1 Dimensional Consolidation

Arul M Britto

October 10, 2013

Contents

1 Introduction

ABAQUS is a general purpose finite element program. It has a large element library and is capable of analysis of a variety of problems. This is a compilation of ABAQUS 6.9 examples which are used to demonstrate the capabilities of the ABAQUS program. The examples given here are of an advanced nature illustrating the type of analyses carried out at Cambridge University.

This document is aimed at user's who already have a basic knowledge of ABAQUS 6.9 and now want to use it for more advanced analysis.

The following example problem is considered here.

- 1 Dimensional Consolidation

2 Problem Definition

Figure 1 shows the problem to be analysed. It is a column of saturated soil of depth 10 m. The water table is at the top in line with the ground level. The column of soil is assumed to be in equilibrium under self weight and a surcharge of 10 kPa. This analysis is to calculate the final settlement under a surcharge increase of 100 kPa (giving a total surcharge of 110 kPa). The base of the soil column is assumed to be impervious. The water table is to be maintained at the top of the soil. This does not mean that it is maintained at the original level but keeps in line with the top of the soil as it settles. Hence the long term pore pressure at the top is assumed to be zero.

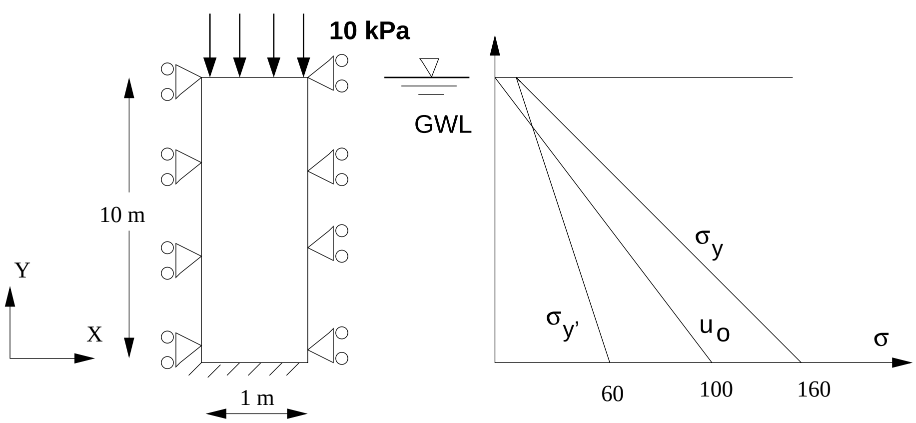

Figure 1: Column of Soil

3 Units

The first task to do is to decide on the units to be used in the analysis. ABAQUS does not have any inherent set of units. The units for Length (L), Force and time have to be decided. The convenient use for Length is Metres. Force is assumed to be in KN (Kilo-Newtons) and the time seconds. The rest of the units are then derived from these.

The units used in the analysis are as follows :

Length	Force	Stress	Time	k_x, k_y	γ_{bulk}	γ_{water}
m	kN	kPa	Sec	(m/s)	kN/m^3	kN/m^3

4 Material Properties

The original Cam Clay model was the basis of the various critical state models (Roscoe, Schofield & Thurairajah (1963), Schofield & Wroth (1968), Schofield (1980)). The historic development of the original Cam Clay model is described by the Principal Architect in a recent book (Schofield, 2005).

In the example considered here the soil is to be modelled using the Modified Cam Clay model (Roscoe and Burland, 1968).

There are other books which also covers critical state soils mechanics concepts (Atkinson & Bransby, 1978, Atkinson, 1981, Bolton, 2000, Wood, 1990, Powrie, 2004).

The soil parameters for this model are as follows :

M	κ	λ	e_{cs}	N	ν	k_x	k_y	γ_{bulk}	γ_{water}
						(m/s)	(m/s)	kN/m^3	kN/m^3
1	0.1	0.5	3.0	4.0	0.3	0.1157E-7	0.1157E-7	15	10

where N = 1 + e_{cs}
M - Critical state ratio
κ - Logarithmic elastic bulk modulus
λ - Logarithmic hardening modulus
ν - Poisson's ratio
k_x, k_y - Permeability (horizontal, vertical respectively, in m/s)
γ_{bulk} - bulk unit weight of soil (kN/m^3)
γ_{water} - unit weight of water (kN/m^3)

Pt	Y	σ'_x	σ'_y	σ'_z	u	p'_c	a_o
	(m)	(kPa)	(kPa)	(kPa)	(kPa)	(kPa)	(kPa)
1	10.	-6.	-10.	-6.	0.	10.	5.
2	0.	-34.	-60.	-34.	100.	58.	29.

Point 1 and 2 represent the initial stresses at the top and bottom of the mesh respectively.
σ'_x - Initial effective horizontal stress (kPa)
σ'_y - Initial effective vertical stress (kPa)
σ'_z - Initial effective out-of-plane stress (kPa)
u - Initial pore pressure
p'_c - Size of the initial yield locus (Cambridge Convention)
a_o - Initial overconsolidation parameter = 0.5 * p'_c

The variation of these stresses and parameters with depth are assumed to be linear.

The notation used here is taken from the book Britto & Gunn (1987) with the exception of specific volume (V = 1 + e) being replaced by voids ratio (e). This is because the voids ratio (e) is readily available as a variable (VOIDR) in ABAQUS.

5 Finite Element Mesh

The width of the soil column was assumed to be 1 m. It was decided to model the soil column with 20 8-noded CPE8P elements of equal height. It is treated as a **plane strain** problem.

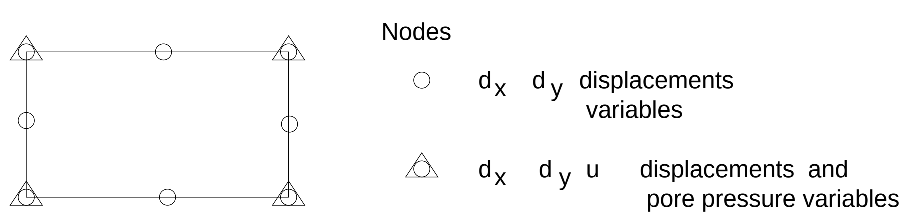

Figure 2: CPE8P Element

As can be seen (Figure 2) only the corner nodes have pore pressure as a variable.

```
*NODE
  1,  0.0,   0.
  2,  0.5,   0.
  3,  1.0,   0.
401,  0.0,  10.
402,  0.5,  10.
403,  1.0,  10.
*********************
```

The nodes along the base and the top are specified first.

Various node and element sets required for specifying the properties and boundary conditions are specified next. All the elements are grouped together into a set called SOIL.

The node set NALL includes all the nodes in the mesh. Node set NALLC includes all the corner nodes ie which have pore pressure as a variable. This enables one to specify the initial pore pressure distribution.

```
*NGEN, NSET=NALL
  1,  401, 10
  2,  402, 20
  3,  403, 10
*NSET, NSET=NALLC, GENERATE
  1,  401, 20
  3,  403, 20
**---------------------
*NSET, NSET=BASE
 1, 2, 3
*NSET, NSET=BASEC
 1, 3
**---------------------
*NSET, NSET=TOP
 401, 402, 403
*NSET, NSET=TOPC
 401, 403
**---------------------
*NSET, NSET=VERBND
   1, 401,  10
   3, 403,  10
**---------------------
```

TOP and BASE refers to the displacement nodes at the top and base of the mesh respectively. TOPC and BASEC refers to the nodes with pore pressure as a variable at the same two boundaries. The only difference between these and the previous sets is the mid-side nodes are missing from the latter group.

The final nodeset VERBND refers to the nodes along the two vertical sides which needs fixing in the horizontal direction.

The first element is specified first as the master element. Then *ELGEN command is used to create the complete element set. All the elements are grouped together into a set called SOIL.

The *SOLID SECTION defines the section and links the element set to the (yet to be specified) Material properties A1.

```
*ELEMENT, TYPE=CPE8P, ELSET=ONE
 1,  1,  3,  23,  21,  2,  13,  22,  11
*ELGEN, ELSET=SOIL
 1, 20, 20
```

```
*ELSET, ELSET=TOP
  20
**----------------------------------------
*SOLID SECTION, MATERIAL=A1, ELSET=SOIL
```

Figure 3 shows the mesh used for the analysis.

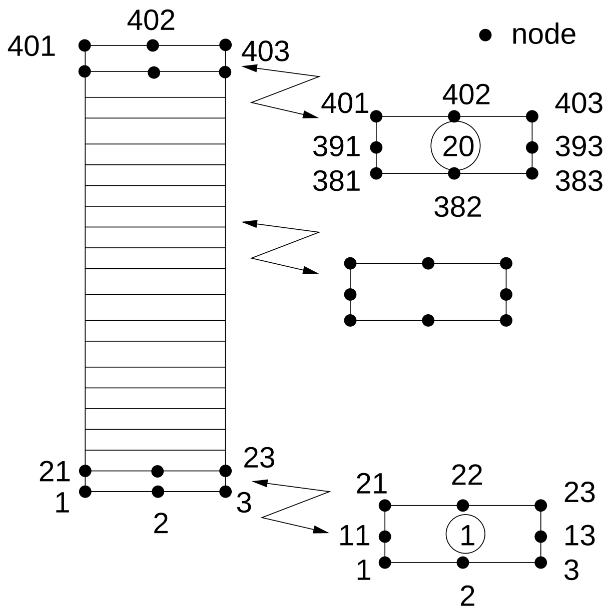

Figure 3: F E Mesh

6 Initial stresses

The Modified Cam Clay model as implemented in ABAQUS is more generalised than the model originally proposed by the Cambridge Soil Mechanics Group (MCC-CSSM) and which can be derived as a special case [2].

First of all to let us try to clear up any misunderstanding due to the use of different notations and parameter definition.

p'_c is used to specify the size of the yield locus in reference 1. This is the value of p′ in the Isotropically normally consolidated line (INCL) which is also known as the virgin consolidation line. Whereas in ABAQUS **a** is used to denote the size of the yield locus but is taken as the value at the critical state. The initial size of the yield locus is denoted by $a_o = \frac{1}{2}(p'_c)$ for the Modified Cam Clay model. This is illustrated in Figure 4.

e_1 is the intercept on the virgin consolidation line. This is not the same as e_{cs} but is related through the following equation :

$$e_1 = e_{cs} + (\lambda - \kappa)ln(2)$$

Substituting for κ, λ and e_{cs} this gives e_1 = 3.2773.

Figure 5 shows the plot in e - ln(p′) space and how the above equation was derived.

As is customary the bulk unit weight of soil (γ_{bulk}) was assumed to be a constant at 15. This is probably an oversimplification and the implication of this will be explored later. However for the initial analysis this is assumed to hold.

```
*MATERIAL, NAME=A1, ELSET=SOIL
```

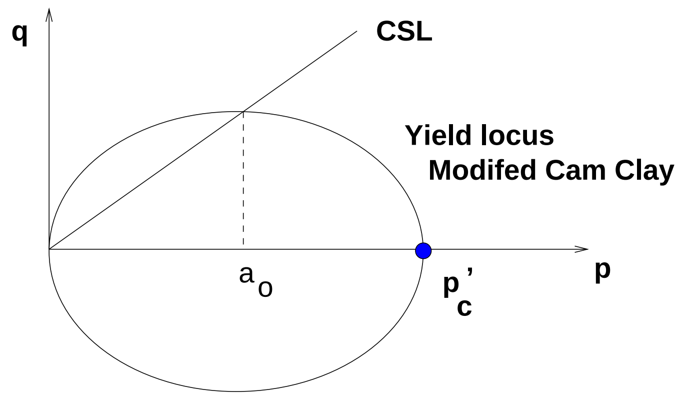

Figure 4: The yield locus for Modified Cam Clay

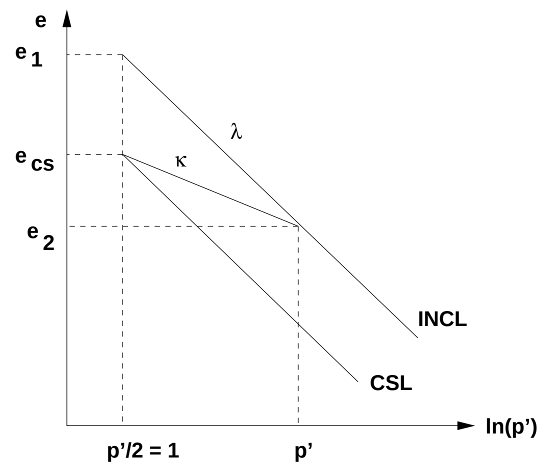

Figure 5: Relationship between e_1 and e_{cs}

```
*POROUS ELASTIC
** kappa v
  0.1,  0.3
```

The above statements specify the κ and ν (Poissons ratio) respectively.

```
*CLAY PLASTICITY, DEPENDENCIES=1
** Lam  M    ao   beta  K  T  f1
  0.5,  1.,  29.,  1.,  1., ,  0.
  0.5,  1.,   5.,  1.,  1., , 10.
```

The above statements are the first attempt to specify the variation of p_c' (a_o) with depth. The values specified in each line are λ, M, a_o, β, K, the temperature (which is omitted) and the field variable. DEPENDENCIES=1 means there is only one field variable.

β - this is used to specify a different yield locus on the wet side. When the same ellipse is used for both dry and wet sides as in the original Cambridge formulation $\beta = 1$.

K - is the ratio of the flow stress in triaxial tension to the flow stress in triaxial compression. This determines the shape of the yield locus in Π plane (plane of the principal deviatoric stresses). K = 1 gives a circle as in the original Cambridge formulation.

It was later discovered that the variation of a_o cannot be specified using a field variable (Section 11.3.4, Users manual Vol.1 Ver 5.7). To quote : "a_o is a function only of the initial conditions; it will not change if temperatures and field variables change during the analysis." In a previous version of this document the author erroneoudly made the statement : "In preventing it from changing during the analysis as a function of field variable it also prevents the specification of it as a function of field variable at in situ stage. That comment was attributed to ABAQUS version 5.7 a decade ago. However with the latest version 6.9 it is allowed to change as a function of the field variable."

There was no restriction in ABAQUS but a misunderstanding on the author's part.

Having specified the variation of the ao value with Field variable 1 the correct way to achieve this variation is by including the following statements

```
*INITIAL CONDITIONS, TYPE=FIELD, VARIABLE=1
ALL, 29., 0., 5., 10.
```

The values following the node set are the size of the yield locus at the Y co-ordinate of 0. (base of the mesh) and a value of 5 the size of the yield locus at y = 10 which corresponds to the surface of the mesh.

This then allows for the correct interpolation of the size of the yield locus.

As an aternative to specifiyimg the variation of the size of yield locus one could specify the intercept of the critical state line. This is then used to calculate the size of the initial yield locus.

The above statements were then replaced with the following statements :

```
*CLAY PLASTICITY, INTERCEPT=3.2773
** Lam  M   ao   beta  K  T
  0.5,  1.,    ,  1.,   1., ,
```

Here the value of a_o is calculated from e_1, which is the intercept and hence the third value is left blank.

It is important to make sure that the initial stresses do not violate the yield condition ie stress state does not lie outside the yield locus. However ABAQUS will issue warning messages when this happens and will automatically adjust the a_o values when this happens.

In the Cambridge version of the Modified Cam Clay the initial yield locus was given by

$$q^2 = M^2 p'(p'_c - p') \tag{1}$$

Rewriting this

$$p'_c = \frac{q^2}{M^2 p'} + p' \tag{2}$$

Substituting for the initial stresses (p_o, q_o) p'_c is replaced by p'_y.

$$p'_y = \frac{q_o^2}{M^2 p'_o} + p'_o \tag{3}$$

Here p'_y represents the yield locus which passes through the initial stress state. Then the variation of p'_y can be plotted with depth and compared with the initial distribution of p'_c. If $p'_y \leq p'_c$ then the yield condition is satisfied everywhere initially. Figure 6 shows the initial stresses :

Figure shows the p'_y, p'_c and a_o :

It can be seen that the values of p'_y and p'_c are very close and that would indicate the initial stress state is very close to yielding for the full depth of the soil.

The permeability (assumed to be a constant here) and the specific weight of water are specified using :

```
*PERMEABILITY, SPECIFIC=10
 0.11574E-7,
```

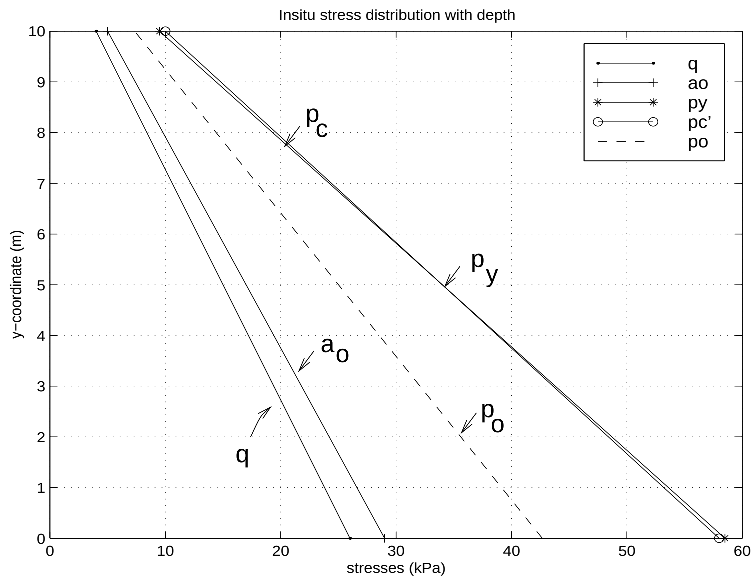

Figure 6: Initial stresses

Notice the comma after the value. If this is omitted the permeability will be interpreted as 0.11574 which would result in erroneous rapid dissipation of the pore pressures. Again this is attributed to ana old version of ABAQUS.

The variation of pore pressure is given as follows:

$$\frac{du_w}{dy} = -\gamma_w \tag{4}$$

Integrating this equation gives :

$$u_w = \gamma_w(y_w^o - y) \tag{5}$$

Here y_w^o represents the Y co-ordinate of the initial level of the Water Table.

The variation of the total vertical stress is given by :

$$\frac{d\sigma_{yy}}{dy} = (\rho g + sn^o \gamma_w) \tag{6}$$

Where :

s - saturation n - porosity ρg - dry density

The total vertical stress σ_{yy} must satisfy the above equation. If saturation is not specified by default it is assumed to be 1. Based on ρ, γ_w and n the variation of the vertical stress is calculated.

Substituting the following equation in equation 3:

Another potential source for confusion is the sign convention used in ABAQUS. As any student of Geotechnics (soil mechanics) would know that compressive stresses are positive and that when you add the effective stress to the pore pressure you get the Total stress. Whereas the following equation (9) states that the effective stress is equal to the total stress plus the the pore pressure.

Both these are equivalent if you take into account the sign convention used by ABAQUS. Consider the situation that the effective stress (compressive) is 30 kPa. The pore pressure is 10 kPa. This would give the total stress as 40 kPa. Considering it is fully saturated s = 1.

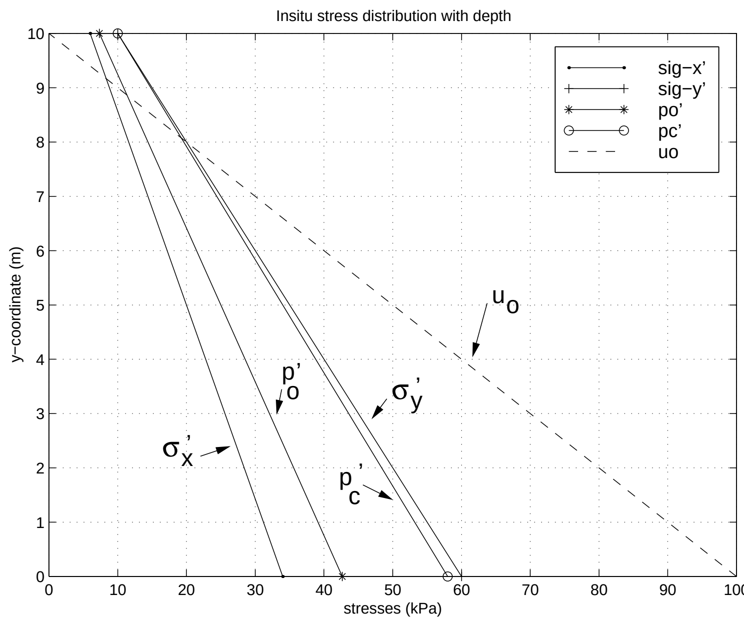

Figure 7: Initial stresses

One final word on pore pressure as to the question of whether u_w is the excess or total pressure is that it is Total pore pressure.

$$\sigma = \overline{\sigma} + u_w I \quad (Soil \;\; Mechanics \;\; Sign \;\; convention) \tag{7}$$

This is illustrated as 40 = 30 + 10

Now returning to ABAQUS sign convention the effective stress being compressive is -30 kPa. Pore pressure is 10 kPa. So the total stress is -40 kPa.

$$-30 = -40 + 10 = -30 \tag{8}$$

```
Or re-writing          -40   =   -30   - (10)
```

The other area of confusion is the interpretation of the output.

The stress quantities S11, S22 and S33 are the direct normal stresses σ'_{xx}, σ'_y and σ'_{zz} are also the EFFECTIVE stresses and not the Total stresses. The pore pressure has the label of POR in the output modules. The variable number for the pore pressure is 8.

Therefore users who require the total stress need to subtract the POR value from the effective stress components (S11, S22 and S33). This is using ABAQUS convention (in contrast to the normal soil mechanics convention of adding the pore pressures to the effective stress).

$$\overline{\sigma} = \sigma + s u_w I \qquad (ABAQUS \;\; Sign \;\; Convention) \tag{9}$$

yields :

$$\frac{d\overline{\sigma}_{yy}}{dy} = \rho g - \gamma_w [s(1 - n^o) - \frac{ds}{dy}(y_w^o - y^o)] \tag{10}$$

Integrating this equation, the effective vertical stress is given by

$$\overline{\sigma}_{yy} = [\rho g - \gamma_w s(1 - n^o)]\,(y - y^o) \tag{11}$$

For fully saturated soils s = 1. Substituting

$$n^o = \frac{e_o}{(1 + e^o)} \tag{12}$$

leads to the equation

$$\overline{\sigma}_{yy} = \left(\rho g - \frac{\gamma_w}{(1 + e_o)}\right)(y - y^o) \tag{13}$$

The relationship between γ_{bulk} and γ_{dry} is derived as follows :

$$\gamma_{bulk} = \frac{(G_s + e)\gamma_w}{1 + e} \tag{14}$$

$$\rho g = \gamma_{dry} = \frac{G_s \gamma_w}{1 + e} \tag{15}$$

$$\rho = \frac{\gamma_{dry}}{g} = \frac{G_s}{1 + e}\frac{\gamma_w}{g} \tag{16}$$

It is ρ which is specified as DENSITY.

$$\gamma_{bulk} = \gamma_{dry}(1 + mc) \tag{17}$$

where **mc** is the moisture content (= wc - water content).

$$mc = \frac{e}{G_s} \tag{18}$$

$$\rho g = \gamma_{dry} \tag{19}$$

Over a range of great depth the simplifying assumption that γ_{bulk} is constant does not hold. The values ρ which is a constant is specified using the *DENSITY keyword. of the soil column and is specified as follows :

```
**
*DENSITY
  0.88

**
USER OUTPUT VARIABLES
  5
```

The above statements allocate storage space for user requested output parameters and this has to be included as part of the material properties. Here five output parameters are allocated space. More on this later.

$$a_o = \frac{1}{2} exp\left(\frac{e_1 - e_o - \kappa ln(p'_o)}{\lambda - \kappa}\right) \tag{20}$$

The above equation relates the initial mean normal stress p_o to the initial voids ratio e_o. Since the variation of the former is known it is possible to plot the variation of the initial voids ratio with depth.

This is derived as follows : In Figure 8, in the e - ln(p′) space between A and B

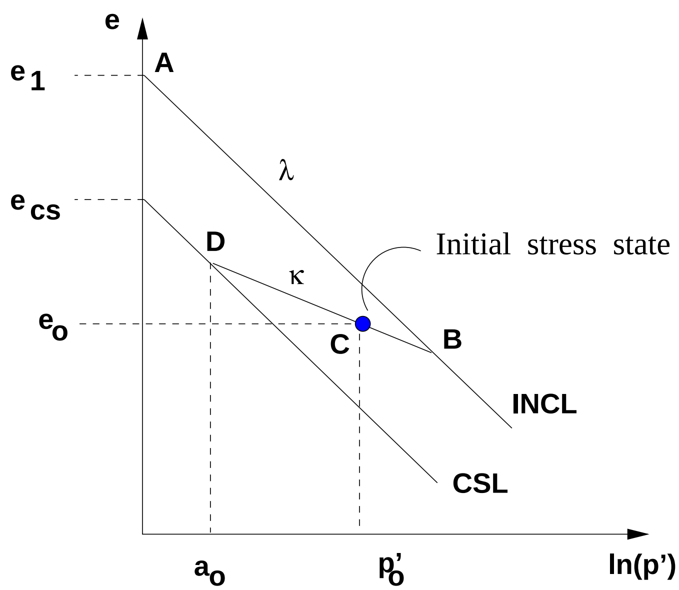

Figure 8: Calculation of the Initial voids ratio

$$e_B = e_1 - \lambda ln(p'_c) \tag{21}$$

Between B and C

$$e_o = e_B + \kappa ln(p'_c)/(p'_o) \tag{22}$$

Substituting for e_B gives

$$e_o = e_1 - \lambda ln(p'_c) + \kappa ln(p'_c)/(p'_o) \tag{23}$$

$$(\lambda - \kappa)(ln(p'_c)) = e_1 - e_o - \kappa ln(p'_o) \tag{24}$$

For Modified Cam-Clay p'_c = 2 a_o. Making this substitution

$$(\lambda - \kappa)ln(2a_o)) = e_1 - e_o - \kappa ln(p'_o) \tag{25}$$

$$a_o = \frac{1}{2}exp\left(\frac{e_1 - e_o - \kappa ln(p'_o)}{\lambda - \kappa}\right) \tag{26}$$

There is a unique relationship between p_o e_o and a_o. One way to specify the variation of a_o is to specify p_o and e_o. This would mean specification of the corresponding initial stresses with depth (σ'_x, σ'_y and σ'_z to match p_o).

```
*INITIAL CONDITION, TYPE=RATIO
NALL, 1.277, 0., 2.157, 10.
```

Here the voids ratio is assumed to vary linearly with depth as a first approximation. Spot values of the voids ratio was calculated at the top and base of the mesh from the stresses at these levels.

The initial stresses are specified as follows (to all the elements in the mesh) :

```
*INITIAL CONDITION, TYPE=STRESS, GEOSTATIC
SOIL, -60., 0., -10., 10., 0.6, 0.6
```

The vertical effective stress is -10 at the top (y = 10) and is -60 at the base (y = 0). The stress ratio K_o = 0.6 for both x and z directions.

The initial pore pressure distribution is specified as follows (to all the nodes which have pore pressure as a variable) :

```
*INITIAL CONDITION, TYPE=PORE PRESSURE
NALLC, 100., 0.,  0., 10.
```

The fixity boundary conditions follow next :

```
*BOUNDARY
BASE, 1,2
VERBND, 1
 401, 8
 403, 8
```

Here the boundary conditions specified above are fixities which remain the same throughout the analysis. Hence these are specified **before** the first step.

This way these need not be specified again in any of the steps.

The nodes at the base of the mesh (BASE) are fixed in both x and y directions. The nodes along the two vertical sides of the mesh (VERBND) are prevented from moving in the x direction. In addition the pore pressure at the top is fixed to be zero throughout the analysis. It is the **total** pore pressure which is specified here. Notice here that in specifiying these boundary conditions the option of not specifying zero value has been exercised.

7 Geostatic step

The first step in analysing geotechnical problems with initial stresses is to carry out a geostatic step. The purpose of this is to ensure that equilibrium is satisfied before the commencement of the analysis proper. The loading which is in effect at in situ stage is specified during this step. For this example it is the surcharge of 10 kPa and the self weight loading under earth's gravity for the soil column.

Elements which are not present at in situ stage can be removed in the geostatic step using

```
*MODEL CHANGE, REMOVE
<element-set-a>
```

Then these elements will not be considered in the calculation of equilibrium stresses.

This is not relevant for this example. It will be useful for example modelling the tunnel lining where the modelling of the excavation of the soil and its replacement in the region occupied by the lining

In the step where the soil is removed include the following statements :

```
*MODEL CHANGE, REMOVE
<element-set-b>
```

In the step where the lining is to be added, include the following statements :

```
*MODEL CHANGE, ADD
<element-set-a>
```

As an alternative to applying the surcharge one could have used elements (for example representing an embankment) to simulate the surcharge. In that case these elements will be removed in the geostatic step and then added in the step where the surcharge is being applied. One of the examples (the fourth set of data given in the associated webpage) uses the embankment construction approach.

Given below is the Geostatic step.

```
**--------------------------------
*RESTART, WRITE, FREQUENCY=20
*STEP,NLGEOM
 ....   GEOSTATIC STEP
*GEOSTATIC
*FIELD,VARIABLE=1, USER
NALL
**
** surcharge load
**
*DLOAD
 20, P3, 10.
**
** Gravity load
**
*DLOAD
SOIL, GRAV,  10., 0., -1., 0.
**-------------------------------
*EL PRINT, ELSET=SOIL
S
*EL FILE, ELSET=SOIL
S
*NODE PRINT,NSET=NALL
U, POR
*NODE FILE,NSET=NALL
U, POR
*MONITOR, NODE=401, DOF=2
*END STEP
**-------------------------------
```

Notice the absence of a RESTART statement. If the analysis is not required to be restarted ie the complete analysis can be finished in a single run, then this statement can be omitted.

Then the restart files (*.stt and *.mdl) are not created conserving disk space.

```
*RESTART, WRITE, FREQUENCY=20
```

The initial stresses specified are assumed to be approximate by ABAQUS. It treats these as an initial guess in calculating the stresses which are in equilibrium. This means ABAQUS attempts to calculate the equilibrium stresses using the user specified stresses asd a first approximation.

If there is a mis-match between the loading (self weight, surcharge - the relevant soil parameters are ρ, e) and the initial stresses the converged stresses can be significantly different from the initial stress estimate. The users are strongly advised to check the equilibrium stresses and also plot the deformed shape of the mesh at the end of the Geostatic step. The resulting deformations in the geostatic step are solely due the difference between the user estimated initial stresses and the converged stresses in the equilibrium state as calculated by ABAQUS. These displacements are not the result of the initial stresses. It is a measure of the error in the user estimated initial stresses. Ideally the equilibrium stress state should be the same as the initial stress state. Also the displacements at the geostatic step should be negligible if not zero.

8 Analysis

The analysis is divided into two further steps. In the first of these the additional surcharge of 100 kPa is applied to the top surface of the soil column. This is achieved in a short space of time (500 secs). Since pressures specified are **total** (accumulated) quantities (not **incremental** changes) a value of 110 is specified.

The following nodal output quantities are requested to be printed in the *.dat file for every 25th increment (FREQ=25):

U - Nodal displacements

RF - Reaction forces at fixed nodes

POR - Pore pressures

RVT - Reaction Total Volume (conjugate to prescribed pore pressure)

PEEQ - represents the a_o values.

In a separate table the nodal displacements and the pore pressures for the nodes at the top (set TOPC) is also requested for every increment in the step.

Notice the output statements for step 3. Use of NODE PRINT and NODE FILE writes to the *.dat file and *.fil file respectively. The remaining output statements does the same for Element related information.

One could write a post-processing program (in FORTRAN) to extract the various information that is written to the *.fil file.

The *.odb (output database) is used for post processing by the Viewer module. However one could write a python based program to extract and manipulate the information held in the output database.

Another observation users would have made is that there are no statements controlling the output written to the output database for this example. This is because there is a default set of parameters which gets written to the output database. However it is possible to control the output written to the output database. More on this later.

```
**
**  step 2 : load application + 100 KPA (Total=110)
**
*STEP,NLGEOM, AMPLITUDE=RAMP
**-------------------------------
 SET UP INITIAL PORE PRESSURES
**-------------------------------
*SOILS,CONSOLIDATION
50.0, 500.0
*DLOAD1
 20, P3, 110.
*NODE PRINT, FREQUENCY=5,NSET=NALL
U, RF, POR, RVT
```

```
*EL PRINT, FREQUENCY=25, POSITION=CENTROID
 S, E
*EL FILE, FREQUENCY=25, POSITION=CENTROID
 S, E
*NODE PRINT, NSET=TOPC
 U, POR
**
*END STEP
**
**  step 3 : consolidation
**
**-------------------------------
** CONSOLIDATION
**-------------------------------
**
*STEP,NLGEOM,INC=200
    consolidation
**
*SOILS,CONSOLIDATION, UTOL=10
 1.E-5, 5.0E8,
**
*BOUNDARY
 TOPC, 8, 8, 0.
**
*NODE FILE, NSET=NALLC, FREQUENCY=20
  U, POR
*NODE PRINT, NSET=NALLC, FREQUENCY=20
  U, POR
*EL PRINT, FREQUENCY=25, POSITION=CENTROID, ELSET=SOIL
 S, E, VOIDR
*EL FILE, FREQUENCY=25, POSITION=CENTROID, ELSET=SOIL
 S, E, VOIDR
*MONITOR, NODE=401, DOF=2
*END STEP
```

```
        DIMENSION FIELD(NSECPT),TIME(2),COORDS(3),TEMP(NSECPT),
C==================================================================
C       The field variable is set equal to the Y co-ordinate.
C       This is turn causes the initial density distribution to
C       vary linearly with depth at the geostatic step.
C==================================================================

     1  DTEMP(NSECPT)
        KFIELD=1
        FIELD(1)=COORDS(2)
C
        RETURN
C
        END
```

The next step is the consolidation period where the excess pore pressures generated in the previous step dissipate. The total number of increments is set to 200. The NLGEOM parameter is specified to take into account the changes in geometry due to large settlement on the stiffness calculations as the analysis progresses.

In this step the pore pressures along the top surface are set to zero. UTOL is specified as 10. The presence of the UTOL parameter invokes the automatic time incrementation. It is set equal to 10 which is the maximum pore pressure change permitted in any increment. ABAQUS will restrict the time step in any increment so that this value is not exceeded at any (pore pressure) node during any increment in the current step. In contrast the absence of the UTOL parameter in the second step means fixed time increments will be used.

When this analysis was run the settlement at the top at the end of the geostatic step was 0.180 m. This is an indication of the error in the user estimated initial stresses. Also the *.msg file had several warning messages to the effect that the values calculated for a_o exceeded the user specified value (size of the initial yield locus). When this happens ABAQUS automatically adjusts the value of a_o.

The program also stopped after the first increment of step 3 because the results were diverging. A look at the *.msg file revealed that the cause of this was the displacement correction being too large compared to the displacement increment. Further investigation revealed that the default value of 0.01 was too stringent a convergence criterion. Consulting the manual this was the second data value for the CONTROL keyword. Comparing the corrections for the iterations in step 3 it was decided that 0.7 would be an acceptable value to provide a solution.

```
**-------------------------------
** CONSOLIDATION
**-------------------------------
**
*STEP,NLGEOM, AMPLITUDE=RAMP, INC=200
    consolidation
**
*SOILS,CONSOLIDATION, UTOL=10
 1.E-8, 5.0E8,
**
*CONTROLS, PARAMETERS=FIELD, FIELD=DISPLACEMENT
 ,0.7,
*BOUNDARY
TOPC, 8, 8, 0.
**
*NODE FILE, NSET=NALLC, FREQUENCY=20
  U, POR
*NODE PRINT, NSET=NALLC, FREQUENCY=20
  U, POR
*EL PRINT, FREQUENCY=25, POSITION=CENTROID, ELSET=SOIL
 S, E, VOIDR
*EL FILE, FREQUENCY=25, POSITION=CENTROID, ELSET=SOIL
 S, E, VOIDR
*MONITOR, NODE=401, DOF=2
*END STEP
```

This resulted in the analysis completing after 104 increments in step 3. The variation of pore pressure at node 401 is plotted against time in Figure 9. The settlement at the top of the soil is plotted against time in Figure 10.

Even though the variation of void ratio is assumed to be linear at in situ stage it is actually non-linear. Similarly there is no direct correspondence between the horizontal and vertical initial stresses. In order to investigate whether the in situ settlement of 0.180 m was due to these inaccuracies the following changes were made and a further analysis was carried out. Both the voids ratio and the initial stresses were calculated using user subroutines.

In the model part of the input the following changes were made :

```
*INITIAL CONDITION, TYPE=STRESS, GEOSTATIC, USER
   < No data card required >
```

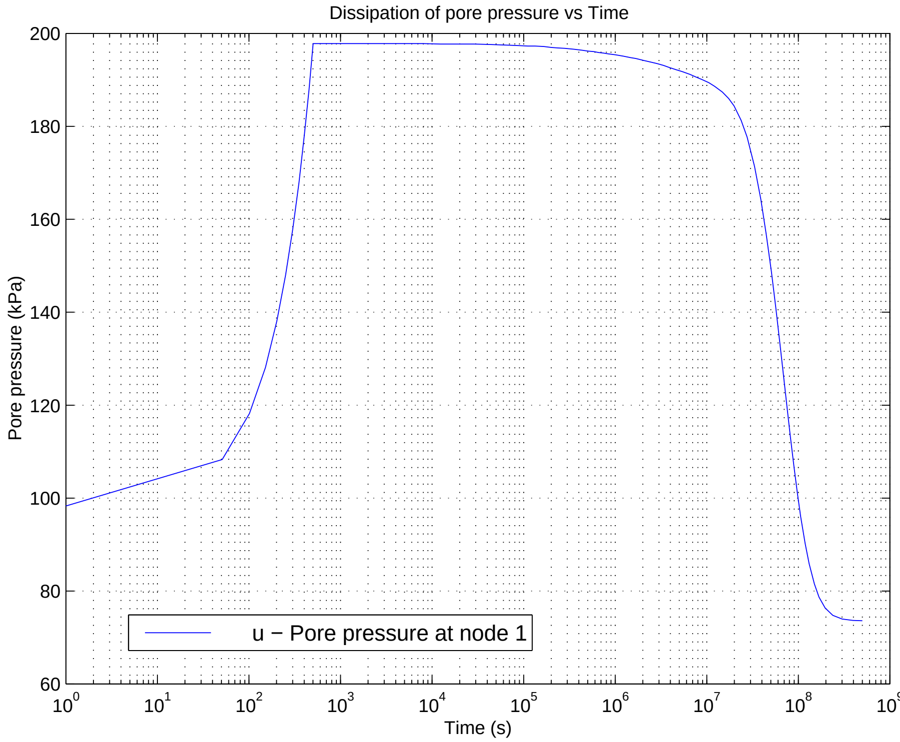

Figure 9: Pore pressure variation with time

```
*INITIAL CONDITION, TYPE=RATIO, USER
   < No data card required >
```

The inclusion of the USER parameters invokes the calls to the subroutines SIGNI and VIODRI respectively.

The following subroutines are included in a separate file (say) usersub.f : This includes the UVARM subroutine which is used to calculate and output the additional parameters that are required. Here it is used to calculate the void ratio, size of initial yield locus and the density and the bulk unit weight. These quantities are plotted in Figure 10.

The subroutine UVARM is invoked by the inclusion of the statement which is followed by the number of output variables requested.

```
*USER OUTPUT VARIABLES
 5
```

All the user subroutines are to be included in a single file with extension .f (extension .for in Windows PC) and the subroutines can appear in any order.

```
      SUBROUTINE VOIDRI(EZERO,COORDS,NOEL)
C************************************************************
C   Routine to calculate an initial void ratio distribution
C   ECS  Not equal to  E1 in ABAQUS (They are not the same).
C   ECS - is on the Critical State line.
C   E1  - is on the INCL (Iso. Normally Consol. line)
C************************************************************
      INCLUDE 'ABA_PARAM.INC'
C
      DIMENSION COORDS(3)
```

```
C
C     PX = PC /2.  (VALUE AT CRITICAL STATE FOR MCC ONLY)
C     PXT - Top.  PXB - Base
      PXT=5.
      PXB=29.
C
      PT=7.333
      PB=42.667
C
      ECS=3.0
C
      HEIGHT=10.
      AKAPPA=0.1
      ALAMDA=0.5
C
      PX = PXT + ((PXB-PXT)/HEIGHT)*(HEIGHT-COORDS(2))
      P  =  PT + ((PB-PT)/HEIGHT)*(HEIGHT-COORDS(2))
C
      EZERO = ECS - (ALAMDA - AKAPPA) * LOG(PX) - AKAPPA * LOG(P)
C
      RETURN
      END
      SUBROUTINE SIGINI(SIGMA,COORDS,NTENS,NCRDS,NOEL,NPT,
     +                  LAYER,KSPT,LREBAR,REBARN)
C************************************************************
C
C     In situ Effective stresses :
C     SIGVET - Vertical Effective Top
C     SIGVEB - Vertical Effective Base
C     SIGHET - Horizontal Effective Top
C     SIGHEB - Horizontal Effective Base
C************************************************************
      INCLUDE 'ABA_PARAM.INC'
C
      DIMENSION SIGMA(NTENS),COORDS(NCRDS)
C
      CHARACTER*8 REBARN
C
      SIGVET=-10.
      SIGVEB=-60.
C
      SIGHET=-6.
      SIGHEB=-34.
C
      HEIGHT=10.
C
      SIGMA(1)=SIGHET + ((SIGHEB-SIGHET)/HEIGHT)*(HEIGHT-COORDS(2))
      SIGMA(2)=SIGVET + ((SIGVEB-SIGVET)/HEIGHT)*(HEIGHT-COORDS(2))
      SIGMA(3)=SIGMA(1)
C
      RETURN
      END
      SUBROUTINE UVARM(UVAR,DIRECT,T,TIME,DTIME,CMNAME,ORNAME,
     1 NUVARM,NOEL,NPT,LAYER,KSPT,KSTEP,KINC,NDI,NSHR,COORD,
```

```
     2 JMAC,JMATYP,MATLAYO,LACCFLA)
C=====================================================================
C
C     Routine to calculate bulk unit weight, dry density, water
C     content, GS (initial GS varies with depth. Because of constant
C     bulk unit weight of 15.0 initially)
C
C=====================================================================
C
      INCLUDE 'ABA_PARAM.INC'
C
      CHARACTER*80 CMNAME,ORNAME
      CHARACTER*3 FLGRAY(15)

      DIMENSION UVAR(NUVARM),DIRECT(3,3),T(3,3),TIME(2)
      DIMENSION ARRAY(15),JARRAY(15),JMAC(*),JMATYP(*),COORD(*)
C
C--------Error counter
      JERROR = 0
C
C---------get co-ordinates
C      CALL GETVRM('COORD',ARRAY,JARRAY,FLGRAY,JRCD)
C      Y=ARRAY(2)
       Y=COORDS(2)
C
C----------get void ratio
      CALL GETVRM('VOIDR',ARRAY,JARRAY,FLGRAY,JRCD)
      E=ARRAY(1)
C
      HEIGHT=10.
C
      BULK=15.
      GAMMAW=10.
C
      GS=(BULK/GAMMAW)*(1.+E)-E
C
      WATC=E/GS
C
      DRY=BULK/(1.+WATC)
C
      UVAR(1)=DRY
      UVAR(2)=BULK
      UVAR(3)=WATC
      UVAR(4)=GS
      UVAR(5)=E
C
      RETURN
      END
```

The command to submit the job is :
abacus job=consol user=user sub interactive
These changes did not make much of a difference to the results. The initial settlement at geostatic step reduced to 0.130 m and the final settlement was 2.72 m compared to the previous 2.64 m.

9 References

1. Atkinson J.H. and Bransby P. L. (1978) **The Mechanics of Soils**, McGraw-Hill.
2. Atkinson J.H. (1981) **Foundations and Slopes**, McGraw-Hill.
3. Bolton M.D. (2000) **A Guide to Soil Mechanics**, McGraw Hill.
4. Britto A.M. and Gunn M.J. (1987) **Critical State Soil Mechanics via Finite Elements**,Ellis Horwood Ltd.
5. Powrie, W (2004) **Soil Mechanics Concepts and Applications**, Taylor & Francis.
6. Roscoe, K.H. and Burland J.B. (1968) **On the generalised stress-strain behaviour of 'wet clay'**, Engineering Plasticity, Cambridge University Press.
7. Roscoe, K.H. Schofield, A.N. and Thurairajah, A.H. (1963) **Yielding of Soils instates wetter than critical**, Geotechnique, **13**, 211-240.
8. Schofield, A.N. and Wroth, C.P. (1968) **Critical State Soil Mechanics**, McGraw Hill, Maidenhead (see http://www-civ.eng.cam.ac.uk/geotech_new/geotech.htm)
9. Schofield, A.N. (1980) **Cambridge Geotechnical Centrifuge Operations**, Rankine lecture, Geotechnique, **30**, 225-268.
10. Schofield, A.N. (2005) **Disturbed Soil Properties and geotechnical Design, Thomas Telford Publications**
11. Wood D. Muir (1990) **Soil behaviour and Critical State Soil Mechanics**

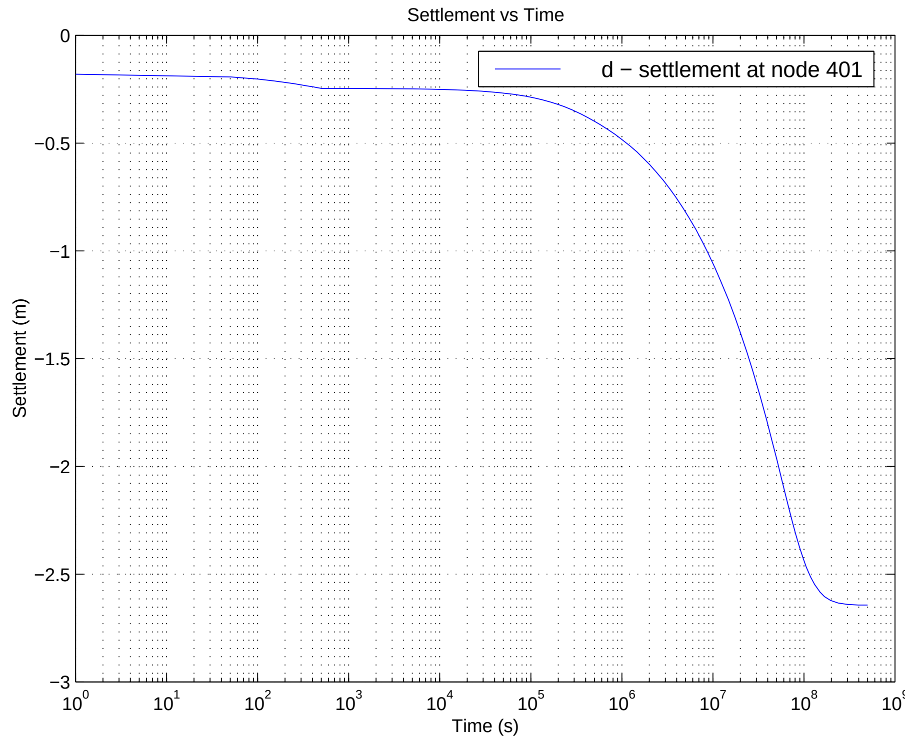

Figure 10: Settlement with time

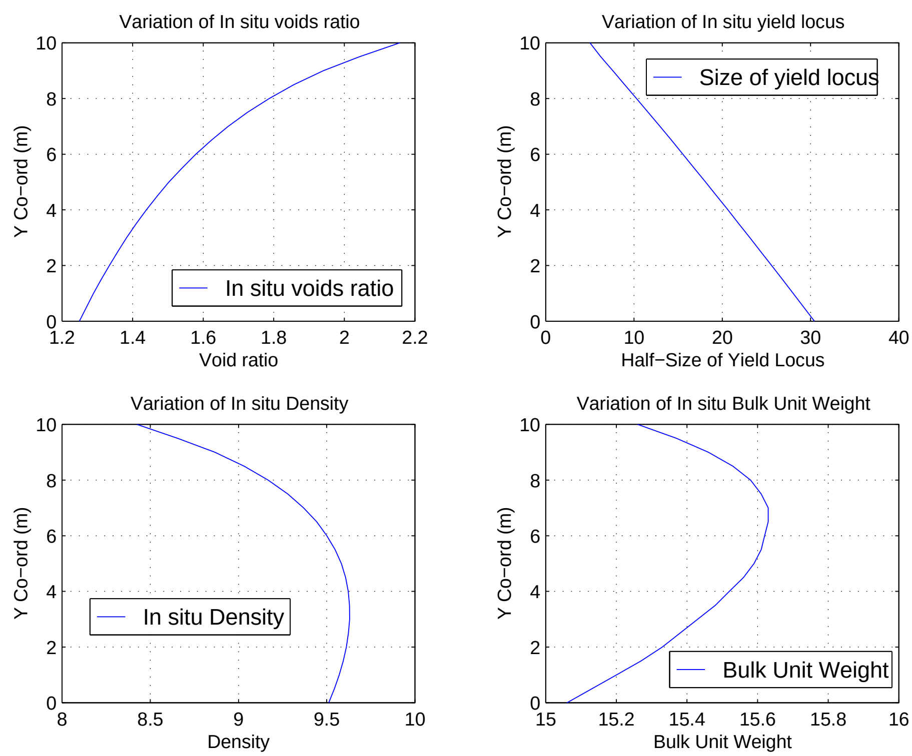

Figure 11: Variation of parameters at In situ Stage.