NUMERICAL ANALYSIS OF A PANEL IN ROBOT SYSTEM .

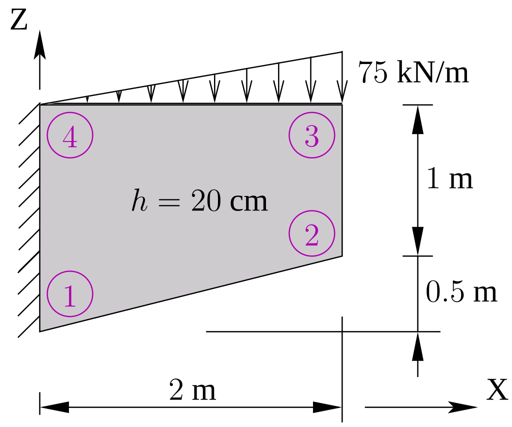
Adam Wosatko and Anna Stankiewicz

v. 2.0, February 2013

Project ”The development of the didactic potential of Cracow University of Technology in the range of modern construction” is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education

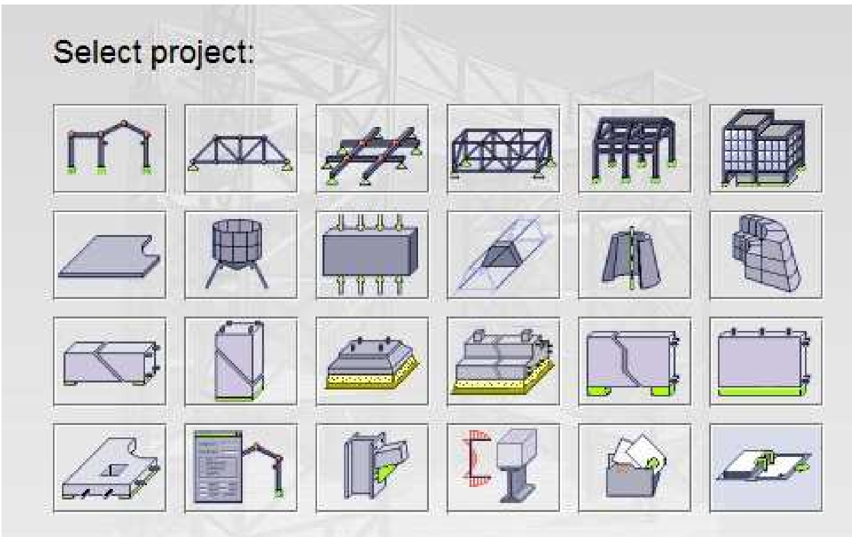
1 Type of project and regional settings

Definition of example. Solve the panel shown in Fig. 1 using the ROBOT package. Geometry and material data are given below.

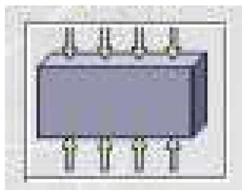


$E = 25$ GPa $\qquad \nu = 0.16$

Figure 1: Geometry and material data.



(a) Window of project selection.



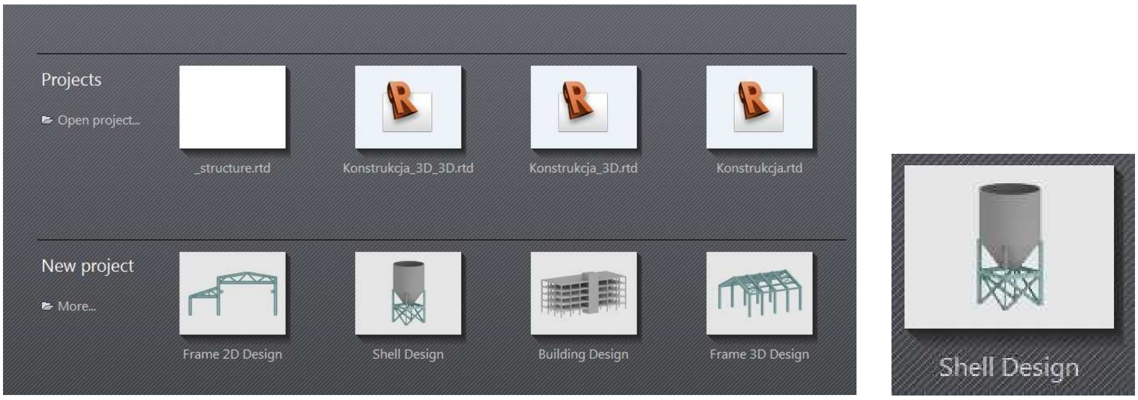
(b) Panel.

Figure 2: Type of project (selection).

Type of project. From the list of possible tasks choose the analysis of a panel structure (cf. Fig. 2). In the case of ROBOT 2012 the shell structure type should be selected (cf. Fig. 3) and as a next step, one has to choose *GEOMETRY* → *STRUCTURE TYPE* and select *Plane stress structure design* as shown in Fig. 4.

Regional settings. Immediately after the project selection select the language and proper standards. From the top menu pick *TOOLS* → *PREFERENCES* and change the following window options: *Regional settings: Eurocode*, *Working language: English*. The printout language self-adapts. Accept these settings (cf. Fig. 5).

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education



(a) Window of project selection. (b) Shell structure.

Figure 3: Type of project (selection) – ROBOT 2012.

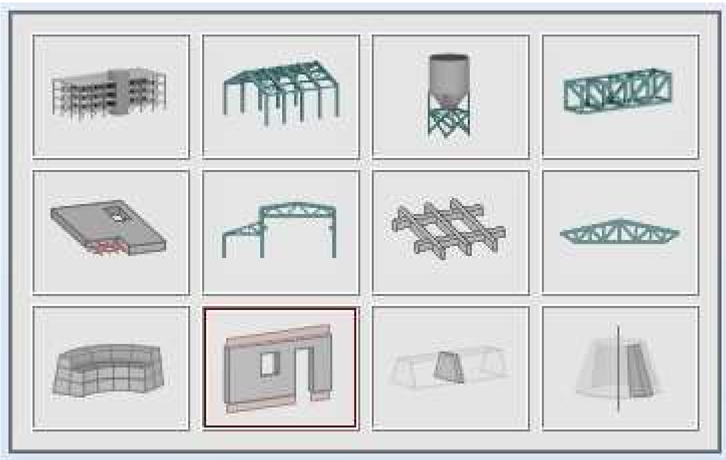


Figure 4: Type of task (selection) – ROBOT 2012.

2 Material and geometry settings

Material. In order to set material data choose *TOOLS* $\rightarrow$ *JOB PREFERENCES*. In the window (cf. Fig. 6) select *Materials* from the list on the left-hand side. Next, click the button *Modification* and choose the *Other* option. Introduce the name of the material (PLSTREmat for example) and choose data for elasticity. Introduce the values of Young modulus E, Poisson coefficient ν and find the value of Kirchhoff coefficient G according to the formula: $G = \frac{E}{2(1+\nu)}$. For the given data set G=10775.861 MPa. To finish the material definition, click *Add*, *OK* and close (click *OK*) the *Job Preferences* window.

Contour definition. To define the geometry of the panel choose *GEOMETRY* $\rightarrow$ *OBJECTS* $\rightarrow$ *POLYLINE* $\rightarrow$ *CONTOUR*. In the window shown in Fig. 7 on the left, choose the *Definition Method* - *Contour* and introduce the coordinates of corners of the considered panel in the part *Geometry*. Confirm the coordinates of the points by clicking the *Add* button every time. To close the window select *Apply*. The window in Fig. 7 on the right shows the defined contour.

Panel properties. Choose *GEOMETRY* $\rightarrow$ *PANELS*. In the window as in Fig. 8 on the left choose the *Contour Type* - *Panel*. Next, set the properties i.e. *Reinforcement* - *None*, *Model* - *Shell*. In order to assign the thickness, click the 'three dots' button and in the new window (cf. Fig. 8(b)) find material PLSTREmat in the list. Then, set the (constant) thickness Th=20 cm. Set the label TH20PS. Click *Add* and *Close* buttons. Now, in the window as in Fig. 8(a) choose *Creation with Internal point*, click anywhere in the area of the panel and close the window.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education

Figure 5: Regional settings.

3 Definition of static and kinematic boundary conditions

Kinematic boundary conditions. In order to prevent the movement of the left edge, choose *GEOMETRY* → *SUPPORTS*. In the window as shown in Fig. 9 select *Linear support* and choose *Pinned* or *Fixed* (in both boundary condition types the UX and UZ displacements are not allowed). Click the left edge of the panel to set the support and close the window.

Load types. Before the loading of the panel is defined, suitable types of loading have to be created. Choose *LOADS* → *LOAD TYPES* to open the window shown in Fig. 10(a). Press the *New* button (or *Add* in the case of ROBOT 2012) to create a new case of loading. Change load nature into *Live* as shown in Fig. 10(b) and click the *New* button again (or *Add* in the case of ROBOT 2012) to create live load. In the case of ROBOT 2012 a loading subnature can be given (cf. Fig. 11). This load remains active, and the window can be closed now. The dead load case is necessary to take the self-weight of the structure into account. Loading of the top edge of the panel is going to be set in the LL1 case. To define the loading of the panel, choose *LOADS* → *LOAD DEFINITION*. The window shown in Fig. 12(a) opens. Select *Surface* loading and introduce data according to the window in Fig. 12(b). Click *Add* and close the previous window.
In Fig. 13 the supported and loaded panel is shown.

4 Mesh generation and FEM analysis

Mesh generation. Open the toolbar Options of FE Mesh Generation shown in Fig. 16. Select the area of the panel (by clicking) and the third icon from the left in the Options of FE Mesh Generation toolbar. In the window shown in Fig. 15 select the *Meshing Method* and *Mesh Generation*. Press the  icon to generate the finite element mesh. It is possible to click *Advanced Options* to set more details in a window as in Fig. 15(b).

Analysis. To perform calculations choose *ANALYSIS* → *CALCULATIONS*.

5 Results

Results – contour plots. Before reading the values of displacements (or stresses) one needs to set the units and formats. Select *TOOLS* → *JOB PREFERENCES* → *UNITS AND FORMATS* → *OTHER* → *DISPLACEMENT*. Compare the window in Fig. 17. To see the displacement values as a text, choose *RESULTS* → *DISPLACEMENT*. The contour plots are available from *RESULTS* → *MAPS*. In Fig. 18 the distribution of stress component σ_{xx} called SXX is shown.

Results – diagram for a cross-section. To draw a diagram of the dependence of one stress tensor component on the coordinate along a selected cross-section choose *RESULTS* → *PANEL CUTS* and set options as shown in Figs. 19 to 21. Make sure that the *Filling the interior* option is not marked in the *Display* menu (available at the right mouse button).

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education

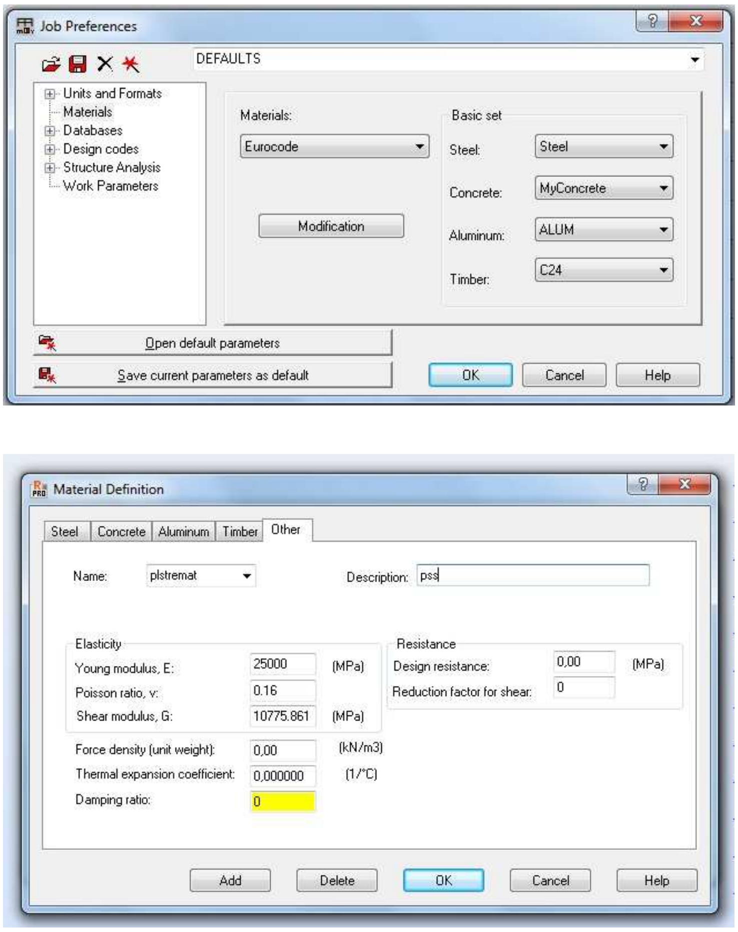


Figure 6: Definition of new material.

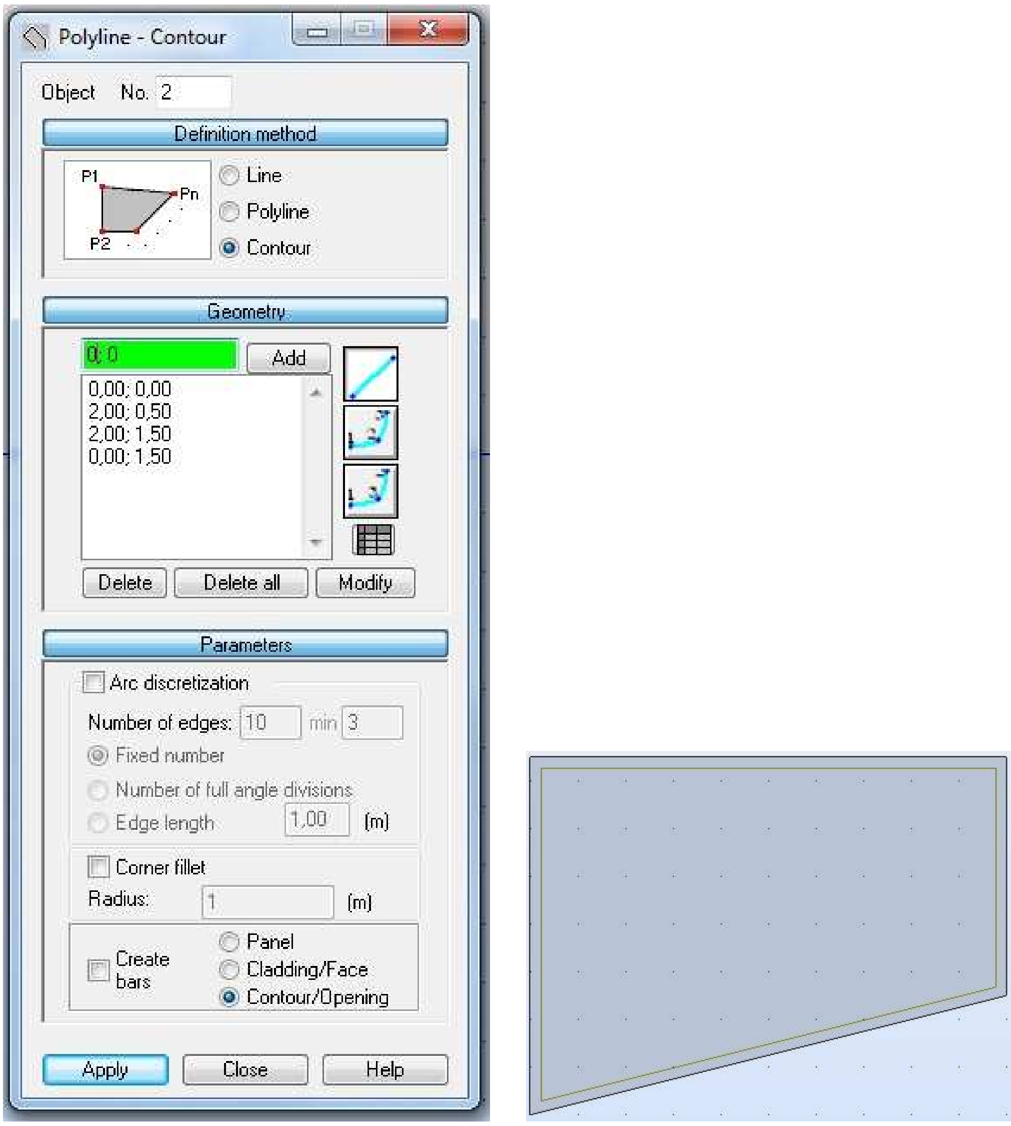
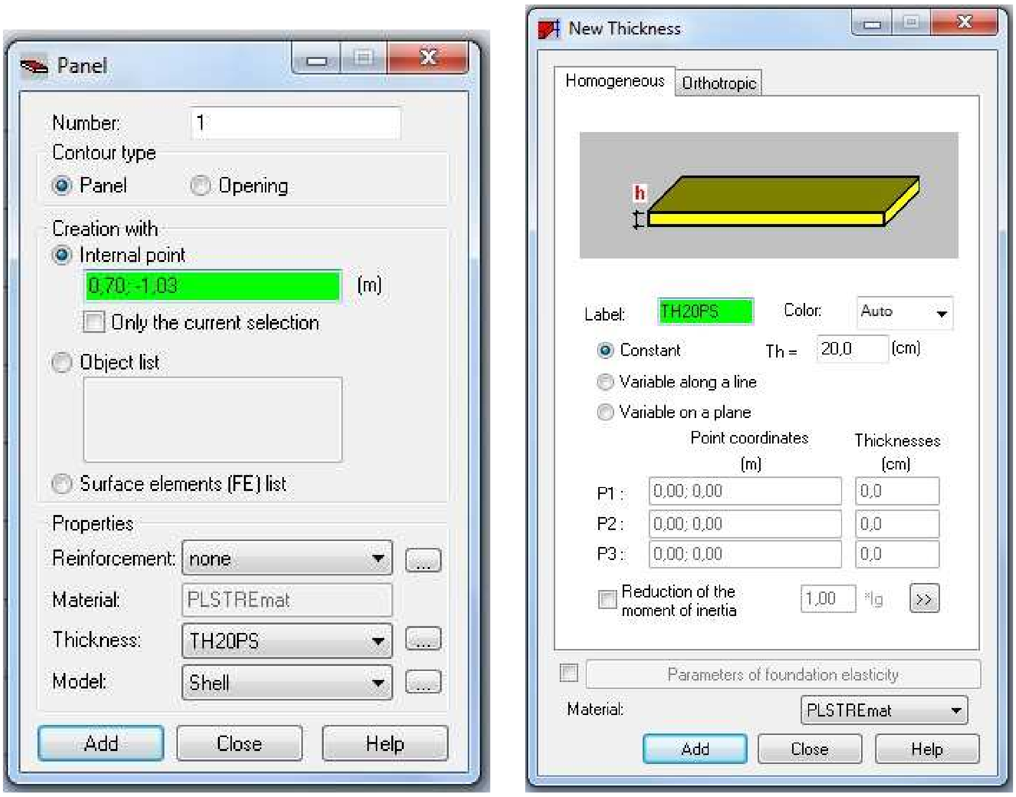


Figure 7: Definition of the panel contour.



(a) (b)

Figure 8: Definition of the panel continued.

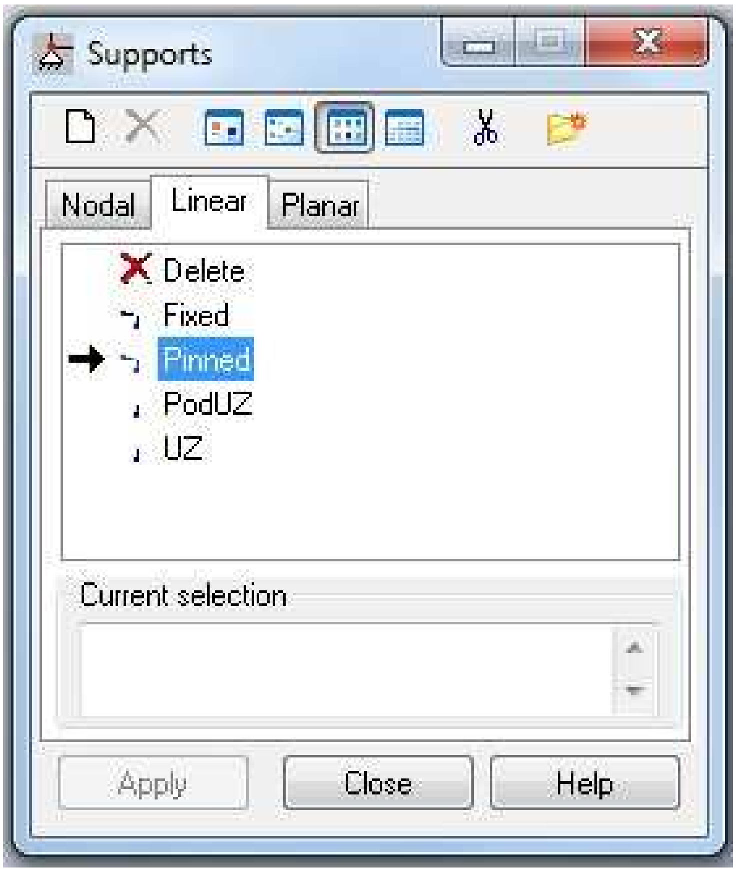
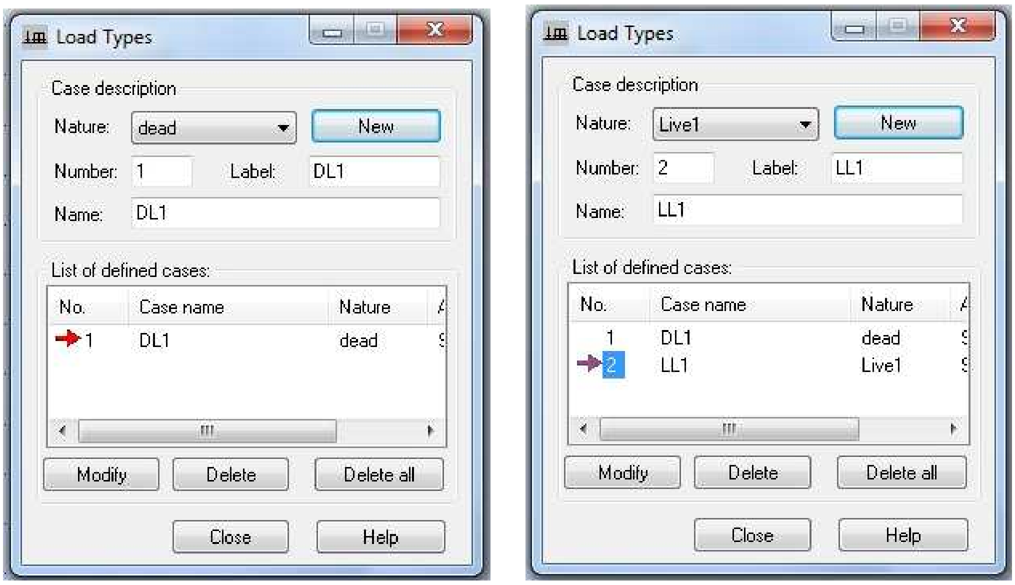


Figure 9: Kinematic boundary conditions.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education



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

Figure 10: Load types window.

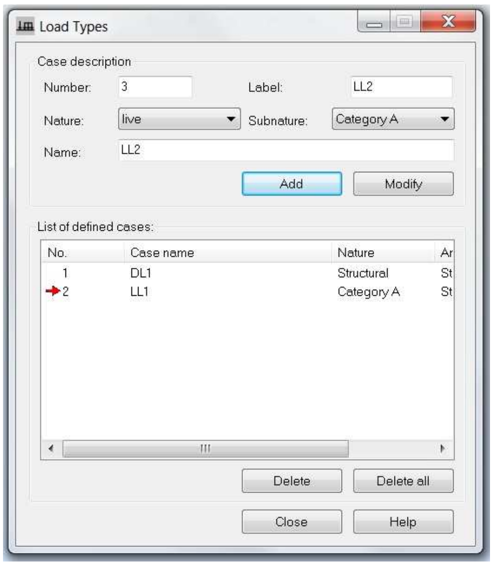
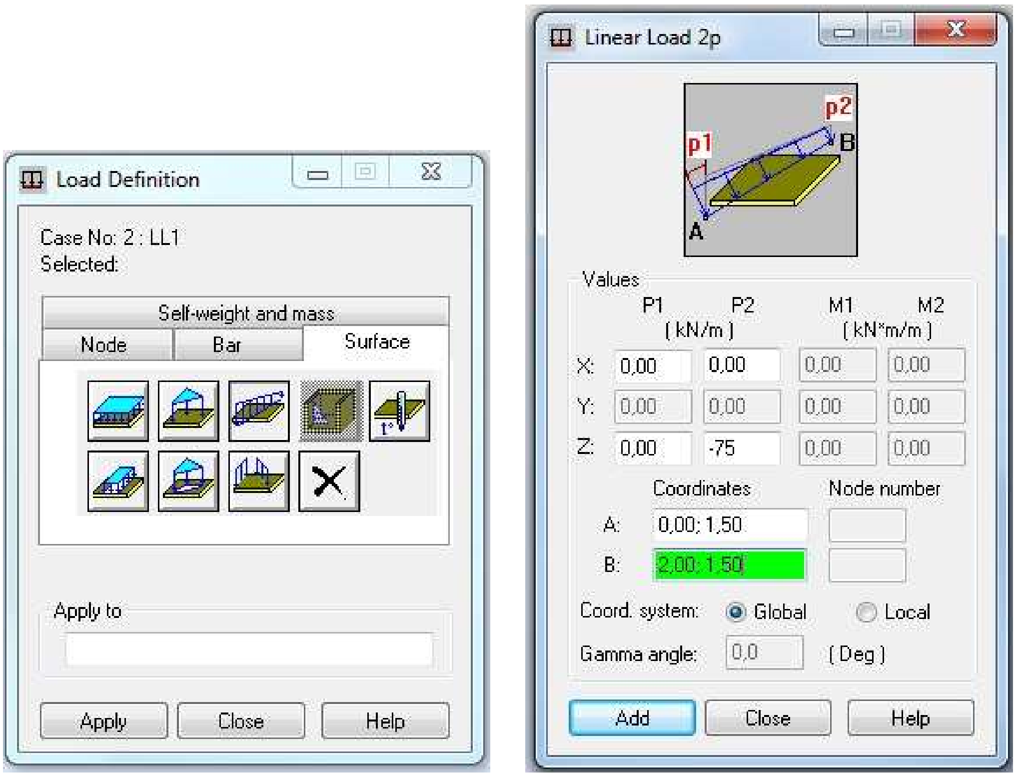


Figure 11: Load subnature option, live load - ROBOT 2012.

Project ”The development of the didactic potential of Cracow University of Technology in the range of modern construction” is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education



(a) Dead load.

(b) Live load.

Figure 12: Load definition.

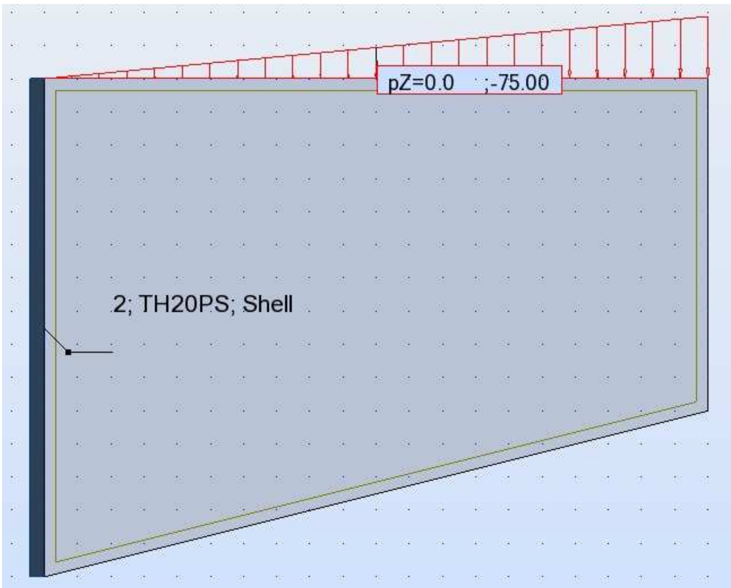
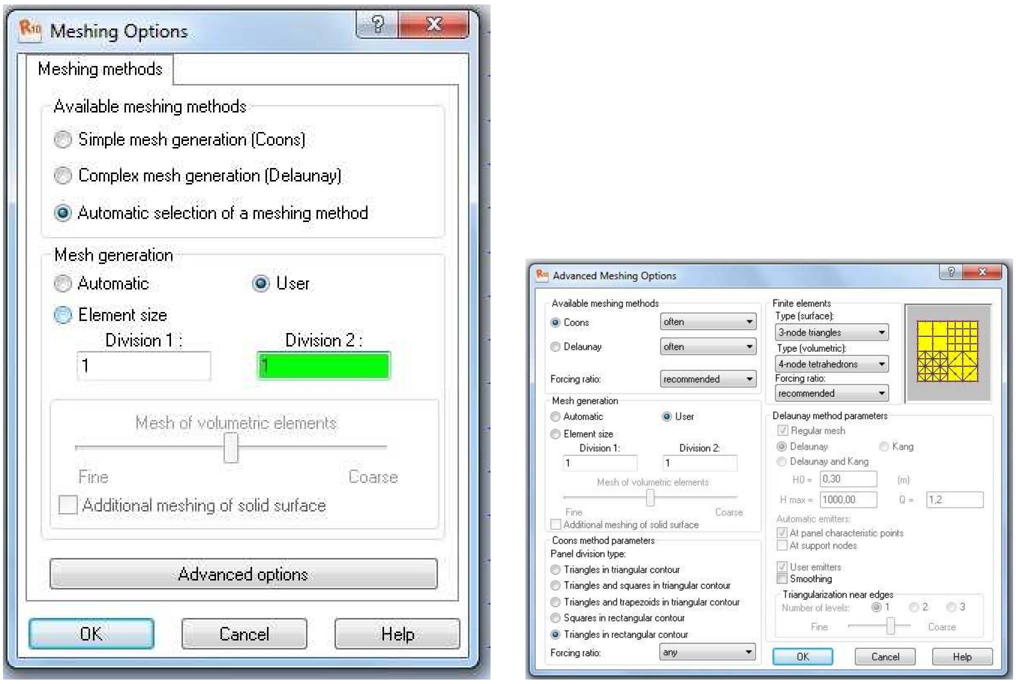


Figure 13: Kinematic and static boundary conditions.

Figure 14: Options of FE Mesh Generation.

Project ”The development of the didactic potential of Cracow University of Technology in the range of modern construction” is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education



(a) (b)

Figure 15: Mesh options.

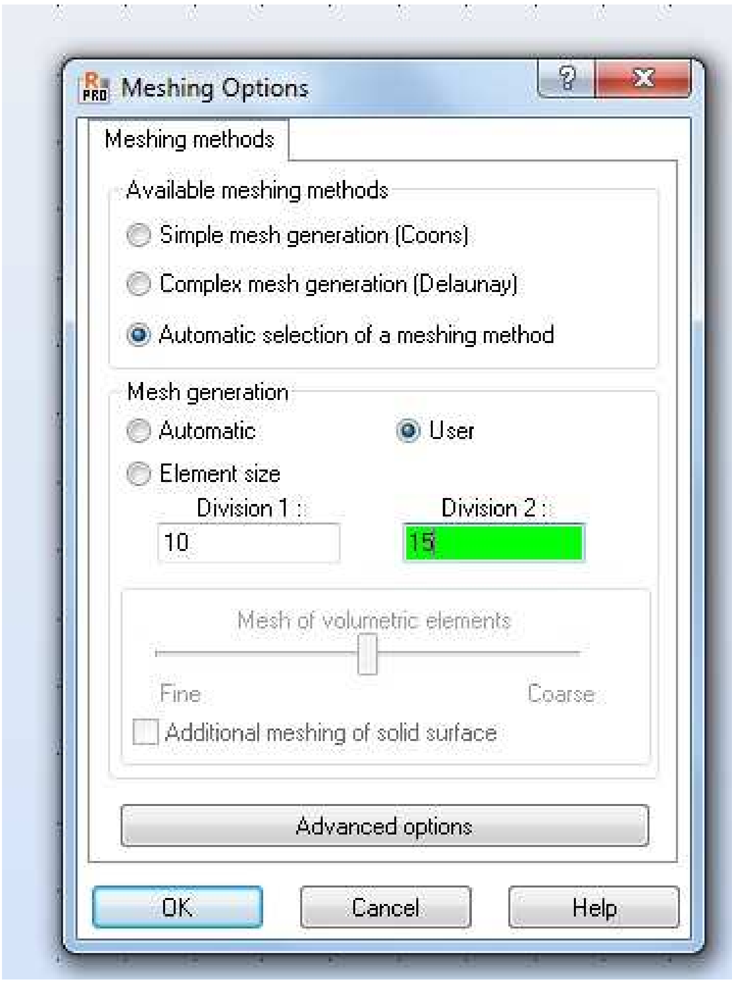


Figure 16: Options of FE Mesh Generation.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education

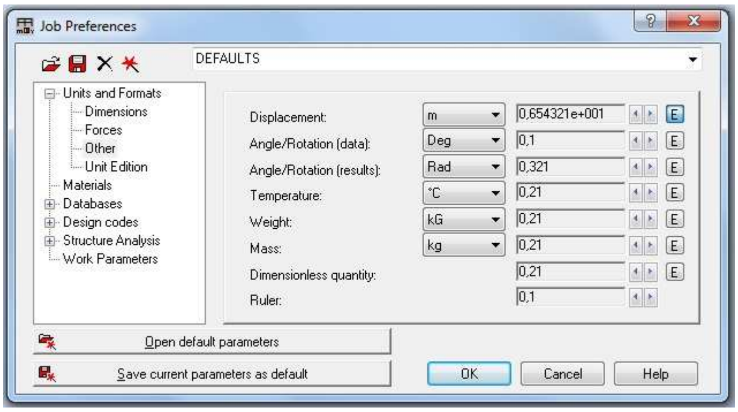
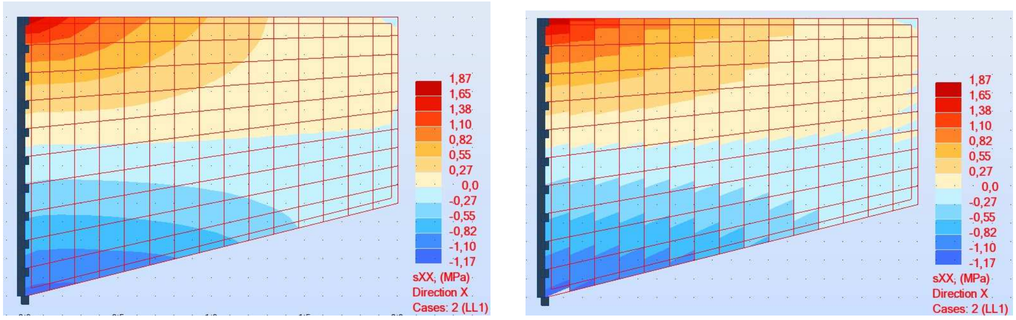


Figure 17: Units and formats setting.



(a) Distribution of σ_{xx} without smoothing. (b) Distribution of σ_{xx} with smoothing.

Figure 18: Contour plots for stress σ_{xx}.

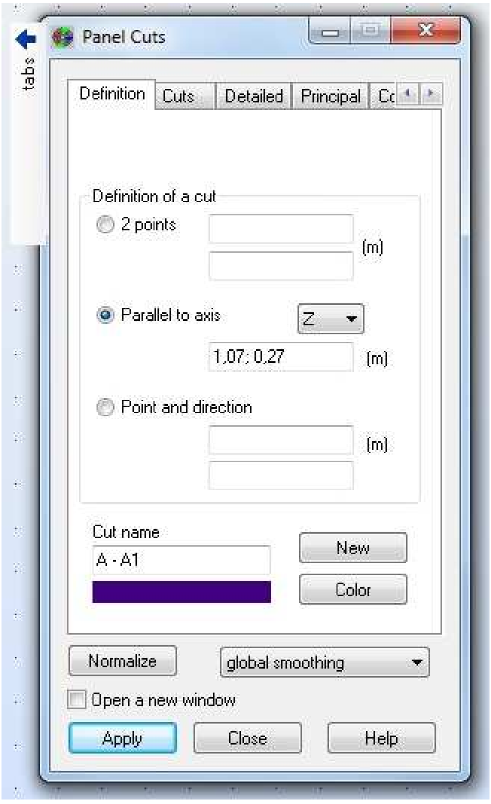


Figure 19: Settings for a diagram.

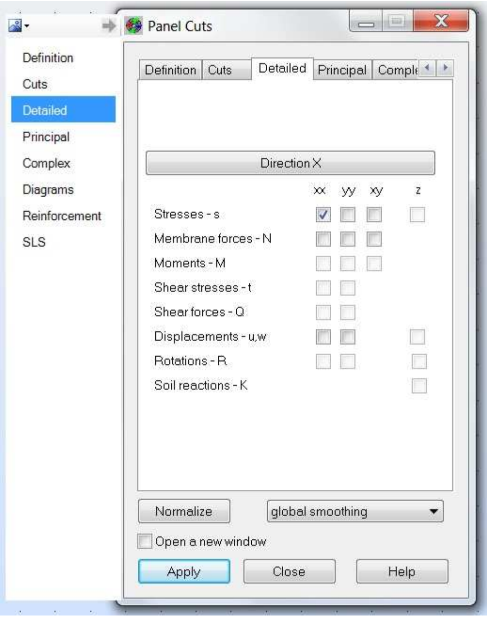


Figure 20: Settings for a diagram.

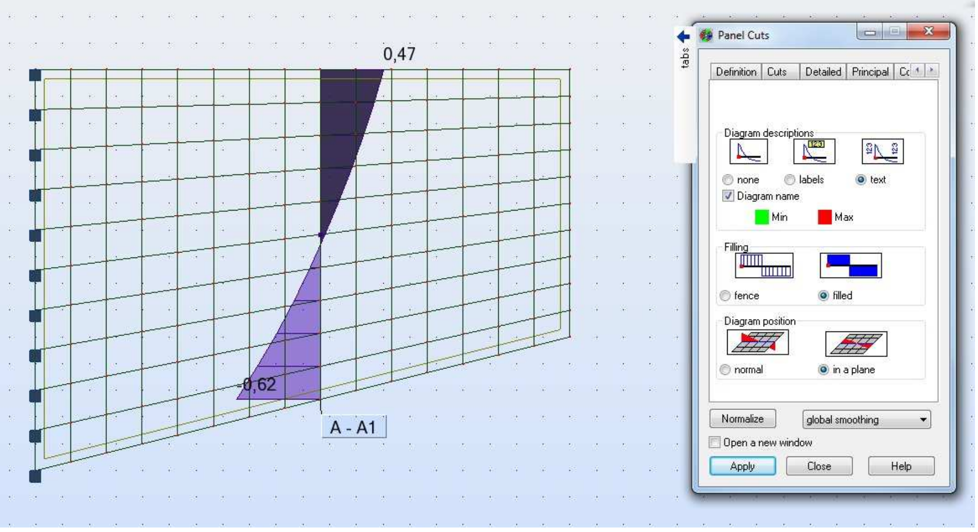


Figure 21: Diagram of σ_{xx} along a selected cross-section.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union
within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education