numgeo

A finite element code for geotechnical simulations

Tutorials
-
Series D: Compression wave propagation

www.numgeo.de
November 26, 2021

History:

2021	Patrick Staubach, Jan Machaček	Initial version

numgeo: Tutorials, Series D: Compression wave propagation

www.numgeo.de
November 26, 2021

Contents

1 Compression wave propagation in a water-saturated column using various finite element formulations 2

1.1 Input file for the `u8p4-sat` and `u8p4-sat-red` elements 3

1.2 Input file for the `u4u4`, `u4u4-red` and `u8u8` elements 6

1.3 Input file for the `u4p4u4-sat`, `u8p4u8-sat`, `u8p4u8-sat-red` and `u8p8u8-sat` elements 7

1.4 Input file for the 3D simulations using the `u8u8-3D-red`, `u20p8-sat`, `u20u20`, `u27p8-sat` and `u20p8u20-sat` elements 8

1.5 Results 8

References 12

1 Compression wave propagation in a water-saturated column using various finite element formulations

In the following, the compression wave propagation in a water-saturated soil column is studied using the various finite element formulations available in `numgeo`. The soil column of 10 m height shown in Figure 1 is composed of 100 elements with dimensions 0.1 m × 0.1 m (× 0.1 m for 3D simulations). The soil within the column is elastic. All elements are given the same elastic parameters (E = 10,000 kPa, $\nu = 0.3$). At the top of the column a Heaviside load is applied which leads to the propagation of a compression wave.

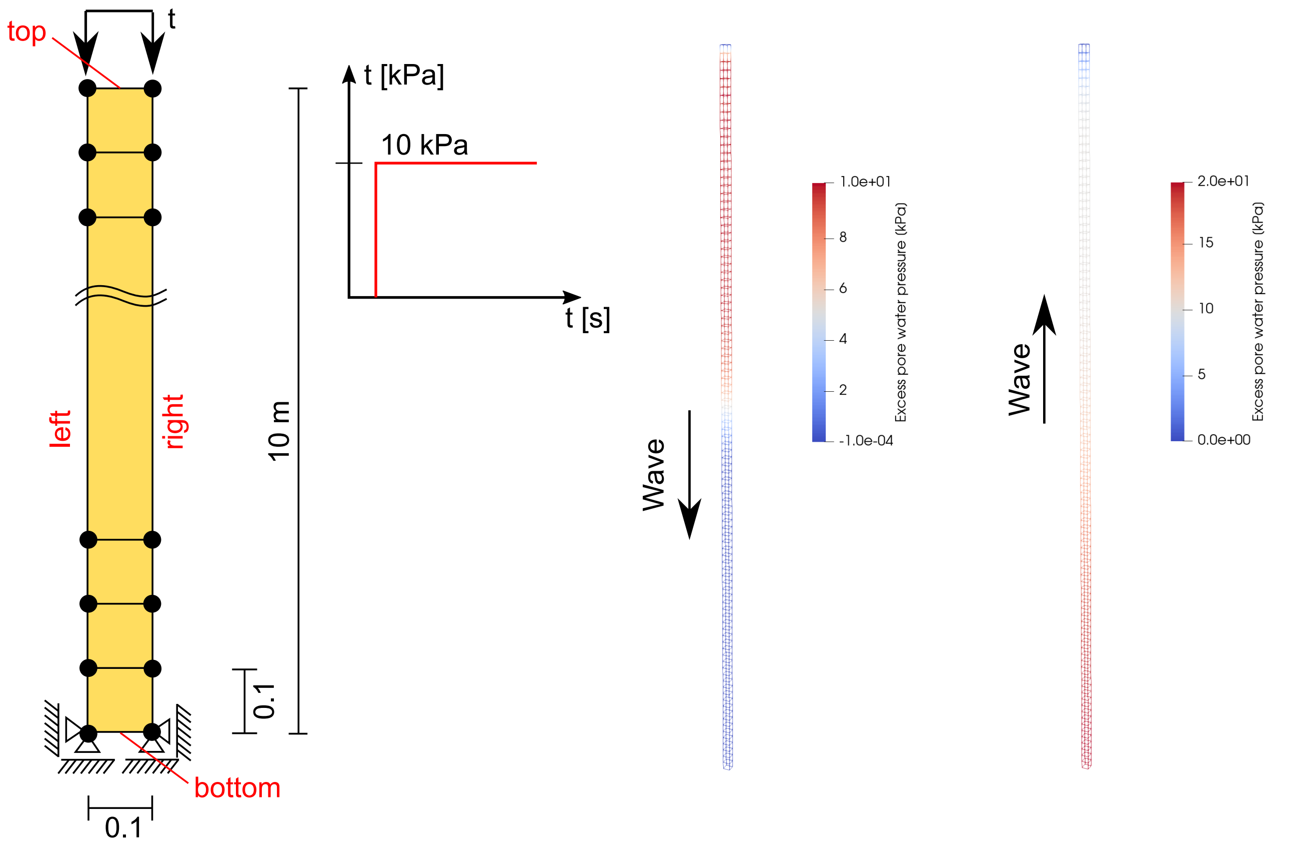

Figure 1: Model, specifications and simulation results of the compression wave propagation example

The naming convention of the different element formulations used is as follows: the number after `u` indicates the number of nodes discretizing the solid displacements, the number after `p` indicates the number of nodes discretizing the pore water pressure and the number after (capital) `U` indicates the number of nodes discretizing the water displacements. `-sat` indicates that the element assumes fully saturated conditions and `-red` that the element uses a reduced integration scheme. The following element formulations are studied

- `u8p4-sat` element: **2D**, **u**-p element formulation, 4 nodes discretising solid displacement, 4 nodes discretising pore water pressure, serendipity formulation, integration with 9 integration points
- `u8p4-sat-red` element: **2D**, **u**-p element formulation, 4 nodes discretising solid displacement, 4 nodes discretising pore water pressure, serendipity formulation, reduced integration with 4 integration points
- `u4p4u4-sat` element: **2D**, **u**-p-**U** element formulation, 4 nodes discretising solid displacement, 4 nodes discretising pore water pressure, 4 nodes discretising water displacement, integration with 4 integration points
- `u8p4u8-sat` element: **2D**, **u**-p-**U** element formulation, 8 nodes discretising solid displacement, 4 nodes discretising pore water pressure, 8 nodes discretising water displacement, serendipity formulation, integration with 9 integration points
- `u8p4u8-sat-red` element: **2D**, **u**-p-**U** element formulation, 8 nodes discretising solid displacement, 4 nodes discretising pore water pressure, 8 nodes discretising water displacement, serendipity formulation, reduced integration with 4 integration points

- `u8p8u8-sat` element: **2D**, **u**-p-**U** element formulation, 8 nodes discretising solid displacement, 8 nodes discretising pore water pressure, 8 nodes discretising water displacement, serendipity formulation, integration with 9 integration points
- `u4u4` element: **2D**, **u**-**U** element formulation, 4 nodes discretising solid displacement, 4 nodes discretising water displacement, integration with 4 integration point
- `u4u4-red` element: **2D**, **u**-**U** element formulation, 4 nodes discretising solid displacement, 4 nodes discretising water displacement, integration with 1 integration point
- `u8u8` element: **2D**, **u**-**U** element formulation, 8 nodes discretising solid displacement, 8 nodes discretising water displacement, serendipity formulation, integration with 9 integration points
- `u8u8-3D-red` element: **3D**, **u**-**U** element formulation, 8 nodes discretising solid displacement, 8 nodes discretising water displacement, integration with 1 integration point
- `u20p8-sat` element: **3D**, **u**-p element formulation, 20 nodes discretising solid displacement, 8 nodes discretising pore water pressure, serendipity formulation, integration with 8 integration points
- `u27p8-sat` element: **3D**, **u**-p element formulation, 27 nodes discretising solid displacement, 8 nodes discretising pore water pressure, bi-quadratic formulation, integration with 27 integration points
- `u20p8u20-sat` element: **3D**, **u**-p-**U** element formulation, 20 nodes discretising solid displacement, 8 nodes discretising pore water pressure, 20 nodes discretising water displacement, serendipity formulation, integration with 8 integration points

An analytical solution for the wave-propagation has been presented in [1], which is used as reference solution for the example.

The creation of the mesh is skipped for this example. The input files including the mesh are supplied together with this tutorial. First, the input file for a simulation using 2D **u**-p element formulations is discussed. Following, the 3D case and the input files for the **u**-**U** and **u**-p-**U** element formulations are given.

1.1 Input file for the `u8p4-sat` and `u8p4-sat-red` elements

Following is an excerpt of the mesh-part of the input file defining `u8p4-sat` elements in line 11:

```
1  *Node
2  1,            0.,           0.
3  2,   0.100000001,           0.
4  3,            0.,  0.100000001
5  .
6  .
7  .
8  501,   0.100000001,   9.94999981
9  502,  0.0500000007,          10.
10 503,            0.,   9.94999981
11 *Element, type=U8P4-sat
12 1,   1,   2,   4,   3, 203, 204, 205, 206
13 2,   3,   4,   6,   5, 205, 207, 208, 209
14 3,   5,   6,   8,   7, 208, 210, 211, 212
15 .
16 .
17 .
18 **---Sets---
19 *Nset, nset=all, generate
20 1,  503,    1
21 *Elset, elset=all, generate
22 1,  100,    1
23 *Nset, nset=bottom
24 1,   2, 203
25 *Nset, nset=bottom_node
26 1,
27 *Elset, elset=bottom
28 1,
29 *Nset, nset=left
30 .
```

```
31 .
32 .
33 **------------------------------- Solid section ------------------------------
34 *Solid Section, elset=all, material=elastic
```

Define an elastic material

```
35 **------------------------------- Materials ------------------------------
36 *material, name = elastic, phases = 2
37 **
38 *mechanical = Linear_Elasticity
39 10000.d0, 0.3d0
40 **
41 *Density
42 2.7, 1.0
43 **
44 *Bulk modulus
45 1.1d6
46 **
47 *Permeability = isotropic
48 1d-10
49 *Dynamic viscosity
50 1d-6
```

with two phases. The permeability is 10^{-10} m^2 and the dynamic viscosity 10^{-6} kPas which results in a hydraulic conductivity of $k^w = 10^{-3}$ m/s with a gravity of 10 m/s^2. The bulk modulus of pore water is set to $K^w = 1.1 \cdot 10^6$ kPa.

The initial void ratio is assumed to be constant over the height of the column and $e = 1$:

```
51 **-------------Initial Conditions-------------**
52 *INITIAL CONDITIONS, TYPE= void ratio, default
53 all, 1.0d0
```

Define the initial (effective) stress conditions:

```
54 *INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC
55 all, 10.0, 0, 0, -85.0, 0.5, 0.5
```

The element set *all* is assigned an initial effective stress state with $\sigma_{22}(y = 0.0 \text{ m}) = -85.0$ kPa ($\gamma' = (1-n) \cdot (\gamma_s - \gamma_w) = \big(1 - \dfrac{e}{1+e}\big) \cdot \big(\gamma_s - \gamma_w\big) = 8.5$ kN/m^3) and $\sigma_{22}(y = 10.0 \text{ m}) = 0.0$ kPa.

The initial pore water distribution has to be defined in addition:

```
56 *INITIAL CONDITIONS, TYPE=PORE WATER PRESSURE
57 all_soil, 10.0, 0.0,0.0, 100.0
```

A hydrostatic initial pore water distribution is defined.
The first step of a geotechnical analysis is usually the so called Geostatic step wherein the initial stress state given in line 21 is checked against the stress state resulting out of gravity (or an external load). An initial stress state that is not in accordance with the gravitational stress state eventually leads to displacements. In general, the less the displacement after the Geostatic step the better the stress state accords to the stress due to gravity. Too large values of displacement indicated a falsification of the initial state which can lead to errors 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 **
4 *BODY FORCE, INSTANT
5 all, GRAV, 10.d0, 0, -1, 0
6 **
7 *BOUNDARY, op=new
8 left, u1, 0.0d0
9 right, u1, 0.0d0
10 bottom, u2, 0.0d0
```

```
11 *boundary,hydrostatic
12 all,pw,10,10
13 **
14 **
15 *output, field, vtk, ASCII
16 *frequency=1
17 *node output, nset = all
18 U, pw
19 *element output, elset = all
20 S
21 **
22 *END STEP
23 **
```

Listing 1: Definition of the Geostatic step using `u8p4-sat` and `u8p4-sat-red` elements

The gravity is applied to all elements of the column in line 5. The column is only free to move vertically. During a static step, all pore water pressure degree-of-freedoms should be constrained. Therefore, the hydrostatic pore water pressure distribution is applied as Dirichlet boundary condition in line 11 to 12.

The dynamic step following the Geostatic step is defined as follows:

```
0 **———————————————————— Steps ————————————————————————
1 *Step, name=step2, inc = 200000
2 *Dynamic
3 0.00005, 0.2, 0.00001, 0.00005
4 *Hilber-Hughes-Taylor = -0.1
5 **
6 *BODY FORCE, INSTANT
7 all, GRAV, 10.d0, 0, -1, 0
8 **
9 *BOUNDARY, op=new
10 left, u1, 0.0d0
11 right, u1, 0.0d0
12 bottom, u2, 0.0d0
13 top, pw, 0.0d0
14 **
15 *DLOAD, INSTANT
16 _top_surf_S3,P3,-10.0
17 **
18 *Output, print
19 *frequency=5
20 *node output, nset = top_node
21 U
22 *node output, nset = bottom_node
23 pw
24 **
25 **
26 *END STEP
27 **
28 *END INPUT
```

Listing 2: Definition of the dynamic step for the compression wave propagation using `u8p4-sat` and `u8p4-sat-red` elements

In a dynamic analysis, physical time is specified. Therefore, 0.2 s of wave propagation are studied according to line 3. In addition, a dynamic analysis requires a time integration scheme (since the displacement is discretised, the velocity and the accelerations are unknown and have to be determined. The Hilber-Hughes-Taylor (HHT) time integration scheme is chosen here and a slight numerical damping with $\alpha = -0.05$ is defined. The HHT time integration is the most frequently applied method in FE codes due to its well proven numerical stability.

The boundary conditions are specified from line 9 to 13. The left and right side of the soil column are constrained horizontally and the bottom is fixed in the vertical direction. The top is open for drainage by prescribing zero pore water pressure. Due to the sudden application of a distributed load of 10 kPa at the top of the column defined in line 16, a compression wave travels downwards.

For the evaluation of the results, the vertical displacement of the top node is written to a print output file in lines 18 to 21. In addition, the pore water pressure at the bottom of the column is requested from line 22 to 23.

1.2 Input file for the `u4u4`, `u4u4-red` and `u8u8` elements

Since the `u4u4-red` element uses an one-point integration, an hourglass stiffness has to be defined in the material section:

```
**------------------------------------------ Materials ------------------------------------------
*material, name = elastic, phases = 2
**
*hourglass, stiffness=100,100
```

100 kPa is set for both the solid and the water phase. For the **u-U** element formulation the boundary conditions in the step definitions have to be changed. Lines 7 to 12 of Listing 1 have to be modified to:

```
0  **------------------------------------------ Steps ------------------------------------------
1  *Step, name=step1, inc = 1
2  .
3  .
4  .
5  *BOUNDARY, op=new
6  left, u1, 0.0d0
7  right, u1, 0.0d0
8  bottom, u2, 0.0d0
9  all,w1,0.0
10 all,w2,0.0
11 **
12 **
13 *output, field, vtk, ASCII
14 *frequency=1
15 *node output, nset = all
16 U
17 *element output, elset = all
18 S, pore_pressure
19 **
```

Listing 3: Definition of the Geostatic step using `u4u4`, `u4u4-red` and `u8u8` elements

For the Geostatic step, the water displacement of all nodes is constrained in both direction in lines 9 to 10. In addition, since the pore water pressure is no longer a nodal value, it has to be requested as element output instead of node output.

For the dynamic step following the Geostatic step the boundary conditions have to be modified as well:

```
**------------------------------------------ Steps ------------------------------------------
*Step, name=step2, inc = 200000
*Dynamic
0.00005, 0.2, 0.00001, 0.00005
*Hilber-Hughes-Taylor = -0.1
**
*BODY FORCE, INSTANT
all, GRAV, 10.d0, 0, -1, 0
**
*BOUNDARY, op=new
left, u1, 0.0d0
right, u1, 0.0d0
bottom, u2, 0.0d0
left, w1, 0.0d0
right, w1, 0.0d0
bottom, w2, 0.0d0
**
*DLOAD, INSTANT
_top_surf_S3,P3,-10.0
**
*Output, print
*frequency=5
*node output, nset = top_node
U
*element output, elset = bottom
pore_pressure
**
**
*END STEP
```

```
**
*END INPUT
```

Listing 4: Definition of the dynamic step for the compression wave propagation using `u4u4`, `u4u4-red` and `u8u8` elements

The water displacement is now only constrained at the bottom and the lateral sides of the column. The top is open for drainage by not prescribing the water displacement. In addition, since for **u**-**U** elements the pore water pressure is an integration point value, the print output has to be modified. The pore water pressure is now written for the (centroid) of the element at the bottom of the column instead of the bottom node using **u**-p elements.

1.3 Input file for the `u4p4u4-sat`, `u8p4u8-sat`, `u8p4u8-sat-red` and `u8p8u8-sat` elements

Compared to Listing 1, the boundary conditions have to be changed for application of the **u**-p-**U** element formulation. Lines 7 to 12 of Listing 1 have to be modified to:

```
0  **———————————————————— Steps ————————————————————
1  *Step, name=step1, inc = 1
2  .
3  .
4  .
5  *BOUNDARY, op=new
6  left, u1, 0.0d0
7  right, u1, 0.0d0
8  bottom, u2, 0.0d0
9  all,w1,0.0
10 all,w2,0.0
11 *boundary,hydrostatic
12 all,pw,10,10
13 .
14 .
15 .
```

Listing 5: Definition of the Geostatic step using `u4p4u4-sat`, `u8p4u8-sat`, `u8p4u8-sat-red` and `u8p8u8-sat` elements

For the Geostatic step, the water displacement of all nodes is constrained in both direction in line 9 to 10. Again, the pore water pressure at the nodes is prescribed by a hydrostatic distribution.
For the dynamic step following the Geostatic step the boundary conditions have to be modified as well:

```
**———————————————————— Steps ————————————————————
*Step, name=step2, inc = 200000
*Dynamic
0.00005, 0.2, 0.00001, 0.00005
*Hilber-Hughes-Taylor = -0.1
**
*BODY FORCE, INSTANT
all, GRAV, 10.d0, 0, -1, 0
**
*BOUNDARY, op=new
left, u1, 0.0d0
right, u1, 0.0d0
bottom, u2, 0.0d0
left, w1, 0.0d0
right, w1, 0.0d0
bottom, w2, 0.0d0
top, pw, 0.0d0
**
*DLOAD, INSTANT
_top_surf_S3,P3,-10.0
**
*Output, print
*frequency=5
*node output, nset = top_node
U
*node output, nset = bottom_node
pw
**
```

```
**
*END STEP
**
*END INPUT
```

Listing 6: Definition of the dynamic step for the compression wave propagation using `u4p4u4-sat`, `u8p4u8-sat`, `u8p4u8-sat-red` and `u8p8u8-sat` elements

For the **u**-p-**U** elements the pore water pressure can be obtained from the bottom node, i.e. the same print output definition as for the **u**-p elements is used.

1.4 Input file for the 3D simulations using the `u8u8-3D-red`, `u20p8-sat`, `u20u20`, `u27p8-sat` and `u20p8u20-sat` elements

For the 3D elements only the boundary conditions have to be modified. We refer to the supplied input files for the necessary adjustments. Note that z-axis is the vertical direction for the 3D input files, for why the definitions of loading and boundary conditions differ to the 2D input files.

1.5 Results

The results in terms of excess pore water pressure at the bottom of the column and settlement of the column evaluated at its top are given in Figs. 2, 3 and 4 as comparison with the results of the analytical approach. As a result of the frequency of approximately 30 Hz of the propagating wave as well as the relatively high hydraulic conductivity ($k^w = 10^{-3}$ m/s), the **u**-p elements show an increasing divergence from the analytical results with ongoing wave propagation. The elements based on the **u**-p-**U** and **u**-**U** element formulation fit well to the analytical results for the entire time period. The relative acceleration is therefore important to be taken into account. Even the linear-interpolated **u**-**U** elements (`u4u4`, `u4u4-red`, `u8u8-3D-red`) are in good accordance with the analytical results despite the lower number of nodes and integration points (the same number of elements has been used for all simulations). The reduced integrated `u4u4-red` element performs slightly better than the fully integrated `u4u4` element which can be traced back to volumetric locking effects for the fully integrated element.

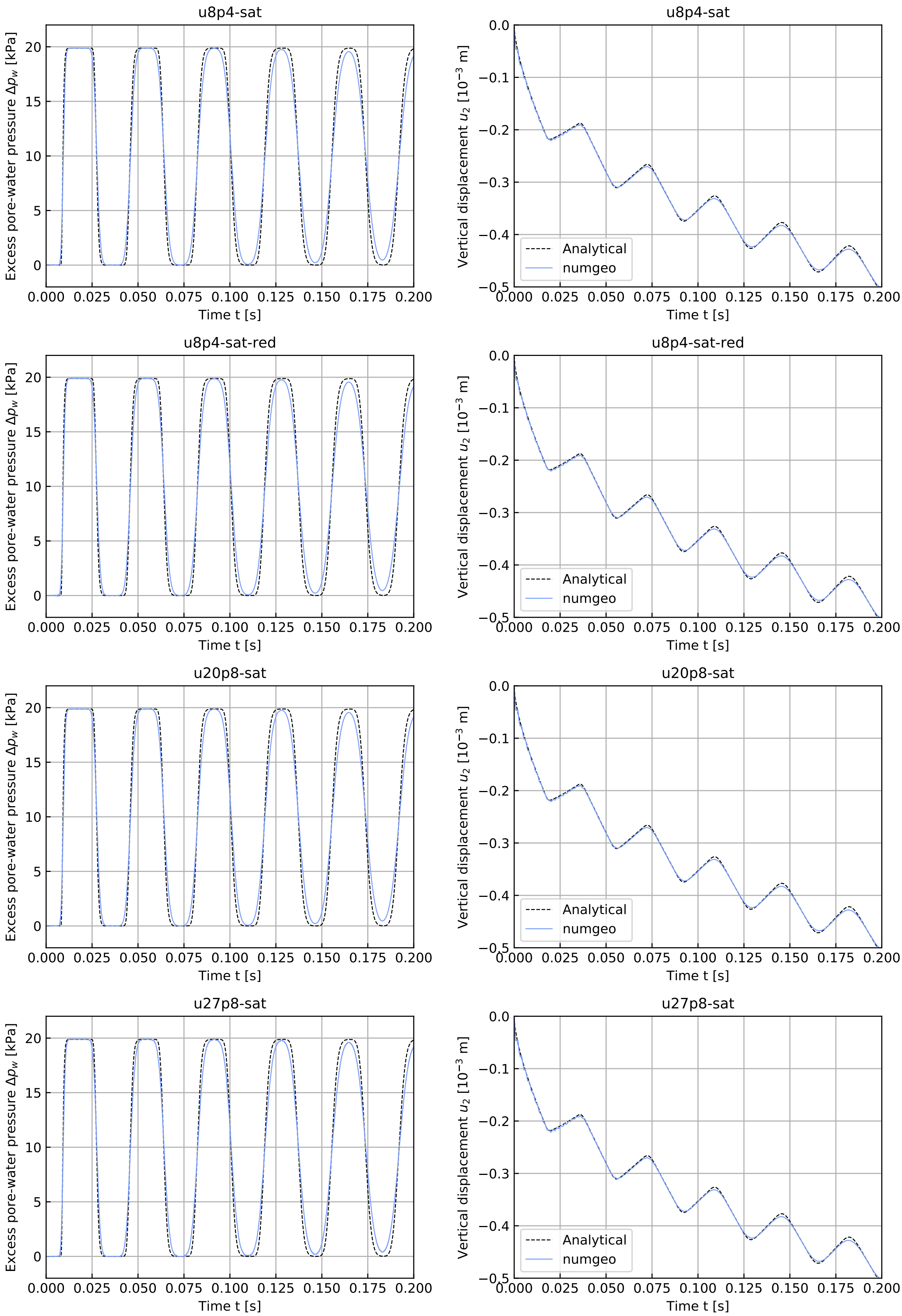

Figure 2: Results in terms of excess pore water pressure at the bottom of the column and vertical displacement at the top using the various **u**-p elements

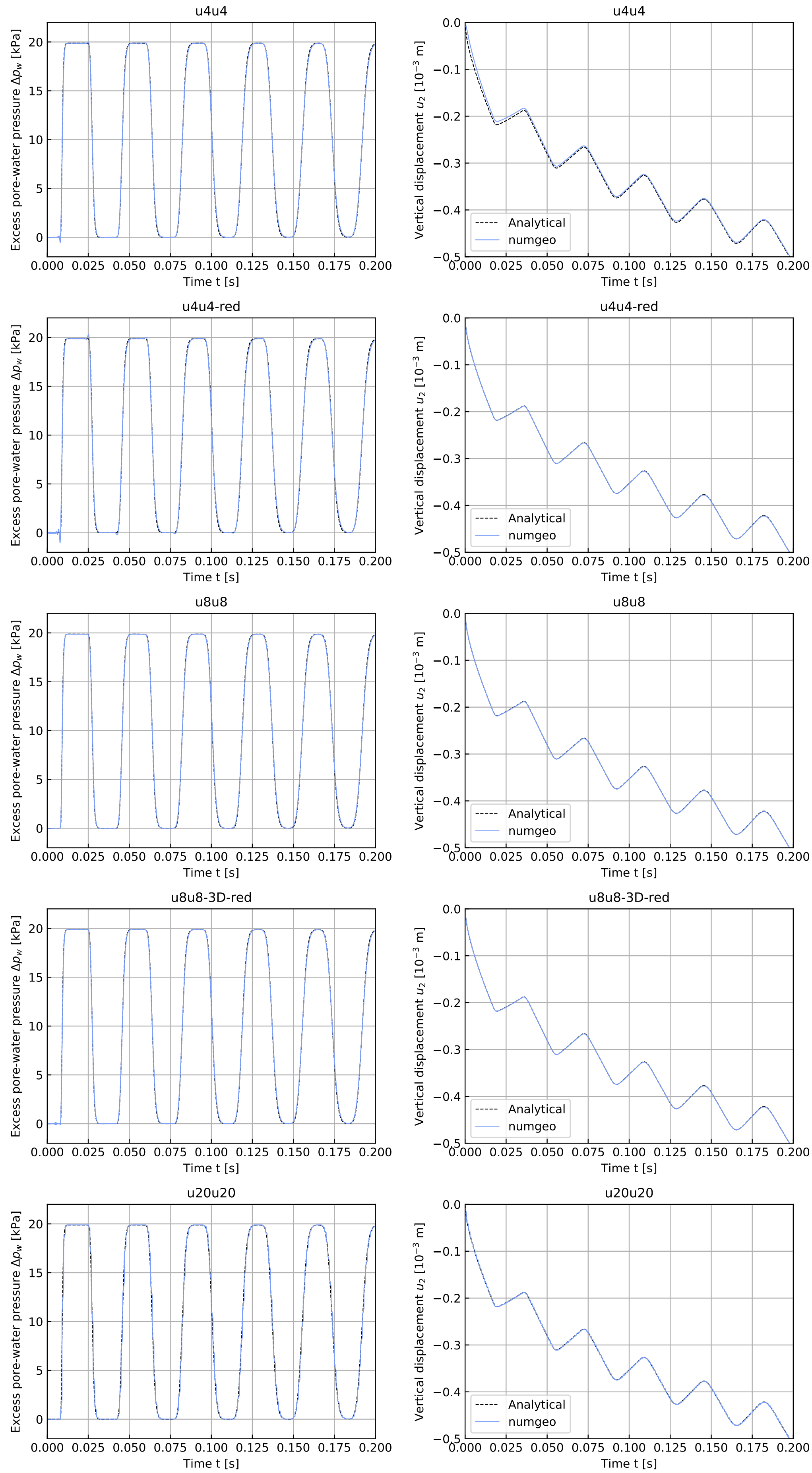

Figure 3: Results in terms of excess pore water pressure at the bottom of the column and vertical displacement at the top using the various **u-U** elements

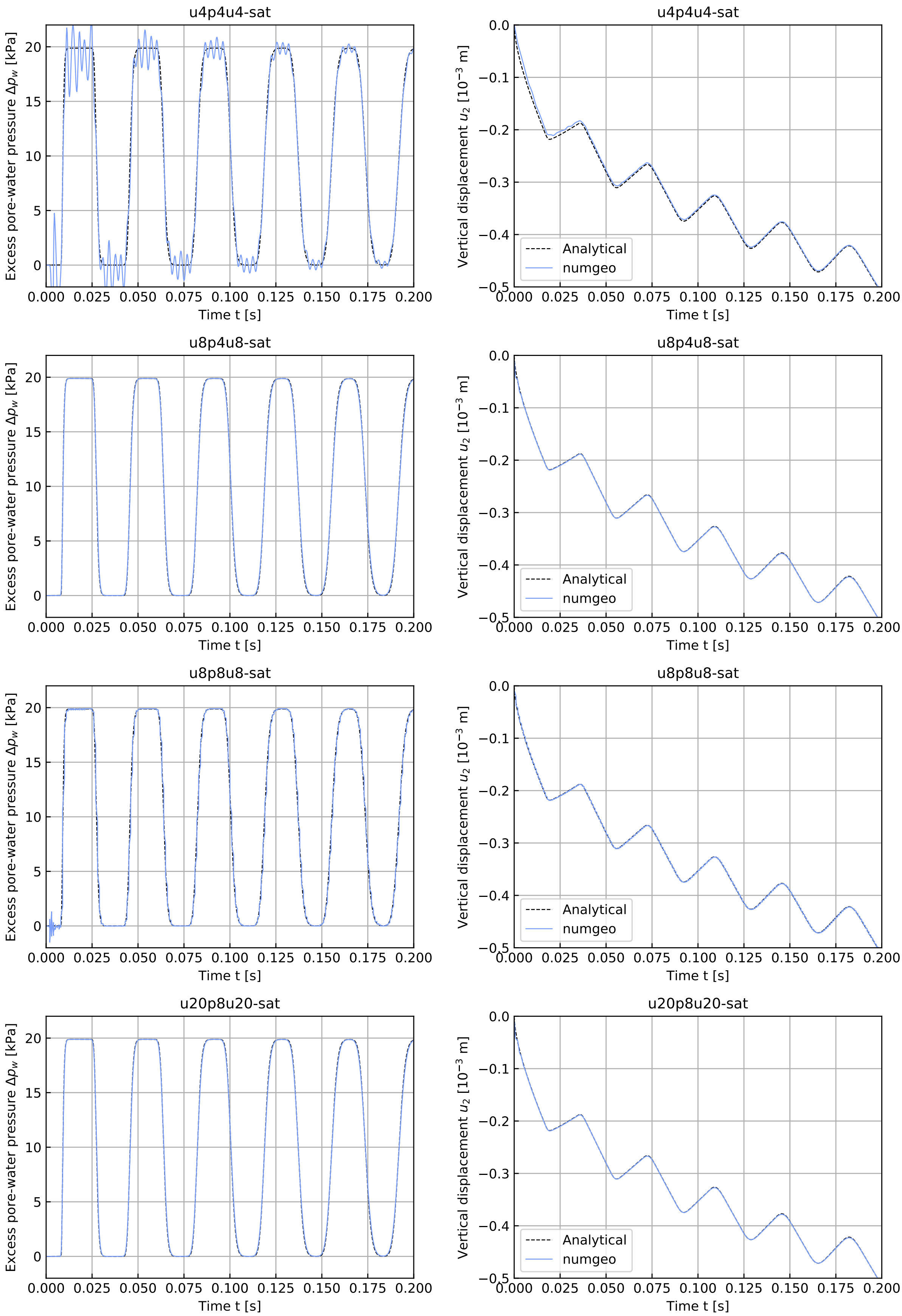

Figure 4: Results in terms of excess pore water pressure at the bottom of the column and vertical displacement at the top using the various **u**-p-**U** elements

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.