ICSBE2016-98

INVERSE MODELLING FOR ESTIMATING AN INTERFACE ELEMENT PROPERTIES IN SOIL-PIPE INTERACTION AN OPTIMIZATION APPROACH

P. Kathirgamanathan[1*] and R. McLachlan [2]

[1]*University of Jaffna, Kilinochi, Sri Lanka*
[2]*OPUS Research, OPUS International Consultants, Wellington, New Zealand*
**E-mail: pkathir @eng.jfn.ac.lk*

Abstract: The development of transmitted stresses onto a pipe from the backfill or in-situ soil is not well known for both static and dynamic load cases. This study was aimed at investigating the application of an inverse modelling technique to determine the material parameters of a thin interface layer which lies between the soil and buried pipe during a seismic event to assess the stress transfer. The model uses measured strain values on a pipe of an axial push test. The process of estimation is mathematically known as an inverse problem and is formulated as a non-linear least squares minimization problem coupled with a finite element model for the soil-pipe interaction. The method involves constructing an iterative procedure using an optimization routine in MATLAB and at every iteration, the finite element problem was solved using the finite element program ABAQUS. Finally, the accuracy of the parameter values are examined by using the measured strain values at various different loadings. This research helps to further the understanding of the soil-pipe load transfer system under various loadings and interface layers in finite element analysis.

Keywords: Finite element modelling; Inverse problem; Soil-Pipe interaction;

1. Introduction

Finite element modelling of a buried pipe may be used during the design process to study the behavior of the pipe during earthquakes. By using finite element simulation, the efficient prediction of pipe deformation and pipe-soil interaction can be studied in detail. This method allows us to determine (a) stress, strain values, (b) influence of pipe dimensions, (c) effect of trench shape geometry and (d) influence of soil material parameters etc.

The performance and behaviour of buried pipe is strongly influenced by soil-pipe interaction. One aspect of this involves the properties and load transfer characteristics of the soil-pipe contact surfaces. The appropriate modelling of the soil-pipe interface is important to ensure that the performance estimation is predicted as accurately as possible. Traditionally, soil-pipe interaction problems are solved for two idealized interface conditions: (a) perfect adhesion of the soil to the pipe structure (the perfectly rough, no-slip): and (b) zero adhesion (the full slip or smooth interface condition). The actual gap and slippage between pipe and soil cannot be modelled using above mentioned techniques. It is expected the actual pipe response in the field is expected to lie somewhere between these two limits.

In 2000, Raul *et al*. (Kudella *et al*, 2002, Raul *et al 2003*) modelled the interaction between a pile and soil using thin continuum elements at a thickness 0.2 of the pile diameter with a reduced strength property. The interface element is the mechanism which allows slippage and gap at the soil-pile contact (Nogami *et al*., 1988).In reality, the material data for the pipe and soil are generally available or can be measured using available apparatus, but the material parameters of interface layer are often unknown. Small errors in these values can cause variations in the estimation of the behaviour. Methods to determine the material parameters of the interface layer are therefore a significant part of modelling underground structures. In this study, local data was not available so field test measurements concerning the soil-pipe interaction from an axial push test result (the average axial strain measurements on pipe at various different locations) was obtained from Chi Fai Ng, (1994). These measurements and material parameters of the soil and pipe were used along with a 3D finite element model of the same axial push test to calculate the material parameters of interface layer. The procedure for estimating the material parameters of the interface layer from experimental measurements reduces to a parameter estimation problem in finite element modelling. The inspiration behind the methodology is based on the work reported in the literature (Kathirgamanathan *et al*2008).

This paper presents three modelling stages: the forward model; interface modelling; and inverse modelling.

2. Forward model

The first requirement in attempting to numerically solve the optimal material parameter estimation problem described above is to be able to model accurately the forward problem i.e.that of finding the stress and strain profile for a given material parameter value and load.

The general-purpose FEA software ABAQUS/Standard has been used to set up the forward finite element model in three dimensions. The investigated problem involves a pipe outer diameter 0.91 m and 14.1 mm in thickness. The pipe is assumed to be continuous with a length 95 m. To reduce the computational effort by making use of the symmetry in the geometry and loading only the half of the model is considered as shown in Figure 1.

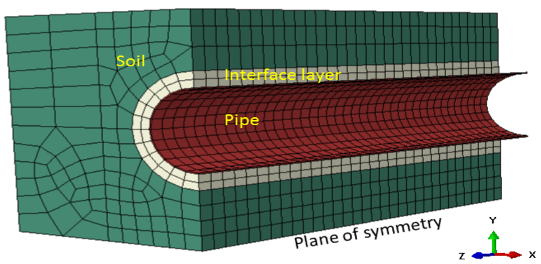

Fig. 1. Finite element model

Two cases of displacement (kinematic) boundary conditions are considered when modelling a pipe-soil model: (a) The plane of symmetry (i.e. the plane where x=0) where movements normal to that plane are prevented by applying a Ux=0 boundary condition, and (b) The bottom of the soil layer is restrained against movements in the x, y and z directions (Ux= Uy= Uz=0 boundary conditions). In the analysis the loads are specified in two consecutive steps. First the soil and pipe are subjected to static gravity loading and then the dynamic loading parallel to pipe's axis. The soil is assumed to be elasto-plastic in behaviour and the Mohr-Coulomb material model is used for representing the behaviour of soil. A linear elastic material model is used for the steel pipe. The following material data values were used in the ABAQUS simulations: Pipe- Modulus= 209000 MPa, Poisson's ratio=0.3, Density=7800 kg/m^3; Soil- Modulus=8.4 MPa, Poisson ratio=0.25, density=1800, Friction angle=39^o, and Cohesion=20.6 kN/m^2.

In pipeline construction it is standard for a specified sandy bedding material to be placed in the trench prior to laying the pipe, for side support and overlay material to be placed, and then the trench backfilled with a fill material. For simplicity, the model has not used three different material layers for bedding, overlay and backfill. The model only consider the pipe and surrounding backfill material. The interaction between pipe and soil are modelled using traditional contact interaction and interface element features. The next section demonstrate the importance of improving the pipe-soil contact interaction using interface element approach.

3. Interface Modelling

The purpose of this section is to study the effect of calculated maximum stress on the pipe wall using traditional contact interaction formulation between soil and pipe and to demonstrate the need for improved pipe-soil contact interaction modelling. The continuum solid interface elements are used between pipe and soil in the interface modelling approach. Various modelling procedures have been developed for simulating soil-pipe interactions under static and seismic loading. Knowledge of soil, interface structure and modelling techniques has been taken from previous literature (Kudella *et al*, 2002, Raul *et al* 2003 and Nogami *et al*., 1988). In the first step, the numerical simulation of the soil-pipe interaction is modelled with traditional, soil-pipe interaction methods and interface elements to study in particular how maximum stress on the pipe changes during the seismic dynamic loading.

The cross-sectional view of the proposed finite element model is shown in Figure 2A. The soil-pipe system is modelled under plain strain conditions. For simplicity the material used to fill the trench is not subdivided into different layers such as bedding, backfill and cover.

The investigated problem involves a pipe outer diameter 0.84 m and 41 mm in thickness.The following material data values were used in simulations: Modulus= 31000 MPa, Poisson's ratio=0.2, density=2643 kg/m³.

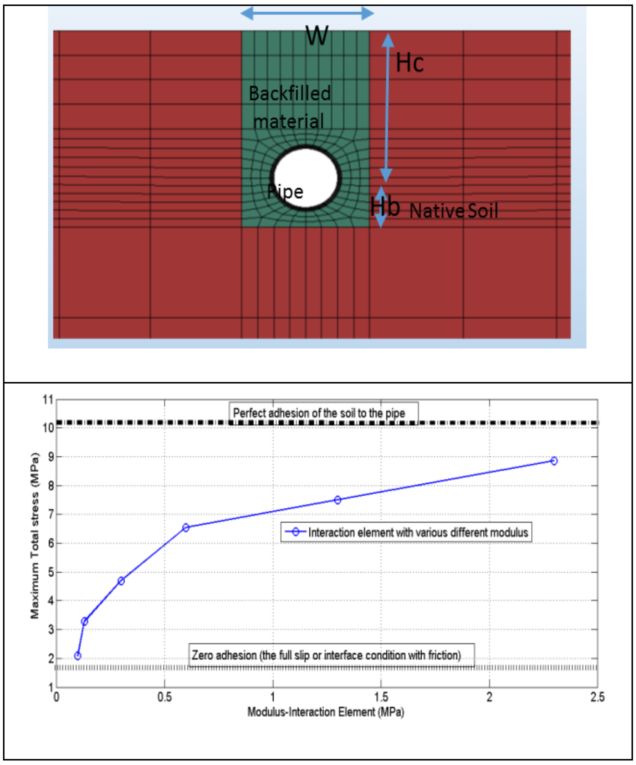

Fig. 2. Soil-Pipe Interaction: (A) Cross-section view, (B) Comparison of interface cases

In modelling with the traditional method of perfect adhesion of the soil to the pipe, the pipe elements and soil elements share the same nodes at the interface and therefore pipe and soil are constrained to have same displacement. In the zero adhesion case the soil-pipe interface is modelled by the contact surface approach. The penalty function method available in ABAQUS is used to model slip and separation between pipe and soil. In the interface element approach, the interface is treated as a solid element with small finite thickness of 0.2 times diameter of pipe (as in Desai *et al*, 1984, Raul *et al*, 2000). The actual material parameter values of interface element are not known. At the first step it is assumed that the material properties of interface element and surrounding soil are the same. Then the 'modulus of interaction' element is changed while other parameters such as Poisson's ratio, friction angle, dilation angle, density are unchanged and equal to surrounding soil parameter values. The results of maximum stress on the pipe during an analysis are calculated and compared against each cases as shown in Figure 2B. It can be seen that the pipe response with interface element is always lie between the pipe responses with traditional approaches.

The variation of maximum stress on the pipe with poisson's ratio is shown in Figure 3A, maximum stress Vs density is shown in Figure 3B, and the maximum stress Vs friction and dilation angle of interface element (while the modulus of interface element remains constant) is shown in figures 3C and 3D.

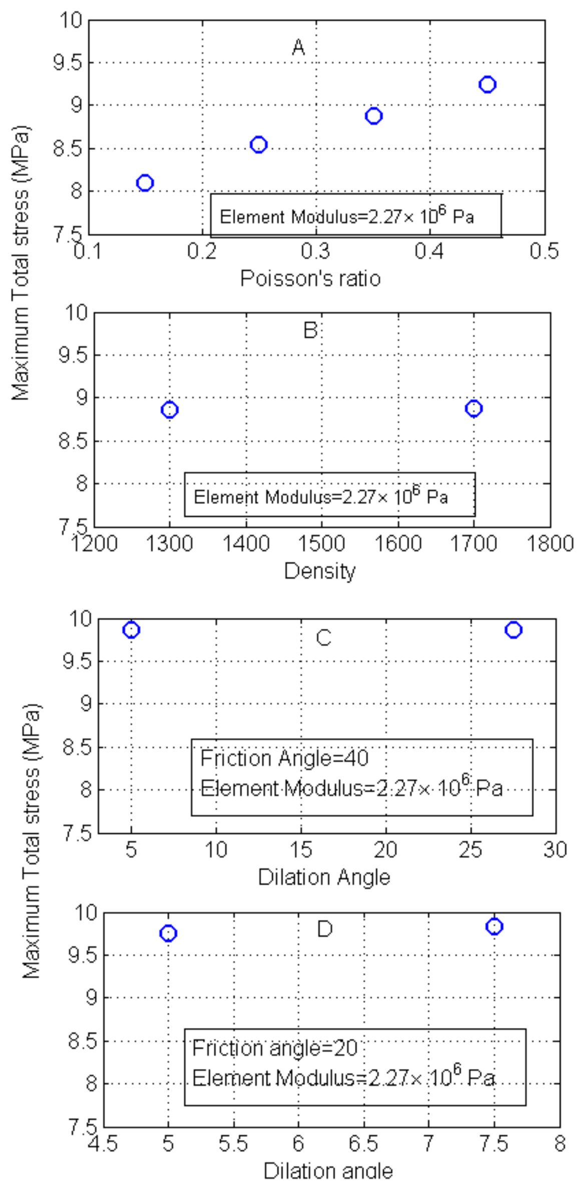

Fig. 3. Interaction with soil parameters

These figures demonstrate how the maximum stress on pipe changes during the loading process, and how this depends on the material properties of the interface element. This behaviour is further supported by Milligan et al. (1989) and Boot et al. (1991). In their findings Milligan et al. (1989) and Boot et al. (1991) suggested the use of packing material with low Poisson's ratio and low stiffness to minimise the stress transfer. Further it can be seen that the error in the modulus and poisson's ratio of interface element increases qudratically with the error in pipe response and demonstrated the importance to determine the material parameters accurately. The objective of next section is to describe an inverse model capable of concurrently estimating the material parameters of interface layer.

4. Inverse Modelling

The intention of the inverse modelling approach is the extraction of model constants from experimental data. It is a discipline, which offers tools for the competent use of data in the estimation of constants used in the models. The strain values are measured at a number of different observation points on pipe. These are the prime unknown in the forward problem. The material parameters are unknown. Our aspiration is to estimate the best estimates of these parameters. This problem is known mathematically as an inverse problem and can be seen as an optimization problem whereby the objective function to minimize is the differences between the measured strain values and the estimated strain values at selected location by finite element simulation.

First of all, a mathematical definition of the differences between the measured and simulated results is required as an objective function. The aim of optimization is to find the best set of constants appearing in the model which minimizes an objective function by improving the performance in the direction of optimas. The intention of the approach is to find the global minimum on a given search space by minimizing the objective function subjected to the given constraints. In the case of a minimization problem, the mathematical formulation of the problem can be stated as follows:

$$\text{Minimize } f(\boldsymbol{p}) = \sqrt{\frac{1}{n}\sum_{i=1}^{n}\left(\frac{\varepsilon_i-\hat{\varepsilon}_i(\boldsymbol{p})}{\varepsilon_i}\right)^2} \qquad \text{Eq 1}$$

Subject to

$$g_j(\boldsymbol{p}) \geq 0, j = 1, \cdots, n1,$$

$$h_k(\boldsymbol{p}) = 0, k = 1, \cdots, n2,$$

where $f(\mathbf{p})$ is the objective function, $g_j(\mathbf{p})$, $h_k(\mathbf{p})$ are constraints function, $\mathbf{p}$ is a vector of constants, n is the number of measured values and ε is the measured strain values and $\hat{\varepsilon}$ is the modelled strain values. For the optimal fit **p** must be varied to minimise *f*.

The estimation procedure is built within a MATLAB platform as shown in Figure 4. The input file defines the physical structure by its dimensions, material properties and boundary conditions. The FEM simulation executes the created input file using ABAQUS. To transfer data between MATLAB and the finite element program ABAQUS, a PYTHON program is used. The minimization process is implemented using MATLABs inbuilt optimization function *lsqnonlin* which uses the *Levenberg-Marquardt* algorithm. This approach supports an easy execution of many runs of ABAQUS within an optimization routine.

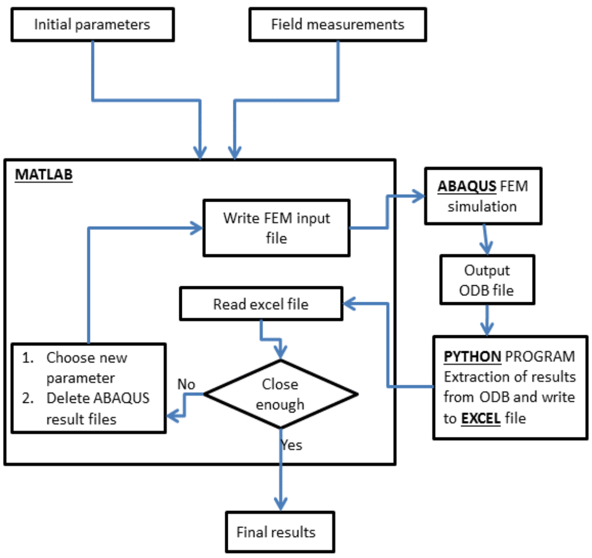

Fig. 4. Technical Procedure

5. Model validation

In this section, numerical simulations are presented to demonstrate the solution process and evaluate the accuracy of the estimated values. To do so, inputs of measured strain values on a pipe at different locations are considered. The experimental data used in this section are taken from a soil-pipe interaction experiment carried out by British Gas at a location north-west of Hilde stone, Staffordshire in 1988 (Chi Fai Ng, 1994).

The experiment involves with a pipe outer diameter 0.91 m, 14.1 mm thickness, length 95 m and surrounded by soil. The soil properties used are: Modulus=8.4 MPa, Poisson ratio=0.25, density=1800, Friction angle=39°, and Cohesion=20.6 kN/m². A total of twenty four strain gauges were installed on the pipe wall for the measurement longitudinal strain variations. The strain gauges were divided into eight groups (A to H) and each group having three gauges arranged as shown in Figure 5. An axial load was applied and the variation in strain along the length of the pipe at locations A, B, C, D, E, F, G and H were recorded.

The finite element program ABAQUS was used to simulate the same experiment described above, and compare the results against the measured values. The experimental values with two different loading conditions are considered with the comparison of average strain values (average of x, y and z) along the pipe with measured values during the optimization shown in Figure 6. The green colored line shows the most optimal fit where as blue clour line is closer to zero adhesion case of traditional approach. This shows the very good agreement between measured and predicted values with most optimised parameter values.

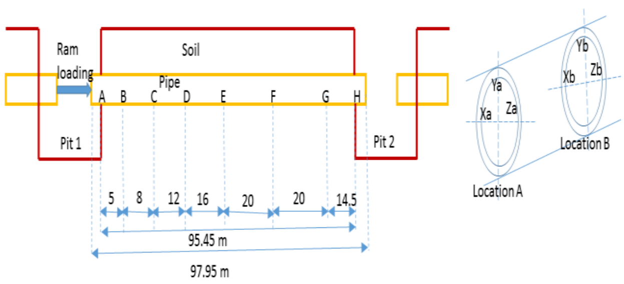

Fig. 5. Side view and location of strain gauges

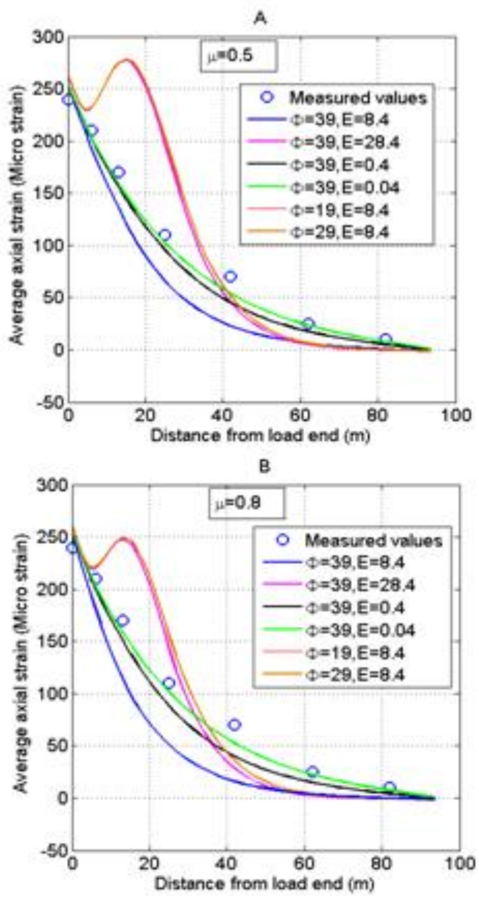

Fig. 6. Comparison of average strain values along the pipe with measured values during the optimization, with (A) Interface layer-Pipe friction=0.5, and (B) Interface layer-Pipe friction=0.8

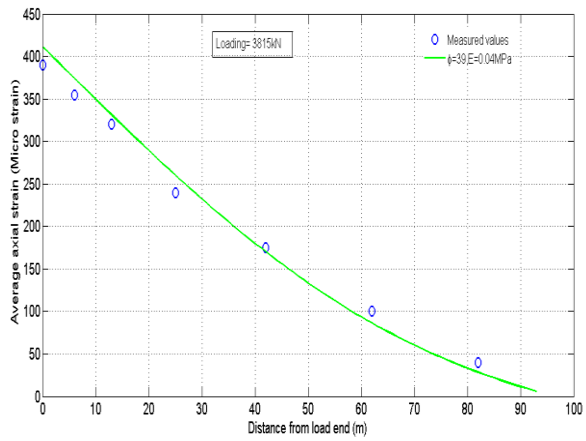

Fig. 07. Average Axial Strain against distance from load

The calculated interface parameter values were also validated by simulating experiments at various different loading rates and comparing to measured values. Figure 7 shows the average axial strain against distance from load used to fit interface parameter along with measured experimental values.

6. Summary and Conclusion

The intention of this paper is to demonstrate the need for modelling of soil-pipe interface layer and show the development of an inverse model capable of simultaneously estimating the material parameter constants of the interface layer between the pipe and soil. The methodology is based on a non-linear least squares estimation using measured axial strain values on a pipe.

Firstly, a pipe-soil finite element model was simulated using ABAQUS and investigated the effectiveness of traditional contact interaction formulation between soil and pipe and demonstrated the importance of improving the pipe-soil contact interaction formulation.

Secondly a numerical technique to estimate the optimal material parameters of the interface element was investigated. It is based on a non-linear least squares coupled with finite element techniques. Nonlinear least squares optimization is carried out by constructing an iterative procedure using MATLAB's inbuilt function *lsqnonlin.* At each iteration the finite element solutions to the pipe-soil interaction were obtained using ABAQUS.

Finally, the accuracy of the model estimated parameter values were validated by simulating experiments at different loading rates and comparing them to independantly obtained experimental data. These results suggests that the developed model is capable of predicting the interface material parameters to a reasonable degree of accuracy.

Acknowledgement

The reported investigation has been sponsored by a grant from Ministry of Business Innovation and Employment, New Zealand. The authors thank the funding body for their generous support.

References

[1]. Boot J. C, and Husein N. M, Vitrified clay pipes subject to jacking forces, Proc. 1st Int. Conference on pipe jacking and micro-tunnelling, p6.i-6.9, Pipe jacking association, London, 1991.

[2]. Chi Fai Ng P, Behavior of buried pipelines subject to external loading, PhD thesis, University of Sheffield, 1994.

[3]. Desai, C.S, Zaman M.M., Lightner, J. G, and Siriwardane H. J, Thin-Layer Element for Interfaces and Joints, Int. J. Numer. Anal. Meth. Geomech, 8, 1984, 19-43.

[4]. Kathirgamanathan P, Neitzert T, Optimal Process Control Parameters Estimation in Aluminum Extrusion for Given Product Characteristics. Proceedings of the World Congress on Engineering, 2008; 2: 1436--1441.

[5]. Kudla P, Reul O, Hypoplastic analysis of piled rafts, Numerical methods in Geotechnical Engineering, Mestat (ed.) 2002.

[6]. Lee H, Finite element analysis of a buried pipeline, PhD thesis, The University of Manchester, 2010.

[7]. McGann C. R, Arduino P, Mackenzie-Helnwein P, Development of Simplified analysis Procedure for Piles in laterally Spreading layered soils, PEER 2012/05, University of Washington, 2012.

[8]. Milligan G. W. E, and Ripley K. J, Packing material in jacked pipe joints, No-Dig 89, Proc. Int. Conf. on Trenchless Technology, p2.1.1.1.1.10, ISTT, London, 1989.

[9]. Nogami T, Konagai K, and Otani J, Nonlinear pile foundation model for Time-Domain dynamic response analysis, Proceedings of 9th world conference on Earthquake Engineering, Tokyo 1988, 593-598.

[10]. Reul O, Randolph M. F, Piled rafts in over consolidated clay: comparison of in situ measurements and numerical analyses, Geotechnique 2003, 53(3), 301-315.