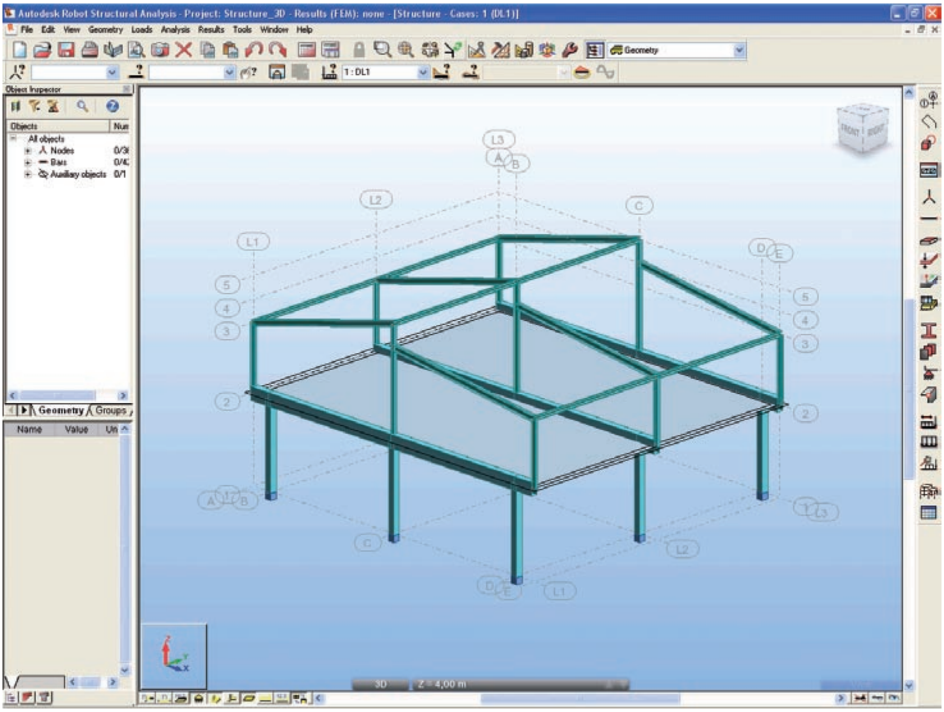


Figure 38 - Structure with slab view.

Offset definition

In this step we define offset between axis slab and RC beams:

1. Press  icon at the bottom of the screen to open **Display** dialog box.

2. Select the **Panels/FE** option (left panel) and tick **Panel thickness** on (right panel) to turn displaying thickness of panels on.

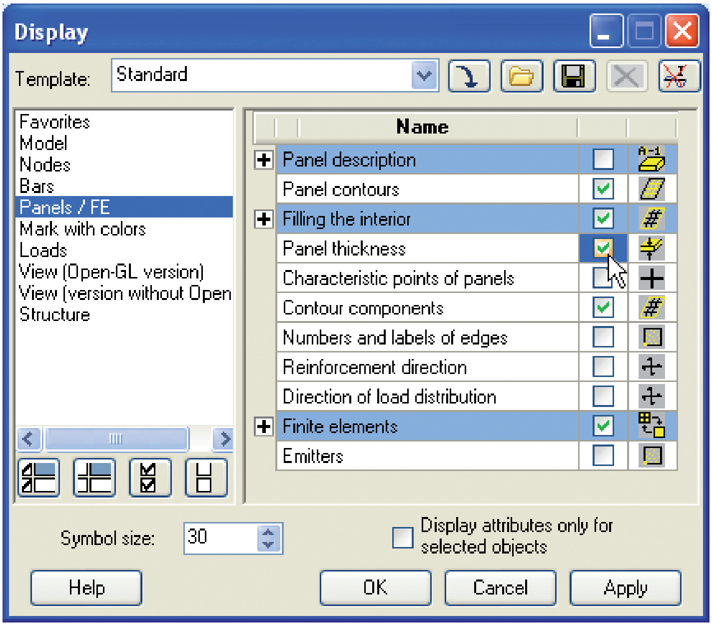


Figure 39 - Switch thickness of panel display on.

3. On the structure drawing, which appears on the screen we can see that neutral axis of the RC beams and RC slab are on the same level with no eccentricity by default.

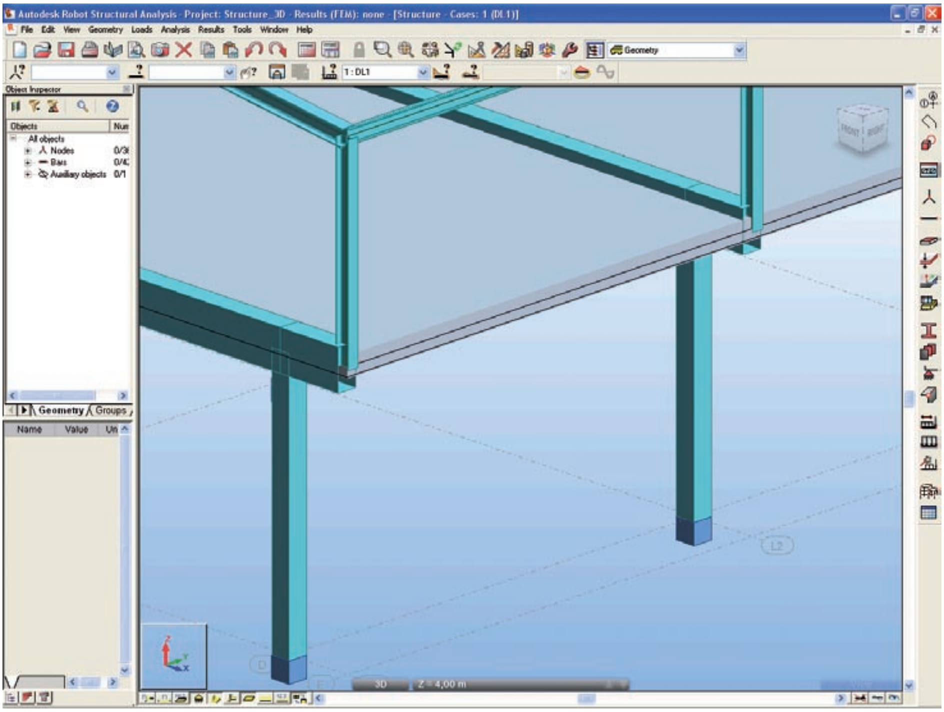


Figure 40 - Default view of RC beams and RC slab on the same level with no eccentricity.

4. Select **Geometry** > **Additional Attributes** > **Offsets…** from the menu then press  **New** offset definition button to open the **New Offset** dialog box.

5. In the **New Offset** dialog box set parameters: **Offset_1** in **Label** field, **-45,24** [cm] (this is half of RC slab height plus half of RC beams height) in the **Beginning** – **UZ**: and **End** - **UZ**: fields and tick **Global** (Coordinate system field) on - see picture below:

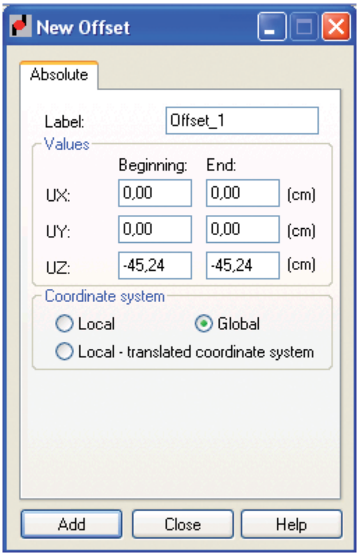


Figure 41 - New offset (between beams and slab) definition.

6. **Add** and **close** the **New Offset** dialog box.

7. Press  icon (left,top corner of the Robot screen) to open the **Selection** dialog box.

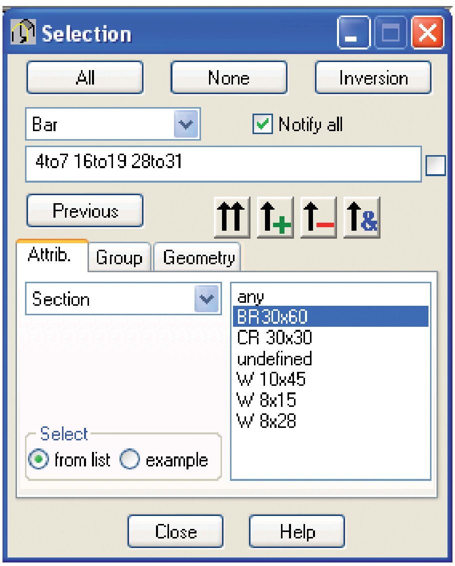


Figure 42 - Selection of RC beams by filters.

8. Indicate **Section** as selection criterion and click **B R30x60** in the list-box above.

9. Press the button to enter defined selection to the edit field and mark red concrete beams, to which offsets will be attributed. **Close** the box.

10. In the **Offsets** dialog box place the cursor in the **Current Selection** field – numbers of selected beams will appear.

11. Press **Apply** and **Close** to finish offset definition.

12. To display defined offset beams press icon at the bottom of the screen to open **Display** dialog box.

13. Select the **Favorites** option (left panel) and tick **Offsets** on (right panel), then press **Apply** and **OK** – result should look as shown below:

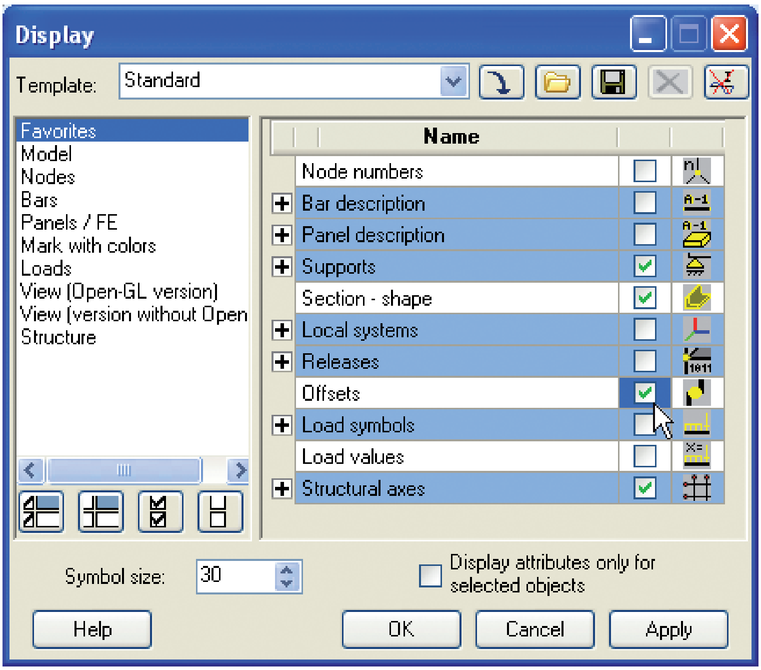


Figure 43 - Switch offsets display on.

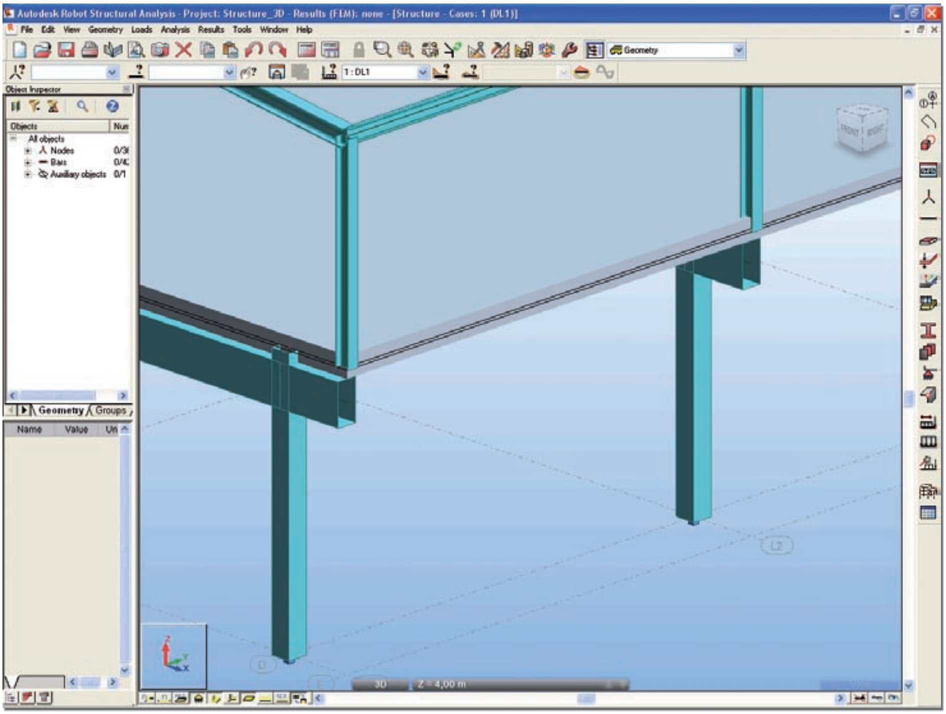


Figure 44 - View of RC beams and RC slab with offset defined.

Definition of a wall

In this step, we learn how to define a wall, first we define a contour then we assign physical properties to it:

1. To set a new work plane press the icon  provided in the bottom left corner of the screen – **View** dialog box will be open.

2. Press **2D** and **YZ** buttons and select **"Structure axis D" 14,00 (m)**. Once these options are selected the structure is set on the YZ plane at the recently defined X coordinate (e.g. X=40); only structure components from this plane are displayed.

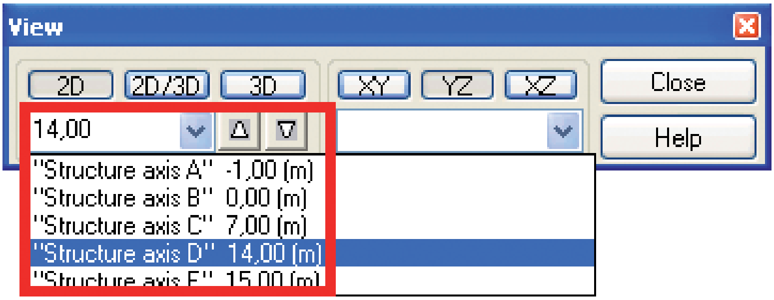
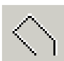


Figure 45 - New work plane definition (in axis D).

3. **Close** the **View** dialog box.

4. Press ,  icons at the bottom of the screen to turn display of supports symbol and sections shape off.

5. Press  icon (right side of the Robot screen) or select **Geometry** > **Objects** > **Polyline-contour** from the menu to open the **Polyline-Contour** dialog box.

6. Indicate required points of the contour in the graphic viewer by cursor as follow:
   - (14; 0; 0)
   - (14; 2; 0)
   - (14; 2; 3)
   - (14; 4; 3)
   - (14; 4; 0)
   - (14; 10; 0)
   - (14; 10; 3)
   - (14; 12; 3)
   - (14; 12; 0)
   - (14; 14; 0)
   - (14; 14; 4)
   - (14; 0; 4)

7. Press **Apply** and close the **Polyline-Contour** dialog box.

8. Defined contour should look as shown below:

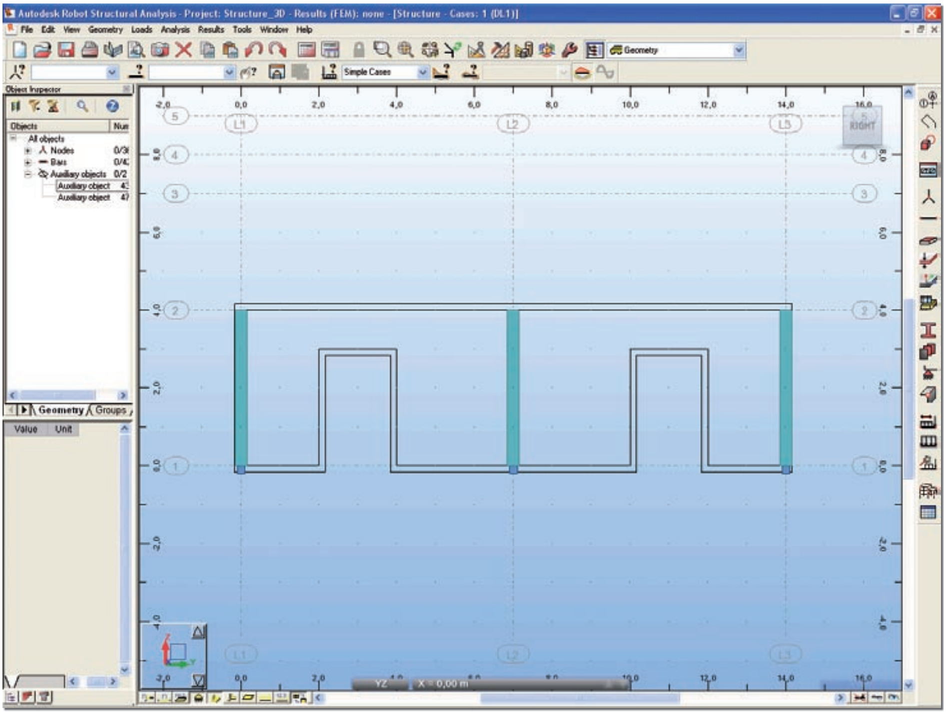


Figure 46 - Contour of defined panel view.

9. Press  icon (right side of the Robot screen) or select **Geometry/Panels** from the menu to open the **Panel** dialog box.

10. In the **Panel** dialog box set parameters: **Panel in Contour type** field, **Internal point** (Creation with field), **none** (Reinforcement:) and **Shell** (Model field).

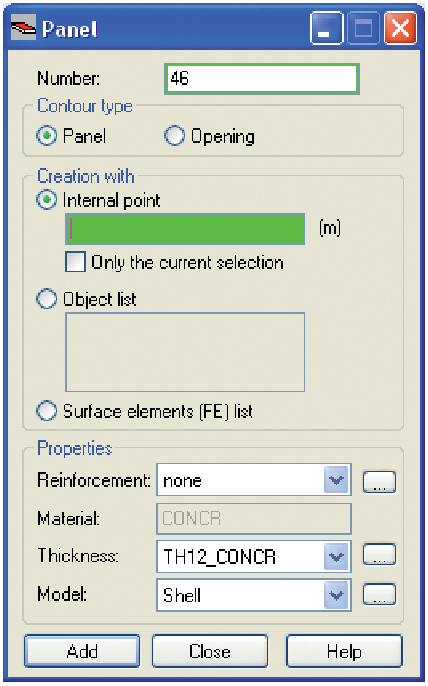


Figure 47 - Panel definition settings.

11. To define thickness of the panel press  button (right to the **Thickness** field), then in the **Th** = field type **30** (cm), in Label: field name **TH30,00** will be automatically updated, **Add** and **close** the **New Thickness** dialog box (see below).

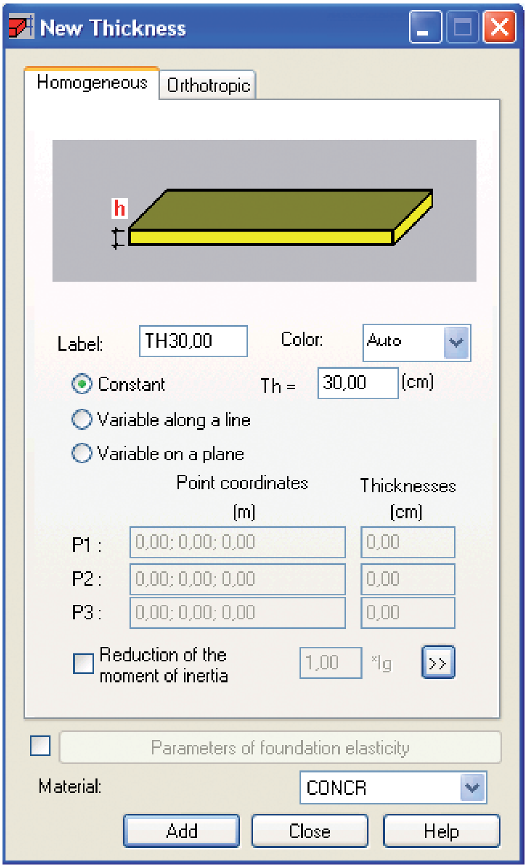


Figure 48 - New panel thickness definition.

12. To apply chosen panel properties to the recently defined contour, in the **Panel** dialog box, set the cursor in the **Internal point** field, then move the cursor to the graphical viewer and indicate a point within the boundaries of the panel contour.

13. Close the box.

Definition of wall support

In this step, we create a line support underneath the wall.

1. Press  icon (right side of the Robot screen) to open the **Supports** dialog box.

2. Go to the **Linear** tab then select the **Fixed** support type.

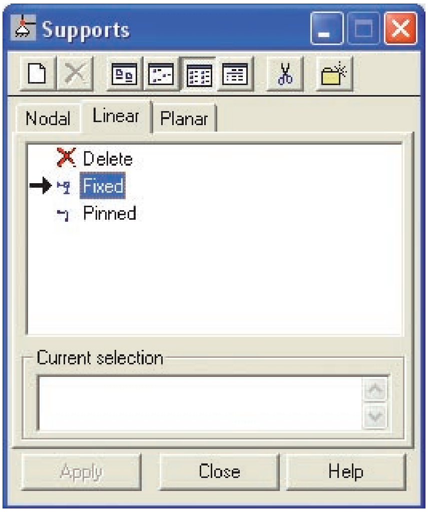


Figure 49 - Linear support of the wall definition.

3. Switch to the graphical viewer, point at the bottom slab edges and click once (when it is highlighted) on the edge.

4. Close the box.

5. To set 3D view of structure click the icon  (bottom left corner of the screen), then in **View** dialog box press **3D** button and close the box.

6. Press ,  icons at the bottom of the screen to turn on display of supports symbol and sections shapes.

7. Defined structure with slab and wall should look as shown below:

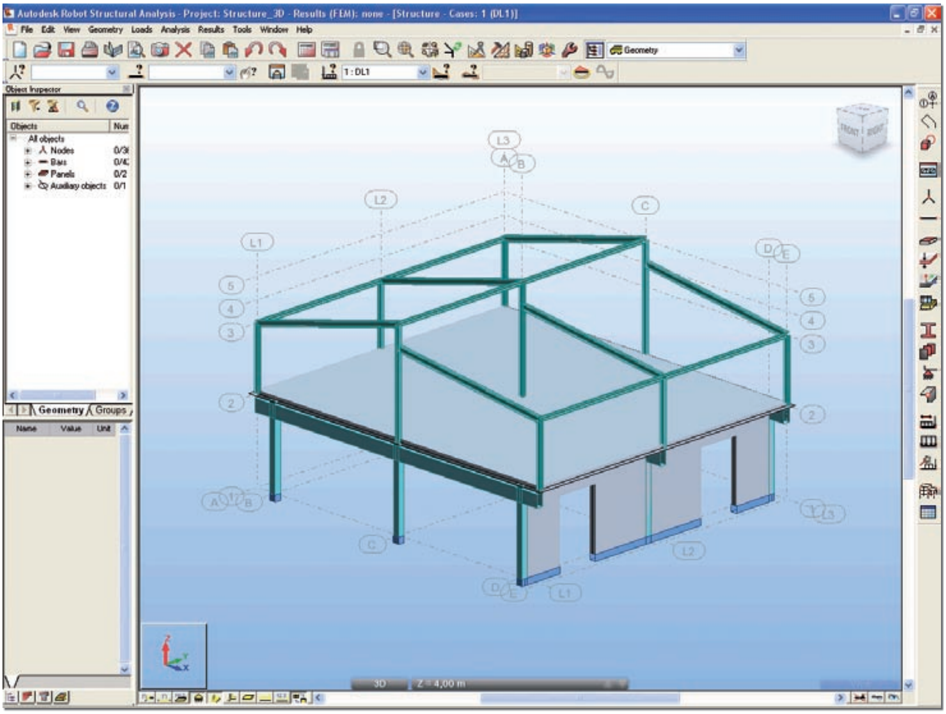
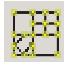


Figure 50 - View of entire structure (with slab and wall defined).

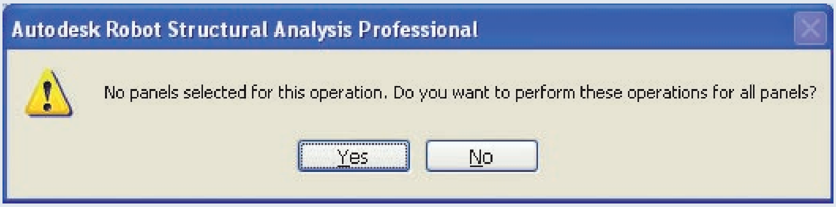
Meshing parameters definition

In this step, we learn how to mesh the panels, since Robot is finite element solution. Robot has extremely powerful meshing algorithms to cope with virtually any shape of structure or mesh requirement.

NOTE:
The meshing of the plates and shells are made automatically (with standard parameters), after starting the calculation process. However it is often desirable to mesh the structure manually - often to obtain better quality meshing we should to modify the standard settings.

1.  **Options of FE Mesh Generation** then  **Meshing Options** – click these icons or select the **Analysis > Calculation model > Meshing Options** option from the menu to open the **Meshing Options** dialog box.

NOTE:
Because there were no panels selected, the program will ask you the question:



Answer **Yes** to select all panels.

2. Select **Complex mesh generation (Delaunay)** to choose meshing method.

3. Type **0,5** [m] to define size of finite elements (as shown below):

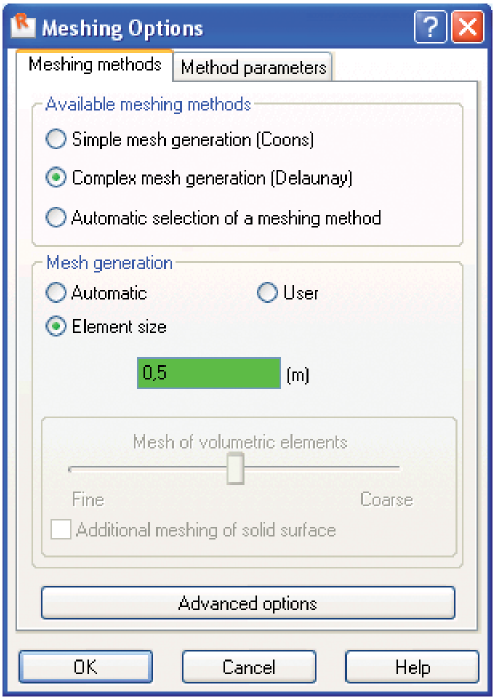


Figure 51 - Meshing method and parametrs definition.

4. Click **OK** to close **Meshing Options** dialog box.

5. From **Options of FE Mesh Generation** toolbar click  **Generation** icon or select the **Analysis** > **Calculation model** > **Generation** option to generate the mesh.

6. Close the **Options of FE Mesh Generation** toolbar.

7. Meshed slab and wall should look as shown below:

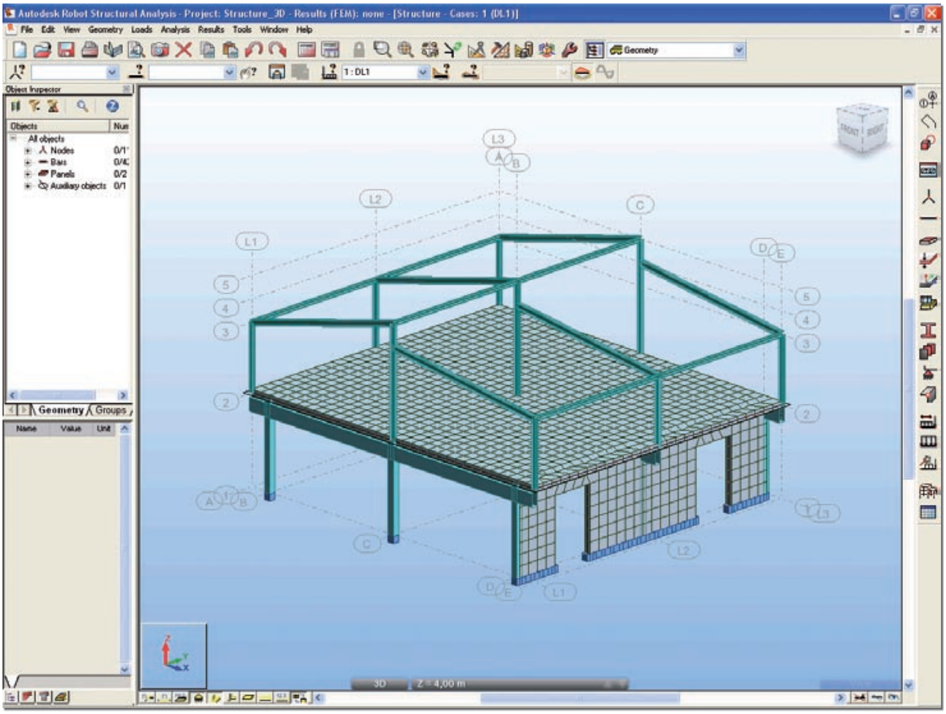


Figure 52 - Structure with meshed panels view.

INFO

Meshing parameters may be defined separately for each panel. This is s times useful if the Engineer is interested in results in a set area and can therefore mesh this area more densely than other areas.

Definition of slab loads

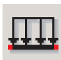
In this step, we define additional loads to be applied to the slab.

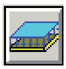
1. Press  icon (right side of the Robot screen) to open the **Load Types** dialog box.

2. Choose in the **Nature** field live, then **New** to define a live load with a standard name **LL3**.

3. To define loads for **LL3** case select this load case in the List of defined cases field.

4. Close the **Load Types** box.

5. Press  icon (right side of the Robot screen) to open the **Load Definition** dialog box.

6. On the **Surface** tab press  icon to open **Uniform Planar Load** dialog box, then type value of load **–10** (kN/m2) in the **pZ** field, click **Add** and **close** the box.

7. In the **Load Definition** dialog box set the cursor in the **Apply to** field, then indicate panel (click when it is highlighted).

8. Press **Apply** and **close Load definition** dialog box.

STRUCTURE ANALYSIS

Here we start the analysis process, but firstly we will define load case combinations:

1. Select **Loads** > **Manual Combinations…** option from the menu to open the **Combination Definition** > **Modification** dialog box. Leave **Combination name** and **Combination type** as set default.

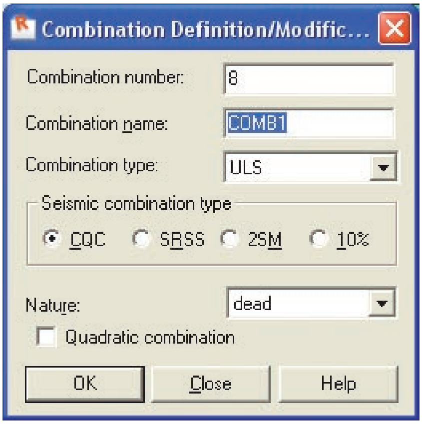


Figure 53 - Definition of combination type.

2. Click **OK** button to open **Combinations** dialog box.

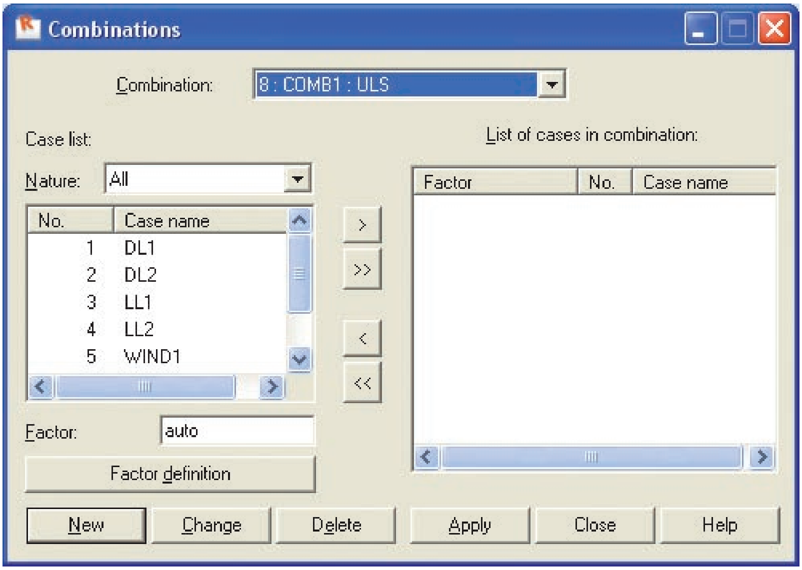
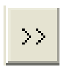


Figure 54 - Definition of cases in combination.

> **INFO**
> Combination factors may be defined manually for each nature of load case (**Factor definition** button) or applied automatically for the selected load nature (option auto in **Factor** field).

3. In this case we use automatic definition of factors (option auto in **Factor** field) so in the **Combinations** dialog box press 

 button to transfer all defined load cases (left panel) to the list of cases in combination (right panel):

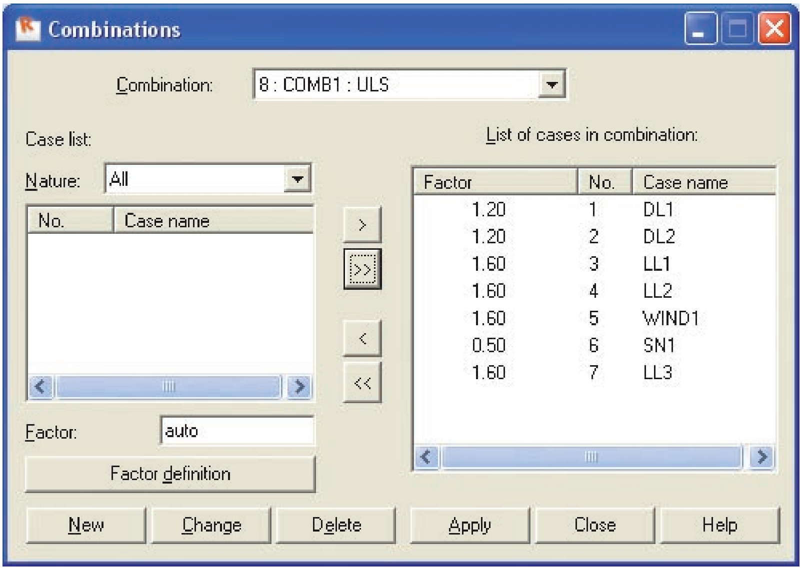


Figure 55 - List of cases in combination with automatically assigned factors.

4. **Apply** and **close** the box.

5. Defined load combination **COMB1** was added to the list of defined load cases:

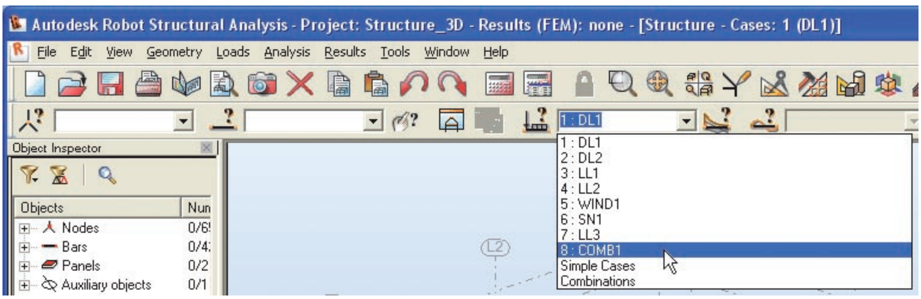


Figure 56 - List of defined load cases updated by load combination COMB1.

6.  **Calculations** – click this icon or select the **Analysis** > **Calculations** option from the menu to start the calculation process.

7. Once the calculations are completed the information: **Results (FEM): Available** should be displayed at the top of the screen.

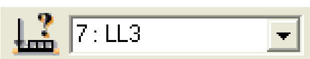
> **INFO**
>
> A key strength of Robot is the possibility to define a wide range of analysis types (linear static, non-linear geometry and material, buckling, modal analysis, harmonic analysis, seismic analysis, time history analysis etc.)
> If the user wishes to see these possibilities, he/she may look in the **Analysis** > **Analysis Types** > **Change analysis** type pull down menu.
> However, for this simple example, we will just assume the default linear static type of analysis.

RESULTS PREVIEW

Panels results in map form

In this step, we learn how to display calculation results on panels as "maps" for selected load cases:

1. In the list of defined load cases choose the load case for which the results will be displayed:



2. Select **Results** > **Maps…** from the menu to open **Maps** dialog box.

3. In the **Detailed** tab check **Displacements – u,w** for **z** direction box.

4. Tick **With FE mesh** on (bottom part of the box):

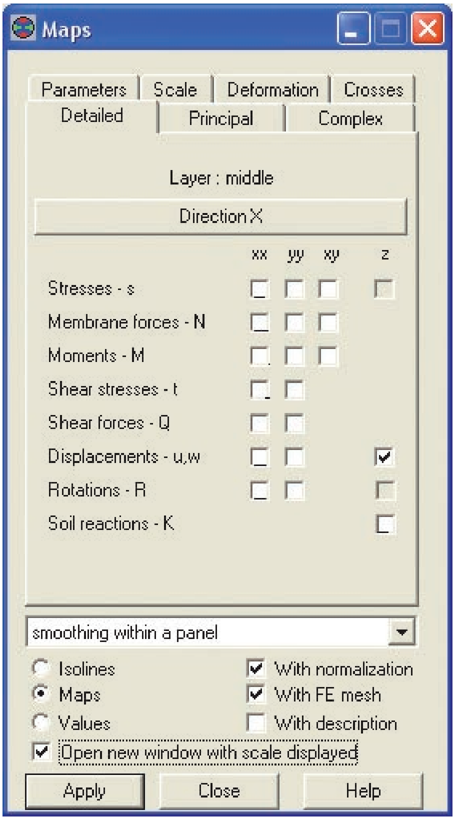


Figure 57 - Parameters of maps on panels definition.

INFO

To display scale of colors, on the **Detailed** tab tick the "Open new window with scale" displayed box . On the **Scale** tab you can change parameters of map presentation (color palette, scale type), in this example, choose **Color palette: 256 colors**.

5. Click **Apply** and **close** the **Maps** dialog box then enlarge the graphical viewer (by stretching) and click  **Zoom All** icon to maximize model view.

6. Map of deflection of the plate should look as shown below:

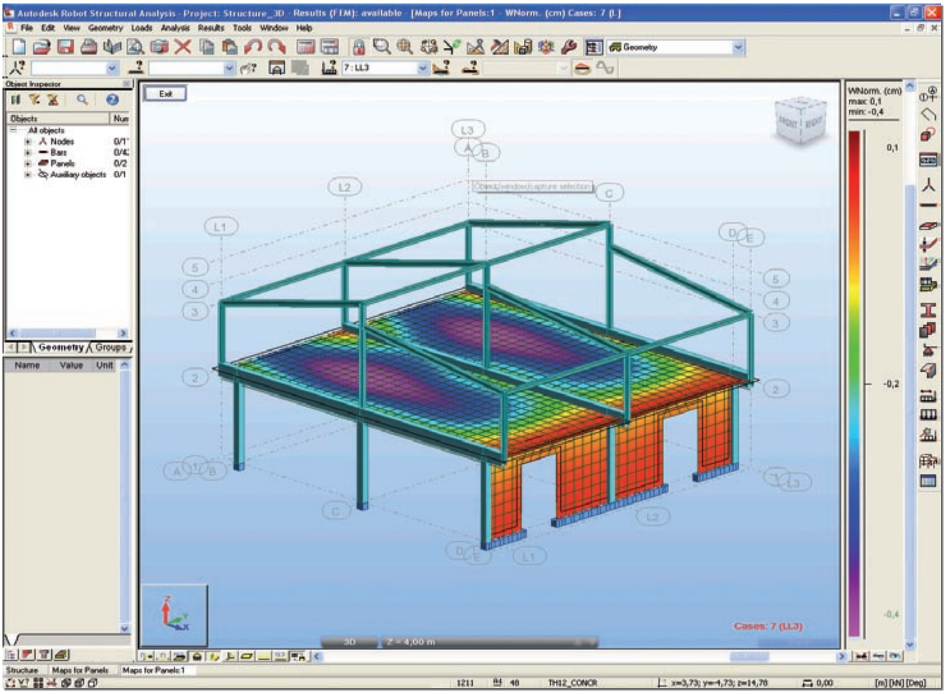


Figure 58 - Panels displacements in z direction in map form.

7. Press **Exit** to close additional window. To switch off the map display, go back to the maps dialogue box, remove the tick and apply

> **NOTE:**
>
> In the similar way, maps that show other values available from the **Detailed** dialog box can also be displayed.

Deformation of the structure

In this step, we learn how to display deformation of the structure.

1. Press  icon at the bottom of the screen to turn displaying sections shape off.

2. Select **Results** > **Diagrams for Bars…** from the menu to open **Diagrams** dialog box.

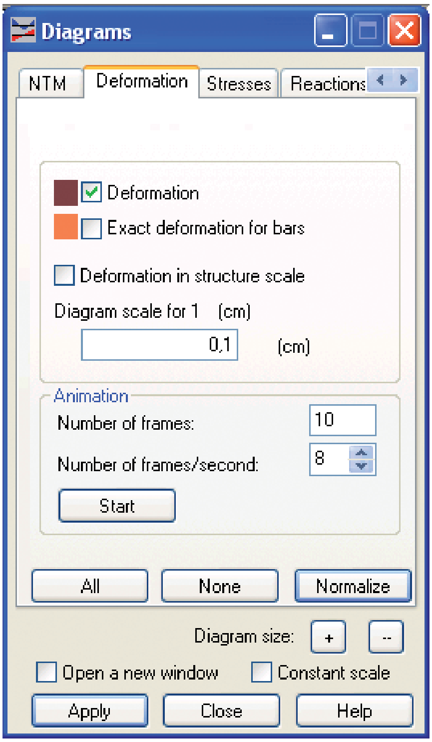


Figure 59 - Presentation parameters of structure deformation.

3. On the **Deformation** tab tick **Deformation** box on, then press **Normalize** button (to auto scale). Deformation of the structure should look as shown below:

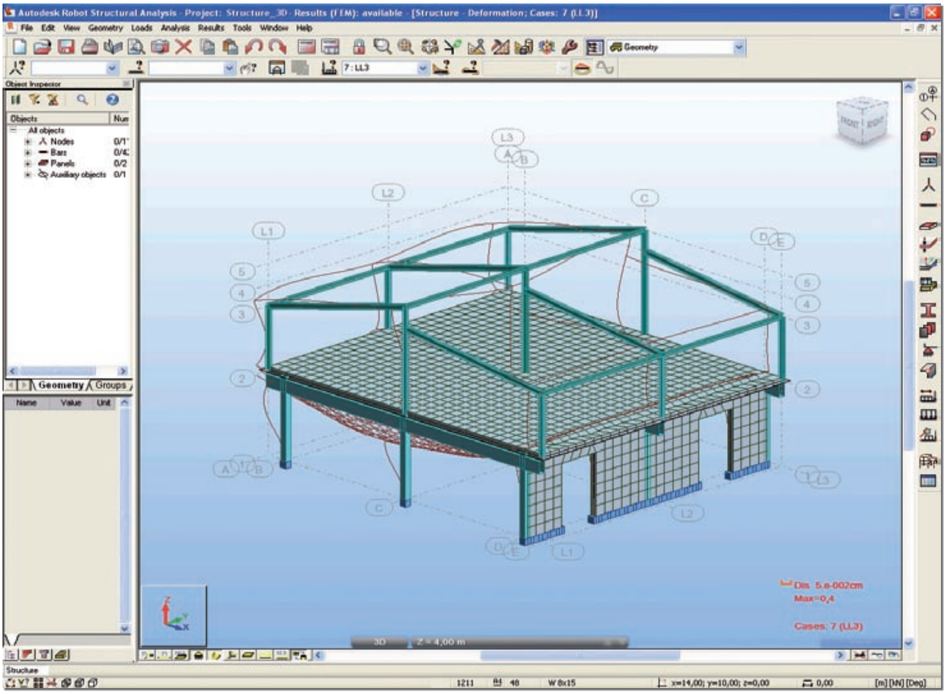


Figure 60 -Deformation of the structure.

NOTE:

In a similar way, diagrams that exhibit other values available from the **NTM** dialog box can be viewed (see below):

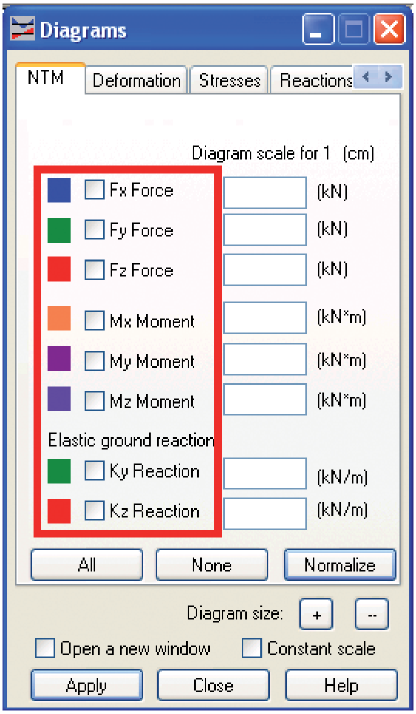
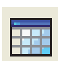


Figure 61 - Diagrams for bars dialog box.

4. Tick off **Deformation** and **Apply**, then **close** the **Diagrams** dialog box.

Results on panels in tabular form

In this step, you learn how to display calculation results as tables:

1. Select **Results > Panel and Shell Results** from the menu to open **FE Results** dialog box, or click  **Tables** icon (toolbar on the right side of the screen) and tick **Results for Plates and Shells** option on, then **OK** to close the box.

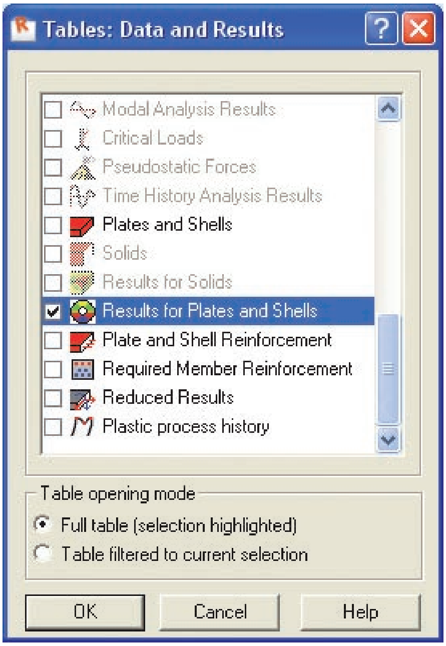


Figure 62 - Selection of data to tabular presentation.

2. Click the right mouse button and select **Table Columns** option to open **Results for Plates and Shells** dialog box.

3. In the **Detailed** tab tick the following options in:

   - **Membrane forces – N:** in direction **xx**
   - **Shear forces – Q:** in direction **xx**
   - **Displacements – u,w:** in direction **z** (as shown below):

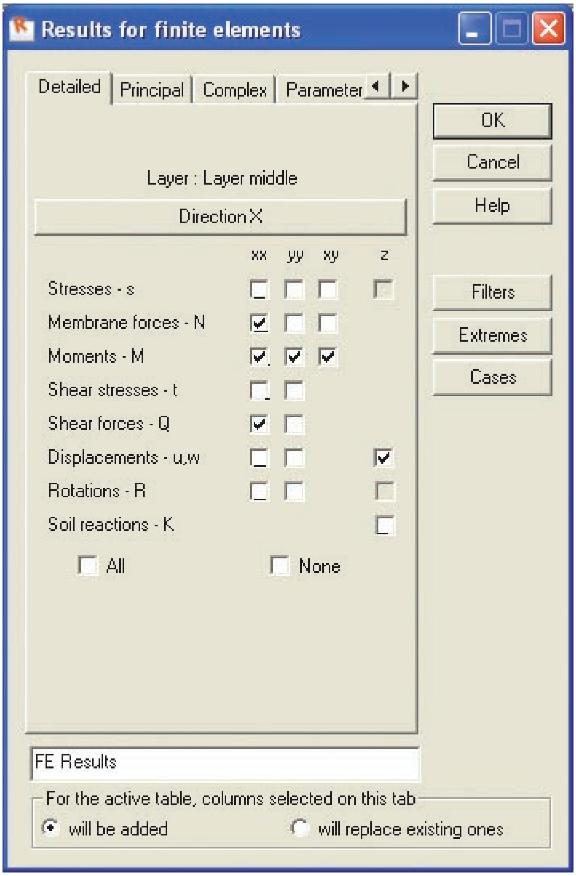


Figure 63 - Table content composition.

4. Click the **Direction X** button and in the opened **Selection of Direction** dialog box tick the **Automatic** option in:

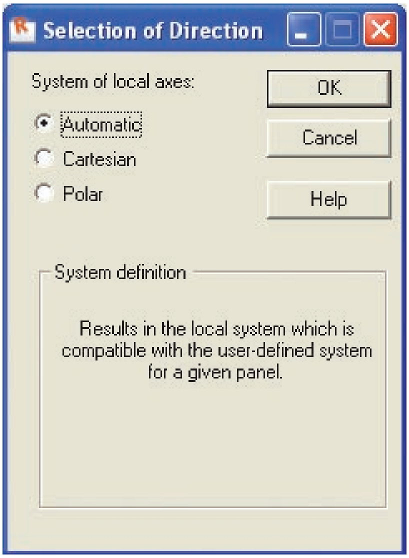


Figure 64 - Definition of the X direction of the local co-ordinate system of panels.

INFO

Selection of Direction window allows the user to define the main direction of the co-ordinate system (more precisely, to define the X-axis), which will be utilized by the user during surface FE result presentation. In this case we using **Automatic** option so the local element system will be automatically adopted according to the system defined for each of the panels.

5. Press **OK** to close **Selection of Direction** dialog box then **OK** to close the **Results for finite elements** box.

6. In the list of defined load cases choose the **8:COMB1** for which the results will be displayed, the results for selected values in tabular form is shown below:

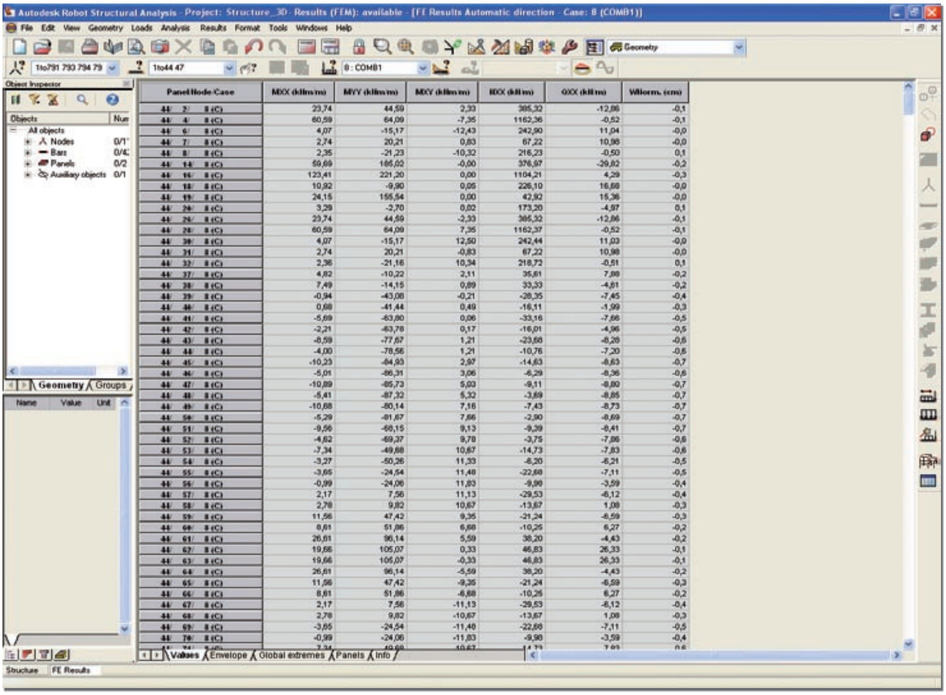


Figure 65 - Example of data presentation in tabular form.

> **INFO**
> There are few tabs on the bottom of the window to display additional data such as **Envelope**, **Global Extremes** and info about **Panels**.

7. In the table, press right mouse button, then "convert to Excel" to extract all data to a spreadsheet

8. Close the **FE Results** table.

INTEGRATION OF AUTODESK ROBOT STRUCTURAL ANALYSIS WITH REVIT® STRUCTURE

This example of a five storey concrete building shows the importance of using a capable analysis solution as part of the BIM process. Such a model cannot be described as "architecturally demanding", yet the analysis model produced from Revit® Structure poses some challenges for a lot of structural analysis solutions - including non-rectangular openings, curved slab edges and cores. However, Robot is ideally suited to such structures and will directly calculate the Revit® Structure generated data, without the need to "hack" the geometry as would be necessary to satisfy the limitations of some other structural software, thereby compromising the overall BIM integrity.

Synopsis:

The purpose of this example is to show the ease of transfer of data from Revit® Structure to Robot and also to show how changes in the model made in Robot can be reflected again in Revit® Structure, thereby maintaining the integrity of the overall Building Information Model. In particular, this example looks at the import of floors and walls from Revit® Structure, in the form of finite element surfaces in Robot.
In addition to showing the user how to interact with and display basic data in Robot, this example also highlights the display of FE results and making a basic FE mesh, plus amending the mesh to suit the Engineer's preferences
For this example, it is assumed that Autodesk Robot Structural Analysis 2010, plus the Extensions For Revit (Integration with Robot Structural Analysis) are installed on the same PC.

EXPORT REVIT MODEL TO ROBOT

Opening project in Revit® Structure

In this step, we open and examine the model in Revit® Structure:

1. Press  icon to run the Revit® Structure program then open the Revit® Structure project: **Revit2Robot building example.rvt**:

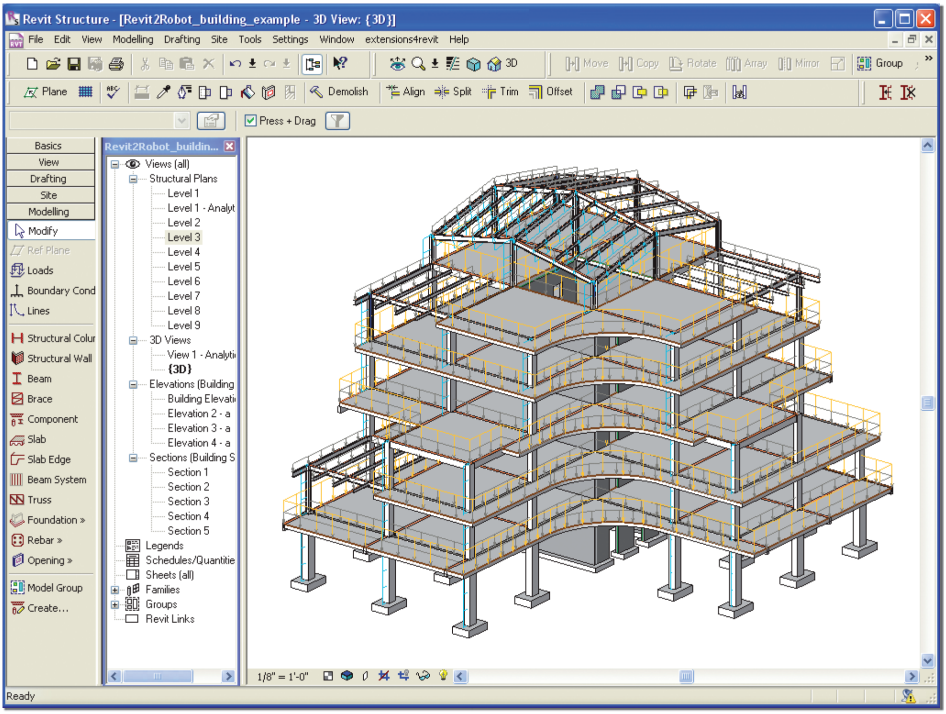


Figure 1 - Model of building in Revit Structure

2. Rotate it to show the type of elements it contains:

   - Sections:
     - steel (according to US AISC and also UK Steel section databases)
     - concrete beams and columns

   - Slabs, walls
   - Foundations:
     - isolated footings - in Robot these will be treated as a fixed support,
     - continuous around wall base - in Robot these will be treated as a line support
   - Loads (Line and Area Load)

Sending data to Robot

In this step, we learn how to export the Revit model to Robot:

> **NOTE:**
>
> A useful feature is that only "selected" structure data in Revit® Structure is transferred to Robot – this allows only part of the model to be transferred to Robot (it could be useful if the user wants to select only part of a structure to analyze).
>
> However, in this example, we want Robot to analyze all of the Revit® Structure data so it is important to ensure that either no data is selected or alternatively all of the structure is selected.

1. Go to the **extensions4revit** menu and select the **Extensions Manager** option:

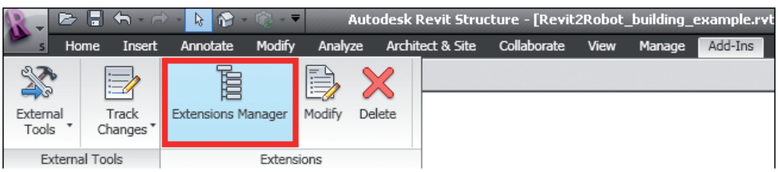


Figure 2 - Extensions4revit menu

> **NOTE:**
>
> Revit® Structure: **Modify** option should be selected in the **Basics** tab of the **Design Bar**.

2. **Extensions Manager** dialog box will appear:

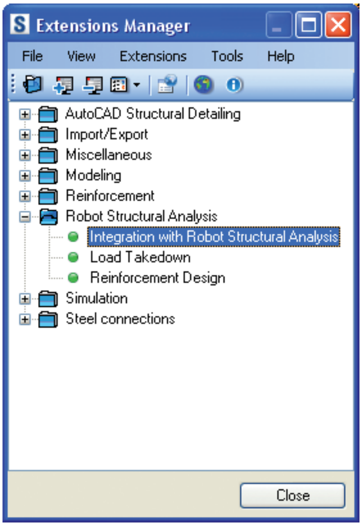


Figure 3 - Extensions Manager dialog box

3. To open **Integration with Robot Structural Analysis** double click the appropriate option (see above).

4. In dialog box shown below leave **Send model to Autodesk Robot Structural Analysis** option active (it is chosen by default) and click OK.

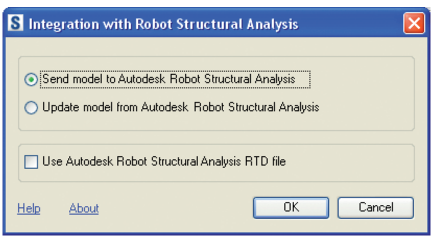


Figure 4 -Integration with Robot Structural Analysis (send model option)

NOTE:

There are 2 ways to send data to Robot:

- dynamic – **Send model to Autodesk Robot Structural Analysis** –this is a dymanic transfer of data between programs
- by file – **Use Autodesk Robot Structural Analysis RTD file** – useful if, for example, Revit® Structure and Robot are on different computers

5. During the transfer process the following splash screen is displayed:

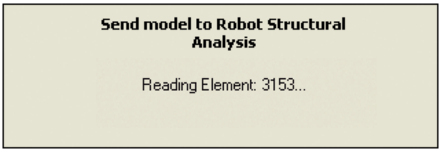


Figure 5 -Send model to Robot splash screen

6. Next, **Integration with Robot Structural Analysis – Send Options** dialog box will appear:

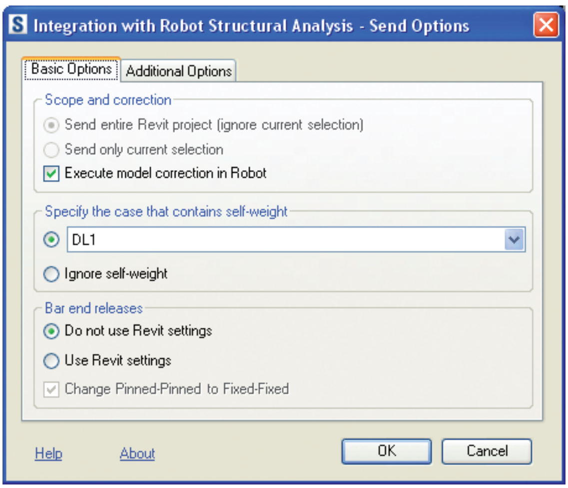


Figure 6 -Integration with Robot Structural Analysis - Send Options dialog box

7. Leave default settings as shown above and click OK.

INFO

Integration with Robot Structural Analysis – Send Options dialog box allows you to configure export parameters

Basic Opitons tab:

Scope and correction group – enables the user to:

- Send entire Revit project (ignore current selection) - sends whole model (this option is selected by default – it prevents exporting of an accidentally selected element or elements.
- Send current selection only - sends only selected elements
- Execute model correction in Robot - decides whether to run “drawing model correction” in Robot automatically, which will serve to join nodes to gether that fall within a certain tolerance

Specify the case that contains self-weight - option allows for defining which load case will be considered in RSA as a self-weight case

- If the first option is chosen, then User is able to specify Revit load case, which will be used in RSA for definition of self-weight. All loads defined in this case remain unchanged – self-weight will be added.
- It is possible to ignore definition of self-weight with using Revit load cases – by choosing second option.

Bar end releases group - allows the user to select the way of handling end releases:

- Do not use Revit settings (assign end releases in Robot) – end releases are manually defined in Robot.
- Use Revit settings – recognizes and assigns bar end releases defined in Revit® Structure
- Change Pinned-Pinned to Fixed-Fixed – all Pinned-Pinned end releases defined in Revit® Structure are changed to Fixed-Fixed. This is often set to avoid mechanisms in the structural model

INFO

Integration with Robot Structural Analysis – Send Options dialog box allows you to configure export parameters

Additional Opitons tab:

Materials group - allows you to select the way of handling materials:

- Use Robot default materials – for each element's material type (steel, concrete, timber) Robot will assign material from Robot database with standard properties
- Define new materials in Robot – creates new materials in Robot with such physical properties as defined in Revit® Structure
- Select material of the best matching parameters - assigns material properties from Robot data base with the nearest values to those defined in Revit® Structure

Curtain walls - option allows for setting sending parameters for Curtain walls. Panels created in RSA have a property (triangular method of load distribution in panel calculation model is set), which allows load distribution to other elements.

- Analytical model only (no system panels, no mullions) – creates one panel in RSA without interior elements (system panels and mullions) – differences in singular panels are neglected
- Analytical model and mullions (no system panels) – creates one panel in RSA (no division to system panels) but also mullions are transferred – differences in singular panels are neglected
- System panels and mullions (detailed model) – sends all interior elements – system panels and mullions are created in RSA – differences in singular panels are considered

Transfer (optionally) – set of additional options of sending

- Use drawing model offsets as analytical – an offset parameter is defined for bar elements in RSA according to Revit's definition
- Reinforcement projects (beams, columns, spread footings) – defined reinforcement in listed elements is transferred to concrete design modules in RSA
- Steel connections – option not available in current version

8. The Send model to Robot splash screen is displayed once again. Information about Robot launching appears. A progress information shows each step of the transfer process (such as reading of necessary Revit® Structure data, creation of Robot project, export of elements and creation of corresponding Robot structural elements).

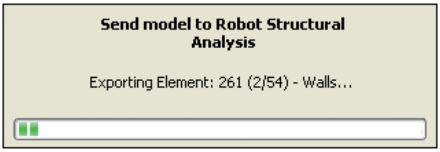


Figure 7 -Send model to Robot splash screen

NOTE:

Occasionally sections may be defined in Revit® Structure that are not defined in Robot. The **Section Database Organizer** (see below) allows the user to add such databases into Robot without disturbing the transfer of data between programs.

9. **Adaptation of Active Section Databases of Robot** dialog box may appear and display (upper, left corner of the box) information about **Unknown** sections (in this case there is one unknown section **178x102x19UB** from the British section database).

NOTE:

Such a message may or may not occur on individual PC's depending on the language, preferences and databases that are used in Robot.

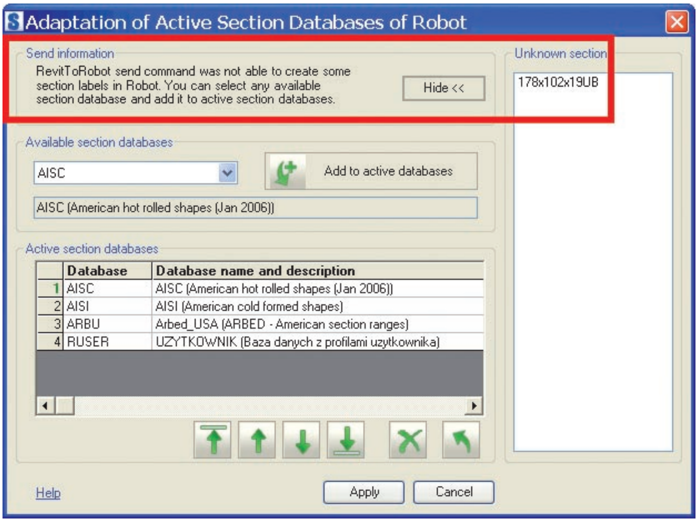


Figure 8 - Adaptation of Active Section Databases of Robot - information about unknown sections

10. In the **Available section databases** pull-down menu select **UKST** (British section database).

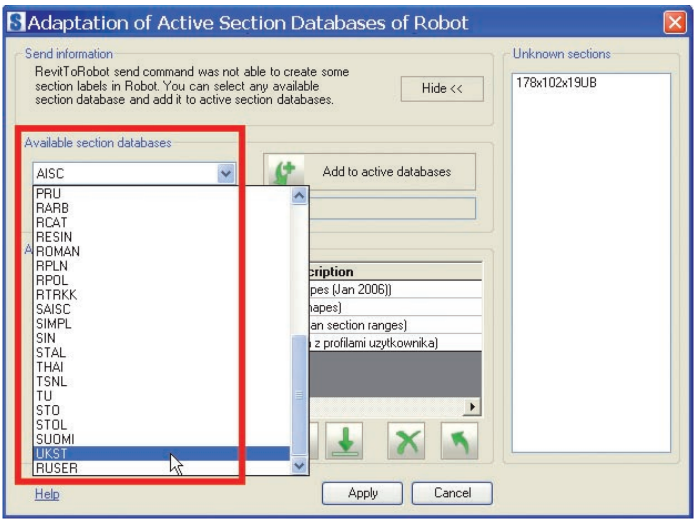


Figure 9 - Adaptation of Active Section Databases of Robot - available section databases pull-down menu

11. Click to **Add** to active databases button, **UKST** database will appear in the **Active section databases** window.

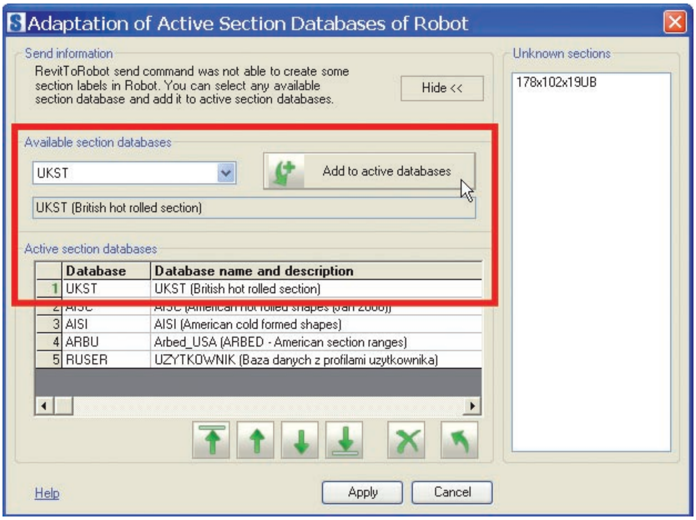


Figure 10 - Adaptation of Active Section Databases of Robot - information about added section database

12. Press **Apply** to close the window and continue exporting process.

13. After data transfer, we can see an events report, to do this click **Yes**.

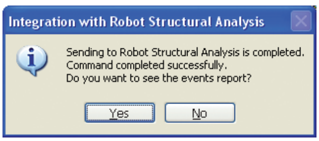


Figure 11 - Events report after exporting data selection.

14. List of messages (and eventual warnings) will appear (see below):

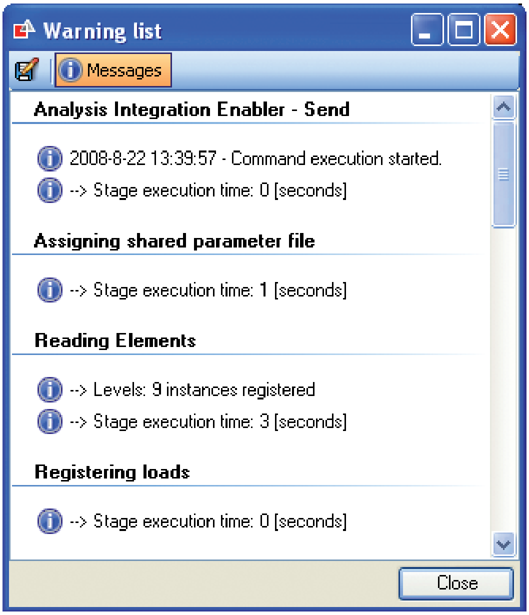


Figure 12 -Warning and messages list

15. Close the **Warning list** to finish sending data to Robot process.

STRUCTURE ANALYSIS IN ROBOT

NOTE:

Change units from imperial to metric: Tools/Job Preferences/Units and Formats/Metric.
You can check active units in right, bottom corner of the screen. In this example should be the display: [m] [kN] [Deg].

Exported Revit® model of structure should appear in Robot as shown below:

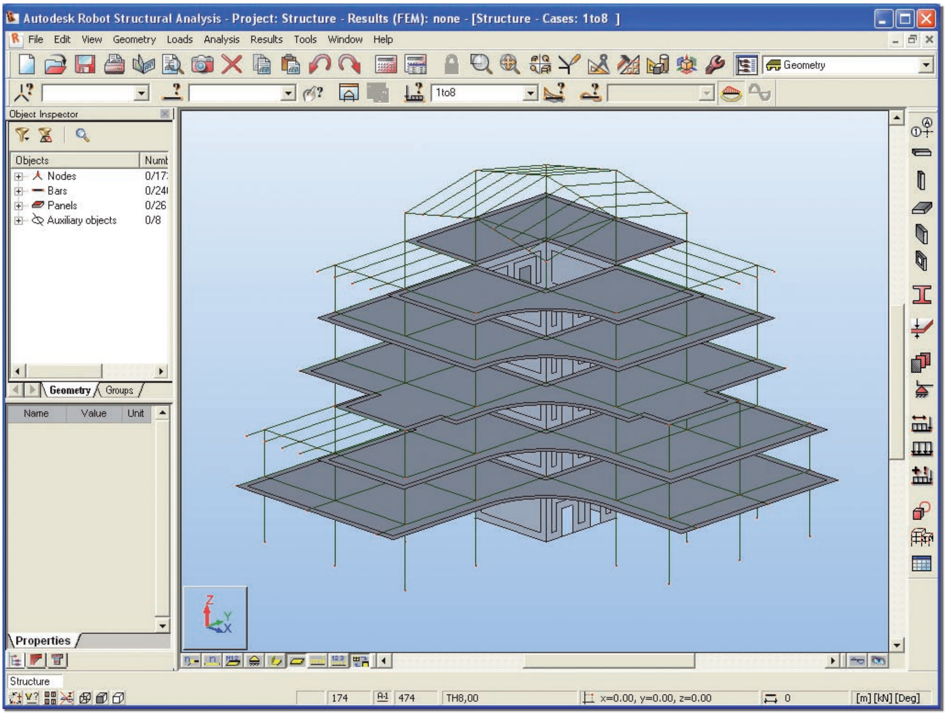


Figure 13 -Exported model of structure in Robot

Displaying items on the screen

In this step, we learn how to display additional attributes on the screen. By selecting icons of the toolbar located under the graphic field display a few more items can be displayed:

 - Support symbols

 - Section shapes

 - Load symbols

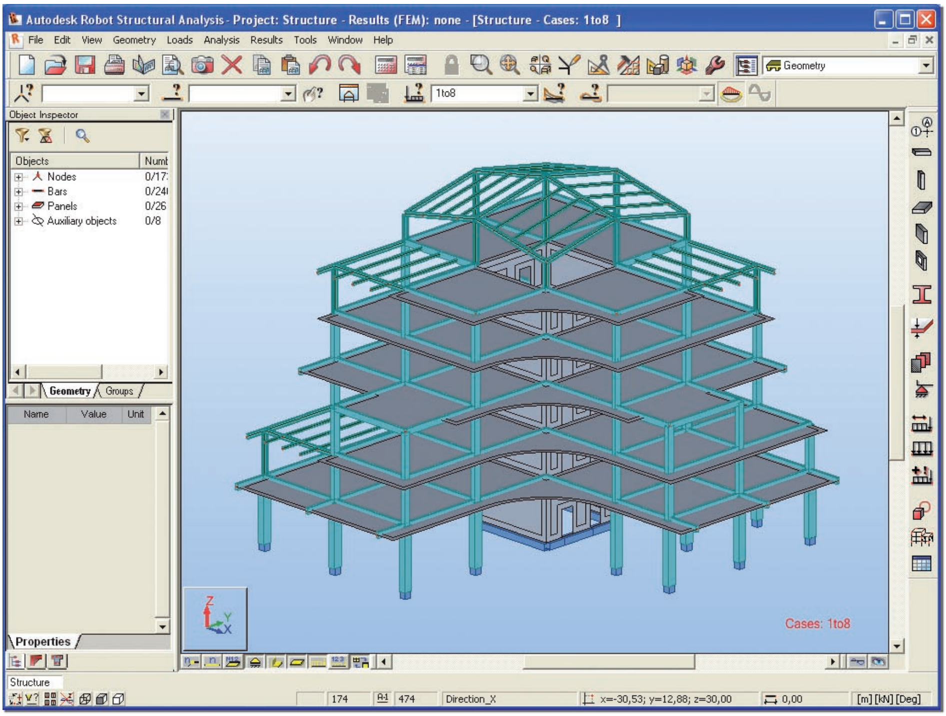


Figure 14 -Model of structure in Robot with additional display attributes

Presentation of load cases transferred from Revit® Structure

In this step, we display the load cases defined in Revit® Structure:

1. From the list of defined load cases choose **2: LL1**

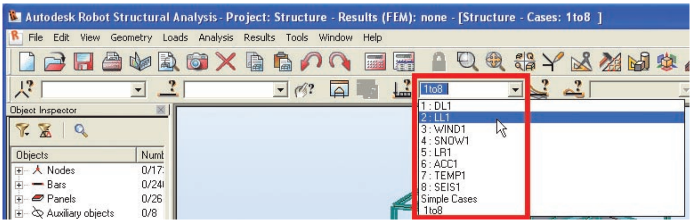


Figure 15 -List of defined load cases

2. Loads belonging to the case **2:LL1** case are shown in the graphic viewer (in the same way display next cases: **3: WIND1** and **4: SNOW1**):

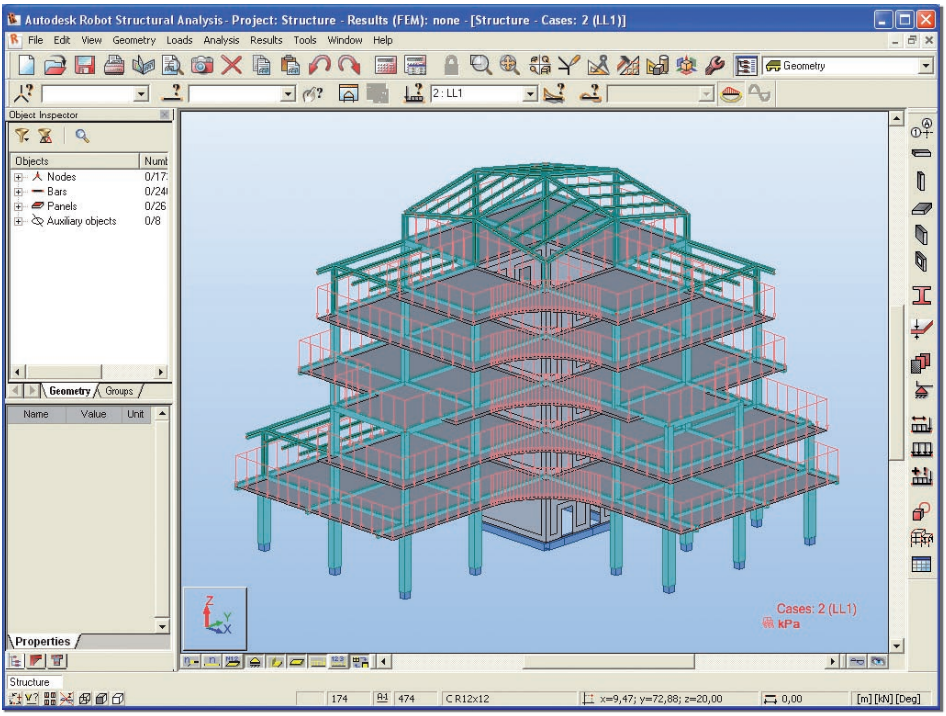


Figure 16 -2:LL1 Load case

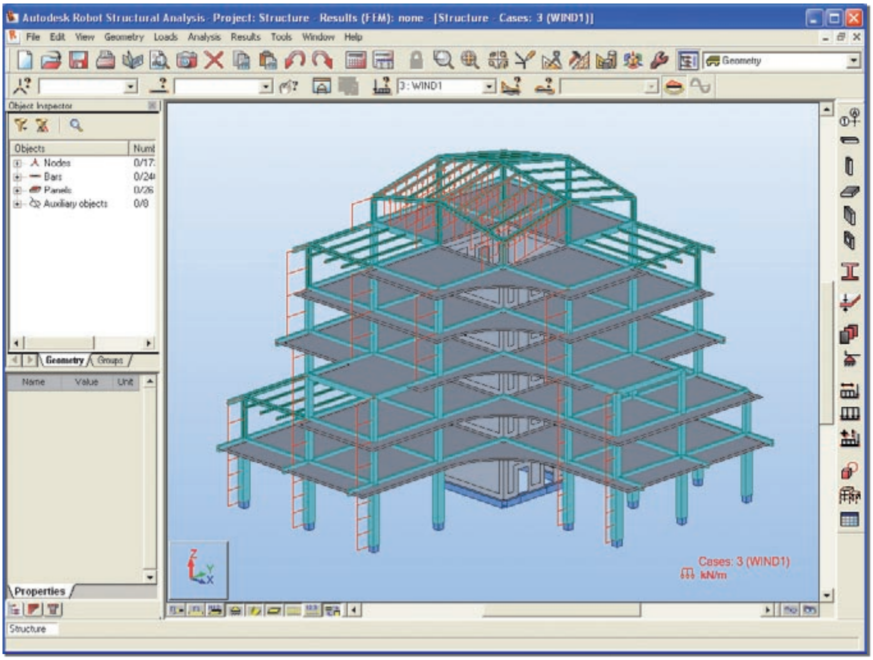


Figure 17 -3:WIND1 Load case

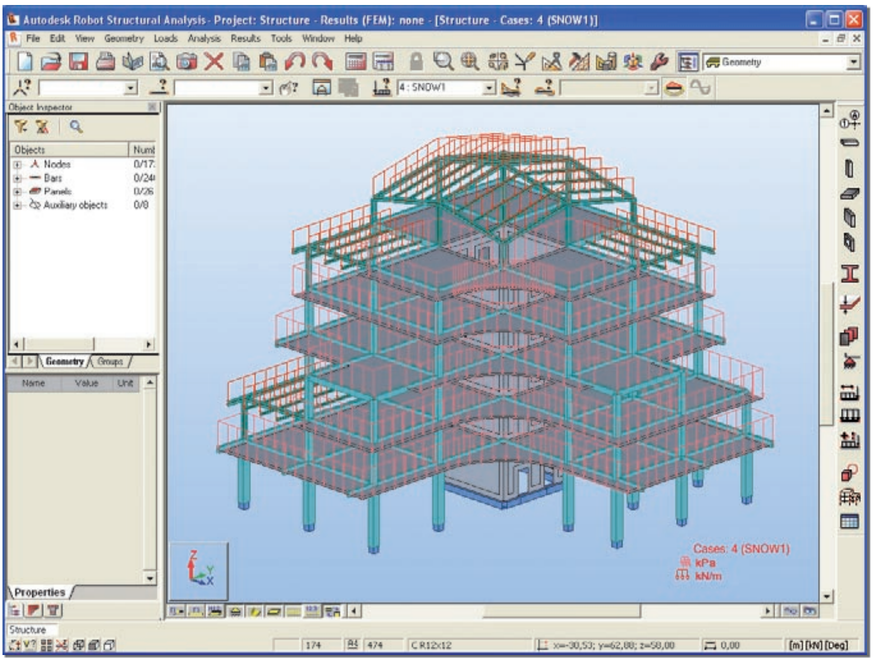


Figure 18 -4:SNOW1 Load case

3. To display numerical values of loads click icon (bottom left corner of the screen).
4. Display the **View** submenu.
5. By means of 3D Orbit option from the **View** toolbar rotate the model to show all the details: Profiles, Panels, Supports, Loads.
6. Click one more time the icons: to turn loads display off.
7. Click one more time the icons: to turn support and sections display off.

Meshing parameters definition

In this step, we learn how to mesh the panels.

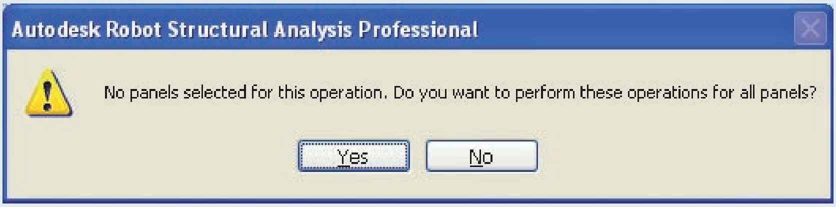
INFO

The meshing of the plates and shells are made automatically (with standard parameters). A key strength of Robot is the ability to rapidly automesh virtually any shape of surface and to edit and refine the mesh as required.

1. **Options of FE Mesh Generation** then **Meshing Options** – click these icons or select the **Analysis** > **Calculation model** > **Meshing Options** option from the menu to open the **Meshing Options** dialog box.

NOTE:

Because there were no panels selected, the program will ask you the question:



Answer **Yes** to select all panels.

2. Select **Complex mesh generation (Delaunay)** to choose meshing method.

3. Type **0,61** [m] to define size of finite elements (as shown below):

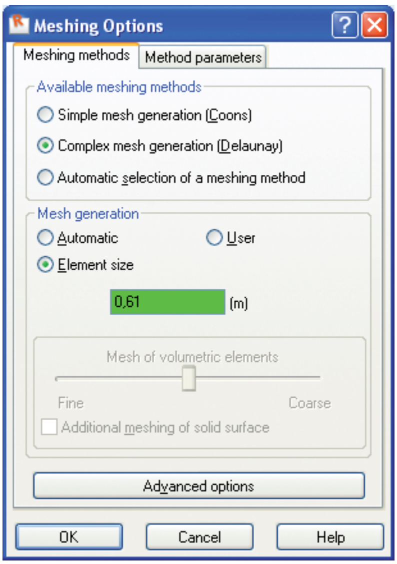
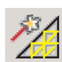


Figure 19 -Meshing method selection

4. Click OK to close **Meshing Options** dialog box.

5. From **Options of FE Mesh Generation** toolbar click  **Generations** icon or select the **Analysis > Calculation model > Generation** option to generate the FE mesh.

6. Close the **Options of FE Mesh Generation** toolbar.

7. Meshed slabs and core walls should appear as shown below:

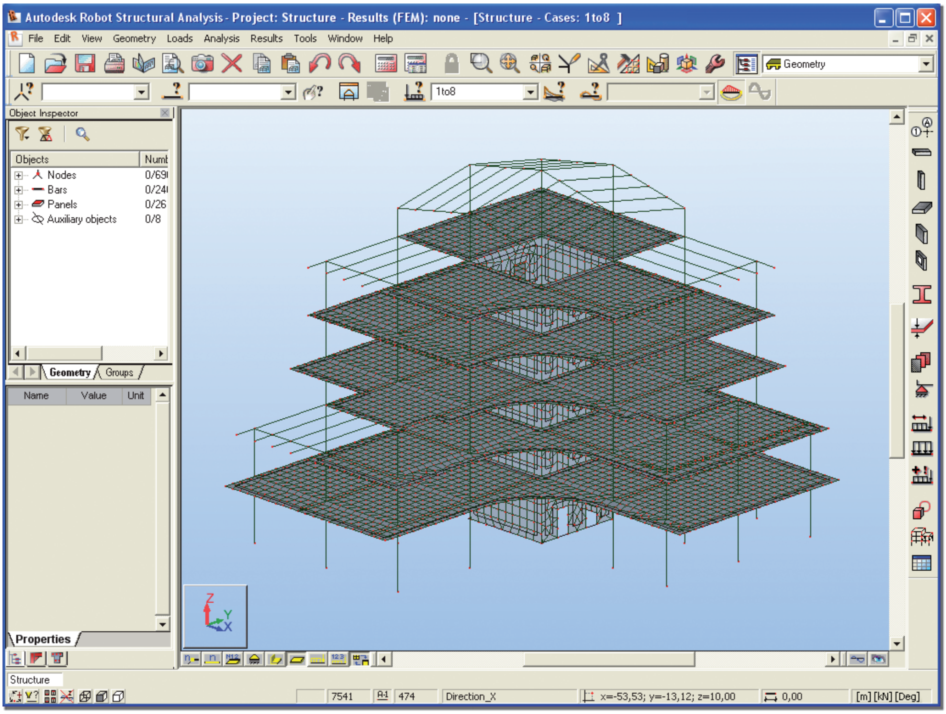


Figure 20 -Example of meshed structure

8. Press  icon at the bottom of the screen to hide the finite element mesh.

INFO

Meshing parameters may be defined separately for each panel.

Calculations

Here we start the analysis process, but firstly we will tell Robot to make automatic code combinations (from any one of the list of Codes in **Job Preferences**):

1. Select **Loads** > **Automatic Combinations…** option from the menu to open the **Load Case Code Combinations** dialog box. Select the **Full automatic combinations** option. The program will now automatically assign a number of combinations to find the most onerous load combination.

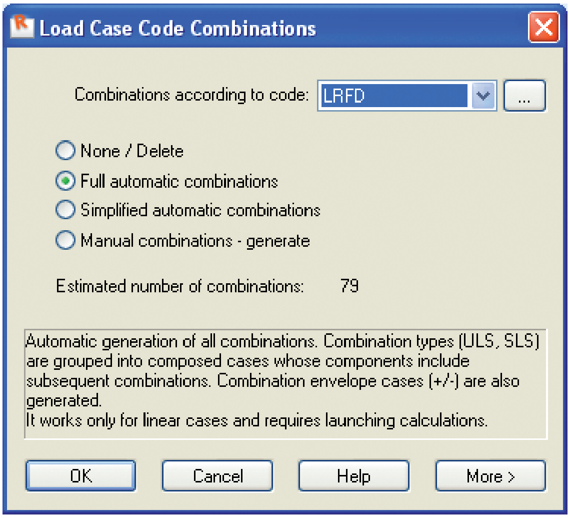


Figure 21 -Code combinations dialog box

2. Click **OK** button – automatic calculation of code combinations will be done.

3.  **Calculations** – click this icon or select the **Analysis** > **Calculations** option from the menu to start the calculation process.

4. Once the calculations are completed the information: **Results (FEM): Available** should be displayed at the top of the screen.

INFO

A key strength of Robot is the possibility to define a wide range of analysis types (linear static, non-linear geometry and material, buckling, modal analysis, harmonic analysis, seismic analysis, time history analysis etc.) If the user wishes to see these possibilities, he/she may look in the **Analysis > Analysis Types > Change analysis type** pull down menu.

However, for this simple example, we will just assume the default linear static type of analysis.

Results preview - displaying panel results in map form

In this step, we learn how to display calculation results on panels as contour maps for selected load cases:

1. In the list of defined load cases choose the load case for which the results will be displayed:

2. Select **Results** > **Maps…** from the menu to open **Maps** dialog box.

3. In the **Detailed** tab check **Displacements – u,w** for **z** direction box in.

4. Tick **With FE mesh** on (bottom part of the box):

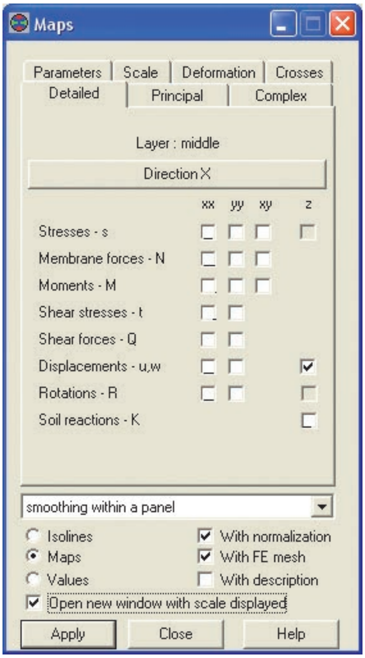


Figure 22 -Maps for surface finite elements (panels) definition

NOTE:

To display scale of colors, on the **Detailed** tab tick the **Open new window with scale displayed** box on. On the **Scale** tab you can change parameters of map presentation (color palette, scale type), in this example choose **Color palette: 256 colors**.

5. Click **Apply** and **Close** the **Maps** dialog box then enlarge the graphical viewer (by stretching) and click  **Zoom All** icon to maximize the model view.

6. Maps for selected value should look as shown below:

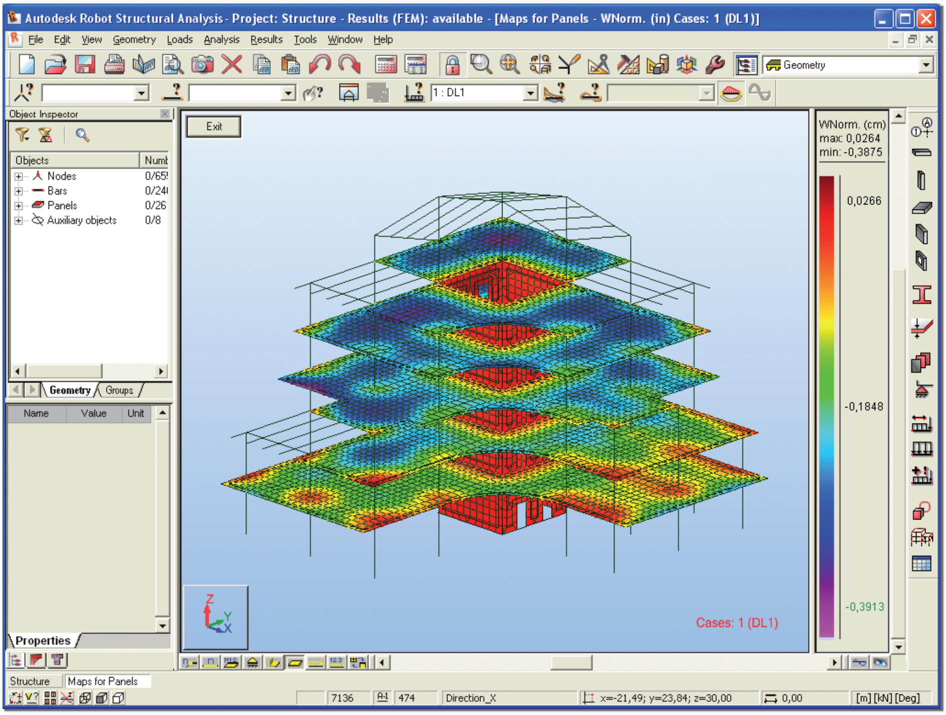


Figure 23 -Example of displacements presentation in map form.

7. Tick off the **Displacement z** check-box and click **Apply** to switch the maps off.

8. Close the **Maps** dialog box.

9. Click **Exit** button (upper, left corner of the Robot screen) to close the window with the scale.

Results preview - displaying results on bars in diagram form

In this step, we learn how to display calculation results on bars as diagrams for selected load cases:

1. In the list of defined load cases choose the load case for which the results will be displayed:

2. Press  icon at the bottom of the screen to turn off display of panel interiors.

3. Select **Results > Diagrams for Bars** option from the menu to open the **Diagrams** dialog box.

4. Select **MY Moment** (this shows the major axis bending moment on the beams) as shown below:

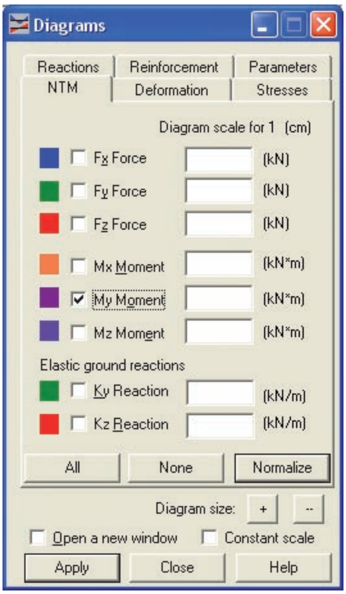


Figure 24 -Diagrams on bars definition (force selection)

5. On the **Parameters** tab tick **differentiated** option (on **Positive and negative values** field) on (see below) and return to **NTM** tab:

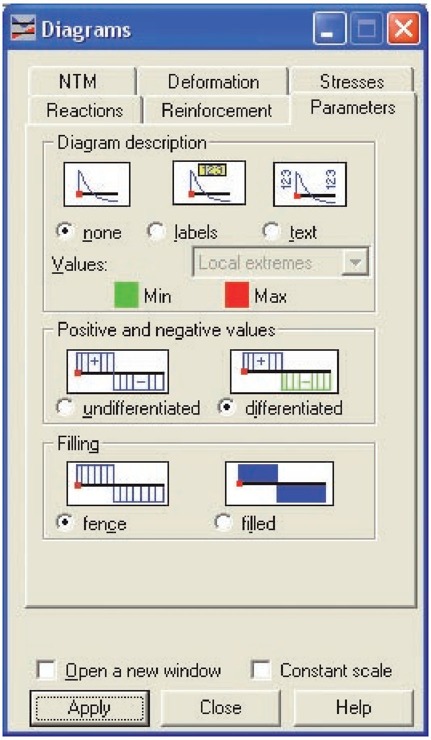


Figure 25 -Diagrams on bars definition (parameters of diagrams displaying)

6. On the **NTM** tab press **Normalize** button (to autoscale) to display bending moment diagrams for bars:

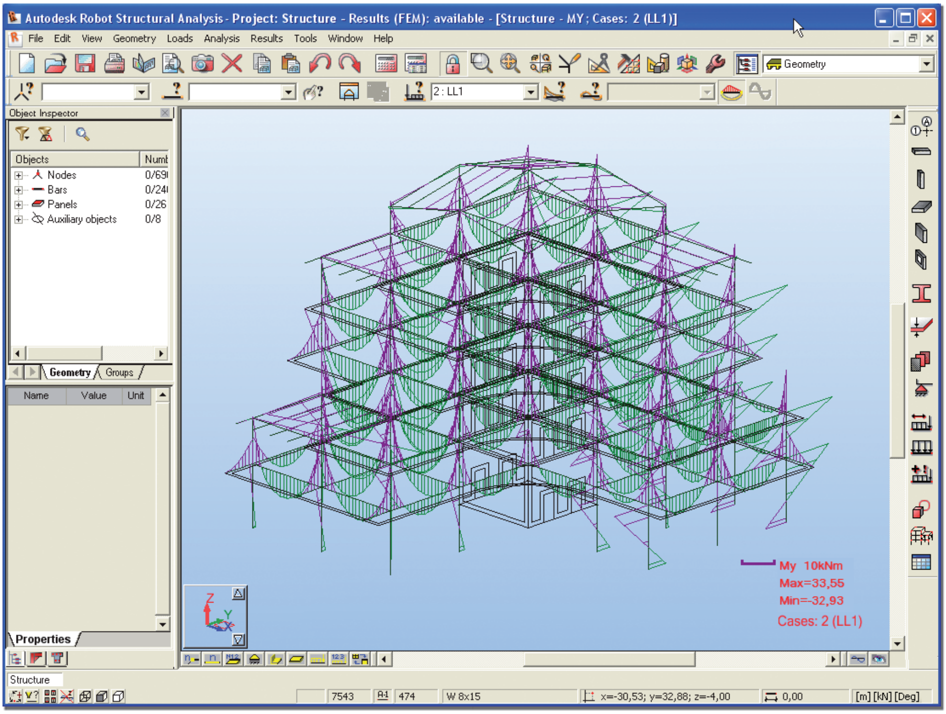


Figure 26 -Example of bending moment presentation in diagram form

7. Tick off **My Moment** and apply to remove the diagram.

8. **Close** the dialog box.

9. Press , ,  icons at the bottom of the screen to turn on display of the supports, section shapes and panel interior.

MODIFICATION OF THE STRUCTURE IN ROBOT

Replacing sections

In this step, we learn how to replace section shapes:

1. Press  icon on top toolbar to open the **Selection** dialog box.
2. Indicate **Section** as selection criterion and click **178x102x19UB**.
3. Press the  button to enter defined selection to the edit field and mark red steel beams (see picture below):

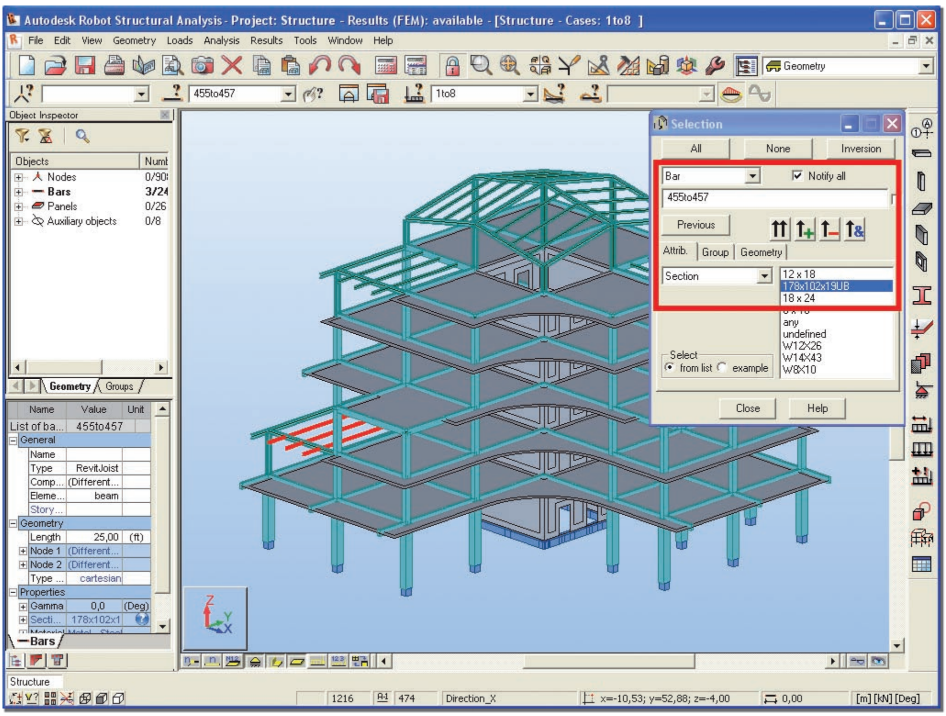


Figure 27 -Selection of beams by filters.

4. Go to the toolbar on the right side of the screen and click  **Bar Sections** icon.

5. In **Sections** dialog box select **W 8x10** section:

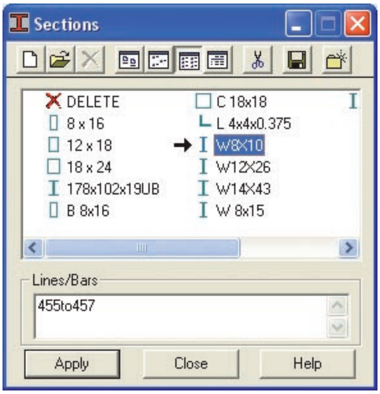
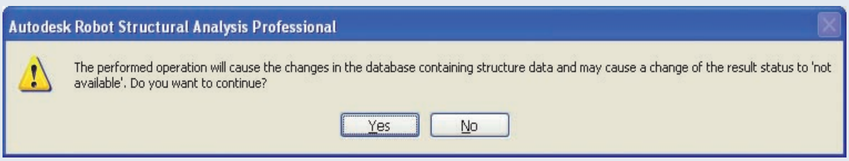


Figure 28 -Section selection.

6. **Apply** and **Close** the box. This changes all the selected sections to **W 8x10**

> **NOTE:**
>
> Because the section exchange implicates changes in the structure stiffness, the program will ask you the question:
>
> 
>
>
> Answer Yes

In the same way we can replace concrete columns **18x24** by sections **C18x18**.

7. Press  icon to open the **Selection** dialog box.

8. Indicate **Sections** as selection criterion and click **18x24**.

9. Press the  button to enter defined selection to the edit field and mark red concrete columns:

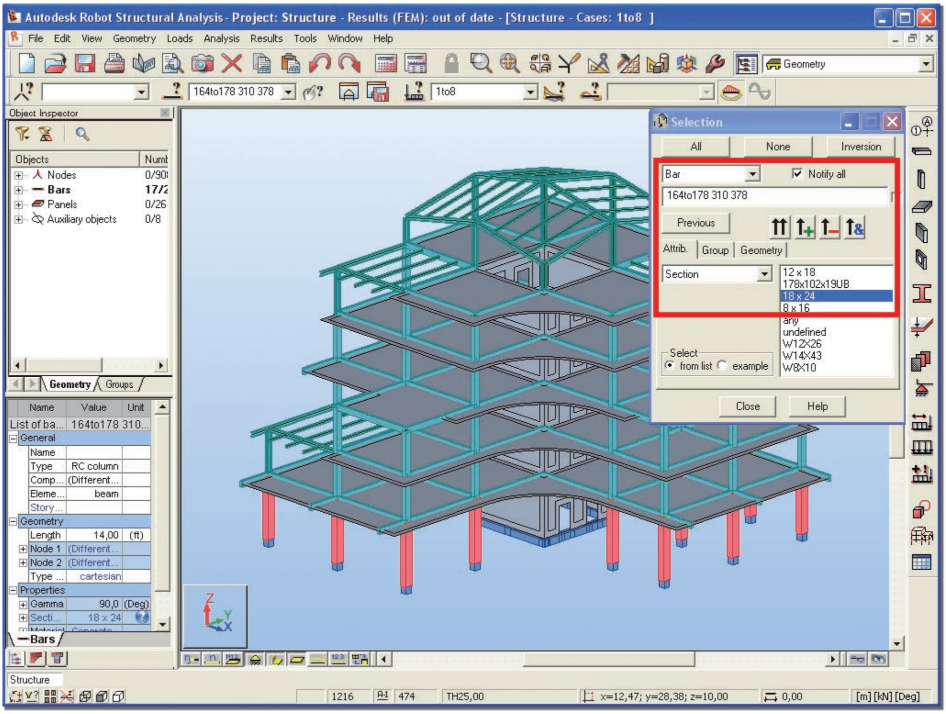


Figure 29 -Selection of columns by filters.

10. Click **Close**.

11. In the **Sections** dialog box choose **C18x18** section as shown below:

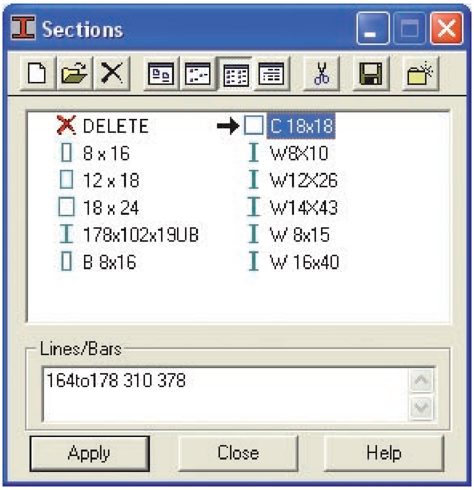


Figure 30 -Section selection.

12. Click **Apply** in Sections dialog box to give the new section type to bars

13. Close the Sections dialog box.

Deleting bars

In this step, we learn how to delete elements:

1. Indicate bars (just click bar when is highlighted) as shown below. To make a multi selection, click while holding down the ctrl button:

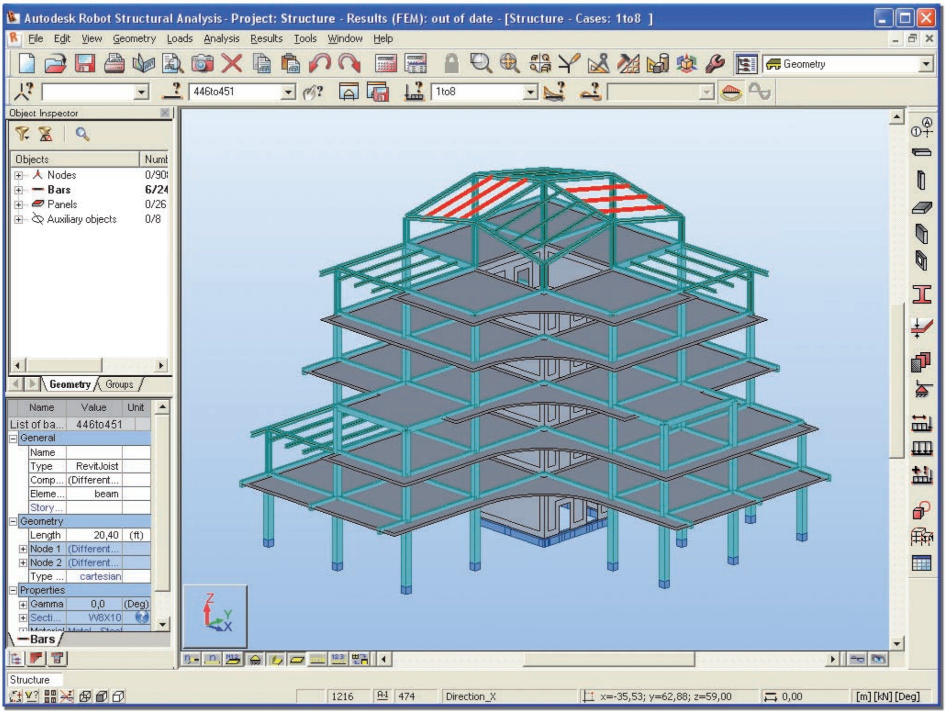
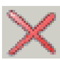


Figure 31 -Selection of beams by coursor.

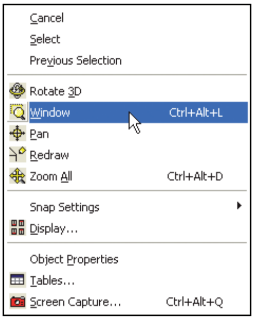
2.  **Delete** – press this icon or select the **Edit** > **Delete** option from the menu to delete selected elements.

Adding new elements

In this step, we learn how to create new elements (in this case we will add cross bracing):

1. Press the icon  to turn sections display off.

2. Press right mouse button on the graphics screen, then in the context menu select **Window** option:



3. Select by dragging window the view of the highest storey as shown below:

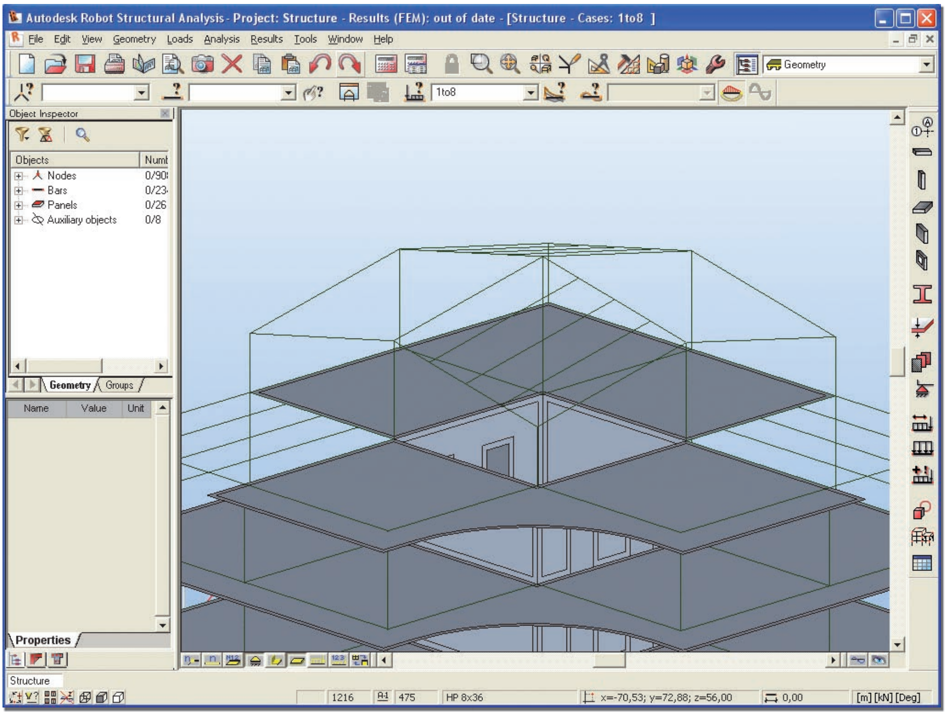


Figure 32 -Top storey view.

4. Select the **Geometry** > **Bars** option from the menu to open the **Bars** dialog box.

5. Set **Bar type:** as **Simple bar** (whatever is selected is not important for analysis, but affects the design parameters for subsequent member design, such a buckling length, position of restraint etc)

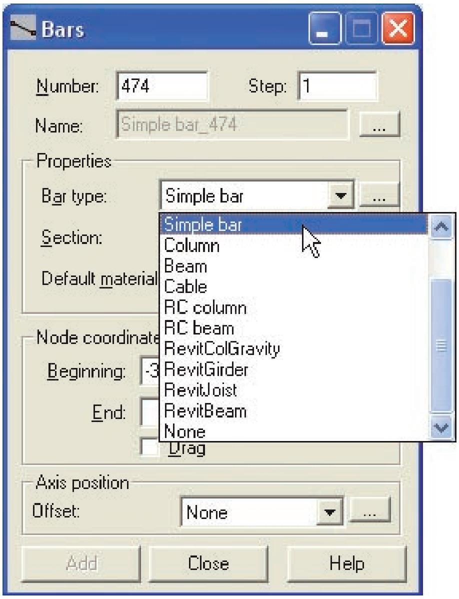


Figure 33 -Bar type definition.

6. Define Section: as **L4x4x0.375**.

7. If the **L4x4x0.375** section is not available on the list, you should click the **(…)** button located beside the **Section** field.

8. In **New section** dialog box, in **Section selection** field for **Database:** select **AISC**, for **Family:** select **L** then in the **Section**: set **L4x4x0.375**:

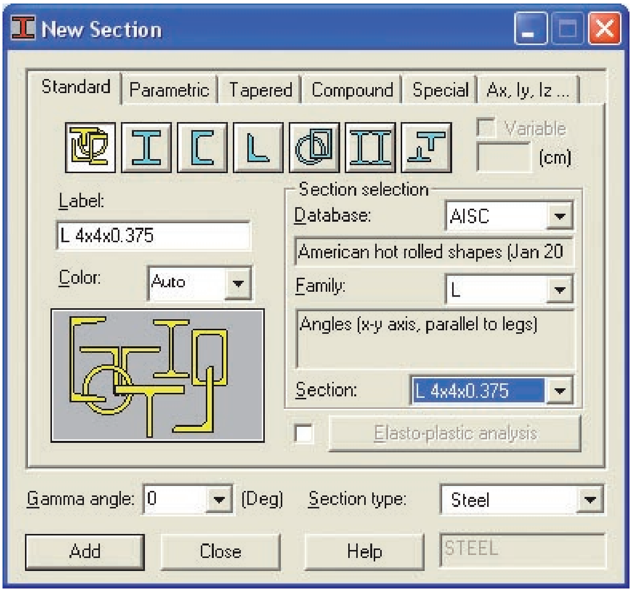


Figure 34 -New section definition

9. Click **Add** and **Close** the box.

INFO

There are many extra options that may be entered for fabricated members, tapering sections and also for beams that the user wants to define as able to exhibit plasticity.

10. Click the following points (their numbers are shown in the picture below) to draw new bracing:

- 137 126 to create 1st bar
- 124 125 to create 2nd bar
- 126 136 to create 3rd bar
- 127 128 to create 4th bar

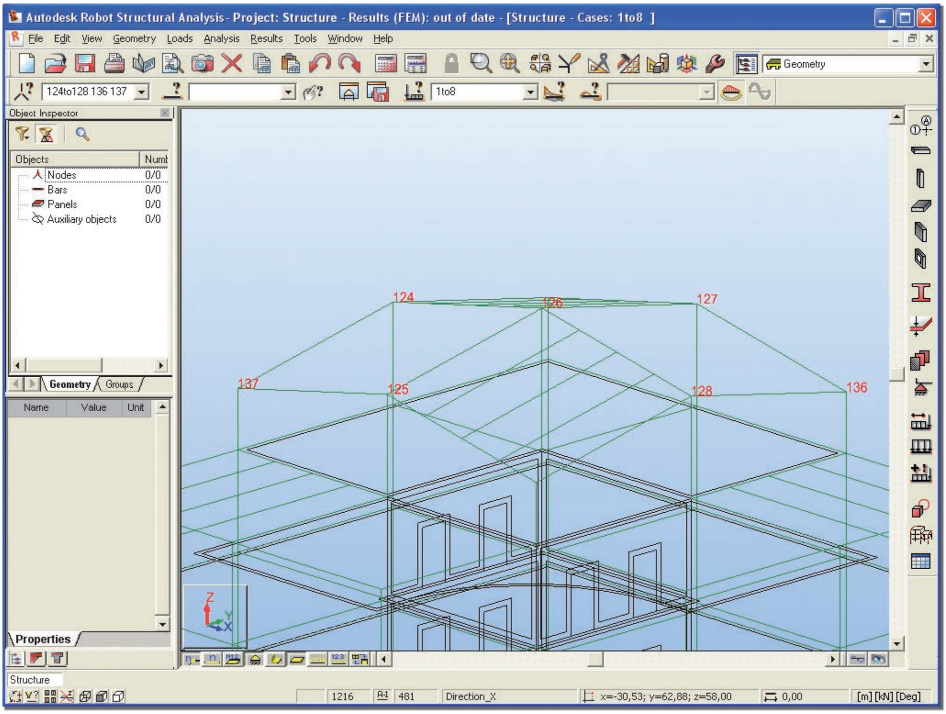


Figure 35 -Nodes numbers necessary to bracing definition.

11. **Close** the **Bars** dialog box.

12. Press the icon  to turn sections display on.

13. Bracings should look as shown below:

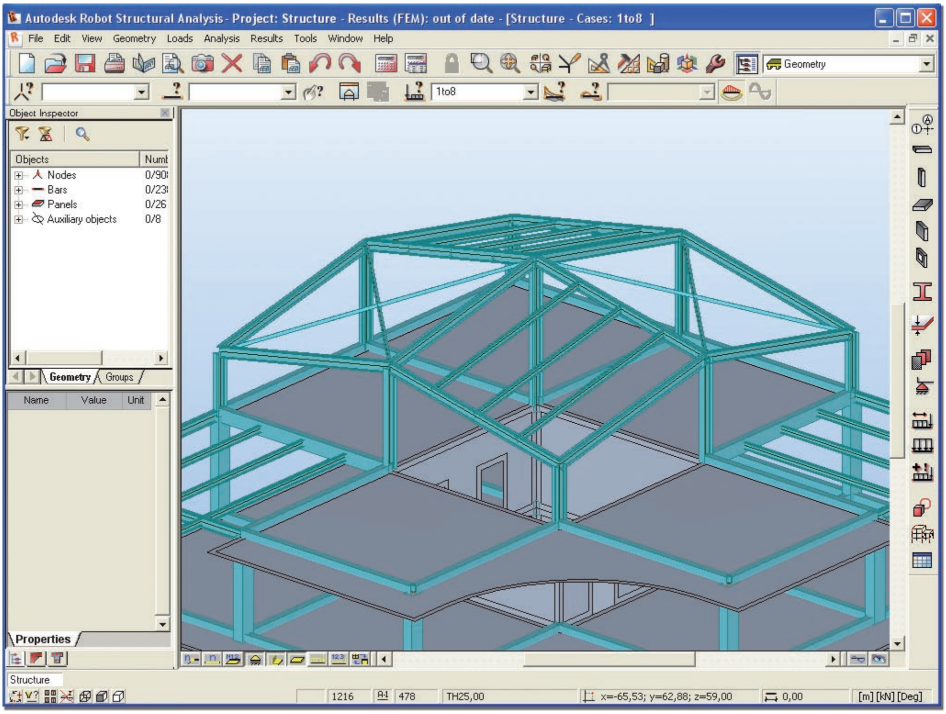


Figure 36 -View of defined bracings.

UPDATE REVIT MODEL FROM ROBOT

Updating Revit® Structure project

In this step, we learn how to update model data after analysis and changes in Robot:

1. Go back to Revit® Structure.

2. Choose the **extensions4revit** menu and select the **Extensions Manager** option:

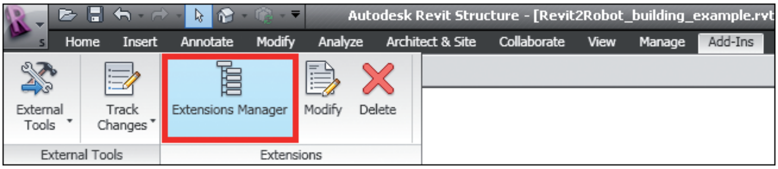


Figure 37 - Extensions4revit menu

3. **Extensions Manager** dialog box will appear (see below):

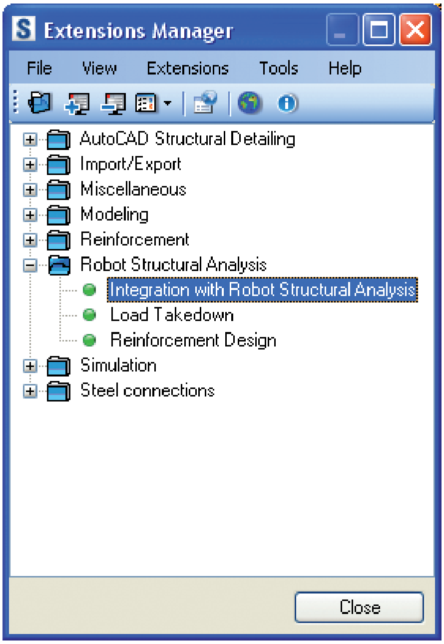


Figure 38 - Extensions Manager dialog box.

4. To open **Integration with Robot Structural Analysis** double click appropriate option (see above)

5. In dialog box shown below click **Update model from Autodesk Robot Structural Analysis** option and click OK:

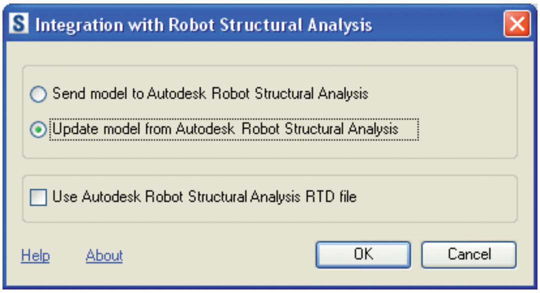


Figure 39 -Integration with Robot Structural Analysis (update model option).

6. Next, **Integration with Robot Structural Analysis – Update Options** dialog box will appear:

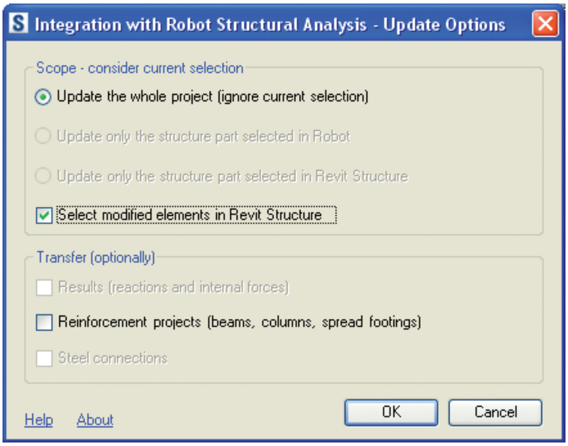


Figure 40 -Integration with Robot Structural Analysis - Update Options dialog box

7. Leave default settings as shown above and click OK.

INFO

Integration with Robot Structural Analysis – Update Options dialog box allows you to configure update parameters

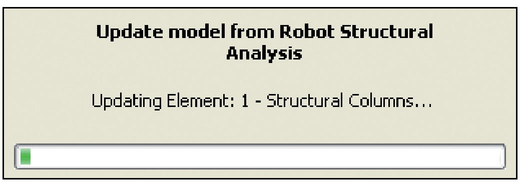
Scope – consider current selection – possibility of update whole structure or selected elements

- Update the whole project (ignore current selection) – all elements will be taken into account in update process
- Update only the structure part selected in Robot – only selected elements in RSA will be updated
- Update only the structure part selected in Revit Structure – only selected elements in Revit will be updated
- Select modified elements in Revit Structure – if some elements are new or changed then they will be selected in Revit after update process

Transfer (optionally)

- Results (reactions and internal forces) – allows to transfer internal forces for Revit load cases in elements (option available only if the structure is calculated in RSA).
- Reinforcement projects (beams, columns, spread footings) – defined reinforcement in listed elements is transferred from concrete design modules in RSA to Revit
- Steel connections – option not available in current version

8. During sending process the following splash screen is displayed:



INFO

It is often the case that the user can generate sections in Robot that are not in the Revit® Structure database. Simple feature allows the user to add this database information without interrupting or abandoning the transfer of data.

9. **Autodesk Robot Structural Analysis To Revit Update Log** dialog box may appear:

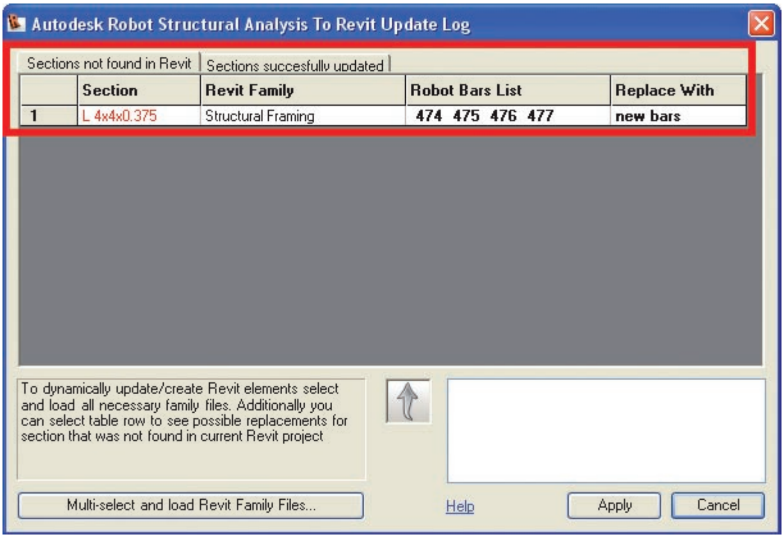


Figure 40 -Unknown section during updating process.

10. Click **Multi-select and load Revit Family Files…** button (bottom, left corner of the box).

11. In the opened dialog box select **L-Angle.rfa family** file (default localization: C:\Documents and Settings\All Users\Application Data\Autodesk\RST 2008\Imperial Library\Structural\Framing\Steel) to load unavailable section family:

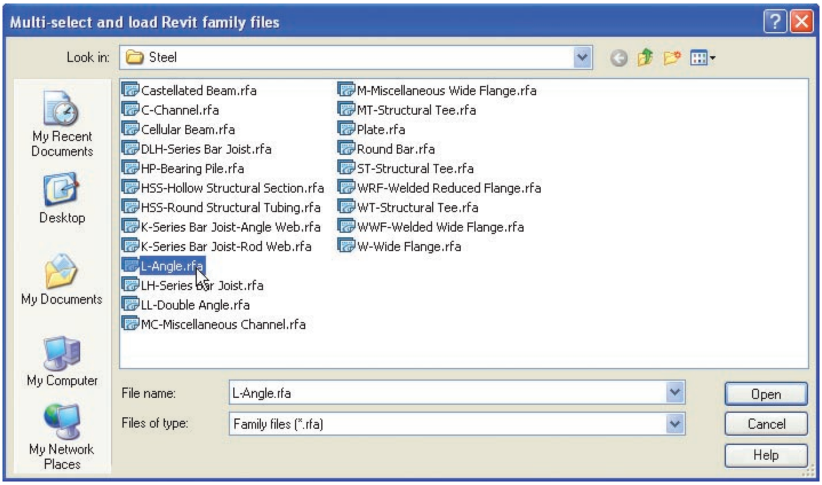


Figure 41 -Selection of missing Revit family file.

12. Press **Open** to open the database and this will close the window and continue the updating process.

13. The **Update model from Robot** splash screen is displayed once again. A progress information shows each step of the transfer process.

14. **Autodesk Robot Structural Analysis To Revit Update Log** dialog box will appear once again showing information about successfully updated sections:

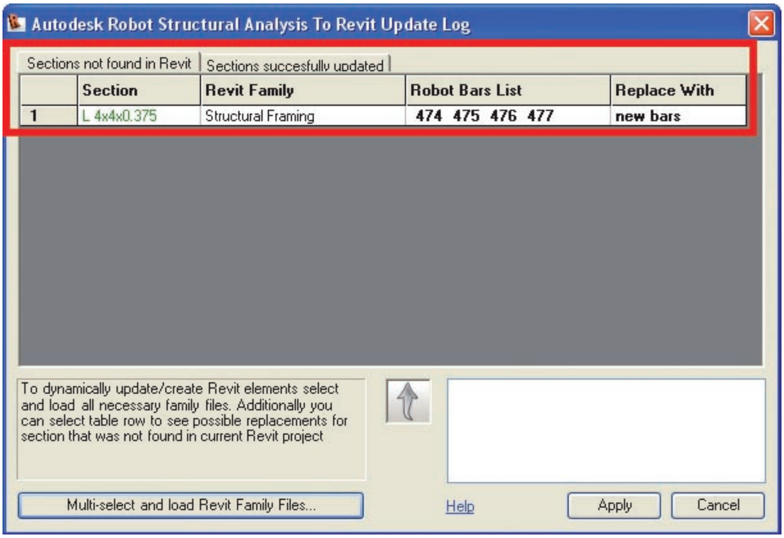


Figure 42 -Sections succesfully updated list.

15. Click **Apply** to close the box.

16. After finish sending data you can see an events report, to do this click **Yes**.

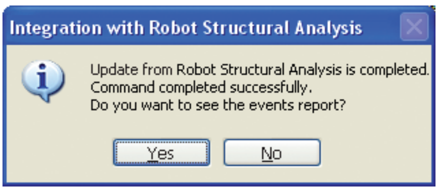


Figure 43 -Events report after updating data selection.

17. List of messages (and also warnings, if any will appear - see below):

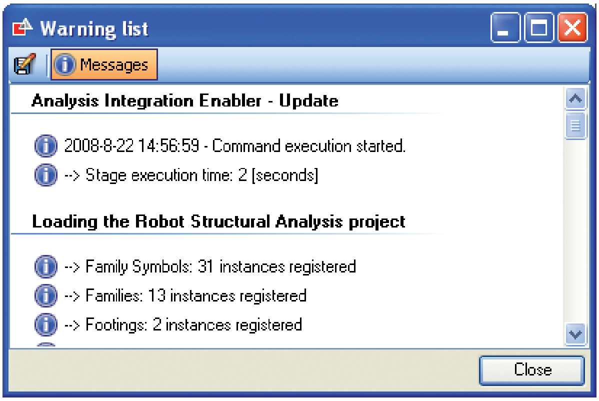


Figure 44 -Events report.

18. **Close** the box.

19. Updated model of structure should look as shown below:

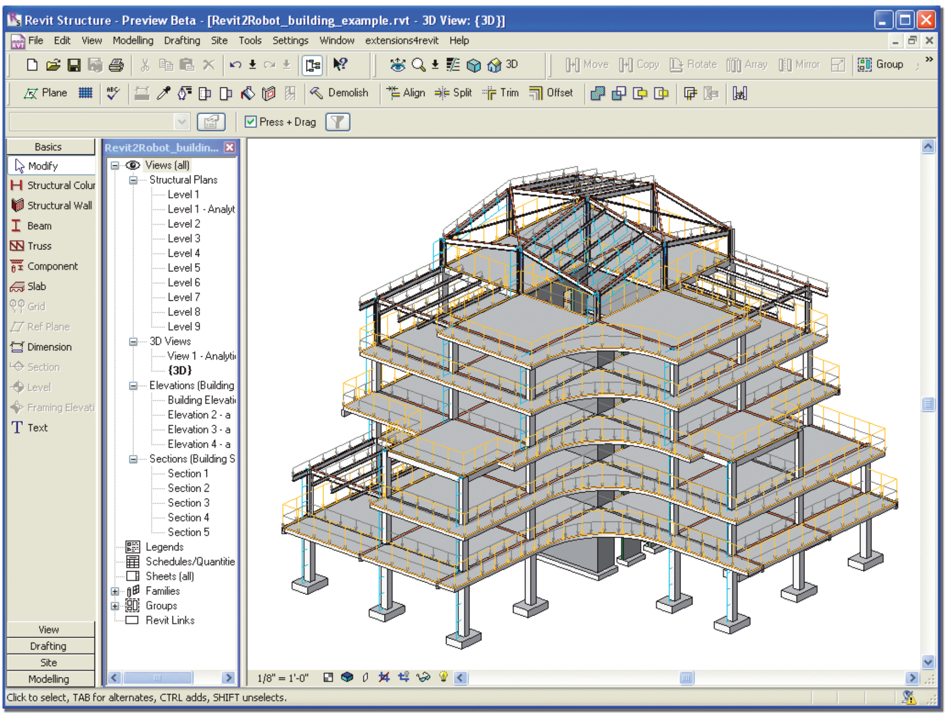


Figure 45 -Updated model of building in Revit Structure.

Model changes presentation

In this step, we can check and display changes in the model:

1. Zoom to view upper storey as shown below, notice that lateral beams **W 8x10** has been deleted and new bracings added (as changed in Robot):

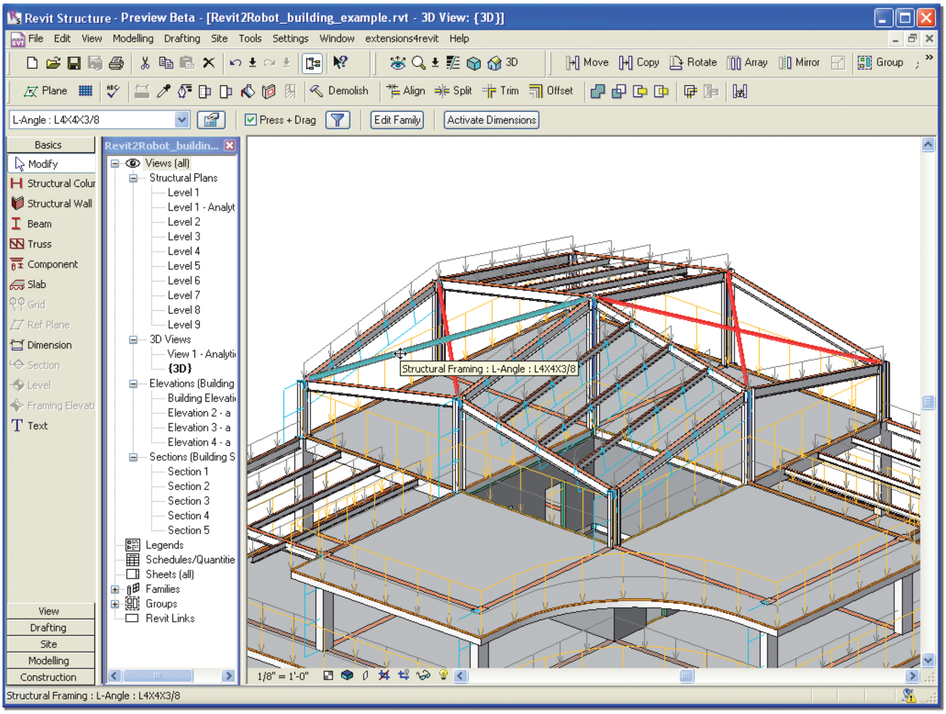


Figure 46 -View of added bracings.

2. Go with the mouse cursor to the one of the new beams and select it to check the section (**L4x4x3/8** should be displayed).

3. Zoom to view second storey on the left side of the building as shown below: