3D FE tool for time dependent settlement predictions

H. P. Jostad
Norwegian Geotechnical Institute (NGI), Norway, hpj@ngi.no

N. Sivasithamparam
NGI, Norway

B. H. Woldeselassie
NGI, Norway

S. Lacasse
NGI, Norway

ABSTRACT
A newly developed calculation tool for fully coupled general 3D finite element consolidation analyses is presented. The development has been part of an ongoing Research and Development project called GeoFuture and the tool is implemented into the commercial geotechnical software Novapoint GeoSuite. There, the tool is part of an integrated software system for handling geotechnical and geometrical data, with wiki based user assistance for selection of material properties and controlling the simulations, and 3D graphical visualization of input data and results. The paper gives the main background and features of the calculation tool. To demonstrate some of the capabilities of the tool, back-calculations of the measured long-term settlements of a heavy building on a deep soft clay layer in Oslo centrum are presented.

Keywords: FEM, Consolidation, Settlements, Creep, Back-calculation

1 INTRODUCTION

Settlements of foundations and embankments on soft ground are in geotechnical engineering often calculated using idealized 1D methods with simplified assumptions or elastic analytical solutions of load spread distribution with depths, pure vertical pore pressure dissipation and compressibility parameters from oedometer tests. Time dependent creep deformations are in Norway generally added by a simple secondary consolidation phase. However, in some projects more accutate settlement predictions are required or the problem is too complex to be idealizezed by a simplified 1D solution. In these cases, analyses using a fully coupled displacement and pore water flow (consolidation) finite element (FE) program with a proper material model is a good approach. Such a general 3D finite element code is developed as part of the Research and Development (R&D) project GeoFuture (www.geofuture.no) and implemented into the commercial software package Novapoint GeoSuite (www.ViaNovasystems.com). The main differences between this code and other existing Finite Element codes as for instance Plaxis (www.plaxis.nl), is that the tool is an integrated part of a system for seamless handling of project related geotechnical and geometrical data. In geotechnical projects, field and laboratory data is generally already stored in the system, Novapoint GeoSuite Presentation. The main features of this integrated geotechnical calculation tool for settlement analyses are described in the following.

2 FINITE ELEMENT CODE

2.1 FE Formulation

A general finite element code, originally developed at NGI in the nineties, is used to solve the governing equations for consolidation problems. The governing differential equations are the (stress) equilibrium and the mass balance (continuity) equation of water. The equilibrium equation is solved by a clasical small strain displacement based finite element formulation, where the external (nodal point) force vector $\boldsymbol{R}_{ext}$ due to body forces and surface loads should be equal to the internal force vector $\boldsymbol{R}_{int}$ from the total stresses (effective stresses plus pore pressure):

$$\boldsymbol{R}_{ext} = \boldsymbol{R}_{int} \tag{1}$$

Were $\boldsymbol{R}_{ext}$ and $\boldsymbol{R}_{int}$ are calculated according to conventional finite element formulations as for instance described in Zienkiewicz (1977). The relationship between effective stresses and strains are defined by a proper material model. This material model may be rate dependent in order to account for time dependent strains (creep). The strains are derived from the calculated displacement field. For 3D problems the displacement vector $\boldsymbol{r}$ is decomposed into the three global directions, r_x, r_y and r_z.
The pore water flow is solved by a weak form of the water balance equation, where the pore water flow out of the model (nodes) $\boldsymbol{F}_{out}$ must be equal to the global volume change ΔV (given by the displacement field $\boldsymbol{r}$):

$$\boldsymbol{F}_{out} = \boldsymbol{L}^T\boldsymbol{r} \tag{2}$$

Where the vector $\boldsymbol{F}_{out}$ and matrix $\boldsymbol{L}$ are calculated by conventional finite element formulations as for instance described in Potts and Zdravkovic (1999). The water is then assumed to be incompressible.
The relationship between the pore pressure field p and the average (superficial) pore water flow velocity q is given by Darcy's law:

$$q = -\frac{1}{\gamma_w} k \ \frac{dp_{excess}}{ds} \tag{3}$$

Where k is the permeability coefficient, γ_w is the unit weight of water, dp_{excess}/ds is the spatial gradient of the excess pore pressure (i.e. in excess to the hydrostatic pore pressure). In the code, the flow velocity is decomposed into the 3 global directions (q_x, q_y and q_z) with the corresponding permeability coefficients (k_x, k_y, k_z).
In order to solve the transient problem, where the pore pressure field is changing with time, a weighted average of the pore pressure field within a time increment Δt is used:

$$p_{av} = p(t) + \theta \cdot p(t+\Delta t) \tag{4}$$

where $p(t)$ and $p(t+\Delta t)$ are the pore pressure in the nodes at the beginning and the end of the time increment. θ is a weighting factor, which for classical consolidation problems is recommended to be chosen larger than 0.5. This means that the coupled consolidation problem is solved by a time stepping procedure starting from an initial state. This also open up for that the external loads and boundary conditions (displacements and pore pressure with known/prescribed values at given nodes) may change with time.
In order to solve the non-linear behaviour of the soil, the governing equations within each time step are solved by a Newton Raphson iteration scheme.

2.2 Main feature of the code

The finite element code used in this development has a modular structure, which makes it easy to include new finite element types, material models and solution algorithms. For coupled consolidation analyses the following elements and features are currently included:

- 20-noded isoprametric brick element with pore pressure degree of freedom in the corner nodes and (2x2x2) reduced or (3x3x3) full Gaussian integration
- 10-noded isoparametric tetrahedral element with pore pressure degree of

freedom in the corner nodes and full Gaussian integration
- An automatic time stepping procedure (described in Jostad and Engin, 2013) controlled by the maximum change in the pore pressure within a time step, in addition to a standard procedure with manual input of the time increments for more complex time histories of loads and boundary conditions
- Spatial lateral interpolation of material properties between a number of input profiles
- Spatial lateral interpolation of steady-state pore pressure between a number of input pore pressure profiles. The steady-state pore pressure field may be given as function of time in order to handle specified or known changes in the ground water table
- A library of different material and permeability models. Some of these material models may account for creep (rate effects). A 3D material model developed at NTNU in a PhD study will be implemented in 2016
- Application of a set of time dependent surface loads

2.3 *Input of topography and soil layers*

For simple geometries (e.g. horizontal soil layers), a cube containing 20-noded brick elements is generated based on input of horizontal and vertical grid lines. In order to generate a 3D finite element model one may simply expand the model/data used in the existing 1D GeoSuite Settlement calculation tool. This means that the graphical user interface (GUI) in this case is the same for 1D and 3D analyses. The 3D finite element model may also be degenerated into a 2D cross-section or a 3D model with only vertical displacement degrees of freedom. For many problems, this will speed up the computation time significantly without significant loss in accuracy.
For more complex geometries, a finite element mesh containing the 10-noded tetrahedral element may be generated by the code Tetgen (2015). An example of a mesh with varying thicknesses of the soil layers and depth to the rock is shown in Figure 1. As bases for generating this model an existing ground observation model (GOM) may be used. The GOM is established based on site specific borehole data and a terrain model. An example of a GOM is shown in Figure 2.

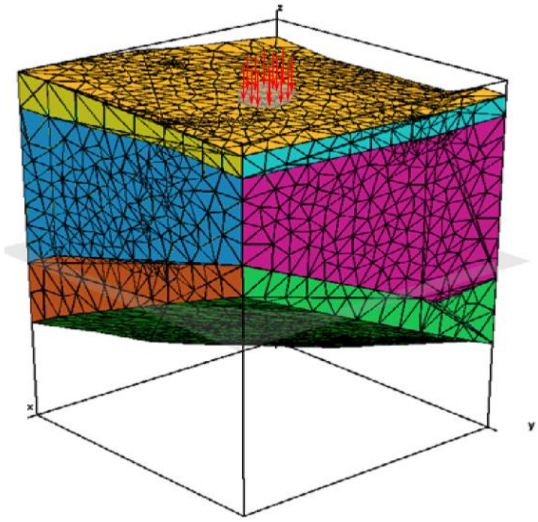

Figure 1 Finite element mesh using 10-noded tetrahedral elements.

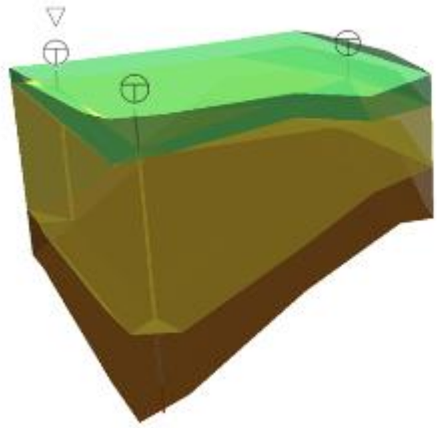

Figure 2 Ground Observation Model (GOM) based on 3 boreholes.

2.4 *Wizards assistance to users*

Lacasse *et al* (2013) described briefly the Wizard function used in GeoSuite. Wizard is an optional, interactive assistance popping up with information on the selection of soil parameters, the interpretation of *in situ* or laboratory test results, the selection of a type of analysis, the features of the analysis itself or the interpretation of the results of an

analysis. Wizard invites the user to note down its comments within the Web site; Wizard makes topic associations with links; Wizard seeks to involve the user in an on-going process of improvement.
As an example, the scheme of a settlement analysis is shown in Figure 3.

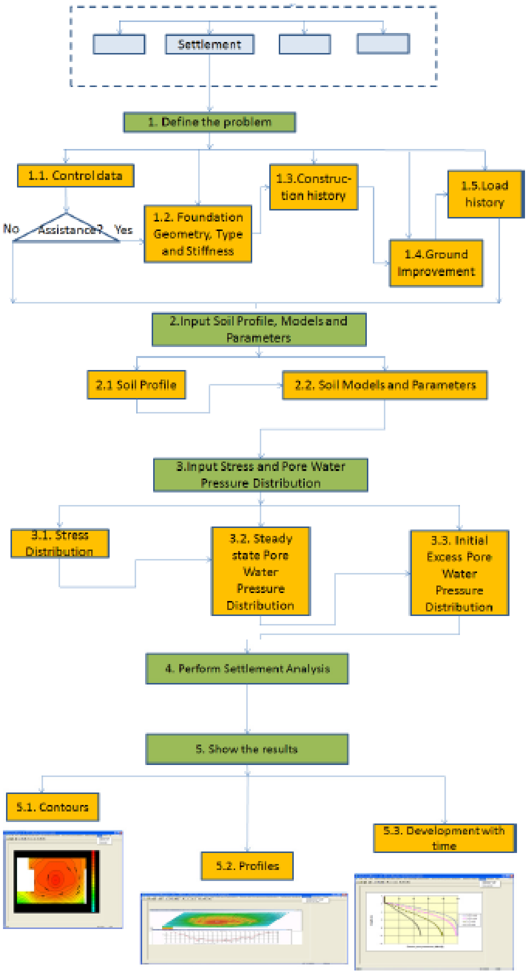

Figure 3 Flow chart for settlement analysis, Step 1 to 5.

3 EXAMPLE

3.1 General background

To demonstrate some of the capabilities of the calculation tool, back-calculations of the measured long-term settlements of a heavy building, Oslo Jernbanetollsted shown in Figure 4, on soft clay in Oslo centrum are presented. Back-calculations of this building have previously been published in Andersen and Clausen (1975) and Svanø et al. (1991). The paper by Andersen and Clausen (1975) gives the details about the soil condition and the construction sequences of the building.

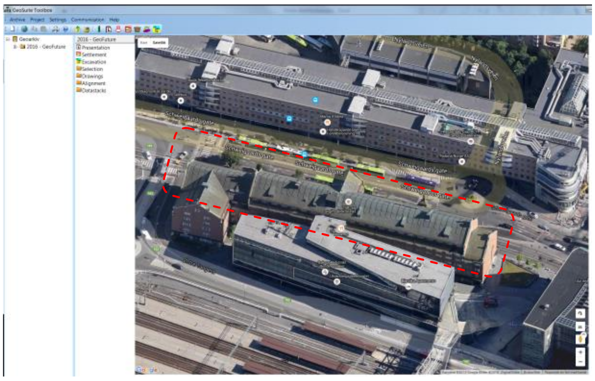

Figure 4 Oslo Jernbanetollsted (second building from the railways). Google picture in the GeoSuite toolbox starting window.

The effects of different available features in the sofware are demonstrated by analyses with increasing degree of complexity.

3.2 Soil condition

The soil consists of a 2.5 m think old fill of gravel and stones, a 7 m thick sandy and silty clay layer with weathering in the upper 2 m, a 1.5 m thick very stiff clay layer, a 22.5 m thick fairly homogenous and nearly normally consolidated marine clay. The depth to the rock is about 80 m. The boring profile is shown in Figure 5. The ground water table is located 0.5 m below the fill.

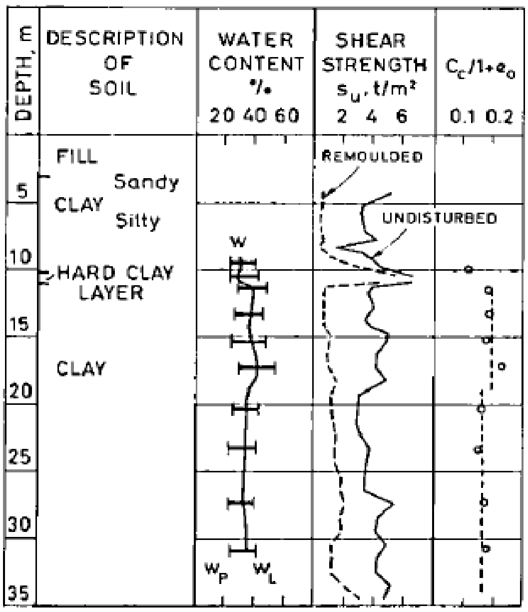

Figure 5 Soil profile close to the building. From Andersen and Clausen (1975).

3.3 Building

The Oslo Jernbanetollsted is a six storeys high building. It covers an area of approximately 140 m times 21 m. The foundation consists of about 5000 wooden piles down to level -9.3 m with a 1-2 m thick reinforced slab at the top, see Figure 6.

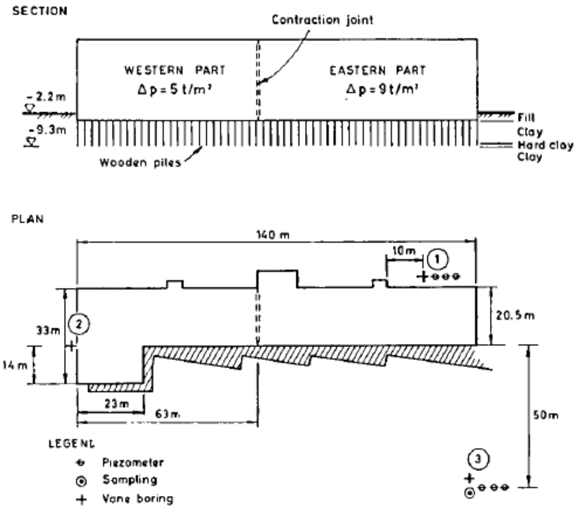

Figure 6 Vertical cross section and plane view of Oslo Jernbanetollsted, From Andersen and Clausen (1975).

The construction of the building started in summer 1920, and finished early in 1924. Two years later the live load had reached the average operational value. The live load was estimated to 50 kPa in the western part and 90 kPa in the eastern part.

3.4 Material model

A material model based on Janbu's resistance concept (Janbu, 1985) is used in the analyses. The material model together with a 1D calculation of the same problem are presented in Svanø et al. 1991.
The constrained (oedometer) vertical strain (rate) $d\varepsilon$ is here given by an effective stress dependent elasto-plastic part $d\varepsilon_{ep}$ and a creep part $d\varepsilon_{creep}$:

$$d\varepsilon = d\varepsilon_{ep} + d\varepsilon_{creep} = d\sigma_v'/M_t + 1/R \qquad (5)$$

The elastic strain for effective vertical stresses σ_v' below the pre-consolidation pressure p_c' is given by an oedometer modulus M_{oc}. For effective stresses in excess of the initial pre-consolidation pressure, the elasto-plastic strain is given by an effective stress dependent tangent oedometer modulus:

$$M_t = m \cdot (\sigma_v' - p_r') \qquad (6)$$

Where m is a dimensionless modulus number and p_r' is a stress intercept that for instance controls the tangential oedometer modulus at p_c'. The time dependent (visco-plastic) creep strain is given by the time resistance:

$$R = R_o + r \cdot (t - t_{ref}) \qquad (7)$$

Where R_o is the time resistance at the reference time t_{ref} (here taken at 24 hours), and r is the dimensionless time resistance number. This means that it is assumed to be zero creep strain at $t = t_{ref}$. However, the creep rate is $1/R_o$ at $t = t_{ref}$. r is varying with the effective vertical stress. It is defined by a value at the in situ condition, at the pre-consolidation pressure p_c' and with a slope after p_c', i.e. the parameters r_o, r_{pc} and $\beta = dr/d\sigma_v'$.

In the present analyses the shear modulus is taken as $G = M_{oc}/3$. For more complex problems (e.g. problems were the soil is loaded to higher shear mobilisation), a more advanced formulation is required for the description of the shear stiffness.
Furthermore, no creep is assumed for horizontal and shear strain components.
The soil parameters used in the analyses are presented in Table 1. The parameters are based on Svanø et al. (1991).

Table 1 Soil properties of the clay layers.

Depth (m)	**OCR**	***m***	***M***	***R_o*** (yrs)	***r_o***	***r_pc***
0–9	1.4	15	5	-		-
9-18	1.4	15	5	0.8	2000	300
18-40	1.4	18	5	0.8	2000	300

The creep parameter β is zero. The (isotropic) permeability k is 0.031 m/years in the top 9 m reducing linearly to 0.022 m/years at 18 m.

3.5 Finite element models

The settlement calculations have been performed with models of increasing complexity:

- 1D models with constant load distribution with depth
- 1D models with load distribution with depth based on Bousinesq (elastic half space solution)
- 2D models of cross sections through the eastern and western part of the building
- 3D model along the centreline of the building (both eastern and western part). This element model is shown in Figure 7

One challenging part here is to calculate the load transfer to the pile tip level. In the finite element analyses a reduced elastic shear stiffness (G = 200 kPa + 100 kPa/m · depth) is used along the periphery of the foundation down to skirt tip level and in the joint between the western and eastern part. The actual stiffness along these soil-soil interfaces affects the loads (pressure) transferred to the skirt tip level. The calculated excess stress distributions versus depth (at maximum load) at the centre of the eastern part are shown in Figure 8.
Drainage is assumed at both the bottom and top of the models. "Roller" boundaries are assumed along the vertical boundaries.

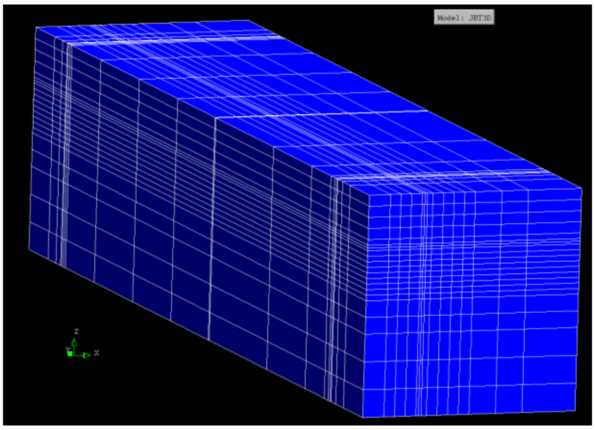

Figure 7 3D FE model of one side of the symmetry plane along the centreline of the foundation.

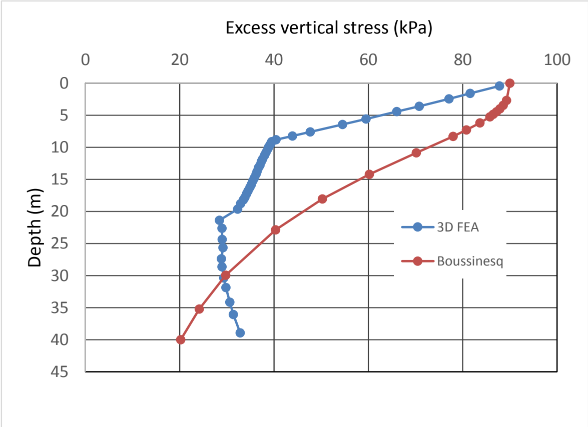

Figure 8 Calculated excess vertical stress versus depth at max loads at the centre of the eastern building.

3.6 Loading history

The actual loading history is sketched in Figure 9. The weight of the old fill (approximately 50 kPa) is in the simulation applied 20 years before construction of the building. This is done in order to account for the increase in pre-consolidation pressure with depth due to creep under this pressure. The maximum foundation loads were reached at the end of 1925.

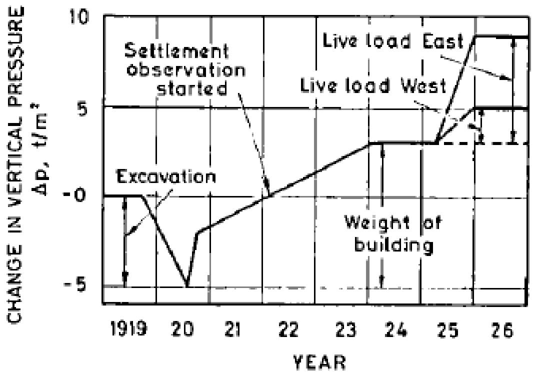

Figure 9 Actual loading history. From Andersen and Clausen (1975).

3.7 Results

The calculated settlements at the centre of western and eastern part of the building are shown in Figure 10.

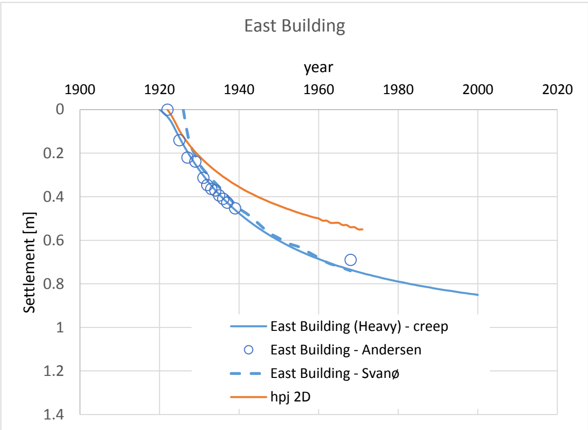

Figure 10 Calculated settlement curves at the centre of the eastern (heaviest) part.

The calculated distribution of the vertical displacements from the 2D cross section trough the eastern part of the building at the end of 1970 is shown in Figure 11.

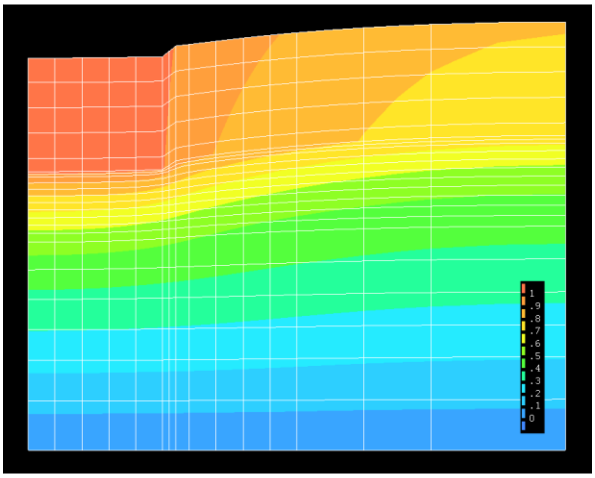

Figure 11 Calculated vertical settlements through the centre of the eastern part at end of 1970, including the settlements due to the old fill. Deformed mesh is magnified by a factor of 10.

The calculated distribution of the vertical displacements from the 3D model at the end of 1970 is shown in Figure 12.
In these calculations the parameters are taken from the paper by Svanø et al. (1991). However, it is clear that by accounting for more advanced calculation of the stress distribution with depth, the parameters used in that idealized 1D calculation should be re-evaluation in order to better fit with the observed settlements.

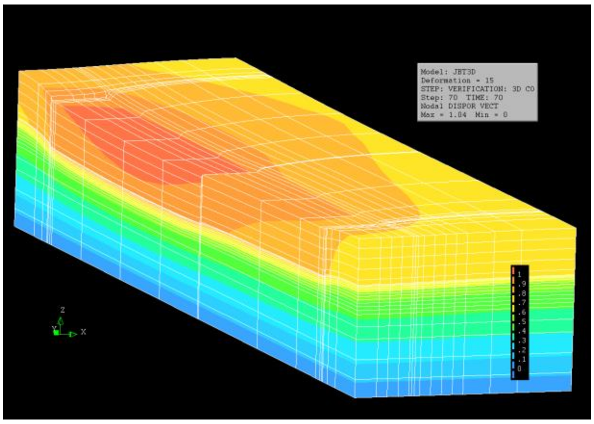

Figure 12 Calculated vertical settlements at end of 1970, including the settlements due to the old fill. Deformed mesh is magnified by a factor of 15. Eastern part toward top left.

4 CONCLUSTIONS

A new calculation tool, based on the finite element method, for time dependent 3D settlement predictions is presented.
The tool may be used to consider the following 3D effects:

- Horizontal pore water flow in two directions
- Stiffness and geometry dependent load (stress) distribution with depth
- Spatial variation in soil properties
- Spatial variation in steady-state pore pressure, which also may vary with time
- More complex soil models accounting for general 3D stress-strain relationships. A 3D material model developed at NTNU will be implemented in 2016

The calculation tool is an integrated part of the Novapoint GeoSuite Toolbox. This means that all relevant project data (e.g. field, laboratory data and soil layering) may be stored in GeoSuite Presentation, imported into BIM (Building Information Modelling) and then used as basis for input to the 3D calculation model.

5 ACKNOWLEDGEMENT

The authors acknowledge the funding of The Research Council of Norway and the technical and financial contributions of the GeoSuite alliance partners: Geovita AS, ViaNova Systems AS, Multiconsult AS, Norconsult AS, ViaNova GeoSuite AB, AG Programutveckling Ekonomisk Förening (AGEF), SINTEF Building and Infrastructure, the Norwegian University of Science and Technology (NTNU), the Norwegian Public Roads Administration (NPRA), the Norwegian Railway Administration (JBV), Skanska AS, Cowi AS, Sweco AS, Rambøll AS and NGI. The contributions A. Rosenquist af Akershult from Vianova GeoSuite AB are gratefully recognized.

6 REFERENCES

Andersen, K.H. & Clausen C.J.F. (1975). A fifty-year settlement record of a heavy building on compressible clay. Settlement of Structures, Conference. Cambridge 1974. Proceedings, pp. 71-78. London, Pentech Press

Janbu, N. (1985). Soil models in offshore engineering. The 25th Rankine lecture. Geotechnique, Vol. 35, No. 3, London, 241-281.

Tetgen (2015). A quality tetrahedral mesh generator and a 3D delaunay triangulator. www.wias-berlin.de/software/tetgen/

Zienkiewicz, O.C. (1977). The finite element method. McGraw-Hill (U.K.)

Potts, D.M. & Zdravlovic, L. (1999). Finite element analysis in geotechnical engineering. Theory. Thomas Telford Ltd (U.K.)

Lacasse, S. Jostad, H.P., Athanasiu C., L'Heureux, J.-S., Sandene. T. and Liu Z.Q. (2013). Assistance for the calculation of settlement. GeoMontréal 2013. Canadian Geotechnical Conference.

Jostad, H.P. & Engin, H.K. (2013). Investigation of different solution strategies for non-linear 3D consolidation problems. Proceedings of the 3rd International Symposium on Computational Geomechanics (COMGEO III), Krakow, Poland, 21-23 August, 2013.

Svanø, G. Christensen, S. & Nordal, S. (1991). A soil model for consolidation and creep. Proc., 10th European Conf. on Soil Mechanics and Foundation Engineering, Vol. 1, Balkema, Rotterdam, Netherlands, 269–272.