

Example 6. Analysis of cylindrical tank

In this lesson you will learn how to:

- generate design model of cylindrical tank with bottom plate;
- define load from dead weight and from the weight of liquid;
- apply local coordinate system of nodes to design model;
- analyse reinforcement.

Description:

Reinforced concrete tank with radius $R = 2$ m and height $H = 3$ m.

Material for tank – reinforced concrete B25.



Thickness of wall $d = 15$ cm and thickness of bottom plate $h = 20$ cm.

Loads:

- load case 1 – dead weight;
- load case 2 – internal water pressure.

- ⇒ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2015** and then click **LIRA-SAPR 2015**.

Step 1. Creating new problem

- ⇒ On the FILE menu, click **New** (button  on the toolbar).
- ⇒ In the **Model type** dialog box (see Fig.6.1) specify the following data:
- problem name – **Example6**;
 - model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).
- ⇒ Click **OK** .

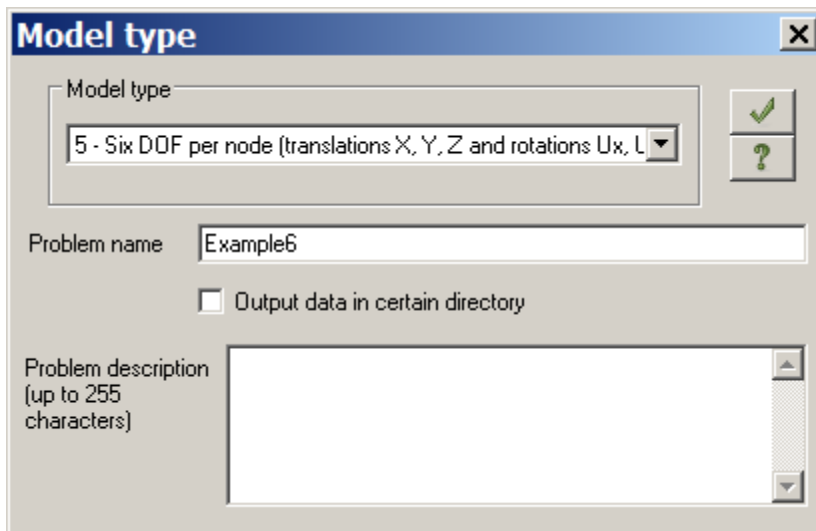



Figure 6.1 **Model type** dialog box



It is also possible to open the **Model type** dialog box with a pre-defined type of model. To do this, on the **LIRA-SAPR menu** (Application menu), point to **New** and click **Model type 5 (Six DOF per node)**

command . One more way to do the same: on the Quick Access Toolbar, click **New** and in the

drop-down menu select **Model type 5 (Six DOF per node)** command . Then you should define only problem name.





To save all output data files for the problem in certain directory, select appropriate check box. The directory name will coincide with the name of the problem. This directory will appear in the directory for files with analysis results. This is helpful if you have to find output data files for certain problem, then transfer these files or review and evaluate them with the help of Windows Explorer or other file managers.

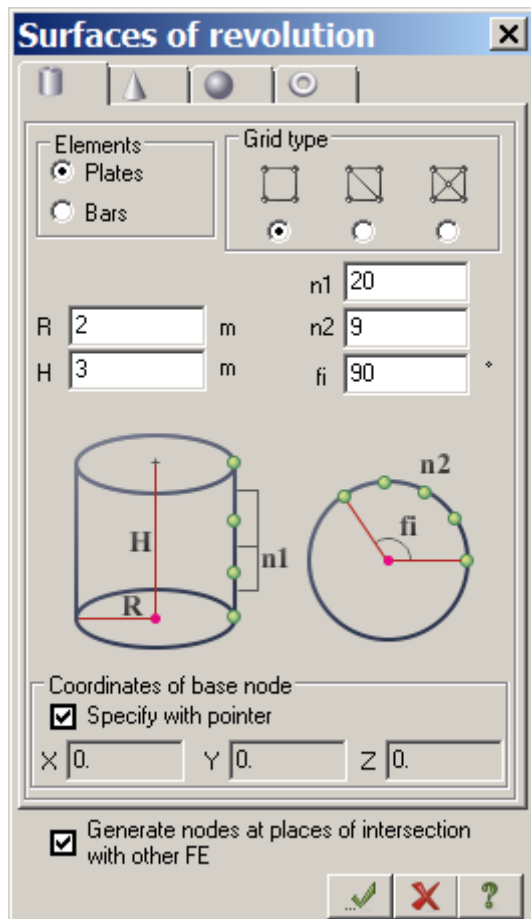
Step 2. Generating model geometry



It is possible to analyse a quarter of a tank because this design model and load are centrally symmetric. In this case it is necessary to assign appropriate restraints on nodes that belong to cut-off planes.

To generate walls of the tank:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Surface of revolution** list and click the **Cylinder** command .
- ⇒ In the **Surfaces of revolution** dialog box (see Fig.6.2), define the data necessary to generate cylinder:
- radius $R = 2$ m;
 - height $H = 3$ m;
 - number of finite elements along the generatrix $n1 = 20$ and around the circle $n2 = 9$;
 - angle of segment $fi = 90^\circ$.
 - other parameters remain by default.
- ⇒ Click **Apply** .

Figure 6.2 **Surfaces of revolution** dialog boxTo generate the tank bottom plate:

- ⇒ In the **Surfaces of revolution** dialog box, click the second tab (see Fig.6.3) and define the data necessary to generate cone:
- minor radius $r = 0$ m;
 - base radius $R = 2$ m;
 - height $H = 0$ m;
 - number of finite elements along the generatrix $n1 = 10$ and around the circle $n2 = 9$;

- angle of segment $fi = 90^\circ$.
- other parameters remain by default.

⇒ Click **Apply** .

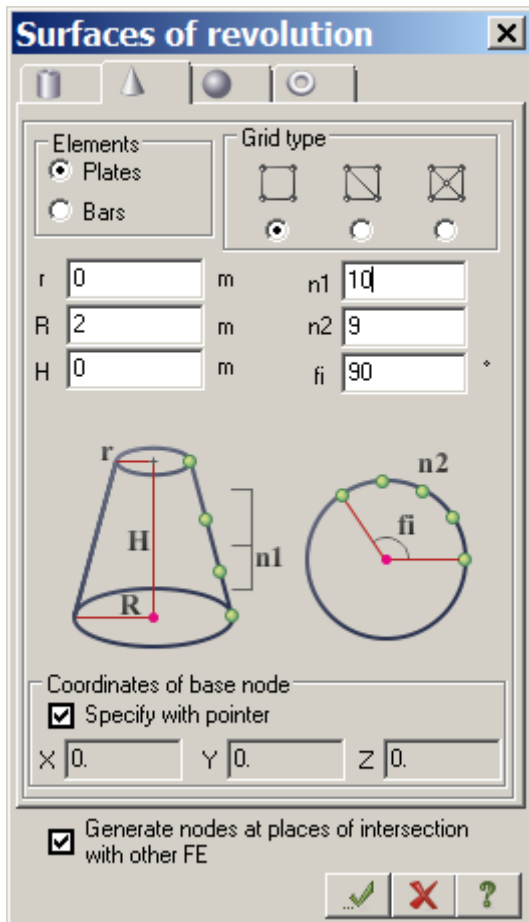




Figure 6.3 **Surfaces of revolution** dialog box

To pack the model:

- ⇒ On the **Create and edit** ribbon tab, on the **Edit** panel, click **Pack model** .
- ⇒ In the **Pack model** dialog box (see Fig.6.4), click **Apply** . It is necessary to pack the model in order to 'throw together' coincident nodes and elements and to eliminate (that is, to remove completely) deleted nodes and elements from design model.

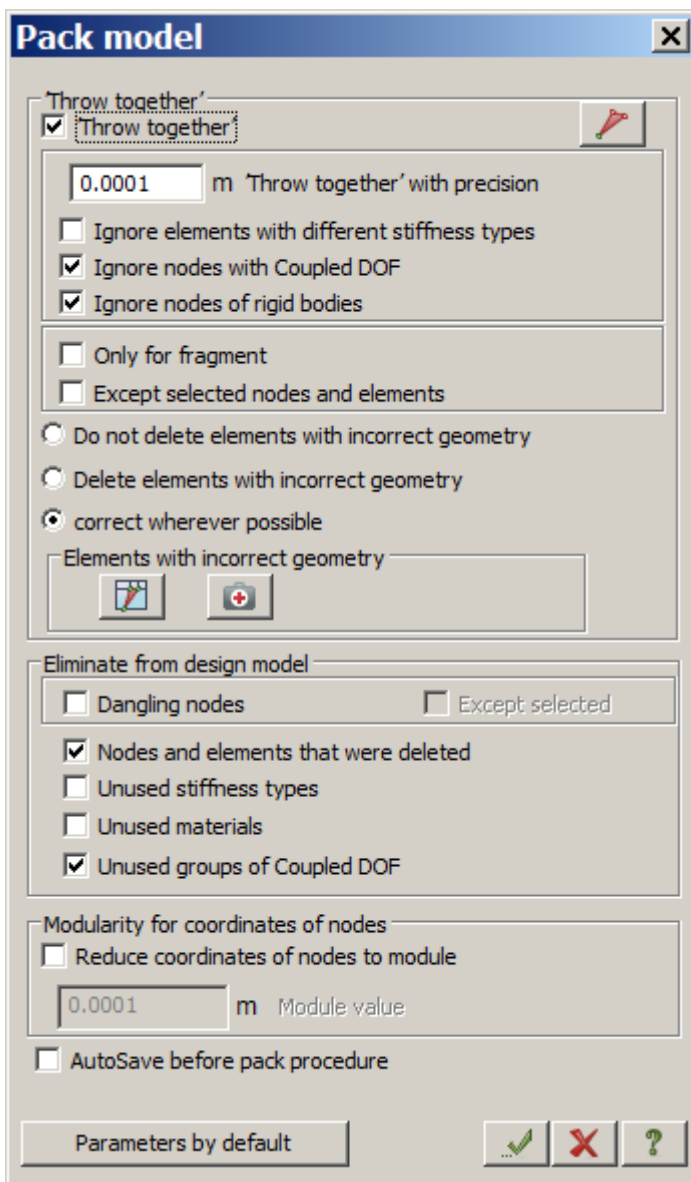



Figure 6.4 Pack model dialog box

To save data about design model:



- ⇒ On the **LIRA-SAPR menu** (Application menu), click **Save** command .
- ⇒ In the **Save as** dialog box specify the following data:
 - file name – **Example6**;
 - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

Step 3. Assigning local coordinate system to nodes of design model

To select nodes:



To assign local coordinate system, it is necessary to select all nodes of design model except central node of bottom plate – node No.301 (0;0;0).

- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **PolyFilter** .
- ⇒ In the **PolyFilter** dialog box, select the **Filter for nodes** tab (see Fig.6.5).
- ⇒ Select **By numbers of nodes** check box and specify numbers of elements 1 – 300.
- ⇒ Click **Apply** .

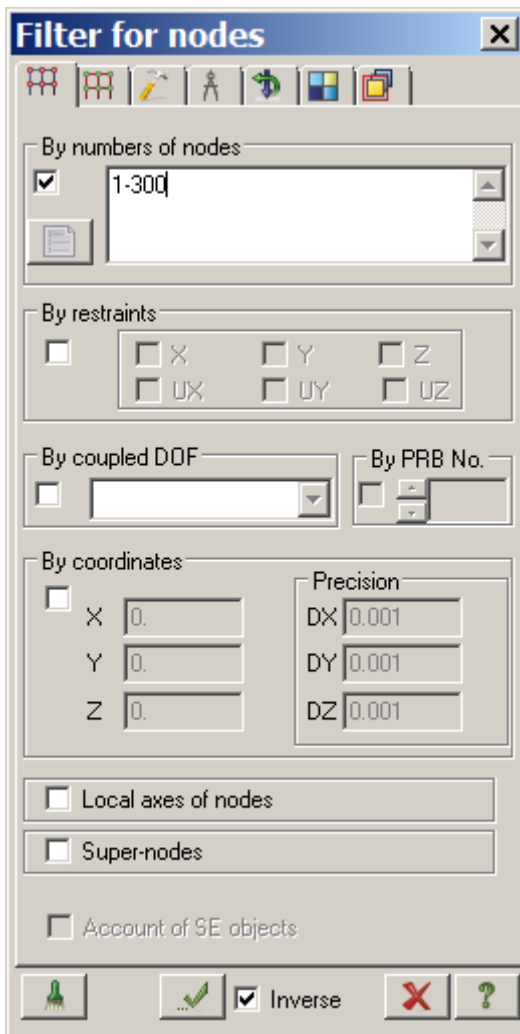




Figure 6.5 **Filter for nodes** tab

To assign local coordinate system:



*When you select nodes or elements on design model, you will see Contextual Tabs on the Ribbon User Interface. Contextual Tabs expose functionality specific only to the object in focus. They remain hidden when the object it works on is not selected. Contextual Tabs are mentioned to work with nodes or elements of the model. They contain commands to create and edit the model and can't be activated from **Results**, **Advanced results** and **Design** ribbon tabs.*

- ⇒ On the **Nodes** contextual tab, on the **Edit nodes** panel, click **Local nodal axes** command .
- ⇒ In the **Local axes of nodes** dialog box (see Fig.6.6), clear the **Z2** coordinate check box (we define coordinate of a point from which local X-axes will be directed. As Z-coordinate is variable along the height, it is necessary to clear appropriate check box).
- ⇒ Click **Apply** .

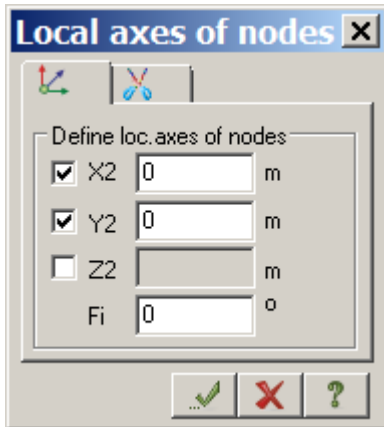


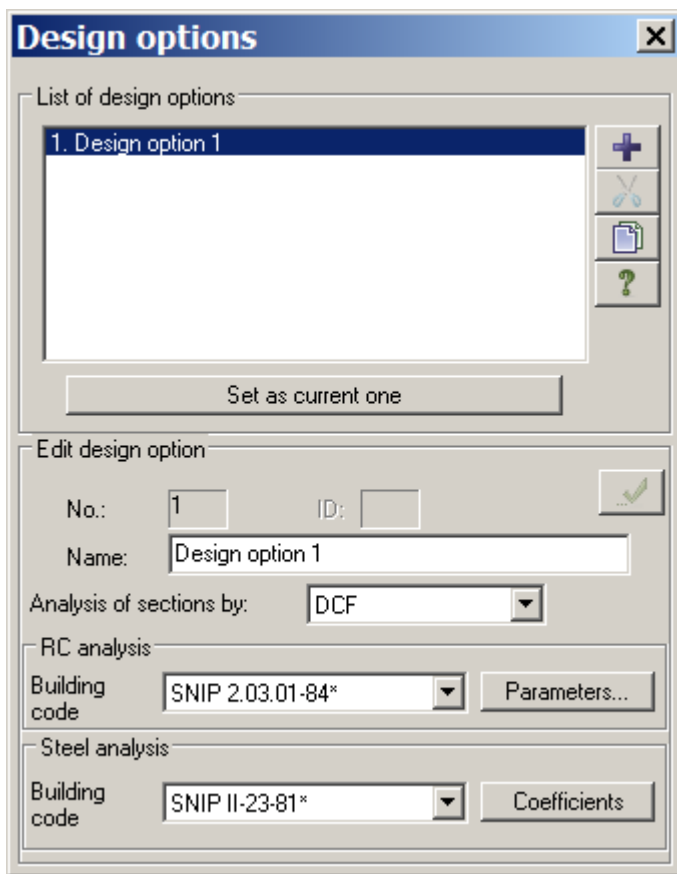


Figure 6.6 **Local axes of nodes** dialog box

Step 4. Defining design options


- ⇒ On the **More edit options** ribbon tab, on the **Design** panel, click **Design options for main model** command .
- ⇒ In the **Design options** dialog box (see Fig.6.7), define parameters for the first design option:
 - in the **Analysis of sections by** list, select **DCF**;
 - other parameters remain by default.
- ⇒ Click **Apply** .

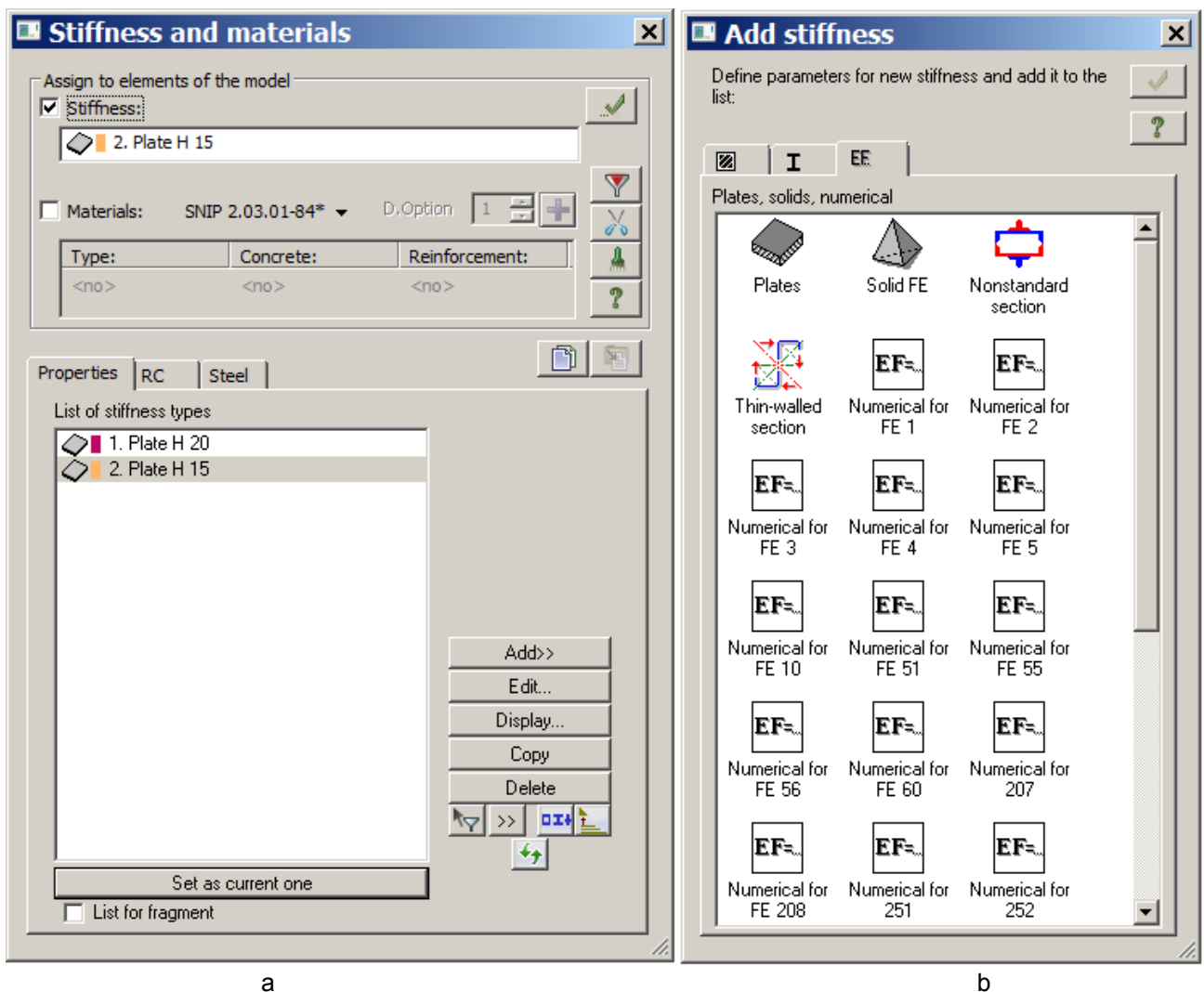
Figure 6.7 **Design options** dialog box

⇒ To close the **Design options** dialog box, click the **Close** button.

Step 5. Defining material properties to elements of tank

To create material data sets:

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Material properties** button .
- ⇒ In the **Stiffness and materials** dialog box (see Fig.6.8a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.6.8b).

Figure 6.8 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**



- ⇒ In the **Stiffness and materials** dialog box, select the third tab **Plates, solids, numerical** and double-click the **Plates** icon in the list.
- ⇒ In the **Stiffness for plates** dialog box (see Fig.6.9), specify the following parameters for **Plate** (bottom plate):
 - modulus of elasticity – $E = 3e6 \text{ t/m}^2$ (for the U.S. keyboard layout);
 - Poisson's ratio – $\nu = 0.2$;
 - thickness – $H = 20 \text{ cm}$;
 - unit weight of material – $R_0 = 2.75 \text{ t/m}^3$.
- ⇒ To confirm the specified data, click **OK** .

Figure 6.9 **Stiffness for plates** dialog box

- ⇒ In the **Stiffness and materials** dialog box, double-click the **Plates** icon in the list once again.
- ⇒ In another **Stiffness for plates** dialog box, specify the following parameters for **Plate** (web):
 - thickness – $H = 15$ cm;
 - other parameters remain as for the previous stiffness.
- ⇒ To confirm the specified data, click **OK** .
- ⇒ To hide library of stiffness properties, in the **Stiffness and materials** dialog box click **Add** unfold button.

To define materials for reinforced concrete (RC) structures:





- ⇒ To define parameters for reinforced concrete structures, in the **Stiffness and materials** dialog box, click the second tab **Reinforced concrete (RC)**.
- ⇒ Select **Type** option and click **Add**.
- ⇒ In the **General parameters** dialog box (see Fig.6.10), define the following parameters for plate elements:
 - in the **Module of reinforcement** list, select **Shell**;
 - in the **Comment** box, type comment - **Shells**;
 - other parameters remain by default.
- ⇒ Click **OK** .

Figure 6.10 General parameters dialog box

- ⇒ In the **Stiffness and materials** dialog box, select the **Concrete** option.
- ⇒ Click **Default** (in this case, concrete B25 is accepted by default).
- ⇒ In the same dialog box, select the **Reinforcement** option and click **Add**.
- ⇒ Click **Default** (in this case, reinforcement A-III is accepted by default).

To assign stiffness and material properties to elements of the tank:



- ⇒ On the **Select** toolbar, click **Select block** button . In this case, make sure that stiffness **2.Plate H 15** is defined as current one in the list of stiffness types and in the list of current materials the following data should be defined as current one: type – **1.shell**, concrete class – **1.B25** and class of reinforcement – **1.A-III**.

- ⇒ Specify with the pointer any node or element of the web (nodes and elements of the wall will be coloured red).
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** . The elements become unselected. It indicates that the current combination of stiffness type and material is assigned to selected elements.
- ⇒ In the **Stiffness and materials** dialog box, on the **Properties** tab, select the stiffness type **1.Plate H 20** in the list.
- ⇒ Click **Set as current type**.
- ⇒ Specify with the pointer any node or element of the tank bottom plate.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .

Step 6. Defining boundary conditions



Local coordinate system is assigned to nodes of design model. Thus, applied boundary conditions will correspond to this coordinate system.

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Restraints** .
- ⇒ In the **Restraints on nodes** dialog box (see Fig.6.11), specify directions along which displacements of nodes are not allowed (Y, UX, UZ). To do this, select appropriate check boxes.
- ⇒ Click **Apply**  (the nodes will be coloured blue).

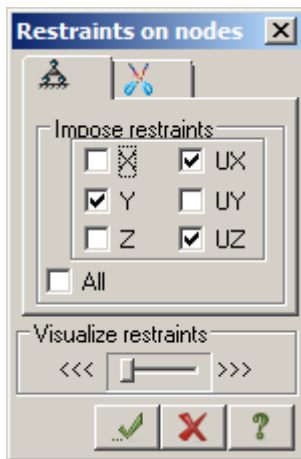









Figure 6.11 **Restraints on nodes** dialog box

- ⇒ To present the model in isometric projection, on the **Projection** toolbar (by default, it is displayed at the bottom of the screen), click **Isometric projection** .
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** command .
- ⇒ On the **Select** toolbar, click **Select by contour** command .
- ⇒ With the left mouse button specify closed contour around the nodes where bottom plate is connected to the web (you could just specify these nodes on the model with the pointer).

- ⇒ In the **Restraints on nodes** dialog box, specify directions along which displacements of nodes are not allowed (Z). To do this, select appropriate check boxes.
- ⇒ Click **Apply** .
- ⇒ To present the model in dymetric projection, on the **Projection** toolbar, click **Dymetric projection** .

Step 7. Applying loads

To create load case No.1:

- ⇒ To define load from dead weight of the slab, on the **Create and edit** ribbon tab, select the **Loads** panel and click **Add dead weight** .
- ⇒ In the **Add dead weight** dialog box (see Fig.6.12), click **All elements** and specify **Load factor** as equal to 1. Then click **Apply**  (dead weight of elements is applied according to the specified unit weight Ro).

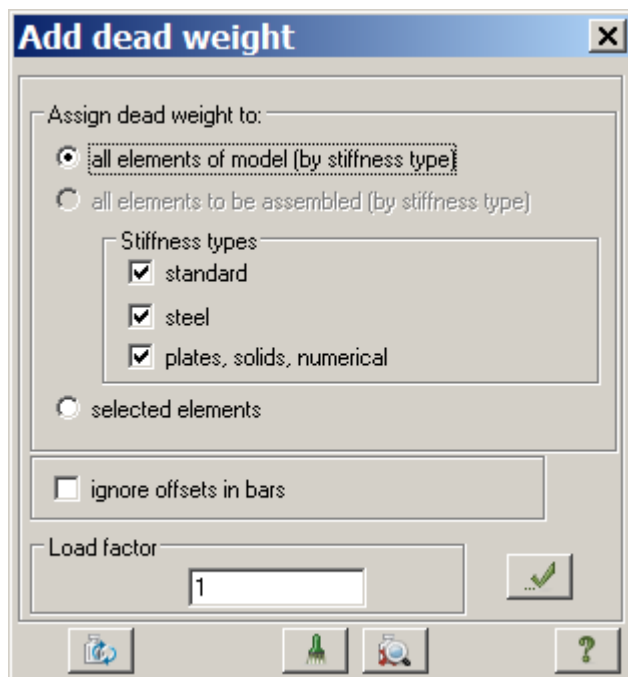





Figure 6.12 **Add dead weight** dialog box

To create load case No.2:

- ⇒ To change the number of the current load case, click the **Next load case** button  located on the Status bar or on the toolbar.
- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Select block** .
- ⇒ Specify with the pointer any node or element of the tank bottom plate.
- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load on plates** command  from the **Loads on nodes and elements** drop-down list.
- ⇒ In the **Define loads** dialog box (see Fig.6.13), specify **Global** coordinate system and direction along the Z-axis (default parameters).

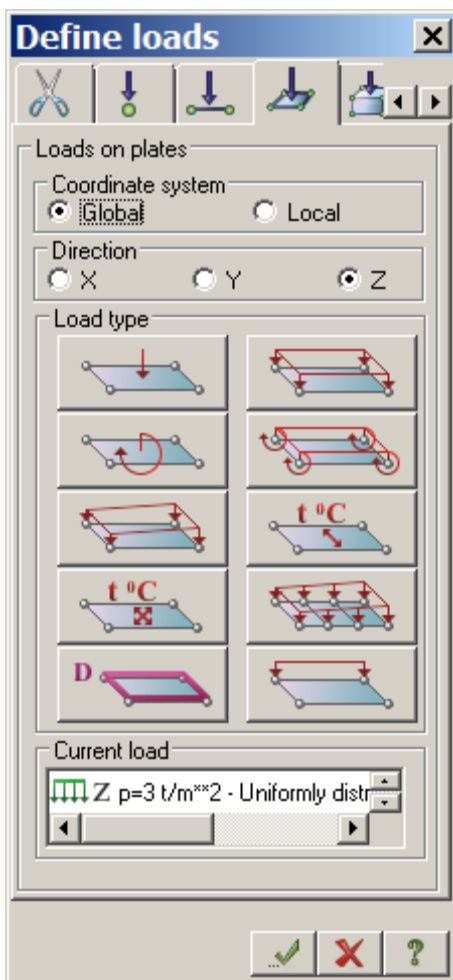
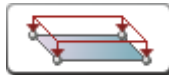



Figure 6.13 Define loads dialog box

- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 3 \text{ t/m}^2$ (see Fig.6.14).
- ⇒ Click **OK** .

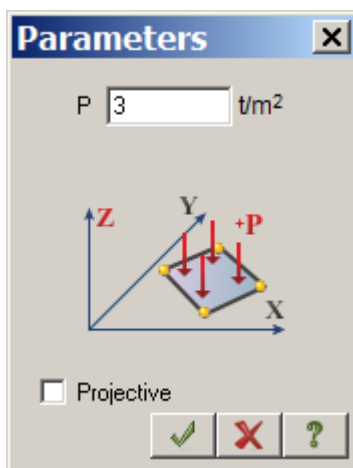
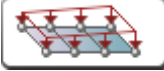


Figure 6.14 Load parameters dialog box

- ⇒ When the **Select block** command is active, specify with the pointer any node or element of the web of a tank.
- ⇒ In the **Define loads** dialog box, on the **Loads on plates** tab, specify **Local** coordinate system.
- ⇒ Click **Trapezoidal load on group of plates** button .
- ⇒ In the **Non-uniformly distributed load** dialog box (see Fig.6.15), specify $P1 = -3 \text{ t/m}^2$ and direction along which the load is changed (Z-axis).
- ⇒ Click **OK**.

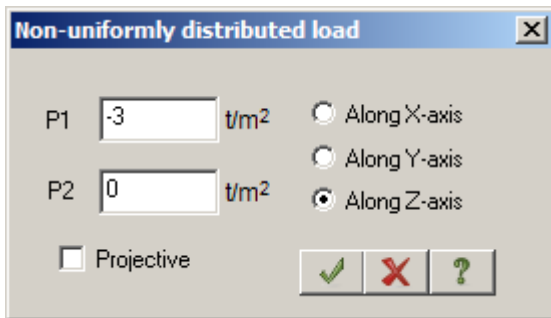



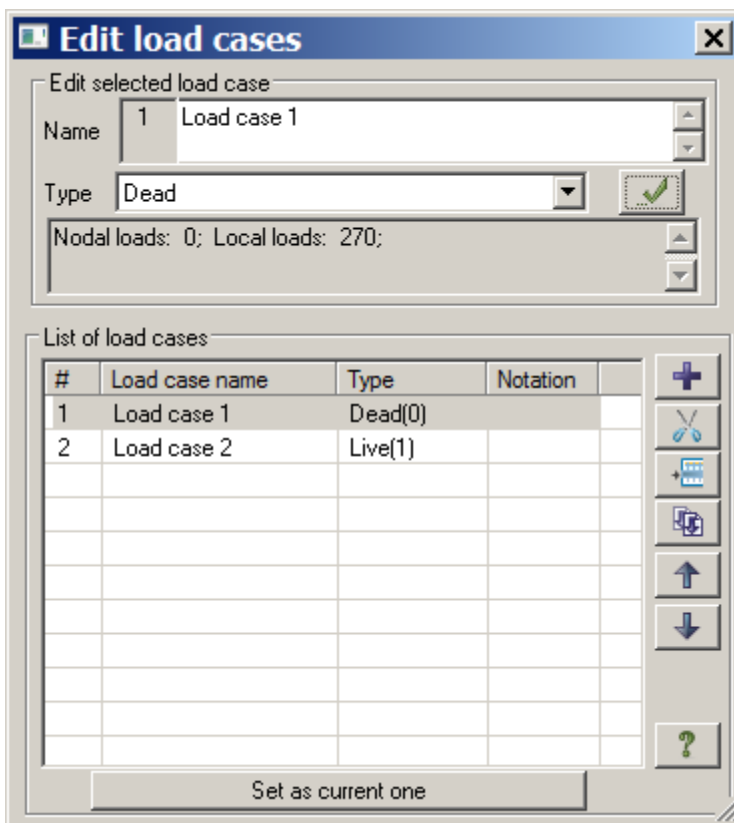


Figure 6.15 **Non-uniformly distributed load** dialog box

- ⇒ To unselect nodes and elements, on the **Select** toolbar, click **Unselect all** button .

To define detailed information about load cases:

- ⇒ On the **Create and edit** ribbon tab, on the **Loads** panel, click **Edit load cases** . The **Edit load cases** dialog box is displayed on the screen (see Fig.6.16).
- ⇒ For load case 1 – in the **Edit selected load case** area, in the **Type** box, select **Dead** and click **Apply** .
- ⇒ For load case 2 – in the **Edit selected load case** area, in the **Type** box, select **Live** and click **Apply** .


Figure 6.16. **Edit load cases** dialog box

Step 8. Generating DCF table

⇒ On the **Analysis** ribbon tab, select the **DCF** panel and click **DCF table** button .



*As the type of load cases was defined in the **Edit load cases** dialog box (see Fig.6.16), the DCF table is generated automatically with parameters accepted by default for every load case. Now you have to modify parameters for the third load case.*

⇒ In the **Design combinations of forces** dialog box (see Fig.6.17), to confirm parameters accepted by default for every load case, just click **OK** .

Design combinations of forces (DCF)

Building code: SNIP 2.01.07-85*

Load case No.: 1 Load case 1

Load case type: Dead(0)

of group of integrated temporary load cases: 0

Account of sign variability: ☐

of the group of mutually exclusive load cases: 0

of the accompanying load: 0 0

Load factor: 1.10

Duration coefficient: 1.00

Do not consider for analysis by SLS: ☐

Restrictions for cranes and brakes: Crane ☐ Brake ☐

DCF coefficients


#	1 main	2 main	Spec.(E)	Spec.(n E)	5 combin.	6 combin.
1	1.00	1.00	0.90	1.00	0.00	0.00
2	1.00	0.95	0.80	0.95	0.00	0.00

Summary table for DCF calculation:

L...	Load case name	Type	DCF parameters	DCF coefficients
1	Load case 1	Dead(0)	0 0 0 0 0 0 1.10 1.00	1.00 1.00 0.90 1.00
2	Load case 2	Live(1)	1 0 0 0 0 0 1.20 1.00	1.00 0.95 0.80 0.95

Figure 6.17 Design combinations of forces dialog box

Step 9. Complete analysis of the model

- ⇒ To carry out complete analysis of the model, on the **Analysis** ribbon tab, select the **Analysis** panel and in the **Analyse** drop-down list, click **Complete analysis** .


Step 10. Review and evaluation of static & dynamic analyses results








When analysis procedure is complete, to review and evaluate results of static and dynamic analyses, select the **Results** ribbon tab.

- ⇒ In the mode of analysis results visualization, by default design model is presented with account of nodal displacements.

To change number of active load case:

- ⇒ On the status bar (displayed at the bottom of the screen), in the **Load case No.** list, select No. **2** and click **Apply** .

To present displacement contour plots along the local X-axis on the deformed web of a tank:

- ⇒ To select elements of web, make sure that **Select block** command  is active (button on the **Select** toolbar) and then specify any node or element of the tank web.
- ⇒ To present on the screen only selected nodes and elements of the web, on the **Select** toolbar, click **Fragmentation** .
- ⇒ To present contour plot of displacements along the local X-axis, on the **Results** ribbon tab, on the **Deformations** panel, select the **Displacement contour plot in local coordinate system** command  in the **Displacement mosaic/contour plot** drop-down list.
- ⇒ Then click **Displacements along X** button  on the same panel.
- ⇒ To present design model in projection on the YOZ-plane (see Fig.6.18), on the **Projection** toolbar (by default, it is displayed at the bottom of the screen), click **Projection on YOZ-plane** .

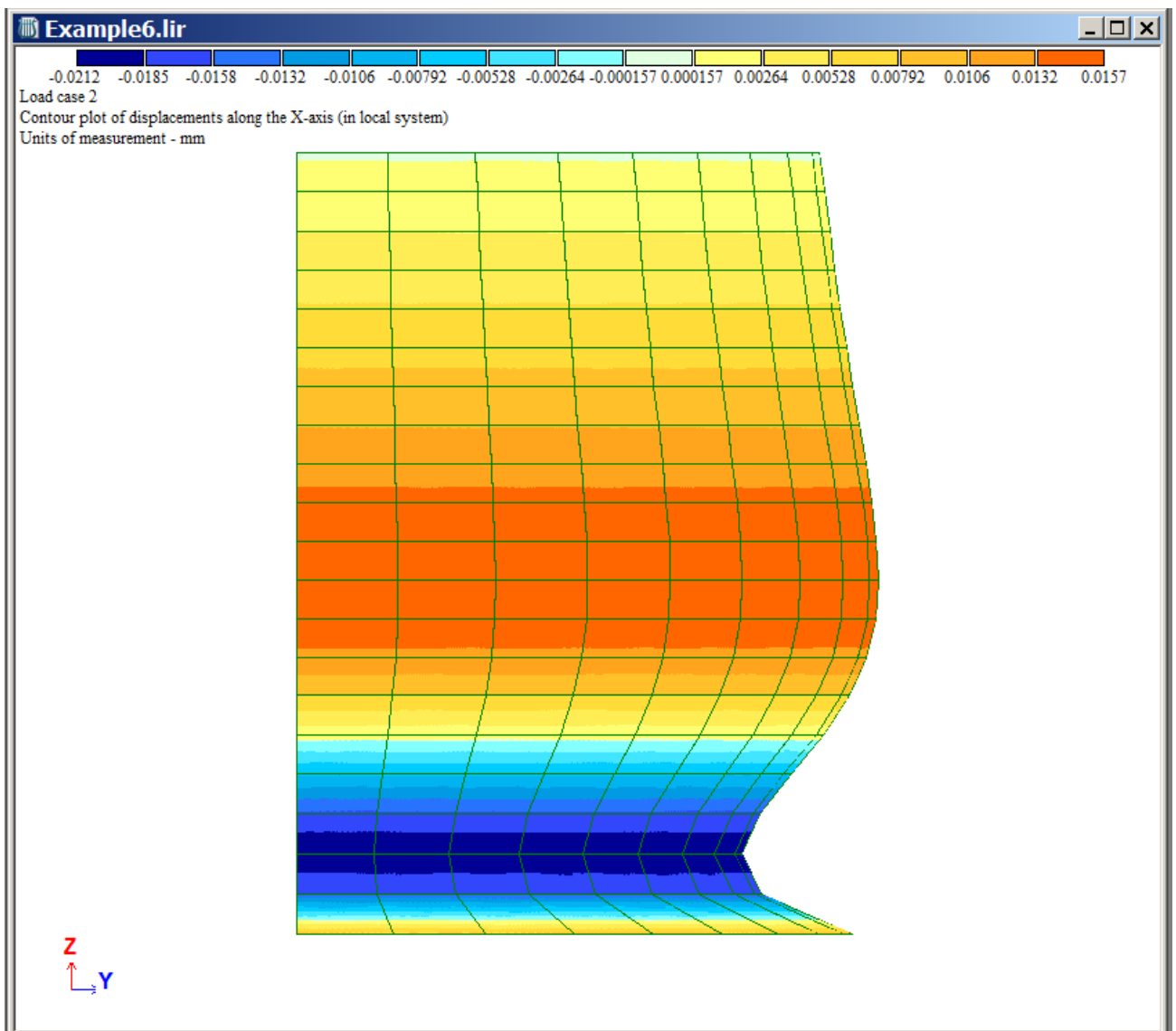








Figure 6.18 Displacement contour plot along the local X-axis in elements of the tank web

- ⇒ To restore design model in initial view, on the **Select** toolbar, click **Restore model** .
- ⇒ To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .

To present contour plots of vertical displacements in tank bottom plate on the deformed shape:

- ⇒ To select elements of tank bottom plate, make sure that **Select block** command  is active (button on the **Select** toolbar) and then specify any node or element of the bottom plate.
- ⇒ To present on the screen only selected nodes and elements of the bottom plate, on the **Select** toolbar, click **Fragmentation** .
- ⇒ To present contour plot of displacements along the global Z-axis, on the **Results** ribbon tab, on the **Deformations** panel, select the **Displacement contour plot in global coordinate system** command  in the **Displacement mosaic/contour plot** drop-down list.
- ⇒ Then click **Displacements along Z** button  on the same panel.

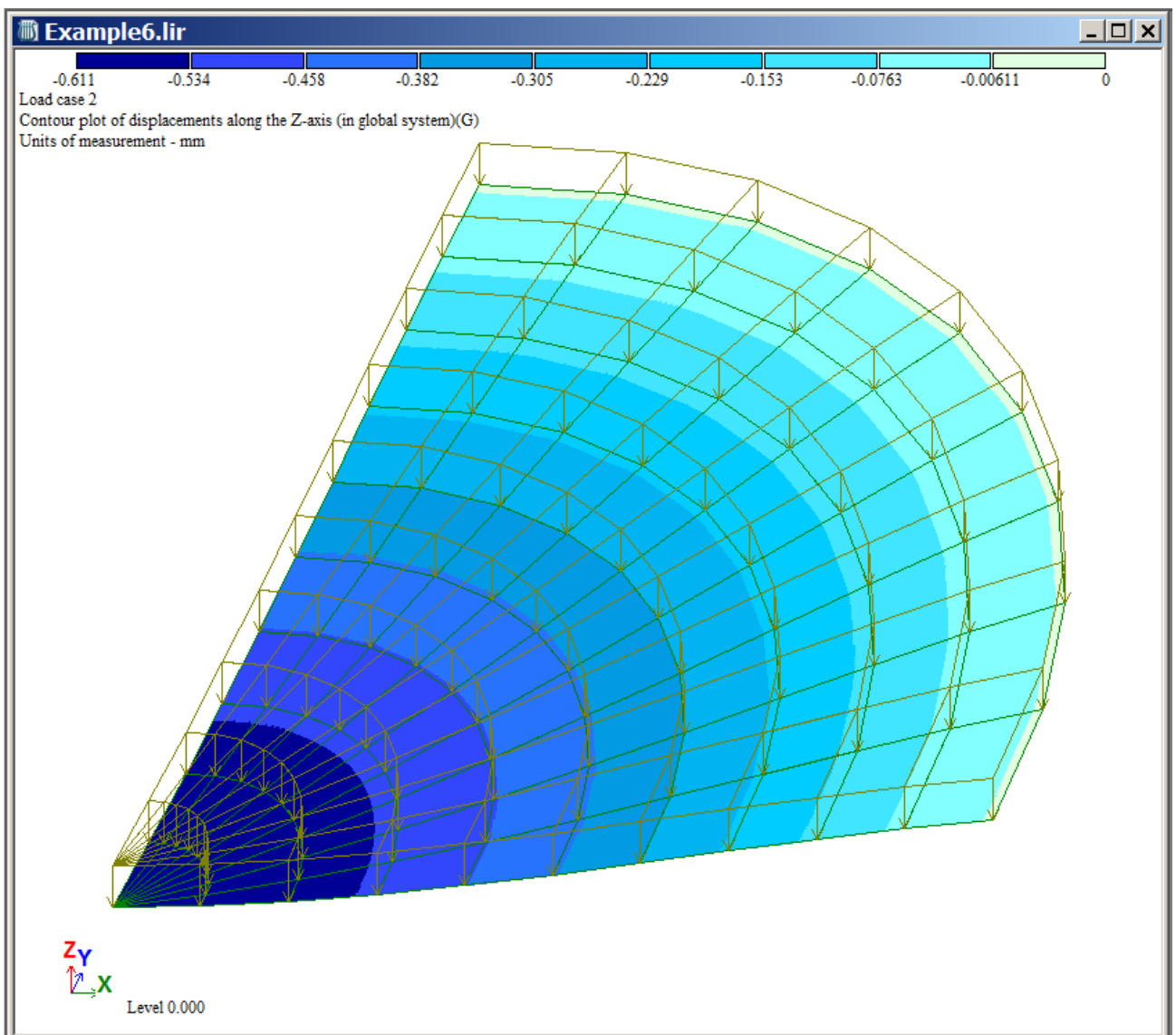









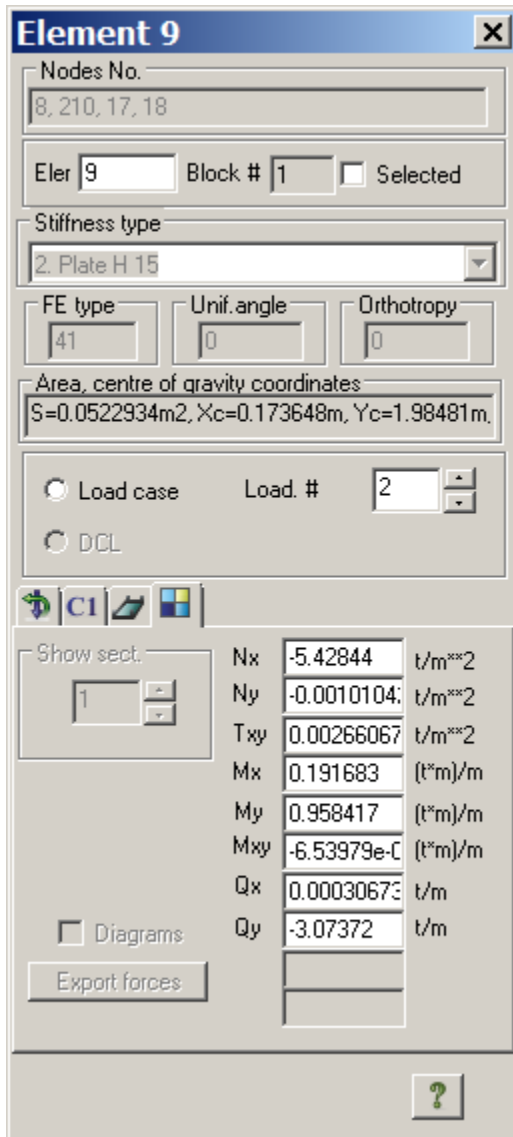
Figure 6.19 Displacement contour plot along the global Z-axis in elements of the tank bottom plate

To present stress mosaic plots:

- ⇒ To present stress mosaic plot for M_x , on the **Results** ribbon tab, on the **Stress in plates and solids** panel, select the **Stress mosaic plot** command  in the **Stress mosaic/contour plots** drop-down list.
- ⇒ Then click **Stress M_x** button  on the same panel.
- ⇒ To present stress mosaic plot for N_x , click **Stress N_x** button  on the same panel.
- ⇒ To restore design model in initial view, on the **Select** toolbar, click **Restore model** .
- ⇒ To display design model and hide the stress mosaic plot N_x , click **Stress N_x** button  once again to make it not active.
- ⇒ To display design model without account of nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

- ⇒ To display information about stress in any element of lower part of tank web, on the **Select** toolbar, click the **Information about nodes and elements** button  and specify with the pointer any element in the lower part of tank web.

The stress values will be displayed in the dialog box (see Fig.6.20).



Element 9

Nodes No.
8, 210, 17, 18

Eler 9 Block # 1 ☐ Selected

Stiffness type
2. Plate H 15

FE type 41 Unif. angle 0 Orthotropy 0

Area, centre of gravity coordinates
S=0.0522934m², Xc=0.173648m, Yc=1.98481m.

☐ Load case Load. # 2
☐ DCL

☐ C1 ☐ ☐ ☐

Show sect.
1

Nx	-5.42844	t/m**2
Ny	-0.0010104	t/m**2
Txy	0.00266067	t/m**2
Mx	0.191683	(t*m)/m
My	0.958417	(t*m)/m
Mxy	-6.53979e-0	(t*m)/m
Qx	0.00030673	t/m
Qy	-3.07372	t/m


☐ Diagrams

Export forces

?

Figure 6.20 **Information about element** dialog box

To generate and review tables of analysis results:

- ⇒ To present table with design combinations of forces in elements of the model, on the **Results** ribbon tab, select **Tables** panel and click **Standard tables**  in the **Documents** drop-down list.
- ⇒ In the **Standard tables** dialog box (see Fig.6.21), select **Design combinations of forces, design values** in the list.
- ⇒ Click **Apply**.



By default, standard tables are generated in the *.csv format. Information presented in these tables is divided into different tabs: input data (optional), e.g. DCF coefficients; output data for bars; output data for plates; etc.

To generate table in *.csv format and add it to the Report Book, select the **Generate updatable table in Report Book** check box. If the table is located in the Report Book, it is possible to update it later (if required) and add it to the report file with the Report Book options.

To modify format of the table, in the **Standard tables** dialog box, click **Select format**. Then in the **Table format** dialog box, select appropriate option and click **OK**. To generate table in *Document Maker (DOC-SAPR module)*, select RPT format.

Selected format is saved and will be applied by default in further work with standard tables.

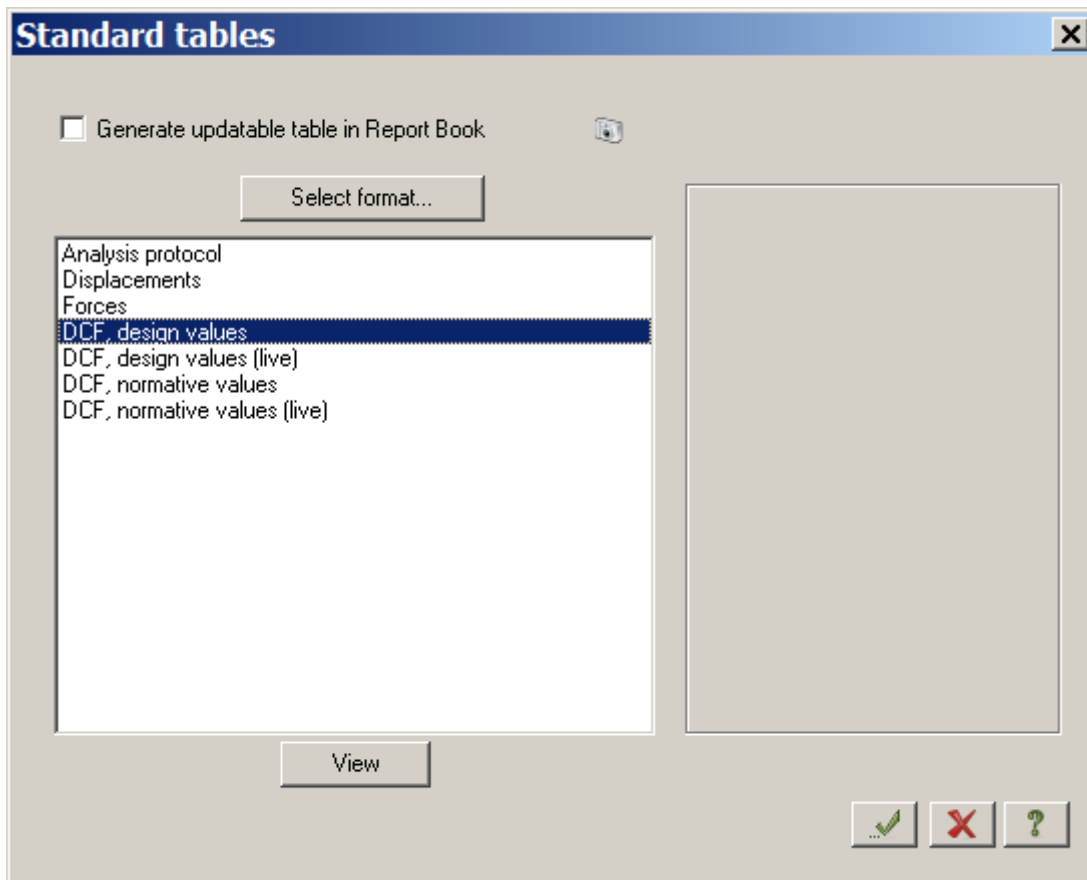


Figure 6.21 **Standard tables** dialog box


⇒ To close the table, on the FILE menu, click **Close**.



Step 11. Review and evaluate results from analysis of reinforcement




When analysis procedure is complete, to review and evaluate analysis results for reinforcement, select the **Design** ribbon tab.

To present results from analysis of reinforcement:

⇒ To present information about determined reinforcement in a certain plate FE, on the **Select** toolbar, click **Information about nodes and elements** button  and specify with a pointer any plate element on the model.

- ⇒ In the dialog box that appears on the screen, select the **Longitudinal reinforcement** tab. This dialog box contains complete information about selected element, including results for reinforcement.
- ⇒ To close the dialog box, click **Close** button.
- ⇒ To display mosaic plot for area of lower reinforcement in plates along the X1-axis, click the **Lower reinforcement in plates along X1** button  (on the **Design** ribbon tab, the **RC: Plates** panel).
- ⇒ To display mosaic plot for area of lower reinforcement in plates along the Y1-axis, click the **Lower reinforcement in plates along Y1** button  (on the **Design** ribbon tab, the **RC: Plates** panel).

To generate and review table with analysis results for reinforcement:

- ⇒ On the **Design** ribbon tab, select the **Tables** panel and click **Analysis results tables for RC** command  in the **Documents** drop-down list.
- ⇒ In the **Tables of analysis results** dialog box (see Fig.6.22), click the **Plates** option in the **Elements** area. Option **For all elements** in the **Create table** area is selected by default.
- ⇒ Click **Table - on the screen** (It is also possible to add the table to the Report Book and select format for the table, in a similar manner to that used for standard tables).

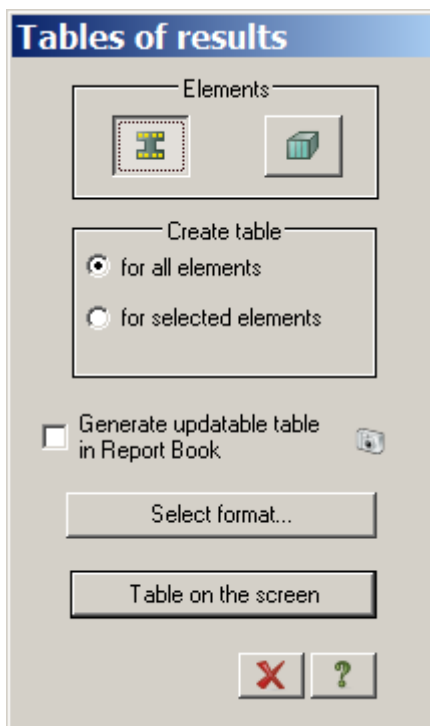


Figure 6.22 **Tables of analysis results** dialog box