Sciences & Technologie **B**– **N°24**, décembre (2005), pp. 69-74.

NUMERICAL MODELING OF AN L – SHAPED VERY STIFF CONCRETE RETAINING WALL

Reçu le23/10/2004 – Accepté le 06/12/2005

Résumé

Un modèle numérique développé en utilisant un programme d'éléments finis ; le Programme Plaxis, est présenté dans cet article, pour la simulation du comportement d'une structure rigide de soutènement en forme de L, supportant du sable. Pour la validation du model proposé, il est fait référence au comportement d'un model réduit testé en centrifugeuse. Le modèle numérique proposé a été utilisé pour prédire le comportement du prototype et de vérifier la validité du concept de modélisation utilisé. En prenant en considération la géométrie et les dimensions du modèle expérimental utilisé dans la centrifugeuse, les conditions de chargement et considérant comme model constitutive du comportement du sol 'the hardening soil model », les résultats de la simulation numérique obtenus en terme de mode et amplitude des déplacements du mur sont très proches aux résultats mesurés expérimentalement dans la centrifugeuse. Une très bonne concordance a également été obtenue entre la pression latérale mesurée et celle calculée par le model numérique proposé, en comparaison à l'approche classique.

Mots clés : *Modèle numérique, Plaxis programme, essais centrifugeuse, prototype, mur de soutènement.*

Abstract

A numerical model developed using the finite element Plaxis program is presented in this paper to simulate the behaviour of a stiff "L" shaped retaining wall supporting sand. For validating the proposed numerical model, reference was made to an experimental model tested in a centrifuge experiment. The proposed numerical model intended to predict the behaviour of a target prototype and to check the validity of the modelling concept used. Taking into account the geometry and dimensions of the wall with respect to the centrifuge scaling law, the loading conditions and considering the hardening soil model, it was found that the predicted pattern and magnitude of the wall displacements were close to the experimentally observed results. A good agreement between the measured and the numerically computed lateral pressures acting on the wall stem was also obtained with comparison to the classical approach.

Keywords: *Numerical model, Plaxis program, Centrifuge testing, Prototype, Retaining Wall.*

A. ROUILI [1]
Y. DJERBIB [2]
M. TOUAHMIA[3]

[1,3]Institut de Génie Civil, Centre Universitaire de Tébessa
Route de Constantine , Tébessa, Algérie
[2] School of Environment and Development, Department of Civil Engineering,
Sheffield Hallam University, City Campus,
Howard Street, Sheffield S1 1WB, UK.

ملخص

في هذا المقال تم تقديم نموذج رقمي تم تطويره بواسطة برنامج الحاسوب PLAXIS الذي يشتغل بمبدأ العناصر المتناهية. هذا البرنامج تم تطويره من أجل دراسة جدار إسناد على شكل حرف « L » ، تمت دراسة نموذج مصغر لهذا الجدار في تجربة مخبرية بواسطة آلة الطرد المركزي بالأخذ بعين الإعتبار الشكل، الأبعاد و الشروط الحدودية. تمت مقارنة نتائج النموذج الرقمي مع نتائج النموذج المخبري. بحيث تم التحصل على نتائج جيدة. كما قارنا مقياس التحركات و الضغط على الجدار بالنسبة كذلك للتقارب الكلاسيكي المبني على كيفية K_0 و طريقة رانكين.

الكلمات المفتاحية : جدار إسناد، تجربة آلة الطرد المركزي، النموذج الرقمي، برنامج PLAXIS

The newly published European geotechnical codes of practice or Eurocode 7 are not as prescriptive as the old codes and allow the designer to choose an appropriate method of analysis. Over the past 20 years, the development of numerical analysis based mainly on the finite element method have provided geotechnical engineer with an extremely powerful analysis tool. As opposed to conventional methods, the use of numerical analysis in geotechnical problems was found to lead to more accurate and economical design, and consequently its application is likely to increase in the future. Parallel developments in computer hardware and more importantly, in geotechnical software, enable the geotechnical engineer, who are not necessarily numerical specialists, to perform very advanced and complicated numerical analyses at low cost and with relatively minimum computational effort.

The Plaxis code is a finite element incorporated programme, developed at Delft University, in the Netherlands, specifically for the numerical analysis of deformation and stability in geotechnical engineering projects. Set out to use realistic soil parameters with sophisticated constitutive models, in order to provide realistic calculations of forces, displacement and stresses. Over the last years, the performance and accuracy of Plaxis code has been fully tested and validated by numerous analyses of problems with known analytical solutions (Binkgreve and Vermeer [1]).

It is suggested in the present investigation to use the Plaxis code in order to develop a numerical model which intends to predict the behaviour of an L-shaped rigid retaining structure made of reinforced concrete. The dimensions, the boundary and the initial loading conditions of the investigated structure are shown in Figure 1; this structure is considered in the following as the prototype.

Identical prototype was previously simulated by a reduced scale model (1/ 60) tested in a centrifuge experiment conducted by Djerbib et al. [2]. For the validation of the proposed numerical model, direct comparison was made with the centrifuge experiment results, up-scaled to the prototype geometry according to the centrifuge scaling laws fully discussed by Schofield [3]. This paper presents the development of the numerical model and the validation of results obtained.

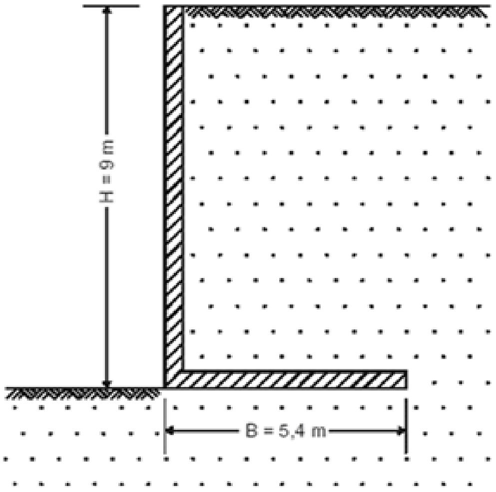


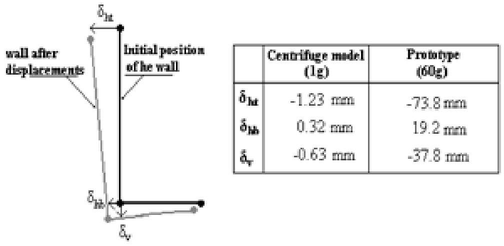
Figure1: Prototype of the L-shaped retaining wall.

THE EXPERIMENTAL MODEL

The reference's experimental model was made of a 22 mm thick mild steel plate, with overall dimensions of 150 mm height and a base 90 mm wide. The centrifuge model was instrumented with several displacements and contact stress transducers to monitor the displacements and the stresses during the course of loading. The soil used in the experiment was the Leighton buzzard sand, according to the British Standard (BS1377, Part 2) this dry sand may be classified on the basis of its particle size distribution as a uniform fine to medium sand with a dry density $\gamma_{unsat} = 17$ kN/ m^3.

The first sequence of the reported centrifuge experiment started by applying an increasing acceleration reaching 60xg and maintained to this level, this stage was supposed to model the backfilling process of the prototype under 1xg. As the deflection of the centrifuge model is negligible, the displacements measured at this step concern the rigid body movements.

3 displacements were computed: δ_{ht} representing the horizontal movement of the top of the centrifuge model stem, δ_v representing the vertical movement of the centrifuge model stem and δ_{hb} representing the horizontal movement of the centrifuge model base. Figure 2 illustrate these displacements and presents the corresponding values converted to the prototype scale.



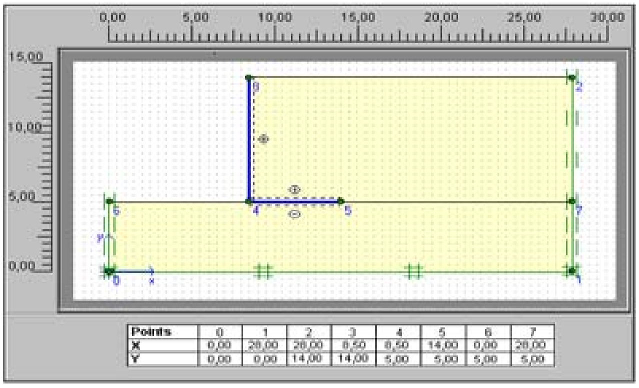
	Centrifuge model (1g)	Prototype (60g)
δ_{ht}	-1.23 mm	-73.8 mm
δ_{hb}	0.32 mm	19.2 mm
δ_v	-0.63 mm	-37.8 mm

Figure 2: Wall displacements measured in the centrifuge (after Djerbib et al.

NUMERICAL ANALYSIS

The geometry of the finite element model was constructed using the graphical input procedure of the Plaxis program. At this stage, the geometry of the numerical model, the material properties and the boundary conditions were specified.

The numerical analysis was carried out in plane strain, as presented in Figure 3, the layout of the numerical model extends 28m horizontally and 14m vertically to model the prototype scale of the centrifuge container, these boundary limits were assumed to be sufficient to avoid border disturbances. Conditions of plain strain were assumed throughout; the vertical boundaries of the model were pinned in the horizontal direction but free to move vertically, and the horizontal boundary at the base of the model was assumed to be pinned in both vertical and the horizontal directions. The retaining wall was defined through an L-shaped beam with a rigid slab footing representing the prototype dimensions of the centrifuge wall-model. Similarly to the centrifuge experiment drained conditions were modelled in the numerical analysis.



Points	0	1	2	3	4	5	6	7
X	0,00	28,00	28,00	8,50	8,50	14,00	0,00	28,00
Y	0,00	0,00	14,00	14,00	5,00	5,00	5,00	5,00

Figure3: Numerical model Geometry and displacements boundary conditions.

Following the setting of the finite element computation parameters, the Plaxis input programme is used for the generation of the model's finite element mesh, a typical mesh is shown in Figure 4. With respect to the centrifuge experiment, all steps going from the installation of the wall in the container to the acceleration phase of the centrifuge were taken into account in the numerical analysis, according to the chart shown in Figure 5.

The calculation phase was preceded by the initial condition stage, simulating the disturbances induced to the soil foundation by the wall weight before the beginning of the backfilling process. This initial condition stage was modelled as shown in Figure 6.

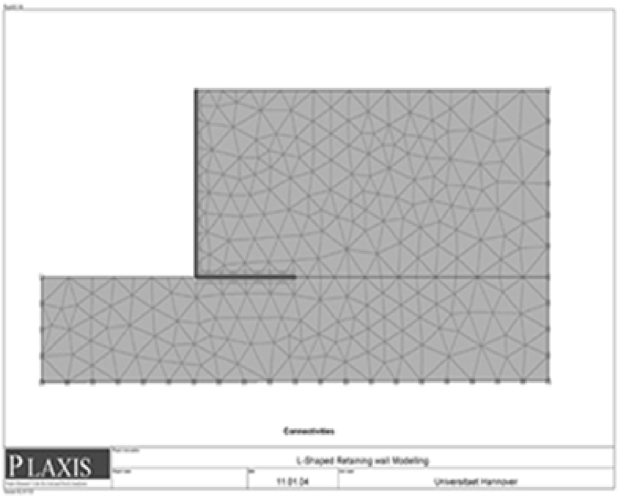


Figure 4: Typical Finite element mesh of the model.

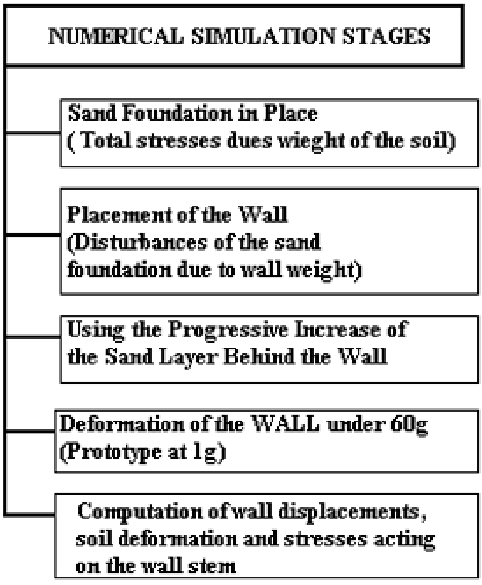


Figure 5: Numerical simulation stages.

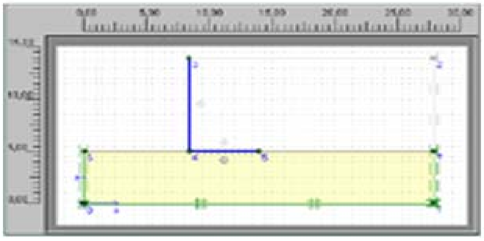


Figure 6 : Initial condition model.

MODELLING MESH DATA

Table 1 gives the modelling mesh data adopted in the finite element computation for the soil, the wall and the interface. The soil model was run with a fine mesh, 15 nodded triangular elements, leading to 344 elements, 2929 nodes and 4128 stress points.

Table 1: Modeling mesh data

Type	Type of element	Type of integration	Total no.
Soil	15 - nodded	12 -point Gauss	344
Wall	5 -node line	4 -point Gauss	10
Interface	1: 5 node line	4 -point Newton-Cotes	14

Modelling experience showed that problems of soil structure interaction might involve points, which require special attention. Corners in stiff structures and an abrupt change in boundary condition (L-shaped geometry of the wall), may lead to peaks in the stresses and strains. Conventional finite elements analyses are not capable of reproducing these sharp peaks and will, as result, produce non-physical stress oscillations. Plaxis program prevents this phenomenon by entering additional interface elements inside the soil body that will enhance the flexibility of the finite element mesh and prevents non-physical result. The theoretical background on the special use of interface element in the modelling of soil-structure interaction has been thoroughly investigated by Van Langen and Vermeer [4]. In the model presentation (Figure 4) the interface element are shown to have finite thickness, but in the Plaxis code formulation the coordinates of each node pair are identical, which means that the element has zero thickness. Each interface has assigned to it a "virtual thickness" which is imaginary dimension used to obtain the material properties of the interface. The thickness is defined as the virtual thickness factor times the average element size. The average element size is determined by the global coarseness setting for the mesh generation. The default value of the virtual thickness factor is 0.1. The value of the interface coefficient R_{inter} was taken equal to one.

WALL MODELLING

The retaining wall structure was simulated with one-dimensional linear beam element that can resist axial load and bending moments. The stiffness for the wall element is represented by means of the flexural rigidity *EI* and the normal stiffness *EA*, where the *A* and *E* are the cross section area and the Young's modulus of the reinforced concrete structure wall. The wall modelling parameters are presented in Table 2.

Table 2. Wall modelling parameters

Type		Wall
Material		Reinforced Concrete
Young's modulus *E*	[kPa]	2.3E7
Poisson's ratio ν	[-]	0.30
Normal stiffness *EA*	[kN/m]	6.9E7
Flexural rigidity *EI*	[kN/m]	5.175E7
Equivalent thickness $d = \sqrt{12 \frac{EI}{EA}}$	[m]	3.00
Mp	[kNm/m]	1E15

SOIL MODELLING

In the present numerical analysis the soil has been modelled using the hardening soil model, incorporated into the Plaxis program, considered in drained conditions. This constitutive model is based on the well-known formulation by Duncan and Chang [5], but formulated within the theory of plasticity. It incorporates shear hardening and volumetric hardening, a stress dependent stiffness for primary loading and unloading/reloading and the stress dilatancy theory by Rowe [6]. Relatively to simple elasto-plastic soil model, the hardening soil model is actually considered as an advanced model for simulating the behaviour of soft and stiff soils and its common use might provide a closer insight into the real soil-structure interaction.

When subjected to primary deviatoric loading, soil shows a decreasing stiffness and simultaneously irreversible plastic strains develop. In the special case of a drained test, the observed relationship between the axial strain and the deviatoric stress can be well approximated by hyperbola. Such a relationship was first formulated by Kondner [7] and later used in the well-known hyperbolic model proposed by Duncun and Chang [5]. The hardening soil model, however, superseded the hyperbolic model by far. Firstly by using the theory of plasticity rather than the theory of elasticity, secondly by including soil dilatancy and thirdly by introducing a yield cap. The basic representative characteristics of the model are:

- Failure occurs according to the Mohr-Coulomb model: represented by c, φ and ψ
- Stress dependent stiffness according to a power law represented by parameter m
- Plastic straining due to primary deviatoric loading represented by the parameter E_{50}^{ref}
- Plastic straining due to primary compression represented by the parameter E_{oed}^{ref}
- Elastic (unloading / reloading) represented by : E_{ur}^{ref} , υ_{ur}

The values of the modeling parameters used in this analysis are presented in Table 3.

Table 3: Soil modelling Data Sets Parameters.

Soil model: *Hardening Soil*	Units	Soil Material: Leighton Buzzard Sand
Type		Drained
γ_{unsat}	[kN/m³]	17.00
γ_{sat}	[kN/m³]	20.00
k_x	[m/day]	1.000
k_y	[m/day]	1.000
E_{50}^{ref}	[kN/m²]	30000.00
E_{oed}^{ref}	[kN/m²]	30000.00
power (m)	[-]	0.50
c_{ref}	[kN/m²]	1.00
φ	[°]	32.00
ψ	[°]	2.00
E_{ur}^{ref}	[kN/m²]	90000.00
$\nu_{ur}^{(nu)}$	[-]	0.200

NUMERICAL RESULTS

When the initial stress conditions are defined, the Plaxis calculation program is loaded and the calculation process of stresses and displacements in the model is started, after completion of the calculation, the Plaxis output generate the computed results. The precision of the calculation was kept equal to the default value of 3%. Prediction of displacements and forces are amongst the key objectives for performing soil-structure interaction analysis. The results of the present numerical analysis are presented mainly in terms of wall displacements and earth pressure computation. Brief indication of the predicted soil movement is discussed.

SOIL DISPLACEMENTS

Figure 7 shows the deformed mesh of the numerical prediction corresponding to the state of end of backfilling. In this figure the displacements are scaled up 10 times to highlight the deformation pattern of the wall (rigid body movements) and the soil mass.

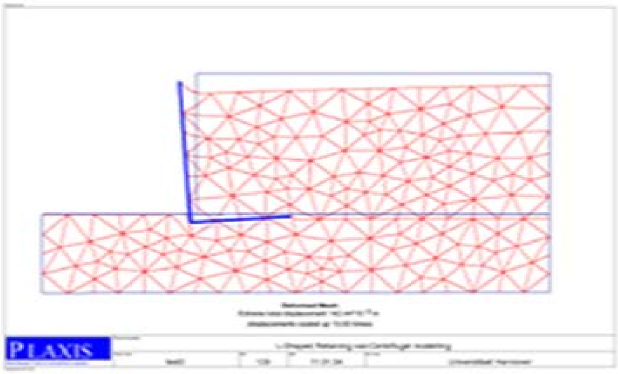


Figure 7: Deformed mesh.

Figure 8 gives an indication of the total displacement vectors of the soil predicted at this stage of loading. It could be noticed that during the backfilling process of the wall, the soil mass is in an active state and is moving with the wall, also, there some concentration of displacement's vectors beneath the wall base and inside the backfill-part, commonly called the virtual wall, this is in contradiction with the design practice of "L-Shaped" retaining walls, where, soil mass resting on the wall base is assumed to be part of the wall, and its displacements are not taken into account.

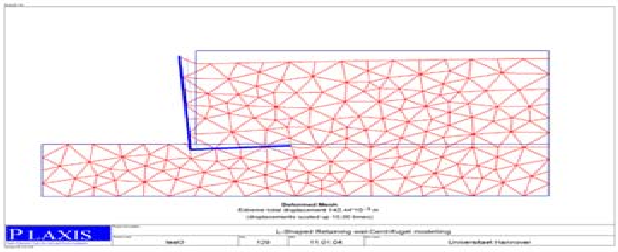


Figure 8: Total displacement vectors of the soil.

WALL DISPLACEMENTS

The resulting, predicted total displacement vectors of the retaining wall is shown in Figure 9. It is clear that the settlement of the wall base was greater at the stem bottom. In addition to the forward tilt of the wall stem away from the original backfill, the base of the wall has also translated forward, the position of the instantaneous center of rotation for this unpropped wall indicate that rotational movements are dominant throughout the test. In addition to the accordance of these observations with the centrifuge experiment, those results remain consistent with previous gravity wall analyses.

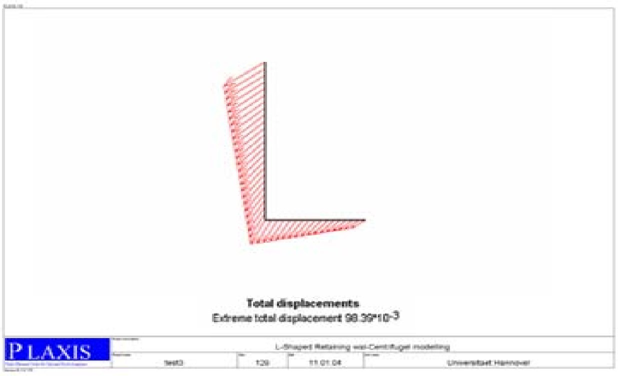
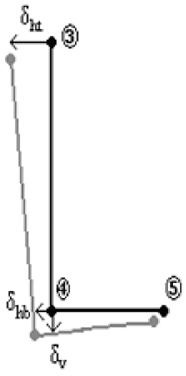


Figure 9: wall total displacement vectors.

Figure 10 shows the computed displacements of the nodes corresponding to the points 3, *4* and *5* representing the geometry of the retaining wall (see Figure 3). The predicted horizontal displacement of the point (*3*) is representative of the displacement of the top of the wall stem, and could therefore, be compared to the experimental value δ_{ht}. The horizontal movement of the bottom of the wall δ_{hb}, is compared to the horizontal displacement (translation) of the point (*4* or *5*). The vertical movement of the wall δ_v is compared to the average displacement of the points (*4*) and (*5*) to account for the rotation of the wall.



	Experimental model	Numerical model
δ_{ht}	-73.8 mm	-82.1 mm
δ_{hb}	19.2 mm	27.5 mm
δ_v	-37.8 mm	-36.0 mm

Figure 10: Comparison of the computed and measured displacements (prototype scale).

The computed wall movement by the proposed numerical model indicates a combination of rotation and translation although the rotation is generally dominant, which is in agreement with the reported behaviour observed throughout the centrifuge test. In most previous investigations, either pure rotation or translation was considered, but it is evident that both rotation and translation occur simultaneously, also, the centre of rotation is usually fixed either at the toe or the top of the wall, the numerical analysis confirms the experimental observation in which rotation takes place about an axis within the displacement vector of the toe.

For comparison, Figure 10 illustrates the correlation obtained between the centrifuge measured (converted to the prototype scale) and the numerically predicted displacements of the wall. As can be seen on this figure, the numerical model proposed was found to be able to produce a very close prediction of both: the displacement pattern and the magnitude of the displacements. The small discrepancy observed between the numerical and the experimental approaches could be attributed to combined factors including limitation of the proposed numerical analysis.

EARTH PRESSURE

Figure 11 shows the associated distribution of the horizontal earth pressures at the vertical plane: immediately adjacent to the wall stem (computed from the stress-points corresponding to the soil elements adjacent to the interface). On this plot the good agreement between the output of the numerical model proposed and the experimental results obtained in the centrifuge on is clearly apparent. Also for the appreciation of the obtained results, the at-rest (K_0), and the classical Rankine active earth pressures (K_a) profiles are also shown on this Figure.

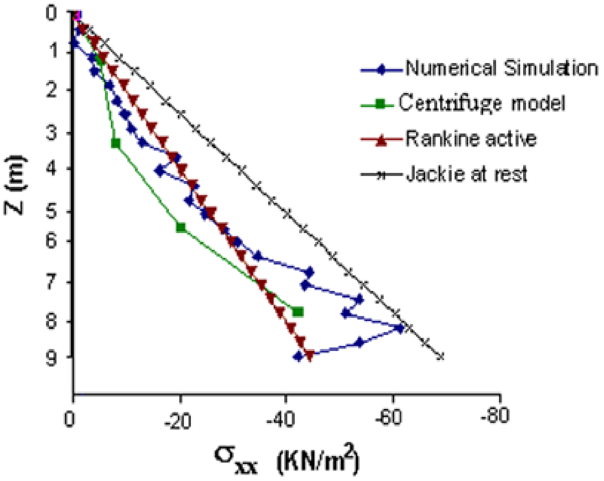


Figure 11: Lateral Pressure acting on the wall stem.

The lateral pressures acting on the wall stem are generally of interest for structural design of reinforced concrete retaining walls. The common practice in design consists of multiplying the active pressures by a load factor that usually has a value greater than unity. The lateral movement of the wall resulted in the reduction of the horizontal earth pressures acting on the wall stem from the at-rest condition for almost the entire height of the wall. The numerically predicted as well as the experimentally measured lateral pressures at the wall stem correspond closely to the classical Rankine active pressures for the top two- thirds of the wall. In the lower third of the stem, the numerically predicted lateral pressures are significantly in excess of the active pressures; this could be attributed to the effect of the wall base.

In practice the lateral pressure distribution is assumed to be linear, however, agreement on the non-linearity of the lateral pressure profile is apparent between the centrifuge measured and the numerically computed results. Due to most current regulations, including the new Eurocode, the structural design of the vertical wall stem of L-shaped retaining walls is mainly based on the assumption of increased earth pressure loading, i. e. the average of Rankine's active earth pressure (K_a) and earth pressure at rest (K_0). From Figure 11, it is clear that with this approach the resultant loading is in most cases overestimated. These few observations are closely consistent with the findings of Goh [8] in a finite element investigation on the behavior of concrete cantilever retaining walls.

LIMITATIONS OF THE NUMERICAL MODELLING PROCEDURE

Although all efforts have been made in order to reproduce numerically the sequences of the centrifuge test referred to in the validation processes, it remains obvious that perfect numerical simulation of geotechnical experimental testing a hard to achieve goal. The experiment difficulties related to the handling of experimental equipment (Djerbib et al. [2]) added to the tolerated finite element computational errors are combined factors that might have introduced uncertainties in the correlation made between the experimental and numerically computed results.

As far as the behaviour of the soil (backfill and foundation) is concerned, it is evident that soil particles in a centrifuge model cannot be scaled down to scale the soil particles of prototype, while other model dimensions can be modelled down. The numerical approach considered skip this grain-size effect by considering the model at prototype scale, and thus, it could be argued that the numerical approach could be assumed to be more precise than the centrifuge experiment in simulating the real soil behaviour surrounding the targeted prototype.

CONCLUSIONS

This paper describe the development of a simple numerical model using the finite element Plaxis program version 8, which could simulate and predict the behaviour observed in a centrifuge experiment of an "L-shaped" rigid retaining walls supporting sand. Indicating a good quality of the numerical model, parametric analysis using this numerical model could be carried to investigate the effect of different parameters (wall height, length of wall base, soil stiffness etc..) on the performance of the L-shaped rigid wall, and to propose more realistic and economical design method.

Using the Plaxis code, the approach followed in the development and validation of present numerical model could be of relevance, since relatively little attention has been paid in the literature on validation of numerical models proposed and on the performances of specific commercial software's, when directly compared to reliable experimental results.

Although, it is recognized that monitoring field observations on real structures or prototypes remains the best way to fully validating numerical simulations, this procedures remain expensive and time consuming, however, the 'modelling of centrifuge experiment' or 'modelling of models' concept used in this work could constitute a simple and useful mean to exploit the considerable centrifuge data gained through the years in investigating the behaviour of retaining structures.

ACKNOWLEDGEMENTS

The numerical model presented was developed using the facilities and the Plaxis licence of IGBE at the University of Hanover, Germany, with a financial support from the DAAD. The authors express their appreciation to Professor Martin Achmus for his many helpful suggestions.

REFERENCES

[1]- Brinkgreve, R.B.J. ; Vermeer, P.A. (2002). "PLAXIS: Finite Element Code for Soil and Rock Analyses" Version 8. A.A. Balkema, Rotterdam / Brookfield.

[2]- Djerbib, Y.; Hird, C. C. and Touahmia, M. (2001). "Centrifugal model tests of uniform surcharge loading on L-shaped retaining walls". 15th International Conference on Soil Mechanics and Foundation Engineering, Istanbul , Vol. 2, pp. 1137-1140.

[3]- Schoffield, A. N. (1980). "Cambridge geotechnical centrifuge operations". 20th Rankine Lecturer.", Geotechnique , Vol.20, No 2, pp. 129-170.

[4]- Van Langen, H. and Vermeer, P. A. (1991). "Interface elements for singular plasticity points". International Journal on Numerical and Analytical Methods in Geomechanics. 15, pp. 301-315.

[5]- Duncan, J. M. and Chang, C.Y. (1970). "Nonlinear Analysis of stress and strain in soil" . ASCE J. of the soil Mech. And Found. Div. Vol. 96, pp. 1629-1653.

[6]- Rowe, P.W. (1962). "The stress - dilatancy relation for static equilibrium of an assembly of particles in contact". Proceedings Royal Society. A269, pp. 500-527.

[7]- Kondner, R. L. (1963). "A Hyperbolic stress strain formulation for sand" . 2nd Pan-American. Iternational conference ICOSEF Brazil, Vol. 1, pp. 289-324. 1963.

[8]- Goh, .T.C. (1993). "Behaviour of cantilever retaining walls" Journal of Geotechnical Engineering. ASCE, Vol 119, N°.11, pp 1751-1770.