ANALYSIS FOR GROUND ANCHOR USING 2D SIMULATION MODEL BY MIDAS/GTS SOFTWARE

MUNIRA SUNGKAR[1,2]*, JASON WU[2]

[1]*Civil Engineering Department, Faculty of Engineering, Syiah Kuala University, Banda Aceh, Indonesia*
Email: sungkar.munira@yahoo.com

[2]*Civil Engineering Department, Chung Hua University, Hsinchu, Taiwan 707 Wufu Rd. Sec. 2, Hsinchu City, Taiwan 30012*

Abstract

Ground anchors are commonly exposed to both acidic and alkaline ground conditions depending on ground mineral content, soil/rock type, age and groundwater flow path. It is essential to understand the limitations to ground anchorage materials especially when exposed to aggressive ground conditions whilst in a stressed state. This study discusses about ground anchors and analysis of the ground anchor failure trigger by corrosion, which analysed by using difference software in previous study (ABAQUS) compare with simulation model analysed by 2D Midas/GTS. Numerical techniques may be very efficient for investigating the load transfer of ground anchors. The results of anchor failures in numerical modeling are different from previous study (field test results), practiced by first cement crack and then wire break. Recommendations are given for applying FEM (finite element method) analysis to the design of anchored stabilization systems, and also for analyzing design variables, and design loads, to achieve more efficient and cost-effective anchored stabilization systems. Several models or simulations are presented to illustrate the displacement and the load that applied to the anchor which analyzed by Midas/GTS software. However, the condition that the numbers of occurrence correspond to load of failures at the time of damages is caused as the consequence of material and model deviation.

Keywords: ground anchor, simulation model, anchor failures, corrosion, Midas/GTS software

Introduction

Ground anchors have been used in the construction of retaining walls and permanent tie-down systems since the turn of the 19th century. The first documented use of ground anchors occurred in 1938 to tie down the reservoir side of Cheurfas Dam in Algeria (Schnabel, 1982). After World War II, uses such as support of temporary cuts, excavation support, landslide mitigation, and dam improvements were implemented. Europe was in the forefront of application.

A system for maintaining anchors needs to be constructed including inspection, integrity investigation, and remedial measures. The objective of the anchor maintenance is to keep the safety and security of the slopes. The system is able to evaluate the slope conditions quantitatively, objectively and properly manage them while using the ground anchors safely for a long time. In addition, also to check the ground anchor durability for long-term purposes in terms of maintenance of ground anchors.

The use of steel ground anchors is often constrained by overall durability in placement (due to weight), and the difficulty in maintaining tension levels in the anchor. In the early years of anchor work, execution and rust-prevention technologies were still under development, and some of the anchors constructed during this period have aged and lost their performance, resulting in slope deformation and exposure of broken anchor heads. The relevant drawings, specifications, and construction records were not stored because anchors were regarded as an auxiliary method for stabilizing slopes (Miyatake, et. al., 2003).

Structurally, anchors stabilize slopes by introducing initial tensioning force during construction, which is gradually released over time through various mechanisms. Thus, anchors must not only be carefully designed and constructed but also vigilantly maintained, especially by monitoring residual tensioning force and checking integrity.

Corrosion in steel tendon ground anchors occurs as a consequence of in-homogeneities or impurities in the steel tendon or grout, or by the existence of salts, sulphates and other dissolved solids present in grout mixtures, soils or groundwater (Sentry, et.al., 2007). Steel tendon corrosion occurs locally where the tendon intersects a crack

in the surrounding grout, or as a result of damage to the corrosion protective sheaths (Weerasinghe and Adams, 1997).

For structural elements, a site to be corrosive if one or more of the following conditions exist for the representative soil and/or water samples taken at the site: Chloride concentration is 500 ppm or greater, sulfate concentration is 2000 ppm or greater, or the pH is 5.5 or less (Miyatake, et.al., 2003).

Methods

This study explains the methods used by previous researcher, Chang et. al. (2008) that conducted the study by using 2D ABAQUS software. In addition, based on the results of previous research it was attempted to create a simulation model using the software Midas/GTS. It aims to compare the results obtained by using 2 different methods or software.

Numerical simulation of ground anchors by ABAQUS software

A ground anchor is much more complicated than a pile mechanism since the load is transferred from the tendon to the grout, and then, to the soil. The load distribution and load transfer mechanism of a ground anchor must be clearly identified to properly design anchored retaining walls or anchored slopes. Furthermore, Kim, et.al. (2007) performed a numerical simulation of a ground anchor to investigate the load transfer mechanism in ground anchors. A procedure of finite-element modeling and beam-column modeling of ground anchors was proposed included the modeling of soil, grout, and strand tendon and the interface modeling of soil–grout and grout–strand in ground anchors.

Numerical techniques maybe very efficient for investigating the load transfer of ground anchors. Desai, et. al. (1986) performed a numerical simulation of a ground anchor to investigate the mechanism of stresses and deformations in an anchor–soil system.

Advanced research conducted by Chang, et.al (2008) by using 2D/3D ABAQUS software. To better understand the failure sequence of the anchor subjected to the pullout load, a 1D multi-degree-of-freedom ground anchor system is adopted herein to analyze the load-displacement relationships of the anchor. In-situ pullout load tests were conducted on two pre-stressed anchors. There are some assessments for this study include: pullout load test prediction for old anchor test and new anchor test. To ensure the feasibility and applicability of the pullout load test prediction in this study, the author uses the design geometry and the assigned soil parameters from the field report, as shown in Table 1.

Table 1. Parameters used in suggested constitutive law for finite element analysis (Chang, et.al., 2008)

Material	Parameter in use						
Steel tendon (Old anchor)	$E\left(kN/m^2\right)$	$D\left(ton/m^3\right)$	$A\left(m^2\right)$	*protruding* $L(m)$	*free* $L(m)$	*fixed* $L(m)$	ξ
	1.965×10^8	7.5	5.574×10^{-4}	0.2	0.0	21.0	0.001
Steel tendon (New anchor)	$E\left(kN/m^2\right)$	$D\left(ton/m^3\right)$	$A\left(m^2\right)$	*protruding* $L(m)$	*free* $L(m)$	*fixed* $L(m)$	ξ
	1.965×10^8	7.5	2.787×10^{-4}	0.2	10.0	10.0	0.001
Grouted-soil interface	$E\left(kN/m^2\right)$	$\tau_1\left(kN/m^2\right)$	$\tau_2\left(kN/m^2\right)$	$s_1(m)$	$s_2(m)$	υ_g	ξ_g
	1.2×10^7	320	300	$\left(\tau_1/K_{ini}\right)$	$1.2\times s_1$	0.17	0.015
Soil (sandstone)	k	$Pa\left(kN/m^3\right)$	n	R_f	$c\left(kN/m^2\right)$	$\varphi(\circ)$	$D_s\left(ton/m^3\right)$	υ_s	ξ_s
	2500	101.325	0.66	0.8	39	19.6	1.95	0.3	0.05

(w/c ratio=0.45 for grouts)

After validation of the pullout test simulations, the parameters to affect the predictions and the signals compared to the field data and the 2D finite element analysis modeling using ABAQUS as shown in Figure 1.

This paper compares the simulation models using 2D ABAQUS software and models are analyzed by 2D Midas/GTS software. The effect of corrosion on the anchor models can be determined and identified by using software Midas/GTS which created three types of treatments on anchor length by reducing the length of the anchor.
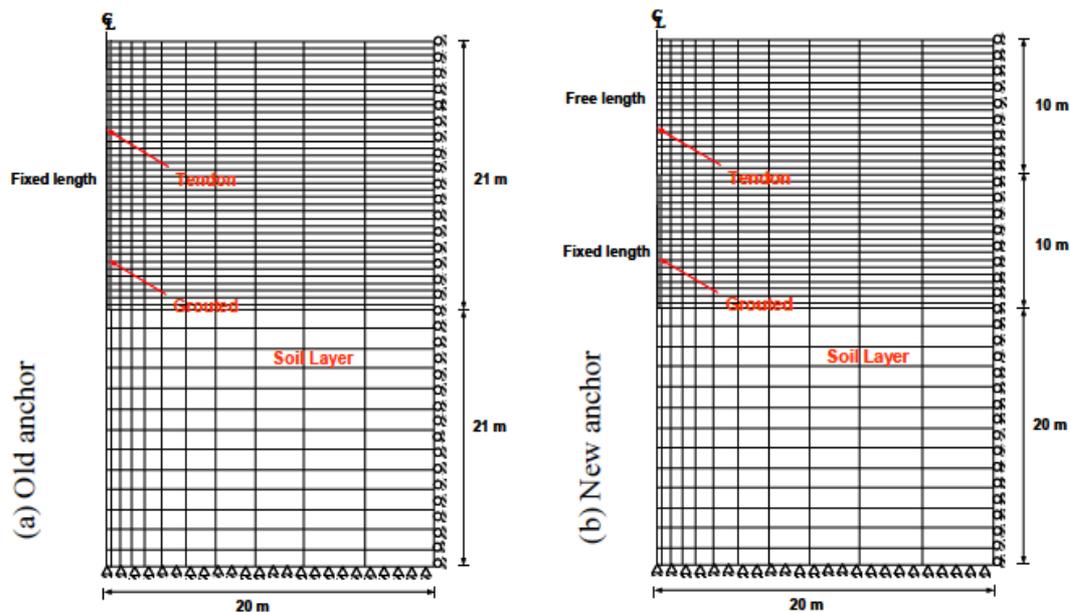
Figure 1. Layout of the 2D FEM mesh for ground anchor system. (Chang, et. al., 2008)

Midas/GTS Software

Midas/GTS is the finite element software for geotechnical structure and tunneling system, which combines the kernel of general finite element analysis with the professional requirements of geotechnical and tunneling system. The function of geometric modeling and mesh generation are very powerful, mainly embodying in the two aspects below:

1. The function of geometric modeling. GTS has operation interface of Windows style. It completes the operation through the clicking the menu, which can easily build the point, line, surface, and body.

2. The function of mesh generation. In addition to faster division of mapped mesh and free mesh, GTS can disjunctive, check, segregate grid manually and so on. It is especially convenient for mesh generation of complex model and inspection (Midas/GTS, 1989).

Modeling and Numerical Analysis using Midas/GTS Software

The soil were assumed to be elasto-plastic material and were computed by constitutive model by using the Mohr-Coulomb strength criterion, moreover tensile yield, elastic-plastic deformation, large deformation of rock mass were taken into account, self weight stress was considered in initial ground stress field of rock mass. The anchor material properties are as follows: the diameter of the anchor (D) = 0.075 m (75 mm), and the cross sectional area of the anchor (A) = 0.00441786467 m^2. The physical and mechanical parameters of soil and anchor were showed in Table 2.

Table 2. Material parameters used for this case study

Name	Sand	Weathered Rock	Grouted Soil	Anchor
Type	Axisymmetry	Axisymmetry	Axisymmetry	Truss
Modulus Elasticity, E_s (kPa)	2500	1.2×10^7	1.2×10^7	1.96×10^8
Poisson's ratio, ν	0.3	0.3	0.17	0.3
Unit weight, γ (kN/m^3)	20	21	20	78
Unit weight sat, γ_{sat} (kN/m^3)	21	22	21	
Cohession, C (kPa)	0	40	18	
Frictional angle, ϕ ($^\circ$)	30	30	30	
Initial stress parameter, Ko	1	1	1	

Models used were adopted from Chang et.al (2008) that have been modified with different soil parameters. The size of numerical model was, X = 20 m and Y = 42 m with one anchor (single anchor) which located in the weathered rock and grouted soil around the anchor, as shown in Figure 2.
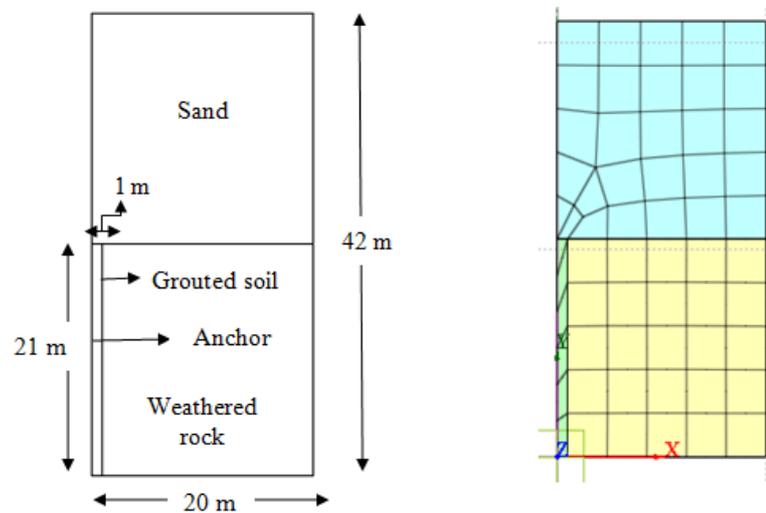

Figure 2. Lay out of model and finite element mesh for 1 (single) anchor length

Hereafter, there are three variations of anchor length, first, one (single anchor) which length is 21 m, and second is 2/3 of anchor length (14 m length) and the last is 1/3 of anchor length (7 m length). All the variations of anchor length are shown in Figure 3.
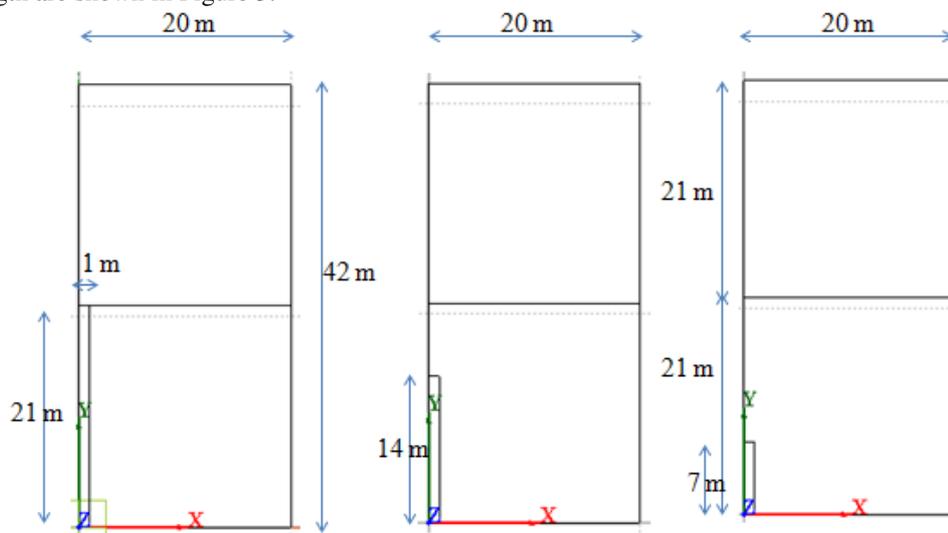
Figure 3. Three variations model of anchor length

The difference of the results from Chang et.al. (2008) with this report are including the soil parameters used and calculation by software for finite element analysis. The results are different, such as the difference of applied load (increment loads) and it implies to displacement occurred.

Results and Discussion

Finite element mesh
In the finite element analysis, the soil was simulated with 2D four-noded axisymmetry bricks elements. The ground anchor was modeled as axisymmetry case. Because of the symmetry about the anchor centerline, only a half plane of the cylinder was considered in the finite element analysis. A refined mesh was adopted to minimize

the effect of mesh efficiency on the finite element analysis, as illustrated in Figure 4. Base on Figure 4, it describes the example for model with mesh, boundary condition and by applying the load (uplift).

In order to simulate the behavior of soil and single anchor, which were resulted from analysis type: nonlinear static analysis, as shown in Figure 5, the displacement and 2D element strain and stress would be obtained.

Displacement, 2D element strain and stress analysis
All the calculation were executed in fully drained condition and the results for single anchor model which length is 21m encompassed the displacement (deformation) for Y direction (Dy), 2D element strain and 2D element stress, as shown from Figure 6.
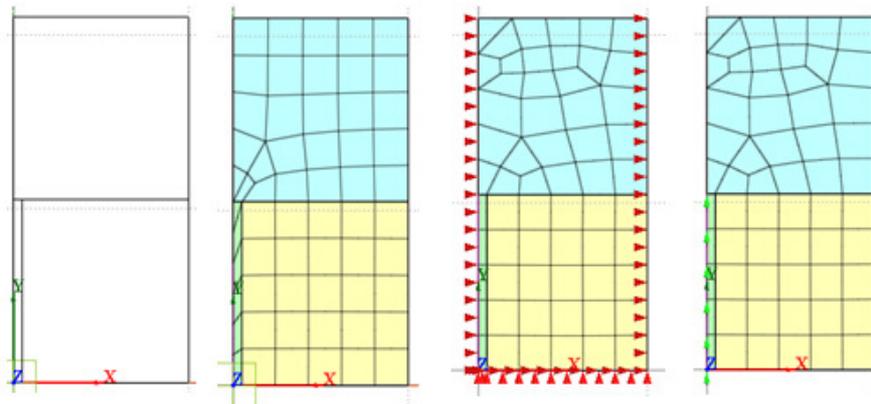

Figure 4. Example for model with mesh, boundary condition and by applying the load (uplift)
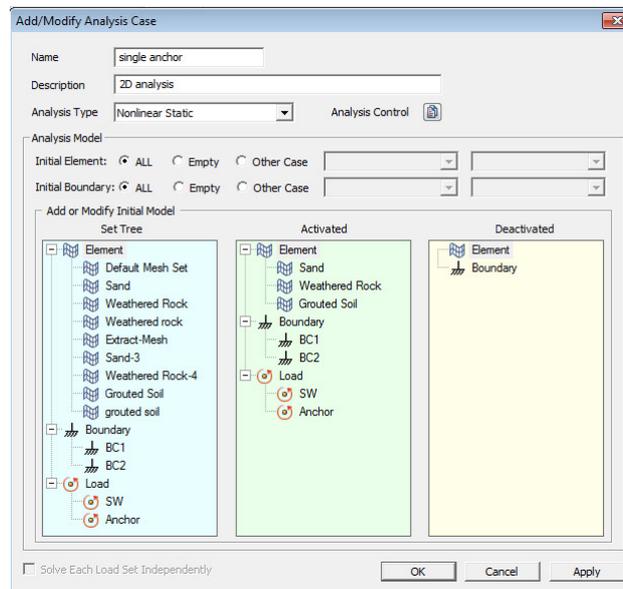

Figure 5. Analysis case for simulation

Vertical displacement distribution is shown in Figure 6, for relative displacement, Dy indicated in-plane sliding, it shows that the maximum displacement was 1.3m which occurred in sand soil (cohesionless soil). While in the area where the anchor located, the displacement relative smaller, it was 8 mm (0.08 m). It is marked by the red colour of the scheme that represent by 29.7% of the area that located in weathered rock.
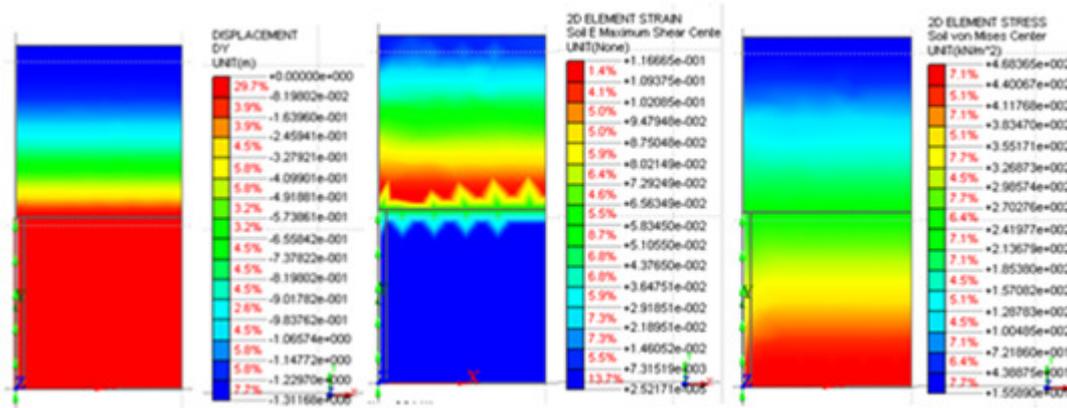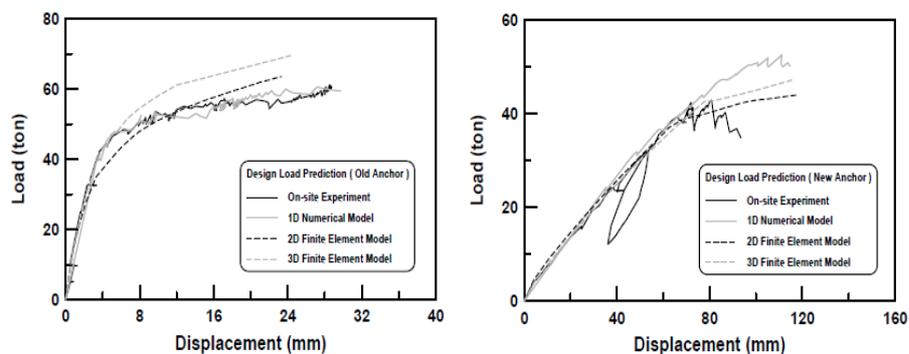
Figure 6. Displacement, 2D element strain and stress analysis case for simulation

Whilst the weathered soil has smaller maximum shear center 0.007, it noted by blue colour around 13.7% of area. This area has smaller maximum shear center due to installation of anchor. On the other hand, the Von Mises equivalent plastic strain ranged between 5.6×10^{-8} to 8.9×10^{-7} which the area where the strain changes is the area around the anchor.

According to Figure 6, it shows 2D element stress, soil Von Mises center. Von Mises stress is considered to be a safe haven for design engineers. Based on this information, it says that the design will fail if maximum value of Von Mises stress induced in the material is more than maximum strength of the material. It works well for most of the cases especially when material is ductile in nature. The Von Mises stress for this model varies from 44 kN/m^2 until 468 kN/m^2 and for the lower part is bigger than the upper part or the area. It indicates that the weathered rock has the maximum value of Von Mises stress.

Comparative finite element analysis
For finite element analysis, the ground anchored system can be simplified as a 2D axisymmetric structural system modeling by using computer program ABAQUS, whilst in this study used 2D Midas/GTS software for simulation and calculation finite element analysis. The difference results from Chang et.al. (2008) with this report are including the soil parameters used and calculation by software for finite element analysis, such as the difference of applied load (increment loads) and it implies to displacement occurred. Comparison of simulation model with ABAQUS and Midas/GTS software can be seen from Figure 7 for relationship of load-displacement curve.

(a) Comparison of load-displacement relationships for 1D modeling and 2D, 3D FEM solutions with the field data (Chang, et. al., 2008)
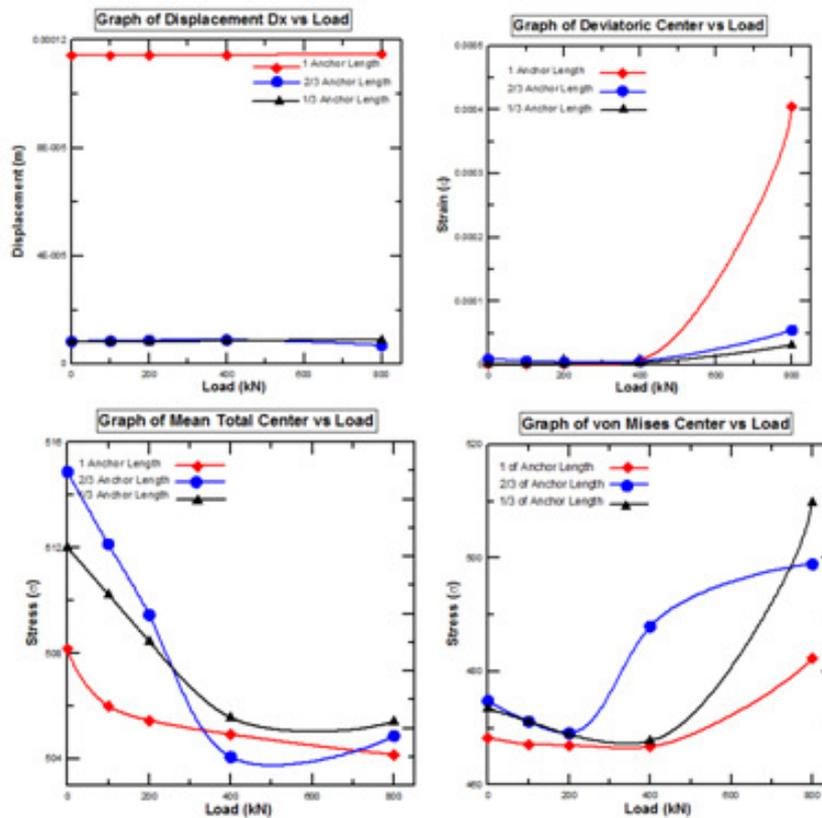
(b) Relationship of load-displacement, 2D element strain, and 2D element stress for three variations anchor length analysis by using Midas/GTS Software

Figure 7. Comparison of simulation model with (a) ABAQUS, and (b) Midas/GTS software

From the result of calculation analysis by using Midas/GTS software, besides displacement-load curve, it also can be seen 2D element strain and stress from the graph. The difference of displacement between this study and the previous study showed very big differences. This is because of using different soil parameters for both model and different load applied for uplift load and compression load.

By comparing the applicability, the characteristics and restraints, and the applications of these methods as shown in Table 4 for establishing load displacement curve of the pullout load test on ground anchors, it can be found which the proposed modeling is rather economic than others. If material properties and model parameters were carefully controlled, then this model perhaps can have great potential in a certain subjects of ground anchor study.

Tabel 4. Comparison of the methods between this study and previous study

Assessment	Applicability	Characteristics and restrains	Applications
Methods			
2D ABAQUS FE Analysis (Previous)	1. Provide rigorous numerical results and detail of the defections 2. Take into account the complexity of the structural system	1. Slow (5 minutes for 2D, 2 hours for 3D) 2. Relative complexity to used 3. Require pre-knowledge of the anchored system 4. Need careful material and model calibrations before simulations 5. Blind monitoring on existing defects	1. Sound basis for applying the monitoring technique 2. Forward modeling 3. Back analysis for data interpretations

2D Midas/GTS FE Analysis (This Study)	1. Proven and robust elements and nonlinear material models 2. More advanced analysis procedure 3. Parallel processing in equation solvers	1. Quite fast (2 minutes) 2. Solving non linear problem efficiently 3. 64-bits solver 4. Need careful material and model calibrations before simulations	1. Access to nonlinear models for structural elements 2. Continuous development 3. Dedicated user-support

Conclusions

The numerical solutions can predict the load-displacement relationships and the relationship between 2D element strain and stress by increment load, the conclusions are suggested as follows:

(1) The results of anchor failures in numerical modeling are different to previous study (field test results), however, the condition that the numbers of occurrence correspond to load of failures at the time of damages is caused as the consequence of material and model deviations.

(2) It is rather convenient to use the 2D analysis because the solution is fast enough but it is needed to select material parameters carefully and model calibrations before simulations.

(3) The selection of soil parameters and applied load is very influential on the results obtained in finite element analysis.

References

Chang, D.W., Cheng, S.H., and Lee, W. F., 2008, Simulation and Interpretation of ground anchor failures from acoustic monitoring test, 3rd Taiwan-Japan Joint Workshop on Geotechnical Natural Hazard, Keelung, Taiwan.

Kim, N.K. et. al., 2007, Numerical Simulation of ground anchors, Computers and Geotechnics, 34, 498-507.

Midas/GTS, 1989, Next Generation Solution for Geotechnical and Tunnel Engineering, Midas Information Technology Co, Ltd. (www.MidasUser.com)

Miyatake, H., et. al., 2003, A New Manual for Ground Anchor Inspection, Integrity Investigation and Remedial Measures.

Mothersille, et. al., 2006, Recent Development in Permanent Ground Anchor Technology, Geoserve Global Ltd. & Geotech Pty. Ltd, p.54.

Schnabel, H. Jr., 1982, Tiebacks in Foundation Engineering and Construction. McGraw-Hill, New York, NY.

Sentry, et.al., 2007, Durability of CFRP Strand for use in high capacity permanent ground anchors, Common Ground-10th Australia New Zealand Conference on Geomechanics. Brisbane.

Weerasinghe, D. R. B. & Adams, M. D., 1997, A Technical Review of Rock Anchorage Practice 1976-1996, Ground Anchorages and Anchored Structures, London, Thomas Telford.