The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM)
ISSN: 2602-3199

The Eurasia Proceedings of Science, Technology, Engineering and Mathematics (EPSTEM), 2025

Volume 38, Pages 30-39

IConTES 2025: International Conference on Technology, Engineering and Science

Numerical Modeling Creep of Two Piles Under Constant Static Loading Using ANSYS

Ghouilem Kamel
Mouloud Mammeri University of Tizi-Ouzou

Belmihoub Hacene
University of Bouira

Abstract: This presents study describes a procedure for modeling and prediction the secondary creep behavior called (Norton Law) of a two concrete structure piles under constant static loading with taking into account the soil structure interaction, during 20000 hours of vertical static loading. In this item, we have applied the Finite Element Method (FEM) based on a Model using the computer code ANSYS® 14.0 shows in equation 16 heading (Norton Law). Contact problems fall into two general classes: rigid-to-flexible and flexible-to-flexible. In general, any time a soft material comes in contact with a hard material, the problem may be assumed to be rigid-to-flexible. To model a contact problem, you first need to identify the parts to be analyzed for their possible interaction. If one of the interactions is at a point, the corresponding component of your model is a node. If one of the interactions is at a surface, the corresponding component of your model is an element. The finite element model recognizes possible contact pairs by the presence of specific contact elements. These contact elements are overlaid on the parts of the model that are being analyzed for interaction. ANSYS ® 14.0 software has been used to perform the numerical calculation in this paper

Keywords: Pile, Creep, Concrete structure, ANSYS, Finite element method (FEM)

Introduction

The purpose of piling a foundation is to transmit forces through a weak stratum to a lower stronger stratum having sufficient bearing capacity to support the structure. The analysis of laterally and vertically loaded piles involves both the response of the soil movement of the pile and the bending deflection. Soil stiffness resists the pile. The initial response of the soil is nearly linear elastic but, as the lateral deflection increases, the soil starts to behave in a more plastic response and the stiffness reduces. An analytical solution may assume a linear elastic soil continuum or an elastic-plastic soil. A number of analytical solutions have been developed by various authors to estimate the response of piles and pile groups which are laterally loaded. These analytical solutions have been developed to provide the design engineer with a realistic and economic method of dealing with laterally loaded piles.

Soil-structure interaction has been extensively investigated regarding axial vertical loads in deep foundations. At the same time, in several circumstances, piles need to be designed to support major static lateral and vertical loads, as a result of the action of, water flow, horizontal earth pressure, earthquakes, and traffic movement. Bridges (Kim, & Jeong, 2011), tall buildings, transmission lines, retaining walls (Gil-Martín et al, 2016), offshore structures (Michel et al, 2018; Arany et al, 2017), wharfs, are a few examples of structures in which lateral loads assume primary significance.

Creep Behavior and Creep Material Model

Creep is a rate-dependent material nonlinearity in which the material continues to deform under a constant load. Conversely, if a displacement is imposed, the reaction force (and stresses) diminish over time (stress relaxation; see Figure 1. The three stages of creep are shown in Figure 2. The program has the capability of modeling the first two stages (primary and secondary). The tertiary stage is usually not analyzed since it implies impending failure

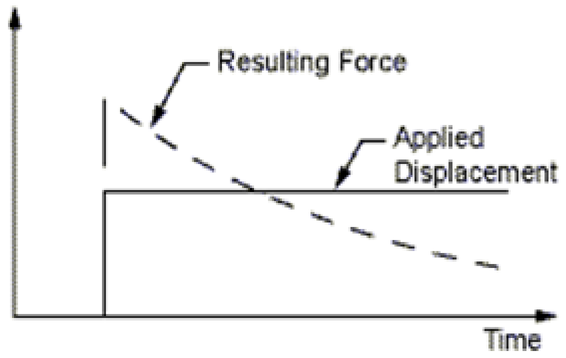

Figure 1. Stress relaxation (ANSYS, 2016)

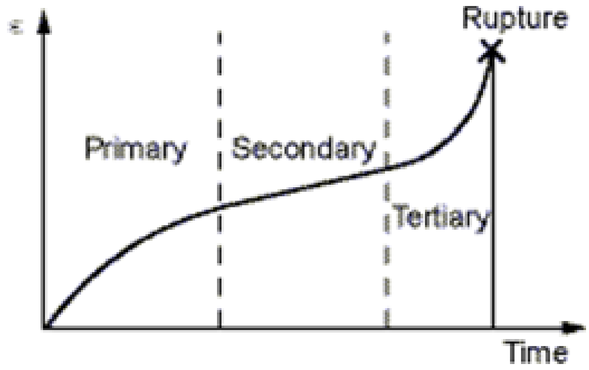

Figure 2. Creep strain stage (ANSYS, 2016)

Creep is important in high temperature stress analyses, such as for nuclear reactors. For example, suppose you apply a preload to some part in a nuclear reactor to keep adjacent parts from moving. Over a period at high temperature, the preload would decrease (stress relaxation) and potentially let the adjacent parts move. Creep can also be significant for some materials such as prestressed concrete. Typically, the creep deformation is permanent. The program analyzes creep using two time-integration methods. Both are applicable to static or transient analyses. The implicit creep method is robust, fast, accurate, and recommended for general use. It can handle temperature dependent creep constants, as well as simultaneous coupling with isotropic hardening plasticity models. The explicit creep method is useful for cases where very small time steps are required. Creep constants cannot be dependent on temperature. Coupling with other plastic models is available by superposition only.

Implicit Creep Procedure

The basic procedure for using the implicit creep method involves issuing the TB command with Lab = CREEP, and choosing a creep equation by specifying a value for TBOPT. The following example input shows the use of the implicit creep method. TBOPT = 1 specifies that the primary creep equation for model 1 is used. Temperature dependency is specified using the TBTEMP command, and the four constants associated with this equation are specified as arguments with the TBDATA command bellow:

```
TB,CREEP,1,1,4,1
TBTEMP,100
TBDATA,1,C1,C2,C3,C4
```

For most materials, the creep strain rate changes significantly at an early stage. Because of this, a general recommendation is to use a small initial incremental time step, then specify a large maximum incremental time step by using solution command DELTIM or NSUBST. For implicit creep, you may need to examine the effect of the time increment on the results carefully because the program does not enforce any creep ratio control by default. You can always enforce a creep limit ratio using the creep ratio control option in commands CRPLIM or CUTCONTROL,CRPLIMIT. A recommended value for a creep limit ratio ranges from 1 to 10. The ratio may vary with materials so your decision on the best value to use should be based on your own experimentation to gain the required performance and accuracy. For larger analyses, a suggestion is to first perform a time increment convergence analysis on a simple small size test.

Explicit Creep Procedure

The basic procedure for using the explicit creep method involves issuing the TB command with Lab = CREEP and choosing a creep equation by adding the appropriate constant as an argument with the TBDATA command.

TBOPT is either left blank or = 0. The following example input uses the explicit creep method. Note that all constants are included as arguments with the TBDATA command, and that there is no temperature dependency.
TB, CREEP,1 TBDATA, 1,C1,C2,C3,C4, ,C6

For the explicit creep method, you can incorporate other creep expressions into the program by using User Programmable Features (see the Guide to ANSYS User Programmable Features). For highly nonlinear creep strain vs. time curves, a small time step must be used with the explicit creep method. Creep strains are not computed if the time step is less than 1.0e-6. A creep time step optimization procedure is available (AUTOTS and CRPLIM) for automatically adjusting the time step as appropriate. In order to modeling the creep bahavior, we have introduced the model equation called (Strain Hardening » model Eq. (1). It is considered that the material is isotropic, and the basic solution method used is that of Newton-Raphson.

$$\dot{\varepsilon}_{cr} = C_1 \sigma^{C_2} t^{C_3} e^{(-C_4/T)} \quad (1)$$

With: $\dot{\varepsilon}_{cr}$: Rate Creep strain, σ : Equivalent stress, t: Time at end of sub –steps), C_1, C_2, C_3, C_4 : Materials Creep parameters (see Table 1), T : temperature (K). The following tables illustrate the materials creep parameters of Clay, Rock and Concrete introduced in the processing creep modeling law. In this studies, parameters is taken equal zero (C4 = Q/K = 0).

Table 1. Elastic and creep parameters properties of the soil layers and Concrete parameters

	Elastic			Creep parameters		
Rock	Young's Modulus	υ	Density	C1(s-1)	C2	C3 (MPa-1)
	2.4×10^{8} Pa	0.45	2.10^{3}kg.m^{-3}	7e-20	2.19	-0.98
Concrete	3.10^{10} Pa	0.23	$2.5.10^{3}$kg.m^{-3}	41e-8	1.48	-0.68
Clay	22.10^{6} Pa	0.25	18 kN.m^{-3}	3.33e-6	1.18	-0.54

The following windows (Figure.3) illustrate the interface procedure for introduction creep equation into the model. The introduction of the Strain Hardening creep equation Law (Eq1) in the model, are illustrated in figures 4 and 5 below. It should be noted that the maximum values of the creep strains in the X direction are concentrated at the level contact zone Clay - rocky substratum. In the y-axis, it can be seen that the minimum creep strain is indicated at the level of the tip pile zone

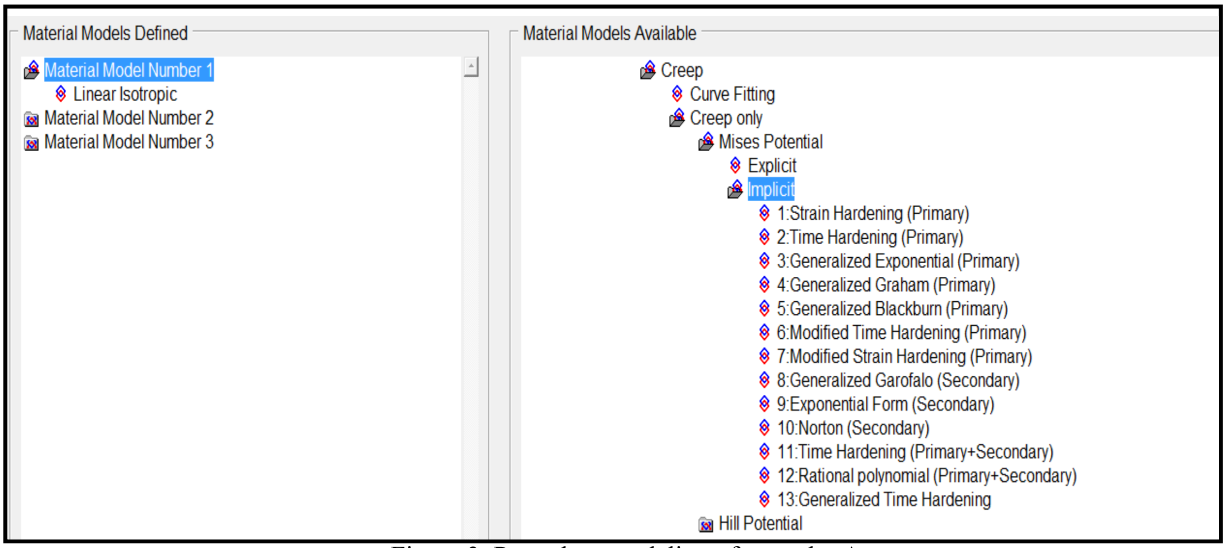

Figure 3. Procedure modeling of creep by Ansys

Contact Behavior and Finite Element Formulation

Using contact elements in a Finite Element Method (FEM) simulation is seldom a simple, painless experience. Changing contact between parts is a common phenomenon which, in some cases, can be treated with rigorous mathematical theory. Formulae for special cases can be found in mac Hine design textbooks (Juvinall, & Marshek1991, Shigley 1972, Timoshenko & Goodier 1970), two spheres, two parallel cylinders, cylinders on a flat plat e, gear teeth, roller bearings, etc. However, these theories only describe the stress in the contact region.

Finding the best choices for contact elements, element options, solver, and solution options can drastically import e the model's performance and reduce the analyst's frustration with a contact simulation model.

This section presented the steps to simulate the contact elements between two the soil and structure (pile) using ANSYS® software. Moreover, it gives more details about the types of contacts and algorithms which are used in this software. The ANSYS® Finite Element Method (FEM) program offers a variety of elements designed to treat cases of changing mechanical contact between the parts of an assembly or between different faces of a single part. Contact elements can be grouped into four general categories based on increasing levels of sophistication or complexity (Ansys, 2011).

Node – to Node Gap Elements: CONTACT12, LINK10, COMBIN40, CONTAC52, CONTA178

Elements CONTACT12 and CONTACT52 simulated node – to node contact in two and three dimensions, respectively. Initially the elements were based upon a penalty function approach and elastic Coulomb friction model. COMBIN7, COMBIN40 and LINK10 elements are included in this list of point-to-point gap elements because they simply connect from one surface node to another, but these elements can have other complicated behaviors. COMBIN7 is a revolute joint element which has "stops" that act like the closing of a contact. COMBIN40 has gap and slider behavior, but also allows two parallel springs, and may have mass and damping for a dynamic simulation.

Node – to Line (or Slide-Line) Contact Elements: CONTAC26

The point-to-line gap element is available for two-dimensional problems only. The node point location on the flexible surface (that is, on the surface of a solid element mesh) is tested with respect to its proximity to some rigid target surface. The analyst defines:

- Stiffness of a closed gap
- The geometry (shape and location) of the rigid surface
- Friction coefficient (if any)
- Stiffness in the sliding direction

Node – to –Surface Contact Elements: CONTAC48, CONTAC49

Ansys. Inc. later developed the CONTACT 48 and CONTACT49 node –to- surface contact elements for general contact problems. The underlying technology is penalty based, but with Lagrange augmentation to enforce compatibility. The elements allow for large sliding. However, these elements are not compatible with higher-order solid elements because the mid-side nodes of the quadratic elements are not used for the contact faces. With these contact elements, the analys defines or selects:

- Stiffness of a closed gap (FKN)
- Formulation
- Penalty method
- Penalty + Lagrange multiplier
- Allowable penetration tolerance (FTOLN)
- Pinball region size (PINB)

Surface-to-Surface Contact Elements: TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174

The surface-to-surface gap elements overcome most of the limitations or restrictions of the other contact elements:

- Two- or three-dimensional systems
- Rigid or flexible target faces can be treated

In this work, it has been used two types of element for contact model as follows:

-"CONTA171": CONTA171 is used to represent contact and sliding between 2-D"target" surfaces (TARGE169) and a deformable surface, defined by this element. The element is applicable to 2-D structural and coupled field contact analyses.

- "TARGE169": TARGE169 is used to represent various 2-D "target" surfaces for the associated contact elements (CONTA171, CONTA172, and CONTA175). The following figures 4 and Figure 5 illustrates some modeling contact elements and PLANE 182 described by ANSYS.

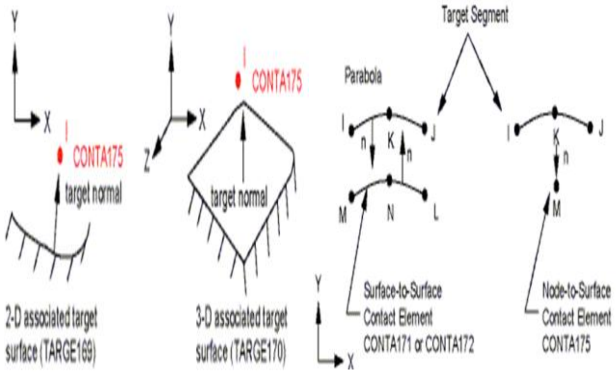

Figure 4. Element contact TARGE 169 and contact 170, 171, 172, 175 (ANSYS contact technology guide)

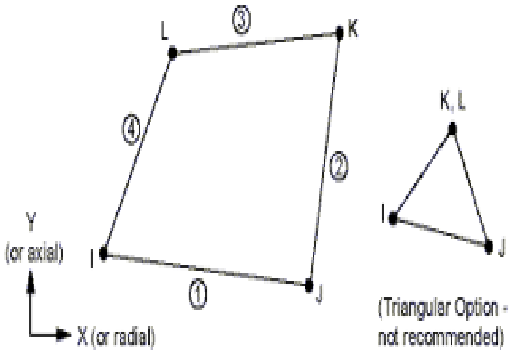

Figure 5. Element PLANE 182 (ANSYS element reference 2016)

The following Figure 6 and Figure 7 bellow illustrates the modeling interface and contact generation soil-Piles with **TARGET169** elements in the **Surface-to-surface contact elements**e.

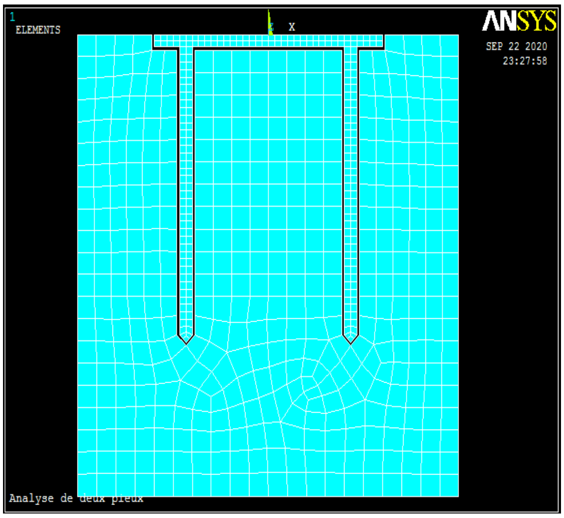

Figure 6. Modeling interface soil pile

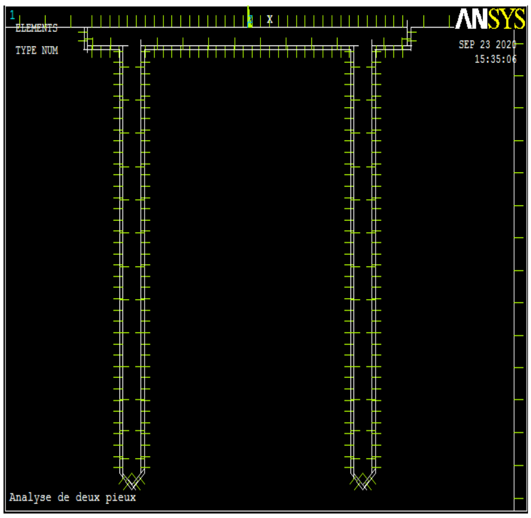

Figure 7. Contact generation between soil structure

Results and Discussion

Finite element analyses of piles in soil are presented by (Ottaviani, 1975; Randolph,1981; Smith I. M. & Griffiths 1988; Selby & Arta,1991), to deal with laterally and static loaded piles. Today powerful computer packages such as Program for Ansys® Finite Element Analysis have been developed which are capable of analyzing 1, 2 and 3 dimensional problems with various types of element. The package uses the Virtual Work theory to evaluate nodal displacements due to applied load vectors, then from the nodal displacements, the strains and stresses are calculated.

ANSYS® is engineering software, worldwide used by researchers for simulation. It develops general purpose of finite element analysis. To create the finite element model in ANSYS there are multiple tasks that have to be completed for the model to run properly. Models can be created using command prompt line input or the Graphical User Interface (GUI). This section describes the different tasks and entries into used to create the Finite Element calibration model. Analysis by ANSYS includes two steps, namely a step of modeling and a calculation step. The first step is to model the finite element structure by choosing an appropriate item to the type of analysis to be performed, such as for example: As part of this work, we limited ourselves to address the problem in two-dimensional finite element discretized by element **PLANE182** with application of boundary conditions (see figure 8 below). In this model, the number of elements structural is approximately 2426 elements and 2429 nodes.

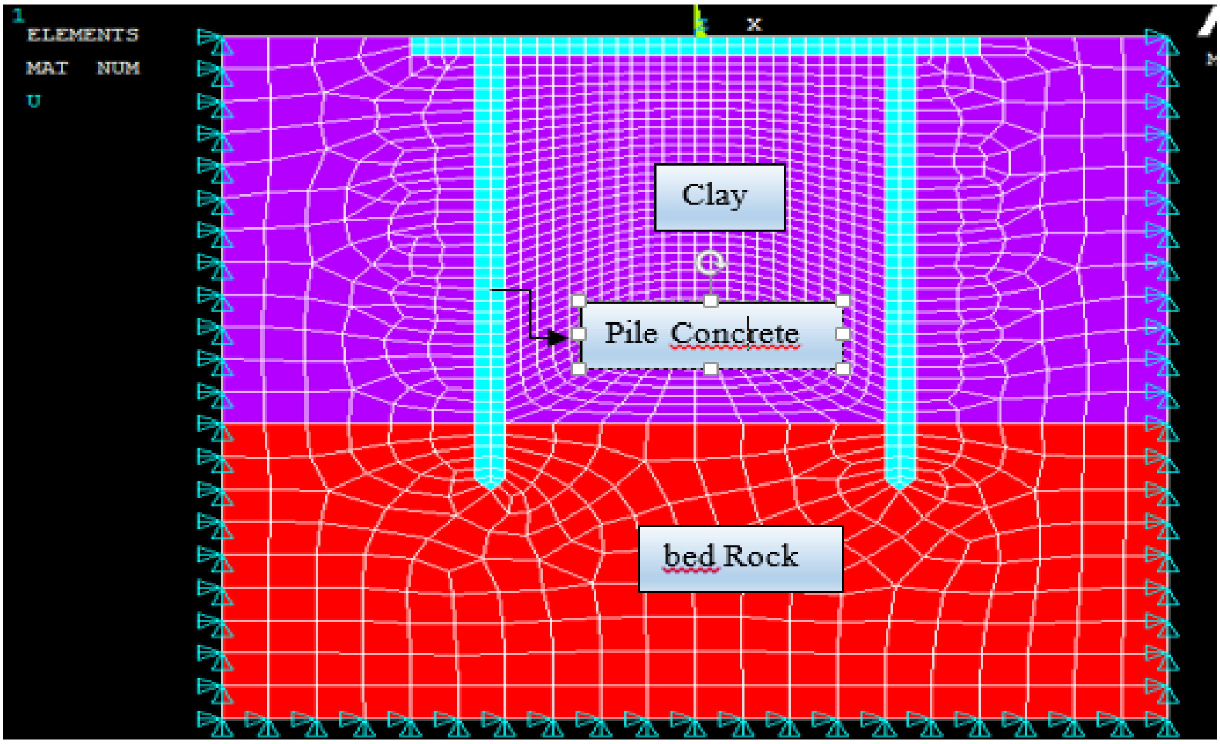

Figure 8. Two – Pile group modeling

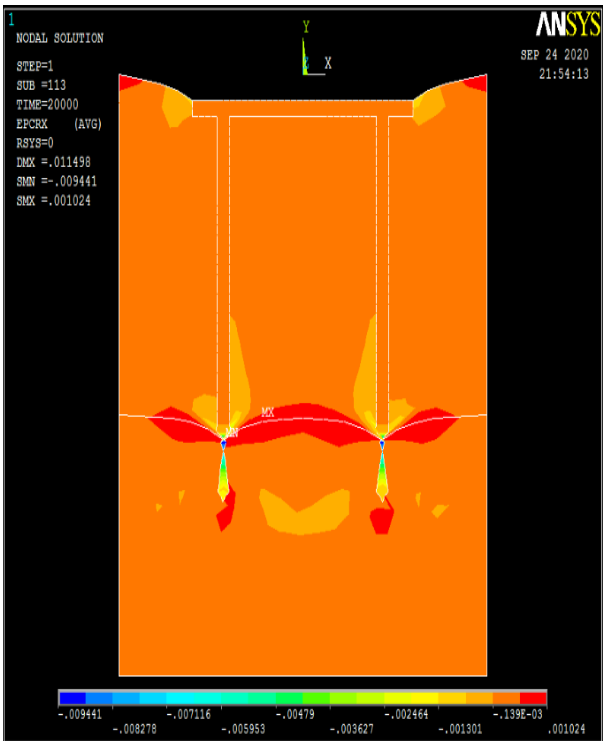

Figure 9. Contour plot of Creep strain ε_X^{cr}

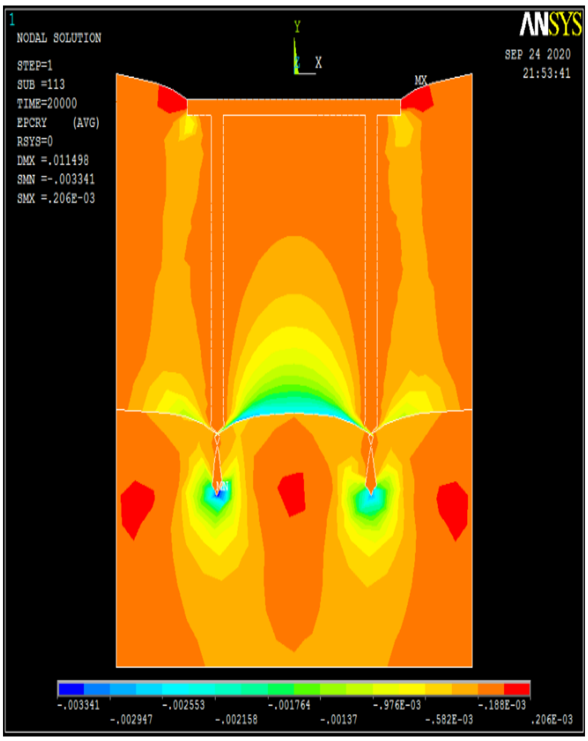

Figure 10. Contour plot of Creep strain ε_Y^{cr}

- The 2nd step is divided into three, namely:

1. Step pre-processor: who is to introduce the geometry of the problem, material properties and Boundary conditions.
2. While in the solution phase, we choose the type of analysis that must be performed.
3. Finally, the results of the completed solution are observed in the post-processing step.

In order to conduct time-dependent numerical modeling, the finite element code ANSYS occurs the different results shown in the Figure 9 and Figure 10. This Item illustrates a time-dependent behavior of two - pile group subjected to a constant static loading during 20000 hours. Figure 9 and Figure 10 illustrate the numerical contour plot results of elastic strain (ε_X^{cr}) and (ε_Y^{cr}) respectively.

The following Figures 11 and Figure 12, illustrates the contour plot elastic shear stress (τ_{XY}^{el}) and the contour plot of Shear Creep strain (ε_{XY}^{cr}) in the soil pile model respectively. It is observed that the shear stresses are focused at the level of the surface of the anchoring zone of the piles in the bedrock. As shown in Figure 11 and Figure 12 that the maximum and minimum values of elastic and creep strain deformations are set in the soil and pile head contact zone.

The following figures illustrate the creep strain in the structure material as (Concrete pile, clay and bedrock), according to the creep equation cited in Equation. 1 Above, called: Strain Hardening » model.

- Clay Material (Figure.13)
- Concrete Pile (Figure 14)
- Bedrock (Figure 15)

These figures illustrate that the Creep Curves Strain is characterized by two steps sum follows:

- 1st phase: 0 < t < 10000 Hours: we note that the experimental results and results numerical are similar in the concrete and bed rock material. But on the other hand, this step is reduced at the interval time: 0 < t < 4000 in the material clay
- 2nd phase: 10000 < t < 20000 hours: The experimental and theorical creep curve is briefly shifted in the three material structure constituting.

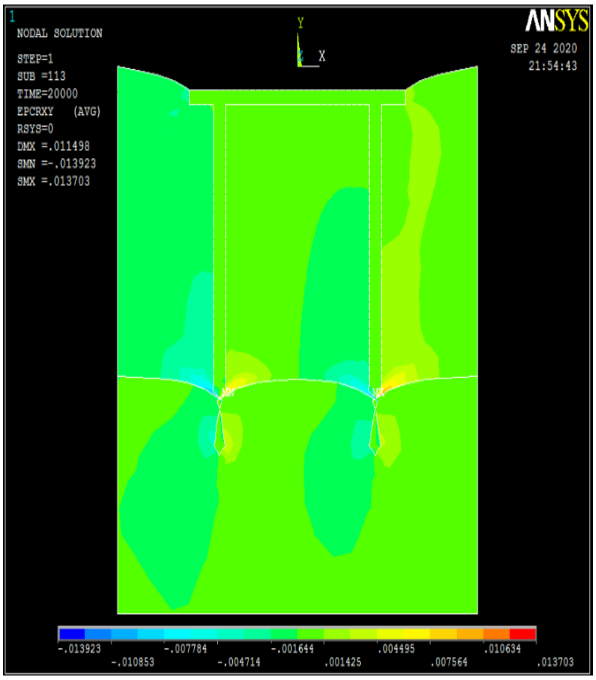

Figure 11. Contour plot of Shear Creep strain ε_{XY}^{cr}

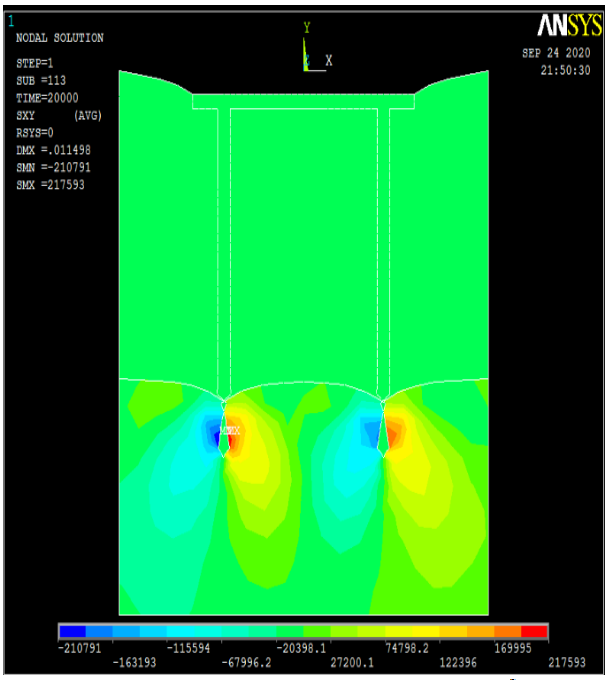

Figure 12. Contour plot of Shear stress (τ_{XY}^{el})

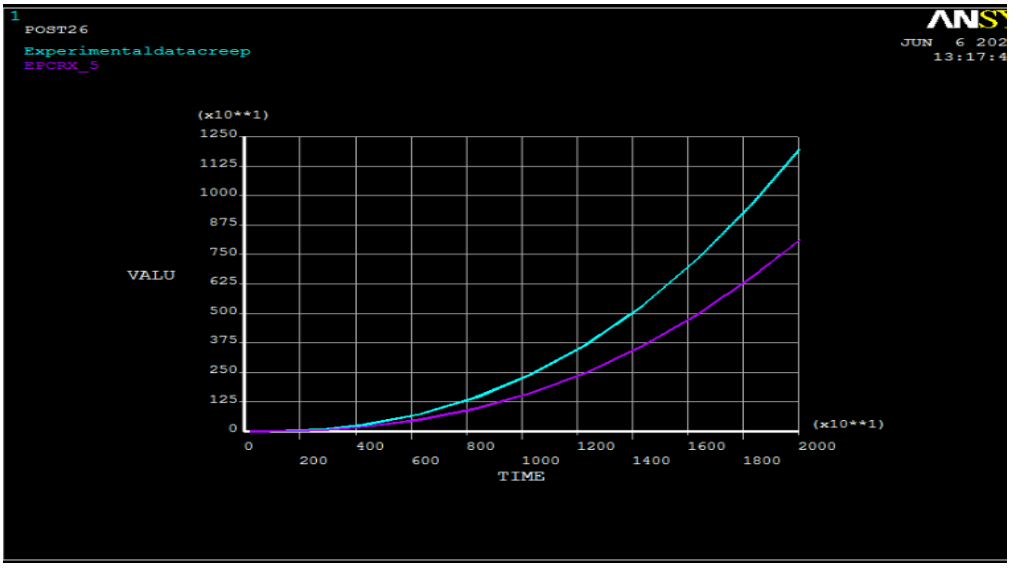

Figure 13. Creep strain curve (Clay)

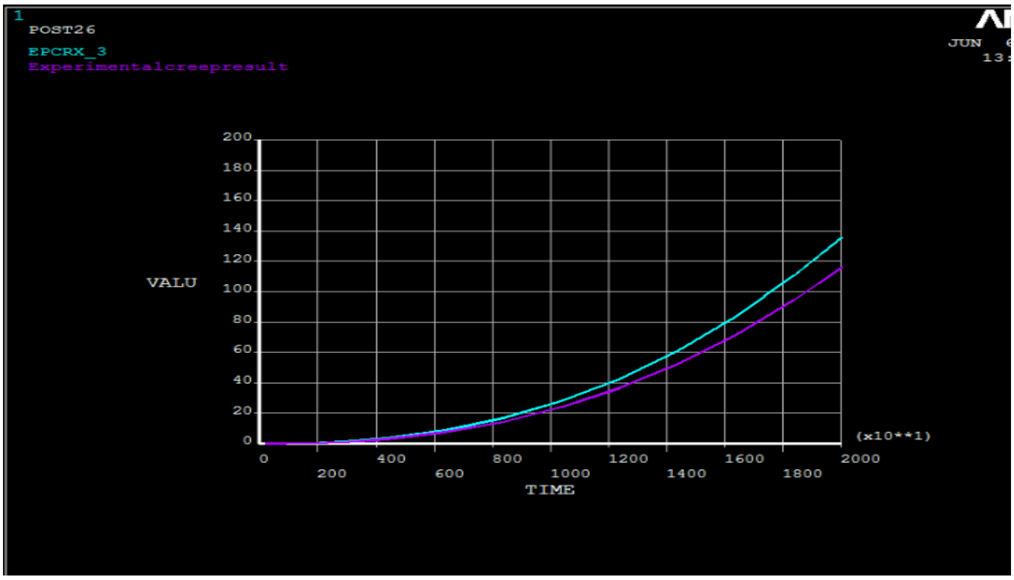

Figure 14. Creep strain curve of pile

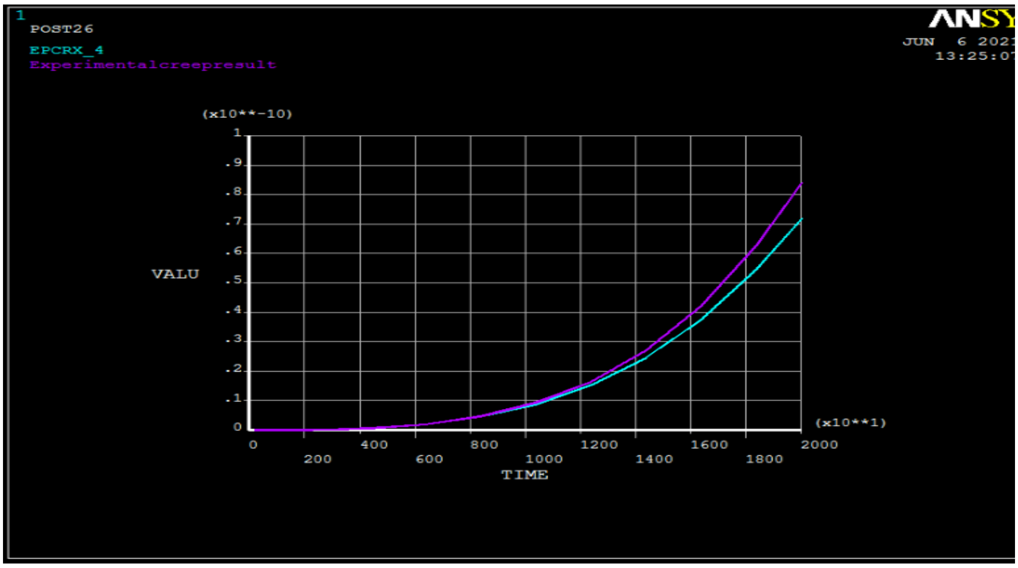

Figure 15. Creep strain curve (bed rock)

Conclusion

The behavior of civil engineering works such as retaining walls, reinforcements, tunnels and foundations is a soil-structure interaction problem. At present, it is commonly accepted that, within the framework of the study of the soil-structure interaction, the transmission of forces, from the structure to the ground, takes place through a thin layer of soil in contact with the structure called "interface". The interface is the site of complex mechanical phenomena: it generates locations of deformation and significant stress concentrations. These phenomena are strongly influenced by the mechanical characteristics of the granular soil and of the structural element, which are generally very contrasting. Breakage is often checked within this layer. He analyzed that, for a correct modeling of a geotechnical structure built in clay and bed rock, it is important to take into account the particular behavior of this interface. The modeling of the soil-structure interface is of capital importance in understanding its influence on the behavior of structures. The ANSYS software offers us this opportunity to achieve the junction between two distinct facets, with appropriate elements. The mains of these papers are to cite the creep behavior with different methods of contact modeling between the interface soil structures. Creep curves strain and shear creep curves numerical results of the different constitutive materials if structure are show's in the above figures. in terms of perspective, we intend to compare the numerical results to the experimental results.

Scientific Ethics Declaration

* The authors declare that the scientific ethical and legal responsibility of this article published in EPSTEM journal belongs to the authors.

Conflict of Interest

* The authors declare that they have no conflicts of interest

Funding

* This research received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

Acknowledgements or Notes

* This article was presented as a poster presentation at the International Conference on Technology, Engineering and Science (www.icontes.net) held in Antalya/Türkiye on November 12-15, 2025.

* This research paper has been developed and produced at the laboratory of geotechnical Environnement and Amenagement (LGEA) of the University of Mouloud Mammeri Tizi – Ouzou, Algeria, for that I would like to thank all reviewers who contributed to its final development.

References

ANSYS, Inc. (2016). *ANSYS contact technology guide* (Release 14.0). ANSYS, Inc.

ANSYS, Inc. (2016). *ANSYS element reference: General element features* (Release 14.0). ANSYS, Inc.

Arany, L., Bhattacharya, S., Macdonald, J., & Hogan, S. J. (2017). Design of monopiles for offshore wind turbines in 10 steps. *Soil Dynamics and Earthquake Engineering, 92*, 126–152.

Coen, M. H., Mozer, M. C., Hasha, R., & Flanagan, J. L. (2002). Room service, AI-style. *IEEE Intelligent Systems, 14*(2), 8–19.

Curnier, A., & Alart, P. (1988). A generalized Newton method of contact problems with friction. *Journal of Theoretical and Applied Mechanics, 7*, 145–160.

Ghouilem, K., Mehaddene, R., & Kadri, M. (2017). Contact friction simulating between two rock bodies using ANSYS. *International Journal of Engineering Research in Africa, 29*, 1–9.

Gil-Martín, L. M., Carbonell-Márquez, J. F., Fernández-Ruíz, M. A., & Hernández-Montes, E. (2016). Theoretical and experimental short-term behavior of non-symmetrical wall pile retaining systems. *Engineering Structures, 112*, 172–183.

Juvinall, R. C., & Marshek, K. M. (1991). *Fundamentals of machine component design* (2nd ed.). John Wiley and Sons.

Kim, Y., & Jeong, S. (2011). Analysis of soil resistance on laterally loaded piles based on 3D soil-pile interaction. *Computers and Geotechnics, 38*(2), 248–257.

Michel, P., Butenweg, C., & Klinkel, S. (2018). Pile-grid foundations of onshore wind turbines considering soil-structure-interaction under seismic loading. *Soil Dynamics and Earthquake Engineering, 109*, 299–311.

Ottaviani, M. (1975). Three-dimensional finite element analysis of vertically loaded pile groups. *Géotechnique, 25*(2), 159–174.

Randolph, M. F. (1981). The response of flexible piles to lateral loading. *Géotechnique, 31*(2), 247–259.

Selby, A. R., & Arta, M. R. (1991). Three dimensional finite element analysis of pile groups under lateral loading. *Computers & Structures, 40*(5), 1329–1336.

Shigley, J. E. (1972). *Mechanical engineering design* (2nd ed.). McGraw-Hill.

Simo, J. C., Wriggers, P., & Taylor, R. L. (1985). A perturbed Lagrangian formulation for the finite element solution of contact problems. *Computer Methods in Applied Mechanics and Engineering, 50*(2), 163–180.

Smith, I. M., & Griffiths, D. V. (1988). *Programming the finite element method* (2nd ed.). John Wiley & Sons.

Timoshenko, S. P., & Goodier, J. N. (1970). *Theory of elasticity* (3rd ed.). McGraw-Hill.

Wriggers, P., & Simo, J. C. (1985). A note on tangent stiffness for fully nonlinear contact problems. *Communications in Applied Numerical Methods, 1*(4), 199–203.

Author(s) Information

Ghouilem Kamel
Mouloud Mammeri University of Tizi-Ouzou
Ummto, BP 17 RP, Tizi-Ouzou 15000, Algeria
Contact e-mail : *kamel.ghouilem@ummto.dz*

Belmihoub Hacene
University of Bouira (Mohand Oulhadj University of Bouira), Bouira, Algeria

To cite this article:

Kamel, G., & Hacene, B. (2025). Numerical modeling creep of two piles under constant static loading using ANSYS. *The Eurasia Proceedings of Science, Technology, Engineering and Mathematics (EPSTEM), 38*, 30-39.