PART I: TASPLAQ MODULE

I.1. INTRODUCTION....7

I.2. THEORETICAL ASPECTS....8

I.2.1. Preliminary notions....8
I.2.1.1. Finite element modelling of a plate....8
I.2.1.2. Generalised use of the Boussinesq model....8

I.2.2. Formulation....9
I.2.2.1. General principle of the method....9
I.2.2.2. Modelling the plate....10
I.2.2.3. Modelling Soil/Plate interaction....12
I.2.2.4. Modelling of supporting soil....13
I.2.2.5. Formulation of problem and results....14

I.2.3. Processing of certain special cases....15
I.2.3.1. Unsticking of plate....15
I.2.3.2. Limitation of soil reaction....15
I.2.3.3. Taking account of an outside load applied to the soil....16
I.2.3.4. Taking account of an initial stress in the soil....16

I.2.4. Taking account of symmetries....16
I.2.4.1. Case of symmetry on one or two planes....17
I.2.4.2. Axisymmetry (or symmetry with respect to an axis)....17
I.2.4.3. Plane deformations....18

I.2.5. Validity limits....19
I.2.5.1. Validity of the Kirchhoff model....19
I.2.5.2. Validity of the Boussinesq model....19
I.2.5.3. Special case of a surface base layer....19

I.3. USER'S MANUAL....20

I.3.1. Notations and units....20
I.3.1.1. Notations and sign conventions....20
I.3.1.2. Units....20

I.3.2. "Parameters" tab....20
I.3.2.1. "General mode" frame....21
I.3.2.2. "Framework" frame....21
I.3.2.3. "Plate support interface" frame....22
I.3.2.4. "Advanced parameters" frame....23

I.3.3. "Layers" tab....24
I.3.3.1. "Soil layers definition" frame....24
I.3.3.2. "External loads on soil" frame....26

I.3.4. "Plate" tab....35

I.3.4.1. Definition of mechanical characteristics of plate 35
I.3.4.2. "Circular plate" and "User-defined plate" wizards 36
I.3.4.3. "Equivalent parameters of a combined section" wizard 38
I.3.5. "Deactivation" tab 40
I.3.5.1. Definition of deactivated zones 40
I.3.5.2. "Deactivation" wizard 41
I.3.6. "Distributed loads" tab 43
I.3.6.1. Distributed loads in the case of 3D projects 43
I.3.6.2. Distributed loads in the case of 2D projects 44
I.3.6.3. Load definition wizards 45
I.3.7. "Linear loads" tab 47
I.3.7.1. Linear loads and stiffnesses in the case of 3D projects 47
I.3.7.2. Linear loads and stiffnesses in the case of 2D projects 48
I.3.8. "Point loads" tab 49
I.3.8.1. "Point loads" wizards 50
I.3.9. "Unsticking/Plastification" tab 53
I.3.10. "Mesh" tab 54
I.3.10.1. "General parameters" frame 54
I.3.10.2. "Refinements" frame 55
I.3.10.3. "User-defined crossover points" frame 55
I.3.11. Calculation and results 56
I.3.11.1. Calculation 56
I.3.11.2. Available results for 3D projects 57
I.3.11.3. Available results for 2D projects 69
I.3.11.4. Export to Tasseldo external wizard 73
I.4. CALCULATION EXAMPLES 74
I.4.1. Example 1: Slab on rigid supports 74
I.4.1.1. Step 1: Slab supported by its two opposite edges 74
I.4.1.2. Step 2: Plate supported by its 4 edges 86
I.4.1.3. Comparison of the results 88
I.4.2. Example 2: Locally loaded rafts 91
I.4.2.1. Step 1: Single raft 91
I.4.2.2. Step 2: Two identical neighbour rafts 106
I.4.3. Example 3: Slab with different systems of joints 112
I.4.3.1. Step 1: Continuous slab laying on multi-layered elastic soil 112
I.4.3.2. Step 2: Slab with partial joints laying on a multi-layered elastic soil 122
I.4.3.3. Step 3: Slab with full joints laying on a multi-layered elastic soil 126
I.4.3.4. Comparison of the three cases 129
I.4.4. Example 4: L-Shaped raft with linear loading and interaction with an embankment 133
I.4.4.1. Step 1: L-Shaped raft with linear loading 133
I.4.4.2. Step 2: Influence of a neighbouring embankment 140
I.4.5. Example 5: Settlements under a circular tank 143
I.4.5.1. Step 1: Construction stage of the tank 144
I.4.5.2. Step 2: "Hydrotest" stage 151
I.4.6. Example 6: Girder cranes and setting of the subgrade reaction coefficient 155
I.4.6.1. Step 1: Locally loaded footing with the consideration of the dead load 155

I.4.6.2. Step 2: Locally loaded footing – definition of the equivalent subgrade reaction coefficient....164
I.4.7. Example 7: 2D calculation of a shallow foundation....167
I.4.7.1. Data input....167
I.4.7.2. Graphical results - Cross-sections....171

TABLE OF FIGURES

Figure I.1 : Schematic diagram 7
Figure I.2 : Application of Steinbrenner's formula 9
Figure I.3 : Principle of the method, system elements 10
Figure I.4 : Finite element plate modelling 10
Figure I.5 : Modelling of holes 11
Figure I.6 : Modelling of a polygonal shape 11
Figure I.7 : Simulation of expansion joints 11
Figure I.8 : Modelling of several separated plaques 11
Figure I.9 : Plate equilibrium 12
Figure I.10 : Distribution of soil reaction under the plate 13
Figure I.11 : Calculation of influence coefficients in the case of a non-horizontal multilayer 14
Figure I.12 : Plate unsticking - Example 15
Figure I.13 : Taking account of an outside load applied directly to the soil 16
Figure I.14 : Taking account of an initial surface stress - Example 16
Figure I.15 : Schematic representation of the symmetry condition with respect to a plane 17
Figure I.16 : Kirchhoff model for circular plates with "annular" elements having 2 nodes and 4 degrees of freedom 17
Figure I.17 : Discretisation into beam elements of 2 nodes and 4 degrees of freedom 18
Figure I.18 : "Parameters" tab 21
Figure I.19 : Help diagram for defining plate coordinates 22
Figure I.20 : Help diagram for choice of symmetry 22
Figure I.21 : "Layers" tab: case of a 3D model 24
Figure I.22 : Help diagram for layer dip along x 25
Figure I.23 : Help diagram for layer dip along y 25
Figure I.24 : Plate: Overall layout of the problem {Plate + Soil + Ext. loads} 26
Figure I.25 : Help diagram for defining external loads in 3D 26
Figure I.26 : Help diagram for definition of external loads in the case of a 2D project with plane deformations 27
Figure I.27 : Help diagram for definition of external loads in the case of an axisymmetric 2D project 27
Figure I.28 : Examples of display of an external load on the soil 28
Figure I.29 : Automatic loads (wizards) 29
Figure I.30 : Wizard: "Uniform circular load" 30
Figure I.31 : "Layers" tab – External loads – After "Uniform circular load" wizard 30
Figure I.32 : Wizard: "Uniform annular load" 31
Figure I.33 : "Layers" tab – External loads - After "Uniform annular load" wizard 32
Figure I.34 : Example of 3D embankment type load 32
Figure I.35 : Wizard: "3D embankment-like load" 33
Figure I.36 : "Layers" tab – External loads – After "3D embankment-like load" wizard 34
Figure I.37 : Loads wizard for Tasplaq – Example of modification of group n°1 35
Figure I.38 : "Plate" tab 35
Figure I.39 : Circular plate wizard 37
Figure I.40 : Wizard for a user-defined plate 38
Figure I.41 : Plate wizard for Tasplaq – Example of modification of group n°2 38
Figure I.42 : Wizard: Equivalent parameters of a combined section 39
Figure I.43 : "Deactivation" tab 40
Figure I.44 : Circular deactivation wizard 41
Figure I.45 : User-defined deactivation wizard 42
Figure I.46 : Deactivation wizard for Tasplaq – Example of group modification n°3 42
Figure I.47 : "Distributed loads" tab – 3D projects 43
Figure I.48 : "Distributed loads" tab – 2D projects 44

Figure I.49 : Distributed loads wizard 45
Figure I.50 : "Circular distributed load" wizard 46
Figure I.51 : "User-defined distributed load" wizard 47
Figure I.52 : "Linear loads" tab – 3D project 48
Figure I.53 : "Linear loads" tab – 2D project 49
Figure I.54 : "Point loads" tab 50
Figure I.55 : Wizard for point loads along a segment 51
Figure I.56 : Wizard for distributed loads on a horizontal rectangle 52
Figure I.57 : Case of a pate partially supported by the soil 53
Figure I.58 : "Unsticking/plastification tab" 53
Figure I.59 : "Mesh" tab 54
Figure I.60 : Calculation in progress/done window 57
Figure I.61 : Results window – 3D project 57
Figure I.62 : Formatted numerical results – 3D project 59
Figure I.63 : Formatted numerical results: Reaction in the surface supports 60
Figure I.64 : Numerical results: Results table 61
Figure I.65 : Cross-section – 3D project – Plate deflection – Example of cross-section along x 63
Figure I.66 : Cross-section – 3D project – Soil settlement – Example of cross-section along y 64
Figure I.67 : Cross-section – 3D project – Soil reaction – Example of cross-section along x 64
Figure I.68 : Shading: Modify bounds of shading 65
Figure I.69 : Shading – 3D project – Moment M_x 65
Figure I.70 : Shading – 3D project – Soil settlement 65
Figure I.71 : 3D Chart – Plate deflection 66
Figure I.72 : 3D Chart – Moment M_x 66
Figure I.73 : Data dishomogenisation wizard 67
Figure I.74 : Data dishomogenisation wizard: results 68
Figure I.75 : Results window – 2D project 69
Figure I.76 : Formatted numerical results – Example for a 2D project in plane deformations 69
Figure I.77 : Numerical results: Results table – 2D projects in plane deformation 70
Figure I.78 : Numerical results: Results table – Axisymmetric 2D projects 71
Figure I.79 : Cross-section: example of 2D axisymmetric project – Plate deflection/soil settlement 72
Figure I.80 : Cross-section: example of 2D project in plane deformation - M_x Moment 72

TABLE OF TABLES

Table I.1 : Notations and sign conventions 20
Table I.2 : Units 20
Table I.3 : "General setting" tab: "Framework" box 22
Table I.4 : "Parameters" tab: "Plate/Support interface" box 23
Table I.5 : "Parameters" tab: "Advanced parameters" frame 24
Table I.6 : Soil layer parameters 24
Table I.7 : Definition of the initial surface stress 25
Table I.8 : Parameters for defining external loads on the soil in the case of a 3D model 27
Table I.9 : Parameters for defining external loads on the soil in the case of a 2D model 28
Table I.10 : Parameters for "Uniform circular load" wizard 29
Table I.11 : Parameters for the "Uniform annular load" wizard 31
Table I.12 : Parameters for "3D Embankment-like load" wizard 34
Table I.13 : Definition of mechanical characteristics of the plate 36
Table I.14 : Parameters for the "Deactivation circular" wizard 36
Table I.15 : Parameters for "User-defined plate" wizard 37
Table I.16 : Parameters of "Combined section" wizard 39
Table I.17 : Deactivation parameters 40
Table I.18 : Parameters for the "Circular deactivation" wizard 41
Table I.19 : Parameters for the "User-defined deactivation" wizard 42
Table I.20 : Parameters for distributed loads in 3D 43
Table I.21 : Parameters for distributed loads in 2D 44
Table I.22 : Parameters for uniform circular load 46
Table I.23 : Parameter for a user-defined distributed load 46
Table I.24 : Parameters for linear loads in 3D 48
Table I.25 : Parameters for linear loads in 2D 49
Table I.26 : Parameters for point loads 50
Table I.27 : Parameters for the point loads along a segment wizard 51
Table I.28 : Parameters for the point loads for a distributed on a horizontal and rectangular load 52
Table I.29 : Parameters for manual definition of unsticking/plastification of the plate 54
Table I.30 : Parameters for "Along X axis" and "Along Y axis" mesh refinement 55
Table I.31 : Parameters for "Along X and Y axes" mesh refinement 55
Table I.32 : Parameters for "Along X axis" and "Along Y axis" user-defined crossover 56
Table I.33 : Parameters for "Along X and Y axes" user-defined crossover 56
Table I.34 : Details of numerical results (settlement and node reaction) 62
Table I.35 : Details of numerical results (plate deflection) 62
Table I.36 : Details of numerical results (bending moment) 63
Table I.37 : Dishomogenisation wizard: "Lower layer" frame 67
Table I.38 : Details of numerical results for the plate (2D projects) 71
Table I.39 : Details of numerical results - Settlement and soil reaction (2D projects) 71
Table I.40 : Details of numerical results - Bending moments (2D projects in plane deformation) 71
Table I.41 : Details of numerical results – Bending moments (axisymmetric 2D projects) 71

I.1. INTRODUCTION

The TASPLAQ module is used for simplified three-dimensional modelling of one or more plates on an elastic supporting soil.

The plate, with linear elastic behaviour, represents an extended foundation element such as a basemat or slab, with irregular inertia and geometry and subject to application of heterogeneous loadings: distributed loads, point loads, moments, etc.

The initial behaviour of the plate support is assumed to be elastic. By default, it is compared to an elastic multilayer medium (horizontal or angled layers, each characterized by a Young's modulus and a Poisson's ratio). The plate can also be supported by a distribution of springs: surface, linear or point, in translation or rotation.

The "support / plate" contact condition authorizes surface unsticking/plastification according to a predetermined criterion.

Plate deflection, settlement and reaction at all points can be obtained using this method, along with the forces in the plate (moments and shear forces).

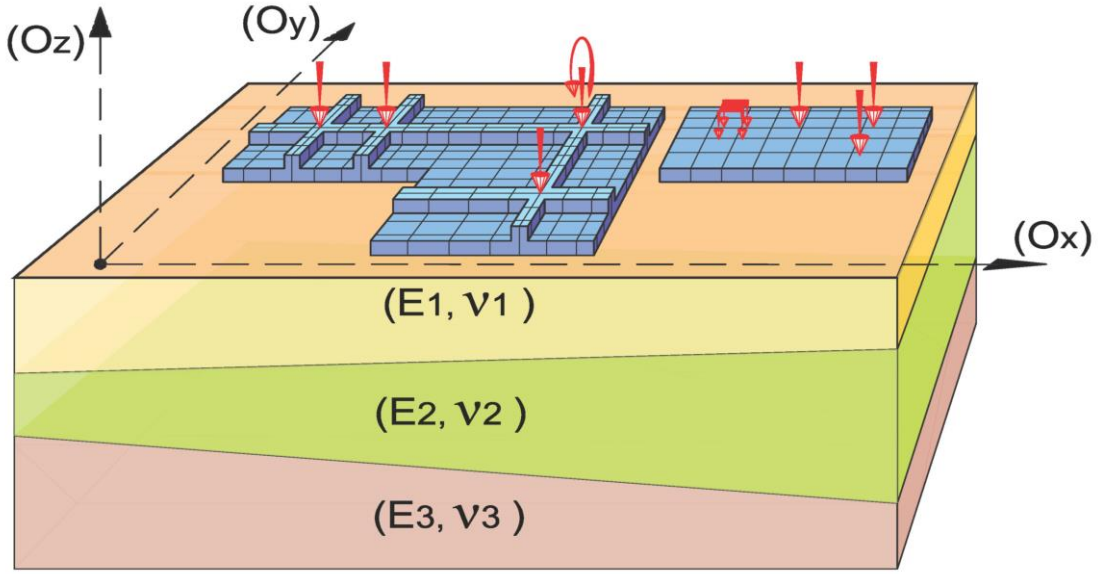


Figure I.1 : Schematic diagram

I.2. THEORETICAL ASPECTS

I.2.1. Preliminary notions

I.2.1.1. Finite element modelling of a plate

The plate is assumed to be homogeneous and isotropic, working in bending alone. We will simply deal with the case of plates which are thin by comparison with the other dimensions, which enables us to adopt the conventional thin plates theory, more precisely the "Kirchhoff discrete model". In the context of said model, only bending strains are taken into account: the contribution of shear strains is ignored.

The plate is assumed to have linear elastic behaviour, the combination of equilibrium equations and behavioural laws thus leads to the following equation, known as the Lagrange equation:

$$\frac{\partial^4 w}{\partial x^4} + 2\frac{\partial^4 w}{\partial x^2 \partial y^2} + \frac{\partial^4 w}{\partial y^4} = q(x,y)\frac{(1-\nu^2)}{EI} \qquad \textbf{(1)}$$

where w refers to plate deflection, q the load density applied to the plate, EI the product of inertia and ν the Poisson's ratio.

This equation can be resolved by introducing a finite element formulation. We choose a "displacements" type model, with rectangular elements with four nodes and 12 degrees of freedom (Zienkiewicz O.C., 1991). Each node thus has three degrees of freedom: one displacement and two rotations. These rotations are simply the first partial derivatives of the deflection (Kirchhoff approximation).

It should be noted that this model is characterised by its simplicity and its reliable and rapid convergence.

I.2.1.2. Generalised use of the Boussinesq model

The soil is compared to an elastic multilayer medium. It is true that all geotechnical engineers know full well that soil is not an elastic material, but Terzaghi himself admitted that when the stresses were below one-third of the limit values, the soil could be considered with sufficient confidence to have elastic behaviour.

Once the elastic behaviour of the soil is accepted, the Boussinesq formulas can be used to calculate surface settlements. It should be recalled that the Boussinesq model was initially established for the case of a homogeneous semi-infinite medium. This model gives us a stress profile that is assumed to be valid for a multilayer case. We therefore accept that the stress distributions are identical in both cases.

Several research projects and approaches (Burland 1977) conducted an in-depth discussion of the conditions of this simplification and concluded that it was in fact acceptable in the majority of situations commonly encountered by the engineer; one significant exception being the case of a stiff layer above more deformable layers.

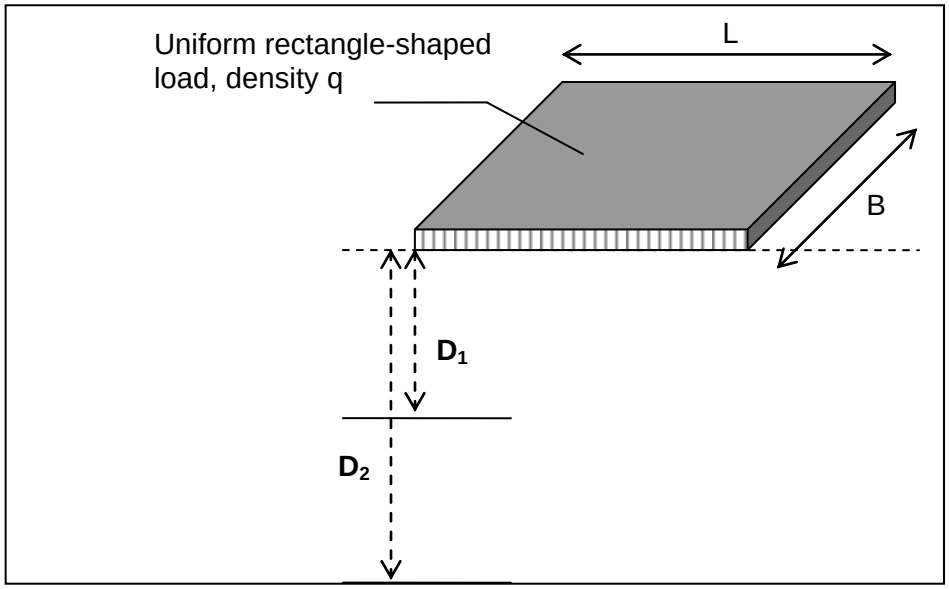
The settlements calculation will be based mainly on the applicable superposition method in the elastic model, as well as on the Steinbrenner formula (3D elastic calculation also used in the Tasseldo module in Foxta) which is derived from the Boussinesq formulas (Terzaghi, 1943). This formula can be used to express settlement, under the corner of a uniform rectangle-shaped load, of a layer situated between depths D1 and D2, with Young's modulus E and Poisson's ratio ν :

$$\rho_{D1-D2} = \frac{qB}{E}\left((1-\nu^2)\left(F_1(d_2)-F_1(d_1)\right)+(1-\nu-2\nu^2)\left(F_2(d_2)-F_2(d_1)\right)\right) \qquad \textbf{(2)}$$

Where:

$$\begin{cases} F_1(d) = \dfrac{1}{\pi}\left(l \ln\left(\dfrac{\left(1+\sqrt{l^2+1}\right)\sqrt{l^2+d^2}}{l\left(1+\sqrt{1+d^2+l^2}\right)} \right) + \ln\left(\dfrac{\left(l+\sqrt{l^2+1}\right)\sqrt{1+d^2}}{\left(l+\sqrt{1+d^2+l^2}\right)} \right) \right) \\ F_2(d) = \dfrac{d}{2\pi} \arctan\left(\dfrac{l}{d\sqrt{1+d^2+l^2}} \right) \qquad \text{with} \quad d = \dfrac{D}{B} \text{ and } l = \dfrac{L}{B} \end{cases} \tag{3}$$

By adding the settlements produced in each layer and by combining with the algebraic superposition method, it is possible to evaluate the settlement induced by one or more rectangle-shaped loads, at all points on the surface of an elastic multi-layer soil.



Uniform rectangle-shaped load, density q

Figure I.2 : Application of Steinbrenner's formula

I.2.2. Formulation

The problem we propose dealing with comprises three unknowns: vertical displacement of the plate, settlement and soil reaction. The Lagrange equation (1) can be used to link plate displacement to soil reaction, a second relation is then obtained by writing the equality between the vertical displacement of the plate and the soil settlement. Finally, applying the Boussinesq model links settlement to soil reaction.

Introducing finite element discretisation for the plate allows a simple matrix formulation of the problem to be resolved.

I.2.2.1. General principle of the method

The proposed method thus consists in coupling three approaches:

- a finite element formulation for the plate;
- discretisation appropriate to the mesh, for the interface pressures;
- application of the Boussinesq formulas for calculation of supporting soil displacements.

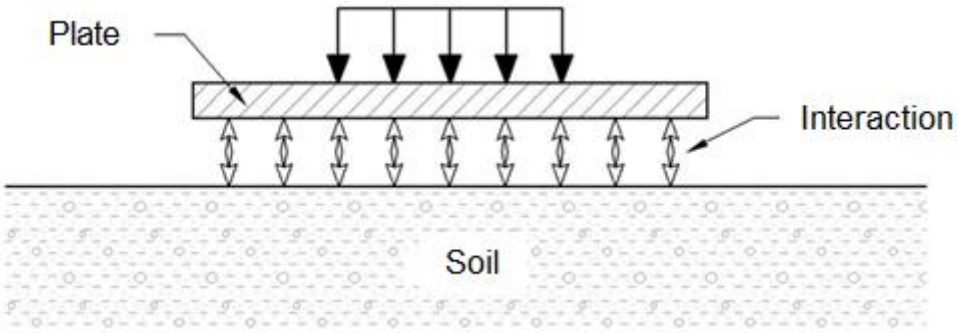


Figure I.3 : Principle of the method, system elements

The principle of the method is to discretise the plate into rectangular finite element. If we accept that the soil reaction is uniformly distributed around each node, we can then express the settlements at the nodes as a function of the soil reaction, using an "influence matrix" which can be calculated on the basis of the characteristics of each layer. The final formulation of the problem is then obtained by writing the equality at each node between soil settlement and plate vertical displacement.
The method was perfected by introducing surface soil plastification thresholds and verifications and by automatically taking account of unsticking and of a plate with irregular geometry.

I.2.2.2. Modelling the plate

The plate is discretised into finite element, using a rectangular mesh for which the "pitch" can vary in the two directions. The elements used are rectangular Kirchhoff elements presented earlier. Each plate element is characterised by a Young's modulus, a thickness and a Poisson's ratio.

The outside loading applied to the plate is introduced by means of vertical distributed loads (associated with each element) and point loads (vertical force + two bending moments), associated with each mesh node.

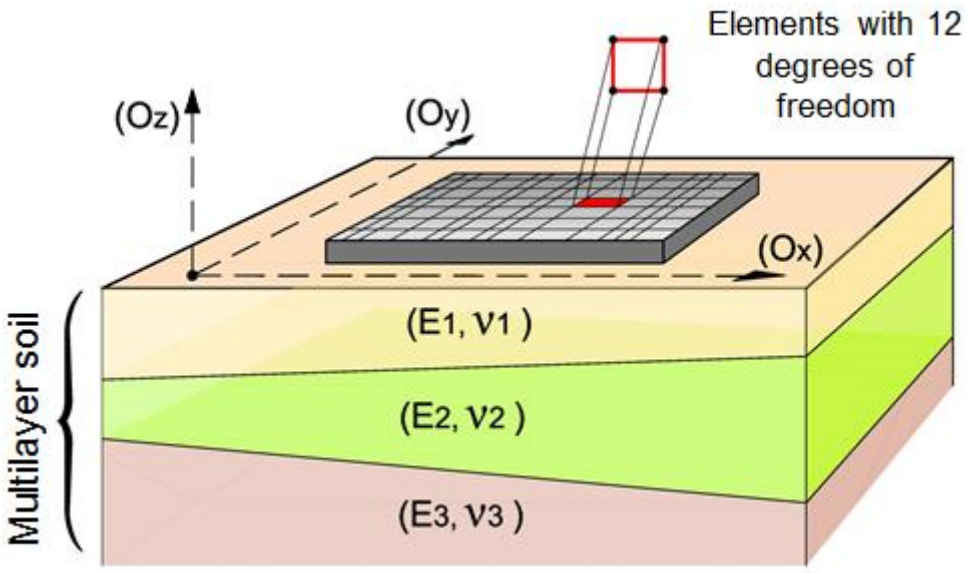


Figure I.4 : Finite element plate modelling

A deactivation technique for the plate elements was also introduced in order to extend the choice of geometry for the plate.

A few examples of possible geometries are given in figures 5 to 8.

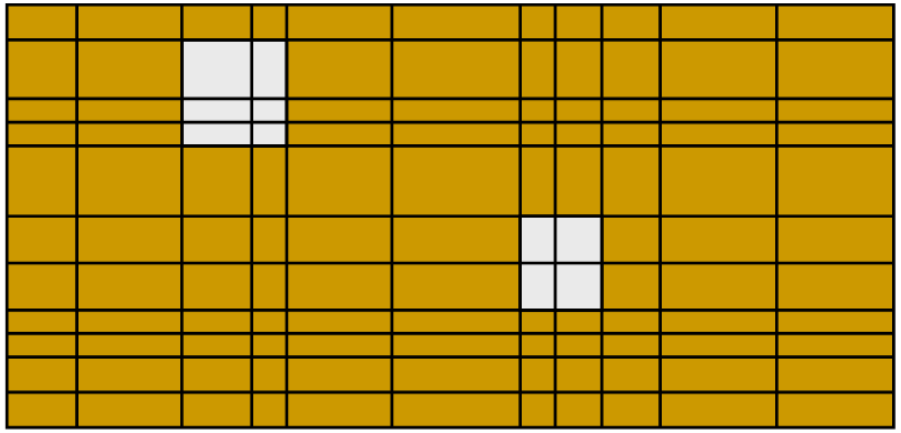


Figure I.5 : Modelling of holes

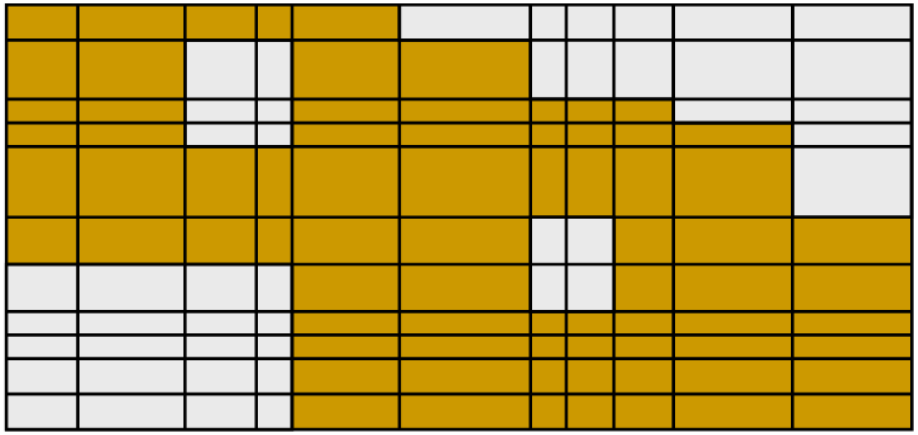


Figure I.6 : Modelling of a polygonal shape

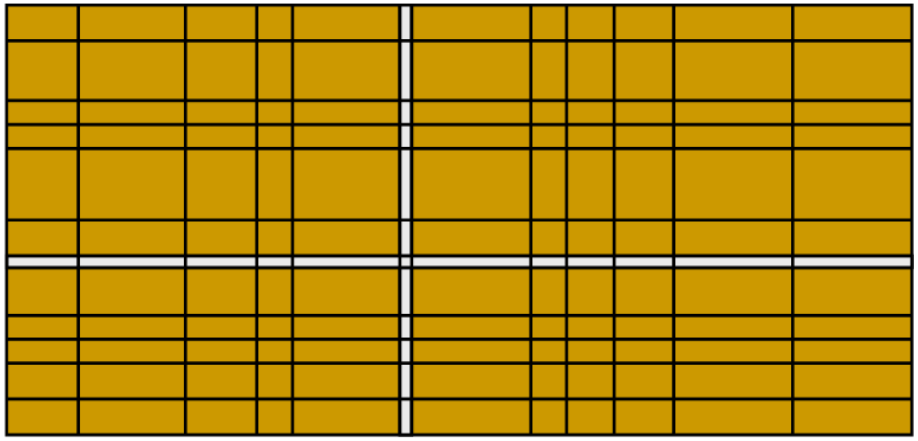


Figure I.7 : Simulation of expansion joints

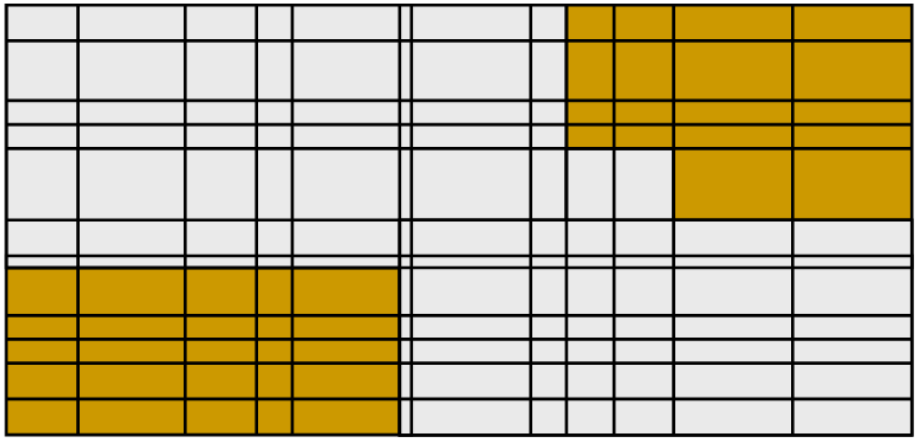


Figure I.8 : Modelling of several separated plaques

Note: The deactivated elements are light and the activated elements dark.

In the rest of this document, "n" indicates the total number of (activated) nodes on the mesh. Using this mesh, the plate equilibrium equation is represented by a linear system:

$$\underline{\underline{\mathbf{K}}}^{e}.\underline{\mathbf{a}}^{e} = \underline{\mathbf{F}}^{e} \quad \textbf{(4)}$$

- first member $\underline{\underline{\mathbf{K}}}^{e}$: plate rigidity matrix (3n x 3n), consisting of an assembly of stiffness matrices associated with each mesh element;
- second member $\underline{\mathbf{F}}^{e}$: equivalent load vector for the plate (3n x 1), constituted by assembling elementary "load vectors";
- static unknown $\underline{\mathbf{a}}^{e}$: equivalent displacement vector for the plate (3n x 1), constituted by displacement and rotations at each node.

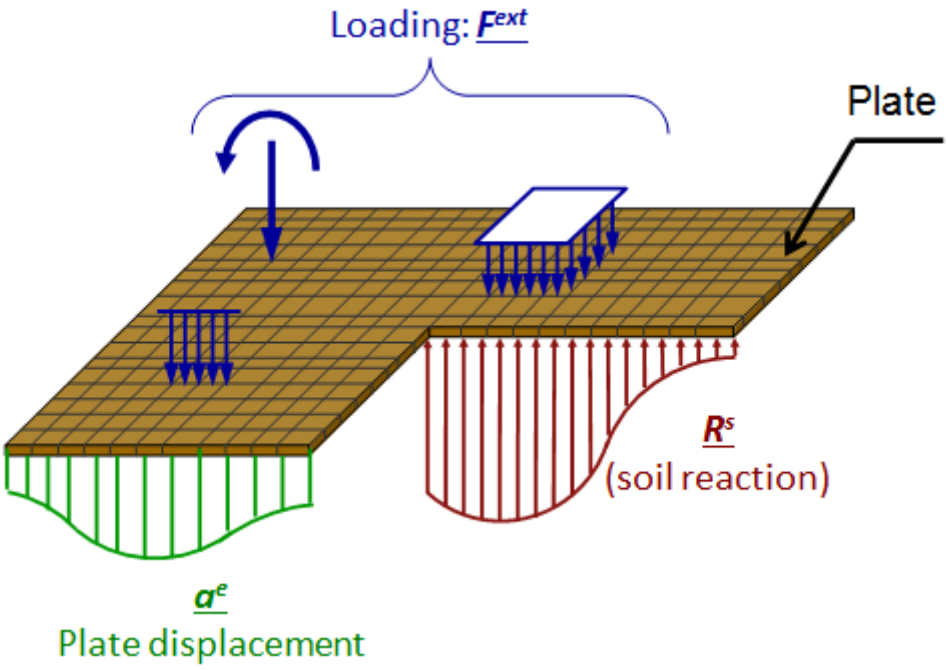


Figure I.9 : Plate equilibrium

By examining the forces balancing the plate, the plate loading vector can be broken down into two parts: a loading vector for outside forces $\underline{\mathbf{F}}^{ext}$, and a loading vector for soil reaction $\underline{\mathbf{R}}^{s}$:

$$\underline{\underline{\mathbf{K}}}^{e}.\underline{\mathbf{a}}^{e} = \underline{\mathbf{F}}^{ext} - \underline{\mathbf{R}}^{s} \quad \textbf{(5)}$$

The first is directly calculated using loads applied to the plate. The second depends on soil stiffness.

Note: The plate global stiffness matrix $\underline{\underline{\mathbf{K}}}^{e}$ already takes account of any point, linear or distributed elastic stiffnesses under the plate. This will make it possible to simulate certain special cases:

- the existence of a very stiff zone under the plate in addition to the soil (e.g. stiff inclusion, pile, etc.), deals with the case of a mixed foundation;
- run a pure strength of materials calculation of the "Plate on elastic or stiff supports" type.

I.2.2.3. Modelling Soil/Plate interaction

The assumption is contact without friction: the forces transmitted to the soil are normal to the plate.

Owing to the principle of action and reaction, the soil reaction is simply the pressure exerted by the plate on it. The basic idea is to assume that this pressure is uniform around each node. More precisely, it is uniform over a rectangle bounded by the centres of the elements connected to this

node. Account must of course be taken of the special case of a node situated on the edge or a node situated on an inside or outside corner.

For a polygonal plate, the interaction pressure distribution should thus be as schematically shown in Figure I.10.

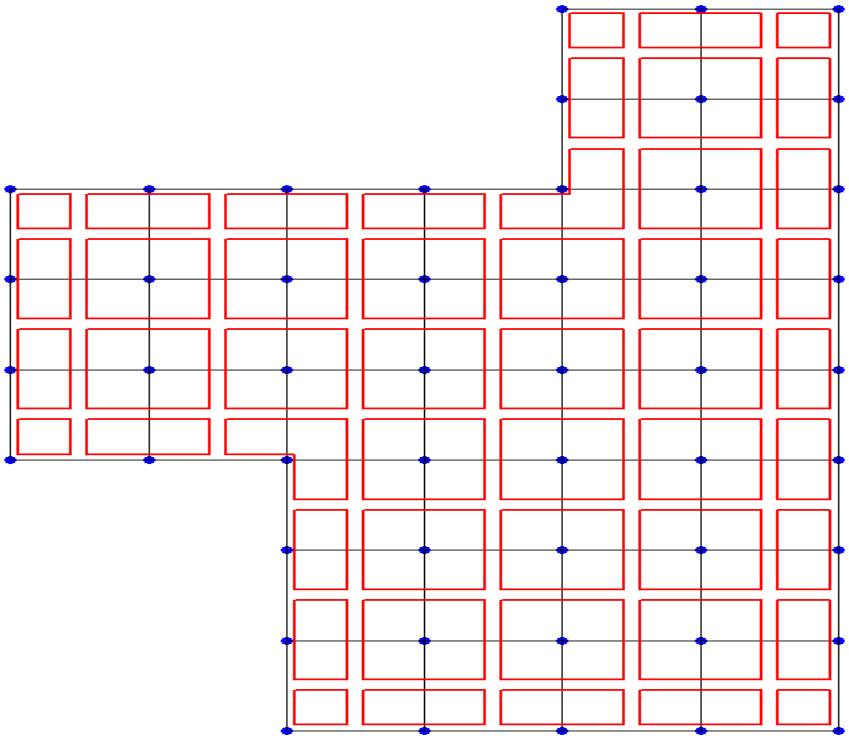


Figure I.10 : Distribution of soil reaction under the plate

I.2.2.4. Modelling of supporting soil

Using the mesh associated with the plate, we define what is known as the "influence matrix", noted $\underline{\underline{\mathbf{T}}}_{\mathbf{inf}}$ (n x n), allowing a linear link to be made between the pressure exerted on the soil and the resulting settlements:

$$\begin{pmatrix} s_1 \\ s_2 \\ \vdots \\ \vdots \\ s_n \end{pmatrix} = \begin{pmatrix} \alpha_{11} & \alpha_{12} & \cdots & \alpha_{1n} \\ \alpha_{21} & \alpha_{22} & & \vdots \\ \vdots & & \ddots & \\ \alpha_{n1} & & \cdots & \alpha_{nn} \end{pmatrix} \times \begin{pmatrix} p_1 \\ p_2 \\ \vdots \\ \vdots \\ p_n \end{pmatrix} \quad \textbf{(6)}$$

Settlements $\underline{\mathbf{S}}$ — Influence matrix $\underline{\underline{\mathbf{T}}}_{\mathbf{inf}}$ — Interaction pressures $\underline{\mathbf{P}}$

In this way the influence coefficient "α_{ij}" is calculated by examining the settlement induced at node "i" by a unit pressure applied around node "j". The calculation is made using the method presented in I.2.1. (Steinbrenner formula).

The case of a non-horizontal multilayer is dealt with by considering, for each of the influence coefficients "α_{ij}", the stratigraphy encountered at point "i" (see following figure).

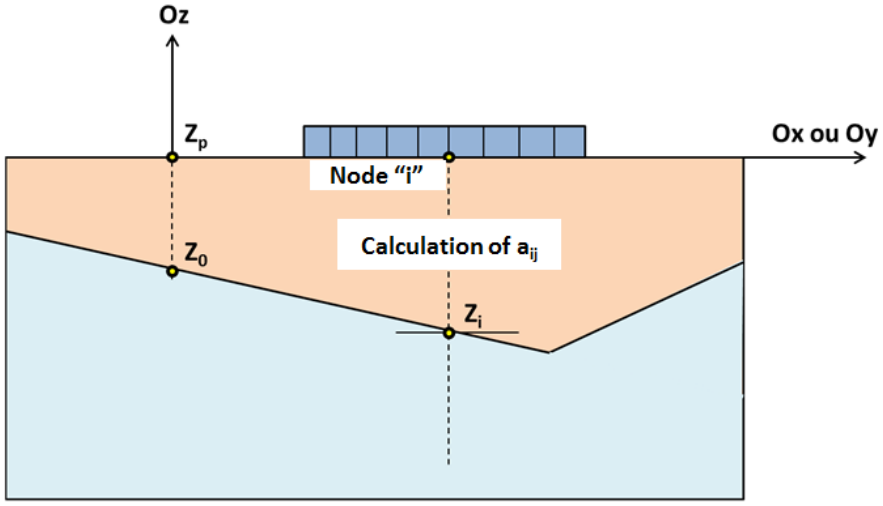


Figure I.11 : Calculation of influence coefficients in the case of a non-horizontal multilayer

I.2.2.5. Formulation of problem and results

Let us return to the plate equilibrium equation: $\underline{\underline{\mathbf{K}}}^{e}.\underline{\mathbf{a}}^{e} = \underline{\mathbf{F}}^{ext} - \underline{\mathbf{R}}^{s}$. Term $\underline{\mathbf{R}}^{s}$ is expressed linearly as a function of soil reaction $\underline{\mathbf{P}}^{\mathbf{s}}$. Then, using the influence matrix, we express the soil reaction as a function of settlement (6). Finally, the ultimate formulation of the problem is obtained by writing the equality at each node between soil settlement and plate vertical displacement:

$$\left(\underline{\underline{\mathbf{K}}}^{e} + \underline{\underline{\mathbf{T}}}^{s}.\left(\underline{\underline{\mathbf{T}}}_{\mathbf{inf}}\right)^{-1}.\underline{\underline{\mathbf{G}}}\right).\underline{\mathbf{a}}^{e} = \underline{\mathbf{F}}^{ext} \qquad \textbf{(7)}$$

Where:

- $\underline{\underline{\mathbf{T}}}^{s}$ is the transfer matrix enabling $\underline{\mathbf{R}}^{s}$ to be expressed as a function of $\underline{\mathbf{P}}^{\mathbf{s}}$.
- $\underline{\underline{\mathbf{G}}}$ is the transfer matrix allowing the transition from $\underline{\mathbf{S}}$ to $\underline{\mathbf{a}}^{e}$, by writing equality at each node between settlement and plate vertical displacement.

Solving this system allows evaluation of the generalised displacement vector and thus the nodal variables associated with each mesh element. This makes it possible to calculate:

- Plate deflection w at any point and thus soil settlement at any point $\underline{\mathbf{S}}$.
- Soil reaction at any point:

$$\underline{\mathbf{P}}^{\mathbf{s}} = \left(\underline{\underline{\mathbf{T}}}_{\mathbf{inf}}\right)^{-1}.\underline{\mathbf{S}} \qquad \textbf{(8)}$$

- Bending moments in the plate, that can be evaluated from the displacement field in each element:

$$\begin{cases} M_x = \dfrac{EI}{(1-\nu^2)}\left(\dfrac{\partial^2 w}{\partial x^2} + \nu\dfrac{\partial^2 w}{\partial y^2}\right) \\ M_y = \dfrac{EI}{(1-\nu^2)}\left(\dfrac{\partial^2 w}{\partial y^2} + \nu\dfrac{\partial^2 w}{\partial x^2}\right) \\ M_{xy} = 2\dfrac{EI}{(1-\nu^2)}\left(\dfrac{\partial^2 w}{\partial xy}\right) \end{cases} \qquad \textbf{(9)}$$

It should be pointed out that the moments are only calculated inside each element (at the integration points and not at the nodes) owing to the "nonconforming" nature of the elements used, which cannot guarantee the continuity of forces when passing from one element to another.

I.2.3. Processing of certain special cases

I.2.3.1. Unsticking of plate

The previous calculation is valid assuming that the plate remains perfectly stuck to the soil. As the soil cannot generate tensile forces, the corresponding nodes must be considered to be "unstuck" if negative reactions are obtained (see Figure I.12 below). This can be done with the help of an automatic iterative calculation, with an unsticking criterion defined by the user.

For the nodes declared as "unstuck":

- plate displacement is no longer equal to soil settlement (see Figure I.12 below);
- soil reaction at these nodes is nil.

If the plate is resting on an elastic multilayer medium, appropriate reprocessing of the influence matrix (6) enables these new conditions to be taken into account automatically.

The automatic unsticking process also applies if the plate is resting on a distribution of surface springs. In this case, appropriate reprocessing of stiffness matrix $\underline{\underline{\mathbf{K}}}^e$ consisting in cancelling the terms equivalent to the springs which "unstick" enables these new conditions to be taken into account.

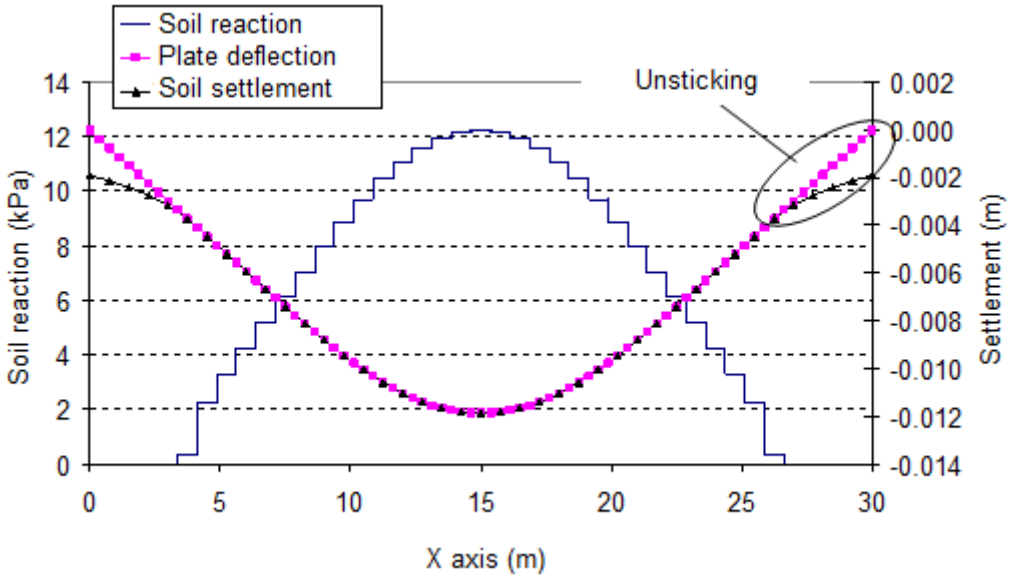


Figure I.12 : Plate unsticking - Example

I.2.3.2. Limitation of soil reaction

In order to extend the model beyond the supporting soil elastic domain, we introduced criteria for local limitation of the pressures at the Soil – Plate interface. This can be done with an automatic iterative calculation, provided there is a "plastification" criterion at the soil-plate interface.

At the nodes declared to be "plastified":

- soil settlement is no longer elastic, but is still equal to plate vertical displacement;
- soil reaction is equal to the plastification boundary pressure (given by the user).

This process applies both to a support defined as an elastic multilayer medium and a support defined as a distribution of surface springs.

I.2.3.3. Taking account of an outside load applied to the soil

This involves dealing with the case of an outside load applied directly to the soil (see Figure I.13 below). A typical case is that of nearby backfill which disrupts the soil/plate interaction. This can be taken into account by adapting relation (6):

$$\underline{\mathbf{S}} = \underline{\underline{\mathbf{T}}}_{\mathbf{inf}} . \left(\underline{\mathbf{P}}^{\mathbf{s}} \right) + \underline{\mathbf{s}}^{ext} \qquad \textbf{(10)}$$

Where $\underline{\mathbf{s}}^{ext}$ designates the settlement produced at each node by the outside load considered.

This function can only be used on a support defined as an elastic multilayer medium.

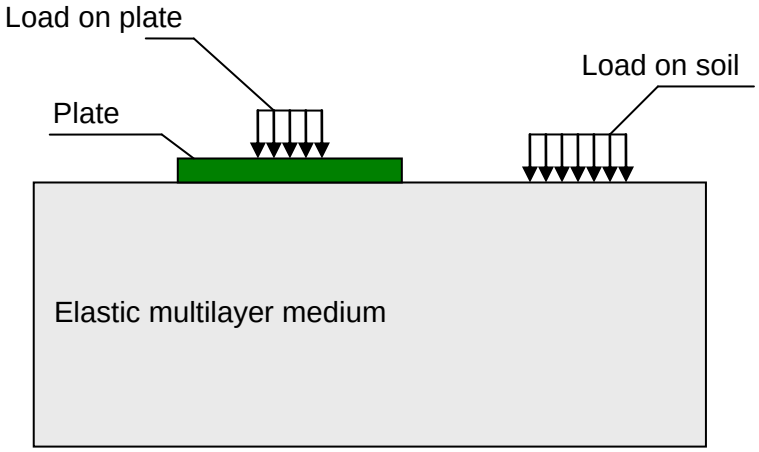


Figure I.13 : Taking account of an outside load applied directly to the soil

I.2.3.4. Taking account of an initial stress in the soil

This involves dealing with the cases of structures placed at the bottom of an excavation (see Figure I.14 below). The assumption adopted consists in ignoring "recompression" settlement under loads less than or equal to the initial weight of earth. This can be taken into account by adapting relation (6):

$$\underline{\mathbf{S}} = \underline{\underline{\mathbf{T}}}_{\mathbf{inf}} . \left(\underline{\mathbf{P}}^{\mathbf{s}} - \boldsymbol{\sigma}_{\mathbf{v}}^{\mathbf{0}} \right) \qquad \textbf{(11)}$$

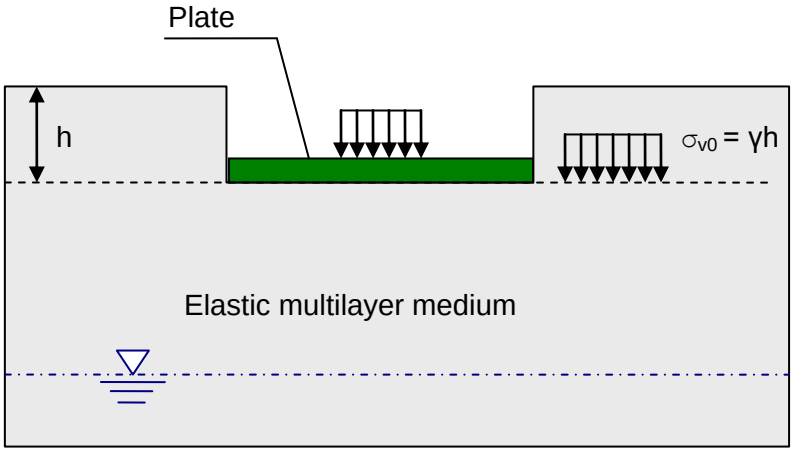


Figure I.14 : Taking account of an initial surface stress - Example

This function can only be used on a support defined as an elastic multilayer medium.

I.2.4. Taking account of symmetries

Any symmetry condition is implicitly assumed to be valid for the plate, for the soil and for the load.

I.2.4.1. Case of symmetry on one or two planes

This concerns the case in which the "plate + soil + load" system comprises one or two vertical planes of symmetry.

In the case of a plate on supports or springs, this type of condition is simply managed by blocking rotation around the horizontal axis generating each plane of symmetry.

In the presence of an elastic medium, blocking rotations at the plate remains necessary but not sufficient. When constituting the soil influence matrix, it is also necessary to take account of the fact that settlement s_{ij} at node "i" under the effect of load "j" is in fact the result of a combination of load "j" and its symmetries, as shown in the following figure.

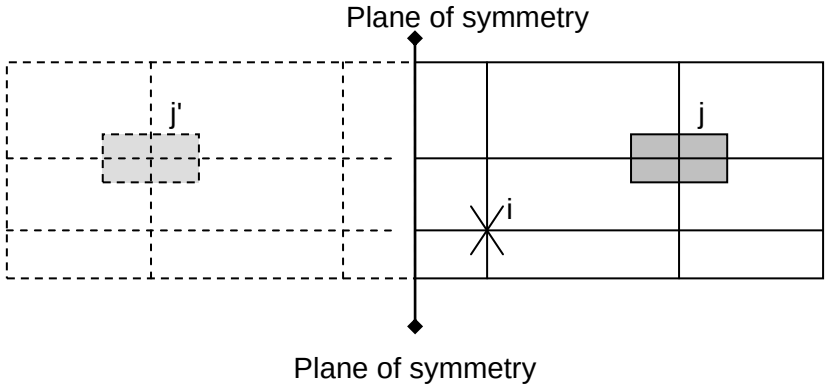


Figure I.15 : Schematic representation of the symmetry condition with respect to a plane

I.2.4.2. Axisymmetry (or symmetry with respect to an axis)

This symmetry condition is appropriate for the case of a circular plate subjected to axisymmetric loading. Taking account of this condition means that particular models must be adopted.

In cylindrical coordinates, the equilibrium of the plate is governed by the following equation:

$$\frac{d^4w}{dr^4}+\frac{2}{r}\frac{d^3w}{dr^3}-\frac{1}{r^2}\frac{d^2w}{dr^2}+\frac{1}{r^3}\frac{dw}{dr}=q(r)\frac{(1-\nu^2)}{EI} \qquad \textbf{(12)}$$

This equation can be resolved by introducing finite element discretisation. For this, the Kirchhoff model is adopted for circular plates with "annular" elements having 2 nodes and 4 degrees of freedom, as shown in the following figure. Any outside load is also axisymmetric: uniform annular surface loads between two given radii and annular linear loads at a given radius.

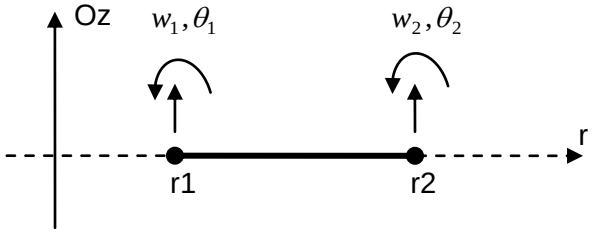


Figure I.16 : Kirchhoff model for circular plates
with "annular" elements having 2 nodes and 4 degrees of freedom

If the supporting soil is input as an elastic multilayer medium, direct numerical integration of the Boussinesq formulas is used to evaluate the influence coefficients and the constitution of the soil flexibility matrix.

The representation by equation and the solution principle are identical to those presented for the general 3D case. The size of the system to be solved is 2n x 2n, where n is the number of "activated" nodes.

All the functionalities previously presented also apply in axisymmetry: unsticking/plastification, deactivation (in annular zones), surface/linear, springs, etc.

This calculation leads to the following results:

- deflection and settlement expressed as a function of r (distance from the axis of symmetry Oz);
- soil reaction as a function of r (defined in annular sections);
- the forces in the plate: radial moment (M_r), tangential moment (M_t) and shear force (T_r).

It should be noted that the one-dimensional nature of the forces makes it easy to evaluate them directly by integration of loads and reactions applied to the plate.

I.2.4.3. Plane deformations

This framework is suited to the case in which bending of the plate can be ignored in one direction. this requires the adoption of special models.

In plane deformations, plate equilibrium is governed by the following equation:

$$\frac{d^4 w}{dx^4} = q(x)\frac{(1-\nu^2)}{EI} \quad \textbf{(13)}$$

The plate is assumed to be infinite in direction (Oy). Its behaviour can be characterised by the laws of thin beams theory. For this, we introduce discretisation into beam elements with 2 nodes and 4 degrees of freedom as shown in the following figure. Any outside load is also "infinite" in direction (Oy): uniform surface loads between two given X-axis points and linear loads at a given X-axis point.

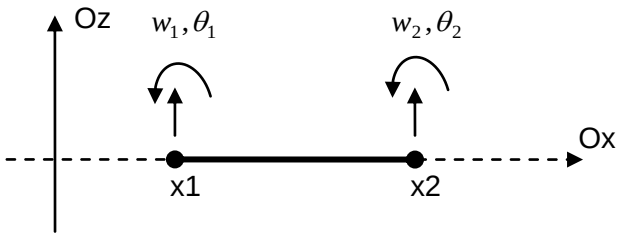


Figure I.17 : Discretisation into beam elements of 2 nodes and 4 degrees of freedom

If the supporting soil is input as an elastic multilayer, the principle of the Steinbrenner model is adapted to the case of a continuous load (along Oy). For this, we can start with formula (2) with B tending towards infinity. Settlement produced in a layer between depths d1 and d2, under the edge of a continuous load of width L, can thus be expressed:

$$\rho_{D1-D2} = \frac{qL}{E}\left((1-\nu^2)\left(F_1(d_2)-F_1(d_1)\right)+(1-\nu-2\nu^2)\left(F_2(d_2)-F_2(d_1)\right)\right) \quad \textbf{(14)}$$

$$F_1(D)=\frac{1}{\pi}\ln\left(1+\frac{D^2}{L^2}\right) \quad \text{and} \quad F_2(D)=\frac{1}{\pi}\frac{D}{L}\arctan\left(\frac{L}{D}\right) \quad \textbf{(15)}$$

The representation by equation and the solution principle are identical to those presented for the general 3D case. The size of the system to be solved is 2n x 2n, where n is the number of "activated" nodes.

All the functionalities previously presented also apply to plane deformations: unsticking / plastification, deactivation (in continuous zones), surface/linear, springs, etc.

This calculation can be used to obtain deflection, settlement and reaction at any point, as well as the forces in the plate: a shear force Tx and a bending moment Mx.

The one-dimensional nature of these forces makes it easy to evaluate them directly by integration of loads and reactions applied to the plate (which limits the influence of precision as linked to the mesh density).

I.2.5. Validity limits

The TASPLAQ module underwent an in-depth validation study first of all concerning the model chosen for the plate and then concerning the general approach considered to take account of the supporting soil. This study showed that in the context of the model validity domain, the results obtained are very similar to those resulting from complete three-dimensional finite element processing.

I.2.5.1. Validity of the Kirchhoff model

The Lagrange equation governing plate equilibrium results from the Kirchhoff model which assumes the thin plates assumption to be valid. It should be pointed out that this assumption consists in ignoring deformations due to shear forces, which remains valid as long as the plate thickness is low by comparison with its other dimensions.

I.2.5.2. Validity of the Boussinesq model

The calculation of the supporting soil deformations in TASPLAQ is based on the generalised Boussinesq model. This consists in assuming that the stresses propagate in the same way in the case of a multilayer as in the case of a homogeneous semi-infinite medium. The validity of this assumption requires low stiffness contrast between the different layers, given that it however remains valid in the case of a deformable layer above a stiffer layer. On the other hand, the case of a relatively stiff layer above a deformable layer cannot be processed in this way and is therefore excluded from the model, with the exception of certain special cases which will be dealt with subsequently.

The various validation tests carried out with respect to complete 3D finite element calculations, showed that the specific resistance of the structural element (plate) considerably attenuates the impact of the Boussinesq model limitation.

I.2.5.3. Special case of a surface base layer

The presence of a low deformability base layer on the surface may limit the validity of the TASPLAQ method, owing to the unsuitability of the Boussinesq model in this particular case. One of the ways of overcoming this limitation is to integrate the base layer into the body of the plate, defining an equivalent inertia for the "plate + base layer" assembly. This leads to a more precise displacement and it is also possible to work back to the loads in the plate from those calculated in the equivalent medium (plate + base layer) by "dishomogenising" it.

I.3. USER'S MANUAL

This chapter presents:

- the Tasplaq module input parameters.
  Some zones can only receive data that make physical sense (for example, a Young's modulus must always be strictly positive).
  The Tasplaq calculation parameters input window consists of 9 separate tabs, including one that is optional (manual management of unsticking/plastification of nodes). The data to be entered in each tab sometimes depend on certain choices made by the user: for example, the data to be filled out for a 2-dimensional calculation with plane deformation will not be strictly the same as for a 3-dimensional calculation.
- the results provided by the Tasplaq module. Here again, they partly depend on the data input by the user, in particular the type of calculation.

This chapter does not detail the actual user interface and its operations (buttons, menus, etc.): these aspects are covered in part I.4 of the manual.

I.3.1. Notations and units

I.3.1.1. Notations and sign conventions

Parameter	Representation	Sign convention
Rotations and moments	θ, M	Trigonometric direction
Plate deflection	w	Positive downwards
Soil settlement	$Tass$	Positive downwards
Shear forces	T	Positive upwards
Vertical load (distributed or concentrated)	q, F	Positive downwards
Soil reaction, interaction pressure	r, p	Positive upwards
Stiffnesses	C, K	Always positive
Stresses	σ	Positive in traction

Table I.1 : Notations and sign conventions

I.3.1.2. Units

Parameter	Unit
Lengths and coordinates	m
Point vertical force F	kN
Moments M	kN.m/ml
Shear forces T	kN/ml
Soil reaction, surface loads	kPa
Displacements (deflection w, settlement s)	m
Rotations	rad
Young's modulus E	kPa
Surface stiffness / reaction coefficient	kPa/m
Point translation stiffness	kN/m
Point rotation stiffness	kN.m/rad

Table I.2 : Units

I.3.2. "Parameters" tab

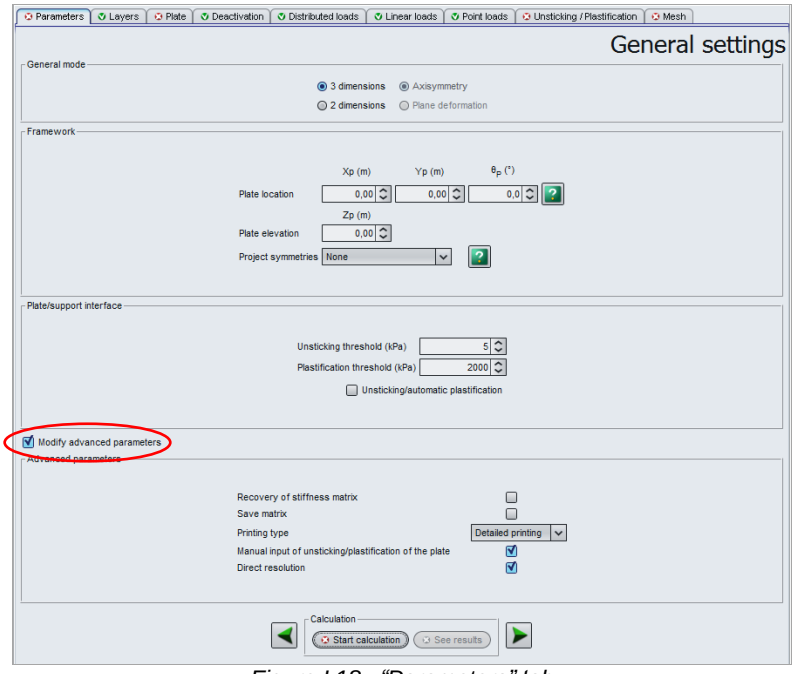


Figure I.18 : "Parameters" tab

I.3.2.1. "General mode" frame

- State whether the calculation is three-dimensional or two-dimensional;
- for a 2D calculation, specify whether it is an axisymmetry or plane deformation calculation.

Depending on the type of calculation chosen, Tasplaq will ask for different settings in the following tabs.

I.3.2.2. "Framework" frame

This will be used to define the local coordinates system of the plate.

The general case comprises 2 coordinates systems:

- the reference coordinates system (O_0x_0, O_0y_0, O_0z_0), in which the plate will be situated, as well as the external loads applying directly to the soil;
- the local coordinates system (Ox, Oy, Oz) associated with the plate, in which the mesh and the various characteristics will be defined. This coordinates system is such that plane (Ox, Oy) is parallel to plane (O_0x_0, O_0y_0). It can thus be perfectly defined with the help of two parameters:
  - ✓ coordinates (x_p, y_p, z_p) of point O in the reference coordinates system. Caution: Z_p is the project reference elevation.
  - ✓ angle θ_p of axis (Ox) with respect to axis (O_0x_0).

It should be noted that point O_0 also acts as the reference when defining the dip angles of the layers when the soil is defined as a non-horizontal multilayer.

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
X_p, Y_p: coordinates in the global coordinates system of the origin of the plate local coordinates system	m	0.00	Only if no symmetry. Else value set at 0 (no modification possible)	Yes	-
Z_p: project reference elevation (elevation of base of plate)	m	0.00	Always	Yes	-
Theta (θ)	°	0	Only if no symmetry. Else value set at 0 (no modification possible)	Yes	-
Symmetries	-	None	Always	Yes	Possible choices: "None", "Along Ox", "Along Oy", "Along Ox and Oy"

Table I.3 : "General setting" tab: "Framework" box

The following help diagrams are available (click the buttons) to illustrate the definition of the plate coordinates and the choice of symmetries applicable to each project:

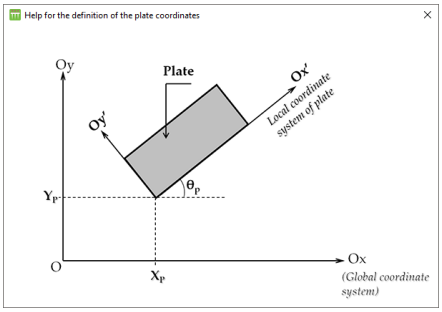


Figure I.19 : Help diagram for defining plate coordinates

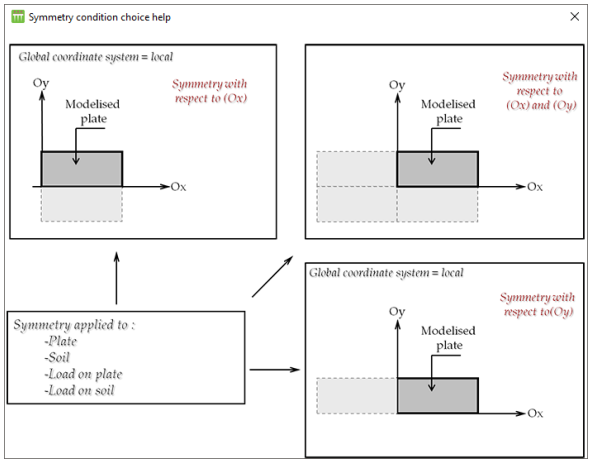


Figure I.20 : Help diagram for choice of symmetry

I.3.2.3. "Plate support interface" frame

This box contains the parameters defining the soil behaviour on the surface and at the plate/support interface. They are only used in the calculation in the case of a plate on supporting soil and only if "unsticking/automatic plastification" calculation was requested.

- Unsticking threshold**:** ultimate **tensile** stress at the Soil-Plate interface, beyond which the corresponding nodes are considered to be "unstuck" (if the "automatic" calculation mode is

ticked). In the event of unsticking, the soil reaction at the "unstuck" nodes is nil and soil settlement and vertical displacement of the plate at these nodes are no longer equal.

- Plastification threshold: ultimate **compression** stress at the Soil/Plate interface, beyond which the corresponding nodes will be considered to be "plastified" (if the "automatic" calculation mode is ticked). In the event of plastification, soil reaction at the "plastified" nodes is imposed (equal to the plastification threshold), but soil settlement and plate vertical displacement are still equal.

In the case of "plate on surface springs" type calculations, these thresholds are automatically used by the programme to manage unsticking/plastification of surface springs. In this case, these thresholds are defined as follows:

- Unsticking threshold**:** ultimate **tensile** stress of the surface springs, beyond which the corresponding elements will be considered "unstuck" (if "automatic" calculation mode is ticked). In the event of unsticking, the surface stiffness under the "unstuck" elements is cancelled.

- Plastification threshold: ultimate **compression** stress of the surface springs, beyond which the corresponding elements will be considered to be "plastified" (if "automatic" calculation mode is ticked). In the event of plastification, the surface stiffness under these elements is replaced by a constant reaction equal to the plastification threshold.

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Unsticking threshold	kPa	5	Always	Yes	≥ 0
Plastification threshold	kPa	2000	Always	Yes	> 10
Unsticking/automatic plastification (as a function of the above unsticking and plastification criteria	-	Unticked	Always	Yes	-

Table I.4 : "Parameters" tab: "Plate/Support interface" box

I.3.2.4. "Advanced parameters" frame

By default, the "Modify advanced parameters" box is unticked. If ticked, the "Advanced parameters" frame appears (see Figure I.18).

- Recovery of stiffness matrix**:** this option is used to recover the influence matrix, (assumed to be already saved) from a previous calculation. This option offers a considerable time saving in the case of a system with several load cases.
- Save matrix**:** used to save the soil influence matrix for a later calculation. This option is useful in the case of a system with several load cases for example.
- Printing type**:** controls printing of the results file. This choice only concerns the data summary:
  - ✓ reduced printing: brief data summary;
  - ✓ detailed printing: detailed data summary.
- Manual input of unsticking/plastification of the plate: when the box is ticked, an additional "Unsticking / Plastification" tab appears (see chapter I.3.9). This tab can be used to force unsticking/plastification of a given zone at the soil/plate interface (manual definition by the user of the nodes to be declared as unstuck or plastified).
- Direct resolution: this option controls the direct or iterative resolution mode of the global system of equations in the case of a plate on a multilayer medium.

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Recovery of stiffness matrix	-	Unticked	Always	Yes	-
Save matrix	-	Unticked	Always	Yes	-
Printing type	-	Detailed printing	Always	Yes	-
Manual input of unsticking/plastification of the plate	-	Unticked	Always (a new tab appears if ticked)	Yes	-
Direct resolution	-	Ticked	Always	Yes	-

Table I.5 : "Parameters" tab: "Advanced parameters" frame

I.3.3. "Layers" tab

This second tab is used to enter parameters concerning behaviour of the soil when defined as an elastic multilayer.

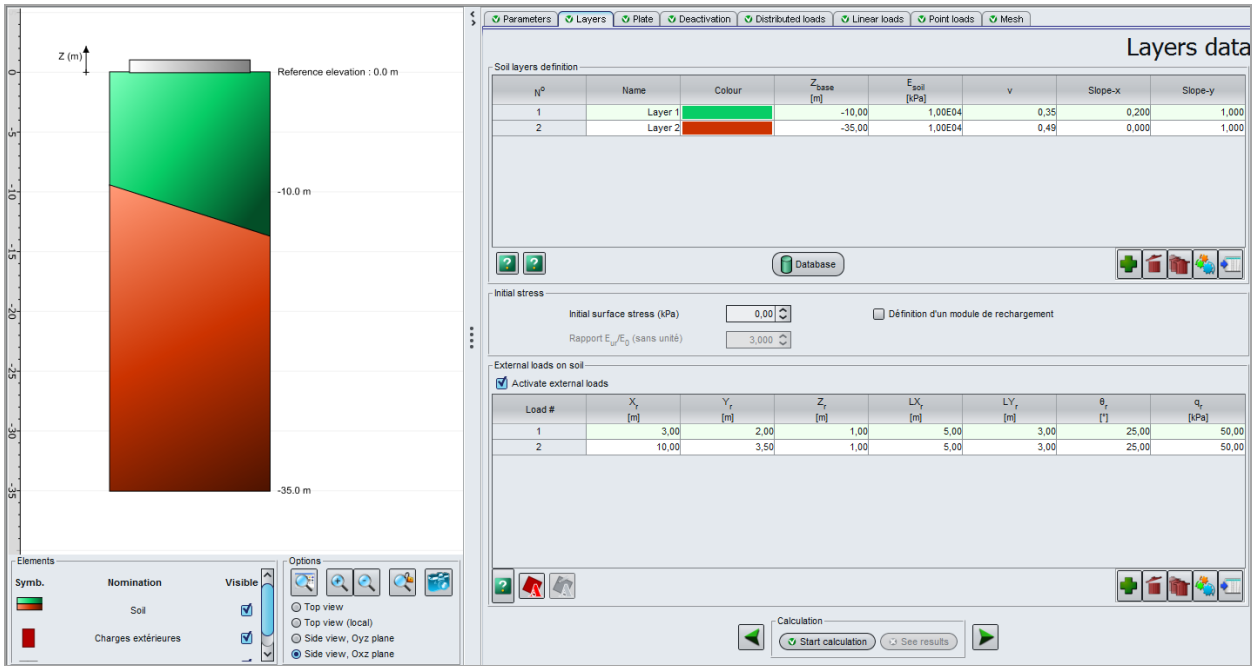


Figure I.21 : "Layers" tab: case of a 3D model

I.3.3.1. "Soil layers definition" frame

Soil layers definition is not a mandatory step in Tasplaq: in the case of a calculation on elastic supports only, for example, soil layers are not defined (see example 1 in this manual).

The following table describes the soil parameters to be defined for each layer:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
$\mathbf{Z_{base}}$: elevation of base of layer	m	1 m under base of layer above	Always	Yes	Strictly descending values
$\mathbf{E_{sol}}$: Young's modulus of layer concerned	kPa	-	Always	Yes	$\nu > 0$
ν: Poisson's ratio	-	-	Always	Yes	$0 \leq \nu < 0.5$
Slope-x	-	0.00	Always	Yes	-
Slope-y	-	0.00	Always	Yes	-

Table I.6 : Soil layer parameters

The "slope-x" and "slope-y" dip parameters designate the X and Y dip of the base of the layer concerned. The elevation of the base "Zbase" thus defined is taken at the reference coordinates system origin O_0. These conventions are illustrated by the following two help diagrams :

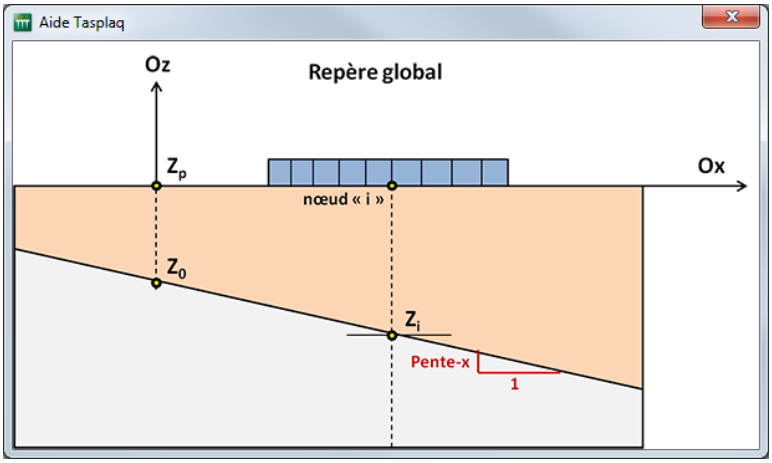


Figure I.22 : Help diagram for layer dip along x

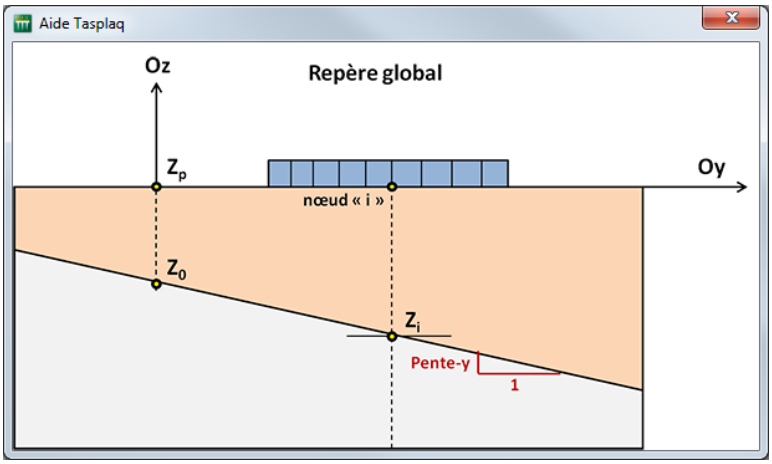


Figure I.23 : Help diagram for layer dip along y

It is important to note that the use of symmetries is incompatible with a dip value other than zero.

The initial surfaces stress for the model should be input under the soil layers definition table:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Initial surface stress	kPa	0	Always	Yes	-

Table I.7 : Definition of the initial surface stress

Foxta can be used to save soil layers in the project soils database and/or in the global soils database, by clicking the Database button.

This saves the soil layers with their parameters and avoids having to input them again when creating a new module in the current project, or for another Foxta project.

Use of the soils database is described in detail in part C of the manual.

I.3.3.2. "External loads on soil" frame

In addition to the pressure(s) exerted by the plate, the soil can be subjected to "direct" external loads. These loads are assumed to be rectangular in shape, positioned and oriented in the global coordinates system. The following figure describes the overall layout of the problem:

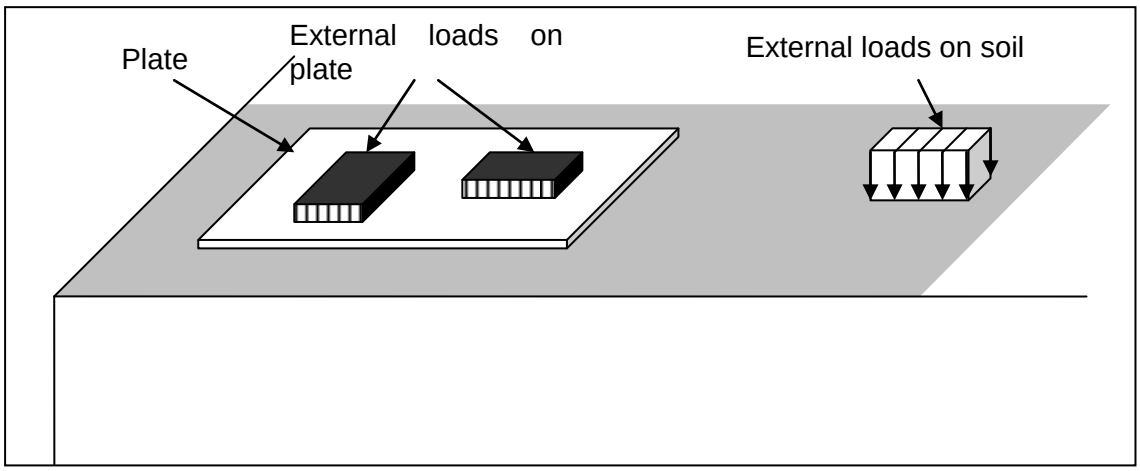


Figure I.24 : Plate: Overall layout of the problem {Plate + Soil + Ext. loads}

The "Activate external loads" box is unticked by default. If ticked, the external loads definition table appears (see Figure I.21).

I.3.3.2.1. Input of external loads (for 3D models)

The data to be input vary according to the type of project chosen in the first tab:

- For a 3D project, each load is characterised by the coordinates of its "bottom-left" corner (X_r, Y_r, Z_r), its dimensions (width LX_r and length LY_r), its orientation (θ_r), as well as by its load density (q_r). The following help diagram, which is available in the software, illustrates these parameters.

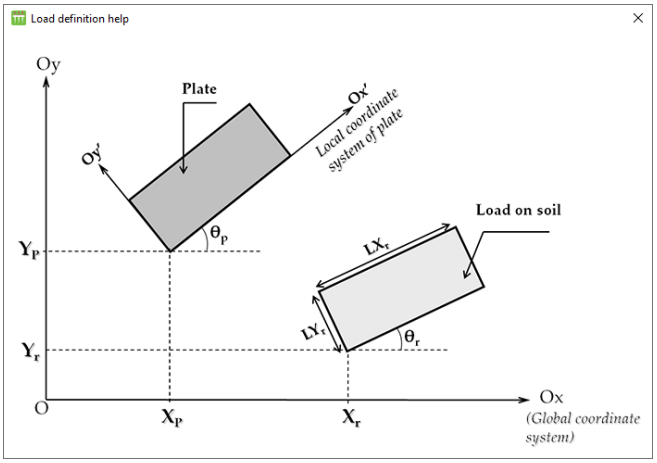


Figure I.25 : Help diagram for defining external loads in 3D

- For a 2D project, each load is characterised by the coordinates of its end in the coordinates system (X_r, Z_r), its width L_r and its load density (q_r). The following help diagrams, which are available in the software, illustrate these parameters.

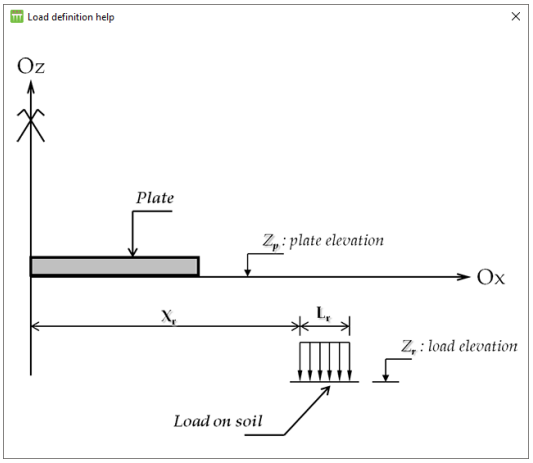


Figure I.26 : Help diagram for definition of external loads in the case of a 2D project with plane deformations

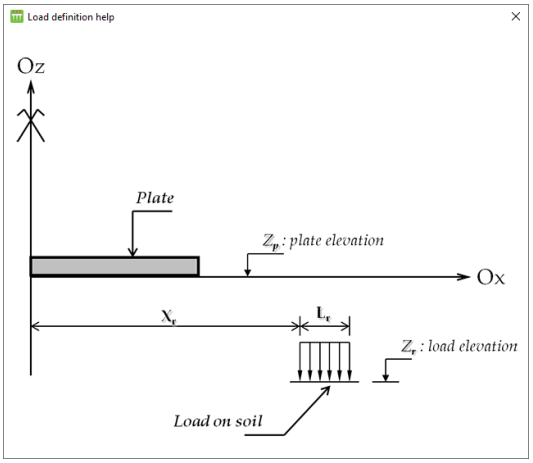


Figure I.27 : Help diagram for definition of external loads in the case of an axisymmetric 2D project

It should be noted that the external loads in 3D are not necessarily oriented parallel with the Ox and Oy axes: they can be placed at any angle with respect to these axes.

The following 2 tables identify the data to be input for the 3D and 2D models respectively.

Designation	**Unit**	**Default value**	**Display condition**	**Mandatory value**	**Local checks**
X_r, Y_r, Z_r: coordinates of reference corner of rectangle in global coordinates system	m	-	Always	Yes	-
LX_r	m	1.00	Only in 3D mode	Yes	> 0
LY_r	m	1.00	Only in 3D mode	Yes	> 0
Theta θ_r	°	-	Only in 3D mode	Yes	-
q_r	kPa	-	Always	Yes	-

Table I.8 : Parameters for defining external loads on the soil in the case of a 3D model

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
X_r, Z_r: coordinates of reference corner of rectangle in global coordinates system	m	-	Always	Yes	-
L_r:	m	1.00	Always	Yes	> 0
q_r	kPa	-	Always	Yes	-

Table I.9 : Parameters for defining external loads on the soil in the case of a 2D model

This tab has a Wizard button to help with definition of loads for the 3D models (see chapter I.3.3.2.2).

To show a particular load, select a row on the table of loads: a green border then appears around the corresponding load on the graphic representation in the left-hand part of the window.

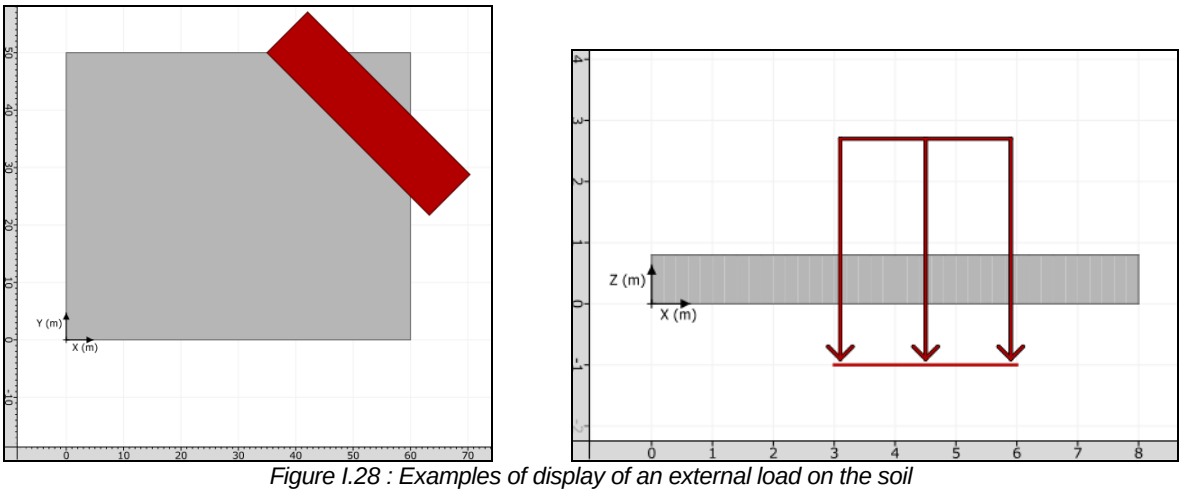


Figure I.28 : Examples of display of an external load on the soil

I.3.3.2.2. Wizards for definition of external loads (for 3D models)

When defining a project in 3 dimensions and to make it easier to define "usual" loads, this tab has an "External loads wizard" button for simple definition of:

- a uniform circular load;
- or a uniform annular load;
- or a 3D embankment-type load.

Figure I.29 illustrates the different wizards available:

- choose the type of load;
- fill out the various input fields;
- click Transfer button.

The use of the windows for the various loads is explained in the following sub-chapters.

Note: several wizards can be used, or the same Loads wizard can be used several times in the context of the same Tasplaq calculation.

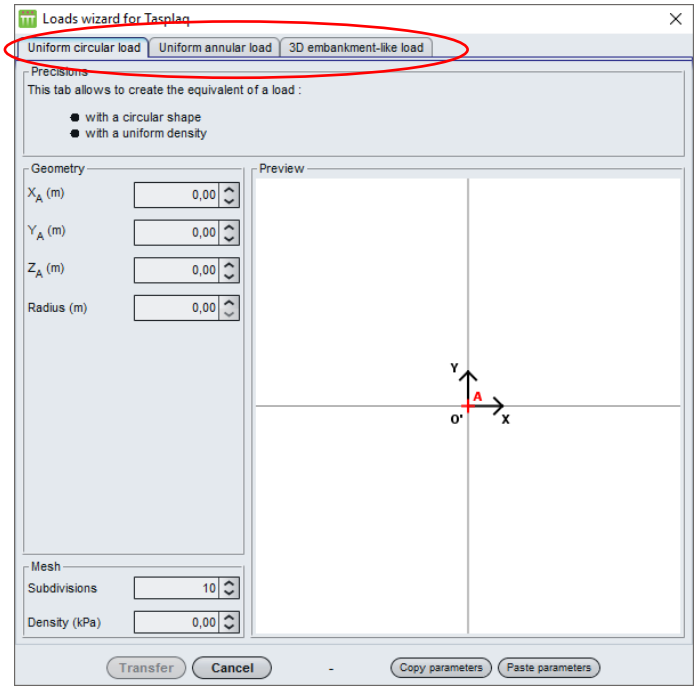


Figure I.29 : Automatic loads (wizards)

Wizard: "Uniform circular load"

This wizard is used to generate a group of rectangle-shaped loads equivalent to a uniform circular load.

The following data must be input:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Point A (X_A, Y_A, Z_A): coordinates of disk centre	m	(0,0,0)	Always	Yes	-
Radius of disk loaded	m	-	Always	Yes	> 0
Subdivisions	-	10	Always	Yes	> 0
Load Density	kPa	-	Always	Yes	> 0

Table I.10 : Parameters for "Uniform circular load" wizard

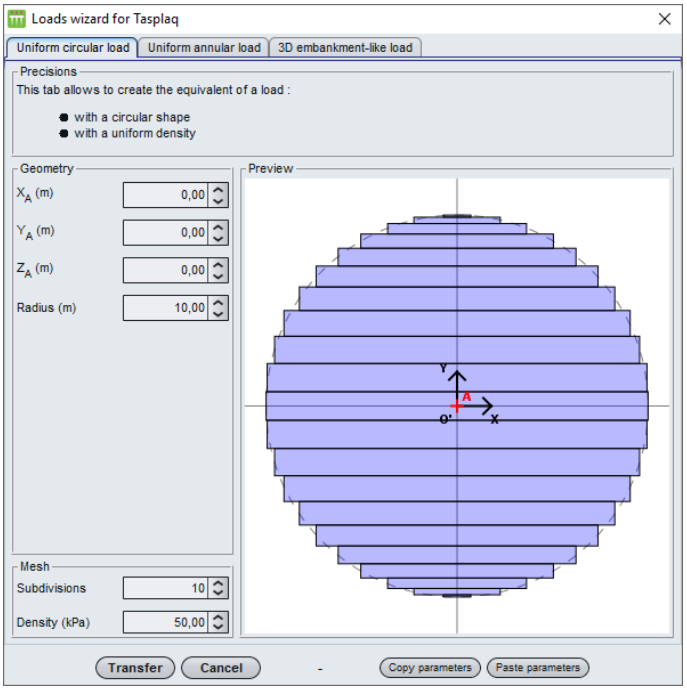


Figure I.30 : Wizard: "Uniform circular load"

The generation of rectangular loads representing the circular load and calculation of their properties X_r, Y_r, Z_r, L_X, L_Y, θ_r and q_r, are activated by clicking the **Transfer** button:

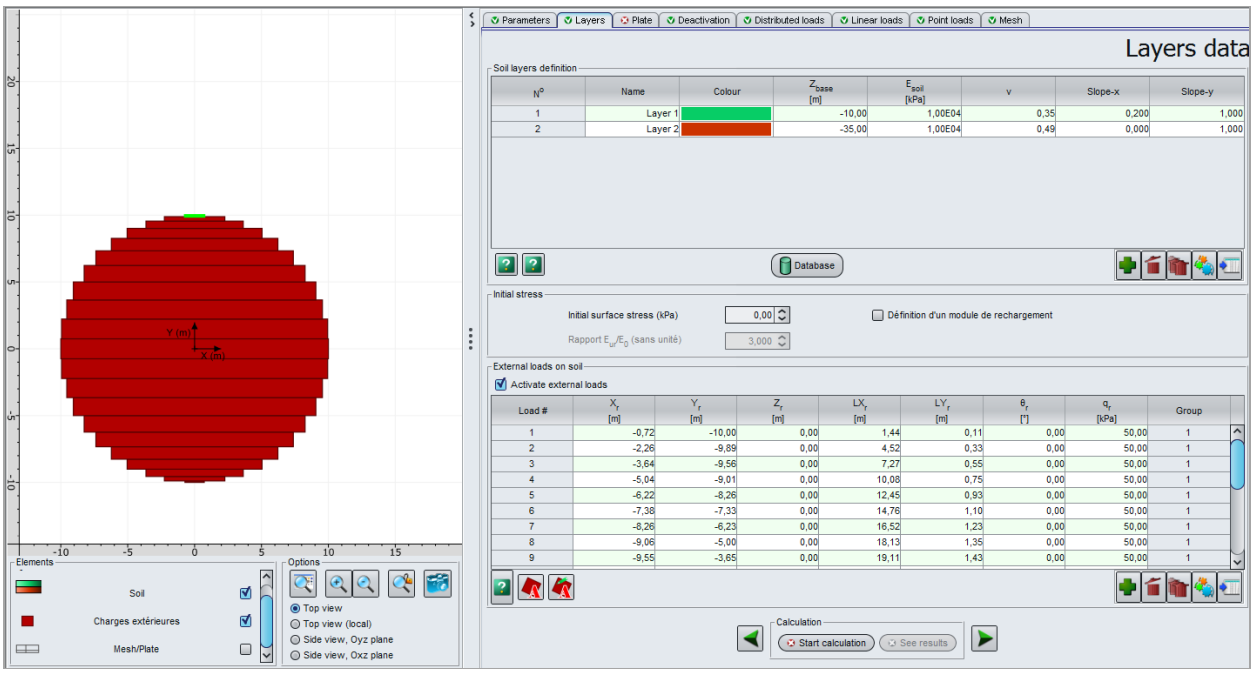


Figure I.31 : "Layers" tab – External loads – After "Uniform circular load" wizard

Wizard: Uniform annular load

This wizard is used to generate a group of rectangular loads equivalent to a uniform annular load.

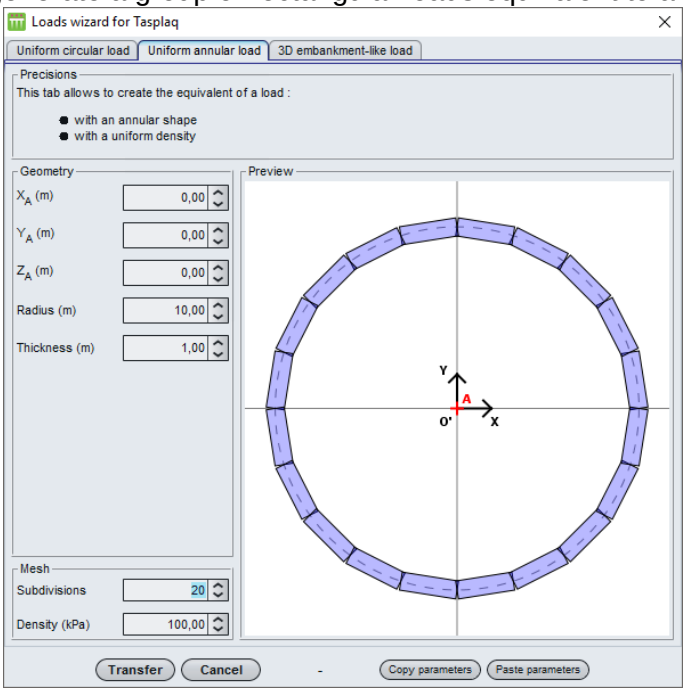


Figure I.32 : Wizard: "Uniform annular load"

The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Point A (X_A, Y_A, Z_A): coordinates of centre of ring	m	(0,0,0)	Always	Yes	-
Mean **Radius** of ring	m	-	Always	Yes	> 0
Thickness of ring	m	-	Always	Yes	> 0
Subdivisions	-	20	Always	Yes	≥ 3
Load **Density**	kPa	-	Always	Yes	> 0

Table I.11 : Parameters for the "Uniform annular load" wizard

Generation of rectangular loads representing the annular load and calculation of their properties X_r, Y_r, Z_r, L_X, L_Y, θ_r and q_r, are activated by clicking the Transfer button:

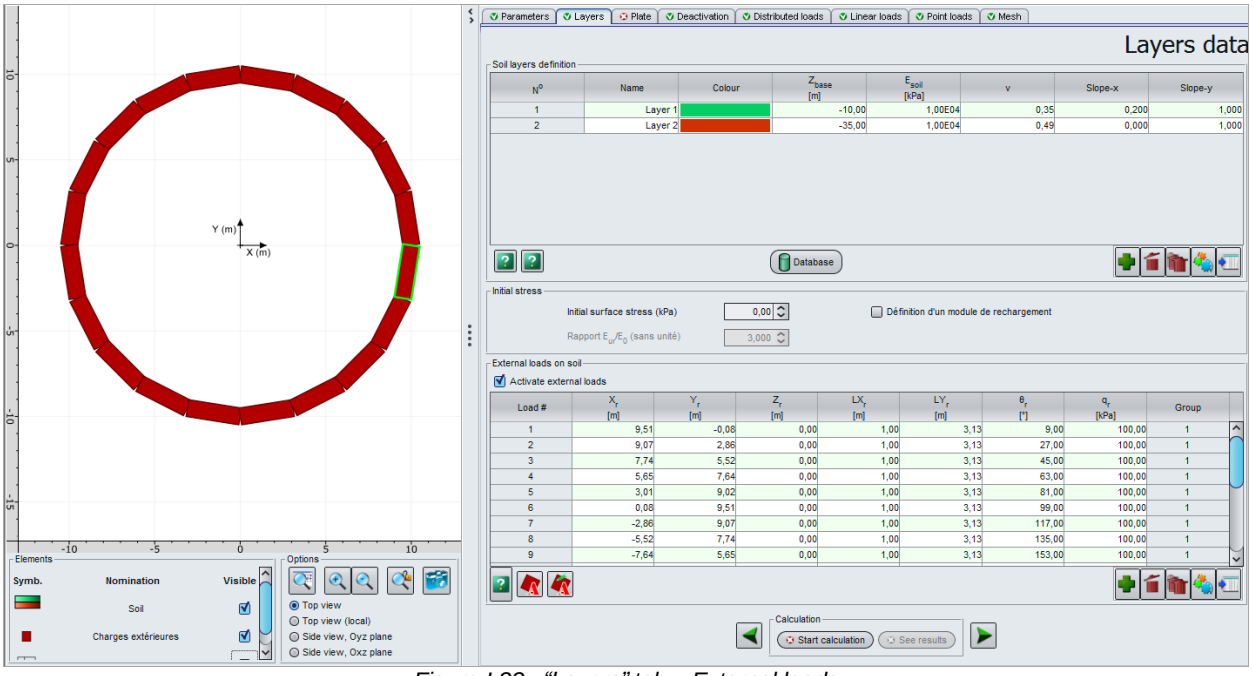


Figure I.33 : "Layers" tab – External loads
- After "Uniform annular load" wizard

Wizard: 3D embankment-like load

This wizard is used to generate a group of rectangular loads equivalent to a three-dimensional embankment.

An example of this type of load is given below, illustrating the general case (different load value at each of the 4 points A, B, C and D).

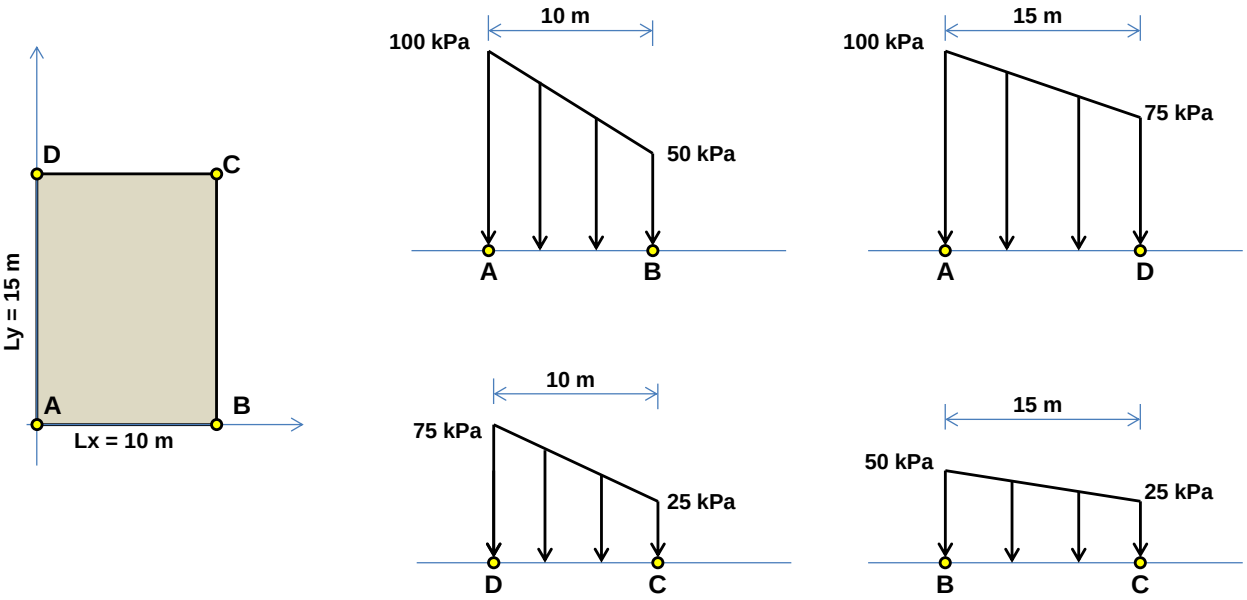


Figure I.34 : Example of 3D embankment type load

The corresponding definition in the wizard is as follows:

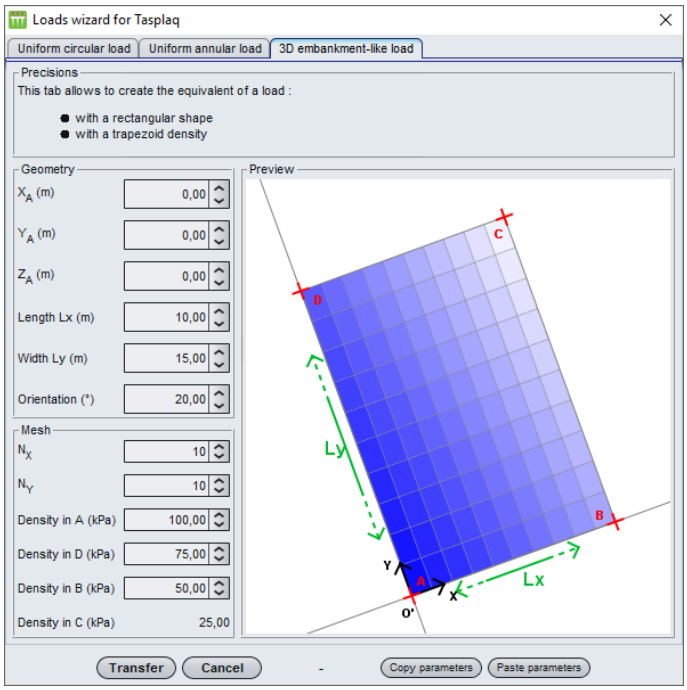


Figure I.35 : Wizard: "3D embankment-like load"

The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Point A (X_A, Y_A, Z_A): Coordinates of lower-left corner of embankment	m	-	Always	Yes	-
L_x: Length of embankment	m	-	Always	Yes	> 0
L_y: Width of embankment	m	-	Always	Yes	> 0
θ : Orientation	°	-	Always	Yes	-
N_x: Subdivisions along X	--	10	Always	Yes	≥ 2
N_y: Subdivisions along Y	--	10	Always	Yes	≥ 2
Load density at A	kPa	-	Always	Yes	At least one of the 3 values must be other than zero
Load density at B	kPa	-	Always	Yes	
Load density at D	kPa	-	Always	Yes	
Load density at C	kPa	-	Always	Yes	Automatically calculated from the 3 previous values

Table I.12 : Parameters for "3D Embankment-like load" wizard

The generation of load rectangles representing the "3D embankment" type load and calculation of their properties X_r, Y_r, Z_r, L_X, L_Y, θ_r and q_r, are activated by clicking the **Transfer** button:

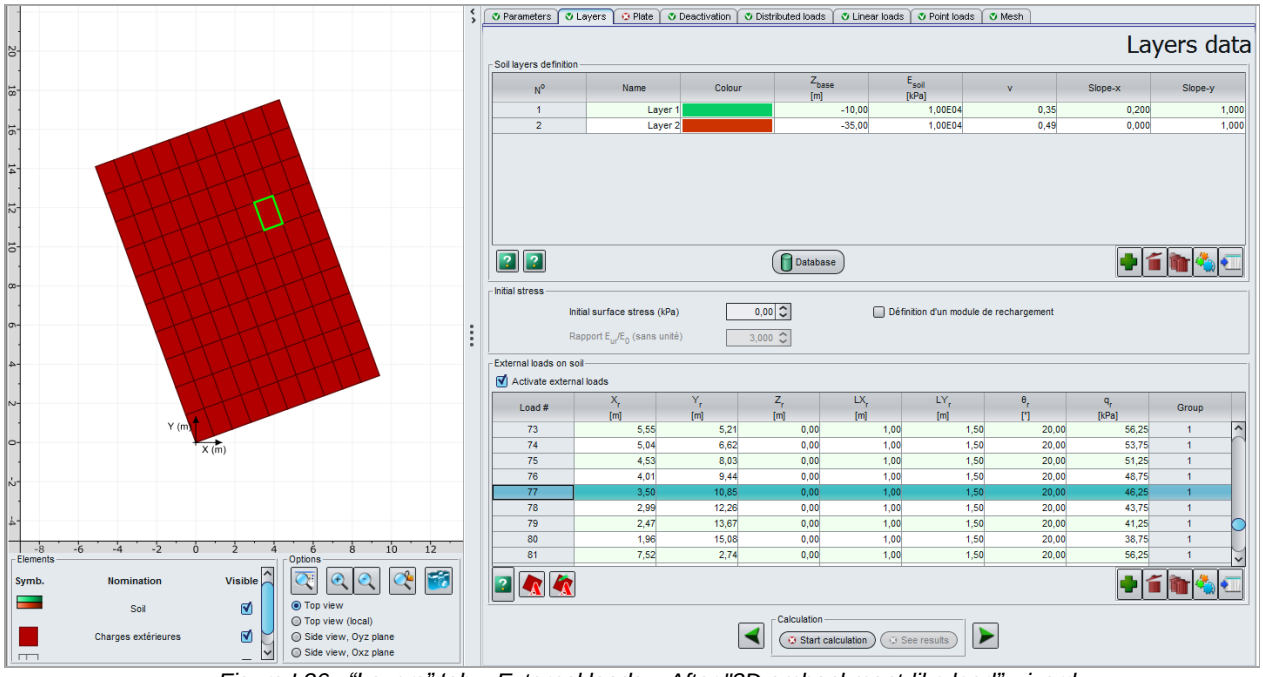


Figure I.36 : "Layers" tab – External loads – After "3D embankment-like load" wizard

Modification of a load group

Selecting a line in the list places a green border around the graphic representation in the left-hand part of the screen and gives access to the selected load group modification wizard :

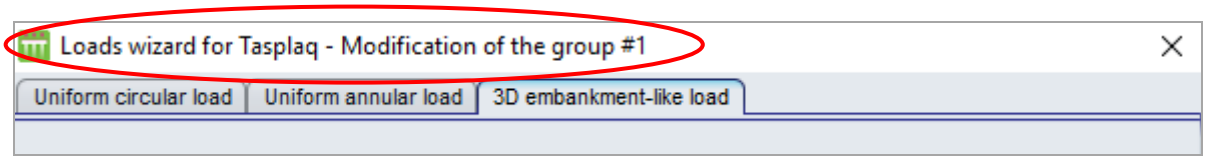


Figure I.37 : Loads wizard for Tasplaq – Example of modification of group n°1

I.3.4. "Plate" tab

The plate properties are assigned by rectangular areas. Each area is characterised by its Young's modulus "E_{plate}", its Poisson's ratio "ν" and its thickness "h".

This step is mandatory. At least one area must be defined in the case of a plate with homogeneous characteristics.

I.3.4.1. Definition of mechanical characteristics of plate

The areas created are bounded by a red line in the application graphics.

When one wishes to define a small area with different characteristics inside a larger area, the larger area should be defined first and then the smaller area with its different mechanical characteristics. The characteristics of the small area "overwrite" and replace those previously defined.

In addition, the fact of allocating characteristics to a deactivated area (see "Deactivation" tab) has no impact on the calculation, because the area is deactivated.

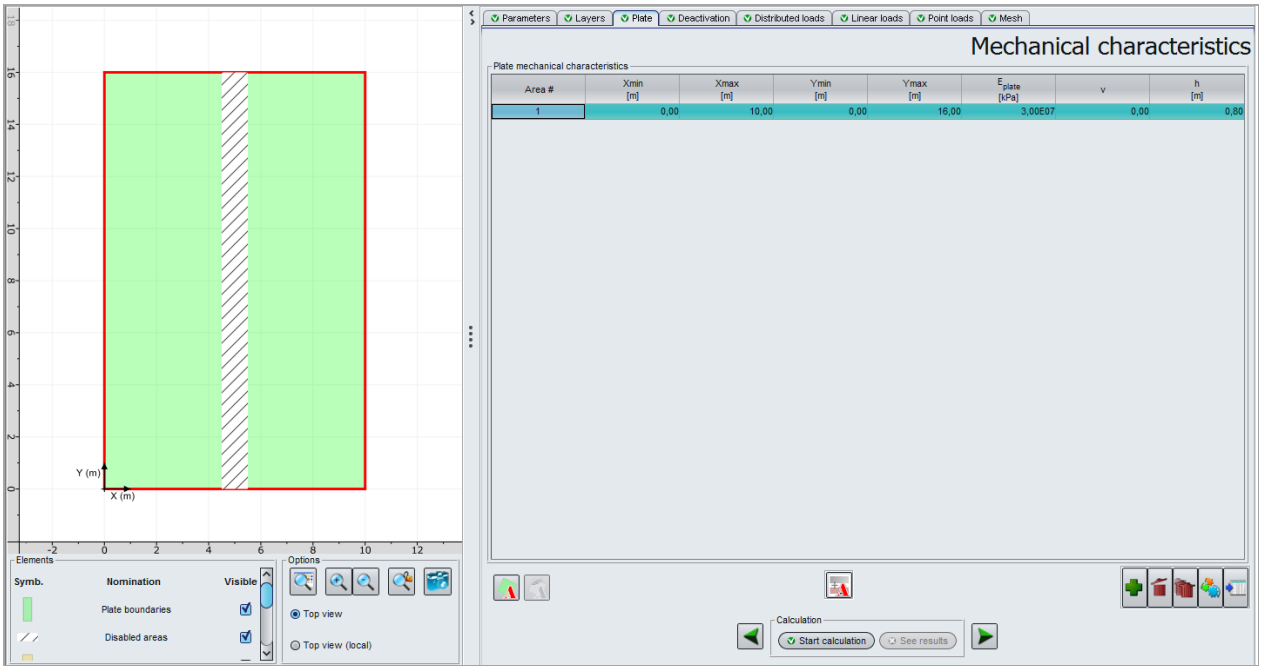


Figure I.38 : "Plate" tab

Below is a summary of the parameters to be input:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Xmin: start of area X-axis	m	Cover the whole plate	Always	Yes	≥ 0
Xmax: end of area X-axis	m		Always	Yes	> Xmin
Ymin: start of area Y-axis	m		Only in 3D mode	Yes	≥ 0
Ymax: end of area Y-axis	m		Only in 3D mode	Yes	> Ymin
$\mathbf{E_{plate}}$: Young's modulus of plate	kPa	-	Always	Yes	> 0
ν: Poisson's ratio	-	-	Always	Yes	$0 \leq \nu < 0.5$
h: plate thickness	m	-	Always	Yes	> 0

Table I.13 : Definition of mechanical characteristics of the plate

The "Plate" tab comprises several wizards: 2 wizards to help the user create geometrical shape plate areas ("Circular plate" wizard and "User-defined plate" wizard, available in 3D only), and 1 wizard to define the plate modulus in the case of a combined section.

I.3.4.2. "Circular plate" and "User-defined plate" wizards

For 3D projects, the plate creation wizard is available by clicking the button. This wizard can be used to define one or more additional plate areas, either circular or user-defined, and transfer them to the project.

This wizard comprises two tabs, described below.

"Circular plate" tab

This wizard is used to generate a group of rectangular areas equivalent to a uniform circular plate area.

The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Point A (X_A, Y_A,): coordinates of centre of circle	m	(0,0)	Always	Yes	-
Radius of plate area	m	0	Always	Yes	> 0
Subdivisions*	-	10	Always	Yes	> 0
$\mathbf{E_{plate}}$: Young's modulus	kPa		Always	Yes	> 0
ν: Poisson's ratio	-		Always	Yes	$0 \leq \nu < 0.5$
h: plate height	m		Always	Yes	> 0

Table I.14 : Parameters for the "Deactivation circular" wizard

* Subdivisions: Constant N. The number of subdivisions will be Nr = 2 x N + 1

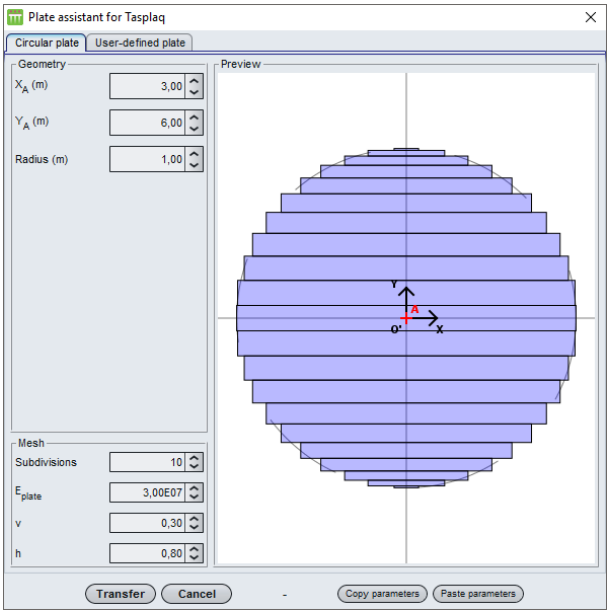


Figure I.39 : Circular plate wizard

"User-defined plate" tab

This wizard is used to generate a group of rectangular areas equivalent to a plate of the configured shape.

The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Coordinates of points (X_A, Y_A, X_B, Y_B, X_C, Y_C, X_D, Y_D,)	M	(0,0,0,0)	Always	Yes	-
Number of elements on X	-	7	Always	Yes	-
Number of elements on Y	-	7	Always	Yes	-
E_{plate}: Young's modulus	kPa	0	Always	Yes	> 0
ν: Poisson's ratio	-	0	Always	Yes	$0 \leq \nu < 0.5$
h: plate thickness	m	0	Always	Yes	> 0

Table I.15 : Parameters for "User-defined plate" wizard

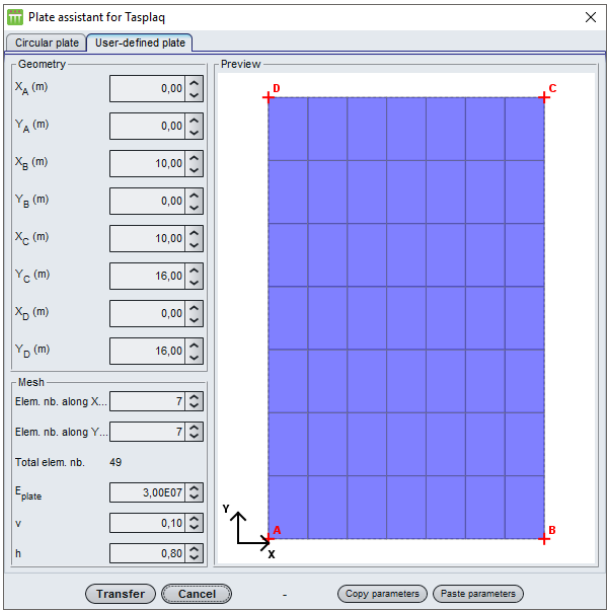


Figure I.40 : Wizard for a user-defined plate

Clicking the Transfer button will import all the corresponding data into the project and close the wizard.

Modification of a group of plates

Selecting a line in the list places a green border around the graphic representation in the left-hand part of the screen and gives access to the selected plate area group modification wizard :

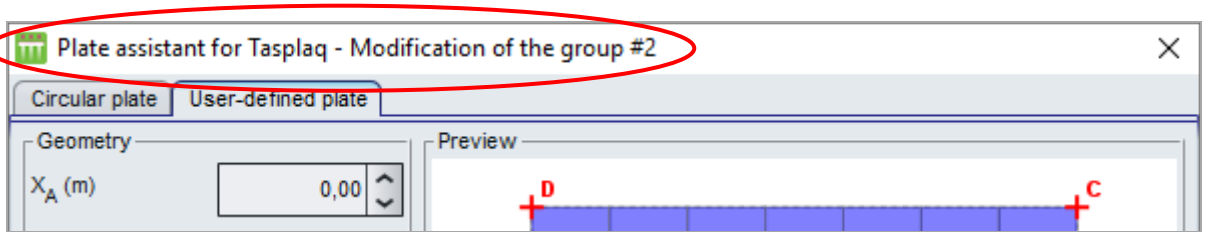


Figure I.41 : Plate wizard for Tasplaq – Example of modification of group n°2

I.3.4.3. "Equivalent parameters of a combined section" wizard

The "Wizard for calculating combined sections" is accessed by the button.

This wizard is used to define equivalent mechanical properties, or what can be called "homogenised parameters", if the plate section is not homogeneous. It should be noted that it may be useful to resort to this technique in certain specific cases:

- Modelling of a stratified slab consisting of a series of concrete layers with different mechanical characteristics;
- Taking account of a very stiff base layer (e.g. cemented gravel) by homogenisation with the concrete layer.

In this wizard, a choice of two contact conditions is proposed:

- Perfect contact: this is represented by continuity of deformation over the height of the section. The equivalent inertia product is obtained by integration with respect to the global neutral plane;
- Slippery contact: in this case, there is no continuity of deformation at the interface between the layers, however the curves are parallel and the equivalent product of inertia is obtained as the sum of two products of inertia "specific" to each section.

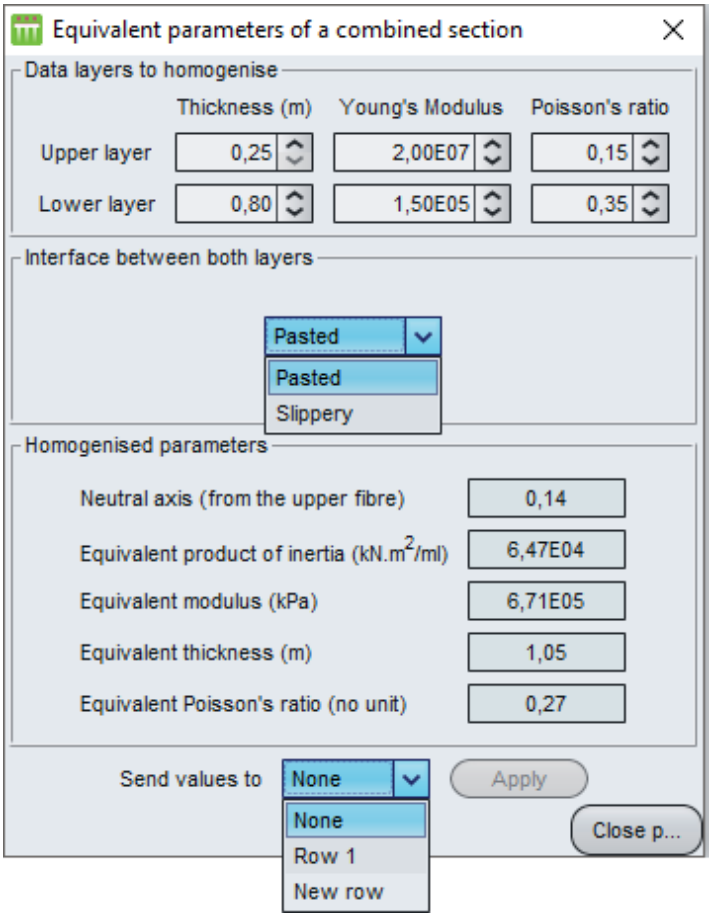


Figure I.42 : Wizard: Equivalent parameters of a combined section

The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Thickness	m	-	Always	Yes, for upper layer and lower layer	> 0
Young's modulus	kPa	-	Always		> 0
Poisson's ratio ν	-	-	Always		$0 \leq \nu < 0.5$
Interface between the 2 layers	-	Pasted	Always	Yes	Possible values: • Pasted • Slippery
Neutral axis (from the upper fibre)	m		Only if "Interface between both layers" pasted	-	Values calculated automatically (non-modifiable)
Equivalent product of inertia	kN.m²/ml		Always	-	
Equivalent modulus	kPa		Always	-	
Equivalent thickness	m		Always	-	
Equivalent Poisson's ratio	-		Always	-	

Table I.16 : Parameters of "Combined section" wizard

If **Send values to** is set to "New row" or a row previously created and selected, the Apply button can be used to import the data into the project.

Click the Close button to close the wizard.

I.3.5. "Deactivation" tab

This tab is used to deactivate a group of rectangular zones in the plates thus defined. This in particular can be used to define joints, non-rectangular plates, multiple separate plates, etc.

This step is not mandatory.

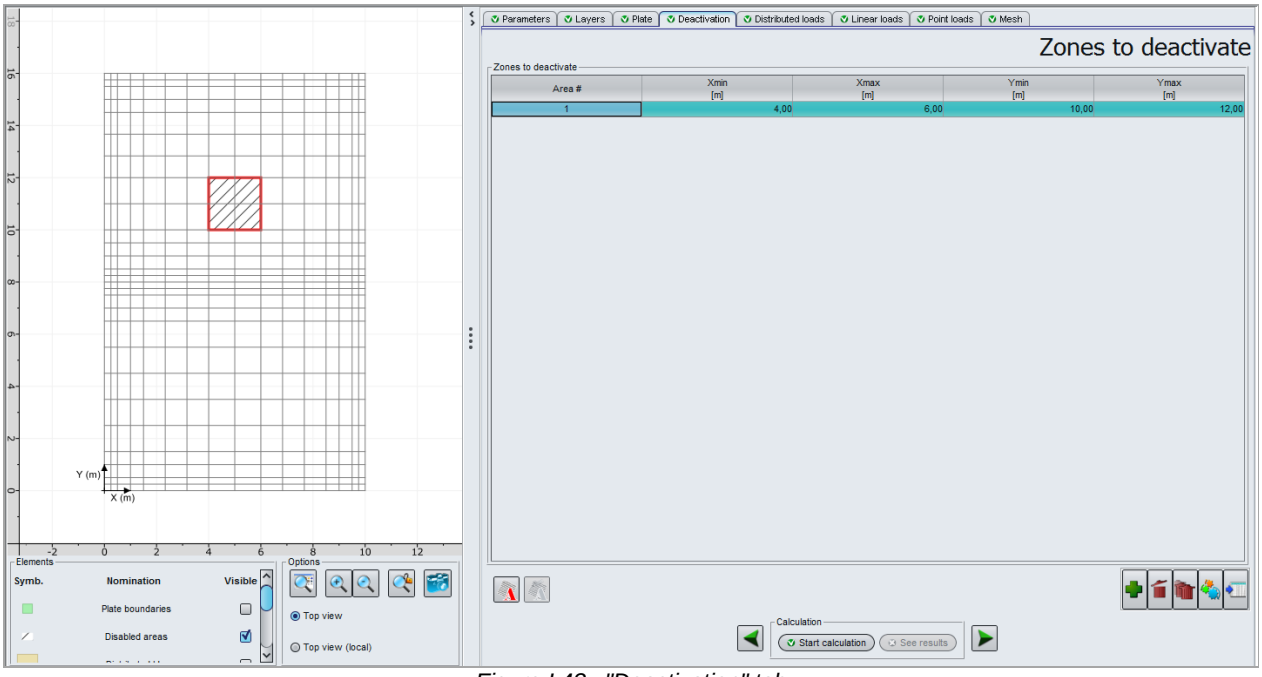


Figure I.43 : "Deactivation" tab

I.3.5.1. Definition of deactivated zones

Each deactivated zone is assumed to have a rectangular shape and be defined by four parameters: Xmin, Xmax, Ymin and Ymax. They can be used to determine the perimeter of the zone along X and Y. Users can easily situate themselves using the graphic representation (Figure I.43).

The deactivation parameters to be defined for each deactivated zone are given below:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Xmin: beginning of zone X-axis	m	0.00	Always	Yes	≥ 0
Xmax: end of zone X-axis	m	0.00	Always	Yes	> Xmin
Ymin: beginning of zone Y-axis	m	0.00	Only in 3D mode	Yes	≥ 0
Ymax: end of zone Y-axis	m	0.00	Only in 3D mode	Yes	> Ymin

Table I.17 : Deactivation parameters

The graphic representation of the plate now takes account of the deactivated zones. These are white and cross-hatched. When a zone is selected from the list it is given a red border in the graphic representation.

The "Deactivation" tab comprises 1 wizard to help the user define geometrical zones that are either a user-defined shape or circular.

I.3.5.2. "Deactivation" wizard

For 3D projects, the deactivation wizard is available by clicking the button. This wizard is used to define the zones to be deactivated and to transfer these selected zones to the project. This wizard comprises two tabs, described below.

I.3.5.2.1. Circular deactivation

This wizard is used to generate a group of rectangular zones equivalent to a uniform circular deactivation.

The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Point A (X_A, Y_A,): coordinates of centre of circle	m	(0,0)	Always	Yes	-
Radius of deactivated zone	m	0	Always	Yes	> 0
Subdivisions*	-	10	Always	Yes	> 0

Table I.18 : Parameters for the "Circular deactivation" wizard

* Subdivisions: Constant N. The number of subdivisions will be Nr = 2 x N + 1

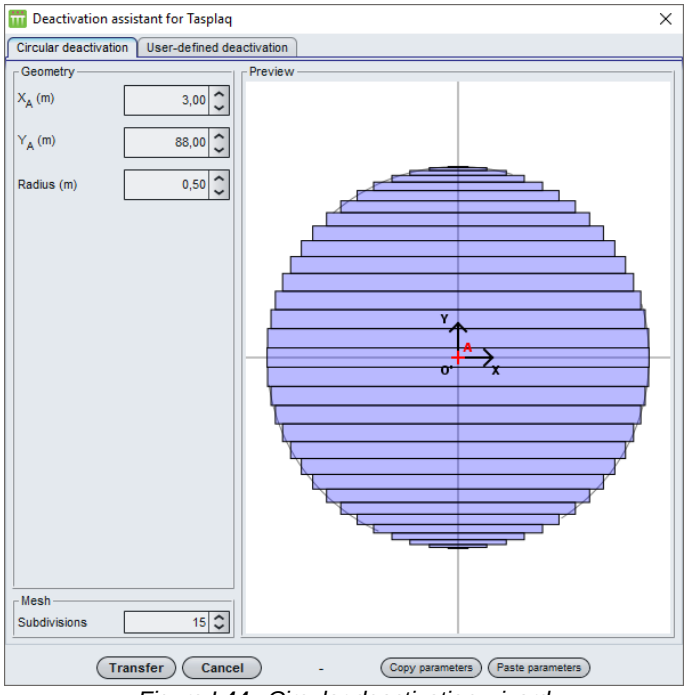


Figure I.44 : Circular deactivation wizard

Clicking the Transfer bouton will import all corresponding data into the project and close the wizard.

I.3.5.2.2. User-defined deactivation

This wizard is used to generate a group of rectangular zones equivalent to deactivation of the configured shape.

The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Coordinates of points (X_A, Y_A, X_B, Y_B, X_C, Y_C, X_D, Y_D,)	m	(0,0,0,0)	Always	Yes	-
Number of elements on X	-	7	Always	Yes	-
Number of elements on Y	-	7	Always	Yes	-

Table I.19 : Parameters for the "User-defined deactivation" wizard

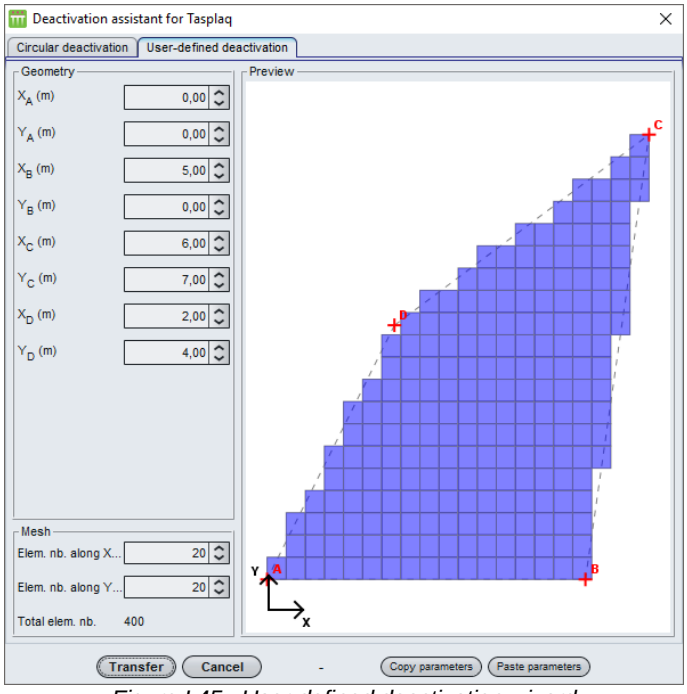


Figure I.45 : User-defined deactivation wizard

Clicking the Transfer button enables all the corresponding data to be input into the project.

Modification of a deactivation group

Selecting a line from the list places a green border around the graphic representation in the left-hand part of the screen and gives access to the selected deactivation group modification wizard :

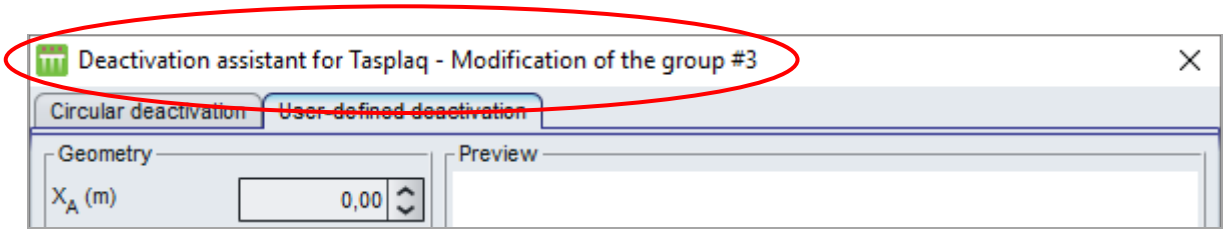


Figure I.46 : Deactivation wizard for Tasplaq – Example of group modification n°3

Management of parameters

For these two wizards, two buttons are used to manage the parameters input, which is useful for complex geometry:

- The Copy parameters button can be used to export the wizard configuration to the Windows® clipboard, in preparation for subsequent import.

The Paste parameters button can be used to import the previously exported wizard configuration from the clipboard.

I.3.6. "Distributed loads" tab

This tab is used to define (in the local coordinates system of the plate) one or more distributed loads applied to the plate, as well as one or more surface stiffnesses, if any, under the plate. This option is available for 3D and 2D projects.

As before, these loads are defined by rectangular zones.

The information to be input differs as a function of the type of calculation chosen.

This step is not mandatory.

I.3.6.1. Distributed loads in the case of 3D projects

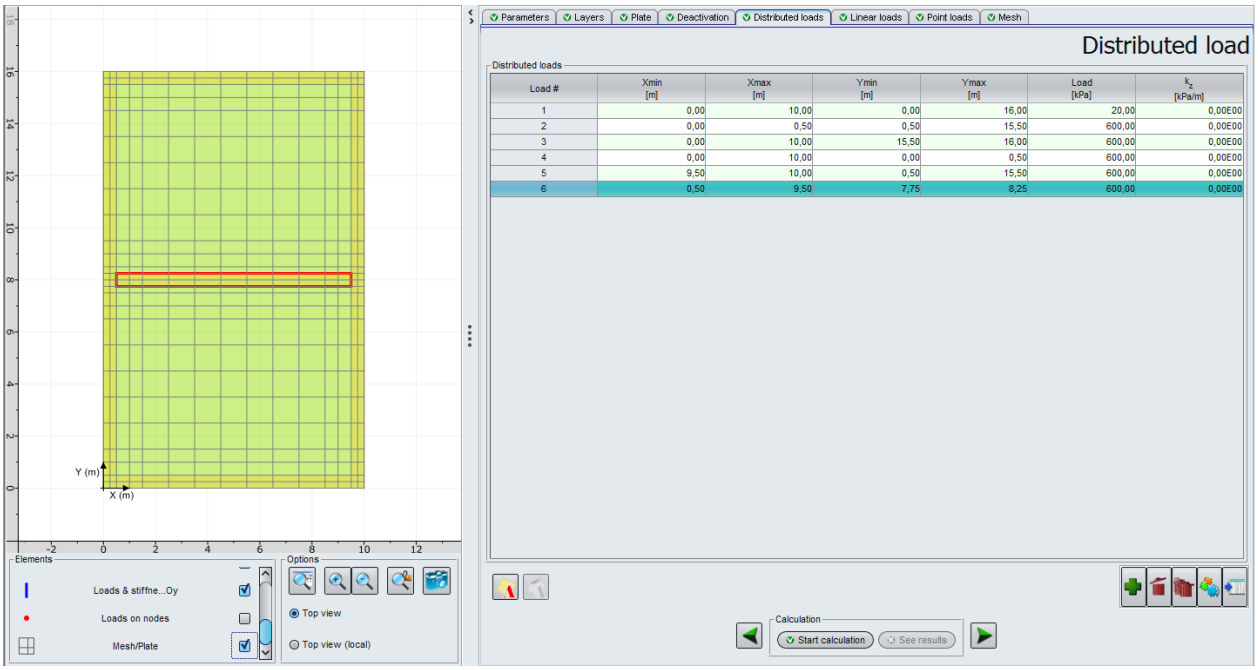


Figure I.47 : "Distributed loads" tab – 3D projects

Below is a summary table of the distributed loads parameters in 3D:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Xmin: beginning of zone X-axis	m	Cover the entire plate	Always	Yes	≥ 0
Xmax: end of zone X-axis	m		Always	Yes	> Xmin
Ymin: beginning of zone Y-axis	m		Always	Yes	≥ 0
Ymax: end of zone Y-axis	m		Always	Yes	> Ymin
Load: uniform density of the distributed load	kPa	0	Always	Yes	One of the 2 values must be other than zero
k_z: surface stiffness* under the plate	kPa/m	0	Always		

Table I.20 : Parameters for distributed loads in 3D

k_z is a surface stiffness in vertical displacement under the plate (reaction coefficient), representative for example of a distribution of juxtaposed springs. This can, for example, be an alternative means of representing the supporting soil **instead** of a representation by an elastic multilayer medium.

Caution, if several loads are defined for the same zone, they are cumulative. The same applies to stiffness values.

I.3.6.2. Distributed loads in the case of 2D projects

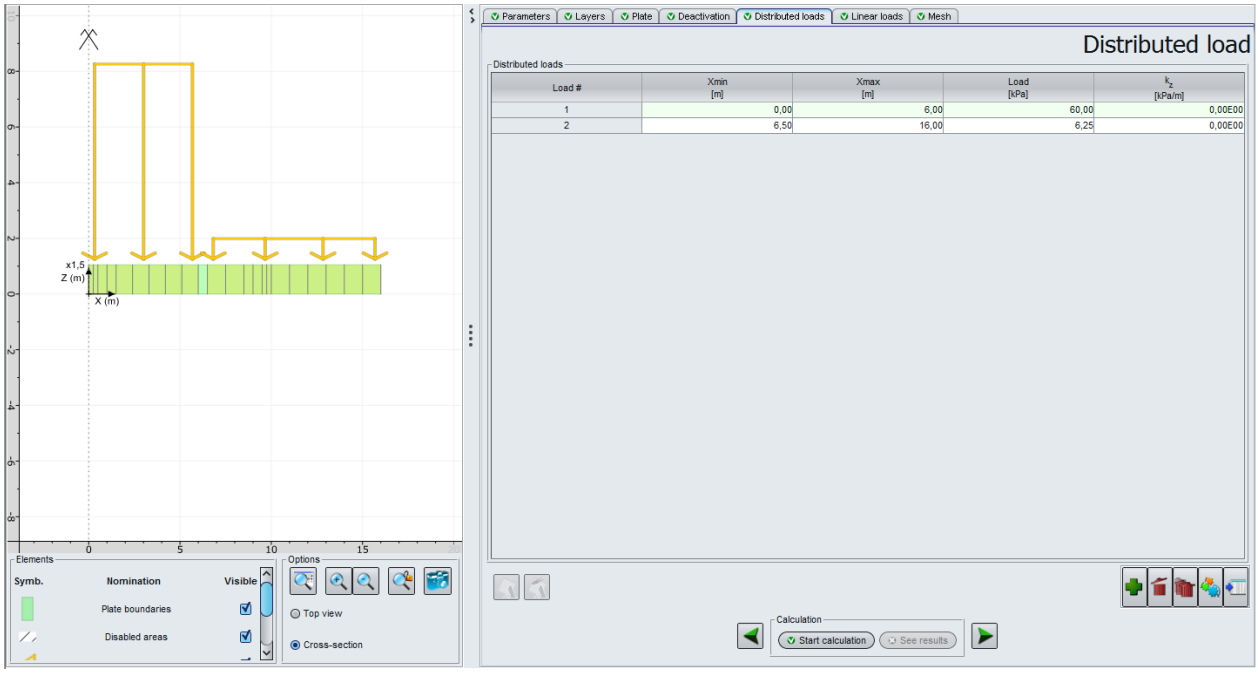


Figure I.48 : "Distributed loads" tab – 2D projects

Below is a summary table of the distributed load parameters in 2D:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Xmin: beginning of zone X-axis	-	Cover the entire plate	Always	Yes	≥ 0
Ymin: end of zone X-axis	-		Always	Yes	> Xmin
$Load_{deb}$: value of distributed load at beginning of zone)	kPa	-	Only for plane deformations	-	At least one of the values displayed must be other than zero
Lod_{fin}: value of distributed load at end of zone	kPa	-	Only for plane deformations	-	
Load: uniform density of the distributed load	kPa	-	Only in axisymmetry	-	
K_z: surface stiffness* under the plate	kPa/m	-	Always	-	

Table I.21 : Parameters for distributed loads in 2D

As above, if several loads are defined for the same zone, they are cumulative. The same applies to stiffness values.

I.3.6.3. Load definition wizards

These wizards are only accessible for 3D projects, defined in 2 dimensions (x, y) – in 2D projects, the loads are defined in a single dimension.

To make it easier to define "normal" loads, this tab has a Wizard button for a simple definition of:

- a circular distributed load;
- a user-defined distributed load;

Figure I.49 illustrates the various available wizards:

- Choose the type of load;
- Fill out the various input fields;
- Click the Transférer button.

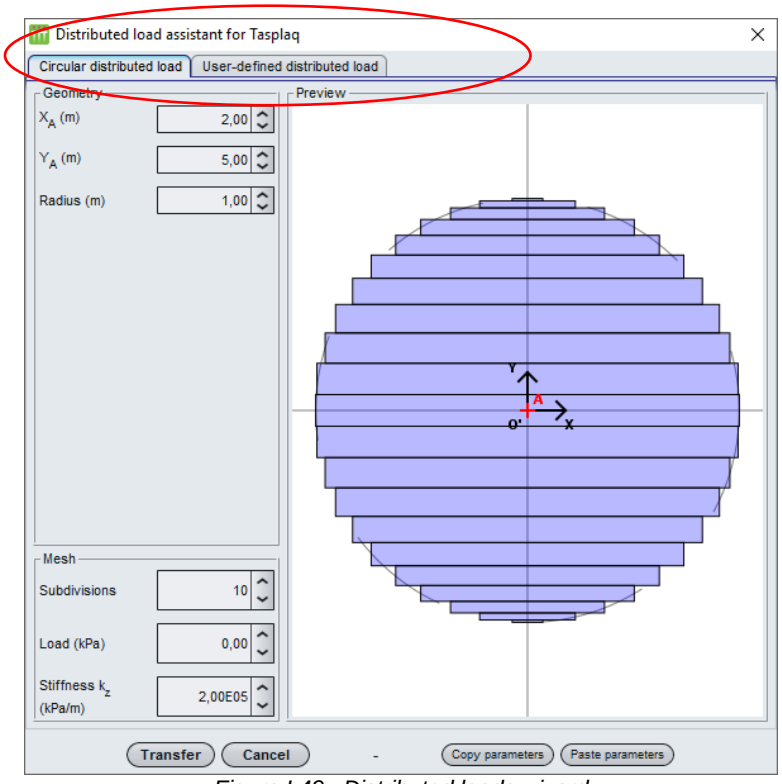


Figure I.49 : Distributed loads wizard

The description of use of the various load windows is explained in the following sub-chapters.

Note: It is possible to use several wizards or the same Loads wizard several times in the context of the same Tasplaq calculation.

I.3.6.3.1. Wizard: "Circular distributed load"

This wizard is used to generate a group of rectangular loads equivalent to a uniform circular load.
The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Point A (X_A, Y_A,): coordinates of centre of disk	m	(0,0)	Always	Yes	-
Radius of loaded disk	m	-	Always	Yes	>0
Subdivisions	-	10	Always	Yes	>0
Stiffness k_z	kPa/m	-	Always	Yes	-

Table I.22 : Parameters for uniform circular load

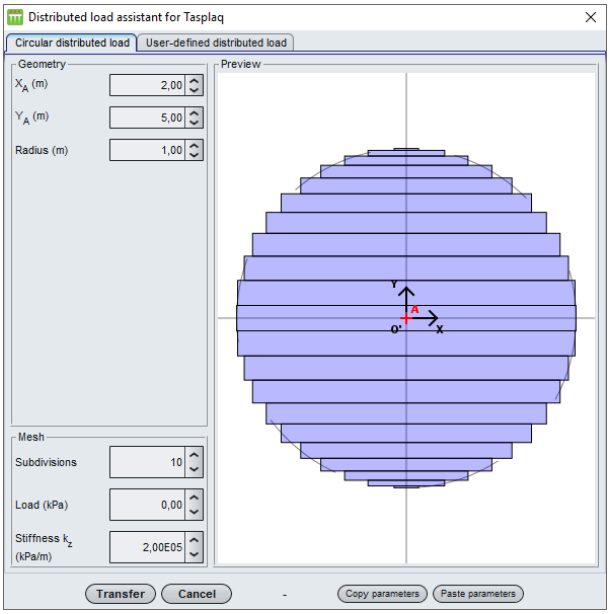


Figure I.50 : "Circular distributed load" wizard

I.3.6.3.2. Wizard: "User-defined distributed load"

This wizard is used to generate a group of rectangular loads equivalent to a user-defined distributed load.

The data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Point A (X_A, Y_A, X_B, Y_B, X_C, Y_C, X_D, Y_D): coordinates of the 4 points	m	(0, 0, 0, 0)	Always	Yes	-
Nbr of elements on X and Y	-	(7, 7)	Always	Yes	-
Density (in A, B, C D)	kPa	0	Always	Yes	-
Stiffness k_Z	kPa/m	0	Always	Yes	-

Table I.23 : Parameter for a user-defined distributed load

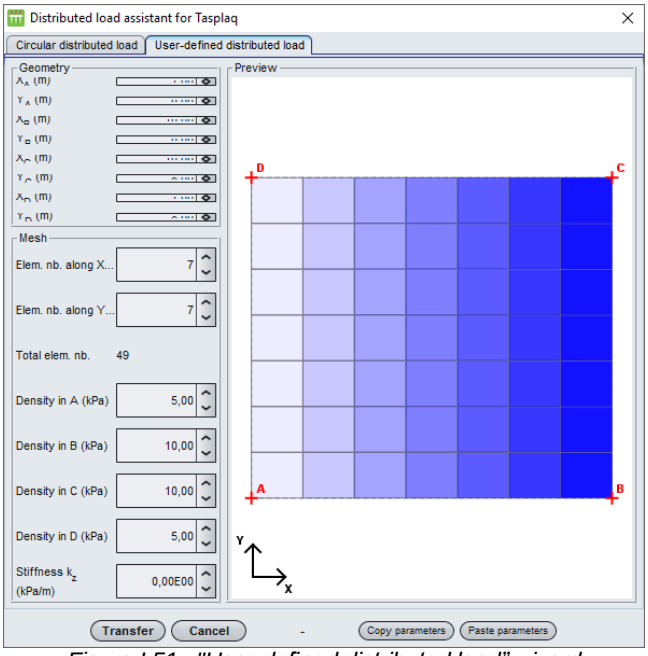


Figure I.51 : "User-defined distributed load" wizard

Generation of rectangular loads, representing the circular load or user-defined load, and calculation of their characteristics Xmin, Xmax, Ymin, Ymax, load and stiffness kz, are activated by clicking the Transfer button.

I.3.7. "Linear loads" tab

This tab is used to define (in the plate local coordinates system) the linear loads for 2D and 3D projects. The information will differ as a function of the type of calculation chosen.

This step is not mandatory.

I.3.7.1. Linear loads and stiffnesses in the case of 3D projects

The plate may be subjected to one or more linear loads along axis Ox, each defined on a segment [Xmin, Xmax]. Each overload is characterised by a linear load density (kN/m) and a possible linear translation stiffness k_z.

A linear load along Ox is parallel to axis Ox, placed along the Y-axis line and bounded by 2 X-axis nodes Xmin and Xmax: the coordinates system principle for each zone is thus similar to that used for the rectangular zones, except that here Ymin = Ymax = Y. The load and/or stiffness k_z is applied to the segment defined in this way.

The plate may also be subjected to one or more linear loads along axis Oy, similarly characterised by a fixed X-axis point equal to X and two Y-axis points Ymin and Ymax.

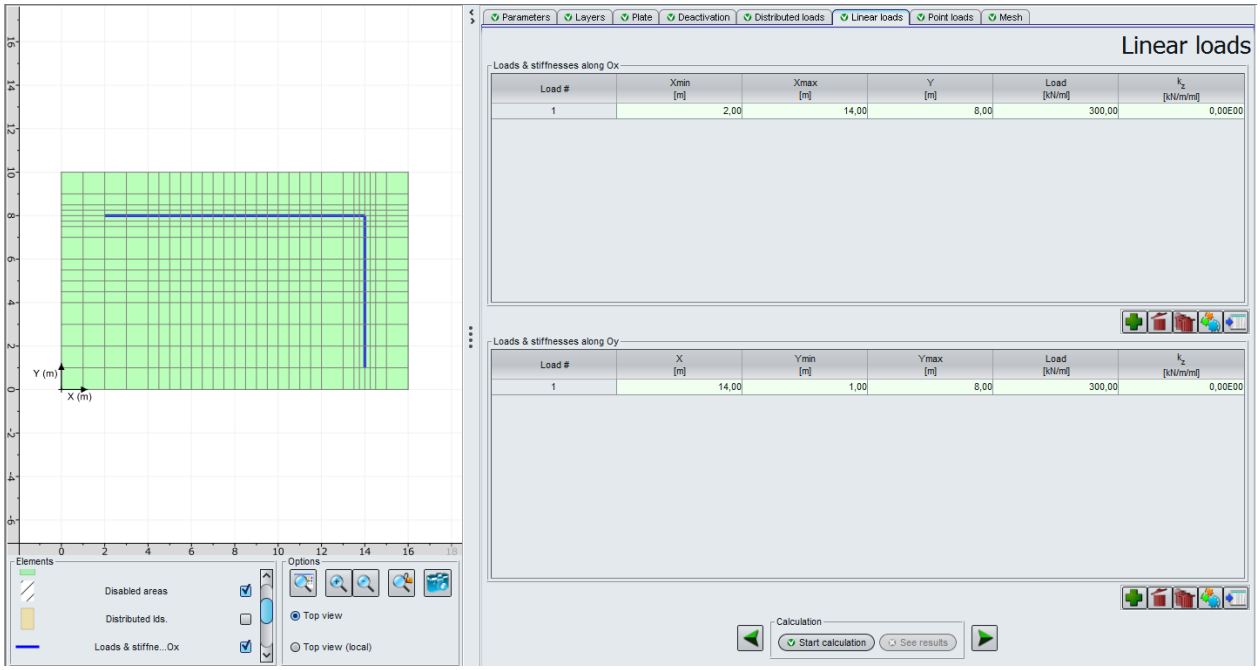


Figure I.52 : "Linear loads" tab – 3D project

Below is the summary table of the linear load parameters in 3D:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Xmin: beginning of zone X-axis	m	0	Along Ox	Yes	≥ 0
Xmax: end of zone X-axis	m	0		Yes	> Xmin
Y: Load Y-axis	m	0		Yes	≥ 0
X: Linear load X-axis	m	0	Along Oy	Yes	≥ 0
Ymin: beginning of zone Y-axis	m	0		Yes	≥ 0
Ymax: end of zone Y-axis	m	1		Yes	> Ymin
Load: linear load uniform density	kN/ml	-	Always	Yes	One of these 2 values must be other than zero
k_z: linear stiffness under plate	kN/m/ml	-	Always	Yes	

Table I.24 : Parameters for linear loads in 3D

I.3.7.2. Linear loads and stiffnesses in the case of 2D projects

For a 2D project, the plate may be subjected to one or more linear loads, each applied to a given X-axis point.

The characteristics of a linear load are as follows: linear stiffness under plate k_z, linear force per node F_z, linear bending moment per node M_x (or M_r) and linear rotation stiffness per node C_x (or C_r).

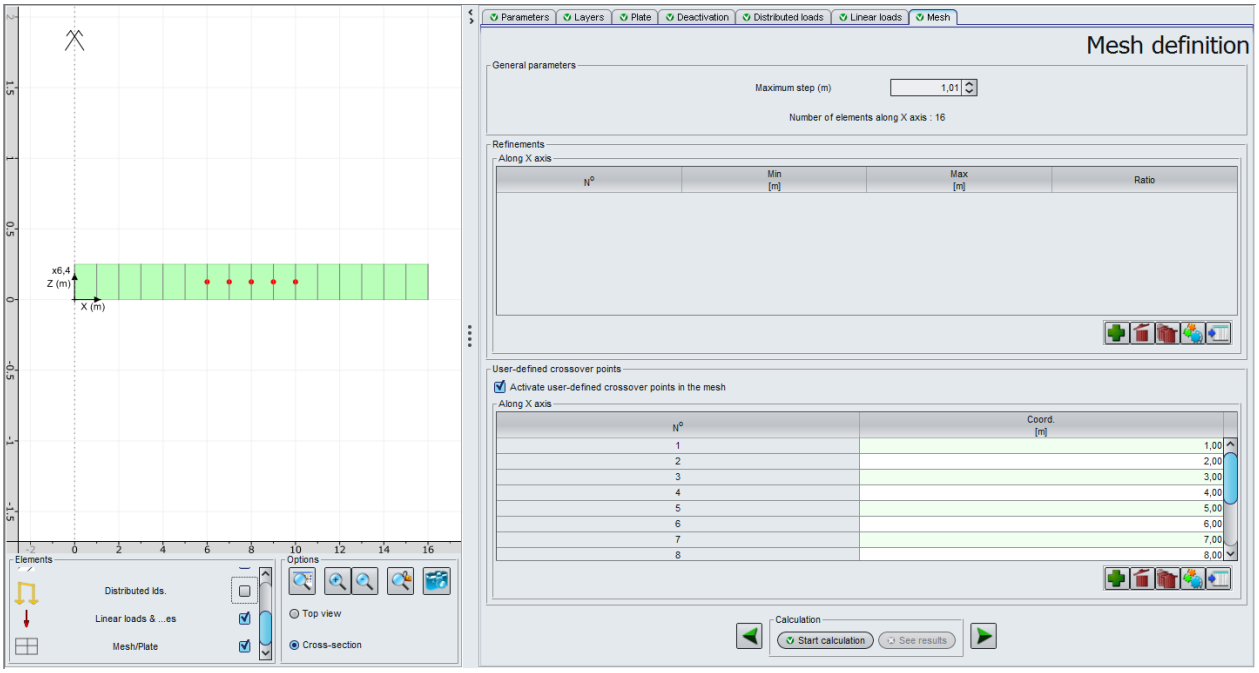


Figure I.53 : "Linear loads" tab – 2D project

Below is the summary table for the linear loads parameters in 2D:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
X: X-axis in local coordinates system	m	0	Always	Yes	≥ 0
F_z: linear force per node	kN/ml	-	Always	Yes	At least one of the 4 values displayed must be other than zero
M_x: Linear bending moment per node	kN.m/ml	-	In plane deformation	Yes	
M_r: Radial linear bending moment per node	kN.m/ml	-	In axisymmetry	Yes	
K_z: linear stiffness per node	kN/m/ml	-	Always	Yes	
C_x: linear stiffness in rotation per node	kN.m/rad/ml	-	In plane deformation	Yes	
C_r: linear stiffness in rotation per node	kN.m/rad/ml	-	In axisymmetry	Yes	

Table I.25 : Parameters for linear loads in 2D

I.3.8. "Point loads" tab

This tab is only visible for 3D projects. This step is not mandatory.

Each point load is constituted by: a point force F_z, two bending moments M_x and M_y, a point stiffness per node k_z, two rotation stiffness values C_x and C_y. These data are allocated to a given node (X, Y).

The input values apply to each node in the defined group.

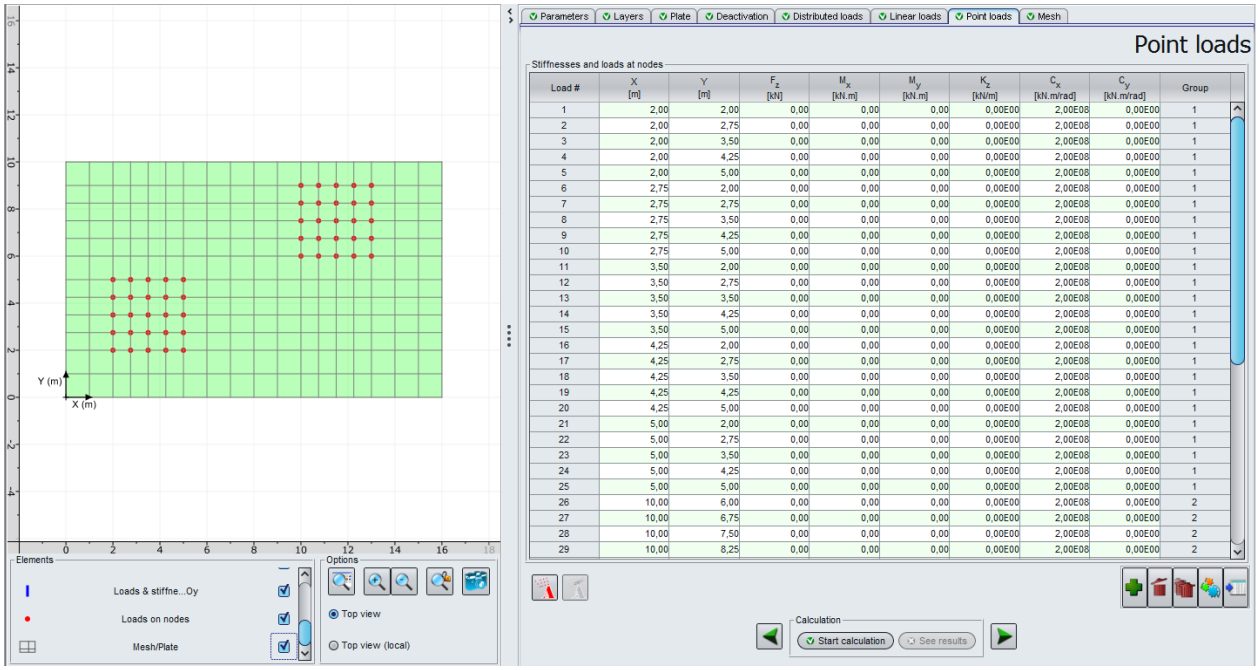


Figure I.54 : "Point loads" tab

Below is a summary of the parameters to be input:

Designation	Unit	Defaul t value	Display condition	Mandatory value	Local checks
X: load X-axis	m	0	Always (3D)	Yes	≥ 0
Y: load Y-axis	m	0	Always (3D)	Yes	≥ 0
F_z: point force	kN	-	Always (3D)	Yes	At least one of these data must be other than zero
M_x: moment around axis (-Oy)	kN.m	-	Always (3D)	Yes	
M_y: moment around axis (Ox)	kN.m	-	Always (3D)	Yes	
K_z: point stiffness per node	kN/m	-	Always (3D)	Yes	
C_x: rotation stiffness with respect to (-Oy)	kN.m/rad	-	Always (3D)	Yes	
C_y: rotation stiffness with respect to Ox	kN.m/rad	-	Always (3D)	Yes	

Table I.26 : Parameters for point loads

I.3.8.1. "Point loads" wizards

These wizards allow automatic generation of aligned or distributed points according to predetermined geometries. They can be accessed by clicking the button.

It is possible to use the two wizards, or the same calculation points wizard several times in the context of the same Tasseldo calculation.

After using at least one "point loads" wizard, the button becomes accessible: it can be used to modify the selected group of calculation points.

I.3.8.1.1. Point loads distributed along a segment

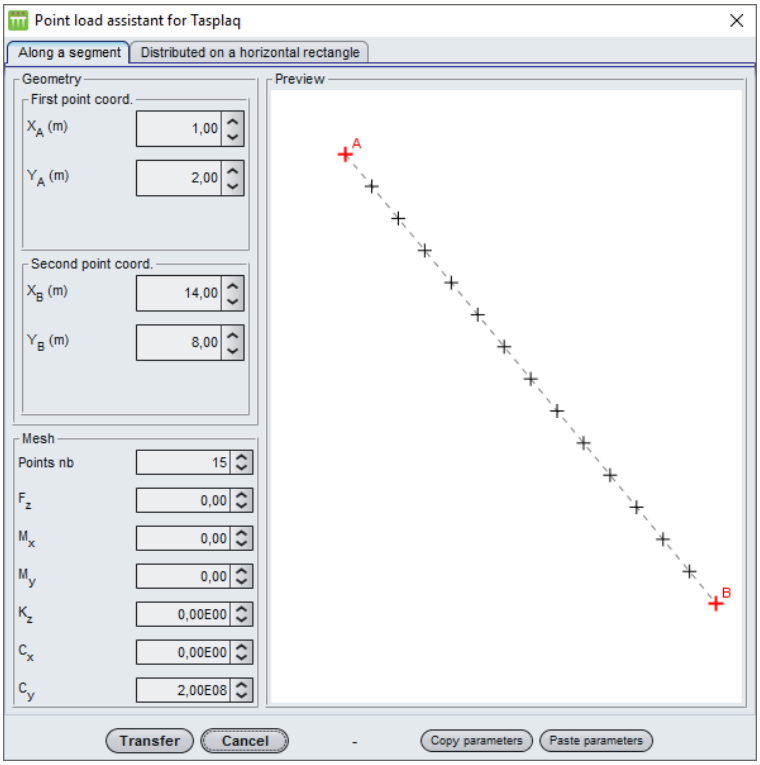


Figure I.55 : Wizard for point loads along a segment

The parameters to be filled out are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
X_A and **Y_A:** coordinates of point A	m		Always (3D)	Yes	≥ 0
X_B and **Y_B:** coordinates of point B	m		Always (3D)	Yes	≥ 0
Number of points	-	10	Always (3D)	Yes	-
F_z: point force	kN	-	Always (3D)	Yes	At least one of these data must be other than zero
M_x: moment around axis (-Oy)	kN.m	-	Always (3D)	Yes	
M_y: moment around axis (Ox)	kN.m	-	Always (3D)	Yes	
K_z: point stiffness per node	kN/m	-	Always (3D)	Yes	
C_x: rotation stiffness with respect to (-Oy)	kN.m/rad	-	Always (3D)	Yes	
C_y: rotation stiffness with respect to Ox	kN.m/rad	-	Always (3D)	Yes	

Table I.27 : Parameters for the point loads along a segment wizard

I.3.8.1.2. Point loads distributed on a horizontal rectangle

This wizard is used to automatically define a mesh of calculation points distributed on a horizontal rectangle [A, B, C, D].

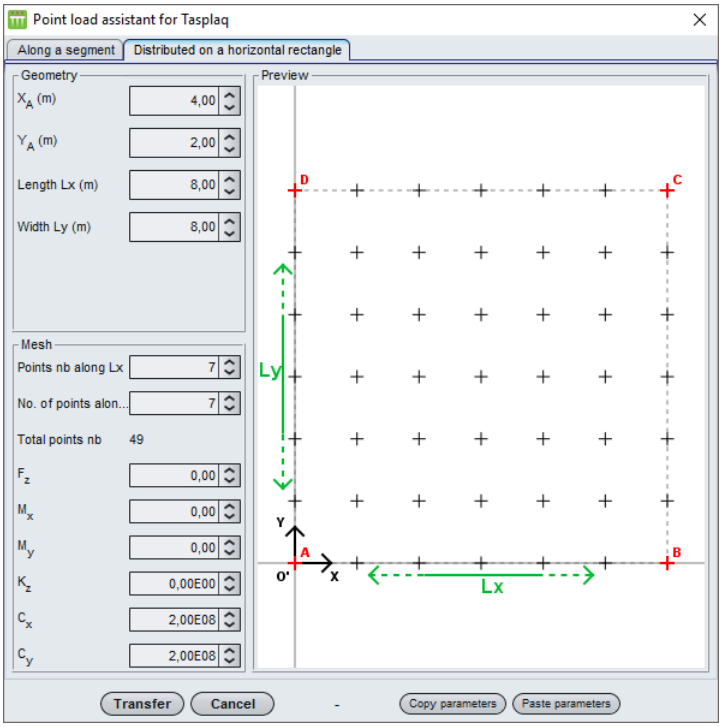


Figure I.56 : Wizard for distributed loads on a horizontal rectangle

The parameters to be filled out are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
X_A and **Y_A:** coordinates of point A	m	0	Always	Yes	-
Length Lx: length of rectangle	m	0	Always	Yes	-
Width Ly: width of rectangle	m	0	Always	Yes	-
Nb of points along Lx	-	0	Always	Yes	-
Nb of points along Ly	-	0	Always	Yes	-
F_z: point force	kN	-	Always (3D)	Yes	At least one of these data must be other than zero
M_x: moment around axis (-Oy)	kN.m	-	Always (3D)	Yes	
M_y: moment around axis (Ox)	kN.m	-	Always (3D)	Yes	
K_z: point stiffness per node	kN/m	-	Always (3D)	Yes	
C_x: rotation stiffness with respect to (-Oy)	kN.m/rad	-	Always (3D)	Yes	
C_y: rotation stiffness with respect to Ox	kN.m/rad	-	Always (3D)	Yes	

Table I.28 : Parameters for the corner points of a distributed load on a horizontal and rectangular area

When the wizard data have been filled out, click the Transfer button: the points generated are then automatically copied into the table of point loads.

I.3.9. "Unsticking/Plastification" tab

This tab appears if the "Manual definition of unsticking/plastification of the plate" option is ticked in the "Advanced parameters" frame of the "Parameters tab", whatever the type of project (2D or 3D).

This tab can be used manually to define:

- The unsticking of certain zones: the soil reaction (multilayer medium) is then equal to 0 and soil settlement is no longer equal to plate vertical displacement. Generally speaking, this option can be used to eliminate the contact between plate and soil under a predetermined zone. This may be useful when simulating certain particular situations, such as that in which the plate only partially rests on the supporting soil, as schematically shown in the following figure.

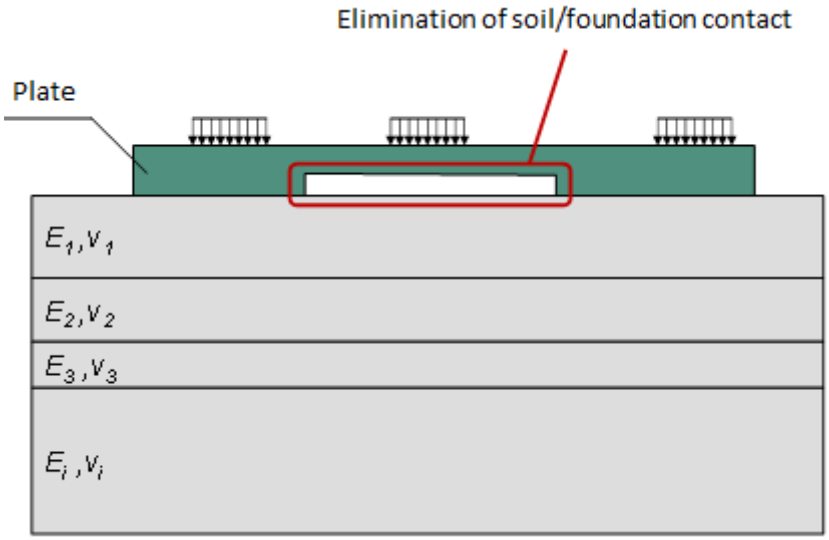


Figure I.57 : Case of a pate partially supported by the soil

- The plastification of certain zones: the soil reaction (multilayer medium) defined is equal to the plastification threshold defined in the "Parameters" tab. Soil settlement and vertical displacement of the plate are thus still equal.

It is possible to combine "Manual definition of unsticking/plastification of the plate" with the "Automatic calculation" option: if the "Automatic calculation" option is activated, Tasplaq checks unsticking/plastification at all the nodes, except those which are situated in the zones declared as unstuck/plastified manually by the user.

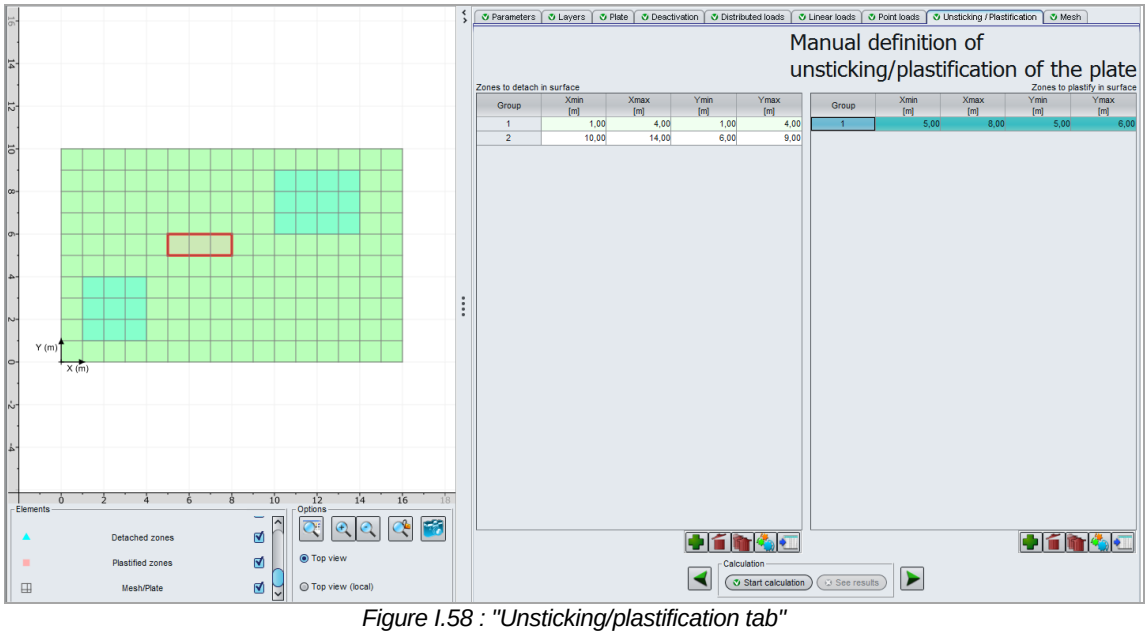


Figure I.58 : "Unsticking/plastification tab"

This option corresponds to advanced use of Tasplaq.

This step is not mandatory.

The types of data to be filled out are the same for the 2 frames: zones to be detached on surface and zones to be plastified on surface. However, the zones defined in the 2 frames may be different.

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Xmin: beginning of zone X-axis	m	0	Always	Yes	≥ 0
Xmax: end of zone X-axis	m	0	Always	Yes	> Xmin
Ymin: beginning of zone Y-axis	m	0	Only in 3D mode	Yes	≥ 0
Ymax: end of zone Y-axis	m	0	Only in 3D mode	Yes	> Ymin

Table I.29 : Parameters for manual definition of unsticking/plastification of the plate

I.3.10. "Mesh" tab

The "Mesh" tab is used to define the mesh for the entire working area, in local coordinates system. 3 frames are used to input the parameters of this mesh.

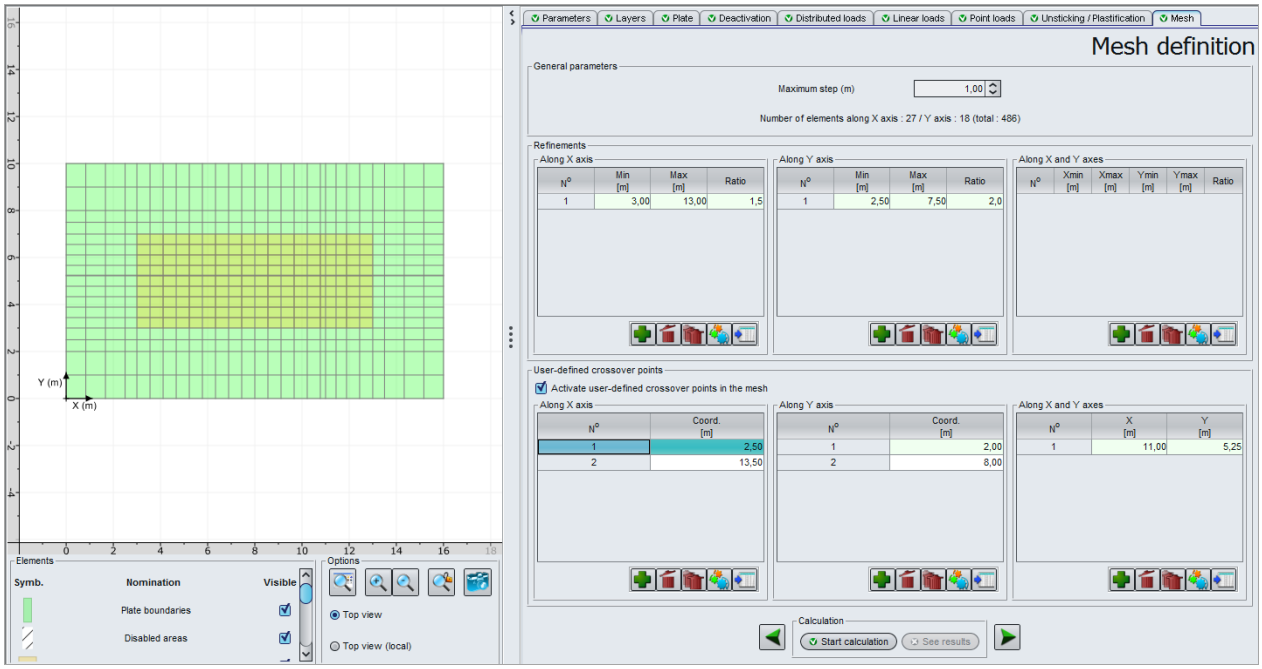


Figure I.59 : "Mesh" tab

I.3.10.1. "General parameters" frame

- Maximum step (pitch) = 1 by default (in metres)

Tasplaq automatically displays the number of elements along X and Y and calculates the total.

Once this parameter has been input, Tasplaq automatically calculates the mesh, taking account of everything defined in the project (plate zones, deactivations, loads, etc.).

If the user wishes greater precision, the automatically calculated mesh can be refined.

I.3.10.2. "Refinements" frame

Tasplaq proposes three mesh refinement options:

- along X axis;
- along Y axis;
- along X and Y axes simultaneously.

The "Along X axis" and "Along Y axis" data to be filled out are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Min: minimum coordinate to which refinement applies	m	0	Always	Yes	≥ 0
Max: maximum coordinate to which refinement applies	m	0	Always	Yes	≥ 0
Ratio*	-	0	Always	Yes	> 0

Table I.30 : Parameters for "Along X axis" and "Along Y axis" mesh refinement

The "Along X and Y axes" data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Xmin: minimum X-axis point to which refinement applies	m	0	Always	Yes	≥ 0
Xmax: maximum X-axis point to which refinement applies	m	0	Always	Yes	≥ 0
Ymin: minimum Y-axis point to which refinement applies	m	0	Always	Yes	≥ 0
Ymax: maximum Y-axis point to which refinement applies	m	0	Always	Yes	≥ 0
Ratio*	-	-	Always	Yes	> 0

Table I.31 : Parameters for "Along X and Y axes" mesh refinement

(*) The ratio is used to determine the maximum step (pitch) applied locally in the refined zone. This maximum local step (pitch) is equal to the maximum general pitch divided by the ratio. It is important to note that the ratios are multiplied if several refined zones are superposed.

I.3.10.3. "User-defined crossover points" frame

For even greater precision in the mesh, Tasplaq can force the mesh to pass through user-defined points. To do this, tick the "Activate user-defined crossover points in the mesh" box.

These can be activated

- along X axis;
- along Y axis;
- along X and Y axes simultaneously.

The "Along X axis" and "Along Y axis" data to be filled out are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
Coordinate: to be defined	m	-	Always	Yes	≥ 0

Table I.32 : Parameters for "Along X axis" and "Along Y axis" user-defined crossover

The "Along X and Y axes" data to be input are as follows:

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
X: X-axis to be defined	m	-	Always	Yes	≥ 0
Y: Y-axis to be defined	m	-	Always	Yes	≥ 0

Table I.33 : Parameters for "Along X and Y axes" user-defined crossover

The coordinate to be defined may be situated outside the framework. Tasplaq will extend automatic meshing accordingly, in order to incorporate this defined coordinate.

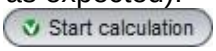
I.3.11. Calculation and results

I.3.11.1. Calculation

Calculation can be started from any tab, provided that the tabs are correctly filled out, in other words when they are all marked by a green tick (for example:

).

They are marked by a red cross (for example:

) until they have been correctly filled out (missing data or data not as expected).
To start calculation, click the 

button.

No numerical limit on the model size is considered in the programme. However, the actual limit could be the maximum memory size that can be allocated to the programme under Microsoft Windows®: this limit is estimated at 4,000 activated elements.

The general calculation process involves the following steps:

1. Read data – Open files;
2. Initialise variables;
3. Construct mesh;
4. Assemble external load vector;
5. Assemble plate stiffness matrix;
6. Calculate soil flexibility matrix (if there is a multilayer medium);
7. Constitute global equations system;
8. Matrix resolution;
9. Calculate displacements and forces in the plate;
10. Calculate settlements and reactions at all points (if there is an elastic multilayer medium);
11. Control unsticking/plastification on surface (if positive, return to step 4);
12. Generate output files (results, graphics);
13. End of programme.

The user is informed about how the various calculation steps are progressing in the following window:

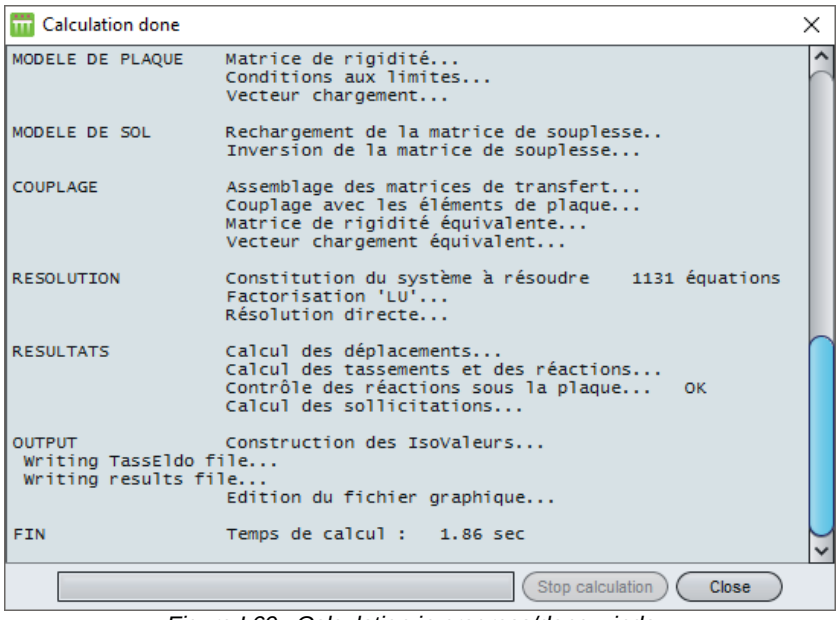


Figure I.60 : Calculation in progress/done window

To display the calculation results, click the Close button then See results.

I.3.11.2. Available results for 3D projects

The following window can be used to choose the type of results to be displayed. Different types of results are accessible in 3D:

- numerical results: formatted results and result tables;
- graphical results: cross-section charts, shadings (in 3D only), 3D charts (in 3D only);
- external wizards: dishomogenisation (in 3D only), export to Tasseldo.

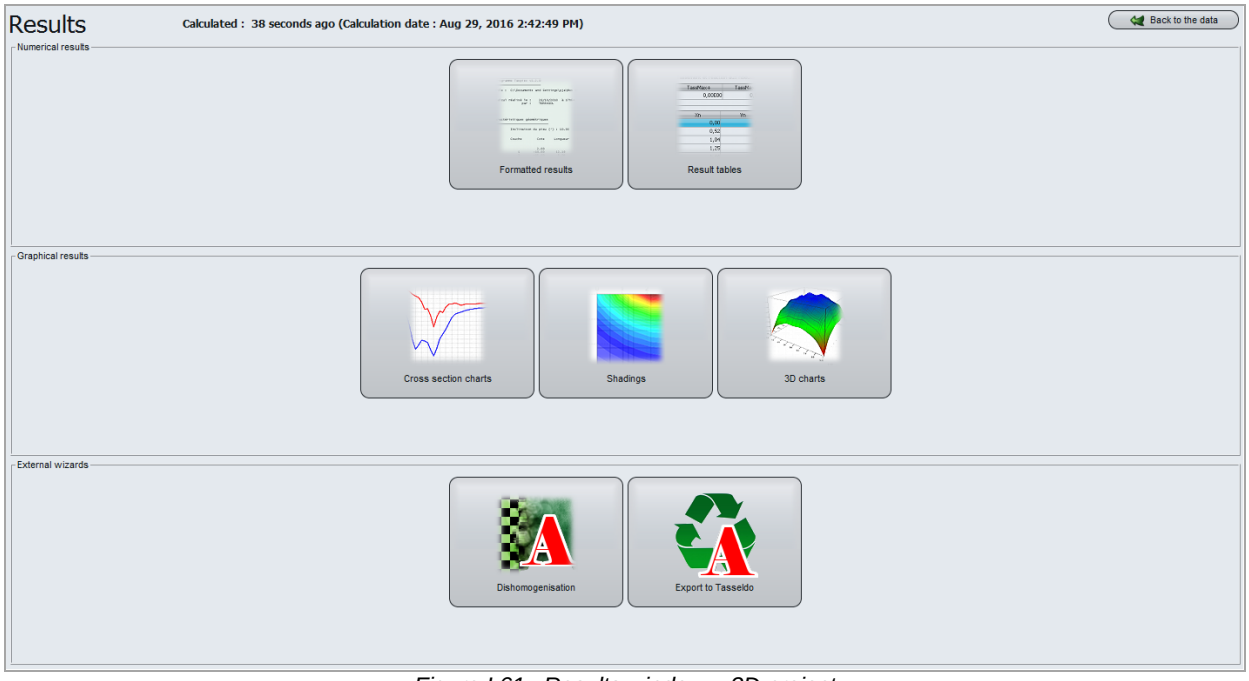


Figure I.61 : Results window – 3D project

I.3.11.2.1. Numerical results: formatted results

Formatted numerical results contain:

- A reminder of the data: general parameters, soil data, plate geometry and distribution, plate characteristics, calculation points, applied loads (and their distribution).
- Results:
  - Displacements and rotations at the activated nodes (**local** coordinates): for each node (x, y), Tasplaq gives:
    - displacement w (m), positive downwards;
    - rotations θ_x and θ_y (°).

    The maximum and minimum values w_{max} and w_{min} (m), and the coordinates of the nodes at which they are obtained are given under this table.
  - Soil reaction and settlement under the nodes (**global** coordinates): for each node, Tasplaq gives:
    - its coordinates in the global coordinates system;
    - its status (plastified or unstuck);
    - pressure (kPa): soil reaction under the plate;
    - soil settlement (m): positive downwards;
    - plate deflection (m): positive downwards.

    The following are specified at the bottom of this table:
    - the maximum and minimum values w_{max} and w_{min} (m) with the coordinates of the nodes at which they are obtained;
    - the maximum and minimum settlement values (m) with the coordinates of the nodes at which they are obtained;
    - the maximum and minimum pressure values (kPa) with the coordinates of the nodes at which they are obtained;
    - the total load transmitted to the soil (kN).

```
PROGRAM TASPLAQ v2.6.3                    (c) TERRASOL 2016

File :  Z:\Logiciel TS\02 - Foxta\Foxta v3\Projets\TASPLAQ\dallage rectangulaire\DALLAGE RECTANGULAIRE\dallage rectangulaire[TQ].resu

Calculation made on :  08/29/2016 at 14h42
                by :  Terrasol

   Detailed printing :          Yes

Iteration :      02

Calculation settings

    Loading stiffness matrices :          No
    Saving stiffness matrices :           No
    Automatic control :                   Yes
    Symmetry :                            no symmetry

    Separation threshold :        -5.000
    Compression threshold :    15000.000
    Initial stress                 0.000

Soil properties

layer      Bottom        Modulus       Poisson's ratio

 01       -10.000       0.100E+05           0.350
 02       -35.000       0.100E+05           0.350

Plate geometry

                        General dimensions
                        ------------------
          XP        YP        Lx        Ly        ZP       teta
        0.00      0.00     16.00     10.00      0.00      0.00

                        No elements deactivation
```

```
                ****************
                ****SOLUTION****
                ****************

          Deflection (w) and rotations of activated nodes (local coordinate system)

          X         Y          W          TetaX      TetaY
        0.000     0.000   0.388446E-02  -0.103823E-02  -0.926110E-03
        0.200     0.000   0.409172E-02  -0.103258E-02  -0.942834E-03
        0.400     0.000   0.429691E-02  -0.101822E-02  -0.959210E-03
        0.600     0.000   0.449866E-02  -0.998517E-03  -0.978017E-03
        0.800     0.000   0.469607E-02  -0.975132E-03  -0.100027E-02
        1.000     0.000   0.488851E-02  -0.948770E-03  -0.102644E-02
        2.000     0.000   0.577865E-02  -0.830763E-03  -0.119846E-02
        3.000     0.000   0.655162E-02  -0.718125E-03  -0.142227E-02
        4.000     0.000   0.721404E-02  -0.607008E-03  -0.166804E-02
        4.500     0.000   0.750216E-02  -0.545148E-03  -0.178843E-02
        5.000     0.000   0.775846E-02  -0.479365E-03  -0.190166E-02
        5.500     0.000   0.798064E-02  -0.408618E-03  -0.200349E-02
        6.000     0.000   0.816624E-02  -0.333091E-03  -0.209063E-02
        6.500     0.000   0.831303E-02  -0.253510E-03  -0.216058E-02
        7.000     0.000   0.841922E-02  -0.170874E-03  -0.221160E-02
        7.500     0.000   0.848356E-02  -0.862464E-04  -0.224259E-02
        8.000     0.000   0.850530E-02  -0.645197E-06  -0.225294E-02
        8.500     0.000   0.848420E-02   0.849707E-04  -0.224247E-02
        9.000     0.000   0.842049E-02   0.169648E-03  -0.221135E-02
        9.500     0.000   0.831489E-02   0.252388E-03  -0.216016E-02
       10.000     0.000   0.816862E-02   0.332158E-03  -0.209000E-02
       10.500     0.000   0.798341E-02   0.408006E-03  -0.200260E-02
       11.000     0.000   0.776142E-02   0.479266E-03  -0.190044E-02
       11.500     0.000   0.750499E-02   0.545830E-03  -0.178679E-02
       12.000     0.000   0.721626E-02   0.608834E-03  -0.166590E-02
       13.000     0.000   0.655031E-02   0.723735E-03  -0.141891E-02
       14.000     0.000   0.576873E-02   0.842950E-03  -0.119381E-02
       15.000     0.000   0.486084E-02   0.974592E-03  -0.102464E-02
       16.000     0.000   0.383698E-02   0.104759E-02  -0.925110E-03
        0.000     1.000   0.478249E-02  -0.111834E-02  -0.838436E-03
        0.200     1.000   0.500653E-02  -0.112137E-02  -0.860913E-03
        0.400     1.000   0.523076E-02  -0.112026E-02  -0.884916E-03
        0.600     1.000   0.545441E-02  -0.111565E-02  -0.910966E-03
        0.800     1.000   0.567686E-02  -0.110852E-02  -0.939380E-03
        1.000     1.000   0.589772E-02  -0.110004E-02  -0.970396E-03
        2.000     1.000   0.696799E-02  -0.103891E-02  -0.116145E-02
        3.000     1.000   0.797222E-02  -0.965924E-03  -0.140582E-02
        4.000     1.000   0.889010E-02  -0.861941E-03  -0.167893E-02
        4.500     1.000   0.930381E-02  -0.790369E-03  -0.181395E-02
        5.000     1.000   0.967813E-02  -0.704578E-03  -0.194050E-02
       12.000     8.000   0.105554E-01   0.117195E-02   0.164124E-02
       13.000     8.000   0.933053E-02   0.126247E-02   0.130750E-02
       14.000     8.000   0.805223E-02   0.128708E-02   0.102411E-02
       15.000     8.000   0.676238E-02   0.129114E-02   0.810614E-03
       16.000     8.000   0.547592E-02   0.127633E-02   0.650813E-03
        0.000     9.000   0.473144E-02  -0.114235E-02   0.834522E-03
        1.000     9.000   0.586738E-02  -0.111823E-02   0.966942E-03
        2.000     9.000   0.695411E-02  -0.105200E-02   0.115785E-02
        3.000     9.000   0.796814E-02  -0.972800E-03   0.140328E-02
        4.000     9.000   0.889073E-02  -0.864839E-03   0.167736E-02
        4.500     9.000   0.930556E-02  -0.792006E-03   0.181277E-02
        5.000     9.000   0.968046E-02  -0.705326E-03   0.193964E-02
        5.500     9.000   0.100088E-01  -0.606146E-03   0.205314E-02
        6.000     9.000   0.102849E-01  -0.496402E-03   0.214951E-02
        6.500     9.000   0.105039E-01  -0.378567E-03   0.222618E-02
        7.000     9.000   0.106625E-01  -0.255153E-03   0.228164E-02
        7.500     9.000   0.107585E-01  -0.128359E-03   0.231514E-02
        8.000     9.000   0.107906E-01   0.319622E-17   0.232633E-02
        8.500     9.000   0.107585E-01   0.128359E-03   0.231514E-02
        9.000     9.000   0.106625E-01   0.255153E-03   0.228164E-02
        9.500     9.000   0.105039E-01   0.378567E-03   0.222618E-02
       10.000     9.000   0.102849E-01   0.496402E-03   0.214951E-02
       10.500     9.000   0.100088E-01   0.606146E-03   0.205314E-02
       11.000     9.000   0.968046E-02   0.705326E-03   0.193964E-02
       11.500     9.000   0.930556E-02   0.792006E-03   0.181277E-02
       12.000     9.000   0.889073E-02   0.864839E-03   0.167736E-02
       13.000     9.000   0.796814E-02   0.972800E-03   0.140328E-02
       14.000     9.000   0.695411E-02   0.105200E-02   0.115785E-02
       15.000     9.000   0.586738E-02   0.111823E-02   0.966942E-03
       16.000     9.000   0.473144E-02   0.114235E-02   0.834522E-03
        0.000    10.000   0.383702E-02  -0.104763E-02   0.925127E-03
        1.000    10.000   0.486091E-02  -0.974625E-03   0.102466E-02
        2.000    10.000   0.576884E-02  -0.842990E-03   0.119384E-02
        3.000    10.000   0.655047E-02  -0.723799E-03   0.141895E-02
        4.000    10.000   0.721650E-02  -0.608944E-03   0.166595E-02
        4.500    10.000   0.750529E-02  -0.545975E-03   0.178685E-02
        5.000    10.000   0.776181E-02  -0.479455E-03   0.190050E-02
        5.500    10.000   0.798391E-02  -0.408250E-03   0.200266E-02
        6.000    10.000   0.816925E-02  -0.332468E-03   0.209006E-02
        6.500    10.000   0.831570E-02  -0.252773E-03   0.216021E-02
        7.000    10.000   0.842151E-02  -0.170118E-03   0.221138E-02
        7.500    10.000   0.848548E-02  -0.855296E-04   0.224248E-02
        8.000    10.000   0.850688E-02  -0.118276E-14   0.225290E-02
        8.500    10.000   0.848548E-02   0.855296E-04   0.224248E-02
        9.000    10.000   0.842151E-02   0.170118E-03   0.221138E-02
        9.500    10.000   0.831570E-02   0.252773E-03   0.216021E-02
       10.000    10.000   0.816925E-02   0.332468E-03   0.209006E-02
       10.500    10.000   0.798391E-02   0.408250E-03   0.200266E-02
       11.000    10.000   0.776181E-02   0.479455E-03   0.190050E-02
       11.500    10.000   0.750529E-02   0.545975E-03   0.178685E-02
       12.000    10.000   0.721650E-02   0.608944E-03   0.166595E-02
       13.000    10.000   0.655047E-02   0.723799E-03   0.141895E-02
       14.000    10.000   0.576884E-02   0.842990E-03   0.119384E-02
       15.000    10.000   0.486091E-02   0.974625E-03   0.102466E-02
       16.000    10.000   0.383702E-02   0.104763E-02   0.925127E-03
      ---------------------------------------------------------------
Wmax   8.000     5.000   0.184718E-01
Wmin  16.000     0.000   0.383702E-02
```

Figure I.62 : Formatted numerical results – 3D project

- Bending moments and twisting moments (**local** coordinates system): for each point X_m and Y_m, Tasplaq gives:
  - bending moment M_x (kN.m/ml);
  - bending moment M_y (kN.m/ml);
  - twisting moment M_{xy} (kN.m/ml).

  At the bottom of this table, the maximum and minimum values of each moment are given, with the coordinates of the points at which they are obtained.

- Shear forces (**local** coordinates system): for each point X_t and Y_t, Tasplaq gives:
  - shear force T_x along x (kN/ml)
  - shear force T_y along y (kN/ml).

  The maximum and minimum values of each shear force are given at the bottom of this table, with the coordinates of the points at which they were obtained.
- Surface support reactions (**local** coordinates system): for each point X_t and Y_t, Tasplaq gives:
  - surface reaction rz (kPa).

  The resultant (kN) of the surface reactions at each of the rectangular zones (groups of elements) defined by the user is given at the bottom of this table.
- Point support reactions: the resultant (kN) of the linear reactions at each of the linear zones (groups of elements) defined by the user is specified.

reaction in distributed supports

Element	Xt	Yt	Reaction	Status
0001	0.500	0.417	0.000	*****
0002	1.500	0.417	0.000	*****
0003	2.500	0.417	0.000	*****
0004	3.333	0.417	0.000	*****
0005	4.000	0.417	0.000	*****
0006	4.667	0.417	0.000	*****
0007	5.333	0.417	0.000	*****
0008	6.000	0.417	0.000	*****
0009	6.667	0.417	0.000	*****
0010	7.333	0.417	0.000	*****
0011	8.000	0.417	0.000	*****
0012	8.667	0.417	0.000	*****
0013	9.333	0.417	0.000	*****
0014	10.000	0.417	0.000	*****
0015	10.667	0.417	0.000	*****
0016	11.333	0.417	0.000	*****
0017	12.000	0.417	0.000	*****
0018	12.667	0.417	0.000	*****
0019	13.500	0.417	0.000	*****
0020	14.500	0.417	0.000	*****
0021	15.500	0.417	0.000	*****
0022	0.500	1.250	0.000	*****
0023	1.500	1.250	0.000	*****
0024	2.500	1.250	0.000	*****
0025	3.333	1.250	0.000	*****
0026	4.000	1.250	0.000	*****
0027	4.667	1.250	0.000	*****
0028	5.333	1.250	0.000	*****
0029	6.000	1.250	0.000	*****
0030	6.667	1.250	0.000	*****

Figure I.63 : Formatted numerical results: Reaction in the surface supports

I.3.11.2.2. Numerical results: results tables

On the whole, these are the same results as those detailed for the formatted results in the previous chapter, but this time presented in table format and without the data reminder part. Some results are however not displayed in these tables (such as rotations and shear forces for example).

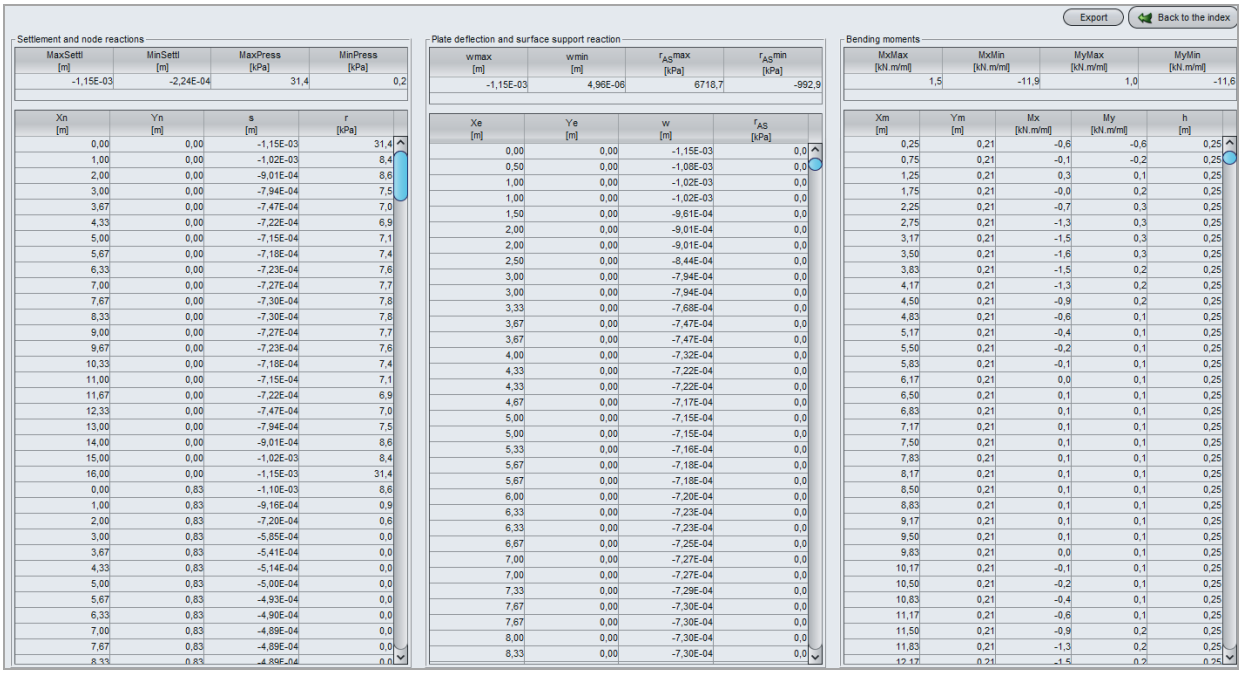


Figure I.64 : Numerical results: Results table

- The first table (left) contains the "settlements and node reactions" for the calculation points as a function of X_n and Y_n (m).
  The first line gives the extreme settlement and node reaction values.

Designation	Unit	Display condition
TassMax: maximum settlement value (negative downwards)	m	Always
TassMin: minimum settlement value (negative downwards)	m	Always
$Press_{max}$: maximum nodes reaction value	kPa	Always
$Press_{min}$: minimum nodes reaction value	kPa	Always
X_n: calculation point X-axis	m	Always
Y_n: calculation point Y-axis	m	Always
s: soil settlement (negative downwards)	m	Always
r: reaction at node	kPa	Always

Table I.34 : Details of numerical results (settlement and node reaction)

- The second table (centre) contains plate deflection and reactions of surface supports as a function of X_e and Y_e (m).
  The first line gives the plate deflection extreme values.

Designation	Unit	Display condition
w_{max}: maximum deflection value (negative downwards)	m	Always
w_{min}: minimum deflection value (negative downwards)	m	Always
r_{Asmax}: maximum surface support reaction value	kPa	Always
r_{Asmin}: minimum surface support reaction value	kPa	Always
X_e: calculation point X-axis	m	Always
Y_e: calculation point Y-axis	m	Always
w: plate deflection (negative downwards)	m	Always
r_{As}: surface supports reaction	kPa	Always

Table I.35 : Details of numerical results (plate deflection)

- The third table (right) contains bending moments at the calculation points as a function of X_m and Y_m (m).
  The first line gives the bending moment extreme values.

Designation	Unit	Display condition
$\mathbf{M_{xmax}}$: maximum bending moment along x	kN.m/ml	Always
$\mathbf{M_{xmin}}$: minimum bending moment along x	kN.m/ml	Always
$\mathbf{M_{ymax}}$: maximum bending moment along y	kN.m/ml	Always
$\mathbf{M_{ymin}}$: minimum bending moment along y	kN.m/ml	Always
$\mathbf{X_m}$: calculation point X-axis	m	Always
$\mathbf{Y_m}$: calculation point Y-axis	m	Always
$\mathbf{M_x}$: moment along x	kN.m/ml	Always
$\mathbf{M_y}$: moment along y	kN.m/ml	Always
h: plate thickness at calculation point	m	Always

Table I.36 : Details of numerical results (bending moment)

I.3.11.2.3. Graphic results: graphic cross-sections

The graphic cross-sections present cross-sections in the plate and display a choice of one of the following 6 quantities: plate deflection, soil settlements (multilayer medium), moments M_x and M_y, soil reaction, or surface supports reaction.

- From the drop-down lists, select the quantity to be presented, the direction of the cross-section (along X or along Y) and its position. The plot of the curve is automatically updated. A final drop-down list can be used to select a 2nd cross-section (same quantity, same cross-section direction, but different position), which allows an easy comparison to be made between the cross-sections obtained at 2 different positions.
- To display the maximum and minimum values obtained for the 6 quantities for the entire model (independently of the cross-sections selected), click the Show button in the "Extreme global values" frame (the Hide (Hide) button can then be used to hide them if necessary).
- To display details of each calculation point for the 2 cross-sections selected, click the Show (show) button in the "Cross sections details" frame (the Hide button can then be used to hide them if necessary).

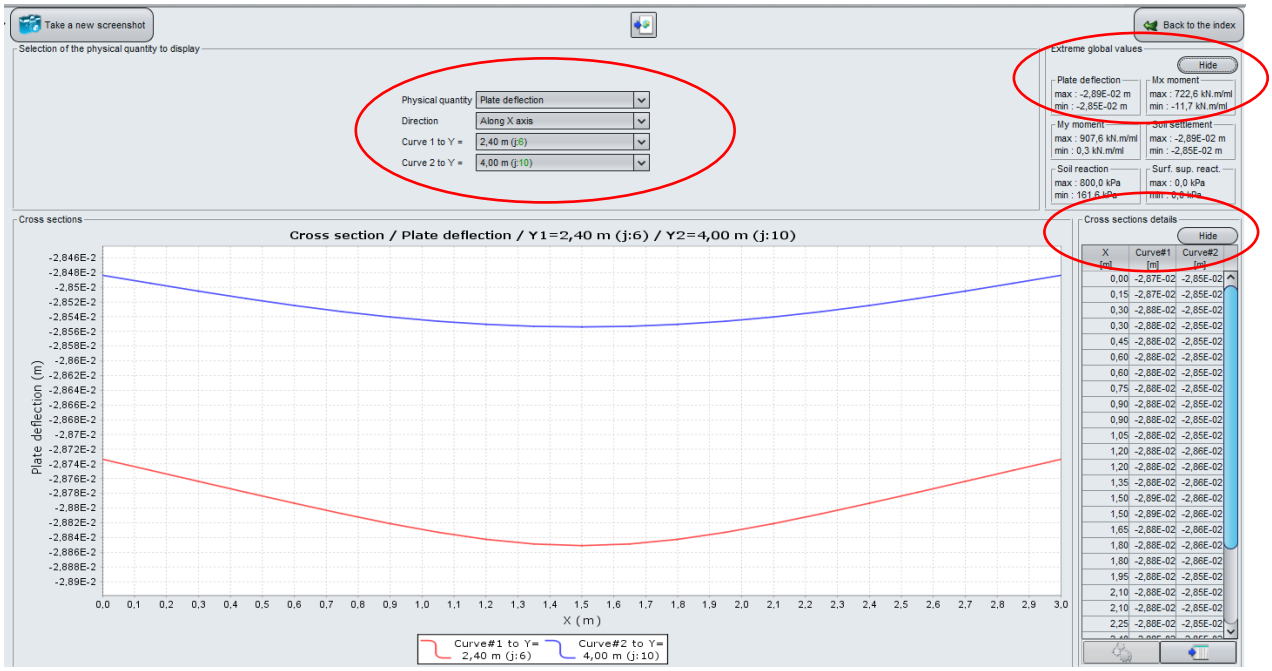


Figure I.65 : Cross-section – 3D project – Plate deflection – Example of cross-section along x

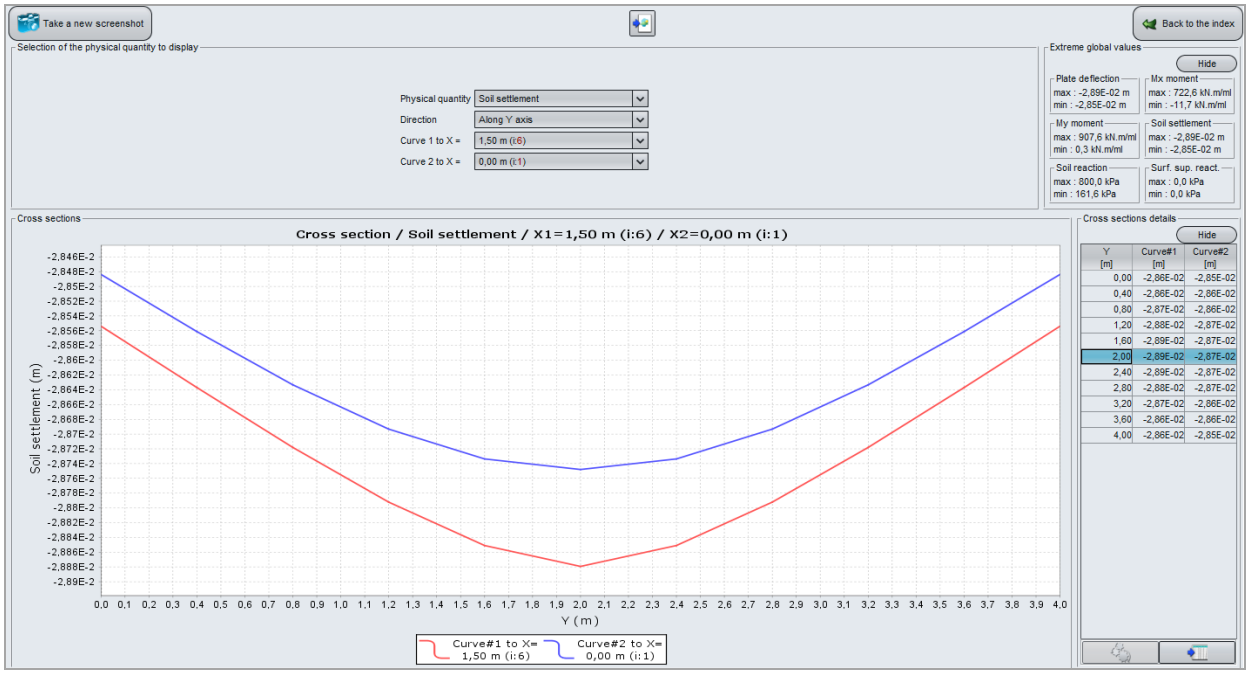


Figure I.66 : Cross-section – 3D project – Soil settlement – Example of cross-section along y

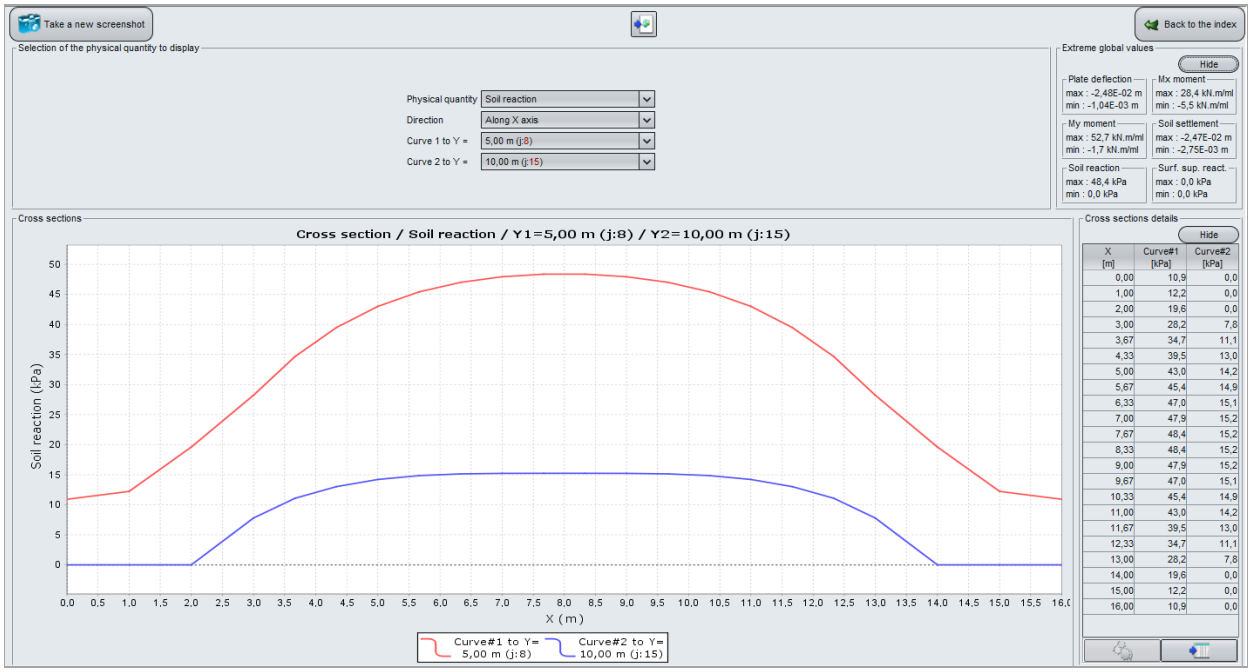


Figure I.67 : Cross-section – 3D project – Soil reaction – Example of cross-section along x

I.3.11.2.4. Graphic result: shading

This option can be used to display different types of results in the form of shadings. The available quantities are the same as for cross-sections: plate deflection, soil settlements, moments M_x and M_y, soil reaction, or surface supports reaction.

The advantage of this window is that it helps the user see the distribution of a given quantity, which in particular helps choose the most relevant cross-sections.

The caption (to the right of the shading) details the various value ranges corresponding to each colour.

Click the Modify the shading... button to modify the bounds of shading:

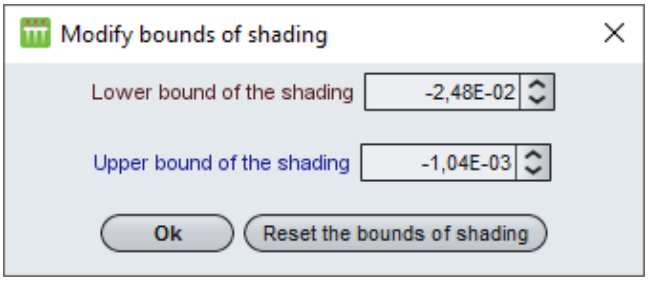


Figure I.68 : Shading: Modify bounds of shading

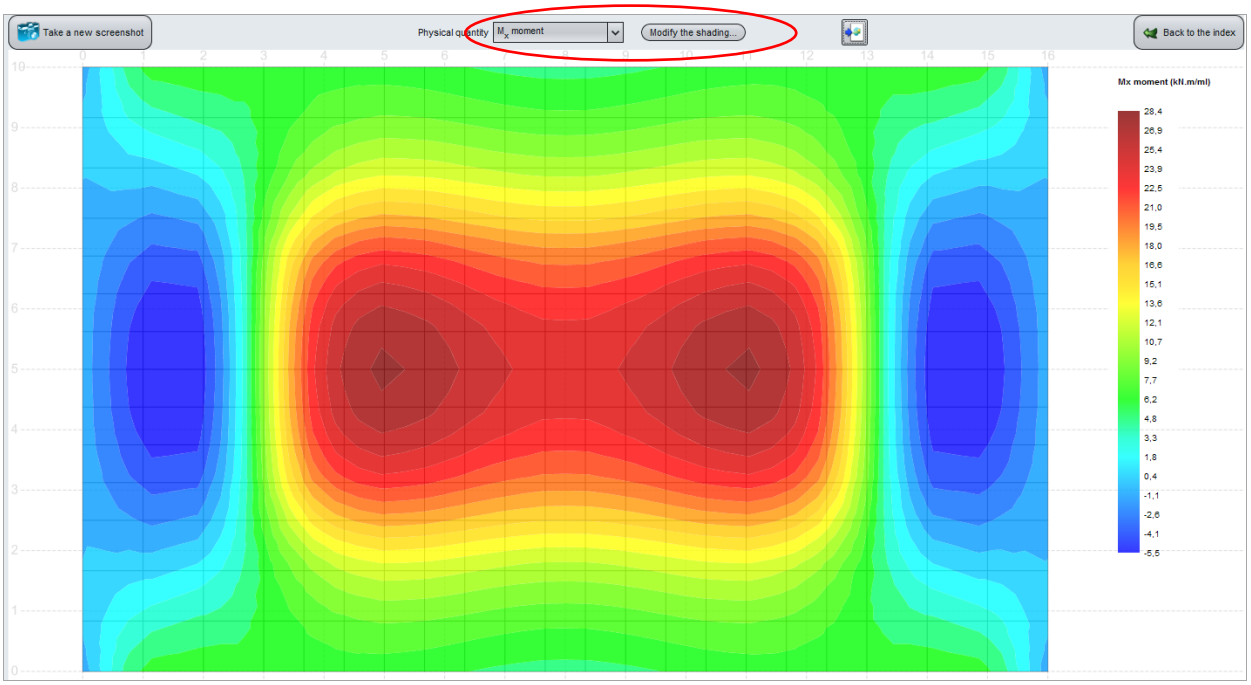


Figure I.69 : Shading – 3D project – Moment M_x

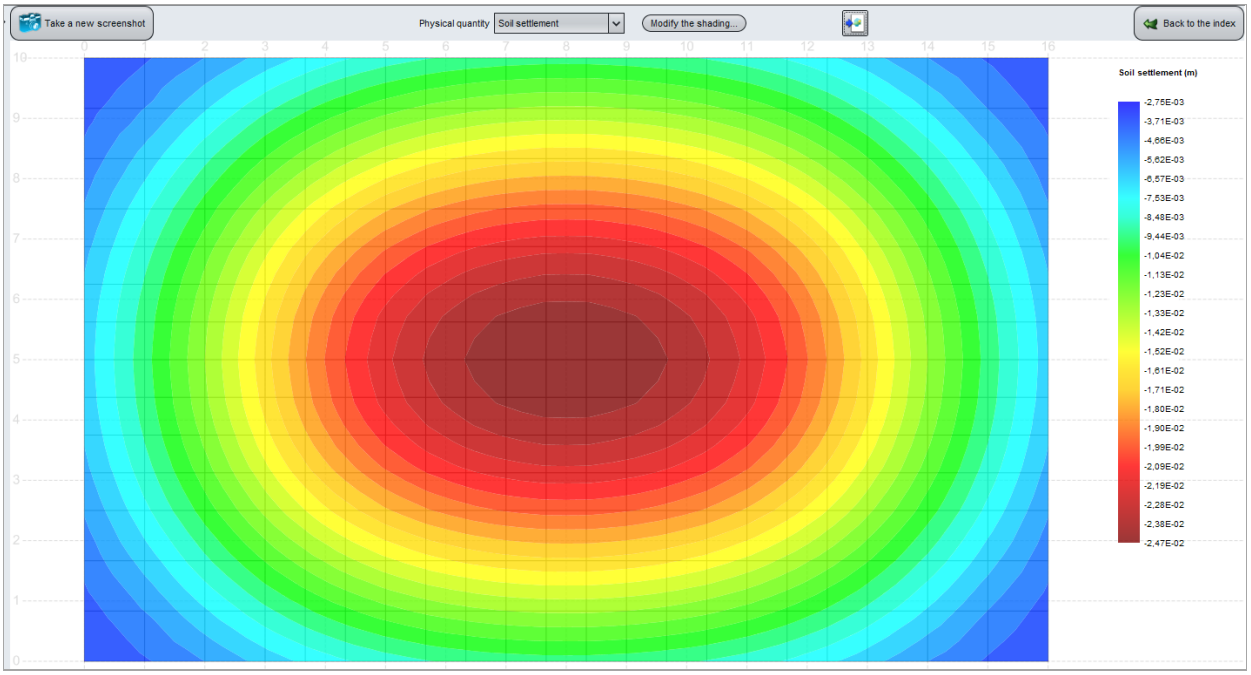


Figure I.70 : Shading – 3D project – Soil settlement

I.3.11.2.5. Graphic results: 3D charts

This option can be used to display different types of results in the form of a three-dimensional surfaces. The available quantities are the same as for cross-sections and shading: plate deflection, soil settlements, moments M_x and M_y, soil reaction, or surface supports reaction.

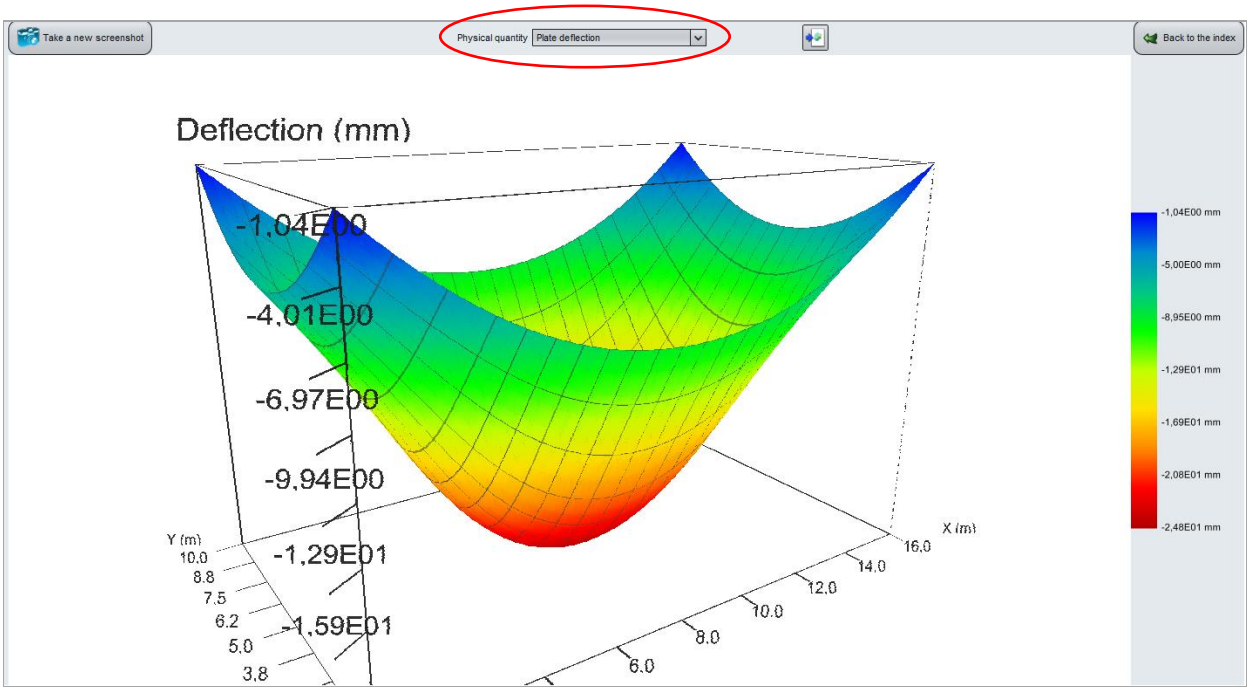


Figure I.71 : 3D Chart – Plate deflection

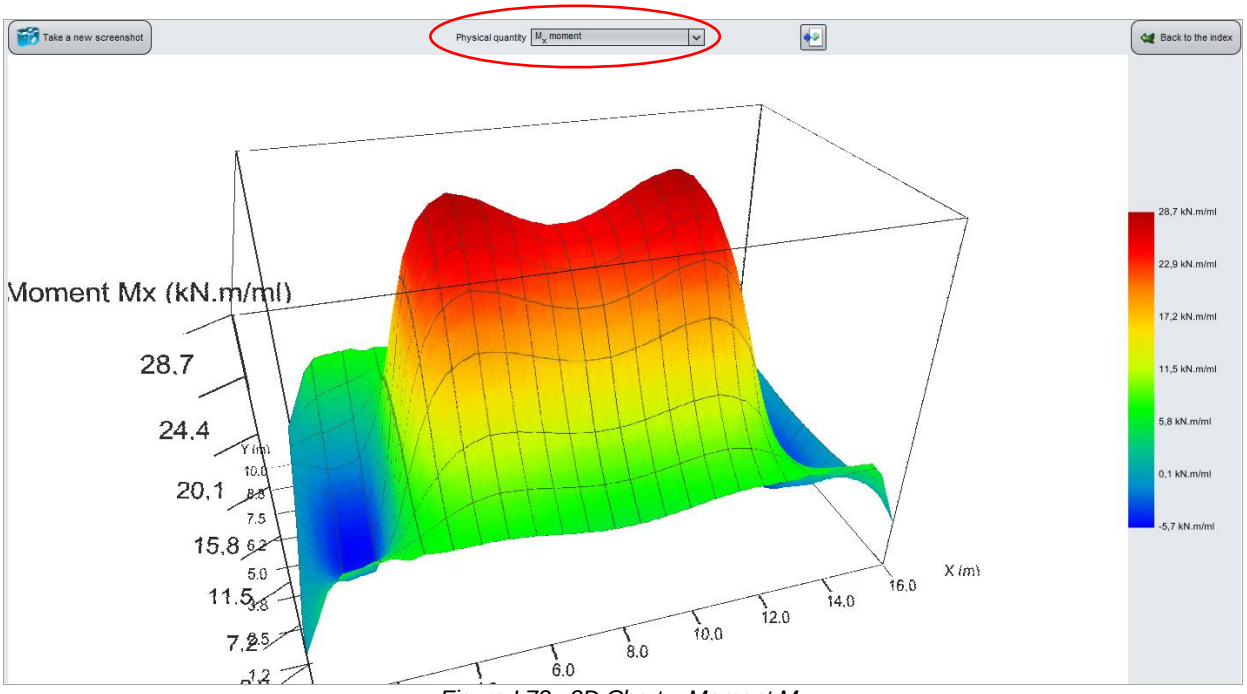


Figure I.72 : 3D Chart – Moment M_x

I.3.11.2.6. Dishomogenisation

Click the button to start the "Dishomogenisation" wizard.

If the user used the "Combined section" wizard in the plate mechanical properties definition step (see chapter I.3.4.3), the "Dishomogenisation" wizard can be used to retrieve the dishomogenised forces.

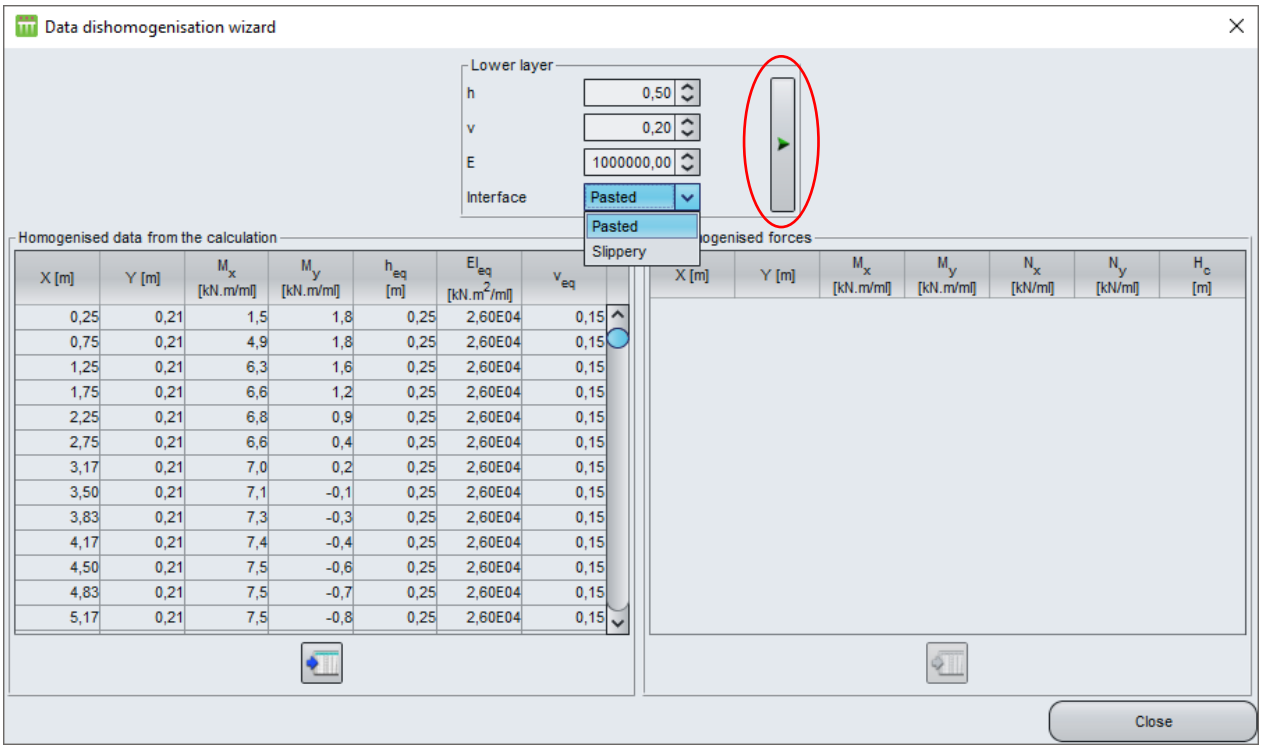


Figure I.73 : Data dishomogenisation wizard

The displayed window comprises three different frames:

"Lower layer" frame:

The data proposed here correspond to those input by the user in the "Combined section". They can be modified.

Designation	Unit	Default value	Display condition	Mandatory value	Local checks
h: thickness	m	Taken from the "Combined section" wizard "	Always	Yes	> 0
ν: Poisson's ratio	-		Always	Yes	$0 \leq \nu < 0,5$
E: Young's modulus	kPa		Always	Yes	> 0
Contact	-		Always	Yes	Possible choices: • Pasted • Slippery

Table I.37 : Dishomogenisation wizard: "Lower layer" frame

"Homogenised data from the calculation" frame:

Here, Tasplaq gives the calculation results. They cannot be modified:

- **X**: X-axis of point (m);
- **Y**: Y-axis of point (m);
- **M_x**: moment around axis -Oy (kN.m/ml);

- **M_y**: moment around axis Ox (kN.m/ml);
- **h_{eq}**: equivalent height of homogenised plate (m);
- **EI_{eq}**: equivalent product of inertia of homogenised plate (kN.m²/ml);
- **ν_{eq}**: equivalent Poisson's ratio of homogenised plate.

Click the "Apply parameters" button to calculate the "Dishomogenised forces"":

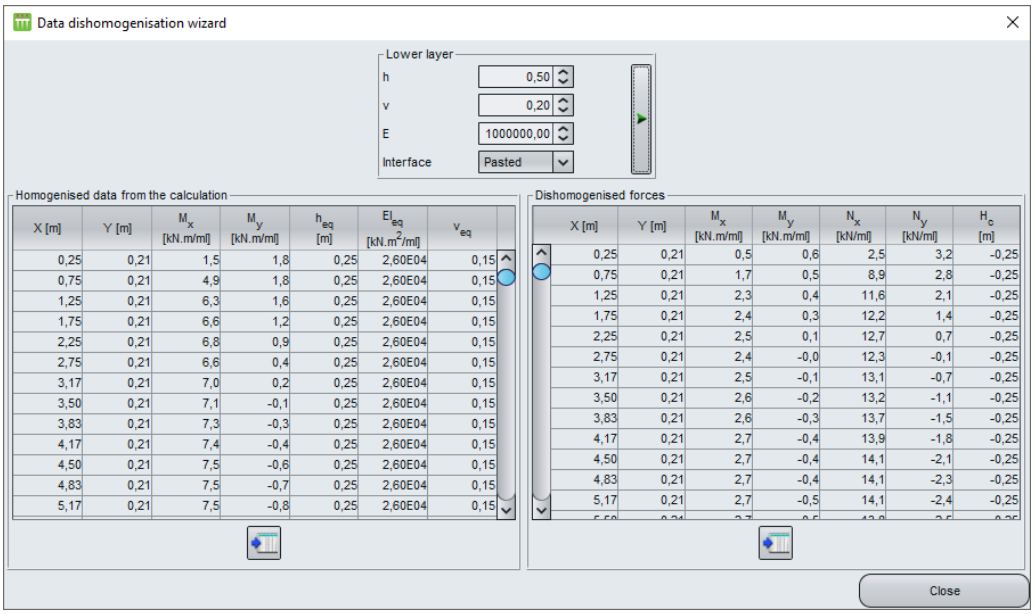


Figure I.74 : Data dishomogenisation wizard: results

"Dishomogenised forces" frame:

Tasplaq displays the calculated dishomogenised values, which cannot be modified: bending moments and normal forces in the upper layer.

- **X**: X-axis of point (m);
- **Y**: Y-axis of point (m);
- **M_x**: moment around axis –(Oy) (kN.m/ml);
- **M_y**: moment around axis (Ox) (kN.m/ml);
- **N_x**: normal force along x (kN/ml);
- **N_y**: normal force along y (kN/ml);
- **H_b**: thickness of upper layer (m).

These data can be exported to MS-Excel© for further analysis by clicking the button. Use of this wizard for export is detailed in part C of the manual.

Click the button to close the window.

I.3.11.3. Available results for 2D projects

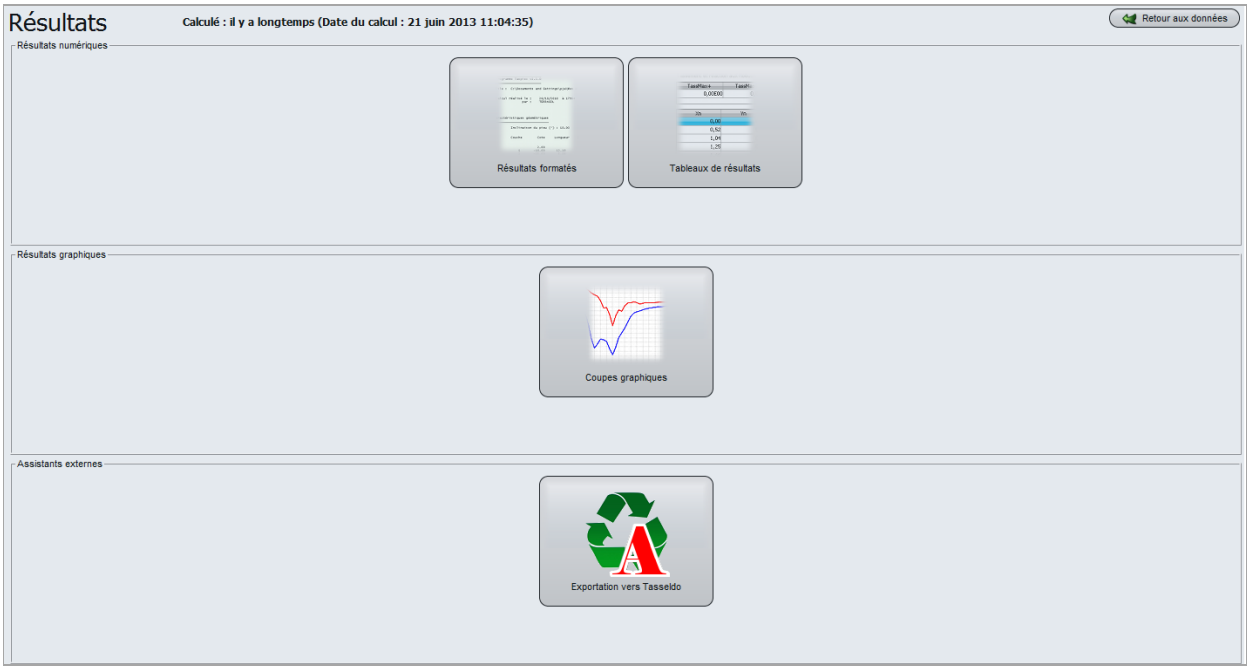
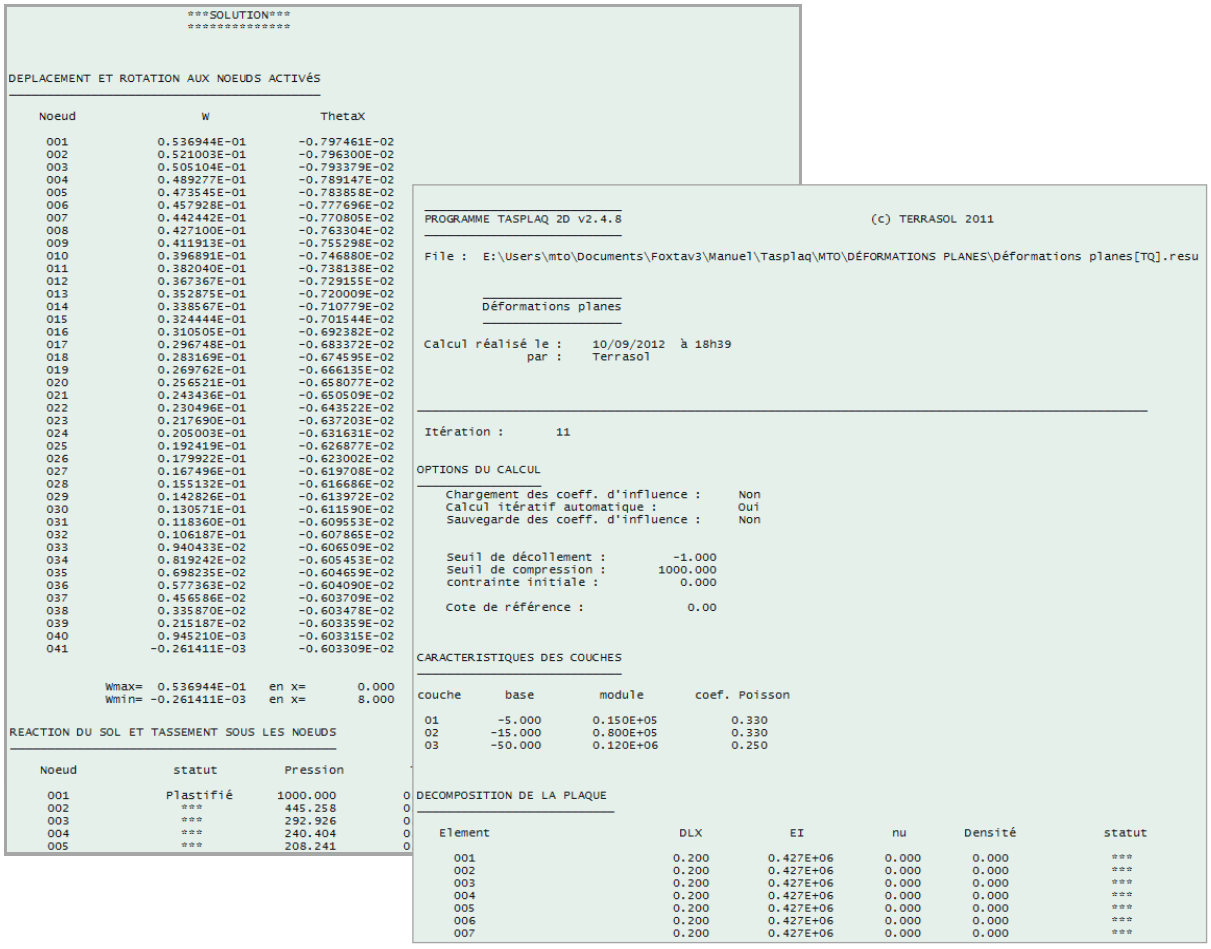


Figure I.75 : Results window – 2D project

I.3.11.3.1. Numerical results: formatted results



```
                    ***SOLUTION***
                    **************


DEPLACEMENT ET ROTATION AUX NOEUDS ACTIVÉS
__________________________________________

    Noeud                W                ThetaX

      001          0.536944E-01       -0.797461E-02
      002          0.521003E-01       -0.796300E-02
      003          0.505104E-01       -0.793379E-02
      004          0.489277E-01       -0.789147E-02
      005          0.473545E-01       -0.783858E-02
      006          0.457928E-01       -0.777696E-02
      007          0.442442E-01       -0.770805E-02
      008          0.427100E-01       -0.763304E-02
      009          0.411913E-01       -0.755298E-02
      010          0.396891E-01       -0.746880E-02
      011          0.382040E-01       -0.738138E-02
      012          0.367367E-01       -0.729155E-02
      013          0.352875E-01       -0.720009E-02
      014          0.338567E-01       -0.710779E-02
      015          0.324444E-01       -0.701544E-02
      016          0.310505E-01       -0.692382E-02
      017          0.296748E-01       -0.683372E-02
      018          0.283169E-01       -0.674595E-02
      019          0.269762E-01       -0.666135E-02
      020          0.256521E-01       -0.658077E-02
      021          0.243436E-01       -0.650509E-02
      022          0.230496E-01       -0.643522E-02
      023          0.217690E-01       -0.637203E-02
      024          0.205003E-01       -0.631631E-02
      025          0.192419E-01       -0.626877E-02
      026          0.179922E-01       -0.623002E-02
      027          0.167496E-01       -0.619708E-02
      028          0.155132E-01       -0.616686E-02
      029          0.142826E-01       -0.613972E-02
      030          0.130571E-01       -0.611590E-02
      031          0.118360E-01       -0.609553E-02
      032          0.106187E-01       -0.607865E-02
      033          0.940433E-02       -0.606509E-02
      034          0.819242E-02       -0.605453E-02
      035          0.698235E-02       -0.604659E-02
      036          0.577363E-02       -0.604090E-02
      037          0.456586E-02       -0.603709E-02
      038          0.335870E-02       -0.603478E-02
      039          0.215187E-02       -0.603359E-02
      040          0.945210E-03       -0.603315E-02
      041         -0.261411E-03       -0.603309E-02


            Wmax=  0.536944E-01   en x=       0.000
            Wmin= -0.261411E-03   en x=       8.000

REACTION DU SOL ET TASSEMENT SOUS LES NOEUDS
____________________________________________

    Noeud            statut          Pression

      001          Plastifié        1000.000          0
      002              ***           445.258          0
      003              ***           292.926          0
      004              ***           240.404          0
      005              ***           208.241          0
```

```
_____________________
PROGRAMME TASPLAQ 2D v2.4.8                                  (c) TERRASOL 2011
_____________________

File :  E:\Users\mto\Documents\Foxtav3\Manuel\Tasplaq\MTO\DÉFORMATIONS PLANES\Déformations planes[TQ].resu


        _____________________
        Déformations planes
        _____________________

Calcul réalisé le :    10/09/2012  à 18h39
             par :    Terrasol


_____________________________________________________________________________

 Itération :       11

OPTIONS DU CALCUL
_________________
    Chargement des coeff. d'influence :     Non
    Calcul itératif automatique :           Oui
    Sauvegarde des coeff. d'influence :     Non


    Seuil de décollement :          -1.000
    Seuil de compression :        1000.000
    contrainte initiale :            0.000

    Cote de référence :              0.00


CARACTERISTIQUES DES COUCHES
____________________________

couche      base          module       coef. Poisson

 01         -5.000       0.150E+05          0.330
 02        -15.000       0.800E+05          0.330
 03        -50.000       0.120E+06          0.250


DECOMPOSITION DE LA PLAQUE
__________________________

   Element                        DLX          EI          nu        Densité          statut

      001                       0.200       0.427E+06     0.000       0.000            ***
      002                       0.200       0.427E+06     0.000       0.000            ***
      003                       0.200       0.427E+06     0.000       0.000            ***
      004                       0.200       0.427E+06     0.000       0.000            ***
      005                       0.200       0.427E+06     0.000       0.000            ***
      006                       0.200       0.427E+06     0.000       0.000            ***
      007                       0.200       0.427E+06     0.000       0.000            ***
```

Figure I.76 : Formatted numerical results – Example for a 2D project in plane deformations

Similarly to the case of 3D projects, the formatted numerical results contain:

- A data reminder: general parameters, soil data, plate geometry and distribution, plate characteristics, calculation points, applied loads (and their distribution).
- The results:
  - Displacements and rotations at the activated nodes: for each node (defined by its number), Tasplaq gives the displacement:
    - displacement w (m), positive downwards;
    - rotation θ_x (2D plane deformations) or θ_R (2D axisymmetric) (in radians).

    The maximum and minimum values w_{max} and w_{min} (m), and the X-axis (2D plane deformations) or the distance to the axis (2D axisymmetric) for which they are obtained are given under this table.
  - Soil reaction and settlement under the nodes: for each node (defined by its number), Tasplaq gives:
    - its status (plastified or unstuck);
    - pressure (kPa): soil reaction under the plate;
    - settlement (m), positive downwards;
    - deflection (m), positive downwards.

    The maximum and minimum settlement values (m) and the X-axis (2D plane deformations) or the distance to the axis (2D axisymmetric) for which they are obtained are given under this table.
  - For 2D models in plane deformation:

    For each calculation point (3 points per element), Tasplaq gives:
    - the element number and its X-axis X_m (m);
    - the bending moment M_x (kN.m/ml);
    - the shear force T_x (kN/ml).
  - For axisymmetric 2D models:
    - ✓ Bending moments: for each calculation point (2 points per element), Tasplaq gives:
      - the element number and the distance X_m from the point to the axis (m);
      - the radial bending moment M_r (kN.m/ml);
      - the tangential bending moment M_t (kN.m/ml);
    - ✓ Shear forces: for each calculation point (3 points per element), Tasplaq gives:
      - the element number and the distance X_t from the point to the axis (m);
      - the shear force T_r (kN/ml).

I.3.11.3.2. Numerical results: results table

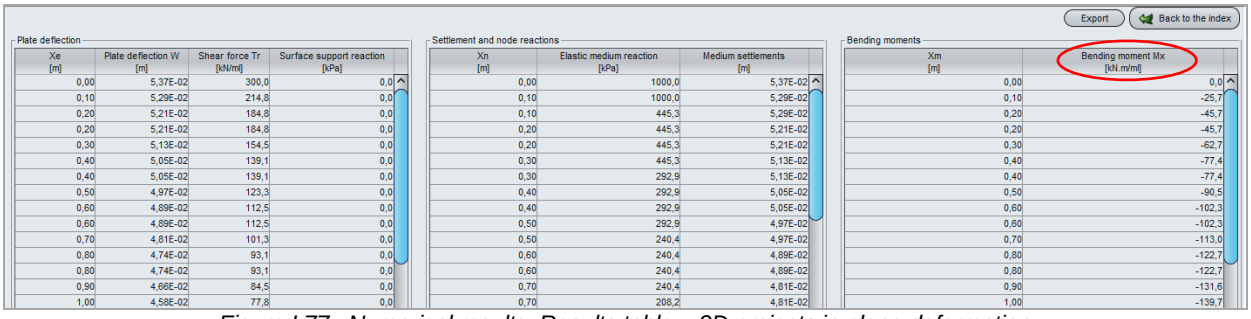


Figure I.77 : Numerical results: Results table – 2D projects in plane deformation

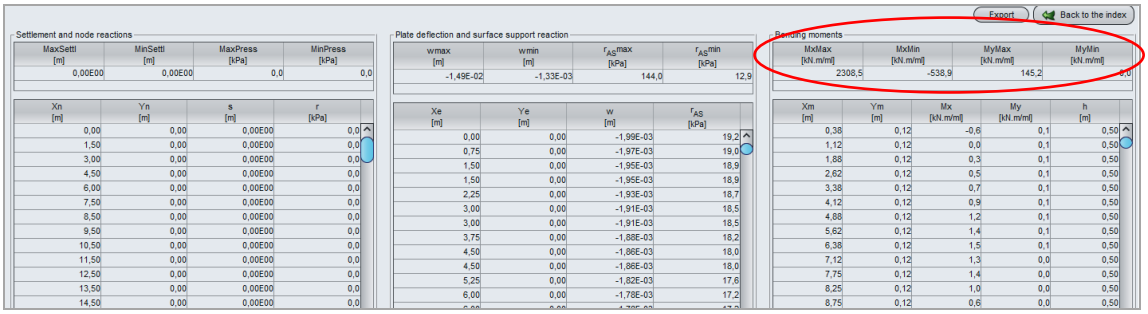


Figure I.78 : Numerical results: Results table – Axisymmetric 2D projects

As in 3D, the results are on the whole the same as those detailed for formatted results, but this time presented in table format and without the data reminder part. Some results are not however shown in these tables (such as rotations for example).

- The first table (left) contains plate deflection, shear force and surface supports reaction at the calculation points as a function of X_e (m).
  The first line gives the extreme settlement and node reaction values.

Designation	Unit	Display condition
X_e: Calculation point X-axis	m	Always
w: plate deflection (positive downwards)	m	Always
Shear force	kN/ml	Always
Surf supp react: reaction of surface supports	kPa	Always

Table I.38 : Details of numerical results for the plate (2D projects)

- The second table (centre) contains settlement and soil reaction at the calculation points as a function of X_n (m).
  The first line gives the plate deflection extreme values.

Designation	Unit	Display condition
X_n: Calculation point X-axis	m	Always
Reaction of elastic medium	kPa	Always
Medium settlement	m	Always

Table I.39 : Details of numerical results - Settlement and soil reaction (2D projects)

- The third table (right) contains the bending moments at the calculation points as a function of X_m (m).
  The first line gives the plate deflection extreme values.

Designation	Unit	Display condition
X_m: Calculation point X-axis	m	Always
M_x: bending moment	kN.m/ml	Always

Table I.40 : Details of numerical results - Bending moments (2D projects in plane deformation)

Designation	Unit	Display condition
X_m: Calculation point X-axis	m	Always
M_r: radial moment	kN.m/ml	Always
M_t: tangential moment	kN.m/ml	Always

Table I.41 : Details of numerical results – Bending moments (axisymmetric 2D projects)

I.3.11.3.3. Graphic results: cross-sections

As in 3D, the graphic cross-sections are used to present cross-sections in the plate, with display of a choice of one of the following quantities:

- plate deflection and soil settlements (shown on the same graph);
- radial and tangential moments (shown on the same graph) in the case of 2D axisymmetric projects, and bending moment M_x in the case of 2D projects in plane deformation;
- soil reaction and surface supports reaction (shown on the same graphic);
- shear force.

The use of this screen is comparable to that described in 3D (see chapter I.3.11.2.3).

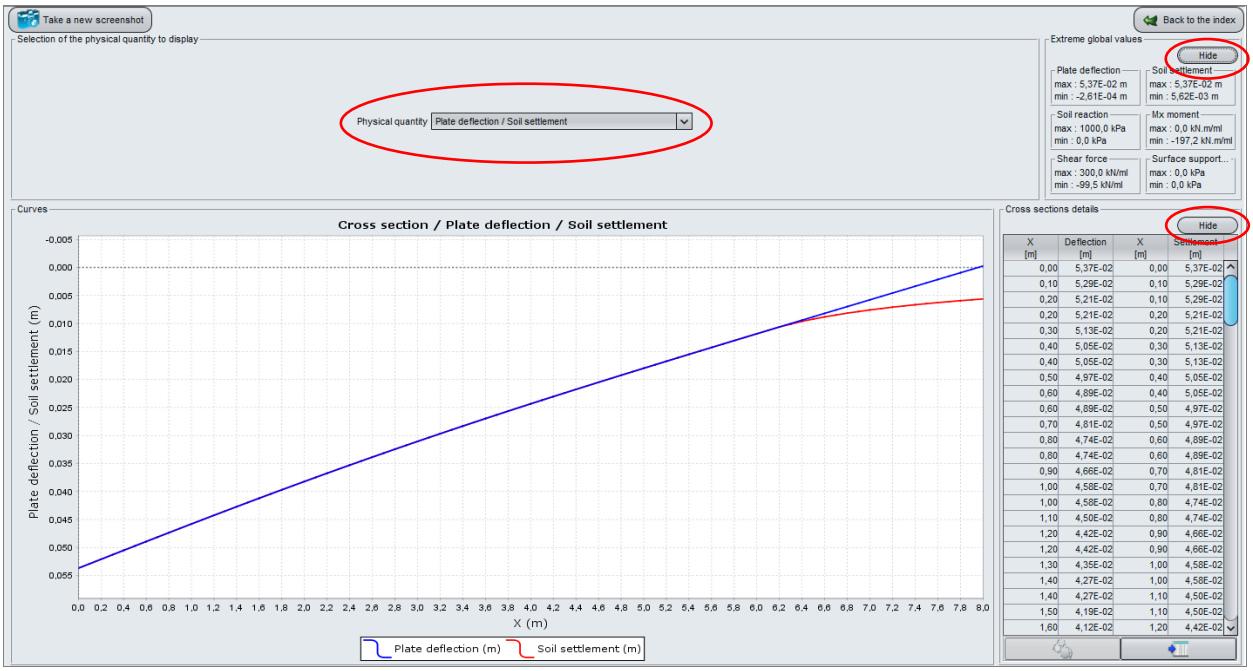


Figure I.79 : Cross-section: example of 2D axisymmetric project – Plate deflection/soil settlement

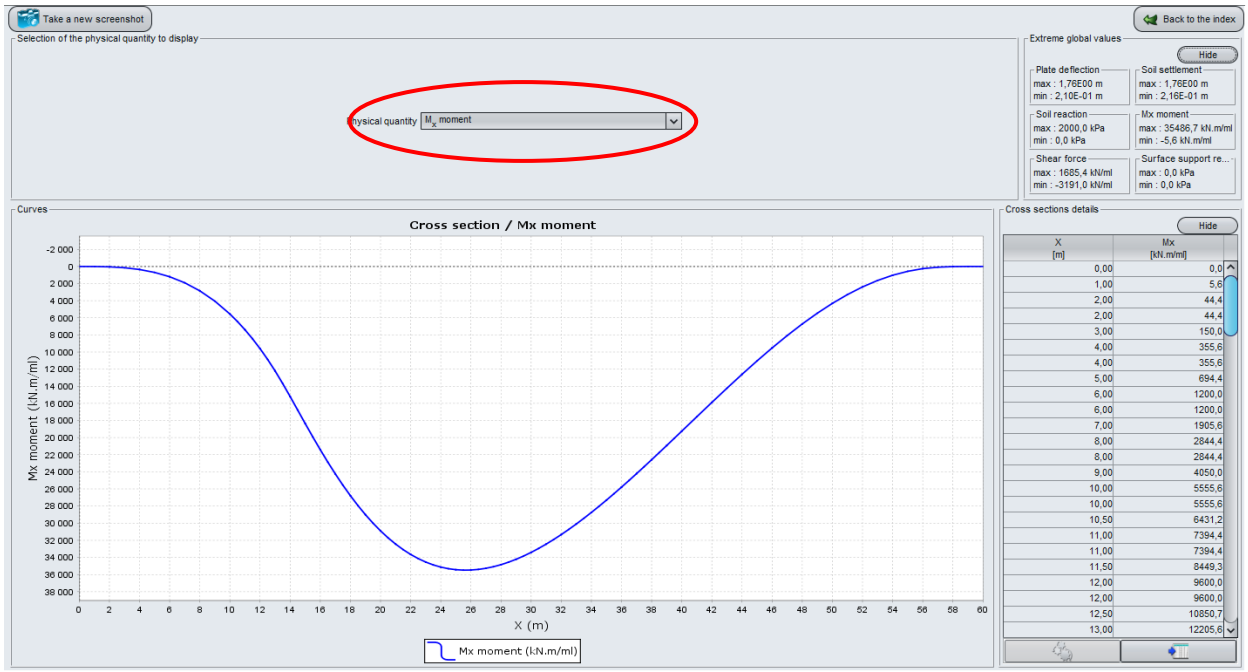


Figure I.80 : Cross-section: example of 2D project in plane deformation - M_x Moment

I.3.11.4. Export to Tasseldo external wizard

The "Export to Tasseldo" wizard, accessed by means of the button, can be used to export all the reactions calculated under the plate, for a calculation with Tasseldo. This wizard is available for 2D and 3D projects.

Using Tasseldo supplements the lessons that can be learned from a Tasplaq model:

- Settlements at depth and at all points on the surface (in particular outside the Tasplaq model);
- Stress distribution at depth.

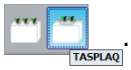
The Tasseldo file generated comprises the following elements:

- definition of soil layers (elastic properties);
- loads exerted on the soil:
  - external loads on the soil;
  - net pressure exerted by the plate at each node (soil reaction);
- calculation points (nodes).

The wizard generates this data file and immediately opens the Tasseldo module.

The Tasplaq and Tasseldo modules are then both activated in the project (the buttons of both modules appear) and the active module is highlighted in blue:

- either at top-right of the window:

or 

. Running the mouse over either button will call up the module name in a tool tip;
- or in the strip to the right of the window:

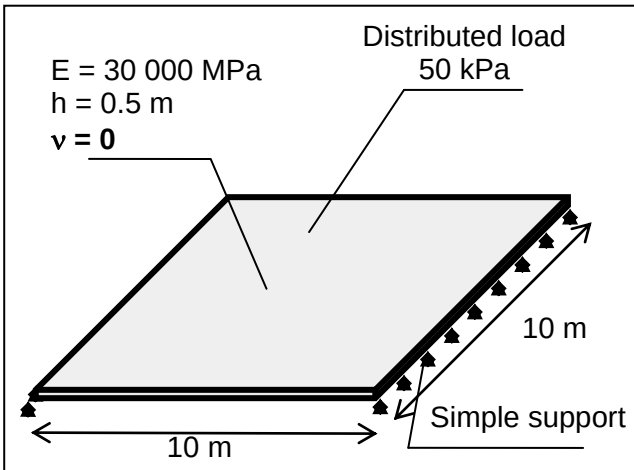
I.4. CALCULATION EXAMPLES

I.4.1. Example 1: Slab on rigid supports

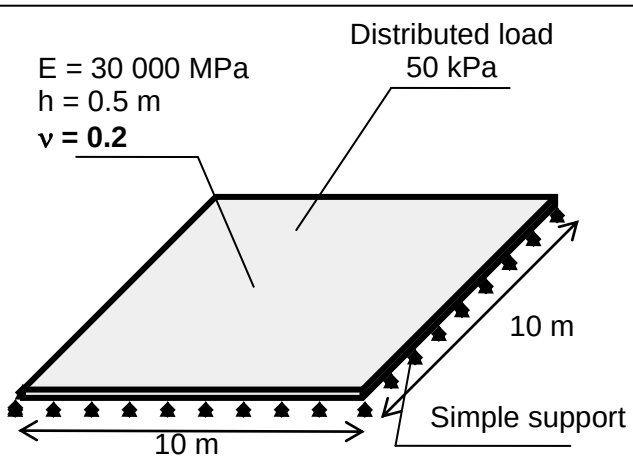
This example deals with the study of a homogeneous square (10 m on each side) slab bearing a distributed load of 50 kPa. We will study two different cases:

- Case of a plate supported by its 2 opposite edges (see the drawing below);
- Case of a plate supported by its 4 edges (see the drawing below).

The input data of the study are detailed in the below drawing:



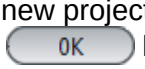
Case (a) – Simple support under 2 opposite edges



Case (b) – Simple support under 4 edges

I.4.1.1. Step 1: Slab supported by its two opposite edges

- To create a new project, click on the

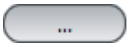
radio-button
- Click on the 

button.

If Foxta is already open, click on "File", then "New project ".

I.4.1.1.1. New project wizard

"File" frame:

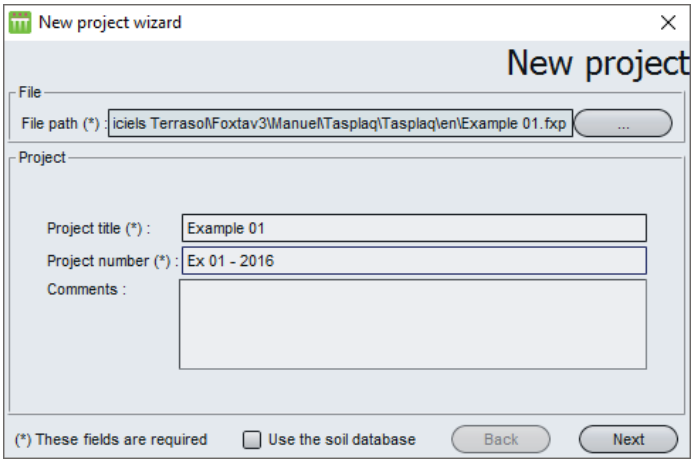
- Click on the 

button to choose the work directory;
- Name and save the file.

"Project" frame:

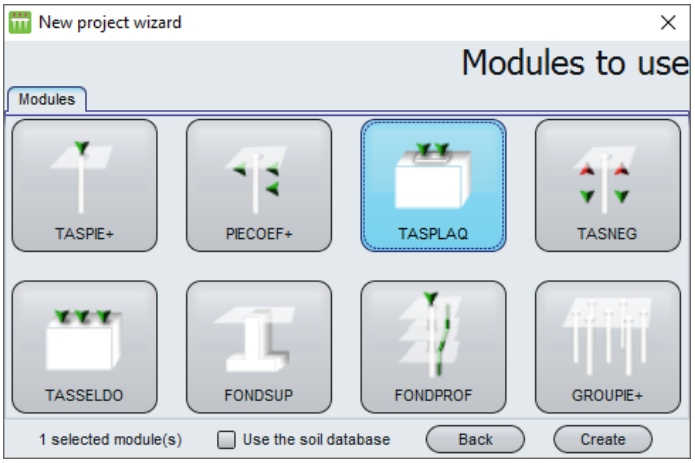
- Define a title for the project;
- Define a project number;
- Complete comments if necessary;
- Leave the "Use the soil database" box unchecked (for this example we will not use the database), and click on the

button.



New project wizard: Module selection

- In the "Modules to use" window, select the TASPLAQ module and click on the Create button·



The input data window of TASPLAQ module is now displayed.
All different data tabs should be completed in the right zone of the screen.

I.4.1.1.2. "Parameters" tab

"General mode" frame:

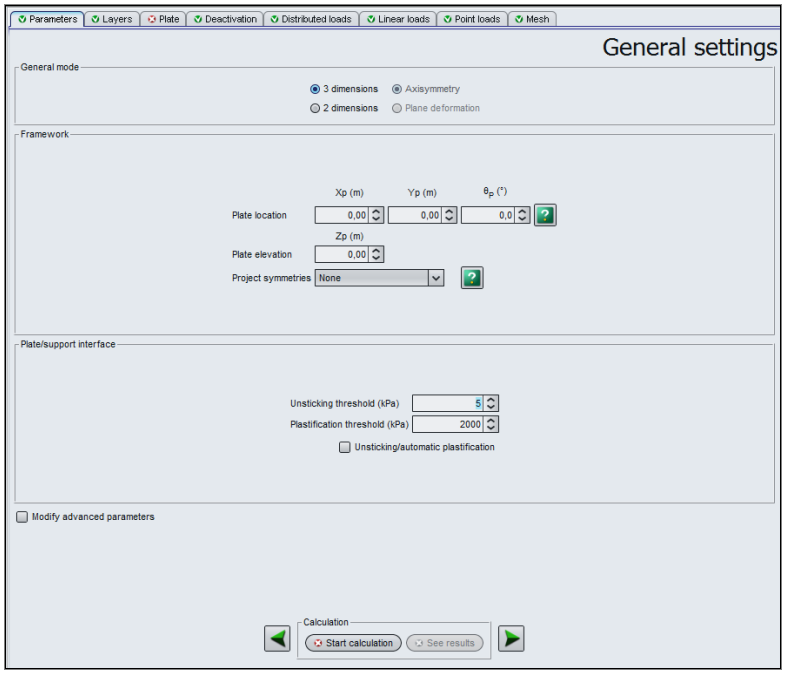
The study model is a three dimensional square plate, so we choose "3 dimensions".

"Framework" frame:

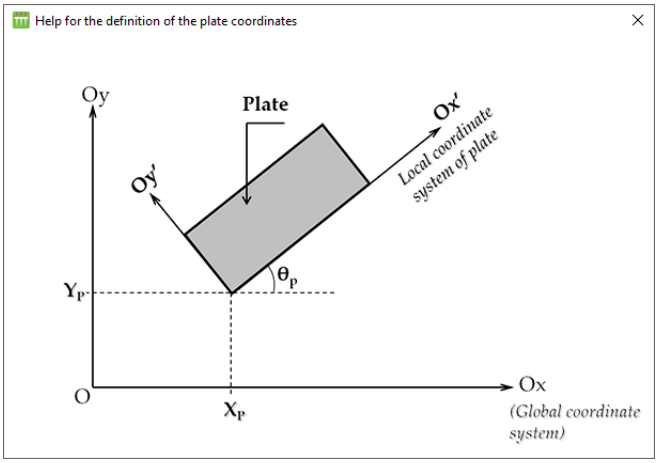
In this example, the local reference system of the plate representing the slab is chosen to be identical to the global one. The (0.0) point represents the "bottom-left" corner of the plate on a plane view.

- Plate location:

X_p (m)	Y_p (m)	θ_p (°)
0.0	0.0	0

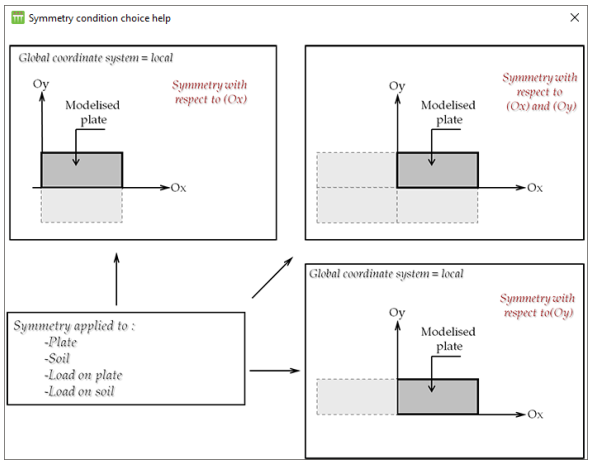


The following help figure is available for the definition of the coordinates of the plate:



- Plate elevation: Zp = 0.0 m (reference elevation = elevation of the underside of the slab).
- Project symmetries: None (here we will deal with the whole model)

  Note: it is possible in Tasplaq to define symmetry axes in order to simplify some models. The following help figure illustrates different possible cases.



"Plate/support interface" frame:

This frame allows the activation of the automatic management of the unsticking and the plastification criteria by using the criteria defined by the user.

- Default values are kept as they are in this example.

Unsticking threshold	5 kPa
Plastification threshold	2000 kPa

- Automatic unsticking/plastification: Unchecked
- Modify advanced parameters: Unchecked

To shift to the next tab, click either on the name of the tab "Layers", or on the button.

I.4.1.1.3. "Layers" tab

This tab includes two different frames:

"Soil layers definition" frame:

- No layers have to be defined. This is a fully structural calculation (plate on supports).
- Initial surface stress: 0 kPa.

"External loads on soil" frame:

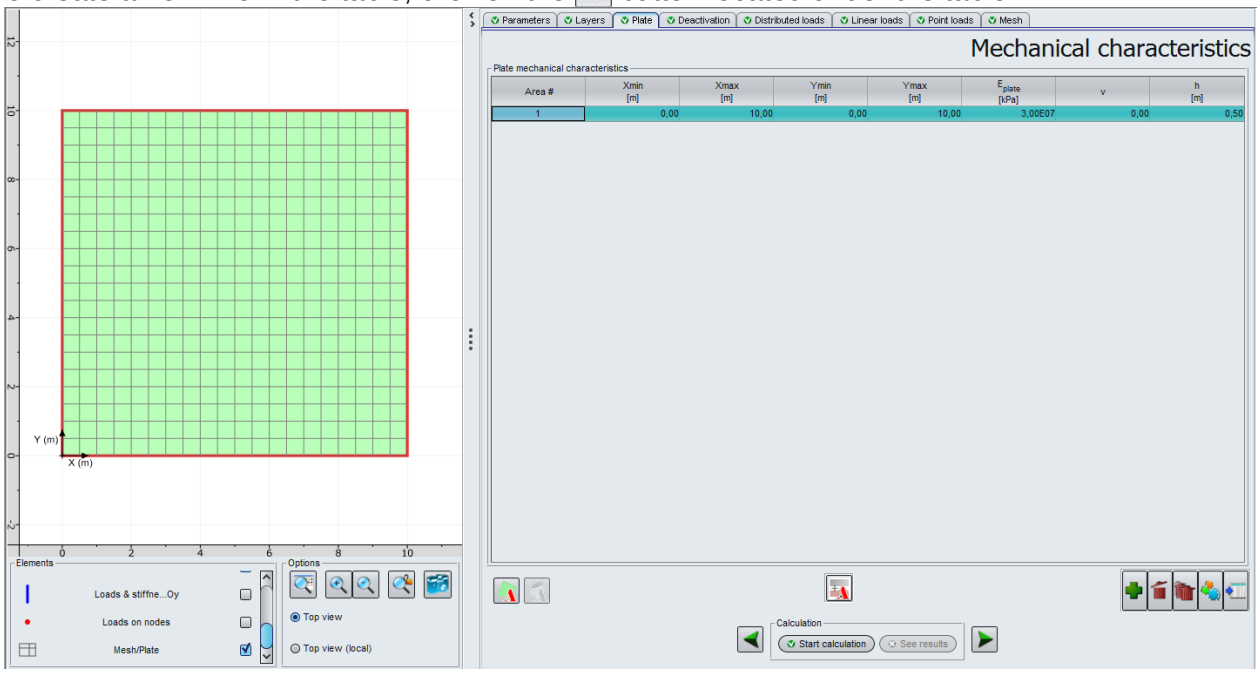
In this example, there is no load applied directly to the soil outside of the plate.

- Activate external loads: Unchecked (by the way, this option is not available in this example because no soil layers are defined here).

I.4.1.1.4. "Plate" tab

In this example, the plate is homogeneous and continuous. To define its characteristics, it is convenient to define one single zone with Xmin = 0 m, Xmax = 10 m, Ymin = 0 m and Ymax = 10 m.

To create a new line in the table, click on the button located under the table.



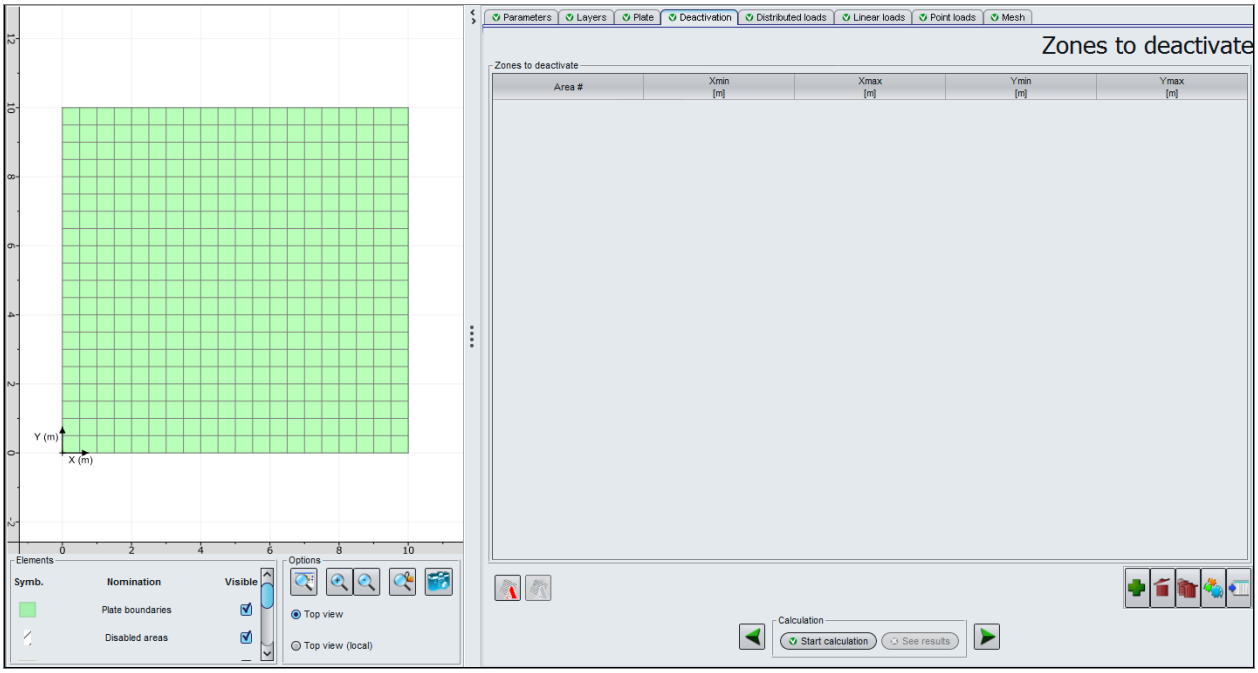
The input data are given in the table below:

Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	10.00	0.00	10.00	3.00E+07	0.00	0.50

The choice of the Poisson ratio equal to zero is made considering the assumption of a plate consisting of reinforced concrete (0.2 value is to be considered when the concrete is not reinforced or fibered: this will be the case in the second step of this example).

I.4.1.1.5. "Deactivation" tab

In this example, there is no element to deactivate, the plate is full and rectangular. Leave the "deactivation" frame empty.



I.4.1.1.6. "Distributed loads" tab

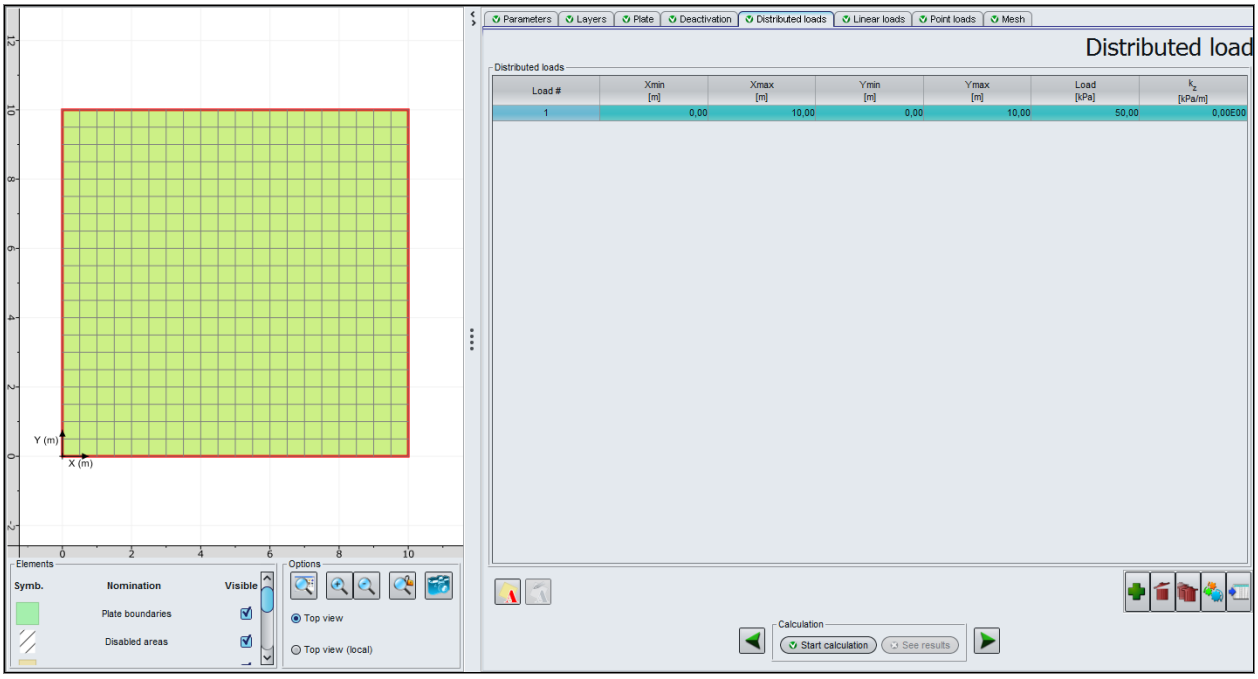
The plate is subjected to an evenly distributed load of 50 kPa. It is convenient to define a load distributed on the whole surface of the plate (from Xmin = 0 to Xmax = 10 m along Ox and from Ymin = 0 to Ymax = 10 m along Oy).

To create a new line in the table, click on the button located under the table.

The "kz" column (surface stiffness) corresponds to the case where a continuous and elastic stiffness is located under the plate (surface springs), which allows the definition of an elastic support with a variable stiffness by zone.

Input the loading characteristics:

Load n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	Load (kPa)	k_z (kPa/m)
1	0.00	10.00	0.00	10.00	50	0

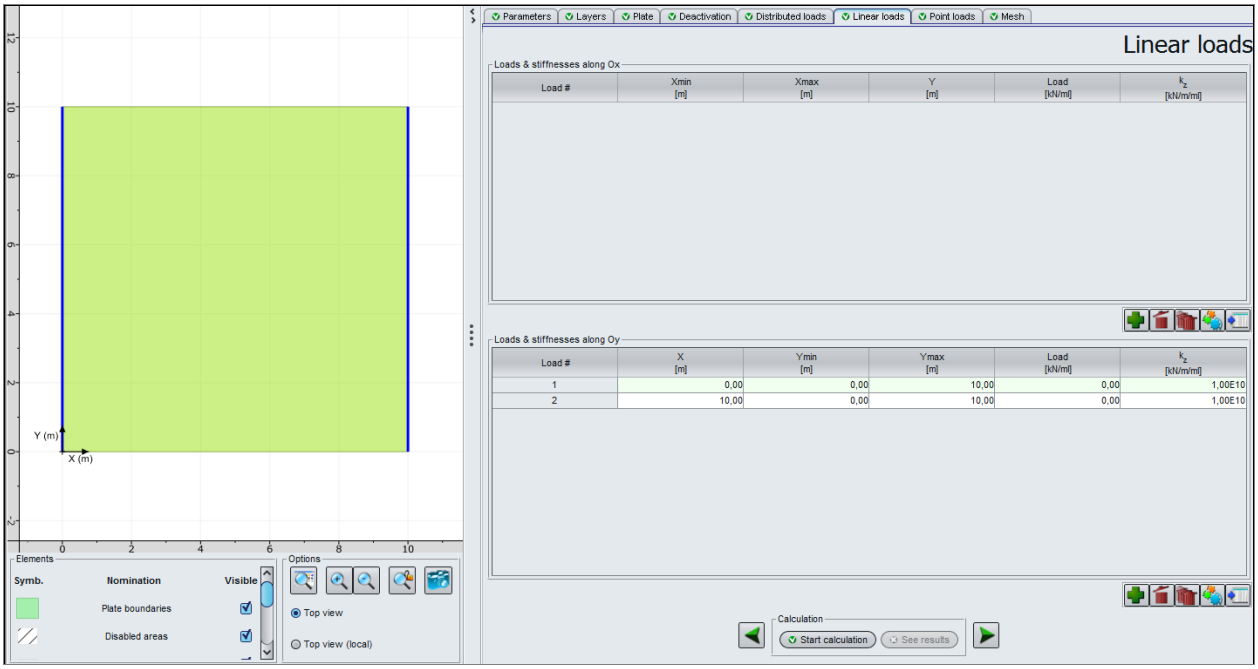


I.4.1.1.7. "Linear loads" tab

The "linear loads" option allows the definition of linear loads applied on the plate, in one hand, and in the other hand this option allows the definition of linear stiffness's which permits the simulation of some particular cases.

We will now use this latest function: in fact, the case of a simple support placed on some edges of the plate (the case of our example) can be modelled by considering a linear support with a translation stiffness high enough (10^{10} kN/m/ml for example).

In our case, 2 supports are to be defined under 2 opposite edges of the plate, thus 2 linear loads to be defined between Ymin = 0 and Ymax = 10 m with respectively X = 0 m and X = 10 m (these are linear stiffness's along Oy).



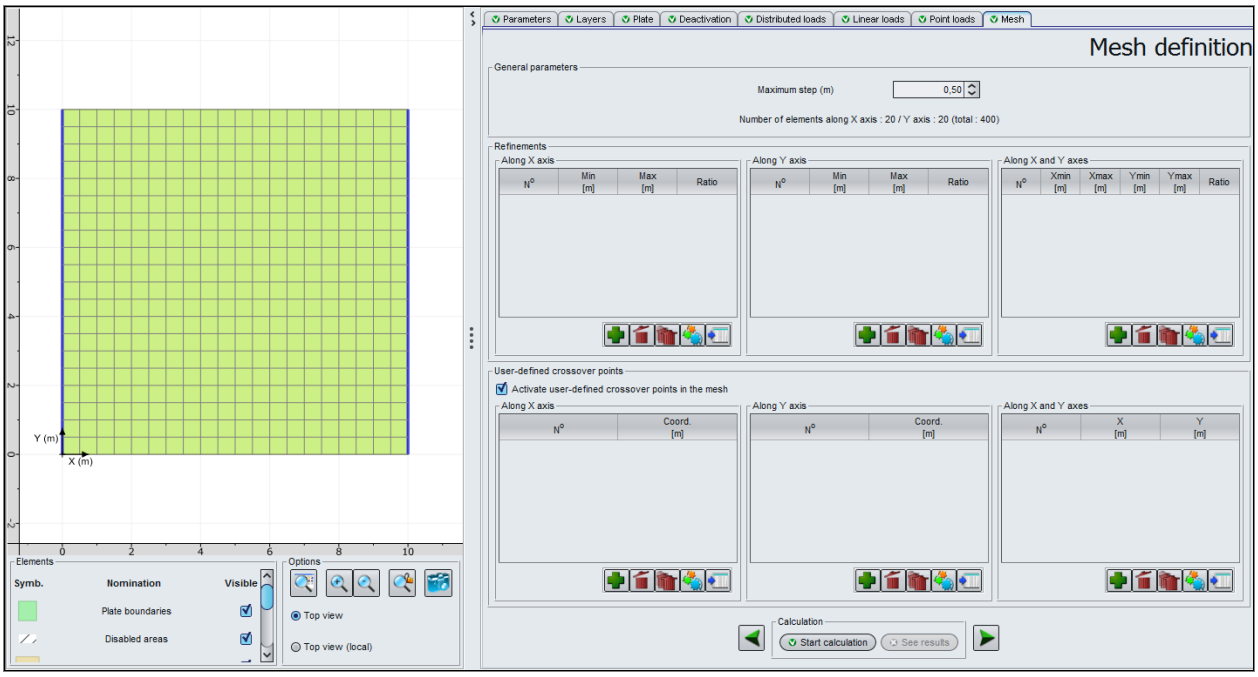
Load n°	X (m)	Ymin (m)	Ymax (m)	Load (kN/ml)	k_z (kN/m/ml)
1	0.00	0.00	10.00	0.00	1.00E10
2	10.00	0.00	10.00	0.00	1.00E10

I.4.1.1.8. "Point loads" tab

There is no point load in this example.

I.4.1.1.9. "Mesh" tab

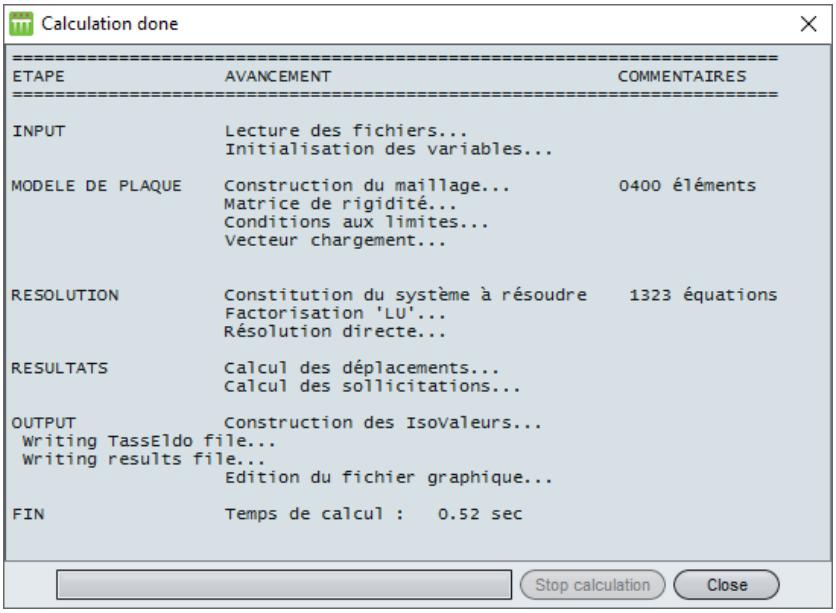
Here we opt for a maximum step of 0.50 m. This leads to a mesh of 20 elements along X axis and 20 elements along Y axis, thus 400 elements in total.



I.4.1.1.10. Calculations and results

Calculation

Click on the Start calculation button. A calculation window appears. It displays the different steps of the calculation and their progress. This is useful especially when the calculation becomes time consuming.



To display the results formatted as tables and graphical views, click on the button.

Results

Numerical results:

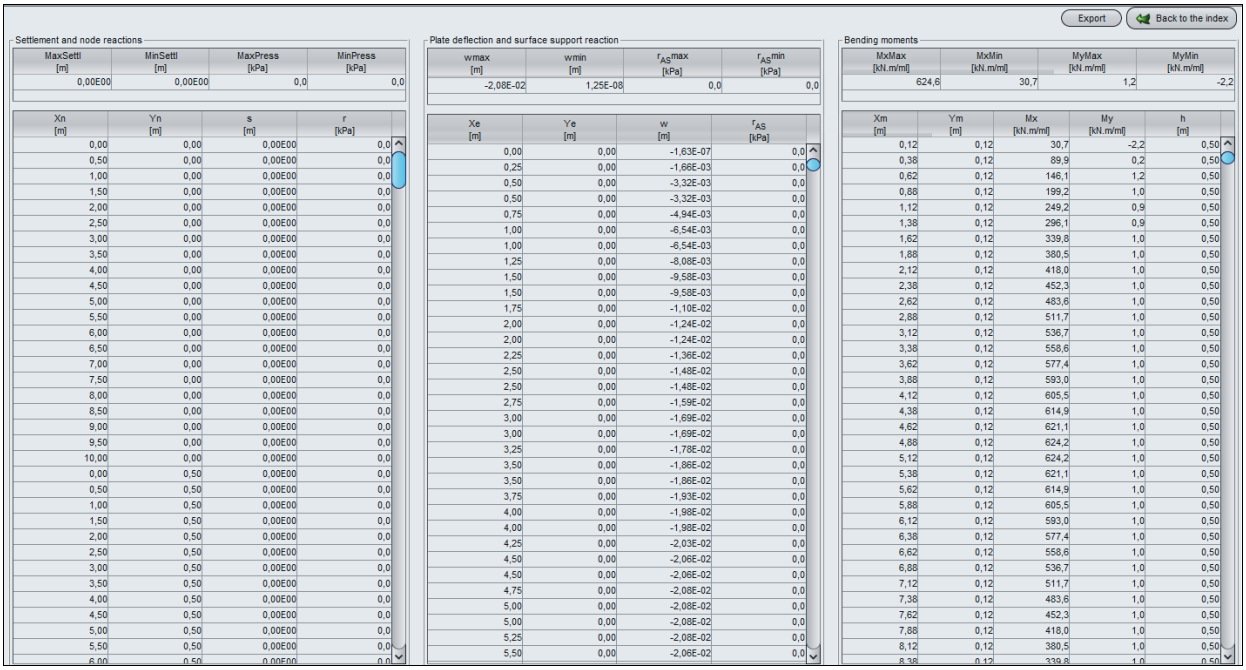
Table of results:

This type of display includes the following results:

- Settlement and node reactions;
- Plate deflection (9 points per element);
- Bending moments (4 points per element).

This window also includes tables showing the maximum and minimum values for each type of result.

The "Settlement and node reactions" tables correspond to the results obtained on the surface of the soil support if this latest has been taken as an elastic multilayer (see examples 2 to 7). If no multi-layered soil has been defined in the project (the case of the present example), displayed values would all be equal to zero.



Graphical results:

Cross-section charts:

This buttons allows to obtain cross-sections within the plate:

The window also includes tables indicating extreme global values of settlements, reactions, bending moments and deflections. To obtain this table, click on the Show button placed in the "Extreme global values" frame.

Four drop down lists are available on top of the window. To display one or two cross-sections, select the result type (physical quantity), direction of the cross-sections and their localisation.

A table showing the detail of the values for the defined cross-section(s) can be displayed at the right of the chart zone: in the frame "Cross-section details", click on the Show button.

Note: the Show buttons are contextual, their name changes once they have been activated and it becomes: Hide.

Plate deflection along X axis:

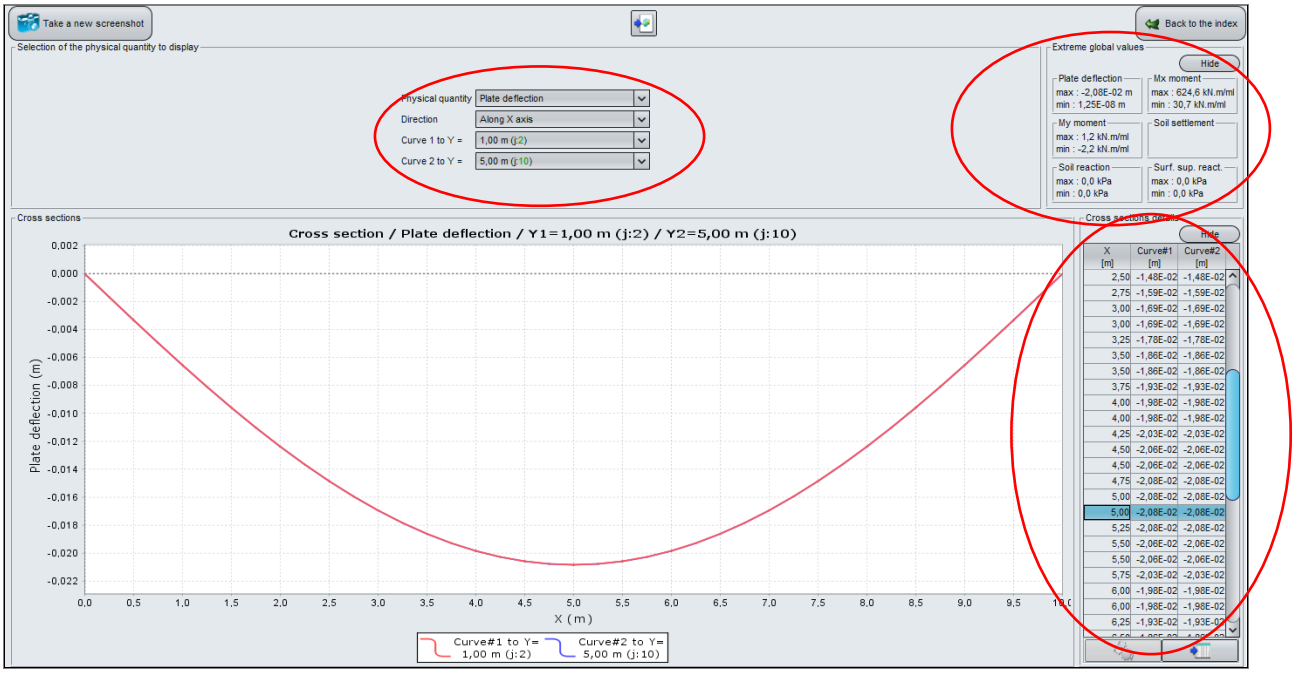
Extreme global values:

- Max: -2.08E-02 m (thus -2.08 cm)
- Min: 1.25E-08 m (thus 0.00 cm)

Cross-section 1 at Y = 1.00 m (j: 2)

Cross-section 2 at Y = 5.00 m (j: 10)

At X = 5 m (centre of the plate), we can read for 2 cross-sections a maximum deflection of 2.08 cm identical for both: the reason is the set of the limit conditions preventing any deflection along Oy direction. Any cross-sections along Oy would give a uniform deflection.



Note: In TASPLAQ, the displacements are positive upwards. Hence, a negative deflection means a "settlement". A positive deflection would correspond to an uplift comparing to the initial elevation.

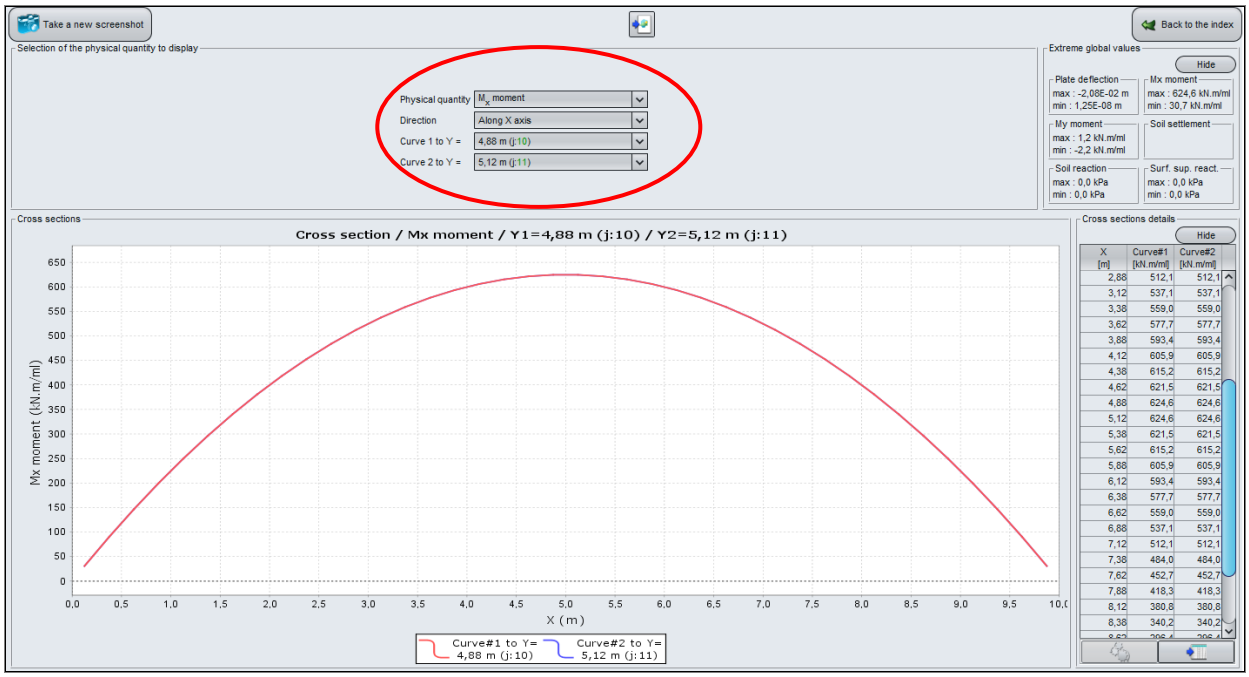
M_x Moment along X axis:

Extreme global values:

- Max: 625 kN.m/ml
- Min: 31 kN.m/ml

Cross-section 1 at Y = 4.88 m (j: 10): at X = 4.88 m / 5.12 m M_x: 624.6 kN.m/ml
Cross-section 5 at Y = 5.12 m (j: 11): at X = 4.88 m / 5.12 m M_x: 624.6 kN.m/ml

Both cross-sections are perfectly identical, the obtained maximum value (625 kN.m/ml) is to be compared to the one obtained for a uniformly loaded beam simply supported at ends.



Note 1: the M_x moment represents the bending moment around the –Oy axis, while the M_y axis represents the bending moment around the +Ox axis.

Note 2: a positive moment value means that the lower fibre is in traction.

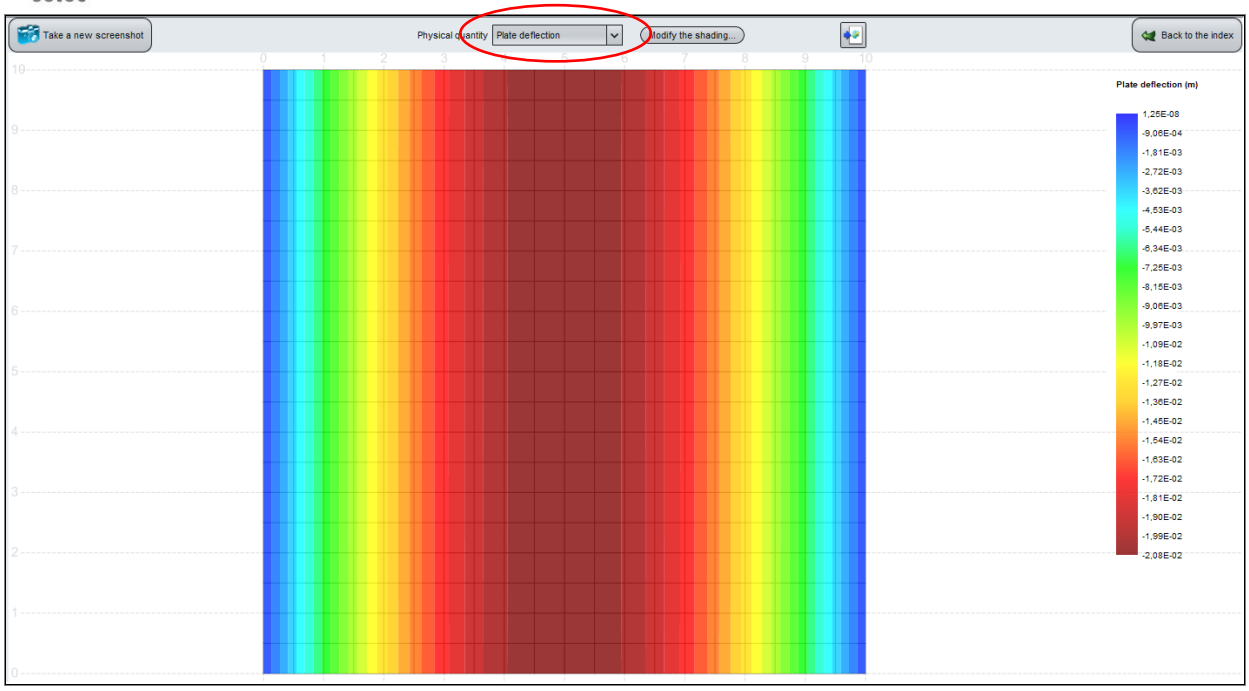
Shadings:

The window includes, on the top, a dropdown list which allows from the selection of the physical quantity to display as shadings.

The aim of this window is to help user to display the distribution of a given physical quantity, that allows in particular the selection of the most relevant cross-sections.

The legend at the right of the shadings shows a range of colours which represents each an interval of intensity of the selected physical quantity.

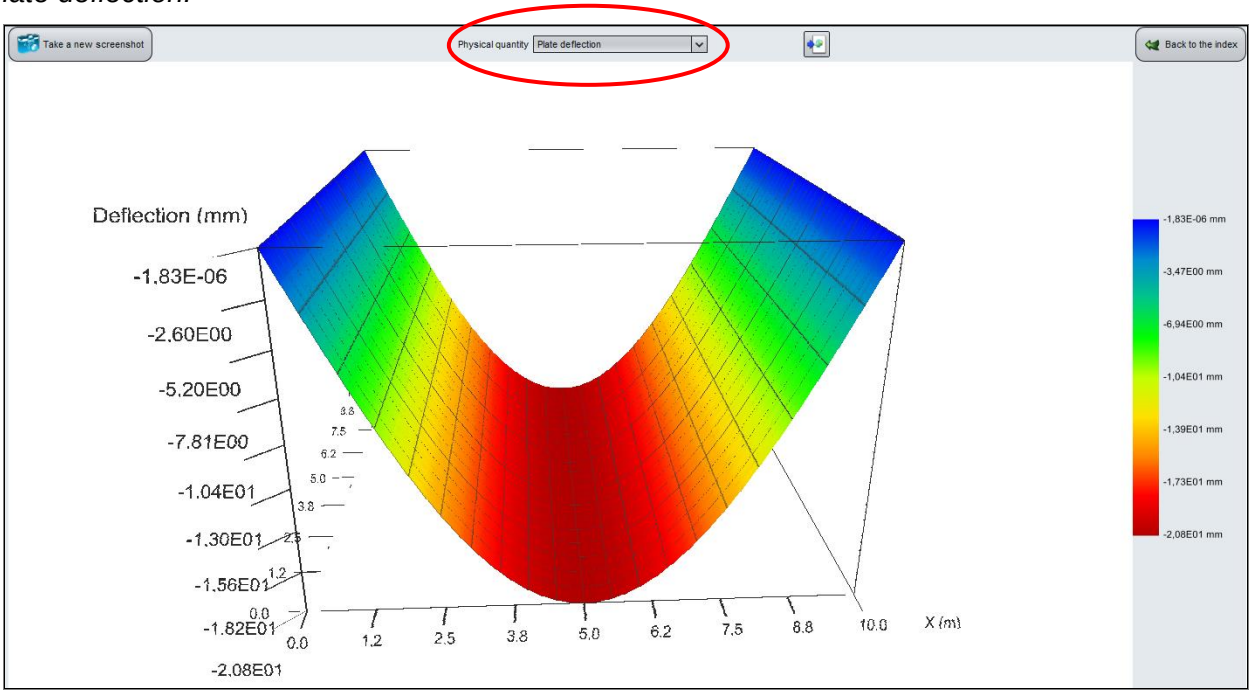
The Modify the shading... button allows the modification of the graduations of shadings in order to emphasise the characteristic values.



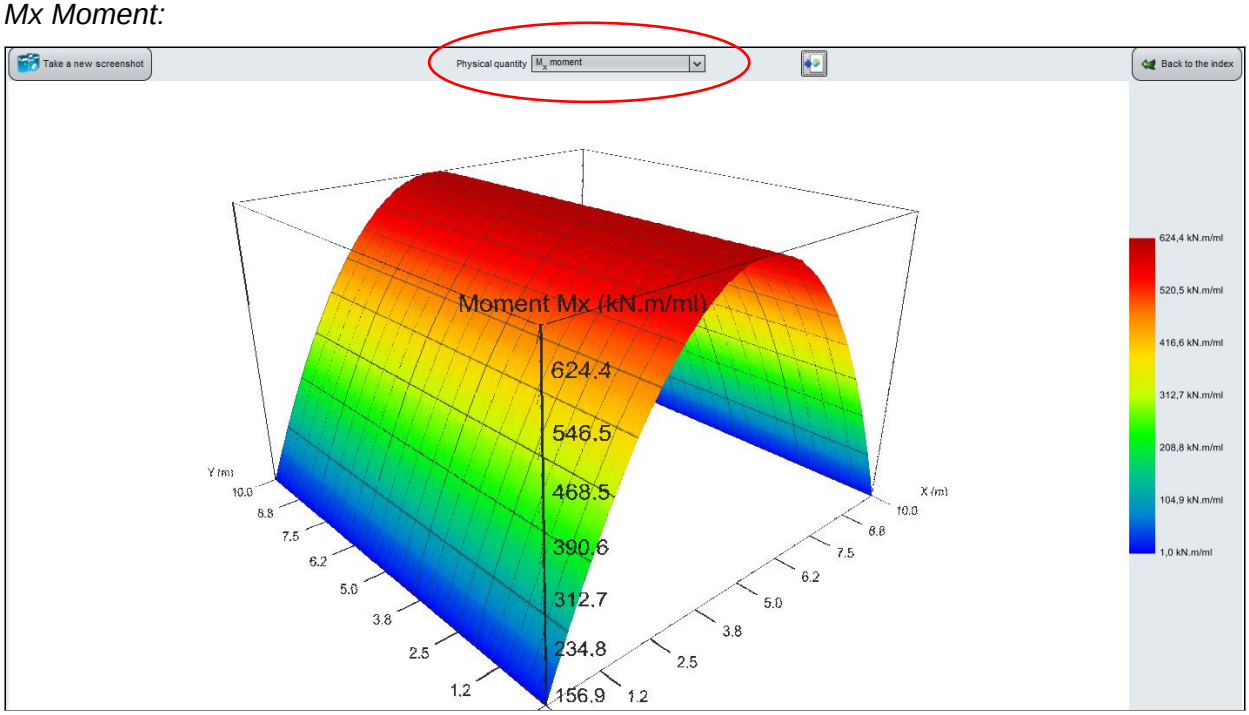
3D Graphics:

This button allows the display of the results as three-dimensional surfaces.
Select the physical quantity that you wish within the dropdown list available at top of the window.

Plate deflection:



Mx Moment:



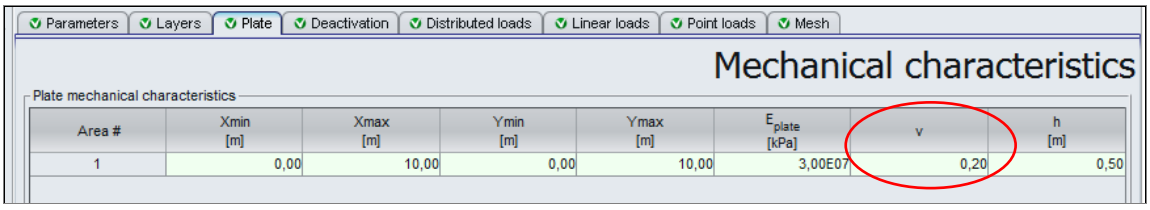
I.4.1.2. Step 2: Plate supported by its 4 edges

To go back to the input mode, click on the Back to the index button then on Back to the data.

Save as the project with another name: "Plate supported by its 4 edges" for example.

I.4.1.2.1. "Plate" tab

We will modify the Poisson ratio for this calculation: $\nu = 0.2$ (applied to whole plate).

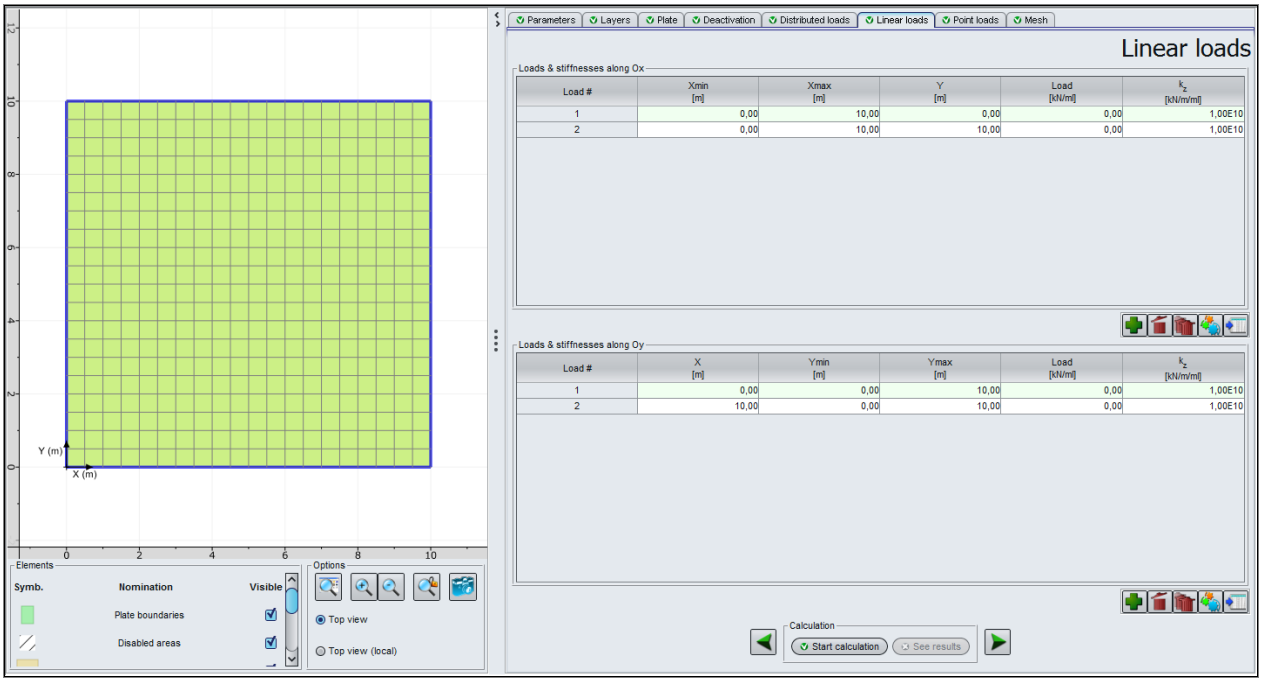


I.4.1.2.2. "Linear loads" tab

This time it suits to define simple supports along 4 edges (in spite of 2 as in the first step).

We add two linear load along Ox:

Load n°	X (m)	Ymin (m)	Ymax (m)	Loads (kN/ml)	k_z (kN/m/ml)
1	0.00	0.00	10.00	0.00	1.00E10
2	10.00	0.00	10.00	0.00	1.00E10

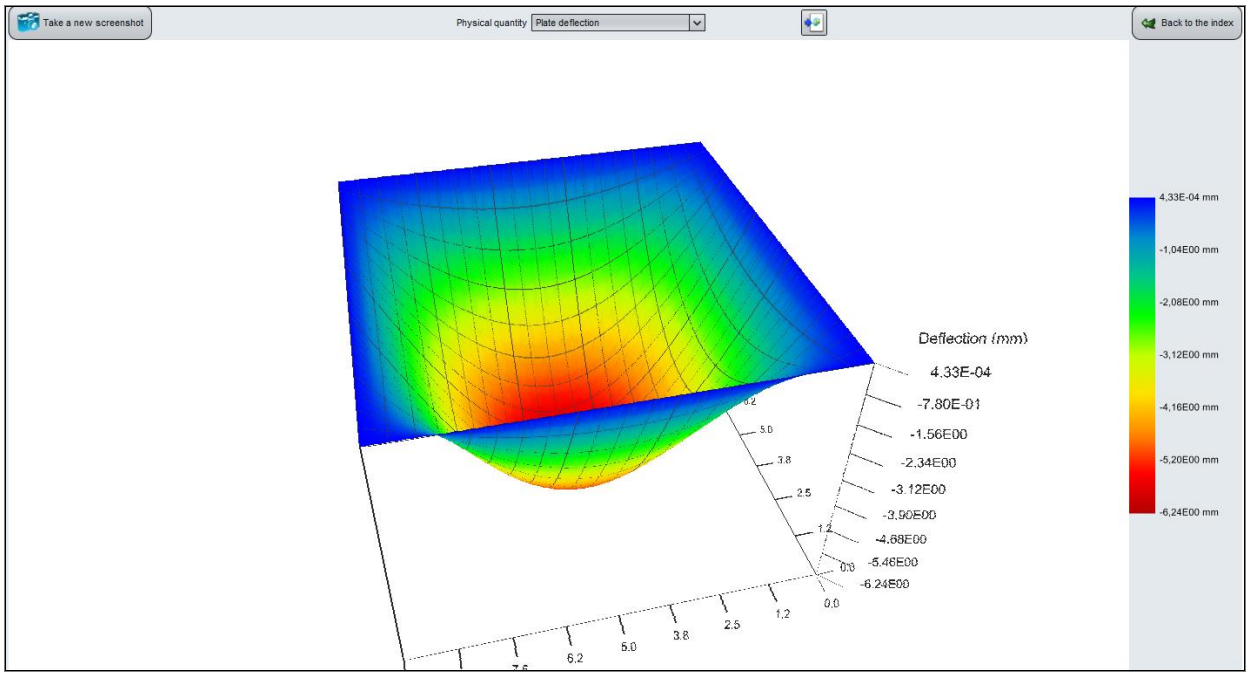


I.4.1.2.3. Calculations and results

Restart calculations by clicking on the Start calculation button.

- The maximum deflection of the plate is 6.3 mm (see extreme global values on the cross-section screen, for example). The display of the deflection as 3D view (see below) shows the expected behaviour of a plate on 4 supports (no displacement on edges, in particular).
- The maximum values of the M_x and M_y moments are identical (as the plate is square and loaded uniformly), with a maximum of 221 kN.m/ml.
  We note that the maximum value of M_x and M_y given in the extreme global values of the cross-sections screen is 220.8 kN.m/ml, whereas the maximum value indicated in M_x and M_y graphs is 221.7 kN.m/ml: this slight gap is due to the interpolation made for the 3D display (calculation of the maximum value at X (resp. Y) = 5.0 m), whereas the screen of the cross-sections shows the extrema obtained amongst the exact calculated values, i.e. at X (resp. Y) = 4.88 m and X (resp. Y) = 5.12 m in this particular case.

3D Results: Plate deflection



I.4.1.3. Comparison of the results

	Maximal Deflection (cm)		Bending moment M_x (kN.m/ml)	
	Min	Max	Min	Max
Plate supported by 2 edges	0.0	-2.08	31	625
Plate supported by 4 edges	0.0	-0.63	2.5	221

The fact of leaning on 4 edges reduces the amplitude of the moments and the displacements of the plate by a factor of 3 comparing to the case where the plate is supported on its 2 opposite edges only.

We can compare the obtained results to the ones we would obtain by using analytical solutions given in the document "Techniques de l'ingénieur" (see below). For the first case, the model is the one of a beam on two simple supports at ends, the second is the one of a square plate supported by its four edges:

A perfect match between both sets of results is to be highlighted.

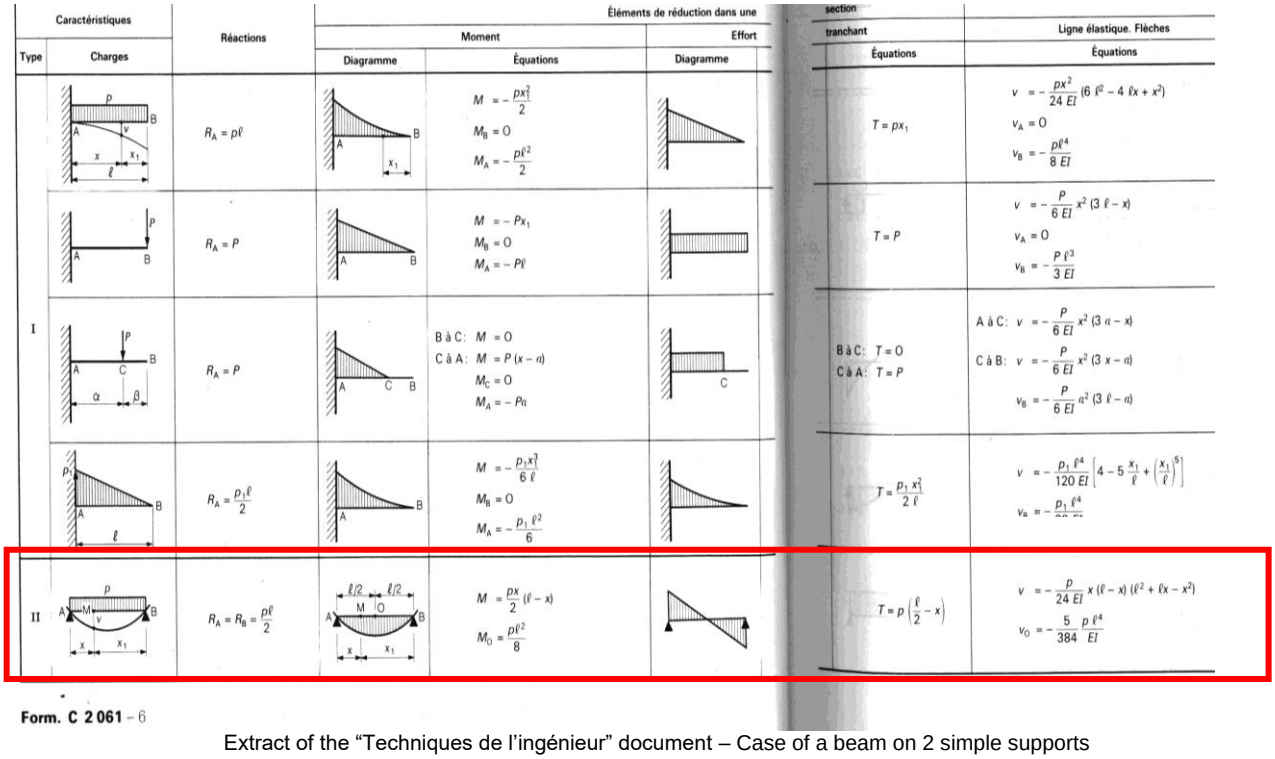
Table 1: comparison of the results – case of a beam supported by its 2 opposite edges:

	Maximal moment (kN.m/ml)	Maximal deflection (mm)
TASPLAQ Results	625	20.8
Reference values (Techniques de l'ingénieur)	625	20.8

Table 2: comparison of the results – case of a plate supported by its 4 edges:

	Maximal moment (kN.m/ml)	Maximal deflection (mm)
TASPLAQ Results	221	6.26
Reference values (Techniques de l'ingénieur)	221	6.24

Both figures hereafter remind the analytical solutions given in the "Techniques de l'ingénieur" document:



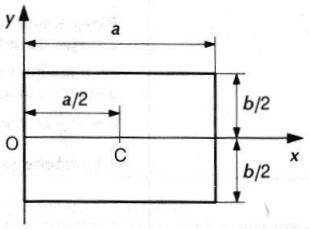
Extract of the "Techniques de l'ingénieur" document – Case of a beam on 2 simple supports

Tableau LI. – Plaques rectangulaires appuyées sur 4 côtés.

La plaque est simplement appuyée sur ses côtés

$x = 0,\ x = a,\ y = -\frac{1}{2}b,\ y = \frac{1}{2}b$

On pose : $\rho = \frac{a}{b}$ et $\lambda = \frac{m\pi b}{2a} = \frac{m\pi}{2\rho}$



Plaque supportant une densité de charge constante *p* sur toute sa surface.

- La **flèche** *w* au centre C de la plaque a pour valeur :

$$w = -\frac{5\,pa^2b^2}{384\,D} f(\rho) = -\frac{5\,pa^4}{384\,D} g(\rho) = -\frac{5\,pb^4}{384\,D} h(\rho)$$

avec : $g(\rho) = 1 - \frac{768}{5} \sum_m{}' \frac{1}{m^5\pi^5} \left(\frac{2 + \lambda \operatorname{th}\lambda}{\operatorname{ch}\lambda}\right) \sin\frac{m\pi}{2}$

$f(\rho) = \rho^2 g(\rho),\quad h(\rho) = \rho^2 f(\rho),\quad f\left(\frac{1}{\rho}\right) = f(\rho),\quad g\left(\frac{1}{\rho}\right) = h(\rho)$

On peut donc supposer $\rho < 1$.

ρ	$f(\rho)$	$g(\rho)$	$h(\rho)$
0	0	1	0
0,1	0,01000	1,00000	0,00010
0,2	0,03985	0,99616	0,00159
0,3	0,08652	0,96135	0,00779
0,4	0,14126	0,88288	0,02260
0,5	0,19447	0,77788	0,04862
0,6	0,23963	0,66565	0,08627
0,7	0,27370	0,55857	0,13411
0,8	0,29624	0,46288	0,18959
0,9	0,30839	0,38073	0,24979
1	0,31199	0,31199	0,31199

- Les **moments fléchissants** au centre C de la plaque ont pour valeurs :

$$M_1 = pab\,[F(\rho) + \nu\,G(\rho)] \quad \text{et} \quad M_2 = pab\,[\nu\,F(\rho) + G(\rho)]$$

avec
$$\begin{cases} F(\rho) = \rho\left[\frac{1}{8} - \sum_m{}' \frac{2}{m^3\pi^3}\left(\frac{2 + \lambda \operatorname{th}\lambda}{\operatorname{ch}\lambda}\right)\sin\frac{m\pi}{2}\right] \\ G(\rho) = \sum_m{}' \frac{1}{m^3\pi^3}\left(\frac{\operatorname{th}\lambda}{\operatorname{ch}\lambda}\right)\sin\frac{m\pi}{2} \end{cases}$$

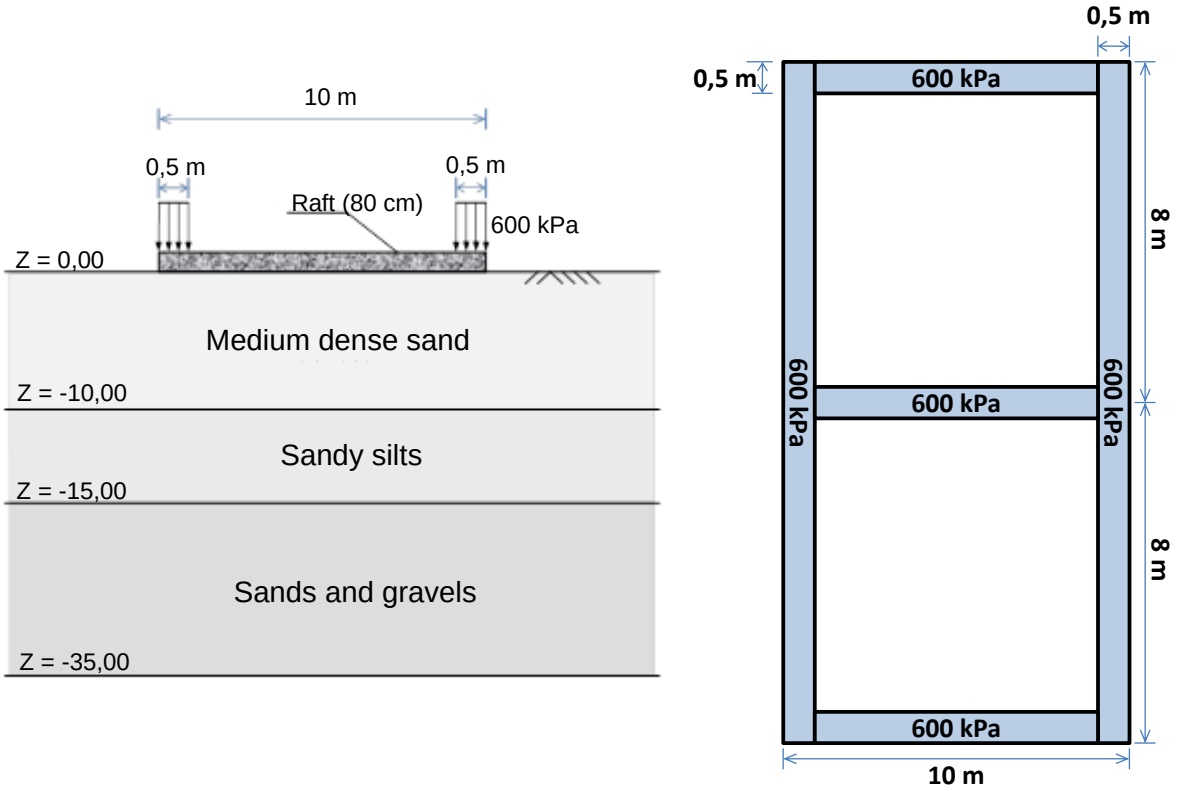
En permutant *a* et *b*, on voit que $F\left(\frac{1}{\rho}\right) = G(\rho)$; on peut donc supposer $\rho < 1$.

ρ	$F(\rho)$	$G(\rho)$	ρ	$F(\rho)$	$G(\rho)$
0	0	0	0,55	0,049086	0,011532
0,05	0,006250	0,000000	0,60	0,049229	0,014540
0,10	0,012500	0,000000	0,65	0,048771	0,017638
0,15	0,018743	0,000006	0,70	0,047821	0,020749
0,20	0,024901	0,000079	0,75	0,046484	0,023805
0,25	0,030751	0,000378	0,80	0,044853	0,026755
0,30	0,036010	0,001078	0,85	0,043009	0,029557
0,35	0,040457	0,002277	0,90	0,041021	0,032183
0,40	0,043980	0,003988	0,95	0,038947	0,034613
0,45	0,046586	0,006159	1	0,036836	0,036836
0,50	0,048230	0,008706			

Extract of the "Techniques de l'ingénieur" document – case of a plate supported by its 4 edges

I.4.2. Example 2: Locally loaded rafts

This example concerns the study of a rectangular and homogeneous raft of 10 m x 16 m, subjected to local loads representing peripheral walls? and a central wall for separation purpose. The input information of the study is described in the figure below:



I.4.2.1. Step 1: Single raft

At the beginning, Foxta proposes:

- to create a new project;
- to open an existing project;
- to open recent project.

In the case of this example:

- To create a new project, click on the (◉ New project) radio-button
- Click on the (OK) button.

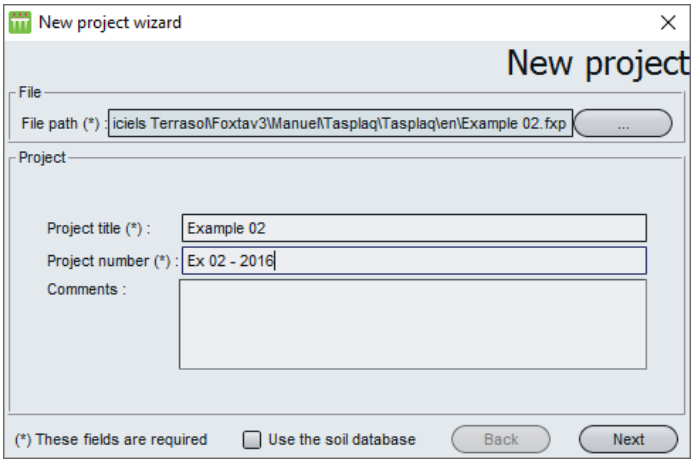
I.4.2.1.1. New project wizard

"File" frame:

- Click on the (...) button to choose the work directory;
- Name and save the file.

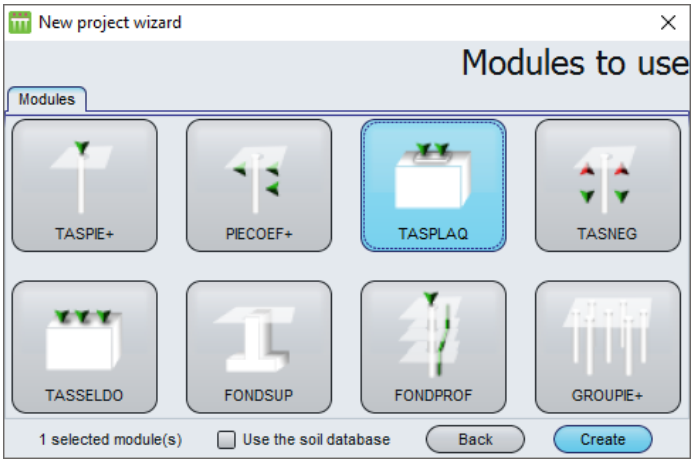
"Project" frame:

- Define a title for the project;
- Define a project number;
- Complete comments if necessary;
- Leave the " Use the soil database" box unchecked (for this example we will not use the database), and click on the Next button.



I.4.2.1.2. New project wizard: Module selection

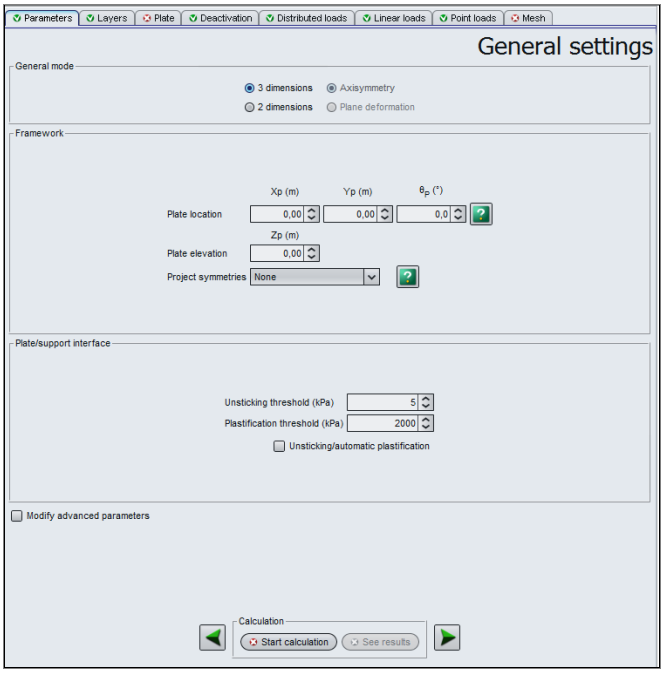
In the "Modules to use" window, select the TASPLAQ module and click on the Create button.



The input data window of TASPLAQ module is now displayed, all different data tabs should be completed.

I.4.2.1.3. "Parameters" tab

This tab includes four different frames:



"General mode" frame:

The study model is three dimensional, click on the radio button "3 dimensions".

"Framework" frame:

- Plate location:
  In this example, the local coordinate system of the plate representing the raft is taken identical to the global one. The (0.0) point represents the "lower-left" corner of the model.

X_p (m)	Y_p (m)	θ_p (°)
0.0	0.0	0.0

- Plate elevation: Zp = 0.0 m (reference elevation = elevation of the underside of the raft).
- Project symmetries: None

"Plate/support interface" frame:

This frame allows the activation of the automatic management of the unsticking and the plastification criteria by using the criteria defined by the user. Default values are kept as they are in this example (option deactivated).

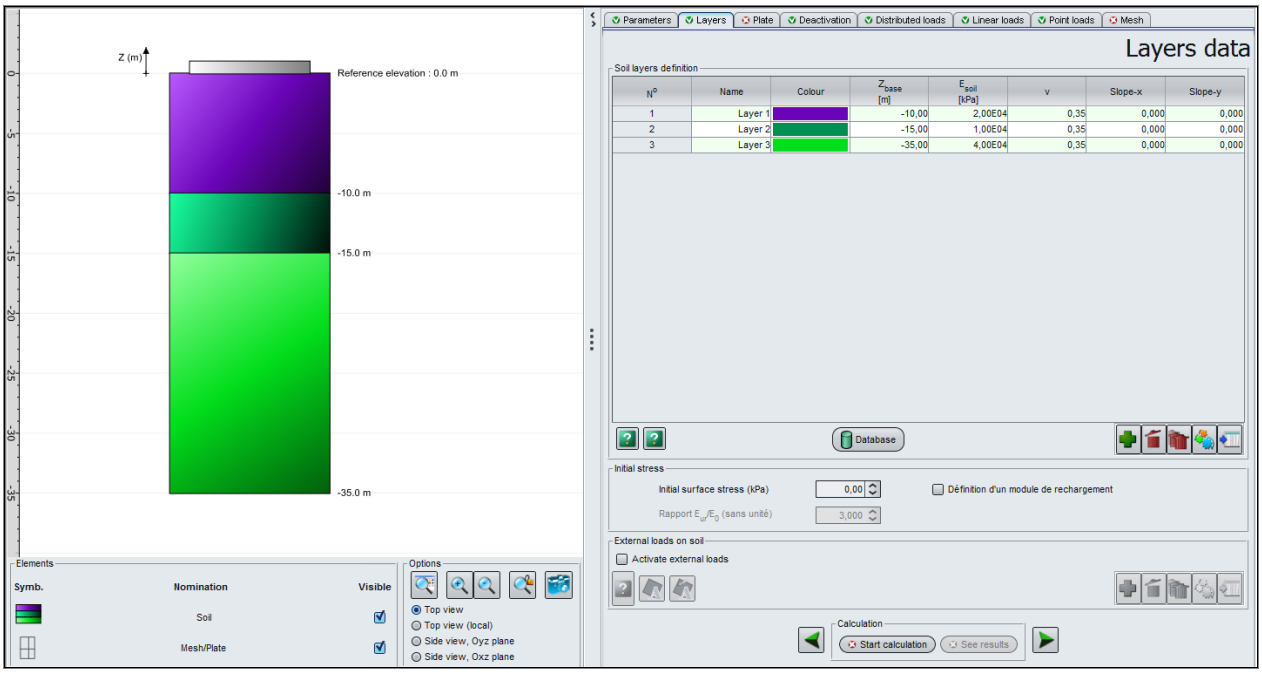
Unsticking threshold	5 kPa
Plastification threshold	2000 kPa

- Automatic unsticking/plastification: Unchecked
- Modify advanced parameters: Unchecked

To shift to the next tab, click either on the name of the tab "Layers", or on the button.

I.4.2.1.4. "Layers" tab

This tab includes two different frames:



"Soil layers definition" frame:

Click on the button to create 3 layers.

The soil will be modelled here as an elastic multilayer (3 layers) block. The input data for different soil layers are the following:

Name	Z_{base} (m)	E_{soil} (kPa)	ν	Slope-x	Slope-y
Layer 1	-10.00	2.00E+04	0.35	0.00	0.00
Layer 2	-15.00	1.00E+04	0.35	0.00	0.00
Layer 3	-35.00	4.00E+04	0.35	0.00	0.00

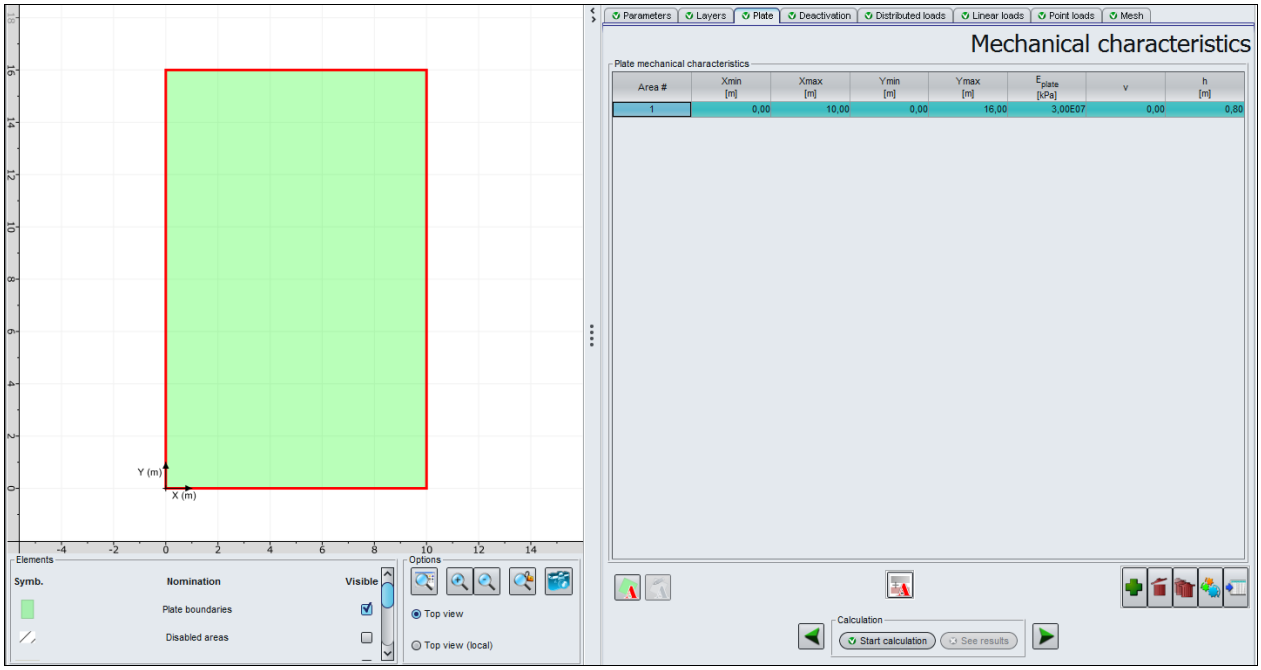
- Initial surface stress: 0 kPa

"External loads on soil" frame:

- Activate external loads: Unchecked

I.4.2.1.5. "Plate" tab

In this example the raft is homogeneous and continuous. To define its characteristics, we just have to define one single zone with Xmin = 0, Xmax = 10 m / Ymin = 0, Ymax = 16 m.



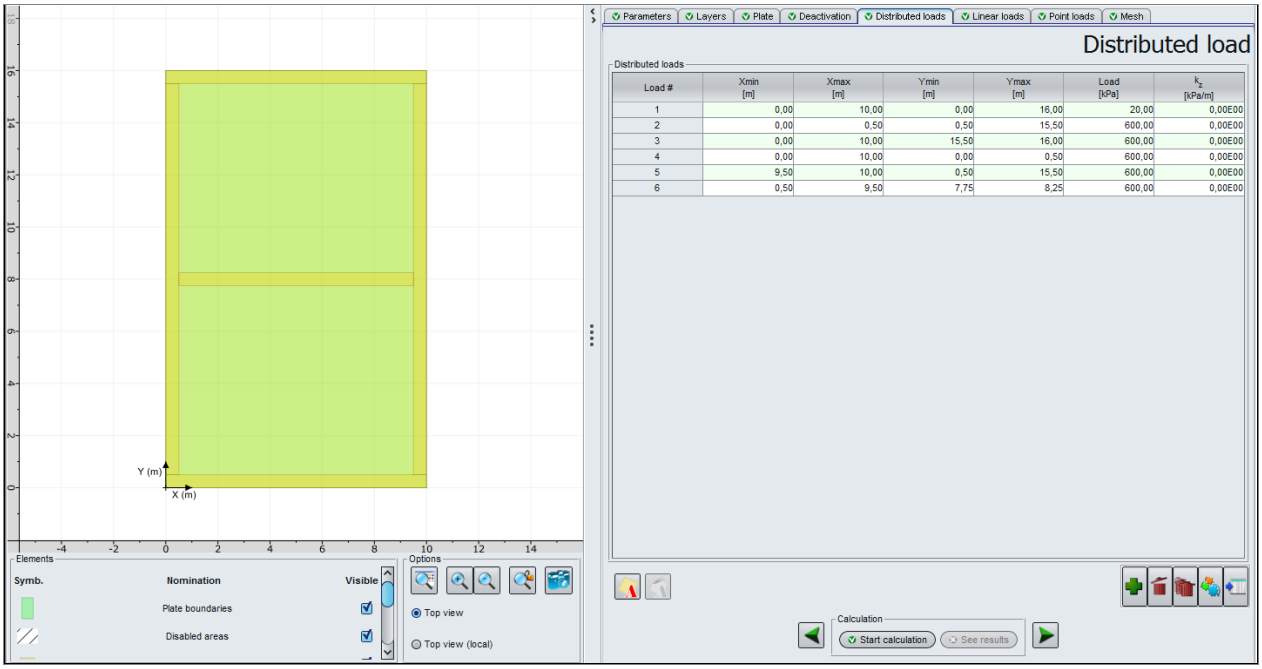
The input data are the following:

Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	10.00	0.00	16.00	3.00E+07	0.00	0.80

The choice of a Poisson ratio equal to zero is due to the fact that the raft is composed of reinforced concrete.

I.4.2.1.6. "Distributed loads" tab

The load includes the dead load of the raft on the entire surface of the plate (20 kPa) and the load brought by the walls (600 kPa along 50 cm large strips).



- Fill in manually the load values in different lines (20 kPa for the global load, and 600 kPa for the zones corresponding to the walls):

Load n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	Load (kPa)	k_z (kPa/m)
1	0.00	10	0.00	16.00	20	0
2	0.00	0.50	0.50	15.50	600	0
3	0.00	10.00	15.50	16.00	600	0
4	0.00	10.00	0.00	0.50	600	0
5	9.50	10.00	0.50	15.50	600	0
6	0.50	9.50	7.75	8.25	600	0

Note: it is worth to note that if several loads are defined in a same zone, they are summed up. Hence, in the calculation, a total load of 620 kPa is assigned to the loading zones representing the walls.

I.4.2.1.7. "Linear loads" tab

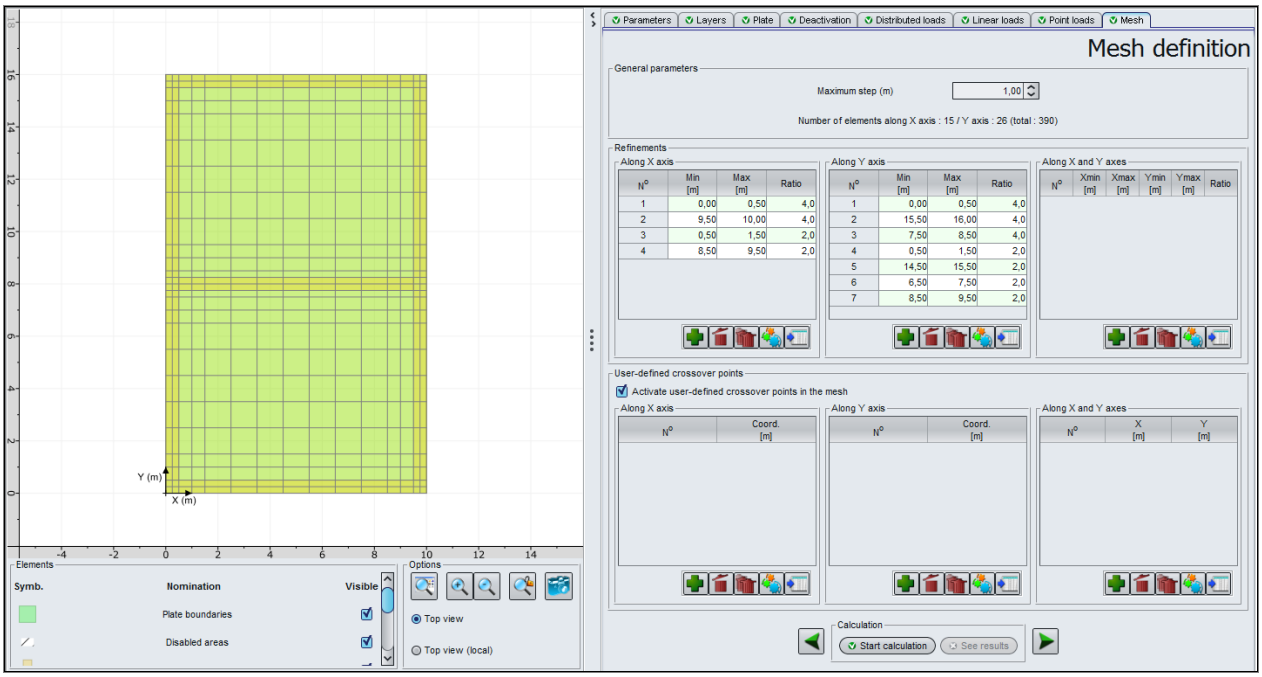
There is no linear load in this example.

I.4.2.1.8. "Point loads" tab

There is no point load in this example.

I.4.2.1.9. "Mesh" tab

Here the mesh is defined with a maximum step of 1 meter and a progressive refinement is applied around the walls.



The ratio column corresponds to the refinement ratio compared to the maximum step. It is aimed to determine the maximum step locally applied in the refined zone.

For example, imposing a ratio of 4 to the zones where local loads are applied implies a maximum step of 1 / 4 = 0.25 m within these zones and along the considered direction.

The input data are the following:

- Maximum step = 1 m

Refinement along X axis:

N°	Min (m)	Max (m)	Ratio
1	0.00	0.50	4.0
2	9.50	10.00	4.0
3	0.50	1.50	2.0
4	8.50	9.50	2.0

Refinement along Y axis:

N°	Min (m)	Max (m)	Ratio
1	0.00	0.50	4.0
2	15.50	16.00	4.0
3	7.50	8.50	4.0
4	0.50	1.50	2.0
5	14.50	15.50	2.0
6	6.50	7.50	2.0
7	8.50	9.50	2.0

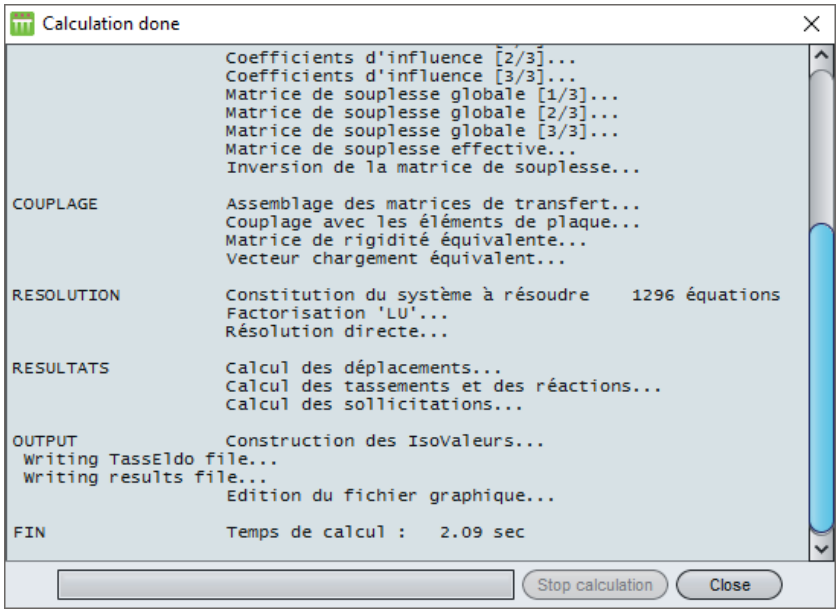
I.4.2.1.10. Calculations and results

Calculation

Click on the Start calculation button·

A click on this button will launch the calculation and will display the following window:

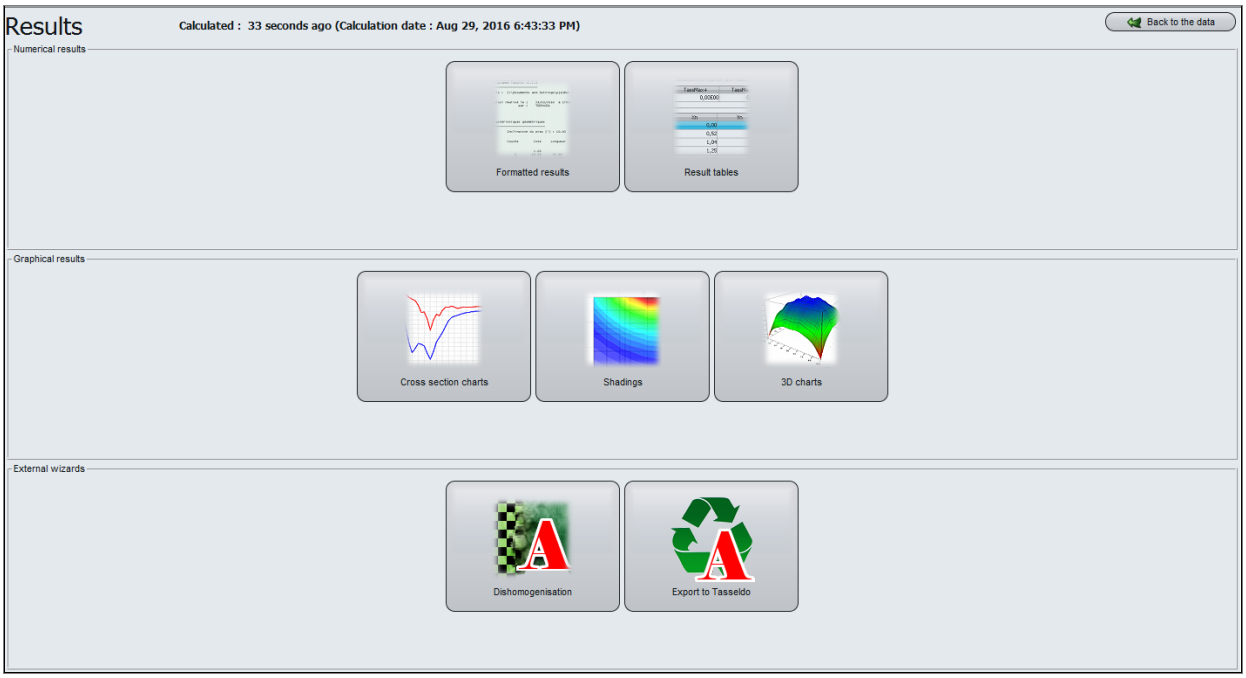
This window allows to follow of the progress of the calculation kernel.



Click on the Close button to close the window at the end of the calculation.
To display the results formatted as tables and graphical views, click on the See results button.

Results

Hereafter are the different types of results output (numerical and graphical):



Numerical results:

Formatted results:

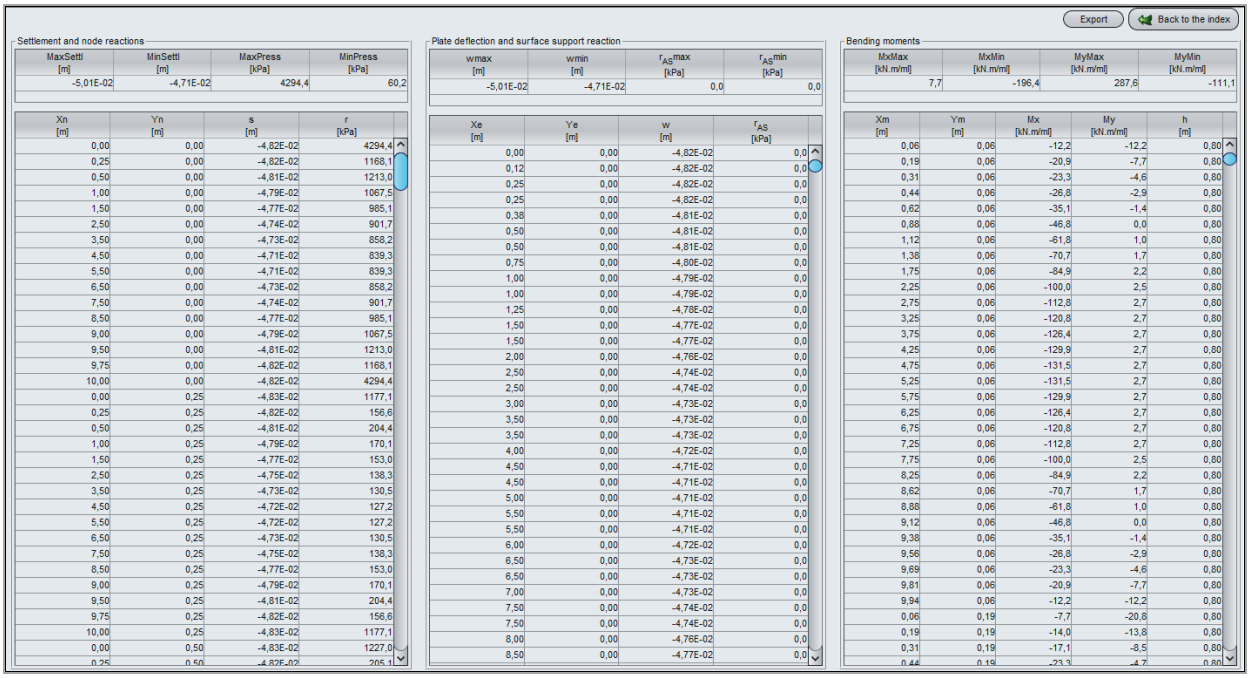
This display includes a summary of the calculation options, input data and results.

Shear forces (local coordinate system)

Xt	Yt	Tx	Ty
0.125	0.125	13.533	13.187
0.375	0.125	16.986	-15.437
0.750	0.125	44.136	-25.909
1.250	0.125	43.505	-19.295
2.000	0.125	32.417	-38.071
3.000	0.125	16.673	-30.722
4.000	0.125	6.896	-27.238
5.000	0.125	0.000	-26.640
6.000	0.125	-6.896	-27.238
7.000	0.125	-16.673	-30.722
8.000	0.125	-32.417	-38.071
8.750	0.125	-43.505	-19.295
9.250	0.125	-44.136	-25.909
9.625	0.125	-16.986	-15.437
9.875	0.125	-13.533	13.187
0.125	0.375	-15.132	14.737
0.375	0.375	23.781	21.397
0.750	0.375	53.775	48.917
1.250	0.375	60.003	59.801
2.000	0.375	37.401	77.110
3.000	0.375	18.229	78.640
4.000	0.375	7.302	80.623
5.000	0.375	0.000	81.482
6.000	0.375	-7.302	80.623
7.000	0.375	-18.229	78.640
8.000	0.375	-37.401	77.110
8.750	0.375	-60.003	59.801
9.250	0.375	-53.775	48.917
9.625	0.375	-23.781	21.397
9.875	0.375	15.132	14.737
0.125	0.750	-25.663	38.234
0.375	0.750	51.144	47.736
0.750	0.750	77.729	71.771
1.250	0.750	53.986	86.106
2.000	0.750	35.452	110.447
3.000	0.750	18.804	113.945
4.000	0.750	7.900	116.265
5.000	0.750	0.000	117.205
6.000	0.750	-7.900	116.265
7.000	0.750	-18.804	113.945
8.000	0.750	-35.452	110.447
8.750	0.750	-53.986	86.106
9.250	0.750	-77.729	71.771
9.625	0.750	-51.144	47.736
9.875	0.750	25.663	38.234
0.125	1.250	-19.372	32.151
0.375	1.250	62.002	48.165
0.750	1.250	92.083	42.289
1.250	1.250	63.298	51.630
2.000	1.250	38.812	66.268
3.000	1.250	20.576	74.699
4.000	1.250	8.831	78.030
5.000	1.250	0.000	78.976
6.000	1.250	-8.831	78.030
9.250	14.750	-92.083	-42.289
9.625	14.750	-62.002	-48.165
9.875	14.750	19.372	-32.151
0.125	15.250	-25.663	-38.234
0.375	15.250	51.144	-47.736
0.750	15.250	77.729	-71.771
1.250	15.250	53.986	-86.106
2.000	15.250	35.452	-110.447
3.000	15.250	18.804	-113.945
4.000	15.250	7.900	-116.265
5.000	15.250	0.000	-117.205
6.000	15.250	-7.900	-116.265
7.000	15.250	-18.804	-113.945
8.000	15.250	-35.452	-110.447
8.750	15.250	-53.986	-86.106
9.250	15.250	-77.729	-71.771
9.625	15.250	-51.144	-47.736
9.875	15.250	25.663	-38.234
0.125	15.625	-15.132	-14.737
0.375	15.625	23.781	-21.397
0.750	15.625	53.775	-48.917
1.250	15.625	60.003	-59.801
2.000	15.625	37.401	-77.110
3.000	15.625	18.229	-78.640
4.000	15.625	7.302	-80.623
5.000	15.625	0.000	-81.482
6.000	15.625	-7.302	-80.623
7.000	15.625	-18.229	-78.640
8.000	15.625	-37.401	-77.110
8.750	15.625	-60.003	-59.801
9.250	15.625	-53.775	-48.917
9.625	15.625	-23.781	-21.397
9.875	15.625	15.132	-14.737
0.125	15.875	13.533	-13.187
0.375	15.875	16.986	15.437
0.750	15.875	44.136	25.909
1.250	15.875	43.505	19.295
2.000	15.875	32.417	38.071
3.000	15.875	16.673	30.722
4.000	15.875	6.896	27.238
5.000	15.875	0.000	26.640
6.000	15.875	-6.896	27.238
7.000	15.875	-16.673	30.722
8.000	15.875	-32.417	38.071
8.750	15.875	-43.505	19.295
9.250	15.875	-44.136	25.909
9.625	15.875	-16.986	15.437
9.875	15.875	-13.533	-13.187

0.750	10.000	127.089	
9.250	10.000	-127.089	
8.000	8.375		144.787
2.000	7.625		-144.787

Result tables:



Graphical results: deflection of the plate

Cross-section graphs:

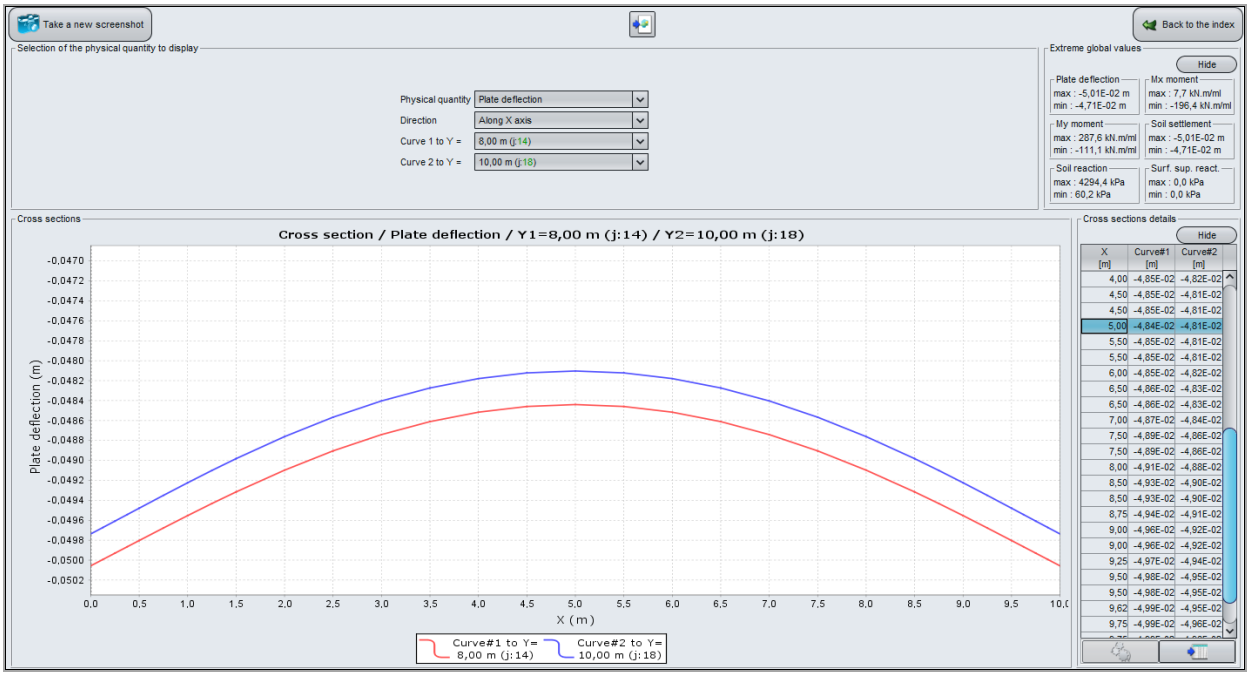
Extreme global values:

- Max: -5.01E-02 m (thus -5.01 cm)
- Min: -4.71E-02 m (thus -4.71 cm)

Cross-section graph along X axis with:

Cross-section 1 at Y = 8.00 m (j: 14): at X = 5.00 m Deflection = - 4.84 cm

Cross-section 2 at Y = 10.00 m (j: 18): at X = 5.00 m Deflection = - 4.81 cm

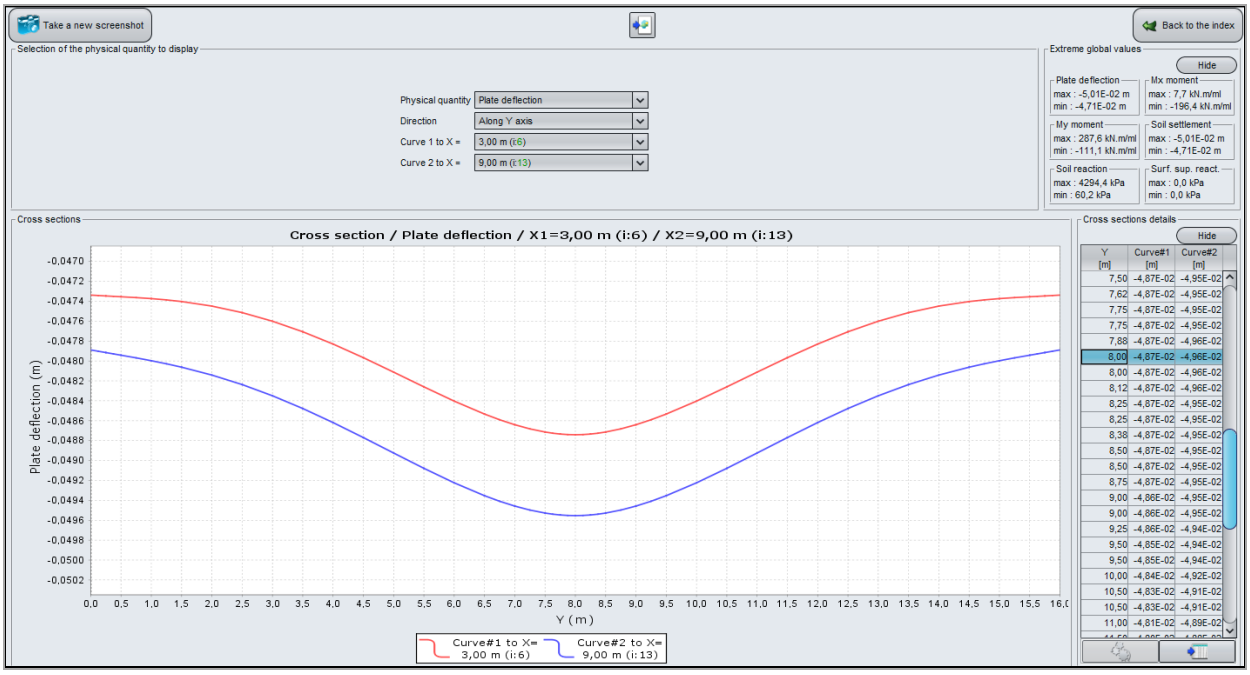


Note: In TASPLAQ, the displacements are positive upwards. Hence, a negative deflection means a "settlement". A positive deflection would correspond to an uplift compared to the initial elevation.

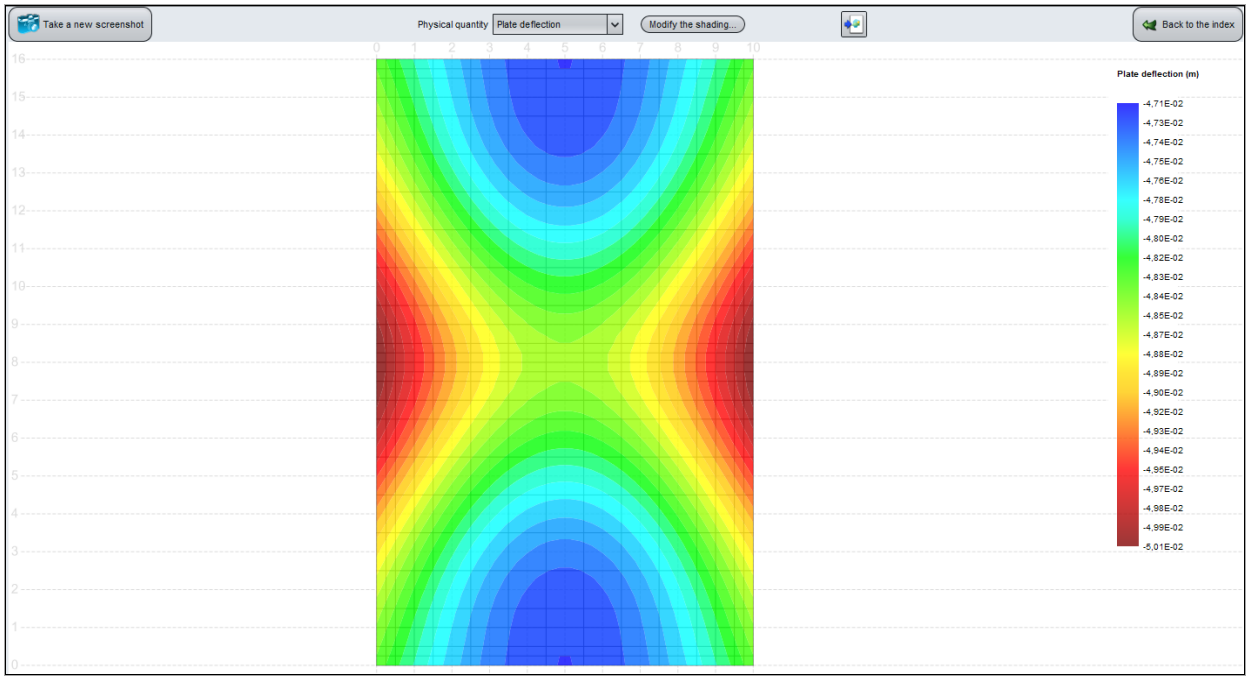
Cross-section graph along Y axis with:

Cross-section 1 at X = 3.00 m (i: 6): at Y = 8.00 m Deflection = - 4.87 cm

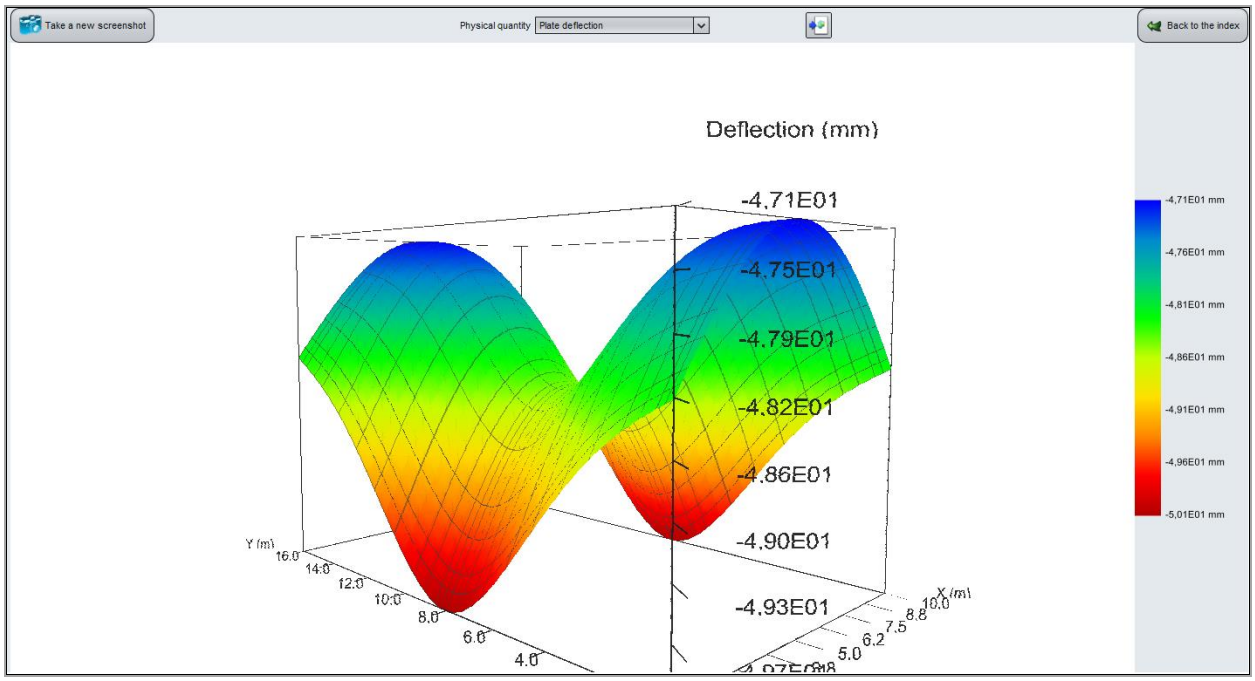
Cross-section 2 at X = 9.00 m (i: 13): at Y = 8.00 m Deflection = - 4.96 cm



Shadings



3D Graph



The maximum deflection is obtained at the centre of the peripheral longitudinal walls.

Graphical results: Moments M_y

Graphical cross-sections

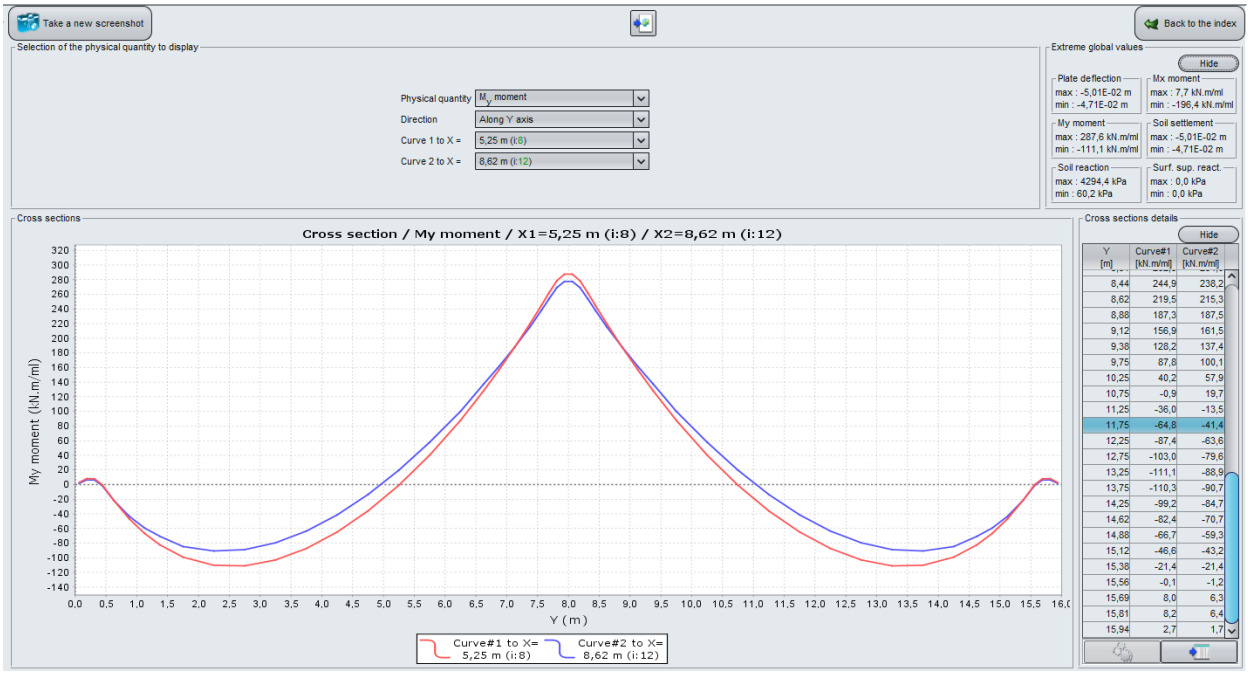
Extreme global values:

- Max: 288 kN.m/ml
- Min: -111 kN.m/ml

Cross-section graph along Y axis with:

Cross-section 1 at X = 5.25 m (i: 8): at Y = 8.06 m M_y = 288 kN.m/ml

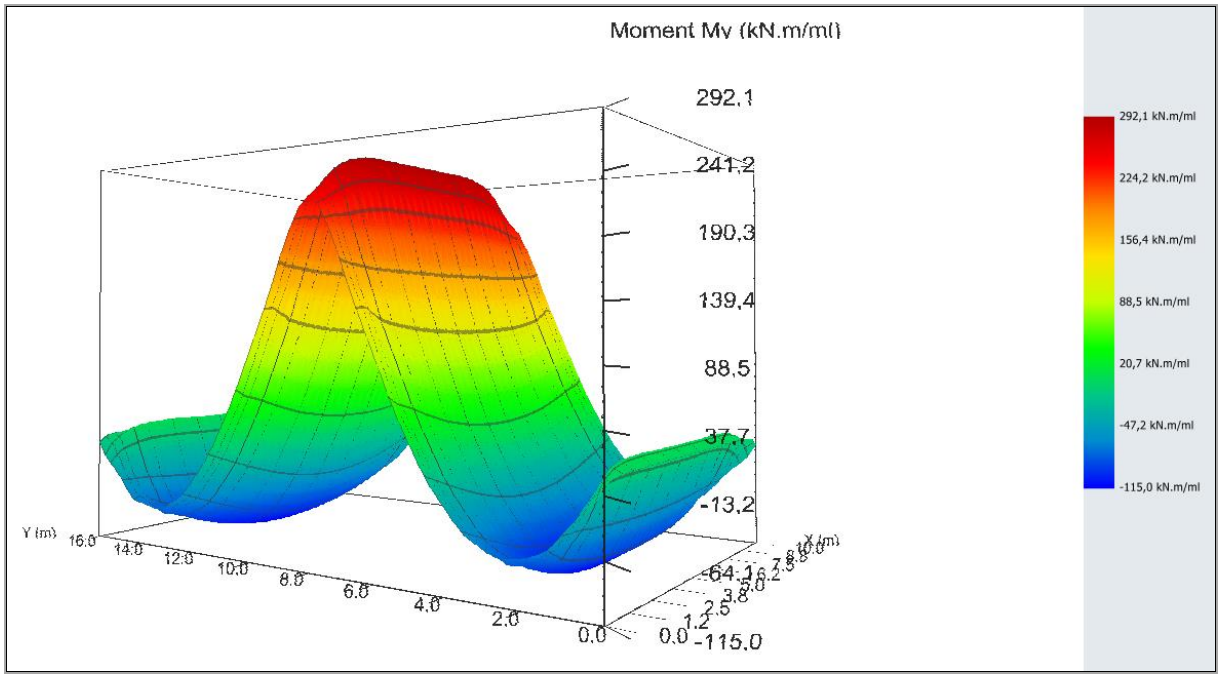
Cross-section 2 at X = 8.62 m (i: 12): at Y = 8.06 m M_y = 278 kN.m/ml



Note 1: the M_x moment represents the bending moment around the –Oy axis, whereas the M_y moment represents the bending moment around the Ox axis.

Note 2: a positive moment value means that the lower fibre is in traction.

3D Graph:



Graphical results: soil reaction

Soil reaction:

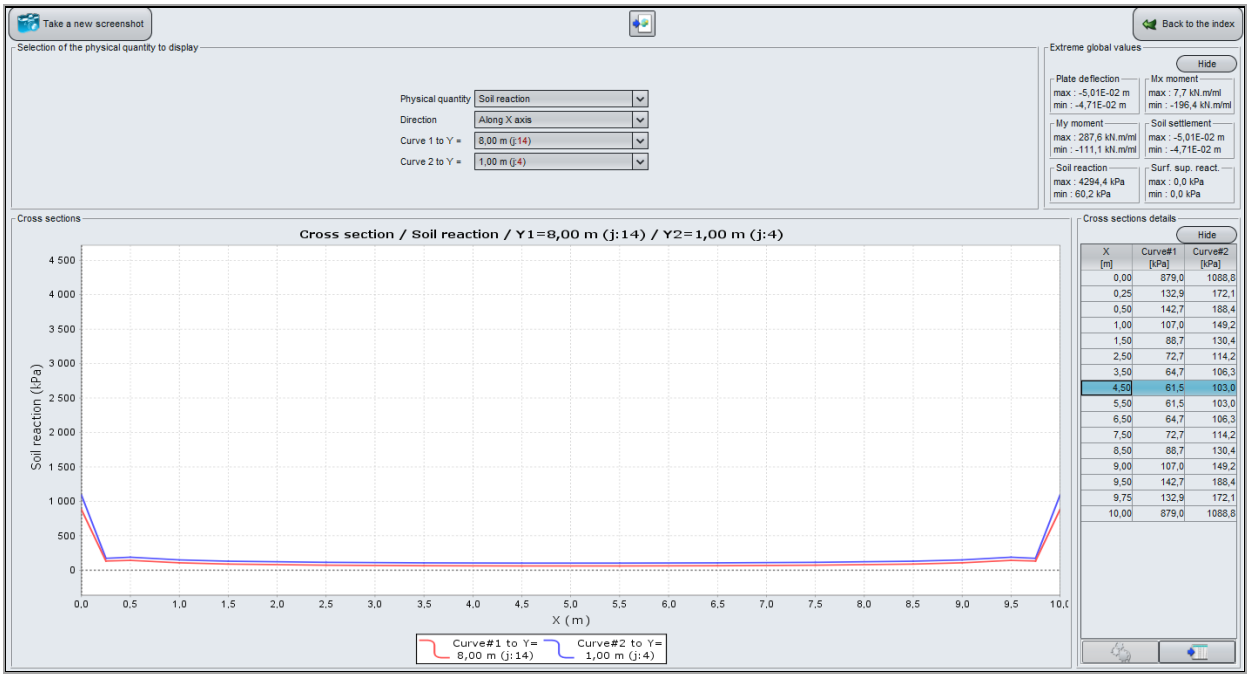
Graphical cross-sections

Global extreme values of the soil reaction:

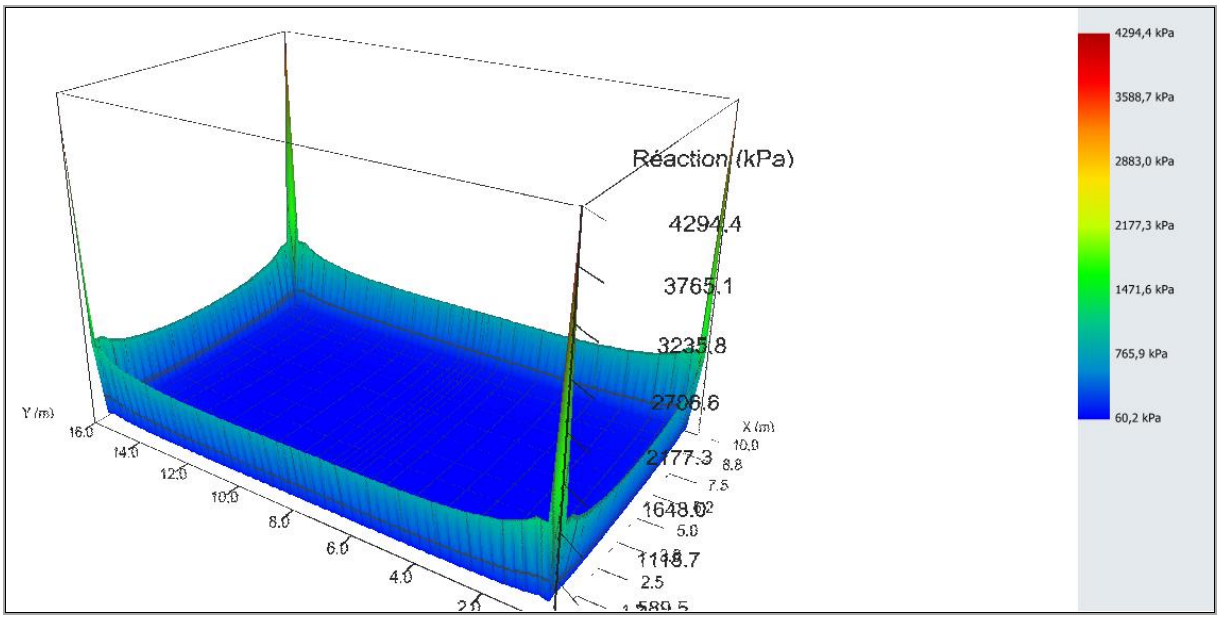
- Max: 4.3 MPa (boundary effect, which can be limited by plastic strains)
- Min: 0.06 MPa

Cross-section graph along X axis with:

Cross-section 1 at Y = 8.00 m (j: 14):	at X = 4.50 m	Soil reaction= 61.5 kPa
Cross-section 2 at Y = 1.00 m (j: 4):	at X = 4.50 m	Soil reaction= 103.0 kPa


Take a new screenshot
Back to the index
Selection of the physical quantity to display
Physical quantity
Soil reaction
Direction
Along X axis
Curve 1 to Y =
8,00 m (j:14)
Curve 2 to Y =
1,00 m (j:4)
Extreme global values
Hide
Plate deflection
max : -5,01E-02 m
min : -4,71E-02 m
Mx moment
max : 7,7 kN.m/ml
min : -196,4 kN.m/ml
My moment
max : 287,6 kN.m/ml
min : -111,1 kN.m/ml
Soil settlement
max : -5,01E-02 m
min : -4,71E-02 m
Soil reaction
max : 4294,4 kPa
min : 60,2 kPa
Surf. sup. react.
max : 0,0 kPa
min : 0,0 kPa
Cross sections
Cross section / Soil reaction / Y1=8,00 m (j:14) / Y2=1,00 m (j:4)
Soil reaction (kPa)
X (m)
Curve#1 to Y= 8,00 m (j:14)
Curve#2 to Y= 1,00 m (j:4)
Cross sections details
Hide
X [m]
Curve#1 [kPa]
Curve#2 [kPa]
0,00 879,0 1088,8
0,25 132,9 172,1
0,50 142,7 188,4
1,00 107,0 149,2
1,50 88,7 130,4
2,50 72,7 114,2
3,50 64,7 106,3
4,50 61,5 103,0
5,50 61,5 103,0
6,50 64,7 106,3
7,50 72,7 114,2
8,50 88,7 130,4
9,00 107,0 149,2
9,50 142,7 188,4
9,75 132,9 172,1
10,00 879,0 1088,8

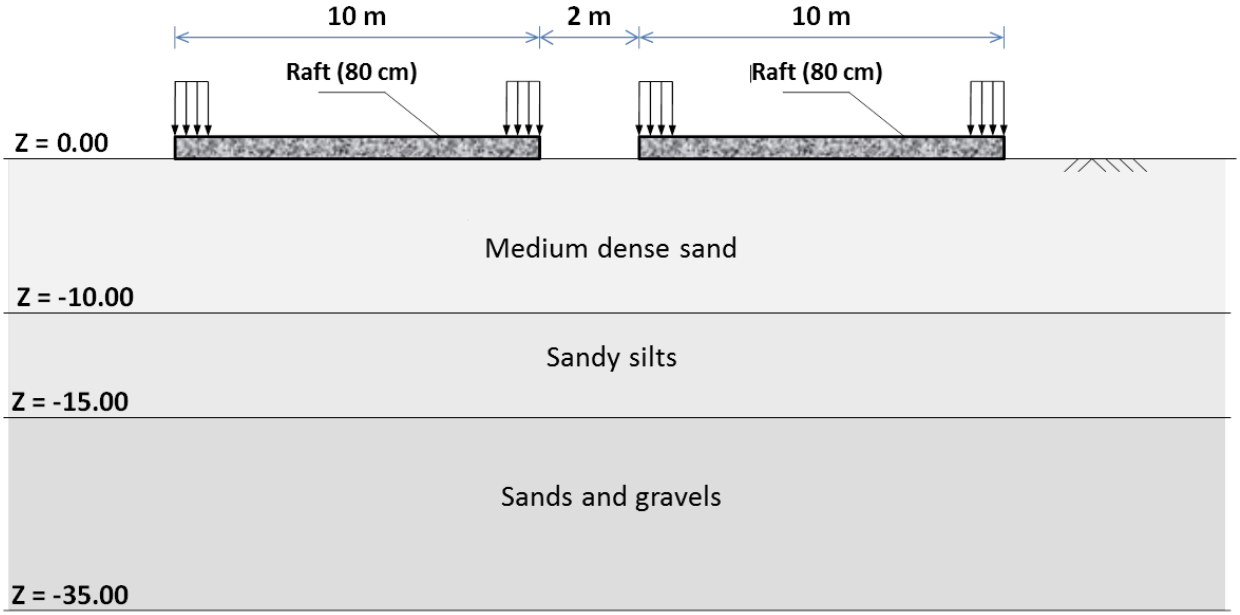
3D Graph



We observe an edge effect which results in reaction peaks at the borders and corners of the raft. This effect arises from the elasticity theory and becomes more significant when the raft becomes more rigid. For a real soil, there is a limitation of this reaction because of plastic strains: this can be managed with the help of the "automatic plastification" option by defining a limit of the plastification. This limitation can correspond, for example, to either creep stress or maximum limit stress, in function of the load combination type.

I.4.2.2. Step 2: Two identical neighbour rafts

We will take over the Single Raft file and modify the input data to study the interaction with the second raft that is identical to the first one and at a distance of 2.0 m as illustrated in the following figure.



To do that, "save as" your project with another name to start.

I.4.2.2.1. "Parameters" tab

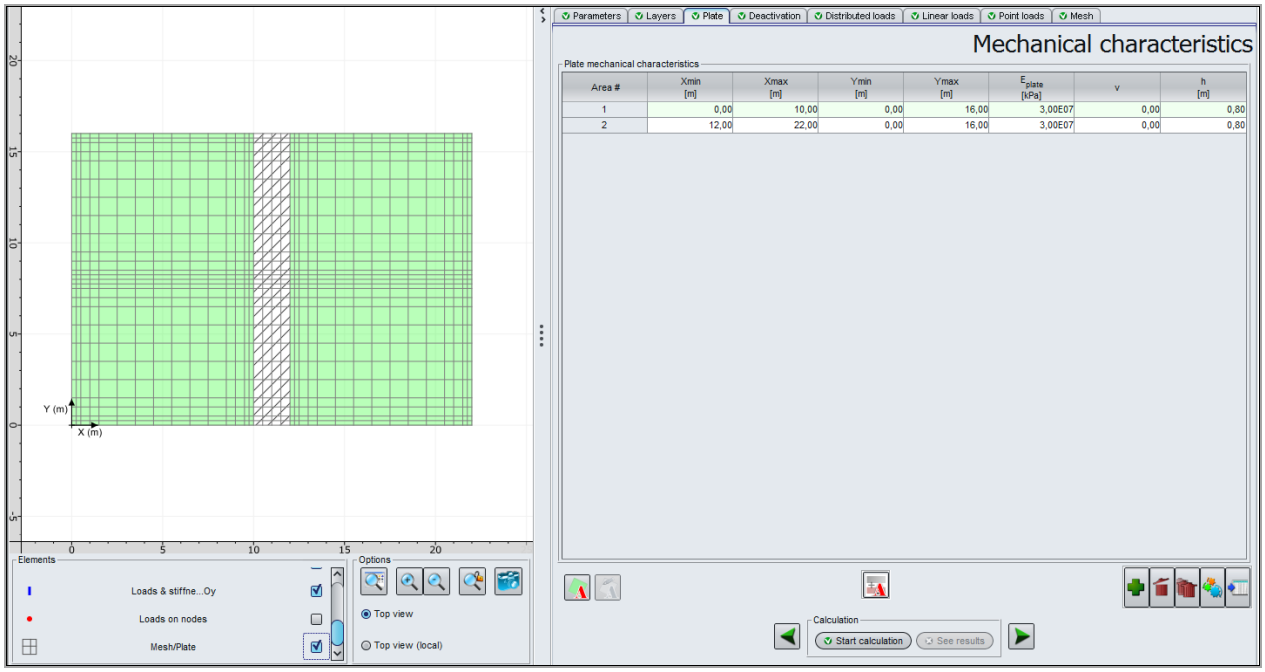
No modification.

I.4.2.2.2. "Layers" tab

No modification.

I.4.2.2.3. "Plate" tab

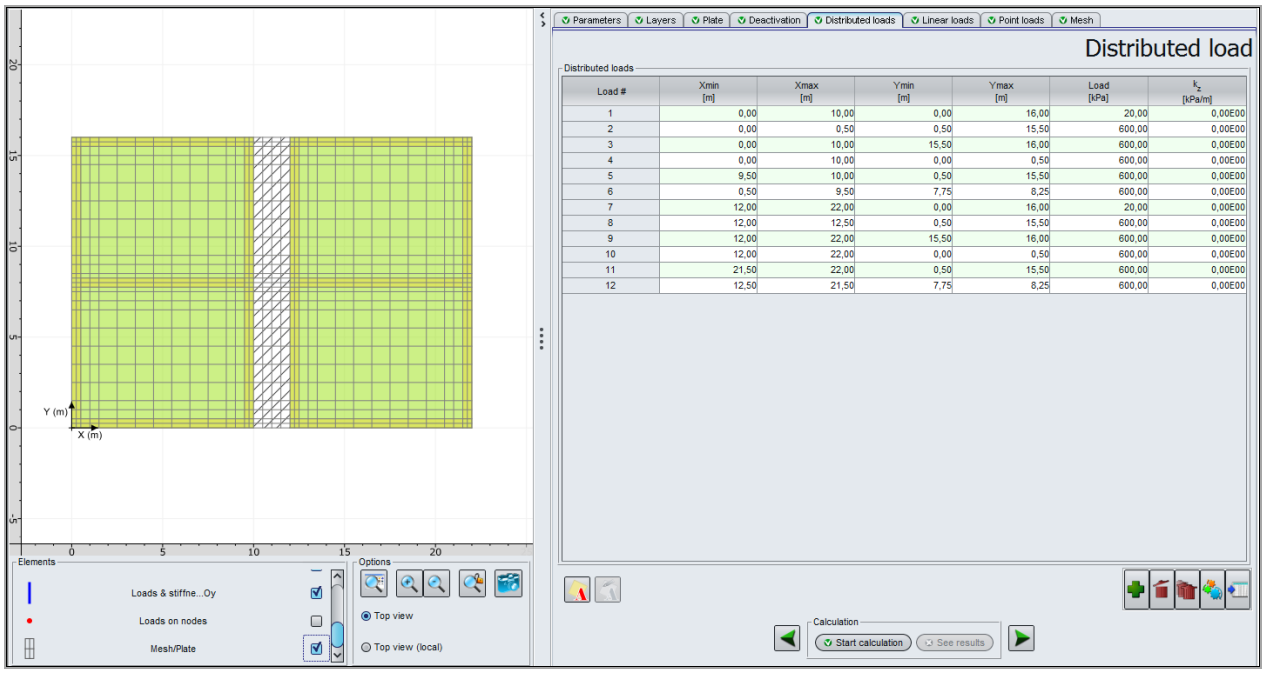
We just have to define a rectangular zone next to the one representing the first raft. The strip of 2 m left "empty" between the two raft zones is automatically recognised by the user interface as a "deactivated" zone.



Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	10.00	0.00	16.00	3.00E+07	0.00	0.80
2	12.00	22.00	0.00	16.00	3.00E+07	0.00	0.80

I.4.2.2.4. "Distributed loads" tab

The loading zones corresponding to the second raft are defined in the same way as for the first raft.



The new complete list of loads is given below:

Load n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	Load (kPa)	k_z (kPa/m)
1	0.00	**10.00**	0.00	16.00	20	0.00
2	0.00	0.50	0.50	15.50	600	0.00
3	0.00	10.00	15.50	16.00	600	0.00
4	0.00	10.00	0.00	0.50	600	0.00
5	9.50	10.00	0.50	15.50	600	0.00
6	0.50	9.50	7.75	8.25	600	0.00
7	**12.00**	**22.00**	**0.00**	**16.00**	**20**	**0.00**
8	**12.00**	**12.50**	**0.50**	**15.50**	**600**	**0.00**
9	**12.00**	**22.00**	**15.50**	**16.00**	**600**	**0.00**
10	**12.00**	**22.00**	**0.00**	**0.50**	**600**	**0.00**
11	**21.50**	**22.00**	**0.50**	**15.50**	**600**	**0.00**
12	**12.50**	**21.50**	**7.75**	**8.25**	**600**	**0.00**

I.4.2.2.5. "Linear loads" tab

No modification.

I.4.2.2.6. "Point loads" tab

No modification.

I.4.2.2.7. "Mesh" tab

Same principle as the first case of the single raft. We propose to apply a refinement with a ratio of 2 in the zone located between the 2 rafts. This will allow to increase the number of calculation points in the interaction zone. We remind that Tasplaq calculates systematically the values of soil settlement at all points of the mesh, including the deactivated zones.

I.4.2.2.8. Calculations and Results

Run the calculation by clicking on the Start calculation button. When the calculation process is over, close the window "Calculation running" then click on the See results button.

Soil settlement

Extreme global values:

- Max: -7.02E-02 m (thus -7.0 cm, thus 2 cm more compared to the single raft case)
- Min: -4.80E-02 m (thus -4.8 cm)

Cross-section along X:

Cross-section 1 at Y = 8.00 m (j: 14): at X = 11 m Soil settlement = -6.17 cm

Cross-section 2 at Y = 2.50 m (j: 6): at X = 11 m Soil settlement = -5.82 cm

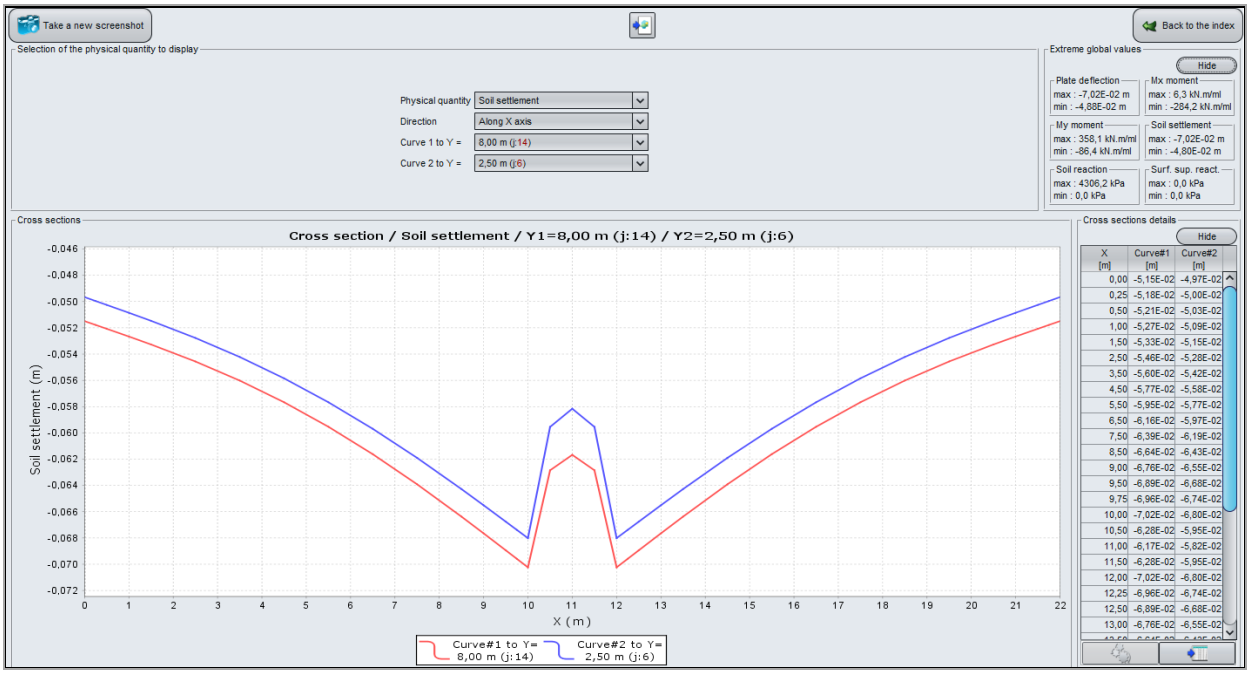


Plate deflection

Extreme global values:

- Max: -7.02E-02 m (thus -7.0 cm, value identical to the soil settlement = no unsticking)
- Min: -4.88E-02 m (thus -4.9 cm)

Cross-section along X:

Cross-section 1 at Y = 8.00 m (j: 14): at X = 8 m Deflection = -6.51 cm

Cross-section 2 at Y = 2.50 m (j: 6): at X = 8 m Deflection = -6.31 cm

Mx Moment:

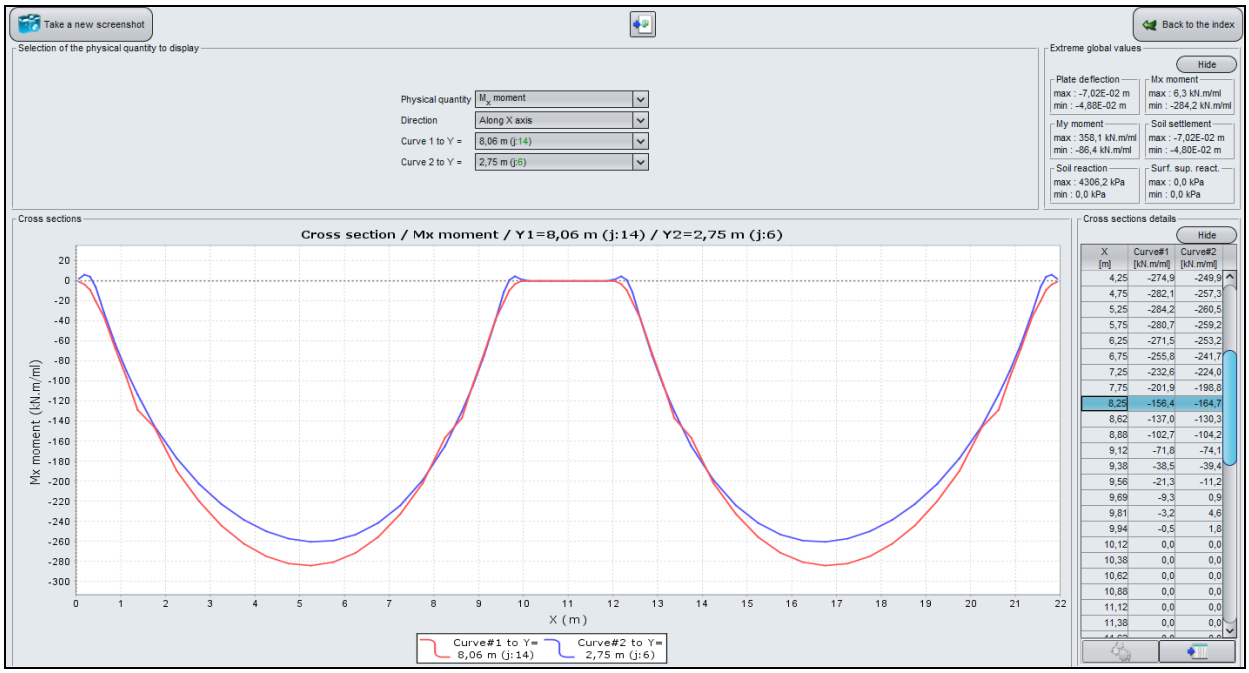
Extreme global values:

- Max: 6 kN.m/ml
- Min: -284 kN.m/ml

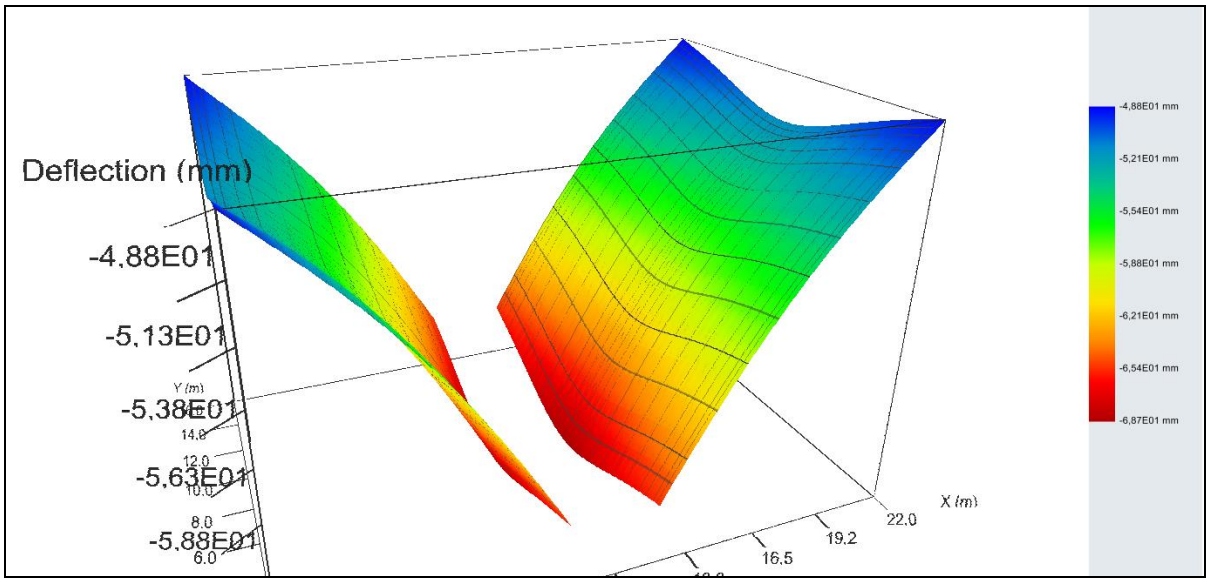
Cross-section graph along X axis with:

Cross-section 1 at Y = 8.06 m (j: 14): at X = 8.25 m M_x = -156 kN.m/ml.

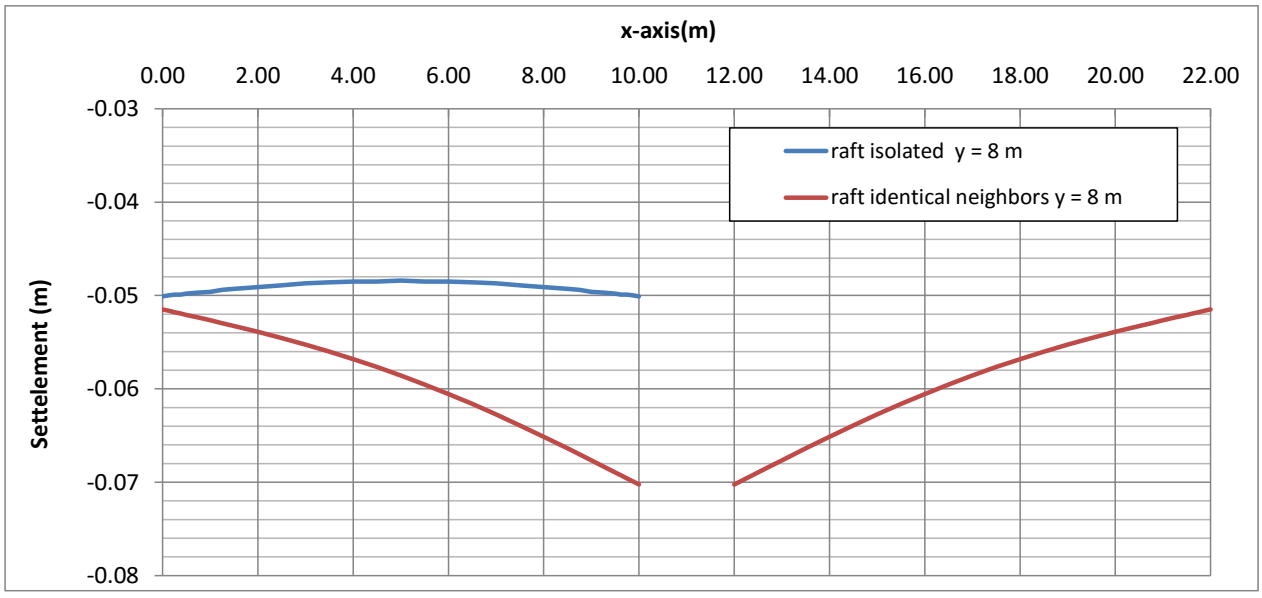
Cross-section 2 at Y = 2.75 m (j: 6): at X = 8.25 m M_x = -165 kN.m/ml.



3D Graph (plate deflection)



The following figure shows the comparison between the settlements obtained in both situations (with and without interaction), and it illustrates a significant amplification of the settlements as a result of the interaction effect between both rafts. The simulation of this phenomenon would not be possible if the soil had been assimilated to a set of juxtaposed springs.

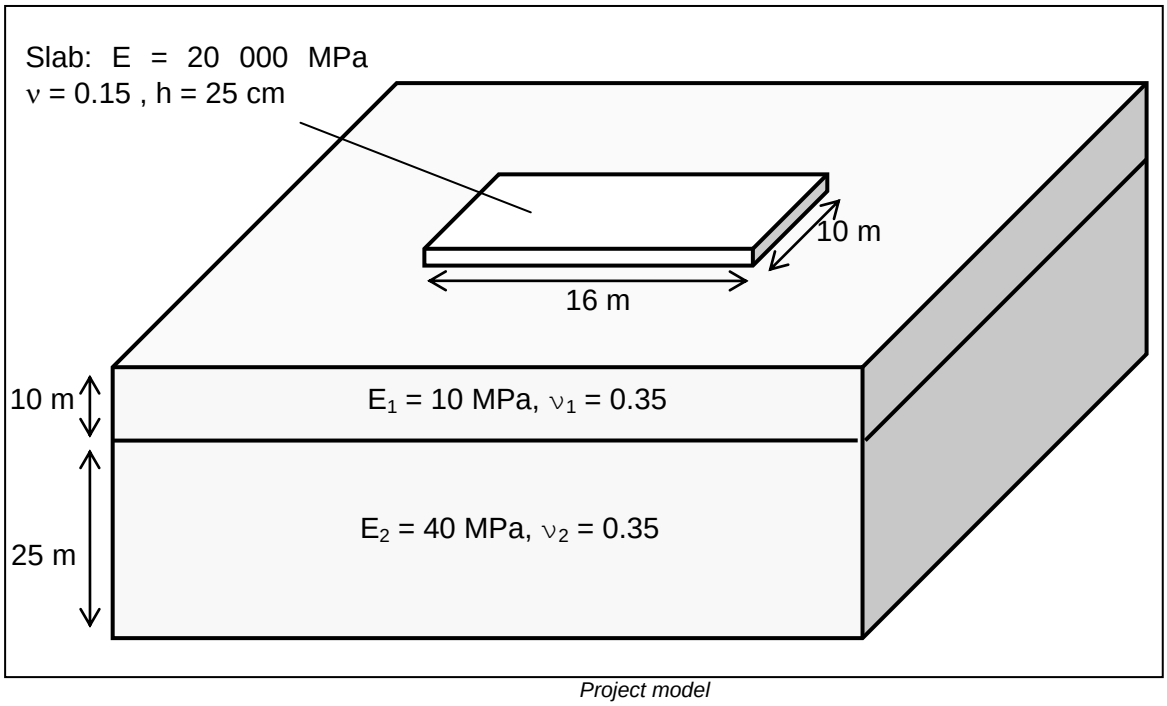


I.4.3. Example 3: Slab with different systems of joints

A 16 m x 10 m rectangular slab is subjected to a load of 50 kPa distributed over a 10 m x 5 m zone at its center in addition to its dead load.

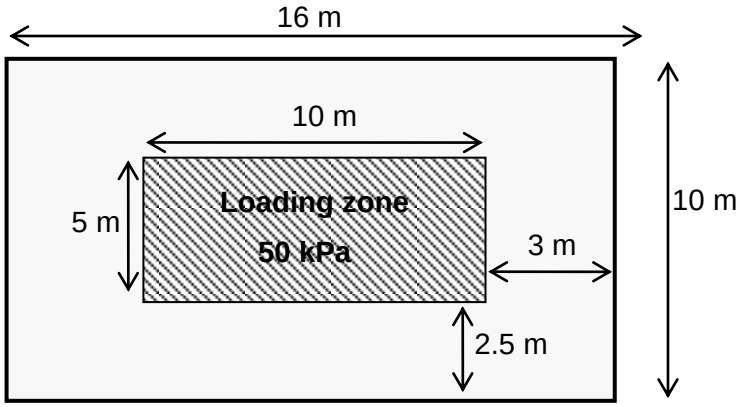
We will consider this example in 3 successive different sitiuations and compare them:

- Step 1: slab supposed to be continuous;
- Step 2: slab with partial joints;
- Step 3: slab with full joints.

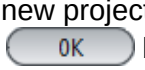


Project model

The following diagram illustrates the localisation of the central load (50 kPa) applied on the slab:



I.4.3.1. Step 1: Continuous slab laying on multi-layered elastic soil

- To create a new project, click on the radio-button
- Click on the  button.

If Foxta is already open, click on "File", then "New project ".

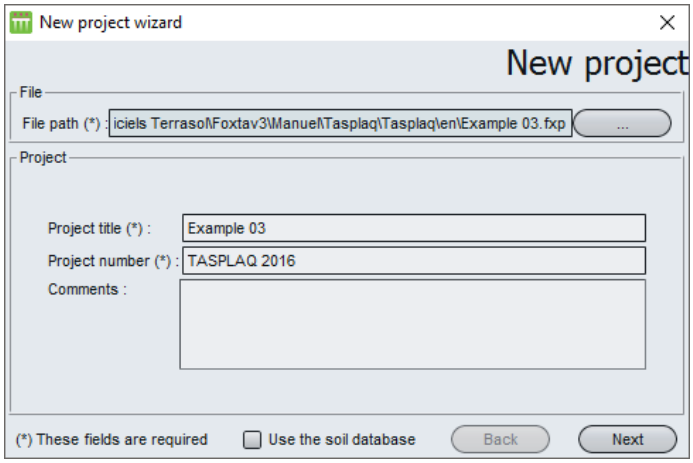
I.4.3.1.1. New project wizard

"File" frame:

- Click on the ... button to choose the work directory;
- Name and save the file.

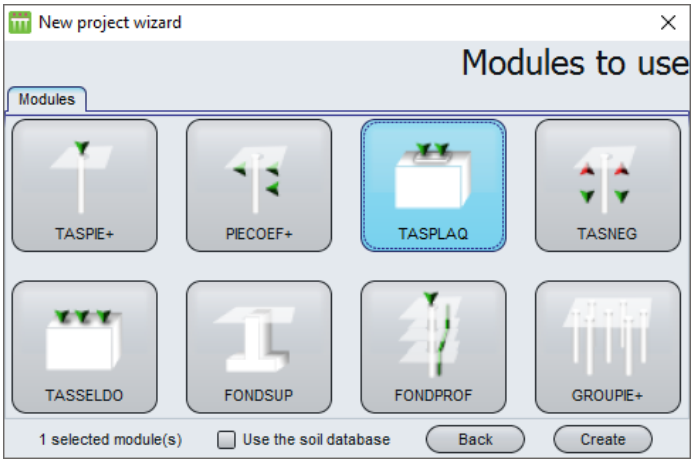
"Project" frame:

- Define a title for the project;
- Define a project number;
- Complete comments if necessary;
- Leave the " Use the soil database" box unchecked (for this example we will not use the database), and click on the Next button.



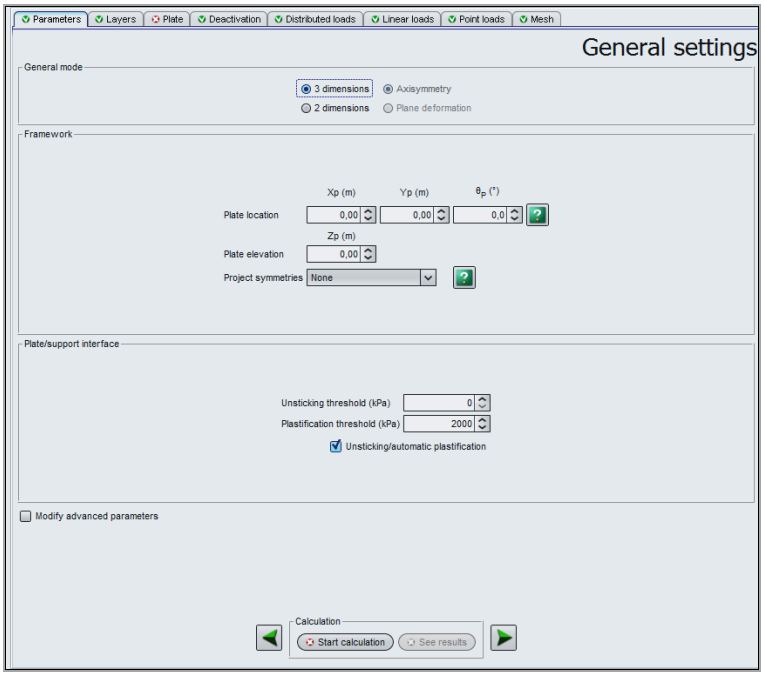
I.4.3.1.2. New project wizard: Module selection

In the "Modules to use" window, select the TASPLAQ module and click on the Create button.



The input data window of TASPLAQ module is now displayed.
All different data tabs should be completed in the right zone of the screen.

I.4.3.1.3. "Parameters" tab



"General mode" frame:
The model is three dimensional, click on the radio button "3 dimensions".

"Framework" frame:
In this example, the local coordinates system of the plate representing the raft is taken identical to the global one. The (0.0) point represents the "lower-left" corner of the model.

X_p (m)	Y_p (m)	θ_p (°)
0.00	0.00	0.00

- Plate elevation: Zp = 0.0 m (reference elevation = elevation of the underside of the plate).
- Project symmetries: None

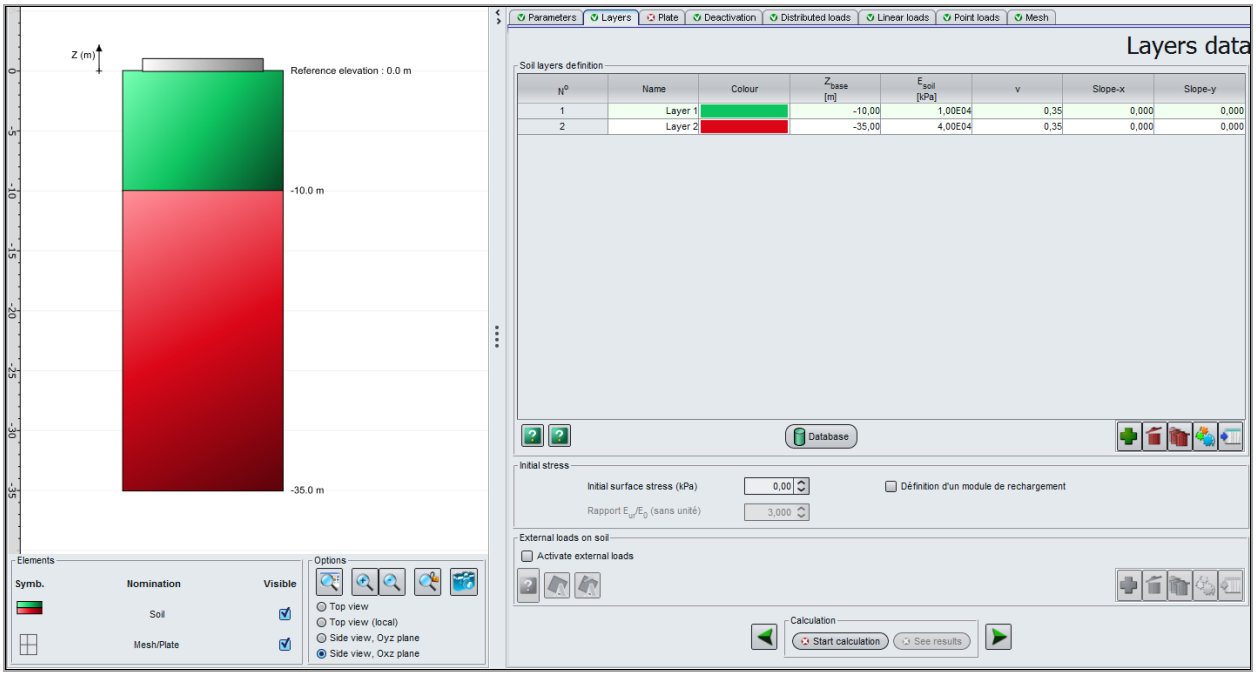
"Plate/support interface" frame:
This frame allows the activation of the automatic management of the unsticking and the plastification criteria by using the criteria defined by the user.

Unsticking threshold	0 kPa (no traction is authorised)
Plastification threshold	2000 kPa

- Automatic unsticking/plastification: Checked
- Modify advanced parameters: Unchecked

To shift to the next tab, click either on the name of the tab "Layers", or on the button.

I.4.3.1.4. "Layers" tab



For this example we have 2 soil layers to define. The values to input are the following:

Name	**Z_{base}** (m)	**E_{sol}** (kPa)	**ν**	**Slope-x**	**Slope-y**
Layer1	-10	1.00E04	0.35	0.00	0.00
Layer 2	-35	4.00E04	0.35	0.00	0.00

Initial surface stress: 0 kPa.

"External loads on soil" frame:

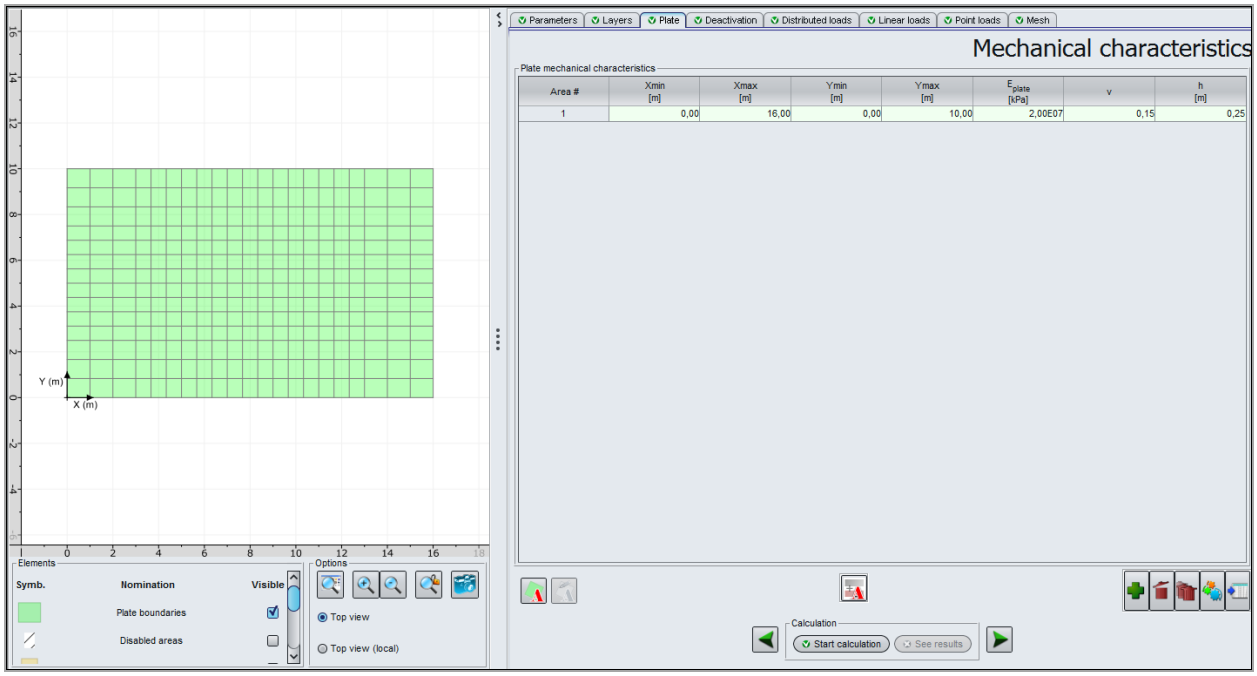
- No external load is to be defined, leave unchecked the "Activate external loads" checkbox.

I.4.3.1.5. "Plate" tab

In this example the plate is homogeneous. One single zone is to be defined: Xmin = 0.00, Xmax = 16.00 m, Ymin = 0.00 and Ymax = 10.00 m.

"Plate mechanical characteristics" frame:

Zone n°	**Xmin (m)**	**Xmax (m)**	**Ymin (m)**	**Ymax (m)**	**E_{plate} (kPa)**	**ν**	**h (m)**
1	0.00	16.00	0.00	10.00	2.00E+07	0.15	0.25



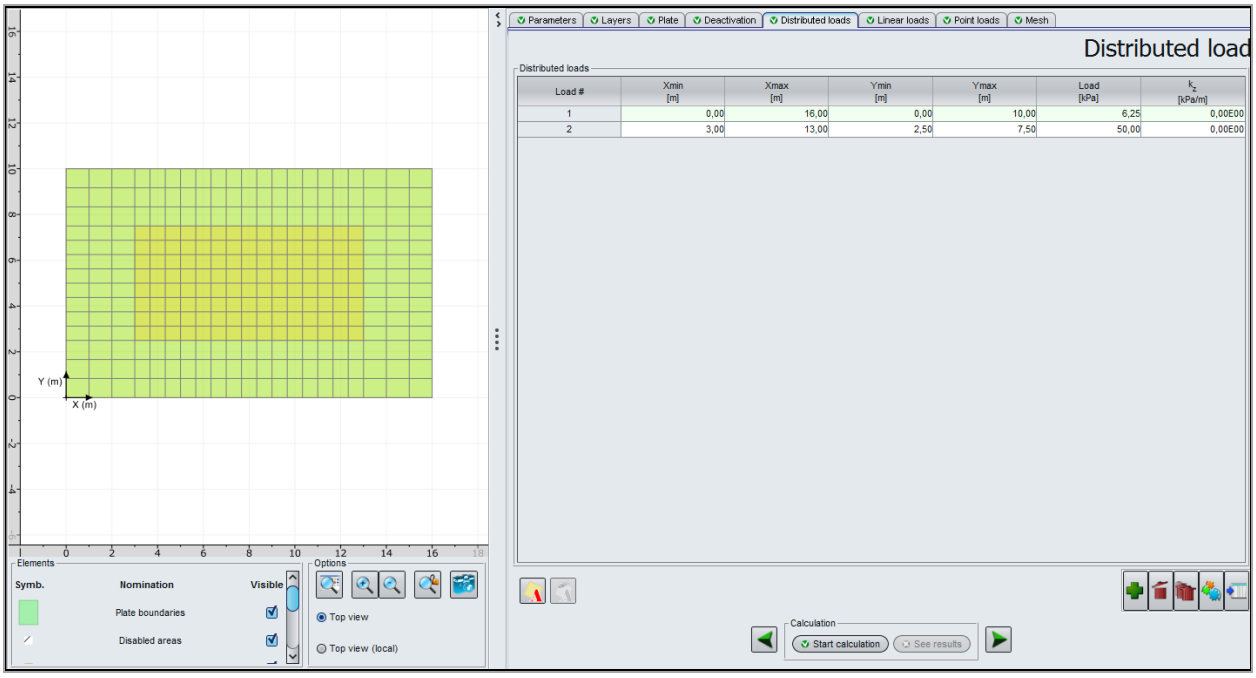
I.4.3.1.6. "Deactivation" tab

In this example, there is no element to deactivate, the zone representing the plate is homogeneous and continuous. The "Deactivation" frame is left empty.

I.4.3.1.7. "Distributed loads" tab

We have to define, for this example, 2 load zones:

- In one hand, the dead load of the slab itself. This corresponds to a distributed load of 6.25 kPa to be applied on the entire surface of the plate,
- In the other hand, the 50 kPa load applied in the central zone of the plate (zone defined from Xmin = 3.00 to Xmax = 13.00 m and from Ymin = 2.50 to Ymax = 7.50 m).



N°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	Load (kPa)	k_z (kPa/m)
1	0.00	16.00	0.00	10.00	6.25	0
2	3.00	13.00	2.50	7.5	50.00	0

I.4.3.1.8. "Linear loads" tab

There is no linear load in this example.

I.4.3.1.9. "Point loads" tab

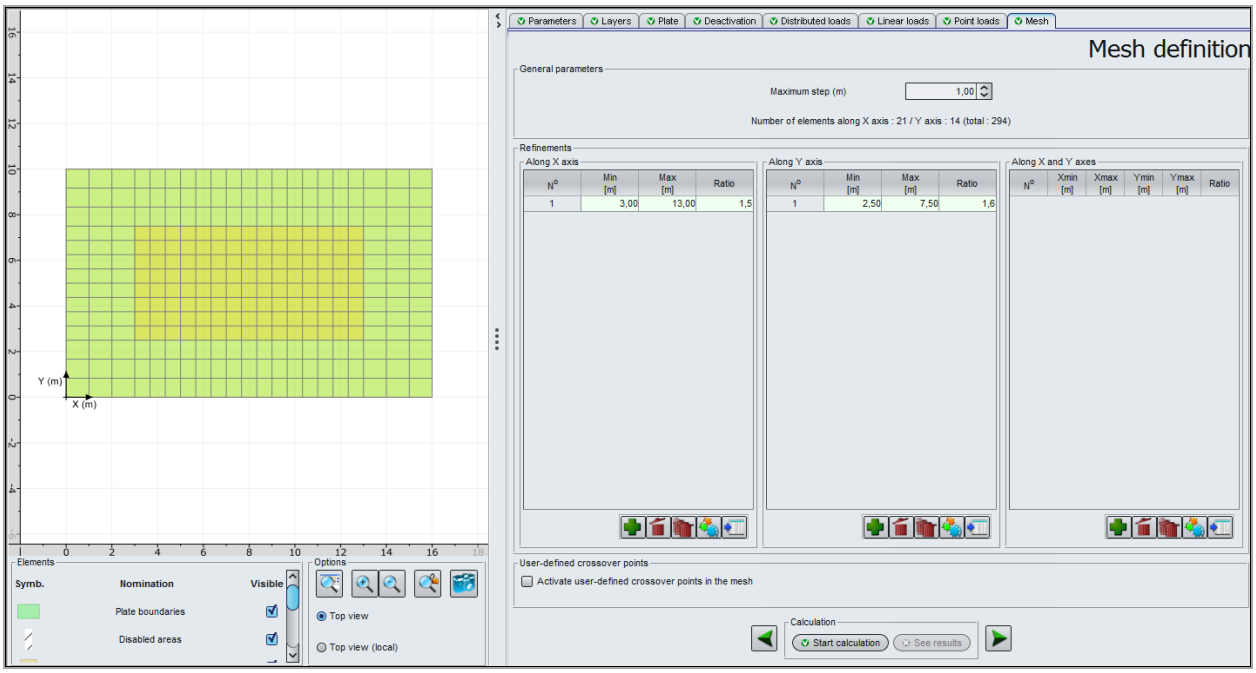
There is no point load in this example.

I.4.3.1.10. "Mesh" tab

A maximum step of 1 meter is to be defined.

In order to obtain more precision in the central zone, we propose a refinement in the footprint of the 50 kPa load, with:

- a ratio of 1.5 along X axis with a Min = 3.00 and a Max = 13.00 m and
- a ratio of 1.6 along Y axis with a Min = 2.50 and a Max = 7.50 m.



The input data are the following:

- Maximum step = 1 (m)

Refinements:

	Min (m)	Max (m)	Ratio
On the X axis	3.00	13.00	1.50
On the Y axis	2.50	7.50	1.60

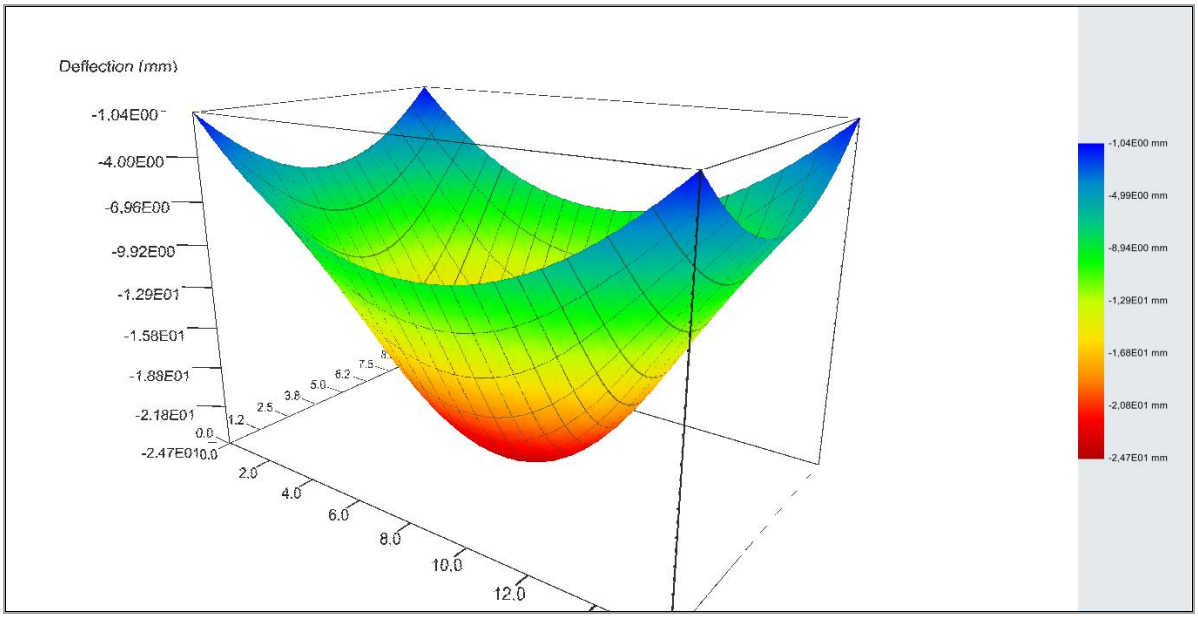
I.4.3.1.11. Calculations and Results

Click on the Start calculation button.

To display the results formatted as tables and graphical views, click on the See results button.

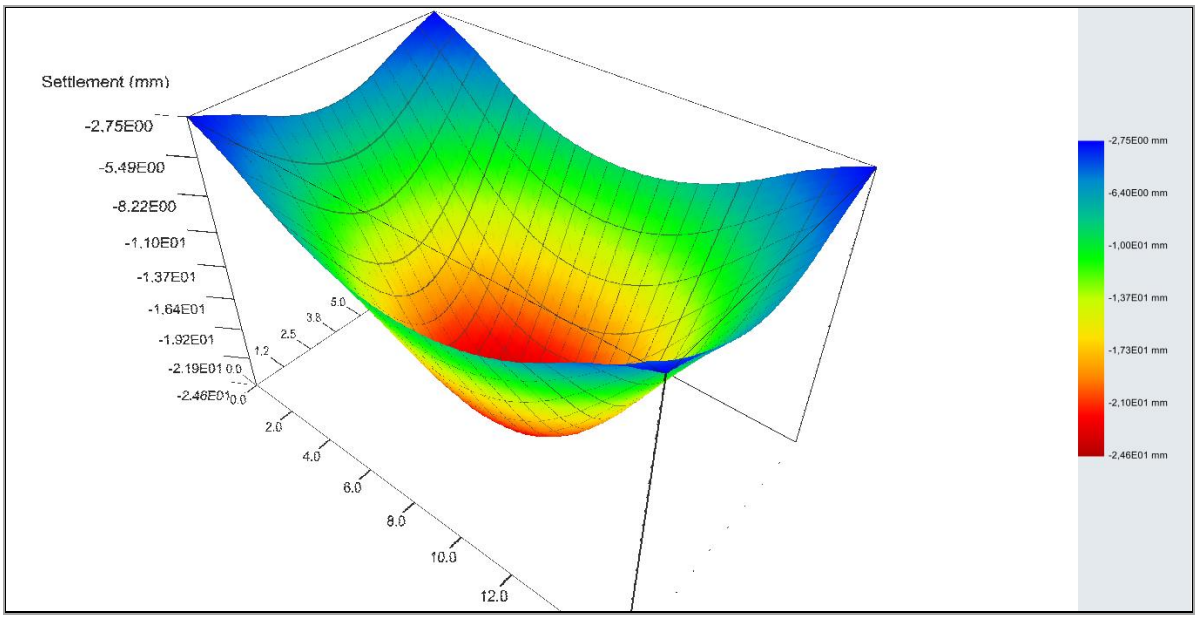
3D Graphics

Plate deflection:



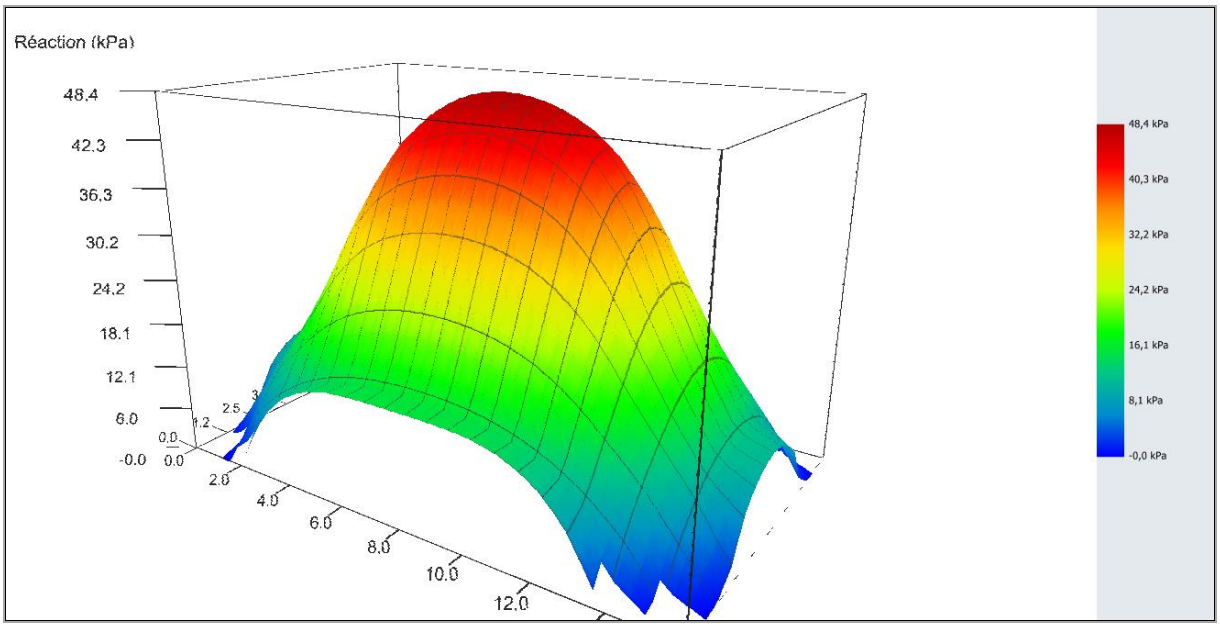
The maximum deflection is about 2.5 cm and it is obtained at the centre of the slab.

Soil settlement:



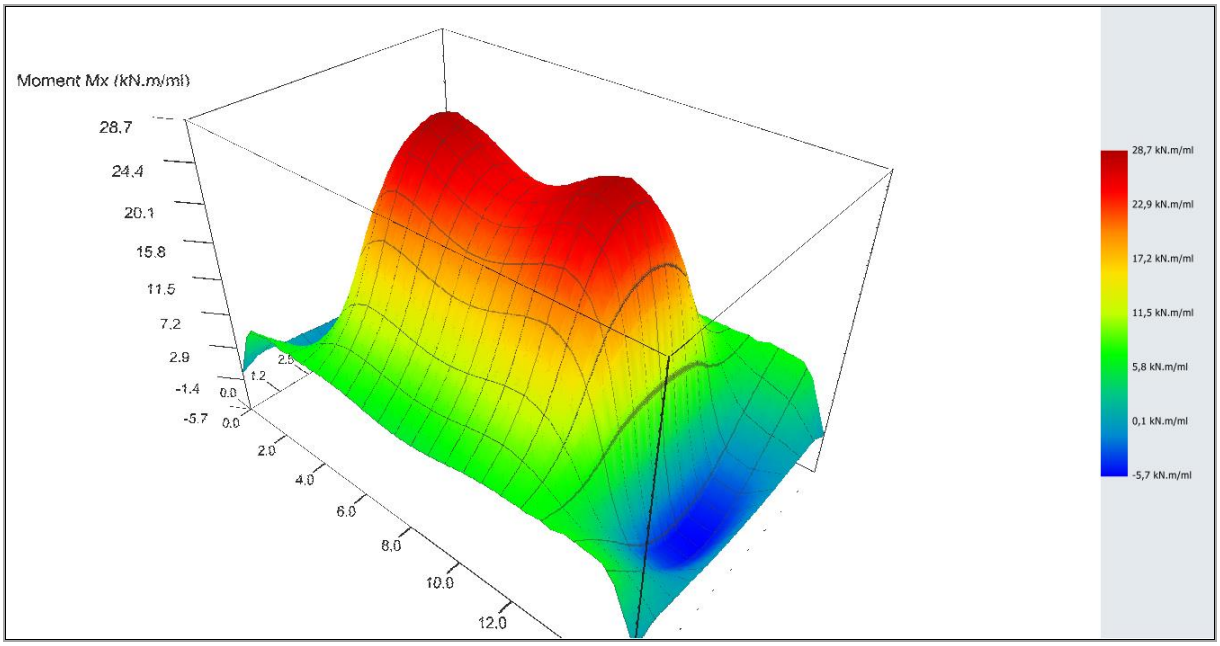
The maximum soil settlement is equal to the maximum deflection and it is obtained at the centre of the slab too.

Soil reaction:

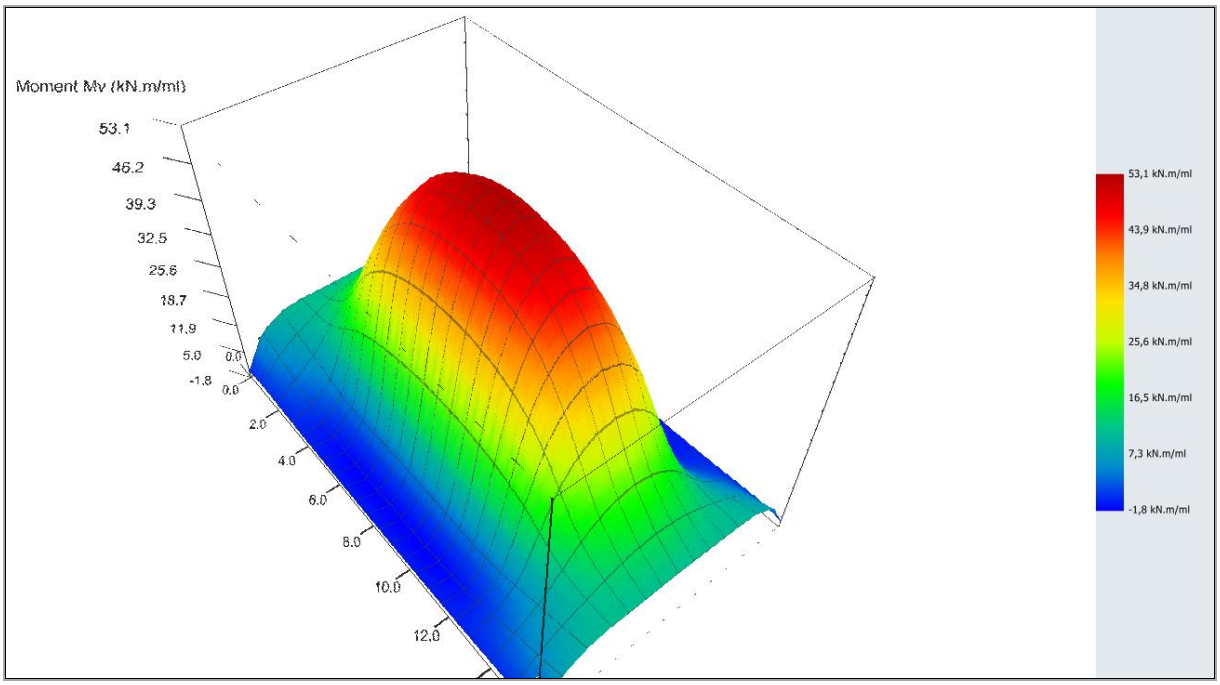


We observe some zones of "zero reaction" appearing at the corners of the slab. This is due to the fact that the calculation resulted in an unsticking phenomenon which also explains the difference that we could obtain between the of soil settlement and the plate deflection profiles.

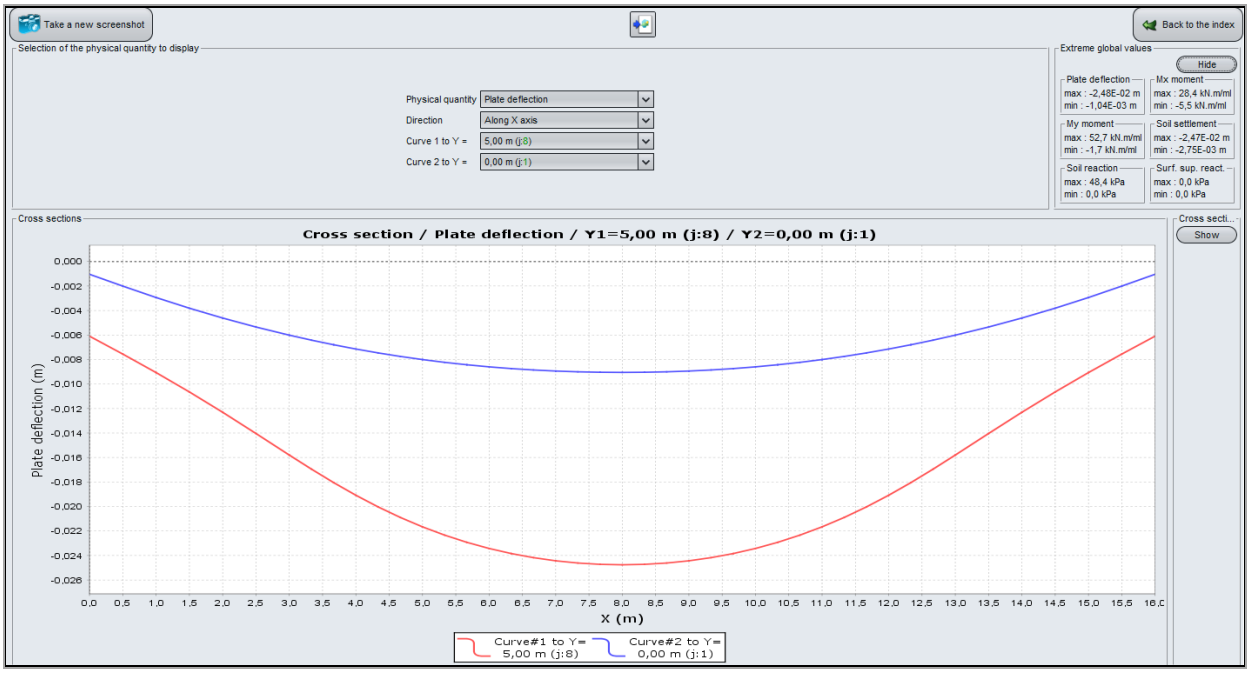
Mx Moment:

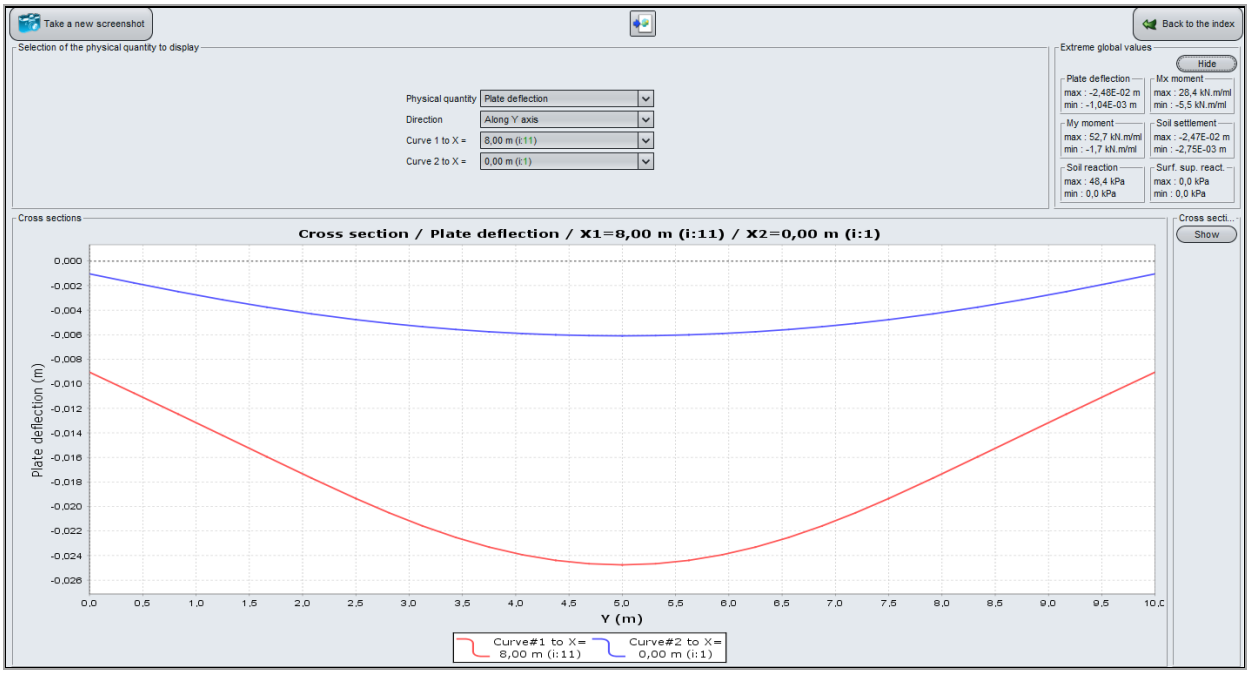


My Moment:



Cross-sections:





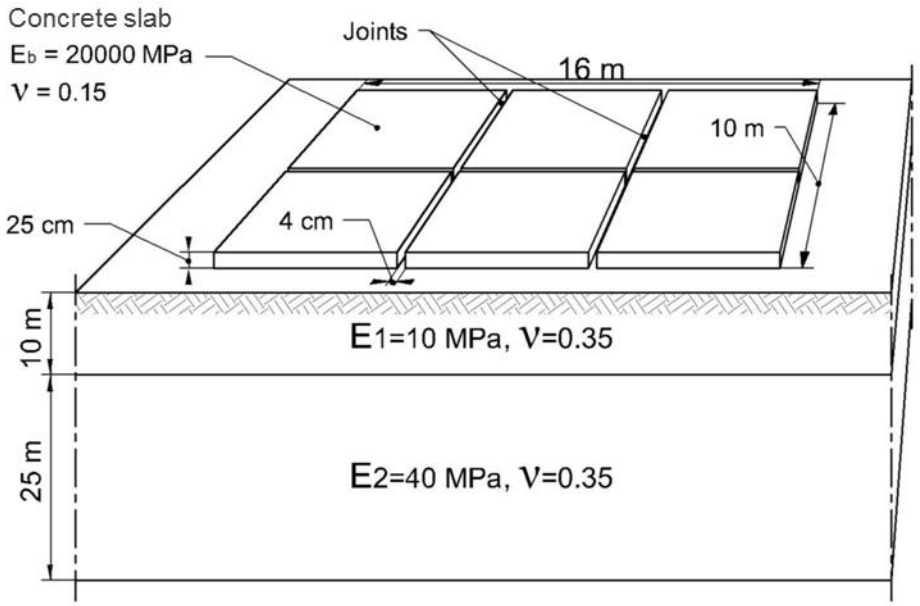
The 2 cross-sections above show the amplitude of the soil settlement along X and Y axis respectively, at the centre and at the edge of the plate in each cases.

Let's keep in mind the following bending moments:

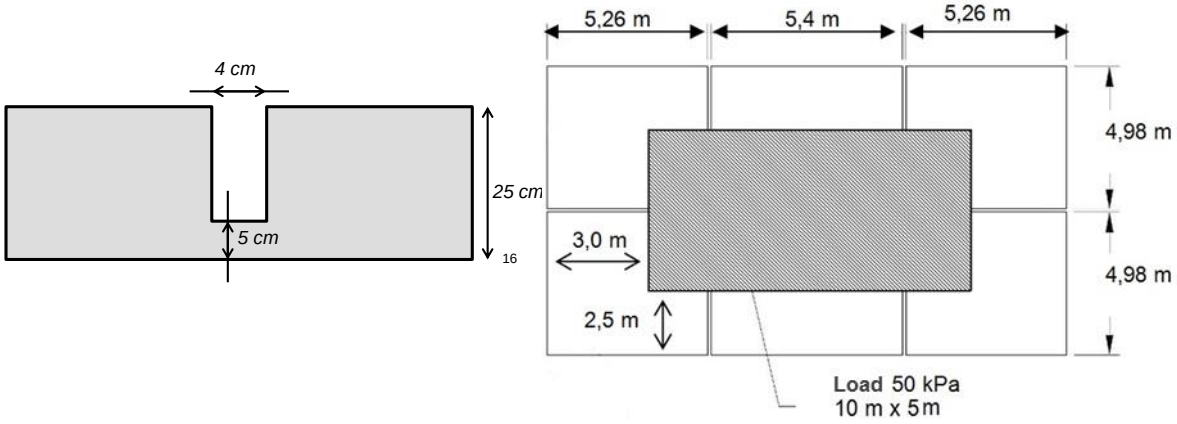
- $M_{x,max}$ = 28 kN.m/ml
- $M_{y,max}$ = 53 kN.m/ml

I.4.3.2. Step 2: Slab with partial joints laying on a multi-layered elastic soil

We take over the case of the step 1. We now consider the presence of the partial joints in the slab as shown in the figure below:



We suppose that the partial joints are obtained by sawing the slab on 4/5th of its thickness. At the place of these joints, the thickness of the slab is thus 5 cm.
The joints are assumed to be 4 cm large, their spacing is illustrated in the figure below.



To modify the input, click on the Back to the data button.

Then "Save as" the project with another name before starting this new step of the example.

We give here the details of the data modified compared to step 1 to take into account the joints.

I.4.3.2.1. "Plate" tab

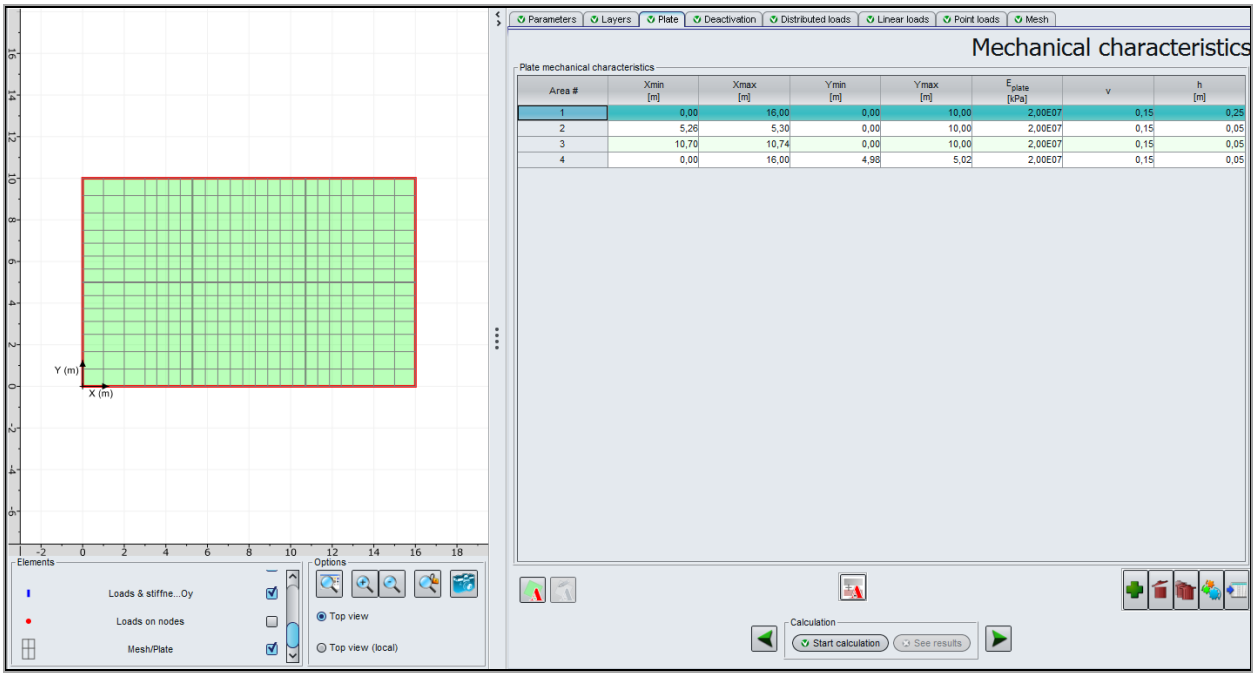
In this step, the plate is not homogeneous anymore (it is thinner at the place of the joints). IWe have to assign a thickness of 5 cm to the zones representing the joints.

Several zones have to be defined to allow the assignment of the different properties to the plate.

Note: if an element belongs to more than one zone, TASPLAQ assigns to this element the mechanical characteristics of the lastly defined zone, which explains the order of the operations hereafter.

We will proceed as follows:

- Definition of a first zone covering the whole plate, to which we assign the mechanical characteristics of the body of the slab (E = 20 000 MPa, ν = 0.15 and h = 0.25 m),
- Then, definition of 3 zones representing the partial joints, to which we assign the mechanical characteristics of the partial joints (E = 20 000 MPa, ν = 0.15 and h = 0.05 m).

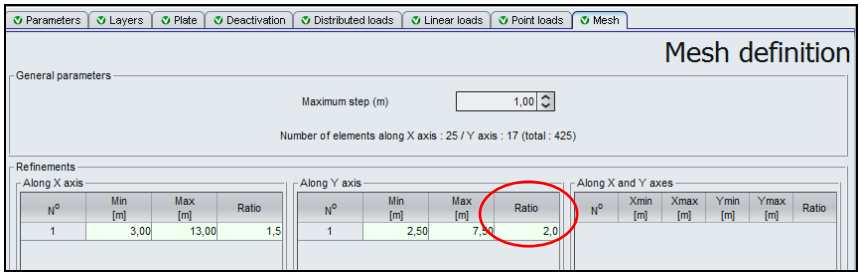


The input data are the following:

Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	16.00	0.00	10.00	2.00E07	0.15	0.25
2	5.26	5.30	0.00	10.00	2.00E07	0.15	0.05
3	10.70	10.74	0.00	10.00	2.00E07	0.15	0.05
4	0.00	16.00	4.98	5.02	2.00E07	0.15	0.05

I.4.3.2.2. "Mesh" tab

We propose a slight refinement of the mesh along Oy: ratio = 2 (previous value was 1.6) at the place of the 50 kPa load zone.



I.4.3.2.3. Calculations and Results

Run the calculations again by clicking on the Start calculation button.

Cross-section charts

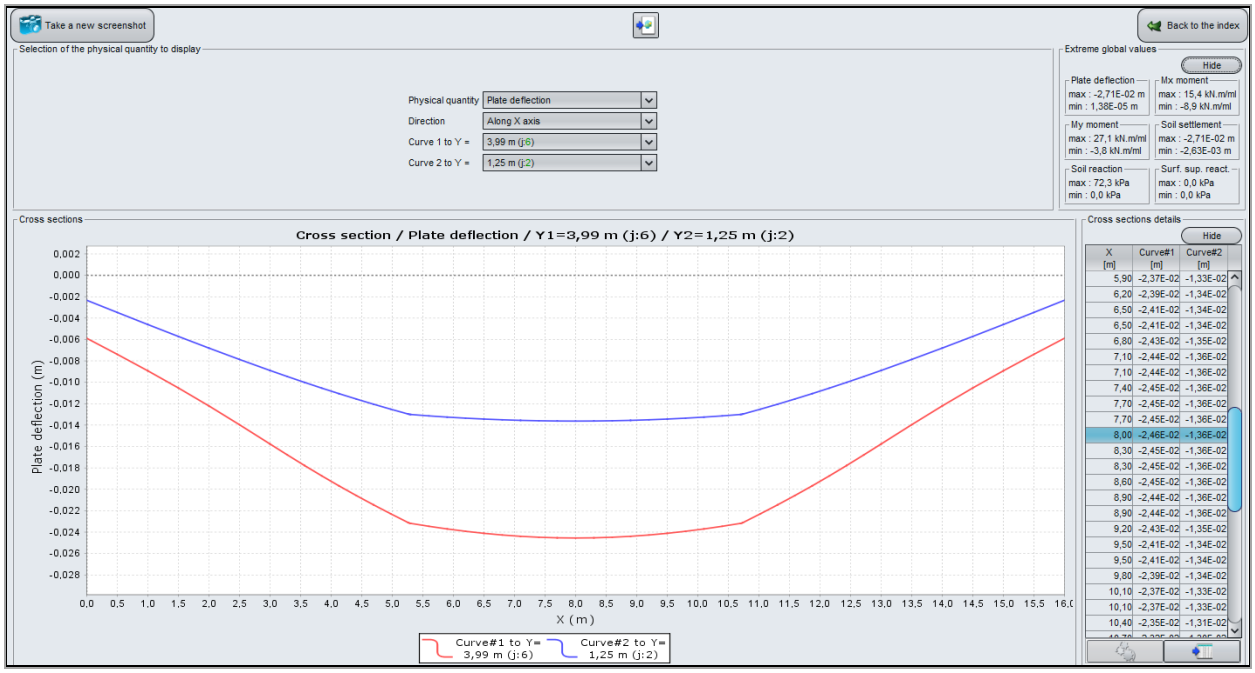
Plate deflection:

Extreme global values:

- Max: -2.71E-02 m (thus -2.7 cm)
- Min: 1.38E-05 m (thus 0.0 cm)

Cross-section along X:

Cross-section 1 at Y = 3.99 m (j: 6): at X = 8 m Deflection = -2.46 cm.
Cross-section 2 at Y = 1.25 m (j: 2): at X = 8 m Deflection = -1.36 cm.



We can observe a "hinged" behaviour at the places of the joints, which is reflected by a discontinuity of the rotations, and a general attenuation of the forces in the slab (see below).

Mx Moment Moment along X axis:

Extreme global values:

- Max: 15 kN.m/ml
- Min: -9 kN.m/ml

Cross-section along X:

Cross-section 1 at Y = 4.11 m (j: 7): at X = 8.15 m M_x = 15 kN.m/ml.
Cross-section 2 at Y = 3.12 m (j: 5): at X = 8.15 m M_x = 12 kN.m/ml.

My Moment along Y:

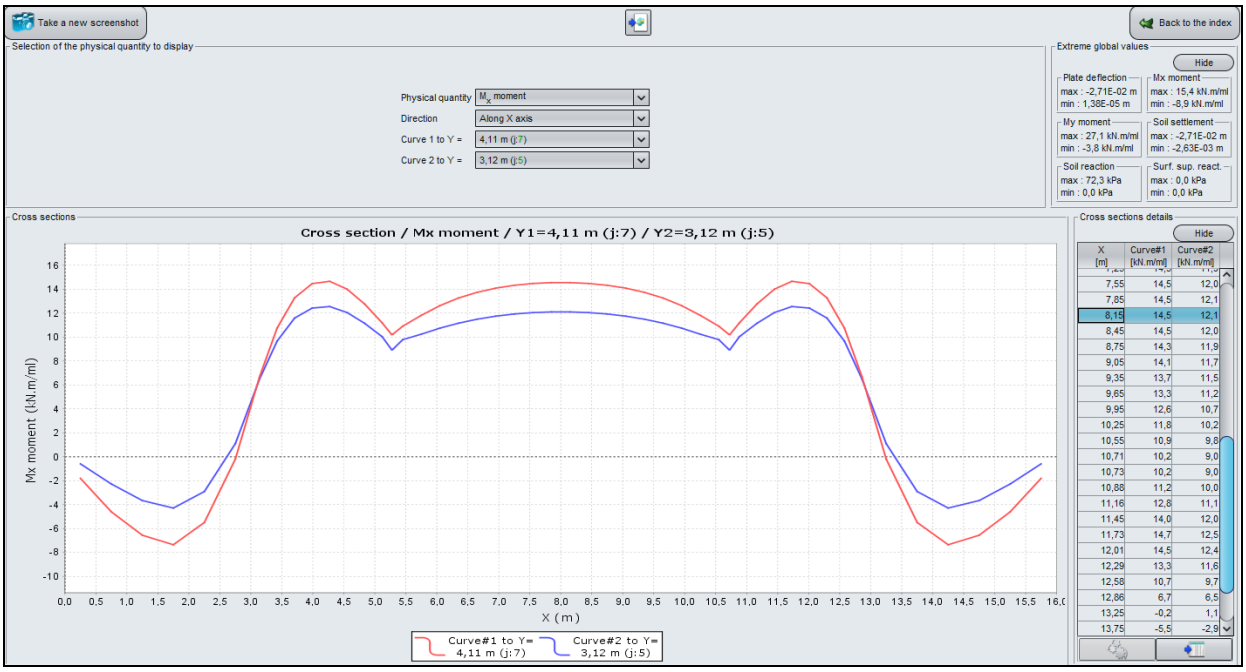
Extreme global values:

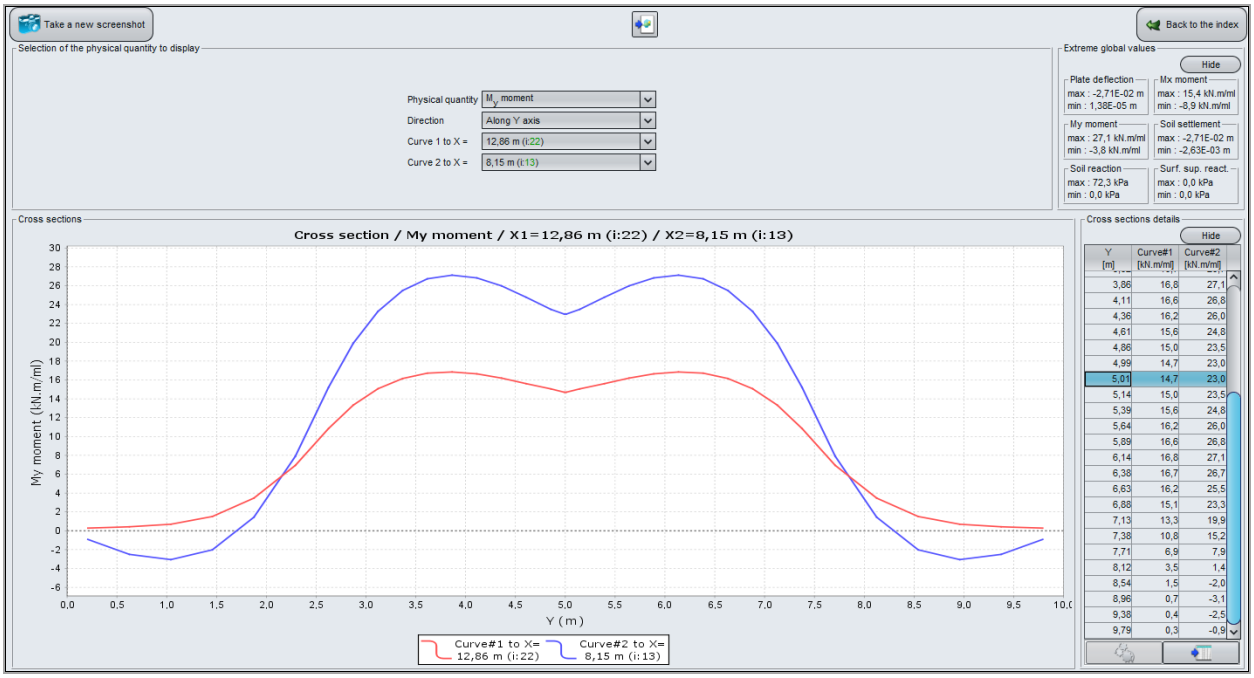
- Max: 27 kN.m/ml
- Min: -4 kN.m/ml

Cross-section along Y:

Cross-section 1 at X = 12.86 m (i: 22): at Y = 5.01 m M_y = 15 kN.m/ml.

Cross-section 2 at X = 8.15 m (i: 13): at Y = 5.01 m M_y = 23 kN.m/ml.





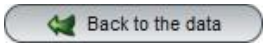
We note the amplitude of the bending moments obtained in this example:

- $M_{x,\,max}$ = 15 kN.m/ml < 28 kN.m/ml for the case of the continuous slab (step 1);
- $M_{y,\,max}$ = 27 kN.m/ml < 53 kN.m/ml for the case of the continuous slab (step 1).

I.4.3.3. Step 3: Slab with full joints laying on a multi-layered elastic soil

We take over the case of the previous step 2. We suppose now that the joints are full (slab sawed on its full thickness).

The joints are assumed to be 4 cm large, their layout is identical to the one in step 2.

To modify the input data, click on the 

button.

Then "Save as" the project with another name before starting.

Again, we give hereafter the details of the modified data only (compared to step 2). The layout of the joints didn't change; we can keep the mesh definition of the previous step.

I.4.3.3.1. "Plate" tab

At this stage, we can go back to a homogeneous definition of the plate for the entire mesh, without making any distinction between joints and plate. Indeed, TASPLAQ assigns the plate characteristics to activated zones only and consequently it excludes automatically deactivated zones (i.e. joints here).

Thus, we only have to keep the first zone defined in step 2, and delete the 3 others corresponding to the joints.

Note: it would be possible also to keep the 4 lines defined in step 2 (including the joints), because the joint zones are deactivated and the 3 last lines would not be taken into account in this case.

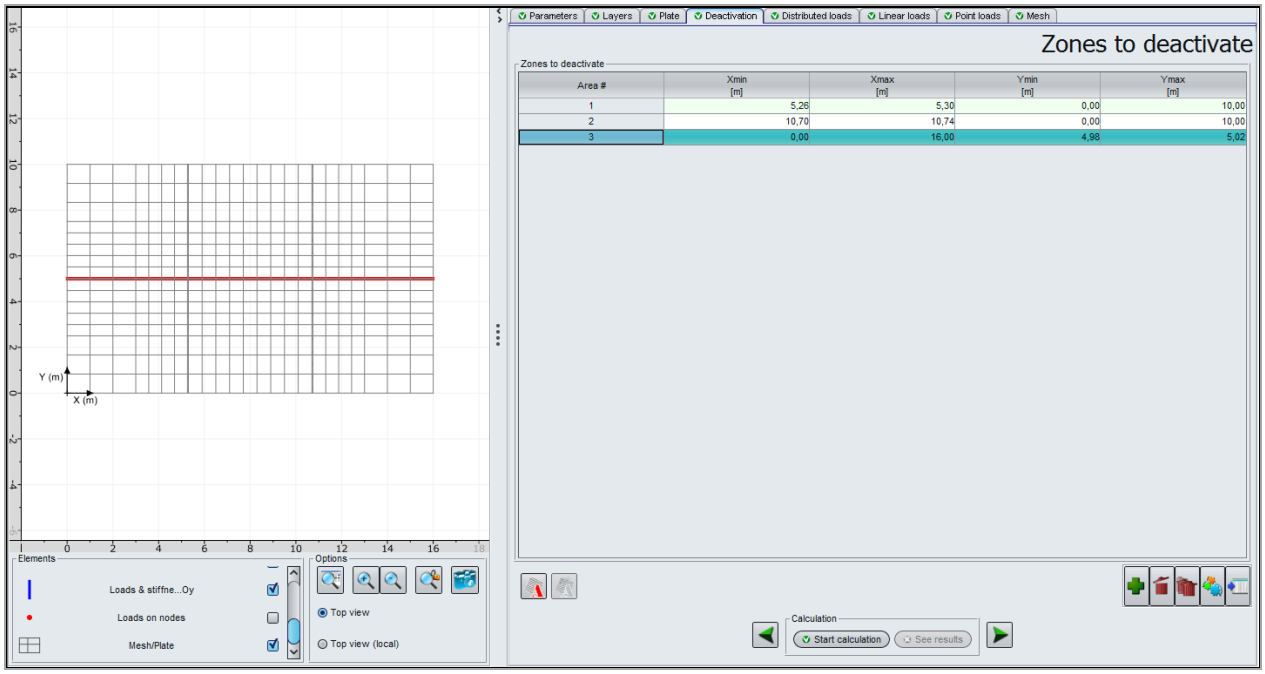
Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	16.00	0.00	10.00	2.00E+07	0.15	0.25

I.4.3.3.2. "Deactivation" tab

Now the zones corresponding to the full joints shall be deactivated.
The aim in this example is to define six divided plates separated by the joints.

To deactivate, the zone corresponding to the first joint for example, define a zone from Xmin = 5.26 to Xmax = 5.30 m and from Ymin = 0.00 to Ymax = 10.00 m.

Proceed in the same way for the 2 other joints.



The input data are the following:

Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)
1	5.26	5.30	0.00	10.00
2	10.70	10.74	0.00	10.00
3	0.00	16.00	4.98	5.02

I.4.3.3.3. Calculations and results

Run the calculation again by clicking on the Start calculation button.

Cross-section charts

To illustrate the results, in terms of the influence of the joints particularly, we can create cross-sections within the plate.

Plate deflection:

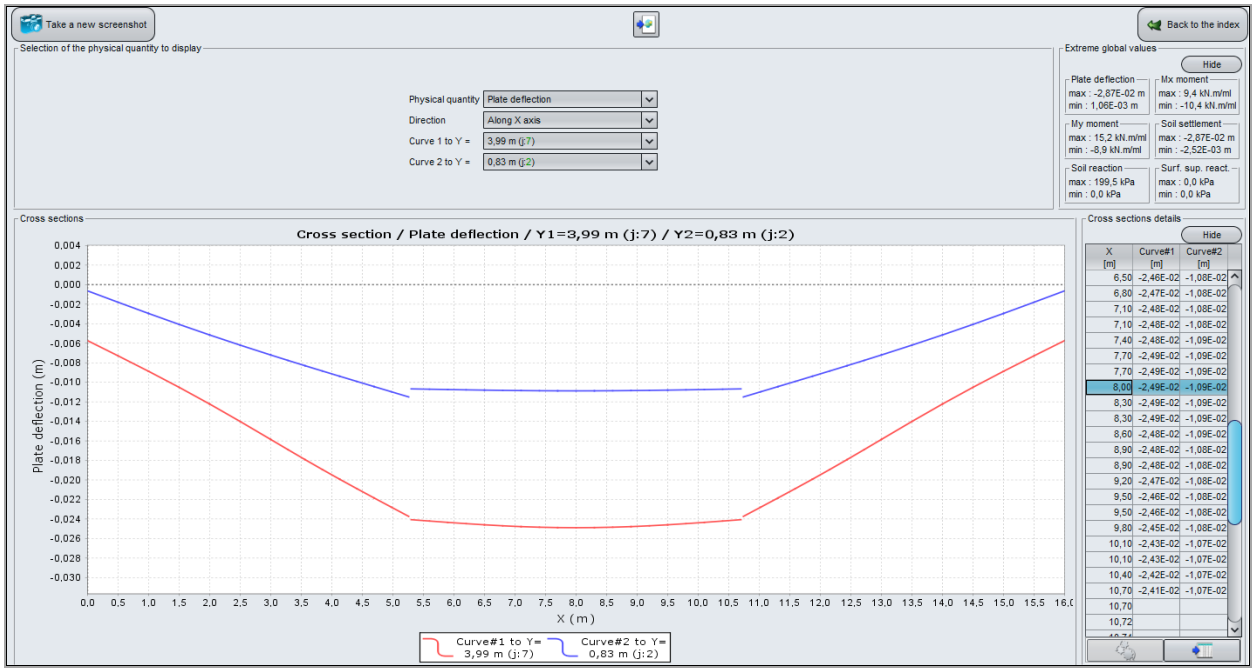
Extreme global values:

- Max: -2.87E-02 m (thus -2.9 cm)
- Min: 1.06E-03 m (thus 0.1 cm)

Cross-section along Ox:

Cross-section 1 at Y = 3.99 m (j: 7): at X = 8 m Deflection = -2.49 cm.

Cross-section 2 at Y = 1.25 m (j: 2): at X = 8 m Deflection = -1.28 cm.



Mx Moment along X axis:

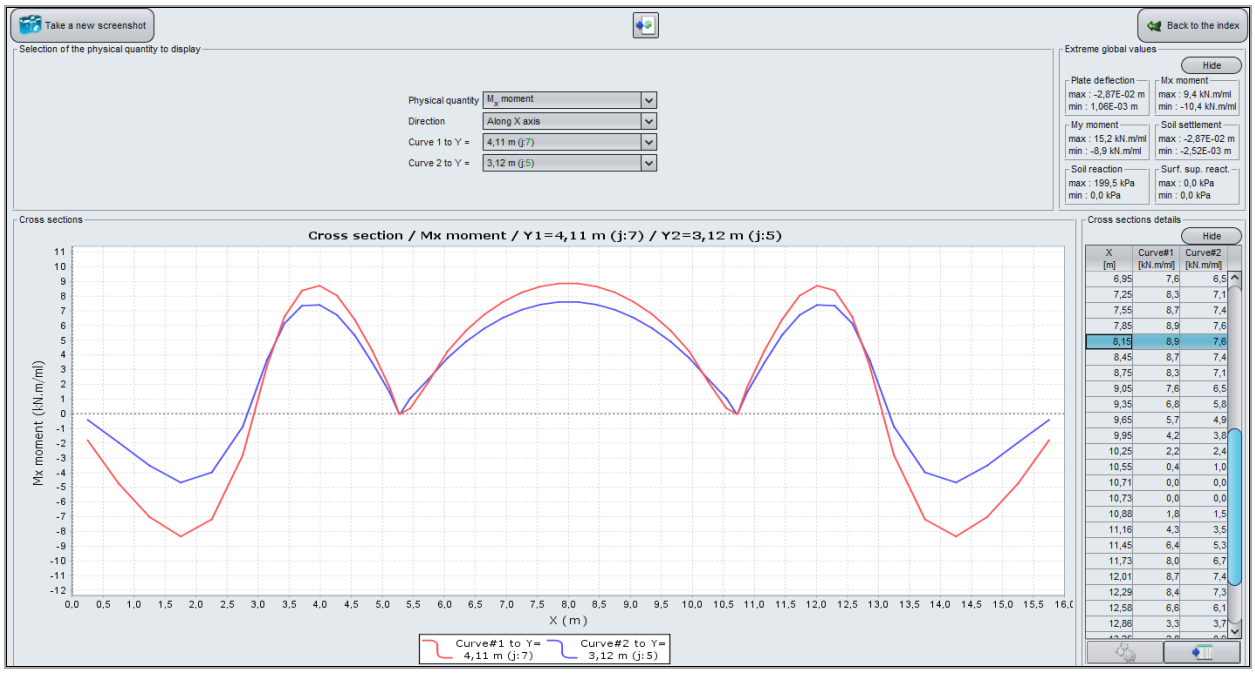
Extreme global values:

- Max: 9 kN.m/ml
- Min: -10 kN.m/ml

Cross-section along Ox:

Cross-section 1 at Y = 4.11 m (j: 7): at X = 8.15 m Mx = 9 kN.m/ml.

Cross-section 2 at Y = 3.12 m (j: 5): at X = 8.15 m Mx = 8 kN.m/ml.



My Moment along Y axis:

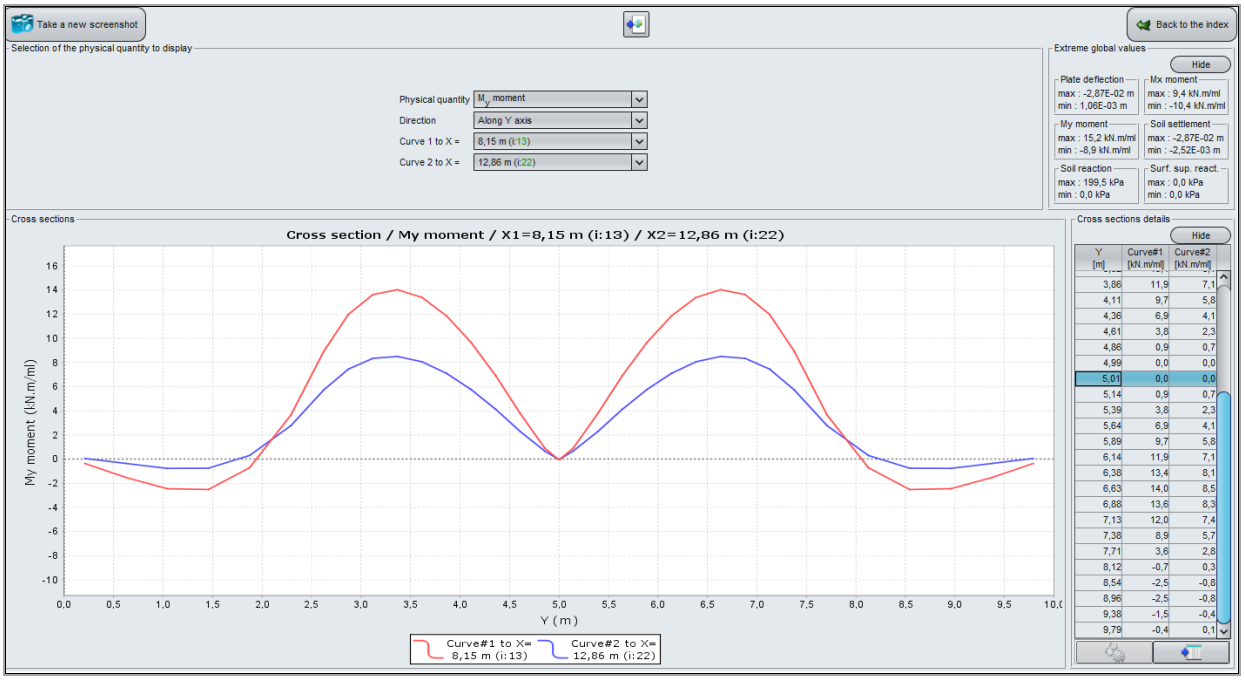
Extreme global values:

- Max: 15 kN.m/ml
- Min: -9 kN.m/ml

Cross-section along Oy:

Cross-section 1 at X = 8.15 m (i: 13): at Y = 5.01 m M_y = 0 kN.m/ml.

Cross-section 2 at X = 12.86 m (i: 22): at Y = 5.01 m M_y = 0 kN.m/ml.

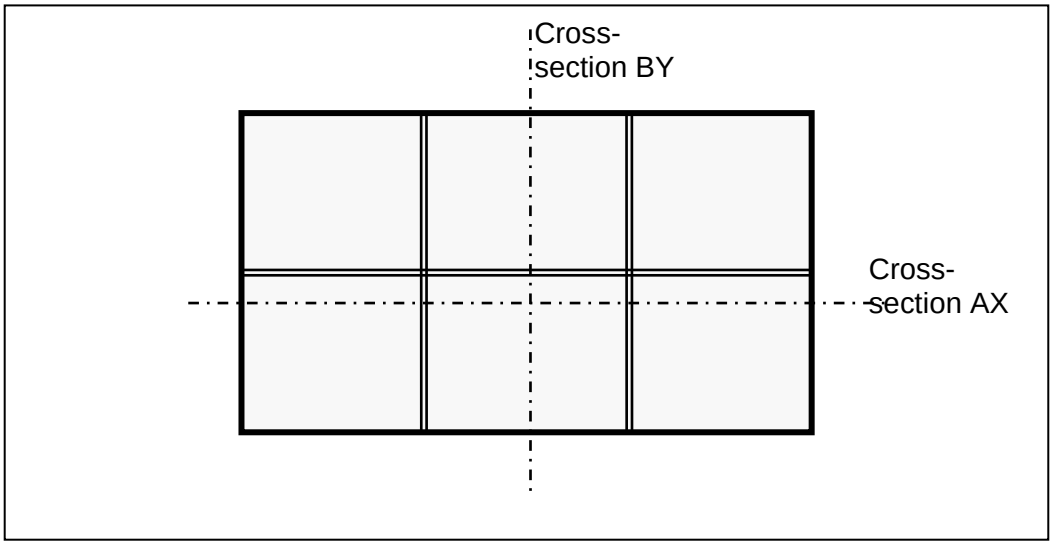


We note the amplitude of the bending moments obtained in this example:

- $M_{x,\,max}$ = 10 kN.m/ml < 0.4 x 28 kN.m/ml for the case of the continuous slab (step 1);
- $M_{y,\,max}$ = 15 kN.m/ml < 0.3 x 53 kN.m/ml for the case of the continuous slab (step 1).

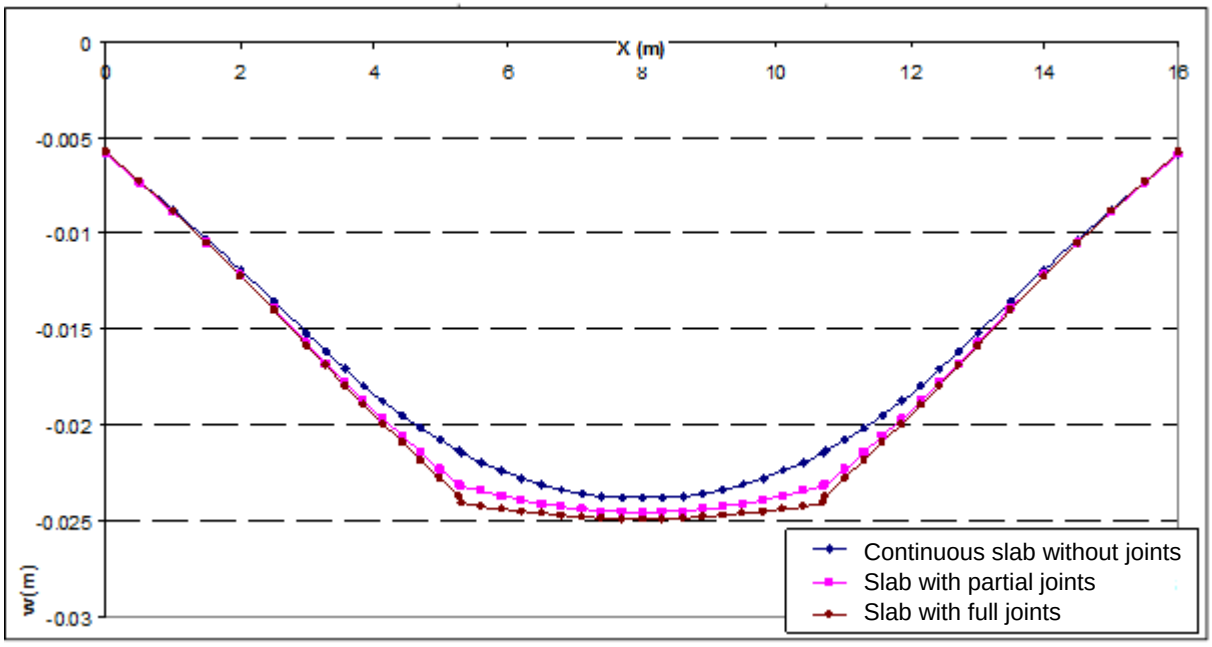
I.4.3.4. Comparison of the three cases

In order to illustrate the incidence of the joints on the shape and the amplitude of the forces in the body of the slab, we propose to superimpose the results obtained for the three cases; i.e. a continuous slab for the first case, a slab with partial joints for the second and a slab with full joints for the latest. The comparison is carried out at the place of the cross-sections presented in the following figure.

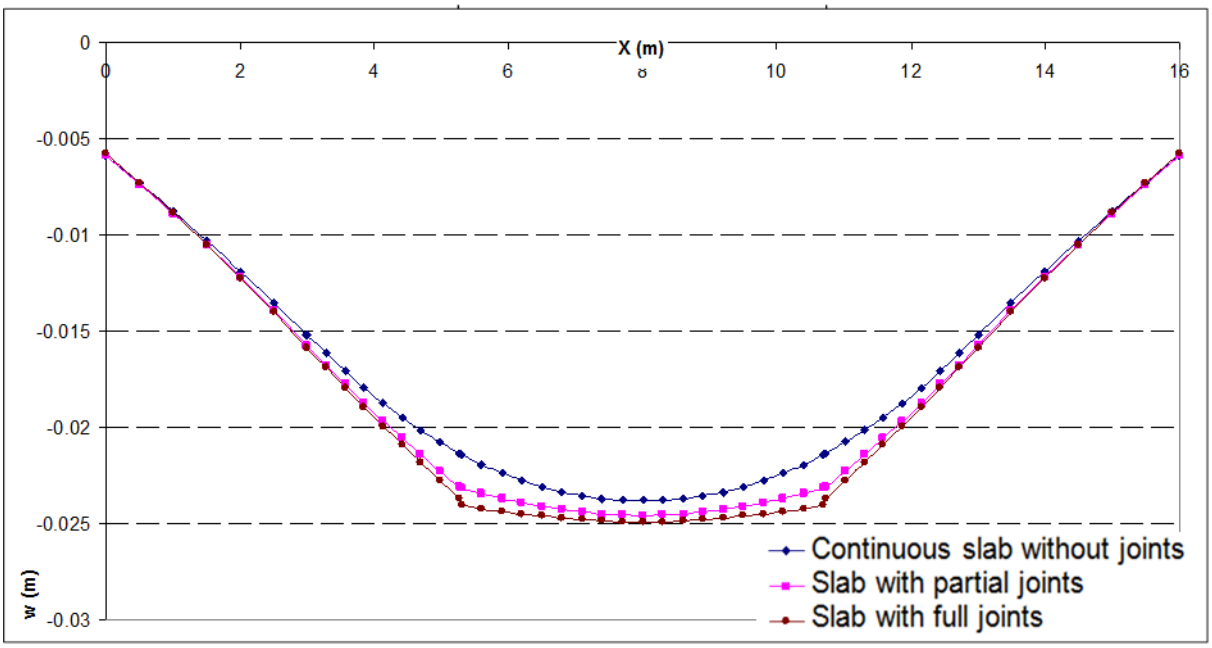


Calculated results are the following:

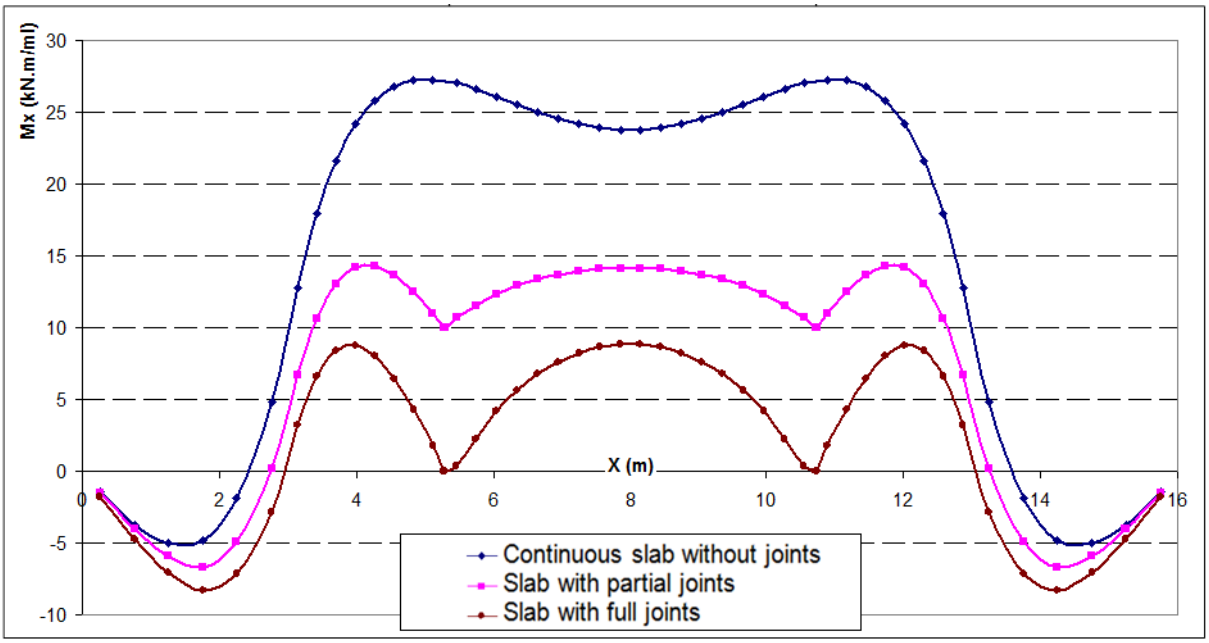
Settlement of the slab along the AX cross-section:



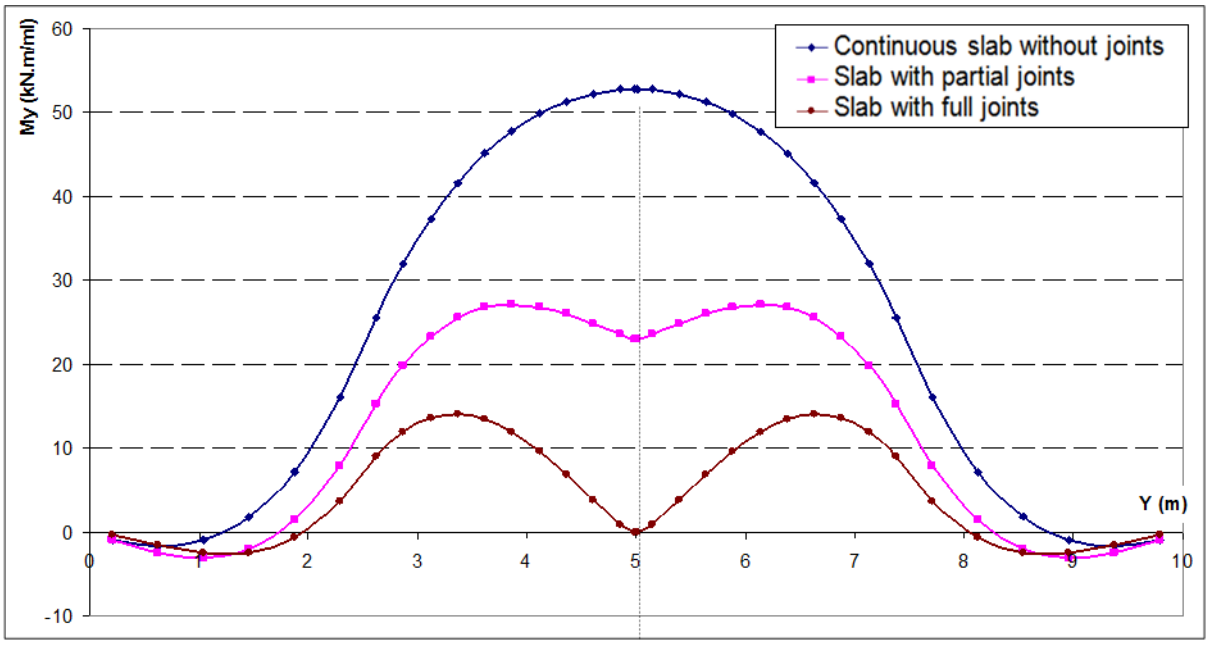
Settlement of the slab along the BY cross-section:



Mx Moment along the AX cross-section:



My Moment along the BY cross-section:



Regarding the displacements, we observe that with the creation of the joints, the behaviour of the slab looks more and more like in the case with 6 separated neighbouring slabs interacting through the soil body. Regarding the bending moments, we note that the introduction of the joints leads to a significant decrease of the amplitude and the shape of the moments in the slab.

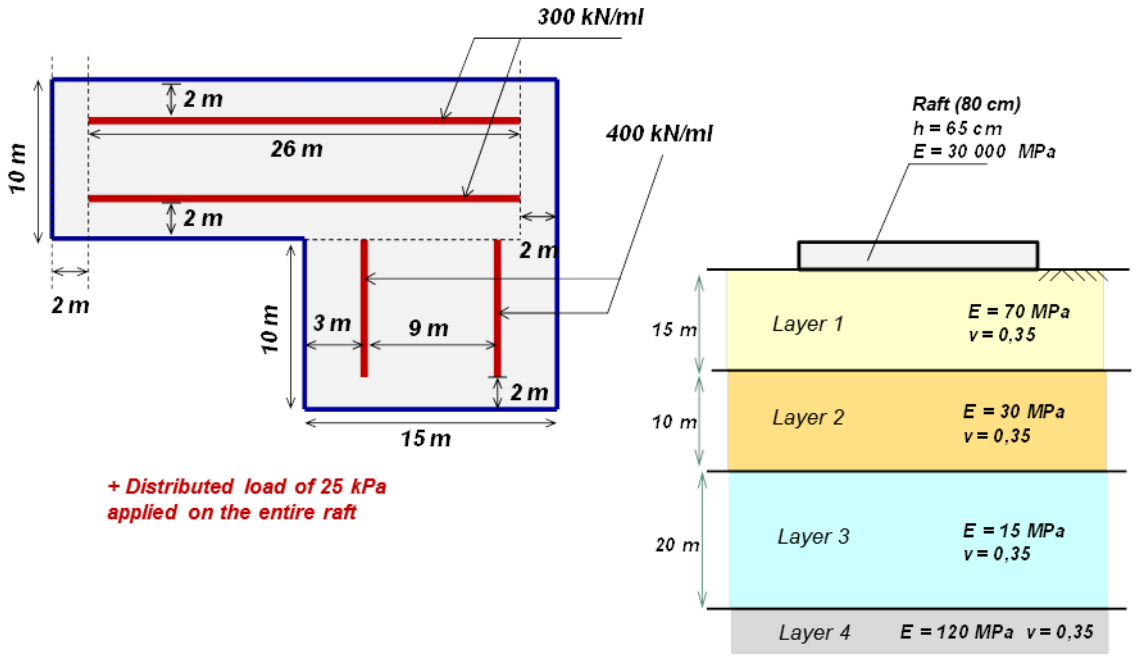
I.4.4. Example 4: L-Shaped raft with linear loading and interaction with an embankment

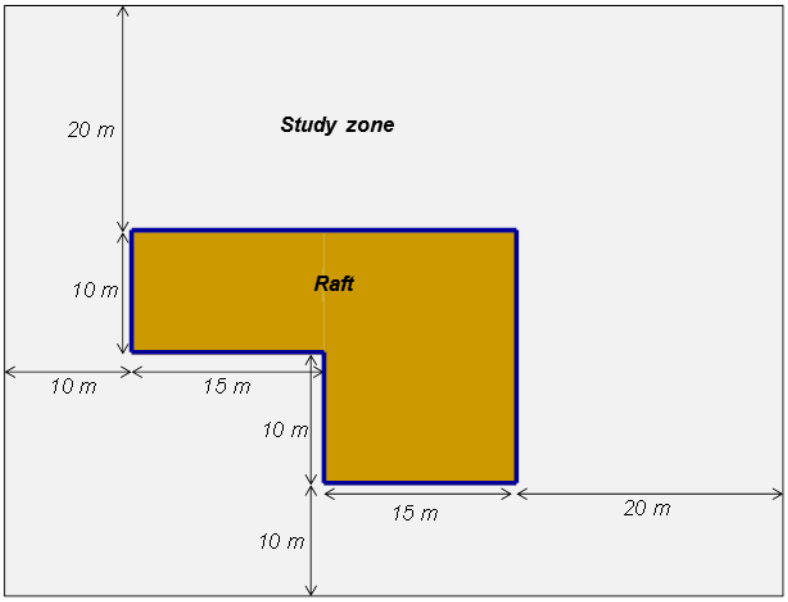
I.4.4.1. Step 1: L-Shaped raft with linear loading

A "L" shaped raft is subjected, in addition to a distributed load of 25 kPa, to two linear loads of 300 kN/ml along Ox axis and two other linear loads of 400 kN/ml along Oy axis.

The soil surrounding the raft is also taken into account to assess the effect of the loading of the raft on potential superstructures around.

The following figures show the input data of the project.



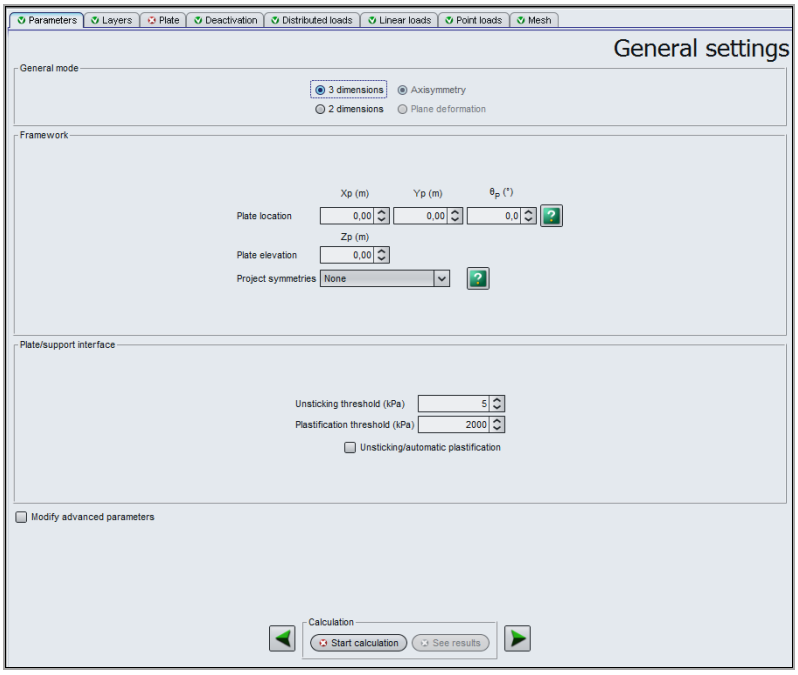


Create a new project (by following the same procedure as described in previous examples).

Select the module TASPLAQ and then click on the Create button.

I.4.4.1.1. "Parameters" tab

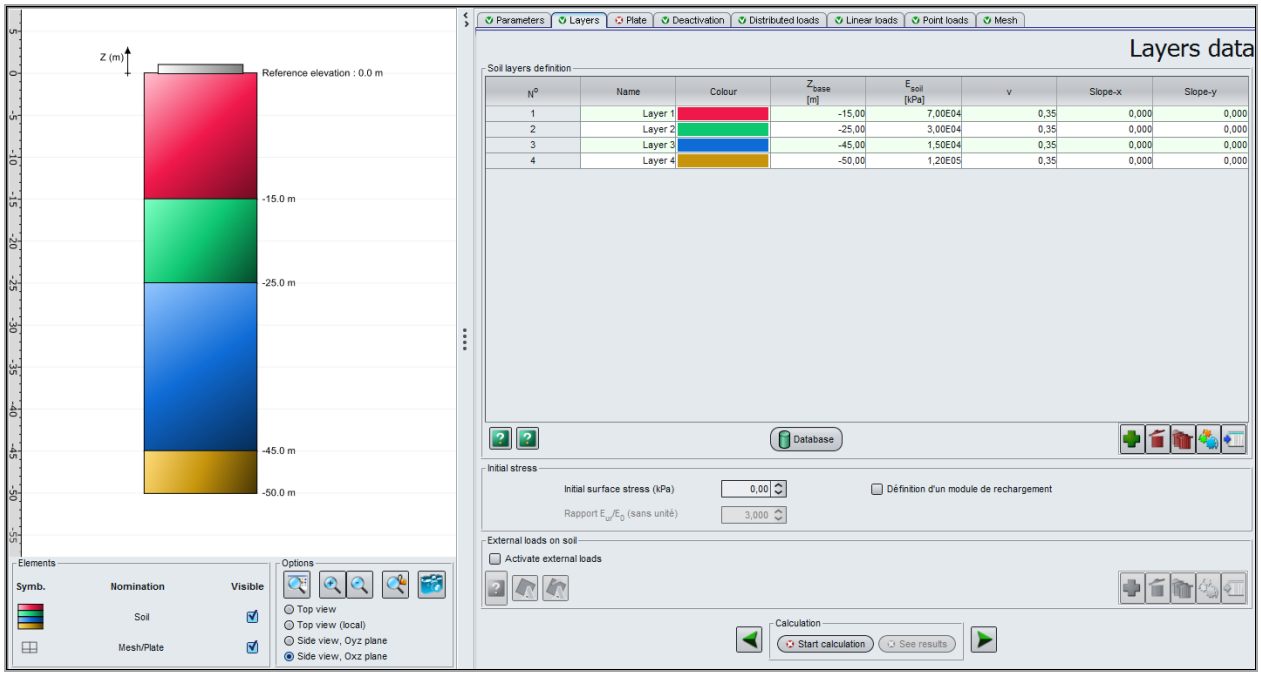
Complete the input data according to the screenshot below (we keep here the choices proposed by default).



I.4.4.1.2. "Layers" tab

Complete the input data according to the screenshot and the table below.

Name	Z_{base} (m)	E_{soil} (kPa)	ν	Slope-x	Slope-y
Layer 1	-15.0	7.00E+04	0.35	0.00	0.00
Layer 2	-25.0	3.00E+04	0.35	0.00	0.00
Layer 3	-45.0	1.50E+04	0.35	0.00	0.00
Layer 4	-50.0	1.20E+05	0.35	0.00	0.00

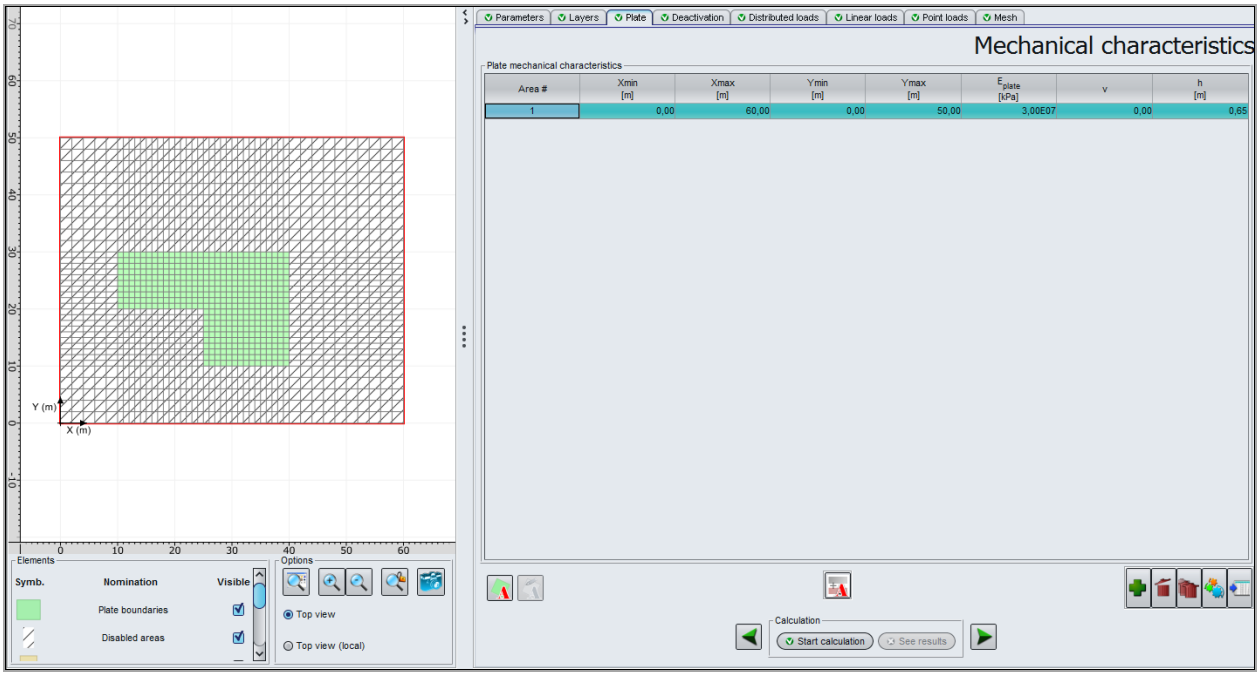


I.4.4.1.3. "Plate" tab

In this example, the raft is homogeneous and continuous. To define its characteristics, we have to define one single zone covering the whole model, knowing that only the activated zones of the plate will actually be assigned with these characteristics.

Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	60.00	0.00	50.00	3.00E+07	0.00	0.65

The choice of the Poisson ratio equal to zero is due to the reinforced concrete raft assumption.

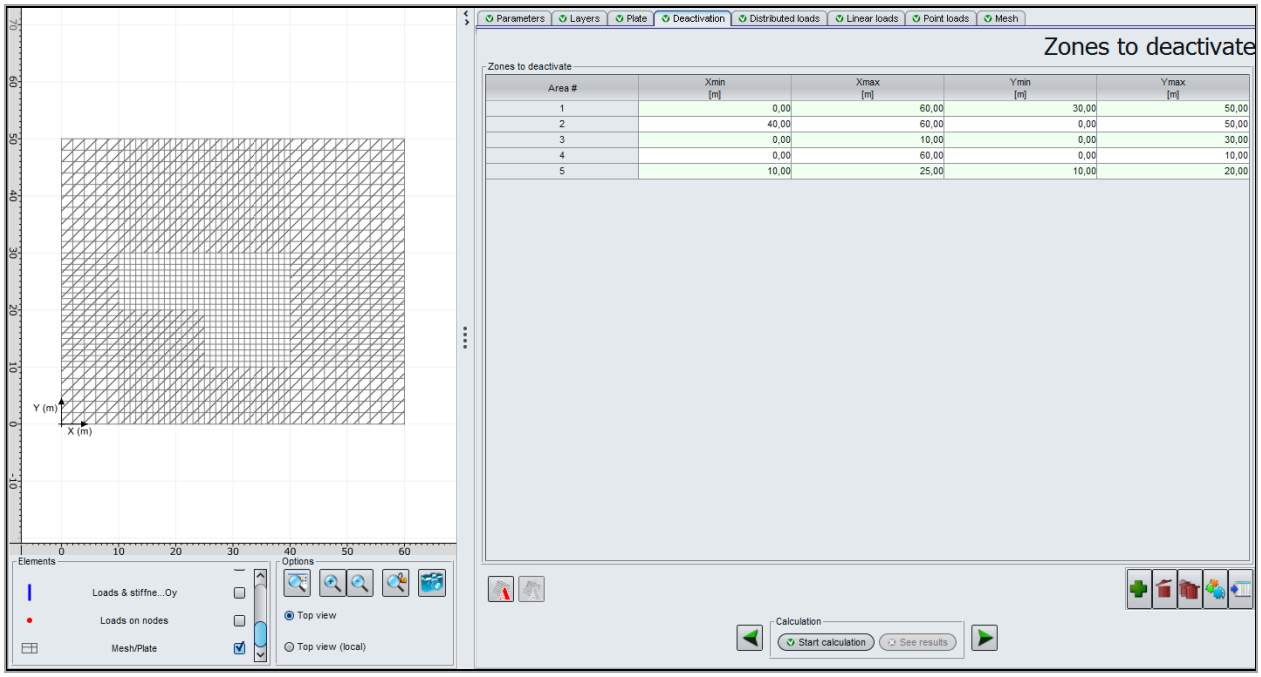


I.4.4.1.4. "Deactivation" tab

We have to deactivate the zones located outside the footprint of the L-shaped raft.

There are several "ways" of deactivation to reach the same result, knowing that an element can be deactivated several times.

The data given below corresponds to one of the possible deactivation methods.

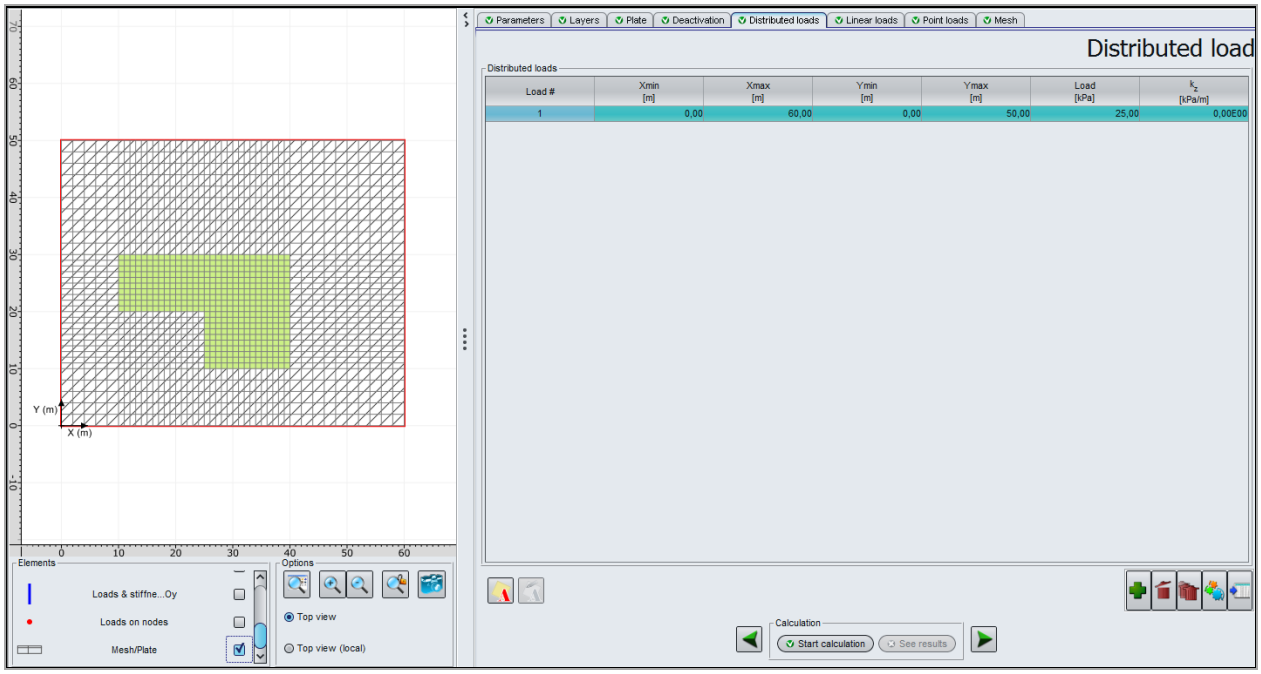


Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)
1	0.00	60.00	30.00	50.00
2	40.00	60.00	0.00	50.00
3	0.00	10.00	0.00	30.00
4	0.00	60.00	0.00	10.00
5	10.00	25.00	10.00	20.00

I.4.4.1.5. "Distributed loads" tab

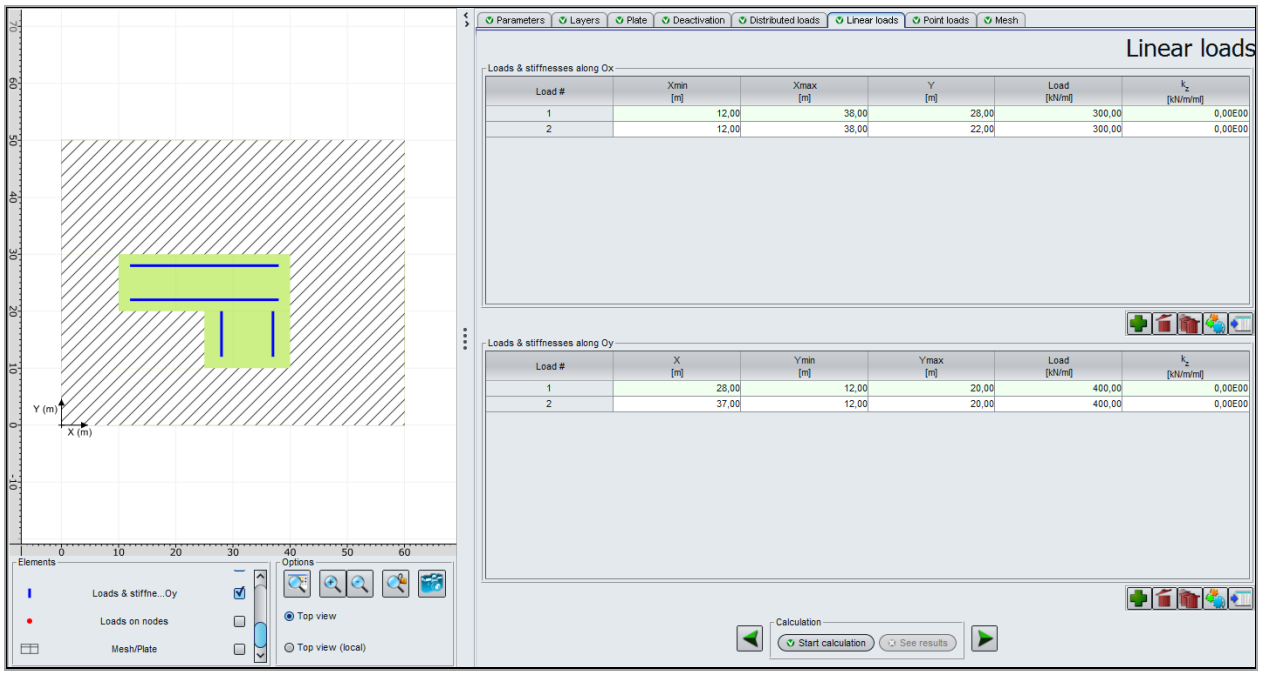
We have to define a distributed load of 25 kPa covering the whole plate: here again, we can define it on a large rectangle covering the whole model: the load will be taken into account only for the active elements of the plate.

Load n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	Load (kPa)	k_z (kPa/m)
1	0.00	60.00	0.00	50.00	25	0



I.4.4.1.6. "Linear loads" tab

Two linear loads along Ox (300 kN/ml) and two linear loads along Oy (400 kN/ml) should be defined on top of the raft.



Loads along X axis:

Load n°	Xmin (m)	Xmax (m)	Y (m)	Load (kPa)	k_z (kPa/m)
1	12.00	38.00	28.00	300	0
2	12.00	38.00	22.00	300	0

Loads along Y axis:

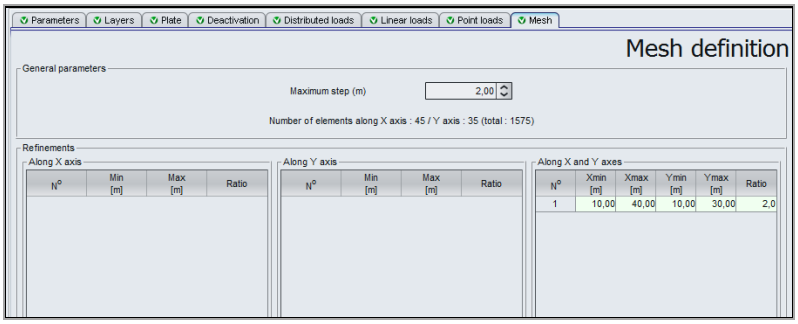
Load n°	Ymin (m)	Ymax (m)	X (m)	Load (kPa)	k_z (kPa/m)
1	28.00	12.00	20.00	400	0
2	37.00	12.00	20.00	400	0

I.4.4.1.7. "Point loads" tab

There is no point load in this example.

I.4.4.1.8. "Mesh" tab

A maximum step of 2 m as well as a refinement (x2) in the zone of the raft are to be defined: a 30 m x 20 m rectangular zone located between Xmin = 10.00 m, Xmax = 40.00 m and Ymin = 10.00 m, Ymax = 30.00 m.



The input data are the following:

- Maximum step = 2 m

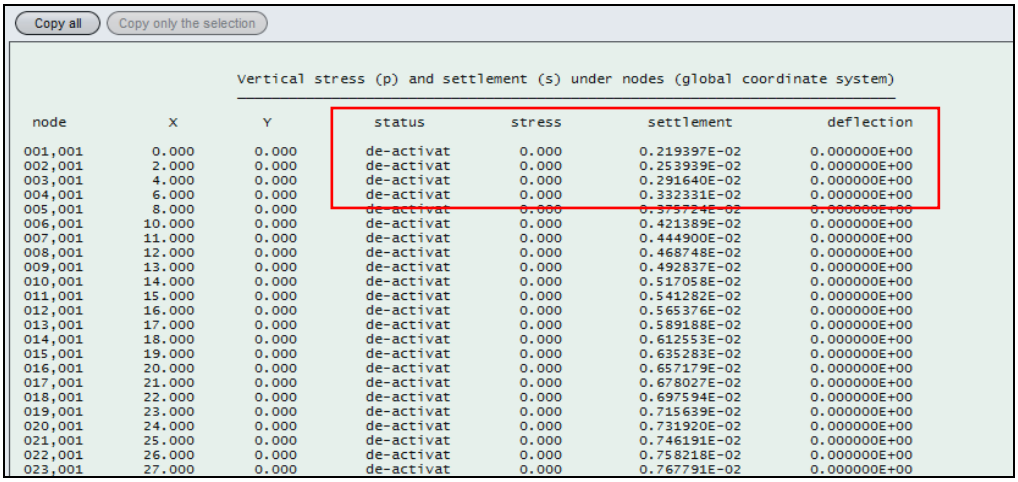
Refinement along X and Y axes:

N°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	Ratio
1	10.00	40.00	10.00	30.00	2.0

I.4.4.1.9. Calculations and Results

Run the calculations and then check the results.

Formatted numerical results



For the deactivated nodes, the reaction of the soil is zero (there is no plate) but a settlement is still calculated; the extension of the model beyond the raft footprint has especially been done for this purpose.

Graphical results – Cross-sections

Soil settlement:

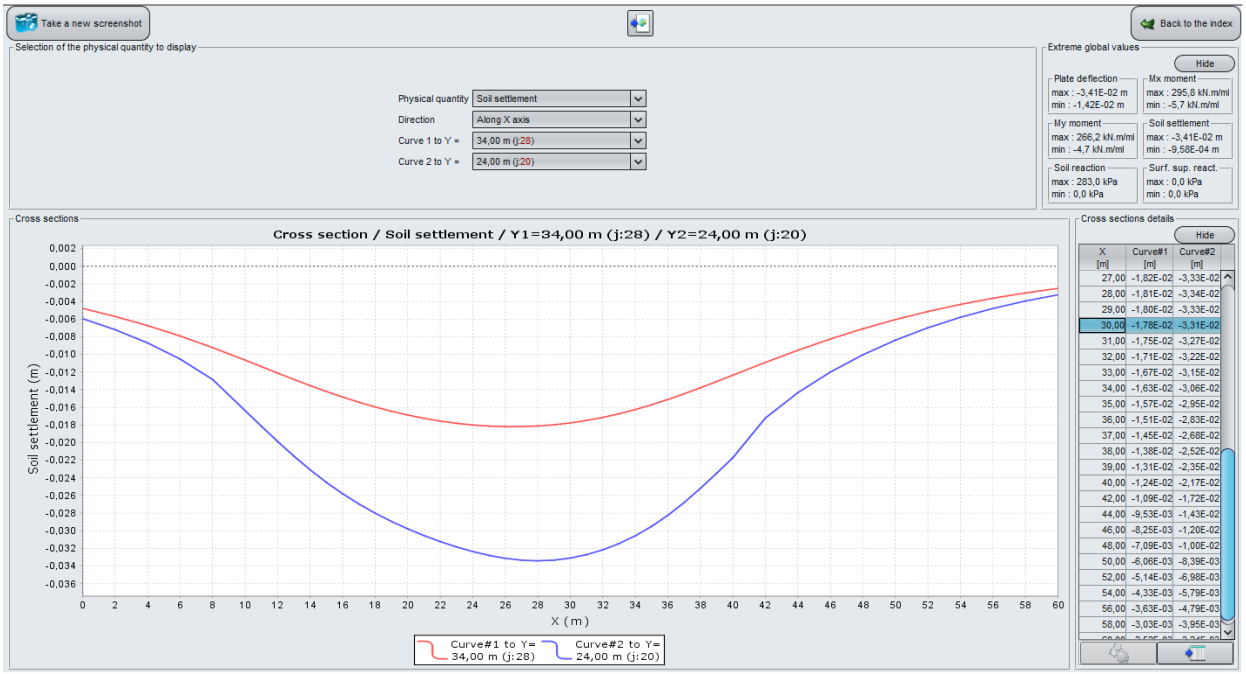
Extreme global values:

- Max: -3.41E-02 m (thus -3.4 cm)
- Min: -9.58E-04 m (thus -0.1 cm)

Cross-section along X axis:

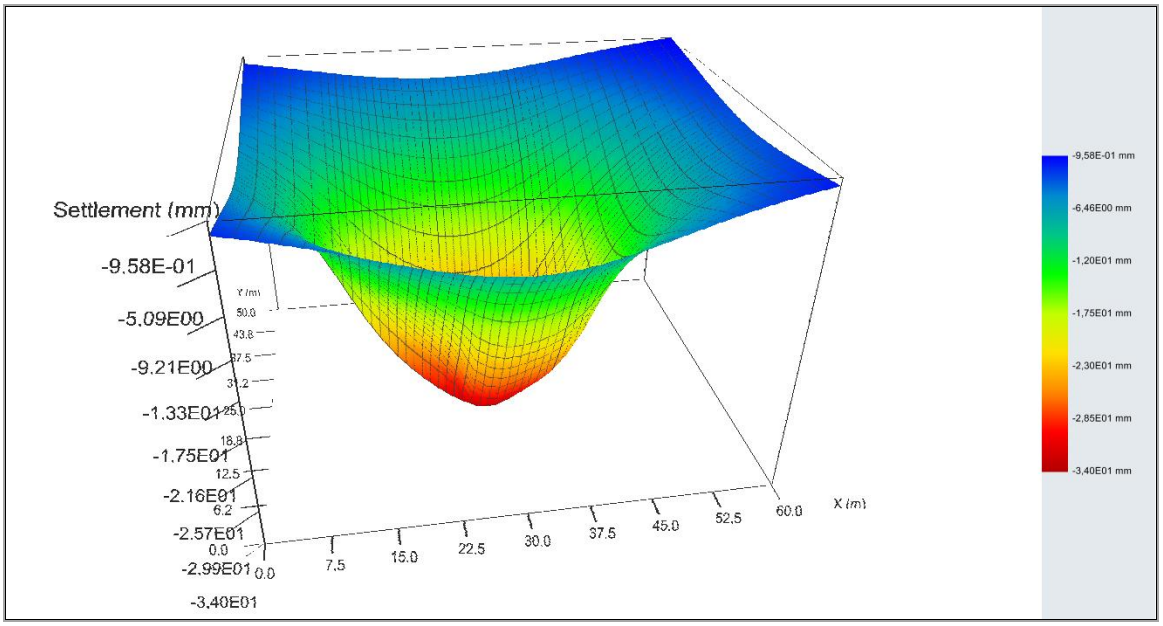
Cross-section 1 at Y = 34 m (j: 28): at X = 30 m Soil settlement = -1.8 cm

Cross-section 2 at Y = 24 m (j: 20): at X = 30 m Soil settlement = -3.3 cm



3D Graphical results

Soil Settlement

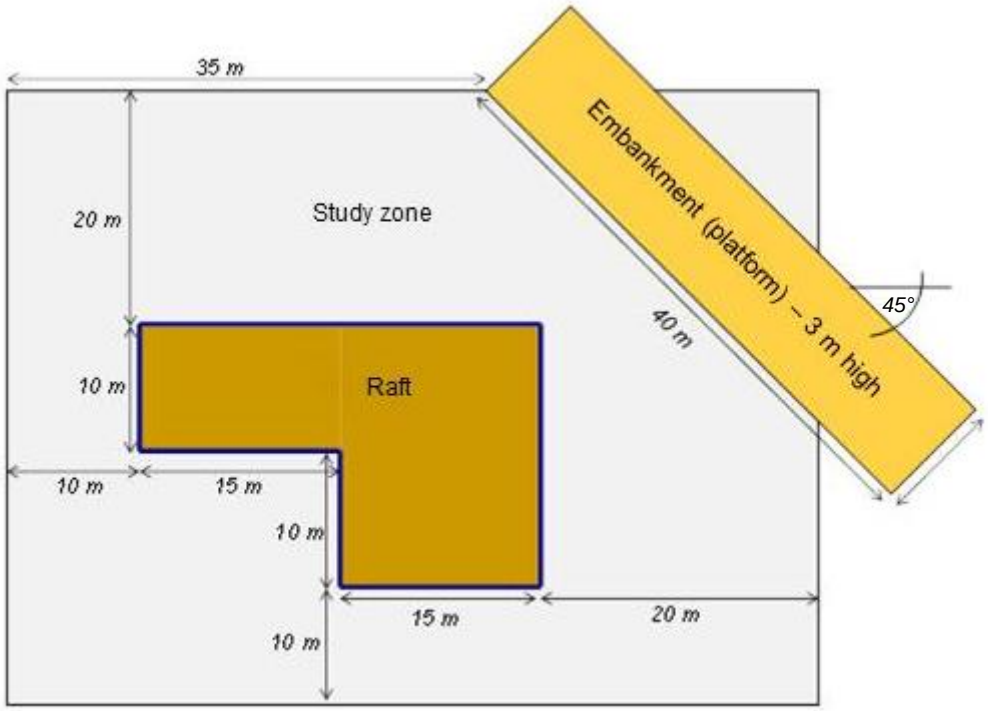


The purpose of this example is to illustrate the ability of TASPLAQ to handle, not only the settlement triggered by the raft in its own footprint, but also the settlement of the surrounding soil.

Please note that the assessment of the surrounding soil settlements would not be possible if the soil had been assimilated, as commonly done, to a set of juxtaposed springs (thus independent springs by definition).

I.4.4.2. Step 2: Influence of a neighbouring embankment

At this stage, we aim to study the influence of an embankment of 3 m high, bringing an extra load of 60 kPa in the vicinity of the raft. The layout geometry of the embankment is given in the figure below.



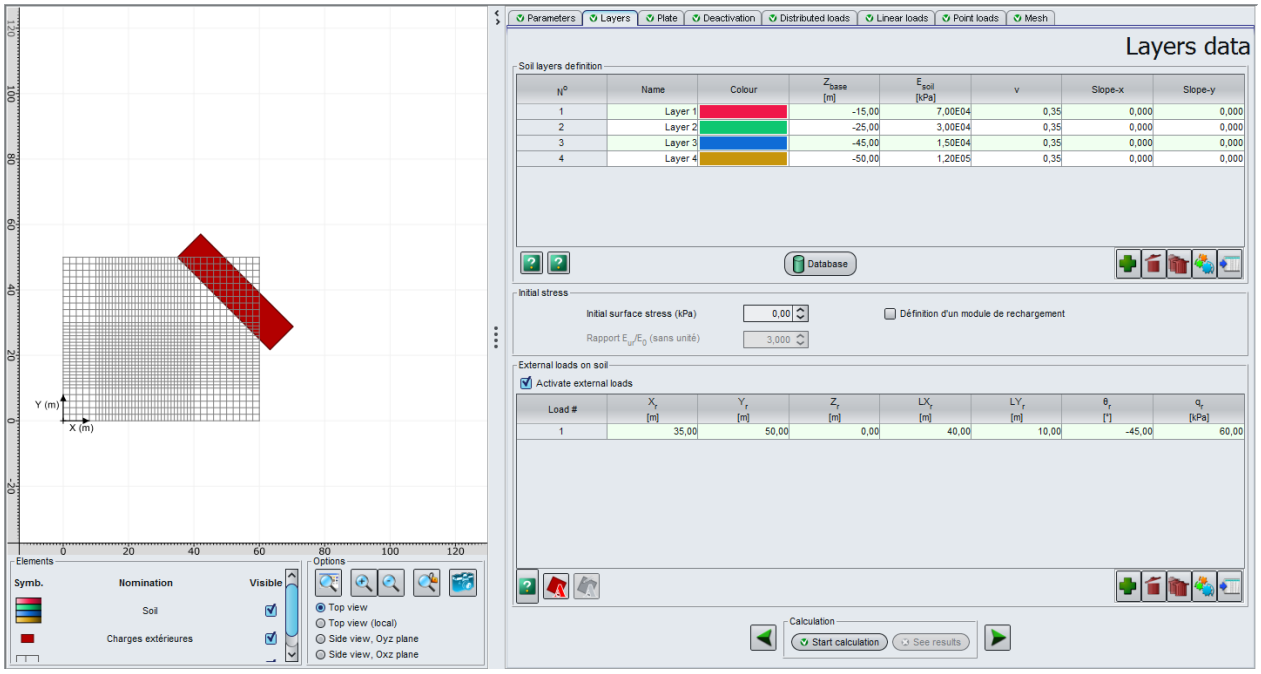
We will take over the previous project (step 1) and add a new load directly on top of the soil, around of the plate. Open the previous project and then save as with a different name.

I.4.4.2.1. "Layers" tab

The presence of an embankment is simulated via the insertion of an external load of 60 kPa (applied directly on top of the soil).

Note: Tasplaq considers implicitly that the load of the embankment and the one of the raft are applied simultaneously!

Load n°	X_r (m)	Y_r (m)	Z_r (m)	LX_r (m)	LY_r (m)	θ (°)	q_r (kPa)
1	35.0	50.0	0.0	40	10	-45	60



I.4.4.2.2. Results

We can now compare the new results to the previous ones, and display the influence of the embankment on the settlements.

<u>Graphical results – Cross-sections</u>

<u>Soil settlement</u>:

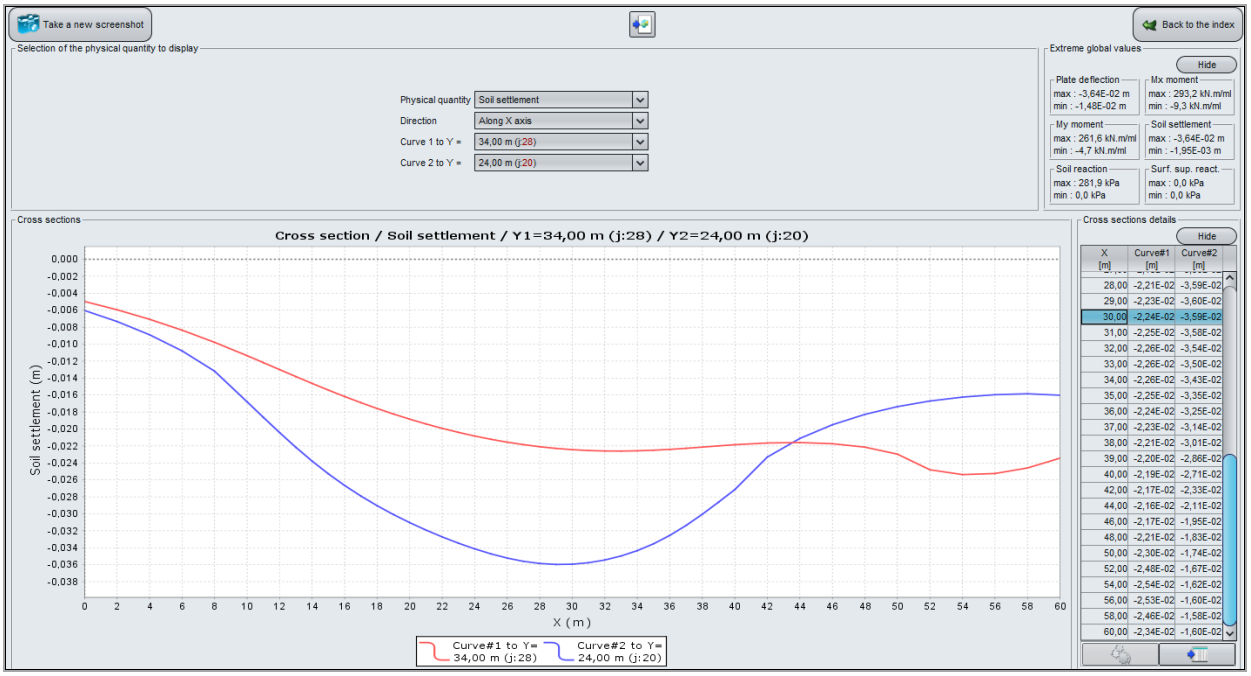
Extreme global values:

- Max: -3.64E-02 m (thus -3.6 cm)
- Min: -1.95E-03 m (thus -0.2 cm)

<u>Cross-section along X</u>:

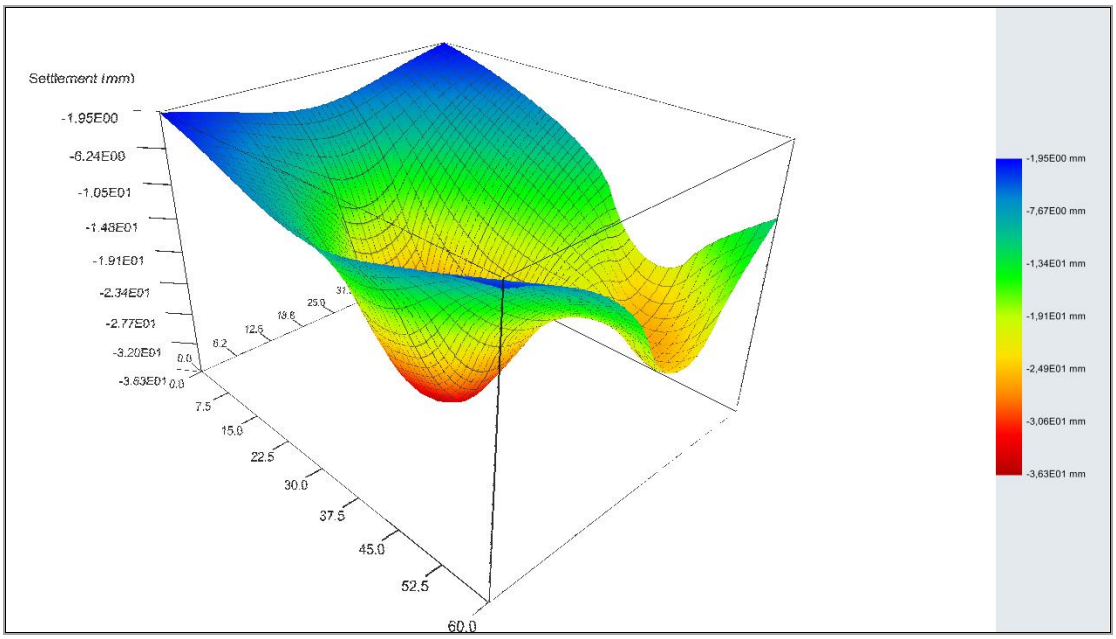
Cross-section 1 at Y = 34 m (j: 28): at X = 30 m Soil settlement = -2.24 cm

Cross-section 2 at Y = 24 m (j: 20): at X = 30 m Soil settlement = -3.59 cm

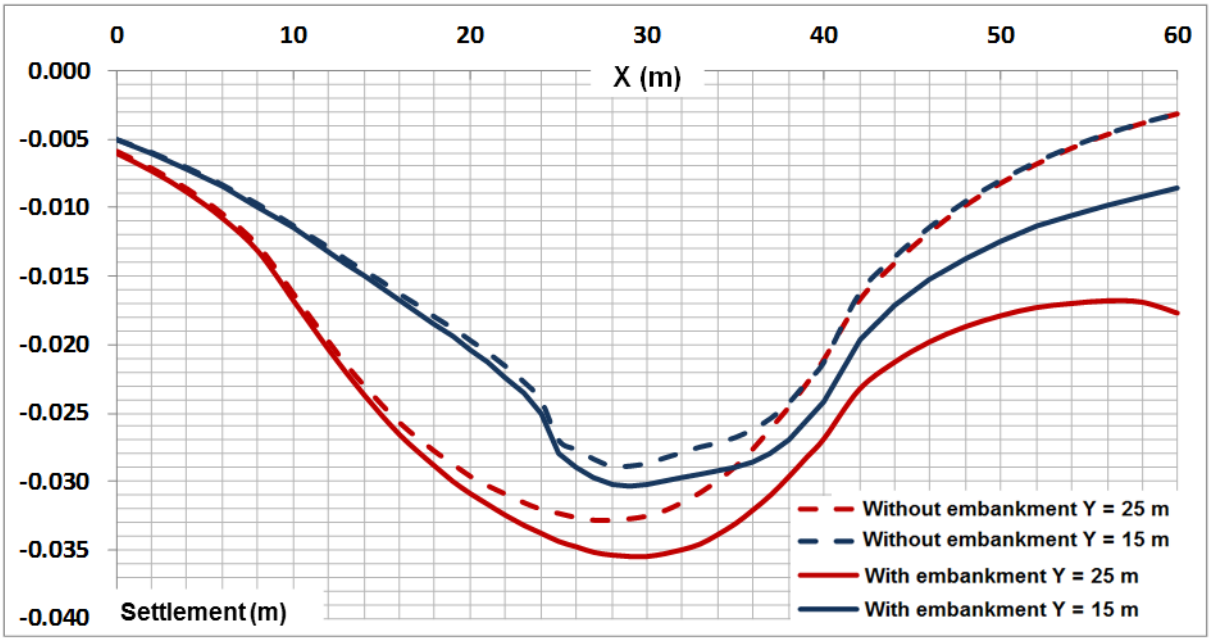


3D graphical results

Soil settlement

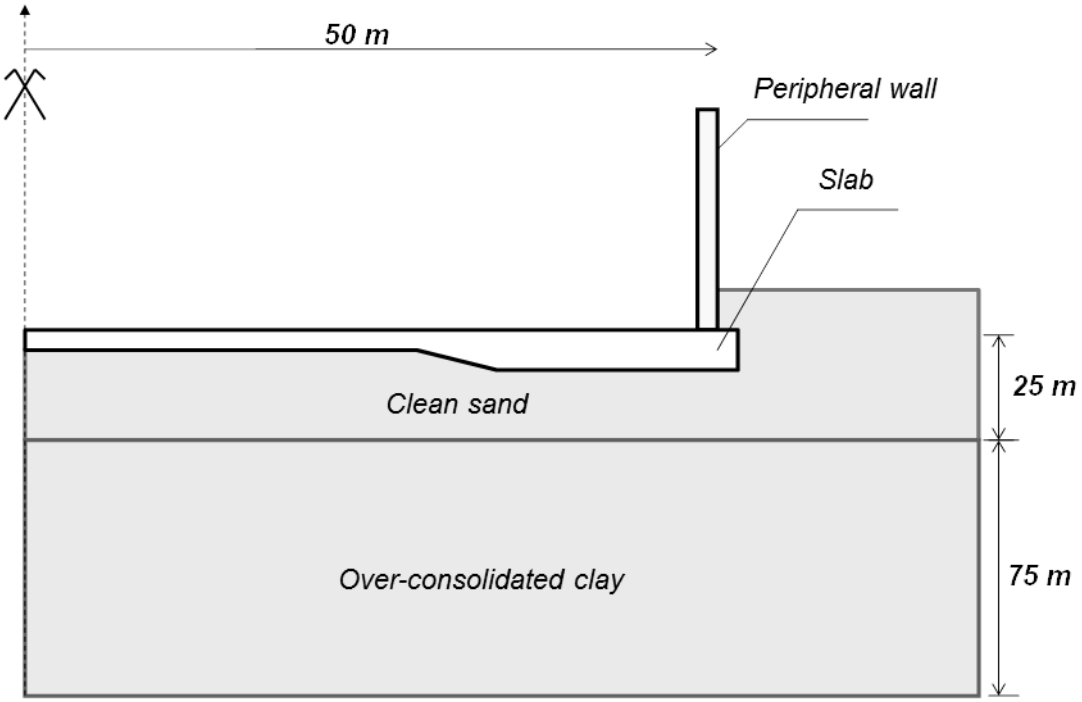


This second step has been considered with the objective of showing the ability of TASPLAQ to handle interaction problems between a raft and an embankment in its vicinity. In this case, the embankment not only generates settlements below itself but also it causes the settlement of the raft. This interaction phenomenon is also illustrated by the graph below.



I.4.5. Example 5: Settlements under a circular tank

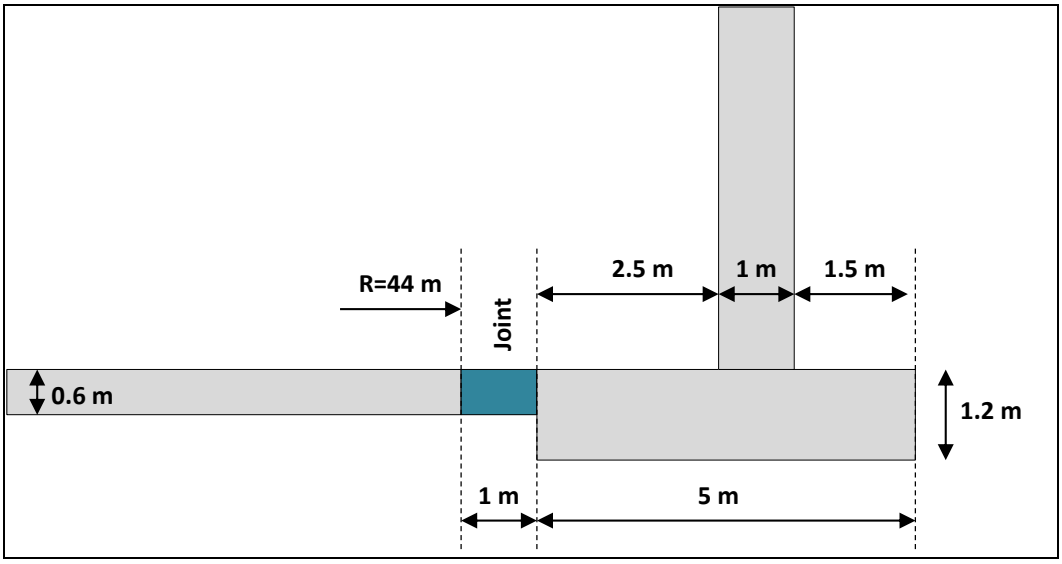
This example presents the settlement calculations under a circular tank of 100 m in diameter. The tank is built on a 25 m thick layer of sandy soil, laying on a very over-consolidated clay layer. The mechanical bedrock is reached at 100 m depth.



The raft includes a 0.6 m thick central zone and a 1.2 m thick peripheral zone. It is partially buried in the soil.

The connexion between the central and peripheral zones is created thanks to a 1 m large joint.

The axis of the peripheral wall is located at 2 m from the edge of the raft. The exact geometry of the structure is given in the figure below.



Geometry of the peripheral part of the structure

Construction and commissioning stages are the following:

- Construction: concrete pouring of the central and the peripheral part of the raft, installation of the peripheral walls and the dome;
- Junction;
- Hydrotest (water filling and discharge of the entire tank);
- Commissioning.

During this analyse, we are interested in settlements generated in the footprint of the tank and its vicinity. We will successively study 2 different loading conditions, corresponding to construction stage and hydrotest respectively.

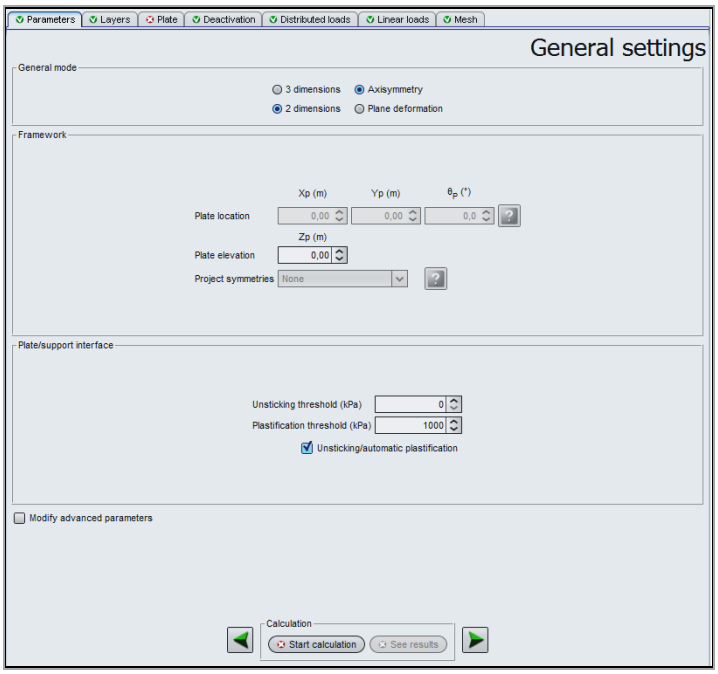
I.4.5.1. Step 1: Construction stage of the tank

Create a new Foxta v3 project and select the TASPLAQ module, then click on the Create button.

I.4.5.1.1. "Parameters" tab

The analyse can be performed by using an axisymmetric model.

Complete this tab as indicated in the screenshot below.

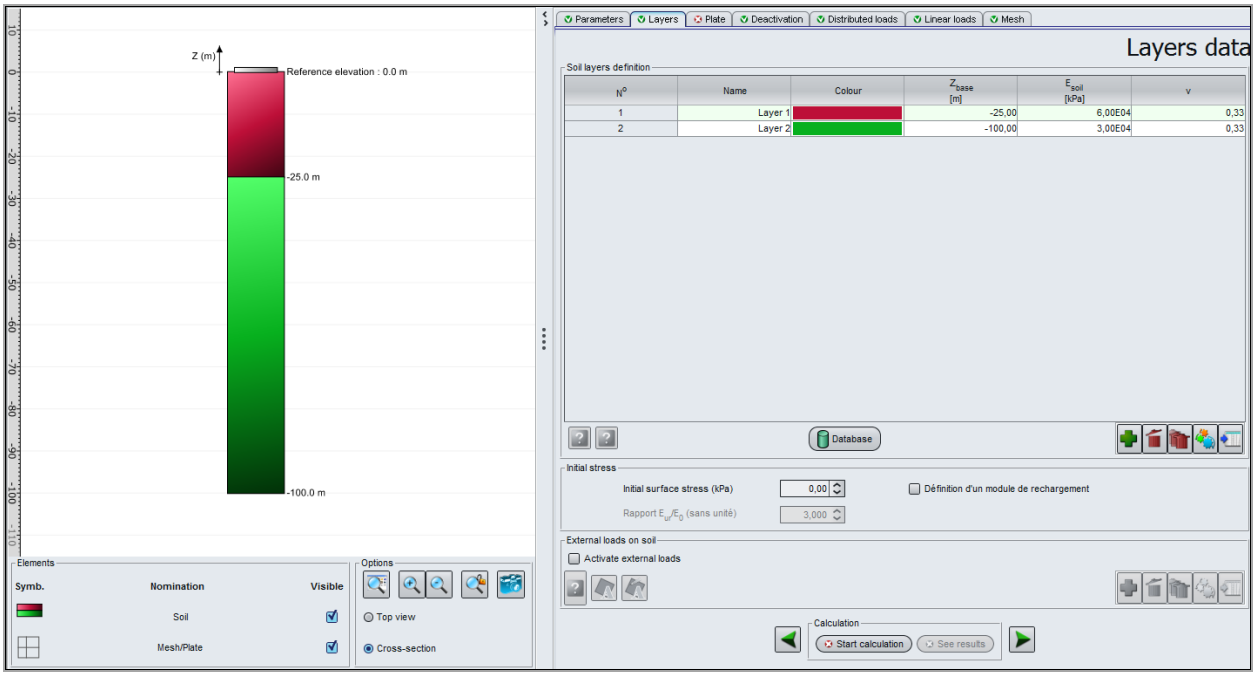


Please note that for this example, we check the "Automatic unsticking/plastification" checkbox, and input the following values:

- Unsticking: 0 kPa
- Plastification: 1000 kPa

I.4.5.1.2. "Layers" tab

Complete this tab as indicated on the following screenshot and the table.

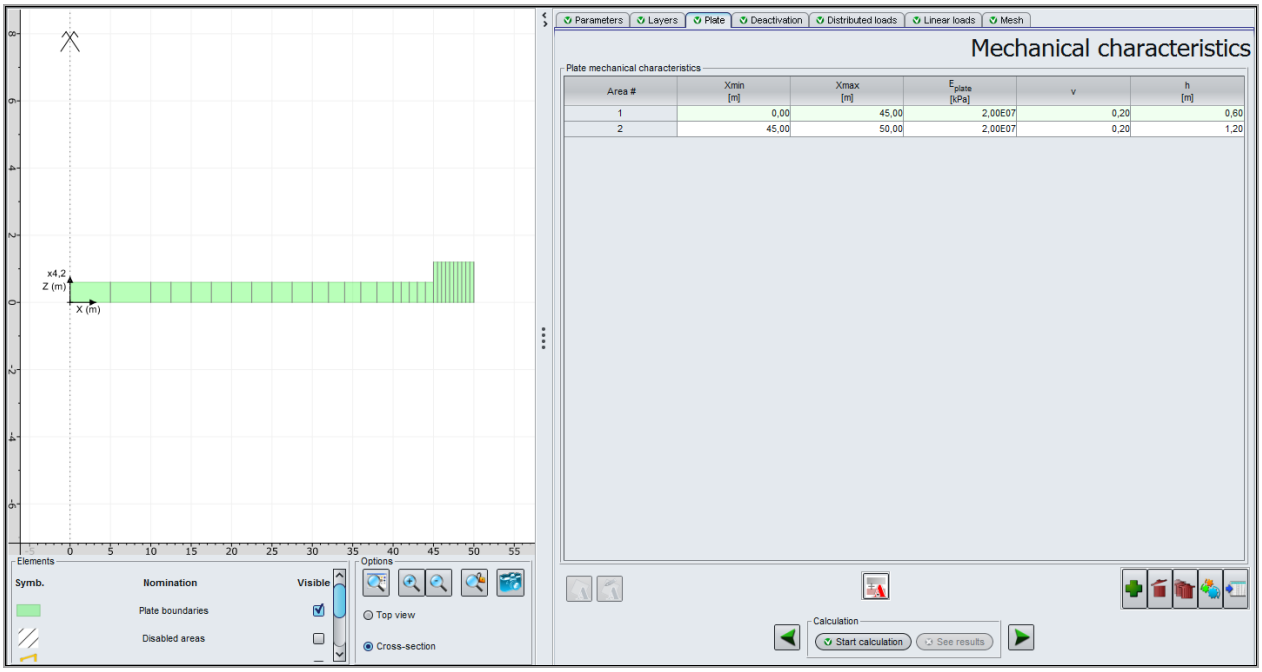


Name	Z_{base} (m)	E_{soil} (kPa)	ν
Layer 1	-25	60 000	0.33
Layer 2	-100	30 000	0.33

I.4.5.1.3. "Plate" tab

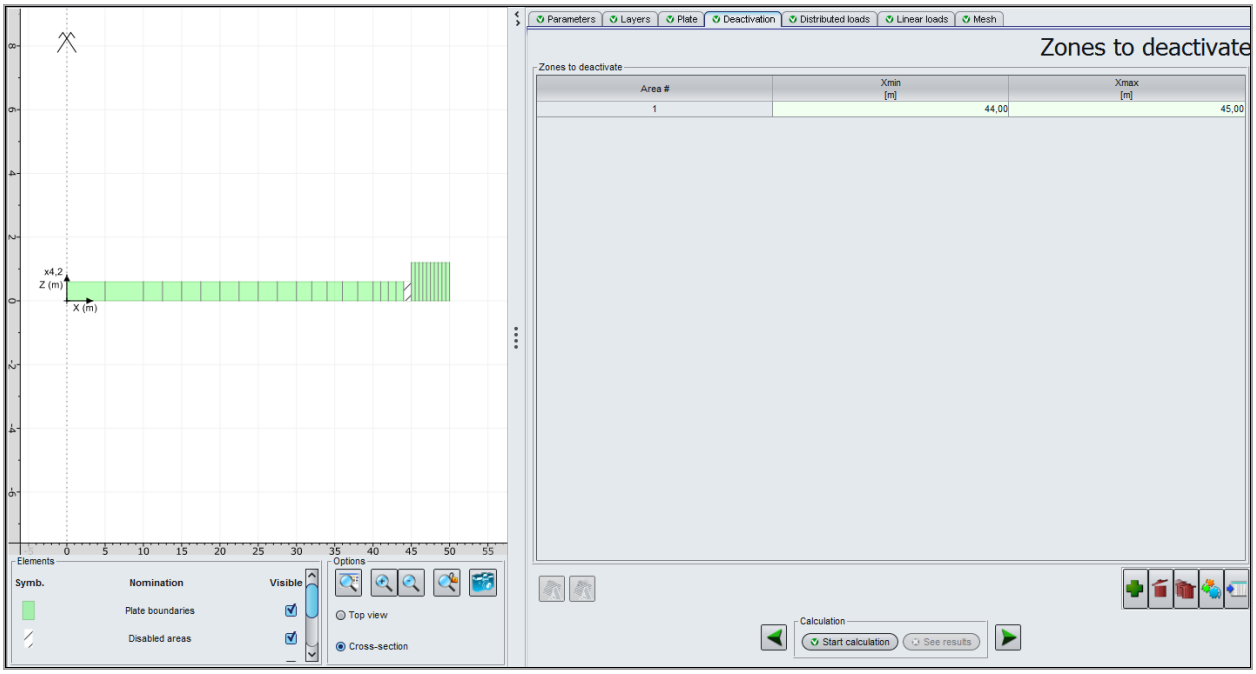
By considering the thickness variation between the central and the peripheral parts of the raft, we distinguish two different zones:

Zone n°	Xmin (m)	Xmax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	45.00	2.00E+07	0.20	0.60
2	45.00	50.00	2.00E+07	0.20	1.20



I.4.5.1.4. "Deactivation" tab

During the construction stage of the raft, the junction joint between the central and peripheral parts is not installed yet. We use the "Deactivation" option in order to take into account this discontinuity. We will deactivate an annular strip zone located between the radii Xmin = 44 m and Xmax = 45 m (from the centre of the raft).



I.4.5.1.5. "Distributed loads" tab

During the construction stage, the load transferred to the soil results only from the dead load of the structure (raft, wall and dome).

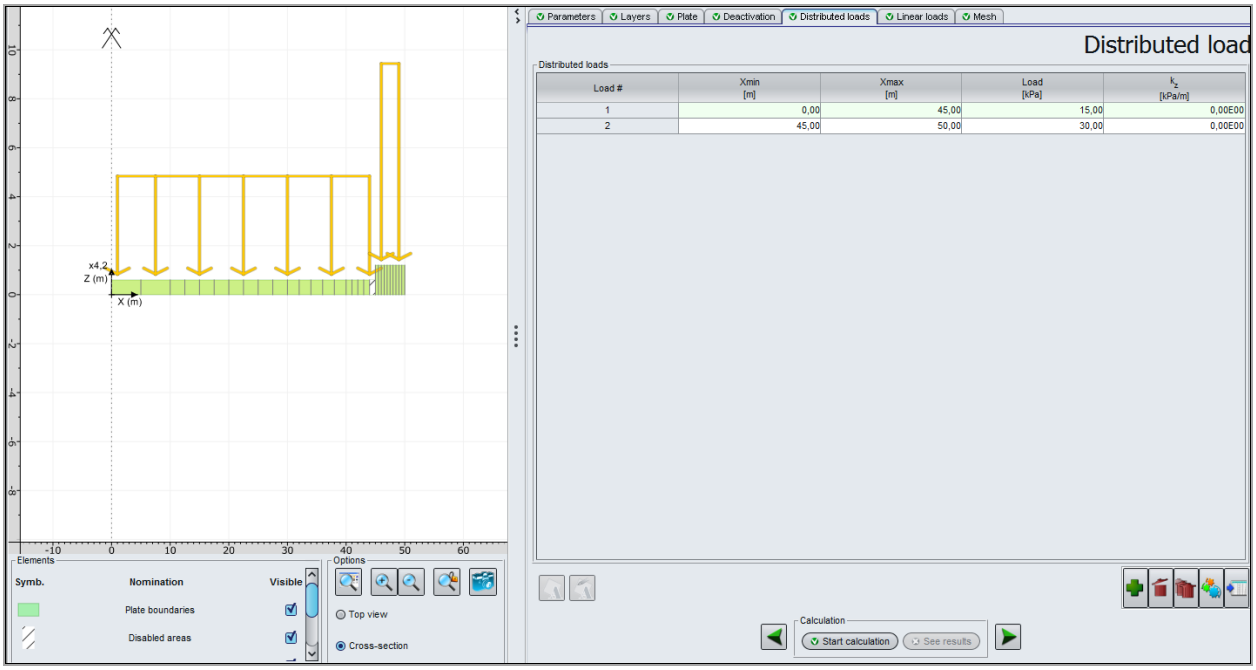
The dead load of the raft is obviously related to its thickness. As the peripheral part of the raft is thicker than the central part, we consider the following loads:

- Load in the central part: 15 kPa;
- Load in the peripheral part: 30 kPa.

Therefore, we define two different load values for plates n°1 and n°2.

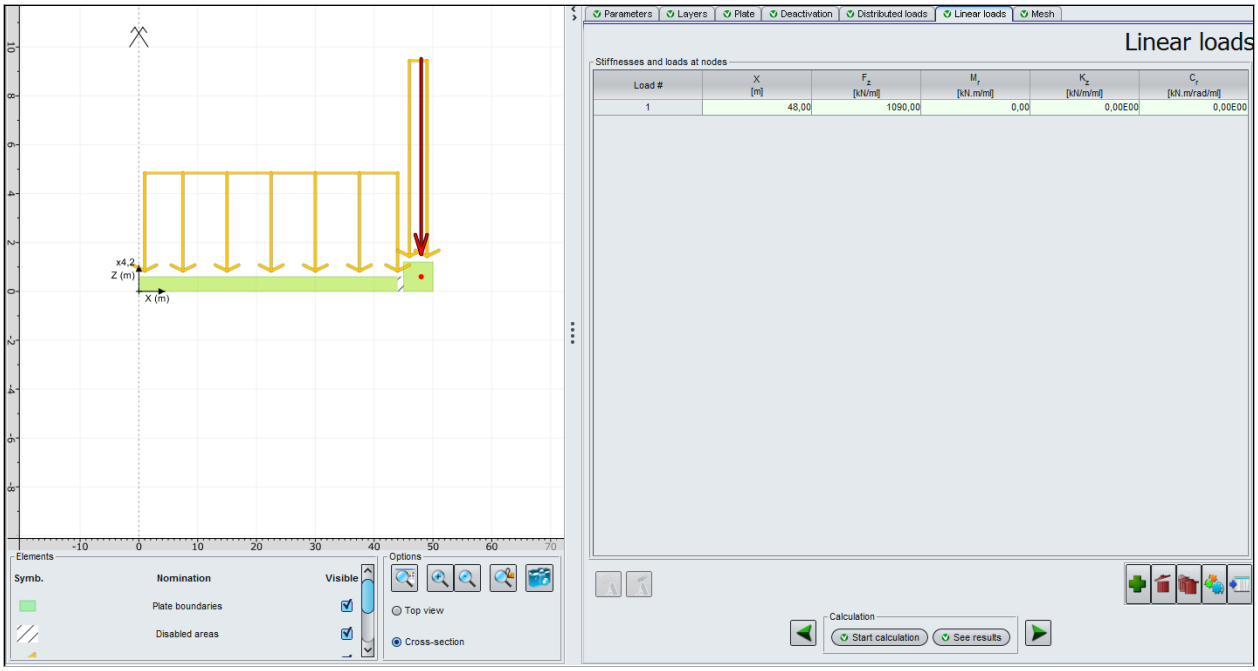
The input data are the following:

Load n°	Xmin (m)	Xmax (m)	Load (kPa)	k_z (kPa/m)
1	0.00	45.00	15	0
2	45.00	50.00	30	0



I.4.5.1.6. "Linear loads" tab

The load generated by the dead load of the superstructure (walls and dome) are transferred to the raft through the peripheral wall. At the axis of the wall, a linear load of 1090 kN/ml shall be applied. The axis of the wall is located at X = 48 m from the centre of the raft.

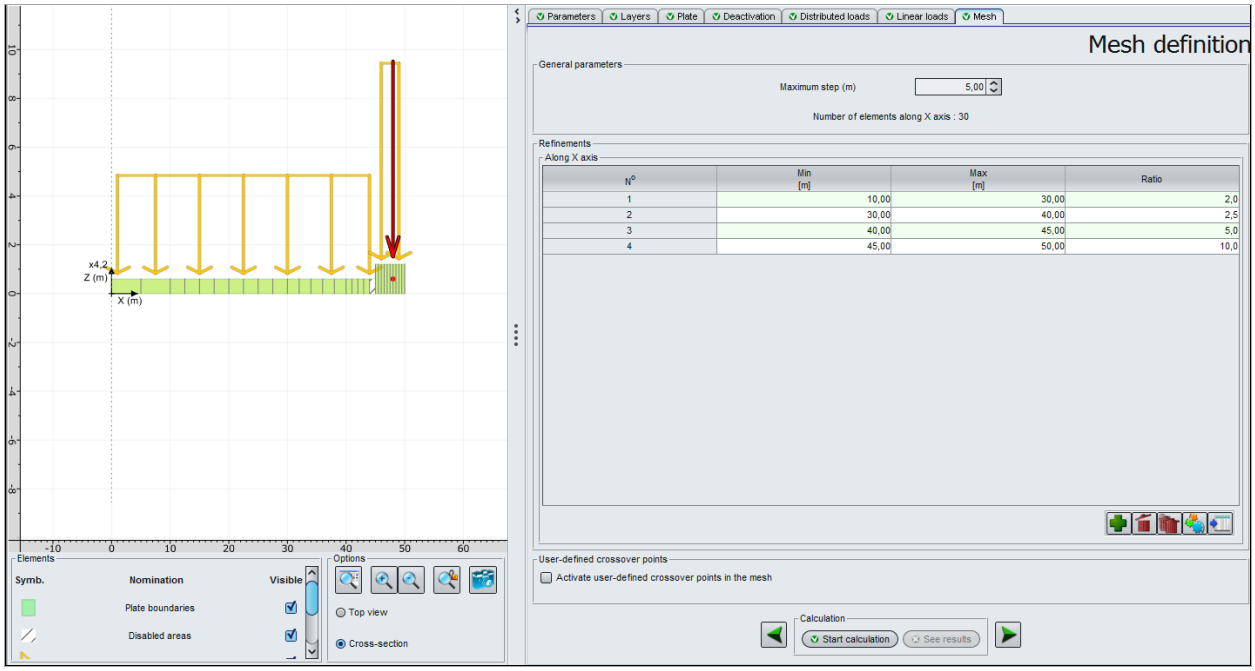


The input data are the following:

Load n°	X (m)	F_z (kN/ml)	M_r (Kn.m/ml)	K_z (kN/m/ml)	C_r (kN.m/rad/ml)
1	48	1090	0	0	0

I.4.5.1.7. "Mesh" tab

We propose the following mesh: maximum step of 5 m with a progressive refinement while getting closer to the periphery.



The input data are the following:

- Maximum step = 5 (m)

Refinement along X axis:

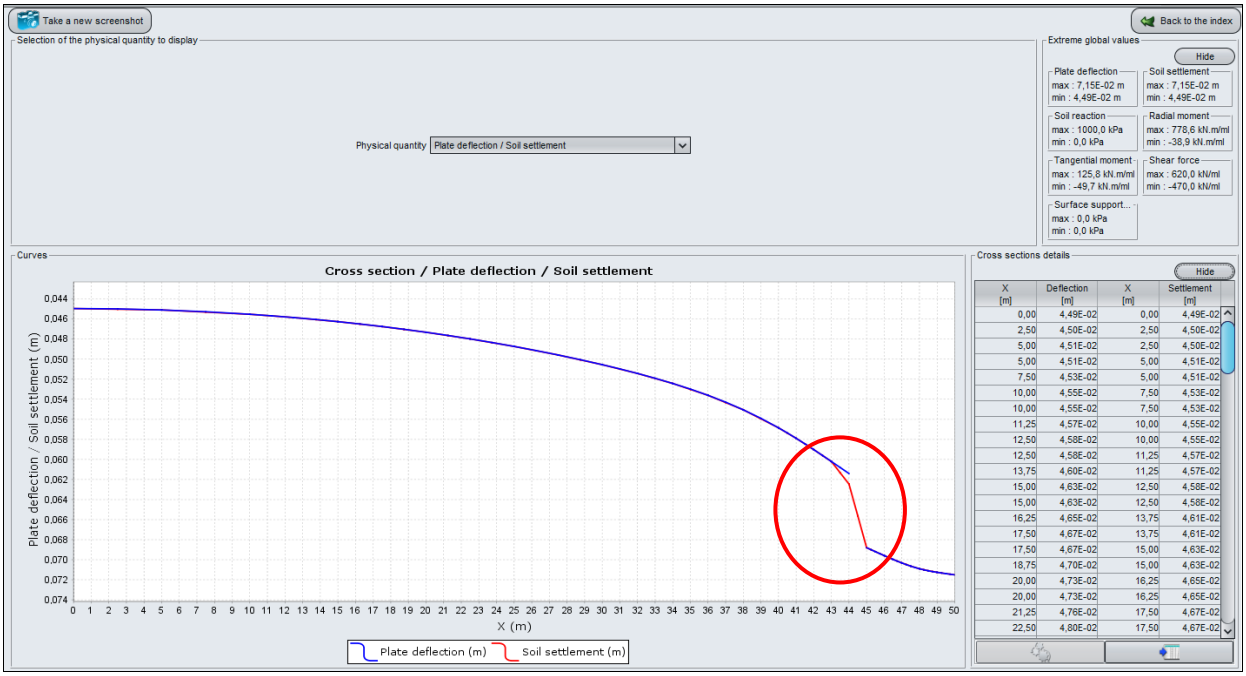
N°	Min (m)	Max (m)	Ratio
1	10.00	30.00	2.0
2	30.00	40.00	2.5
3	40.00	45.00	5.0
4	45.00	50.00	10.0

I.4.5.1.8. Graphical results – Cross-sections

Plate deflection:

By using the drop-down list located at the middle of the frame, we will display the deflection and settlement results.

The results show a localised unsticking zone close to the junction joint.



Soil settlement:

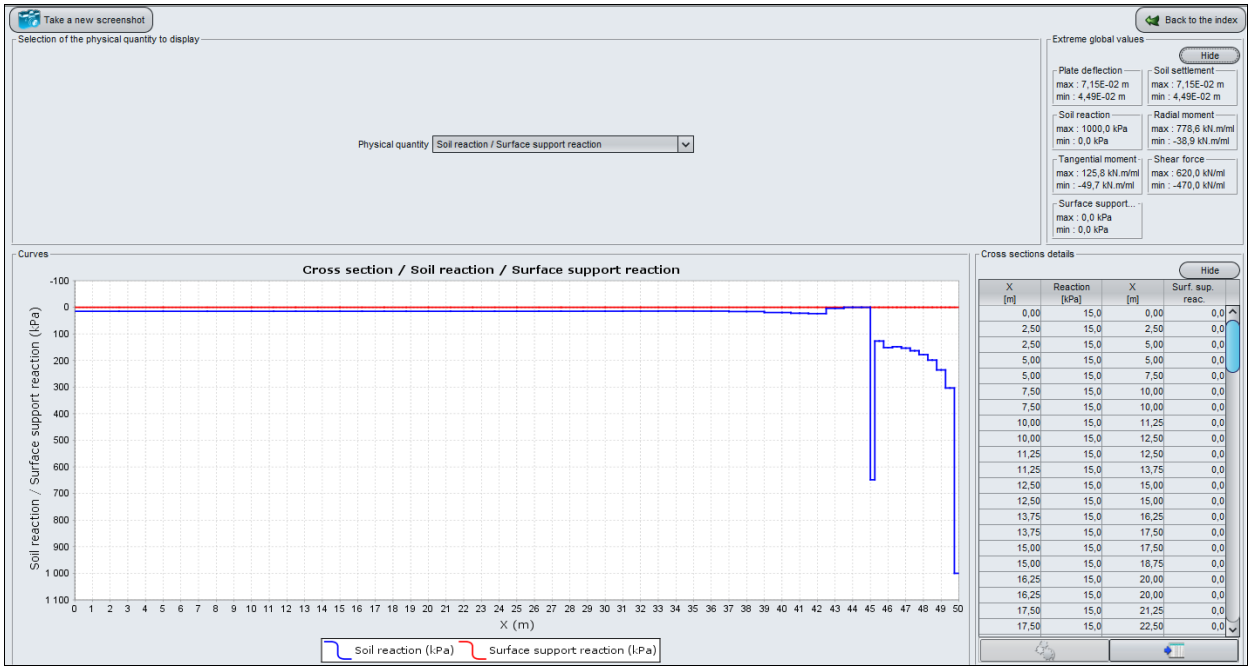
Extreme global values:

- Max: 7.15E-02 m (thus 7.2 cm)
- Min: 4.49E-02 m (thus 4.5 cm)

The examination of the deflection on both sides of the junction allows the determination of the camber to anticipate, which eases the casing-pouring operations during the later junction joint installation. In the case of this example, the camber value is about one millimetre.

Soil reaction / Surface support reaction:

By using the drop-down list located in the middle of the frame, we will shift to reaction results.



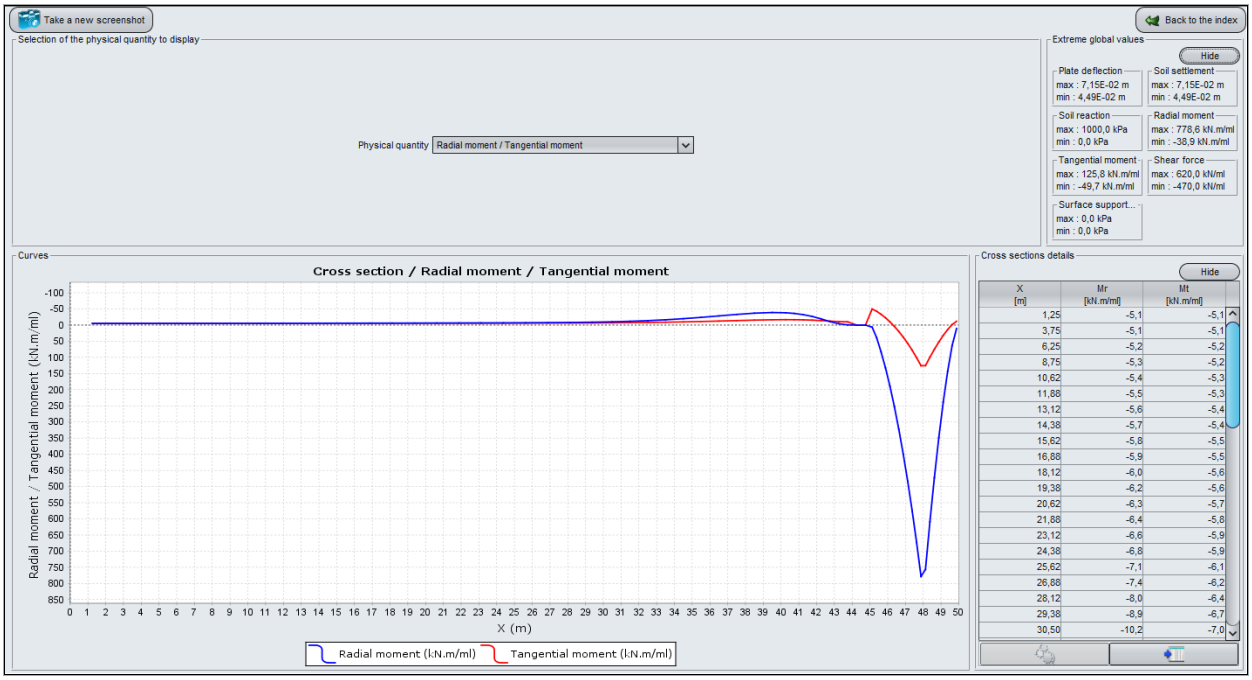
The calculated results demonstrate an edge effect resulting in "peaks of reaction" on the two extremities of the peripheral part of the raft. The more rigid the raft is, the greater these peaks of

reaction will be. In the central part, the soil reaction is almost uniform and its value is 15 kPa, which corresponds to the load applied on the plate surface.

Please note that at the edge of the tank, the calculated reaction is 1000 kPa (maximum value input previously); this corresponds to a localised plastification in this zone.

Radial moment / Tangential moment:

Finally, we can display the curves of radial and tangential moments.



The radial moment turns out to be preponderant. The calculated extreme global values are the following:

Radial moment:

- Max = 779 kN.m/ml
- Min = -39 kN.m/ml

Tangential moment:

- Max = 126 kN.m/ml
- Min = -50 kN.m/ml

I.4.5.2. Step 2: "Hydrotest" stage

We propose at this stage to carry out the same calculation for the hydrotest stage.

"Save as" the project with another name, "Circular tank2" for example.

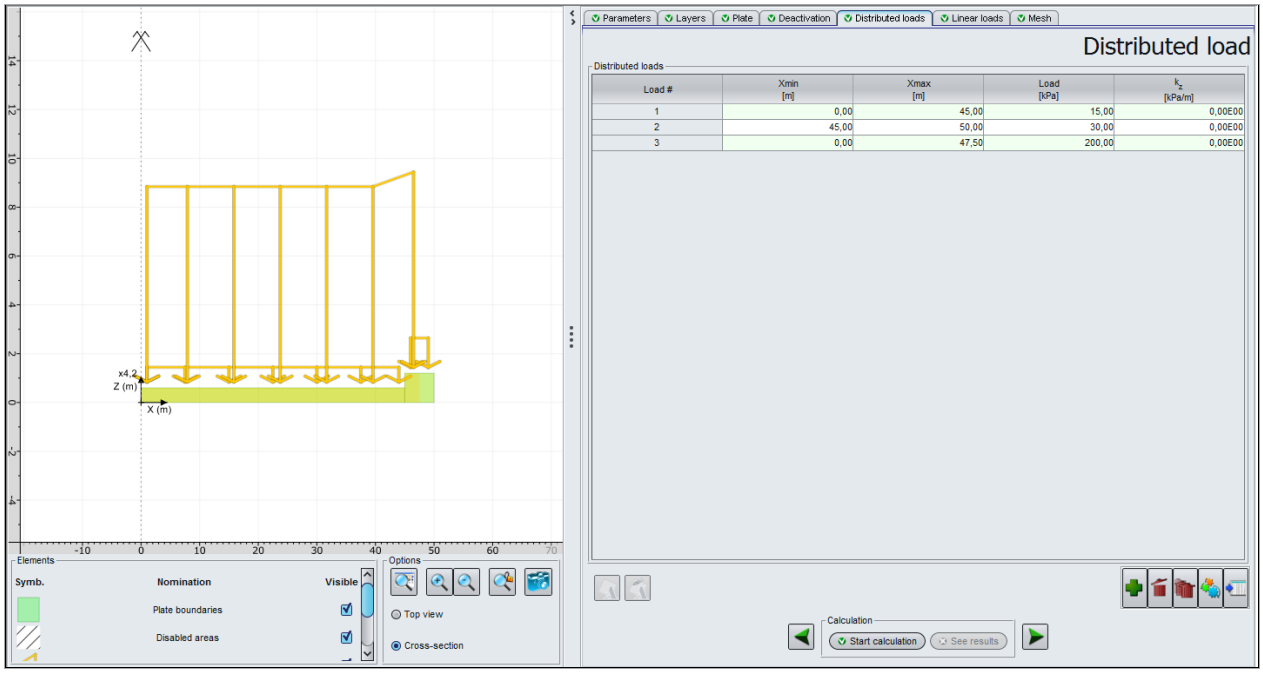
I.4.5.2.1. "Deactivation" tab

In the hydrotest stage, the junction will already exist and thus the joint between the central part and the peripheral part will be filled in. No zone shall be deactivated.

- Select the line and click on the "Delete" button .

I.4.5.2.2. "Distributed loads" tab

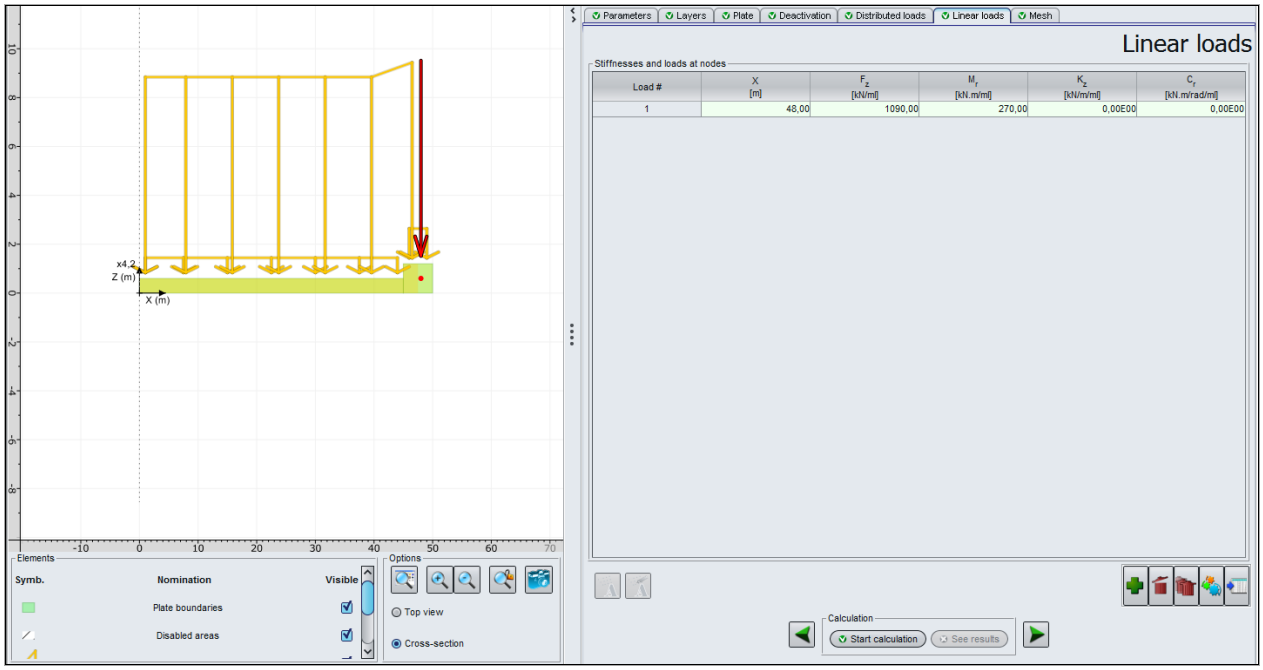
Add a third load of 200 kPa, which corresponds to the hydrotest load, between Xmin = 0 and Xmax = 47.5 m (interior of the tank).



Load n°	Xmin (m)	Xmax (m)	Load (kPa)	k_z (kPa/m)
1	0.00	45.00	15	0
2	45.00	50.00	30	0
3	**0.00**	**47.50**	**200**	**0**

I.4.5.2.3. "Linear loads" tab

While the tank is getting filled, a hydrostatic pressure is applied on the interior side of the peripheral walls. This load results in the creation of a bending moment which is transmitted at the base of the wall at the junction area with the raft. This fact will be taken into account by introducing a linear radial moment, applied at X = 48 m. The value to input is M_r = 270 kN.m/ml.

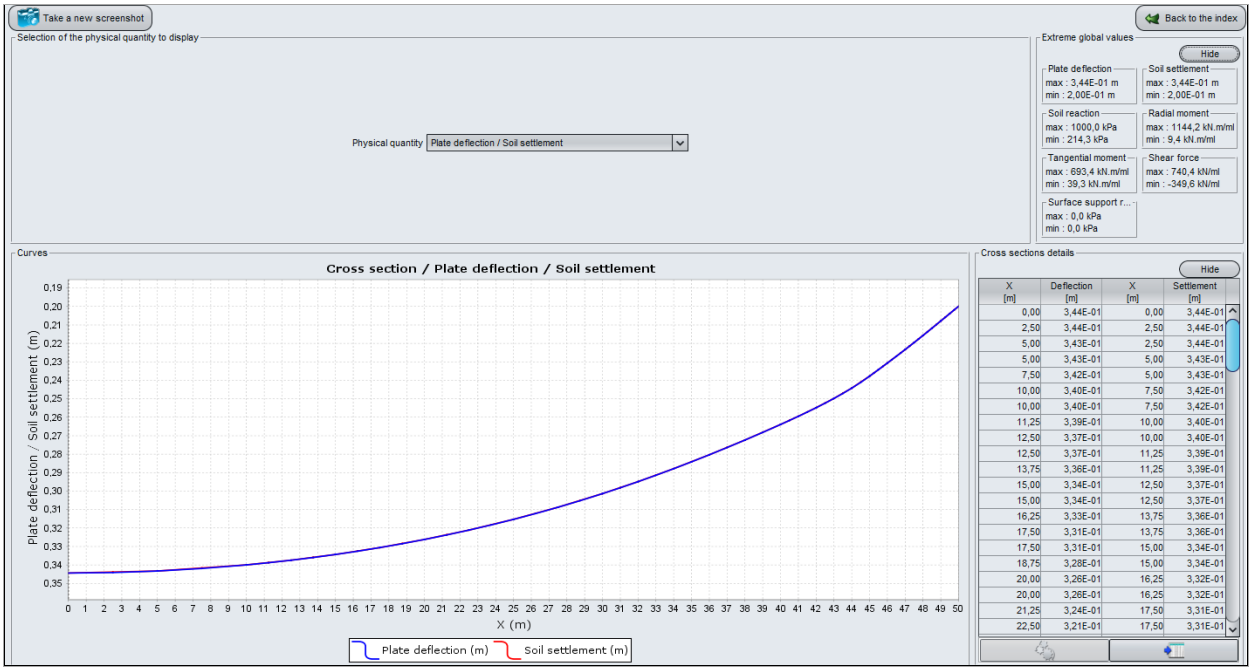


Load n°	X (m)	F_z (kN/ml)	Mr (kN.m/ml)	K_z (kN/m/ml)	C_r (kN.m/rad/ml)
1	48	1090	270	0	0

I.4.5.2.4. Graphical results - Cross-sections

Plate deflection / Soil settlement:

By using the drop-down list located in the middle of the frame, we will display the deflection / settlement results.



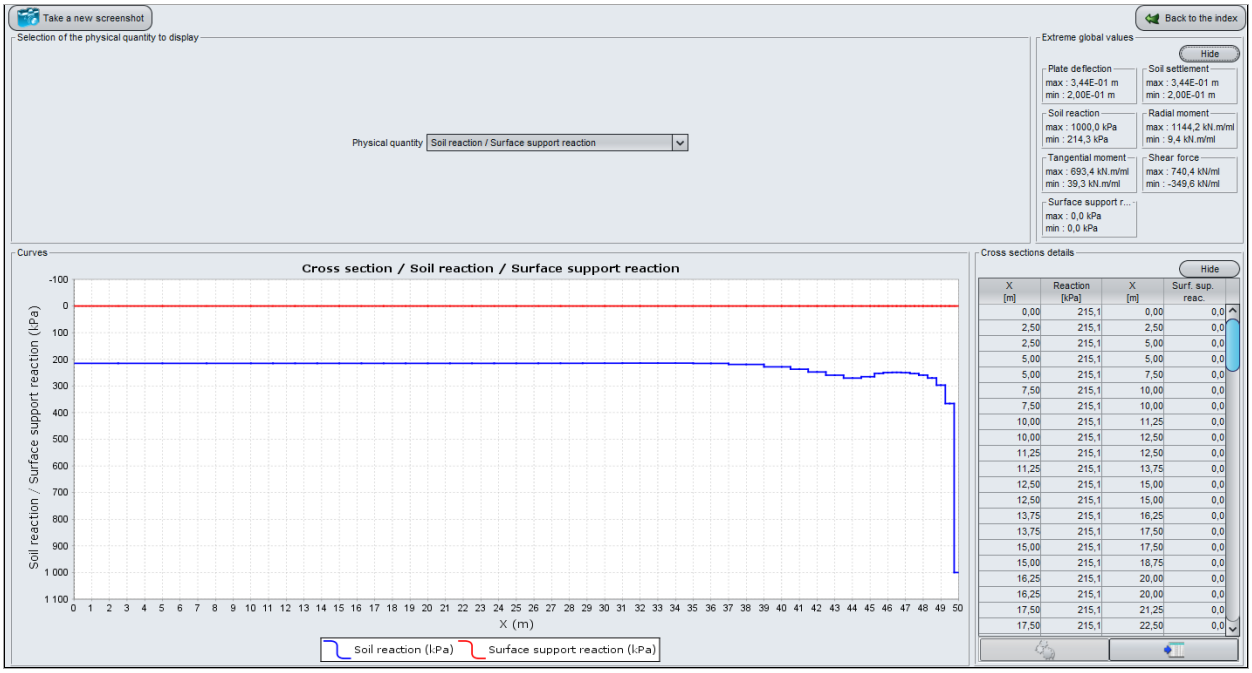
The plate deflection and soil settlement values are strictly identical; from that we can deduce that there is no unsticking. We summarise hereafter the global extreme values (deflection and soil settlement):

- Max: 3.44E-01 m (thus 34.4 cm)
- Min: 2.00E-01 m (thus 20.0 cm)

We deduce a differential settlement of 15 cm between the centre and the edge of the tank.

Soil reaction / Surface support reaction:

By using the drop-down list located in the middle of the frame, we will display the reaction results.



We observe that the soil reaction value is almost uniform (about 215 kPa) under the central part. The edge effect at the extremity of the peripheral raft results in a peak of reaction which is capped at 1000 kPa (input data).

The calculated extreme global values are given below.

Soil reaction:

- Max = 1000 kPa
- Min = 214 kPa

Surface support reaction: not applicable

Radial and tangential moment: Finally, we display the charts of radial and tangential moments.

The extreme global values are the following:

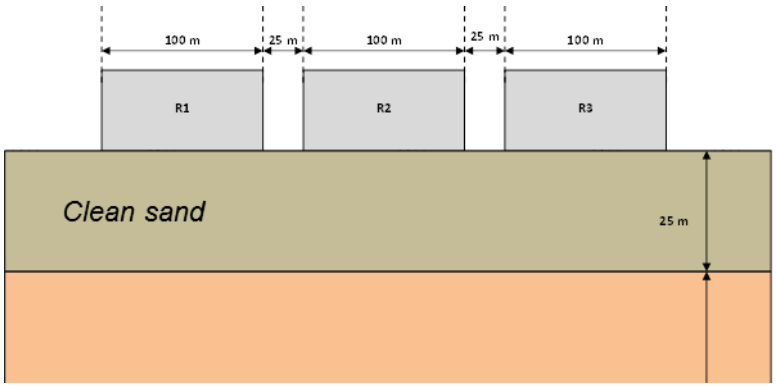
Radial moment:

- Max = 1144 kN.m/ml
- Min = 9 kN.m/ml

Tangential moment:

- Max = 693 kN.m/ml
- Min = 39 kN.m/ml

This example could be extended, for example, to the case of settlements under several neighbouring tanks, taking account of the interaction between the tanks by exporting the interaction stresses into the Tasseldo module.

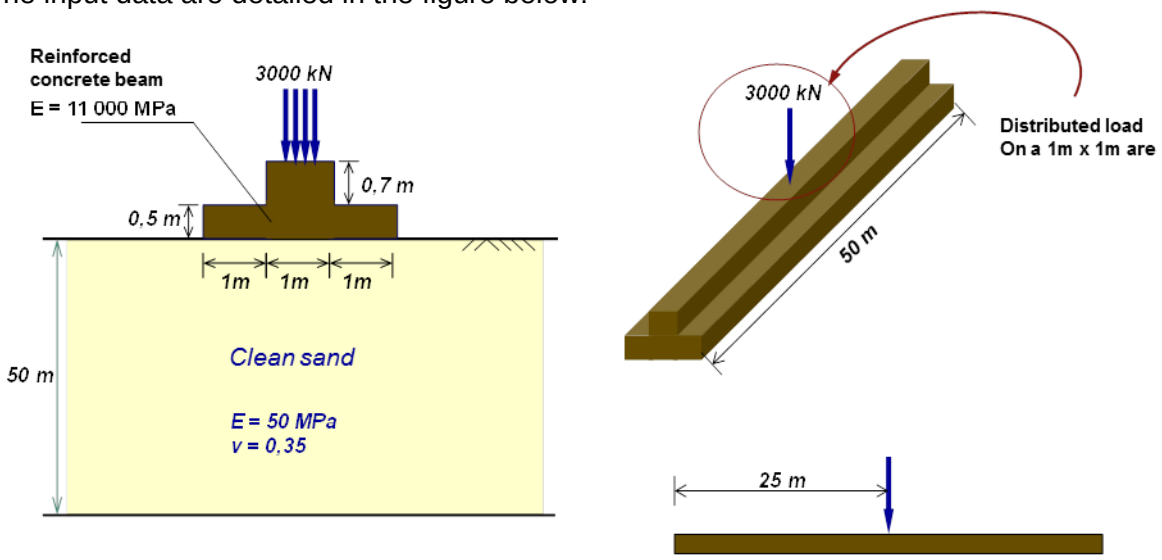


I.4.6. Example 6: Girder cranes and setting of the subgrade reaction coefficient

I.4.6.1. Step 1: Locally loaded footing with the consideration of the dead load

The study case of this example consists in a 50 m long, 3 m large footing subjected to a load of 3000 kN applied on the central beam. The footing lays on a homogeneous elastic soil body.

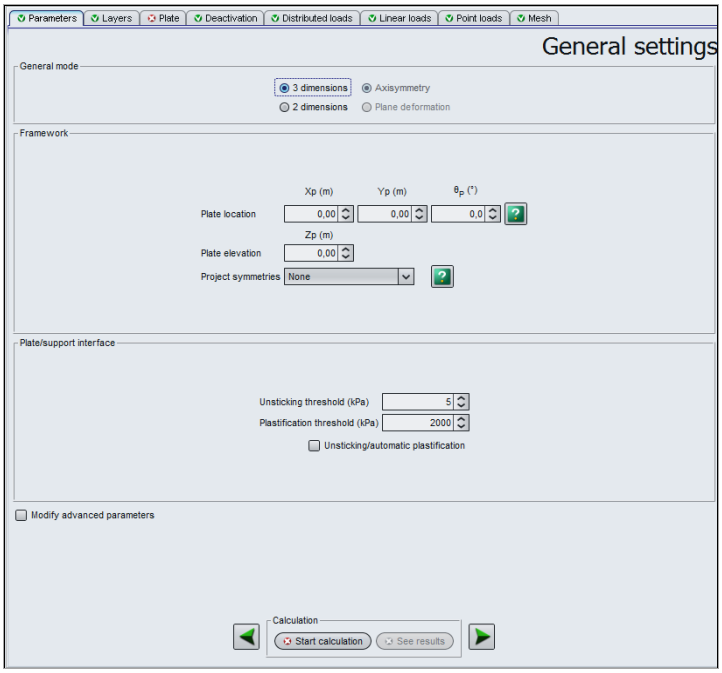
The input data are detailed in the figure below:



Create a new Foxta v3 project and select TASPLAQ module.

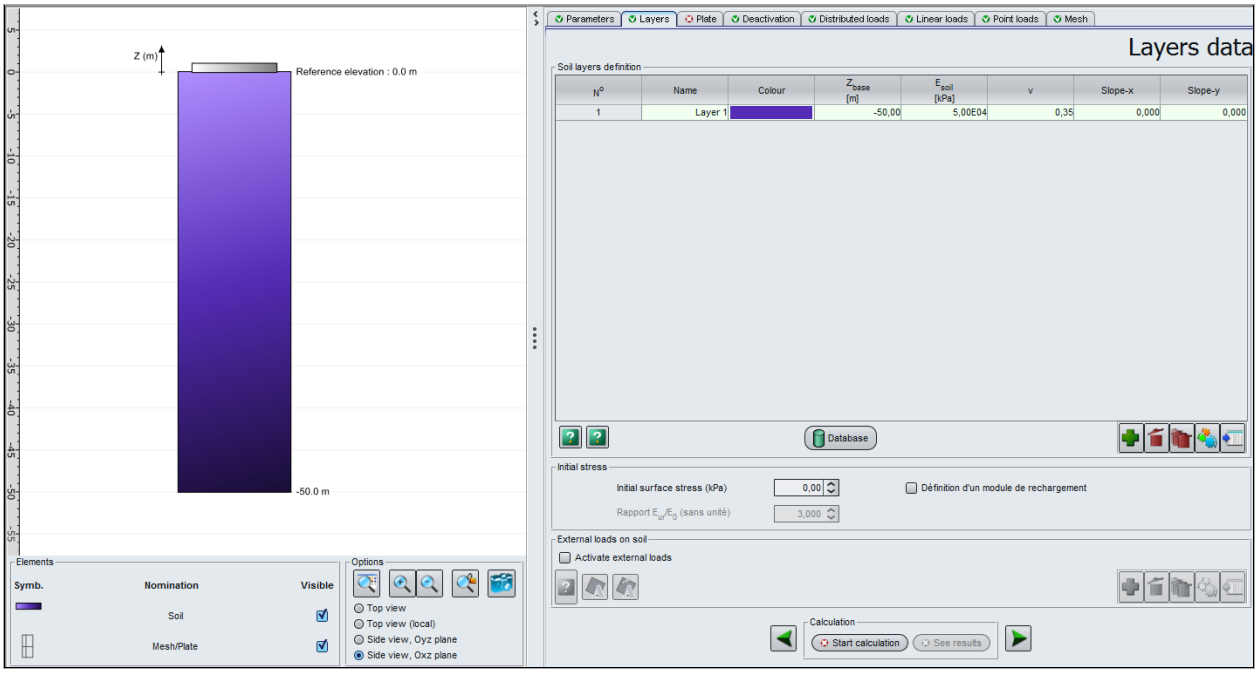
I.4.6.1.1. "Parameters" tab

Complete the input data according to the screenshot below (we keep here by default options).



I.4.6.1.2. "Layers" tab

Complete the input data according to the screenshot and the table below.

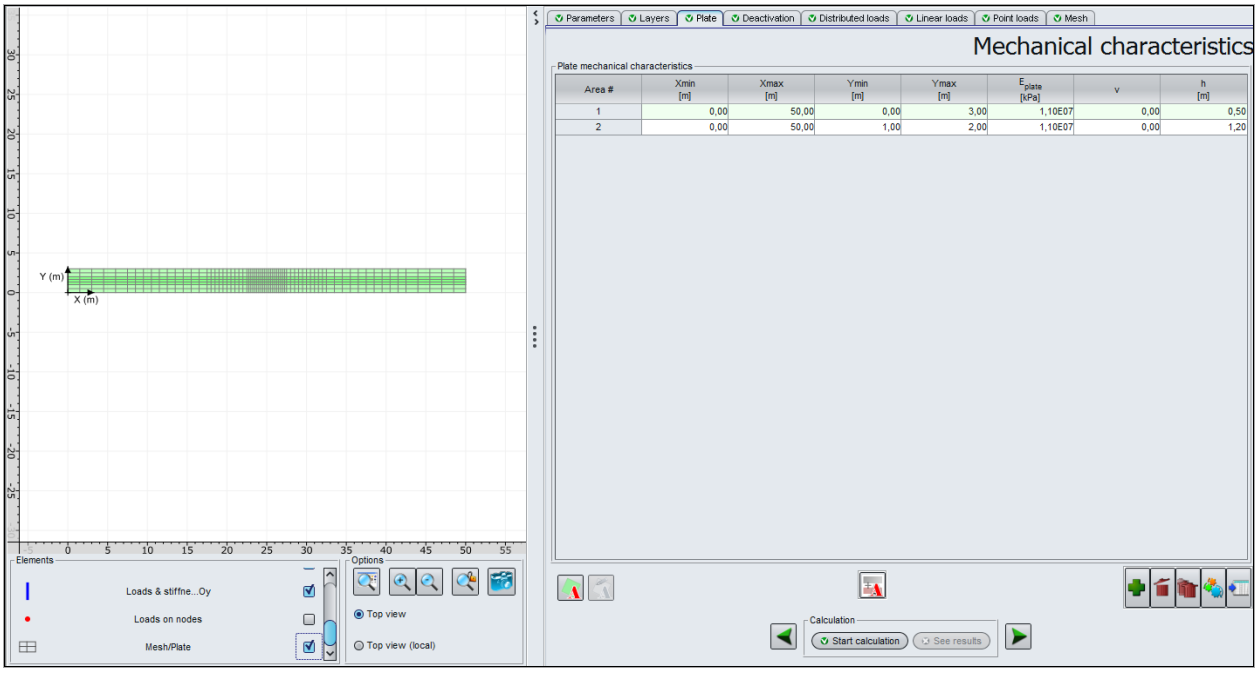


The input data are the following:

Name	Z_{base} (m)	E_{soil} (kPa)	ν	Slope-x	Slope-y
Layer1	-50	5.00E04	0.35	0.00	0.00

I.4.6.1.3. "Plate" tab

Several solutions are possible to define the mechanical characteristics of the footing in function of its different zones. Here, the most intuitive one consists in distinguishing two zones: first we define the characteristics of the "thin" plate, that we assign to the whole plate; then we overwrite (replace) these characteristics in the zone of the central beam, i.e. "thick" plate.



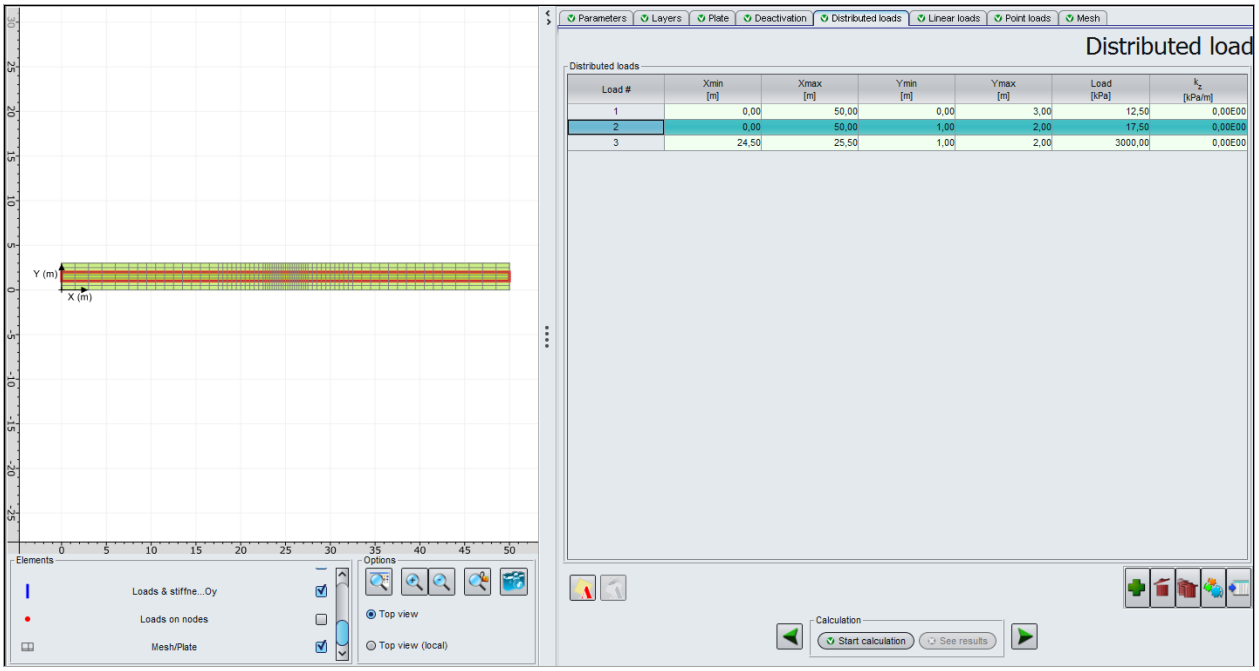
Zone n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	50.00	0.00	3.00	1.10E+07	0.00	0.5
2	0.00	50.00	1.00	2.00	1.10E+07	0.00	1.2

I.4.6.1.4. "Distributed loads" tab

The dead load of the beam is to be taken into account.

Three loading zones are to be defined:

- 12.50 kPa load, to assign to a zone covering the entire plate (Xmin = 0.00 to Xmax = 50.00 m and from Ymin = 0.00 to Ymax = 3.00 m) (dead load of the 0.5 m thick beam);
- 17.50 kPa load, to assign to the central zone, thus from Xmin = 0 to Xmax = 50 m and from Ymin = 1 to Ymax = 2 m (supplement of the dead load in the central zone, corresponding to the extra thickness of 0.7 m);
- 3000 kPa load to assign to the zone located between Xmin = 24.50 to Xmax = 25.50 m and from Ymin = 1.00 to Ymax = 2.00 m (1 m x 1 m loaded zone).



Load n°	Xmin (m)	Xmax (m)	Ymin (m)	Ymax (m)	Load (kPa)	k_z (kPa/m)
1	0.00	50.00	0.00	3.00	12.5	0
2	0.00	50.00	1.00	2.00	17.5	0
3	24.50	25.50	1.00	2.00	3000	0

I.4.6.1.5. "Linear loads" tab

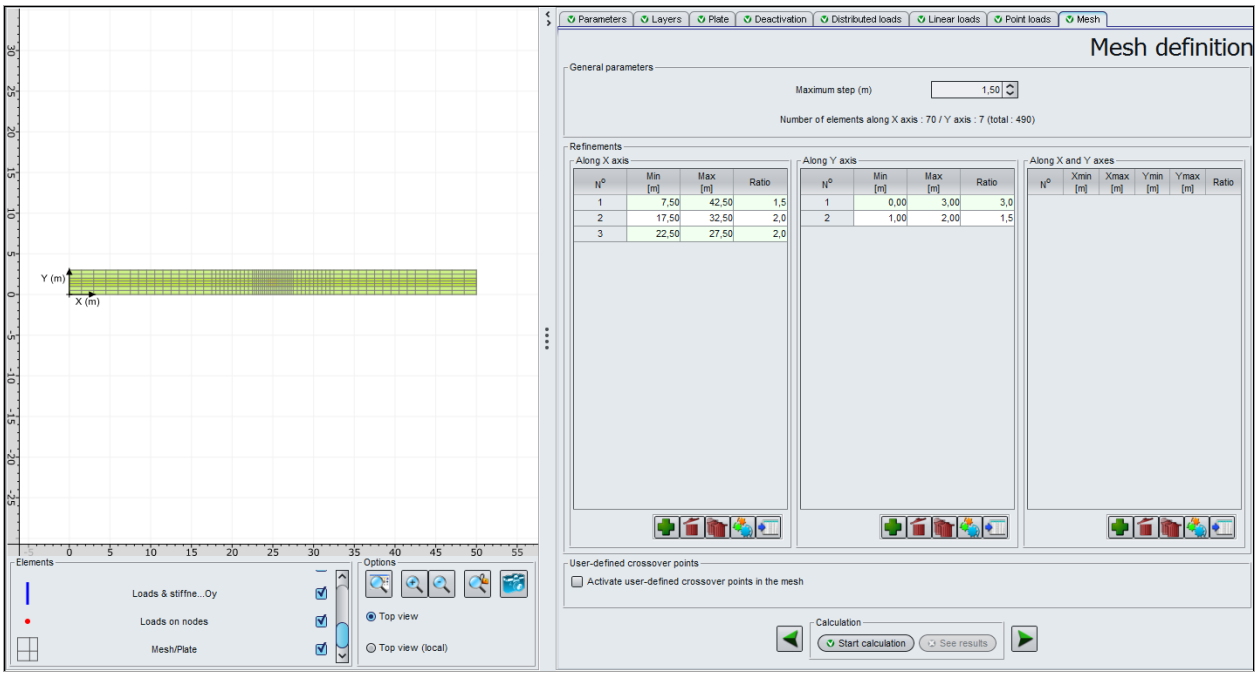
There is no linear load in this example.

I.4.6.1.6. "Point loads" tab

There is no point load in this example.

I.4.6.1.7. "Mesh" tab

We propose the following mesh: maximum step of 1.50 m with a progressive refinement around the local load. Please note that when several refinement zones are superimposed, the ratio taken into account in the calculation is the multiplication of the ratios defined for each zone.



The input data are the following:

- Maximum step = 1.50 (m)

Refinement along X axis:

N°	Min (m)	Max (m)	Ratio
1	7.50	42.50	1.5
2	17.50	32.50	2.0
3	22.50	27.50	2.0

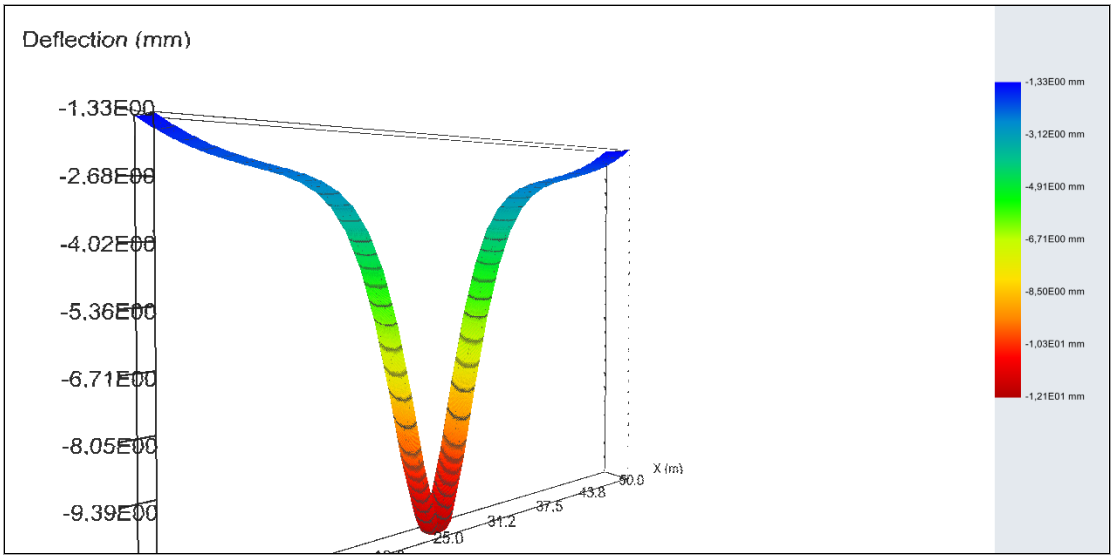
Refinement along Y axis:

N°	Min (m)	Max (m)	Ratio
1	0.00	3.00	3.0
2	1.00	2.00	1.5

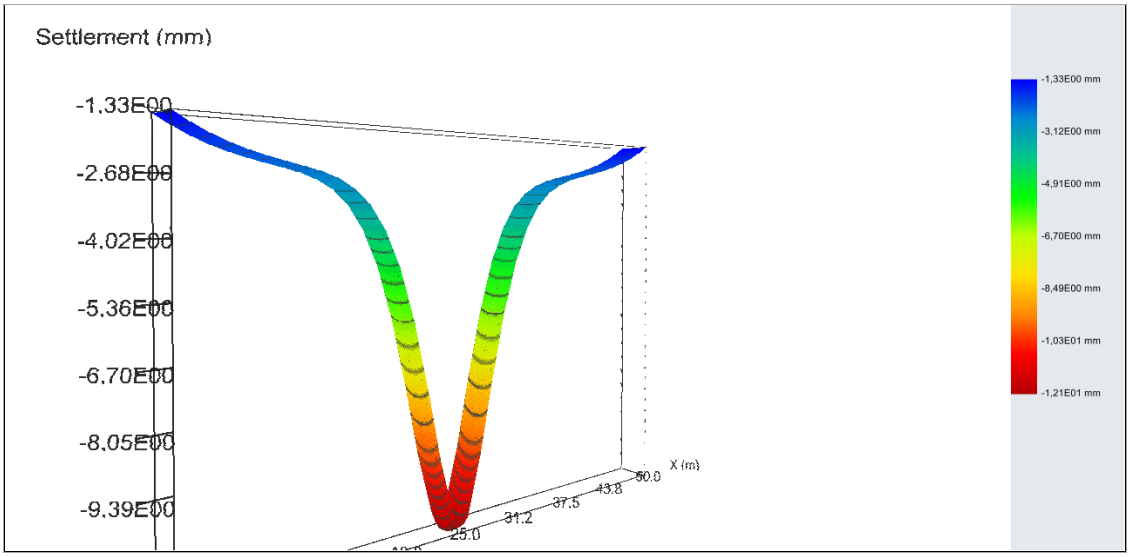
I.4.6.1.8. Results

3D Graphical results

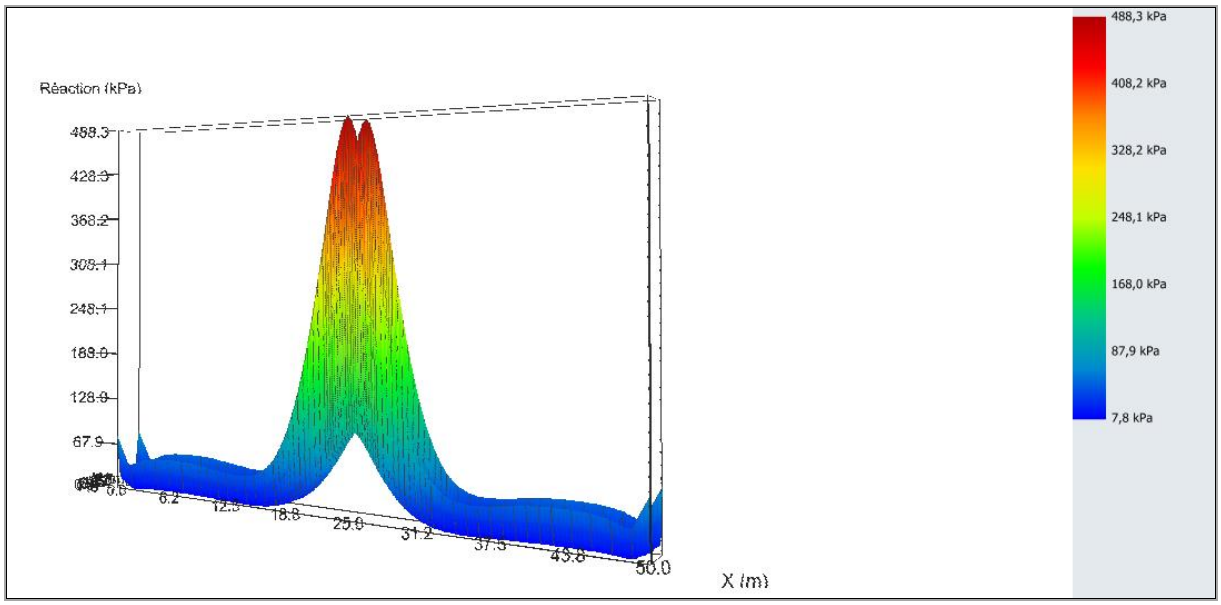
Deflection of the plate



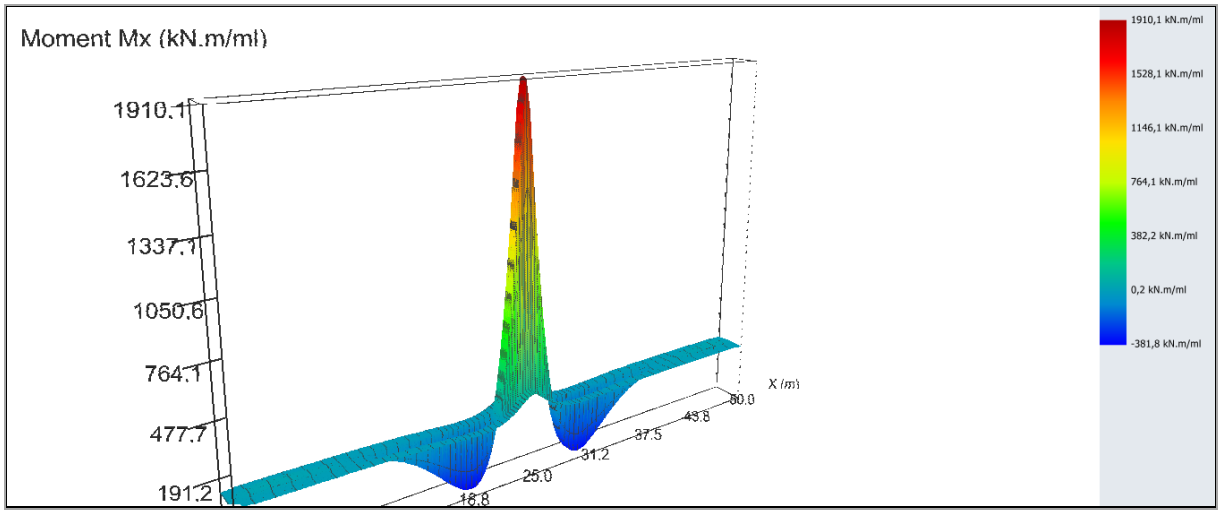
Soil settlement (mm)



Soil reaction (kPa)



Mx Moment (kN.m/ml)



Graphical 3D representation of the bending moment M_x shows that the latest is produced mainly in the central beam.

Graphical results - Cross-sections

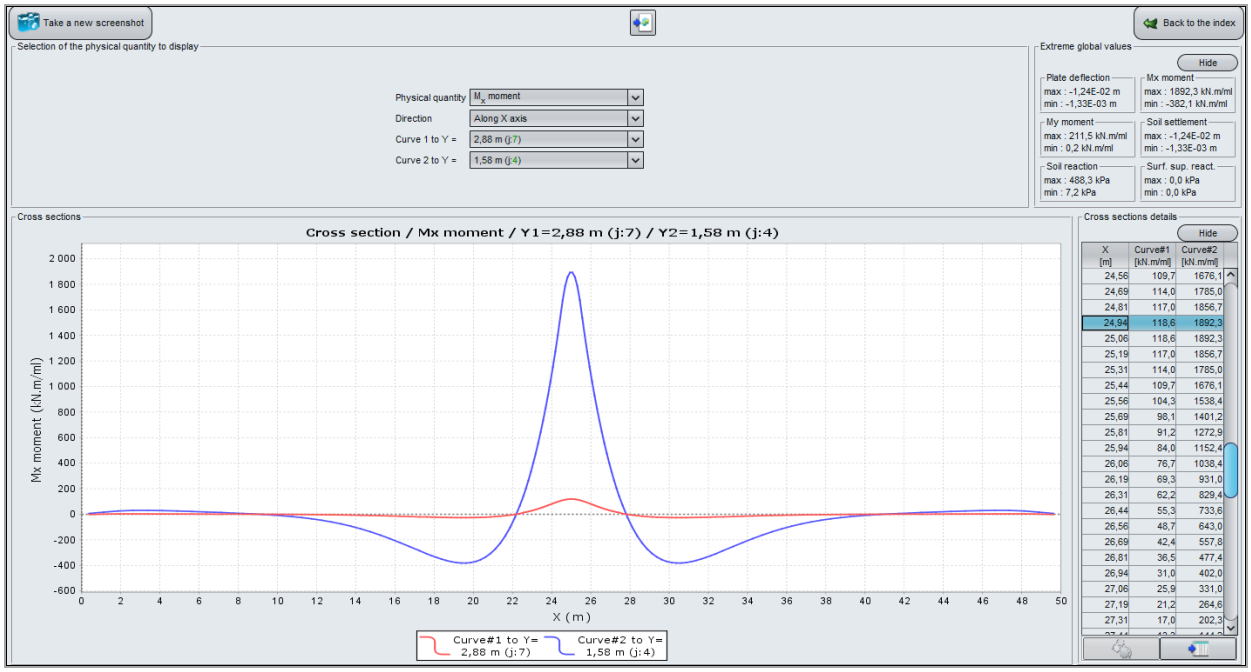
Mx Moment:

Extreme global values:

- Max: 1892 kN.m/ml
- Min: -382 kN.m/ml

Cross-section graph along X axis with:

Cross-section 1 at Y = 2.88 m (j: 7):	at X = 24.94 m	M_x = 119 kN.m/ml.
Cross-section 2 at Y = 1.58 m (j: 4):	at X = 24.94 m	M_x = 1892 kN.m/ml.



Soil reaction along X

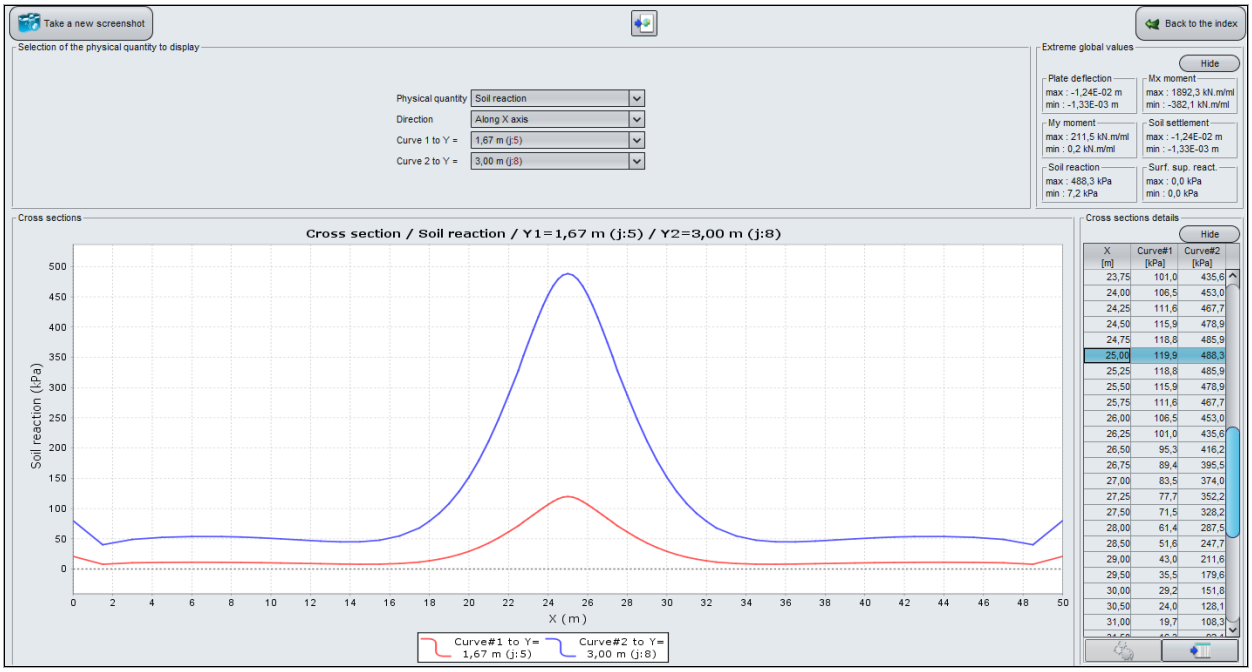
Extreme global values:

- Max: 488 kPa
- Min: 7 kPa

Cross-section graph along X axis with:

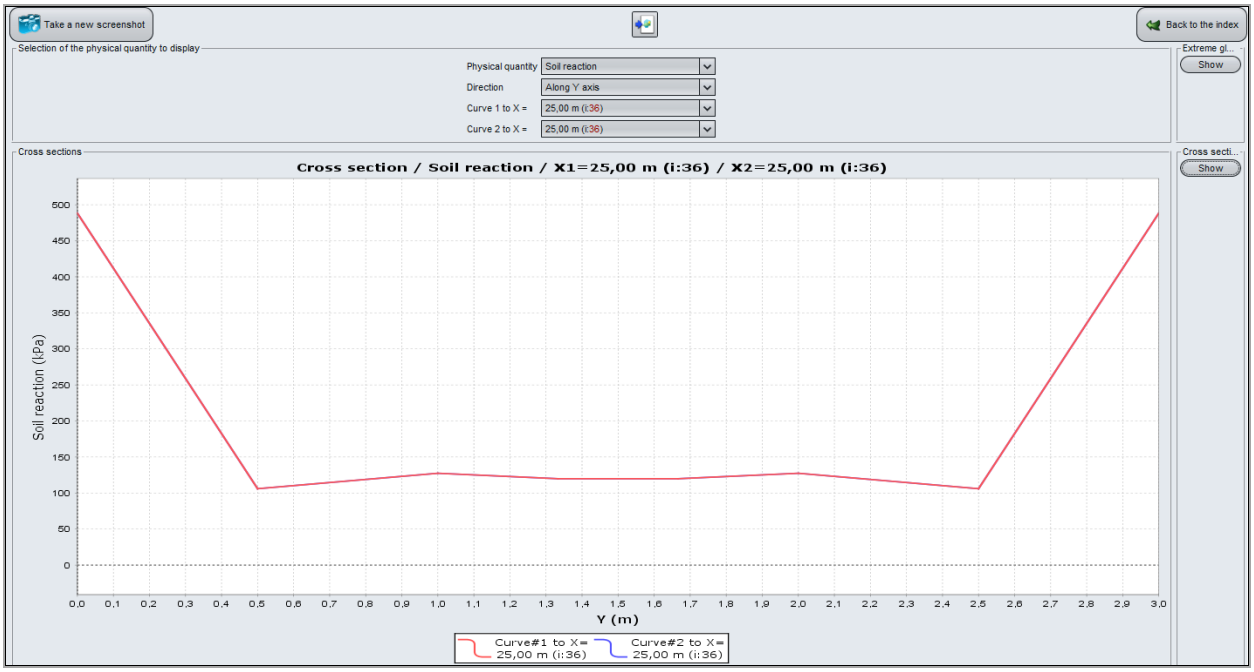
Cross-section 1 at Y = 1.67 m (j: 5): at X = 25.0 m Reaction = 120 kPa.

Cross-section 2 at Y = 3.00 m (j: 8): at X = 25.0 m Reaction = 488 kPa.



Soil reaction along Y

We notice on the cross-section below that the soil reaction is almost uniform between Y = 0.5 m and Y = 2.5 m (we can easily distinguish the influence of the discretisation of the mesh along Y axis).



Formatted numerical results

Reaction and settlement at the centre of the footing:

Vertical stress (p) and settlement (s) under nodes (global coordinate system)

node	X	Y	status	stress	settlement	deflection
001,001	0.000	0.000	***	79.988	0.133234E-02	0.133234E-02
002,001	1.500	0.000	***	40.181	0.158784E-02	0.158784E-02
003,001	3.000	0.000	***	49.039	0.180975E-02	0.180975E-02
004,001	4.500	0.000	***	52.397	0.198536E-02	0.198536E-02
005,001	6.000	0.000	***	53.767	0.212375E-02	0.212375E-02
006,001	7.500	0.000	***	53.680	0.223636E-02	0.223636E-02
007,001	8.500	0.000	***	52.954	0.230297E-02	0.230297E-02
008,001	9.500	0.000	***	51.604	0.236639E-02	0.236639E-02
009,001	10.500	0.000	***	49.927	0.243294E-02	0.243294E-02
010,001	11.500	0.000	***	48.039	0.250948E-02	0.250948E-02
011,001	12.500	0.000	***	46.214	0.260576E-02	0.260576E-02
012,001	13.500	0.000	***	44.947	0.273552E-02	0.273552E-02
013,001	14.500	0.000	***	45.045	0.291772E-02	0.291772E-02
014,001	15.500	0.000	***	47.733	0.317730E-02	0.317730E-02
015,001	16.500	0.000	***	54.754	0.354535E-02	0.354535E-02
016,001	17.500	0.000	***	67.935	0.405935E-02	0.405935E-02
017,001	18.000	0.000	***	79.065	0.438316E-02	0.438316E-02
018,001	18.500	0.000	***	92.094	0.475653E-02	0.475653E-02
019,001	19.000	0.000	***	108.320	0.518377E-02	0.518377E-02
020,001	19.500	0.000	***	128.104	0.566762E-02	0.566762E-02
021,001	20.000	0.000	***	151.774	0.620900E-02	0.620900E-02
022,001	20.500	0.000	***	179.567	0.680645E-02	0.680645E-02
023,001	21.000	0.000	***	211.572	0.745529E-02	0.745529E-02
024,001	21.500	0.000	***	247.651	0.814663E-02	0.814663E-02
025,001	22.000	0.000	***	287.492	0.886637E-02	0.886637E-02
026,001	22.500	0.000	***	328.215	0.959439E-02	0.959439E-02
027,001	22.750					
028,001	23.000					
029,001	23.250					
030,001	23.500					
031,001	23.750					
032,001	24.000					
033,001	24.250					
034,001	24.500					
035,001	24.750					
036,001	25.000					
037,001	25.250					
038,001	25.500					
039,001	25.750					
040,001	26.000					
041,001	26.250					
042,001	26.500					
043,001	26.750					
044,001	27.000					
045,001	27.250					
046,001	27.500					
047,001	28.000					
048,001	28.500					
049,001	29.000					
050,001	29.500					
051,001	30.000					
052,001	30.500					
053,001	31.000					
054,001	31.500					
055,001	32.000					
054,003	31.500	1.000	***	17.955	0.481616E-02	0.481616E-02
055,003	32.000	1.000	***	15.073	0.443270E-02	0.443270E-02
056,003	32.500	1.000	***	12.700	0.410146E-02	0.410146E-02
057,003	33.500	1.000	***	10.108	0.357656E-02	0.357656E-02
058,003	34.500	1.000	***	8.904	0.320181E-02	0.320181E-02
059,003	35.500	1.000	***	8.683	0.293927E-02	0.293927E-02
060,003	36.500	1.000	***	8.996	0.275636E-02	0.275636E-02
061,003	37.500	1.000	***	9.553	0.262710E-02	0.262710E-02
062,003	38.500	1.000	***	10.174	0.253187E-02	0.253187E-02
063,003	39.500	1.000	***	10.756	0.245649E-02	0.245649E-02
064,003	40.500	1.000	***	11.246	0.239091E-02	0.239091E-02
065,003	41.500	1.000	***	11.620	0.232800E-02	0.232800E-02
066,003	42.500	1.000	***	11.895	0.226257E-02	0.226257E-02
067,003	44.000	1.000	***	12.015	0.215043E-02	0.215043E-02
068,003	45.500	1.000	***	11.749	0.201061E-02	0.201061E-02
069,003	47.000	1.000	***	11.066	0.183283E-02	0.183283E-02
070,003	48.500	1.000	***	8.524	0.161287E-02	0.161287E-02
071,003	50.000	1.000	***	22.175	0.136375E-02	0.136375E-02
001,004	0.000	1.333	***	20.964	0.136469E-02	0.136469E-02
002,004	1.500	1.333	***	7.905	0.161353E-02	0.161353E-02
003,004	3.000	1.333	***	10.364	0.183344E-02	0.183344E-02
004,004	4.500	1.333	***	10.992	0.201130E-02	0.201130E-02
005,004	6.000	1.333	***	11.230	0.215116E-02	0.215116E-02
006,004	7.500	1.333	***	11.108	0.226328E-02	0.226328E-02
007,004	8.500	1.333	***	10.846	0.232868E-02	0.232868E-02
008,004	9.500	1.333	***	10.485	0.239157E-02	0.239157E-02
009,004	10.500	1.333	***	10.014	0.245712E-02	0.245712E-02
010,004	11.500	1.333	***	9.452	0.253247E-02	0.253247E-02
011,004	12.500	1.333	***	8.846	0.262766E-02	0.262766E-02
012,004	13.500	1.333	***	8.289	0.275690E-02	0.275690E-02
013,004	14.500	1.333	***	7.946	0.293983E-02	0.293983E-02
014,004	15.500	1.333	***	8.089	0.320244E-02	0.320244E-02
015,004	16.500	1.333	***	9.135	0.357739E-02	0.357739E-02
016,004	17.500	1.333	***	11.467	0.410261E-02	0.410261E-02
017,004	18.000	1.333	***	13.651	0.443407E-02	0.443407E-02
018,004	18.500	1.333	***	16.303	0.481783E-02	0.481783E-02
019,004	19.000	1.333	***	19.714	0.525781E-02	0.525781E-02
020,004	19.500	1.333	***	23.985	0.575686E-02	0.575686E-02
021,004	20.000	1.333	***	29.227	0.631610E-02	0.631610E-02
022,004	20.500	1.333	***	35.533	0.693423E-02	0.693423E-02
023,004	21.000	1.333	***	42.974	0.760671E-02	0.760671E-02
024,004	21.500	1.333	***	51.576	0.832465E-02	0.832465E-02
025,004	22.000	1.333	***	61.351	0.907370E-02	0.907370E-02
026,004	22.500	1.333	***	71.537	0.983265E-02	0.983265E-02
027,004	22.750	1.333	***	77.722	0.102070E-01	0.102070E-01
028,004	23.000	1.333	***	83.515	0.105721E-01	0.105721E-01
029,004	23.250	1.333	***	89.402	0.109228E-01	0.109228E-01
030,004	23.500	1.333	***	95.280	0.112530E-01	0.112530E-01
031,004	23.750	1.333	***	101.042	0.115559E-01	0.115559E-01
032,004	24.000	1.333	***	106.548	0.118242E-01	0.118242E-01
033,004	24.250	1.333	***	111.602	0.120491E-01	0.120491E-01
034,004	24.500	1.333	***	115.906	0.122213E-01	0.122213E-01
035,004	24.750	1.333	***	118.850	0.123301E-01	0.123301E-01
036,004	25.000	1.333	***	119.876	0.123673E-01	0.123673E-01
037,004	25.250	1.333	***	118.850	0.123301E-01	0.123301E-01
038,004	25.500	1.333	***	115.906	0.122213E-01	0.122213E-01
039,004	25.750	1.333	***	111.602	0.120491E-01	0.120491E-01
040,004	26.000	1.333	***	106.548	0.118242E-01	0.118242E-01
041,004	26.250	1.333	***	101.042	0.115559E-01	0.115559E-01
042,004	26.500	1.333	***	95.280	0.112530E-01	0.112530E-01

Maximum moment:

Bending and torsion moments (local coordinate system)

Xm	Ym	Mx	My	Mxy
0.375	0.125	-2.439	0.209	0.046
0.375	0.375	-1.465		
1.125	0.125	1.343		
1.125	0.375	1.173		
1.875	0.125	2.695		
1.875	0.375	2.554		
2.625	0.125	2.439		
2.625	0.375	2.341		
3.375	0.125	2.189		
3.375	0.375	2.152		
4.125	0.125	2.048		
4.125	0.375	2.061		
4.875	0.125	1.769		
4.875	0.375	1.780		
5.625	0.125	1.536		
5.625	0.375	1.567		
6.375	0.125	1.222		
6.375	0.375	1.215		
7.125	0.125	0.385		
7.125	0.375	0.627		
7.750	0.125	1.322		
7.750	0.375	1.030		
8.250	0.125	0.476		
8.250	0.375	0.494		
8.750	0.125	0.152		
8.750	0.375	0.137		
9.250	0.125	-0.191		
9.250	0.375	-0.166		
9.750	0.125	-0.502		
9.750	0.375	-0.492		
10.250	0.125	-0.908		
10.250	0.375	-0.895		
10.750	0.125	-1.348		
10.750	0.375	-1.345		
11.250	0.125	-1.923		
11.250	0.375	-1.923		
11.750	0.125	-2.564		
11.750	0.375	-2.572		
12.250	0.125	-3.366		
12.250	0.375	-3.383		
12.750	0.125	-4.283		
12.750	0.375	-4.305		
13.250	0.125	-5.369		
13.250	0.375	-5.415		
13.750	0.125	-6.640		
13.750	0.375	-6.680		

	35.750	2.625	-8.137	2.222	-0.167
	35.750	2.875	-8.044	0.455	-0.186
	36.250	2.625	-6.680	2.167	-0.034
	36.250	2.875	-6.640	0.436	-0.053
	36.750	2.625	-5.415	2.141	0.037
	36.750	2.875	-5.369	0.423	0.037
	37.250	2.625	-4.305	2.145	0.105
	37.250	2.875	-4.283	0.416	0.104
	37.750	2.625	-3.383	2.162	0.136
	37.750	2.875	-3.366	0.413	0.144
	38.250	2.625	-2.572	2.194	0.161
	38.250	2.875	-2.564	0.414	0.169
	38.750	2.625	-1.923	2.229	0.166
	38.750	2.875	-1.923	0.415	0.177
	39.250	2.625	-1.345	2.268	0.163
	39.250	2.875	-1.348	0.418	0.174
	39.750	2.625	-0.895	2.303	0.147
	39.750	2.875	-0.908	0.420	0.155
	40.250	2.625	-0.492	2.334	0.124
	40.250	2.875	-0.502	0.422	0.132
	40.750	2.625	-0.166	2.367	0.079
	40.750	2.875	-0.191	0.425	0.089
	41.250	2.625	0.137	2.405	0.069
	41.250	2.875	0.152	0.430	0.079
	41.750	2.625	0.494	2.437	0.000
	41.750	2.875	0.476	0.435	0.008
	42.250	2.625	1.030	2.465	0.274
	42.250	2.875	1.322	0.441	0.283
	42.875	2.625	0.627	2.489	0.190
	42.875	2.875	0.385	0.446	0.194
	43.625	2.625	1.215	2.510	-0.162
	43.625	2.875	1.222	0.451	-0.158
	44.375	2.625	1.567	2.489	-0.106
	44.375	2.875	1.536	0.447	-0.119
	45.125	2.625	1.780	2.426	-0.171
	45.125	2.875	1.769	0.436	-0.184
	45.875	2.625	2.061	2.380	-0.223
	45.875	2.875	2.048	0.431	-0.227
	46.625	2.625	2.152	2.349	-0.185
	46.625	2.875	2.189	0.434	-0.190
	47.375	2.625	2.341	2.406	-0.003
	47.375	2.875	2.439	0.493	0.041
	48.125	2.625	2.554	2.550	0.356
	48.125	2.875	2.695	0.609	0.399
	48.875	2.625	1.173	2.551	1.328
	48.875	2.875	1.343	0.514	1.254
	49.625	2.625	-1.465	2.409	0.121
	49.625	2.875	-2.439	0.209	0.046
MxMax+	24.938	1.583	1892.323		
MxMax-	30.375	1.583	-382.096		
MyMax+	24.938	1.583		211.516	
MyMax-	49.625	2.875		0.209	
MxyMax+	24.312	1.917			28.772
MxyMax-	25.688	1.917			-28.772

Soil settlement:

Vertical stress (p) and settlement (s) under nodes (global coordinate system)

node	X	Y	status	stress	settlement	deflection
001,001	0.000	0.000	***	79.988	0.133234E-02	0.133234E-02
002,001	1.500	0.000	***			
003,001	3.000	0.000	***			
004,001	4.500	0.000	***			
005,001	6.000	0.000	***			
006,001	7.500	0.000	***			
007,001	8.500	0.000	***			
008,001	9.500	0.000	***			
009,001	10.500	0.000	***			
010,001	11.500	0.000	***			
011,001	12.500	0.000	***			
012,001	13.500	0.000	***			
013,001	14.500	0.000	***			
014,001	15.500	0.000	***			
015,001	16.500	0.000	***			
016,001	17.500	0.000	***			
017,001	18.000	0.000	***			
018,001	18.500	0.000	***			
019,001	19.000	0.000	***			
020,001	19.500	0.000	***			
021,001	20.000	0.000	***			
022,001	20.500	0.000	***			
023,001	21.000	0.000	***			
024,001	21.500	0.000	***			
025,001	22.000	0.000	***			
026,001	22.500	0.000	***			
027,001	22.750	0.000	***			
028,001	23.000	0.000	***			
029,001	23.250	0.000	***			
030,001	23.500	0.000	***			
031,001	23.750	0.000	***			
032,001	24.000	0.000	***			
033,001	24.250	0.000	***			
034,001	24.500	0.000	***			
035,001	24.750	0.000	***			
036,001	25.000	0.000	***			
037,001	25.250	0.000	***			
038,001	25.500	0.000	***			
039,001	25.750	0.000	***			
040,001	26.000	0.000	***			
041,001	26.250	0.000	***			
042,001	26.500	0.000	***			
043,001	26.750	0.000	***			
044,001	27.000	0.000	***			
045,001	27.250	0.000	***			
046,001	27.500	0.000	***			
047,001	28.000	0.000	***			
048,001	28.500	0.000	***			
049,001	29.000	0.000	***			
050,001	29.500	0.000	***			
051,001	30.000	0.000	***			
052,001	30.500	0.000	***			
053,001	31.000	0.000	***			
054,001	31.500	0.000	***			
055,001	32.000	0.000	***			

026,008	22.500	3.000	***	328.215	0.959439E-02	0.959439E-02
027,008	22.750	3.000	***	352.202	0.995250E-02	0.995250E-02
028,008	23.000	3.000	***	374.000	0.102999E-01	0.102999E-01
029,008	23.250	3.000	***	395.524	0.106316E-01	0.106316E-01
030,008	23.500	3.000	***	416.248	0.109416E-01	0.109416E-01
031,008	23.750	3.000	***	435.641	0.112231E-01	0.112231E-01
032,008	24.000	3.000	***	453.048	0.114686E-01	0.114686E-01
033,008	24.250	3.000	***	467.725	0.116704E-01	0.116704E-01
034,008	24.500	3.000	***	478.905	0.118211E-01	0.118211E-01
035,008	24.750	3.000	***	485.926	0.119144E-01	0.119144E-01
036,008	25.000	3.000	***	488.320	0.119459E-01	0.119459E-01
037,008	25.250	3.000	***	485.926	0.119144E-01	0.119144E-01
038,008	25.500	3.000	***	478.905	0.118211E-01	0.118211E-01
039,008	25.750	3.000	***	467.725	0.116704E-01	0.116704E-01
040,008	26.000	3.000	***	453.048	0.114686E-01	0.114686E-01
041,008	26.250	3.000	***	435.641	0.112231E-01	0.112231E-01
042,008	26.500	3.000	***	416.248	0.109416E-01	0.109416E-01
043,008	26.750	3.000	***	395.524	0.106316E-01	0.106316E-01
044,008	27.000	3.000	***	374.000	0.102999E-01	0.102999E-01
045,008	27.250	3.000	***	352.202	0.995250E-02	0.995250E-02
046,008	27.500	3.000	***	328.215	0.959439E-02	0.959439E-02
047,008	28.000	3.000	***	287.492	0.886637E-02	0.886637E-02
048,008	28.500	3.000	***	247.651	0.814663E-02	0.814663E-02
049,008	29.000	3.000	***	211.572	0.745529E-02	0.745529E-02
050,008	29.500	3.000	***	179.567	0.680645E-02	0.680645E-02
051,008	30.000	3.000	***	151.774	0.620900E-02	0.620900E-02
052,008	30.500	3.000	***	128.104	0.566762E-02	0.566762E-02
053,008	31.000	3.000	***	108.320	0.518377E-02	0.518377E-02
054,008	31.500	3.000	***	92.094	0.475653E-02	0.475653E-02
055,008	32.000	3.000	***	79.065	0.438316E-02	0.438316E-02
056,008	32.500	3.000	***	67.935	0.405935E-02	0.405935E-02
057,008	33.500	3.000	***	54.754	0.354535E-02	0.354535E-02
058,008	34.500	3.000	***	47.733	0.317730E-02	0.317730E-02
059,008	35.500	3.000	***	45.045	0.291772E-02	0.291772E-02
060,008	36.500	3.000	***	44.947	0.273552E-02	0.273552E-02
061,008	37.500	3.000	***	46.214	0.260576E-02	0.260576E-02
062,008	38.500	3.000	***	48.039	0.250948E-02	0.250948E-02
063,008	39.500	3.000	***	49.927	0.243294E-02	0.243294E-02
064,008	40.500	3.000	***	51.604	0.236639E-02	0.236639E-02
065,008	41.500	3.000	***	52.954	0.230297E-02	0.230297E-02
066,008	42.500	3.000	***	53.680	0.223636E-02	0.223636E-02
067,008	44.000	3.000	***	53.767	0.212375E-02	0.212375E-02
068,008	45.500	3.000	***	52.397	0.198536E-02	0.198536E-02
069,008	47.000	3.000	***	49.039	0.180975E-02	0.180975E-02
070,008	48.500	3.000	***	40.181	0.158784E-02	0.158784E-02
071,008	50.000	3.000	***	79.988	0.133234E-02	0.133234E-02
w_max.	25.000	1.667				0.123673E-01
w_min.	50.000	3.000				0.133234E-02
s_Max.	25.000	1.667			0.123673E-01	
s_Min.	50.000	3.000			0.133234E-02	
p_Max.	25.000	3.000		488.320		
p_Min.	48.500	2.500		7.158		

Total load on soil: 5750.00

We get:

- Maximum moment Mx: 1892 kN.m/ml
- Maximum settlement: 1.24 cm
- Soil reaction at the centre: 120 kPa approximately

Note: the reaction shall be raised at the centre of the footing, i.e. at Y = 1.5 m. However, to get that, it would be necessary to define a discretisation of, for example, 4 instead of 3 along Y within the central zone of the footing (1 m). Because of the discretisation of 3, we only have access to the results at Y = 1.33 m or Y = 1.67 m (reaction: 119.9 kPa at X = 25 m and for these values of Y). But as we previously noticed that the reaction is almost uniform in this zone of the footing, we can accept this approximation.

Save the project, then "Save as" with another name before moving to the step 2.

I.4.6.2. Step 2: Locally loaded footing – definition of the equivalent subgrade reaction coefficient

Based on the results of the previous step, we propose here to define an equivalent subgrade reaction coefficient by taking as reference the amplitude of the M_x moment previously calculated.

The proposed calculation type is "Plate on continuous elastic supports". A first estimation of the equivalent subgrade reaction coefficient can be made by using the ratio:

$$k_{z0} = q / s$$

Where s is the maximum settlement of the footing calculated in the previous step (1.24 cm), and q the subgrade reaction at the centre of the footing (120 kPa): k_{z0}= 9677 kPa/m.

Therefore, we will take over the input data of the previous example, and bring the modifications described hereafter.

I.4.6.2.1. "Layers" tab

The present calculation is "Plate on continuous elastic supports" type, where the soil is modelled as equivalent springs and not as an elastic body.

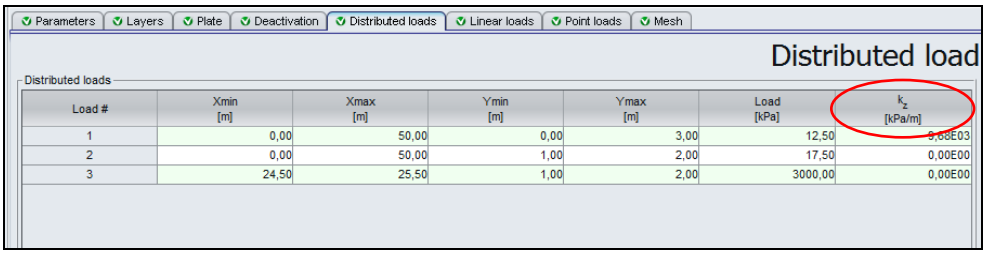
- Delete the soils layer.

I.4.6.2.2. "Distributed loads" tab

In addition to the loading sets defined in the previous step (dead load of the footing and 3000 kPa load), we should insert the subgrade reaction modulus as a distributed (surface) vertical stiffness of 9677 kPa/m.

We assign a stiffness covering the whole plate:

- k_z = 9677 kPa/m.



Load #	Xmin [m]	Xmax [m]	Ymin [m]	Ymax [m]	Load [kPa]	k_z [kPa/m]
1	0,00	50,00	0,00	3,00	12,50	9,68E03
2	0,00	50,00	1,00	2,00	17,50	0,00E00
3	24,50	25,50	1,00	2,00	3000,00	0,00E00

I.4.6.2.3. Results

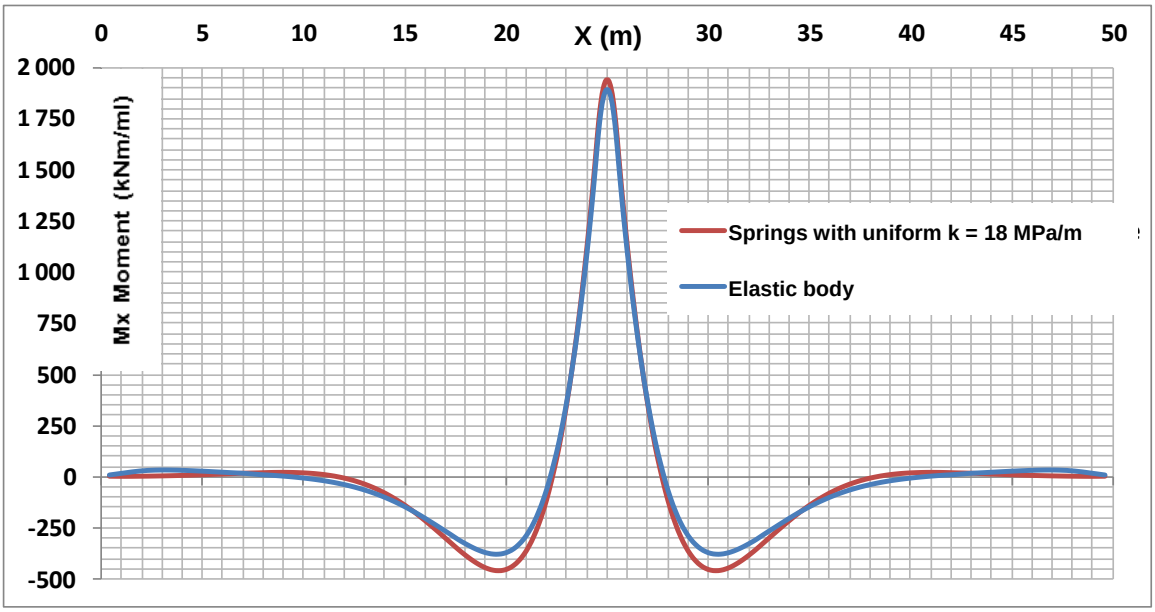
We are interested only in the maximum value of the M_x moment that we want to set equal to the originally calculated value (1892 kN.m/ml). This value can be read directly in the formatted results for example. We find out: $M_{x,max}$ = 2308 kN.m/ml.

This value is higher than the targeted one. In order to decrease the maximum moment in the footing, it is convenient to increase the subgrade reaction coefficient. Hence, as a first iteration, we propose to increase it by a ratio of 1.2 (≈ 2308/1892), and thus define this time k_z ≈ 12000 kPa/m.

The following table summarises the successive iterations carried out to reach the targeted value of M_x (1892 kN.m/ml):

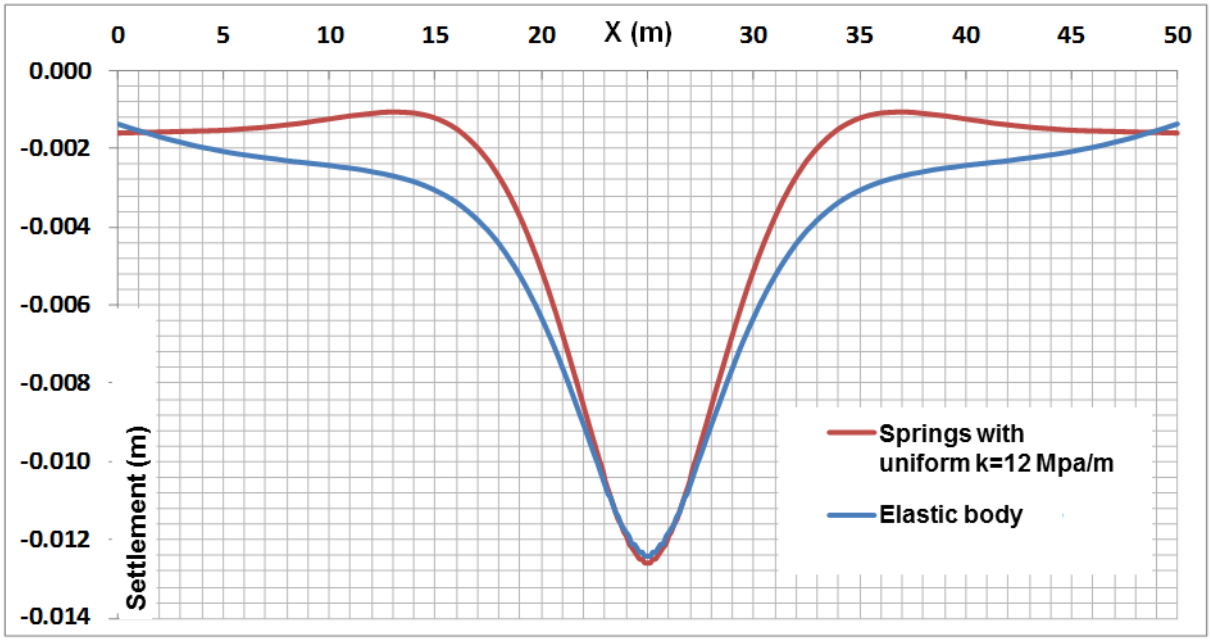
Iteration	Subgrade reaction coefficient kz (kPa/m)	Calculated maximum moment (kN.m/ml)
1	9677	2308
2	12000	2173
3	14000	2081
4	16000	2004
5	18000	1938

The figure below shows the calculated moment curve with kz = 18 MPa/m together with the reference one:



Finally, a coefficient of 18000 kPa/m turns out to be the most representative of the actual soil body in terms of the maximum forces observed in the central beam.

It is worth noting that the subgrade reaction coefficient will not necessarily provide consistent results in terms of displacements. This requires another specific fitting study, on the base of equal maximum settlement this time: this would lead to an equivalent subgrade reaction coefficient equal to 12 MPa/m as shown in the following figure.


0
5
10
15
20
X (m)
30
35
40
45
50
0.000
-0.002
-0.004
-0.006
-0.008
-0.010
-0.012
-0.014
Settlement (m)
Springs with uniform k=12 Mpa/m
Elastic body

I.4.7. Example 7: 2D calculation of a shallow foundation

The purpose of this example is to deal with an 80 cm thick and 8 m large footing with a great length (L). The bending of the footing is neglected in L direction, which allows a 2D calculation (plane strain).

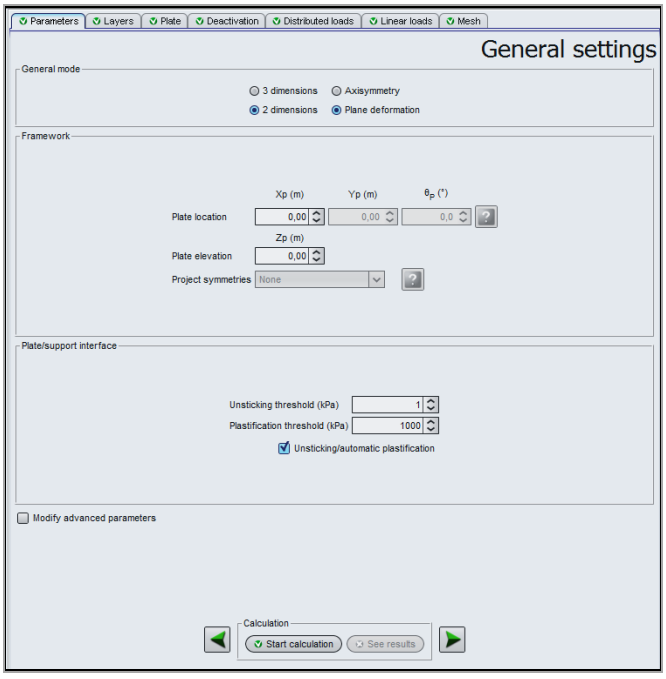
Loading conditions are illustrated in the figure below.

Create a new Foxta v3 project and select the TASPLAQ module.

I.4.7.1. Data input

I.4.7.1.1. "Parameters" tab

Complete the data according to the screenshot below.

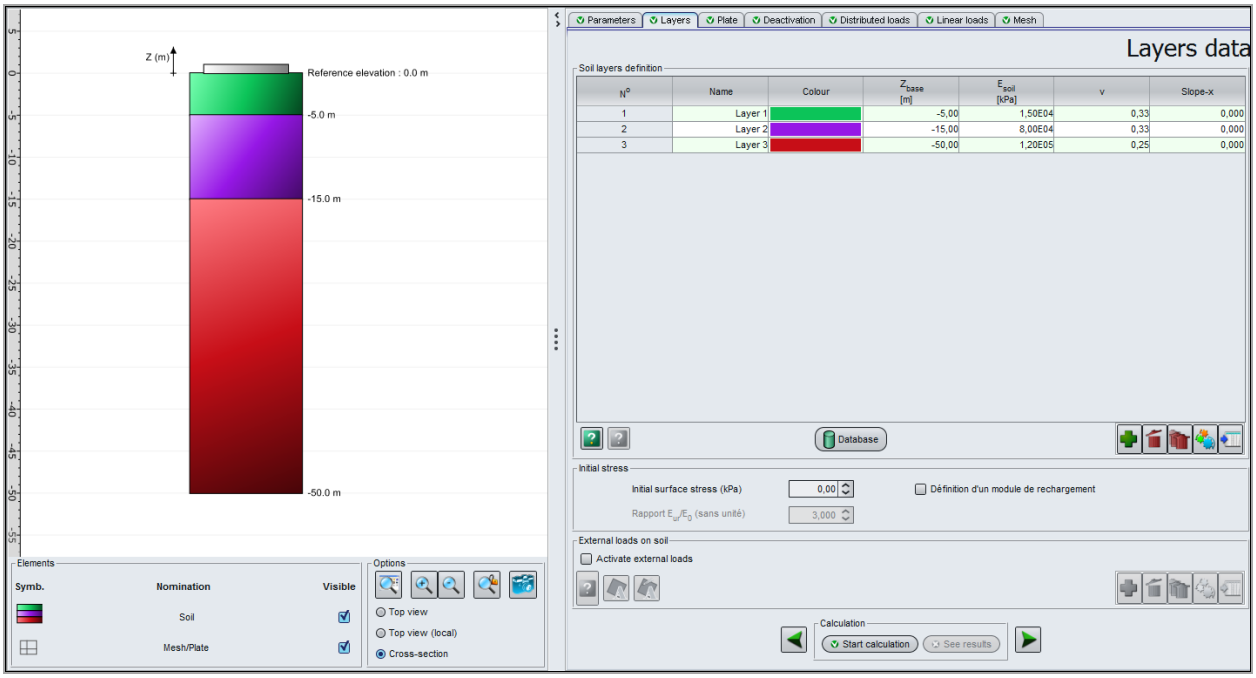


Please note that for this example, we check the "automatic unsticking/plastification" checkbox, with the following values:

- Unsticking: 1 kPa (i.e. unsticking when the tensile stress is higher than 1 kPa)
- Plastification: 1000 kPa (i.e. limitation of the soil reaction to 1000 kPa)

I.4.7.1.2. "Layers" tab

Complete the input data according to the screenshot and the table below.

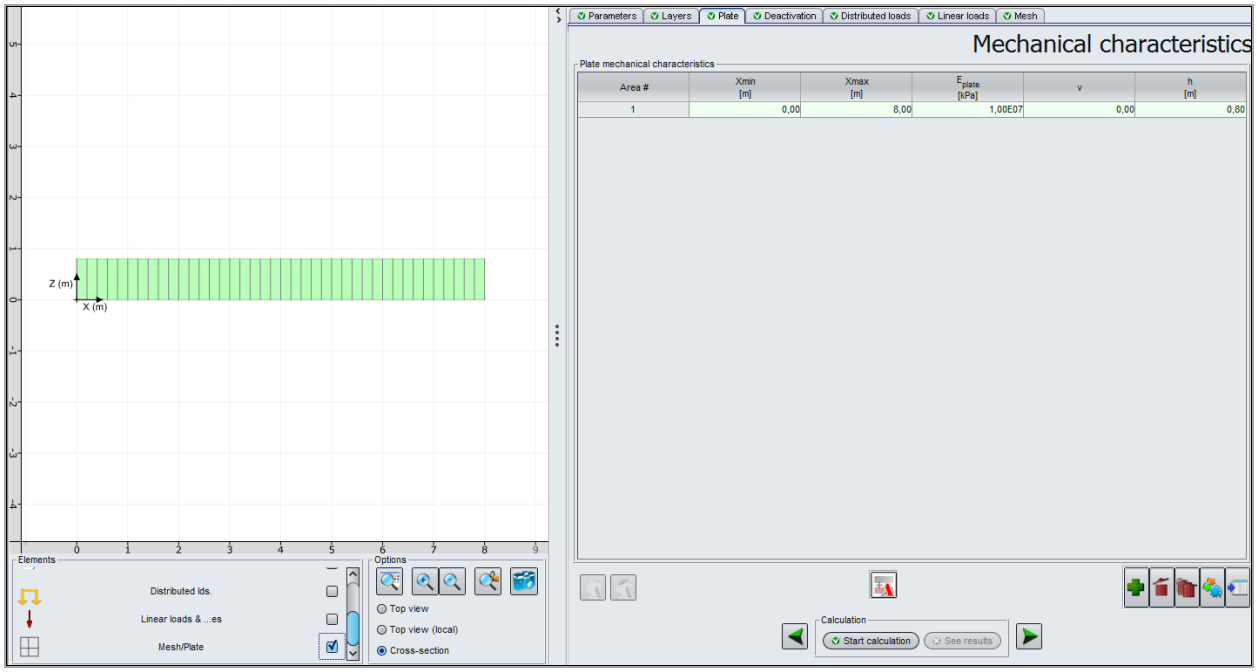


Name	Z_{base} (m)	E_{soil} (kPa)	ν	Slope-x
Layer1	-5.0	1.50E04	0.33	0.00
Layer 2	-15.0	8.00E04	0.33	0.00
Layer 3	-50.0	1.20E05	0.25	0.00

I.4.7.1.3. "Plate" tab

In this example, the plate is homogeneous and its thickness is 0.8 m. Thus, we have to define one single zone from Xmin = 0 to Xmax = 8 m.

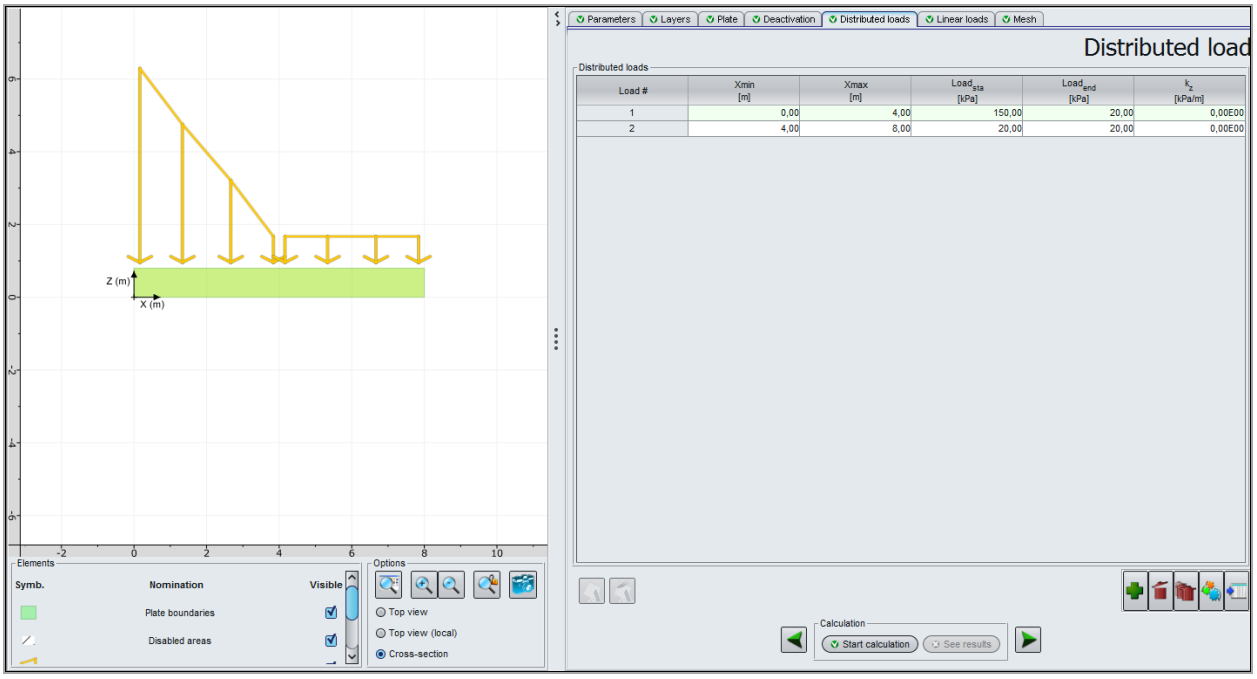
Zone n°	Xmin (m)	Xmax (m)	E_{plate} (kPa)	ν	h (m)
1	0.00	8.00	1.00E+07	0.00	0.80



I.4.7.1.4. "Distributed loads" tab

Two loading zones are to be defined:

- a first trapezoidal load along the "left" half of the plate (150 kPa at Xmin = 0 down to 20 kPa at Xmax = 4 m);
- then a second uniform load of 20 kPa along the other half of the plate (from Xmin = 4 m to Xmax = 8 m).



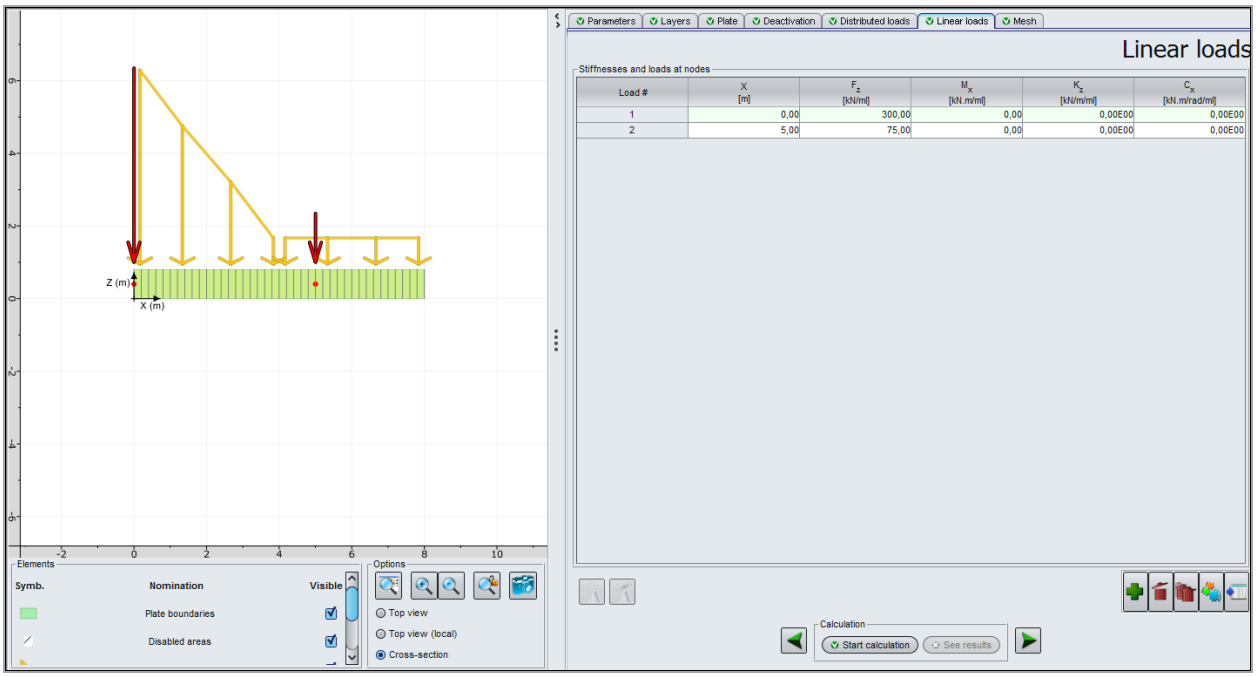
Load n°	Xmin (m)	Xmax (m)	$Load_{left}$ (kPa)	$Load_{right}$ (kPa)	k_z (kPa/m)
1	0.00	4.00	150	20	0
2	4.00	8.00	20	20	0

I.4.7.1.5. "Linear loads" tab

Two linear loads are to be defined for this example:

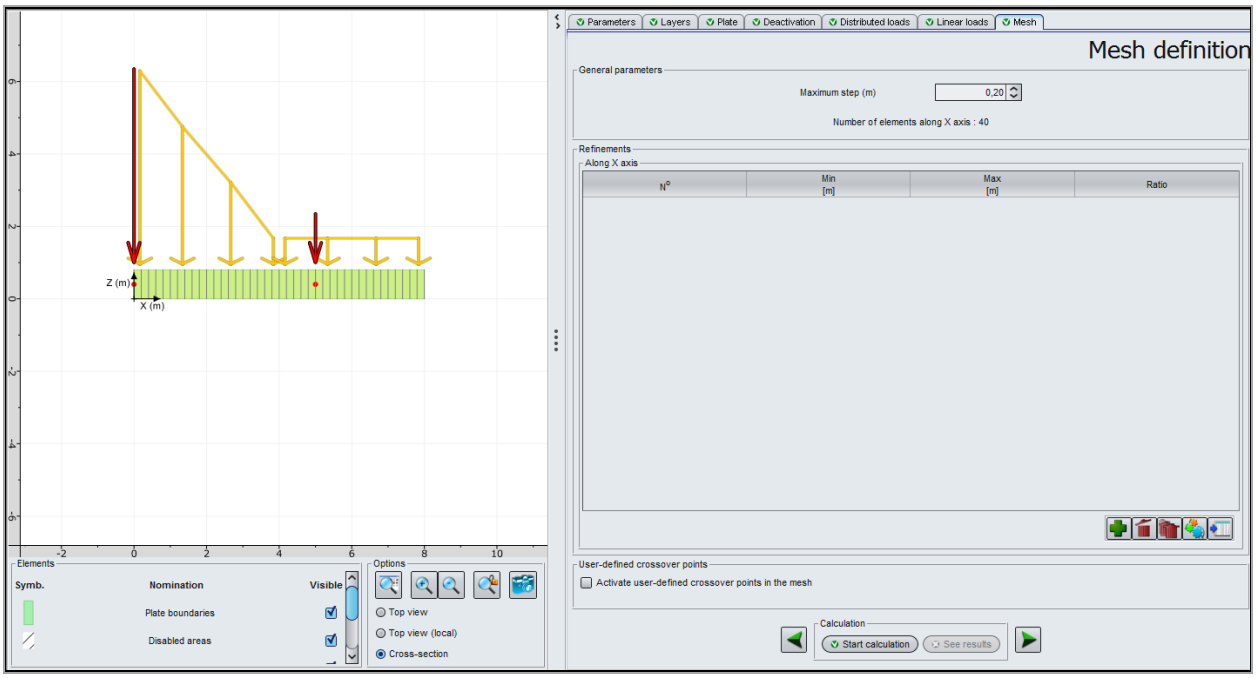
- F_z from 300 kN/ml to X = 0;
- F_z from 75 kN/ml to X = 5 m.

Load n°	X (m)	F_Z (kN/ml)	M_X (kN.m/ml)	K_Z (kN/m/ml)	C_X (kN.m/rad/ml)
1	0.00	300	0	0	0
2	5.00	75	0	0	0



I.4.7.1.6. "Mesh" tab

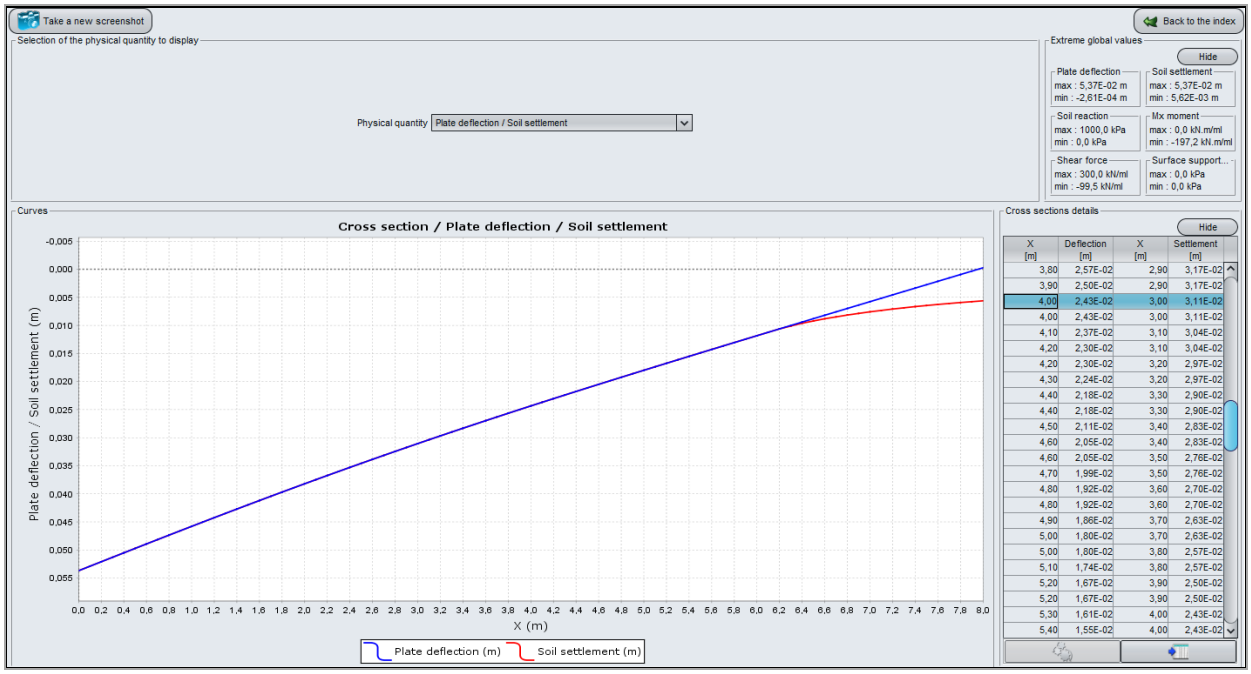
We propose a uniform meshing with 0.2 m step.



I.4.7.2. Graphical results - Cross-sections

<u>Plate deflection/ Soil settlement</u>

We observe an unsticking at the right extremity of the footing due to the dissymmetric nature of the applied load.

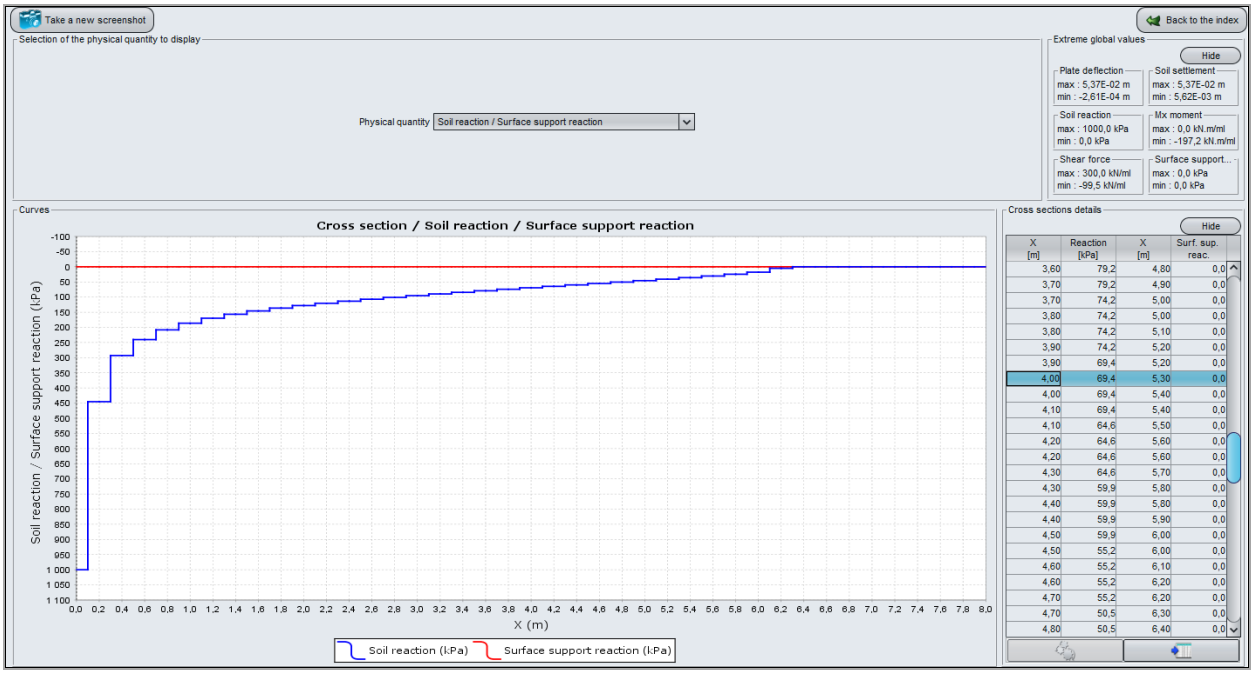


<u>Soil reaction</u>

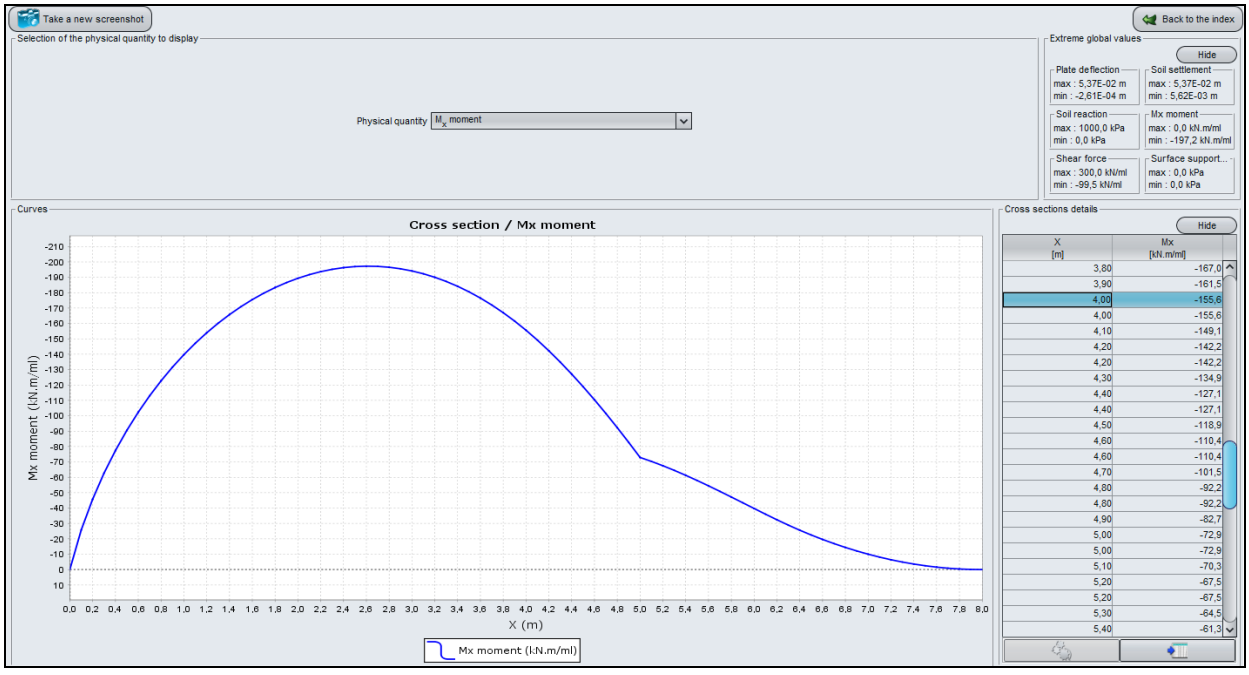
Extreme global values:

- Max: 1000 kPa
- Min: 0 kPa

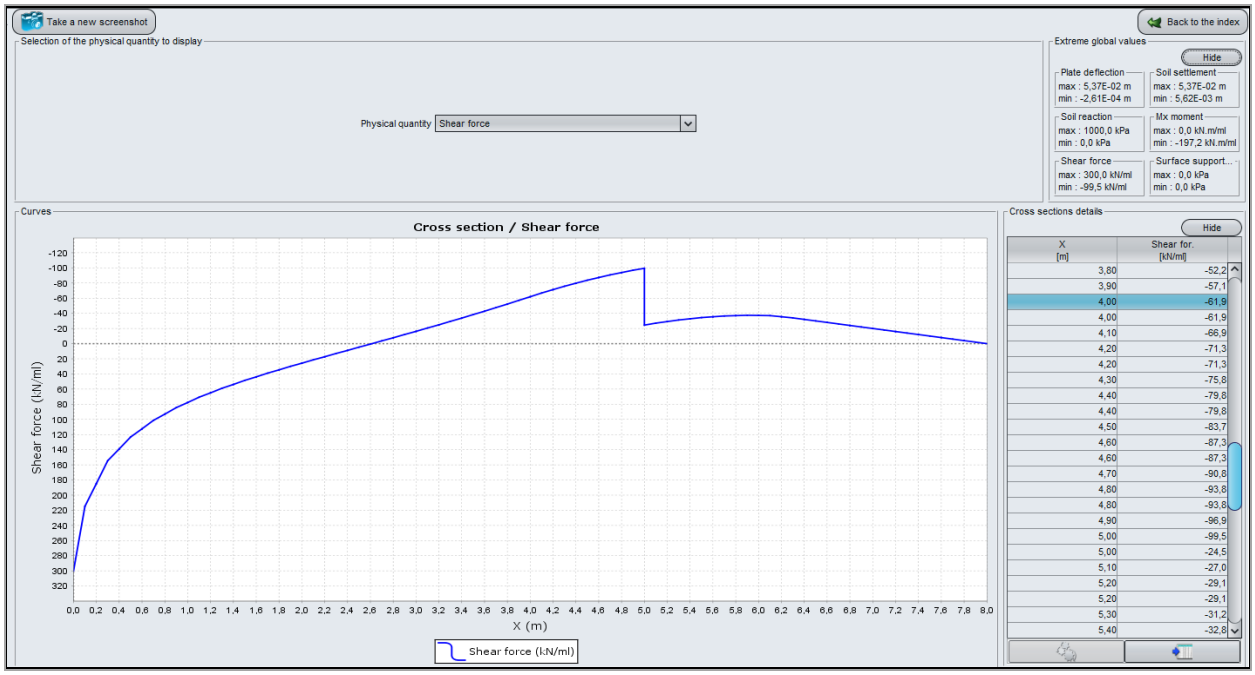
The maximum reaction is calculated at the left extremity of the footing (where the load is the greatest). This maximum value is 1000 kPa, it corresponds to the input value assumed for this calculation: so there is plastification (which is managed automatically by the software). The unsticking zone can be observed here as the zone where the soil reaction is zero, thus graphically starting from X = 6.50 m.



Mx Moment



Shear force



The maximum shear force is reached at X = 0 m (left extremity of the footing): its value is 300 kN/ml and it corresponds to the linear load value applied at X = 0 m.

The minimum shear force is reached at X = 5 m: at this location we take note of a gap of shear force of 75 kN/ml, equal to the linear load applied at X = 5 m.