TWO-DIMENSIONAL FRAME
– SOLUTION IN ROBOT SYSTEM

HUMAN CAPITAL
NATIONAL COHESION STRATEGY

EUROPEAN UNION
EUROPEAN
SOCIAL FUND

Adam Wosatko

v. 1.0, February 2012

1. Type of project and regional settings

Definition of example. Solve the given portal frame plotted in Fig. 1 using the ROBOT package. Assume:

- material: concrete C20/25 ($E = 30$ GPa)
- RC beam: rectangular $A_1 = 0.3$ m $\times 0.5$ m
- RC column: rectangular $A_2 = 0.3$ m $\times 0.3$ m

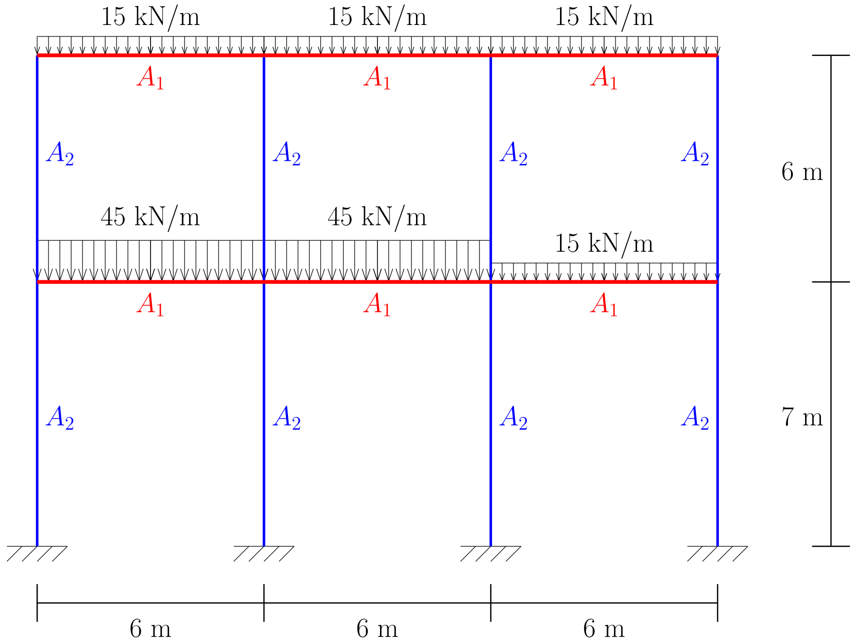
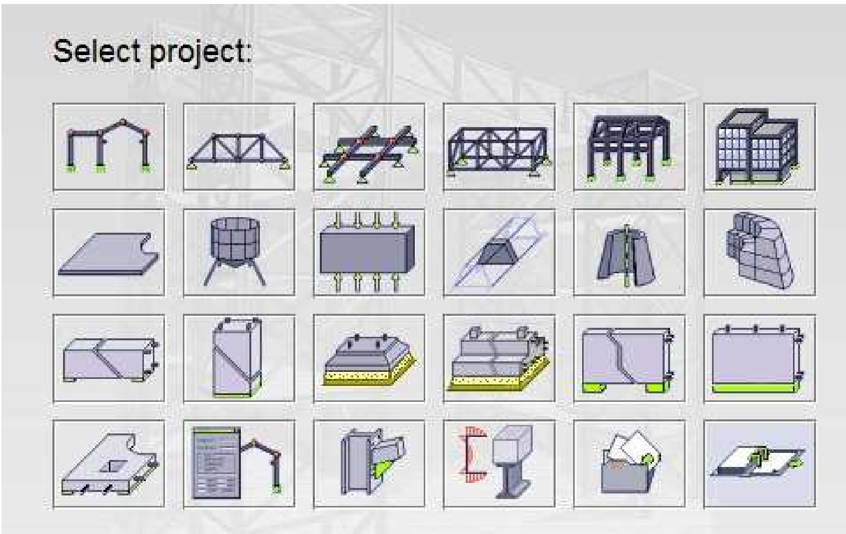
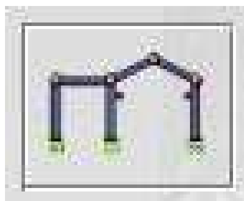


Fig.1. Diagram of portal frame.



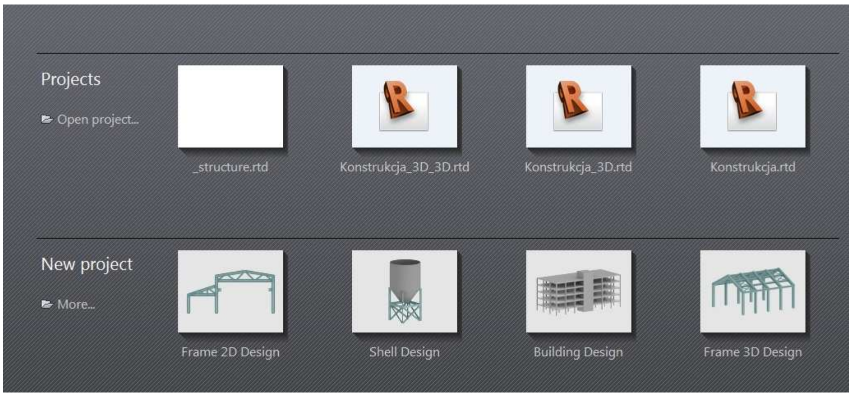
(a) Window of project selection.



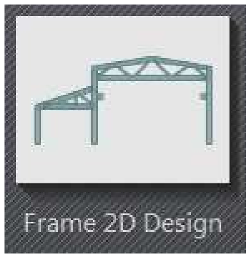
(b) Frame 2D icon.

Fig.2. Type of project (selection).

Type of project. Firstly we select the analysis of two-dimensional frame from among possible tasks (cf. Fig. 2(a) or Fig. 3(a) in case of ROBOT 2012). In this case button like in Fig. 2(b) or Fig. 3(b) should be pressed.



(a) Window of project selection.



(b) Frame 2D icon.

Fig.3. Type of project (selection) – ROBOT 2012.

Fig.4. Regional settings.

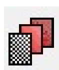
Regional settings. Immediately after project selection we select language and proper standards. From the top menu we pick *Tools* → *Preferences* and change the following window options: *Regional settings: Eurocode*, *Working language: English*. Printout language self-adapts. We accept this settings (cf. Fig. 4).

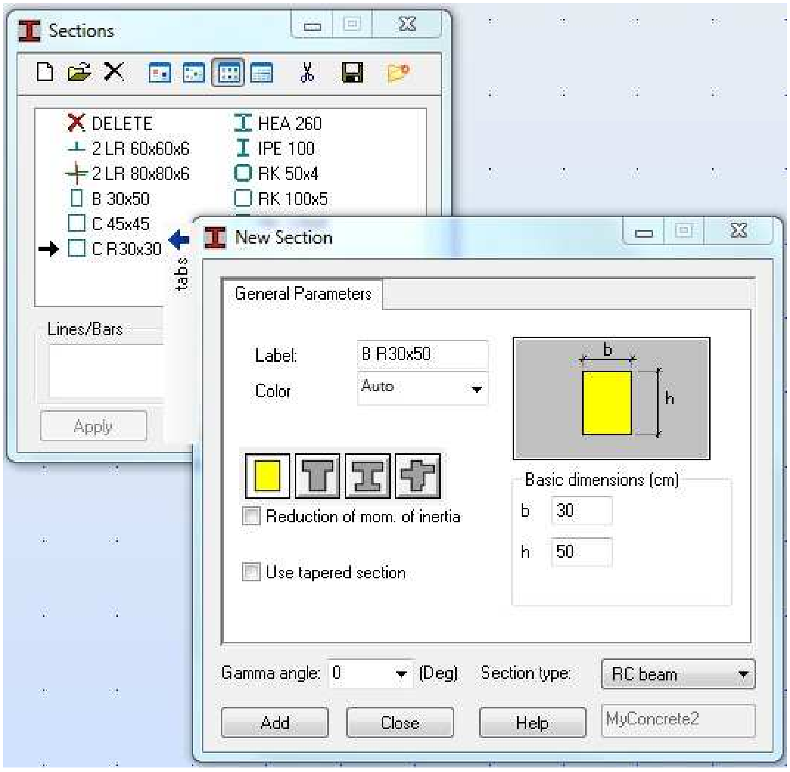
2. Sections and material definition

Before we construct the model of the portal frame we should determine necessary bar cross-sections and the type of material.

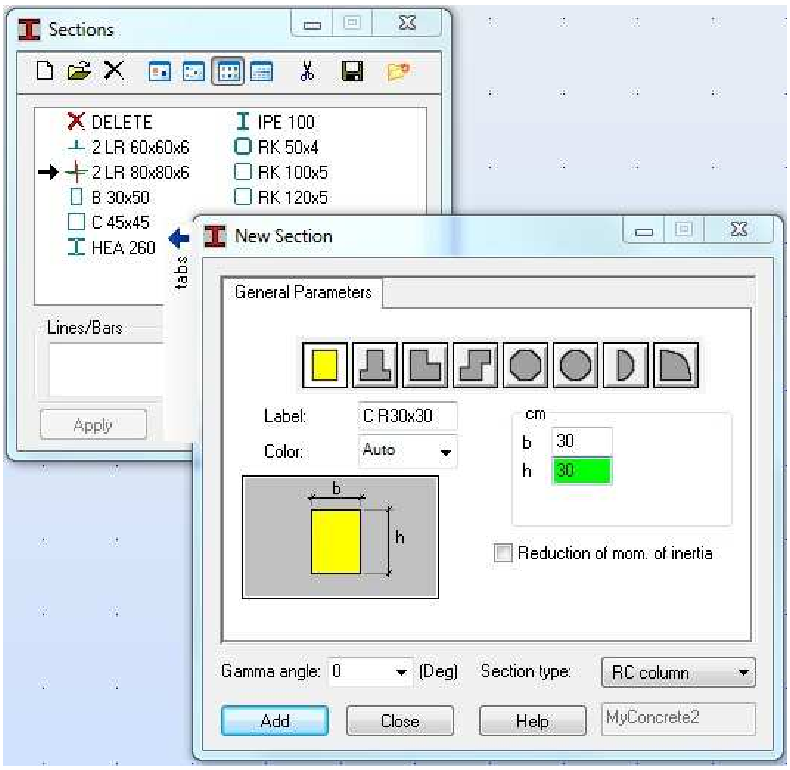
Sections. We choose from the top menu *Geometry* → *Properties* → *Sections* or the icon  called *Bar Sections* from the toolbar on the right. In the window we define *New Section* – this is the icon with small white blank page. In the next window we set the section type as *RC beam*, rectangular, and complete dimensions: $b = 30$ cm, $h = 50$ cm. Label *BR 30x50* remains without change. At the end press button *Add*. In case of problems please refer to Fig. 5(a).

Analogically we define the next section as rectangular *RC column* with dimensions $b = 30$ cm, $h = 30$ cm. Label *CR 30x30* is the default (cf. Fig. 5(b)). It is important that button *Add* and button *Close* should store the settings and turn off the windows.

Material. Now we can assign a material to the sections defined previously. We choose the icon  called *Materials* from the right-hand toolbar or the following options from the top menu: *Geometry* → *Materials*. In popup window we select concrete *C20/25* as material and next point to the suitable label of section (*BR 30x50* and then *CR 30x30*). *Apply* confirms our selection, then press *Close*.



(a) RC beam section.



(b) RC column section.

Fig.5. Definition of sections.

3. Construction of structure together with supports

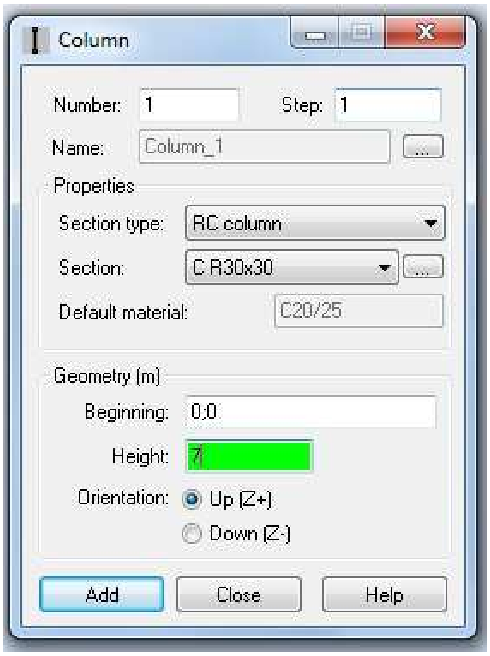
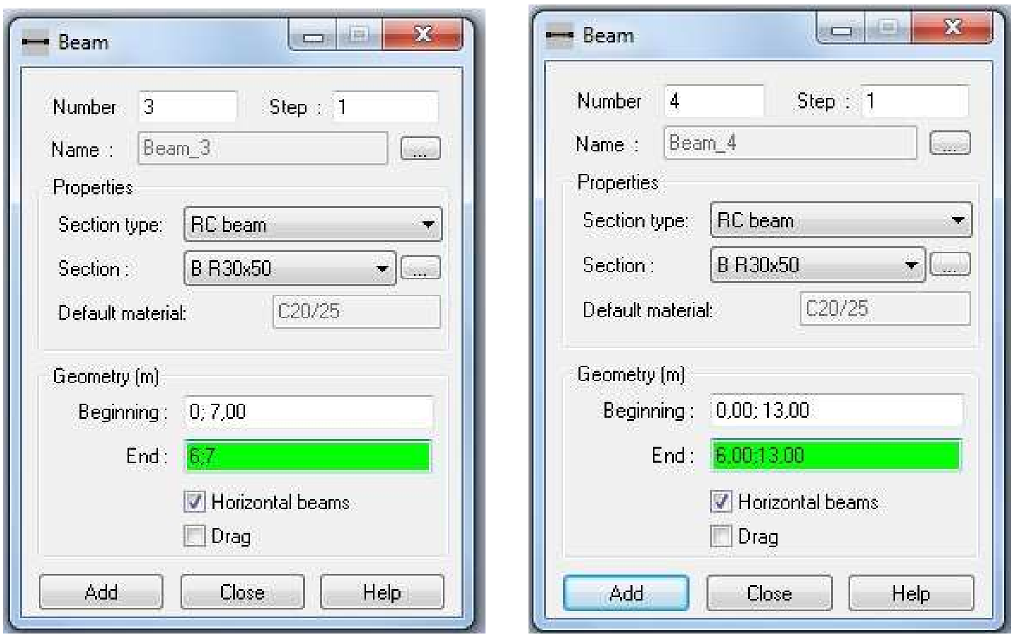


Fig.6. Definition of column.

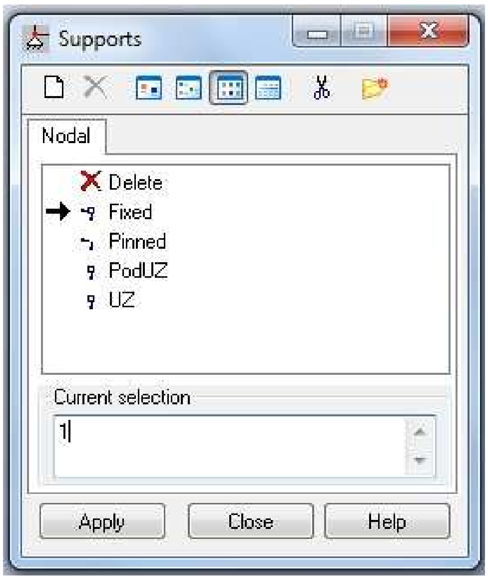
First building block. To construct the structure model in a proper way we open the *Column* window starting from the top menu *Geometry* → *Columns*. In the window we pick properties, write geometry parameters according to Fig. 6 and *Add* this element. We do not close the window. Now we can change insertion point and height in geometry parameters – *Beginning* has coordinates 0; 7 and column *Height* is equal to 6. We *Add* second column and *Close* the window.

Analogical operations are performed for the beams. Starting from *Geometry* → *Beams* we open the window and introduce all settings as in Fig. 7 for both lower and upper beams.

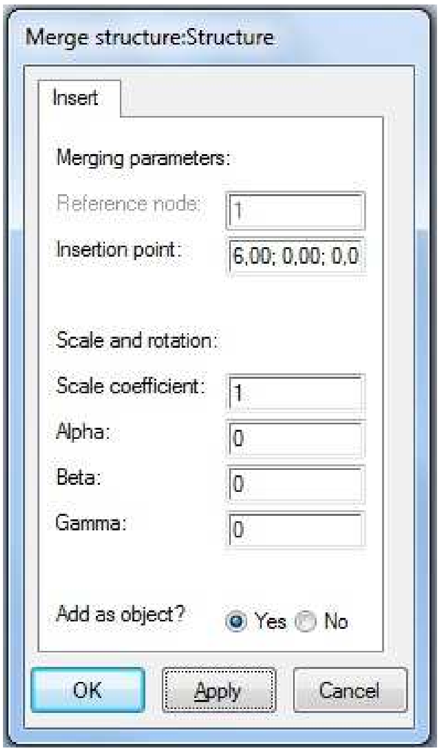


(a) Lower beam. (b) Upper beam.

Fig.7. Definition of beams.

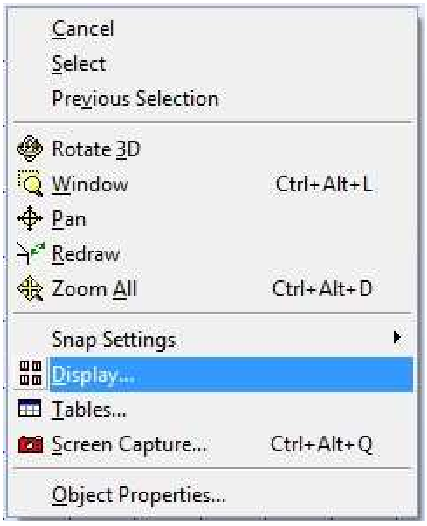


(a) Support definition.

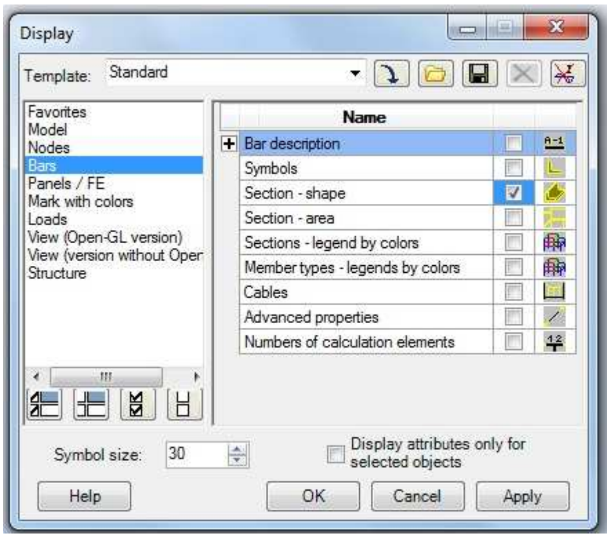


(b) Merging structure window.

Fig.8. Support and merging structure windows.



(a) Workspace window.



(b) Display window.

Fig.9. Displaying different attributes.

To define boundary conditions we select from the top menu *Geometry* → *Supports*, pick *Fixed* option in the window as in Fig. 8(a) and assign to node 1 (coordinates $0; 0; 0$) using mouse or writing node number. The small square denotes that the rotation as well as horizontal and vertical translation are blocked (all degrees of freedom).

Copying building block. Firstly, we should mark all elements of the structure which have been created until now. We can use shortcut key *Ctrl + A* or from the top menu *Edit* → *Select All*. Copy–paste operation works as usual (*Ctrl + C* – *Ctrl + V* or *Edit* → *Copy* – *Edit* → *Paste*), but we decide where the duplicated part should be inserted. In the window as in Fig. 8(b) we introduce insertion point $6; 0; 0$ and choose *Yes* answering the question *Add as object?*. To confirm we press button *Apply*. We

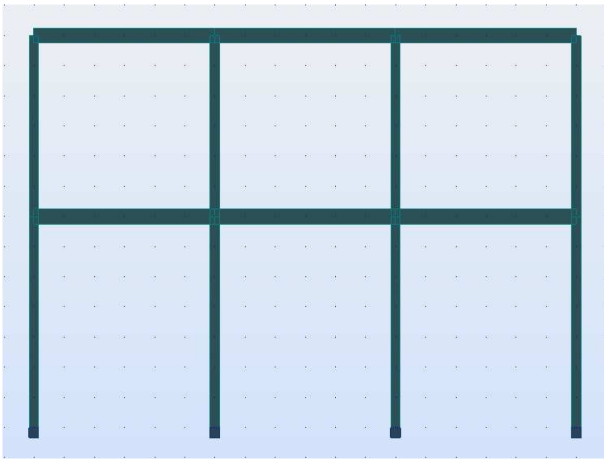


Fig.10. Display of section-shape attribute of the frame.

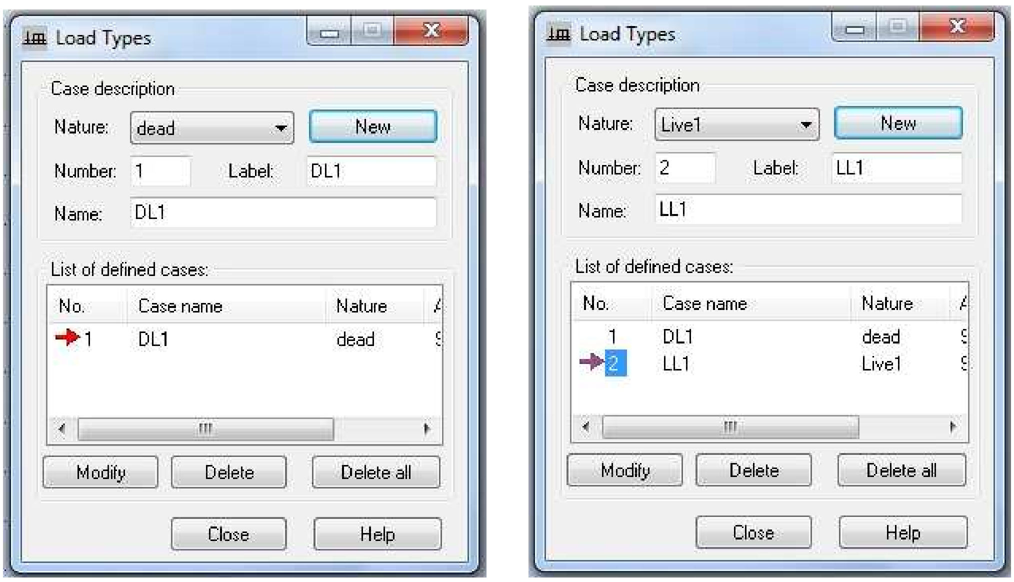
repeat the whole procedure of copying, but the second time the insertion point is 12; 0; 0. After pressing *Apply* we confirm using *OK*. Now we should unmark the structure (*Esc*).

The last operation is copying the originally first and second columns together with the support at node 1. We mark this part of the structure, run copy–paste, introduce insertion point 18; 0; 0 and again select *Yes* answering the question *Add as object?*. Buttons *Apply* and *OK* confirm our action.

Division of building blocks. Using *Edit* → *Substructure modification* → *Explode Object* from the top menu we decompose the whole structure in order to have capability to modify of each element separately.

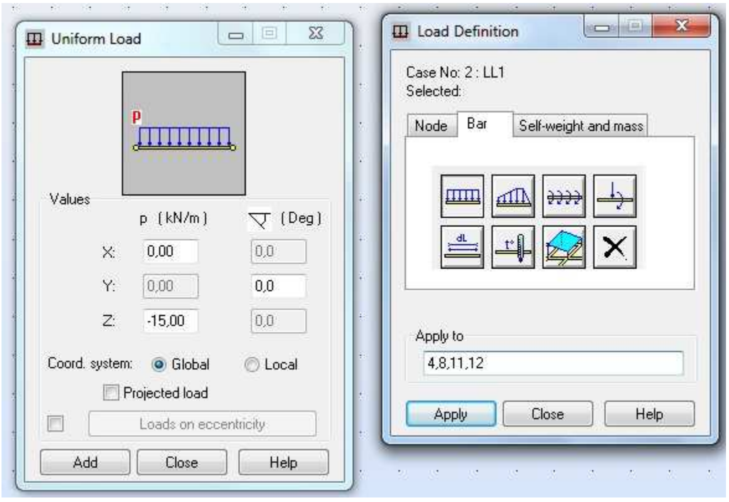
Display of attributes of the structure. If we would like to display e.g. the section shape for all elements of the frame we should press the right mouse button and select in the window *Display...* option as in Fig. 9(a). In the second window presented in Fig. 9(b) among *Bars* attributes we tick *Section-shape* and confirm using *Apply* and *OK*. The structure will be shown as in Fig. 10. In a similar way we should untick attributes in order to return to the original view.

4. Loading

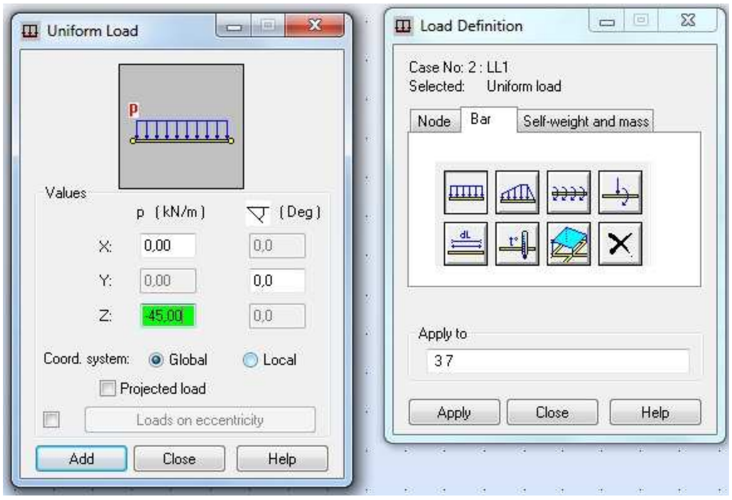


(a) Dead load. (b) Live load.

Fig.11. Load types window.



(a) $p = 15$ kN/m.



(b) $p = 45$ kN/m.

Fig.12. Loading definition.

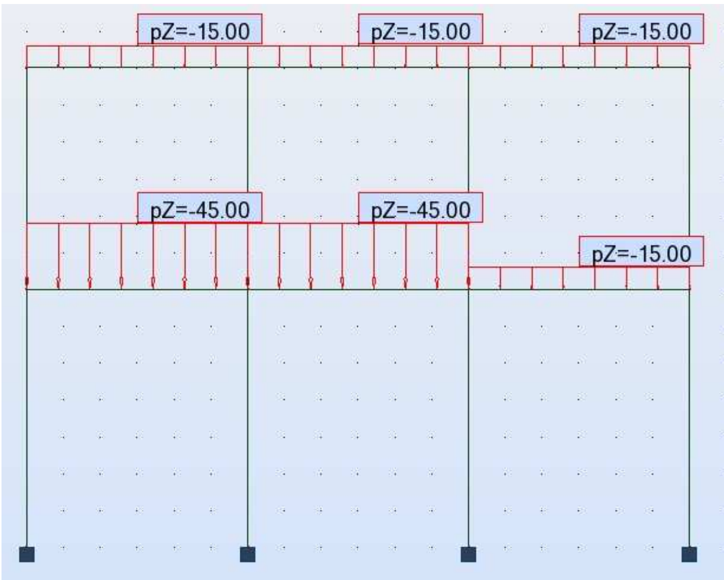


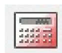
Fig.13. Model of frame with loading and supports.

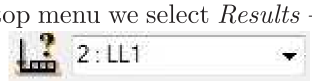
Load types. Before we define the loading of the frame structure we should create suitable types of loading. Firstly, we generate dead load to avoid an automatic addition of this load to the next type. Starting from the top menu *Loads* → *Load Types* we open a window and push button *New* (or *Add* in case of ROBOT 2012). Now the dead load is created. We change load nature into *Live* similarly to Fig. 11(b) and once again push button *New* (or *Add* in case of ROBOT 2012) to create live load. This load remains active, so we can *Close* the window.

Load definition. We execute *Loads* → *Load Definition* from the top menu or right-hand toolbar. In the window we choose *Bar* options and pick first icon *Uniform load* as shown in Fig. 12(a). In the new window we enter −15 kN/m in direction *Z* and confirm this value by button *Add*. Now we tick elements in the workspace which should have this loading applied to, or write in the previous window numbers of elements in the field *Apply to*, cf. Fig. 12(a). Analogical operation should be performed for the larger load equal to −45 kN/m in direction *Z*, cf. Fig. 12(b). The model with loading and supports is presented in Fig. 13.

5. Calculation and results

Our analysis is static, elastic and the most important aim is the presentation of generalized stresses (cross-section forces) as a result.

Calculation. We choose from the top menu *Analysis* → *Calculations* or press the icon  called *Calculations* in order to compute the structure.

Results – diagrams. The results for the portal frame can be presented if we open *Diagrams* window. From the top menu we select *Results* → *Diagrams for Bar*. Before we draw the diagrams we should check if live load 

is active. Next we can set the last option *Parameters* according to Fig. 14. Going back to *NTM* option we can draw suitable diagrams for shearing forces or/and bending moments or/and normal forces. Please, see Fig. 15 to verify your computations.

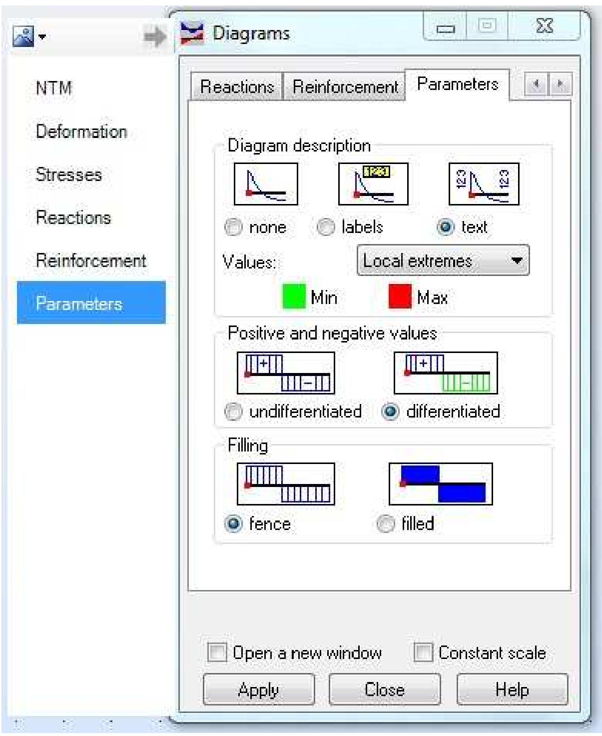
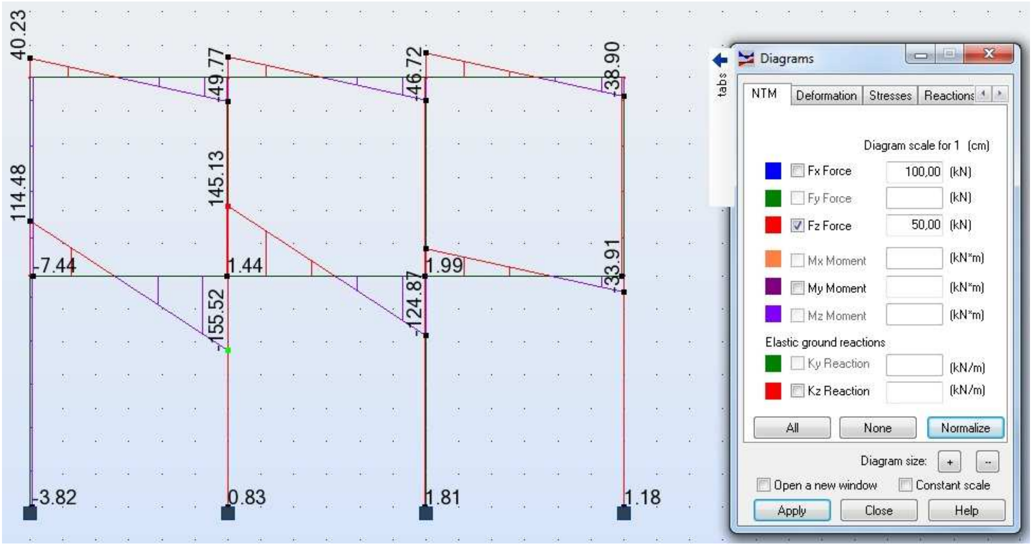
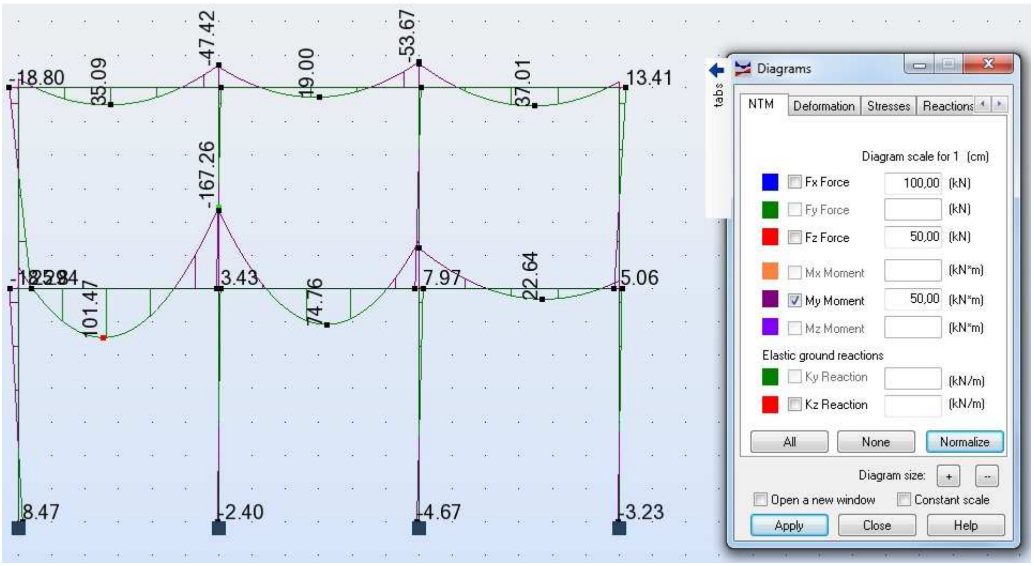


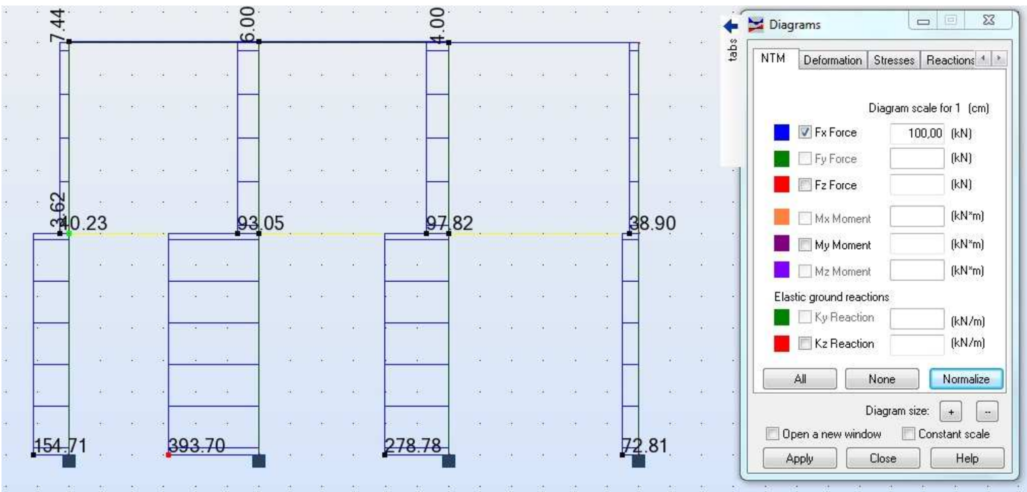
Fig.14. Parameter settings for diagrams.



(a) Shearing forces F_z.

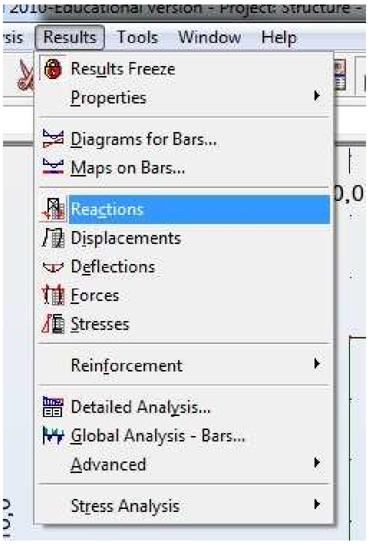


(b) Bending moments M_y.

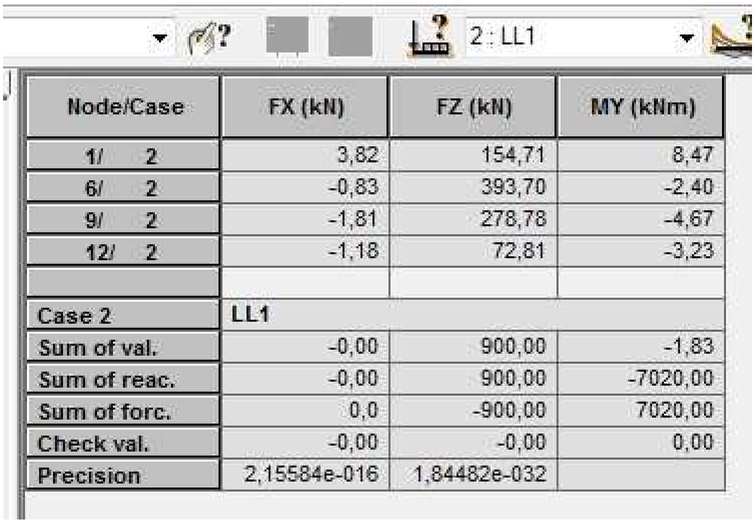


(c) Normal forces F_x.

Fig.15. Diagrams of forces.



(a) How to display reactions in table?



Node/Case	FX (kN)	FZ (kN)	MY (kNm)
1/ 2	3,82	154,71	8,47
6/ 2	-0,83	393,70	-2,40
9/ 2	-1,81	278,78	-4,67
12/ 2	-1,18	72,81	-3,23
Case 2	LL1		
Sum of val.	-0,00	900,00	-1,83
Sum of reac.	-0,00	900,00	-7020,00
Sum of forc.	0,0	-900,00	7020,00
Check val.	-0,00	-0,00	0,00
Precision	2,15584e-016	1,84482e-032	

(b) Displaying of reactions.

Fig.16. Reactions in table.

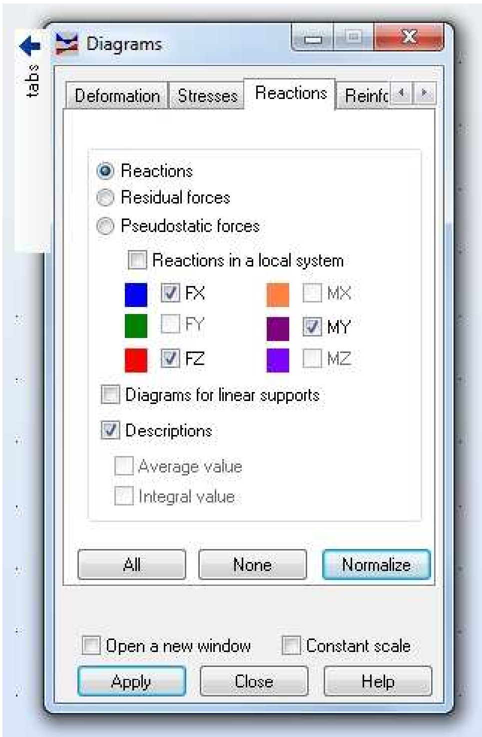


Fig.17. Setup of reactions – options in diagrams window.

Results – reactions. Reactions can be observed in two manners. If we choose from the top menu *Results* → *Reactions* like in Fig. 16(a) then the table given in Fig. 16(b) appears. The other way is running *Results* → *Diagrams for Bar* and opening option of *Reactions* in *Diagrams* window, cf. Fig. 17. Next we select suitable settings and analyze all reactions. Fig. 18 shows reactions together with loading.

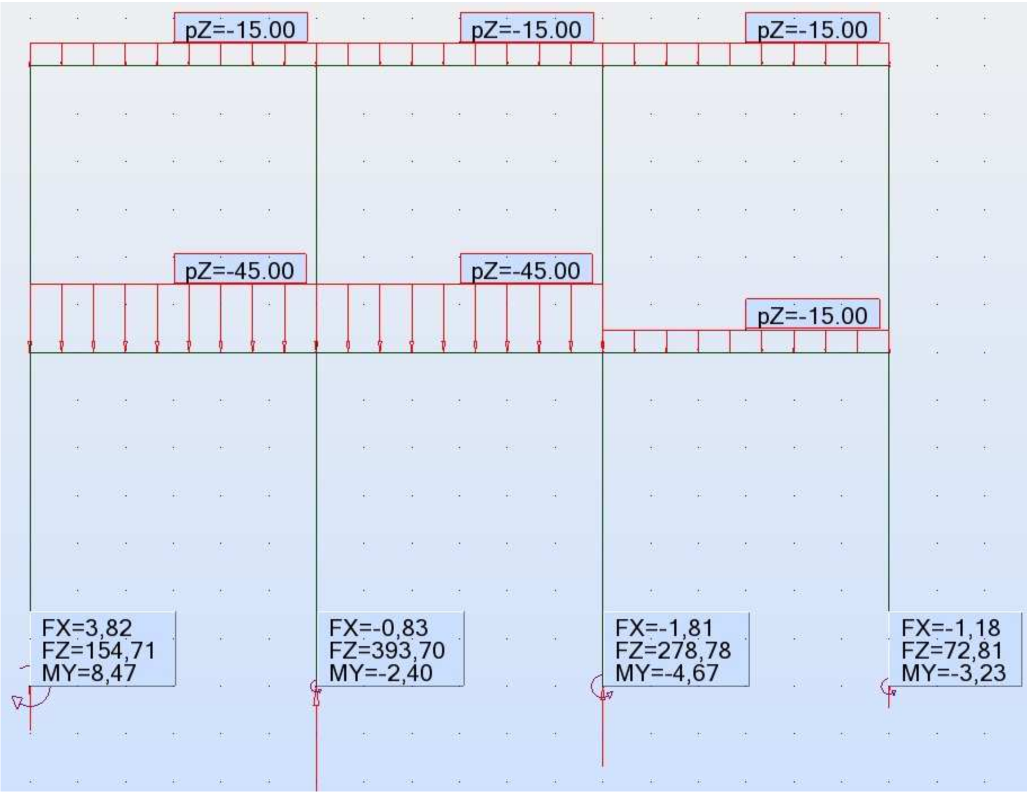


Fig.18. Reactions and loading.