A finite element code for geotechnical simulations

Tutorials
-
Series E: Monopile

©Jan Machaček, Patrick Staubach
www.numgeo.de
September 30, 2022

History:

2022	Patrick Staubach, Jan Machaček	Initial version

`numgeo`: Tutorials

www.numgeo.de
September 30, 2022

Contents

1 Monopile under high-cyclic loading

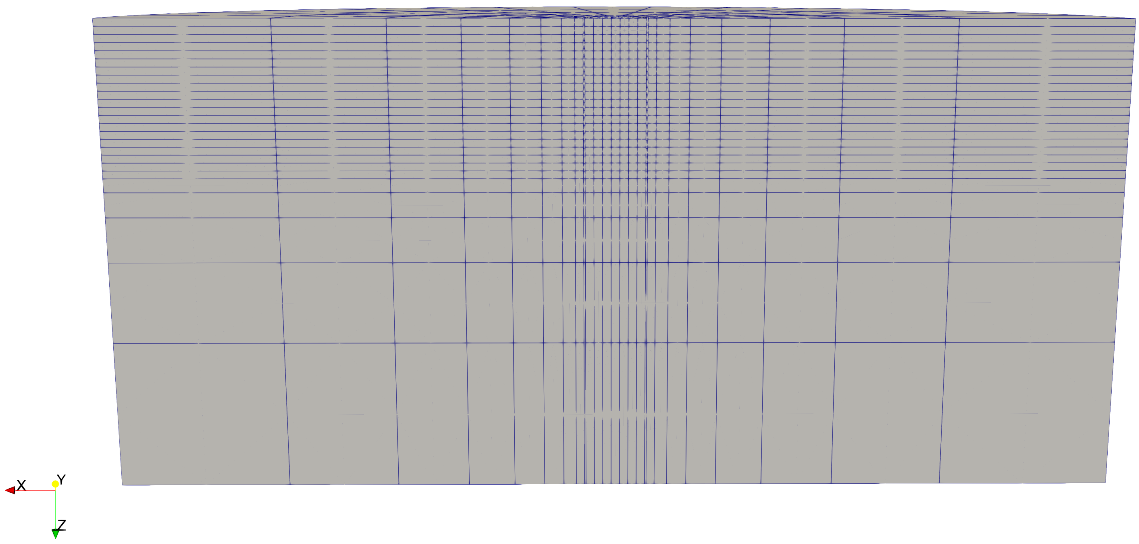

Figure 1: Model of the monopile (only the corner nodes of the elements are visible)

This example explains the simulation of the long-term response of monopile foundations using the HCA model in combination with the hypoplastic model with intergranular strain. The mesh generation is skipped for this example. However, the mesh is contained in the folder shipped together with this document.

The adopted finite element model is given in Fig. 1. The pile has a length of 10 m and a diameter of 4 m. The file containing the mesh (*mesh.inp*) is discussed first and then the step file (*step_tutorial.inp*) is addressed.

The simulation needs quite a bit of RAM. If your simulation is killed, the RAM of your computer is not enough. Note that Windows Subsystem for Linux (WSL) users may have only half of their installed RAM available. It is possible to use nearly 100 % of your installed RAM by setting the memory in the .wslconfig file in your windows home folder.

1.1 Mesh

For the soil, three-dimensional **u**-p finite elements with 27 nodes discretising the displacement **u** of the solid phase and 8 nodes discretising the pore water pressure p^w are utilised (termed `u27p8` element, obtained using the `bi-quadratic` option in the element definition since the pre-processor generated `u20` elements).

```
.
.
*Element, type=u20p8-sat, bi-quadratic
.
.
```

Hence, quadratic interpolation functions are used for the solid displacement while linear interpolation functions are applied for the pore water pressure (i.e. Taylor-Hood element formulation [4]). The `u27p8` element is superior to the `u20p8` element in contact analyses as has been demonstrated in [1].

The pile is modelled with single phase elements with 27 nodes (`u27-solid` obtained again using the `bi-quadratic` option).

```
.
.
*Element, type=u20-solid, bi-quadratic
```

```
.
.
```

1.2 Properties, Initial conditions and step definitions

1.2.1 Properties, contact and Initial conditions

The step file (*step_tutorial.inp*) first includes the mesh file by defining:

```
*include=mesh
```

Then the material definitions are given. The pile is composed of two materials: the regular steel material and a much stiffer material used only for the uppermost row of elements where the load is applied to the pile. This is done to avoid excessive deformations of the pile at the point of load application.

```
*Material, name=Elastisch_Stahl, phases = 1
*Density
30.9416d0
*Mechanical = Linear_Elasticity
2.1e+8, 0.3
**
*Material, name=Stahl_kopf, phases = 1
*Density
30.9416d0
*Mechanical = Linear_Elasticity
2.1e+12, 0.3
```

Note that the density is set higher in order to consider an additional length of the pile not modelled.

For the low-cycle phase of the HCA model, the Hypoplastic model with intergranular strain extension is used (see Listing 1). The parameters calibrated for "Karlsruhe fine sand", as reported in [2], are adopted. The parameters for the HCA model are provided in Table 1.

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 1: Parameters of the HCA model for "Karlsruhe Fine Sand"

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} kPas) multiplied with the hydraulic conductivity k^w and divided by the unit weight of water ($\gamma_w = 10$ kN/m^3). A high hydraulic conductivity of $k^w = 1 \cdot 10^{-3}$ m/s is defined here. The resulting permeability of 10^{-10} m^2 is given in line 29.

To avoid positive mean effective stresses (in mechanical sign convention), a minimum value is set in line 19-20. In addition, an artificial viscosity is used to achieve better convergence. However, it is not necessarily required and also doesn't influence the results.

In addition to the material parameters, the HCA model requires the declaration of the step types. The first four steps are low-cycle ("implicit") steps. The first sinusoidal load is applied in *step*4. In *step*5 the second cycle is applied, for which the strain path is recorded. The actual HCA phase is performed in *step*6. The period of the cycles is 1 s for why in line 40 to 41 the cycle time is set to 1 s.

```
0  **———————————————————————————————————
1  *Material, name = hypo, phases = 2
2  *Mechanical = HCA_Hypoplasticity
3  **phi, nu, hS, n, ed0, ec0, ei0, alpha
4  **beta, mT, mR, R, betaR, chi, Kw
5  0.577, 0.0, 4d6, 0.27, 0.677, 1.054, 1.15, 0.14
6  2.5, 1.2, 2.4, 1d-4, 0.1, 6.0 , 0.
7  *******
8  **CN1, CN2, CN3, Campl, Ce, 0, Cp2, 0
9  **Cy2, Cpi1, Cpi2, Cpi3, patm, eref, eps^ampl_ref, A
10 **a, n, nu, phic
11 0.000295, 0.41, 0.000019, 1.33, 0.6, 0, 0.23, 0
12 1.68, 0, 0, 0, 100, 1.054, 0.0001, 1209
13 1.63, 0.5, 0.32, 0.577
14
```

```
*Density
2.7d0,  1.0


*minpressure
-0.05

*mechanical viscosity = linear
0.5,2.0,1.1,1.5,1.1,1.5

*Bulk modulus
2.2d5

*Permeability=ISOTROPIC
1.0e-10
** k = 1.0e-10 * 1.0e-6 * 10 = 1.0e-3
*Dynamic viscosity
1.0e-6

*Implicit hca steps
step1,step2,step3,step4
*Recording hca steps
step5
*Explicit hca steps
step6
*HCA cycle time
1
```

Listing 1: Definition of the soil material

The penalty method is used to enforce the contact constraints as is given in Listing 2. No separation between soil and pile is possible once the contact is active. A simple Coulomb friction model with a friction coefficient of 0.3 is used.

The contact between soil and pile is discretised using a mortar contact discretisation technique (line 5), which implementation is discussed in detail in [3].

A small value for the contact clearance is defined, which leads to a small value of Initial contact pressure. This is beneficial since the Initial stress state is closer to equilibrium, in which the earth pressure is acting as contact pressure.

```
0 **---------------------------------------- Contact ----------------------------------------
1 *Interaction,name=pen,MECHANICAL=penalty,no separation
2 *friction, model=MC
3 0.3,0,0
4
5 *Contact Pair,interaction=pen,discretisation=element mortar
6 BodenMonopileAussenSurface,MonopileAussenSurface
7 BodenMonopileInnenSurface,MonopileInnenSurface
8 BodenPfahlfussSurface,MonopileUntenSurface
9 *contact options, name=pen, clearance=1d-6
```

Listing 2: Definition of the contact

The Initial conditions for stress and state variables are defined in user subroutines (see the files shipped with this document). The void ratio used for the calculation of the density is given in line 10. This value does not necessary have to coincide with the value given in the user Initial state file. A hydrostatic pore water pressure is defined in line 13 for the entire soil mass.

```
0 **---------------------------------------- Initial conditions ----------------------------------------
1 **
2 *Initial CONDITIONS,Type=STRESS,user
3
4 *Initial CONDITIONS,Type=state variables, USER
5 Geometrie_Bodenkoerper_Aussen4
6 Geometrie_Bodenkoerper_Aussen5
7 Geometrie_Bodenkoerper_Innen4
8
9 *Initial conditions, type=void ratio, default
10 BodenInstance.BodenAlle,0.7524d0
11
12 *Initial conditions, type=pore water pressure, default
13 BodenInstance.BodenAlle, 0.0d0, 0.0d0, 50.0d0, 500.0d0
```

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

```
**------------------------------------ Define Amplitudes ------------------------------
*Amplitude, Name = LoadingRamp , Type = RAMP
0.0, 0.0, 1.0d0, 1.0d0
*Amplitude, Name = Sinus1Hz , Type = PERIODIC
1,0.0,0.0,6.28
0,1
```

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 $t_2 = 1$. To apply the cyclic loading, a sinusoidal Amplitude with a frequency of 1 Hz is defined in addition.

1.2.2 Steps and loading definitions

The first step is defined in Listing 3. Line 3 starts the step environment and defines the name of the step. The analysis type of the step is given in line 4, which will be a static step considering non-linear geometry effects (`nlgeom`). The keyword **`Body force`** imposes a gravitational force which is applied instantaneous (`Instant`). The element sets *BodenInstance.BodenAlle* and *MonopileInstance.MonopileAlle* are loaded by the gravity (Amplitude 10 m/s^2, directed downwards with the normalized vector of the gravity $\vec{\mathbf{b}} = \{0, 0, 1\}$).

The boundary conditions are specified from line 16 to line 24. All nodes of the pile and the soil are constraint in 1-, 2- and 3-direction. Therefore, no displacement is possible for any Node in the model. The pore water pressure is prescribed hydrostatically using a user subroutine (see the files shipped with this document). The error controls are modified in lines 26 to 27. Only the local convergence controls are set active.

The output demand is specified from line 29 to line 48. The output is written in the vtk format (suitable for `ParaView`) and is of ASCII type. The Node output includes the displacement (`U`) and pore water pressure (`PW`) of the nodes. The element output includes the stress (`S`) as well as the void ratio (`Void_ratio`) and the contact output variables (`Contact`). The contact output contains contact stresses as well as contact distances for each contact Node. In addition, a print output is defined in line 34. The displacement and the pore water pressure is printed for several nodes of the model.

```
0  **------------------------------------ Steps ------------------------------------
1  ** STEP: Eigengewicht_Boden
2  **
3  *Step, name=step1,inc=1,nlgeom
4  *static
5  1,1,1d-4,1
6
7  ** LOADS
8
9
10 *Body force, Instant
11 BodenInstance.BodenAlle, GRAV, 10., 0., 0., 1.
12
13 *Body force, Instant
14 MonopileInstance.MonopileAlle, GRAV, 10., 0., 0., 1.
15
16 *Boundary
17 MonopileInstance.MonopileAlle, u1,0.0d0
18 MonopileInstance.MonopileAlle, u2,0.0d0
19 MonopileInstance.MonopileAlle, u3,0.0d0
20 BodenInstance.BodenAlle, u1,0.0d0
21 BodenInstance.BodenAlle, u2,0.0d0
22 BodenInstance.BodenAlle, u3,0.0d0
23 *Uboundary
24 BodenInstance.BodenAlle, pw,1.0d0
25
26 *Controls, global, deactivate
27 *Controls, u, activate
28
29 *output, field, vtk, ASCII
30 *Node output, nset = BodenInstance.BodenAlle
31 U,pw
32 *element output, elset = BodenInstance.BodenAlle
33 S, Contact, void_ratio
34 *output, print
35 *Node output, nset = Knoten_Monopile_Aussen_Oben_Links
```

```
U
*Node output , nset = Knoten_Monopile_Aussen_Oben_Links
U
*Node output , nset = Monopile_Symmetrie_Vorne
U, coords
*Node output , nset = PWD_rechts_oben
U, coords , pw
*Node output , nset = PWD_links_oben
U, coords , pw
*Node output , nset = PWD_rechts_unten
U, coords , pw
*Node output , nset = PWD_links_unten
U, coords , pw
*End STEP
```

Listing 3: Definition of the first step

In the second step given in Listing 4, the correct boundary conditions of the model are established. The back-side of the soil is constraint in the two horizontal directions in lines 16-17. The bottom is constraint in the vertical direction in line 18. The symmetry axis of both parts of the model is constraint in the y-direction (see Fig. 1) in lines 19 to 22.

A vertical loading is applied in line 29 to 32. The left and the right top Node of the pile are gradually loaded by 500 kN over the course of the step. An additional surface pressure on the top of the model is applied in lines 34 to 37. The output remains identical to the first step.

```
0 ****------------------------------ Steps ------------------------------
1 **
2 *Step , name=step2 , inc=100,nlgeom
3 *static
4 0.5 ,1 ,1d-4,1
5
6 ** LOADS
7
8 *Body force , Instant
9 BodenInstance . BodenAlle , GRAV, 10. , 0. , 0. , 1.
10
11 *Body force , Instant
12 MonopileInstance . MonopileAlle , GRAV, 10. , 0. , 0. , 1.
13
14 ** BOUNDARY CONDITIONS
15 *Boundary
16 Boden_Hinten ,u1 ,0.0d0
17 Boden_Hinten ,u2 ,0.0d0
18 Boden_Unten_S5 ,u3 ,0.0d0
19 Boden_Symmetrie_Vorne_Links , u2 ,0.0d0
20 Boden_Symmetrie_Vorne_Mitte , u2 ,0.0d0
21 Monopile_Symmetrie_Vorne ,u2 ,0.0d0
22 Boden_Symmetrie_Vorne_Rechts , u2 ,0.0d0
23 *Uboundary
24 BodenInstance . BodenAlle , pw,1.0d0
25
26 ** LOADS
27
28
29 *Cload , Amplitude=LoadingRamp
30 Knoten_Monopile_Aussen_Oben_Rechts , 3 , 500
31 *Cload , Amplitude=LoadingRamp
32 Knoten_Monopile_Aussen_Oben_Links , 3 , 500
33
34 *Dload , Instant
35 _BodenObenS1Surface_S2 ,P2,-3d0
36 *Dload , Instant
37 _BodenObenS1Surface_S3 ,P3,-3d0
38
39
40 *Controls , global , deactivate
41 *Controls , u, modify
42 0.05 ,0.05 ,0.08 ,1e-7,1e-7
43
44 *output , field , vtk , ASCII
45 *Node output , nset = BodenInstance . BodenAlle
46 U,pw
```

```
*element output, elset = BodenInstance.BodenAlle
S, Contact, void_ratio
*output, print
*Node output, nset = Knoten_Monopile_Aussen_Oben_Links
U
*Node output, nset = Knoten_Monopile_Aussen_Oben_Links
U
*Node output, nset = Monopile_Symmetrie_Vorne
U, coords
*Node output, nset = PWD_rechts_oben
U, coords, pw
*Node output, nset = PWD_links_oben
U, coords, pw
*Node output, nset = PWD_rechts_unten
U, coords, pw
*Node output, nset = PWD_links_unten
U, coords, pw
*End STEP
```

Listing 4: Definition of the second step

In the third step given in Listing 5 the mean value of horizontal loading is applied in line 35 to 45. The left and the right top Node of the pile is gradually loaded by 50 kN in horizontal direction and a moment (represented by a vertical force pair) of $600 \cdot 2 \cdot 2 = 2400$ kNm over the course of the step. The other definitions remain identical to the previous step.

```
0 *——————————————————— Steps ———————————————————
1 **
2 *Step, name=step3,inc=100,nlgeom
3 *static
4 0.05,1.0d0,1d-4,0.05
5 **
6 ** LOADS
7
8 *Body force, Instant
9 BodenInstance.BodenAlle, GRAV, 10., 0., 0., 1.
10
11 *Body force, Instant
12 MonopileInstance.MonopileAlle, GRAV, 10., 0., 0., 1.
13
14 ** BOUNDARY CONDITIONS
15 *Boundary
16 Boden_Hinten,u1,0.0d0
17 Boden_Hinten,u2,0.0d0
18 Boden_Unten_S5,u3,0.0d0
19 Boden_Symmetrie_Vorne_Links, u2,0.0d0
20 Boden_Symmetrie_Vorne_Mitte, u2,0.0d0
21 Monopile_Symmetrie_Vorne,u2,0.0d0
22 Boden_Symmetrie_Vorne_Rechts, u2,0.0d0
23
24 *Uboundary
25 BodenInstance.BodenAlle, pw,1.0d0
26
27 **
28 ** LOADS
29 **
30 *cload,Instant
31 Knoten_Monopile_Aussen_Oben_Rechts, 3, 500
32 *cload,Instant
33 Knoten_Monopile_Aussen_Oben_Links, 3, 500
34
35 *Cload,Amplitude=LoadingRamp
36 Knoten_Monopile_Aussen_Oben_Rechts, 1, -50
37
38 *Cload,Amplitude=LoadingRamp
39 Knoten_Monopile_Aussen_Oben_Links, 1, -50
40
41 *Cload,Amplitude=LoadingRamp
42 Knoten_Monopile_Aussen_Oben_Rechts, 3, 600
43
44 *Cload,Amplitude=LoadingRamp
45 Knoten_Monopile_Aussen_Oben_Links, 3, -600
46
```

```
*Dload,Instant
_BodenObenS1Surface_S2,P2,-3d0
*Dload ,Instant
_BodenObenS1Surface_S3,P3,-3d0
**
*Controls, global, deactivate
*Controls, u, activate
*Controls, pw, deactivate
**
*output, field, vtk, ASCII
*Node output, nset = BodenInstance.BodenAlle
U,pw,V,A
*element output, elset = BodenInstance.BodenAlle
S, Contact, void_ratio,contact_diagnostic,VIS_STRESS11,VIS_STRESS22,VIS_STRESS33
*output, print
*Node output, nset = Knoten_Monopile_Aussen_Oben_Links
U
*Node output, nset = Monopile_Symmetrie_Vorne
U, coords
*Node output, nset = PWD_rechts_oben
U, coords, pw
*Node output, nset = PWD_links_oben
U, coords, pw
*Node output, nset = PWD_rechts_unten
U, coords, pw
*Node output, nset = PWD_links_unten
U, coords, pw
*End STEP
```

Listing 5: Definition of the third step

The fourth and fifth step are identical (expect of the step name). *step4* is given in Listing 6. The horizontal sinusoidal load is applied, which also has a component in horizontal direction and a moment component. This is defined in lines 46 to 56. Note that for this example no effects from excess pore water pressure and consolidation are accounted for. This is defined by setting the step to static and constraining the pore water pressure in the entire model. If one would want to consider these effects, the type of analysis in line 2 has to be changed to a transient calculation, including pore water pressure change and water flow (consolidation). In addition, the boundary condition in lines 63 to 64 would have to be deleted. The boundary conditions in lines 22 to 26 would in this case allow for drainage at the top and the back of the model. In the current definition as static step they are obsolete.

```
0  **
1  *Step, name=step4,inc=1000, no cut back,nlgeom
2  *static
3  0.05,1.0d0,1d-4,0.05
4  **
5  ** LOADS
6  
7  *Body force, Instant
8  BodenInstance.BodenAlle, GRAV, 10., 0., 0., 1.
9  
10 *Body force, Instant
11 MonopileInstance.MonopileAlle, GRAV, 10., 0., 0., 1.
12 
13 ** BOUNDARY CONDITIONS
14 *Boundary
15 Boden_Hinten,u1,0.0d0
16 Boden_Hinten,u2,0.0d0
17 Boden_Unten_S5,u3,0.0d0
18 Boden_Symmetrie_Vorne_Links, u2,0.0d0
19 Boden_Symmetrie_Vorne_Mitte, u2,0.0d0
20 Monopile_Symmetrie_Vorne,u2,0.0d0
21 Boden_Symmetrie_Vorne_Rechts, u2,0.0d0
22 *boundary
23 Boden_Oben_S1, pw,0.0d0
24 *uboundary
25 Boden_hinten, pw,1.0d0
26 Boden_Unten_S5, pw,1.0d0
27 **
28 ** LOADS
29 **
```

```
*cload,Instant
Knoten_Monopile_Aussen_Oben_Rechts, 3, 500
*cload,Instant
Knoten_Monopile_Aussen_Oben_Links, 3, 500
*Cload,Instant
Knoten_Monopile_Aussen_Oben_Rechts, 1, -50

*Cload,Instant
Knoten_Monopile_Aussen_Oben_Links, 1, -50

*Cload,Instant
Knoten_Monopile_Aussen_Oben_Rechts, 3, 600

*Cload,Instant
Knoten_Monopile_Aussen_Oben_Links, 3, -600

*Cload, Amplitude=Sinus1Hz
Knoten_Monopile_Aussen_Oben_Rechts, 3, 600

*Cload, Amplitude=Sinus1Hz
Knoten_Monopile_Aussen_Oben_Links, 3, -600

*Cload, Amplitude=Sinus1Hz
Knoten_Monopile_Aussen_Oben_Rechts, 1, -50

*Cload, Amplitude=Sinus1Hz
Knoten_Monopile_Aussen_Oben_Links, 1, -50
*Dload,Instant
_BodenObenS1Surface_S2,P2,-3d0
*Dload ,Instant
_BodenObenS1Surface_S3,P3,-3d0
*Uboundary
BodenInstance.BodenAlle, pw,1.0d0

**
**
*Controls, global, deactivate
*Controls, u, modify
0.03,0.03,0.04,1e-7,1e-7
*Controls, pw, deactivate
**
*output, field, vtk, ASCII
*frequency=5
*Node output, nset = BodenInstance.BodenAlle
U,pw,V,A
*element output, elset = BodenInstance.BodenAlle
S, Contact, void_ratio
*output, print
*Node output, nset = Knoten_Monopile_Aussen_Oben_Links
U
*Node output, nset = Monopile_Symmetrie_Vorne
U, coords
*Node output, nset = PWD_rechts_oben
U, coords, pw
*Node output, nset = PWD_links_oben
U, coords, pw
*Node output, nset = PWD_rechts_unten
U, coords, pw
*Node output, nset = PWD_links_unten
U, coords, pw
*End STEP
```

Listing 6: Definition of the fourth and fifth step

Lastly, the HCA step is defined in step 6 (Listing 7) simulating 10^6 further cycles. Only the mean value of load is applied in this step. The strain Amplitude was automatically calculated at the End of step 5. Some HCA-specific output variables are requested in line 66.

```
0 **
1 *Step, name=step6,inc=100000,nlgeom
2 *static
3 1,1d6,1d-4,1d4
4 **
```

```
** LOADS

*Body force, Instant
BodenInstance.BodenAlle, GRAV, 10., 0., 0., 1.

*Body force, Instant
MonopileInstance.MonopileAlle, GRAV, 10., 0., 0., 1.

** BOUNDARY CONDITIONS
*Boundary
Boden_Hinten,u1,0.0d0
Boden_Hinten,u2,0.0d0
Boden_Unten_S5,u3,0.0d0
Boden_Symmetrie_Vorne_Links, u2,0.0d0
Boden_Symmetrie_Vorne_Mitte, u2,0.0d0
Monopile_Symmetrie_Vorne,u2,0.0d0
Boden_Symmetrie_Vorne_Rechts, u2,0.0d0
*boundary
Boden_Oben_S1, pw,0.0d0
*uboundary
Boden_hinten, pw,1.0d0
Boden_Unten_S5, pw,1.0d0
**
** LOADS
**
*cload,Instant
Knoten_Monopile_Aussen_Oben_Rechts, 3, 500
*cload,Instant
Knoten_Monopile_Aussen_Oben_Links, 3, 500

*Cload,Instant
Knoten_Monopile_Aussen_Oben_Rechts, 1, -50

*Cload,Instant
Knoten_Monopile_Aussen_Oben_Links, 1, -50

*Cload,Instant
Knoten_Monopile_Aussen_Oben_Rechts, 3, 600

*Cload,Instant
Knoten_Monopile_Aussen_Oben_Links, 3, -600
*Dload,Instant
_BodenObenS1Surface_S2,P2,-3d0
*Dload ,Instant
_BodenObenS1Surface_S3,P3,-3d0
*Uboundary
BodenInstance.BodenAlle, pw,1.0d0
**


**
*Controls, global, deactivate
*Controls, u, modify
0.12,0.12,0.08,1e-7,1e-7
**
*output, field, vtk, ASCII
*frequency=5
*Node output, nset = BodenInstance.BodenAlle
U,pw
*element output, elset = BodenInstance.BodenAlle
S, Contact, void_ratio, Strain_ampl, Strain_acc, cyclic_history, fe, fampl, fy, fp,
*output, print
*Node output, nset = Knoten_Monopile_Aussen_Oben_Links
U
*Node output, nset = Monopile_Symmetrie_Vorne
U, coords
*Node output, nset = PWD_rechts_oben
U, coords, pw
*Node output, nset = PWD_links_oben
U, coords, pw
*Node output, nset = PWD_rechts_unten
U, coords, pw
*Node output, nset = PWD_links_unten
U, coords, pw
```

```
*End STEP
```

Listing 7: Definition of the sixth step

The calculation is started by calling `numgeo` over the command line, specifying the name of the input file *step* (without ending), confirming by pressing enter, specifying the number of CPUs (depending on the hardware, 4 CPUs are appropriated here) and pressing enter again.

1.3 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 and if any error files were generated in the calculation folder.

To see the output of the calculation, start `ParaView` and open the pvd file (in the right upper options bar: "File" → "Open...").

The monopile pile head rotation is depicted in Fig. 2 using the python script shipped with this file. The field of the strain Amplitude and the final deformed configuration after 10^6 cycles (5-times enhanced) is given in Fig. 3.

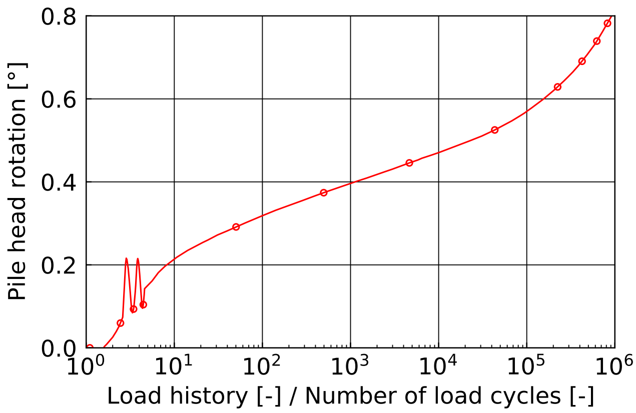

Figure 2: Pile head rotation vs. time or number of load cycles

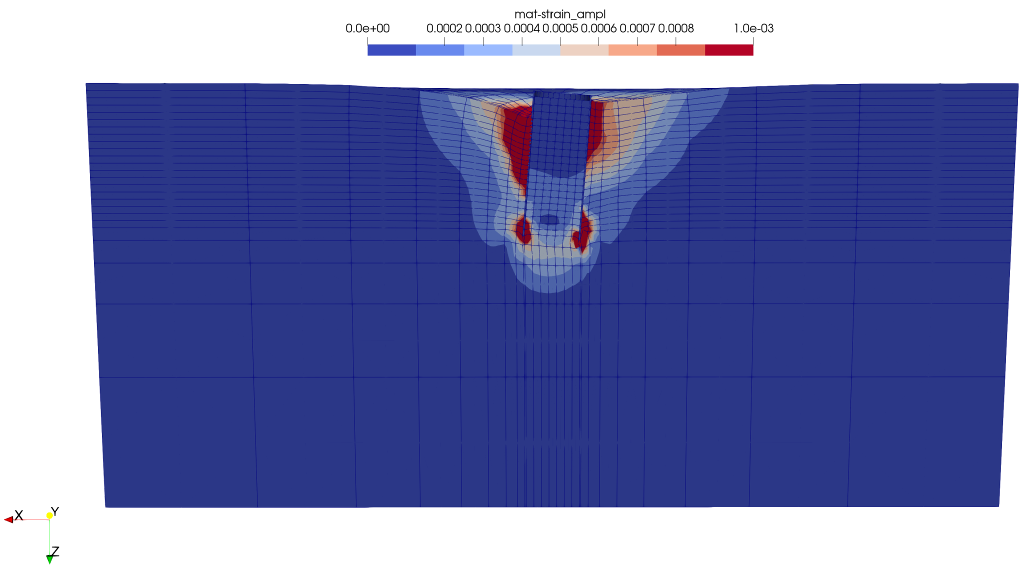

Figure 3: Field of the strain Amplitude and final pattern of deformations (5-times enhanced)

References

[1] Staubach, P., Machaček, J., Tschirschky, L., and Wichtmann, T. "Enhancement of a high-cycle accumulation model by an adaptive strain amplitude and its application to monopile foundations". In: *International Journal for Numerical and Analytical Methods in Geomechanics* 46.2 (Feb. 2022), pp. 315–338. ISSN: 0363-9061. DOI: 10.1002/nag.3301.

[2] Staubach, P. and Wichtmann, T. "Long-term deformations of monopile foundations for offshore wind turbines studied with a high-cycle accumulation model". In: *Computers and Geotechnics* 124 (Aug. 2020), p. 103553. ISSN: 0266352X. DOI: 10.1016/j.compgeo.2020.103553.

[3] Staubach, P., Machaček, J., and Wichtmann, T. "Novel approach to apply existing constitutive soil models to the modelling of interfaces". In: *International Journal for Numerical and Analytical Methods in Geomechanics* 46.7 (May 2022), pp. 1241–1271. ISSN: 0363-9061. DOI: 10.1002/nag.3344.

[4] Taylor, C. and Hood, P. "A numerical solution of the Navier-Stokes equations using the finite element technique". In: *Computers and Fluids* 1.1 (1973), pp. 73–100. ISSN: 00457930. DOI: 10.1016/0045-7930(73)90027-3.