One-dimensional consolidation of a linear elastic two-layered soil profile

Question: Consider a $5m$-thick layer of clay underlain by a $5m$-thick layer of sand, above stiff, impermeable bedrock. An extensive layer of granular fill is placed on the soil surface, exerting a pressure of $50kPa$. The placement of the fill is assumed instantaneous.

1. Determine the settlement vs time curve.

2. Plot isochrones of excess pore pressure at the end of the 2nd, the 5th and the 10th day.

Both soils are assumed linear elastic. The clay parameters are $E = 10MPa$ and $\nu = 0.30$, and its permeability is $k = 10^{-8}m/sec$. The sand parameters are $E = 50MPa$ and $\nu = 0.35$, and its permeability is $k = 10^{-5}m/sec$.

Although for a soil profile of this thickness it is not a realistic assumption, to keep the analysis simple we will ignore gravity and thus use incremental effective stresses and excess pore pressures.

Get the units straight! In this case we use m as a unit of length and kN as a unit of force. Therefore the unit of stress will be $kN/m^2 = kPa$, of unit weights kN/m^3, and of densities Mg/m^3. Although at first sight the implied unit of time appears to be sec, since $kN = Mg \cdot m/sec^2$, the unit of time is determined by the units of permeability: sec if permeability is entered in m/sec, day if entered in m/day, etc. In the following we will convert permeability values to m/day for convenience, hence the unit of time will be day.

Start a new model Start ABAQUS/CAE, create a new database and a new model. By default it will be named “Model-1”. Click “Model/Rename/Model-1” and enter “Two_layers” as the new name of the model.

Create a part Go to module “Part” and create a new part (“Part/Create”) to represent the geometry. This should be in “2D planar” modelling space, of the “Deformable” type and with “Shell” as its base feature. This is meant to be a one-dimensional problem, so the width of the part is irrelevant. Create a rectangular part that is $10m$ high and $1m$ (unit) wide: in the sketcher select “Add/Line/Rectangle” and type the following coordinates for the two corners of the part: $0, 0$ and $1, -10$. Click the red “x” and then “Done”.

Create partitions To accommodate two different materials, the part must be partitioned to two regions, one for each material. Select “Tools/Partition”, then “Face” and “Sketch” in the window that appears. Select “Edit/Sketcher options”, then “Align grid/Origin” and click on the top left corner of the rectangular part: this resets the coordinate system to start at that point.

To create the partition select “Add line/Connected lines” and enter the following sets of coordinates to create the clay-sand interface: $0, -5$ and $1, -5$. Click the red “x” and then “Done” to exit the sketcher, then “Done” again to create the partition.

The part should now appear split in two equal parts by a horizontal line.

Define material properties For the clay: In the “Property” module select “Material/Create” to create a new material named “clay”. Assign a linear elastic model using “Mechanical/Elasticity/Elastic” and entering the properties $E = 10000$ (kPa) and $\nu = 0.3$. Then select “Other/Pore fluid/Permeability” and enter the permeability for the clay in m/day: $k =$

$10^{-8} m/sec = 8.64 \cdot 10^{-4} m/day$. Use as the corresponding value of void ratio $e = 1.0$. The actual value of void ratio you enter is not important as long as it is positive, as we are providing a singe data line and thus defining a constant value for the permeability. You also need to enter the specific (unit) weight of the fluid. You need to be careful to be consistent with units here; in this case it should be prescribed in kN/m^2. Enter 10 in the box.

For the sand: In the "Property" module select "Material/Create" to create a new material named "sand". Assign a linear elastic model using "Mechanical/Elasticity/Elastic" and entering the properties $E = 50000$ (kPa) and $\nu = 0.35$. Then select "Other/Pore fluid/Permeability" and enter $k = 0.864$ as the permeability of the sand (which is $10^{-5} m/sec$ in m/day) and a corresponding value of void ratio $e = 1.0$. You also need to enter the specific (unit) weight of the fluid as 10.

Create sections Go to "Section/Create", name the new section "clay", select "Solid" and "Homogeneous" and click "Continue". Select "clay" as the material and click "OK". Then repeat the process to create a new section named "sand", to which you assign the "sand" material.

Assign sections to the part Go to "Assign/Section", click on the top partition then "Done". In the window that appears select "clay" as the section and click "Done". This assigns the clay material to the top partition. Repeat the process to assign the "sand" material to the bottom partition.

Assemble parts Go to module "Assembly" and introduce an instance of the part you just created into the assembly, by selecting "Instance/Create", selecting "Part-1", selecting "Independent" and then "OK".

Create the analysis steps Go to module "Step". We will need two different steps: a *Geostatic* step to establish initial equilibrium, and a transient *Soils* step to carry out the consolidation analysis. Although in this case the Geostatic step is not necessary as all initial conditions are zero, it is a good habit to have it in the analysis.

Click on "Step/Create" and select "Geostatic" from the list of different types of Step. Select "Continue…". Make sure that "Nlgeom" is off and select "OK".

Create a second step now, selecting "Soils". Make sure "Transient consolidation" is selected in the "Basic" tab, "Nlgeom" is off and "Include creep/swelling" is left unticked. As time period for the analysis enter 60, i.e. the analysis will attempt to calculate the response for the first 60 days.

In the "Incrementation" tab you can select the number of increments in which ABAQUS will attempt to subdivide the Step, and the rules for the maximum and minimum time each increment should correspond to. Use "Automatic" incrementation, with initial increment size 0.1, minimum 0.1 and maximum 5. Abaqus will adjust the time increment during the calculation to save computational time and increase accuracy, but it will honour these constraints in the process. The initial and the minimum increment size have been chosen so that the timestep is always greater than the minimum acceptable value for the mesh used (see the lecture notes for the relevant expression!)

Tick "Max pore pressure change per increment" and enter 50. This allows you to control the accuracy of time integration by requiring that the pore pressure does not change more than a limit at each increment. In this case we expect a change of $50kPa$ to take place within a single increment, as drainage drops excess pore pressure to zero from the initial value of $50kPa$. Anything less that 50 will lead to (apparent) lack of convergence. You can also experiment with larger values.

Click "OK" to continue.

Here you can also specify the type and amount of output that you would like generated at the end of the analysis. You can do that by selecting "Output/Field Output Requests/Edit/F-Output-1". Stress, strain and displacements are pre-selected by default, however you may want to add fluid velocity FLVEL to the list; it is under "Porous media/Fluids".

Prescribe initial conditions, loads and boundary supports Go to module "Load".

Create the supports To create the lateral supports select "BC/Create", name the boundary condition "lateral", select the "Initial" step and the "Displacement/Rotation" type. Click "Continue", select all four left and right edges (using the Shift key while clicking, to add to the selection each time) and click "Done". In the menu that appears select $U1$ and "OK". Only zero boundary conditions are acceptable in the "Initial" step, so zero will be assumed as the boundary condition value.

Following the same steps, create the bottom support by selecting the bottom edge of the model and constraining $U2$.

Create the load We impose a downwards distributed load of $50kPa$ at the top surface of the layer. Select "Load/Create" and select the Soils step (Step-2) as this is when the load becomes active. Select "Pressure" as the load type and click "Continue". Select the top edge of the model, then "Done". In the menu that appears enter "50" as the magnitude and click "OK". You should now see the load appear as a group of downward arrows.

Create the drainage boundary During the consolidation stage we want to allow drainage from the top surface. From "BC/Create", choose the Soils step (Step-2) and tick "Other" as the category of boundary condition you wish to apply. Select "Pore pressure" and click "Continue". Select the top surface of the model, "Done", and enter 0 as the magnitude in the menu that appears. You should now see a set of squares appear at the top edge, signifying the existence of a pore pressure boundary condition.

Prescribe initial conditions We need to establish initial conditions for the void ratio, as it is used to define the permeability. Select "Predefined Field/Create". From the roll-down menu select step "Initial", category "Other" and type "Void ratio". Click "Continue", select the whole model and click "Done". In the window that appears select "Constant through region", enter "1.0" as magnitude and click "OK". You should now see a set of squares appear all over the model, signifying the existence of a predefined field.

In this particular case it is not necessary to prescribe an initial profile of pore pressure or of effective stress, as both are assumed zero. If gravity were considered, however, along with total pore pressures and a geostatic profile of effective stress, these would need to be prescribed using the same procedure but different options.

Create the mesh Go to module "Mesh".

Type of meshing Select "Mesh/Controls" and then a "Quad" element shape and the "Structured" technique.

Type of element Select "Mesh/Element Type", select both partitions of the model and click "Done". In the window that appears select "Standard" (which is the name of the programme that we will use.) We want to use biquadratic diplacement, bilinear-pore pressure quadrilaterals, so tick "Quadratic" for geometric order and select the "Quad" tab. Scroll down in the "Family" section and select "Pore Fluid/Stress" as the type of element. We can use either full or reduced integration; in this case use reduced integration by making sure the appropriate box is ticked. Leave everything else at default values. The element chosen should appear lower in the window as "CPE8RP". Select OK to continue, then "Done".

Mesh seeds and meshing Select "Seed/Instance". In the window that appears, enter an approximate global seed size of 0.5, so that the default element size will be $0.50m \times 0.50m$, and click "OK" and "Done".

Select "Mesh/Instance" and answer "Yes" in the question of whether you want to mesh this part instance. A mesh should appear, consisting of 2×20 quadrilateral elements.

Running the analysis Go to module "Job" and select "Job/Create". Name the new job "two_layers", select the model "Two_layers" in the lower part of the window, and click "Continue". In the window that appears make sure "Full Analysis" is selected and press "OK" to create the job.

Submit the job Select "Job/Submit/two_layers"; this submits the job for analysis. Then immediately select "Job/Monitor/two_layers" and a monitoring window will appear, showing you how the analysis progresses. If everything went well, the first step will appear on the top row to have ran in a single increment. The log at the bottom of the window will show at what time the job was submitted and eventually "Completed: Abaqus/Standard". At that stage the analysis will have finished and results should be available. You can select "Dismiss" to close the monitoring window.

Post processing the results While still in the "Job" module, select "Job/Results" to look at the results of the analysis.

Deformed shape To examine the deformed shape of the model select "Plot/Deformed Shape".You can change the magnification factor used for the deformed shape if you select 'Options/Common" and on the "Basic" tab select e.g. "Uniform" and type in the desired value.

Vector plots To plot results in vector form select "Plot/Symbols/On Undeformed Shape". Vector plots showing the direction and relative magnitude of the principal stresses should appear by default. You can plot a different variable by selecting "Result/Field Output" and picking the variable of interest in the window that appears. (One example is the flow velocity FLVEL.) From "Options/Symbol" you can fine tune different properties of the vectors and the maximum and minimum values represented by the size of the vectors plotted.

XY-Data You can plot data as curves of a quantity (e.g. "U2"') vs. a variable (e.g. "Time"); ABAQUS calls this type of data "XY-Data". There are different ways of creating XY-Data.

You can create XY-Data by extracting the value of a variable (e.g. displacement at a node) over time by selecting "Tools/XY Data/Create" and ticking "ODB field output" as the source. When you press "Continue" a window appears that allows you to choose what data you want extracted and from what parts of the model. Select “Unique nodal” as the type of quantity and click on “U2” in the list that appears. You may need to click on the black arrows to reveal individual components of vector quantities. Once you have selected the quantities you want reported, click on the “Elements/Nodes” tab. Click on the “Pick from viewport” method and “Edit selection”. Select a node to report its results, e.g. the middle node of the top edge, then click “Done”, “Save”, “OK” and “Dismiss”. The relevant curve, which shows the vertical displacement of that node over time, has now been created and you can display it by selecting “Tools/XY Data/Plot” and then the name of the curve. You can change the way the curve looks by going to “Options/XY Options/”.

A different way of creating XY-Data is using a path, so that the spatial distribution of a quantity can be plotted at a given time step. We can use a path to create isochrones of excess pore pressure at specific times.

To create a path first select “Plot/Undeformed shape” and, once the mesh is visible, “Tools/Path/Create”. Select “Point list” and “Continue”. Enter the coordinates of the start of the path, in this case “0.5,0”, press “enter” and in the second line enter the coordinates of the end of the path - in this case “0.5,-10”. Click ’OK’.

To extract the pore pressure along the path, select “Tools/XY Data/Create”, then “Path” in the list that appears, and click “Continue”. Select “Undeformed” shape, then “Path points”, “Include intersections” and “True distance”.

Click on “Field output” and select the quantity of interest - in this case POR, the excess pore pressure. By clicking the icon next to the “Frame” indicator, you can choose which timestep you want data extracted from. For this example, select the line corresponding to step-time $t = 3$, which is 3 days after the load was applied. Click “OK” twice, then “Save as...”, provide the name “por03” for the new curve and click “OK”. Select “Tools/XY Data/Plot/por03”to plot the curve.

Using the same process you can create different isochrones; to plot many of them together from “Tools/XY Data/Manager”.