

## **OBSAH**

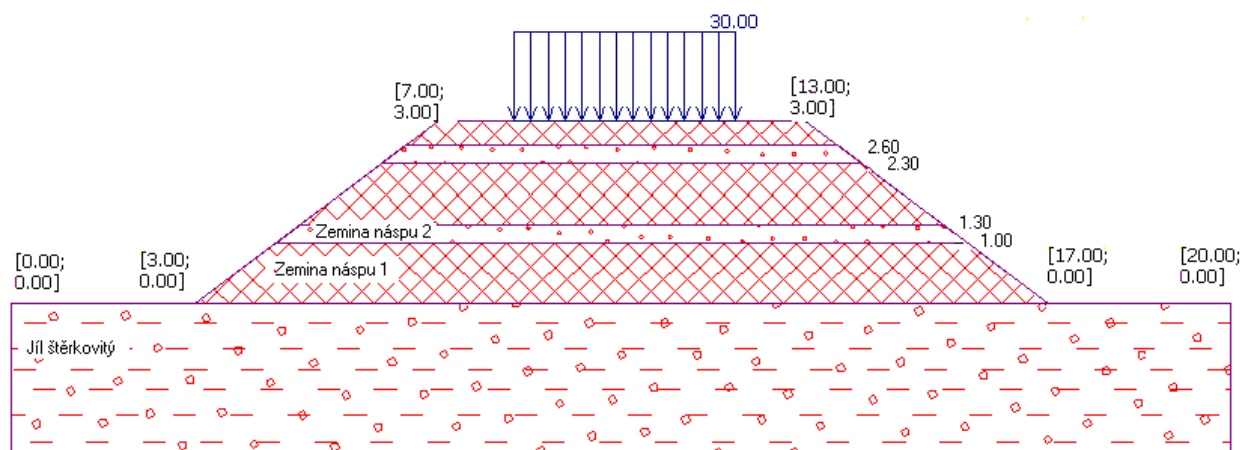
<b>5. Terrain settlement – program „Depression“.....</b>	<b>2</b>
<b>5.1 Example .....</b>	<b>2</b>
<b>5.2 Setting up the original ground .....</b>	<b>3</b>
<b>5.3 Embankment setup and settlement computation.....</b>	<b>6</b>
<b>5.4 Adding surcharge, overall settlement.....</b>	<b>9</b>

## 5. Terrain settlement – program „Depression“

The following chapter outlines computation of the terrain settlement under the embankment. A special attention is paid to the description of the output window, which enables to store various views of the structure. This window is also used in other programs of the GEO4 system, e.g., the programs BEAM, PLATE, etc.

### 5.1 Example

Compute the terrain settlement caused by an embankment and the overall terrain settlement after introducing a surcharge displayed in **Fig. 5.1**.



**Fig. 5.1 Example setup**

Soil parameters:

Name	$\gamma$ [kN/m <sup>3</sup> ]	$m$ [-]	$E_{def}$ [kPa]	$\nu$ [-]
Gravelly soil	20.00	0.20	25.00	0.30
Embankment soil 1	19.00	0.30	14.00	0.35
Embankment soil 2	20.00	0.30	13.00	0.35

## 5.2 Setting up the original ground

To create the original ground you proceed as in the program Slope stability. Details are given in Chapter 2. The present chapter reviews only the specific features of the program Depression. As a first step when creating the ground you enter the world coordinates – 0-20m in the present example (Fig. 5.2).

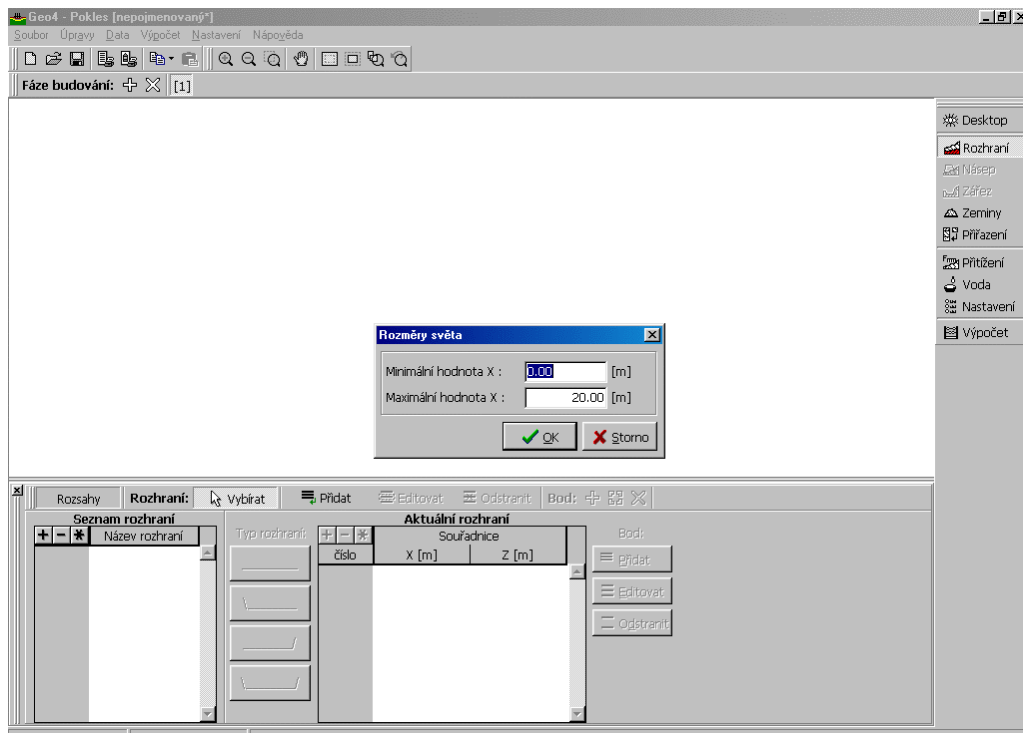


Fig. 5.2 Dialog window to enter world coordinates

Then you specify the shape of the ground (Fig. 5.3).

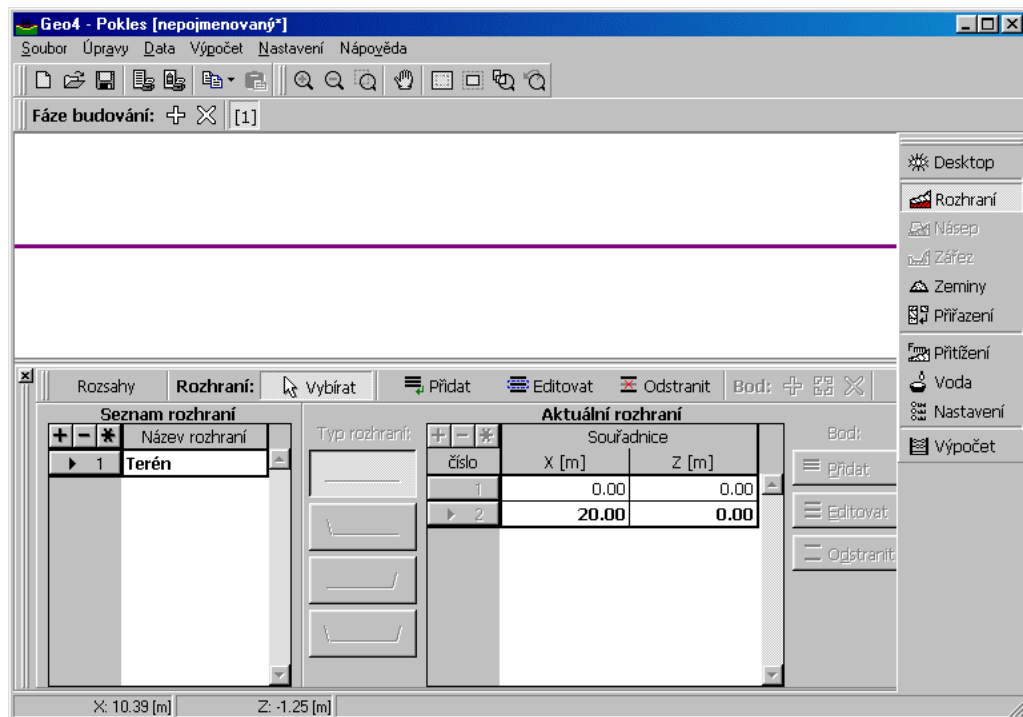
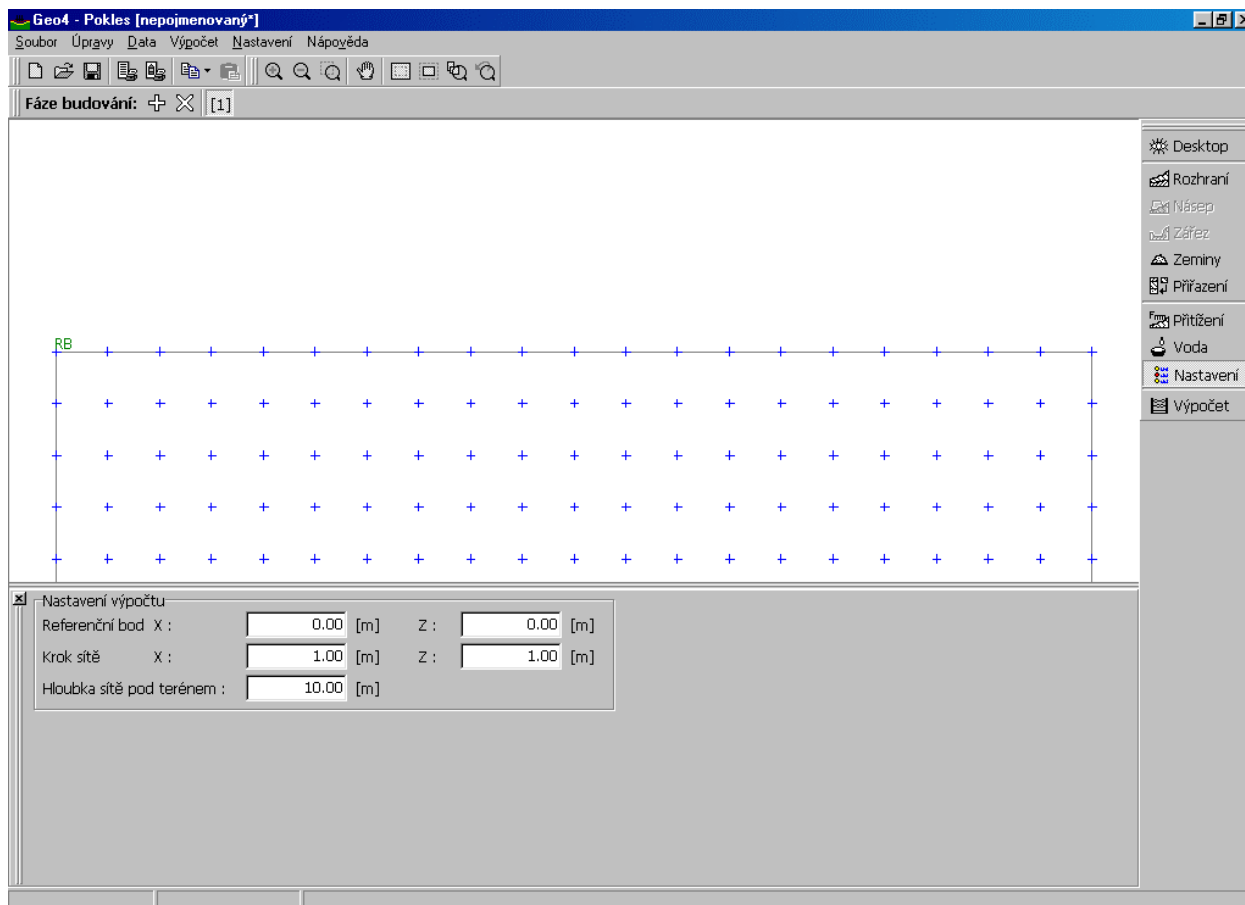


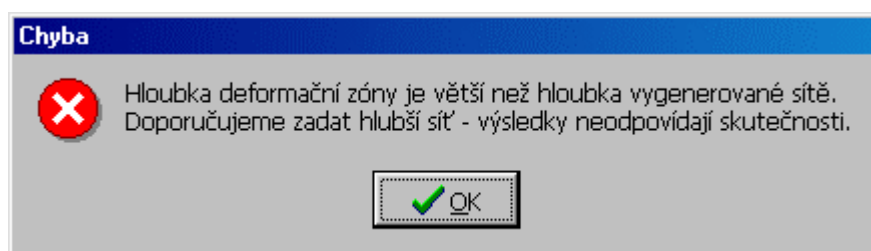
Fig. 5.3 Ground shape setup

After specifying the soil parameters (at the same time you are required to define all soils which may appear in subsequent stages of construction – embankment soil in particular) you assign the selected soil into the profile. Neither surcharge nor the ground water is present in the first stage of construction. As a next step you define a mesh of points at which you wish to calculate the stress state and settlement of individual layers.



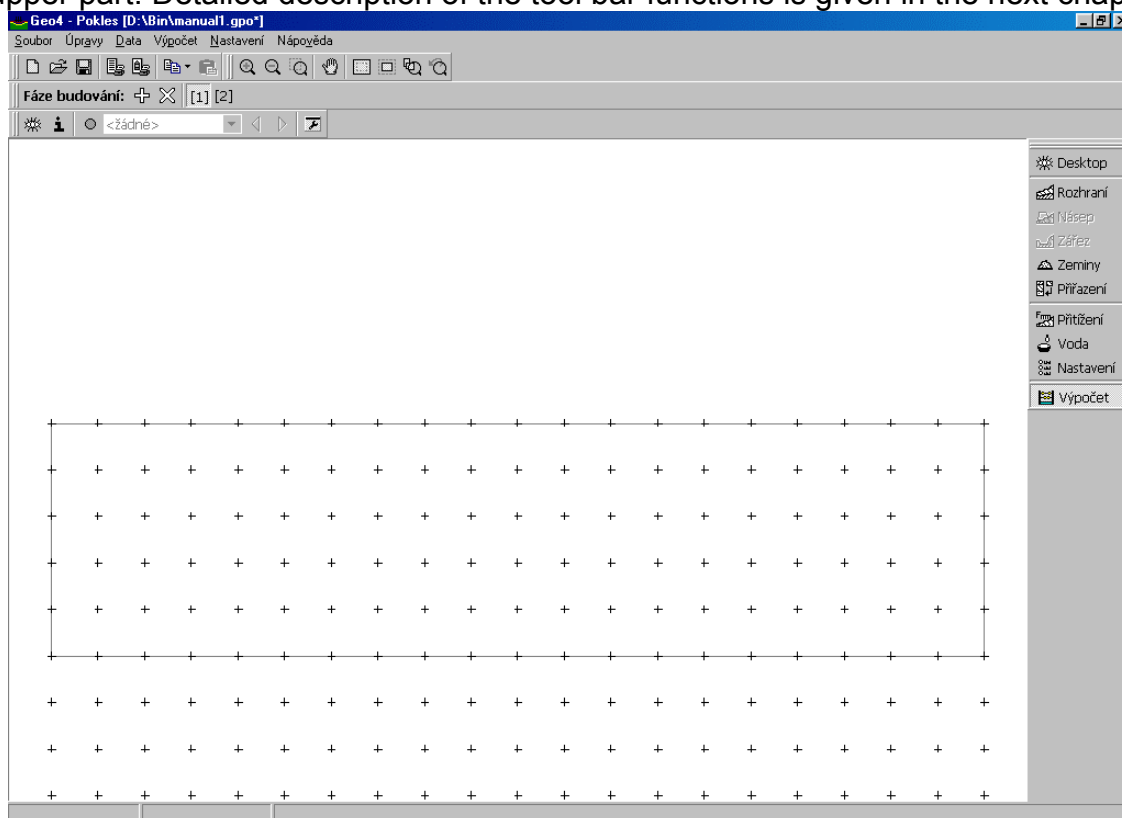
**Fig. 5.4 Mesh of points for stress and settlement computation**

To proceed, you select the reference point (determines the mesh location) and the mesh step in both the horizontal and vertical directions. Note that finer mesh subdivision provides more accurate results. The computational effort, however, increases and the resulting output becomes less lucid. In this regard, the mesh of 25 by 25 points appears optimal. This window further serves to specify the depth of mesh under the ground. If the influence zone (zone of deformation) exceeds the current mesh depth, the program prompts an error message, **Fig. 5.5**, and you are required to increase the mesh depth.



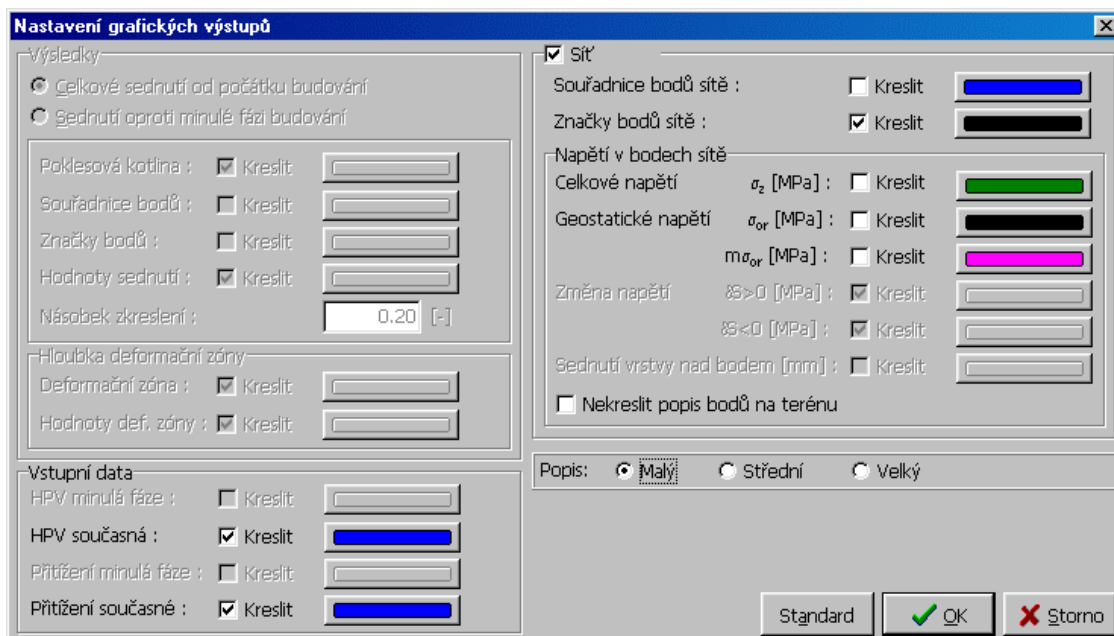
**Fig. 5.5 Error message – insufficient depth of mesh**

After pressing the „**Analysis**“ button, the screen changes and a new tool bar appears in its upper part. Detailed description of the tool bar functions is given in the next chapter.



**Fig 5.6 Screen after analysis**

Pressing the window button on this tool bar opens the dialog window for setting up the graphical output, **Fig 5.7**.



**Fig. 5.7 Dialog window to set up parameters of graphical output**

As we are still in the first stage of construction (original ground setup) the program shows only the initial state of stress of the soil body.

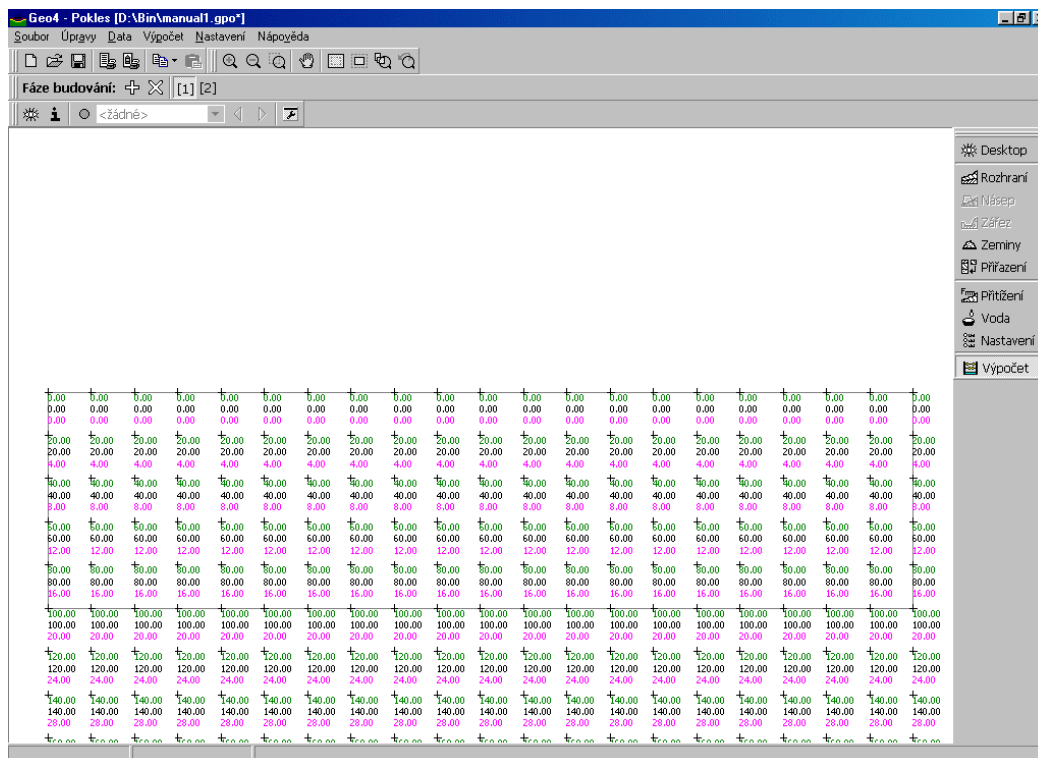


Fig. 5.8 Presentation of results – stresses at mesh points

### 5.3 Embankment setup and settlement computation

To add a new stage of construction, press the „+“ button on the „Construction stage“ tool bar. The „Embankment“ and „Cut“ buttons appear on the tool bar, while the „Interface“, „Soils“ and „Setup“ buttons are deactivated (these buttons are accessible only in the first stage of construction).

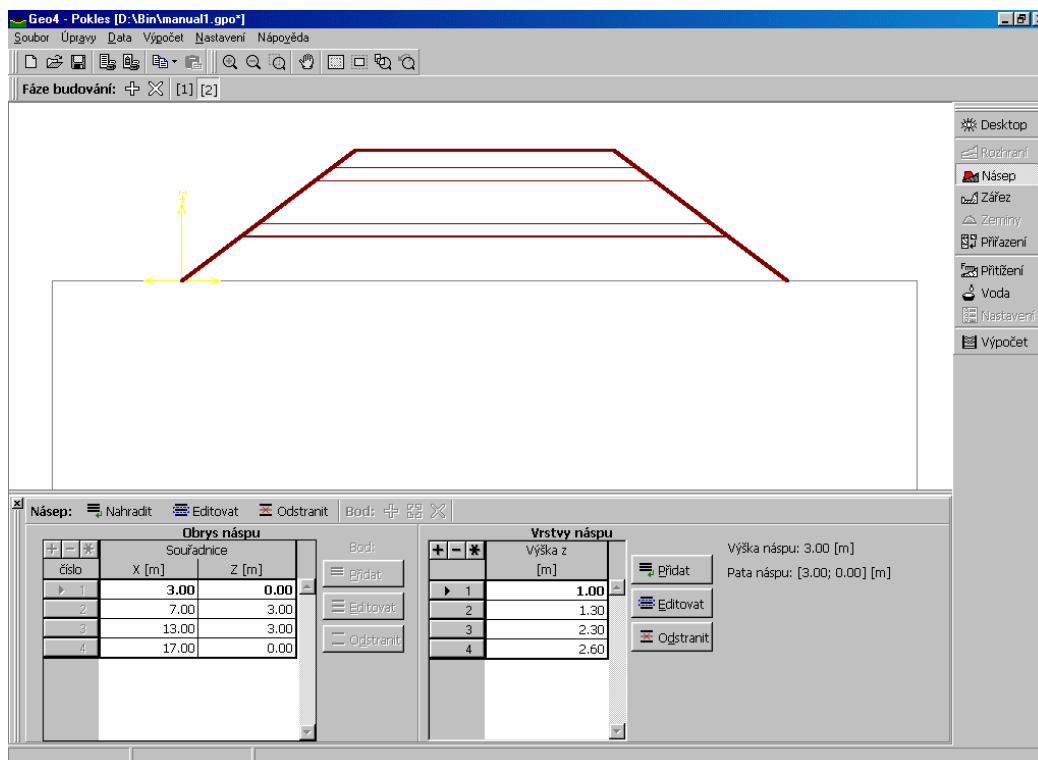


Fig. 5.9 Dialog window to define embankment

Now you may proceed to define an embankment. First you specify the embankment shape and then thickness of individual layers, **Fig. 5.9**. After assigning soils into the embankment you run the analysis (**Fig 5.10**).

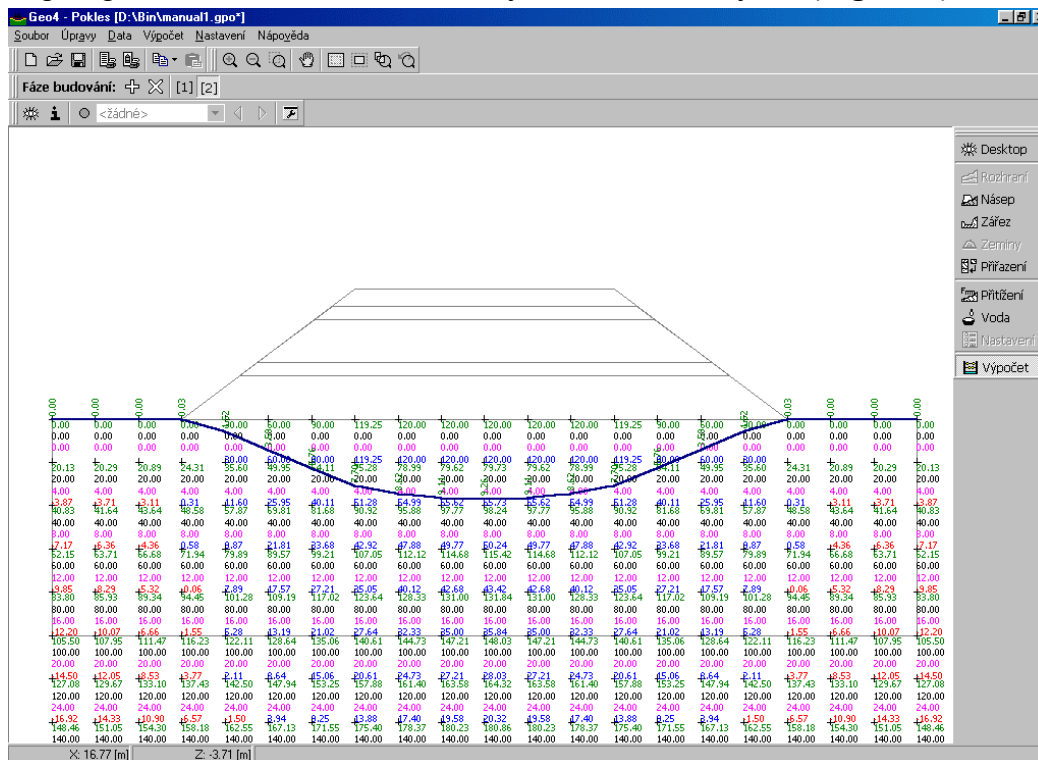


Fig. 5.10 Analysis results

The figure shows depression and stresses at points of the mesh. To plot depression and shape of the influence zone only, **Fig. 5.11**, change the plotting parameters.

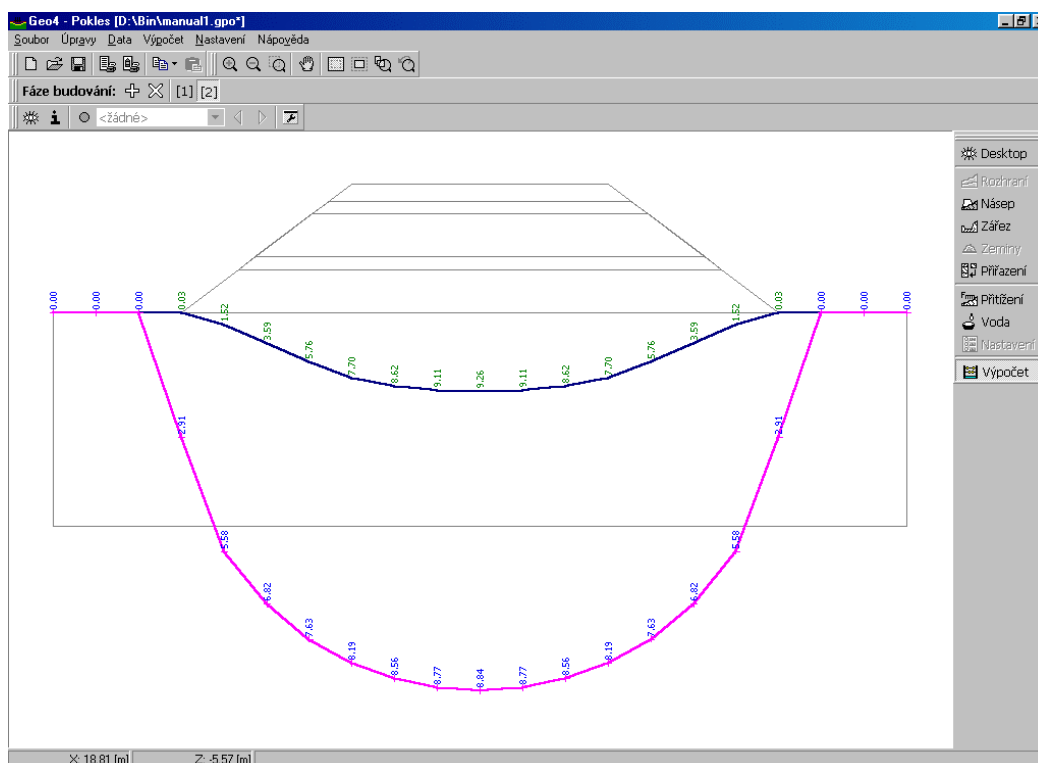


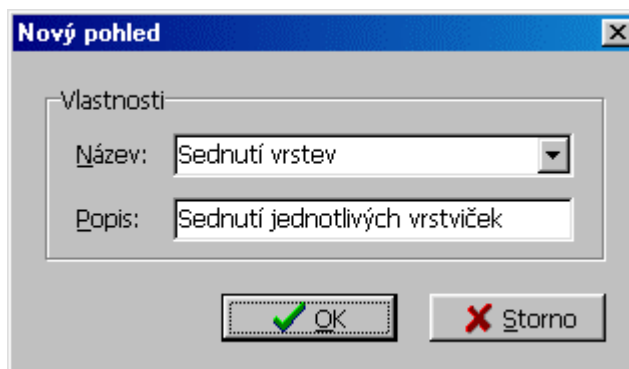
Fig. 5.11 Depression and influence zone

Since defining individual views is time consuming, the program enables their storage for later use. The tool bar displayed in **Fig. 5.12** serves to that purpose.



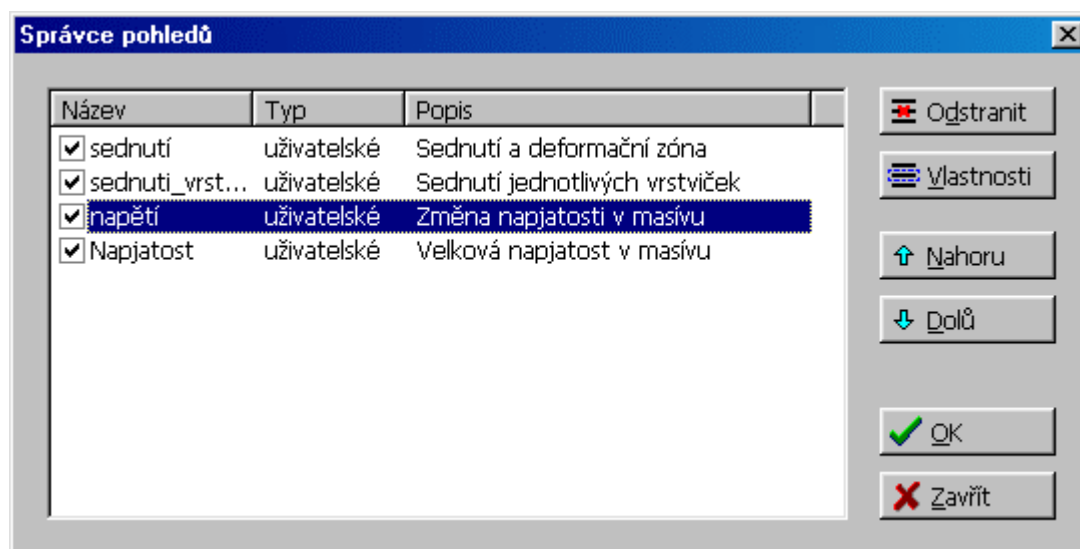
**Fig. 5.12** Tool bar to explore stored views

**Fig. 5.13** shows properties to be specified when storing a view.



**Fig. 5.13** Dialog window to enter name and description of a view

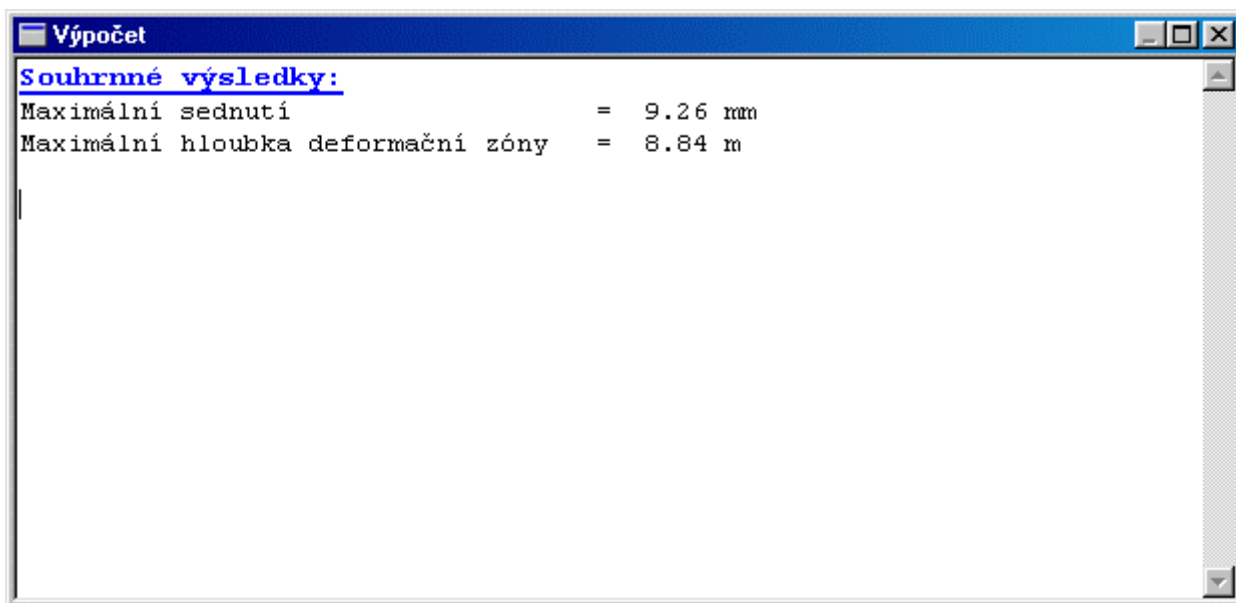
An arbitrary number of views can be stored in the program. When running the analysis again all views are automatically redrawn. Tool bar arrows or the view manager serve to jump between individual views, **Fig. 5.14**.



**Fig. 5.14** Dialog window „View manager“

Using the „**Graphical printout**“ option in the menu bar allows printout of all selected views in the current stage of construction.

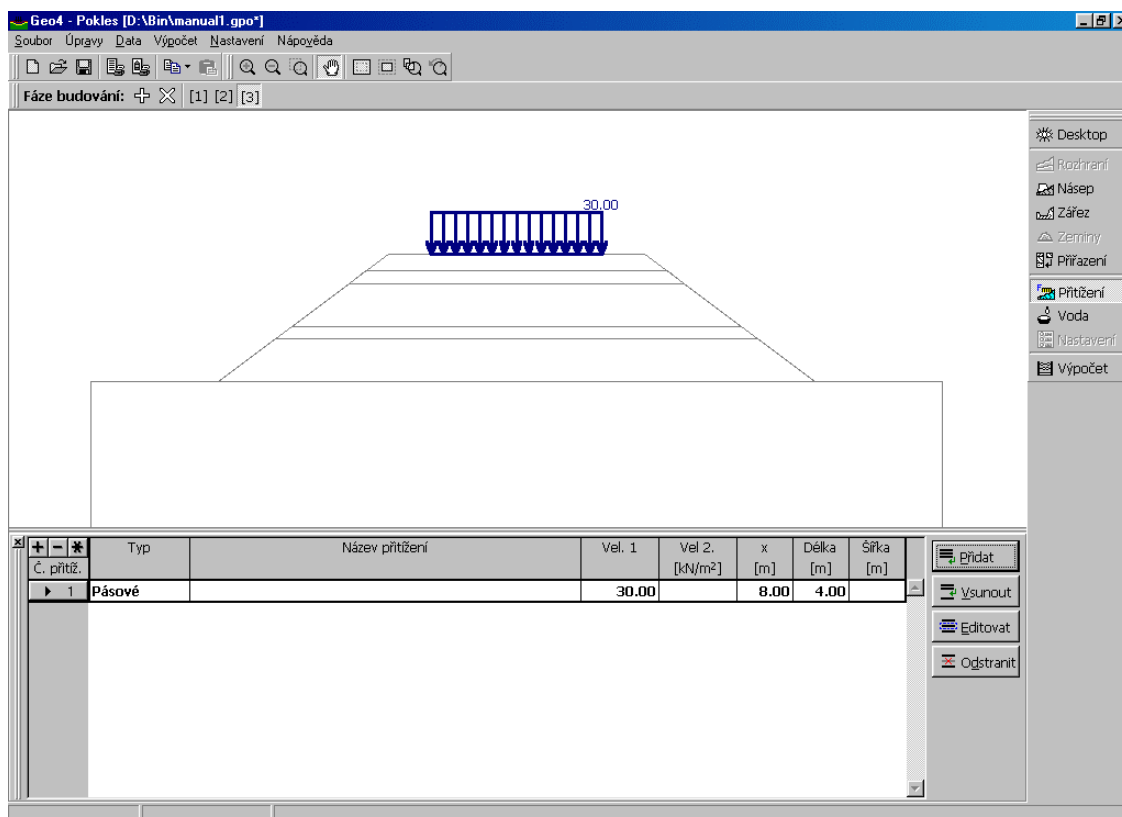
Text information (**Fig. 5.15**) about the performed analysis can be displayed on screen by pressing the button „f“.



**Fig. 5.15** Text information

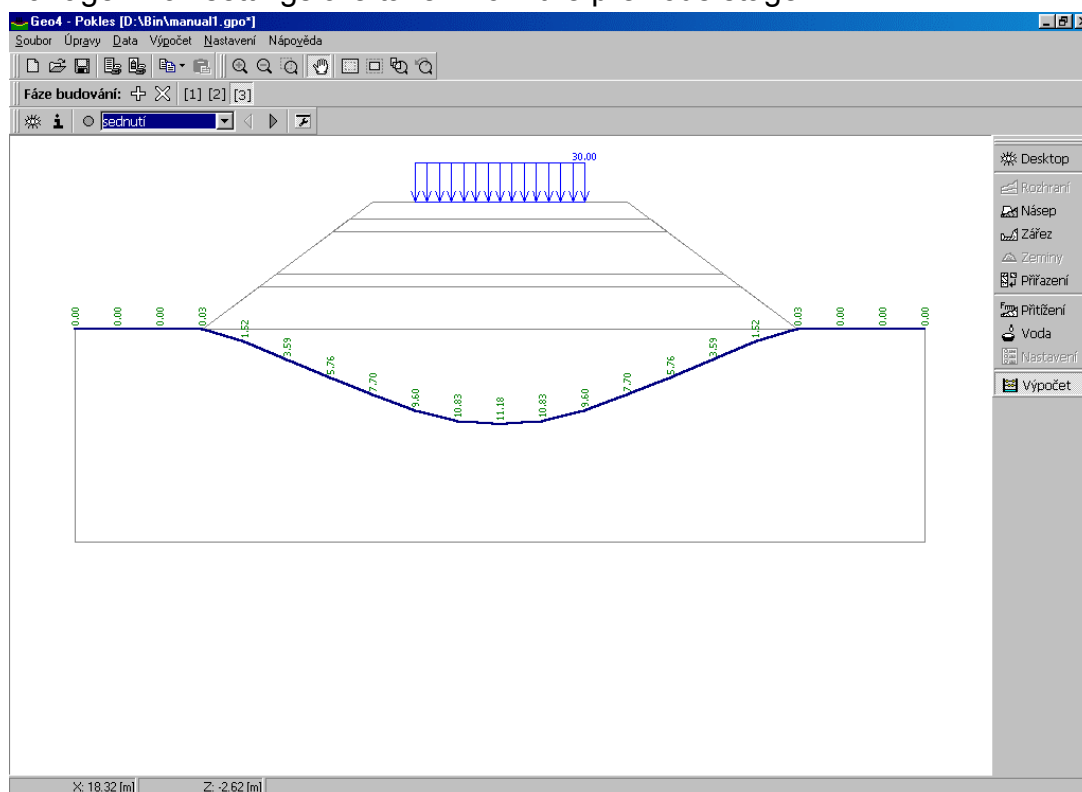
#### 5.4 Adding surcharge, overall settlement.

Add a new construction stage and introduce the desired surcharge (**Fig. 5.16**).



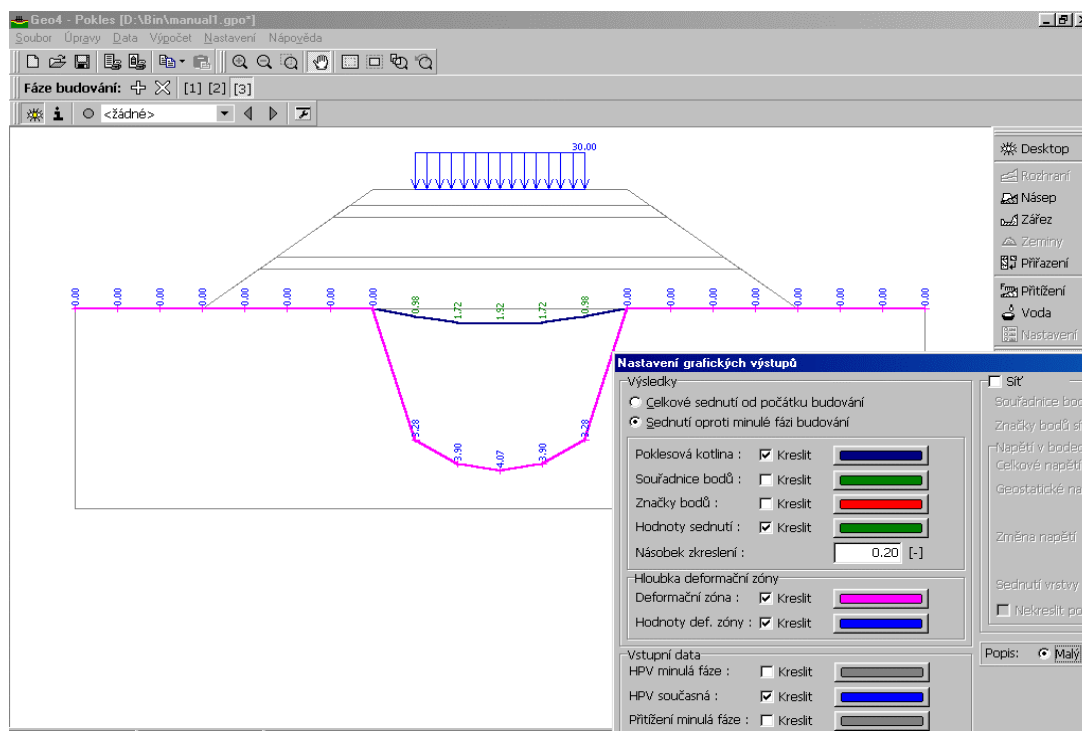
**Fig. 5.16** Additional stage of construction – embankment surcharge

After running the analysis (Fig. 5.17) you may immediately view the results using the view manager – all settings are taken from the previous stage.



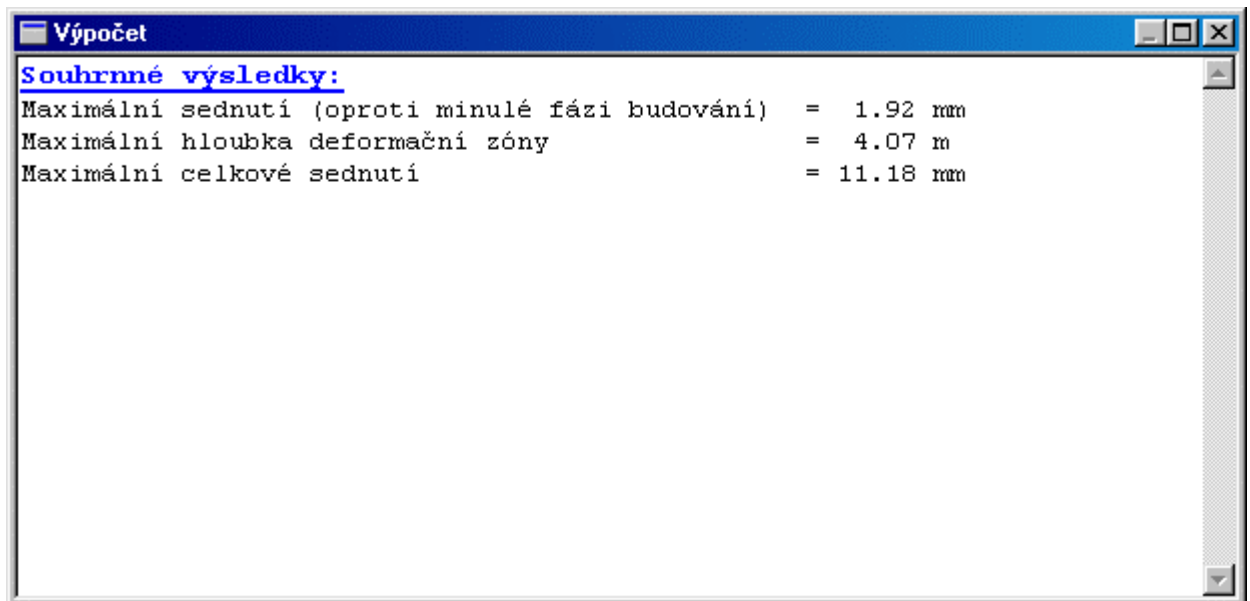
**Fig. 5.17 Analysis after adding a surcharge**

This final stage of construction allows you to view both the overall (overall settlement from the beginning of construction) and partial results (settlement and change of stress from stage to stage). To move between individual results use the „Graphical output setting“ dialog window, Fig. 5.18.



**Fig. 5.18 Additional stage of construction – embankment surcharge**

Note that it is not possible to view stresses at mesh points and the depth of influence zone when plotting the overall settlement. Pressing the “7” button opens the text printout window with the maximum values of settlement, **Fig. 5.19**.



**Fig. 5.19 Text printout**