TASPLAQ module

B. User Manual

1. SIGN NOTATIONS AND CONVENTIONS 3

1.1 SIGN NOTATIONS AND CONVENTIONS 3
1.2 UNITS 3

2. GLOBAL PRESENTATION OF THE USER INTERFACE 4

3. DATA INPUT 5

3.1 GENERAL OPERATION OF DATA INPUT 5
3.2 GENERAL SETTINGS 6
3.2.1 Calculation settings 6
3.2.2 Elastic thresholds for soil-plate interaction 6
3.2.3 Geometry dimensions 7
3.3 DEFINING THE LAYERS 10
3.4 MESH ALONG X-AXIS 12
3.5 MESH ALONG Y-AXIS 14
3.6 DEACTIVATING ELEMENTS 16
3.7 DEFINING THE MECHANICAL PROPERTIES OF THE PLATE 19
3.8 DEFINING THE LOAD DISTRIBUTED ON THE PLATE 21
3.9 DEFINING LOAD ON NODES 24
3.10 DEFINING EXTERNAL LOADS APPLIED TO THE SOIL 26
3.11 MANUAL CONTROL OF SEPARATED AND PLASTIC NODES (OPTIONAL) 28

4. CALCULATIONS 30

5. RESULTS 32

5.1 RESULT FILE 32
5.2 EXPORT TO A NEW SPREADSHEET 33
5.3 CROSS SECTIONS 34
5.4 2D SCATTER POINTS 35
5.5 3D GRAPH WIZARD 35
5.6 DEHOMOGENISATION 36

6. INPUT AND OUTPUT FILES 38

6.1 INPUT: CONSTITUTION OF THE INPUT DATA FILE (TPL) 38
6.2 OUTPUT FILES 40
6.2.1 Result file 40
6.2.2 TASSELDO file 41
6.2.3 Influence matrix temporary backup file 41
6.2.4 File for use under Microsoft Excel® 41

LIST OF FIGURES

FIGURE 1: HOME PAGE OF TASPLAQ INPUT ... 4
FIGURE 2: GENERAL PARAMETERS - EXAMPLE ... 8
FIGURE 3: DEFINITION OF GEOMETRY DIMENSIONS ... 9
FIGURE 4: WORK COORDINATE SYSTEM ... 9
FIGURE 5: LAYER DEFINITION ... 10
FIGURE 6: EXAMPLE OF LAYERS DEFINITION ... 11
FIGURE7: MESH ALONG X-AXIS – MODELLING PRINCIPLES ... 12
FIGURE 8: EXAMPLE OF MESH ALONG X-AXIS ... 13
FIGURE 9: MESH ALONG Y-AXIS – MODELLING PRINCIPLES ... 14
FIGURE 10: EXAMPLE OF MESH ALONG Y-AXIS ... 15
FIGURE 11: GLOBAL MESH ... 16
FIGURE 12: ELEMENT DEACTIVATION TECHNIQUE ... 17
FIGURE 13: ELEMENTS DEACTIVATED (EXAMPLE) ... 17
FIGURE 14: EXAMPLE OF ELEMENT DEACTIVATION ... 18
FIGURE 15: EXAMPLE OF DEFINITION OF THE MECHANICAL PROPERTIES OF THE PLATE ... 20
FIGURE 16: COMBINED SECTION CALCULATION ... 21
FIGURE 17: EXAMPLE OF LOAD DISTRIBUTED ON THE PLATE ... 23
FIGURE 18: EXAMPLE OF POINT LOAD ... 25
FIGURE 19: GLOBAL LAYOUT OF THE PROBLEM {PLATE + SOIL + EXTERNAL LOADS} ... 26
FIGURE 20: COORDINATES OF EXTERNAL LOADS ... 26
FIGURE 21: EXAMPLE OF EXTERNAL LOADS ON THE SOIL ... 27
FIGURE 22: MANUAL DEFINITION OF NODE SEPARATION / PLASTIFICATION - EXAMPLE ... 28
FIGURE 23: HOMEPAGE OF TASPLAQ INPUT ... 30
FIGURE 24: CALCULATION WINDOW ... 31
FIGURE 25: HOME PAGE OF TASPLAQ OUTPUT ... 32
*FIGURE 26: *.RESU FILE - EXAMPLE* ... 33
FIGURE 27: EXPORTING THE FILE UNDER MICROSOFT EXCEL® ... 33
FIGURE 28: SETTLEMENT ALONG X IN Y = 5 AND Y = 7 ... 34
FIGURE 29: 2D SCATTER POINTS ... 35
FIGURE 30: HOME WINDOW OF TASPLAQ GRAPHIQUE3D.XLS ... 35
FIGURE 31: 3D GRAPH WINDOW ... 36
FIGURE 32: HOME WINDOW OF TASPLAQ DESHOMOGENISATION.XLS ... 36
FIGURE 33: WINDOW [CHARACTERISTICS OF THE LOWER LAYER] ... 37
FIGURE 34: DEHOMOGENISATION ... 37

LIST OF TABLES

TABLE 1: SIGN NOTATIONS AND CONVENTIONS ... 3
TABLE 2: UNITS ... 3
TABLE 3: SUMMARY OF GENERAL SETTINGS ... 9
TABLE 4: SUMMARY OF PARAMETERS REQUIRED FOR SOIL DEFINITION ... 10
TABLE 5: PARAMETERS REQUIRED TO DEFINE MESH ALONG X-AXIS ... 12
TABLE 6: DEACTIVATION PARAMETERS ... 16
TABLE 7: PARAMETERS FOR ALLOCATING MECHANICAL CHARACTERISTICS ... 19
TABLE 8: PARAMETERS FOR DISTRIBUTED LOAD ... 22
TABLE 9: PARAMETERS FOR LOADS ON NODES ... 24
TABLE 10: SETTING OF EXTERNAL LOADS ON THE SOIL. ... 28
TABLE 11: PARAMETERS FOR MANUAL SEPARATION/PLASTIFICATION MANAGEMENT ... 29

1. SIGN NOTATIONS AND CONVENTIONS

1.1 Sign notations and conventions

Magnitude	Representation	Sign convention
Rotations and moments	$\theta_x, \theta_y, \theta_p, \theta_r \quad M_x, M_y, M_{xy}$	Trigonometric meaning
Plate deflection	w	Positive downwards
Soil settlement	Tass	Positive downwards
Shear forces	T_x, T_y	Positive upwards
Vertical load (distributed or point)	q, F_z	Positive downwards
Soil reaction, interaction pressure	p_s	Positive upwards
Springs	C_x, C_y, K_z, Ks_z	Always positive
Loads	$\sigma_x, \sigma_y, \sigma_{xy}$	Positive in traction

Table 1: Sign notations and conventions

1.2 Units

Magnitude	Unit
Lengths and coordinates	m
Vertical point load Fz	kN
Moments (Mx, My, Mxy)	kN.m/ml
Shear forces (Tx, Ty)	kN/ml
Soil reaction, distributed loads	kPa
Displacements (deflection w, settlement s)	m
Rotations	rad
Young’s modulus E	kPa
Distributed springs / subgrade reaction	kPa/m
Linear springs	kN/m
Rotation springs	kN.m/rad

Table 2: Units

2. GLOBAL PRESENTATION OF THE USER INTERFACE

The application's interface was developed under Microsoft Excel®. When opening the TASPLAQ_vx.x.xls file, a home page appears (Figure 1).
It allows to select an existing file or create a new one. The work directory can also be configured.
The installation directory is entered automatically.

From this interface, you can:
- Access data entry ([Start modelling]) ;
- Launch the calculation: the interface then calls upon TASPLAQ's calculation engine to run the .tpl file created during modelling ;
- Display results: the calculation results are accessible from the TASPLAQ Output_vx.x.xls file (Microsoft Excel®).

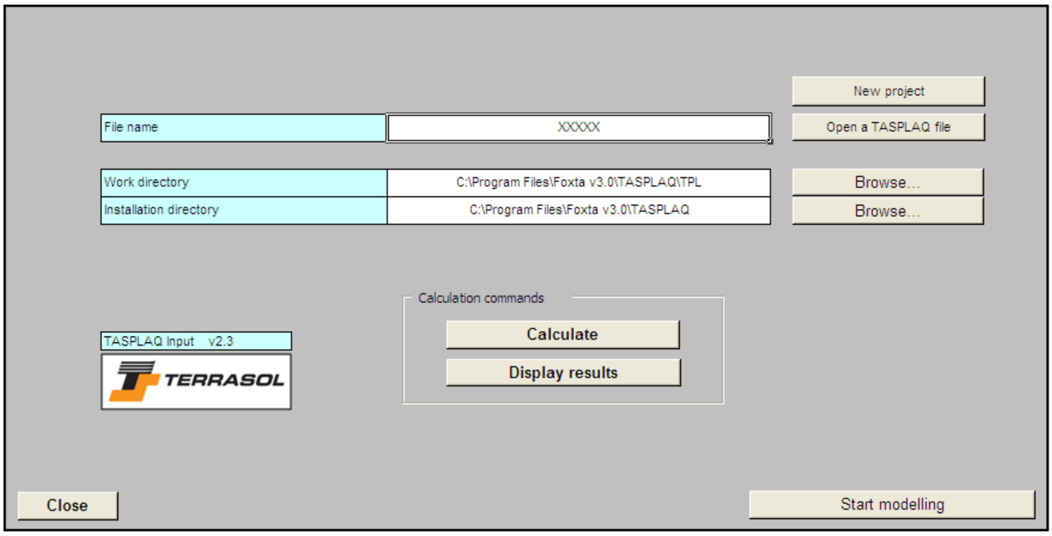


Figure 1: home page of Tasplaq input

3. DATA INPUT

To access data input or modification press the [Start modelling] button.

3.1 General operation of data input

Data input is performed by following the steps described in the next paragraphs. These steps correspond to the different types of data to be defined.

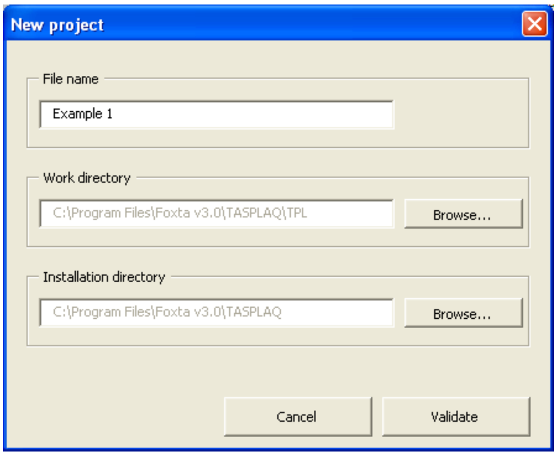
This input is accompanied with graphic viewing updated automatically when adding information.

Screenshots of the application illustrate each of the steps in making the model.

Once the TASPLAQ_vx.x.xls file is open, you can either create a new calculation file, or open an existing file.

To create a new calculation file:

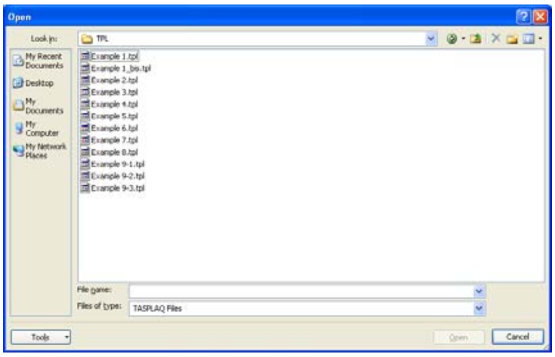
- Click the [New project] button, a new window opens.



- Enter the name of the file to create.
- Click the [Browse…] button to choose the work directory to save the *.tpl file.
- The installation directory is configured automatically.
- Click the [Validate] button to return to the home page.

To open an existing calculation file:

- Click the [Open a TASPLAQ file] button, a new window opens.



- Choose the directory containing the *.tpl file required.
- Select the file, then click the [Open] button.

- Click the [Start modelling] button to input your data.

Data is entered in the module using 9 tabs filled successively. Move between the tabs using the [Next] and [Previous] buttons. We recommend following the tab sequence.

Once data has been input in the 9 tabs, a window opens to save a data file in the .tpl format. The content of this file is explained in paragraph 6.

3.2 General settings

The following are the general settings to be entered.

3.2.1 Calculation settings

- **Import stiffness matrices**: allows importing the influence matrix of a previous calculation, presumably saved beforehand. It ensures a major time gain in the case of a system with several loading cases.

- **Save stiffness matrices**: used to save the soil influence matrix for a subsequent calculation. This option is used in the case of a system with several loading cases for example.

- **Automatic control:** allows automatic consideration of separation and/or plastification to points as per the separation and plastification criteria defined in the section 'thresholds for soil-plate interface'.

- **Symmetries:** allows considering symmetries, along x-axis or/and along y-axis.

- **Printing content:** controls printing of the results file. This choice is related only with the data summary: Reduced print = short summary of the data / Detailed printing = detailed summary of the data.

3.2.2 Elastic thresholds for soil-plate interaction

These parameters concern surface soil only. They intervene in the calculation only in the case of a plate on supporting soil, and only if automatic calculation has been requested.

- **Separation threshold (kPa)**: limit stress in **traction** at the Soil-Plate interface, beyond which the corresponding points are considered as being 'separated'. Soil reaction beside these points is hence zero, and there is no longer equality between soil settlement and the vertical displacement of the plate.

- **Plastification threshold (kPa)**: limit stress in **compression** at the Soil-Plate interface, beyond which the corresponding points are considered as being 'plastified'. The soil's reaction beside these points is imposed (equal to the plastification threshold), but equality between soil settlement and vertical displacement of the plate is always ensured.

3.2.3 Geometry dimensions

This means defining the local coordinate system of the plate. Therefore, the general case includes two coordinate systems:

- A reference coordinate system (O_0x_0, O_0y_0, O_0z_0), containing the plate as well as the external loads applied directly to the soil.
- A local coordinate system (Ox, Oy, Oz) associated with the plate, defining the mesh, as well as different characteristics. This coordinate system is such that the plane (Ox, Oy) is parallel to (O_0x_0, O_0y_0). Hence it can be defined perfectly using two parameters:
    - **The coordinates** (x_p, y_p, z_p) of point O in the reference coordinate system. Beware! *Zp* is the reference level of the project.
    - **The rotation angle** θ_p of the axis (Ox) in respect of the (O_0x_0) axis.

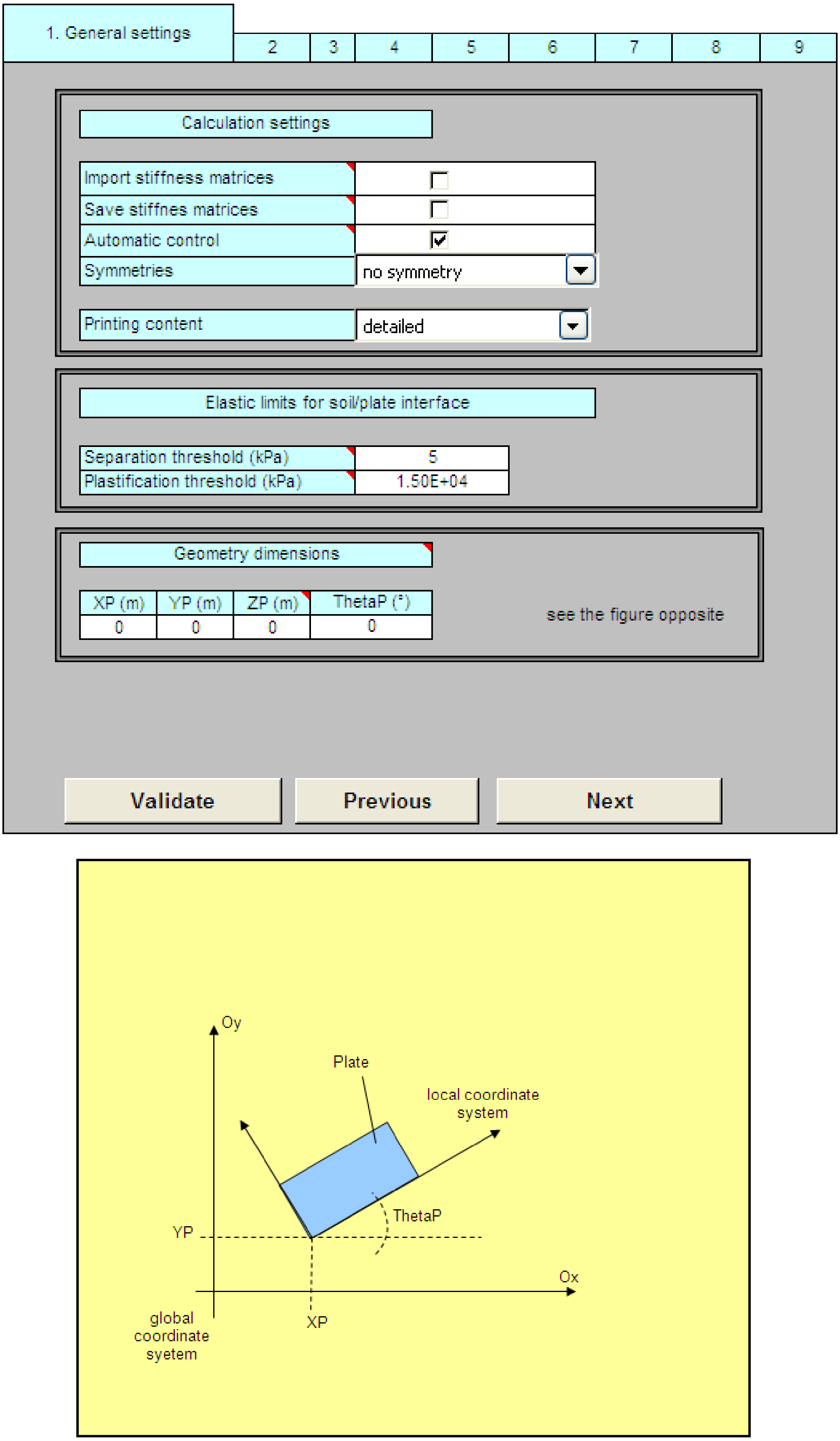


Figure 2: General parameters - example

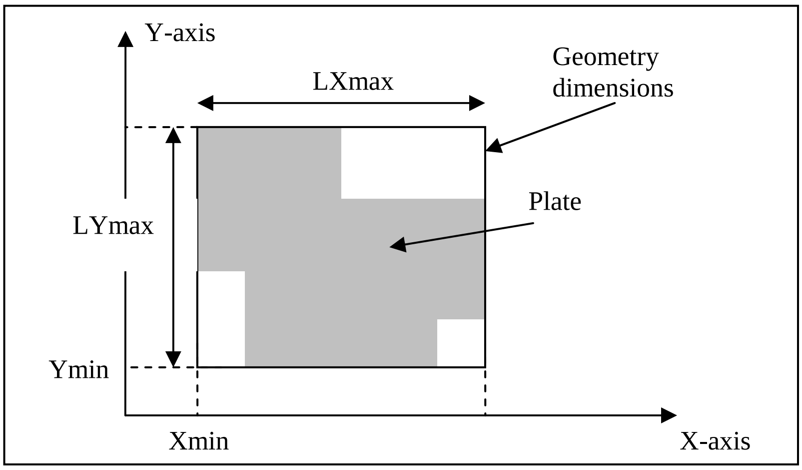


Figure 3: definition of geometry dimensions

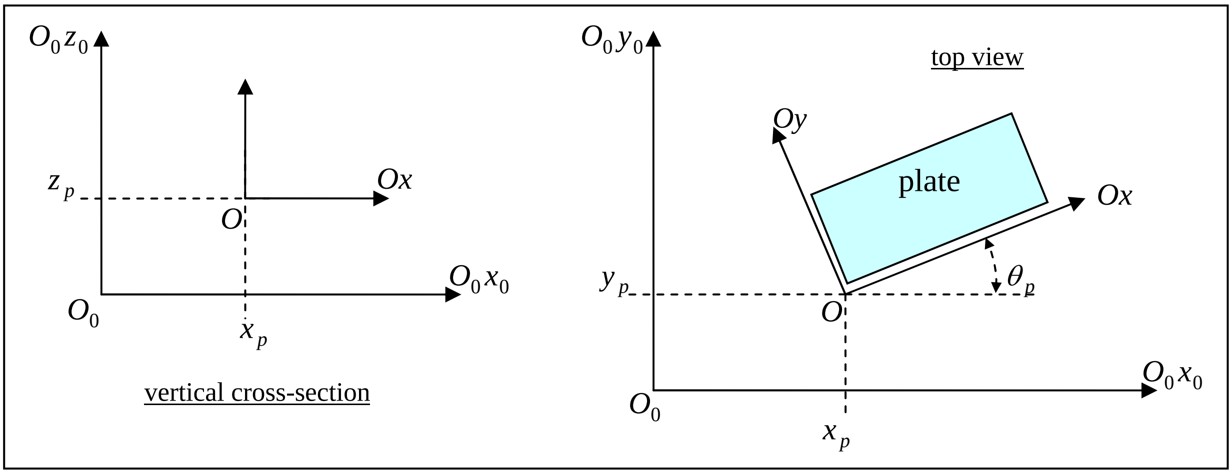


Figure 4: work coordinate system

Designation	Unit	Default value	Display condition	Mandatory value
Import stiffness matrices	None	Unchecked	Always	Yes
Save stiffness matrices	None	Unchecked	Always	Yes
Automatic control	None	Unchecked	Always	Yes
Symmetries	None	No symmetries	Always	Yes
Printing content	None	Detailed printing	Always	Yes
Separation threshold	kPa	5	Always	Yes
Plastification threshold	kPa	10000	Always	Yes
XP	m	0	Only if there are no more symmetries. Otherwise, value set to 0 (no modification possible)	Yes
YP	m	0	Only if there are no more symmetries. Otherwise, value set to 0 (no modification possible)	Yes
ZP	m	0	Always	Yes
Theta	°	0	Only if there are no more symmetries. Otherwise, value set to 0 (no modification possible)	Yes

Table 3: summary of general settings

3.3 Defining the layers

The soil is made of a series of horizontal layers, each characterised by its Young's modulus, its Poisson's ratio and the level of its base. Hence, the 'i' layer is located between the $(z = z_{i-1})$ and $(z = z_i)$ planes. Conventionally, we take z_0 equal to z_p, level of the plate (Figure 5).

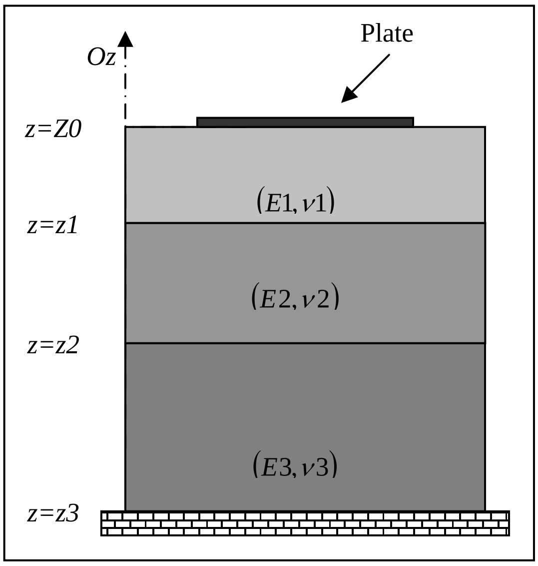


Figure 5: Layer definition

The following figure details the parameters required to define the layers. The user can view the layout of the layers in the form of a vertical cross-section. In this diagram, the soil's surface is taken equal to the level of the ZP plate defined in general settings.

This step is not mandatory: e.g. in the case of a calculation on elastic supports only, no soil layers are defined.

The table below summarises the layer definition parameters:

	Designation	Unit	Value by default	Condition of display	Mandatory value
For each layer, enter:	Layer name	None	---	Always	No
	Level of the base of the layer	m	---	Always	Yes
	Young's modulus of the layer	kPa	---	Always	Yes
	Poisson's ratio	None	---	Always	Yes
Enter once	Initial vertical stress on surface	kPa	0	Always	Yes

Table 4: summary of parameters required for soil definition

1	2. Definition of soil layers	3	4	5	6	7	8	9

Soil layer	Name	Bottom level Zb(m)	Modulus (kPa)	Poisson's ratio
1	Soil 1	-10	1.00E+04	0.35
2	Soil 2	-35	4.00E+04	0.35
3				
4				
5				
6				
7				
8				
9				
10				
11				
12				
13				
14				
15				
16				
17				
18				
19				
20				

Initial stress at groundlevel	0

Validate   Previous   Next

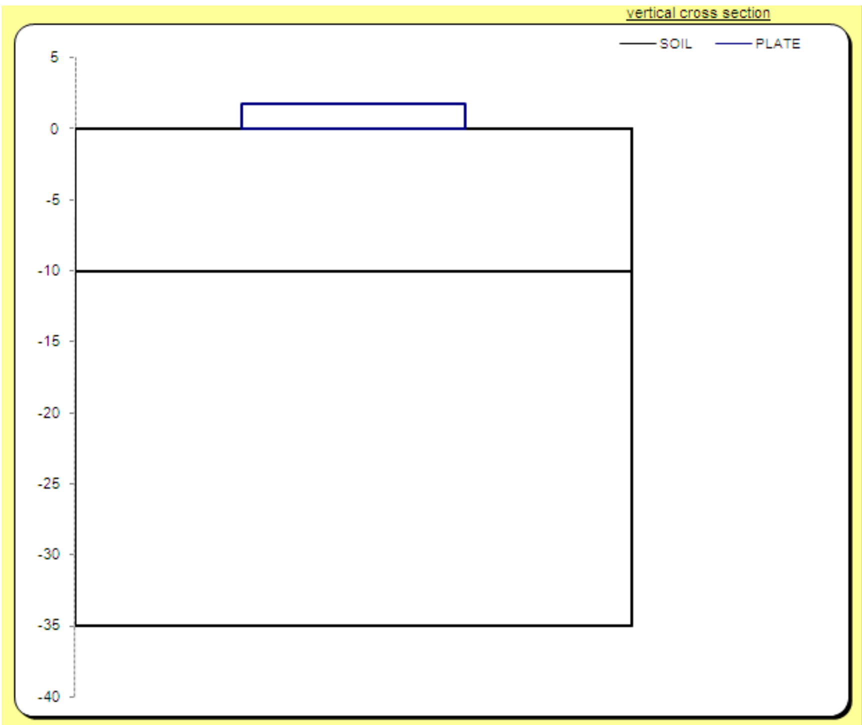


Figure 6: Example of layers definition

3.4 Mesh along x-axis

We then go to the local coordinate system of the plate. The mesh is defined in two steps corresponding to the x-axis and y-axis directions. At first, we look at the mesh along x-axis.

The plate is divided into one or several clusters along the x-axis. Each cluster is characterised by its length Lx(i) and associated number of subdivisions Nx(i) as shown in the diagram below.

This step is mandatory, at least one cluster must be defined.

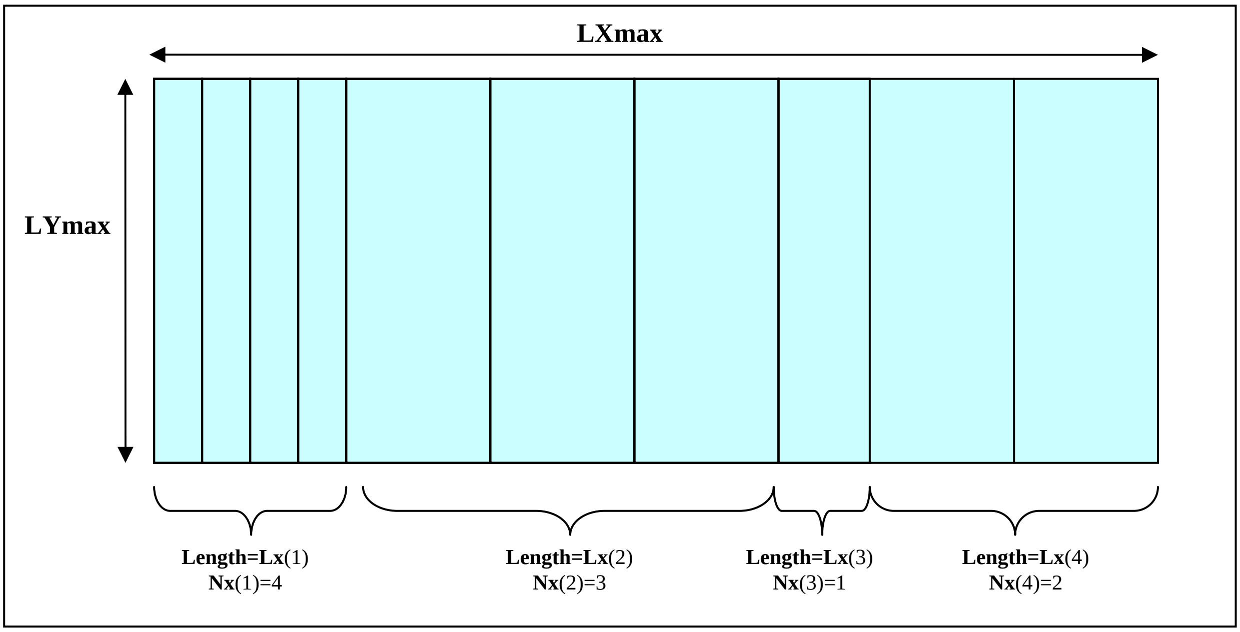


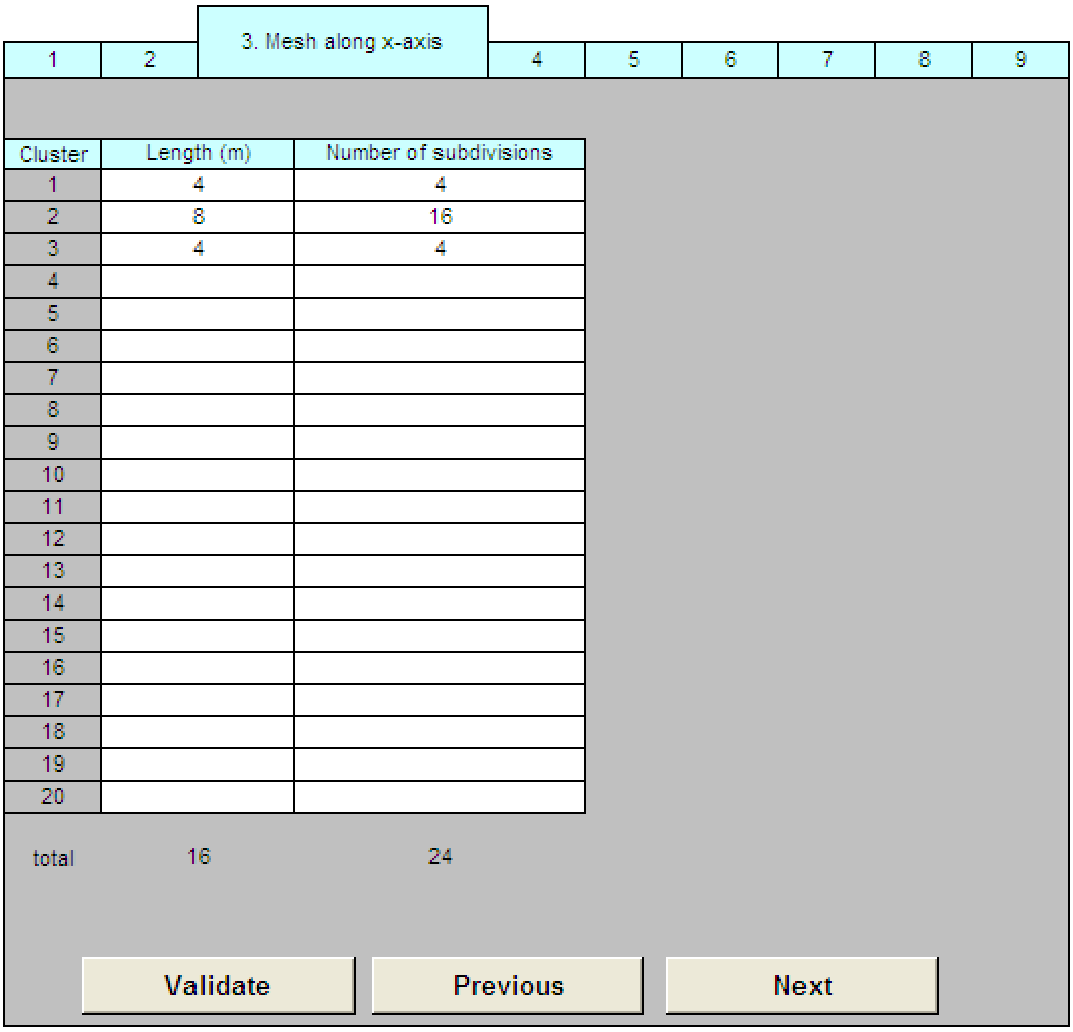
Figure7: Mesh along x-axis – Modelling principles

In the graphic window, the plate is shown by a top view: the user can view the discretization defined upon entry.

The table below summarises the parameters required:

Designation	Unit	Default value	Display condition	Mandatory value
Length of the cluster	m	---	Always	Yes
Number of subdivisions	None	---	Always	Yes

Table 5: Parameters required to define mesh along x-axis



1 | 2 | 3. Mesh along x-axis | 4 | 5 | 6 | 7 | 8 | 9

Cluster	Length (m)	Number of subdivisions
1	4	4
2	8	16
3	4	4
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		
18		
19		
20		
total	16	24

Validate | Previous | Next

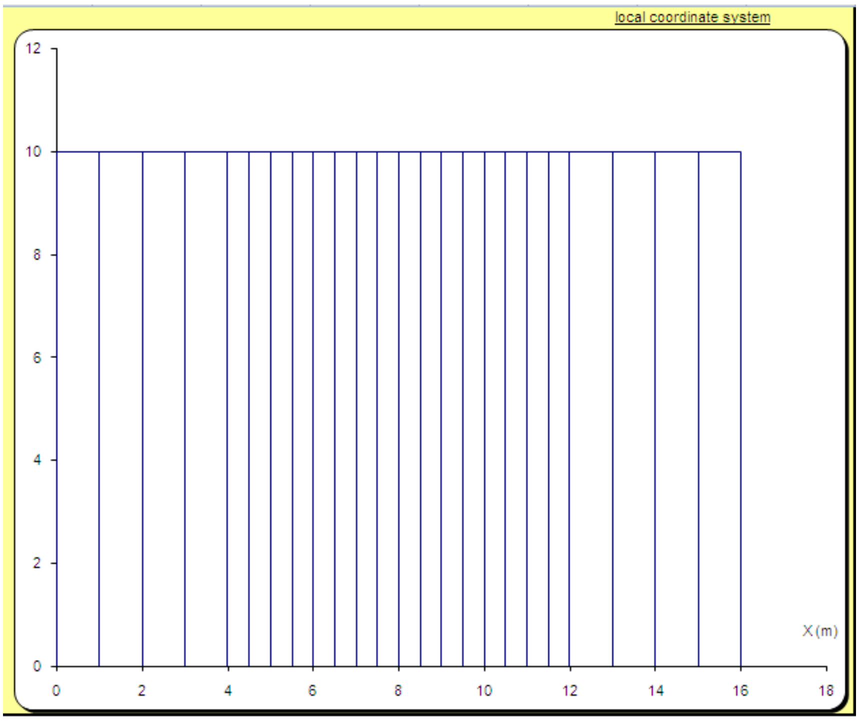


Figure 8: Example of mesh along x-axis

3.5 Mesh along y-axis

As the mesh is defined along the x-axis direction, a discretization along y-axis is superimposed according to the same principle, as shown in the figure below.

This step is mandatory, at least one cluster must be defined.

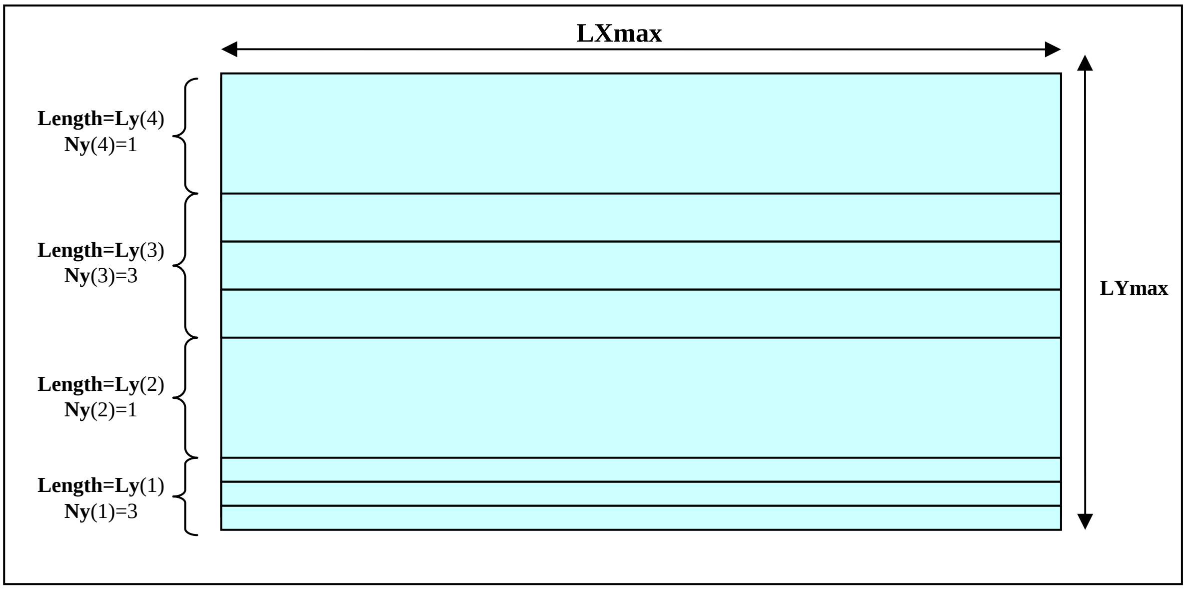
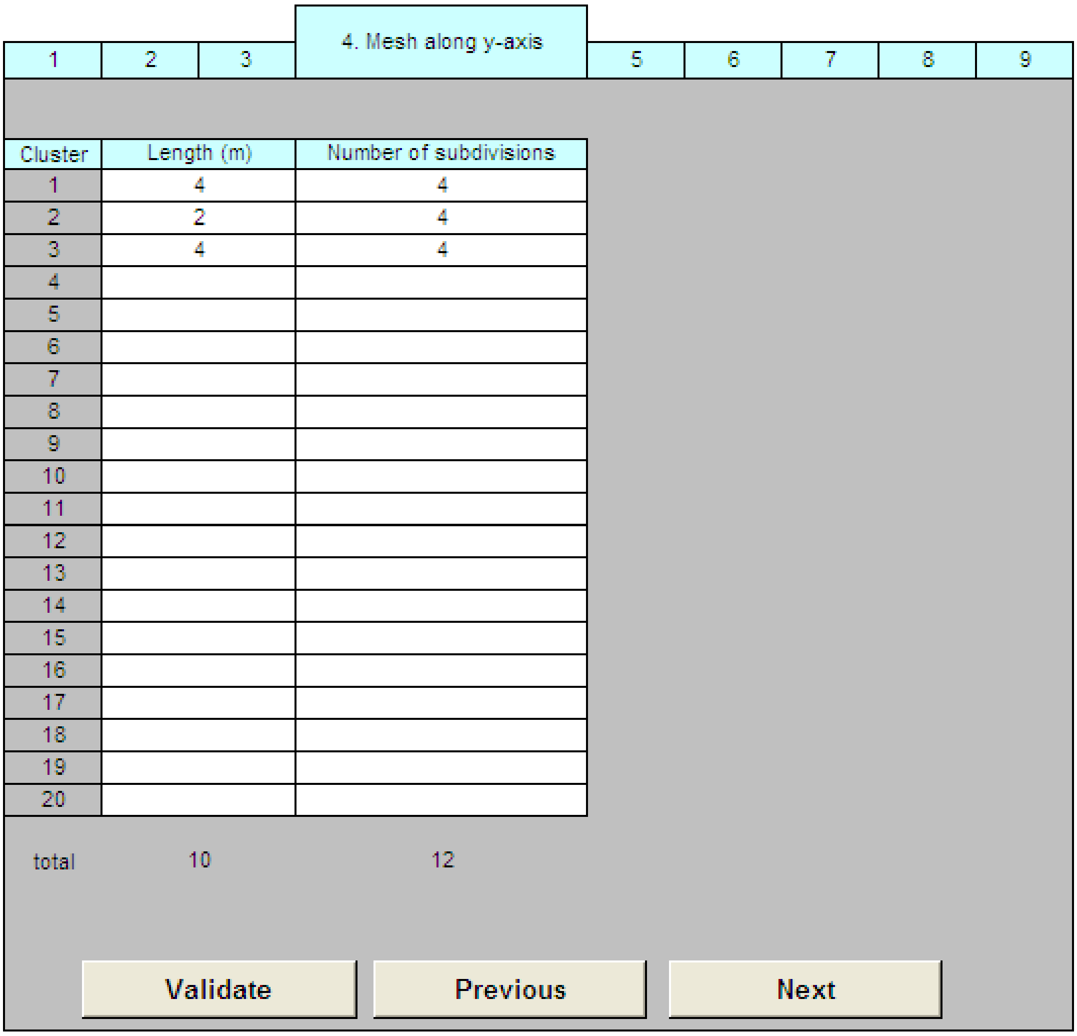


Figure 9: Mesh along y-axis – Modelling principles

The principle of discretization is identical to that considered for the x-axis direction: the pitch is defined by cluster, each cluster being characterised by its length Ly(i) and the associated number of subdivisions Ny(i) as shown in the diagram above.

Caution! The total number of Nx $\times$ Ny elements must be below 2500 (maximum manageable by Windows environment).



1	2	3	4. Mesh along y-axis	5	6	7	8	9

Cluster	Length (m)	Number of subdivisions
1	4	4
2	2	4
3	4	4
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		
18		
19		
20		
total	10	12

Validate Previous Next

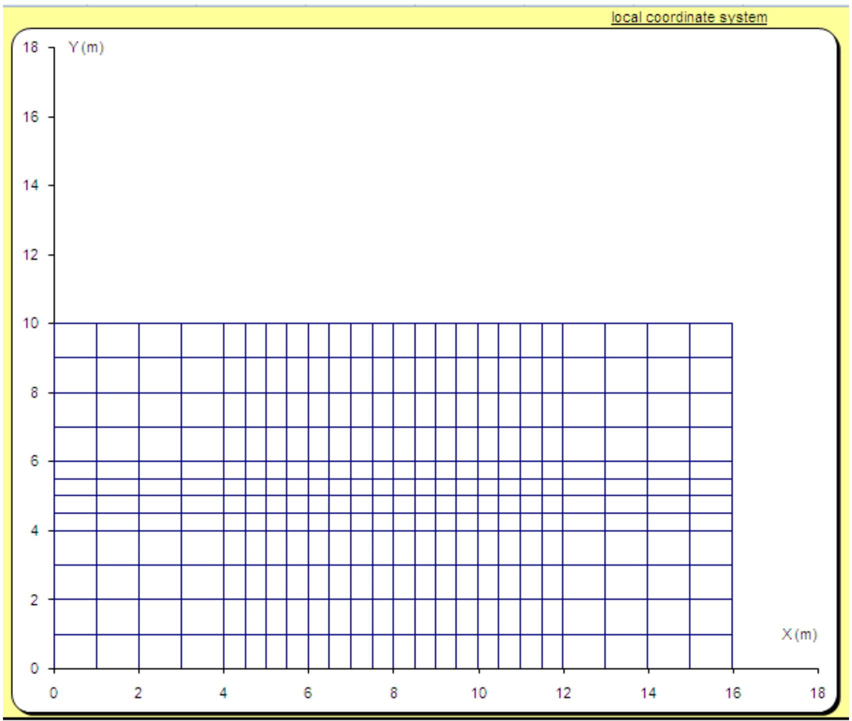


Figure 10: Example of mesh along y-axis

Superimposing the two x-axis and y-axis meshes leads to the final mesh.

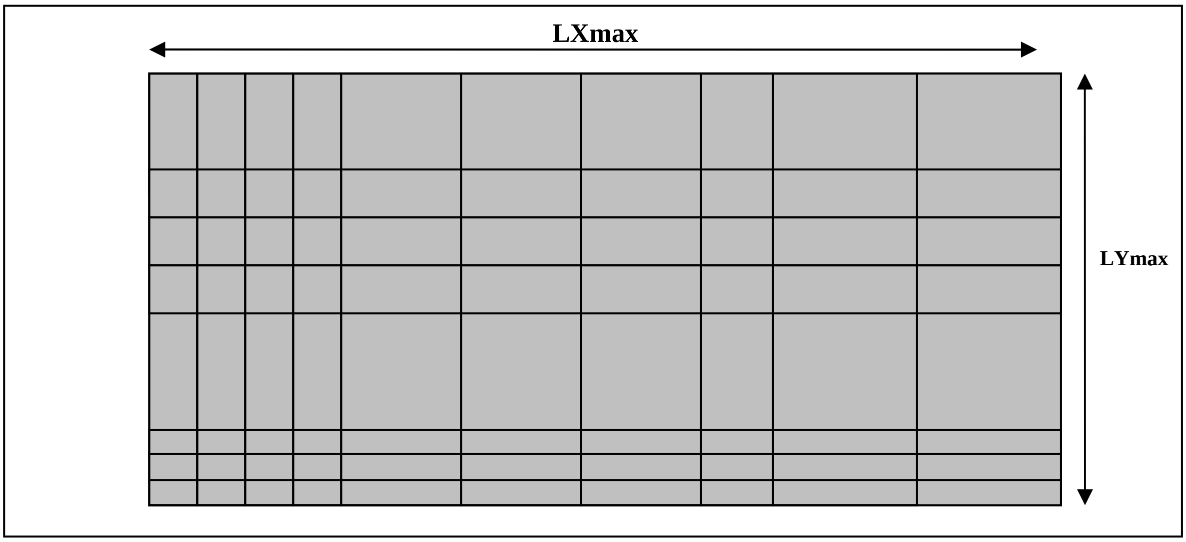


Figure 11: Global mesh

3.6 Deactivating elements

Once the mesh has been defined, the 'effective' plate geometry must be set. Indeed, complex plate geometries can be modelled, using the element deactivation option.

This step is not mandatory: if no element is deactivated, the plate is assumed to cover the entire mesh.

Each deactivated cluster of the plate is defined by a group of elements corresponding to a rectangular cluster.
The groups of elements themselves are defined using a numbering system: elements are numbered in each direction to facilitate group selection in the form 'i1 i2 j1 j2'.

Note: the element numbering system in each direction appears in Figure 13.

The following table lists the parameters required:

Designation	Unit	Value by default	Condition of display	Mandatory value	Local checks
i1*	None	---	Always	Yes	>0 and <= total number of subdivisions along x-axis
i2*	None	---	Always	Yes	>=i1 and <= total number of subdivisions along x-axis
j1*	None	---	Always	Yes	>0 and <= total number of subdivisions along (y-axis)
j2*	None	---	Always	Yes	>=j1 and <= total number of subdivisions along x-axis

*: i1, i2, j1 and j2 are the basic coordinates of the deactivated cluster (Figure 13)

Table 6: Deactivation parameters

One or several clusters can be deactivated.

The clusters deactivated are outlined by a red line in the drawing.

The following figures show a few possible cases.

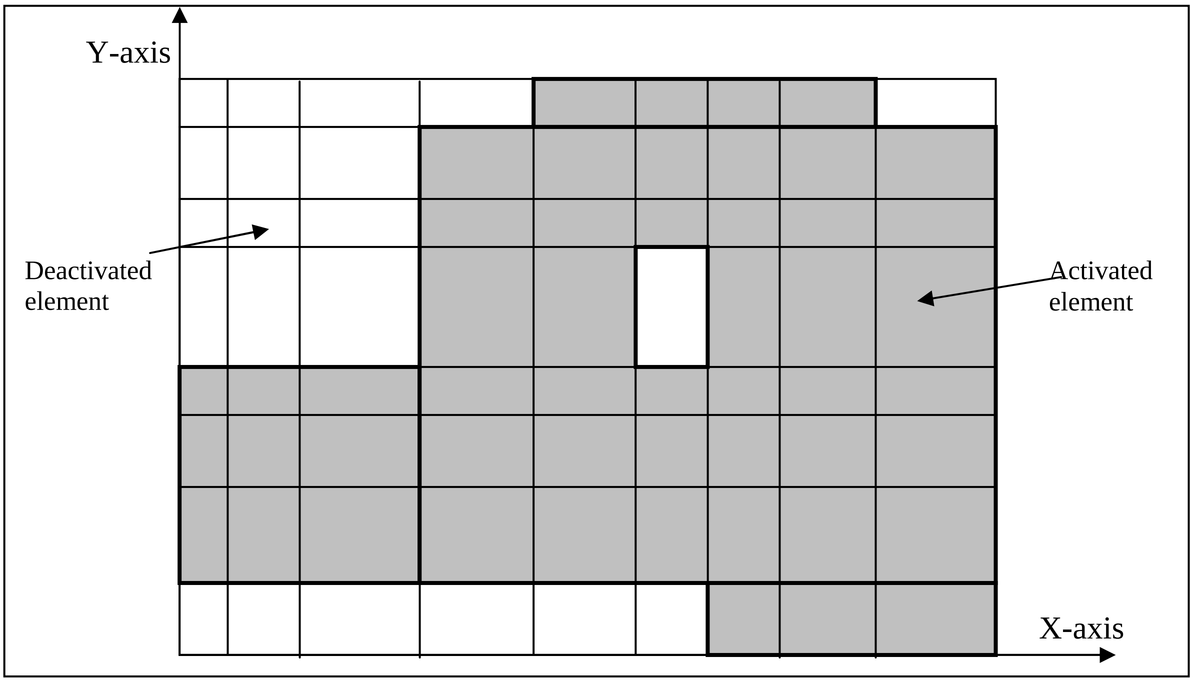


Figure 12: Element deactivation technique

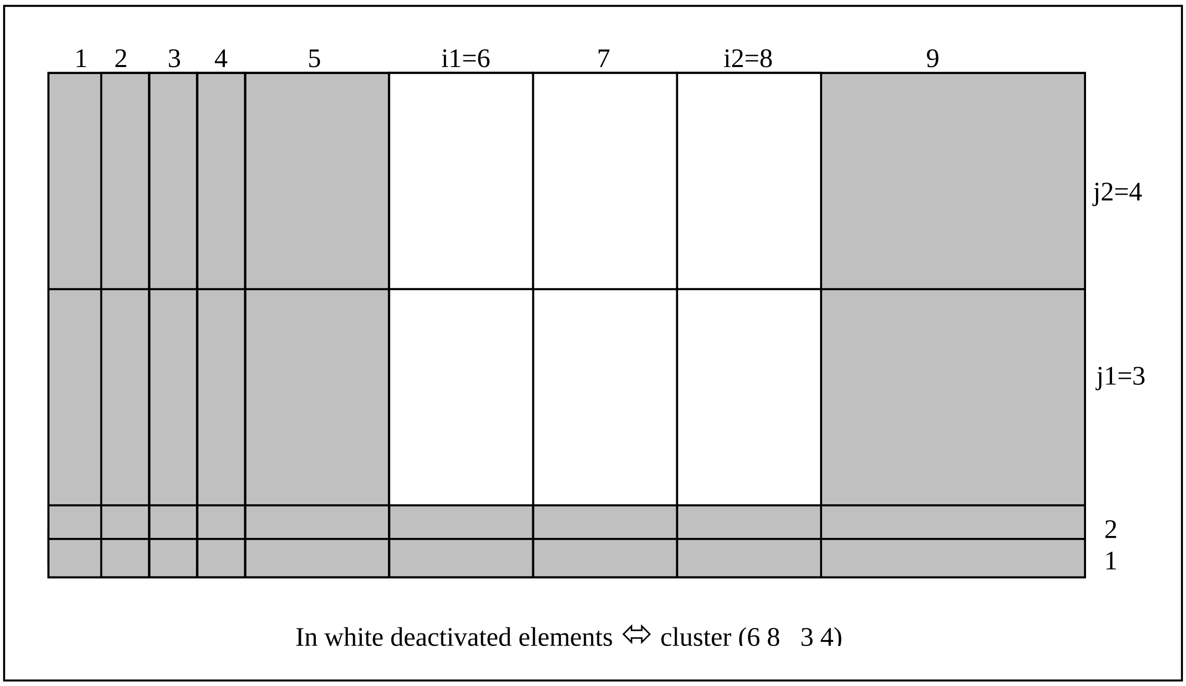
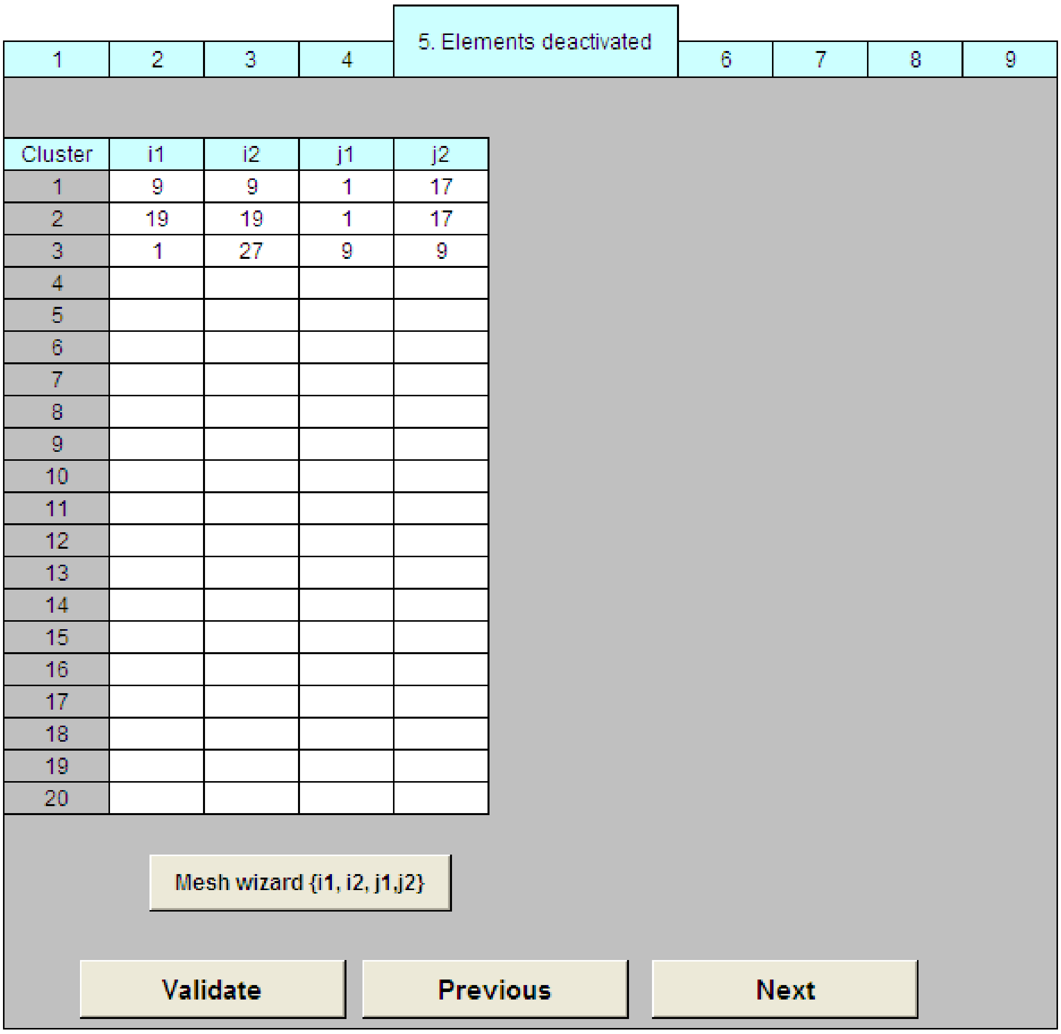


Figure 13: Elements deactivated (example)



Cluster	i1	i2	j1	j2
1	9	9	1	17
2	19	19	1	17
3	1	27	9	9
4				
5				
6				
7				
8				
9				
10				
11				
12				
13				
14				
15				
16				
17				
18				
19				
20				

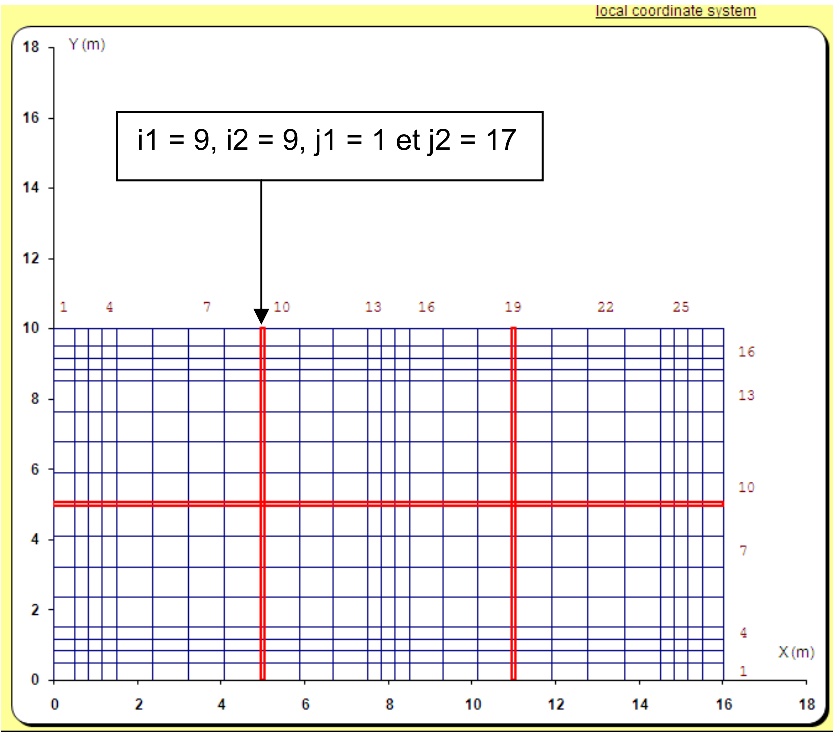


Figure 14: Example of element deactivation

3.7 Defining the mechanical properties of the plate

The properties of the plate are presumed uniform for each element. Each element is characterised by its Young's modulus 'E', its 'bare' Poisson's ratio, as well as its thickness 'h'. This data can be assigned by groups of elements. The allocation principle is identical to that used for deactivating elements: i.e. allocation by groups of elements.

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

Here again we use group definitions of the 'i1 i2 j1 j2' type.

The clusters created are outlined by a red line in the graph of the application.

When defining a small cluster with different characteristics inside a larger cluster, first define the larger cluster, then the smaller cluster with its different characteristics. The characteristics of the small cluster 'overwrite' and replace those defined previously.

The table below summarises the parameters to enter:

Designation	Unit	Value by default	Condition of display	Mandatory value	Local checks
i1	None	---	Always	Yes	>0 and <= Total number of subdivisions along x-axis
i2	None	---	Always	Yes	>=i1 and <= Total number of subdivisions along x-axis
j1	None	---	Always	Yes	>0 and <= Total number of subdivisions along (y-axis)
j2	None	---	Always	Yes	>=j1 and <= Total number of subdivisions along x-axis
Young's modulus of the plate	kPa	---	Always	Yes	>0
Poisson's ratio	None	---	Always	Yes	>0 and < 0.5
Plate thickness	m	---	Always	Yes	>0

Table 7: Parameters for allocating mechanical characteristics

1	2	3	4	5	6. Mechanical properties of the plate	7	8 9
Cluster	i1	i2	j1	j2	Modulus E (kPa)	Poisson's ratio	Thickness (m)
1	1	36	1	28	1.00E+07	0.2	0.5
2	1	8	3	10	1.00E+07	0.2	1
3							
4							
5							
6							
7							
8							
9							
10							
11							
12							
13							
14							
15							
16							
17							
18							
19							
20							

Mesh wizard {i1, i2, j1,j2}

Calculation wizard for combined section

Validate

Previous

Next

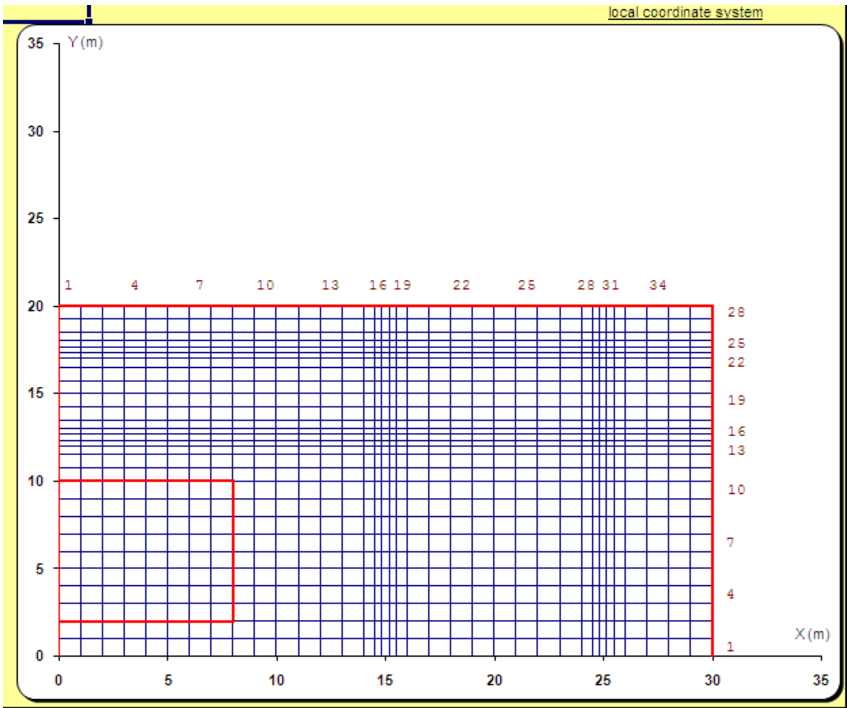


Figure 15: Example of definition of the mechanical properties of the plate

The ‘Calculation wizard for combined section’ button starts the wizard for calculating a combined section using the data to be entered as shown in the figure below.

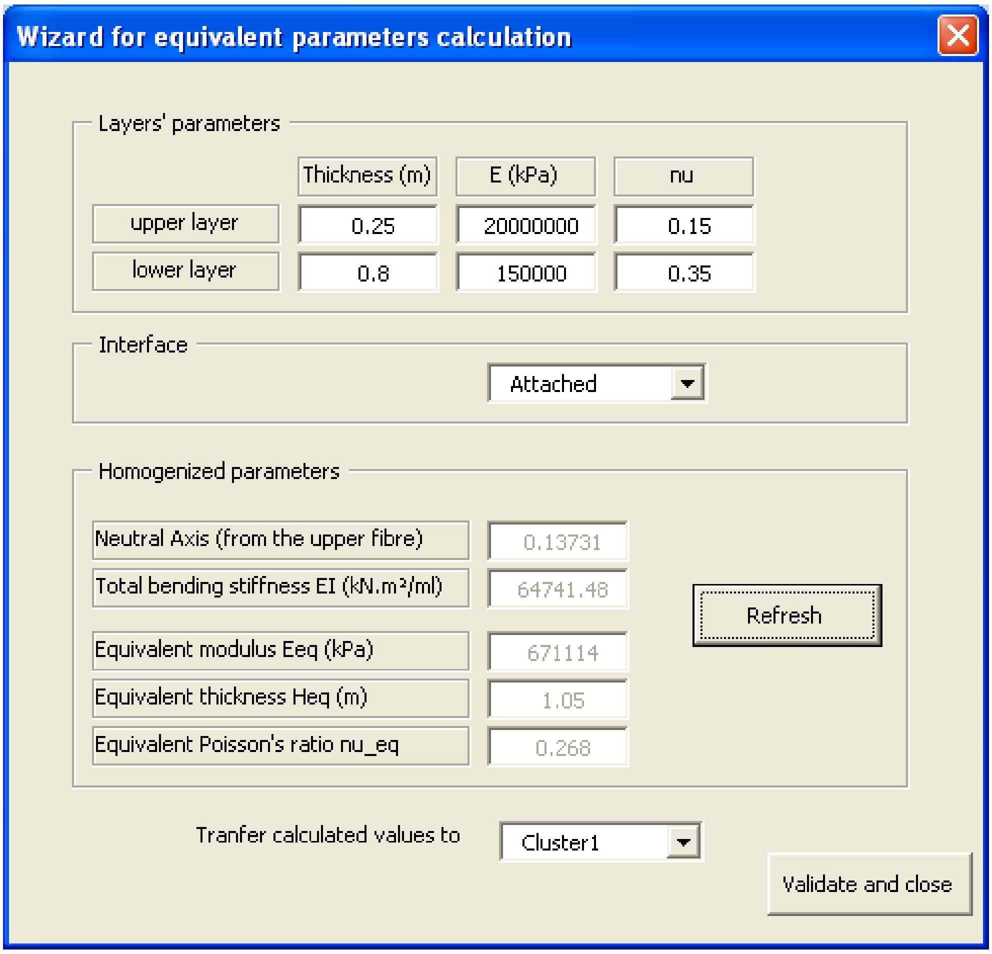


Figure 16: combined section calculation

This wizard allows defining equivalent mechanical properties or what can be called ‘homogenised parameters’, if the plate section is not homogeneous. Please note that the use of this technique may be ‘useful’ in certain specific cases, as the one described in tutorial 8.

3.8 Defining the load distributed on the plate

This tab allows defining one or several loads distributed over the plate, as well as any one or several distributed springs under the plate. As previously, this load is defined by groups of elements.

This step is not mandatory.

Designation	Unit	Value by default	Condition of display	Mandatory value	Local checks
i1	None	---	Always	Yes	>0 and <= total number of subdivisions along x-axis
i2	None	---	Always	Yes	>=i1 and <= total number of subdivisions along x-axis
j1	None	---	Always	Yes	>0 and <= total number of subdivisions along (y-axis)
j2	None	---	Always	Yes	>=j1 and <= total number of subdivisions along x-axis
Load distributed vertically on the plate	kPa	---	Always	Yes	---
Springs*	kPa/m	---	Always	Yes	Positive

*: stiffness distributed in displacement under the plate, for example representative of a distribution of juxtaposed springs

Table 8: Parameters for distributed load

If several loads are defined over the same cluster, they are added. The operation is the same for springs.

1	2	3	4	5	6	7. Distributed load	8	9

Cluster	i1	i2	j1	j2	Load (kPa)	Stiffness (kPa/m)
1	2	4	2	4	200	0
2	24	26	2	4	200	0
3	2	4	14	16	200	0
4	24	26	14	16	200	0
5	13	15	2	4	200	0
6	13	15	14	16	200	0
7	1	27	1	17	6.25	0
8	7	21	7	11	40	0
9						
10						
11						
12						
13						
14						
15						
16						
17						
18						
19						
20						

Mesh wizard {i1, i2, j1,j2}

Validate | Previous | Next

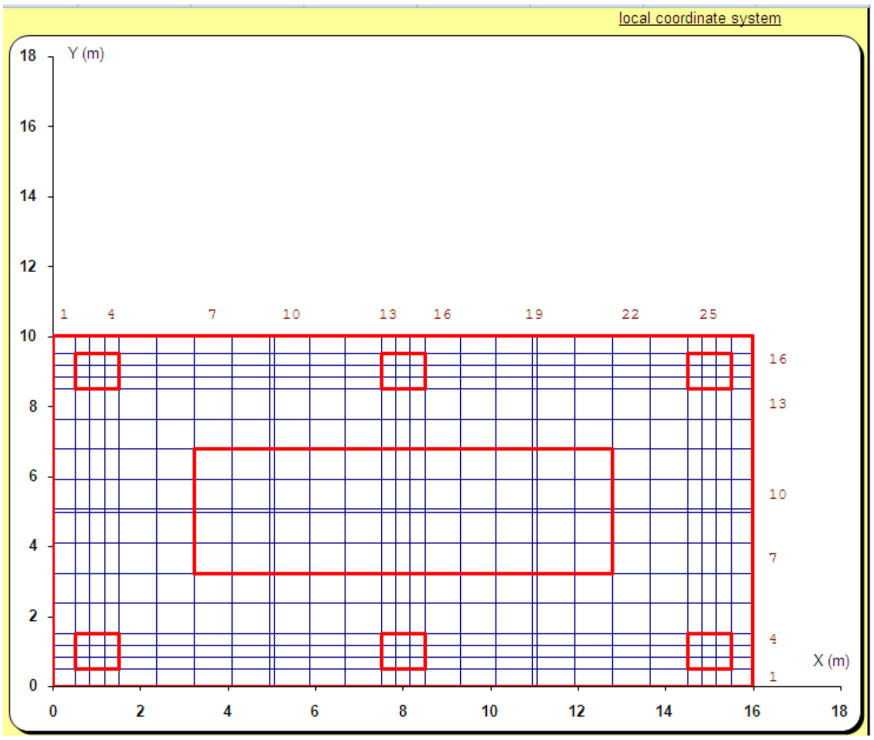


Figure 17: Example of load distributed on the plate

3.9 Defining load on nodes

Each load on nodes is made of a vertical load, two bending moments, one spring in translation, two springs in rotation. This data is assigned by groups of nodes. Each is defined using nodes with maximum/minimum index. The principle for each group's coordinates is similar to that used for groups of elements.

The values entered apply to each of the nodes in the cluster.

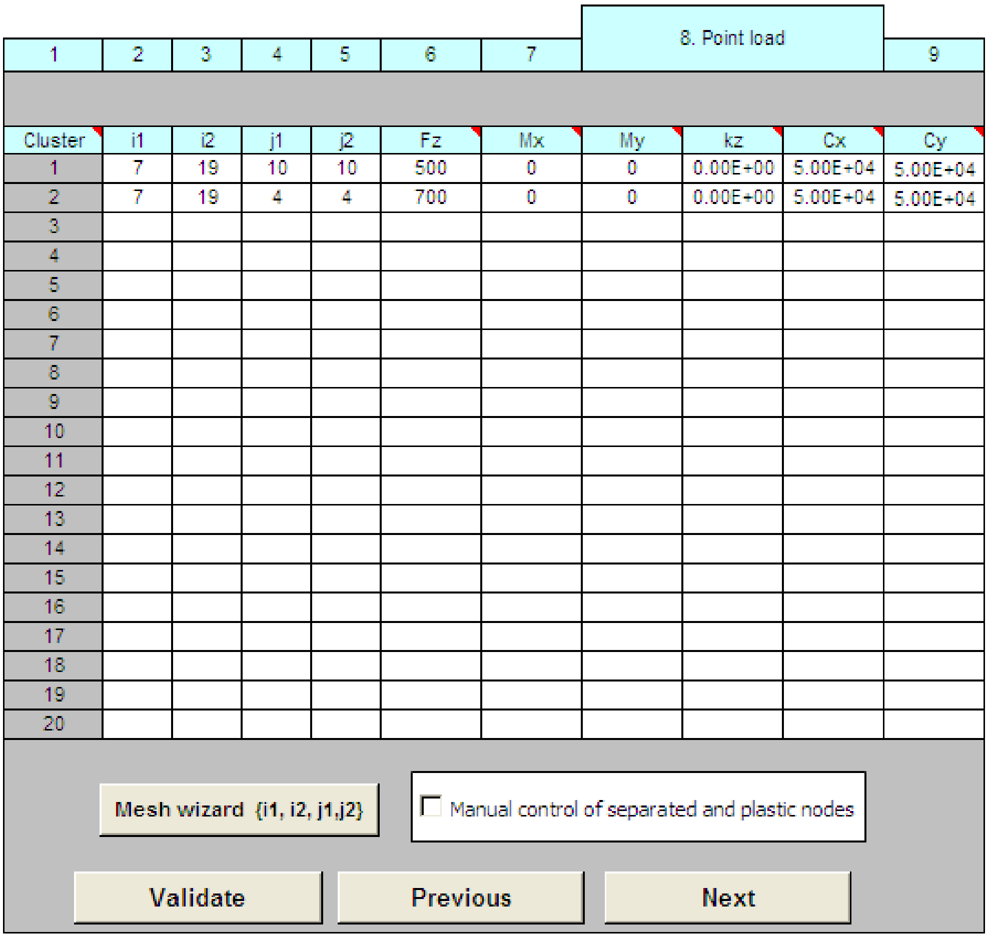
This step is not mandatory.

The table below summarises the parameters to enter:

Designation	Unit	Value by default	Condition of display	Mandatory value	Local checks
i1	None	---	Always	Yes	>0 and <= total number of subdivisions along x-axis + 1
i2	None	---	Always	Yes	>=i1 and <= total number of subdivisions along x-axis +1
j1	None	---	Always	Yes	>0 and <= total number of subdivisions along y-axis +1
j2	None	---	Always	Yes	>=j1 and <= total number of subdivisions along x-axis +1
Fz (vertical point load)	kN	---	Always	Yes	---
Mx (moment around the y-axis)	kN.m	---	Always	Yes	---
My (moment around the x-axis)	kN.m	---	Always	Yes	---
Kz (linear spring under the plate)	kN/m	---	Always	Yes	Positive
Cx (rotation spring around the y-axis)	kN.m/ rad	---	Always	Yes	Positive
Cy (rotation spring around the x-axis)	kN.m/ rad	---	Always	Yes	Positive
Manual management of Node Separation/Plastification	---	Unchecked	Always	---	The number of soil layers must be positive

Table 9: Parameters for loads on nodes

The 'Manual control of separated and plastic points' option allows the user to define manually the nodes to declare as separated or plastified. In this case, a new tab 'nodes to separate / plastify' appears (see § 3.11).



1	2	3	4	5	6	7	8. Point load			9
Cluster	i1	i2	j1	j2	Fz	Mx	My	kz	Cx	Cy
1	7	19	10	10	500	0	0	0.00E+00	5.00E+04	5.00E+04
2	7	19	4	4	700	0	0	0.00E+00	5.00E+04	5.00E+04
3										
4										
5										
6										
7										
8										
9										
10										
11										
12										
13										
14										
15										
16										
17										
18										
19										
20										

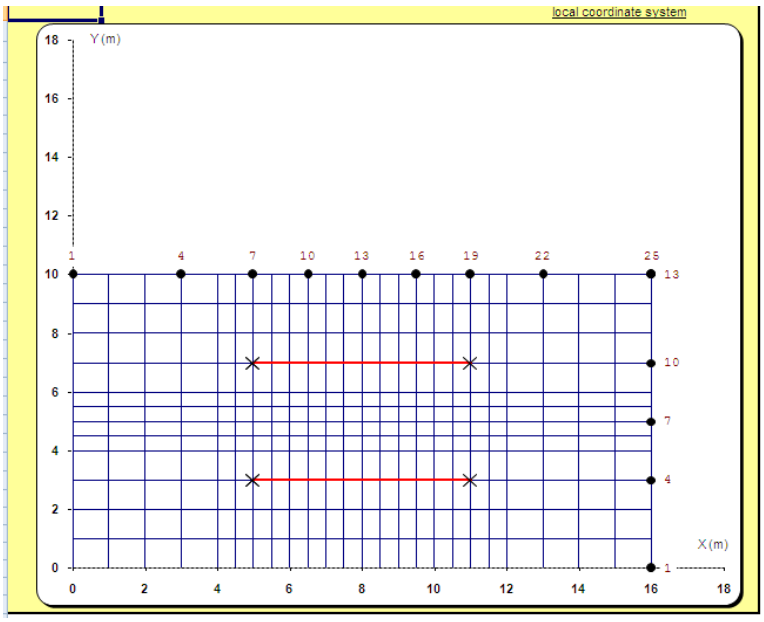


Figure 18: Example of point load

3.10 Defining external loads applied to the soil

In addition to pressure applied by the plate, the soil may be subject to 'direct' external loads. These loads are presumed rectangular shaped, positioned and turned in the global coordinate system.

The following figure describes the global position of the problem:

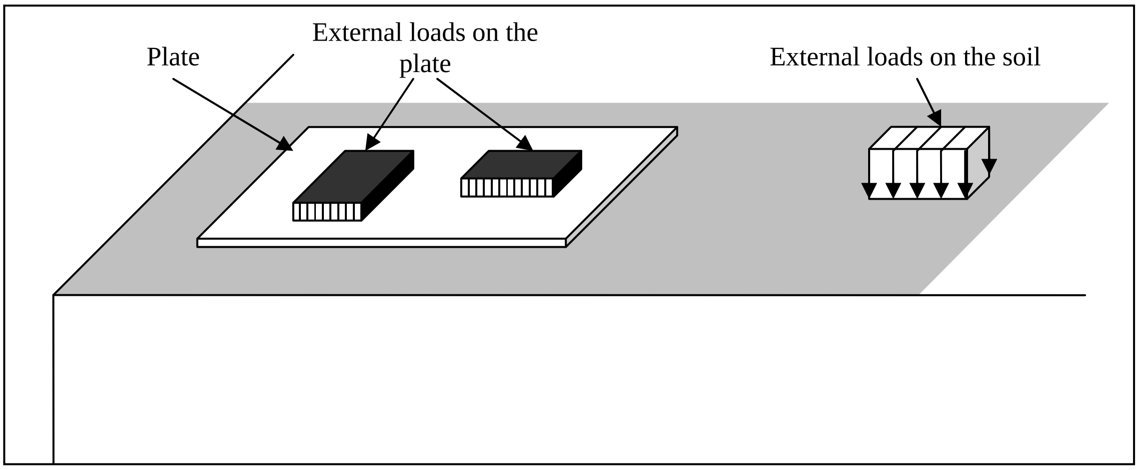


Figure 19: Global layout of the problem {Plate + Soil + External loads}

Each load is characterised by the coordinates of its 'lower – left' top (Xr, Yr, Zr), dimensions (DLX width and DLY length), orientation (θr), as well as its load (qr).

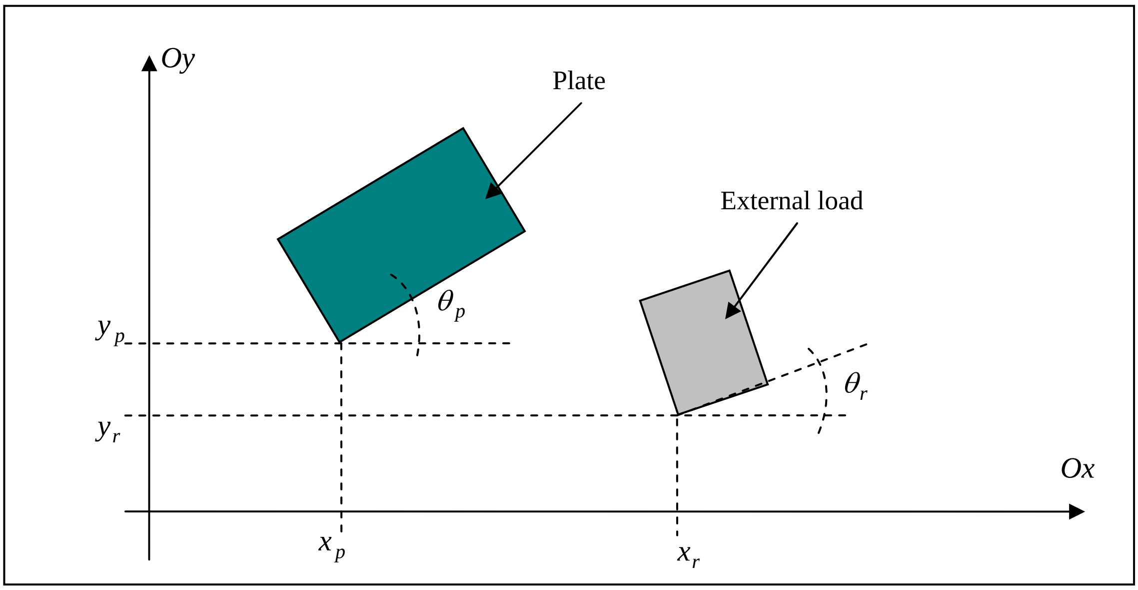


Figure 20: Coordinates of external loads

Tasplaq proposes a top view of these loads, as well as of the plate. We can note that the external loads are not always oriented in parallel with the x-axis and y-axis axes (Figure 20): they can be placed with any angle in respect of these axes.

This step is not mandatory.

1	2	3	4	5	6	7	8	9. Loads on surrounding soil

Load	XR (m)	YR (m)	ZR (m)	DLX (m)	DLY (m)	Teta (°)	qr (kPa)
1	20	10	25	10	8	10	30
2	5	6	0	3	8	0	25
3							
4							
5							
6							
7							
8							
9							
10							
11							
12							
13							
14							
15							
16							
17							
18							
19							
20							

Validate Previous Next

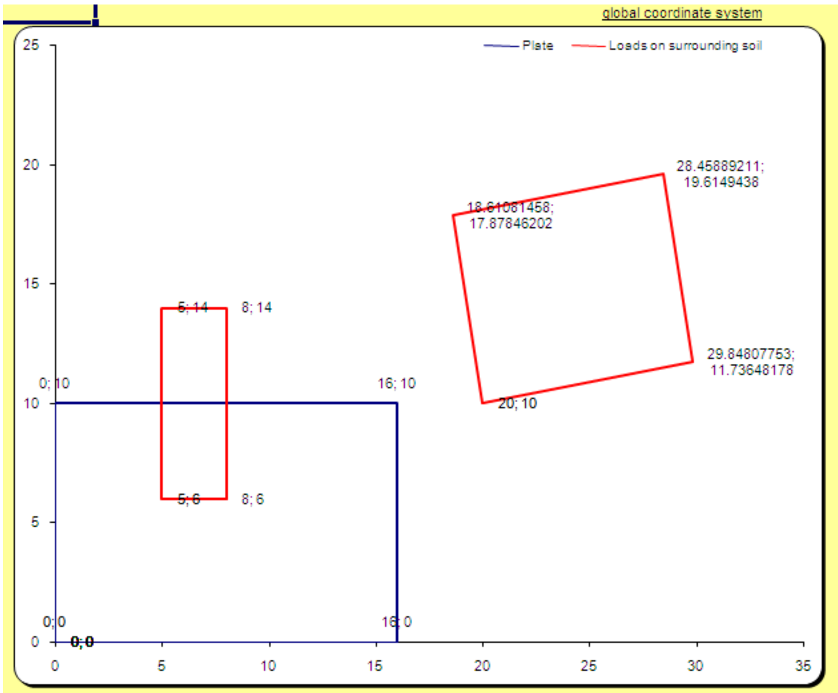


Figure 21: Example of external loads on the soil

Designation	Unit	Value by default	Condition of display	Mandatory value	Local checks
XR	m	---	Always	Yes	---
YR	m	---	Always	Yes	---
ZR	m	---	Always	Yes	---
DLX	m	---	Always	Yes	>0
DLY	m	---	Always	Yes	>0
Theta	°	---	Always	Yes	---
qr	kPa	---	Always	Yes	---

Table 10: Setting of external loads on the soil.

3.11 Manual control of separated and plastic nodes (optional)

This button allows to impose the following manually:

- Separation of certain nodes: the soil's reaction then equals 0 and soil settlement no longer equals the vertical displacement of the plate.
- Plastification of certain nodes: the soil's reaction imposed equals the plastification threshold defined in 'general settings'. Equality between soil settlement and the vertical displacement of the plate is always ensured.

The 'separation/plastification manual management' can be combined with the 'automatic calculation': Indeed, if the 'automatic calculation' option is activated, TASPLAQ checks separation/plastification beside all nodes, except those declared separated/plastified manually by the user.

This option corresponds to an advanced use of Tasplaq.

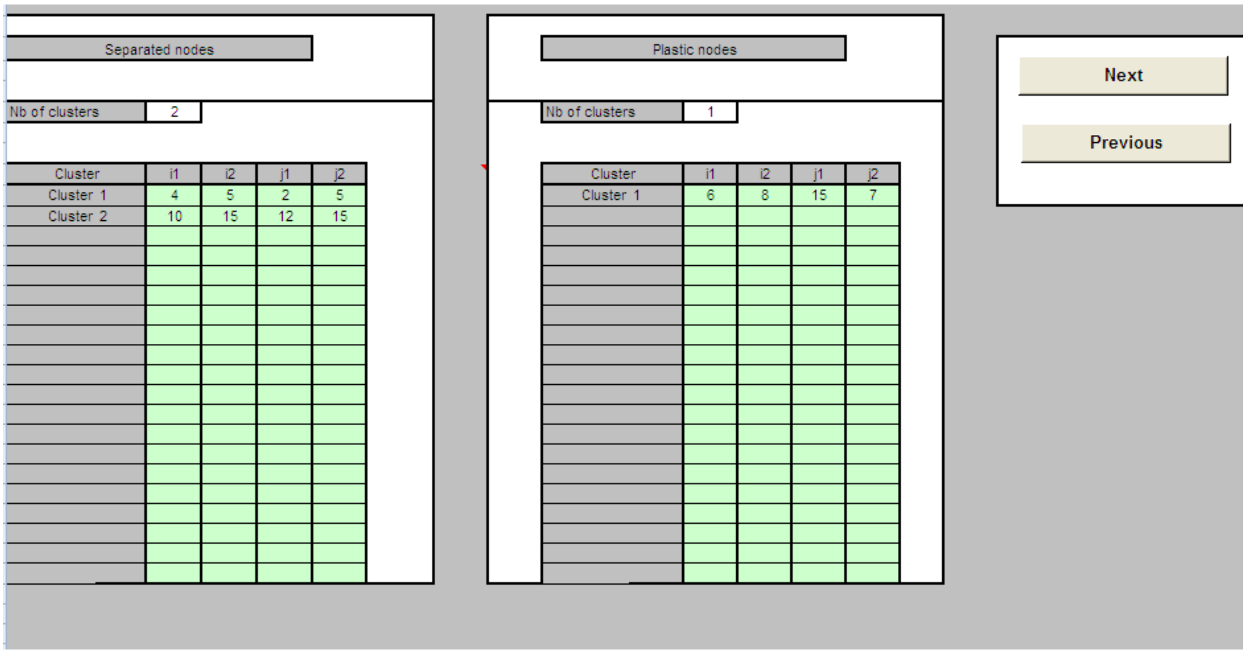


Figure 22: manual definition of node separation / plastification - example

If this option is not activated, the number of nodes separated and plastified is reset to zero.

Of course, this step is not mandatory.

Designation	Unit	Value by default	Condition of display	Mandatory value	Local checks
i1	None	---	Always	Yes	>0 and <= total number of subdivisions along x-axis + 1
i2	None	---	Always	Yes	>=i1 and <= total number of subdivisions along x-axis +1
j1	None	---	Always	Yes	>0 and <= total number of subdivisions along y-axis +1
j2	None	---	Always	Yes	>=j1 and <= total number of subdivisions along x-axis +1
Number of clusters	None	---	Always	Yes	>= 0

Table 11: Parameters for manual separation/plastification management

4. CALCULATIONS

No calculation is performed under Microsoft Excel® interactive. The latter allows only to generate the data file (Filename.tpl) to be read and run by the TASPLAQ.exe calculation engine (then use the results returned by the calculation engine).

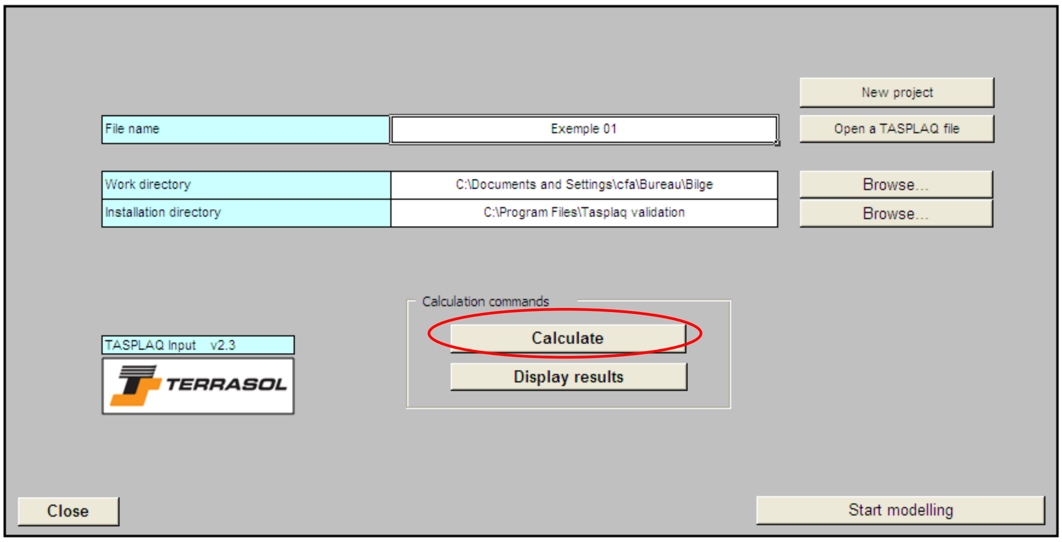


Figure 23: homepage of Tasplaq input

The calculation engine is developed under Visual Compaq Fortran. The matrix systems are resolved directly. Non linear procedures (separation, plastification…) are managed iteratively.

No digital limit is considered in the program in terms of model size. However, a limit may exist due to the maximum memory size which can be assigned to the program under Microsoft Windows: this limit is estimated at à 2500 activated elements.

The general calculation process is led according to the following steps:

1. Read the data – Open the files
2. Initialise the variables
3. Construct the mesh
4. Assemble the external load vector
5. Assemble the plate's rigidity matrix
6. Calculate the soil's flexibility matrix (if there is a soil)
7. Construct the global equation system
8. Matrix resolution
9. Calculate displacements and forces in the plate
10. Calculate settlements and reactions in all nodes (if there is a soil)
11. Check separation/plastification on surface (if positive, back to step 4)
12. Generate output files (results, graphs)
13. End of program.

The user is informed of progress of the different calculation steps through a DOS window (next figure).

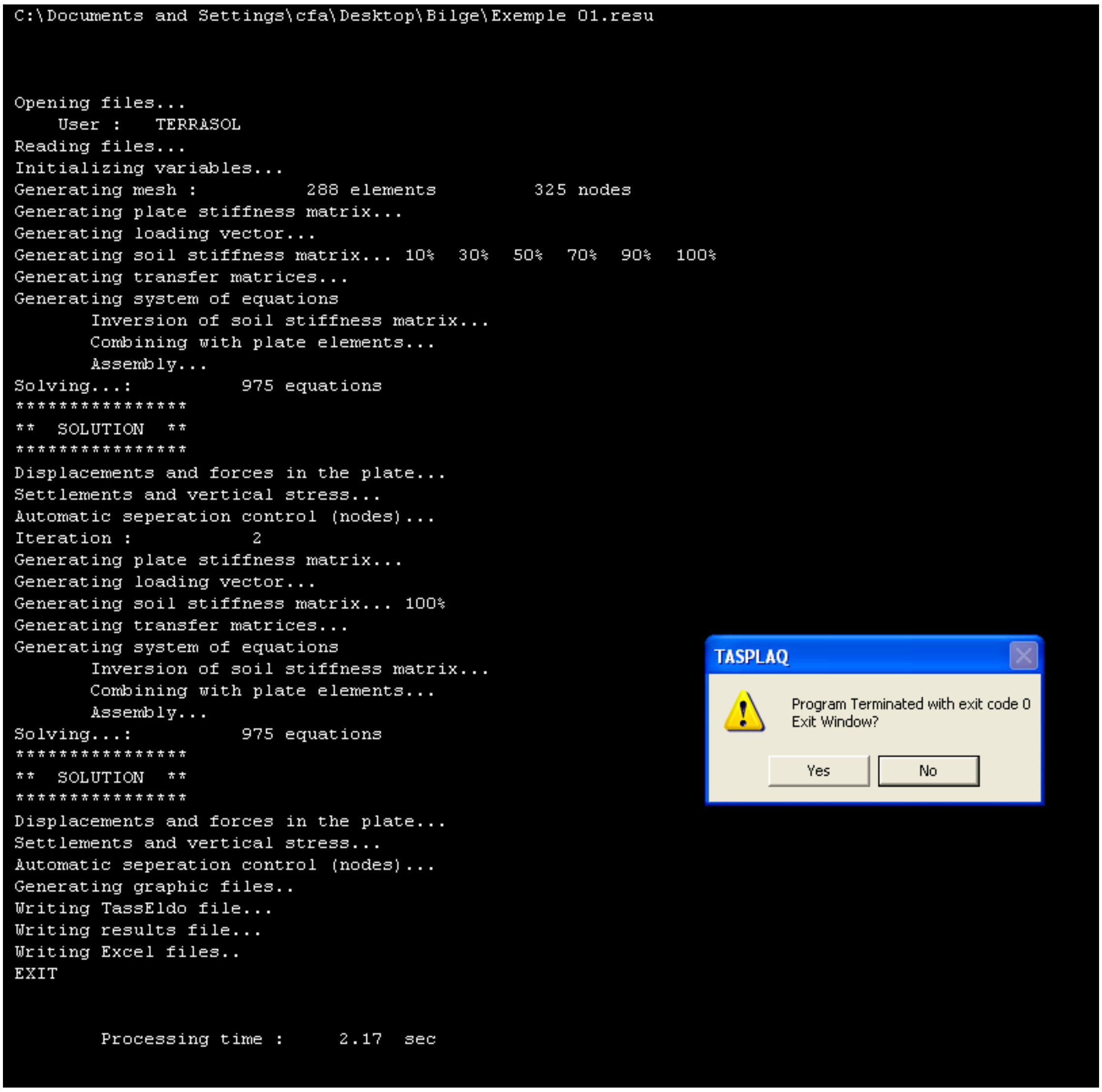


Figure 24: calculation window

At the end of the calculation, just click the [Yes] button.

5. RESULTS

The results can be viewed by clicking the [Display results] button of the TASPLAQ_vx.x.xls file. The Microsoft Excel® TASPLAQ Output_vx.x.xls file opens:

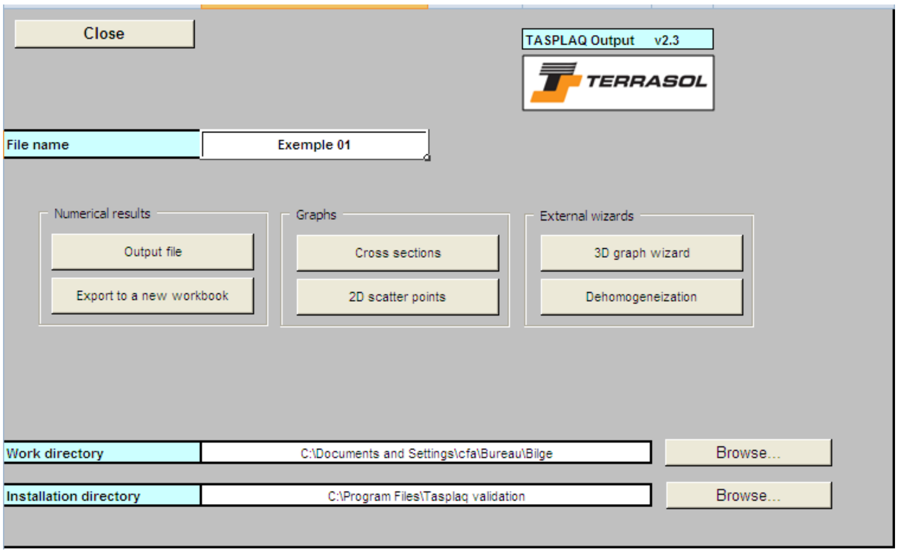


Figure 25: home page of Tasplaq output

6 types of results are available.

5.1 Result file

This button provides access to the content of the Filename.resu file in the text format (Notepad).
This file contains a summary of the project's data, as well as the results.

Exemple 01.resu - Notepad

File Edit Format View Help

```
PROGRAM TASPLAQ v2.3.5                          (c) TERRASOL 2009

File :  C:\Documents and Settings\cfa\Desktop\Bilge\Exemple 01.resu

Calculation made on :  06/17/2009 at 17h17
              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       0.400E+05        0.350

Plate geometry

                               General dimensions
                               ------------------
       XP           YP          LX           LY          ZP        rotation(°)
      0.00         0.00        16.00        10.00       0.00         0.00

                              No elements deactivation

Plate properties (local coordinate system)

Cluster     Xmin        Xmax       Ymin        Ymax        Modulus       nu       Tickness     density

 01         0.00        16.00      0.00        10.00       0.200E+08    0.150      0.25         0.00

Decomposition of the plate

   Element             DLX          DLY           EI            nu       Density          status

 ( 001 , 001 )        1.000        1.000       0.260E+05      0.150      0.000             ***
```

Figure 26: *.resu file - example

5.2 Export to a new spreadsheet

This button exports the digital results to a new Microsoft Excel® spreadsheet.
This new spreadsheet contains the results at each calculation point issued from the mesh defined beforehand, as well as tables indicating the maximum and minimum values for settlement, reactions, moments, and deflection.

Settlement (s) and Vertical stress (p)

s_max+ (m)	-1.44E-02
s_max- (m)	-1.71E-03
p_max+ (kPa)	31.3
p_max- (kPa)	0.0

Plate deflection (w)

w_max+ (m)	-1.44E-02
w_max- (m)	-1.51E-03

Bending moments

MxMax+ (kN.m/ml)	21.0
MxMax- (kN.m/ml)	-4.4
MyMax+ (kN.m/ml)	48.8
MyMax- (kN.m/ml)	-2.1

Numerical results - Local coordinate system

Xn	Yn	s	p	Xe	Ye	w	Xm	Ym	Mx	My	H
0	0	-0.00171	0	0	0	-0.00151	0.25	0.25	0.894785	1.16127	0.25
1	0	-0.00231	2.2388	0.5	0	-0.00192	0.75	0.25	2.51392	1.14972	0.25
2	0	-0.00301	6.47203	1	0	-0.00231	1.25	0.25	2.97624	0.98233	0.25
3	0	-0.00359	8.15301	1	0	-0.00231	1.75	0.25	2.7893	0.735231	0.25
4	0	-0.00409	8.95222	1.5	0	-0.00268	2.25	0.25	2.58126	0.526311	0.25
4.5	0	-0.0043	9.22434	2	0	-0.00301	2.75	0.25	2.30201	0.348049	0.25
5	0	-0.00448	9.39529	2	0	-0.00301	3.25	0.25	2.29576	0.188488	0.25
5.5	0	-0.00465	9.51871	2.5	0	-0.00332	3.75	0.25	2.28374	0.005816	0.25
6	0	-0.00478	9.60539	3	0	-0.00359	4.125	0.25	2.50177	-0.10124	0.25
6.5	0	-0.00489	9.66392	3	0	-0.00359	4.375	0.25	2.57639	-0.18869	0.25
7	0	-0.00497	9.70063	3.5	0	-0.00385	4.625	0.25	2.69991	-0.26575	0.25
7.5	0	-0.00501	9.72042	4	0	-0.00409	4.875	0.25	2.8117	-0.34153	0.25
8	0	-0.00503	9.7266	4	0	-0.00409	5.125	0.25	2.94836	-0.40778	0.25

Figure 27: Exporting the file under Microsoft Excel®

5.3 Cross sections

This button shows different magnitudes according to the cross-sections through the plate.

The [Cross-sections] window reminds maximum values of the project for settlement, reactions, moments, and deflection.

The right-hand side shows four drop-down lists used to configure the cross-sections displayed.
The first 3 lists are used to select the magnitude to represent, the cross-section direction and its localization. The graphic plot of the cross-section is updated automatically.
The fourth list allows selecting the localization of a potential 2nd cross-section, which will be superimposed onto the drawing at the first (allowing very easy comparisons).

For example, to compare settlement along the x-axis in Y = 5 and Y = 7, select the 'Settlement' magnitude, cross-section along X for values Y = 5 and Y = 7 (Figure 28).

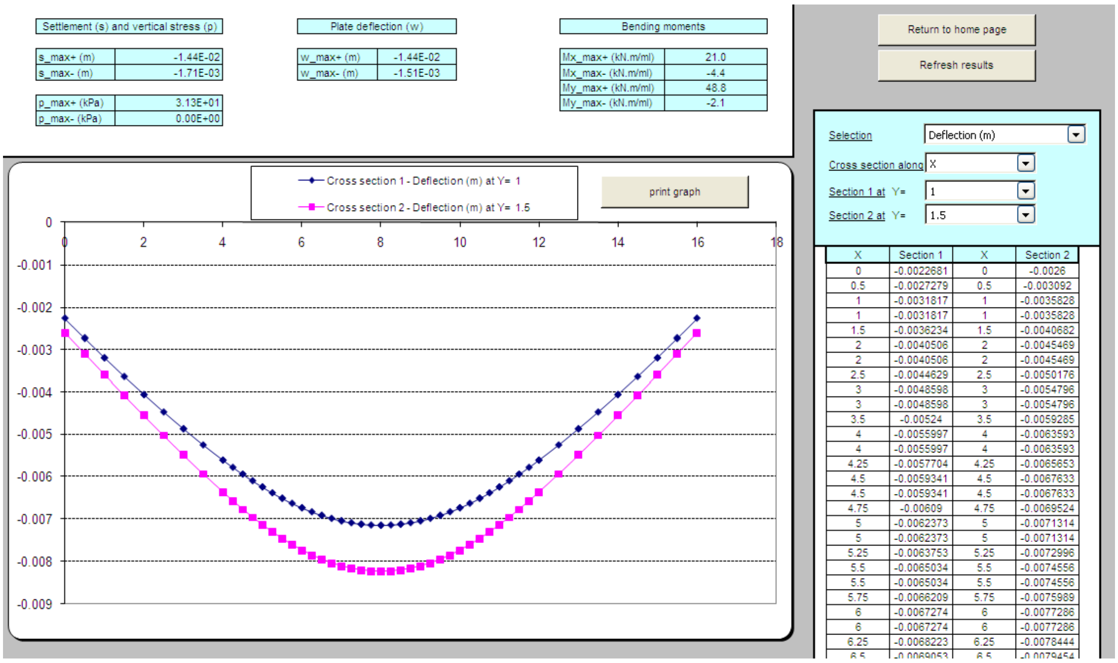


Figure 28: Settlement along X in Y = 5 and Y = 7

5.4 2D scatter points

This option allows to display the different magnitudes calculated in the form of scatter points.

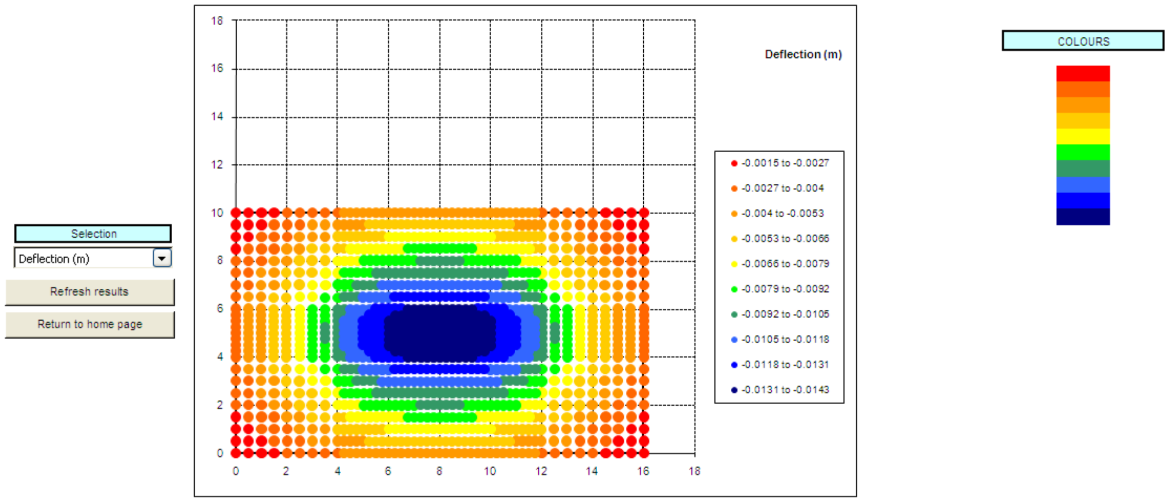


Figure 29: 2D scatter points

The left part includes a drop-down list, allowing to choose the magnitude to represent: the scatter point drawing is updated automatically after selection.

The point of this window is to help the user to view the distribution of a given magnitude, allowing notably to choose the most appropriate cross sections.
The caption to the right of the scatter points details the different ranges of values matching each colour.

5.5 3D Graph wizard

This option is used to represent the results in the form of a 3D surface.
The appropriate button allows to open the Microsoft Excel® TASPLAQ Graphique3D.xls file:

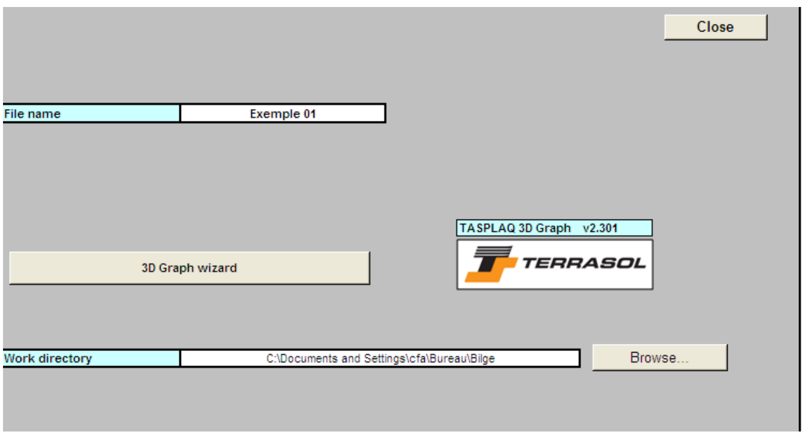


Figure 30: Home window of TASPLAQ Graphique3D.xls

The 3D Graph window is composed of two drop-down lists.
To create a view, select in the ‘View’ drop-down list a 3D view or plane view, and in the ‘Selection’ drop-down list, select the quantity to show.

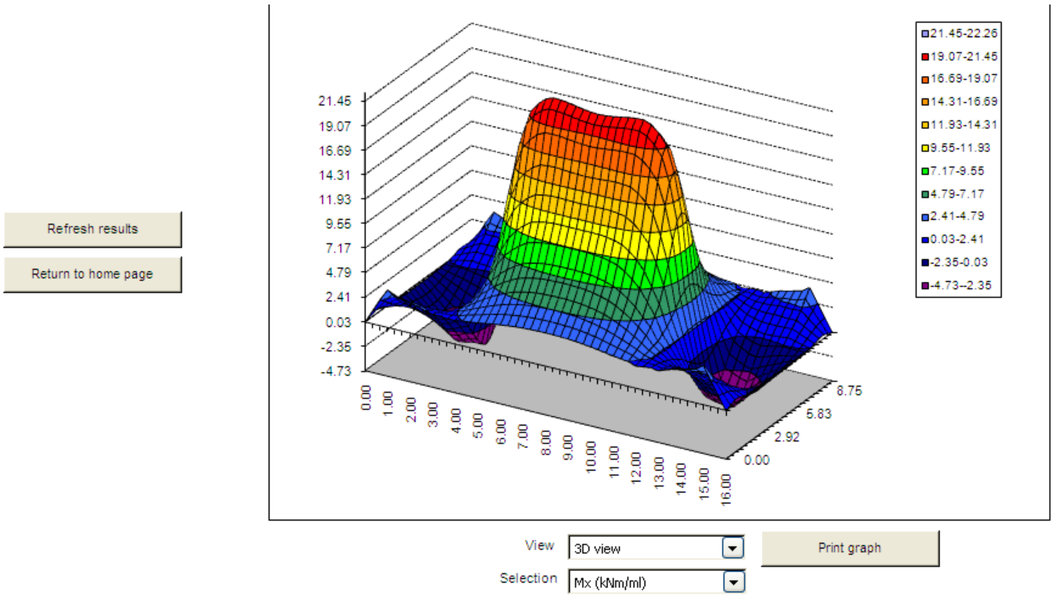


Figure 31: 3D Graph window

5.6 Dehomogenisation

This option can be used only when using the 'Combined section' wizard.

Click the [Dehomogenisation] button. A Microsoft Excel® file opens.

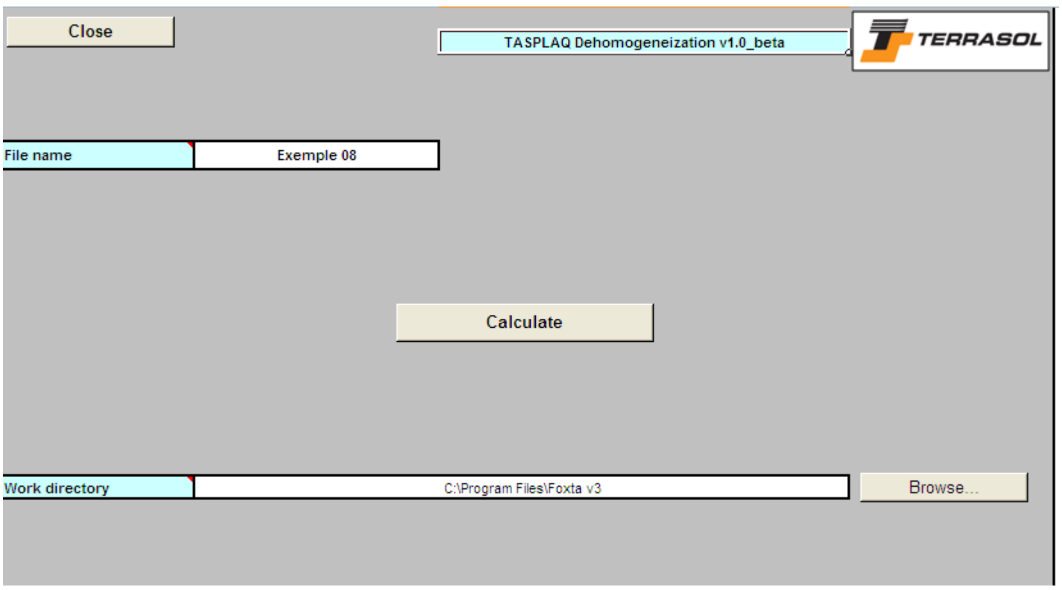


Figure 32: Home window of TASPLAQ Deshomogenisation.xls

Enter the parameters requested (thickness, Poisson's ratio and Young's modulus) for the lower layer, as well as the nature of the interface between the two layers.

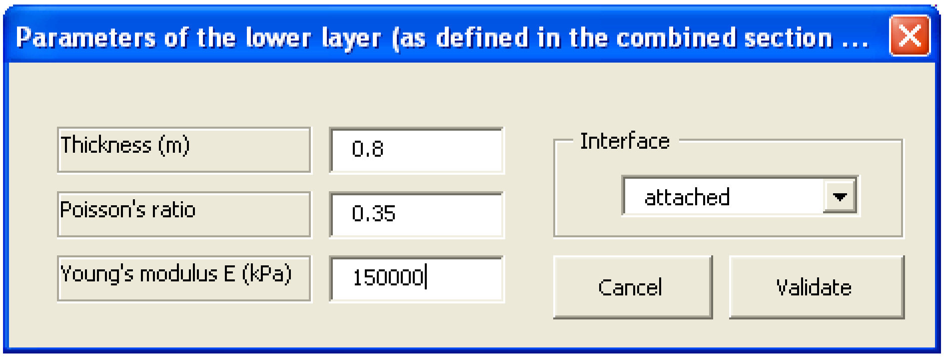


Figure 33: Window [Characteristics of the lower layer]

After validation, another Microsoft Excel® file is opened: it contains the homogenised data issued from the calculation, as well as the dehomogenised loads: bending moments and axial forces in concrete (upper layer). The occurrence of axial forces is due to the fact the neutral plane of the equivalent plate does not always match the centre of the homogenised area.

Homogenized values in the combined section

k0	1440
4*ke	640

Xm	Ym	M1	M2	h_eq	Ei_eq	Nu_eq
0.25	0.25	0.936738	1.60184	1.05	64741.1	0.268
0.75	0.25	3.14167	1.62264	1.05	64741.1	0.268
1.25	0.25	4.12742	1.412	1.05	64741.1	0.268
1.75	0.25	4.56847	1.15067	1.05	64741.1	0.268
2.25	0.25	4.98189	0.923307	1.05	64741.1	0.268
2.75	0.25	5.31249	0.715124	1.05	64741.1	0.268
3.25	0.25	5.91043	0.543375	1.05	64741.1	0.268
3.75	0.25	6.49207	0.332053	1.05	64741.1	0.268
4.25	0.25	7.34551	0.183007	1.05	64741.1	0.268
4.75	0.25	8.09455	-0.00458	1.05	64741.1	0.268
5.25	0.25	8.97463	-0.12324	1.05	64741.1	0.268
5.75	0.25	9.65964	-0.26037	1.05	64741.1	0.268
6.25	0.25	10.3225	-0.34162	1.05	64741.1	0.268
6.75	0.25	10.77	-0.41874	1.05	64741.1	0.268
7.25	0.25	11.1062	-0.45934	1.05	64741.1	0.268
7.75	0.25	11.2569	-0.48328	1.05	64741.1	0.268
8.25	0.25	11.2569	-0.48328	1.05	64741.1	0.268
8.75	0.25	11.1062	-0.45934	1.05	64741.1	0.268
9.25	0.25	10.77	-0.41874	1.05	64741.1	0.268
9.75	0.25	10.3225	-0.34162	1.05	64741.1	0.268
10.25	0.25	9.65964	-0.26037	1.05	64741.1	0.268
10.75	0.25	8.97463	-0.12324	1.05	64741.1	0.268
11.25	0.25	8.09455	-0.00458	1.05	64741.1	0.268
11.75	0.25	7.34551	0.183007	1.05	64741.1	0.268
12.25	0.25	6.49207	0.332053	1.05	64741.1	0.268
12.75	0.25	5.91043	0.543375	1.05	64741.1	0.268
13.25	0.25	5.31249	0.715124	1.05	64741.1	0.268
13.75	0.25	4.98189	0.923307	1.05	64741.1	0.268
14.25	0.25	4.56847	1.15067	1.05	64741.1	0.268
14.75	0.25	4.12742	1.412	1.05	64741.1	0.268
15.25	0.25	3.14167	1.62264	1.05	64741.1	0.268

Parameters of the lower layer (as defined in the combined section wizard)

Thickness (m)	0.8
Poisson's ratio	0.35
Young's modulus E (kPa)	150000

Refresh

Return to home page

Bending moments and axial forces in the upper layer (concrete)

Mx	My	Nx	Ny		Xm	Ym	hc
0.29	0.59	0.69	1.39		0.25	0.25	0.25
1.16	0.49	2.75	1.16		0.75	0.25	0.25
1.56	0.36	3.69	0.85		1.25	0.25	0.25
1.75	0.24	4.13	0.56		1.75	0.25	0.25
1.92	0.13	4.54	0.30		2.25	0.25	0.25
2.06	0.03	4.88	0.07		2.75	0.25	0.25
2.31	-0.07	5.45	-0.16		3.25	0.25	0.25
2.55	-0.18	6.02	-0.42		3.75	0.25	0.25
2.89	-0.28	6.83	-0.66		4.25	0.25	0.25
3.20	-0.39	7.55	-0.92		4.75	0.25	0.25
3.55	-0.48	8.39	-1.13		5.25	0.25	0.25
3.83	-0.56	9.04	-1.33		5.75	0.25	0.25
4.09	-0.63	9.67	-1.48		6.25	0.25	0.25
4.27	-0.68	10.10	-1.61		6.75	0.25	0.25
4.41	-0.71	10.42	-1.68		7.25	0.25	0.25
4.47	-0.73	10.56	-1.72		7.75	0.25	0.25
4.47	-0.73	10.56	-1.72		8.25	0.25	0.25
4.41	-0.71	10.42	-1.68		8.75	0.25	0.25
4.27	-0.68	10.10	-1.61		9.25	0.25	0.25
4.09	-0.63	9.67	-1.48		9.75	0.25	0.25
3.83	-0.56	9.04	-1.33		10.25	0.25	0.25
3.55	-0.48	8.39	-1.13		10.75	0.25	0.25
3.20	-0.39	7.55	-0.92		11.25	0.25	0.25
2.89	-0.28	6.83	-0.66		11.75	0.25	0.25
2.55	-0.18	6.02	-0.42		12.25	0.25	0.25
2.31	-0.07	5.45	-0.16		12.75	0.25	0.25
2.06	0.03	4.88	0.07		13.25	0.25	0.25
1.92	0.13	4.54	0.30		13.75	0.25	0.25
1.75	0.24	4.13	0.56		14.25	0.25	0.25
1.56	0.36	3.69	0.85		14.75	0.25	0.25
1.16	0.49	2.75	1.16		15.25	0.25	0.25

Figure 34: Dehomogenisation

6. INPUT AND OUTPUT FILES

6.1 Input: constitution of the INPUT data file (TPL)

The data file must have the tpl extension (name of the type 'filename.tpl').
This file corresponds to the following syntax (specified here for information).

- **Itype Isev Isym Iauto Iedit Nx Ny**
  - ***Itype:*** *code related with the type of calculation.*
    =0 for an initial calculation
    =1 for a calculation importing the influence matrix
  - ***Isev:*** *code related with saving the influence matrix*
    =0 do not save
    =1 save the matrix (.temp01)
  - ***Isym**:* *code related with consideration of symmetries*
    =0 no symmetries
    =1 symmetry in respect of x-axis
    =2 symmetry in respect of y-axis
    =3 symmetry in respect of x-axis and y-axis
  - ***Iauto**:* *code related with the iterative calculation*
    =0 for a normal calculation
    =1 for an automatic iterative calculation
  - ***Iedit** :* *code related with result printing*
    =0 for short printing
    =1 for detailed printing
  - ***Nx** :* *= 2× Total nr of elements along x-axis*
  - ***Ny** :* *= 2× Total nr of elements along y-axis*
- **XP YP ZP Theta Sd Sp**
  - ***XP,YP,ZP*** *Coordinates related with the geometry dimensions*
  - ***Theta*** *Plate orientation in the reference coordinates system*
  - ***Sd*** *separation threshold*
  - ***Sp*** *plastification threshold*
- **N_CLUSTERS_MAILLAGE_X** *Number of mesh clusters along x-axis*
- **LX(i) NX(i)**
  - ***LX(i)*** *Length of the 'i' cluster along x-axis*
  - ***NX(i)*** *Number of subdivisions*
- **N_CLUSTERS_MAILLAGE_Y** *Number of mesh clusters along y-axis*
- **LY(j) NY(j)**
  - ***LY(j)*** *Length of the 'j' cluster along y-axis*
  - ***NY(j)*** *Number of subdivisions*
- **N_CLUSTERS_DESACTIVEES** *Number of element clusters to deactivate*
- **I1(k) I2(k) J1(k) J2(k)** *Localization of deactivated clusters*

- *(I1(k), J1(k))* — *Minimum index of the 'k' cluster, (bottom left)*
- *(I2(k), J2(k))* — *Maximum index of the 'k' cluster, (top right)*

- **N_CLUSTERS_MATERIAU** — *Number of clusters for properties of the material*

- **I1(k) I2(k) J1(k) J2(k) E(k) NU(k) H(k) RHO(k)**
  - ***(I1(k), J1(k))*** — *Minimum index of the 'k' cluster, (bottom left)*
  - ***(I2(k), J2(k))*** — *Maximum index of the 'k' cluster, (top right)*
  - ***E(k)*** — *Young's modulus for the 'k' cluster*
  - ***NU(k)*** — *Poisson's ratio*
  - ***H(k)*** — *Thickness*
  - ***RHO(k)*** — *Density*

- **N_CLUSTER_CHARGE_REPARTIE** — *Number of clusters for distributed load*

- **I1(k) I2(k) J1(k) J2(k) PE(k) KS(k)**
  - ***(I1(k), J1(k))*** — *Minimum index of the 'k' cluster, (bottom left)*
  - ***(I2(k), J2(k))*** — *Maximum index of the 'k' cluster, (top right)*
  - ***PE(k)*** — *Load distributed over the cluster*
  - ***KS(k)*** — *Distributed springs over the cluster*

- **N_CLUSTER_CHARGE_NOEUD** — *Number of clusters for node load*

- **I1(k) I2(k) J1(k) J2(k) FZ(k) Mx(k) My(k) Kz(k) Cx(k) Cy(k)**
  - ***(I1(k), J1(k))*** — *Minimum index of the 'k' group, (bottom left)*
  - ***(I2(k), J2(k))*** — *Maximum index of the group, (top right)*
  - ***FZ(k)*** — *Point load applied to each node in the group*
  - ***Mx(k)*** — *Moment around the y-axis*
  - ***My(k)*** — *Moment around the x-axis*
  - ***Kz(k)*** — *Linear spring per node*
  - ***Cx(k)*** — *Rotation spring around the y-axis*
  - ***Cy(k)*** — *Rotation spring around the x-axis*

- **N_COUCHES_SOL** — *Number of layers in the soil*

- **Zs(i) Es(i) NUS(i)**
  - ***Zs(i)*** — *Level of the base of the 'I' layer*
  - ***Es(i)*** — *Young's modulus of the layer*
  - ***NUS(i)*** — *Young's modulus of the layer*

- **N_CHARGES_EXT_SOL** — *Number of external loads on the soil*

- **Xr(i) Yr(i) Zr(i) LXr(i) LYr(i) Theta(i) Qr(i)**
  - ***Xr(i),Yr(i), Zr(i)*** — *Coordinate of the load in the reference coordinates system*
  - ***LXr(i), LYr(i)*** — *Load size: width and length*
  - ***Theta(i)*** — *Orientation in the reference coordinates system*
  - ***qr(i)*** — *Load density, presumed uniform*

- **N_NOEUDS_DECOLLEES** — *Number of clusters to separate*

- **I1(k) I2(k) J1(k) J2(k)** — *Localisation of the cluster to separate*
    - ***(I1(k), J1(k))*** — *Minimum index of the 'k' cluster, (bottom left)*
    - ***(I2(k), J2(k))*** — *Maximum index of the 'k' cluster, (top right)*
- **N_NOEUDS_PLASTIQUES** — *Number of clusters to 'plastify'‘*
- **I1(k) I2(k) J1(k) J2(k)** — *Localization of the cluster to plastify*
    - ***(I1(k), J1(k))*** — *Minimum index of the 'k' cluster, (bottom left)*
    - ***(I2(k), J2(k))*** — *Maximum index of the 'k' cluster, (top right)*

6.2 Output files

There are five output files in total:

- Results file, named 'filename.resu'.
- TASSELDO file, named 'filename.tso'.
- Influence matrix save file, named 'filename.temp01'.
- 'filename.sci' to use the results under Microsoft Excel®.
- 'filename.log' to save the calculation process. It may be useful for debugging.

6.2.1 Result file

The results generated are:

- A reminder of the calculation data
    - Calculation settings
    - Characteristics of the layers
    - External loads on the soil
    - Plate geometry (geometry dimensions + deactivated clusters)
    - Plate material
    - Plate decomposition
    - Calculation points
    - Load distributed on the plate
    - Load on nodes
    - Point elastic supports

- Results
    - Deflection and rotations at the nodes
    - Soil settlement and reaction at the nodes
    - Bending moments and torsion moment, evaluated in four points in each element
    - Shear forces: estimate at one point of each element

6.2.2 TASSELDO file

This file is to be reread by the TASSELDO module (FOXTA v2 software). It is an optional step.

This file includes:

- Definition of soil layers
- Loads applied to the soil
    - External loads on the soil
    - Pressure applied by the plate in each node
- Calculation points (nodes)

6.2.3 Influence matrix temporary backup file

This file contains the influence matrix of the calculation. Importing this file (option in general settings, chapter 3.2.1) allows launching a new calculation without having to recalculate the influence ratios (allowing to gain time).

Matrix importing is valid only when the influence ratios remain unchanged, i.e. if:

- The layer data is unchanged.
- The mesh geometry is unchanged.

6.2.4 File for use under Microsoft Excel®

This file contains all digital results (raw) for the TASPLAQ Output vx.x.xls interactive.

It contains the following, in sequence:

- **NOMBRE_DE_NOEUD_X** *Number of nodes in the x-axis direction (global mesh)*
- **NOMBRE_DE_NOEUD_Y** *Number of nodes in the y-axis direction (global mesh)*

Then for each 'i' calculation:

- **Xe(i) Ye(i) W(i)** *Deflection calculated in 9 points per element*
- **Xm(i) Ym(i) Mx(i) My(i) Mxy(i)** *Moments calculated in 4 points per element*
- **Xt(i) Yt(i) Tx(i) Ty(i)** *Shear forces calculated in 1 point per element*
- **Xn(i) Yn(i) Tass(i) Ps(i)** *Settlement and reaction at nodes*