

TOPIC 6

ANALYSIS OF STRESS DISTRIBUTION IN A LINEARLY DEFORMABLE SOIL LAYER MODEL

For a more visual analysis of the impact of the resolution of the soil model, we will replace the type of foundation of the building with a slab foundation. First, we will remove the previously created models of individual foundations for columns. For the convenience of working in the filter panel, we will disable the display of volumetric elements on the working field.

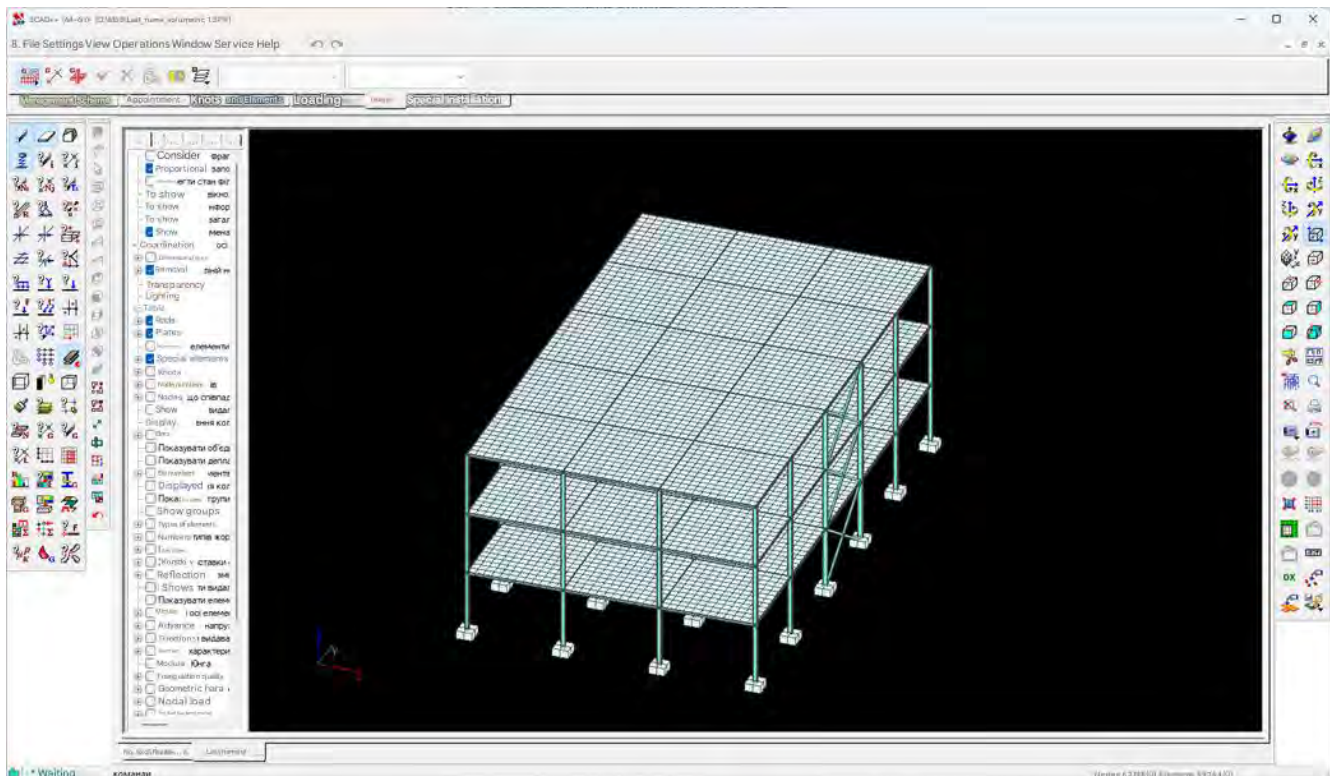


Figure 6.1 – Design scheme with the display of volumetric elements (soil massif) disabled

By returning the diagram to a convenient position for selecting foundations using the "Remove Elements" tool, select and delete the models of the foundations of the building (Fig. 6.2). After selecting the corresponding elements, press the confirm or "Enter" button on the keyboard. Return to the original display of the diagram (using the visualization panel) and enable the display of volumetric elements (soil mass) in the filter panel. For the convenience of work, we perform a cross-section with the plane of the HOU along the upper plane of the soil mass (using the function already known to us on the visualization panel). Go to the "Nodes and Elements" tab and run the "Add Plates" tool.

To ensure the joint operation of the "foundation-foundation-structure" system, we crush the foundation slab, taking into account the intermediate nodes (bases of columns and nodes of the soil mass model) using the appropriate tool on the "Nodes and Elements" tab. Click the corresponding button  on the toolbar and select the previously created plate and confirm the operation.

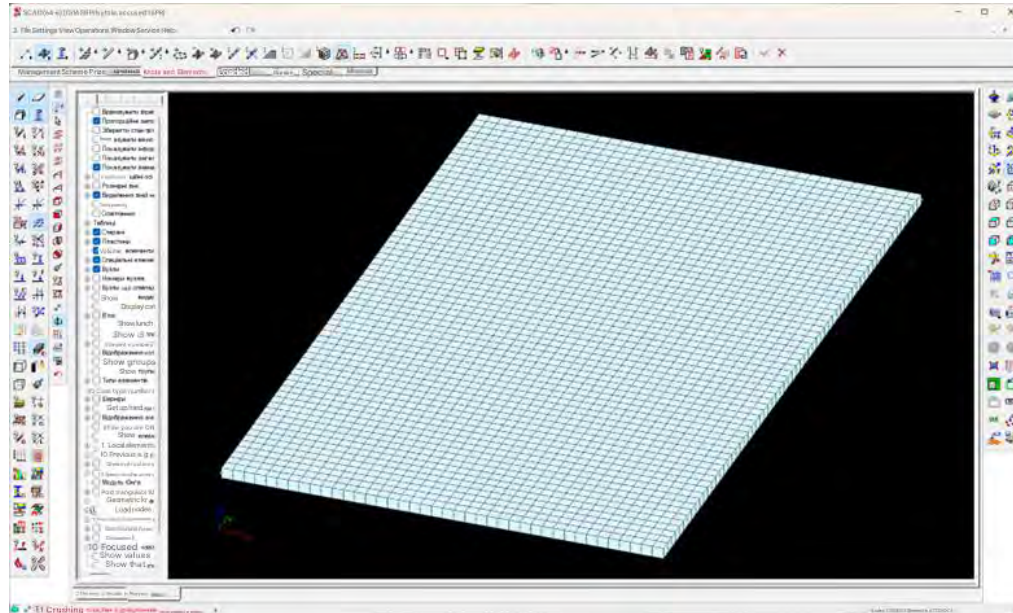


Figure 6.4 – The result of crushing the foundation slab model for linking with other elements of the "foundation-foundation-structure" system model

Using the visualization panel, we return to the original display of the design scheme. Go to the project management screen and perform the calculation.

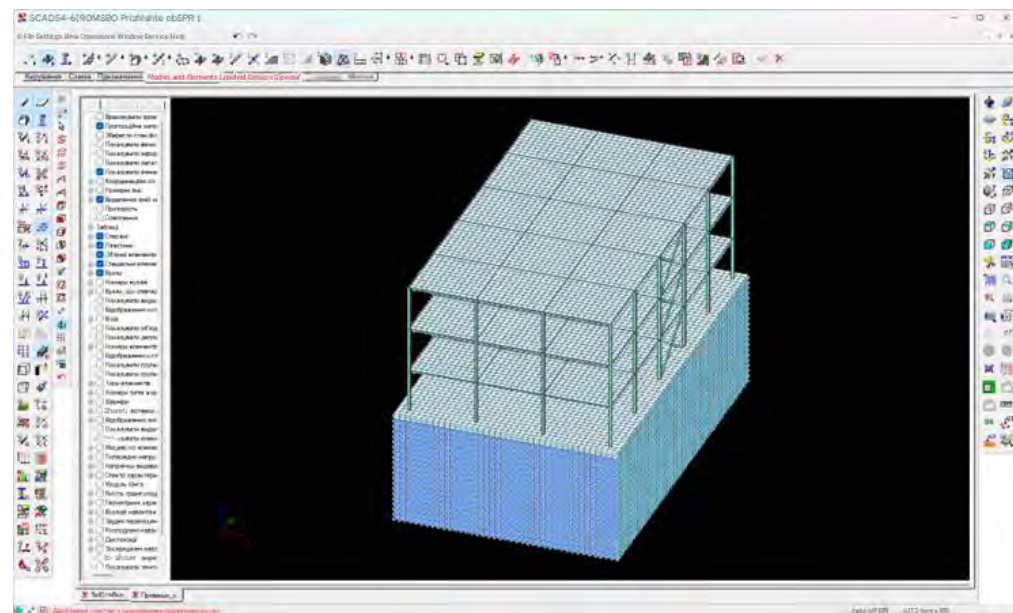


Figure 6.5 – Full display of the design scheme with a modified foundation

We go to the mode of graphical analysis of the results to the "Displacement" tab, where we record the magnitude and nature of the subsidence of the slab foundation of the building from the combination of forces created for the calculation of the slab foundation using the coefficients of the bedding. For the convenience of analysis, we perform a cross-section with the plane of the HOU along the plane of the slab. We determine the average subsidence S_{ser} .

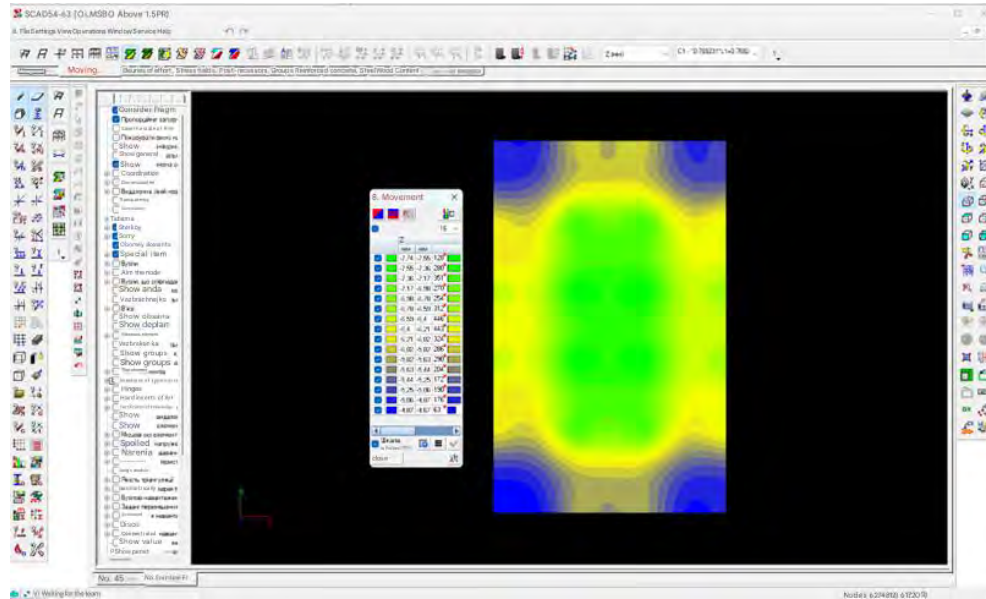


Figure 6.6 – Analysis of the subsidence of the foundation slab on the soil model represented by volumetric elements

Go to the "Stress fields" tab and determine the maximum values of the moments M_x and M_y in the slab foundation for this design scheme.

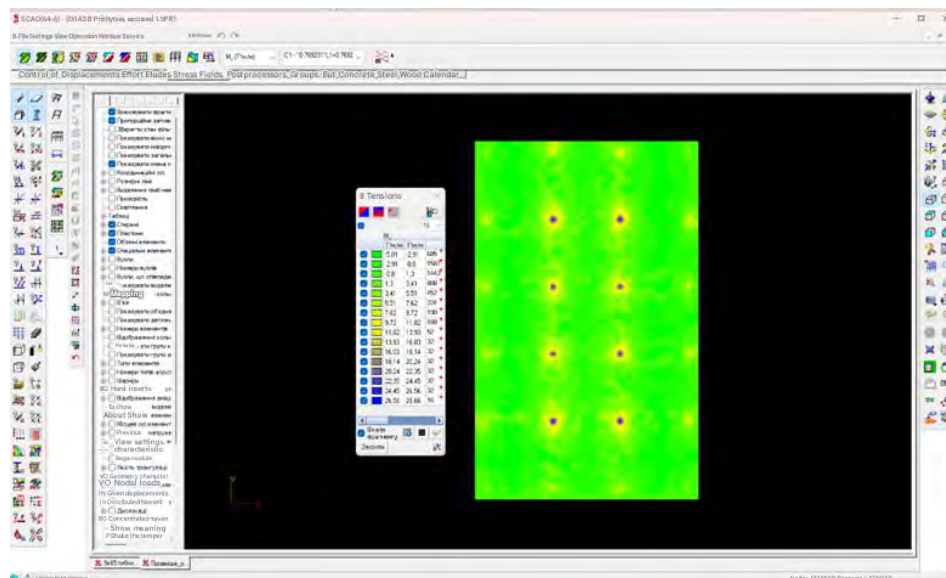


Figure 6.7 – Isopole of moments M_x in a slab foundation

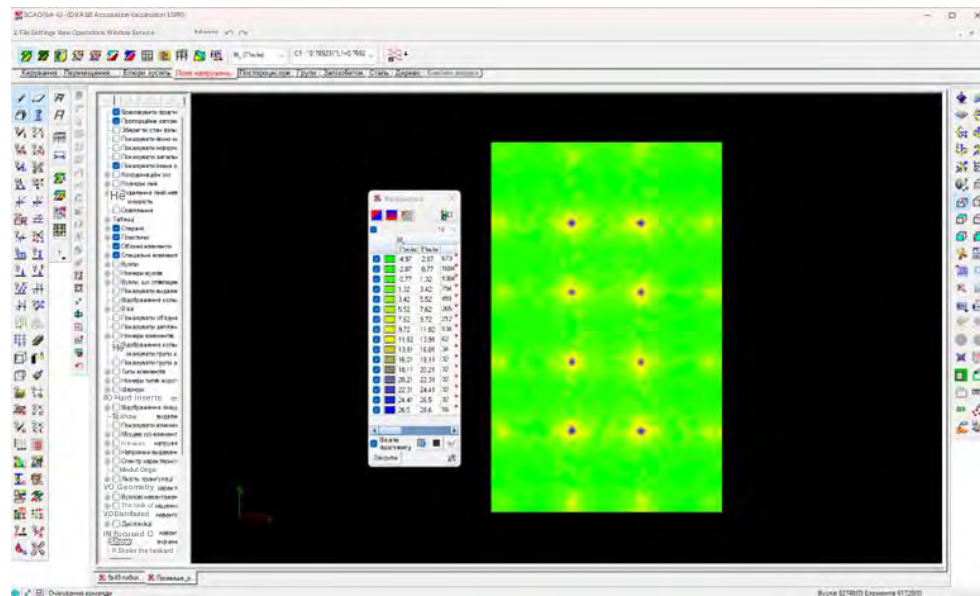


Figure 6.8 – Isopoles of moments M_y in a slab foundation

Write the results in Table 6.1. Save the calculation option under the first cipher of Table 6.1. Copy the file and save it under the second cipher of the same table. Then work with it, modifying the calculation scheme for the distribution angle $\alpha = 25^\circ$ (choose the optimal distribution angle according to [2]).

Table 6.1 – Results of the analysis of the influence of boundary conditions of the model of the soil basis of finite resolution.

Schema code	Average subsidence, mm	Maximum torques +/-, kN/m
Last name HB=5 a=0		
Last name HB=5 a=25		
Last name HB=5 a=45		

Through the project management screen, we return to the mode of building the design scheme and extend the boundaries of the soil base model to fulfill the angle condition from the distribution $\alpha = 25^\circ$. The first step is to remove the previously set boundary conditions (elms in the nodes of the soil model). Uncheck all the boxes in the elm dialog box and, having selected the entire soil massif, click the confirm button.

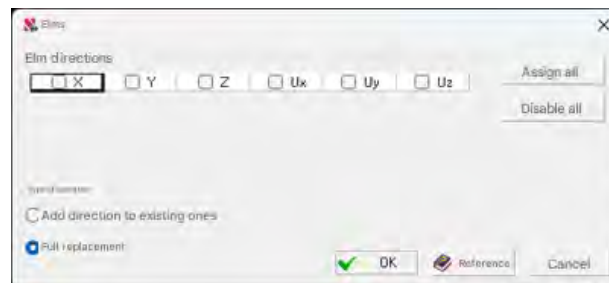


Figure 6.9 – Disabling Elms in the Dialog Box

For further convenience of working with the soil mass model, turn off the display of rods and plates on the filter panel.

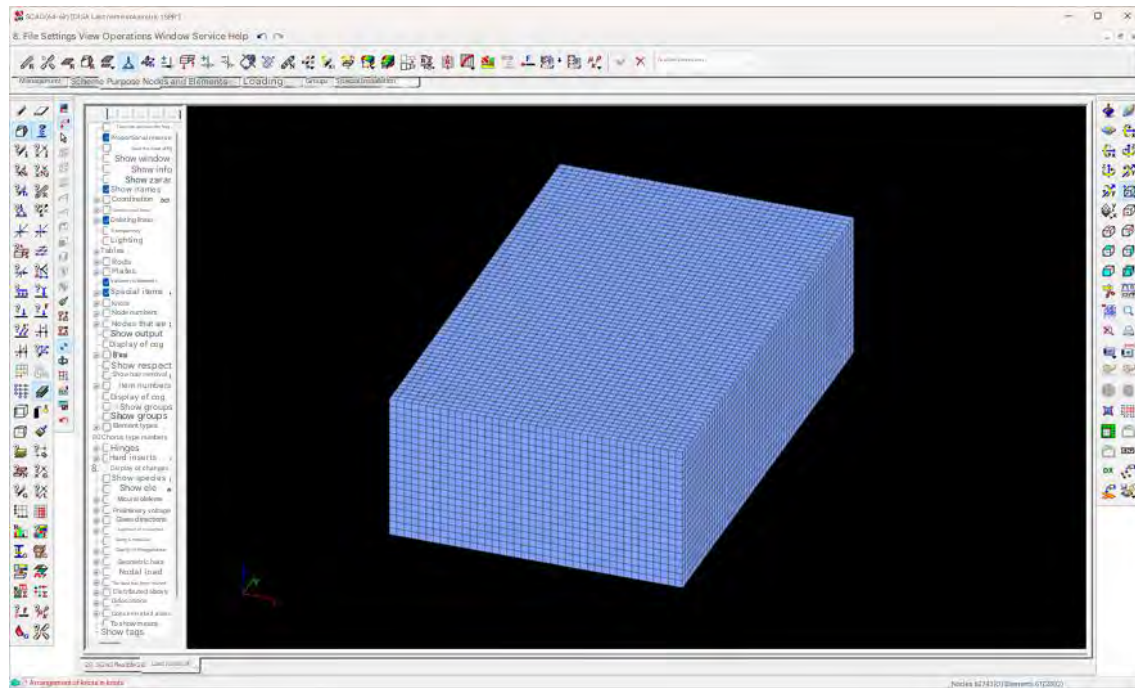


Figure 6.10 – Soil model on the working field of the program

We make a projection of the soil model onto the plane of the HOU and turn on the display of the coordination axes. We calculate the required size of the expansion of the soil model in the plan in accordance with its geometry with the task of achieving a distribution angle α of 25° using geometric construction, or trigonometric dependencies, for example, as in Figure 6.11.

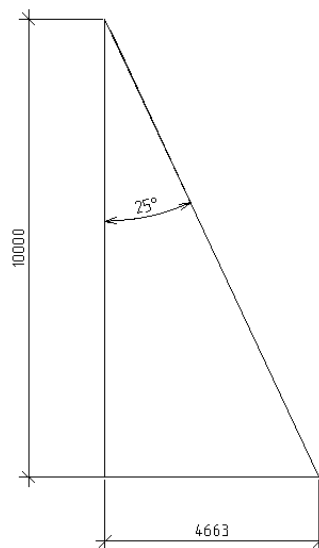


Figure 6.11 – Scheme for calculating the geometry of the soil massif.

We round the obtained value in accordance with the value of the triangulation step of your scheme and copy or complete the required number of elements of the soil massif. We return to the initial display of the scheme, check the result and set the boundary conditions (elms in the nodes on the edges of the soil massif) similar to the actions discussed in Topic 5.

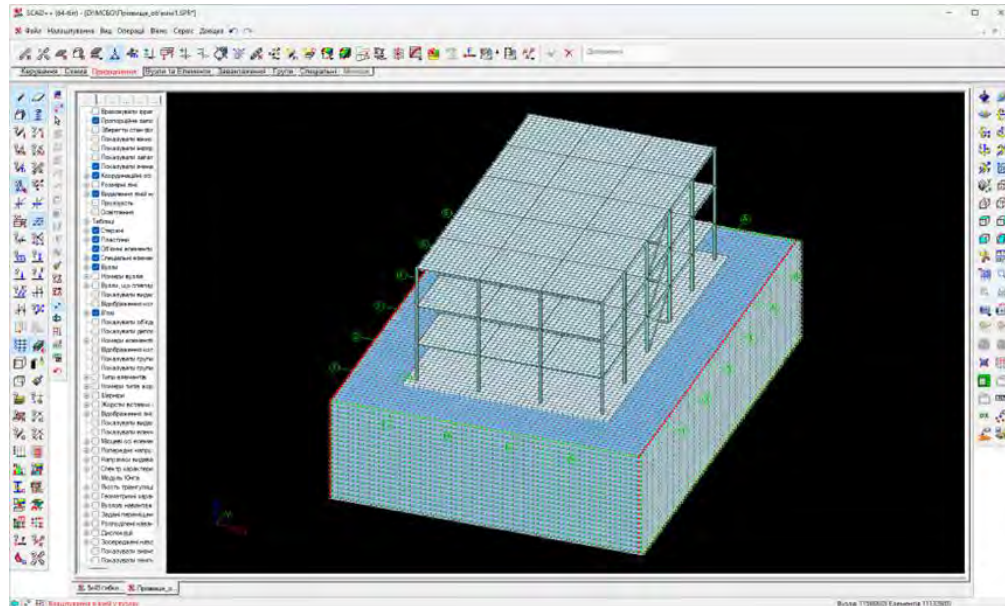


Figure 6.12 – The calculation scheme is prepared for calculation

On the project management screen, perform a linear calculation and switch to the graphical analysis mode.

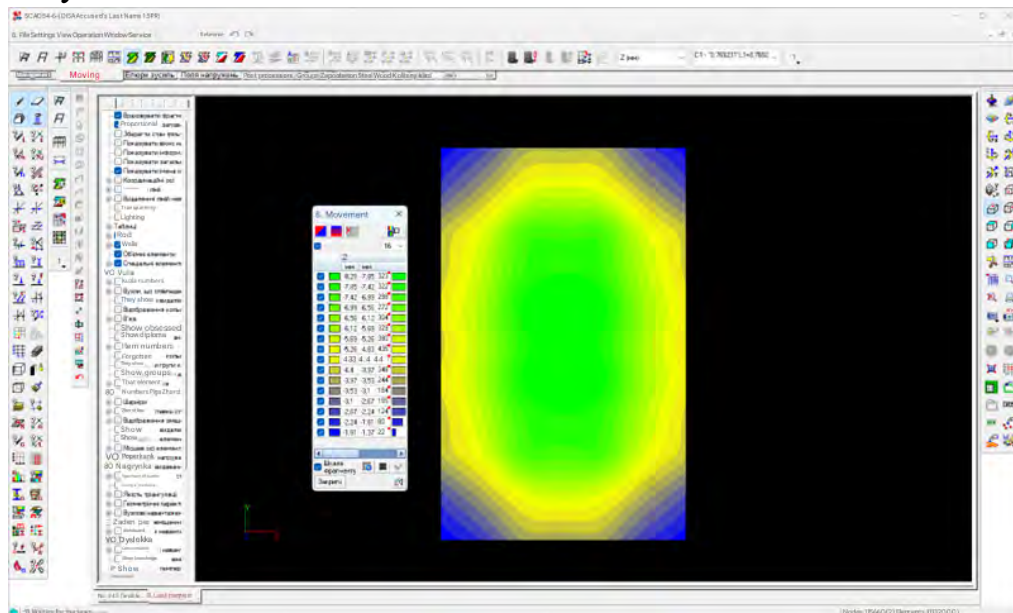


Figure 6.13 – Isopoles of displacements (subsidence) of the foundation slab

The screenshot shows the SAP2000 software interface. The main window displays a stress field plot for a reinforced concrete slab. The plot is color-coded, with a central yellow region indicating high stress and surrounding green and blue areas indicating lower stress. A 'Stress' dialog box is open, showing a table of stress values for various elements. The table has columns for 'Mx (T/m)', 'My (T/m)', and 'Mz (T/m)'. The values are as follows:

Element	Mx (T/m)	My (T/m)	Mz (T/m)
1	3.62	1.52	592 *
2	1.52	0.58	175 *
3	0.58	2.68	110 *
4	2.68	4.77	802 *
5	4.77	6.87	441 *
6	6.87	8.97	238 *
7	8.97	11.07	184 *
8	11.07	13.17	154 *
9	13.17	15.27	124 *
10	15.27	17.37	94 *
11	17.37	19.47	64 *
12	19.47	21.57	34 *
13	21.57	23.67	4 *
14	23.67	25.77	32 *
15	25.77	27.86	32 *
16	27.86	29.96	18 *

The screenshot shows the ANSYS Workbench environment. The main window displays a 2D stress analysis of a mechanical part. The stress distribution is visualized using a color scale from green (low stress) to yellow (high stress). A 'Tension' table is open, showing the following data:

Element	Min	Max	Avg	Std
1	1.7	0.4	1.72	0.0
2	0.4	0.4	0.4	0.0
3	0.4	0.4	0.4	0.0
4	0.4	0.4	0.4	0.0
5	0.4	0.4	0.4	0.0
6	0.4	0.4	0.4	0.0
7	0.4	0.4	0.4	0.0
8	0.4	0.4	0.4	0.0
9	0.4	0.4	0.4	0.0
10	0.4	0.4	0.4	0.0
11	0.4	0.4	0.4	0.0
12	0.4	0.4	0.4	0.0
13	0.4	0.4	0.4	0.0
14	0.4	0.4	0.4	0.0
15	0.4	0.4	0.4	0.0
16	0.4	0.4	0.4	0.0
17	0.4	0.4	0.4	0.0
18	0.4	0.4	0.4	0.0
19	0.4	0.4	0.4	0.0
20	0.4	0.4	0.4	0.0
21	0.4	0.4	0.4	0.0
22	0.4	0.4	0.4	0.0
23	0.4	0.4	0.4	0.0
24	0.4	0.4	0.4	0.0
25	0.4	0.4	0.4	0.0
26	0.4	0.4	0.4	0.0
27	0.4	0.4	0.4	0.0
28	0.4	0.4	0.4	0.0
29	0.4	0.4	0.4	0.0
30	0.4	0.4	0.4	0.0
31	0.4	0.4	0.4	0.0
32	0.4	0.4	0.4	0.0
33	0.4	0.4	0.4	0.0
34	0.4	0.4	0.4	0.0
35	0.4	0.4	0.4	0.0
36	0.4	0.4	0.4	0.0
37	0.4	0.4	0.4	0.0
38	0.4	0.4	0.4	0.0
39	0.4	0.4	0.4	0.0
40	0.4	0.4	0.4	0.0
41	0.4	0.4	0.4	0.0
42	0.4	0.4	0.4	0.0
43	0.4	0.4	0.4	0.0
44	0.4	0.4	0.4	0.0
45	0.4	0.4	0.4	0.0
46	0.4	0.4	0.4	0.0
47	0.4	0.4	0.4	0.0
48	0.4	0.4	0.4	0.0
49	0.4	0.4	0.4	0.0
50	0.4	0.4	0.4	0.0
51	0.4	0.4	0.4	0.0
52	0.4	0.4	0.4	0.0
53	0.4	0.4	0.4	0.0
54	0.4	0.4	0.4	0.0
55	0.4	0.4	0.4	0.0
56	0.4	0.4	0.4	0.0
57	0.4	0.4	0.4	0.0
58	0.4	0.4	0.4	0.0
59	0.4	0.4	0.4	0.0
60	0.4	0.4	0.4	0.0
61	0.4	0.4	0.4	0.0
62	0.4	0.4	0.4	0.0
63	0.4	0.4	0.4	0.0
64	0.4	0.4	0.4	0.0
65	0.4	0.4	0.4	0.0
66	0.4	0.4	0.4	0.0
67	0.4	0.4	0.4	0.0
68	0.4	0.4	0.4	0.0
69	0.4	0.4	0.4	0.0
70	0.4	0.4	0.4	0.0
71	0.4	0.4	0.4	0.0
72	0.4	0.4	0.4	0.0
73	0.4	0.4	0.4	0.0
74	0.4	0.4	0.4	0.0
75	0.4	0.4	0.4	0.0
76	0.4	0.4	0.4	0.0
77	0.4	0.4	0.4	

134

The results are entered in Table 6.1.

We save the next version of the calculation file and perform the described similar procedures, changing the distribution angle to $\alpha=45^\circ$. We repeat the calculation, enter the results of the calculation of subsidence and bending moments in the plate in Table 6.1. We analyze the results obtained, draw conclusions.

Tasks for independent work on topic 6.

Perform the calculation of the subsidence of the foundation according to the model of a linear-deformed medium (in the program "REQUEST") using identical parameters of dimensions, soil characteristics and load. Compare the results of the calculation with the results of determining the subsidence for different distribution angles from Table 6.1.

To perform an examination of reinforced concrete and metal elements of the building frame for three options for the resolution of the soil base. Analyze the critical factors of the elements in all three versions of topic 6 and in the version from topic 5. If necessary, select reinforcement, change the grade of concrete, sections of metal elements, etc. Explain the results obtained.