Table of contents

3. Analyses of walls and supporting constructions

The previous chapter described analyses of earth-pressures. This chapter outlines analysis of supporting structures, e.g., gravity and cantilever walls. Common features of all programs are discussed first (**Chapters 3.1 – 3.6**). Specifics of individual programs are explained next.

3.1 Resistance on front face of a construction

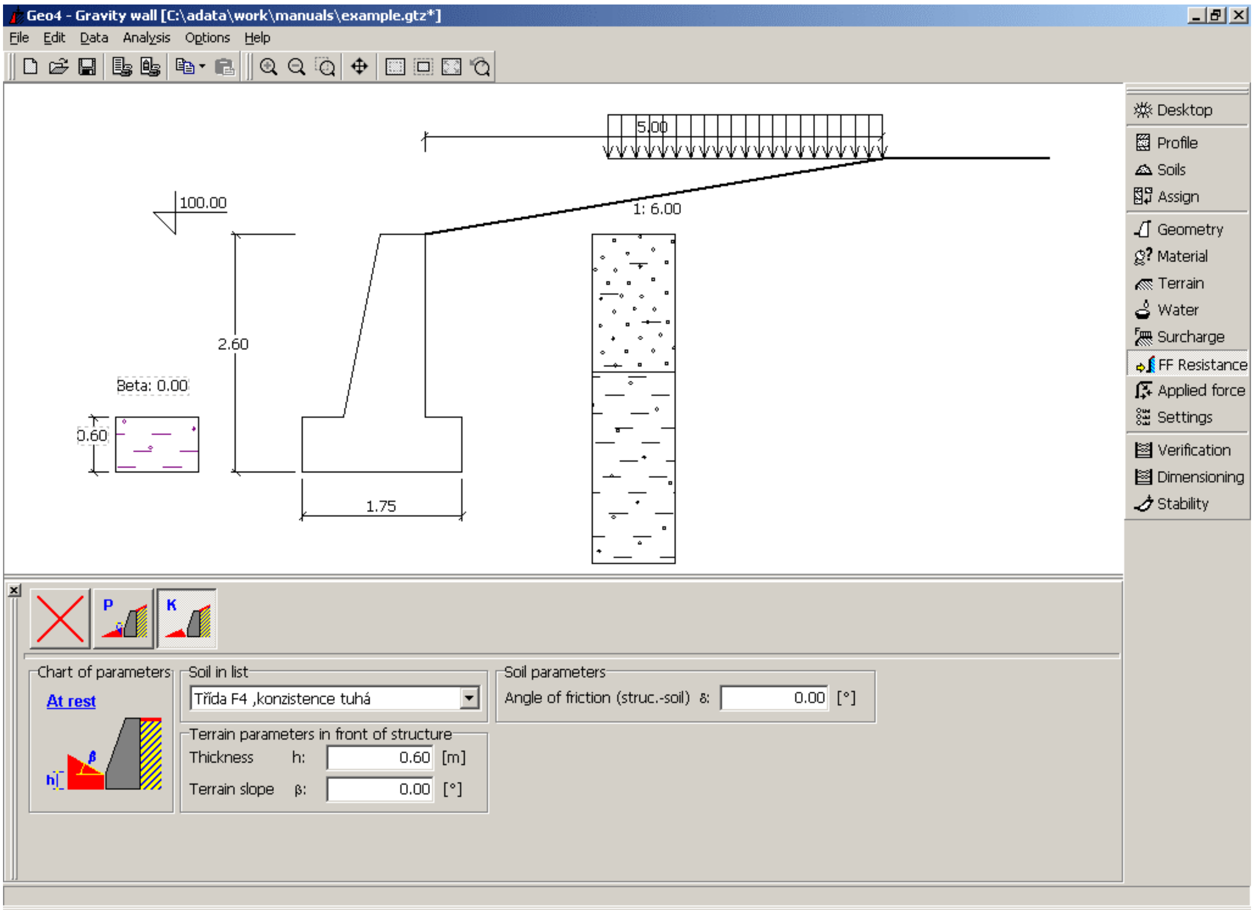
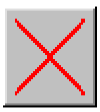


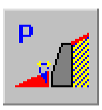
Fig. 3.1 Dialog window to specify parameters of front face resistance

Use the "**FF Resistance**" dialog window to specify the terrain profile, soil and its influence on a construction (**see Fig. 3.1**).

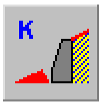
Three options are available:



The soil in front of the construction is not considered, so that the front face resistance is not realized.



The FF resistance is considered a passive pressure. In such a case, you specify the terrain profile, soil and the angle of friction construction-soil. When selecting this option, one should check whether the structure deformation is sufficient to mobilize the passive pressure (see **Chapter 2.2 "Types of earth-pressures**" for details). For supporting constructions the authors recommend to reduce the resulting passive pressure by the design coefficient, which can be inputted in the verification or dimensioning dialog window, respectively (see **Chapter 3.3**). Note that reducing the magnitude of passive pressure by 1/2 reduces the structure deformation up to 1/5.



The FF resistance is considered as a pressure at rest. In this particular case you specify the terrain profile and soil in front of the construction.

The earth-pressures are computed with reduction of input parameters of soil according to the option selected in „**Settings - Analysis**“.

3.2 Inserted forces

The program enables the user to put in an arbitrary number of forces entering the analysis (loading at the wall crest, anchoring, etc.). The forces are introduced in the „**Applied force**“ dialog widow - **Fig. 3.2**.

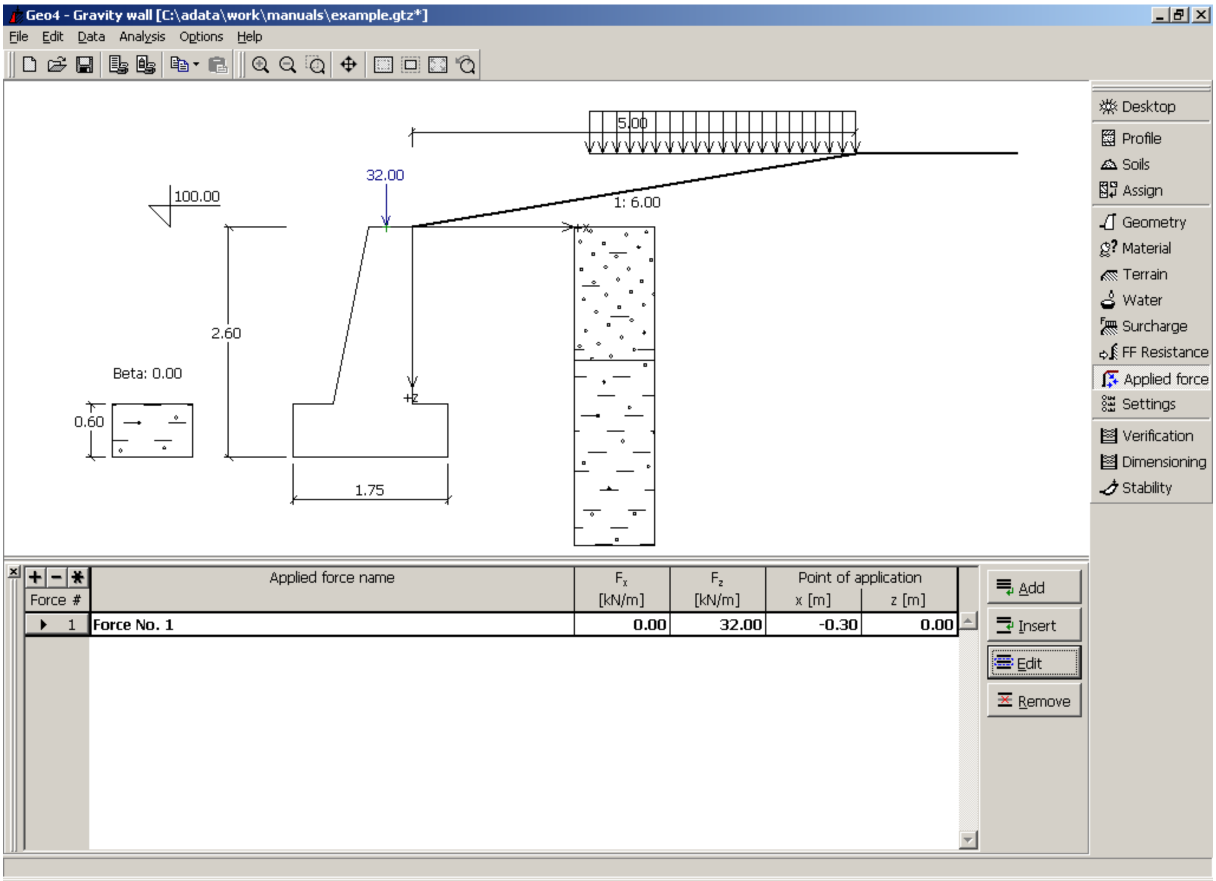


Fig. 3.2 Dialog window for input of forces

Verification analysis includes all the inputted forces (see **Fig. 3.3**), while dimensioning takes into account only forces above the verified joint. The forces are not multiplied by any design coefficient.

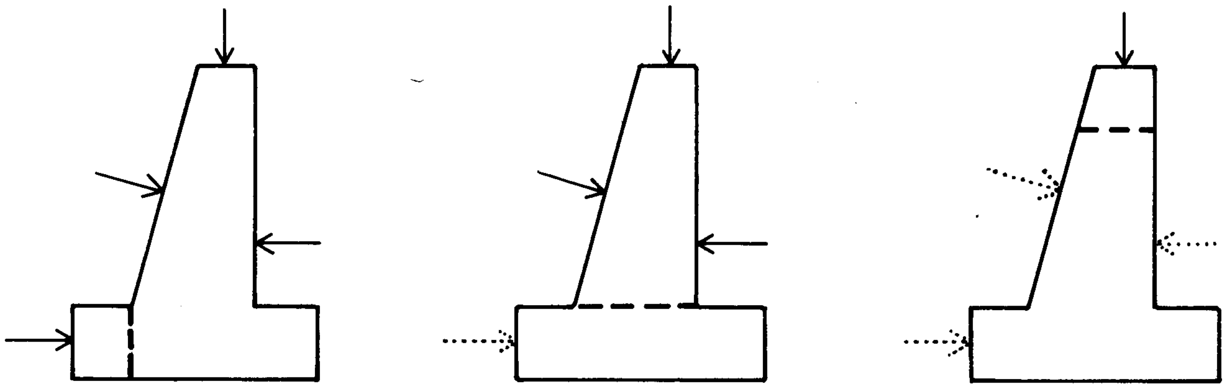


Fig. 3.3 Forces entering the analysis

3.3 Forces on construction, coefficients of forces

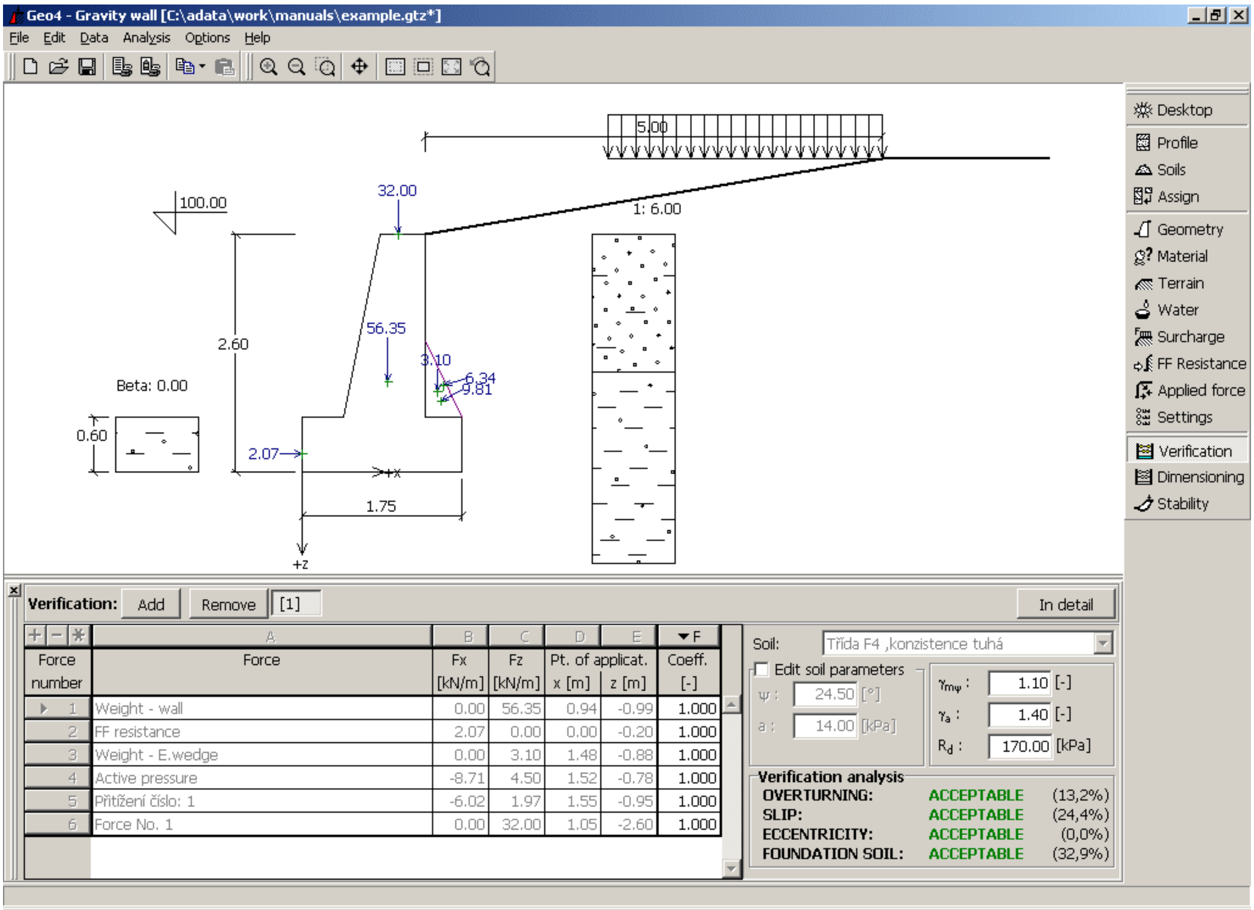


Fig. 3.4 The dialog window „Verification“ – input of coefficients of forces

The particular analysis is carried out after pressing the buttons „**Verification**“ and „**Dimensioning**“, respectively. The forces acting upon a given construction are displayed (**Fig. 3.4**).

The forces and their locations are listed in the table in the left lower part of the window. The computation of forces is performed gradually, and the forces are ordered as follows:

- **Wall tensile force**
  It depends on the shape and bulk weight of a wall (for input use the "**Material**" dialog window). It is not multiplied by any design coefficient. If the wall is below the water table, the analysis assumes uplift pressure of *10 kN/m*^{3}.
- **Front face resistance**
  The force acts, when putting in the FF resistance, as a pressure at rest or as a passive pressure, respectively (see Chapter 3.2).
- **Tensile forces of earth wedges**
  The number of such forces depends on the shape of a construction. Their computation is described in Chapter 3.4.
- **Active pressure and pressure at rest, respectively, acting on a structure**
  Is the basic loading on a construction caused by soil pressure (see Chapters 2.3 resp.2.5, 2.6, 2.7, 2.8, for details). This force includes water and pore pressure influence. According to the selected option in the „Analysis setup", the pressure might be computed with the reduction of input parameters of soil.
- **Forces due to surcharge**
  Each surcharge corresponds to one particular force. If the magnitude of force is zero (no effect of surcharge), then the force is not plotted on the picture, but appears only in the table (see Chapter 2.8 for details).
- **Inputted forces**
  The forces that enter the analysis are displayed - see Chapter 3.2.

The table contains the column „**Coeff.**" that serves to specify the magnitude of the coefficient used to multiply a respective force during the verification or dimensioning analysis, respectively. The default value of each coefficient is one.

The coefficients may serve to, e.g.:

- Reduction of passive pressure acting on a front face (see **Chapter 3.1**)
- Reduction of magnitudes of forces due to surcharges when combining more surcharges at the same time
- Introducing new design coefficients
- Excluding a force from the analysis (*coeff.=0*)

The forces displayed on the screen are already multiplied by corresponding coefficient. When the coefficient equals 0, the force is not shown at all.

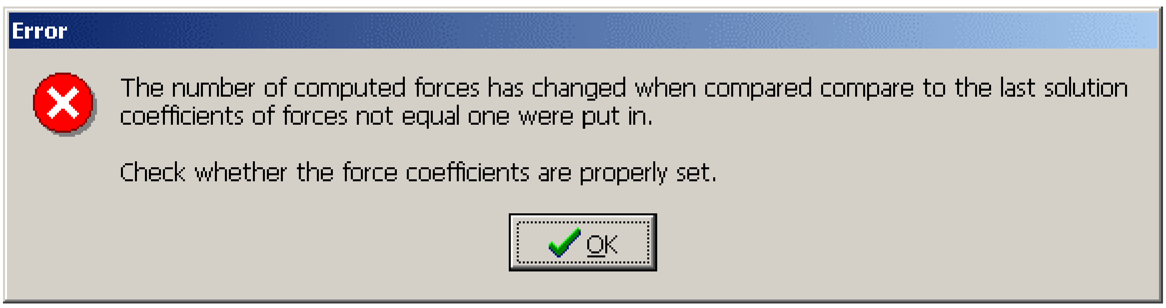


Fig. 3.5 Warning for possible change of coefficients references

Coefficients are saved into data and their magnitudes are kept even during new calculations – but they are not strictly tied to a given force. When the number of the forces changes, the inputted coefficient can move to another force. When a coefficient differs from 1, the program prompts a warning message (**Fig. 3.5**). In that case, we recommend the user to check, whether the coefficients of forces are inputted correctly.

3.4 Determination of earth wedges

Procedure for computation of active pressure:

First, assign the shape of wedge behind the wall. For individual layers of soil above the wall foundation the program computes the angle υ_a, which serves to determine the angle υ_{as}, see **Fig. 3.6**.

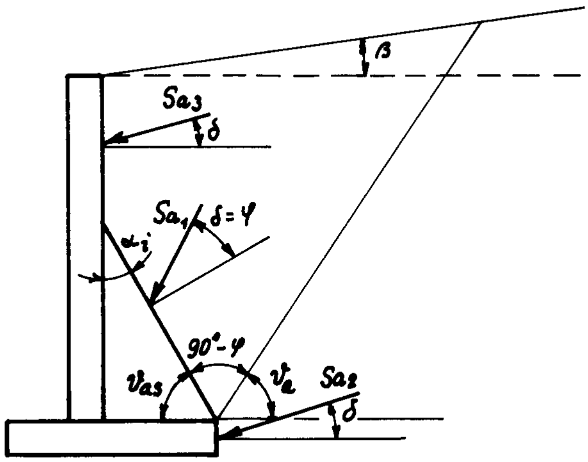


Fig. 3.6 Earth wedge for active pressure

The angle υ_a is given by

$$\upsilon_a = \varphi + \varepsilon$$

where:

$$\tan\varepsilon = \frac{\cos(\varphi-\alpha)\sin(\varphi-\beta)\cos(\alpha+\delta) + B\cos(\varphi-\beta-\alpha-\delta)}{\sin(\varphi-\alpha)\sin(\varphi-\beta)\cos(\alpha+\delta) + B\sin(\varphi-\beta-\alpha-\delta) + M}$$

$$M = \sqrt{(\sin(\varphi-\beta)\cos(\beta-\alpha)+B).(\sin(\varphi+\delta)\cos(\alpha+\delta)+B)}$$

$$B = \frac{2c\cos\alpha\cos(\beta-\alpha)\cos\varphi}{\gamma h\cos(\beta-\alpha) + \dfrac{2\sigma_z\cos\alpha\cos\beta}{\gamma h}}$$

In the next step, the program determines an intersection of a line drawn under the angle υ_{as} from an upper right point of a foundation block with the next layer. The procedure continues by drawing another line starting from the previously determined intersection and inclined by the angle υ_{as} of a next layer. The procedure is terminated when the line intersects the terrain or wall surface, respectively. The wedge shape is further assumed in the form of triangle (intersection with wall) or rectangle (intersection with terrain).
The wedge self weight is determined from inserted values of bulk weight γ or γ_{su} if a given layer is below the ground water table (see **Chapter 2.1**).

When computing the pressure at rest the earth wedge is vertical – see **Fig. 3.7**.

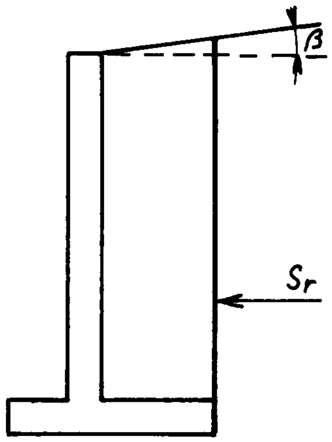


Fig. 3.7 Shape of earth-pressure wedge when computing the pressure at rest

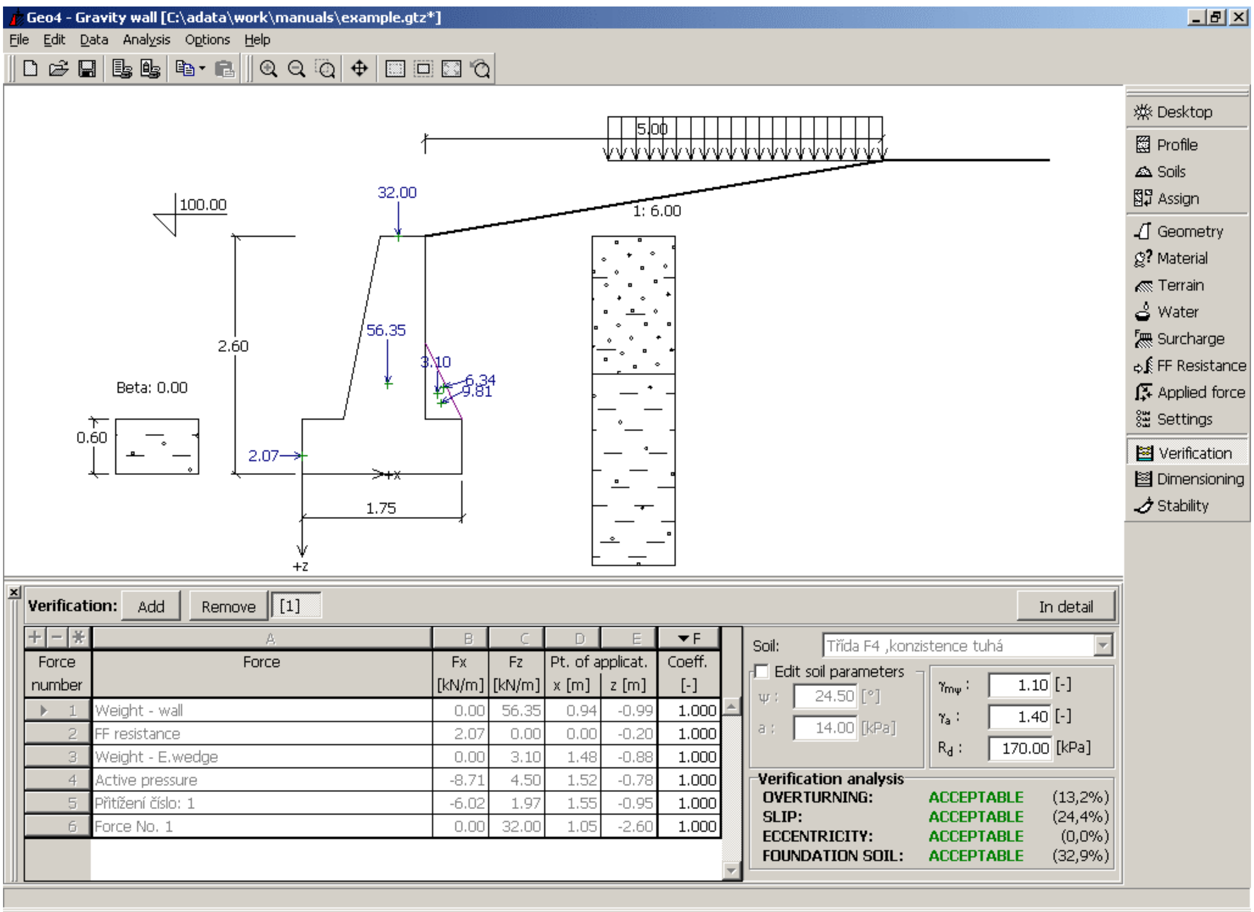


Fig. 3.8 The dialog window „Verification" – input of coefficients of forces

3.5 Verification of wall against sliding and overturning, and of bearing capacity of foundation soil

During the verification analysis the screen appears as in **Fig. 3.8**. The soil below foundation and input parameters for verification, angle of friction construction-soil (ψ), cohesion construction-soil (a), are found in the right part of the window. Standard values of the angle of internal friction (φ) and cohesion (c) of soil below the bottom of a ditch are used. When computing according to the limit states (see **Chapter 2.6**), the input parameters also include coefficients of reduction of parameters of composite action of soil and foundation $\gamma_{m\psi}$ and γ_{ma} (standard values correspond to magnitudes $\gamma_{m\varphi}$ a γ_{mc}).

The design bearing capacity of foundation R_d must also be put in. If desired it can be determined by running the program GEO 4 – Spread foundation.
The verification analysis computes first the forces acting on a wall (F_{ver} – overall vertical force, F_{hor} – overall horizontal force) and overall moments with respect to the left bottom point of the wall foundation (M_{ovr} – overturning moment, M_{res} – resisting moment) due to acting forces (see **Chapter 3.3**).

Then the program determines the overall forces acting at the foundation joint:

$$T = F_{ver} \sin\alpha + F_{hor} \cos\alpha$$

$$N = F_{ver} \cos\alpha + F_{hor} \sin\alpha$$

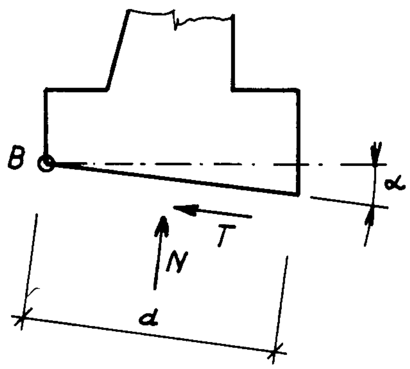


Fig. 3.9 Forces acting at the foundation joint

3.5.1 Verification – limit states

The verification analysis is performed using the following expressions:

Check for overturning stability:

$$M_{ovr}\gamma < M_{res}$$

Check for slip:

$$(N \tan(\psi_d) + a_d (d - 2e))\gamma < T$$

where eccentricity *e* is given by

$$e = \frac{M_{ovr} + M_{res} + \frac{N.d}{2}}{N}$$

Check for bearing capacity of foundation soil:

$$\sigma = \frac{N}{d - 2e} < R_d$$

$$e < \frac{1}{3} d = e_{aloow}$$

The coefficient of reliability γ, when checking for the overturning stability and slip, is assumed equal to 0.9.

3.5.2 Verification – safety factors

The verification analysis is performed using the following expressions:

Check for overturning stability:

$$\frac{M_{res}}{M_{ovr}} < FS_{over.}$$

Check for slip :

$$\frac{(N \tan(\psi) + a(d - 2e))}{T} < FS_{slip}$$

where eccentricity *e* is given by:

$$e = \frac{M_{ovr} + M_{res} + \frac{N.d}{2}}{N}$$

The „**Settings**“ dialog window serves to assign the safety factors SF (standard values are set to *1,5*).

$$\sigma = \frac{N}{d - 2e} < R_d$$

Verification of bearing capacity of foundation soil:

–

3.6 External stability of a construction

The program enables the user to verify the external stability of the entire construction. By pressing the bottom „**Stability**", you switch to the stability program (**Fig. 3.10**) – the construction is generated, and parameters of soils and surcharges are transferred into the program. **Inputted forces, however, are not transferred.** Therefore, their probable effect must be modeled in the program stability of slopes. The actual analysis of stability of slopes is described in **Chapter 4 „Stability of slopes**".

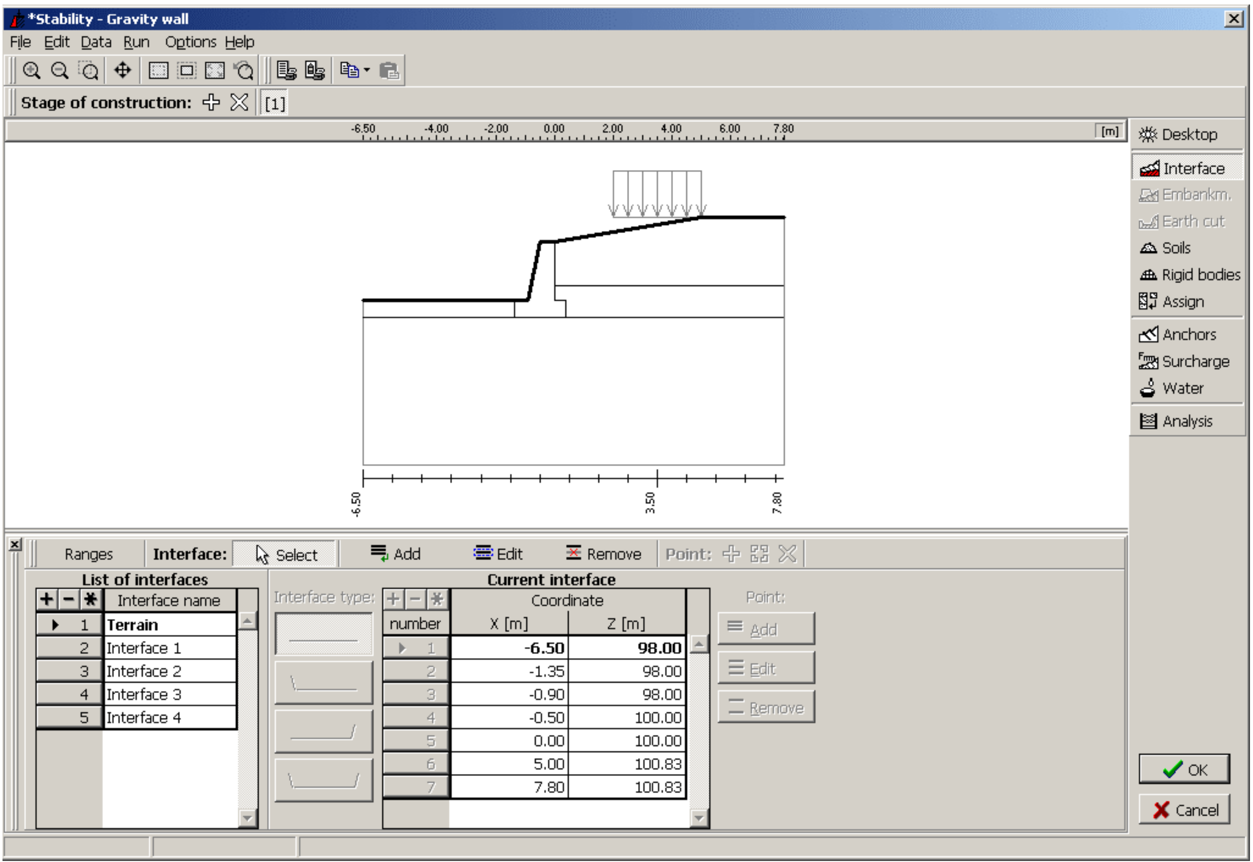


Fig. 3.10 Verification of external stability of a construction

3.7 Program „Gravity wall“

3.7.1 Analysis settings

The gravity wall is verified for the load due active earth pressure. The verification can be carried out according to the limit states or using the safety factors, respectively, depending on the input in the “**Settings**” dialog window (**Fig. 3.11**).

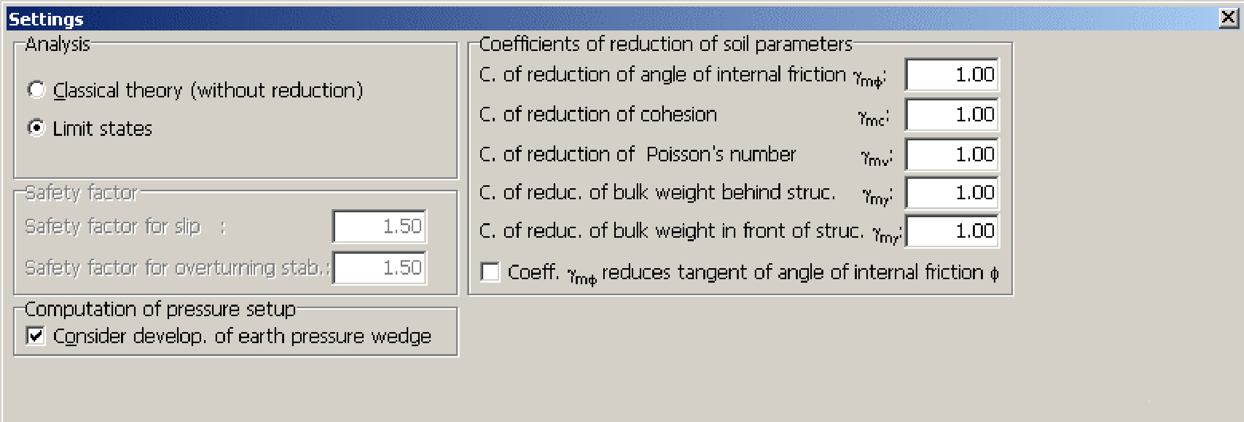


Fig. 3.11 Dialog window „Settings“

In this window, you may also turn on the switch „**Consider develop. of earth pressure wedge**“ (see **chapter 2.3**). Verification of the entire wall is discussed in **Chapter 3.5**.

3.7.2 Verification of the wall material

Verification of the wall stem design and construction joints:

Load on the wall stem is considered as an active pressure.
Verification of a cross-section is carried out according to various standards (EC, IS, PN, CSN.).
The dimensioning using the various standards are describes in the guide appendix.

Verification of the wall front jump:

The front jump of a wall is verified for the load due moment and shear force, which are given by

$$M = d_v^2 \sigma$$

$$Q = d_v \sigma$$

where

σ - maximum stress in construction joint
d_v - joint length.

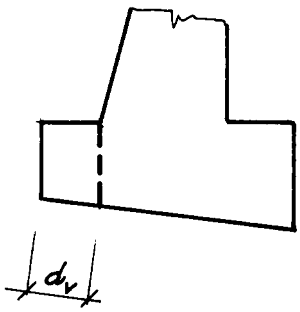


Fig. 3.12 Front jump of a wall

3.8 Program „Cantilever wall“

3.8.1 Analysis settings, verification

The cantilever wall is verified for the load due to active pressure (the wall can deform) or pressure at rest (deformation is constrained) depending on the input in the “**Settings**” dialog window. The scheme of development of earth-wedges and analysis of earth-pressures is explained in **Chapter 3.4**. The verification can be carried out either according to the limit states or using the factors of safety.

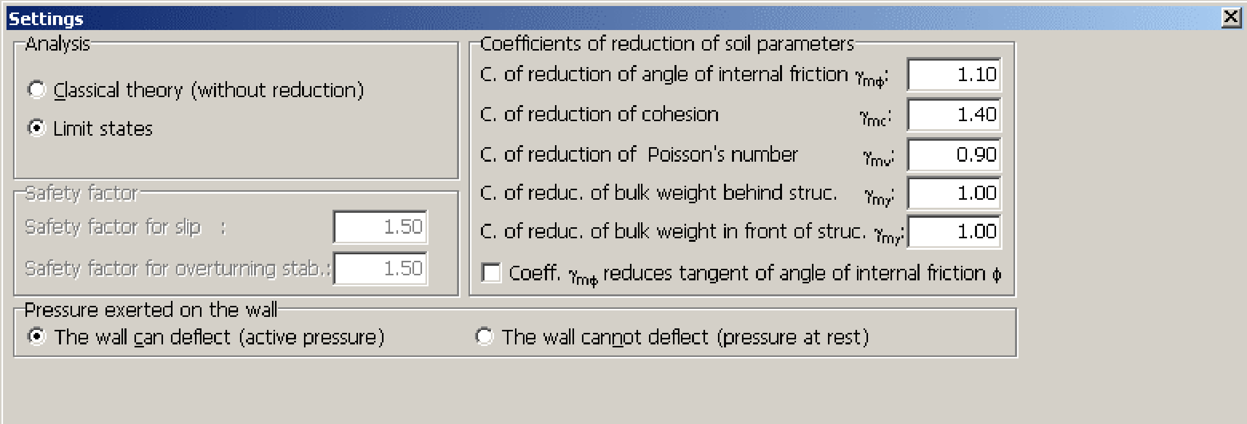


Fig. 3.13 Dialog window „Settings“

The back jump of a wall is accounted for such that an inclined foundation joint between the most front point of a foundation and front point of the jump is considered.

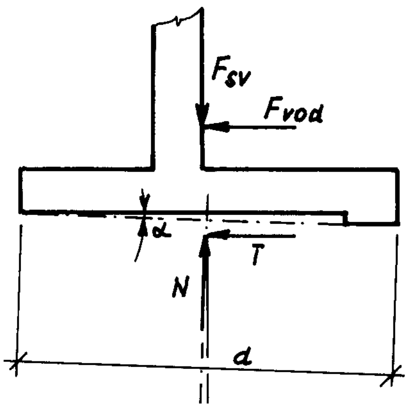


Fig. 3.14 Back jump of a wall

3.8.2 Dimensioning of reinforcement

Verification of the wall stem design and construction joints:

Load on the wall stem for reinforcement dimensioning is considered as a pressure at rest. The earth pressure at rest is computed including the reduction of input parameters of soil depending on the input in the „**Setting**“ dialog window.
Verification of a cross-section is carried out according to various standards (EC, IS, PN, CSN.).
The dimensioning using the various standards are describes in the guide appendix.

Verification of the wall front jump:

The front jump of a wall is verified for the load due moment given by

$$M = d_v^2 \sigma$$

where

σ - maximum stress in construction joint

d_v - joint length.

3.9 Program „Earth pressure“

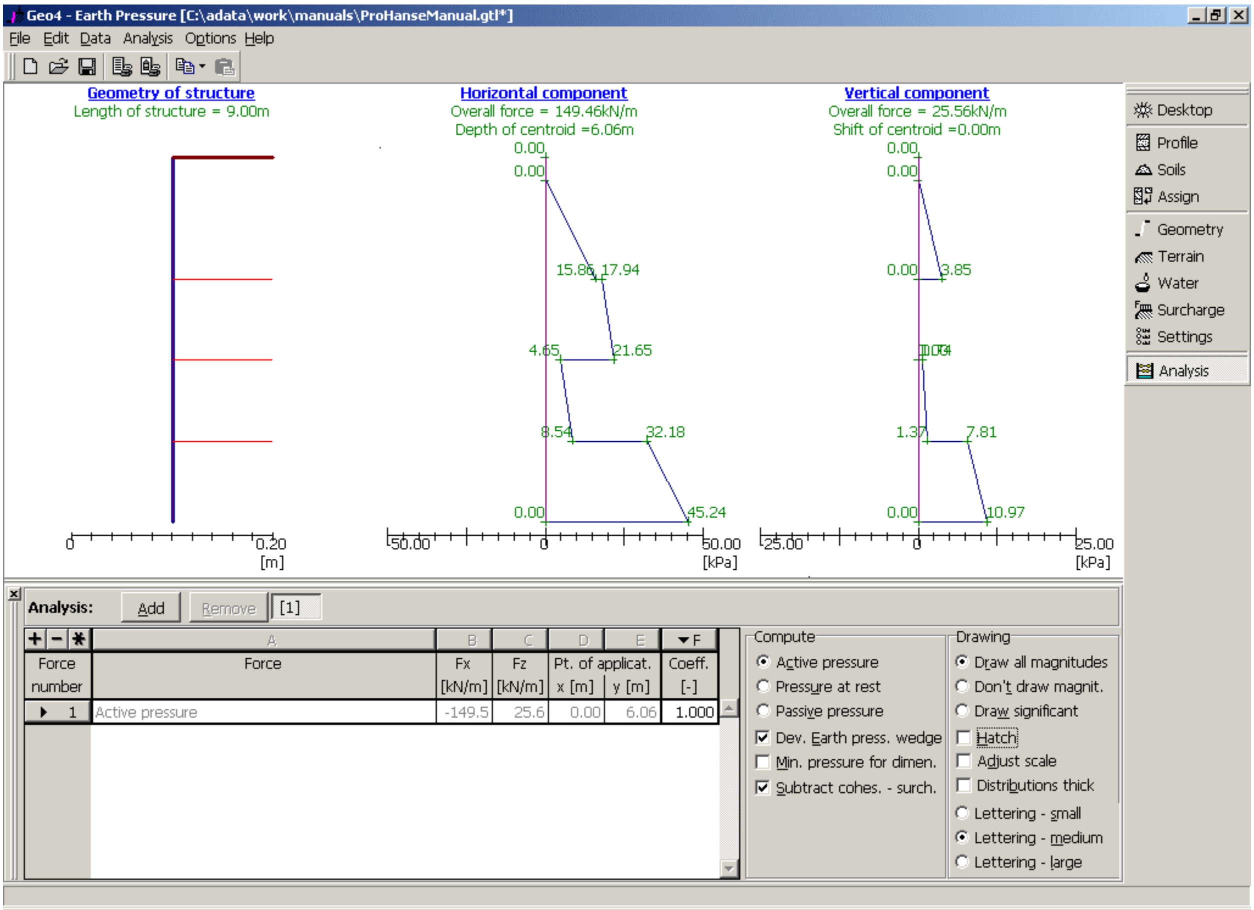


Fig. 3.15 „Analysis“ dialog window

Fig. 3.15 shows a typical window setup in the „**Earth pressure**“ program:
The computed pressure is plotted in the upper part, while the list of overall forces, their location and corresponding multipliers (see **Chapter 3.3**) are displayed in the lower part. Recall that computation of earth pressures is discussed in detail in **Chapter 2.** The only new switch is „**Subtract cohesion - surcharge**“ when working with active pressure. As default this parameter is turned on – cohesion is subtracted.

The process of computation of the overall active pressure is as follows:

- -The active pressure is computed (including negative values – tensile stresses in the soil are considered) and then multiplied by a given coefficient.
- -Pressures due to surcharges are computed and multiplied by corresponding coefficients.
- -All stresses are summed up.
- -The tension is excluded (zero points are inserted into the distribution and zero values are assigned to tensile values of the overall stress)

If the parameter „**Subtract cohesion - surcharge**“ is switched off, then the tension is excluded directly during the computation of active pressure. Stresses due to surcharges are then added.

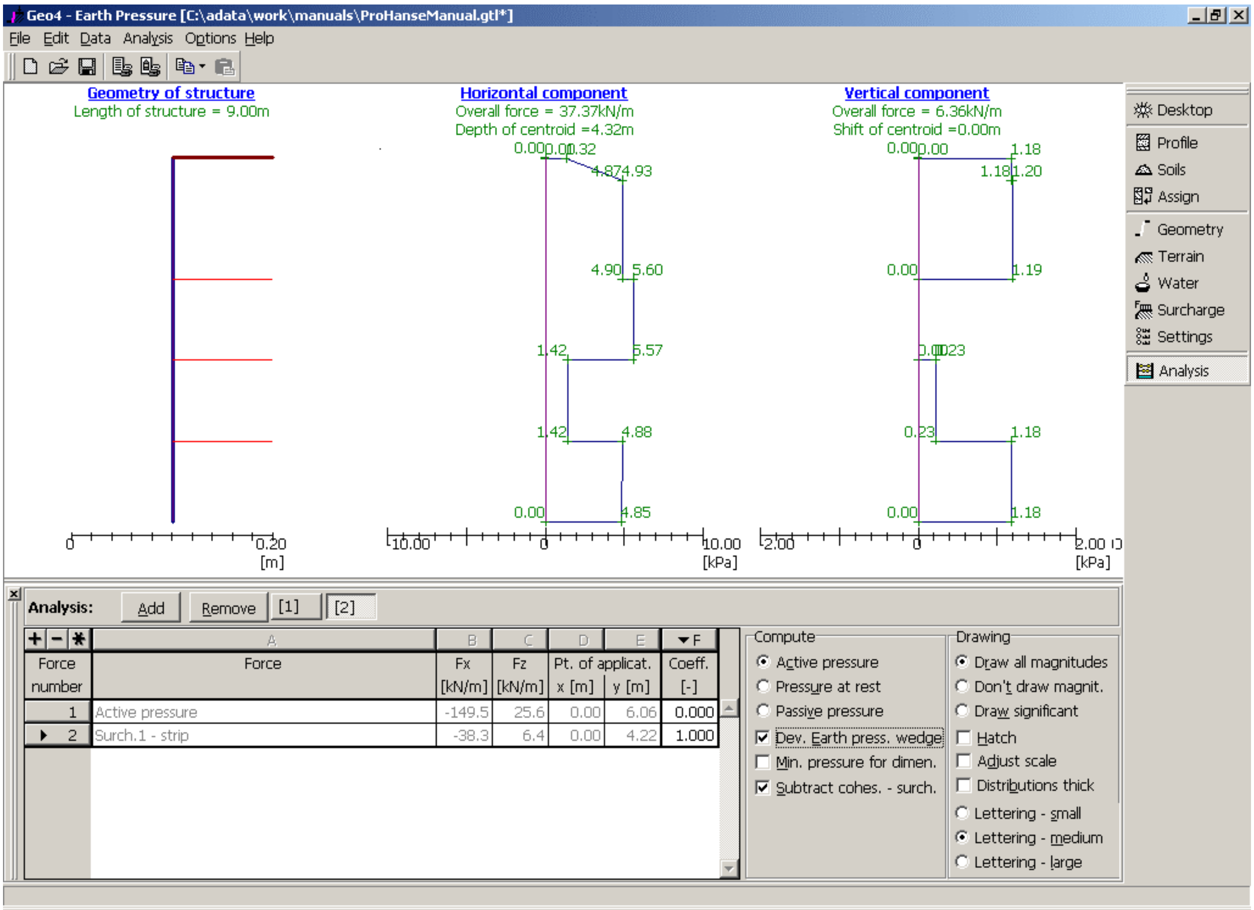


Fig. 3.16 Pressures due to surcharges only (without subtracting cohesion)

This is useful, particularly when the distribution due to a single surcharge is of interest.

Note that allowable values of input parameters might be exceeded during the analysis of individual pressures. In such a case, the program performs the computation using the largest allowable magnitude. A warning message „**MODIFIED**“ then appears in the output protocol in section partial results – pressures.

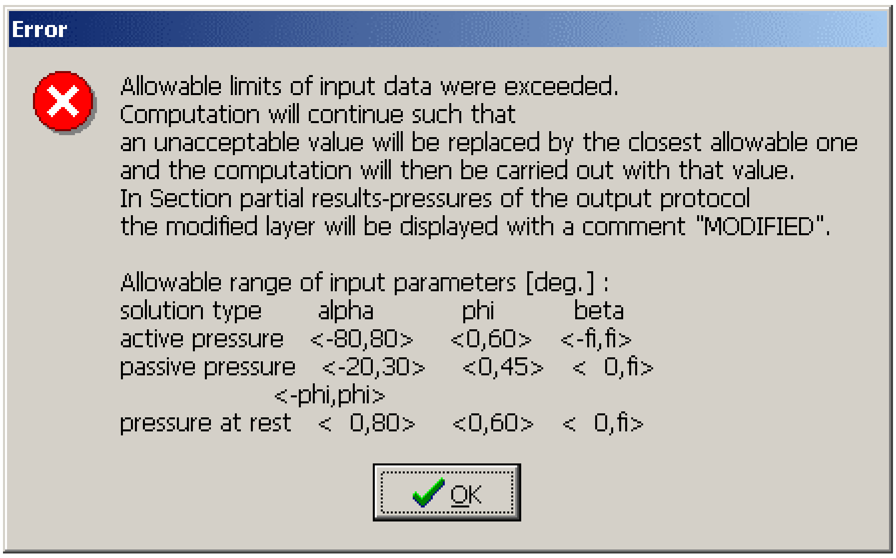


Fig. 3.17 Warning: allowable values of input parameters exceeded

3.10 Program „Nailed slopes"

The program „**Nailed slopes**" is build upon the well-known engineering approaches, thus no standards are followed. When analyzing nailed slopes, one should realize that this approach is suitable for cohesive soils with cohesion $c>10kPa$. Crucial problem of the verification of nailed slopes is the determination of bearing capacity of a single nail, which must be passed into the program as an input item.

The nail bearing capacity, determined e.g. experimentally, is given by the minimum value of:

- Force at the breaking limit,
- Force at the limit of pulling the nail out of the soil,
- Force at the limit of tearing the nail head out of its anchorage.

The nail bearing capacity against pull-out is computed by the program (details are given in **Section 3.10.3 „Verification of the nail bearing capacity**" – this value, however, is only approximate as it is computed for the driven in nails and should be used with caution. The actual bearing capacity is usually significantly higher and it is the user responsibility to select and assign an appropriate value. The computed bearing capacity should serve more or less for comparison only and to check expected values.

3.10.1 Determination of shape of fictitious wall, verification of the wall

To verify an external stability you first create a fictitious structure – a wall, which is then subject to verification. The wall is formed by a front face and by a line connecting ends of individual nails. A vertical line is drawn from the first nail up to the wall depth, so that the lower edge of the fictitious wall is always horizontal.
The structure is assumed to be loaded by an active pressure. Two options are available to run the analysis depending on the selection in the „**Settings**" window. The limit state analysis or the analysis based on the factor of safety is applicable. Details are given in **Section 3.5**.

3.10.2 Verification of internal stability

Two types of slip surfaces are used to verify the internal stability:

a) Straight-line type of slip surface (**Fig. 3.18**)

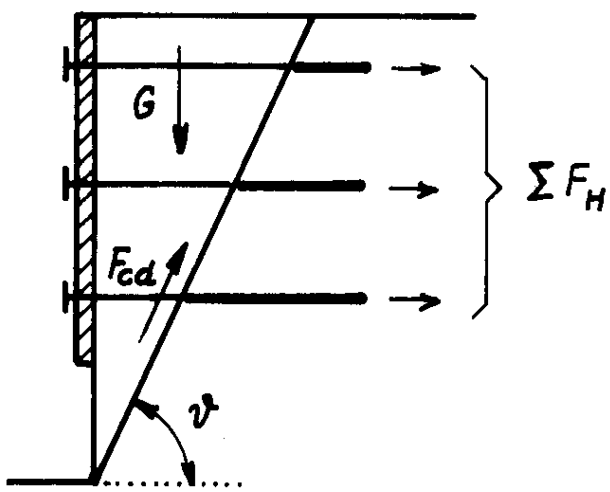


Fig. 3.18 Straight-line type of slip surface

b) Polygonal slip surface (**Fig. 3.19**)

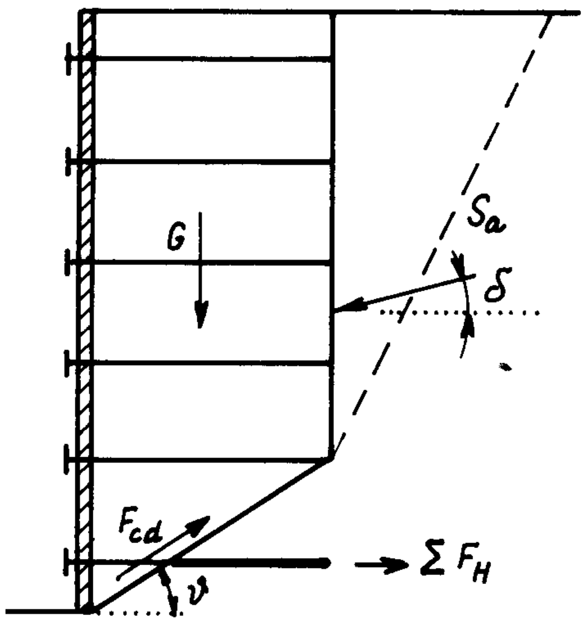


Fig. 3.19 Polygonal slip surface

In either case the stability analysis is carried out assuming variation of angle υ.

During optimization, the stability analysis is performed for all segments with variation of slip surface angle υ from *1* to *89* degrees with *1*-degree step.
Verification of internal stability is performed using the factor of safety – **input parameters are not reduced** in the computational run. Therefore, the analysis of internal stability does not depend on input of parameters in the "**Analysis setting**" dialog window. The allowable safety factors can be assigned in the "**Internal stability**" dialog window.
The verification analysis checks the condition that ratio of resisting and active forces on the slip surface does not exceed the respective factor of safety.

Active forces:

- Component of the gravity force in the direction of slip surface
- In case of polygonal slip surface also a component of active earth pressure acting on the vertical part of a structure in the direction of slip surface (pressure derived without reduction of input parameters).

Resisting forces:

- Friction and cohesion of soil on slip surface,
- Resultant force transmitted by nails.

Magnitude of a force transmitted by a single nail is found as length of a part of nail behind the slip surface multiplied by the nail bearing capacity ad divided by the overall length of a given nail.

The factor of safety thus follows from:

$$SF = \frac{F_h \cos(\upsilon + \alpha) + F_{cd}}{(G + S_{a,sv})\sin(\upsilon) + S_{a,vod}\cos(\upsilon)}$$

where:

$$F_h = \sum F_{h,n}$$

$$F_{cd} = \sum \frac{d_i}{d}((G\cos(\upsilon) + F_h \sin(\upsilon + \alpha))\tan\phi_i + \sum d_i c_i$$

where:

G	- gravity force,
$S_{a,sv}$	- vertical component of active pressure,
$S_{a,vod}$	- horizontal component of active pressure,
D_i	- length of the i-th part of slip surface,
D	- length of slip surface,
$F_{h,n}$	- bearing capacity of the n-th nail behind slip surface per one meter,
C_i	- cohesion of the i-th soil layer,
φ_i	- coefficient of internal friction of the i-th soil layer,
υ	- inclination of slip surface,
α	- inclination of nails from horizontal.

3.10.3 Verification of bearing capacity of nails

Determination of forces in individual nails is performed by assigning the computed earth pressure to individual soil layers. Each nail transmits only a portion of active pressure that acts within a given layer. The magnitude of active pressure is lowered by coefficient $K_n=0,85$ derived experimentally.

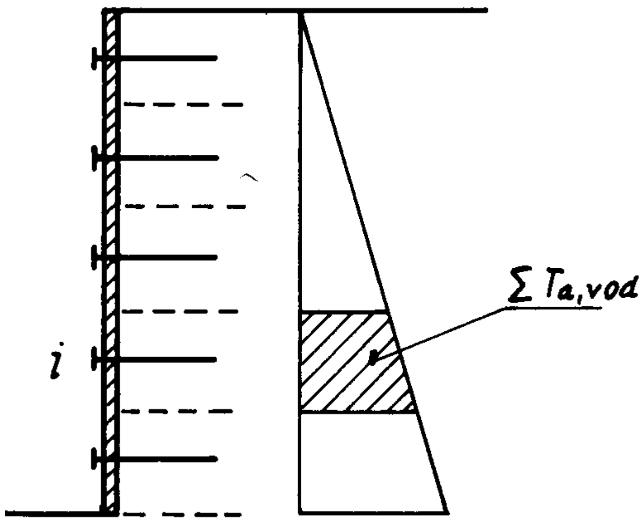


Fig. 3.20 Computation of forces corresponding to a single nail

$$F_i = \frac{b \sum T_{a,vod}}{\cos\alpha}$$

where:

B - spacing of nails,
α - inclination of nails.

The output window also lists approximate values bearing capacity of driven in nails against pullout (skin friction).

$$F_u = 2\pi f l \frac{1}{2(1-\upsilon)} \sigma_z \tan\varphi$$

where:

L - nail length,
R - nail radius,
υ - Poisson number of surrounding soil,
φ - angle of internal friction of surrounding soil,
σ - geostatic pressure at a depth of nail.

Recall that the actual bearing capacity is usually significantly higher. The computed bearing capacity should serve for comparison only and to check expected values.

3.10.4 Dimensioning of concrete cover

Dimensioning of a concrete cover of nailed slope is carried out with respect active pressure. For the design purposes it is assumed that the structure is subdivided into individual dimensioning strips separated by supports. The nail head is modeled as a smooth pin and joint between layers as an internal hinge. Default setup assumes structure with five supports loaded by uniform distribution of active pressure found at a depth of nail head, **Fig. 3.21**.

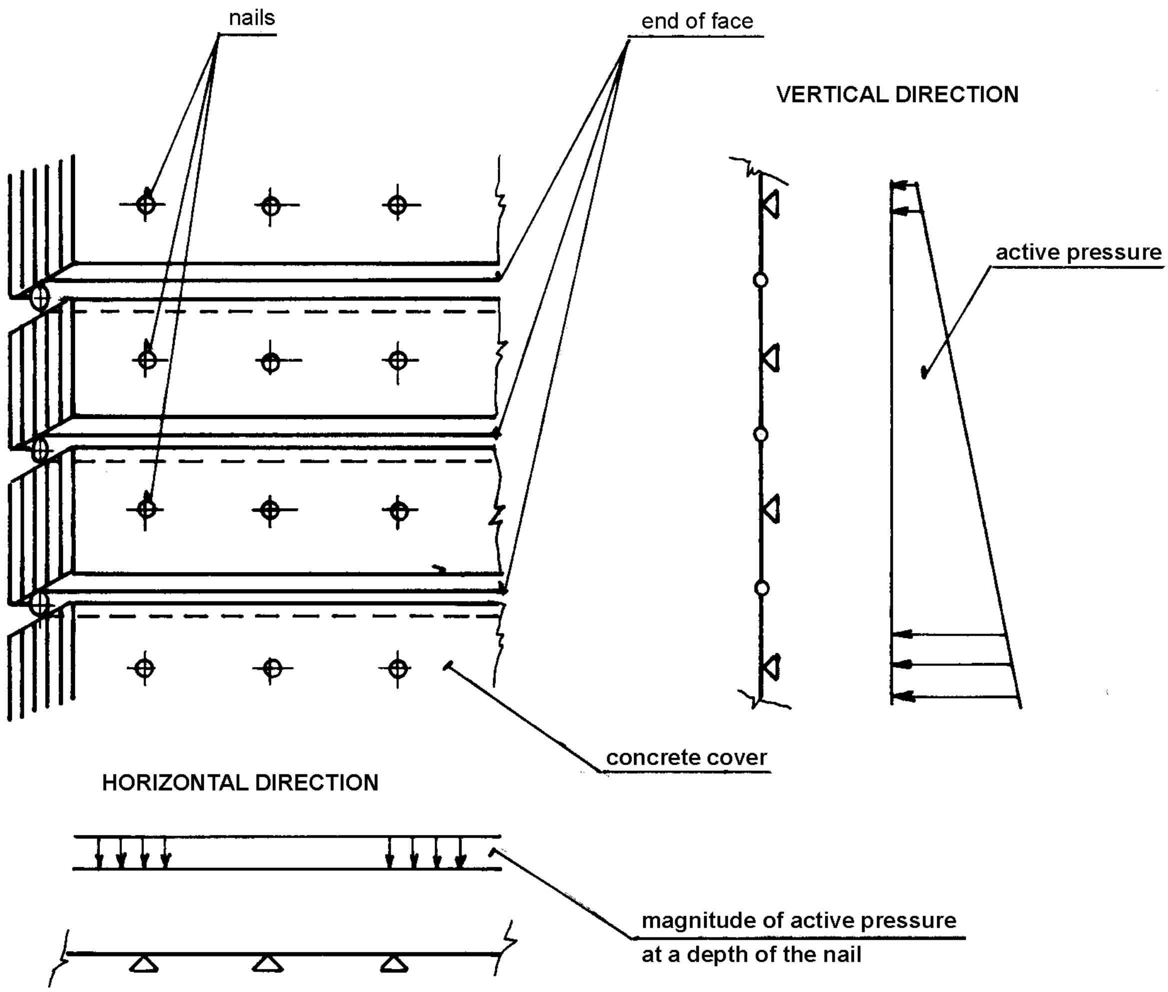


Fig. 3.21 Dimensioning of concrete cover

Verification of the cross-sections is carried out according to various standards (EC, IS, PN, CSN.). The dimensioning using the various standards are describes in the guide appendix.

3.11 Program „Gabion"

3.11.1 Verification of the whole structure

Verification of gabion constructions can be performed either according to the limit states or using the safety factors depending on the input in the "**Settings**" dialog window (**Fig. 3.22**).

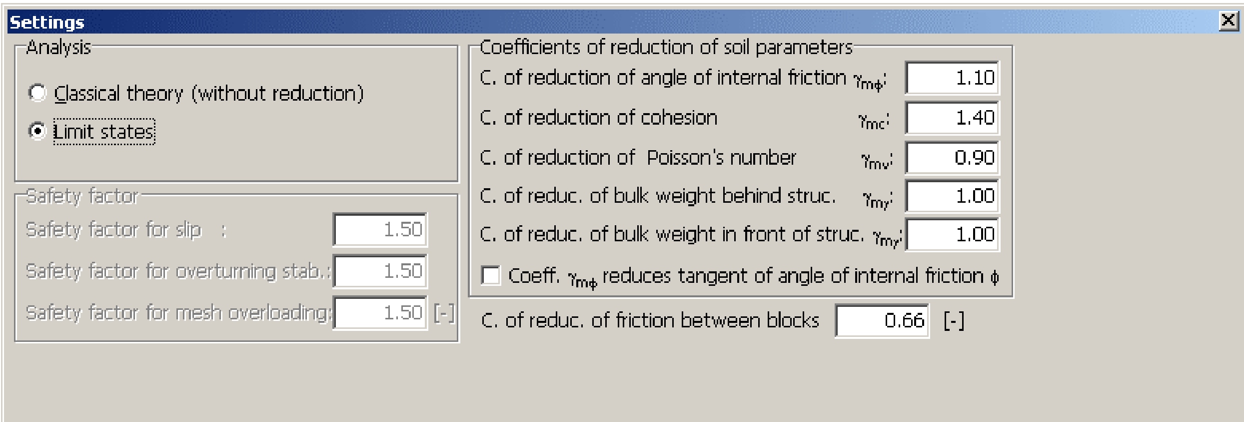


Fig. 3.22 Dialog window „Settings"

Verification analysis is carried out for the load due to active earth pressure (see **Chapter 3.5**). Providing the input assumes overhangs of meshes behind the blocks, the analysis further accounts for anchor forces as shown in **Fig. 3.23**.

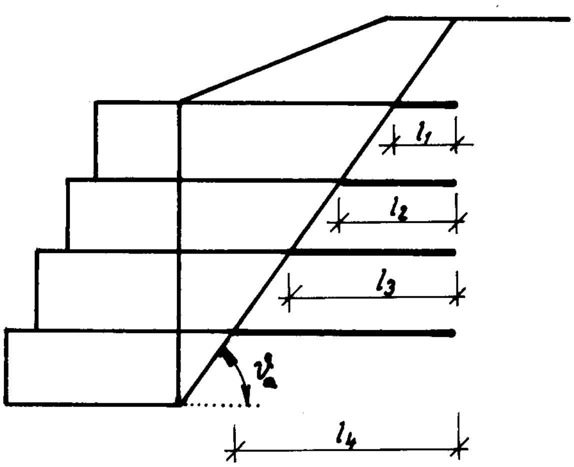


Fig. 3.23 Overhangs of meshes

Computation of the angle υ_a is described in **Chapter 3.4**.

Forces due to anchorage assume the form

$$F_i = l_i F_{1i}$$

Where

l_i -mesh length behind the slip surface [*m*]

F_l -bearing capacity of 1 unit meter of mesh against pullout out of the soil [kN/m^2] (inputted separately for each block in the „**Geometry**“ dialog window – input entry „**Mesh bearing capacity**“)

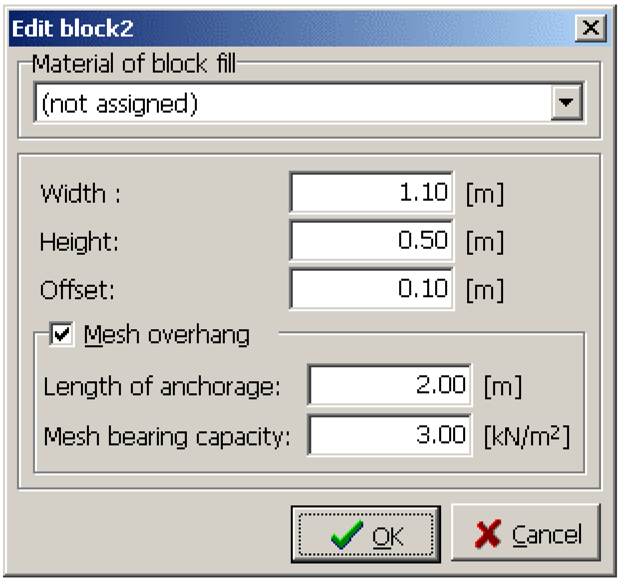


Fig. 3.24 Dialog window „Geometry - edit block”

The program also makes sure that the force F_i will not exceed the allowable strength of mesh (default value is *40 kN/m*). For input, use the “**Material**“ dialog window, **Fig. 3.25**.

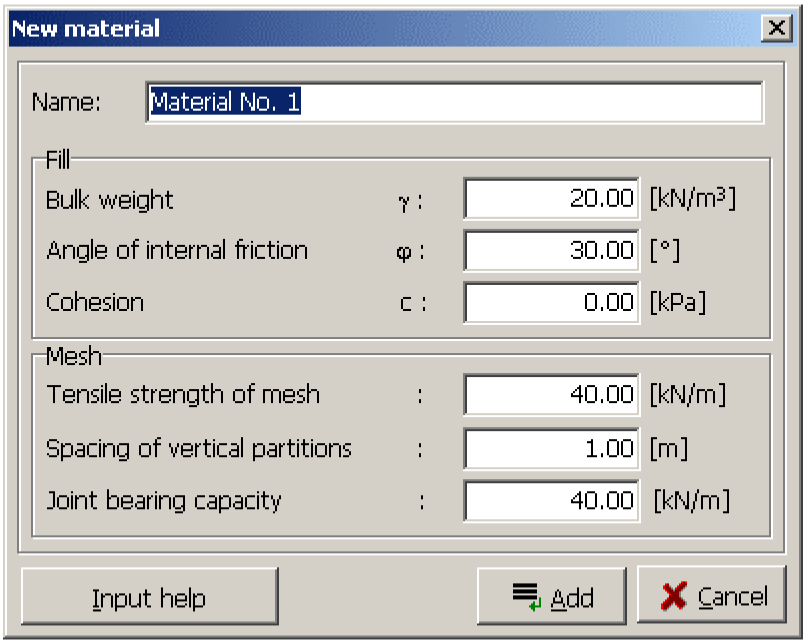


Fig. 3.25 Dialog window „Material“

The program enables the user to analyzed gabions with simple and double meshes between blocks. In case of the double mesh (**Fig. 3.26**), the tensile strength of mesh is assumed twice as big as for the simple mesh (the „**Edit material**“ dialog window).

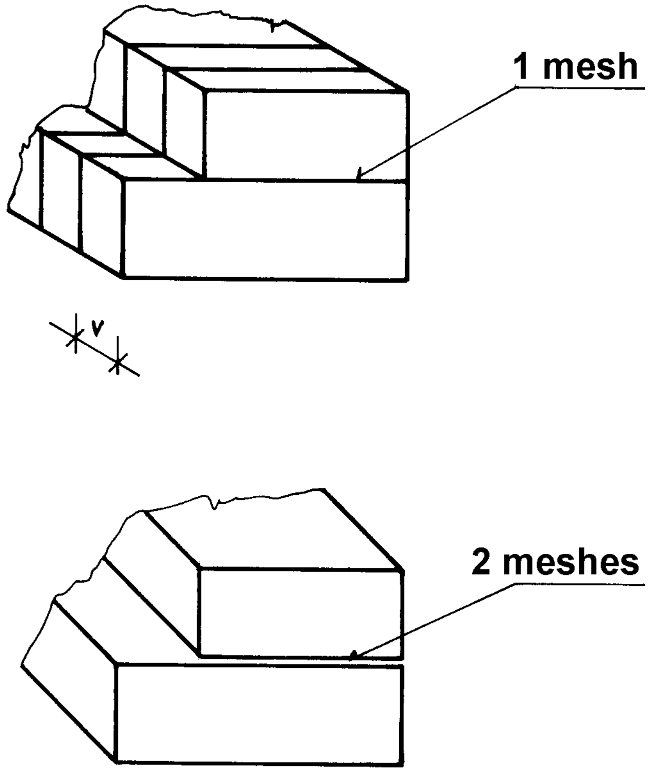


Fig. 3.26 Geometry of gabions

3.11.2 Internal stability of gabion wall

Verification of joints between individual blocks is performed in the „**Dimensioning**" dialog window. The structure above the block is loaded by active pressure and corresponding forces are determined in the same way as for the verification of the entire wall (**Chapters 3.5. a 3.11.1**). A loose filling is used in the analysis – not placed rockfill – but its effect can be simulated using a very high angle of internal friction. It can be assumed that after some time due to keying action of filling aggregate the stress in meshes will drop down. Individual sections of the gabion wall are checked for the maximum normal and shear stress. With the help of these variables it is possible to modify the slope of structure face by creating terraces or by increasing the slope of face of wall (α).

Loading applied to the bottom block is depicted in **Fig. 3.27**:

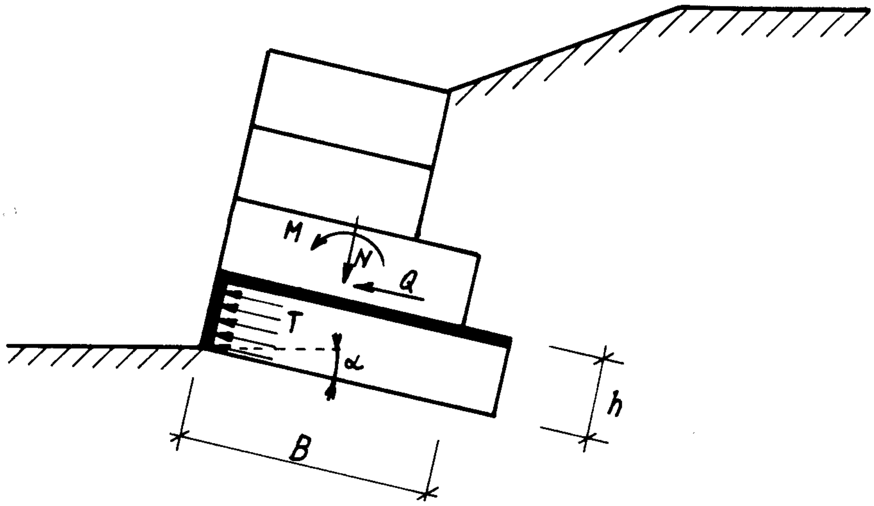


Fig. 3.27 Loading of the bottom block

Normal stress in the block center is given by

$$\sigma = \frac{N}{B-2e} + \frac{\gamma h \cos\alpha}{2}$$

$$e = \frac{M}{N}$$

Pressure acting upon the wall of bottom block is determined an increased active pressure:

$$T = 0.5T_r + 0{,}5T_a$$

$$T_r = \sigma K_r$$

$$T_a = \sigma K_a - 2c_d\sqrt{K_a}$$

$$K_r = 1 - \sin\varphi_d$$

$$K_a = tg^2(45 - \frac{\varphi_d}{2})$$

where

- φ_d - design angle of internal friction of material of the bottom block [°],
- c_d - design cohesion of material of the bottom block [*kPa*],
- γ - bulk weight of material of the bottom block [kN/m^3],
- h - height of the bottom block [*m*],
- B - width of the bottom block [*m*],
- α - slope of gabion [°],
- T - an average value of pressure acting upon the face of bottom block [*kPa*],
- σ - maximum normal stress acting upon the bottom block [*kPa*].

Widths of meshes of the bottom block per 1 unit meter of the gabion wall are

$$D_{hor} = 1$$

$$D_{all} = \frac{h}{v} + 1$$

where

D_{upp}	- length of upper mesh between blocks loaded in tension [m],
D_{all}	- overall length of mesh loaded in tension T [m],
v	- distance between vertical meshes [m],
h	- height of the bottom block [m].

Verification – limit states

Reduced parameters of gabion material set depending on coefficients set in the „**Settings**" dialog window are used in the verification analysis.

a) Check for overturning stability:

$$M_{ovr} < M_{res}$$

b) Check for slip:

$$N.tg\varphi_d + c_d B < Q$$

c) Check for bearing capacity with respect to the lateral pressure

$$S < S_u$$

$$S = \frac{Tbh}{D_{all}}$$

where

T - an average value of pressure acting upon the face of bottom block [*kPa*],
S - force per 1 unit meter pf joint [*kN/m*],
S_u - joint bearing capacity [*kN/m*] (for input use the "**Material**" dialog window),
b - width = 1 unit meter.

d) Check for bearing capacity of joint between blocks

$$N_d < N_u$$

$$N_d = \frac{Tbh}{D_{all}} + \frac{Max(0, Q - Q_{tr})}{D_{hor}}$$

$$Q_{tr} = k_t N tg\varphi_d + c_d B$$

where

N_d - tensile force per 1 unit meter of upper joint of the bottom block [*kN/m*],
N_u - strength of mesh [*kN/m*] (for input use the "**Material**" dialog window),
Q_{tr} - shear force transmitted by friction and cohesion between blocks [*kN/m*],
K_t - coefficient of reduction of friction between blocks (for input use the „**Setting**" dialog window – default value is *0,66*).

Verification – safety factor

a) Check for overturning stability:

$$\frac{M_{vzd}}{M_{kl}} < FS_{over.}$$

b) Check for slip:

$$\frac{Ntg\varphi + cB}{Q} < FS_{slip}$$

c) Check for bearing capacity with respect to the lateral pressure

$$\frac{}{S} < FS_{sit}$$

$$S = \frac{Tbh}{D_{all}}$$

where

T - an average value of pressure acting upon the face of bottom block [*kPa*],
S - force per 1 unit meter pf joint [*kN/m*],
S_u - joint bearing capacity [*kN/m*] (for input use the "**Material**" dialog window),
FS_{sit} - factor of safety of mesh stressing (for input use the „**Setting**" dialog window – default value is *1.5*),
b - width = 1 unit meter.

d) Check for bearing capacity of joint between blocks

$$\frac{_{u}}{N} < FS_{sit}$$

$$N_d = \frac{Tbh}{D_{all}} + \frac{Max(0, Q - Q_{tr})}{D_{hor}}$$

$$Q_{tr} = k_t Ntg\varphi + cB$$

where

N_d - tensile force per 1 unit meter of upper joint [*kN/m*],
N_u *e* - strength of mesh [*kN/m*] (for input use the "**Material**" dialog window),
SFsi - factor of safety of mesh stressing (for input use the „Setting" dialog window – default value is 1.5),
Qtr - shear force transmitted by friction and cohesion between blocks [kN/m],
Kt - coefficient of reduction of friction between blocks (for input use the „Setting" dialog window – default value is 0,66).

3.12 Program „Abutment“

The program „**Abutment**“ serves to design and verify bridge abutments. It allows stability analysis and verification of foundation soil according to the limit states or using the factor of safety. The critical sections of the bridge abutment (made from steel reinforced concrete as well as from plain concrete) can be verified according to various standards.

3.12.1 Computation of forces acting upon an abutment

An abutment is verified per 1 unit meter of its length. All forces entering the analysis are derived as follows:

- **-Abutment self weight** is computed from the specified abutment transverse section per 1 unit meter.
- **-Reactions inserted by bridge and transitional plate** are inputted in [kN] using the values for the whole abutment, these values are in the analysis divided by the **abutment length,**
- **-Soil pressure** is determined per 1 unit meter and then reduced by the ratio **length of load due to soil / abutment length,**
- -Weight of soil wedges is determined per 1 unit meter and then reduced by the ratio length of load due to soil / abutment length,
- -**Surcharge** is determined per 1 unit meter and then reduced by the ratio **length of load due to soil / abutment length,**
- **-Inputted forces, front face resistance** are assumed per 1 unit meter and are not reduced in any way,
- **-Wingwalls** – the self-weight of wing walls is computed from their geometry, for the stem design and verification of the foundation the value of self-weight is divided by the **abutment length** (it is the user responsibility to either include or exclude the effect of wing walls in from the analysis).

The above forces, having units [*kN/m*], are displayed in the „**Verification**“ and „**Dimensioning**“ dialog windows.

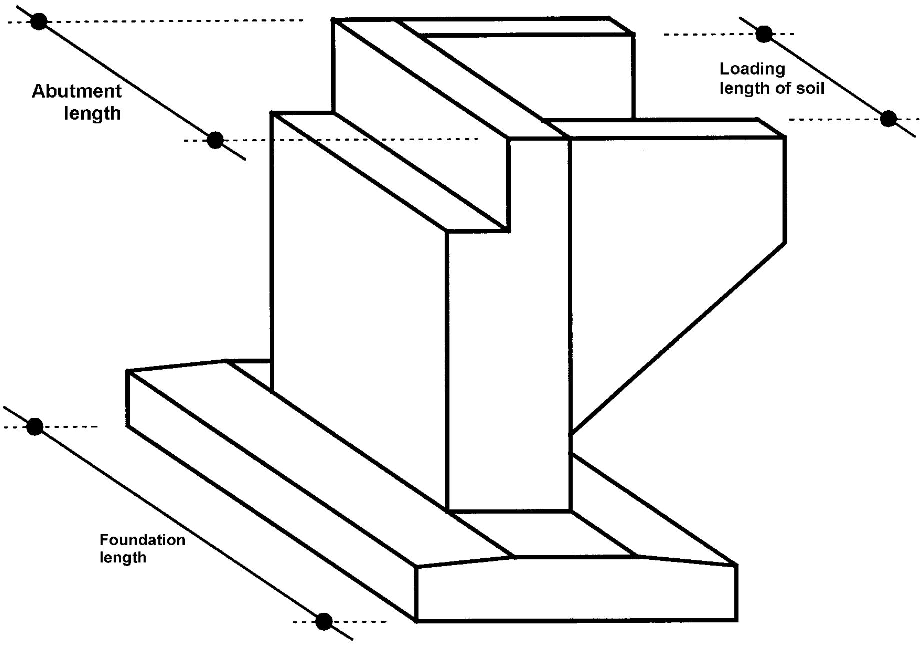


Fig. 3.28 Geometry of bridge abutment

All forces acting at the foundation joint, which enter the verification analysis (overall verification of an abutment), are reduced by the ratio **abutment length / foundation length**.

In case of prolonged wing walls it is necessary to put in the prolongation of foundation under the wing walls. Such jumps of foundations are reflected in the analysis by computing a fictitious width of the foundation

$$d_{fict} = \frac{A_{all}}{s}$$

Where

A_{all} - overall area of foundation including all jumps,
s - length of abutment foundation,
d_{fict} - fictitious width of foundation for verification analysis

The foundation is then considered as rectangular, which is simplified but rather conservative assumption.

3.12.2 Load cases in analyses

When performing the verification or dimensioning analysis it is possible to choose from three load cases depending on gradual construction of a bridge abutment.

1.LC – construction state, abutment without closure wall

This load case is always analyzed for the load due to active earth pressure. Bridge and transitional plate reactions are not considered. The flat terrain is assumed behind the structure. Only surcharges acting in the 1st LC are considered.

2.LC – service state, without live short-time loading

This load case is analyzed for the load due to either active earth pressure or pressure at rest depending on the input in the "**Setting**" dialog window. Bridge reactions are derived without accounting for live short-time loading; as for the reaction exerted by transitional plate only the vertical component is considered. Only surcharges acting in the 2nd LC are considered.

3.LC – service state, overall loading

This load case is analyzed for the load due to either active earth pressure or pressure at rest depending on the input in the "**Setting**" dialog window. The overall reaction exerted by bridge is considered (permanent + live long-time + live short-time loading); as for the reaction exerted by transitional plate both the vertical and horizontal components are considered. Only surcharges acting in the 3rd LC are considered. For certain cases the user is allowed to create special load cases – e.g., an abutment without backfill can be modeled by inputting a layer of soil having $\gamma=0$ kN/m^3.

Note:

Reaction of bridge and transitional plate – providing the analysis is carried out according to the limit states (overall verification), the forces are multiplied by the design coefficients inputted in the "**Loading**" dialog window.

3.12.3 Abutment verification

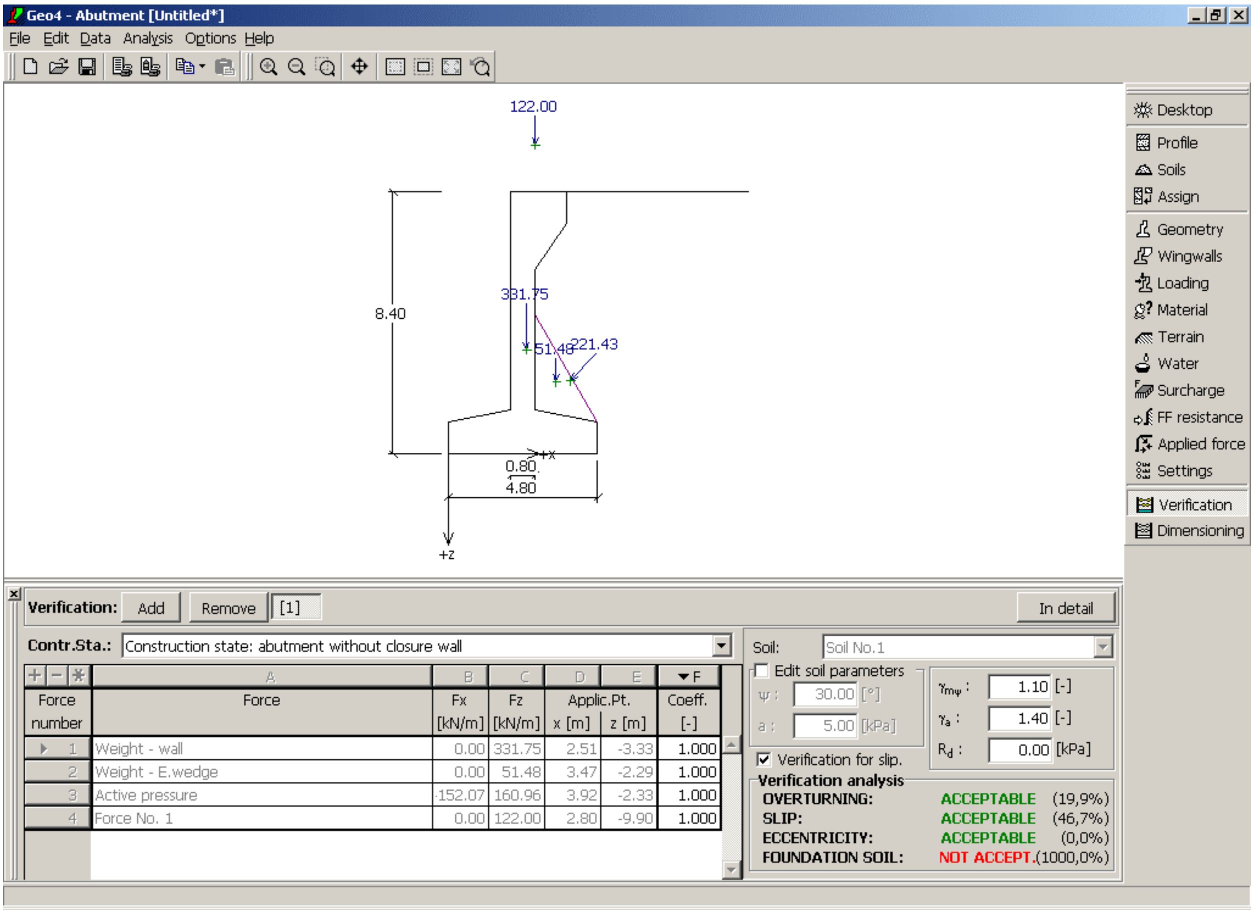


Fig. 3.29 Abutment verification

Verification of the entire structure is analogous to the verification of walls. Details are provided in **Chapter 3.5** – here we present only some differences.

In this dialog window you select the load case – the construction load case is always analyzed for the load due to active earth pressure, while the service state depends on the input in the "Setting" dialog window (either pressure at rest or active pressure is considered).

The verification analysis can be performed either according to the limit states or using the factor of safety depending on the input in the "**Setting**" dialog window. Providing the analysis according to the limit states is selected, the bridge and transitional plate reactions are multiplied by the design coefficients inputted in the "**Loading**" dialog window.

The "**Verification**" dialog window contains also the switch "**Verification for slip**". If the displacement is constrained (e.g., foundation on piles), this type of analysis can be switched off.

3.12.4 Dimensioning of critical joints of abutment

The verification analysis assumes either loading due to active pressure (construction LC) or due to pressure at rest or active pressure, respectively, depending on the input in the "**Setting**" dialog window. Pressures are considered without reduction of input parameters. Bridge and transitional plate reactions are assumed as standard (without multiplying by the design coefficients).

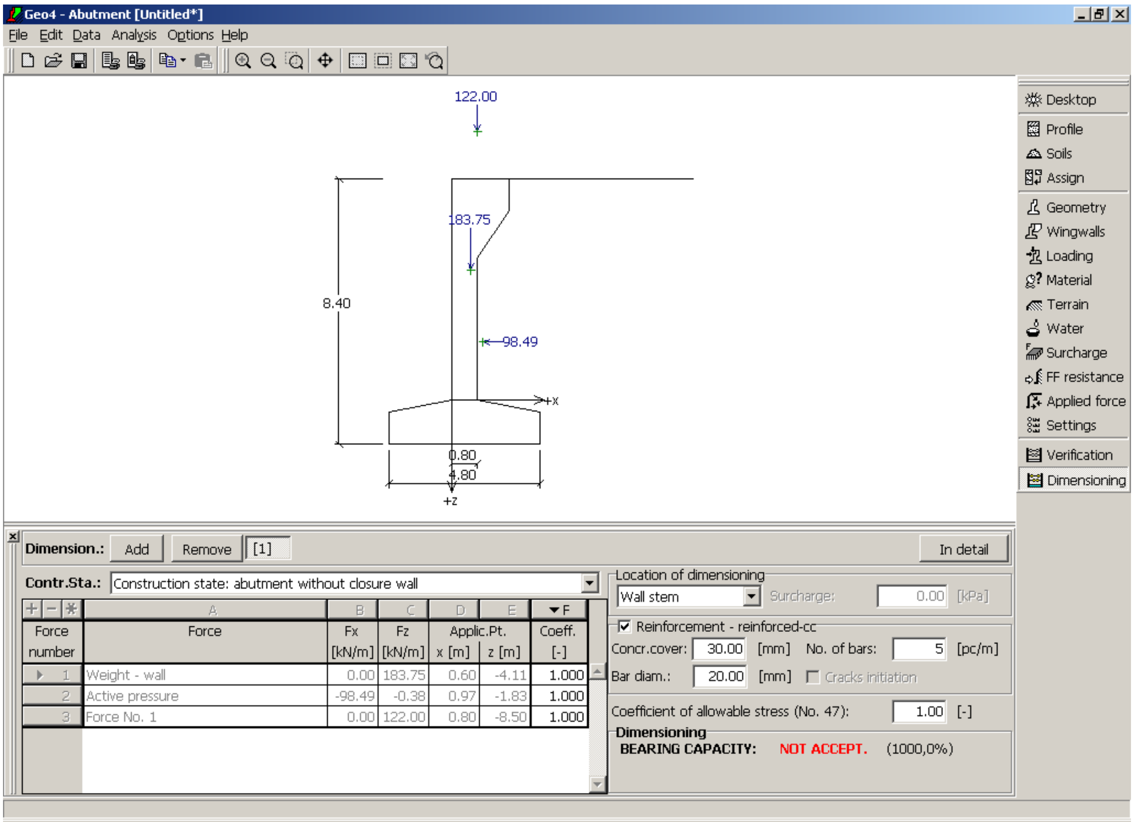


Fig. 3.30 Abutment dimensioning

The program allows verification of cross-sections made from plain concrete or cross-sections reinforced on one side.

The „**Dimensioning**" dialog window serves to enter individual joints, a specific load case and reinforcements parameters.

The program allows verification of the following joints:

1)Abutment stem – foundation

The cross-section can be made either from plain or reinforced concrete. The joint is verified for the load due to normal force and bending moment.

2)Construction joint is verified as above.

3)Closure wall – bearing block

The cross-section is verified for the load due to normal force and bending moment. The reinforced concrete cross-section is always assumed.

4)Wing wall – abutment
The cross-section wing wall-abutment is verified for the load due to bending moment. The wing wall is loaded by active pressure of soil behind abutment. In the dialog window the user is allowed to enter a surface load to determine the pressure acting on wing wall. In such a case a surcharge, which might be introduced in the "**Surcharge**" dialog window, in not considered and the terrain behind abutment is flat. The overall moment loading the joint is found by multiplying the magnitude of pressure acting upon abutment and by the difference centers of pressure resultant and joint.
The default length of cross-section for dimensioning is taken equal to the wing wall height. The program, however, enables to change this value in the "**Wing walls**" dialog window. A uniform distribution of moment along the joint is assumed.
The cross section can be made either from plain or reinforced concrete.

5)Front jump of abutment cross-section
The jump is analyzed as a cantilever beam loaded by the reaction (stress) due to foundation soil. The joint can be made either from plain or reinforced concrete. The program checks for the allowable stress developed in concrete and steel and for the loading of concrete section in compression. As for reinforced concrete, the program also checks the degree of reinforcement.

Verification of the cross-sections is carried out according to various standards (EC, IS, PN, CSN.). The dimensioning using the various standards are describes in the guide appendix.