A finite element code for geotechnical simulations

Tutorials
-
Series B: Strip footing

©Jan Machaček, Patrick Staubach
WWW.NUMGEO.DE
November 26, 2021

History:

2021	Jan Machaček, Patrick Staubach	Initial version

`numgeo`: Keyword Reference Manual
©Jan Machaček, Patrick Staubach (Editors)
www.numgeo.de
November 26, 2021

Contents

1 Rigid strip foundation with consolidation and high-cyclic loading **2**
1.1 Introduction . . . 2
1.2 Model generation with Salome . . . 2
1.3 Dry simulation . . . 8
1.3.1 Definitions in the input file . . . 8
1.3.2 Results of the simulation . . . 11
1.4 Fully-coupled simulation including the consolidation process . . . 12
1.4.1 Definitions in the input file . . . 12
1.4.2 Results of the simulation . . . 16
1.5 Fully-coupled simulation using the HCA model . . . 17

References **21**

1 Rigid strip foundation with consolidation and high-cyclic loading

1.1 Introduction

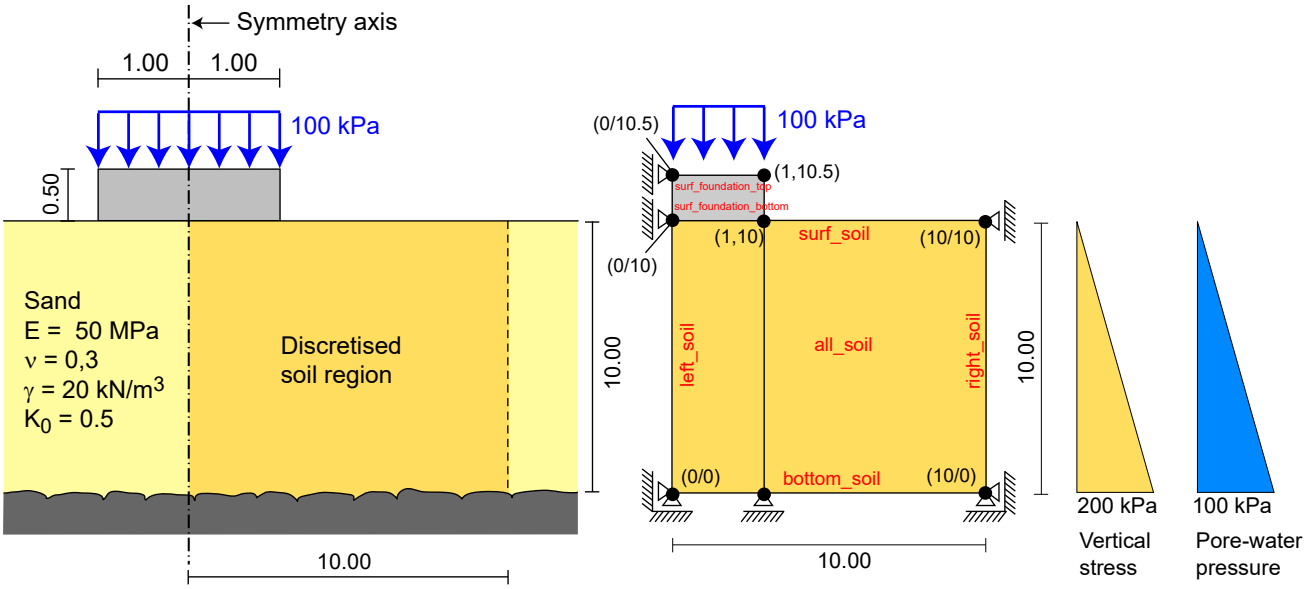

Figure 1: Model and specifications of the rigid foundation

In this example a rigid foundation on an elastic soil layer will be studied. The model specifications are shown in Figure 1. The foundation is loaded by a distributed load along the top surface of the foundation and the settlement is studied. The contact will be enforced using the penalty method in conjunction with the mortar approach for the discretisation of the contact area. In addition, simulations considering the pore water pressure and the consolidation will be performed using coupled elements. The excess pore water pressure due to the loading and the settlement of the foundation caused by the consolidation process will be studied. Different hydraulic conductivities will be applied.

1.2 Model generation with Salome

To create the model, open Salome and proceed as explained below.

- Module `Geometry`
  - Change to the `Geometry` module (see Figure 2, (step 1)) in the upper drop-down menu and choose "2D Sketch Construction" (step 2) to create the geometry of the soil area. In the opening window change the "Element Type" to the rectangle (step 3). The coordinates of the left lower and the right upper corner can be left unchanged. Click on "Apply and Close" to finish the sketch
  - Build a face from the sketch by picking "Build face" (see Figure 3). Leave all options unchanged and mark the line of the previously created rectangle. The face should now be painted magenta and *Sketch_1* should be present in the "Objects" argument. Click on "Apply and Close" to finish the face creation
  - Repeat the same steps to create the foundation
    - Choose "2D Sketch Construction"
    - Change the "Element Type" to the rectangle
    - Change the coordinates to $X1 = 0$, $Y1 = 10$ and $X2 = 1$, $Y2 = 10.5$
    - Finish the sketch
    - Build a face from the sketch
  - Create a partition of the face of the soil (see Figure 4)
    - Choose "Create a point"
    - In the opening window changes the coordinates to $X = 1$, $Y = -1$

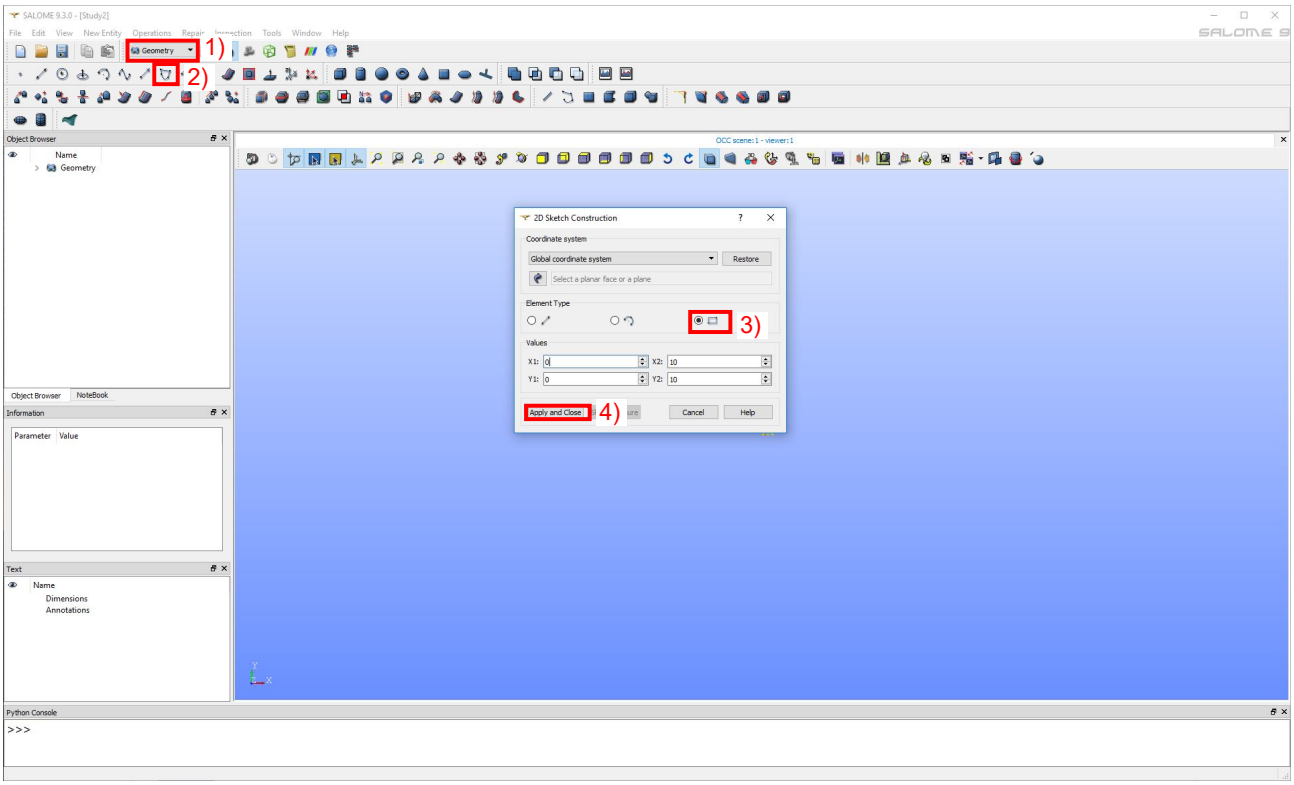

Figure 2: Sketch creation in the `Geometry` module

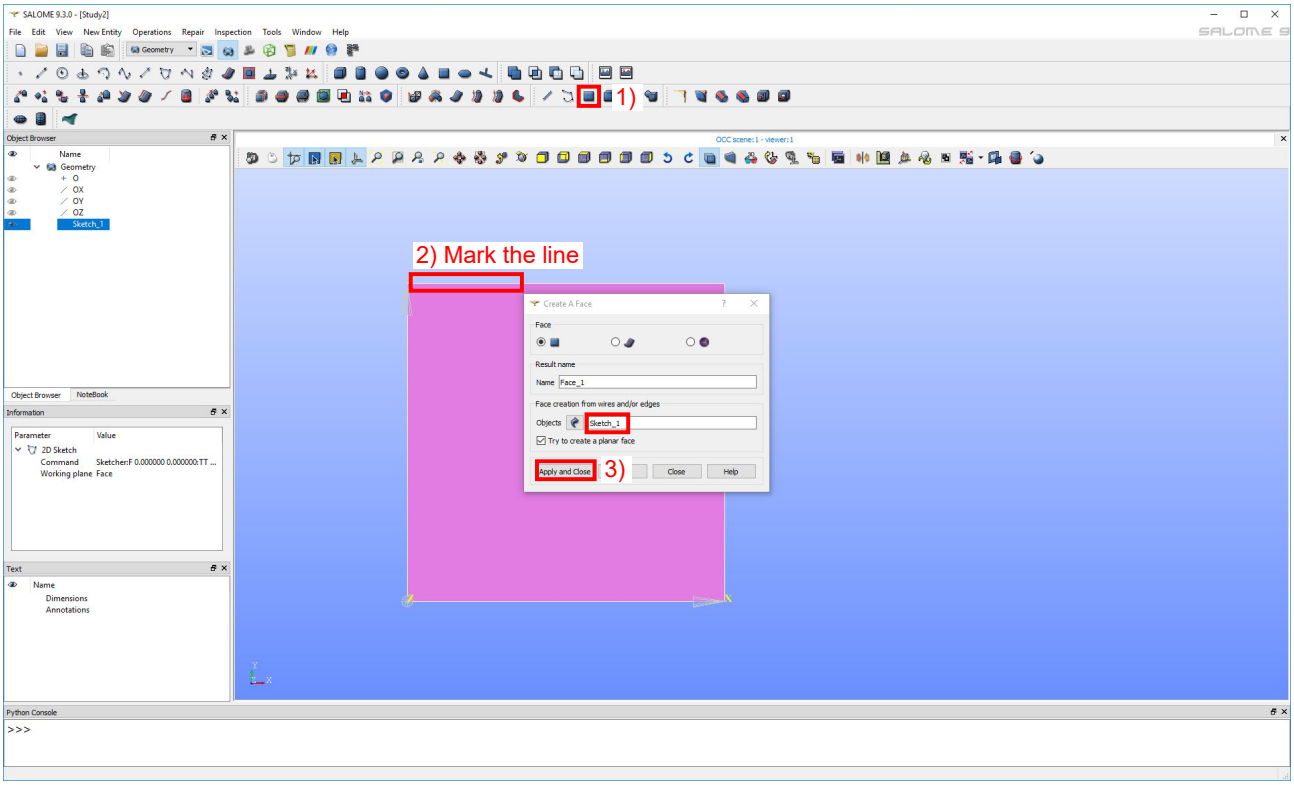

Figure 3: Creation of the face of the soil geometry

* Create the point using "Apply and Close"
* Create a second point with the coordinates $X = 1, Y = 11$
* Choose "Create a line"

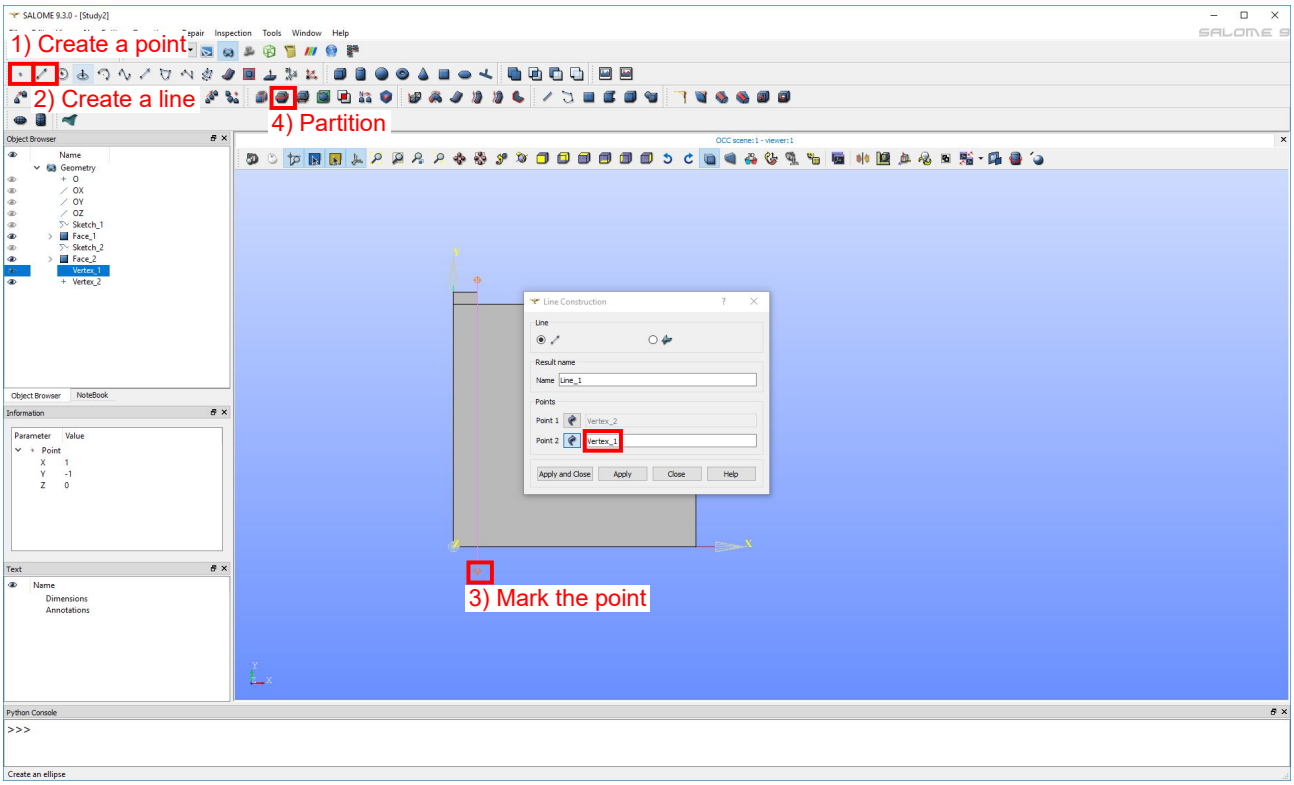

Figure 4: Creation of the line used for the partition of the soil

- Pick the lower point and add it to "Point 2" in the opening window
- Create the line by clicking "Apply and Close"
- Open the window "Partition" (see step 4 of Figure 4)
- In the opening window click on the defined line to choose it as "Tool Object" (see Figure 5)
- Finish the partition by clicking "Apply and Close"

– Create a compound object combining the soil and the foundation face using "Build compound" (three symbols right to "Build Face"). Mark *Face_2* and *Partition_1* (using ctrl) and create the compound

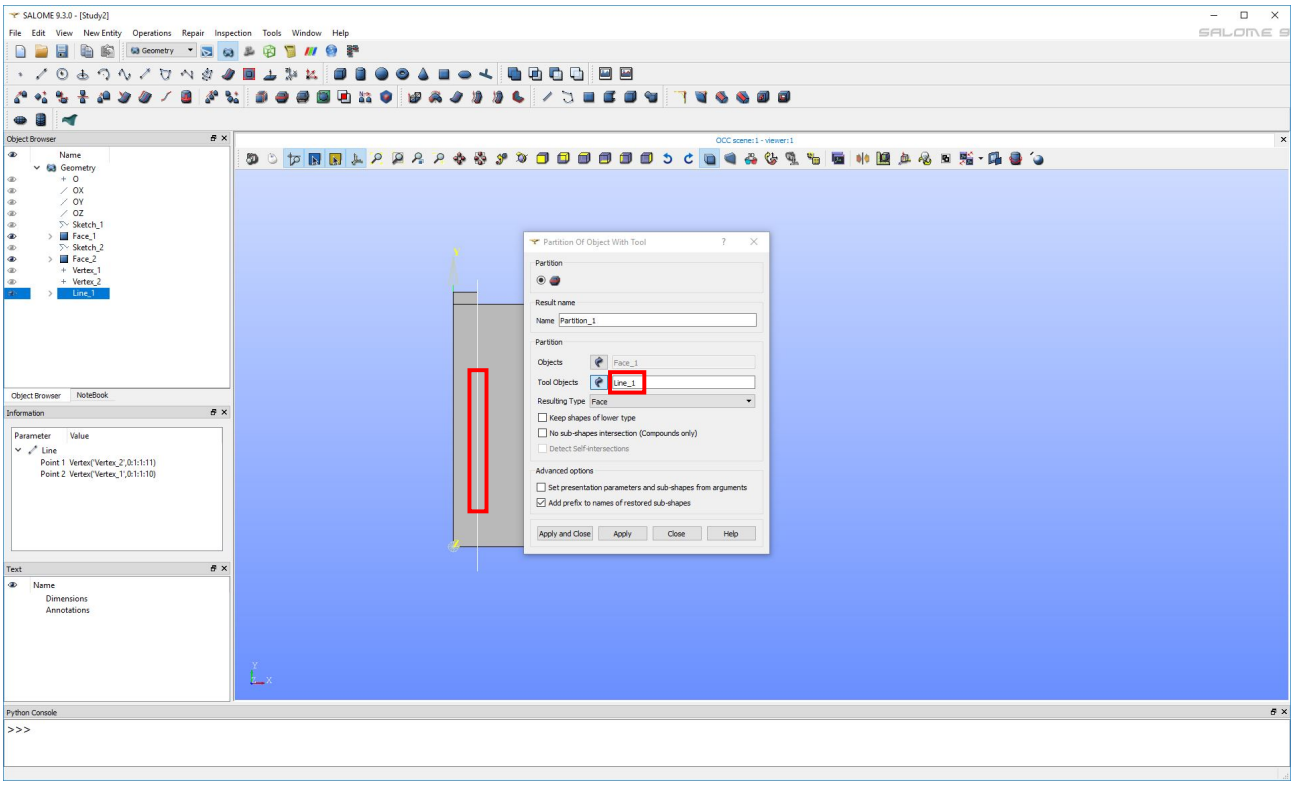

Figure 5: Creation of the partition of the soil

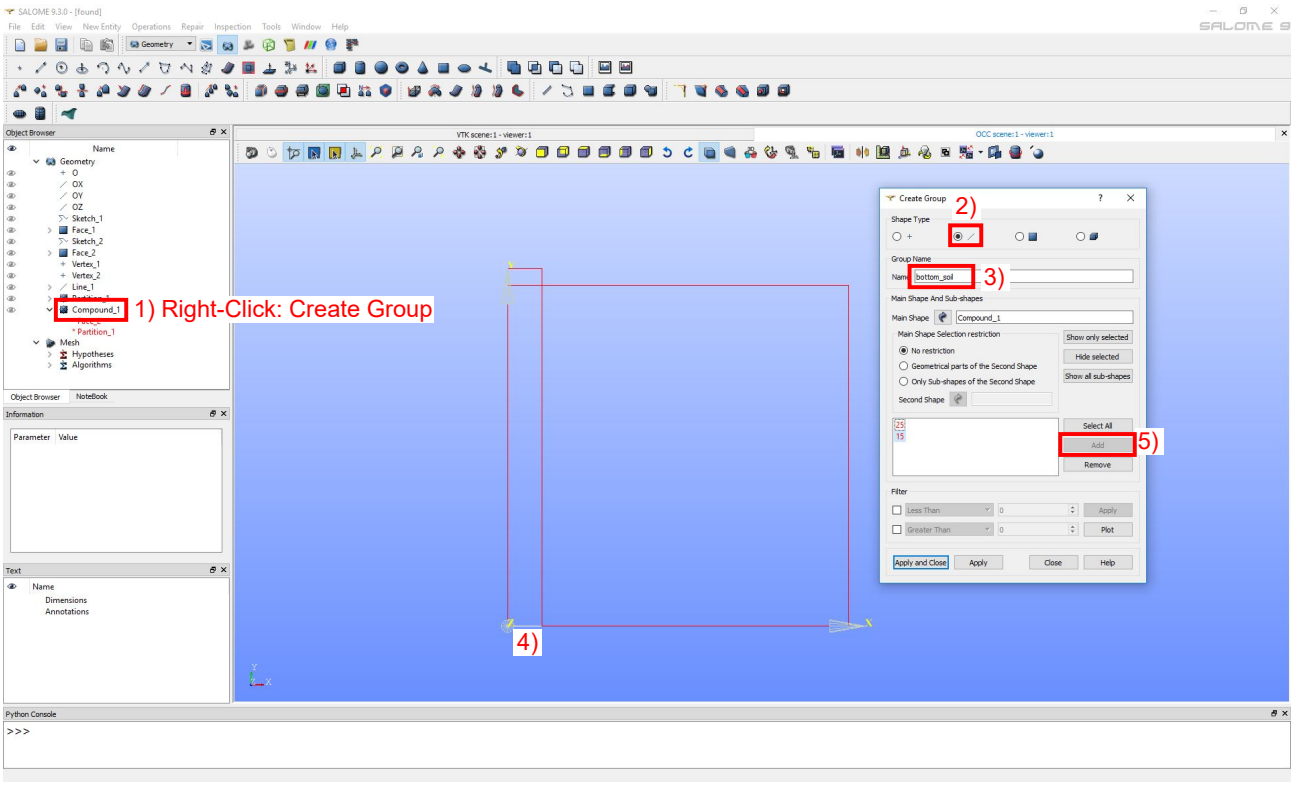

Figure 6: Creation of the line groups (for the face groups change in step 2 to the rectangular blue face)

– Define different groups of geometric entities by right-clicking on *Compound_1* and choosing "Create Group". In the opening window change the "Shape Type" to edge (inclined line) and define the following groups (see Figure 6)

* The line group *bottom_soil*, containing the entire bottom edge of the soil (two individual edges)
* The line group *left_soil*, containing the entire left edge of the soil (one individual edge)
* The line group *left_foundation*, containing the entire left edge of the foundation (one individual edge)
* The line group *right_soil*, containing the entire right edge of the soil (one individual edge)
* The line group *surf_soil*, containing the entire top edge of the soil (two individual edges). Under "Main Shape Selection restriction" switch to "Geometrical parts of the second Shape" and add *Partition_1* in the "Object Browser". Now only the soil should be displayed, such that no irritation with the edge of the foundation exists
* The line group *surf_foundation_bottom*, containing the lower edge of the foundation (one individual edge). Under "Main Shape Selection restriction" switch to "Geometrical parts of the second Shape" and add *Face_2* in the "Object Browser"
* The line group *surf_foundation_top*, containing the top edge of the foundation (one individual edge)
* The line group *surf_soil_found*, containing the top left edge of the soil below the foundation (one individual edge)
* The face group *all_soil* (change the "Face Type" to the 2D face), containing the face of the soil (two individual faces)
* The face group *all_foundation* (change the "Face Type" to the 2D face), containing the face of the foundation (one individual face)

- Module `Mesh`
  - Change to the `Mesh` module (see Figure 7, (step 1)) in the upper drop-down menu and change the view orientation (step 2). Click on "Create Mesh" (step 3) and mark *Compound_1*. Choose "Quadrangle: Mapping" as "Algorithm". Change to 1D and choose:
    * "Algorithm": "Wire Discretisation"
    * In "Hypothesis" click on the gearwheel, select "Local Length" and define 0.2 as "Length"
    * "Apply and Close"

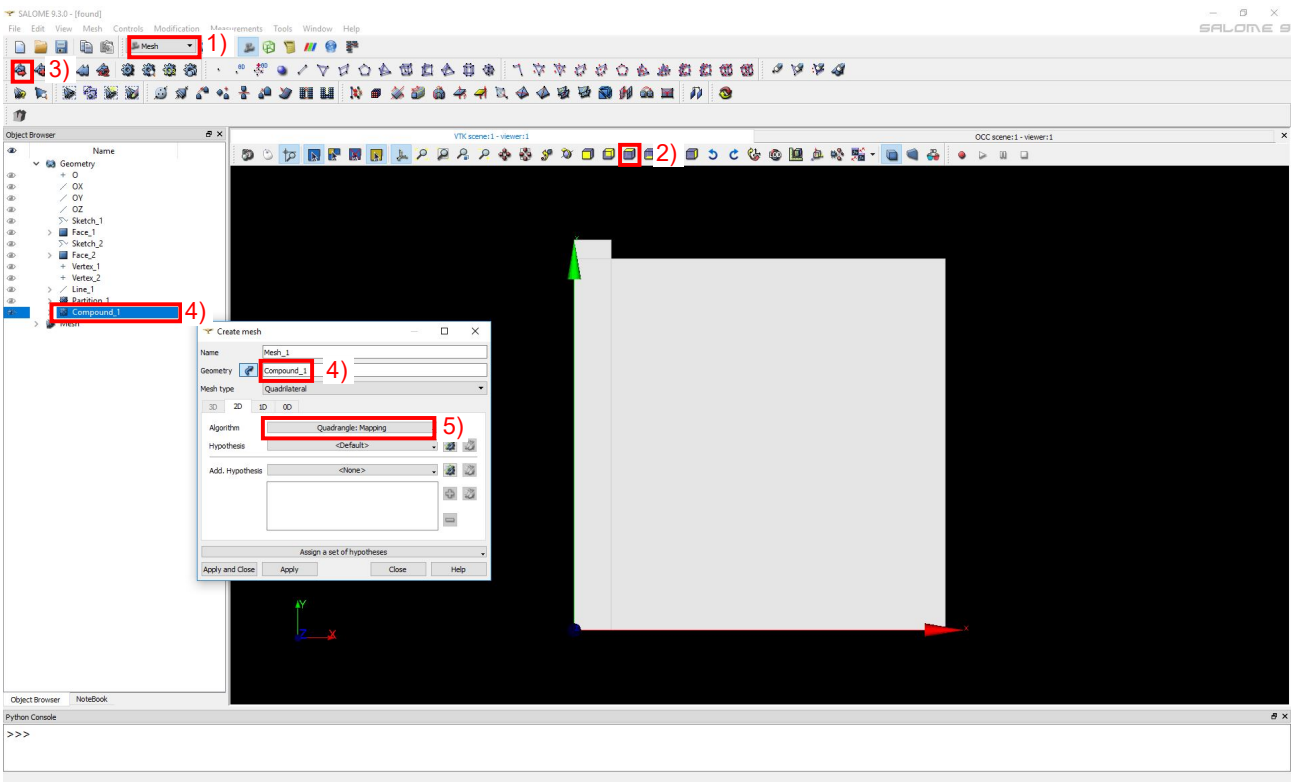

Figure 7: Generation of the mesh

- Right-click on *Mesh_1*, "Compute"
- Import the previous defined groups by right-clicking on *Mesh_1* and choose "Create Groups from Geometry" (see Figure 8). Mark the groups shown in Figure 8 (using ctrl) in the "Object Browser" from *Compound_1* and add them to the elements as well as to the nodes

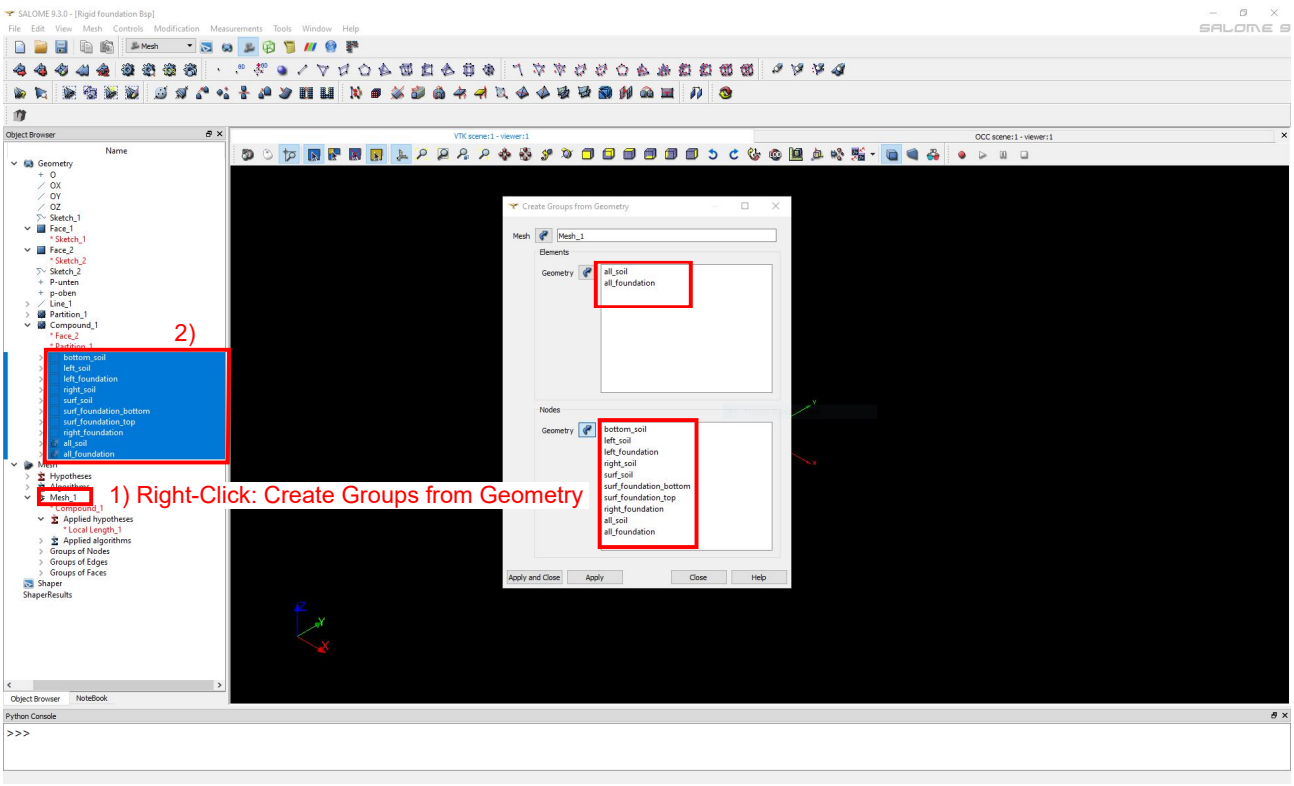

Figure 8: Importing the groups into the mesh

- Right-click on *Mesh_1* → "Convert to/from quadratic", pick "Convert to quadratic"
- Export the model by selecting *Mesh_1* in the "Object Browser" and go to File → Export → UNV file and save the file to the desired destination

- To translate the .UNV file to an input file in the format of `numgeo`, the `numgeo-python-API` is used
  - Copy the files "numgeoUtils.py", "pynumgeo.py" and "example.py" to the same directory as the .UNV file
  - Change the filename in "example.py" to the name of the .UNV file and rename part and materials as wished:

```
import os
import numpy as np
from pynumgeo import numgeo

# Specify some filenames
inpName = './Mesh_1'
fileFormat = 'unv'
outName = './Mesh_1'

# Initialise the numgeo object
numgeo = numgeo()

# We now create our first part

# The finite element mesh as well as the group definitions is read from a *.unv file
# created with Salome. Unfortunately this files contains a mixture of line elements (1
    D),
# shell elements (2D) and volume elements (3D). By choosing the "Dimension" we define
    which
```

```
# of these element types we want to extract
Dimension = '2D'

# For better readability we now name our part
PartName = 'Soil-Foundation'

# Choose the element type used for this part. You only have to choose the family of
    element
# formulations. The final element type in "numgeo-language" is automatically detected
    based
# on the element shape (e.g. 20 node brick element) and the ElementType (e.g. u-solid)
# For the present example this will result in an element type "u20-solid"
ElementType = 'u-solid'

# Now create the part from the .unv-File
numgeo.createPart(inpName,fileFormat,Dimension,PartName,ElementType)

# Just look up what element sets are defined
numgeo.Parts[0].printSets()


numgeo.Parts[0].addSolidSection(ElementSet='Soil-Foundation.all_soil', MaterialName='
    soil')
numgeo.Parts[0].addSolidSection(ElementSet='Soil-Foundation.all_foundation',
    MaterialName='foundation')

# Write an input file readable by numgeo containing the geometry (parts, nodes, mesh,
    sets, ...)
# as well as material and step definitions
numgeo.writeInput(outName)
```

Listing 1: "example.py" used to generate an input file from the .unv file

  - Run the script using the IDE Sypder, a python console or jupyter notebook
  - A file with the same name as the .UNV file but with the .inp ending should have been created

- In some cases, SALOME meshes with a clockwise arrangement of nodes for the individual elements. This will result in a negative element area and therefore an error. `numgeo` will in this case write a message in the console notifying the user that the jacobian determinate of the element is negative. This error can be resolved using the option "Orientation" in the `Mesh` module of SALOME. Choose this option, add the entire mesh (Apply to all) and apply the change of orientation

1.3 Dry simulation

1.3.1 Definitions in the input file

Open the previous created .inp file using a text editor, e.g. Notepad or Geany. Then scroll down to the end of the file and define the following additional node set before `*End Instance`:

```
*nset, find, nset=found_top_left_node
0,10.5,0
**
*End Instance
**
*End Assembly
**
```

This node will later be used for the output of the vertical settlement of the foundation. After the node set definition, add the following surface definitions:

```
**----------------Surfaces----------------**
*Surface, type=node, name=surf_soil_top
Soil-Foundation.surf_soil_found
*Surface, type=node, name=surf_foundation_bottom
Soil-Foundation.surf_foundation_bottom
*Surface, type=node, name=surf_foundation_top
Soil-Foundation.surf_foundation_top
```

Each surface definition contains an underlying node set defined in the instance $Soil - Foundation$. Two solid sections have already been generated automatically by the API:

```
**------------------Solid Section------------------**
*Solid Section, Elset = Soil-Foundation.all_soil, Material = soil
**
*Solid Section, Elset = Soil-Foundation.all_foundation, Material = foundation
```

Since two different materials are used, two separate sections have to be generated. The materials are defined in the next step:

```
**------------------Materials------------------**
*material, name = soil, phases = 1
*Mechanical = Linear_Elasticity
50000, 0.3
*Density
2.0
*material, name = foundation, phases = 1
*Mechanical = Linear_Elasticity
50000000, 0.3
*Density
0.00001
```

For the dry case, both materials are composed by just one phase. Both materials are assumed to behave elastically, using a Young's modulus of 50,000 kPa and a Poisson's ratio of 0.3 for the soil and 50,000,000 kPa and 0.3 for the foundation. The soil has a density of 2 g/cm^3 and the foundation has a negligible density since its self weight will be applied by a distributed load rather than gravity.

The contact conditions are defined in the following:

```
**------------------Contact------------------**
*INTERACTION,name=penalty, MECHANICAL=Penalty, no separation
*Contact Pair, interaction=penalty, discretisation=ELEMENTMORTAR
surf_soil_top, surf_foundation_bottom
```

A penalty contact enforcement is chosen (`MECHANICAL=Penalty`) and after an initial active contact no separation is possible (`no separation`). This means that contact tension is also possible. It is chosen here because it makes the calculation more stable.

The penalty factor is automatically chosen by the code. It is used to "punish" a deviation from the non-penetration condition between the contacting surfaces. If wished, the penalty factor can also be chosen by the user by setting it in the line following the `*INTERACTION` command. The higher the penalty factor, the more rigorously the contact condition will be enforced. A too little value will lead to a great overlapping between the contacting surfaces. A too great value leads to numerical instabilities. A good first approximation of the penalty factor is ten times the stiffness of the less stiff material. However, one should still check that the penetration is not too great.

In line 24, the previously defined interaction is used in a `Contact Pair` definition. A contact pair is a set of surfaces constrained by the defined interaction and discretised by the chosen `discretisation`. In the present case, an element-based mortar (`ELEMENTMORTAR`) discretisation is used. The contact discretisation is used to evaluate the contact distance, compute the contact quantities and integrate the contact contributions to the overall system of equations. In line 25, the surfaces of the contact pair are defined. The first surface is denoted as slave surface while the latter one is the master surface. Since the mortar approach performs the segmentation over the slave surface, the finer meshed surface should always be defined as slave surface.

The initial stress state is defined as K_0 stress state using:

```
**------------------Initial Conditions------------------**
*initial conditions, type=stress, geostatic
Soil-Foundation.all_soil, 10, 0, 0, -200, 0.5, 0.5
```

Therein the element set $Soil - Foundation.all_soil$ is assigned an initial stress state with $\sigma_{22}(y = 10.0\text{ m}) = 0.0$ kPa and $\sigma_{22}(y = 0.0\text{ m}) = -200$ kPa ($\gamma = 20$ kN/m^3). K_0 is 0.5 in x as well as in z direction. The initial stress of the foundation is not necessarily to be defined and is zero by default. After the initial conditions of the stress state, the initial void ratio is set:

```
*initial conditions, type=void ratio, default
Soil-Foundation.all_soil, 1.0d0
```

A constant void ratio of 1 is assumed for the soil.

For the analysis, different time distributions of boundary and loading conditions have to be defined. This is done using the following amplitude definitions:

```
31 **------------------Amplitudes------------------**
32 *AMPLITUDE, NAME = LoadingRamp , TYPE = RAMP
33 0.0, 0.0, 1.0, 1.0
```

The amplitude *LoadingRamp* defines a linear increase of a quantity beginning with the relative value 0 at the time $t_1 = 0$ and the relative value 1 at the time $t_2 = 1$.

The first step of a geotechnical analysis is usually the so called `Geostatic` step wherein the initial stress state given in line 31 is checked against the stress state resulting out of gravity. An initial stress state that is not in accordance with the gravitational stress state eventually leads to displacements. In general, the smaller the displacement after the Geostatic step, the better the stress state accords the stress due to gravity. Too large values of displacement indicate a falsification of the initial state that is crucial in case of path dependent constitutive models such as the hypoplasticity.

The definition of the Geostatic step is given as follows:

```
0  **------------------Steps------------------**
1  *Step, name=step1, inc = 1
2  *Geostatic
3  1,1.0,1,1
4  **
5  *SOLVER,MUMPS
6  *BODY FORCE, INSTANT
7  Soil-Foundation.all_soil, GRAV, 10, 0, -1, 0
8  *BODY FORCE, INSTANT
9  Soil-Foundation.all_foundation, GRAV, 10, 0, -1, 0
10 **
11 *BOUNDARY
12 Soil-Foundation.all_foundation,u1, 0.0d0
13 Soil-Foundation.all_foundation,u2,0.0d0
14 Soil-Foundation.left_soil,u1,0.0d0
15 Soil-Foundation.right_soil,u1,0.0d0
16 Soil-Foundation.bottom_soil,u2,0.0d0
17 **
18 *output, field, vtk, ASCII
19 *node output, nset = Soil-Foundation.all_soil
20 U
21 *element output, elset = Soil-Foundation.all_soil
22 S,Contact
23 **
24 *END STEP
```

Listing 2: Definition of the Geostatic step

Line 1 starts the step environment and defines the name of the step. The analysis type of the step is given in line 2, which will be a Geostatic step. In line 6, the solver for the system of equations is specified, which will be the MUMPS solver in this calculation. The keyword `Body force` imposes a gravitational force which is applied instantaneous (`Instant`). The element sets $Soil - Foundation.all_soil$ and $Soil - Foundation.all_foundation$ are loaded by the gravity (amplitude 10 m/s^2, directed downwards with the normalized vector of the gravity $\vec{\mathbf{b}} = \{0, -1, 0\}$).

The boundary conditions are specified from line 11 to line 16. All nodes of the foundation are constraint in 1- as well as in 2-direction in the Geostatic step. Therefore, no displacement of the foundation is possible. The soil area is only constrained at the bottom in vertical direction and at both lateral sides in horizontal direction.

The output demand is specified from line 18 to line 22. The output is written in the vtk format (suitable for `ParaView`) and is of ASCII type. The node output includes the displacement (`U`) of the nodes and the element output includes the stress (`S`) as well as the contact output variables (`Contact`). The contact output contains of contact stresses as well as of contact distances for each contact node.

In the second step, the foundation is released and loaded. This is defined as follows:

```
0  **------------------Steps------------------**
1  *Step, name=step2, inc = 10000
2  *STATIC
```

```
3  0.01,1.0,0.0001,0.1
4  **
5  *SOLVER,MUMPS
6  *BODY FORCE, INSTANT
7  Soil-Foundation.all_soil, GRAV, 10, 0, -1, 0
8  *BODY FORCE, INSTANT
9  Soil-Foundation.all_foundation, GRAV, 10, 0, -1, 0
10 **
11 *BOUNDARY
12 Soil-Foundation.left_foundation,u1,0.0d0
13 Soil-Foundation.left_soil,u1,0.0d0
14 Soil-Foundation.right_soil,u1,0.0d0
15 Soil-Foundation.bottom_soil,u2,0.0d0
16 **
17 *DSLOAD, INSTANT
18 surf_foundation_top, -5.0d0
19 *DSLOAD, Amplitude=LoadingRamp
20 surf_foundation_top, -100.0d0
21 **
22 *Controls, global, deactivate
23 *Controls, u, activate
24 **
25 *output, field, vtk, ASCII
26 *node output, nset = Soil-Foundation.all_soil
27 U
28 *element output, elset = Soil-Foundation.all_soil
29 S,Contact
30 **
31 *output,print
32 *node output, nset =Soil-Foundation.found_top_left_node
33 U
34 *END STEP
35 *END INPUT
```

Listing 3: Definition of the second step defining the loading of the foundation

A static analysis type is chosen, where an incrementation scheme as given in line 3 is used. The initial increment is 0.01, the total step time is 1, the minimum allowed increment is 0.0001 and the maximum increment is 0.1. Note that time does not correspond to physical time in a static step but rather is the fraction of an applied load or boundary condition. The static analysis type indicates that no inertia forces (and therefore no physical time dependencies) are considered.

In `numgeo`, previous imposed loads have to be redefined in every subsequent step in which they are supposed to be active. Therefore, the gravity is applied again in line 6 to 9. The boundary conditions for the soil remain the same as in the first step but the foundation is now only constrained horizontally in the symmetry axis. The foundation is now free to penetrate into the soil vertically.

The top of the foundation is loaded as defined in line 17 to 20. An instant distributed load of 5 kPa is applied as self weight of the foundation and an additional load of 100 kPa is increased linearly over the step time. At the end of the step, the foundation is loaded by a total of 105 kPa.

The error controls are modified in line 30 and 31. Only the local convergence controls are set active. From line 31 to 33 a print output is defined for the top left node of the foundation. A print output directly writes the requested variable with respect to the calculation time in a separate file.

The calculation is started by calling `numgeo` over the command line, specifying the name of the input file (without ending), confirming by pressing enter, specifying the number of CPUs (depending on the hardware, 2 CPUs are appropriated here) and pressing enter again.

1.3.2 Results of the simulation

After the calculation is finished (the command window is ready for a new command), open the .sta file first and check that the simulation was successful by identifying that both steps have been completed successfully. If the calculation immediately stops, check the error message in the .log file.
To see the output of a step, start `ParaView` and open the group of .vtu files (in the right upper options bar: "File" → "Open...") that is present in the calculation folder and has the number of the step in its name. Alternatively, you may open the .pvd file in the simulation folder to open the entire simulation results (all steps).

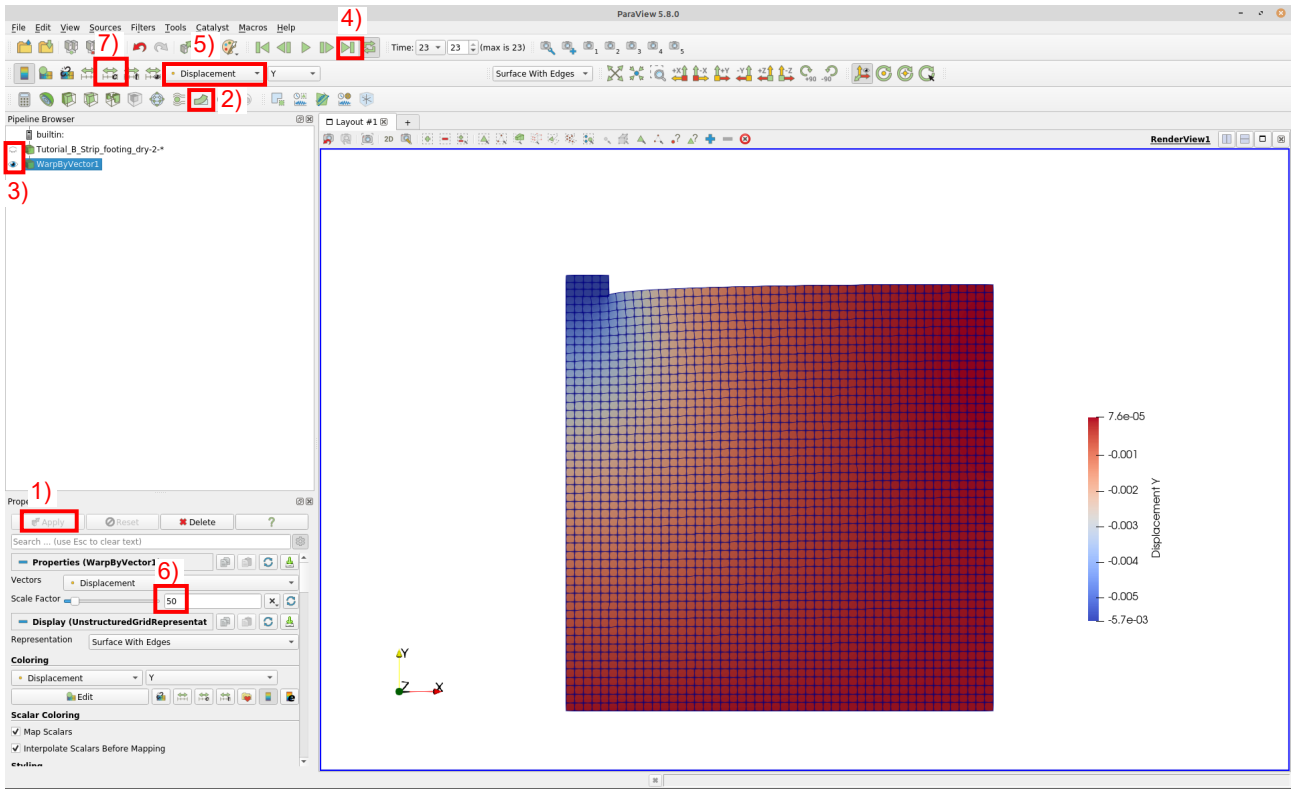

Figure 9: Vertical displacement at the end of the second step

To display the deformed system at the end of the second step, follow the steps shown in Figure 9. Note that in (step 6) a deformation scale factor of 50 is chosen. This means that the displayed state is depicted with 50 times more displacement. The legend tells that the foundation settles 5 mm in total. To evaluate the stress under the foundation, the contact stress is given in Figure 10. Tt can be seen that the right element of the foundation penetrates into the soil. This can be reduced by choosing a higher value for the penalty factor. However, one has to keep in mind that the deformation is displayed 50 times higher here. As indicated by the analytical from Boussinesq, a concentration of stress is identified under the right corner of the foundation.

1.4 Fully-coupled simulation including the consolidation process

To study the effect of the pore water pressure change and the consolidation, a simulation incorporating so called coupled elements is conducted. Coupled elements do not only discretise the displacement of the solid phase but also the pore water pressure (u-p elements), the displacement of the water (u-U elements) or both the pore water pressure and the displacement of the water (u-p-U elements). For the present simulation the u-p elements will be used. More on different geotechnical element formulations can be found in e.g. [1].
The ground water level will be set to the ground surface and drainage alongside the top surface will be assumed. The load of the foundation is again applied within 1 s (physical time now since a transient process is studied). After the application of the load, the consolidation process (the dissipation of the excess pore water pressure build under the foundation due to the loading) is studied in a third step where a time of 10,000 s is used.

1.4.1 Definitions in the input file

Use the input file from the dry calculation, search the file from the top for the keyword `Element`. The first 16 elements belong to the foundation. All remaining elements are soil elements. Therefore, after the last element of the foundation, add a new element type for all soil elements.

```
0 .
1 .
2 .
3   281,      273,       17,        3,       16,     2909,     2634,     2633,     2913
4 *Element, Type = u8p4-sat
5   282,        5,       75,      274,       23,     2696,     2915,     2914,     2642
```

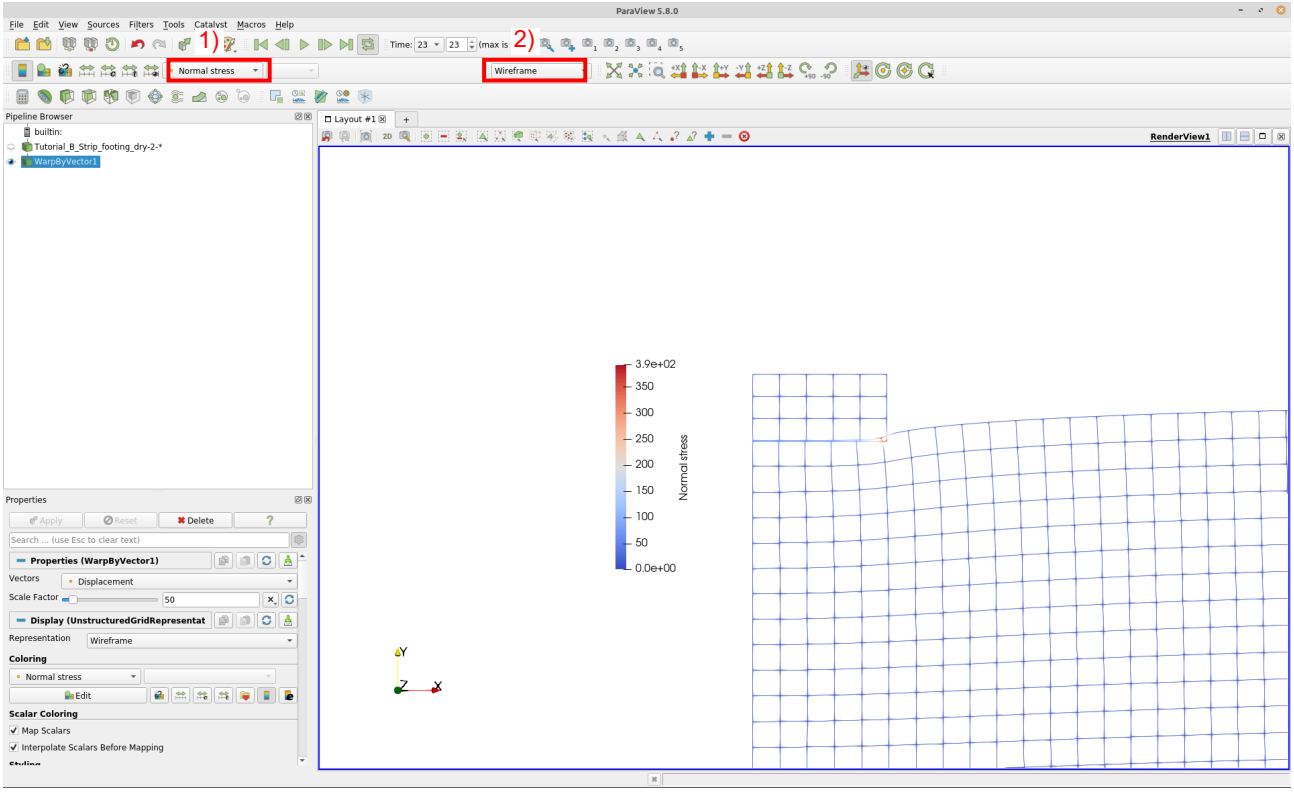

Figure 10: Normal stress at the end of the second step (deformation scale factor of 50)

```
.
.
.
```

`U8P4-sat` elements (the element has 8 nodes with solid displacement degree of freedoms (dofs), 4 pore water pressure dofs and assumes a fully saturated state (`-sat`)) are used. In addition to the consideration of pore water, the Sanisand model is used now. For the Sanisand model the parameters calibrated for "Karlsruhe fine sand", as reported in [2], are adopted. They have been derived from an extensive laboratory testing program. The parameters are provided in Table 1.

p_a	e_0	λ_c	ξ	M_c	c	m	G_0
100 kPa	1.103	0.122	0.205	1.34	0.7	0.05	150.0
ν	h_0	c_h	n_b	A_0	n_d	$z_{\max}$	c_z
0.05	10.5	0.75	1.2	0.9	2.0	20.0	10000

Table 1: Parameters of the Sanisand model for "Karlsruhe Fine Sand"

The material definition has to be changed in order to define a two-phase material (solid grains plus water) now:

```
**------------------------------------ Materials ------------------------------------
*material, name = soil, phases = 2
*Mechanical = SANISAND
** Sanisand parameters for Karlsruhe fine sand
**p_a, e0, lambda, xi, M_c, M_e, mm, G0
**nu, h0, ch, n_b, A0, n_d, z_max, c_z
100, 1.103, 0.122, 0.205, 1.34, 0.938, 0.05, 150
0.05, 10.5, 0.75, 1.2, 0.9, 2.0, 20, 10000
*Density
2.65,1.0
**
*Bulk modulus
2.2d6
**
```

```
14 *Permeability = isotropic
15 1.0d-12
16 **
17 *dynamic viscosity
18 1d-6
19 **
```

Here, two densities have to be given which are the density of the solid grains and the density of water. In addition, the bulk modulus of the pore water and the permeability have to be defined. In `numgeo`, the permeability equals the dynamic viscosity (set to 10^{-6}) multiplied with the hydraulic conductivity k^w and divided by the unit weight of water ($\gamma_w = 10$ kN/m^3). A moderate hydraulic conductivity of $k^w = 1 \cdot 10^{-5}$ m/s is defined here. The resulting permeability of 10^{-12} is given in line 25 and 26.
Since effective stresses are used now, the initial stress condition of the soil has to be changed:

```
20 **----------Initial Conditions----------**
21 *initial conditions, type=stress, geostatic
22 Soil-Foundation.all_soil, 10, 0, 0, -82.5, 0.5, 0.5
```

Therein the element set *all_soil* is assigned an initial effective stress state with $\sigma_{22}(y = 0.0 \text{ m}) = -82.5$ kPa ($\gamma' = (1-n) \cdot (\gamma_s - \gamma_w) = \left(1 - \frac{e}{1+e}\right) \cdot (\gamma_s - \gamma_w) = 8.25$ kN/m^3) and $\sigma_{22}(y = 10.0 \text{ m}) = 0.0$ kPa.
In addition, the initial pore water pressure has to be set:

```
23 **----------Initial Conditions----------**
24 *initial conditions, type=pore water pressure, default
25 all_soil, 0.0d0, 100.0d0,10.0d0, 0.0d0
```

The soil has 100 kPa pore water pressure at $y = 0.0$ m and zero pressure at $y = 10.0$ m.
The Sanisand model requires the definition of the initial void ratio, assumed to 1 in the present case:

```
26 **----------Initial Conditions----------**
27 *initial conditions, type=state variables, default
28 Soil-Foundation.all_soil, void_ratio, 1.0d0
```

In the first step, only little changes have to be made compared to the dry case:

```
0  **----------Steps----------**
1  *Step, name=step1, inc = 1
2  *GEOSTATIC
3  **
4  *SOLVER,MUMPS
5  **
6  *BODY FORCE, INSTANT
7  Soil-Foundation.all_soil, GRAV, 10, 0, -1, 0
8  *BODY FORCE, INSTANT
9  Soil-Foundation.all_foundation, GRAV, 10, 0, -1, 0
10 **
11 *BOUNDARY
12 Soil-Foundation.all_foundation,u1, 0.0d0
13 Soil-Foundation.all_foundation,u2,0.0d0
14 Soil-Foundation.left_soil,u1,0.0d0
15 Soil-Foundation.right_soil,u1,0.0d0
16 Soil-Foundation.bottom_soil,u2,0.0d0
17 *BOUNDARY,type=hydrostatic
18 Soil-Foundation.all_soil,pw,10,10
19 **
20 *output, field, vtk, ASCII
21 *node output, nset = Soil-Foundation.all_soil
22 U
23 *element output, elset = Soil-Foundation.all_soil
24 S,Contact
25 **
26 *END STEP
```

Listing 4: Definition of the Geostatic step using coupled elements

The boundary condition for the pore water pressure is added in line 17 to 18. The pore water pressure is set to zero for the nodes of the foundation and a hydrostatic distribution is defined for all soil nodes. The distribution defined here coincides with the defined initial conditions. The first value in line 18 is the unit weight of water γ_w and the second 10 is the height of the ground water level. This defines a linear distribution of pore water pressure. For the output the pore water pressure is added in line 23 using `Pw`.

The second step is defined as follows:

```
0  **----------------Steps----------------**
1  *Step, name=step2, inc = 10000
2  *transient
3  0.01,1.0,0.0001,0.1
4  **
5  *SOLVER,MUMPS
6  **
7  *BODY FORCE, INSTANT
8  Soil-Foundation.all_soil, GRAV, 10, 0, -1, 0
9  *BODY FORCE, INSTANT
10 Soil-Foundation.all_foundation, GRAV, 10, 0, -1, 0
11 **
12 *BOUNDARY
13 Soil-Foundation.left_foundation, u1, 0.0d0
14 Soil-Foundation.left_soil,u1,0.0d0
15 Soil-Foundation.right_soil,u1,0.0d0
16 Soil-Foundation.bottom_soil,u2,0.0d0
17 *BOUNDARY
18 Soil-Foundation.surf_soil,pw,0.0d0
19 **
20 *DSLOAD, INSTANT
21 surf_foundation_top, -5.0d0
22 *DSLOAD, Amplitude=LoadingRamp
23 surf_foundation_top, -100.0d0
24 **
25 *Controls, global, deactivate
26 *Controls, u, activate
27 *Controls, pw, deactivate
28 **
29 *output, field, vtk, ASCII
30 *node output, nset = Soil-Foundation.all_soil
31 U,pw
32 *element output, elset = Soil-Foundation.all_soil
33 S,Contact
34 **
35 *output,print
36 *node output, nset = Soil-Foundation.found_top_left_node
37 U
38 *END STEP
```

Listing 5: Definition of the second step defining the loading of the foundation using coupled elements

The type of analysis in line 2 was changed to a transient calculation, including pore water pressure change and water flow (consolidation). Note that physical time is now used for this step. Therefore, the step is 1 s long as defined in line 3. The boundary conditions for the pore water pressure of the soil is now redefined for the top node set only. This means that this edge is now permeable and open for water flow. The rest of the step remains the same as in the dry case.

To study the consolidation process after the load of the foundation is applied, a third step is created:

```
0  **----------------Steps----------------**
1  *Step, name=step3, inc = 10000
2  *transient
3  0.05,10000.0,0.0001,100
4  **
5  *SOLVER,MUMPS
6  **
7  *BODY FORCE, INSTANT
8  Soil-Foundation.all_soil, GRAV, 10, 0, -1, 0
9  *BODY FORCE, INSTANT
10 Soil-Foundation.all_foundation, GRAV, 10, 0, -1, 0
11 **
12 *BOUNDARY
13 Soil-Foundation.left_foundation, u1, 0.0d0
14 Soil-Foundation.left_soil,u1,0.0d0
15 Soil-Foundation.right_soil,u1,0.0d0
16 Soil-Foundation.bottom_soil,u2,0.0d0
17 *BOUNDARY
18 Soil-Foundation.surf_soil,pw,0.0d0
19 **
20 *DSLOAD, INSTANT
```

```
surf_foundation_top, -5.0d0
*DSLOAD, INSTANT
surf_foundation_top, -100.0d0
**
*Controls, global, deactivate
*Controls, u, activate
*Controls, pw, deactivate
**
*output, field, vtk, ASCII
*node output, nset = Soil-Foundation.all_soil
U,pw
*element output, elset = Soil-Foundation.all_soil
S,Contact
**
*output,print
*node output, nset = Soil-Foundation.found_top_left_node
U
*END STEP
*END INPUT
```

Listing 6: Definition of the third step defining the consolidation step

The total time of this step is 10,000 s and the maximum time increment is 100 s. The load of the foundation is now applied instantly for both loads, as the load was already brought into equilibrium with the model in the second step.

1.4.2 Results of the simulation

The simulation is conducted for two different hydraulic conductivities, $k^w = 1 \cdot 10^{-5}$ m/s and $k^w = 1 \cdot 10^{-3}$ m/s. The results of the field of pore water pressure at the end of step 2 are shown in Figure 11. As expected, the lower hydraulic conductivity leads to more building-up of excess pore water pressure under the foundation than the higher hydraulic conductivity. Since the entire top of the soil is permeable, zero pore water pressure is encountered directly under the foundation. Due to the lower hydraulic conductivity, the consolidation process is slower in case of $k^w = 1 \cdot 10^{-5}$ m/s and less pore water flow is happening during the second step. Figure

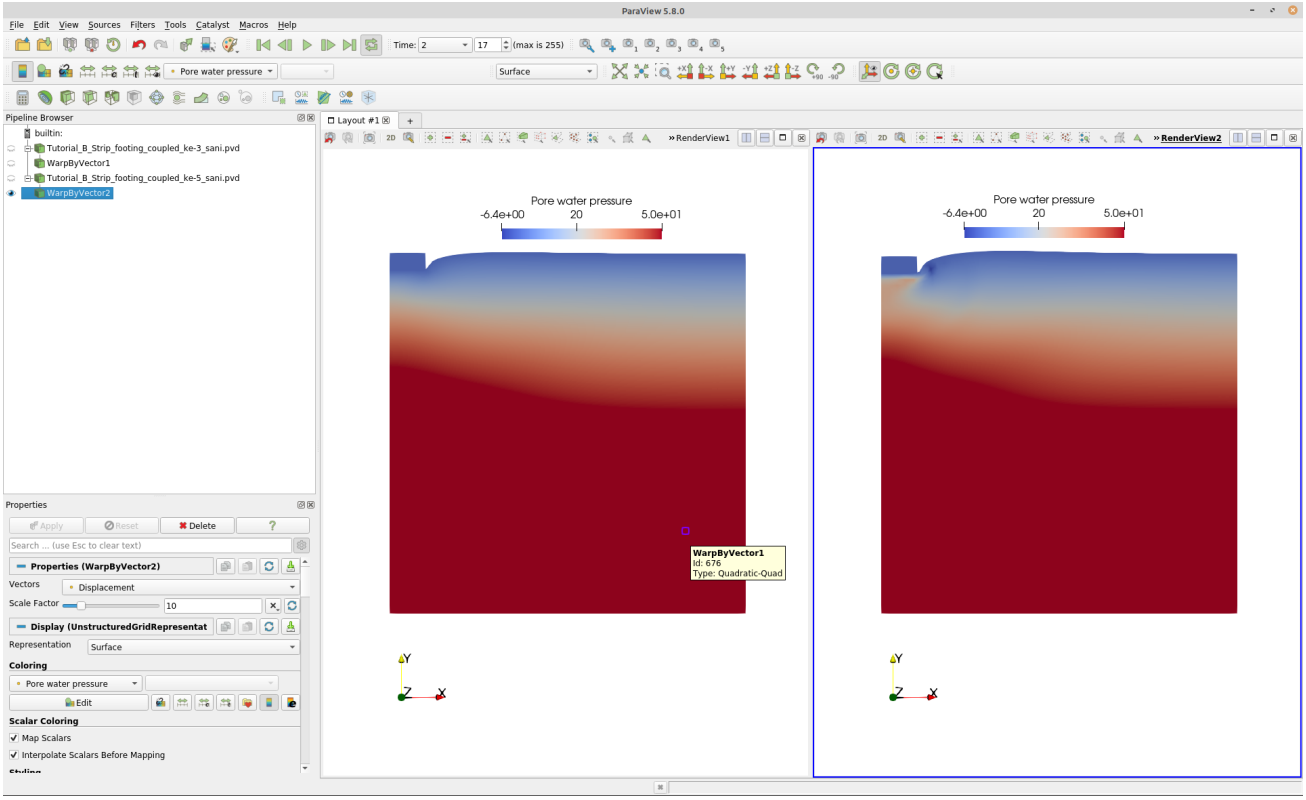

Figure 11: Comparison of the pore water pressure after the application of the complete foundation load for the two different hydraulic conductivities (left: $k^w = 10^{-3}$ m/s; right: $k^w = 10^{-5}$ m/s

12 compares the vertical displacement of the foundation for the two different hydraulic conductivities in the third step. Since larger excess pore water pressure occurs during the loading phase (step 2) in case of the lower hydraulic conductivity, larger settlement of the foundation occurs.

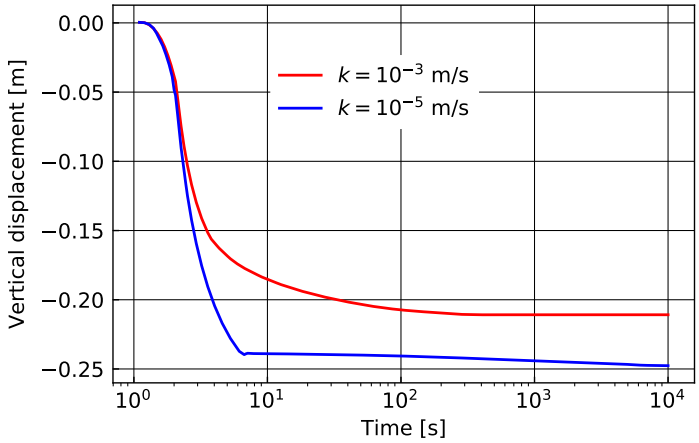

Figure 12: Comparison of the vertical displacement of the foundation in the third step for two different hydraulic conductivities

1.5 Fully-coupled simulation using the HCA model

A cyclic loading with an amplitude of 25 kPa and 10^6 cycles is studied in the following using the high-cycle accumulation model. For the low-cycle phase of the HCA model, the Sanisand model is used. For the HCA model the parameters calibrated for "Karlsruhe fine sand", as reported in [2], are adopted. The parameters are provided in Table 1 and Table 2.

C_{ampl}	C_e	C_p	C_Y	C_{N1}	C_{N2}	C_{N3}
1.33	0.60	0.23	1.68	$2.95{\cdot}10^{-4}$	0.41	$1.9 \cdot 10^{-5}$

Table 2: Parameters of the HCA model for "Karlsruhe Fine Sand"

The material definition has to be changed in order to use the HCA model with Sanisand:

```
**------------------------------------ Materials ------------------------------------
*material, name = soil, phases = 2
*Mechanical = HCA_SANISAND
** Sanisand parameters for Karlsruhe fine sand
**p_a, e0, lambda, xi, M_c, M_e, mm, G0
**nu, h0, ch, n_b, A0, n_d, z_max, c_z
100, 1.103, 0.122, 0.205, 1.34, 0.938, 0.05, 150
0.05, 10.5, 0.75, 1.2, 0.9, 2.0, 20, 10000
** HCA parameters for Karlsruhe fine sand
0.000295,      0.41,   1.9e-05,      1.33,       0.6,        0.,      0.23,        0.
1.68,        0.,         0.,         0.,      100.,     1.054,    0.0001,     3000.,
1.63,       0.5,      0.32,     0.578
**
*Implicit hca steps
step1,step2,step3
*Recording hca steps
step4
*Explicit hca steps
step5
*HCA cycle time
1
*Density
2.0,1.0
**
*Bulk modulus
2.2d6
**
```

```
*Permeability = isotropic
1.0d-10
*dynamic viscosity
1d-6
**
```

In addition to the material parameters, the HCA model requires the declaration of the step types. The first three steps are low-cycle ("implicit") steps. Opposite to the previous calculation, the first sinusoidal load is applied in *step*3. In *step*4 the second cycle is applied, for which the strain path is recorded. The actual HCA phase is performed in *step*5. The period of the cycles is 1 s for why in line 19 to 20 the cycle time is set to 1 s.
To apply the cyclic loading, a sinusoidal amplitude is defined:

```
**------------------------------------ Define Amplitudes ------------------------------------
*AMPLITUDE, NAME = Sinus1Hz , TYPE = PERIODIC
1,0.0,0.0,6.28
0,1
```

The steps 3, 4 and 5 are defined as follows.

```
**------------------------------------ Steps ------------------------------------
*Step, name=step3, inc = 10000
*transient
0.05,1.0,0.0001,0.05
**
*SOLVER,MUMPS-mpi=1
**
*BODY FORCE, INSTANT
Soil-Foundation.all_soil, GRAV, 10, 0, -1, 0
*BODY FORCE, INSTANT
Soil-Foundation.all_foundation, GRAV, 10, 0, -1, 0
**
*BOUNDARY
Soil-Foundation.left_foundation, u1, 0.0d0
Soil-Foundation.left_soil,u1,0.0d0
Soil-Foundation.right_soil,u1,0.0d0
Soil-Foundation.bottom_soil,u2,0.0d0
*BOUNDARY
Soil-Foundation.surf_soil,pw,0.0d0
**
*DSLOAD, INSTANT
surf_foundation_top, -5.0d0
*DSLOAD, INSTANT
surf_foundation_top, -100.0d0
*DSLOAD, amplitude = Sinus1Hz
surf_foundation_top, -25.0d0
**
*Controls, global, deactivate
*Controls, u, activate
*Controls, pw, deactivate
**
*output, field, vtk, ASCII
*node output, nset = Soil-Foundation.all_soil
U,pw
*element output, elset = Soil-Foundation.all_soil
S,Contact
**
*output,print
*node output, nset =Soil-Foundation.found_top_left_node
U
*END STEP
**------------------------------------ Steps ------------------------------------
*Step, name=step4, inc = 10000
*transient
0.05,1.0,0.0001,0.05
**
*SOLVER,MUMPS-mpi=1
**
*BODY FORCE, INSTANT
Soil-Foundation.all_soil, GRAV, 10, 0, -1, 0
*BODY FORCE, INSTANT
Soil-Foundation.all_foundation, GRAV, 10, 0, -1, 0
**
```

```
53 *BOUNDARY
54 Soil-Foundation.left_foundation, u1, 0.0d0
55 Soil-Foundation.left_soil,u1,0.0d0
56 Soil-Foundation.right_soil,u1,0.0d0
57 Soil-Foundation.bottom_soil,u2,0.0d0
58 *BOUNDARY
59 Soil-Foundation.surf_soil,pw,0.0d0
60 **
61 *DSLOAD, INSTANT
62 surf_foundation_top, -5.0d0
63 *DSLOAD, INSTANT
64 surf_foundation_top, -100.0d0
65 *DSLOAD, amplitude = Sinus1Hz
66 surf_foundation_top, -25.0d0
67 **
68 *Controls, global, deactivate
69 *Controls, u, activate
70 *Controls, pw, deactivate
71 **
72 *output, field, vtk, ASCII
73 *node output, nset = Soil-Foundation.all_soil
74 U,pw
75 *element output, elset = Soil-Foundation.all_soil
76 S,Contact
77 **
78 *output,print
79 *node output, nset =Soil-Foundation.found_top_left_node
80 U
81 *END STEP
82 ** ---------------------------------------- Steps ----------------------------------------
83 *Step, name=step5, inc = 10000
84 *transient
85 0.05,1d6,0.0001,1d4
86 **
87 *SOLVER,MUMPS-mpi=1
88 **
89 *BODY FORCE, INSTANT
90 Soil-Foundation.all_soil, GRAV, 10, 0, -1, 0
91 *BODY FORCE, INSTANT
92 Soil-Foundation.all_foundation, GRAV, 10, 0, -1, 0
93 **
94 *BOUNDARY
95 Soil-Foundation.left_foundation, u1, 0.0d0
96 Soil-Foundation.left_soil,u1,0.0d0
97 Soil-Foundation.right_soil,u1,0.0d0
98 Soil-Foundation.bottom_soil,u2,0.0d0
99 *BOUNDARY
100 Soil-Foundation.surf_soil,pw,0.0d0
101 **
102 *DSLOAD, INSTANT
103 surf_foundation_top, -5.0d0
104 *DSLOAD, INSTANT
105 surf_foundation_top, -100.0d0
106 **
107 *Controls, global, deactivate
108 *Controls, u, activate
109 *Controls, pw, deactivate
110 **
111 *output, field, vtk, ASCII
112 *node output, nset = Soil-Foundation.all_soil
113 U,pw
114 *element output, elset = Soil-Foundation.all_soil
115 S,Contact,strain_ampl
116 **
117 *output,print
118 *node output, nset =Soil-Foundation.found_top_left_node
119 U
120 *END STEP
121 *END INPUT
```

In *step*3 the first cycle is applied in lines 24 and 25. *step*4 is a repetition of *step*3. The actual HCA simulation is performed in *step*5. In line 85, a total time of 10^6 s is defined for the step. Since each cycle has a period of 1 s, 10^6 cycles are simulated. Note that no sinusoidal load is active in *step*5, only the average loading is applied.

The strain amplitude is demanded as output in line 115.

The field of the strain amplitude calculated based on the recorded strain path of step 4 is given in Fig. 13. The deformed configuration corresponds to the end of the HCA phase. Figure 14 depicts the vertical displacement of the foundation vs. time (equivalent to number of load cycles).

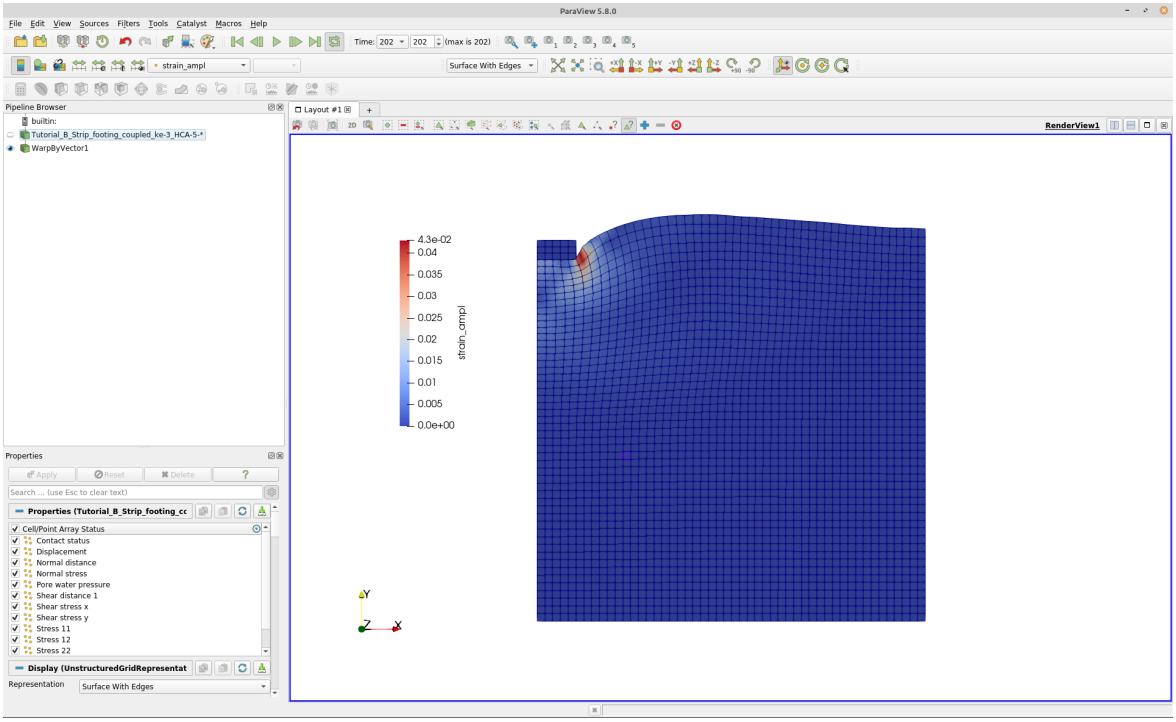

Figure 13: Field of the strain amplitude and deformed configuration (no scale factor) after 10^6 cycles calculated with the HCA model

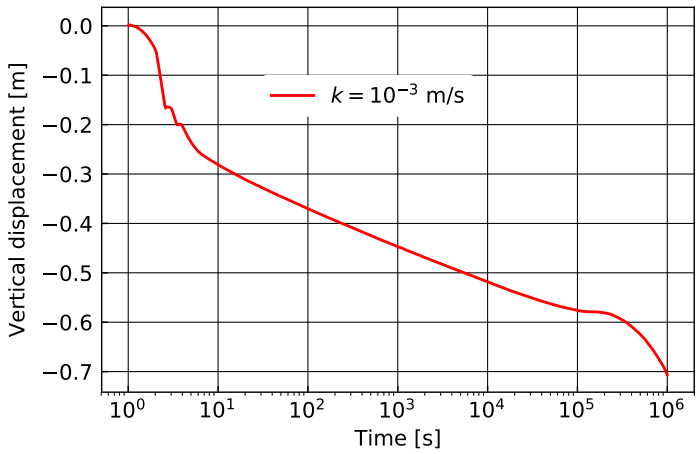

Figure 14: Vertical displacement of the foundation obtained from the simulation with the HCA model and a hydraulic conductivity of $k^w = 10^{-3}$ m/s

References

[1] Staubach, P. and Machaček, J. "Influence of relative acceleration in saturated sand: Analytical approach and simulation of vibratory pile driving tests". In: *Computers and Geotechnics* 112 (Aug. 2019), pp. 173–184. ISSN: 0266-352X. DOI: 10.1016/j.compgeo.2019.03.027. URL: http://www.sciencedirect.com/science/article/pii/S0266352X19301028.

[2] Wichtmann, T. "Soil behaviour under cyclic loading-experimental observations, constitutive description and applications". Veröffentlichung des Instituts für Bodenmechanik und Felsmechanik am Karlsruher Institut für Technologie (KIT), Heft 181, 2016.