

OPTUM COMPUTATIONAL ENGINEERING

Optum^{G2}

ANALYSIS



Version 2016

Contents

1 GENERAL INFORMATION	5
2 MESH	7
Settings	7
3 SEEPAGE	8
Settings	8
Mesh	8
3.1 Notes	8
4 INITIAL STRESSES	9
Settings	10
4.1 Choice of earth pressure coefficient	10
5 ELASTIC	11
Settings	11
Mesh	12
5.1 Notes	12
Initial Stresses	12
6 LIMIT ANALYSIS	13
Settings	13
Mesh	14
Safety	15
6.1 Notes	16
Initial Stresses	16
7 STRENGTH REDUCTION	17
Settings	18
Mesh, Safety	18
7.1 Notes	19
Initial Stresses	19
8 ELASTOPLASTIC	20
Settings	20
Mesh	20
8.1 Notes	21
Initial Stresses	21
9 MULTIPLIER ELASTOPLASTIC	22
Settings	22
Load Stepping	23
Mesh	23
Advanced Settings	24

10 CONSOLIDATION	26
Settings	26
Time Stepping	26
Advanced	26

1 GENERAL INFORMATION

Optum G2 is a finite element program for strength and deformation analysis of geotechnical boundary value problems. It has a number of overall principles in common with other programs available on the market, but also differs fundamentally on a number of points. For example, it is possible to compute limit loads, or bearing capacities, directly without having to perform a traditional step-by-step elastoplastic analysis. Moreover, it is possible to compute rigorous upper and lower bounds to the limit load, thus bracketing the exact solution to a usually rather narrow interval. Conversely, given a set of fixed loads, it is possible to compute upper and lower bounds to the exact strength reduction factor, i.e. the factor by which the material strengths need to be reduced in order to cause collapse. Besides these unique features, Optum G2 also allows for more traditional analyses including step-by-step elastoplasticity, seepage, and consolidation.

The type of analysis to be conducted for a given stage is selected via the Stage Manager (see Figure 1.1). The Stage Manager window is divided horizontally into two parts. The upper part contains three columns. The first column contains the stage name, which can be modified as desired. The second column contains a drop-down menu where the From stage can be selected. The From stage precedes the current stage and all state variables (stresses, displacements, etc) are transferred to the current stage as the initial state. Changes to geometry, loading, materials, etc are identified automatically and no other provisions than selecting the relevant From stage need to be taken in order to link stages. Finally, the last column contains a drop-down menu for selection of the analysis for the current stage.

The lower half of the Stage Manager window contains the various options and settings relevant to the type of analysis that is being conducted. These are particular to a given stage and may differ from stage to stage for the same analysis type.

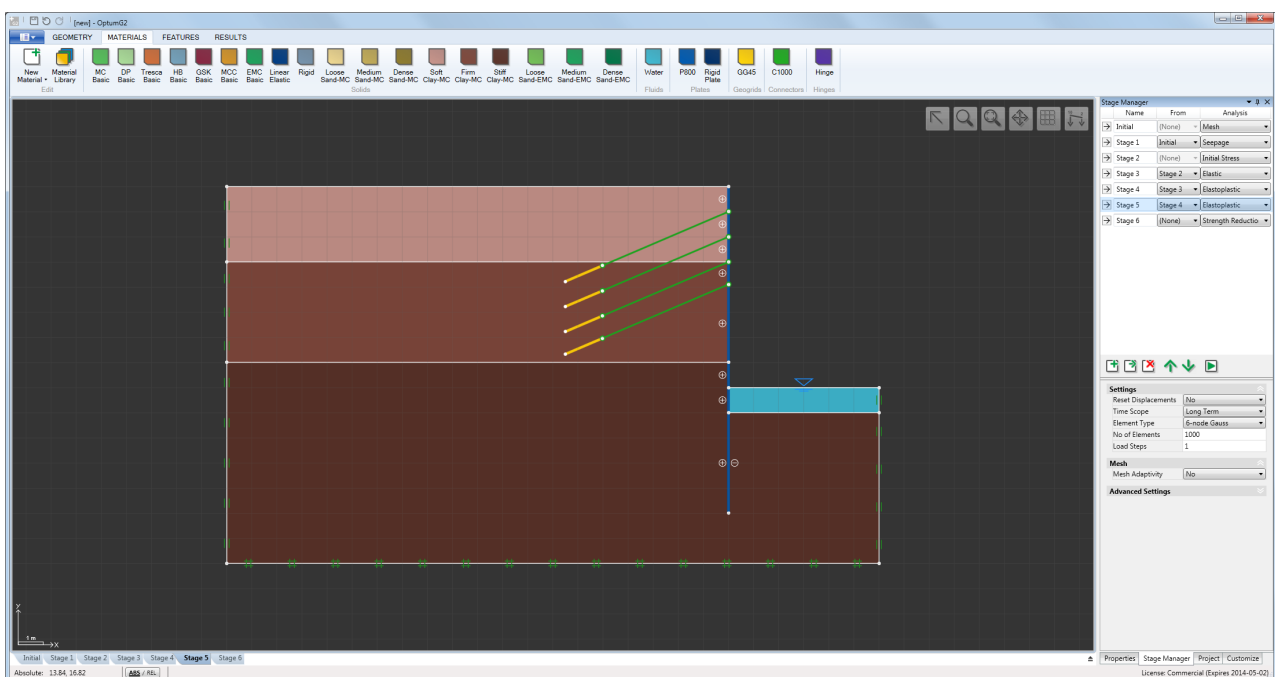


Figure 1.1: Optum G2 showing the Stage Manager on the right.

In between the two parts of the Stage Manager window is a row of control buttons (Figure 1.2). These controls allow for various manipulations: adding new stages, cloning existing stages, removing stages and moving stages up and down in the list of stages to be processed. The last button executes the calculations and the stages are processed in the order in which they appear in the list.

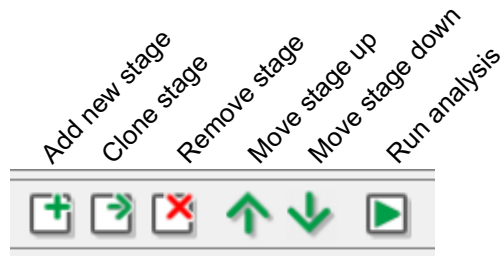


Figure 1.2: Stage Manager control buttons.

The analysis types available in Optum G2 are:

- Mesh
- Seepage
- Initial Stresses
- Elastic
- Limit Analysis
- Strength Reduction
- Elastoplastic
- Multiplier Elastoplastic

In the following, each of these different analysis types are documented in detail.

2 MESH

This analysis type is atypical in that it involves no physics but only generates a mesh.

Settings

This category contains a single field:

- No of elements: approximate number of elements in the mesh.

In addition, the mesh will be affected by the possible presence of Mesh Sizes and Mesh Fans as specified via the relevant tools in the Features ribbon (see the Features Manual and the Examples Manual).

Note: all analyses involve the automatic generation of a mesh. As such the Mesh analysis type is meant only to gauge the eventual mesh that would be used in an analysis with the same mesh settings.

3 SEEPAGE

This analysis performs seepage in accordance with the material properties and seepage boundary conditions. No a priori distinction is made between confined and unconfined seepage.

Settings

- No of Elements: this specifies the approximate number of elements used in the analysis (always three-node linear pressure elements). If mesh adaptivity is used (see below), it specifies the number of elements in the final mesh.

Mesh

It is possible to use adaptive mesh refinement using the following settings:

- Mesh Adaptivity (Yes/No).
- Adaptivity Iterations: number of adaptive refinement steps.
- Start Elements: number of elements in the initial mesh.
- Adaptivity Control (Flow, Flow Energy, Saturation): control variable on the basis of which the mesh is adapted.

Adaptivity Iterations specifies the number of adaptivity iterations. Usually 3 or 4 are appropriate.

Start Elements specifies the number of elements in the first adaptivity iteration.

The three possible control variables are described below.

Adaptivity Control = Saturation leads to the mesh being refined around any free seepage surfaces that may exist within the domain.

Adaptivity Control = Flow Energy leads to the mesh being refined according to the distribution of the 'flow energy' $\frac{1}{2}\mathbf{q}\mathbf{k}^{-1}\mathbf{q}$ where \mathbf{q} is the flux and \mathbf{k} is the hydraulic conductivity modulus.

Adaptivity Control = Flow uses a combination of Saturation and Flow Energy. This is the default setting.

3.1 Notes

Hydraulic boundary conditions applied via the tools located in the Features ribbon (see the Features Manual). However, it should be noted that all external boundaries are seepage faces by default and remain so unless another boundary condition is imposed.

4 INITIAL STRESSES

The initial stresses in the ground are in many cases an important aspect of strength and deformation analysis. In geotechnics, the initial stress state is usually characterized by the earth pressure coefficient:

$$K_0 = \frac{\sigma'_v}{\sigma'_h} \quad (4.1)$$

where σ'_v and σ'_h are the effective vertical and horizontal stresses respectively. In Optum G2, it is assumed that gravity acts opposite to the y -direction and we thus have

$$K_0 = \frac{\sigma'_x}{\sigma'_y} = \frac{\sigma'_z}{\sigma'_y} \quad (4.2)$$

For simple rectangular domains supported at the bottom and on the sides, the horizontal stresses σ'_y must be such that equilibrium equations are satisfied. For a constant unit weight this implies a linear variation of σ_y with depth. The vertical stresses, σ'_x and σ'_z can then be specified as $\sigma'_x = \sigma'_z = K_0\sigma'_y$ where K_0 must be such that the yield conditions are satisfied. For a purely frictional Mohr-Coulomb material, the bounds on K_0 are the well-known active and passive earth pressure coefficients:

$$\frac{1 - \sin \phi}{1 + \sin \phi} \leq K_0 \leq \frac{1 + \sin \phi}{1 - \sin \phi} \quad (4.3)$$

For problems involving more complex geometries and/or constitutive models, it generally not possible to prescribe the initial stresses in this simple way. In such cases, the Initial Stresses analysis available in Optum G2 is convenient. This analysis aims to determine a stress field that:

- Satisfies the equilibrium and boundary conditions.
- Satisfies the yield conditions.
- Satisfies the earth pressure conditions (4.2) 'to the greatest possible extent'.

Whereas the two first conditions cannot be negotiated, there is some flexibility with respect to the last one. In Optum G2, it is implemented by minimizing the following error measure:

$$\epsilon = \int_V \|\sigma'_x - K_0\sigma'_y\| + \|\sigma'_z - K_0\sigma'_y\| dV \quad (4.4)$$

subject to equilibrium, boundary and yield conditions. Under the aforementioned conditions of a rectangular domain of homogeneous material the error will be zero, i.e. the earth pressure condition (4.2) is satisfied exactly. Under more general conditions, the error will be finite, but usually of a moderate magnitude.

If seepage boundary conditions are applied, the seepage pressures are determined before commencing the Initial Stress procedure. Note that K_0 refers to the ratio between the *effective* horizontal and vertical stresses.

It should be noted that Fixed loads (shown in green) are included into the Initial Stress procedure. These loads should therefore be regarded as having always been present, i.e. they are not a result of any known process of load application. If the fixed loads are meant to account for surcharge, i.e. soil whose strength and stiffness are neglected but whose weight is included in the analysis, this is

a reasonable assumption. In other cases where the initial Fixed loads are the result of a deliberate application of load through some known process, it is advisable to account for this process in a separate Elastoplastic analysis stage.

An exception to the above described procedure occurs for the analysis types Elastic and Elastoplastic (see Sections 5 and 8 respectively). For these analysis types, only gravity is included in the Initial Stress analysis. The Fixed loads are then applied in a number of steps that can be set by the user.

Where the constitutive model involves hardening parameters, e.g. Modified Cam Clay and HMC, the yield condition enforced in the Initial Stress procedure is the limiting failure condition experienced at large strains. So for Modified Cam Clay, $F = q - Mp \leq 0$ is enforced and for HMC the usual Mohr-Coulomb criterion is used.

Settings

This category contains two fields:

- Element Type.
- No of Elements.

The recommended Element Types are Lower, 6-node Gauss, or higher order elements from the Gauss family. Usually, the number of elements is not particularly critical and the computed stress fields will rarely improve significantly by increasing the number of elements above 1,000.

4.1 Choice of earth pressure coefficient

For granular materials, it is common practice to prescribe the earth pressure at rest as:

$$K_0 = 1 - \sin \phi \quad (4.5)$$

This means that K_0 will decrease as ϕ increases or, in other words, that denser soils imply a smaller K_0 than looser soils. While this assumption to some extent is supported both theoretically and experimentally, there is also experimental evidence to the contrary (see e.g. ?).

For fine grained soils such as clays, the earth pressure coefficient is usually taken as

$$K_0^{\text{oc}} = K_0^{\text{nc}} \text{OCR}^{\sin \phi} \quad (4.6)$$

where K_0^{oc} is the earth pressure coefficient for the soil in its present, possibly overconsolidated state, K_0^{nc} is the earth pressure coefficient of the soil in its original, normally consolidated, state and OCR is the overconsolidation ratio. Again, K_0^{nc} is usually taken as:

$$K_0^{\text{nc}} = 1 - \sin \phi \quad (4.7)$$

In summary: the prescription of a reasonable earth pressure coefficient is a non-trivial task subject to considerable uncertainty. Under drained conditions, only the deformations will be affected by the exact choice of earth pressure coefficient while the ultimate limit state is unaffected. Under undrained conditions, both the deformations and ultimate limit state depend directly on the earth pressure coefficient. The exact effect is problem dependent and it is therefore advisable to scrutinize the effects of the choice of K_0 appropriately.

5 ELASTIC

The Elastic analysis type performs elastic analysis only, i.e. yield conditions, tension cut-offs, and similar features that imply a constraint on the stresses are ignored. In Elastic analysis only Fixed loads are processed while Multiplier loads are ignored.

Settings

This category contains three fields:

- Time Scope (Long Term/Short Term).
- Element Type.
- No of Elements.

Regarding the element type, all elements are in principle applicable. However, as is well known, some elements are better than others. The 6-node FE and higher-order elements from the Gauss family are particularly suitable. With these elements, the elastic energy of the system is underestimated, i.e. the elements are 'too stiff'. Generally speaking, this means the deformations are underestimated. Conversely, with the Lower bound element the elastic energy is over-estimated, i.e. the element is 'too flexible' and the displacements are generally over-estimated. An example is shown in Figure 5.1. The error for the finest mesh is here less than 1%. In other words, the actual, exact, displacement has been determined to with a tolerance not exceeding 1%.

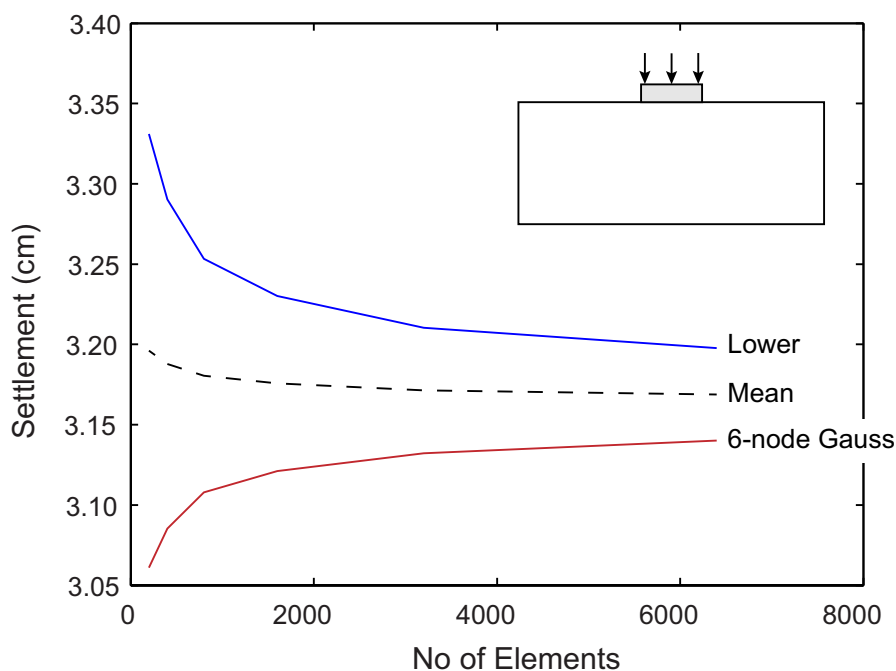


Figure 5.1: Upper and lower bounds on elastic settlement of rigid footing (soil domain = 2×5 m, footing width = 1 m, load = 100 kN/m^2 , $E = 30 \text{ MPa}$, $\nu = 0.25$).

Mesh

It is possible to use adaptive mesh refinement using the following settings:

- Mesh Adaptivity (Yes/No).
- Adaptivity Iterations: number of adaptive refinement steps.
- Start Elements: number of elements in the initial mesh.
- Adaptivity Control (Elastic Energy, Strain): control variable on the basis of which the mesh is adapted.

Adaptivity Iterations specifies the number of adaptivity iterations. Usually 3 or 4 are appropriate.

Start Elements specifies the number of elements in the first adaptivity iteration.

The two possible control variables are described below.

Adaptivity Control = Elastic Energy leads to the mesh being refined according to the distribution of the elastic complementary energy $\frac{1}{2}\boldsymbol{\sigma}^T\mathbf{C}\boldsymbol{\sigma}'$.

Adaptivity Control = Strain leads to the mesh being refined according to the distribution of the the norm of the strain vector, $\|\boldsymbol{\varepsilon}\|$.

5.1 Notes

Initial Stresses

For Elastic stages without a From stage, the initial stresses are calculated using the Initial Stress procedure described in Section 4. However, Fixed loads are not taken into account but rather applied as part of the Elastic analysis itself.

Seepage pressures are computed automatically if any Flow BCs are specified.

6 LIMIT ANALYSIS

Limit Analysis allows for the rapid assessment of the stability or bearing capacity of geostructures without having to perform an exhaustive step-by-step elastoplastic analysis.

Both Fixed and Multiplier loads are applicable. In Limit Analysis, the former are kept constant while the latter are amplified until a state of incipient collapse is attained. The factor by which the multiplier loads need to be amplified in order to cause collapse is also referred to as the collapse multiplier.

In addition to determining the collapse multiplier for a set of external loads, it is also possible to compute the factor by which gravity should be amplified in order to achieve a state of collapse. This feature is useful for example in connection with slope stability.

A unique feature of Optum G2 is the ability to compute upper and lower bounds on the collapse load. This is achieved simply by choosing the appropriate element type – Upper or Lower respectively.

Settings

This category contains four fields:

- Multiplier (Load/Gravity).
- Time Scope (Long Term/Short Term).
- Element Type.
- No of Elements.

For Multiplier = Load, the collapse multiplier amplifies the Multiplier loads (shown in red) while Fixed loads (shown in green) and gravity are kept constant at their specified values. For Multiplier = Gravity, Multiplier loads are ignored, Fixed loads are kept constant, and the gravitational acceleration is amplified until a state of collapse is attained. These rules are summarized in Table 6.1. Some examples are shown in Figure 6.1

	Multiplier = Load	Multiplier = Gravity
Multiplier loads	Amplified	Ignored
Fixed loads	Constant	Constant
Gravity	Constant	Amplified

Table 6.1: Status of Multiplier loads, Fixed loads and Gravity as function of Multiplier setting.

Regarding the Time Scope setting, please refer to the Materials Manual.

For Element Type = Lower, rigorous lower bounds are computed whereas upper bounds result from Element Type = Upper.

No of Elements specifies the number of elements (and the number of elements in the final mesh if mesh adaptivity is used).

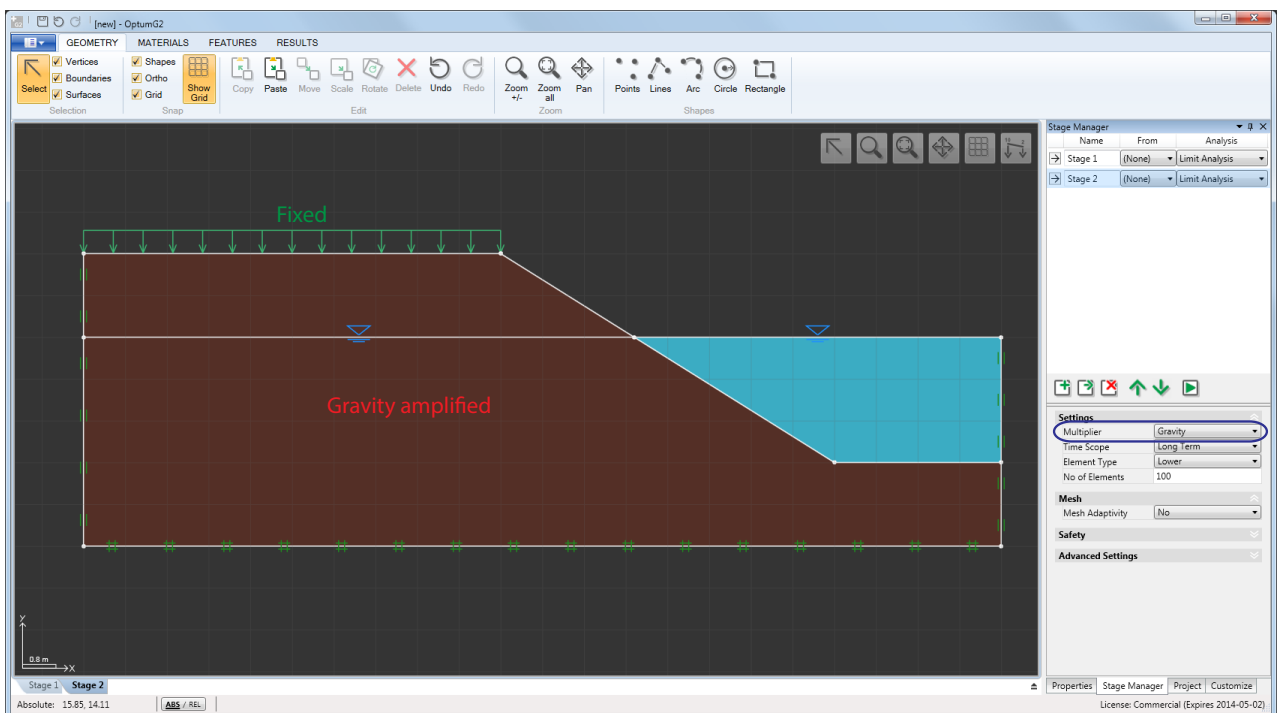
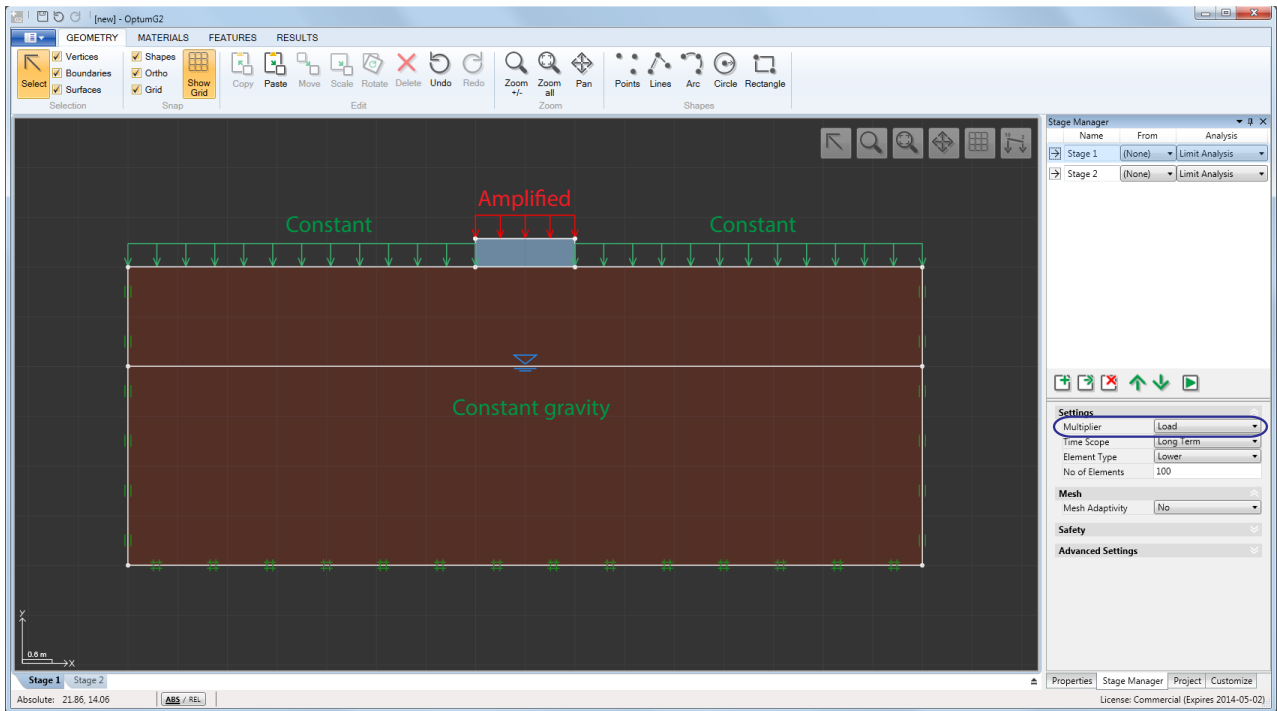


Figure 6.1: Examples of Limit Analysis with Multiplier = Load (top) and Multiplier = Gravity (bottom).

Mesh

It is possible to use adaptive mesh refinement using the following settings:

- Mesh Adaptivity (Yes/No).
- Adaptivity Iterations: number of adaptive refinement steps.

- Start Elements: number of elements in the initial mesh.
- Adaptivity Control (Shear Dissipation, Total Dissipation, Strain, Plastic Multiplier): control variable on the basis of which the mesh is adapted.

Adaptivity Iterations specifies the number of adaptivity iterations. Usually 3 or 4 are appropriate.

Start Elements specifies the number of elements in the first adaptivity iteration.

The four possible control variables are described below.

Adaptivity Control = Shear Dissipation leads to the mesh being refined according to the distribution of the plastic shear dissipation $D_s = (\boldsymbol{\sigma} - \mathbf{m}p)^T(\boldsymbol{\varepsilon} - \mathbf{m}\varepsilon_v)$ where p is the mean stress, ε_v is the volumetric strain, and $\mathbf{m} = (1, 1, 1, 0, 0, 0)^T$. This control variable is generally the most efficient and reliable one for Limit Analysis.

Adaptivity Control = Total Dissipation leads to the mesh being refined according to the distribution of the total plastic dissipation $D = \boldsymbol{\sigma}^T \boldsymbol{\varepsilon}$. This control variable is not appropriate for purely frictional materials.

Adaptivity Control = Strain leads to the mesh being refined according to the distribution of the norm of the strain vector $\|\boldsymbol{\varepsilon}\|$.

Adaptivity Control = Plastic Multiplier leads to the mesh being refined according to the distribution of the plastic multiplier λ (see the Theory Manual).

Safety

Limit Analysis requires specification of a Design Approach. Following Eurocode 7, loads are classified as being either favorable or unfavorable to the structural integrity of the system under consideration. Moreover, within these two classifications, the loads must further be classified as being either permanent, variable, or accidental. Depending on the classification a partial factor is attached to the load. In the limit analysis, this factor is automatically multiplied onto the load magnitude entered in the property window.

On the materials side, Eurocode operates with partial factors on the parameters c , $\tan \phi$ and c_u . In OptumG2, models operating with c and ϕ are: Mohr-Coulomb, Modified Cam Clay, and MHC while Tresca is the only model that makes use of c_u . In OptumG2, the original material parameters are automatically factored before the limit analysis, leading to reduced parameters:

$$\begin{aligned} c_{\text{red}} &= c/F_c \\ \phi_{\text{red}} &= \arctan[(\tan \phi)/F_\phi] \\ c_{u,\text{ref}} &= c_u/F_{cu} \end{aligned} \tag{6.1}$$

where F_c , F_ϕ and F_{cu} are the partial factors.

A total of seven Design Approaches are available: Unity, DA1/1, DA1/2, and User 1-4. The partial factors attached to each of these Design Approaches can be viewed and edited via the Design

Approaches button under Project.

Design Approaches							
	Unity	DA1/1	DA1/2	User 1	User 2	User 3	User 4
Unit Weight - Unfavourable:							
Permanent	1	1.35	1	1	1	1	1
Unit Weight - Favourable:							
Permanent	1	1	1	1	1	1	1
Loads - Unfavourable:							
Permanent	1	1.35	1	1	1	1	1
Variable	1	1.5	1.3	1	1	1	1
Accidental	1	1	1	1	1	1	1
Loads - Favourable:							
Permanent	1	1	1	1	1	1	1
Variable	1	0	0	1	1	1	1
Accidental	1	0	0	1	1	1	1
Materials:							
c	1	1	1.25	1	1	1	1
ϕ	1	1	1.25	1	1	1	1
c_u (Tresca)	1	1	1.4	1	1	1	1

Figure 6.2: Design Approaches table for defining partial factors for loads and materials (available under Project).

The default Design Approach is Unity meaning that a factor of unity is applied to all loads and material parameters.

The two predefined Design Approaches, DA1/1 and DA1/2, correspond to those adopted by the UK, Ireland, Italy (except for slopes), Poland and Belgium among others. Other Design Approaches (e.g. Design Approach 3 of Eurocode 7) can be defined in the User 1-4 columns.

6.1 Notes

Initial Stresses

For limit analysis under conditions where no excess pore pressures are generated (Always Drained material regardless of Time Scope or Drained/Undrained materials for Time Scope = Long Term), the solution is independent of the initial stresses. This is a unique feature of limit analysis.

For cases where excess pore pressures are generated (Time Scope = Short Term for Drained/Undrained materials), the solution will depend on the initial stresses. For such analysis, a From stage may either be specified in the Stage Manager or, if no From stage is specified, the initial stresses will be calculated automatically using the Initial Stress analysis described in Section 4.

Seepage pressures are always calculated automatically and accounted for in the subsequent limit analysis.

7 STRENGTH REDUCTION

While Limit Analysis deals with the problem of determining the ultimate magnitude of a set of reference loads, Strength Reduction analysis deals the complementary problem of determining the strength necessary to prevent collapse given a set of actual loads.

The Strength Reduction analysis in Optum G2 proceeds by computing a strength reduction factor by which the material parameters need to be reduced in order to attain a state of incipient collapse. A factor greater than 1 thus implies a stable system while a factor less than 1 implies that additional strength is required to prevent collapse.

Depending on the model in question, one or more of the parameters are reduced in a particular way that affects the value of the parameters actually leading to collapse. For example, for Mohr-Coulomb, c and $\tan \phi$ are reduced by the same amount. So for an initial set of $c = 10$ kPa and $\phi = 25^\circ$, a reduction factor equal to 1.3, gives reduced strengths of $c_{\text{red}} = c/1.3 = 7.7$ kPa and $\phi_{\text{red}} = \arctan[(\tan 25^\circ)/1.3] = 19.7^\circ$. This reduction strategy is consistent with the commonly used definition of the factor of safety (FS):

$$\text{FS} = \frac{c}{c_{\text{red}}} = \frac{\tan \phi}{\tan \phi_{\text{red}}} \quad (7.1)$$

In other words, the strength reduction factor may be viewed as the factor of safety. Table 7.1 provides an overview of the reduction strategies used for the different models available in Optum G2.

	Parameters reduced
Mohr-Coulomb	$c, \tan \phi$
Drucker-Prager	k, M
Tresca	c_u
Hoek-Brown	σ_{ci}, m_i
GSK	$c, \tan \phi_1, \tan \phi_2$
Modified Cam Clay	$\tan \phi$
MHC	$c, \tan \phi$

Table 7.1: Parameters reduced in Strength Reduction analysis.

In addition to the strategy of defining the strength reduction factor in terms of soil parameters, it may also in some cases, be convenient to use the reduction in strength of certain structural element necessary to cause collapse as a measure of the factor of safety of the system. For example, for an excavation supported by a sheet pile wall, the factor of safety could be defined as

$$\text{FS} = \frac{m_p}{m_{p,\text{red}}} \quad (7.2)$$

where m_p is the plastic moment (yield moment).

The Strength Reduction analysis available in Optum G2 allows for a reduction in either the strength of the solids according to Table (7.1) or a reduction in the strength of the structural elements (Beams, Geogrids, Anchors, Connectors). In the latter case, reduction strategy is as summarized in Table 7.2.

	Parameters reduced
Beams	n_p, m_p (Set A) or σ_0 (Set B)
Geogrids	n_p (Set A) or σ_0 (Set B)
Fixed-End Anchors	n_p (Set A) or σ_0 (Set B)
Connectors	n_p (Set A) or σ_0 (Set B)

Table 7.2: Parameters reduced in Strength Reduction analysis (Structs).

Settings

This category contains four fields:

- Reduce Strength in (Solids/Structs)
- Time Scope (Long Term/Short Term).
- Element Type.
- No of Elements.

Regarding the first setting, the option Solids reduces the solid materials strengths according to Table 7.1 while the strengths of the structural elements are kept fixed. Conversely, the option Structs, implies a reduction of the structural elements strengths according to Table 7.2, keeping the solid strengths fixed.

For the Time Scope setting, please refer to the Materials Manual.

For Element Type = Lower, rigorous lower bounds are computed whereas upper bounds result from Element Type = Upper. That is, the strength reduction factor will be less than the exact one in the former case and greater than the exact one in the latter case.

No of Elements specifies the number of elements (and the number of elements in the final mesh if mesh adaptivity is used).

Mesh, Safety

See Section 6. Regarding Safety, the loads and material parameters are factored according to the Design Approach and the Load Type and Category following the table in Figure 6.2.

7.1 Notes

Initial Stresses

As for limit analysis, under conditions where no excess pore pressures are generated (Always Drained material regardless of Time Scope or Drained/Undrained materials for Time Scope = Long Term), the solution is independent of the initial stresses which will not be computed.

For cases where excess pore pressures are generated (Time Scope = Short Term for Drained/Undrained materials), the solution will depend on the initial stresses. For such analysis, a From stage may either be specified in the Stage Manager or, if no From stage is specified, the initial stresses will be calculated automatically using the Initial Stress analysis described in Section 4. Note that any Fixed loads present in the problem will enter into the Initial Stress analysis. As such, if the Fixed loads are applied in the short term, a separate From stage without the Fixed loads is necessary to establish the initial stress state.

Seepage pressures are always calculated automatically and accounted for in the subsequent Strength Reduction analysis.

8 ELASTOPLASTIC

Elastoplastic analysis is similar to Elastic analysis, the difference being that failure criteria for all solid and structural elements are included in the analysis. As in Elastic analysis, only Fixed loads are processed while Multiplier loads are ignored. Gravity is automatically included. Elastoplastic is the appropriate analysis for both the situation where the deformations in response to a load of a given magnitude are to be computed and in the case where the analysis involves several linked stages in a staged construction scenario (e.g. excavation, embankment construction, etc).

Settings

This category contains four fields:

- Time Scope.
- Element Type.
- No of Elements.
- Load Steps.

For the Time Scope setting, please refer to the Materials Manual.

In Elastoplastic analysis either of the Gauss elements (6-node or 15-node) are usually appropriate choices. However, the elements Lower and Upper may also be used to compute a response that underestimates (Lower) or overestimates (Upper) the true strength as well as stiffness. It should be noted, though, that these bounds are, strictly speaking, only valid for materials with an associated flow rule. For materials with a nonassociated flow rule, it may sometimes be observed that elements of the type Lower lead to higher apparent strengths than Upper. However, this is quite rare and as a general rule, elements of the type Lower will underestimate both the strength and the stiffness as compared to all the other available elements.

No of Elements specifies the number of elements (and the number of elements in the final mesh if mesh adaptivity is used).

Load Steps specifies the number of load steps. A value greater than 1 may be used both where a Fixed load is to be incremented and in the case where deformations occur as a result of changes in geometry (e.g. in excavation analysis). In the latter case, the out-of-balance forces are reduced in a number of steps within the same stage. Generally, however, it is not recommended to link stages with differences in geometry that would lead to a large dependence on the number of load steps.

Mesh

It is possible to use adaptive mesh refinement using the following settings:

- Mesh Adaptivity (Yes/No).
- Adaptivity Iterations: number of adaptive refinement steps.
- Adaptivity Frequency.
- Start Elements: number of elements in the initial mesh.

- Adaptivity Control (Shear Dissipation, Total Dissipation, Strain, Plastic Multiplier): control variable on the basis of which the mesh is adapted.

Adaptivity Iterations specifies the number of adaptivity iterations. Usually 3 or 4 are appropriate.

Adaptivity Frequency specifies how often the mesh is adapted for analysis with Load Steps > 1 . The default setting Adaptivity Frequency = 3 thus implies that the mesh is adapted after load steps 1, 4, 7, etc.

Start Elements specifies the number of elements in the first adaptivity iteration.

The adaptivity controls are the same as those for Limit Analysis (see Section 6) although all are combined with Elastic Energy (see Section 5) to take both elastic and plastic deformations into account. The default option Shear Dissipation is usually the most reliable and efficient control variable.

8.1 Notes

Initial Stresses

For Elastoplastic stages without a From stage, the initial stresses are calculated using the Initial Stress procedure described in Section 4. However, Fixed loads are not taken into account but rather applied as part of the Elastic analysis itself.

Seepage pressures are computed automatically if any Flow BCs are specified.

9 MULTIPLIER ELASTOPLASTIC

Multiplier Elastoplastic analysis may be seen as combining the Limit Analysis and Elastoplastic analysis types. As in Limit Analysis, the Multiplier loads are amplified until collapse while Fixed loads and gravity are kept constant. This is done in a step-by-step elastoplastic manner with deformations computed at each load step.

Settings

This category contains three fields:

- Time Scope.
- Element Type.
- No of Elements.

For the Time Scope setting, please refer to the Materials Manual.

The Element Type considerations follow those of Elastoplastic analysis (see the previous section). The recommended elements are 6-node Gauss or 15-node Gauss, but any of the available elements may in principle be used, including Lower with which a response that underestimates both the stiffness and the strength of the system will be computed. An example is shown in Figure 9.1.

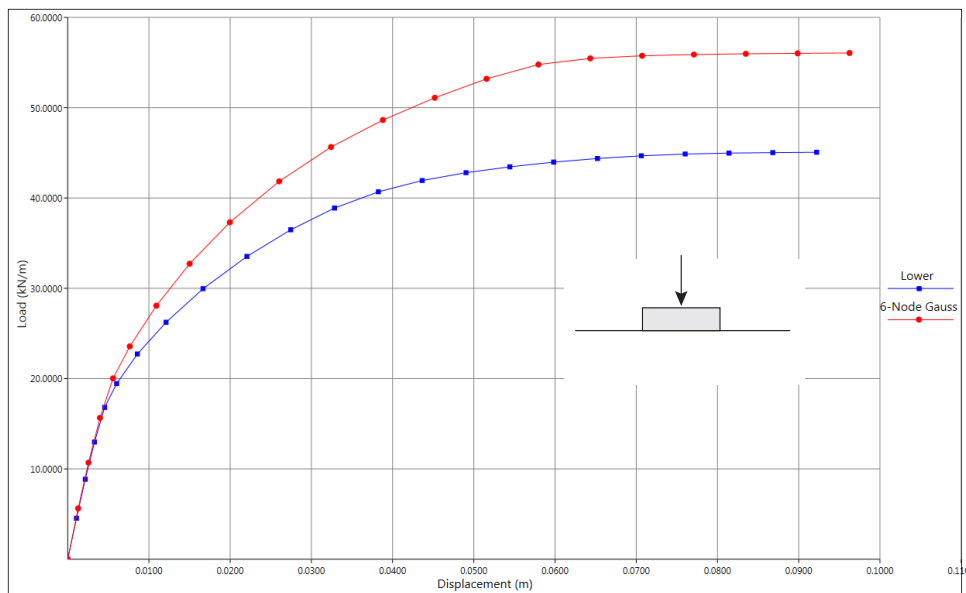


Figure 9.1: Load-displacement curves for strip footing on Tresca material using Lower and 6-node Gauss elements.

No of Elements specifies the number of elements (the number of elements in the final mesh if mesh adaptivity is used).

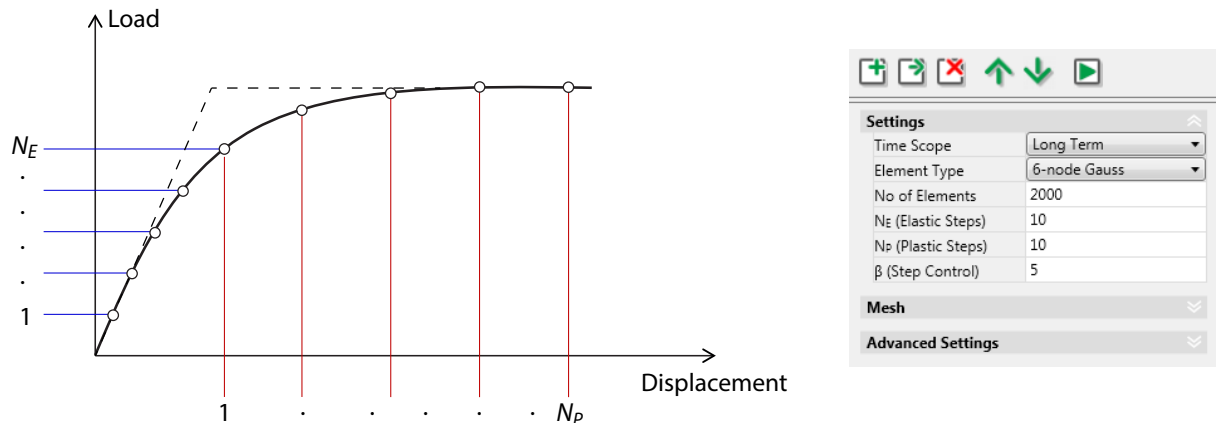


Figure 9.2: Typical load-displacement curve (left) and Settings (Stage Manager).

Load Stepping

Two different types of load stepping schemes are available: Auto and Target.

For Scheme = Auto, three additional fields appear:

- N_E (Elastic Steps).
- N_P (Plastic Steps).
- β (Step Control).

These are the three parameters that control the Auto load stepping scheme. Considering a typical load-displacement curve as shown in Figure 9.2, the response will initially be approximately linear and then gradually become more and more plastic. N_E and N_P specifies the approximate number of steps in these two regions. The parameter β is used for automatic adjustment of the load step from step to step. For $\beta = 1$, the step size (measured in terms of work) will not vary while higher values of β implies a more aggressive strategy with the step size being increased for parts of the curve where there is little change. For most problems, the default parameters, $N_E = 10$, $N_P = 10$, and $\beta = 5$, are a good starting point and often little improvement is observed as a result of increasing the number of steps, i.e. increasing N_E and N_P .

For Scheme = Auto, three additional fields appear:

- Target (Displacement, Multiplier, Work).
- u_{target} , α_{target} or w_{target} depending on Target.
- N (Steps).

The idea with the Target scheme is to continue the load stepping until the specified target has been reached. This target may be specified as a displacement (maximum absolute displacement in the problem), as a load multiplier, or as a work (integral of reference multiplier load time times displacement). The load stepping is performed in N steps of equal magnitude.

Mesh

See Section 8.

Advanced Settings

This category contains three fields:

- Initial State (Default, Auto, User).
 - For Initial State = User: Start Multiplier
- Seepage Pressures.

The first option concerns the initial state. For stages linked to a From stage, this option is not relevant: the initial stresses will simply be transferred from the From stage to the current stage.

For stages without a From stage, the Initial State setting is used to determine how the initial stresses are computed. For Initial State = Default, a standard Initial Stress analysis as described in Section 4 is carried. For many problems, this will lead to a realistic and satisfactory initial state. The load stepping then following from an initial load multiplier equal to zero.

There are, however, situations where the Initial Stress analysis will fail due to the initial configuration of the problem being unstable. An example is shown in Figure 9.3. This problem, which aims to determine the passive earth pressure on the wall, is initially unstable since the wall is not supported horizontally and Multiplier loads are ignored in the Initial Stress analysis. In such cases, Initial State = Auto may be used. This setting implies that the initial stress state will be found by an elastoplastic analysis in which the load multiplier attains the minimum value necessary to make the problem stable. Once this state is found, the actual load stepping begins from a non-zero value of the initial load multiplier.

Finally, by the option Initial State = User, it is possible to specify the initial value of the load multiplier manually after which the initial stresses will be determined in an elastoplastic analysis using this

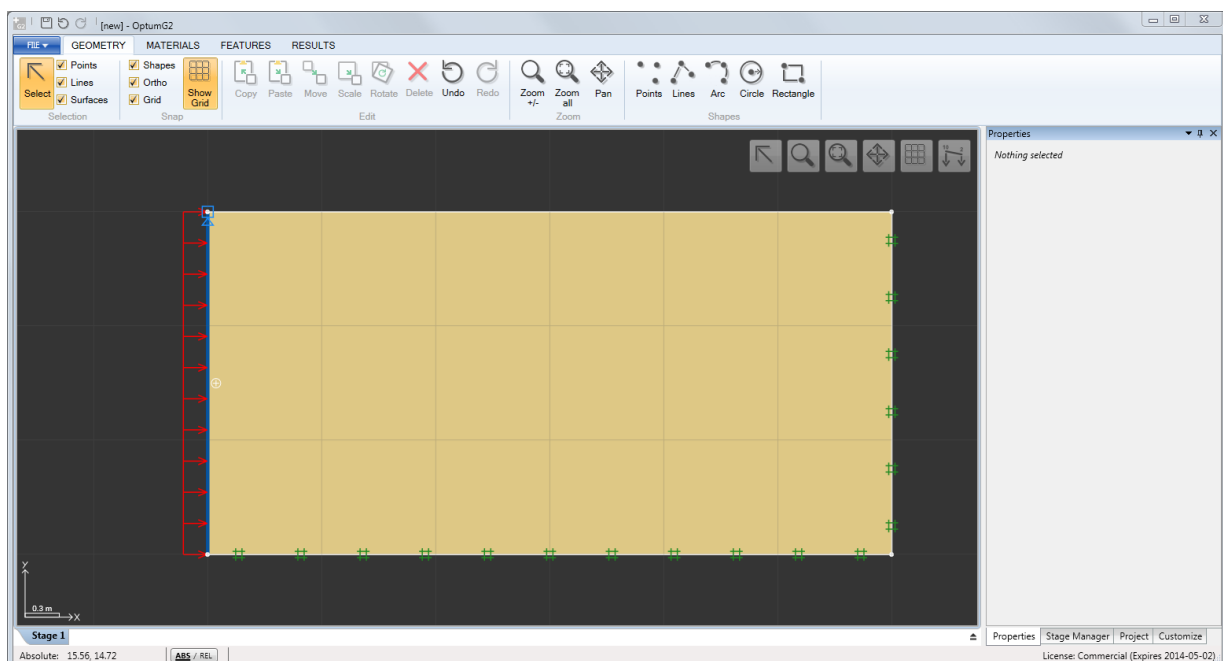


Figure 9.3: Initially unstable Multiplier Elastoplastic problem requiring either use of a separate From stage or Initial State = Auto/User to compute the initial stress state.

value of the multiplier.

The last field allows for a user defined steady state seepage pressure distribution (see the Materials Manual for user defined distributions).

10 CONSOLIDATION

Consolidation analysis is applicable to situations where the generation and dissipation of excess pore pressures with time are of interest. Examples include rapidly loaded foundations, embankment construction, excavations and various other staged construction analyses. It is not recommended to probe the ultimate limit state by means of Consolidation analysis. For such information, Limit Analysis, Strength Reduction or Multiplier Elastoplastic analysis should be used.

Settings

This category contains two fields:

- Element Type.
- No of Elements.

The Element Type considerations follow those of Multiplier Elastoplastic analysis (see the previous section). The recommended elements are 6-node Gauss or 15-node Gauss, but any of the available elements may in principle be used, including Lower with which a response that underestimates both the stiffness and the strength of the system will be computed.

No of Elements specifies the number of elements (the number of elements in the final mesh if mesh adaptivity is used).

Time Stepping

Two different types of load stepping schemes are available: Auto and Target.

For Scheme = Auto, one additional field appears:

- No of Steps

The Auto scheme works by continuing the stepping to a point where the state is close to being drained. This is done by automatically adjusting the time step in the course of the specified number of steps.

For Scheme = Target, three additional fields appear:

- Target (Time, Degree)
- No of Steps
- t_{target} or U_{target} depending on Target.

The idea with this scheme is similar to that of the Multiplier Elastoplastic Target scheme: a quantity to monitor (time or degree of consolidation) is specified and the time stepping continues until the specified target is reached. The number of steps is specified in the second field. For Target = Time, a number of time steps of equal magnitude is used while for Target = Degree, the time step is adjusted automatically to reach the target degree of consolidation (approximately) in the course of the number of steps specified.

Advanced

As for Elastoplastic analysis (see Section 8).