

1 SHALLOW FOUNDATION 1

This example deals with the an eccentrically loaded foundation as shown in Figure 1.1. The soil is saturated clay and the analysis is to be performed assuming undrained conditions. For this purpose a total stress analysis approach is adopted. The soil is modeled by means of the Tresca model with an undrained shear strength $s_u = 30$ kPa and an undrained Young's modulus of $E_u = 40$ MPa. The foundation is modeled as Rigid material with a unit weight of 24 kN/m³. The material properties are shown in the property window on the right in Figure 1.1.

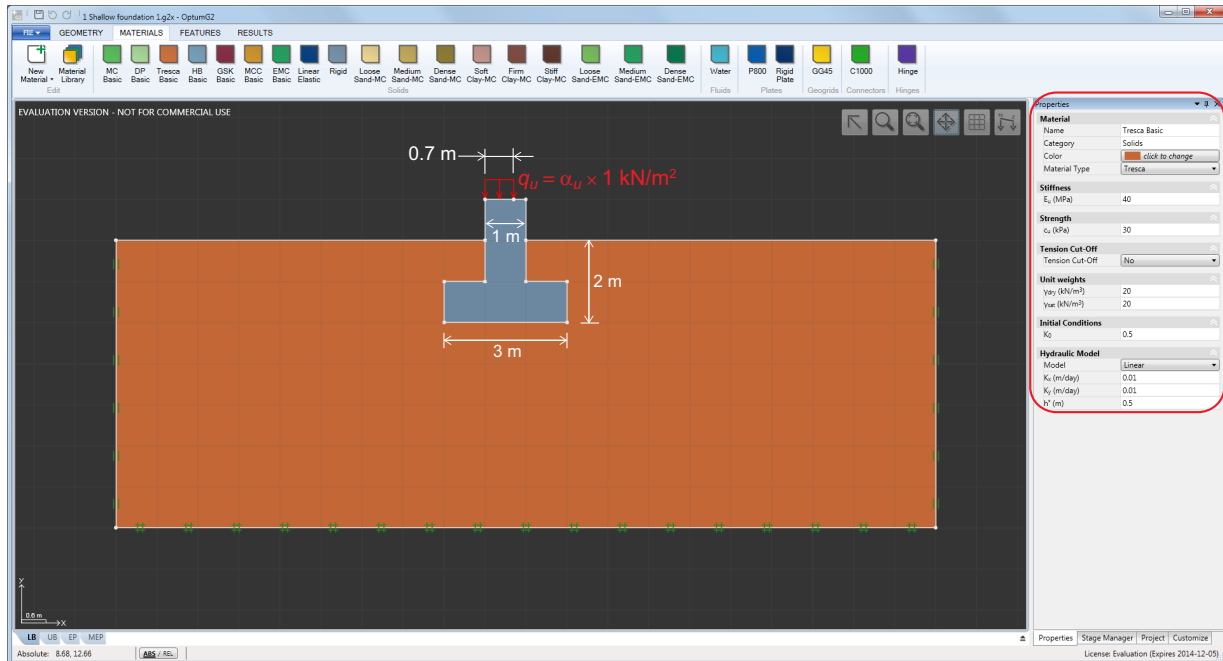


Figure 1.1: Shallow foundation in Tresca USS soil.

The task of setting up the problem proceeds by creating the geometry and then assigning materials and load. The boundary conditions are then applied by the Standard Fixities button in the Features ribbon.

1.1 Limit analysis

The first goal of the the analysis is to determine the ultimate magnitude, α_u , of the vertical reference load of 1 kN/m² working on the foundation. For this purpose Limit Analysis is used. The result of this analysis is the load multiplier α_u , i.e. the factor by which the multiplier load (shown in red) should be magnified in order to induce a state of collapse.

In the Stage Manager, Limit Analysis is chosen as the relevant analysis. Under Settings in the lower half of the Stage Manager window, the particular settings of the stage are specified. For the present analysis Multiplier should be set to Load since the aim is to determine the ultimate magnitude of an external load. The Time Scope is in this case (for the Tresca model) irrelevant and may be set to Long Term.

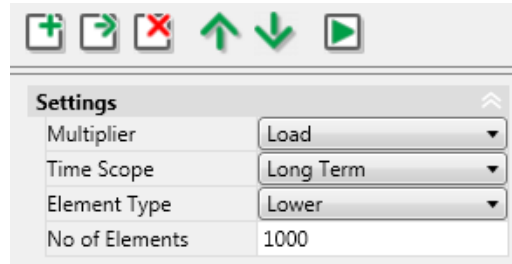


Figure 1.2: Stage settings for lower bound limit analysis. The Time Scope is irrelevant for the Tresca model.

Rather than determine an approximate solution to the problem, upper and lower bounds on the exact bearing capacity will be computed. This requires two separate calculations which may be organized in two stages with Element Type = Lower and Upper respectively. For both analyses, the number of elements (No of Elements in Settings) is set to 1,000.

Running the analyses results in lower and upper bound collapse multipliers of 851.1 and 1017.4 respectively. In other words, the maximum vertical load that can be sustained is:

$$851.1 \times 1 \text{ kN/m}^2 \leq q_u \leq 1017.4 \times 1 \text{ kN/m}^2 \quad (1.1)$$

or, in terms of total force (the load works over 0.8 m):

$$680.9 \text{ kN/m} \leq Q_u \leq 813.9 \text{ kN/m} \quad (1.2)$$

The result may also be stated as

$$q_u = 934.2 \text{ kN/m}^2 \pm 8.9\% \quad (1.3)$$

In other words, the error in the mean value between the upper and lower bounds is $\pm 8.9\%$.

1.1.1 Mesh adaptivity

The gap between the upper and lower bounds can be narrowed either by increasing the number of elements or by using mesh adaptivity. In the following we opt for the latter.

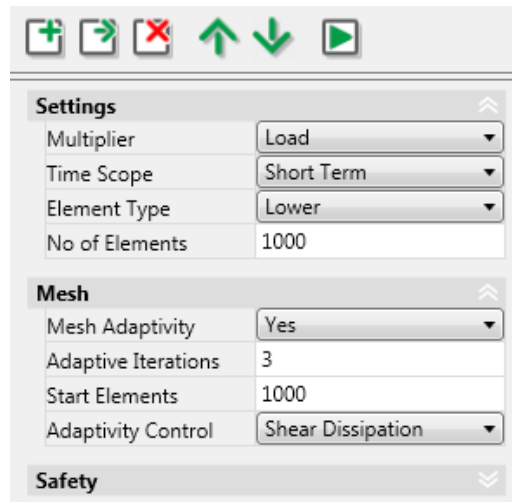


Figure 1.3: Stage settings for lower bound limit analysis mesh adaptivity.

Mesh adaptivity is defined under the category Mesh in the Stage Manager (see Figure 1.3). In the following, we will use 3 adaptivity steps together with the default option of Shear Dissipation as adaptivity control. This means that a total of 3 calculations will be carried out, each with a mesh adapted according to the previous distribution of the shear dissipation and such that the number of elements in the final mesh is equal to the number of elements specified in Settings (1,000 as before).

The results of the analyses are:

$$860.0 \text{ kN/m}^2 \leq q_u \leq 930.0 \text{ kN/m}^2 \quad (1.4)$$

or:

$$q_u = 895.0 \text{ kN/m}^2 \pm 3.9\% \quad (1.5)$$

which is a substantial improvement on the previous solution. Further improvements – at the expense of computational cost – can be achieved by increasing the number of elements.

The initial and adapted meshes for 1,000 elements are shown in Figure 1.4 along with the collapse solution.

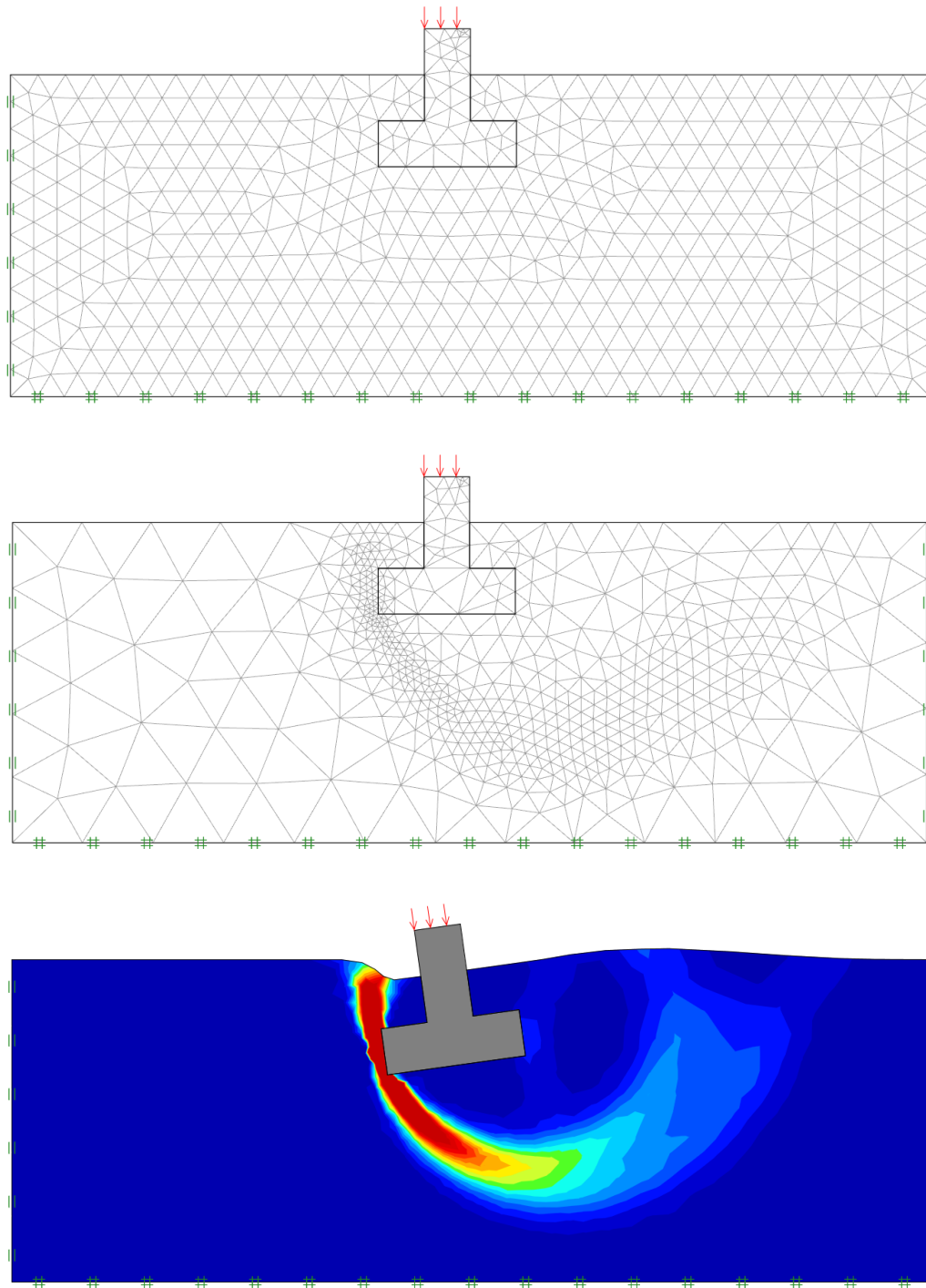
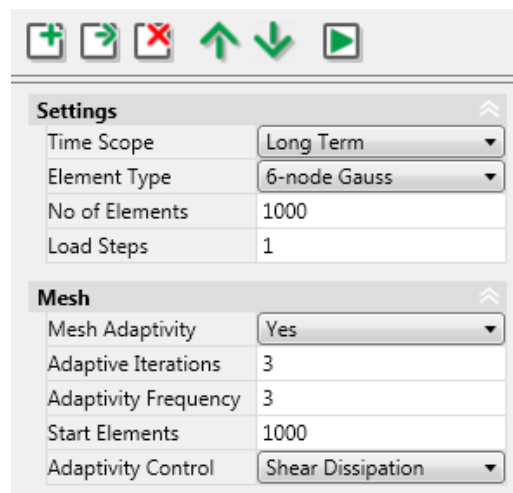


Figure 1.4: Initial and adapted meshes and collapse solution with intensity of dissipation (Upper element).

1.2 Elastoplastic analysis

Next, with the information that the collapse load is approximately 895 kN/m^2 , the deformations for a fixed load of 600 kN/m^2 are to be determined. For this purpose an Elastoplastic analysis is carried out. It is most convenient to clone the last stage and specify Elastoplastic in the Analysis column in the upper half of the Stage Manager window. In the lower half, the stage settings then appear. The Time Scope is again irrelevant. The Element Type is selected as 6-node Gauss which is well suited for deformation analysis. The No of Elements is set to 1,000. The number of Load Steps is set to 1. This means that the whole load is applied in a single step. For loads relatively far from collapse such as the present one (600 kN/m^2 vs a collapse load of 895 kN/m^2), this is usually adequate. Note: in contrast to the previous Limit Analysis, the loads of the current analysis are Fixed (shown in green).



The screenshot shows a software interface for setting analysis parameters. At the top, there are six icons: a green plus, a green right arrow, a red X, a green up arrow, a green down arrow, and a green play button. Below these icons is a 'Settings' section with a scroll-up arrow. It contains four rows: 'Time Scope' with a dropdown menu set to 'Long Term', 'Element Type' with a dropdown menu set to '6-node Gauss', 'No of Elements' with the value '1000', and 'Load Steps' with the value '1'. Below the 'Settings' section is a 'Mesh' section with a scroll-up arrow. It contains five rows: 'Mesh Adaptivity' with a dropdown menu set to 'Yes', 'Adaptive Iterations' with the value '3', 'Adaptivity Frequency' with the value '3', 'Start Elements' with the value '1000', and 'Adaptivity Control' with a dropdown menu set to 'Shear Dissipation'.

Figure 1.5: Stage settings Elastoplastic analysis with mesh adaptivity. The Time Scope is irrelevant for the Tresca model.

As for Limit Analysis, mesh adaptivity can be used. Again, this feature is activated by setting Mesh Adaptivity = Yes. A number of fields then appears. Adaptivity Iterations has the same meaning as before and is set to 3. Adaptivity Frequency is relevant only if more than one load step is used and is left at the default value of 3. And as before, the Adaptivity Control is set to Shear Dissipation. In the case of Elastoplastic analysis, the control variable incorporates both shear dissipation and elastic energy.

Any elastoplastic analysis requires an initial state of stress. In the present example, no From stage is specified, and consequently, the initial stresses are calculated automatically (see Section I.II).

The deformed configuration is shown in Figure 1.6 along with the distributions of shear dissipation and elastic energy. As expected, the plastic zones are less developed than at full collapse (compare to Figure 1.4).

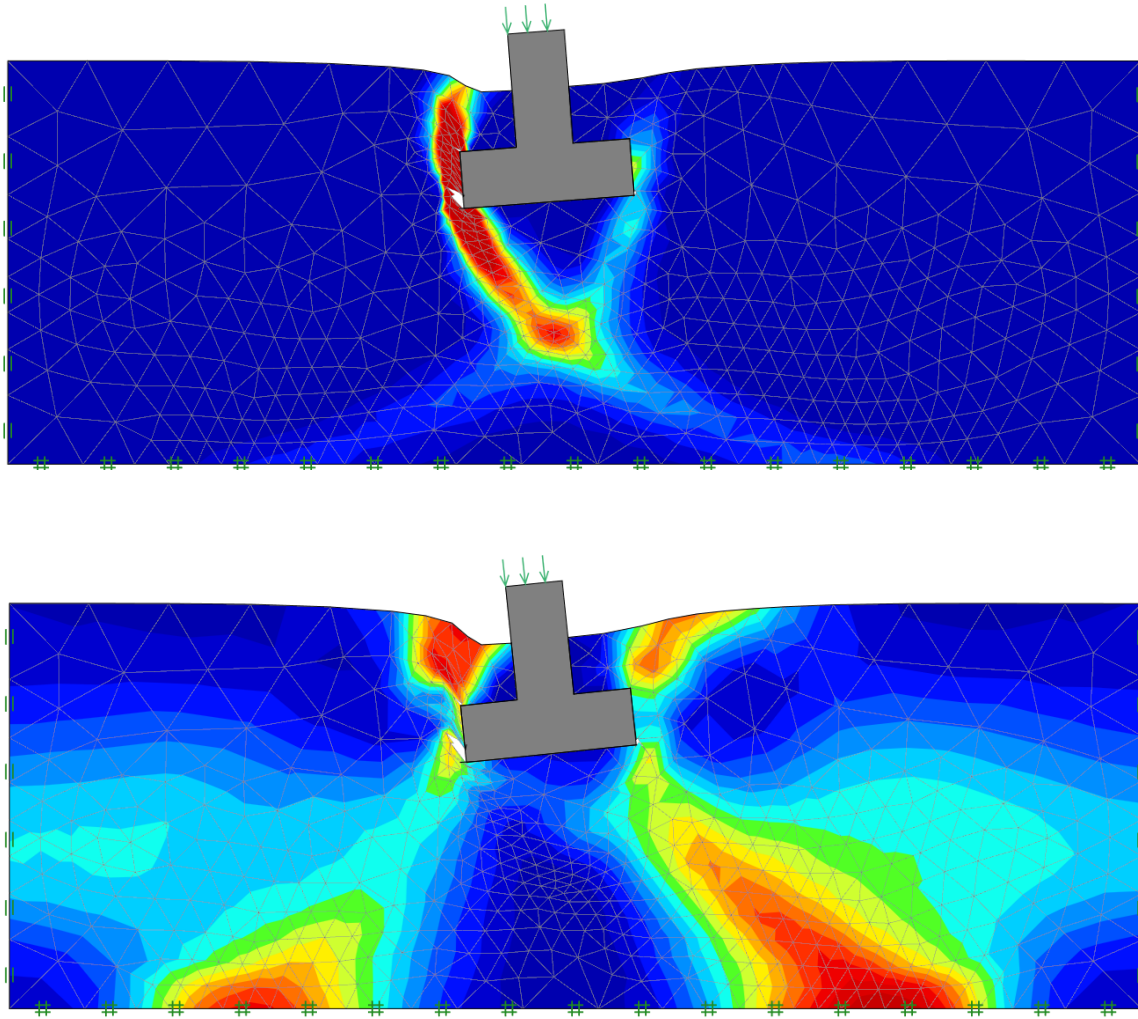


Figure 1.6: Deformations and distribution of shear dissipation (top) and elastic energy (bottom) from Elastoplastic analysis (displacements scaled by a factor of 30).

The displacements at selected points can be accessed by mouse click. In this way, the displacements at the upper left edge of the foundation are found as:

$$\begin{aligned} u_x &= -6.0 \text{ mm} \\ u_y &= -13.8 \text{ mm} \end{aligned} \quad (1.6)$$

These results may be improved slightly by increasing the number of elements and the number of load steps.

1.3 Multiplier Elastoplastic analysis

Besides determining the ultimate bearing capacity and the deformations under serviceability conditions in a direct and rapid manner, OptumG2 also allows for the full load-displacement response to be traced. Such analyses are carried out using the Multiplier Elastoplastic analysis type. This analysis type may be thought of as combining the two previous analysis types. As in Limit Analysis, a set of Multiplier Loads (shown in red) are incremented in a sequence of steps until collapse. And as in Elastoplastic analysis, the deformations are determined for each load step.

Settings	
Time Scope	Long Term
Element Type	6-node Gauss
No of Elements	1000

Load Stepping	
Scheme	Auto
N_E (Elastic Steps)	10
N_P (Plastic Steps)	12
β (Step Control)	5

Mesh	
Mesh Adaptivity	Yes
Adaptive Iterations	3
Adaptivity Frequency	3
Start Elements	1000
Adaptivity Control	Shear Dissipation

Figure 1.7: Stage settings for Multiplier Elastoplastic analysis with mesh adaptivity. The Time Scope is irrelevant for the Tresca model.

In the following, we apply a multiplier load of 600 kN/m^2 (such that a multiplier $\alpha = 1$ corresponds to the state arrived at in the previous analysis). All other parameters are left at their default values except that the No of Elements is set to 1,000 and Mesh Adaptivity is used, again with default values. The Adaptivity Frequency ($= 3$) here indicates that the mesh is adapted in load steps 1, 4, 7, etc. The specification of initial stresses follows that of the previous Elastoplastic analysis. No From stage is specified, implying that the initial stresses will be calculated automatically. For further details on Multiplier Elastoplastic analysis, please refer to the Analysis Manual.

The results of the analysis in terms of the displacement, stress, etc versus load multiplier can be plotted using the XY Plots tool located in the Results ribbon. In order to specify a point at which to collect such data during the analysis, the Result Point tool located in the Features ribbon can be used. In this case, a Result Point is defined (prior to running the analysis) at the top left corner of the foundation (see Figure 1.8).

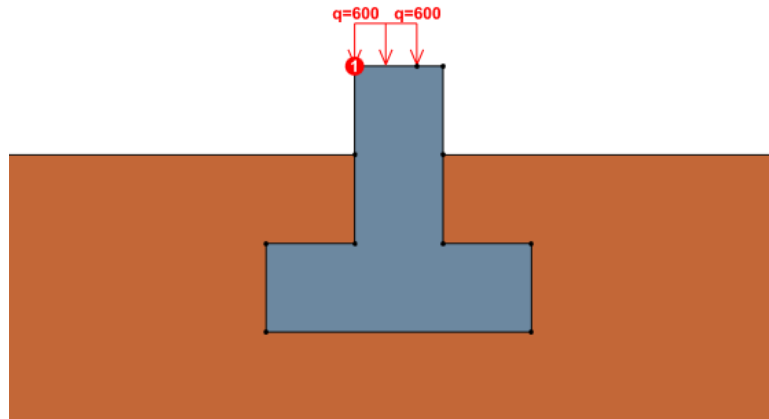


Figure 1.8: Setup for Multiplier Elastoplastic analysis: Multiplier Distributed Load of 600 kN/m² and Result Point located at the top left corner of the foundation (only a section of the full problem domain is shown).

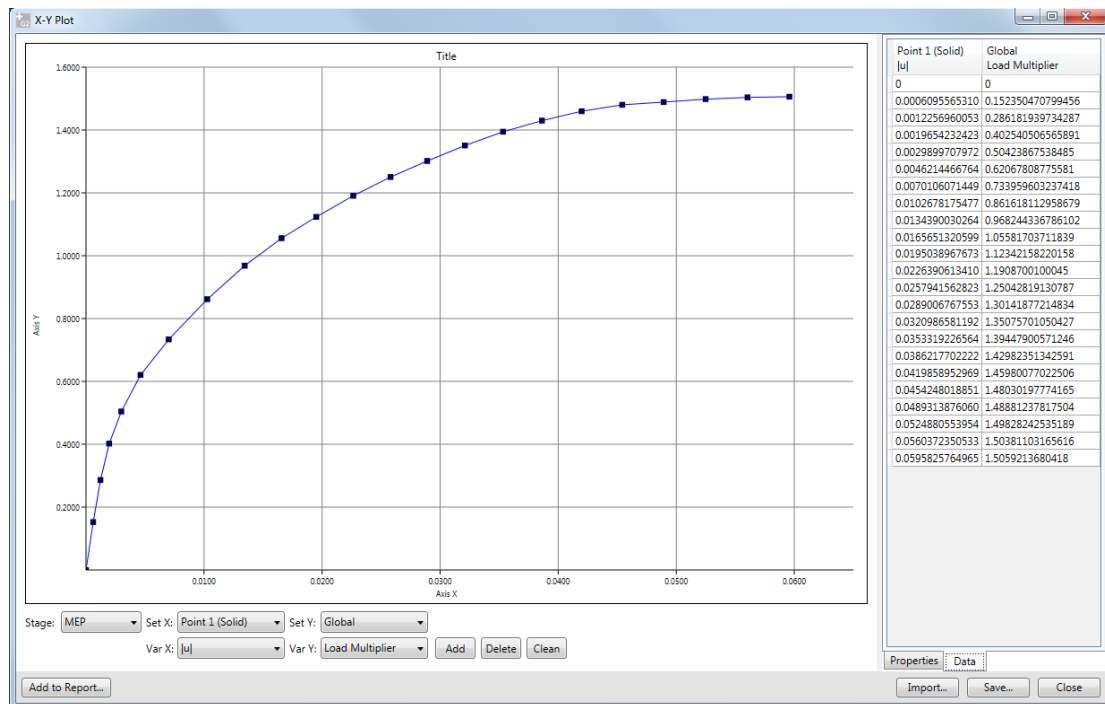


Figure 1.9: Load-displacement curve created by the XY Plots tool. The actual data can be accessed via the Data tab that appears in the right bottom corner when the curve is selected.

Using the XY Plots tool, the displacement $|u| = \sqrt{u_x^2 + u_y^2}$ is plotted as function of the load multiplier as shown in Figure 1.9. We note that the result previously found by means of Elastoplastic analysis (using a single load step), $|u| = \sqrt{0.006^2 + 0.0138^2} = 0.01477$ is in good agreement with the result of the Multiplier Elastoplastic analysis (which uses 8 load steps to reach a load multiplier of 1 versus only a single step in the previous analysis).

Similarly, the final load multiplier of around 1.5, corresponding to a total load of $1.5 \times 600 = 900$ kN/m², is in good agreement with the results of the Limit Analyses ($q_u = 895$ kN/m² \pm 3.9%).

1.4 Variation of undrained shear strength with depth

The use of a constant undrained shear strength is often a rather crude approximation to reality where one will usually observe an increase of shear strength with depth. In OptumG2, linear variations of all parameters can be specified via the righthand side icon that appears when any parameter field is selected (see Figure 1.10).

In the following, a shear strength varying from $s_u = 15$ kPa at the top surface (at level of $y = 16$ m) and increasing by 5 kPa/m with depth is used. Such a variation is can be defined using the Material Parameter dialog shown in Figure (1.10).

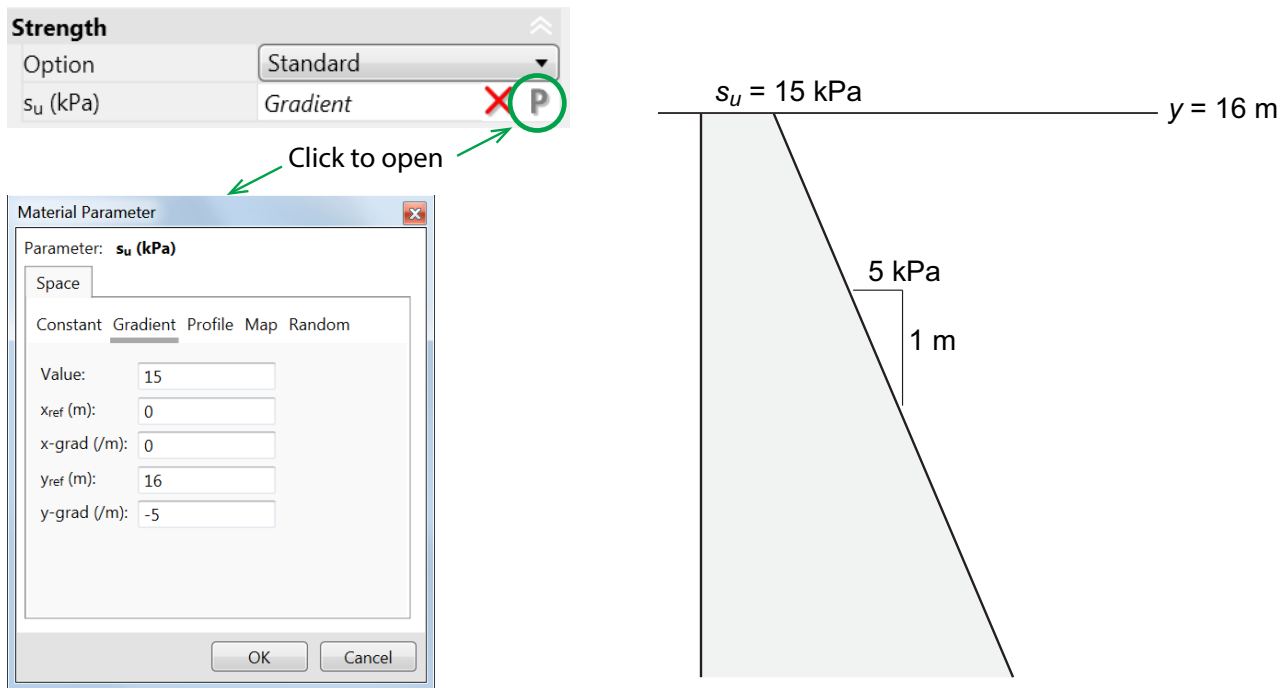


Figure 1.10: Specification of linear distribution of s_u .

Running upper and lower bound limit analysis for this problem gives:

$$q_u = 833.5 \pm 3.5\% \text{ kN/m}^2 \quad (1.7)$$

as compared to the value of $q_u = 895.0 \text{ kN/m}^2$ for a constant $s_u = 30$ kPa.

Finally, as a check that the correct distribution of s_u has been specified, the distribution of all material parameters can be visualized under Results (see Figure 1.11).

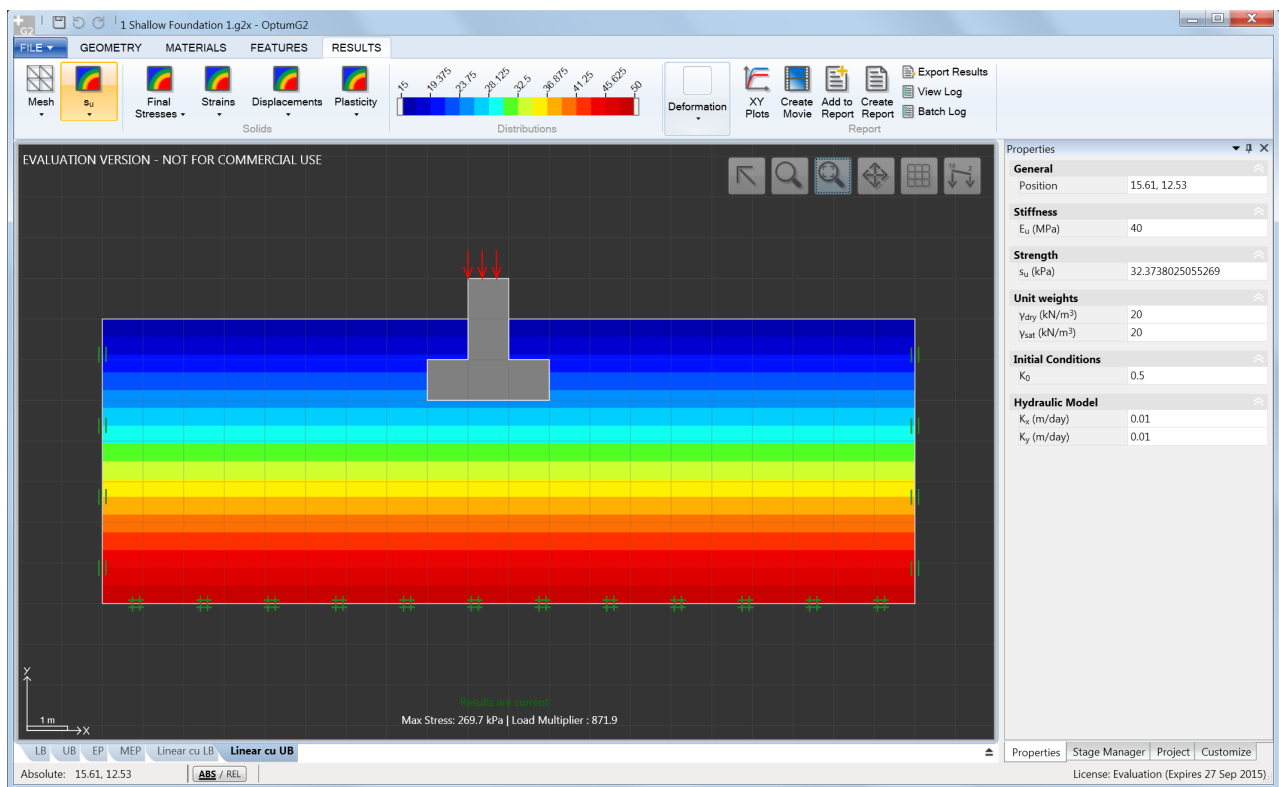


Figure 1.11: Variation of s_u .