Hydro-mechanically coupled explicit analysis using Abaqus

Patrick Staubach*

March 3, 2024

Version		
02.2021	:	Initial version
12.2021	:	Added description of `VUMAT` routine for Abaqus 2020 and newer. Added warning for usage of large bulk moduli. Added additional output
03.2022	:	Added information about defining the hydrostatic pore water pressure for the `VUMAT` routine for Abaqus 2020
11.2023	:	Calculation of correct friction with field variables
12.2023	:	Added warning if too many elements are considered in subroutine `VUSDFLD`

1 Theory

This document explains how to conduct a hydro-mechanically coupled explicit simulation using Abaqus according to the approach documented in [2, 3] (note that the approach is based on the work by Hamann [1]). The theory is explained first, the necessary settings in the input file are introduced subsequently.

The balance of mass of the pore water is given by:

$$\varphi^w \frac{1}{\bar{K}^w} \frac{dp^w}{dt} + \varphi^w \mathrm{div}(\boldsymbol{v}^w) + (1-n)\ \mathrm{div}(\boldsymbol{v}^s) = 0 \tag{1}$$

The generalized Darcy law is used to evaluate the relative velocity between solid skeleton and pore water which is derived from the linear momentum of the pore water:

$$\boldsymbol{w}^w = \frac{\boldsymbol{K}^{\mathrm{Perm}}}{\eta^w} \cdot \left(-\mathrm{grad}(p^w) + \bar{\rho}^w \left(\boldsymbol{b} - \ddot{\boldsymbol{u}}\right)\right) \tag{2}$$

Combined these equations yield:

$$\varphi^w \frac{1}{\bar{K}^w} \frac{dp^w}{dt} - \frac{\boldsymbol{K}^{\mathrm{Perm}}}{\eta^w} \cdot \mathrm{div}\left(-\mathrm{grad}(p^w) + \bar{\rho}^w \left(\boldsymbol{b} - \ddot{\boldsymbol{u}}\right)\right) - \mathrm{div}(\boldsymbol{v}^s) = 0 \tag{3}$$

The energy balance is:

$$\rho c \dot{U} + k\ \mathrm{div}\Big(\mathrm{grad}(\theta)\Big) = -m_T \tag{4}$$

Combining Equation 3 and 4 one obtains:

$$c = \varphi^w \frac{1}{\bar{K}^w \rho} \tag{5}$$

$$k\ \mathrm{div}\Big(\mathrm{grad}(\theta)\Big) = \frac{\boldsymbol{K}^{\mathrm{Perm}}}{\eta^w} \cdot \mathrm{div}\Big(-\mathrm{grad}(p^w) + \bar{\rho}^w \left(\boldsymbol{b}\right)\Big) \tag{6}$$

$$m_T = -\mathrm{div}(\boldsymbol{v}^s) + \frac{\boldsymbol{K}^{\mathrm{Perm}}}{\eta^w} \mathrm{div}(\ddot{\boldsymbol{u}}) \tag{7}$$

*Chair of Geotechnics, Bauhaus-University Weimar, Germany / Ruhr-University Bochum, Germany. Email: patrick.staubach@uni-weimar.de

2 Definition of the input file

A coupled-thermal analysis with Abaqus/Explicit is performed, where the specific heat and the thermal conductivity are modified in such a way that the mass balance of pore water is derived. The constant c is given in the input file by:

```
*SPECIFIC HEAT
1.03744454d-7
```

where $c = \frac{\varphi^w}{\bar{K}^w \rho} = \frac{0.48}{2200000 \cdot 1.884} = 1.03744454^{-7}$ was set exemplary. φ^w is the porosity, $\bar{K}^w$ the bulk modulus of water and ρ the total density.

It is not recommended to use the bulk modulus of "pure" water in general. This will lead to very small time increments. Better consider some trapped air bubbles in the soil, which will reduce the bulk modulus considerably.

k is given by using:

```
*CONDUCTIVITY, TYPE=ISO
1d-4
```

where $\frac{K^{\text{Perm}}}{\eta^w} = \frac{10^{-10}}{10^{-6}}$ was set exemplary.

In addition, elements with thermal degree of freedom have to be used, e.g. CPE4RT (Lagrangian analysis) or EC3D8RT (Eulerian analysis) and

```
*INELASTIC HEAT FRACTION
1.0
```

has to be defined. Using the `VUMAT` together with the hypoplastic constitutive model, the following additional definitions are necessary

```
*Depvar
35
*User Material, constants=15, unsymm
0.578,      0.37,      4e+06,      0.27,      0.677,      1.054,      1.212,      0.14
2.5,        1.2,       2.4,       0.0001,     0.1,        6.,         0.
```

where the parameters for Karlsruhe fine sand have been used. The initial state variables have to initialized as well:

```
*Initial conditions,type=solution
Part-1-1.all,1.0d0,0,0,-0.0001d0,0,0,0
0,0,0,0,0,0,0,0,0
0,0,0,0,0,0,0,0,0
0,0,0,0,0,0,0,0,0
0,0,0,0,0
```

The element set *Part-1-1.all* has an uniform initial void ratio of 1 and a fully mobilized intergranular strain in z-direction of -0.0001.

In order to conduct a fully coupled dynamic analysis, the following type of analysis has to be defined in the step definition:

```
*Dynamic TEMPERATURE-DISPLACEMENT, explicit
```

Within the `VUMAT`, the internal energy m_T caused by inelastic heat generation is used to add the volumetric terms of the actual mass balance to the energy balance Abaqus enforces. As Abaqus requires the dissipated energy per unit mass, the terms are divided by the density of the mixture. As only the

excess pore water pressure is described, the hydrostatic pore water pressure has to be taken into account separately. The definition of the ground water table is discussed in the following.

Two `VUMAT` routines are supplied: one for all Abaqus versions older then Abaqus 2020 and one for Abaqus 2020 and newer. You need only one of them.

Within the `VUMAT` for Abaqus 2020 and newer, the following additional parameters have to be set:

```
KPerm       = 1.00d-10 ! permeability
viscosity   = 1.00d-6  ! dynamic viscosity
density2    = 1.884d0  ! density
gamma_w     = 10.0d0   ! dead weight water
gravity     = 10.0d0   ! gravity
KO          = 0.5d0    ! lateral stress coefficient
water_table = 10.0d0   ! water table height
dir_grav    = 2        ! direction of gravity
cavitation  = -100.0d0 ! Total pore water pressure at which cavitation occurs
```

In addition, one has to make sure that the direction of gravity used for the calculation of the hydrostatic pore water pressure in line 155 or line 161

```
statev(24) = (-coords(int(dir_grav)) + water_table)*gamma_w
```

is correct for the numerical model used.

Opposite to that, the `VUMAT` for older Abaqus versions needs only the following settings:

```
KPerm       = 1.00d-10! permeability
viscosity   = 1.00d-6 ! dynamic viscosity
density2    = 1.884d0 ! density
gamma_w     = 10.0d0  ! dead weight water
gravity     = 10.0d0  ! gravity
KO          = 0.5d0   ! lateral stress coefficient
```

This is because for the `VUMAT` for older versions, the initial pore water pressure is calculated based on the given stress conditions. However, the `VUMAT` version for Abaqus 2020 and newer should generally be preferred since it allows to consider the hydrostatic pore water pressure as constant with respect to the global coordinates.

In the `VUMAT` for older versions, the value of hydrostatic pore water pressure moves with the material. The excess pore water pressure can be viewed in the odb using:

```
*Node Output
NT
```

In addition, the hydrostatic pore water pressure is obtained using:

```
*Element Output
SDV24
```

The effective stresses are obtained by:

```
*Element Output
SDV27,SDV28,SDV29,SDV30,SDV31,SDV32
```

The simulation has to be started using `user=call` or `user=call2020`, depending on the `VUMAT` version used.

3 Calculation of friction with effective contact stresses

Two additional user files are required if effective contact stresses are used to calculate friction: `VUFIELD` and `VUSDFLD`.

3.1 Statements in the input file

The following additional considerations in the material definition are required:

```
*Depvar
36
*USER DEFINED FIELD
```

When defining the interaction behavior, the following definition of the friction coefficient is required:

```
*Friction ,DEPENDENCIES = 1
0.0, , , ,0
0.25, , , ,0.5
```

Here, the 'real' friction coefficient is 0.25, i.e. in the simulation this friction coefficient is active.

For all steps, the following commands have to be added:

```
*field ,user
*field ,user, NUMBER =2
Set-euler_all
```

Here, *Set-euler_all* is a node set containing all nodes of the eulerian region.

To check if the field variable used to calculate the ratio of effective to total contact stresses is calculated correctly, the field variable is added to the output:

```
*Element Output
FV
```

3.2 Statements in the user files

Additional considerations are required for the files `VUFIELD` and `VUSDFLD`. At the start of `VUSDFLD`, the following variables have to be set:

```
pile_radius       = 0.0165d0
min_coords3       = 0.5d0
pile_center(1:2) = [0.0d0,0.0d0]
```

By default, a pile driving analyses is considered. To save time, only in a zone 1.4 times the pile radius, the field variable is assigned. In addition, for z-coordinates lower than *min_coords3* the field variable is not assigned. Both is done to save time and can be modified for other problem positions. The variable *pile_center* denotes the central axis of the pile, to which all normal vectors of the pile shaft are assumed to point.

At the moment, the files `VUFIELD` and `VUSDFLD` shipped together with this document only support analyses with a maximum of 6 threads.

At the start of `VUFIELD`, the following variables have to be set:

```
pile_radius       = 0.0165d0
min_coords3       = 0.5d0
pile_center(1:2) = [0.0d0,0.0d0]
frequency_update = 10
bytes_per_line    = 62
```

In addition to the aforementioned variables, *frequency_update* and *bytes_per_line* have to be set. *frequency_update* denotes the frequency with which the field variable is updated. To save time, the field variable is only updated every 10th increment. Because there seems to be a general problem with reading files with Fortran within subroutines of Abaqus, *bytes_per_line*, setting the number of bytes

of one line of the files that the `VUSDFLD` routine generates is required. As this number stays constant for all analyses, it is not required to be modified. However, one should check that the correct number of lines is read.

Independent of the number of threads used, Abaqus crashes if the files written by the user routines contain to many elements. A critical number was found to be around 5000. If your simulation crashes and reports a dumped core in the main executable, reduce the number of elements included in the region of the model considered by the subroutines, e.g. reduce the multiplier of the pile radius.

3.3 Benchmark

In order to show how field variables can be used to modify the friction coefficient, a large scale interface shear test is simulated using the CEL method. The wall is meshed with Lagrangian elements and the soil with Eulerian elements. Only elastic materials and no influence from pore fluid pressure is considered. After establishing the normal contact stresses within the first 4 s of the simulation, the wall is moved downwards with a constant velocity. During this, the field variable is changed such that the friction coefficient changes with time. Two different functions for the field variable are considered, which are depicted in Fig. 1. As can be seen from Fig. 1 the field variable gives the correct ratio of friction forces to normal forces by means of the friction coefficient.

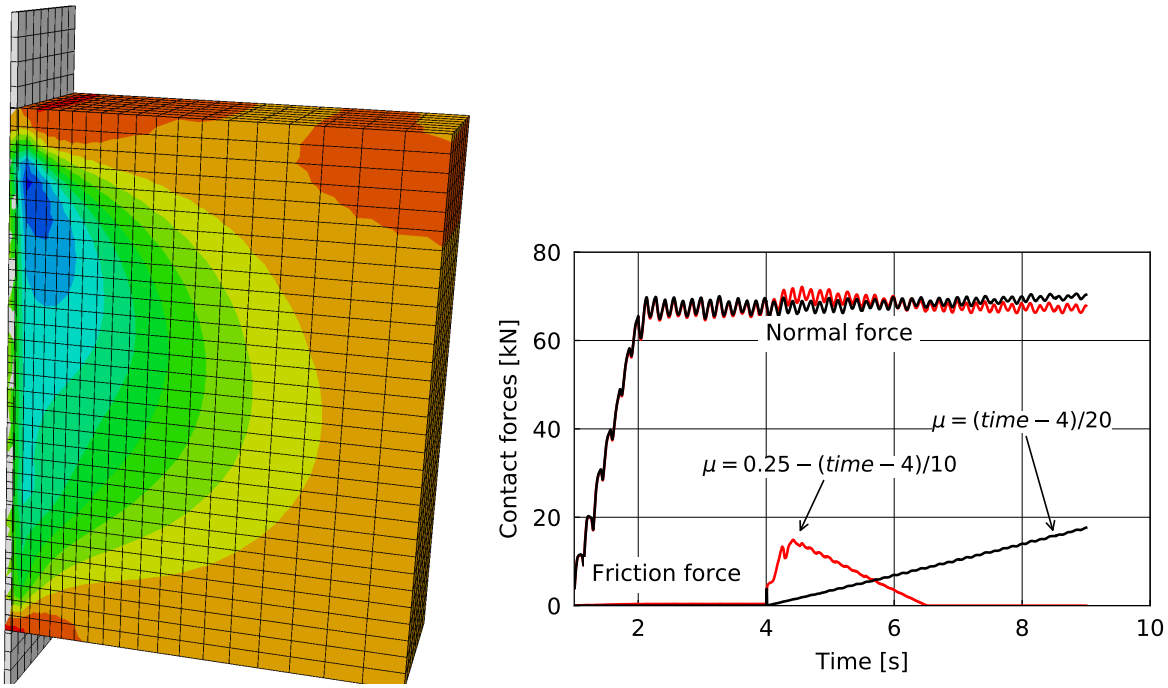

Figure 1: Interface shear test

References

[1] T. Hamann. “Zur Modellierung wassergesättigter Böden unter dynamischer Belastung und großen Bodenverformungen am Beispiel der Pfahleinbringung”. Publications of the Institute of Geotechnical Engineering and Construction Management, University of Hamburg, 2015.

[2] P. Staubach, J. Machaček, M. C. Moscoso, and T. Wichtmann. “Impact of the installation on the long-term cyclic behaviour of piles in sand: A numerical study”. In: *Soil Dynamics and Earthquake Engineering* 138 (2020), p. 106223. ISSN: 02677261. DOI: 10.1016/j.soildyn.2020.106223.

[3] P. Staubach, J. Machaček, J. Skowronek, and T. Wichtmann. "Vibratory pile driving in water-saturated sand: Back-analysis of model tests using a hydro-mechanically coupled CEL method". In: *Soils and Foundations* 61 (1 Feb. 2021), pp. 144–159. ISSN: 00380806. DOI: 10.1016/j.sandf.2020.11.005.