A finite element code for geotechnical simulations

Tutorials
-
Series E: Elastic pile

©Jan Machaček, Patrick Staubach
WWW.NUMGEO.DE
September 29, 2022

History:

2022	Patrick Staubach, Jan Machaček	Initial version

numgeo: Tutorials

www.numgeo.de
September 29, 2022

Contents

1 Elastic pile: numgeo vs. Abaqus

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.

In order to show the performance of `numgeo` in three-dimensional contact analyses, the cyclic loading of a pile in elastic soil using `numgeo` and `Abaqus` is investigated. Figure 1 shows the numerical models and the comparison of the spatial distribution of normalised horizontal stress ($\sigma_{11}/\sigma_{11,max}$) and the deformed configuration at the maximum loading. Note that the mesh is identical in both simulations.

Only elastic materials are considered in this benchmark. The following properties are used:

```
*material, name = soil, phases = 1
*Mechanical = Linear_Elasticity
20000, 0.3
*Density
1.56
*material, name = pile, phases = 1
*Mechanical = Linear_Elasticity
1e8, 0.3
*Density
8.96
```

In both programs the penalty method is used to enforce the contact between soil and pile. To be comparable, a fixed factor of $2 \cdot 10^6$ kN/m^3 is used:

```
*INTERACTION,name=penalty,MECHANICAL=penalty
2e6
**
*Contact Pair,interaction=penalty,discretisation=element mortar
surf_Soil_Inner_Face,surf_Monopile_Outer_Face
surf_Inner_Soil_Outer_Face,surf_Monopile_Inner_Face
surf_Soil_PileTip_Face, surf_MonopileTip
```

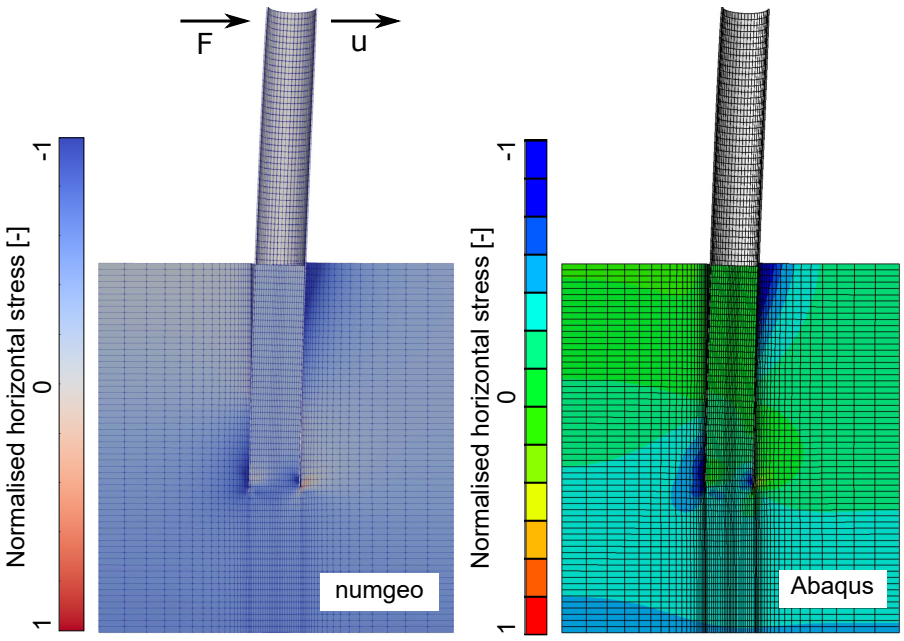

Figure 1: Models and spatial distribution of normalised horizontal stress ($\sigma_{11,max}$ is identical for the results of both programs) at the peak loading using `numgeo` and `Abaqus`, respectively

A sinusoidal load is applied to the pile head in *step4*. The explanation of the first three steps is skipped here.

In these steps the initial stress is initialized, the contact is initialized and the average value of lateral loading is applied.

```
**------------------------------------ Steps ------------------------------------
*Step, name=step4, inc = 100000
*Static
0.05,1.,1e-6,0.05
**

*BODY FORCE, INSTANT
soil-1.soil, GRAV, 10, 0, 0, -1
*BODY FORCE, INSTANT
pile-1.pile_all, GRAV, 10, 0, 0, -1
**
*Dload, INSTANT
soil-1.surf_Soil_Top_F4, P4,-1.0
*Dload, INSTANT
soil-1.surf_Soil_Top_F5, p5,-1.0
*Dload, INSTANT
soil-1.surf_Soil_Top_F6, p6,-1.0
*Dload, INSTANT
soil-1.surf_Inner_Soil_Top_F4, p4,-1.0
*Dload, INSTANT
soil-1.surf_Inner_Soil_Top_F5, p5,-1.0
*Dload, INSTANT
soil-1.surf_Inner_Soil_Top_F6, p6,-1.0
**
*boundary
soil-1.surf_Soil_Back,u2,0.0d0
soil-1.surf_Soil_Front,u2,0.0d0
soil-1.surf_Soil_Left,u1,0.0d0
soil-1.surf_Soil_Right,u1,0.0d0
soil-1.surf_Soil_Bottomsoil,u3,0.0d0
soil-1.surf_Inner_Soil_Front,u2,0.0d0
pile-1.surf_Monopile_Front_Face,u2,0.0d0
*cload,instant
pile-1.pile_weight,3,-0.035
*Cload, instant
pile-1.pile_load,1,0.011325
**
*Cload, amplitude =Sinus1Hz
pile-1.pile_load,1,0.011325
**
*Controls, global, deactivate
*Controls, u, activate
**
*output, field, vtk, ASCII
*frequency=10
*node output, nset = soil-1.soil
U
*element output, elset = soil-1.soil
S, Contact, void_ratio
*output, print
*node output, nset = pile-1.pile_load
U
*node output, nset = pile-1.pile_mud
U
*END STEP
```

The pile head displacement for both programs is depicted in Fig. 2. A good accordance of the two FE programs is observed.

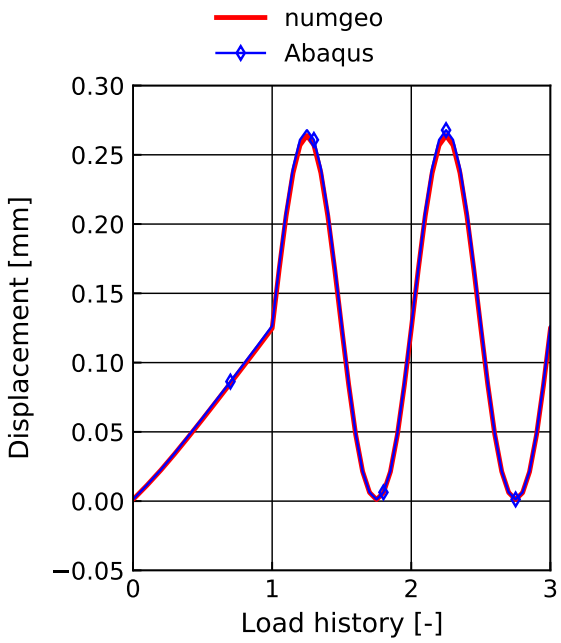

Figure 2: Horizontal displacement using `numgeo` and `Abaqus`, respectively