UPDATED MESH ANALYSIS OF A CANTILEVER BEAM

The range of problems with known analytical solutions involving large displacement effects, which could be used to test the large displacement options in PLAXIS, is very limited. However, the large displacement elastic bending of a cantilever beam is well suited as a large displacement benchmark problem since a known analytical solution exists (Mattiasson, 1981).

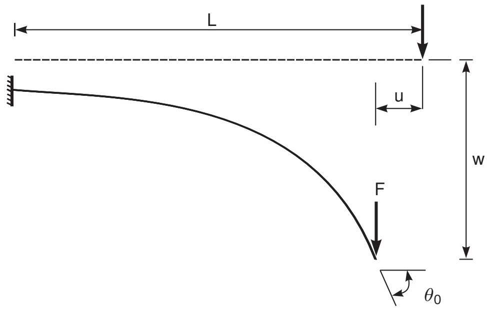

Figure 1 True deformation of elastic cantilever beam

Geometry non-linearity is of major importance in problems involving slender structural members like beams, plates and shells. Phenomena like buckling and bulging cannot be described without considering geometry changes. However, soil bodies are far from slender structures and consequently, most finite element formulations disregard changes in geometry. This also applies to conventional PLAXIS calculations. Despite the fact that most practical cases imply very small changes in geometry, in some particular cases the changes in geometry may be significant. Users should check calculation results by considering the truly deformed mesh.

For special problems of extreme large deformation an *Updated mesh analysis* is needed. For details on the implementation the reader may refer to the PhD thesis by Van Langen (1991). An *Updated mesh analysis* in PLAXIS is based on the *Updated Lagrangian formulation* as described by McMeeking & Rice (1975).

Used version:

- PLAXIS 2D - Version 2018.0

Geometry: The *Updated mesh analysis* is related to the calculation of the horizontal and vertical tip displacement of the cantilever beam illustrated in Figure 1. Two different ways of formulation are used for the cantilever beam:

- The beam is modelled by use of beam elements with thickness of 0.01 m.
- The beam is modelled as a soil layer composed of triangular elements with thickness of 0.01 m.

For a more efficient model both cases are considered in a single project. The model geometry is presented in Figure 2. The length of the beam equals 1 m. A soil cluster of 1 m $\times$ 1 m is used to generate the mesh elements. Note that the properties of the material assigned to the soil below the beam do not affect the results as this cluster will be deactivated in the calculation phases.

In case that the beam is composed of beam elements, a point load equal to 1 kN/m is used acting at the right edge (tip) of the beam. The left edge is clamped by use of a point displacement with fixed rotation and translations.

In case that the beam is composed of soil elements, a vertical line load is used at the right side, over the height of the beam. Due to the fact that load's length equals 0.01 m, its magnitude is set equal to 100 kN/m/m to simulate the same boundary conditions as above. The left side of the beam is clamped by use of a vertical line displacement with fixed translations. Its length equals 0.01 m as well.

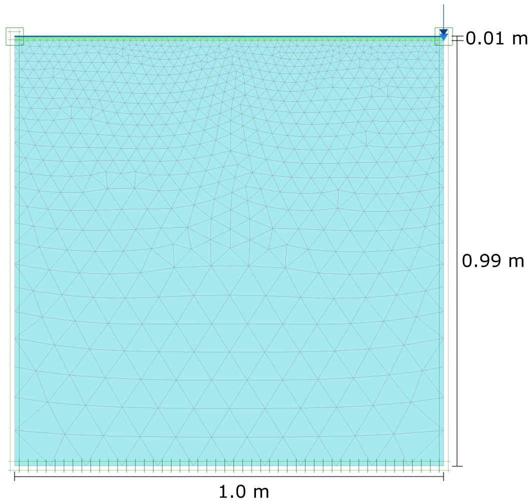

Figure 2 Model geometry and finite element mesh

Materials: The adopted material parameters for the cantilever beam are:

Soil:	Linear elastic	Drained	E'=10^6 kN/m^2	ν'=0.0
Plate:	Elastic (isotropic)	EA=10^4 kN/m^2	EI=0.08333 kN/m^2/m	ν=0.0

Meshing: The *Medium* option is selected for the *Element distribution*. The mesh is locally refined with a *Coarseness factor* of 0.1 at the line in which the plate material is assigned (top boundary of the model). The generated mesh is shown in Figure 2.

Calculations: In the Initial phase, zero initial stresses are generated by using the *K0 procedure* (γ = 0). A new calculation phase is introduced (Phase 1) and the *Calculation type* is set to *Plastic analysis*. In this phase all soil clusters are deactivated and the plate representing the cantilever is activated, together with the point load and the point displacement. Starting from the Initial phase, another calculation phase is introduced (Phase 2) and its *Calculation type* is set to *Plastic analysis* as well. In Phase 2, only the top soil layer is activated (representing the cantilever beam), while the bottom soil layer is deactivated. Plate elements are kept deactivated as well. The line load and the line displacement are activated for this phase.

For both Phases 1 and 2, the *Reset displacements to zero* is selected and the *Tolerated error* is set to 0.001. The *Updated mesh* option is selected in the *Deformation control parameters* window. The *Max steps* options is set equal to 1000 and 2000 in Phase 1 and 2 respectively. The boundary conditions (*Deformations* option in *Model conditions*) are set to inactive for both Phases 1 and 2.

Output: The resulting deformed mesh (true scale) is presented in Figure 3 for the cantilever formed of beam and soil elements respectively. The maximum reached value of deformation $|u|$ at the tip of the cantilever is 1.019 m in both cases.

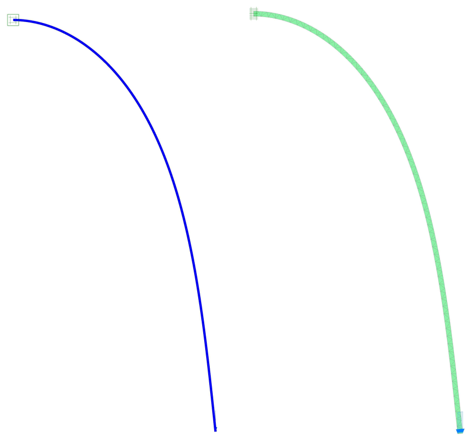

Figure 3 Deformed mesh (true scale) in case of beam elements (left) and soil elements (right)

Verification: The analytical solution presented by Mattiasson (1981) is compared to PLAXIS results. The vertical w and horizontal u displacement of the cantilever tip are normalized over the length L and plot against the applied load F, normalized over the factor EI/L^2. Load-displacement curves are presented in Figures 4 and 5. It is concluded that PLAXIS results, considering both soil and beam elements, are in close agreement with the analytical solution.

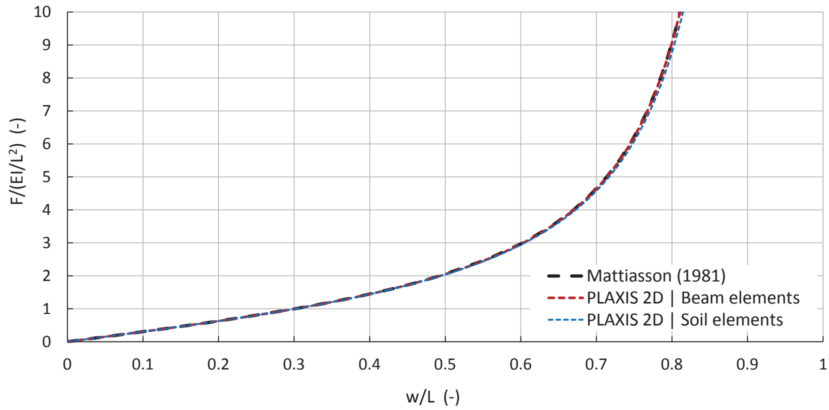

Figure 4 Variation of normalized load with vertical displacement

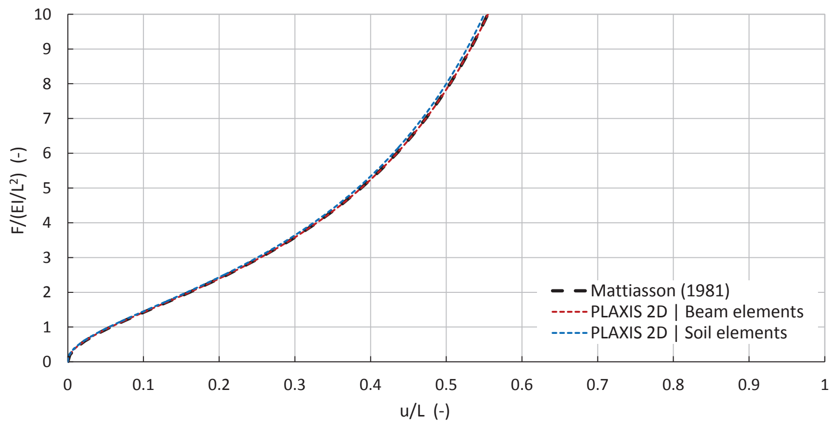

Figure 5 Variation of normalized load with horizontal displacement

REFERENCES

[1] Mattiasson, K. (1981). Numerical results from large deflection beam and frame problems analyzed by means of elliptic integrals. Int. J. Numer. Methods Eng., 17, 145–153.

[2] McMeeking, R.M., Rice, J.R. (1975). Finite-element formulations for problems of large elastic-plastic deformation. Int. J. Solids Struct., 11, 606–616.

[3] Van Langen, H. (1991). Numerical Analysis of Soil-Structure Interaction. Phd thesis, Delft University of Technology. Plaxis users can request copies.