Technical Report ITL-94-2
March 1994

US Army Corps of Engineers
Waterways Experiment Station

AD-A279 570

(1)

Computer-Aided Structural Engineering (CASE) Project

User's Guide for the Incremental Construction, Soil-Structure Interaction Program SOILSTRUCT with Far-Field Boundary Elements

by T. Kuppusamy, Mark A. Zarco
Virginia Polytechnic Institute and State University

Robert M. Ebeling

94-15532

417556 1098

WES

Approved For Public Release; Distribution Is Unlimited

94 5 23 116

Prepared for Headquarters, U.S. Army Corps of Engineers

The contents of this report are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such commercial products.

Computer-Aided Structural
Engineering (CASE) Project

Technical Report ITL-94-2
March 1994

User's Guide for the Incremental Construction, Soil-Structure Interaction Program SOILSTRUCT with Far-Field Boundary Elements

by T. Kuppusamy, Mark A. Zarco

Charles Via Department of Civil Engineering
Virginia Polytechnic Institute and State University
Blacksburg, VA 24061

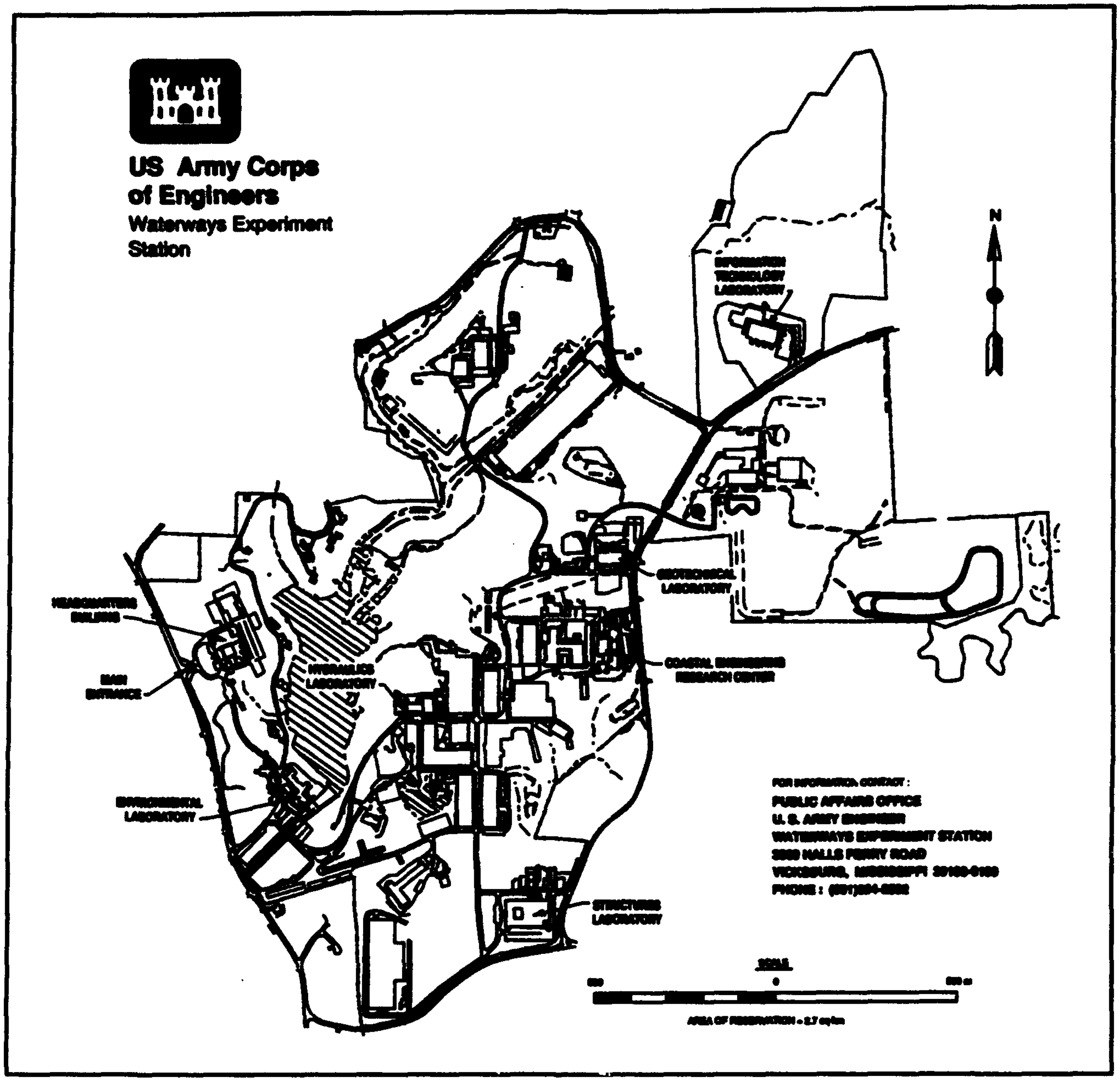
Robert M. Ebeling

U.S. Army Corps of Engineers
Waterways Experiment Station
3909 Halls Ferry Road
Vicksburg, MS 39180-6199

Final report

Approved for public release; distribution is unlimited

Prepared for U.S. Army Corps of Engineers
Washington, DC 20314-1000



Waterways Experiment Station Cataloging-in-Publication Data

Kuppusamy, T.
User's guide for the incremental construction, soil-structure interaction program SOILSTRUCT with far-field boundary elements / by T. Kuppusamy, Mark A. Zarco, Robert M. Ebeling ; prepared for Headquarters, U.S. Army Corps of Engineers.
81p. : ill. ; 28 cm. — (Technical report ; ITL-94-2
Includes bibliographic references.
1. Boundary element method. 2. Finite element method. 3. Soil-structure interaction — Computer programs. 4. SOILSTRUCT (Computer program). I. United States. Army. Corps of Engineers. II. U.S. Army Engineer Waterways Experiment Station. III. Title. IV. Computer-Aided Structural Engineering (CASE) Project. V. Series: Technical report (U.S. Army Engineer Waterways Experiment Station) ; ITL-94-2.
TA7 W34 no.ITL-94-2

PREFACE

This report describes the finite element computer program SOILSTRUCT used in the evaluation of soil-structure interaction of earth retaining structures. The initial version of the program was developed by Professors G. W. Clough and J. M. Duncan in 1969 and has been enhanced during the last 20 years by Professor Clough and his coworkers. This report documents the version of the program in which a procedure for performing soil-structure interaction analysis using the coupled-boundary element/finite element method is incorporated. The work was performed by Mark Zarco, Ph.D., student at Virginia Tech, under the guidance of Professor T. Kuppusamy, Professor of Civil Engineering, Virginia Tech. This work was funded by the U.S. Army Engineer Waterways Experiment Station (WES) under Contract No. DACW39-91-C-0085. Funding for the adaptation of the program and documentation was provided by Headquarters, U.S. Army Corps of Engineers (HQUSACE), under the Computer-Aided Structural Engineering (CASE) Project.

This report was prepared by Dr. T. Kuppusamy, Dr. Mark A. Zarco, and Dr. Robert Ebeling, Scientific and Engineering Applications Center, Computer-Aided Engineering Division (CAED), Information Technology Laboratory (ITL), WES. The work was managed and coordinated by Dr. Reed L. Mosher, Interdesciplinary Research Group, CAED, ITL, and Dr. Ebeling. All the work was accomplished under the general supervision of Dr. Reed Mosher, Acting Chief, CAED, and Dr. N. Radhakrishnan, Director, ITL.

At the time of publication of this report, Director of WES was Dr. Robert W. Whalin. Commander was COL Bruce K. Howard, EN.

CONTENTS

Page

CONVERSION FACTORS, NON-SI TO SI (METRIC) UNITS OF MEASUREMENT

Non-SI units of measurement used in this report can be converted to SI (metric) units as follows:

Multiply	By	To Obtain
cubic feet	0.2831685	cubic metres
feet	0.3048	metres
pounds (mass) per cubic foot	16.01846	kilograms per cubic metre
pounds (force) per square foot	47.88026	pascals
pounds (force) per square inch	6.894757	kilopascals
square feet	0.09290304	square metres
square inches	6.4516	square centimeters

USER'S GUIDE FOR THE INCREMENTAL CONSTRUCTION SOIL-STRUCTURE INTERACTION PROGRAM SOILSTRUCT WITH FAR-FIELD BOUNDARY ELEMENTS

PART I: DESCRIPTION OF PROGRAM SOILSTRUCT

Introduction

1. SOILSTRUCT is a general purpose finite element program for two-dimensional plane strain analysis of soil-structure interaction problems. It calculates displacements and stresses due to incremental construction and/or load application and is capable of modeling nonlinear stress-strain material behavior. The simulation of incremental construction may include embankment construction or backfilling, excavation, installation of a strut or tie-back anchor excavation support system, removal of the same system, and the placement of concrete or other construction materials. The incremental loading simulation may consist of the application of concentrated loads, boundary pressures, or loads due to temperature changes in non-soil materials.

2. The initial version of SOILSTRUCT was developed by Professors G. W. Clough and J. M. Duncan for use in the analysis of Port Allen and Old River U-frame locks (Clough and Duncan 1969). Since then, the program has been modified to expand the capabilities of the finite element constitutive models, load vector formulation algorithm, the size of the problem which may be analyzed, and the transfer of input, output, restart, and plot data files by means of disc storage. These modifications have been made in conjunction with a number of projects at the US Army Engineer Waterways Experiment Station (Ebeling 1990, Ebeling, Duncan and Clough 1990, Ebeling, Clough, Duncan and Brandon 1992, Regalado, Duncan and Clough 1992, Ebeling, Mosher, Abraham and Peters 1993). This version of the program reflects modifications made by Mark Zarco under the supervision of Professor T. Kuppusamy, to incorporate the capability of modeling far-field effects by boundary elements.

3. SOILSTRUCT has been coded in FORTRAN 77 language and consists of a main program and 33 subroutines named: DETNA, INITAL, STRSTF, QUAD, BAREL, EXCAV, EQNDFO, SURFLD, JTSTF, SUBSTP, JSTRES, SEEP, MODCAL, BUILD, OPTSOL, AUXOUT, STRESS, PRNCIP, PRNTFD, GETFIL, NOTENS, UNBALS, PRINTF0, PRNTJDM INTERF, PRINTFD, BEMSTF, MATRX, FUNC, MELAN, SETCON, NMATRX, and STFSYM. A user's guide for the SOILSTRUCT Program can be found in Appendix A. Appendix B contains the sequence of operations for the SOILSTRUCT Program.

Previous Contributions to SOILSTRUCT

4. Numerous versions of the SOILSTRUCT code have been written since it was introduced by Clough and Duncan in 1969. Most modifications were used to solve a specific soil-structure interaction problem. The resulting computer code continued to be referred to as SOILSTRUCT. Because of the lack of documentation within the computer code, the authors of this report were not able to identify the early contributions contained within the version that was used as the basis for the computer code described in this report. Since 1985, four analytical enhancements have been made to the version of SOILSTRUCT described in this report, two are known as "the alpha method," the third is a shear stress-deformation model for interface elements, and the fourth is the modeling of the far-field domain using boundary elements.

5. The alpha method was first developed for analyzing walls that are laterally loaded so heavily that gaps develop along the interface between the base and the foundation (Ebeling, Duncan and Clough 1990, and Ebeling, Clough, Duncan and Brandon 1992). This numerical method was later extended to two-dimensional elements and used to reduce numerical inaccuracies such as overshoot stresses in nonlinear soil elements which are in failure or near failure (Regalado, Duncan and Clough 1992). This situation occurs when walls retaining backfills that are at, or near, an active state of stress are analyzed.

6. A third enhancement contained within this version of SOILSTRUCT is the reintroduction of the hyperbolic shear-stress displacement relationship for the interface element (Ebeling, Peters, and Clough 1990). Although present in the original version of SOILSTRUCT, this nonlinear displacement relationship was missing from the version of computer code in which the alpha method was incorporated.

7. Coupling of the boundary element method of analysis with the finite element method is described in Part IV of this report.

Finite Elements and Boundary Elements Employed

8. Three types of finite elements are used to represent the behavior of different materials: (a) a two-dimensional continua element, (b) an interface element, and (c) a one-dimensional bar element.

9. A two-dimensional, subparametric, quadrilateral element (QM5) is used to represent the soil and most structural materials. Structural supports, such as the struts or tieback components of an excavation support system, are typically modeled as a spring support using bar elements. However, two-dimensional elements have been used to model these supports. The geometry of this element, developed by Doherty, Wilson

and Taylor (1969) is defined by four external nodes, while the displacement functions include an internal fifth node. To improve flexural response, a constant shear strain, calculated at the location of the internal fifth node, is imposed throughout the element. The QM5 element can be allowed to degrade to a triangular element by letting two adjacent nodes of the quadrilateral coincide.

10. The Goodman, Taylor and Breeke (1968) interface element is used to allow for relative movement between different materials such as between a soil backfill and a support wall. This element is defined by four nodes, with each of the two pairs of nodes having the same coordinates; thus, this type of element has no thickness.

11. One dimensional, two node bar or spring elements are used to model the behavior of a variety of structural systems. This includes the modeling of structural supports such as braces or tiebacks or the modeling of reinforcement placed within a soil backfill.

12. Linear boundary elements based on the Melan fundamental solution (Telles and Brebbia 1981), are used to model the far-field boundary.

PART II: MATERIAL STRESS-STRAIN BEHAVIOR

Introduction

13. Several modes of stress-strain behavior are utilized to represent the response of soil, construction materials, and the interface region between different materials.

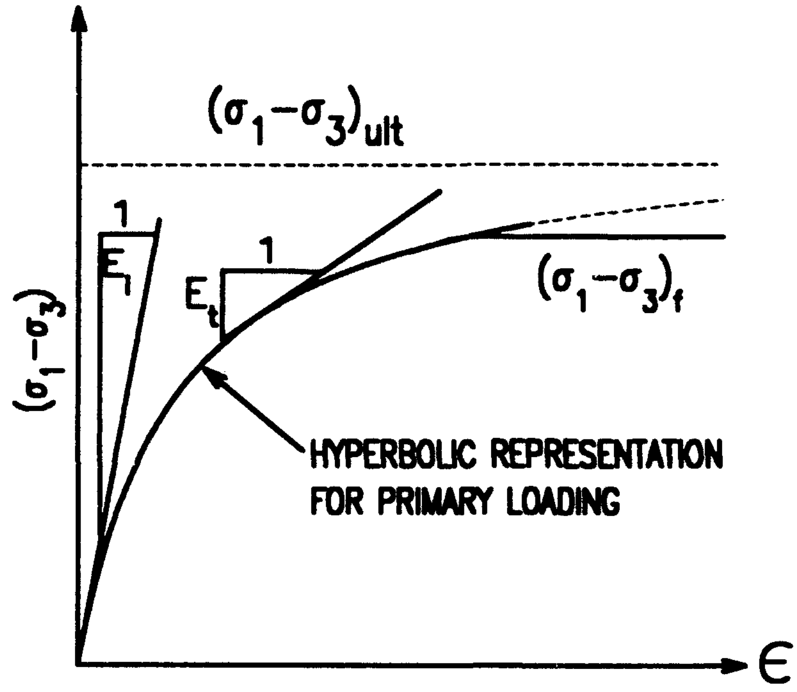
14. The constitutive relationship used for all two-dimensional elements is Hooke's law. SOILSTRUCT uses an incremental, equivalent linear method of analysis to model nonlinear material behavior. In this type of analysis, the incremental changes in stresses are related to the incremental strains through a linear relationship. This relationship is defined for each structural element by two engineering constants, the Young's moduli and the Poison's ratio.

Nonlinear Stress-Strain Response of Soil

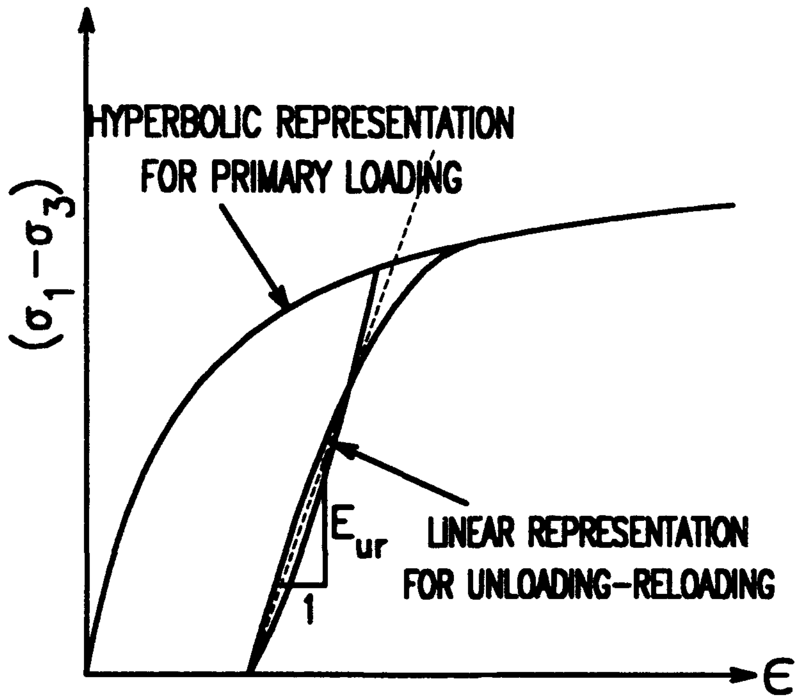
15. A plane strain, isotropic drained or undrained stress-strain soil model is incorporated within soil SOILSTRUCT. The program uses a nonlinear, stress-dependent hyperbolic curve to represent the relationship between stress-strain response during primary loading of the soil (Figure 1a) and a linear stress-strain response during unloading or reloading of the soil (Figure 1b). The unload-reload stress-strain response is applicable when the current stress state is less than that which has been applied previously; otherwise, the primary loading stress-strain is appropriate. Laboratory testing and interpretation procedures for determining the parameters used to define the soil model are described in Duncan, Byrne, Wong and Mabry (1978). A brief review of the hyperbolic model is given in paragraphs 18 to 22.

16. The nonlinear soil response to loading is modeled by performing a series of analyses in which each load is applied incrementally, with the total change in stress computed at the center of each soil element being equal to the sum of the incremental changes in stress over all the load steps. In general, the greater the curvature of the stress-strain relationship or the larger the magnitude of the applied load, the greater the number of load steps required to accurately model the nonlinear soil response. This may be achieved in two ways using SOILSTRUCT; either the total load approach using a greater number of incremental loadings, or during the course of each load case analysis, the load vector may be applied in a series of increments using the substep option.

17. Application of each loading in the finite element analysis results in a change in stress within each of the soil elements. In addition to the change in stress, there is a corresponding change in stiffness. Since each incremental analysis is performed assuming equivalent linear element response, SOILSTRUCT updates the value of the elastic moduli assigned to each soil element. To account for the change in stiffness that occurs during the application of a load increment, each incremental load calculation may



a. Hyperbolic representation of stress-strain curve for primary loading



b. Linear unloading-reloading stress-strain relationship

Figure 1. Hyperbolic model for stress-strain behavior (after Duncan, Byrne, Wong, and Mabry 1978)

be repeated using the iteration option. When the iteration option is invoked, the load vector is reapplied with a revised value for the element stiffness. The value assigned for the stiffness of the soil element reflect the average of the stress state developing at the end of the previous load case, or substep, and that which develops during the current iteration. However, when only one iteration is specified, the modulus values are calculated using the stresses developing at the end of the previous load increment. Upon completion of the last iteration for each load case or substep, the arrays tabulating the values of the total nodal point displacements and total element stresses are updated with the computed incremental values.

Primary Loading - Young's Moduli

18. Prior to each analysis a tangent Young's modulus E_t is assigned to each soil element. The stress-dependent value of E_t is computed using the relationship

$$E_t = E_i\left(1 - R_f \cdot SL\right)^2 \quad (1)$$

where

E_i = initial Young's modulus

R_f = failure ratio

SL = stress level

The initial Young's modulus E_i is equal to

$$E_i = KP_a\left(\frac{\sigma_3}{P_a}\right)^n \quad (2)$$

where

K = modulus number

P_a = atmospheric pressure

n = modulus exponent

σ_3 = minor principal stress

19. The proportion of mobilized shear strength for each soil element is reflected in the value of the stress level SL. SL is equal to the current deviator stress $(\sigma_1 - \sigma_3)$ divided by the deviator stress at failure $(\sigma_1 - \sigma_3)_f$, denoted by the subscript f.

$$SL = \frac{\sigma_1 - \sigma_3}{(\sigma_1 - \sigma_3)_f} \tag{3}$$

where σ_1 = major principal stress. The value of SL ranges from a value equal to zero to a value equal to unity. SL equal to zero indicates an isotropic stress state, while SL equal to unity corresponds to the complete mobilization of shear resistance within the soil element.

20. This version of SOILSTRUCT defines the deviator stress at failure using the original Duncan formulation (Duncan and Chang, 1970). In the original Duncan formulation, the value of the minor principal stress at failure is set equal to the current minor principal stress. The deviator stress at failure is given by

$$(\sigma_1 - \sigma_3)_f = \frac{2c\cos\phi + 2\sigma_3 \sin\phi}{1 - \sin\phi} \tag{4}$$

where

c = cohesion intercept

ϕ = angle of internal friction

21. The failure ratio R_f relates the ultimate deviator stress $(\sigma_1-\sigma_3)_f$.

$$(\sigma_1 - \sigma_3)_f = R_f(\sigma_1 - \sigma_3)_{ult} \tag{5}$$

The ultimate deviator stress is the asymptote to the stress-strain hyperbola, as in Figure 1a. The value of R_f is always less than unity and varies from 0.5 to 0.9 for most soils.

Unload-Reload Stress-Strain Behavior - Young's Modulus

22. During unloading or reloading, when the current deviator stress is less than that which has been applied during previous loadings, a stress-dependent, linear response is assumed, as shown in Figure 1b. In this case, the value of E_{ur} is computed using

$$E_{ur} = K_{ur} P_a \left(\frac{\sigma_3}{P_a}\right)^n \tag{6}$$

where K_{ur} = unload-reload modulus number.

Poison's Ratio

23. The second elastic parameter used to define the material behavior of soil is the Poison's ratio ν. When using Poison's ratio, two values are specified: a constant value which is applicable for all states of stress prior to failure, $SL < 1$, and the value of Poison's ratio applicable when the shear strength of the soil is fully mobilized, $SL = 1$.

24. When computing initial stress by gravity turn-on analysis, the value of Poison's ratio used for the soil model may not be suitable for initial stress conditions. For example, the Poison's ratio for undrained analysis is generally taken to be nearly 0.5 because saturated soil is near incompressible. By contrast, the initial stress conditions should be fully drained. Therefore, the initial stress computations are based on a value of Poison's ratio that gives the correct ratio σ'_v/σ'_h for level ground conditions; that is $\sigma'_v/\sigma'_h = K_O$. Using the value of K_O input, the Poison's ratio used for initial conditions is given by

$$\nu = \frac{K_o}{1+K_o} \tag{7}$$

Jaky (1948) suggested that K_O may be approximated using the relationship

$$K_o = 1 - \sin\phi \tag{8}$$

where ϕ is the angle of internal friction.

Overshoot in Soil Elements - ALPHA Method

25. The alpha method is used to analyze walls retaining backfills containing soil elements which are in or near failure (Regalado, Duncan and Clough 1992). The alpha method is used to reduce numerical inaccuracies such as overshoot stresses in these nonlinear soil elements.

26. The finite element program SOILSTRUCT employs hyperbolic relationships to model the nonlinear stress-strain behavior of soils. As discussed previously, nonlinear behavior is approximated by performing the analyses in a series of loading increments or "steps". Within each step of the analysis the materials are treated as if their behavior is linear. The modulus values assigned to each element are adjusted in accordance with their stresses to simulate nonlinear behavior.

27. The way the stresses in an element might change during one step of an analysis is shown in Figure 2. Figure 2(a) shows the state of stress at the center of an element

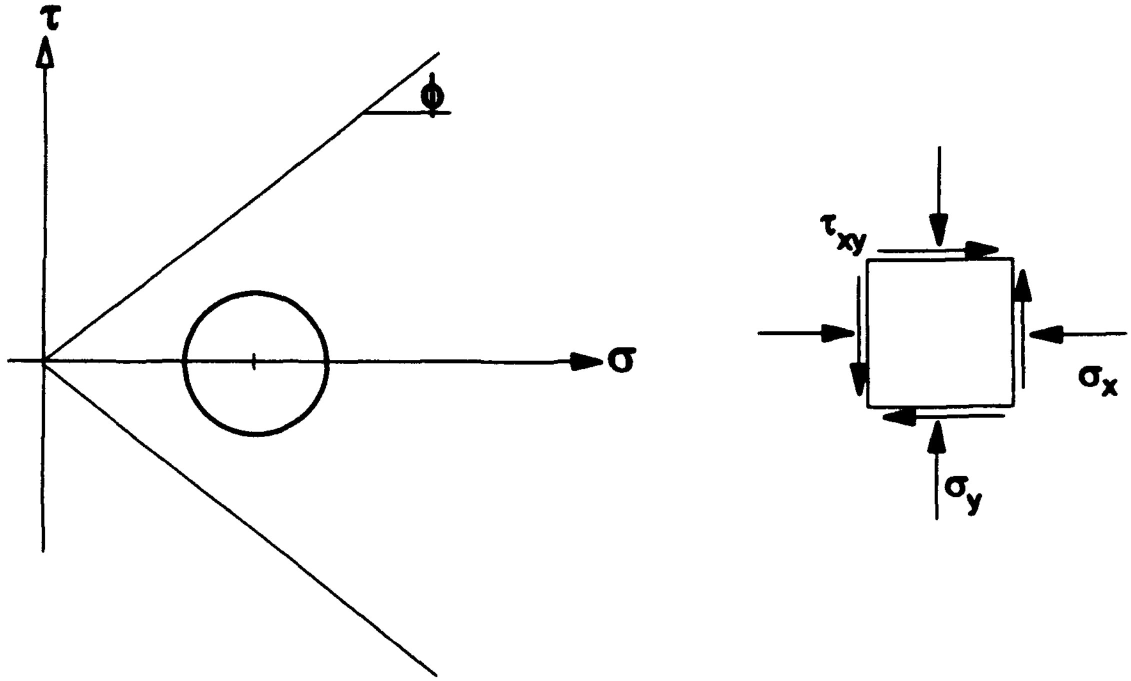
and the corresponding Mohr's circle at the beginning of a load step. Figure 2(b) shows the state of stress and the Mohr's circle for the same element after a load step that brought about the changes in stresses $\Delta\sigma_x$, $\Delta\sigma_y$ and $\Delta\tau_{xy}$. Figure 2(b) illustrates that after the addition of the stress increments, the resulting combination of stresses may result in an unacceptable state of stress. A Mohr's circle of stress above the failure envelope is indicative of overshoot.

28. The ideal situation would be that the load applied during a load step was just large enough to bring the most severely stresses element to failure. This situation would correspond to a Mohr's circle which was tangent to the failure envelope. The purpose of the alpha method is to achieve results closer to the ideal situation, in which no element is overstressed. The basic idea behind the method is to reduce the magnitudes of the loads applied in any step of the analysis to a point that no soil element incurs a stress level greater than unity.

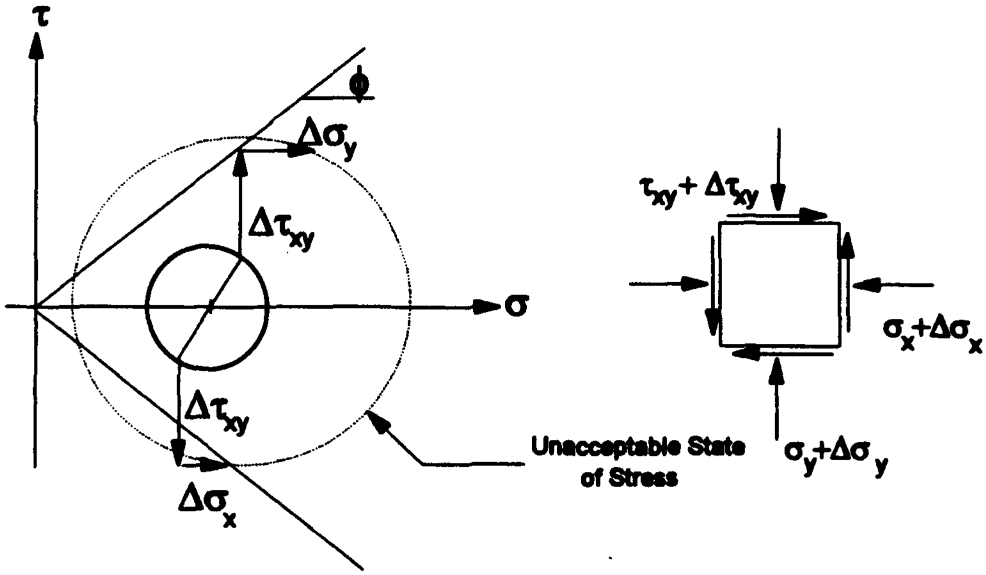
29. Figure 3(a) illustrates the same situation shown in Figure 2(b) in which the stress increments during a load step result in overshoot in a soil element. Figure 3(b) shows a Mohr's circle which has been reduced in size so that it is tangent to the failure envelope. This reduction can be accomplished by multiplying the magnitudes of all the stress change components ($\Delta\sigma_x$, $\Delta\sigma_y$ and $\Delta\tau_{xy}$) by a factor α (hence the name "alpha" method). Reducing the stresses by this factor in all the elements in the mesh can be accomplished by reducing the load by the same factor, since the system beh[illegible] linearly during each step of the analysis.

30. A situation may arise in which several elements fail, with each one requiring a different value of the reduction factor alpha. When this situation occurs, the most severely stressed element will require the smallest value of alpha. Using the alpha value to reduce the load will ensure that failure takes place only in the worst stresses element and nowhere else in the mesh. This worst stressed element is then assigned properties that reduce its ability to carry more load. After this change in the properties of the most severely stressed element, the remaining portion of the load for this particular load step is applied. This process of reducing the load in order to allow only one element to fail at a time is repeated until the entire load for the load step has been applied.

31. In the process of assigning properties to a failed element, consideration is given to avoiding too abrupt changes in modulus value. When an element first incurs a stress level that indicates failure, its modulus is reduced to one-tenth its value before failure. This process of reducing the modulus is repeated in subsequent loadings for as long as the element is in a state of failure. In order for the modulus value not to become unrealistically small, the program user can specify a lower limit for the modulus to be assigned to failed elements. When the process of continuous reduction in the value of modulus for a failed element results in a value smaller than the lower limit, its value is kept at the lower limit. It is also at this stage that the element is assigned a failure

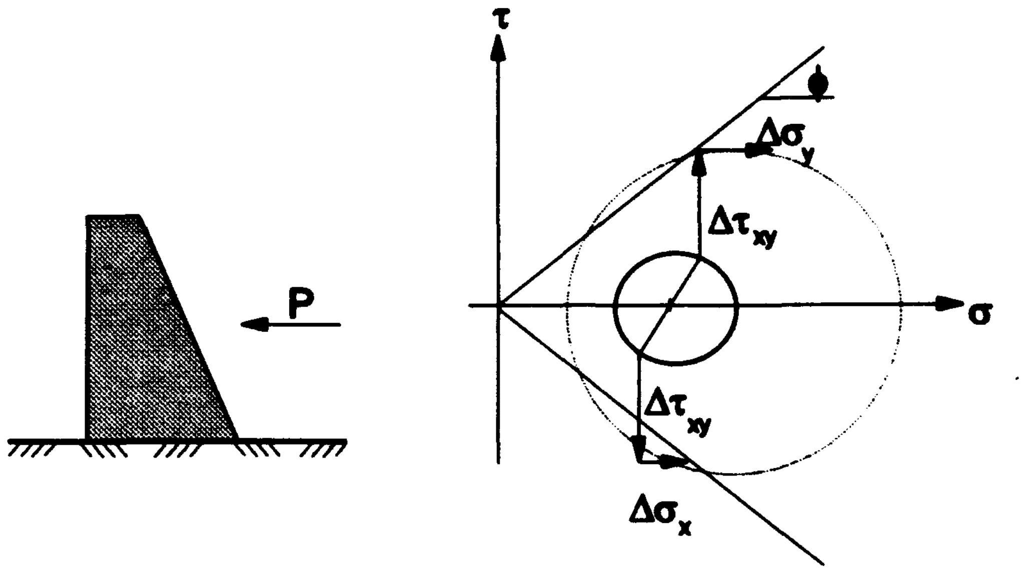


(a) Before Load Step

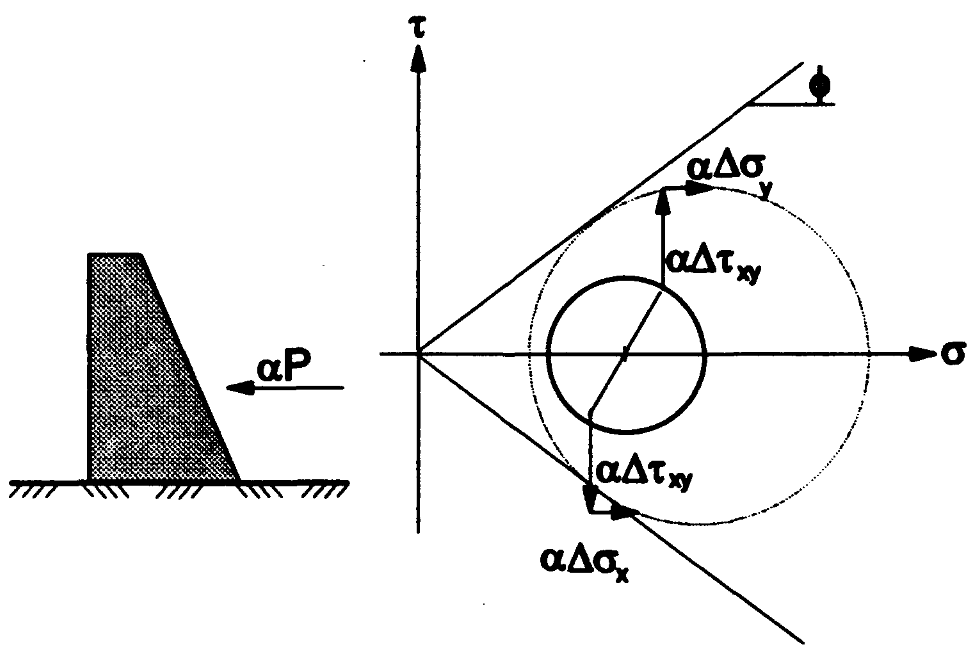


(b) After Load Step

Figure 2. Stress Changes in a Soil Element during a Load Step and the Occurrence of Overshoot (Regalado, Duncan, and Clough, 1992).



(a) Application of Full load and Corresponding Stress Component Increases in a Soil Element



(b) Factored Load and Corresponding Stress Component Increases in Soil Element

Figure 3. Factoring the Load to Eliminate Overshoot (Regalado, Duncan and Clough. 1992)

value of Poisson's ratio. The element retains these property values for as long as it remains in failure.

32. Although this procedure allows a gradual transition to failure, it does not completely remove the load carrying capacity of a failed element. Consequently, failed elements are still able to carry some additional load, making it possible for them to exceed a stress level of unity by some usually small amount.

33. In summary, the steps in the alpha method are as follows: a. if overshoot occurs in any soil element, reduce the stresses to eliminate overshoot in the most severely stressed element; b. assign post-failure properties to this element; and c. apply the remaining portion of the load and continue the analysis. The data input for the alpha method are described in Section 4. of Appendix A. For additional details regarding the use of the alpha method for controlling overshoot in two-dimensional elements, see Chapter 2 of Regalado, Duncan and Clough, 1992.

PART III: MODELING STRUCTURAL ELEMENTS

Introduction

34. This chapter describes how the stress-strain behavior of different structural elements used are modeled. Included is a discussion on material response of structural elements modeled using two-dimensional QM5, bar and interface elements. Also discussed is the alpha method for modeling base separation.

Structural Material Response

35. Structural materials such as wood, concrete, or steel are modeled using two-dimensional QM5 elements with linear elastic stress-strain behavior assumed. Support elements such as struts or anchors are typically modeled using one-dimensional bar members, and are also assumed to behave linearly. Bar elements as formulated within in SOILSTRUCT have the capability to respond in compression only, in tension only, or both tension and compression. In addition, slack in the support system at the time of installation may be accounted for by specifying an initial value of displacement for the bar element.

Interface Response

36. Interface elements are used to allow for relative movement between different material regions, such as between a soil backfill and a support wall. These element are defined by four nodes, each node having two degrees of freedom; each of the two pairs of nodes sharing the same coordinates. The interface element, therefore, is of finite length but zero thickness.

37. The properties of interface elements are defined by an interface normal stiffness k_n and an interface shear stiffness k_s. These values of stiffness relate the average relative displacement normal to the interface element Δ_n and average relative shear displacement Δ_s to the corresponding normal stress σ_n and shear stress τ by the equations

$$\sigma_n = k_n \Delta_n \tag{9}$$

and

$$\tau = k_s \Delta_s \tag{10}$$

The units of k_n and k_s are force per cubic length.

38. The initial value of k_n is set equal to 1×10^8 within the program. This value for k_n ensures that the normal relative displacement of the interface element is insignificant when English units (feet, pounds) or SI units (meters, kilonewtons) are used. If other units are used, the value of the normal stiffness may need to be changed to a higher value.

39. Two types of interface shear response are modeled, a bilinear shear stress-displacement relationship shown in Figure 4a, and a hyperbolic shear stress-displacement relationship shown in Figure 4b. In the bilinear model, the value assigned to k_s is a constant so long as the average shear stress τ along the interface is less than the shear strength. If the shear strength of the interface element is fully mobilized, which occurs when τ is equal to τ_f, k_s is set equal to zero. When the normal stress σ_n is greater than or equal to zero, the value of τ_f is given by the relationship

$$\tau_f = c_i + \sigma_n \tan \delta \tag{11}$$

where

c_i = cohesion intercept along the interface

δ = angle of internal friction along the interface

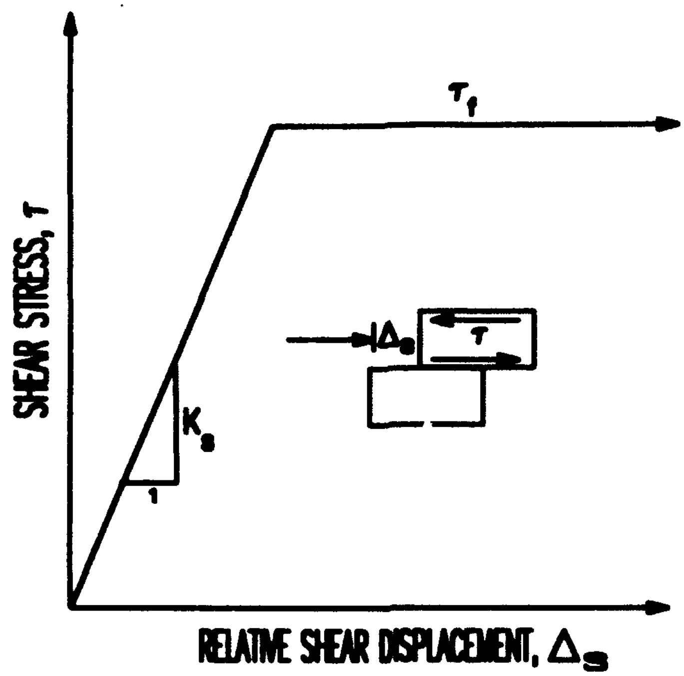
and shown in Figure 5a. When σ_n is less than zero, τ_f is computed using

$$\tau_f = \sigma_n \left(\frac{c_i}{\sigma_t} \right) \tag{12}$$

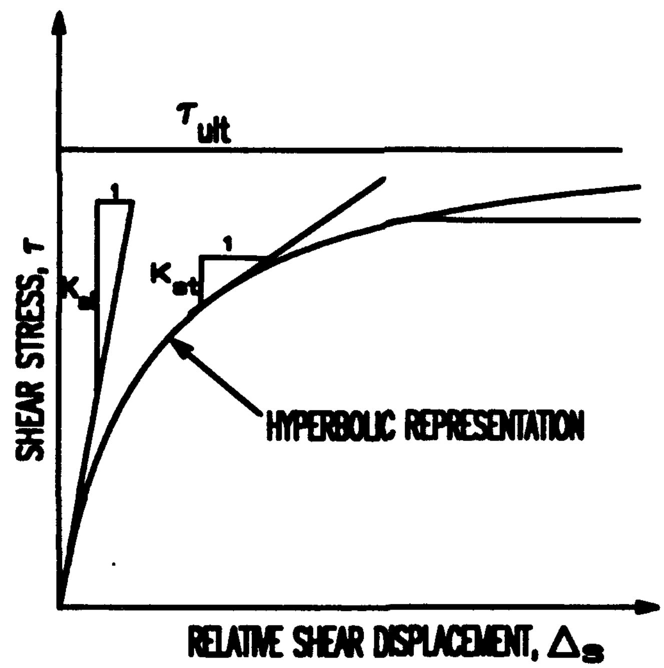
where σ_t = tensile strength.

40. Direct shear test results on soil-to-concrete interface and soil-to steel interface by Potyondy (1961), Clough and Duncan (1969), and Peterson et al. (1976) have shown that the value of δ is proportional to the angle of internal friction of the soil. The value of the constant of proportionality is dependent upon both the type of soil and the type of material comprising the surface of the structure.

41. The direct shear tests performed by Clough and Duncan (1969) and Peterson et al. (1976) have shown that for some materials, such as sand-to-concrete interfaces, the interface response during shear is nonlinear and dependent upon the normal stress. A nonlinear, stress-dependent hyperbolic curve is used to represent the relationship between shear stress and average relative shear displacement developing during primary loading of the interface (Figure 5a) and a linear shear stress-relative displacement response during unloading or reloading of the interface. The stress-dependent value of k_{st} is computed using the relationship



a. Bilinear stress-strain model representing interface behavior



b. Hyperbolic representation of the variation of shear stress with relative shear displacement (k_{st} = tangent interface shear stiffness, k_{si} = initial interface shear stiffness)

Figure 4. Bilinear and hyperbolic models for interface shear stress-relative shear displacement behavior after Clough and Duncan(1969)

$$k_{st} = k_{si}\left(1 - R_{fi} \cdot SL_i\right)^2 \qquad (13)$$

where

k_{si} = initial interface shear stiffness
R_{fi} = failure ratio
SL_i = stress level

The initial interface shear stiffness k_{si} is equal to

$$k_{si} = K_j \gamma_w \left(\frac{\sigma_n}{P_a}\right)^{n_j} \qquad (14)$$

where

K_j = interface modulus number
γ_w = unit weight of water
n_j = interface modulus exponent

42. The proportion of mobilized shear strength for each interface element is reflected in the value of the stress level SL_i. SL_i is equal to the current shear stress τ divided by the stress at failure, τ_f.

$$SL_i = \frac{\tau}{\tau_f} \qquad (15)$$

τ_f is computed using either Equation 11 or 12. SL_i ranges in value between zero and one.

43. The failure ratio R_{fi} relates the ultimate shear stress τ_{ult} to the shear stress at failure

$$\tau_f = R_{fi} \cdot \tau_{ult} \qquad (16)$$

The ultimate shear stress is the asymptote to the shear stress-relative shear displacement hyperbola, as shown in Figure 4.b. Direct shear stress tests on sand-to-concrete interfaces by Peterson et al. (1976) have shown the value of R_{fi} typically ranges in value from 0.3 to 1.0.

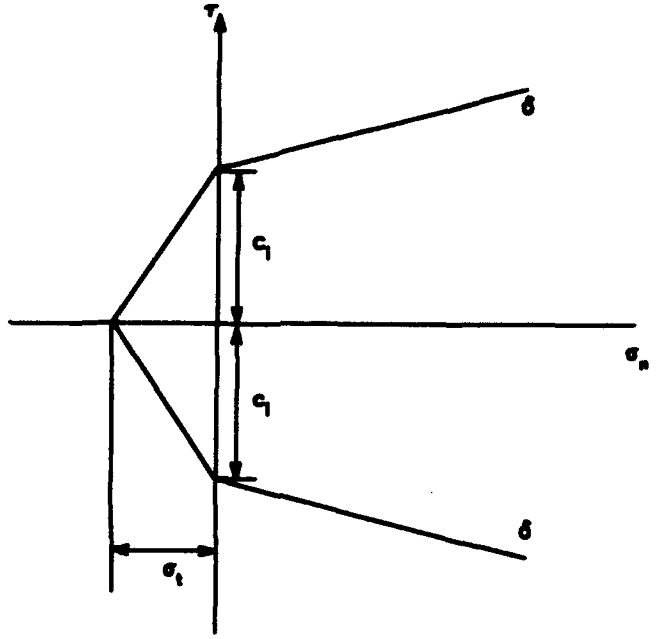
44. The relationship between the average normal stress along the interface and the tensile strength is shown in Figure 5b. The value of k_n is a constant value equal to 1×10^8 when σ_n is greater than or equal to σ_t. If σ_n is less than σ_t, k_n is set equal to zero, assuring that additional tensile stresses do not accrue upon subsequent loadings. This procedure allows for separation to occur between two adjacent regions of mesh along interface elements during the course of an incremental analysis. If the separation closes during subsequent loadings, k_n is reset to 1×10^8.

Base Separation Analysis - ALPHA Method

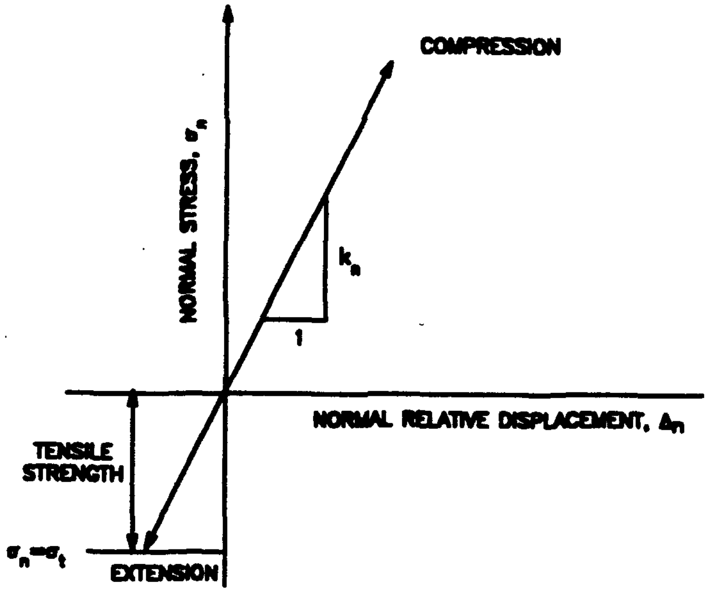
45. The alpha method is used to analyze walls that are loaded so heavily horizontally that gaps develop along the interface between the base and the foundation (Ebeling, Duncan, and Clough, 1990 and Ebeling, Clough, Duncan and Brandon 1992). In the finite element model of the soil-structure interaction problems, the boundary between the base of the monolith and its foundation is modeled using interface elements. This version of SOILSTRUCT checks for separation of the base of the wall from its foundation after each increment of loading. Base separation occurs when tensile stresses within an interface element center are computed to exceed the tensile strength along the interface. At this stage in the analysis, the alpha method is initiated by the computer code. The purpose of the alpha method is to redistribute the tensile stress and shear stress along the separated interface element to the adjacent elements. Details regarding this numerical method are summarized in the following paragraphs.

46. After each load step is completed, all interface elements are checked for the development of tensile stress at their centers. This occurs prior to updating the stress, displacement, and modulus arrays. If no tensile stresses are found, the analysis proceeds with the array updates and the next increment of load as usual. When tensile stresses are observed in the interface element(s), the following series of computations (idealized in Figure 6 for a single interface element developing tension after application of the full load increment $\{Q_0\}$) would be made prior to any array updates or the analyze for the next load increment:

a. For each interface element that develops tensile stress at its center, the fraction of the applied force that would result in zero normal center stress is computed. This fraction is referred to as alpha and is computed for all elements that develop tensile stress at their centers. Each of these alpha values may be described as the fraction of the total force which results in zero overshoot normal force for that interface element. The linear relationship is due to the incremental linear elastic analysis procedure used in the program. Due to the mechanics of the crack development, the interface element closest to the heel, which has not developed tensile stress at its center in a previous load increment, possesses the smallest value of alpha.

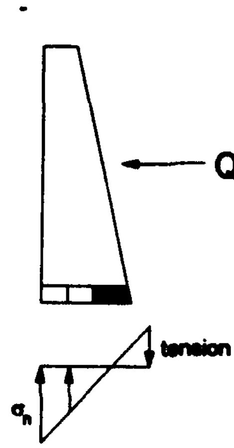


a. Strength criteria



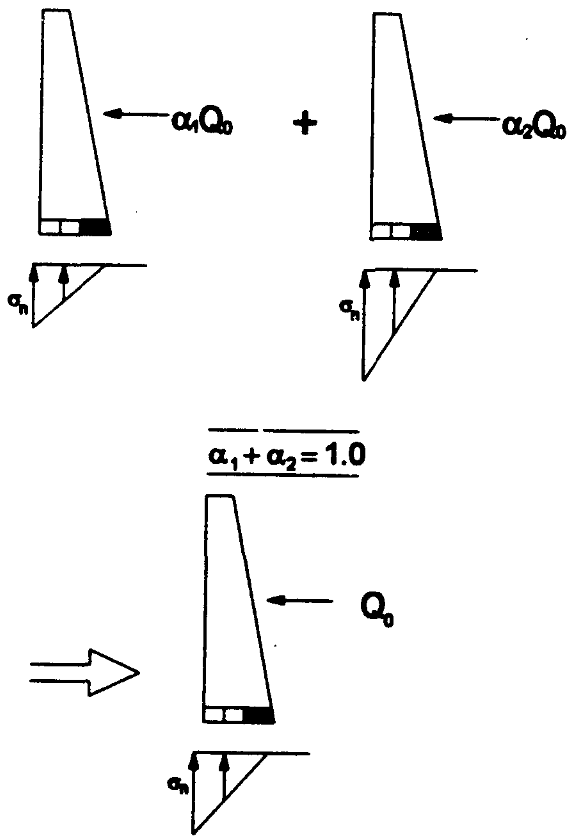
b. Normal stress-normal relative displacement behavior

Figure 5. Interface strength criteria and normal stress-normal relative displacement relationship.



<u>**Alpha Method**</u>

(a) Apply full load increment {Q_0} to develop tension in single interface element.



(b) Apply force loads: $\alpha_1 Q_0 + \alpha_2 Q_0$

Figure 6. Idealization of the alpha method.
(Ebeling, Clough, Duncan and Brandon, 1992)

b. The load vector $\{Q\}$ is then redefined as the product of the smallest value of alpha, α_1, and the original load vector for this load step $\{Q_0\}$.

$$\{Q\} = \alpha_1 \{Q_o\} \qquad (17)$$

A new analysis is conducted using the reduced load vector, and all displacements and stresses are updated.

c. Zero normal and shear stiffness is then assigned to the interface element. This prevents the accrual of stress described in step d.

d. For the critical interface element, the tension stress at its center is now zero, but artificial restraining forces are needed to reduce the built-in shear stress to zero, The artificial restraining force Q_S is computed for the element. Using the unbalanced load procedure, Q_S is:

$$Q_s = \iiint_V [B]^T \{\sigma_s\} dV \qquad (18)$$

where $\{\sigma_S\}$ is the element shear stress to be restrained by the nodal forces. It is equal to the actual linear shear stress distribution across the element. $[B]^T$ is the transformation matrix that relates element strains to the nodal point displacements. In the program code, this is accomplished by subtracting this equivalent shear stress from the stress regime existing across the entire element.

e. Since the force Q_S does not actually exist, an analysis is made of the entire mesh to these nodal point forces applied in the opposite direction at the nodes. All displacements and stresses are updated. Essentially, the shear stresses are redistributed. This step, applied in conjunction with step d, maintains equilibrium within the system.

f. The next fraction of the initial load to be applied is computed. The load vector $\{Q\}$ is defined as;

$$\{Q\} = \alpha_i \cdot \left(1 - \sum \alpha\right) \cdot \{Q_o\} \qquad (19)$$

where

α_i = the smallest of the remaining values of alpha

$\Sigma\alpha$ = the sum of previously applied alpha values

$\{Q_0\}$ = the smallest of the remaining values of alpha

The analysis is then conducted using $\{Q\}$, and all resulting displacements and stresses are updated.

g. Steps c through f are repeated until the total original load vector for this load step $\{Q_0\}$ is applied. Then the conventional analysis is resumed with the application of the next load increment.

For additional details regarding the alpha method and the unbalanced load procedure, see Part IV of Ebeling, Clough, Duncan and Brandon, 1992.

47. A bilinear shear stress-displacement relationship shown in Figure 4a and a bilinear normal stress-displacement are used for the interface elements along the base of the wall. Use of the alpha method is restricted to walls with horizontal wall to foundation interfaces. The data input for the base separation analysis using the alpha method is described in Section 4 of Appendix A. These interface elements must be assigned consecutive numbers along the base of the structure.

PART IV: MODELING THE FAR-FIELD BOUNDARY

Introduction

48. This version of SOILSTRUCT has the capability of taking into consideration the effects of the far-field domain by using boundary elements based on a half-plane fundamental solution. Figure 7 illustrates a typical soil-structure interaction problem where far-field domain is modeled using boundary elements., while the structure and adjacent soil domain is modeled using finite elements.

49. This sections describes the procedure used to model the far-field of a soil-structure interaction problem using boundary elements. A brief description of the boundary element used is discussed in the next section. The substructure method for coupling the boundary element and finite element system is described in succeeding sections.

Boundary Elements

50. The boundary element method is a numerical technique for solving boundary value problems. For an elasticity problem, the method involves discretizing the boundary of the problem into a series of elements. Over each of these elements, the displacements and tractions are chosen to be piecewise interpolated between element nodal points. An integral equation is obtained from the governing differential equations by successive application of integration by parts. This integral equation is applied to each nodal point on the boundary and the resulting integrals are evaluated (usually using a numerical quadrature) over each element. A system of linear equations in terms of the nodal boundary displacements and tractions results. Prescribed boundary conditions are then applied. The resulting system of equations is then solved for the unknown nodal boundary displacements and tractions.

51. SOILSTRUCT uses the direct boundary element method. This method results in a system of equations of the form:

$$[H]\{u\} = [G]\{p\} \tag{20}$$

where $\{u\}$ are the nodal boundary displacements and $\{p\}$ are the nodal boundary tractions. The matrices $[H]$ and $[G]$ are called the matrices of *influence coefficients* and depend on the type of *fundamental solution* used as well as the manner in which the displacements and tractions vary between nodal points. A more detailed discussion of the direct boundary element method can be obtained from Brebbia (1984).

52. The boundary element formulation used for SOILSTRUCT is based on the infinite half-plane fundamental solution. This fundamental solution gives the displacements and stresses that result due to the application of a concentrated load to the interior of an elastic half-plane. The solution to this problem was first presented by

Melan (1932). However, in the original work, there seem to be errors which were pointed out and corrected by Telles and Brebbia (1981).

53. In this version of SOILSTRUCT, linear isoparametric boundary elements are used. The geometry of the element, as shown in Figure 8, is defined using two points (x_1,y_1) and (x_2,y_2). Within the element, the cartesian coordinates (x,y) interpolated from the local coordinate ξ using the relationship

$$x=\sum_{i=1}^{2} x_i\,\psi(\xi)_i \qquad\qquad y=\sum_{i=1}^{2} y_i\,\psi(\xi)_i \tag{21}$$

where $\psi_1(\xi)$ are the lagrangian interpolating functions given by

$$\psi_1=\frac{(1-\xi)}{2} \qquad\qquad \psi_2=\frac{(1+\xi)}{2} \tag{22}$$

In a similar manner, the displacements and tractions along the boundary are interpolated in terms of the nodal values using the relationship

$$u_x=\sum_{i=1}^{2} u_{xi}\,\psi(\xi)_i \qquad\qquad u_y=\sum_{i=1}^{2} u_{yi}\,\psi(\xi)_i \tag{23}$$

$$t_x=\sum_{i=1}^{2} t_{xi}\,\psi(\xi)_i \qquad\qquad t_y=\sum_{i=1}^{2} t_{yi}\,\psi(\xi)_i \tag{24}$$

where u_{xi} and u_{yi} are the x and y components of the displacement at node i. Similarly, t_{xi} and t_{yi} are the x and y components of the traction at node i.

54. One limitation of the boundary element method is that for problems involving non-homogenous or non-linear material properties, the solution procedure becomes extremely complicated. To keep the solution procedure simple, the far field is assumed to be a homogenous linearly elastic (isotropic) half-plane. A single value of E and ν is specified for the entire far field.

Coupled Boundary Element-Finite Element Method

55. To couple the boundary element method with the finite element method, the entire far field is treated as a substructure. This is done by converting system resulting from the direct boundary element method as described in equation (20) into the form of the direct stiffness method:

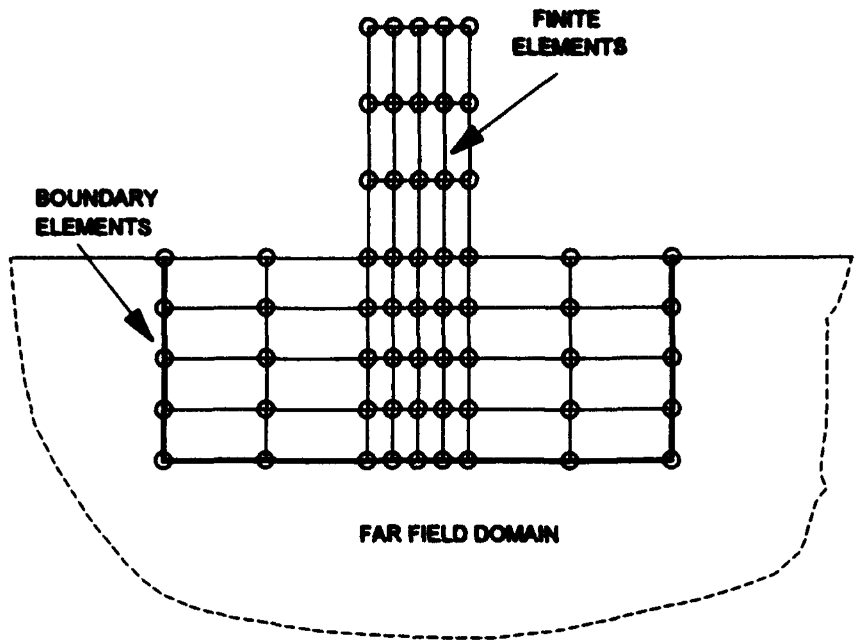


Figure 7. A typical soil-structure interaction problem with the far field-domain modeled using boundary elements.

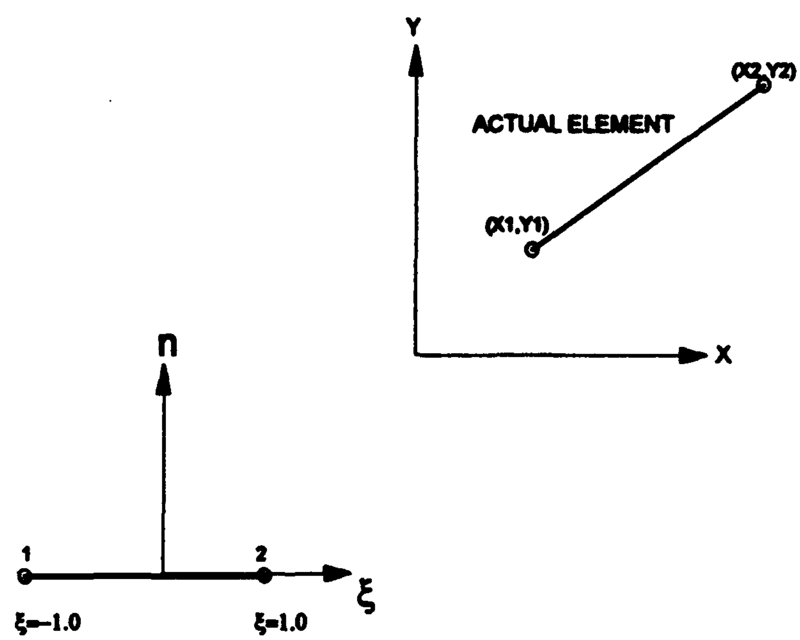


Figure 8. Linear boundary element in cartersian and local coordinates.

$$[\hat{K}]\{u\} = \{F\} \quad (25)$$

$\{u\}$ are the nodal displacements, $\{F\}$ are the nodal forces, and $[\hat{K}]$ is the equivalent stiffness matrix for the boundary element system. In this form, the boundary element stiffness matrix can be assembled into the global stiffness matrix just like any finite element.

56. The equivalent stiffness matrix for the boundary element system is given by

$$[\hat{K}] = [M][G]^{-1}[H] \quad (26)$$

where $[G]$ and $[H]$ are the matrices of influence coefficients described in equation (20), and $[N]$ is a matrix used to convert nodal tractions to nodal forces using the following relationship

$$\{F\} = [M]\{p\} \quad (27)$$

57. For a single boundary element with linearly varying tractions, the transformation matrix $[M]$ is given by

$$M = \frac{l}{6}\begin{bmatrix} 2 & 0 & 1 & 0 \\ 0 & 2 & 0 & 1 \\ 1 & 0 & 2 & 0 \\ 0 & 1 & 0 & 2 \end{bmatrix} \quad (28)$$

where $l = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}$. The M matrix for the entire system may be obtained by assembling together M matrices of the individual boundary elements.

58. SOILSTRUCT is an incremental, equivalent linear analysis program. The mesh used must have a sufficient number of elements so that two dimensional finite elements are used to model the region in which nonlinear stress-strain response is to be anticipated while boundary elements are used to model the region where linear stress-strain response is to be anticipated.

59. The stiffness matrix $[\hat{K}]$ is always unsymmetric. To preserve the symmetry of the finite element system, only the symmetric part of $[\hat{K}]$ is used in SOILSTRUCT.

The (i,j)th element of the symmetric stiffness matrix $[\hat{K}]^s$ can be obtained using the following relationship:

$$\hat{k}_{ij}^s = \frac{1}{2}\left(\hat{k}_{ij} + \hat{k}_{ji}\right) \tag{29}$$

Brebbia and Walker (1972) have shown that the symmetric matrix $[\hat{K}]^s$ resulting from equation (29) corresponds to a least squares approximation to the unsymmetric matrix $[\hat{K}]$.

60. Zarco (1993) pointed out that for a boundary element formulation using the Melan fundamental solution, the elements of the skew-symmetric part of $[\hat{K}]$ when normalized with respect to the diagonal terms of $[\hat{K}]$ in general are very small:

$$\left|\frac{\hat{k}_{ij} - \hat{k}_{ji}}{\hat{k}_{ii}}\right| \leq 0.3\% \quad \forall i,j \tag{30}$$

For a problem with symmetric geometry and loading, if only half of the problem's domain is modeled using the "Method of Images", the resulting stiffness matrix $[\hat{K}]$ becomes more unsymmetric :

$$\left|\frac{\hat{k}_{ij} - \hat{k}_{ji}}{\hat{k}_{ii}}\right| \leq 1.0\% \quad \forall i,j \tag{31}$$

However, the additional asymmetry introduced by the "Method of Images" does not have a very significant effect when $[\hat{K}]$ is assembled into the global finite element stiffness matrix $[K]$. This is due to the symmetric positive definite property of the finite element stiffness matrix. If the unsymmetric stiffness matrix $[\hat{K}]$ were assembled in to the global finite element stiffness matrix, the skew-symmetric terms of the resulting stiffness matrix $[K]$ would be very small:

$$\left|\frac{k_{ij} - k_{ji}}{k_{ii}}\right| \leq 0.01\% \quad \forall i,j \tag{32}$$

where k_{ij} is the (i,j)th element of the global stiffness matrix $[K]$ resulting from the assembly of $[\hat{K}]$ into the global finite element stiffness matrix.

61. Zarco (1993) developed another coupled BEM-FEM program, BEFEC, which uses the full unsymmetric stiffness matrix $[\hat{K}]$. This program is capable of solving non-associated plasticity problems using the Newton-Raphson method. For non-associated plasticity problems, it is necessary to retain the unsymmetric portion $[\hat{K}]$ of because the global stiffness matrix $[K]$ resulting from the finite element method is also unsymmetric. For the linear case, analyses performed by Zarco using BEFEC indicate discarding the skew-symmetric part of $[\hat{K}]$ has very little effect on the resulting displacements and stresses.

62. SOILSTRUCT models the nonlinear stress-strain behavior using a series of equivalent linear analyses in which the finite element stiffness matrix is symmetric. Only the symmetric part of $[\hat{K}]$ is used in SOILSTRUCT so as to preserve the symmetry of the finite element system. Because the analyses are linear, discarding the skew-symmetric part of $[\hat{K}]$ does not significantly affect the resulting displacement stresses.

PART V: USE OF SOILSTRUCT PROGRAM

Sign Convention and Coordinate System

63. All input data and results are specified using a right-hand coordinate system; the x-axis being horizontal and positive to the right and the y-axis being vertical and positive upwards. The sign convention for stresses acting at the center of a two-dimensional element is shown in figure 9a. Compressive stresses are taken to be positive.

64. Stresses for interface elements are defined with respect to their local axes along the length of the interface x' as defined by the I and J nodes, and normal to the element y' as shown in Figure 9b. Positive normal stresses are compressive. Positive shear stresses act in the positive x' direction along the length of the interface as shown in Figure 9b.

65. Positive forces are taken to be compressive in all one-dimensional bar elements with the exception of bar reinforcement elements. For these elements, positive forces are taken to be tensile.

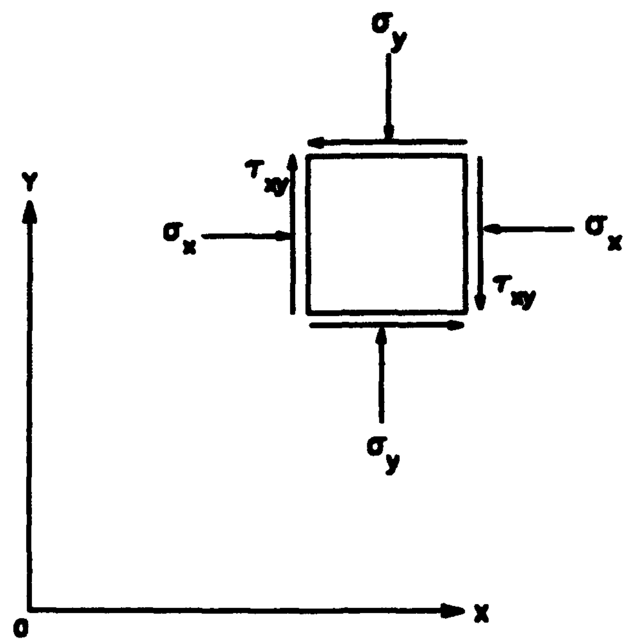
Units

66. Any consistent set of units can be used with SOILSTRUCT, with one cautionary note. The normal stiffness of interface elements is arbitrarily set to a value of 1×10^8, independent of units, as discussed in paragraph 38.

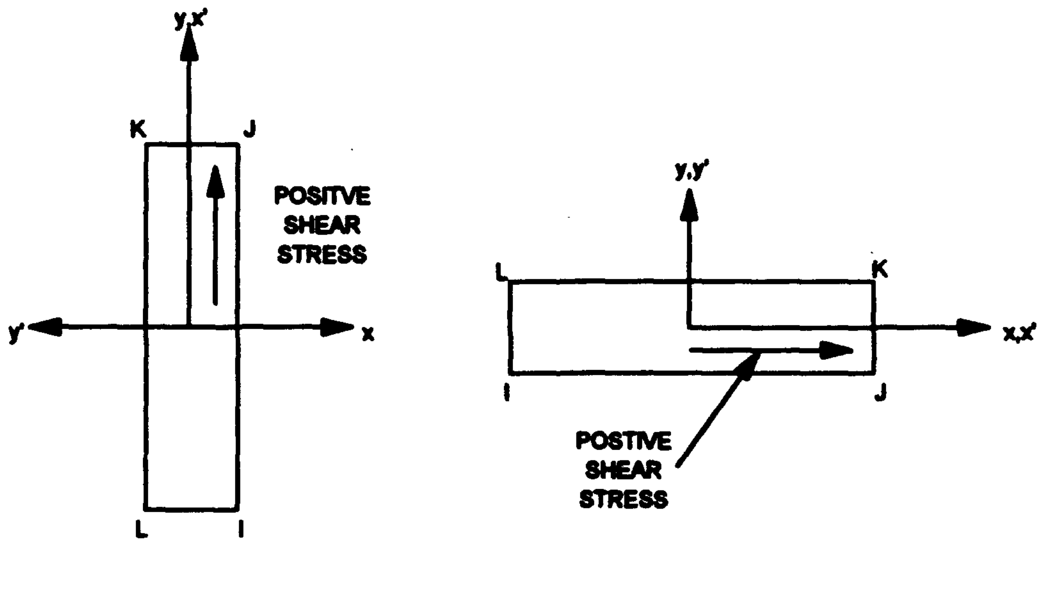
Capacity

67. The capacity of the program is determined by the size of variables in the common block and dimension statements. The global stiffness matrix [K] is stored within the program vector SN. The dimension SN is equal to 1,600,000, and the maximum number of either nodes or two-dimensional elements is 3,000. the dimension for all of the arrays are given throughout the input guide. Double precision is specified for all arrays.

68. The global stiffness matrix is assembled in vector form using a modified skyline procedure. Therefore, there is no direct correlation between the size of SN and the maximum number of nodes or elements. If the required size of SN exceeds 1,600,000 during execution of SOILSTRUCT, the program will terminate execution and print the size required for the SN vector. If this occurs, two statements must be modified within



a. Coordinate system and sign convention for positive stresses for two-dimensional elements



b. Definition of interface local axes and sign convention for positive shear stress

Figure 9 Sign conventions and coordinate axes for stress and displacements (Clough 1984) the computer code; (a) the dimension of the SN vector, found in the main program, must be increased to the required value, and (b) a call statement, comparing the required size of the SN vector for the problem being analyzed to the actual dimension of SN, must be revised in subroutine DETNA.

REFERENCES

Brebbia, C. A., Telles, J. C. F., Wrobel, L. C. 1984 "Boundary Element Techniques: Theory and Applications in Engineering," Springer Verlag, Berlin, 464 p.

Clough, G. W. 1984. "User's Manual for Program SOILSTRUCT," Virginia Polytechnic Institute and State University, Blacksburg, VA.

Clough, G. W., and Duncan, J. M. 1969 (Sep). "Finite Element Analyses of Port Allen and Old River Locks," Contact Report S-69-6, U.S. Army Engineers Waterways Experiment Station, Vicksburg, MS.

Clough, G. W., and Duncan, J. M. 1969 (Sep). "Finite Element Analyses of Port Allen and Old River Locks," Report No. TE 69-3, College of Engineering, Office of Research Services, University of Califonia Berkely, CA.

Clough, G. W., and Duncan, J. M. 1969 (Sep). "Finite Element Analysis of Port Allen and Old River Locks," Contract Report 2-69-6, U.S. Army Engineers Waterways Experiment Station, Vicksburg, MS.

Doherty, W. P., Wilson, E. L., and Taylor, R. L. 1969. "Stress Analysis of Axisymmetric Solids Utilizing Higher-Order Quadrilateral FiniteElements," Report No. SESM 69-3, Structural Engineering Laboratory, University of California, Berkely, CA.

Duncan, J. M. Byrne, P., Wong, K. S., and Mabry, P. 1978 (Apr). "Strength, Stress-Strain and Bulk Modulus Parameters for Finite Element Analyses of Stresses and Movements in Soil Masses," Report No. UCB/GT/78-02, College of Engineering, Office of Research Servicesm Univeristy of California, Berkely, CA.

Duncan, J. M., and Chang, C. Y. 1970 (Sep). "Nonlinear Analysis of Stress and Strains in Soils," *Journal of the Soil Mechanics and FoundationsDivision, ASCE,* Vol. 96, No. SM5, Proc. Paper 7513, pp 1629-1653.

Ebeling, R. M. 1990 (Dec). "Review of Finite Element Procedures for Earth Retaining Structures," Miscellaneous Paper ITL-90-5, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.

Ebeling, R. M., Duncan, J.M., and Clough, G. W. 1990 (Oct). "Methods of Evaluating the Stability and Safety of Gravity Each Retaining Structures Founded on Rock - Phase 2 Study," Technical Report REMR CS-29, U.S. Waterways Experiment Station, Vicksburg, MS.

Ebeling, R. M., Clough, G. W., Duncan, J. M., and Brandon T. L. 1992 (March). "Methods of Evaluating the Stability and Safety of Gravity Earth Retaining Structures Founded on Rock," Technical Safety Report REMR CS-29, U.S. Waterways Experiment Station, Vicksburg, MS.

Ebeling, R. M., Mosher, R. M., Abraham, K. and Peters, J. F. 1993 (Jan). "Soil-Structure Interaction Study of Red River Lock and Dam No. 1 subjected to Sediment Loading," Technical Report ITL-93-, U.S. Army Engineers Waterways Experiment Station, Vicksburg, MS.

Ebeling, R. M., Duncan, J. M., and Clough, G. W. 1990 (Oct) "Methods of Evaluating the Stability and Safety of Gravity Earth Retaining Structures Founded on Rock-Phase 2 Study, " Technical Report ITL-90-7, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.

Ebeling, R. M., Peters, J., Clough, G. W. 1992 (May). "Users Guide for the Incremental Construction, Stoil-Structure Interactiuon Program SOILSTRUCT," Technical Report ITL-90-6, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.

Goodman, R. E., Taylor, R. L., and Breeke, T. L. 1968. "A Model for the Mechanics of Jointed Rock," *Journal of the Soil Mechanics and Foundations Division, ASCE*, No. SM3.

Jaky, J. 1948. "Pressure in Silos," *Proceedings, 2nd International Conference on Soil Mechanics and Foundation Engineering*, Vol. 1, pp 103-107.

Melan E. 1932, "Der Spannungszustand der durch eine Einselkraft im Innern beanspruchten Halbscheibe," Z. Angew. Math Mech, Vol 12. pp 343-346.

Peters, J. F., Holmes, T. L., and Leavell, D. A. 1986. "Finite Element Analysis of Williamson CBD Sheetpile Cell Floodwall, TASK 1,". letter report to Department of Army COE, Huntington District, Huntington, WV.

Potyondy, J. G. 1961 (Dec). "Skin Friction Between Various Soils and Construction Materials," *Geotechnique*, Vol ii, No. 4 pp 339-353.

Regalado, L. R., Duncan, J. M., and Clough, G. W. 1992 (Feb). "Finite Element Analyses of Gravity Earth Retaining Structures Founded on Soil," Technical Report to the U.S. Army Engineers Waterways Experiment Station, Virginia Polytechnic Institute and State University, Blacksburg, VA.

Telles, J. C. F. and Brebbia, C. A. 1981. "Boundary Element Solution of Half-Plane Problems," *Internation Journal of Solids and Structures*, Vol. 17, pp 1149-1158.

Zarco, M. 1993. "Solution of Soil-Structure Interaction Problems by Coupled Boundary Element-Finite Element Method,". Ph.D. Dissertation, Virginia Polytechnic Institute and State University, Blacksburg, VA.

APPENDIX A: USER'S GUIDE FOR PROGRAM SOILSTRUCT

1. IDENTIFICATION CARD FORMAT (20A4)

Column	Variable	Explanation
1-80	HED	Analysis identification

2. DATA CONTROL CARD FORMAT(14I5)

(a) MESH PARAMETERS: All nodes, two-dimensional, interface and boundary elements to be used in the analysis must initially be included in the mesh; additions or deletions are not allowed, but the material parameters can be changed to make them inactive. One-dimensional bar elements usually are added in subsequent construction steps, but may initially be included in the mesh.

Column	Variable	Explanation
1-5	NUMNP	Number of nodal points (3000 maximum).
6-10	NUMEL	Number of elements initially in the mesh, excluding bar, boundary and beam elements, but including interface elements. NUMEL, then includes NUMJT (3000 maximum). Interface elements should be numbered first.
11-15	NUMJT	Number of interface elements (200 maximum)
16-20	NUMBAR	Number of bar elements initially in the mesh (60, including elements assessed in subsequent construction steps).
21-25	NBEM	Number of boundary elements (200 maximum)

(b) ANALYSIS PARAMETERS

Column	Variable	Explanation
26-30	NC	Number of loading construction steps (60 maximum)
31-35	NMOD	Modulus specification code = 0 if modulus calculation codes input with loading information card. = 1 if modulus calculation codes input with modulus calculation card.

Column	Variable	Explanation
36-40	INIT	Initial stress input code = 0 if external input from cards or tape, included in input sequence. Use for reading initial stresses and initial displacements from a restart file. = 1 if internally generated from gravity turn-on analysis. = 2 if the initial stresses and displacements are to be set equal to zero within SOILSTRUCT
41-45	KI	Equal to zero (interface is automatically activated). KI is used when INIT is not equal to 0. If INIT = 0, then KI can be left blank.
46-50	IHORIZ	Ground surface inclination code = 0 horizontal ground surface. Vertical stresses computed from gravity turn-on. Horizontal stresses are compute assuming $K_0 = \nu/(1-\nu)$. Unless K_0 is specified. = 1 sloping ground surface. Vertical and horizontal stresses are calculated from gravity turn-on analysis linear elastic response soil - i.e., $K_0 = \nu/(1-\nu)$.

NOTE: If INIT is not equal to 0, 1 or 2, initial stresses are generated assuming a horizontal ground surface, horizontal water table, and $\sigma_x = K_0 \sigma_y$.

(c) OUTPUT PARAMETERS

Column	Variable	Explanation
51-55	ITRD	Analysis printout code - = 0 if initial stresses are to be printed. = 2 if initial stresses are <u>not</u> to be printed.
56-60	ILIST	Element and nodal point card data printout code - = 0 if not printed = 1 if printed
61-65	IPUNCH	Code for a Restart (tape 9)- = 0 Restart file not created = 1 Restart file is created

(d) BASIC PARAMETERS **FORMAT(2D10.5)**

Column	Variable	Explanation
1-10	GAMW	Unit weight of water
11-20	PATM	Atmospheric pressure

The unit weight of water and the atmospheric pressure are included as basic parameters. Either English or SI units can be used. All data must be compatible with input coordinate, pressure and material property parameters.

3. MATERIAL ALLOCATION CARD **FORMAT(F6I5)**

Column	Variable	Explanation
1-5	NUMMAT	Total number of material types, including both two-dimensional soil or construction material types and interface material types. (40 maximum)
6-10	NUMSOL	Total number of material types excluding the interface material types. Thus (NUMAT-NUMSOL) must equal the number of interface material types.
11-15	NATYP	Material type number assigned to 2D elements having the properties of air. Usually, elements that will be built are initially identified as air elements.
16-20	NATYP2	Material type number assigned to interface elements having the properties of air. Usually, elements that will be built are initially identified as air elements.
21-25	NCTYP	Structural material type, such as concrete or sheetpiling
26-30	NB1TYP	Backfill material type 1. (refer to 15b on fill or concrete placement)
31-35	NB2TYP	Backfill material type 2. (refer to 15b on fill or concrete placement)

4. ALPHA METHOD PARAMTERS FOR BASE SEPARATION AND FOR 2-D SOIL ELEMENTS

The following set of variables control the model of the loss of contact between a structure and its foundation and to reduce overshoot stresses in 2-D soil elements using the alpha method. Note that only horizontal interface elements may be used along the base of the structure and these interface elements must be assigned consecutive numbers. If this base separation routine is not required, assign a value of zero to NLOOP and proceed to group 3 input data.

(a) FIRST CARD **FORMAT(2I5,20X,4I5)**

Column	Variable	Explanation
1-5	NLOOP	Maximum number of iterations using alpha method for interface elements for each load step.
6-10	KEYFRC	Linear stress increment applied to interface elements during transfer of shear stress to adjacent elements = 0 = 1 for uniform
31-35	NOELF1	First horizontal interface element checked for tension.
36-40	NOELF2	Last horizontal interface element checked for tension.
41-45	NGROUP	Number of groups of interface elements for which resultant forces are to be computed.
46-50	NLOOP2	Maximum number of iterations using alpha method for 2-D soil elements for each loading step.

(if the alpha method for 2-D soil elements is not required, assign a value of zero to NLOOP2)

NOTE: If both NLOOP and NLOOP2 are non-zero, then the smaller of the two will determine the maximum number of iterations using the alpha method.

Input SECOND and THIRD cards for each plot group, NGROUP.

(b) SECOND CARD **FORMAT(2I5)**

Column	Variable	Explanation
1-5	NJELGRP	Number of elements in Plot group. (40 maximum)
6-10	NORIGIN	Node about which Moments will be summed.

(c) THIRD CARD **FORMAT(16I5)**

Column	Variable	Explanation
1-5	NJEL (NJELGRP)	Element Numbers in Plot Group NJELGRP. (800 maximum) Repeat until NJELGRP elements have been inputted.

4.1 CONTROL PARAMETERS USED IN CHECKING EXCESSIVE INCREMENTAL DISPLACEMENTS

The following variables control the routines that check if too-large displacements have occurred in the mesh.

(a) FIRST CARD **FORMAT(3I5,2D10.1,I5)**

Column	Variable	Explanation
1-5	ICKCOL	Code that specifies the type of incremental displacement check to be performed. = 0 if no check is to be made = 1 if a check is to be made for excessive x-displacements = 2 if a check is to be made for excessive y- displacements = 3 if a check is to be made for excessive incremental x or y-displacements
6-10	LODCHK	Number of the first load step to check for excessive incremental displacements. The same check will be performed in all subsequent load steps.
11-15	NNDCHK	Total number of nodal points to be checked for excessive incremental displacements. If ICKCOL=0, the value of NNDCHK will not be used. If ICKCOL does not equal 0 and NNDCHK=0, all nodes in the mesh will be checked for excessive incremental displacements.
16-20	XTOL	Tolerance value for nodal incremental x-displacement
21-25	YTOL	Tolerance value for nodal incremental y-displacement
26-30	ICOLST	Stop run code = 0 Program will not be stopped = 1 Program will not be stopped if incremental displacements of specified nodes exceed the values given by XTOL and YTOL. The current stresses and displacements will be output prior to the termination of the analysis. (The results will be those just before the occurrence of excessive displacements.)

(b) SECOND CARD **FORMAT(16I5)**

Required only if NNDCHK does not equal 0. The nodal point numbers of nodes (a total of NNDCHK) to be checked for excessive displacements are specified in the following cards. A total of 16 nodal points numbers are supplied on one card. The number can be input in any order.

Column	Variable	Explanation
1-5	NDCHK(I)	Code that specifies the type of incremental displacement check to be performed. = 0 if no check is to be made = 1 if a check is to be made for excessive x-displacements

5. LOADING INFORMATION CARD FORMAT(6I3,2X,15A4)

One card is supplied for each loading step. One to three loading/construction modes can be included in each loading step. The loading or construction mode codes include:

KCS(NC,I)	DESCRIPTION
1	Excavation (equivalent nodal loads can be applied to equal increments)
2	Fill placement (subroutine SUBSTP cannot be used in conjunction with the fill placement procedures of subroutine BUILD)
3	Seepage loading (equivalent nodal loads can be applied in equal increments)
4	Deletion of bar element (force in the element can be applied in equal increments)
5	Installation of bar element (prestress force can be applied in equal increments)
6	Boundary pressure loading (equivalent nodal loads can be applied in equal increments)
7	Temperature loading (the total temperature change can be applied in equal increments)
8	Support displacement (the total displacement can be applied in equal increments)
9	Concentrated nodal loads (can be applied in equal increments)
10	Element material type change

As indicated in the listing, input loads, displacements or temperature changes can be analyzed in equal increments, or substeps. Subroutine SUBSTP generates the increments then main analyzes all increments prior to analyzing the next load step. With one exception, all loading/construction modes that can be applied in increments or substeps, can also be applied in any combination in any load step. The number of substeps, however, will be the same for all loading or construction modes included in the load step. The exception is temperature loading; if a temperature change is specified, and a given number of substeps is specified, then only temperature loading can be specified in the loading step - i.e., KCS(N,2) and KCS(N,3) must be set equal to zero, or left blank. If the number of substeps, NSBSP, equal to zero, then temperature loading can be included with other loading/construction modes in a load step.

Since the same input format is used and similar operation are performed for loading or construction modes 8 and 9, the following rules in the usage of these two codes apply. If only concentrated nodal loads are specified, use mode 9. If only support displacements are specified use mode 8. If both loads and displacements are specified, use mode 8.

Column	Variable	Explanation
1	NALFPT(NC)	Printout code for alpha method. = 1 A message that gives details on the alpha method analyses will be output for this load step. Details include which element(s) failed, the highest stressed element, and the value of alpha for this element.
2-3	KCS(NC,1)	First loading/construction mode code
4-6	KCS(NC,2)	Second loading/construction mode code
7-9	KCS(NC,3)	Third loading/construction mode code
10-12	NUMIT(NC)	Number of iteration for the loading step. NUMIT(NC) applies to <u>each</u> substep if substeps are specified. A maximum of 10 iterations can be specified. NUMIT(NC) = 0 is the same as NUMIT(NC) = 1
13-15	NUMSS(NC)	Number of substeps. A maximum of 10 substeps can be specified. NUMSS(NC) can not be zero.
16-18	MOD(1,NC)	Modulus calculation code- = 1 if a loading modulus is to be calculated = 2 if an unload reload modulus is to be calculated. = 0 if the computer code is to decide the type of modulus to be calculated. In this case, if the most recently calculated maximum shear stress for an element is less than all previous values of maximum shear stress, an unload-reload modulus is calculated. Otherwise a primary loading modulus is calculated

Input is required here only if NMOD = 0. All material types other than interface or bar elements, are given one of the above codes. If NMOD = 0 and NC = 0, as might be the case for an analysis of initial stresses, MOD(1,1) is set equal to zero, or the SOILSTRUCT code decides.

Column	Variable	Explanation
19	IPRT(1,NC)	Print code- = 0 print displacements and stresses for final iteration only = 1 print interface stresses for each iteration, and stresses for all elements and displacements for final iteration only = 2 print stresses for all elements for each iteration, and displacements for final iteration only = 3 print stresses and displacements for each iteration
20	IPLT(1,NC)	Code for creating a NISA plot files (tapes 12 and 22) = 0 NISA plot file not created = 1 NISA plot file created NISA geometry on tape 12 NISA geometry and output on tape 22
21-80	HEDCS(NC)	Description of loading step

KCS(NC,1), KCS(NC,2) and KCS(NC,3) can be input in any numerical order, but the modes are processed in ascending numerical order: If second and/or third loading construction nodes are not required then KCS(NC,2), KCS(NC,3) should be set equal to zero or left blank.

6. MODULUS SPECIFICATION CARD FORMAT(40I2)

This card is required only if NMOD = 1 and NC not equal to 0. A card is required for each loading step, 1 to NC. In this option, values of the modulus specification code are specified for each material type (and thus each element, excluding bar and interface elements), regardless of the change in maximum shear strain that may have occurred.

Column	Variable	Explanation
2,4,6,..	MOD(I,NC)	Modulus calculation code for each material type (1 to NUMSOL) for the first load step. Separate cards are required for each load step. Columns not used can be left blank.

7. MATERIAL PROPERTY CARDS

These cards are used only for two-dimensional elements; the first and second cards must be supplied for each material type, excluding bar and interface material types. The first and second cards, as a pair, are supplied in order of material type number N = 1 to NUMSOL. Information or properties not required for a material type can be left blank.

(a) FIRST CARD **FORMAT(7D10.5,I10)**

Column	Variable	Explanation
1-10	GUI(N)	Poison's ratio before failure
11-20	GUF(N)	Poison's ratio at failure (no greater than 0.49)
21-30	GAM(N)	Total or bouyant unit weight (always specified regardless of drained or undrained material behavior)
31-40	FR(N)	The failure ratio R_f.
41-50	AO(N)	Coefficient of lateral earth pressure at rest, K_o, as pertaining to effective stresses.
51-60	PHI(N)	Friction angle in degrees
61-70	XXP(N)	Exponent n is the initial tangent and unload-reload modulus expressions. Its value is assumed to be independent of loading mode. For a linear elastic material n must be 0. For saturated soils when PHI(N) = 0., N should normally be 0.0001.
71-80	IDRAIN(N)	Material behavior code - = 0 if undrained = 1 if drained

(b) SECOND CARD **FORMAT(3D10.3,D20.5,3D10.0)**

Column	Variable	Explanation
1-10	HCOEF(N)	Coefficient K_m in the initial tangent modulus expression $E_i = P_a K_m (\sigma_3/P_a)^n$
11-20	ULCOEFF(N)	Coefficient K_{ur} in the unload-reload modulus expression $E_{ur}\ P_a K_{ur} (\sigma_3/P_a)^n$
21-30	COHE(N)	Undrained strength or cohesion
31-50	E(N)	Tangent modulus at failure for isotropic nonlinear (non-elastic) materials. Young's modulus for elastic materials
51-60	ALPHA(N)	Coefficient of linear thermal expansion for structural element; zero or blank otherwise

Column	Variable	Explanation
61-70	EIMIN(N)	Minimum initial tangent modulus for isotropic nonlinear (non-elastic) materials. Zero or left blank for elastic materials
71-80	TENS(N)	Minimum allowable value of the minor principle stress for isotropic nonlinear (non-elastic) materials. If tensile, input as a negative value. Zero or blank for elastic materials.

8. INTERFACE PROPERTY CARDS

One pair of cards are supplied for each interface material type, N = 1 to NUMJT. If no interface materials, no cards are required.

(a) FIRST CARD FORMAT(6D10.5,I5)

Column	Variable	Explanation
1-10	PHJ(N)	Interface friction angle in degrees.
11-20	RKS(N)	Shear stiffness k_s before failure
21-30	RKN(N)	Normal stiffness k_n before failure
31-40	COJ(N)	Interface cohesion
41-50	FRJ(N)	Failure ratio R_{fi}, FRJ = 0.0 for bilinear
51-60	TENSJ(N)	Tensile strength of interface material σ_t.
61-65	IADJMT(N)	Material number of 2-D soil element next to the interface

(b) SECOND CARD FORMAT(4D10.5)

Column	Variable	Explanation
1-10	STFSMN(N)	Shear stiffness after failure.
11-20	STFMN(N)	Normal stiffness after failure.
21-30	RKJ(N)	Modulus Number for interface element K_j.
31-40	XXPJ(N)	Modulus Exponent for interface element n_i.

9. NODAL POINT CARDS **FORMAT(I10,4D10.2)**

One card is supplied for each node. The numbering of nodal points must be sequential and some of the nodes can be omitted. Those nodes omitted are automatically generated by the program at equal spacings between those specified. The first and last nodes must always be specified. Note that DP(N) and PP(N) are automatically generated in equal increments for those nodes omitted (see Figure A1)

Column	Variable	Explanation
1-10	N	Nodal point number.
11-20	X(N)	X coordinate, positive to right
21-30	Y(N)	Y coordinate, positive upward
31-40	PP(N)	Pore pressure in head of water; zero or blank of not specified. Pore pressure must not be specified for undrained materials, but must be specified for drained materials.
41-50	DP(N)	Change in pore pressure in head of water for soil elements; change in temperature for linear elastic structural material; zero or blank otherwise.

10. BOUNDARY CONDITION CARDS

Cards 1 through 8 are supplied as required to specify restraints of boundary nodes. If there are no nodes restrained in the mode specified for cards 2, 3, or 4, then the card is not required. For a given card, specified nodes N=1, to NOY, NOX, or NOXY must be in sequential order.

(a) FIRST CARD **FORMAT(3I5)**

Column	Variable	Explanation
1-5	NOY	Number of nodal points fixed against Y-movement only.
6-10	NOX	Number of nodal points fixed against X-movement only.
11-15	NOXY	Number of nodal points fixed against both X- and Y-movement.

(b) SECOND CARD **FORMAT(16I5)**

Column	Variable	Explanation
1-5	IC(N)	Nodal point number of the first nodal point fixed against Y-movement

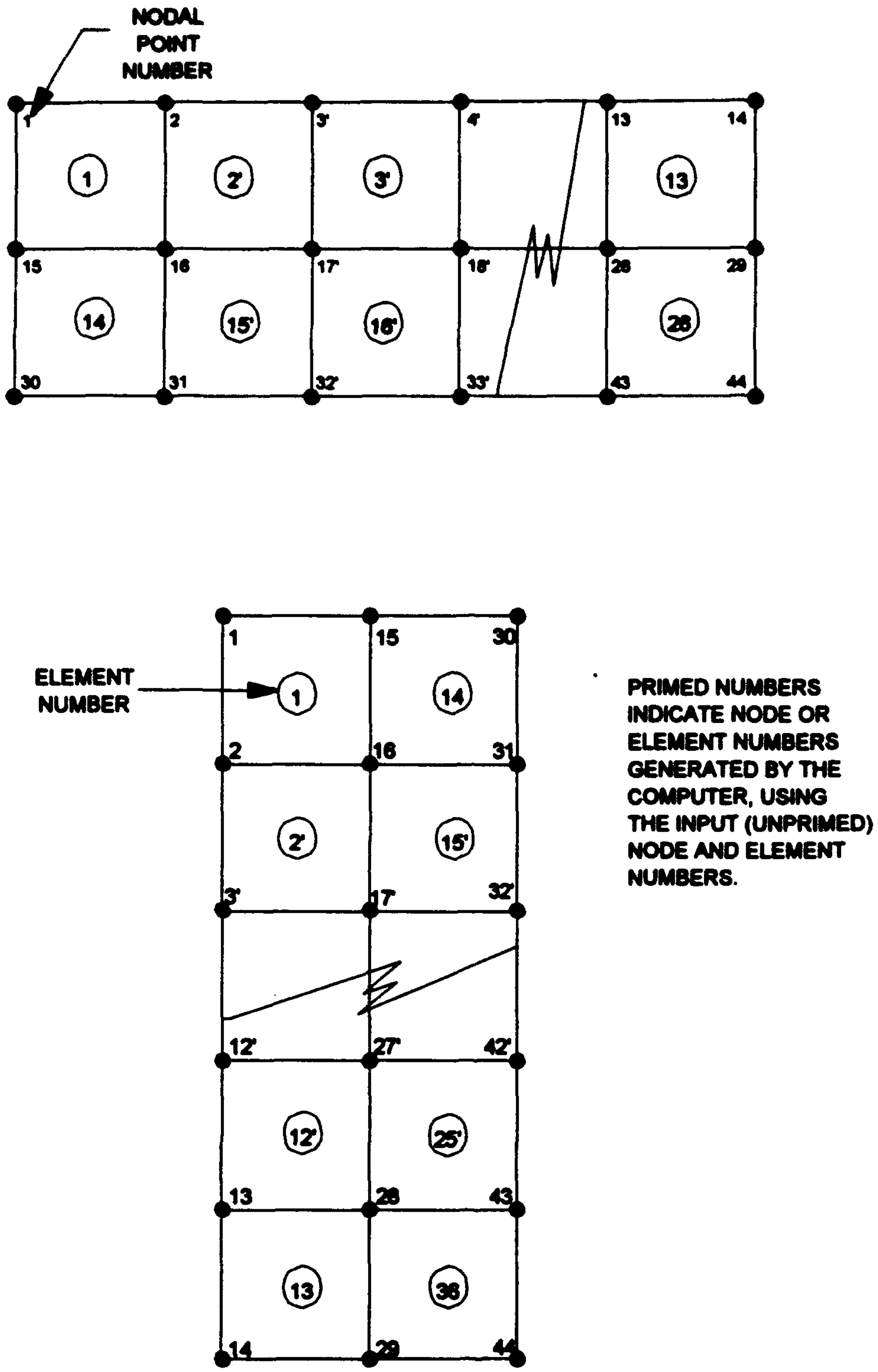


Figure A1. Examples of number nodes and elements if generated by the program (Clough 1984)

Additional nodal points fixed against Y-movement, N = 2 to NOY, are specified in the next 15 five-column fields and on additional cards as required.

(c) THIRD CARD **FORMAT(16I5)**

Column	Variable	Explanation
1-5	IC(N)	Nodal point number of the first nodal point fixed against X-movement

Additional nodal points fixed against X-movement, N = 2 to NOX, are specified in the next 15 five-column fields and on additional cards as required.

(d) FOURTH CARD **FORMAT(16I5)**

Column	Variable	Explanation
1-5	IC(N)	Nodal point number of the first nodal point fixed against both X- and Y-movement

Additional nodal points fixed against both X- and Y-movement, N = 2 to NOXY, are specified in the next 15 five-column fields and on additional cards as required.

11. ELEMENT CARD **FORMAT(6I5)**

Once card is supplied for each interface or two-dimensional element; bar elements are not included in this series of cards. All interface elements are supplied in sequential order first, followed by two dimensional elements, also in sequential order. (If 'Build' is used; place interface elements which will be built up at the end of joint list. i.e. start numbering these elements with N = NUMJT - NJTFIL). Thus, interface elements must be numbered from N = 1 to NUMJT, and two-dimensional elements from N = (NUMJT+1) to NUMEL.

Nodal point numbers must be specified consecutively, processing counter-clockwise around the element as shown in Figure A2 The first and last nodal point numbers specified for interface elements must have the same coordinates. Triangular two-dimensional elements having four different nodal point numbers may not be used; the first and last point numbers of a triangular element must be identical.

Element numbers in a row may be omitted, in which case the omitted elements will be generated by incrementing the element number N and the nodal point numbers I, J, K, and L by one, and by assigning the same material type number as specified for the last element. The first and last elements in the row must be specified. If no elements are omitted, the element numbering may be done in any order, provided all interface elements are numbered first.

Column	Variable	Explanation
1-5	N	Element Number
6-10	IL(N,1)	Number of Nodal Point I.
11-15	IL(N,2)	Number of Nodal Point J.
16-20	IL(N,3)	Number of Nodal Point K.
21-25	IL(N,4)	Number of Nodal Point L.
26-30	IL(N,5)	Material Type Number

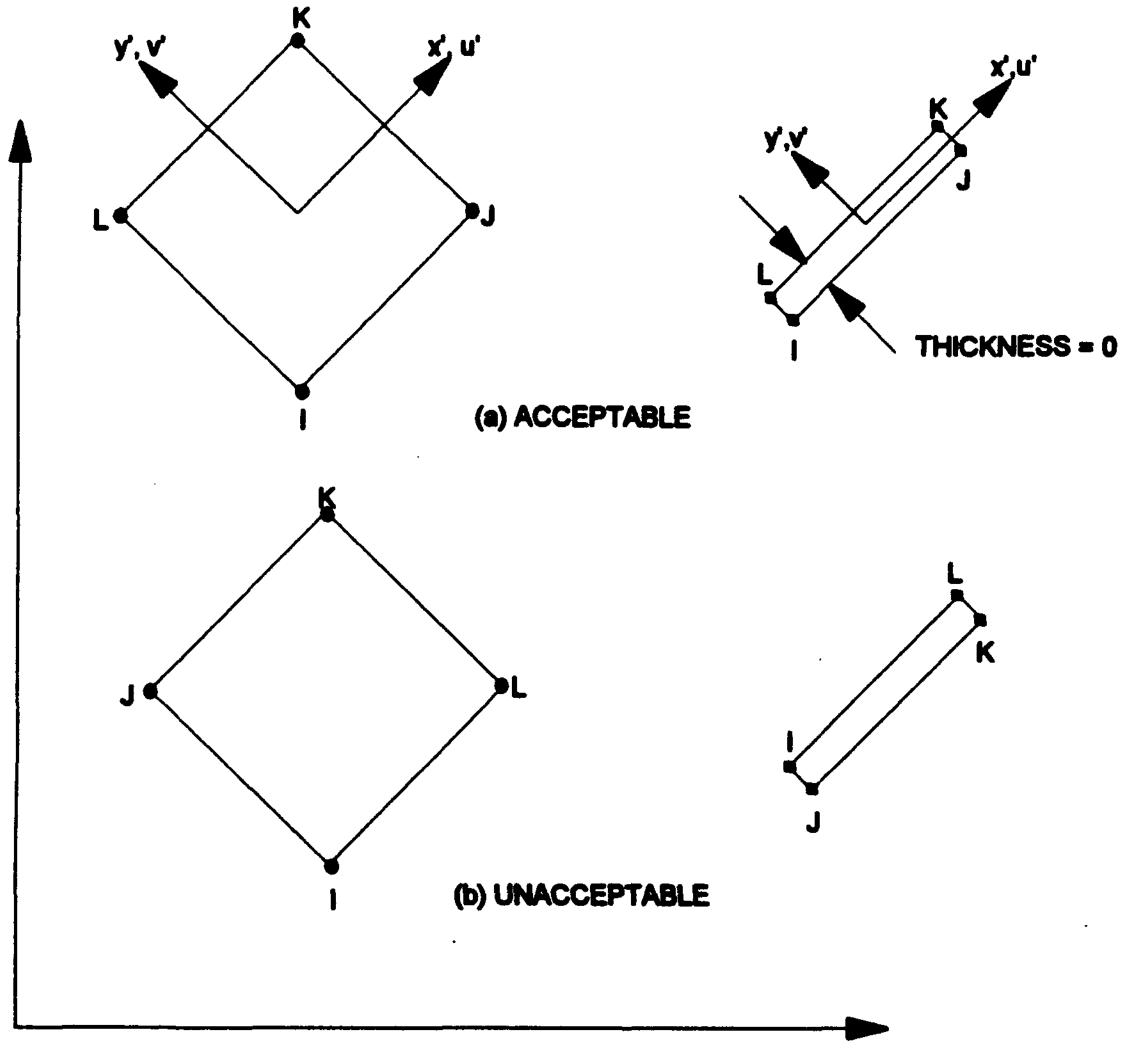


Figure A2 Nodal point numbering of isoparametric QM5 elements and interface elements (Clough 1984)

12. BAR ELEMENT CARD FORMAT(4I5,2D10.7,2D10.1,D10.5)

One card is supplied for each bar element initially in the mesh or, if a continuation analysis, added in an incremental loading step of the previous analysis. Note that for a continuation analysis this card is not automatically generated. Elements are numbered sequentially from N = 1 to NMBAR

Column	Variable	Explanation
1-5	N	Element Number
6-10	IB(N,1)	Number of nodal point I
11-15	IB(N,2)	Number of nodal point J
16-20	IB(N,3)	Spring response type code- = 1 if both compression and tension of bar allowed = 2 if only compression allowed. = 3 if only tension allowed.
21-30	BAR(N,1)	cos α
31-40	BAR(N,2)	sin α
41-50	BAR(N,3)	Prestress force in the bar element. The force must be applied as loads at nodes I and J using loading/construction mode 9 in a loading step.
51-60	BAR(N,4)	Stiffness of bar element. This usually computed as AE/L, but the mesh length (distance from node I to node J) need not, and usually does not, correspond to the actual length.
61-70	BAR(N,5)	Displacement of bar element at activation. This allows for a specified degree of slack at the strut connection; the bar will deform BAR(N,5) prior to its stiffness being activated.

Bar elements can function as either anchors or strut (spring) supports. The required parameters are dependent on the type of bar element specified.

If a strut support is specified, nodal point J is a node fixed against x- or y- movement, depending on the orientation of the strut being modeled. For program storage efficiency the number of node J should be as close as possible to the number of node I. Nodal point I represents the point of connection between the wall and the actual strut. Nodal points I and J, then, are not necessarily physically connected, since the element stiffness is input independently. Nodal point J allows the force at the J node to be carried into the system as a reaction at a fixed node. This is consistent with the typical mesh representation of one half of a symmetric excavation. The value of sin α and cos α are specified according to the sign convention shown in Figure A3. The values are input to represent the line of action of the strut support, and do not need to correspond to the relative positions of the I and J nodes.

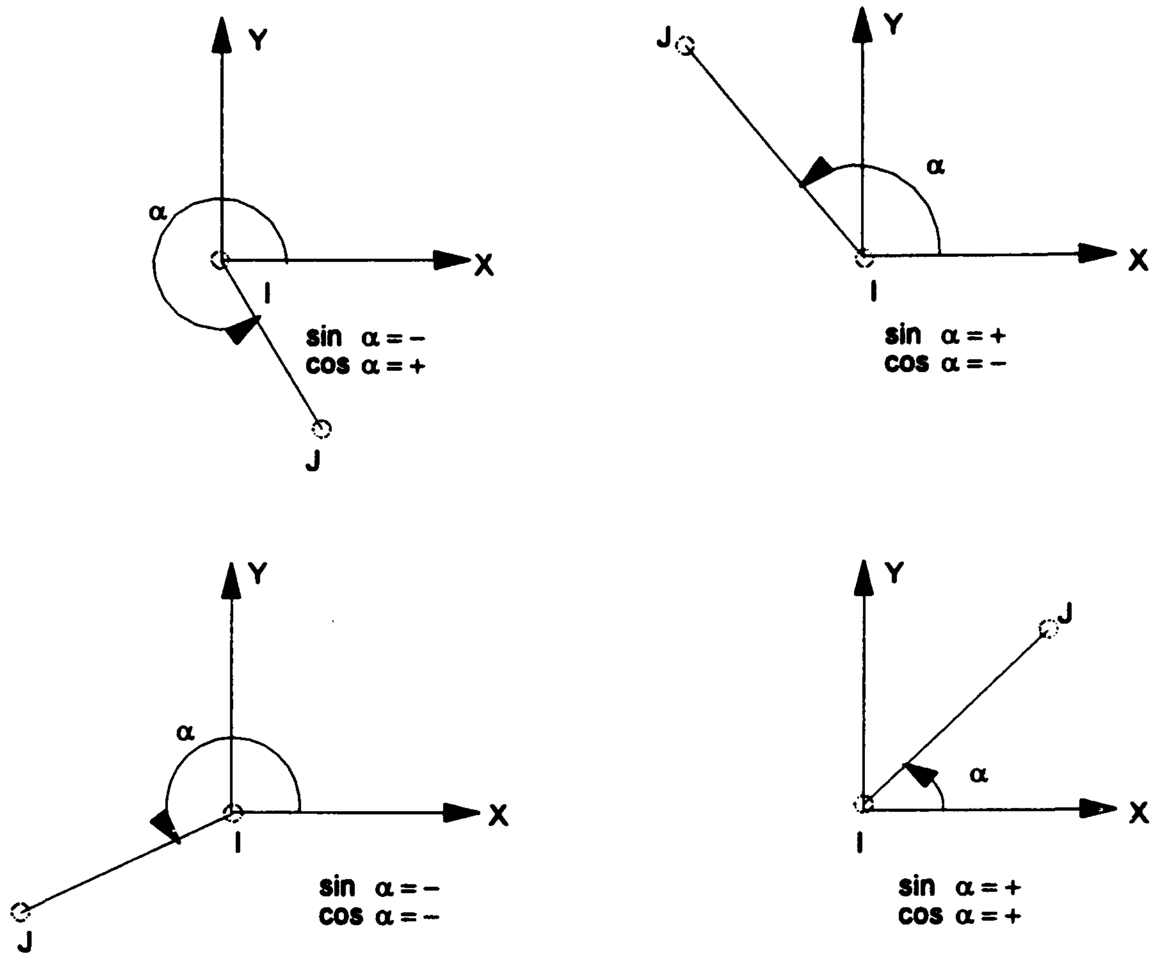


Figure A3. Sign convention and definition of local axes for bar element (Clough 1984)

If an anchor is specified, nodal points I and J physically represent two ends of the anchor, and must be restrained appropriately. The values of sin α and cos α must correspond to the relative position of the I and J nodes representing the ends of the anchor. Stiffness is computed as AE/L, with L being the distance between nodes I and J and either A or E altered to give the correct stiffness. Stiffness for an anchor or a strut support is input as force per length of wall.

For either element type, specifying the prestress force does not apply the force. The concentrated force loading/construction mode must be used for this purpose. Thus, bar elements initially in the mesh cannot carry prestress force, since it is not applied by a gravity turn on analysis.

13. BOUNDARY ELEMENT CARD

The maximum number of boundary elements that can be specified is 200. Figure A4 shows that the nodes within each boundary element are numbered so that normal to each towards the element domain. The local numbering of the nodes in boundary element system must conform to this confluency convention illustrated in Figure A5.

(a) FIRST CARD (NODAL COORDINATE DATA) FORMAT(16I5)

One card is supplied to specify the coordinates of the global nodal point numbers in the boundary element system. The specified nodes N=1,[NBEM+1] must be in sequential order.

Column	Variable	Explanation
1-5	IX(N)	Global nodal point number of first node in boundary element system.

Additional global nodal point numbers, N=2 to NBEM, are specified in the next 15 five column fields and on additional cards as required.

(b) SECOND CARD (SYMMETRY CONDITIONS) FORMAT(2I5)

One card is supplied to specify the symmetry conditions of the boundary element system.Because of the formulation which assumes a half-plane with the surface parallel to the x-axis as the domain for the boundary element system, only symmetry about the y-axis is considered.

Column	Variable	Explanation
1-5	NSYM	Number of nodes in boundary element system lying on axis of symmetry (Y-axis). If NSYM = 0, no symmetry in problem.

(c) THIRD CARD (SYMMETRY COORDINATE DATA) FORMAT(16I5)

One card is supplied to specify the local nodal point numbers in the boundary element system which lie on the axis of symmetry. If NSYM = 0, skip this card. The specified nodes N=1,NSYM can be input in any order.

Column	Variable	Explanation
1-5	ISYM(N)	Local nodal point number of node in boundary element system lying on axis of symmetry (Y-axis), $1 \leq$ ISYM(N) $\leq$ [NBEM+1]

Additional local nodal point numbers in boundary element system lying on the axis of symmetry are specified in the next 15 five-column fields and on additional cards as required.

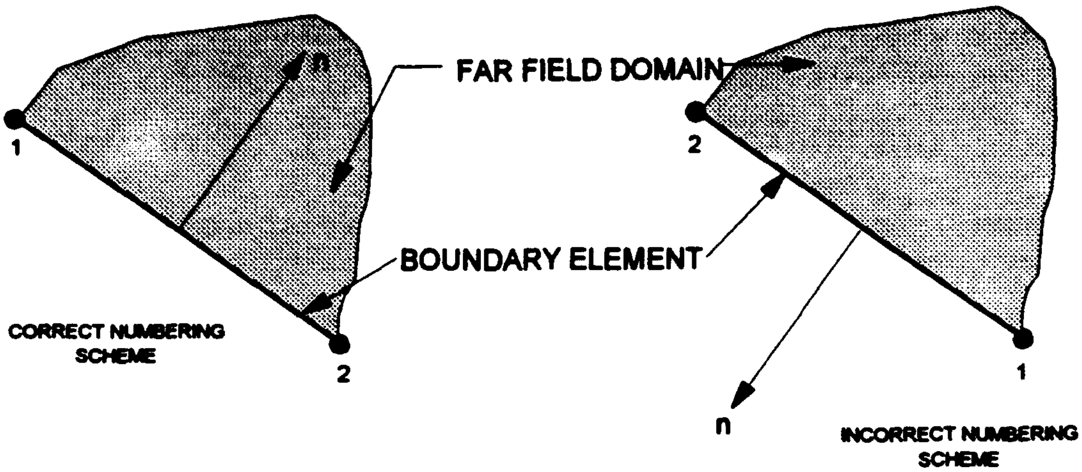


Figure A4. Nodal point numbering of linear boundary element.

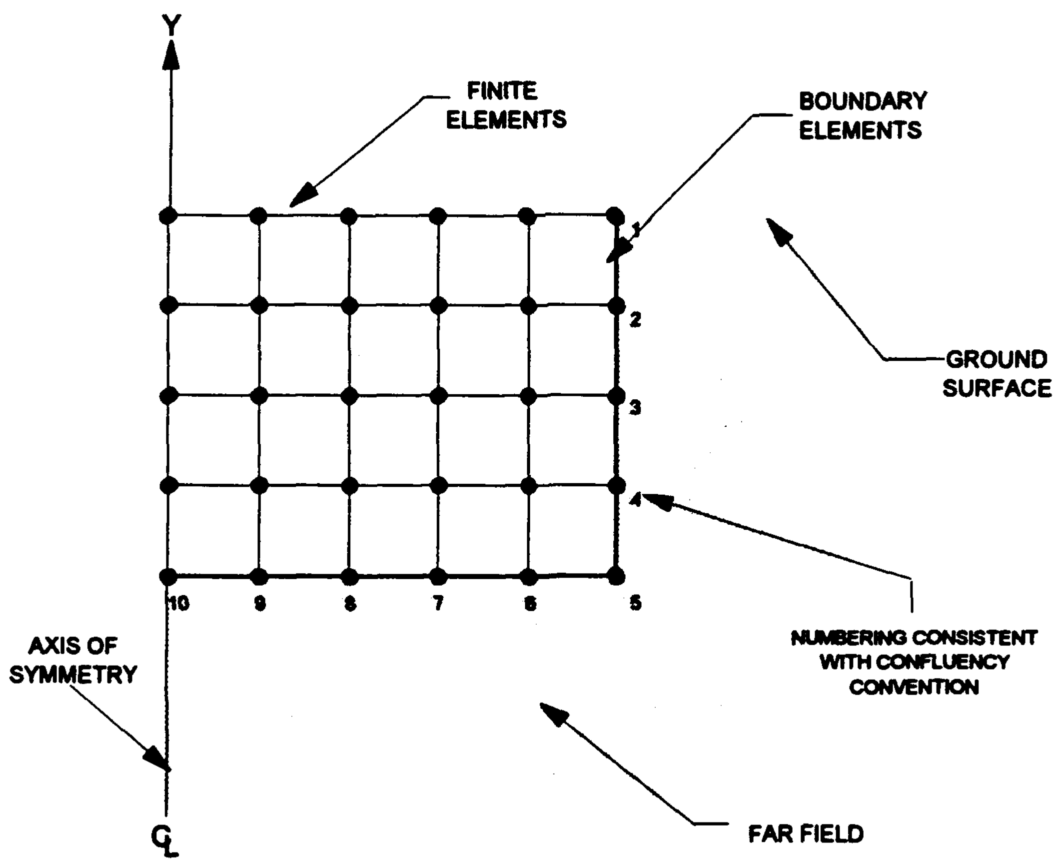


Figure A5. Local numbering of boundary element nodal points in a coupled boundary-finite element system.

(d)FOURTH CARD (MATERIAL PROPERTIES) FORMAT(2D10.3)

One card is supplied to specify the material properties of the entire boundary element system.

Column	Variable	Explanation
1-10	E	Young's Modulus for far field.
11-20	PR	Poison's ratio for far field.

Since only one set of E and ν must be specified for the entire far field, it is suggested that the average value of E and ν at the bottom of the mesh be used.

14. CONTINUATION OR INITIALIZATION CARDS

This input is applied only if INIT = 0; it can be supplied from external disk storage or punched cards input from a preceding analysis. Input format is the same for both input modes. This option is provided so that a required sequence of loading steps can be stopped at an intermediate step, then restarted from that step without repeating the complete analysis. These cards may also be used to specify values of particular variables in an initial analysis without using the gravity turn-on procedure followed with INIT = 1 or the special procedure. Similarly, values assigned to specific variables can be changed if the sequence of loading is stopped, prior to restart analysis.

(a) FIRST CARD (ELEMENT INFORMATION) FORMAT(5I5)

Column	Variable	Explanation
1-5	NUMEL	Number of elements in the mesh, excluding bar elements, but including interface elements.
6-10	NUMJT	Number of interface elements.
11-15	NUMBAR	Number of bar elements, including those initially in the mesh and those added in previous loading steps (if a restart analysis).
16-20	NUMNP	Number of nodal points.
21-25	NSTART	Interface element code- = 0 if all interface elements are active (i.e. no shear or tension failures). IFLOLD and IFAIL keys are set equal to zero for all interface elements within the program. See card groups g and h. = 1 if IFLOLD and IFAIL keys are to be read for all interface elements was generated.

When NSTART = 1, the keys IFLOLD and IFAIL have in a previous analysis and are included as standard output to tape 9 when the disc storage option (see LOADING INFORMATION CARD) was activated.

(b) SECOND CARD (STRESS INFORMATION)

INTERFACE ELEMENTS **FORMAT(2(1P,1D13.6,2X))**

2-DIM ELEMENTS **FORMAT(4(1P,1D13.6,2X))**

Three cards are supplied for each interface element, in numerical sequence N = 1 to NUMJT. For each interface element, the pair of normal and shear stresses at node I (SIGI), the center (SIG), and node J (SIGJ) are supplied.

Column	Variable	Explanation
1-13	SIG(N,1)	Stress in the x-direction for a two-dimensional element, σ_x, normal stress for an interface element
16-28	SIG(N,2)	Stress in the y-direction for a two-dimensional element, σ_y, shear stress for an interface element
31-43	SIG(N,3)	x-y shear stress for a two-dimensional element, τ_{xy}, zero or blank for interface element.
46-58	SIG(N,4)	Maximum previous value of x-y shear stress for a two-dimensional element, zero or blank for interface element

One card is supplied for each two-dimensional element, in numerical sequence N = (NUMJT + 1) to NUMEL.

(c) THIRD CARD (NODAL POINT INFORMATION)

FORMAT(3(1P1D14.7,1X)

Information for three nodal points is supplied on each card. Nodal points are specified in numerical order, N = 1 to NUMNP

Column	Variable	Explanation
1-14	DISPX(N)	Displacement in x-direction
16-29	DISPY(N)	Displacement in y-direction
31-44	PP(N)	Pore pressure in head of water

(d) FOURTH CARD (MATERIAL TYPE DESIGNATION) **FORMAT(15I5)**

Material type number for 15 interface or two-dimensional elements are specified on each card. Material type numbers for elements in numerical sequence, N = 1 to NUMEL, are specified.

Column	Variable	Explanation
1-5	IL(N,5)	Material type number

Material type number for the next 14 elements are supplied in the next 14 five-column fields.

Note that material type numbers supplied on these cards supersedes the material type numbers specified on the element card (section 11). Thus material type changes can be made as part of a restart analysis rather than including such changes in a loading step of an analysis.

(e) FIFTH CARD (BAR ELEMENT INFORMATION)
FORMAT(3I5,2D10.7,2D10.1,D10.5)

This card is supplied only if bar elements are included (NUMBAR > 0). Information for bar elements is specified on each card. Information is supplied for bar elements in numerical sequence, N = 1 to NUMBAR.

Column	Variable	Explanation
1-5	IB(N,1)	Number of nodal point I.
6-10	IB(N,2)	Number of nodal point J.
11-15	IB(N,3)	Spring response type code - = 1 if both compression and tension of bar allowed = 2 if only compression allowed = 3 if only tension allowed
16-25	BAR(N,1)	cos α
26-35	BAR(N,2)	sin α
36-45	BAR(N,3)	Prestress force in bar element.
46-55	BAR(N,4)	Stiffness of bar element, (AE/L).
56-65	BAR(N,5)	Displacement of bar element at activation.

Note that these parameters, if changed for a restart analysis supersede those specified on the bar element card (section 12). Also, if bar elements are initially included in the mesh, and an initialization procedure is used, then this card must be included, duplicating the information specified in the Bar Element Card.

(f) SIXTH CARD (INTERFACE INFORMATION) FORMAT(8D10.4)

This card is supplied only if interface elements are included (NUMJT > 0). Information for four interface elements is specified on each card. Information is supplied for interface elements in numerical sequence, N = 1 to NUMJT. If no interface elements are present, proceed to group 12 input data.

Column	Variable	Explanation
1-10	STFS(N)	Shear Stiffness of first interface element
11-20	STFN(N)	Normal stiffness of first interface element

Note that the value of the shear stiffness, if changed for a restart analysis, supersedes the value specified on the Interface Property Card (section 8). Thus the interface stiffness can be changed as part of a restart analysis.

(g) SEVENTH CARD (INTERFACE KEY) FORMAT(15I5)

This card is supplied if interface elements are included (NUMJT > 0) and NSTART = 1. Information for fifteen interface elements is specified on each card. Information is supplied for interface elements in numerical sequence, N = 1 to NUMJT. If no interface elements are present or NSTART = 0, proceed to group 12 input data

Column	Variable	Explanation
1-5	IFLOLD(N)	Modulus assignment key - = 0 if interface element N is active = 1 if tension failure occurred = 2 if shear failure occurred

(h) EIGHTH CARD (INTERFACE KEY) FORMAT(15I5)

This card is supplied only if NSTART = 1 and for only those interface elements along the horizontal base of the structure used to model the base separation, with element numbers from NOELF1 to NOELF2. Information for fifteen interface elements is specified on each card.

Column	Variable	Explanation
1-5	IFAIL(N)	Stress transfer key- = 0 if interface element number N is active = 1 if transfer of normal and shear stress in failed interface element number N has been completed.

15. LOADING STEP CARDS

These cards are required only if NC is not equal to 0. Cards are assembled in the order specified on the Loading Information Card (section 5): cards for the first loading/construction mode specified for the first loading step, and so on, to the cards for the last loading/construction mode specified for the last construction step.

For a given loading step, the lowest numbered loading/construction mode is processed first, but the analysis of the loading step is made for the combined effect of all loading/construction modes included in the loading step. Care must therefore be exercised in specifying some loading/construction modes, such as material type changes or concentrated forces representing prestress forces, in the same loading step with other loading/construction modes.

(a) EXCAVATION

These cards are supplied only if KCS(N,1), KCS(N,2), or KCS(N,3) = 1. Input is handled by subroutine EXCAV. Free excavated elements and common excavated elements are input separately. A free excavated element is an element specified to be excavated that has no boundary in common with element specified to be excavated in the loading step. A common excavated element, therefore, has at least one boundary in common with an unexcavated element (Figure A6)

Interface elements can only be included as free excavated elements, even if they have a boundary in common with an unexcavated element. Interface element cannot be used as interpolation elements. Free excavated elements (other than interpolation elements) can be used as interpolation elements, though common excavated elements are more commonly used.

If possible, adjacent common excavated elements should be input sequentially as this procedure avoids repetitive computation; nodal loads for a nodal point common to the two sequential elements will only be calculated once.

(1) FIRST CARD (CONTROL DATA) **FORMAT(2I5)**

Column	Variable	Explanation
1-5	NFXEL	Number of free excavated elements.
6-10	NXELCB	Number of common excavated elements.

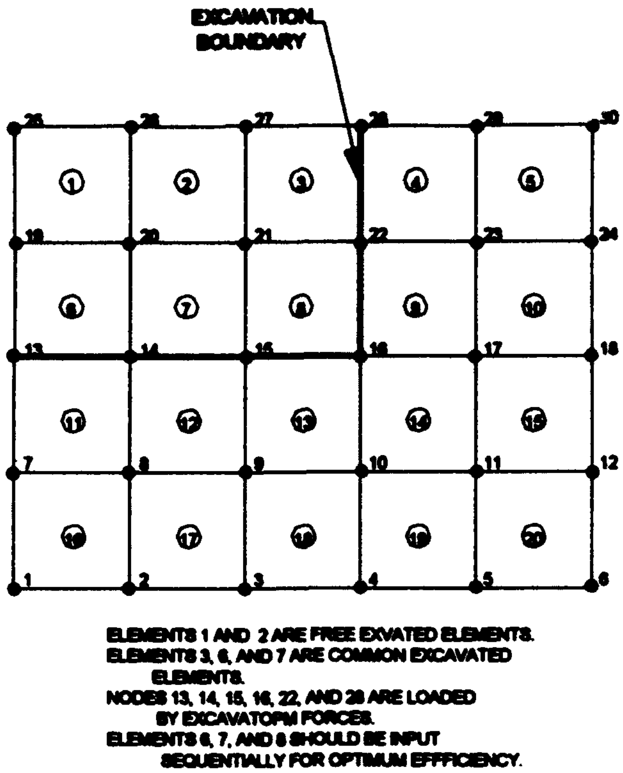


Figure A6 Example excavation load step defining free- and common-excavated elements in relation to the excavation boundary (Clough 1984)

(2) SECOND CARD (FREE EXCAVATED ELEMENT DATA)

FORMAT(16I5)

Element numbers of 16 free excavated elements can be supplied on one card. A maximum of 50 can be specified in on loading step. Element numbers of all free excavated elements, N = 1 to NFXEL are to be specified.

Column	Variable	Explanation
1-5	LNXEL(N)	Element number of first free excavated element.

Information for the next 15 free excavated elements is supplied in the next 15 five-column fields.

(3) THIRD CARD (COMMON EXCAVATED ELEMENT DATA)

FORMAT(8I5)

One card is supplied for each common excavated element, N = 1 TO NXELCB. A maximum of 50 common excavated elements can be specified in on loading step. Loading codes include:

0 - The node is not loaded by excavation forces and is not common to both an excavated and an unexcavated element.

1 - The node is loaded by excavation forces and is common to both an excavated and an unexcavated element.

Note that I, J, K, and L refer to the same nodes I, J, K, and L specified on the element card (section 11)

Interpolation element should be specified in a criss-cross fashion as shown in Figure A7. Further, x- and y-coordinates of diagonal elements must not be the same. If these rules are not adhered to, the interpolation routine will detect singularity and processing will stop.

Column	Variable	Explanation
1-5	LUL(N,1)	Element number of the first common excavated element. This element is also the first interpolation element.
6-10	LUL(N,2)	Element number of second interpolation element.
11-15	LUL(N,3)	Element number of third interpolation element.
16-20	LUL(N,4)	Element number of fourth interpolation element
21-25	LUL(N,5)	Loading code for node I.
26-30	LUL(N,6)	Loading code for node J.
31-35	LUL(N,7)	Loading code for node K.
36-40	LUL(N,8)	Loading code for node L.

Loading codes are specified for the nodes of the element to be excavated specified in columns 1-5.

(b) FILL OR CONCRETE PLACEMENT

These cards are supplied only if KCS(N,1), KCS(N,2), OR KCS(N,3) = 2. Input is handles by subroutine BUILD. The types of elements that can be placed include structural, soil and interface elements. Subroutine SUBSTP cannot be used in conjunction with subroutine BUILD.

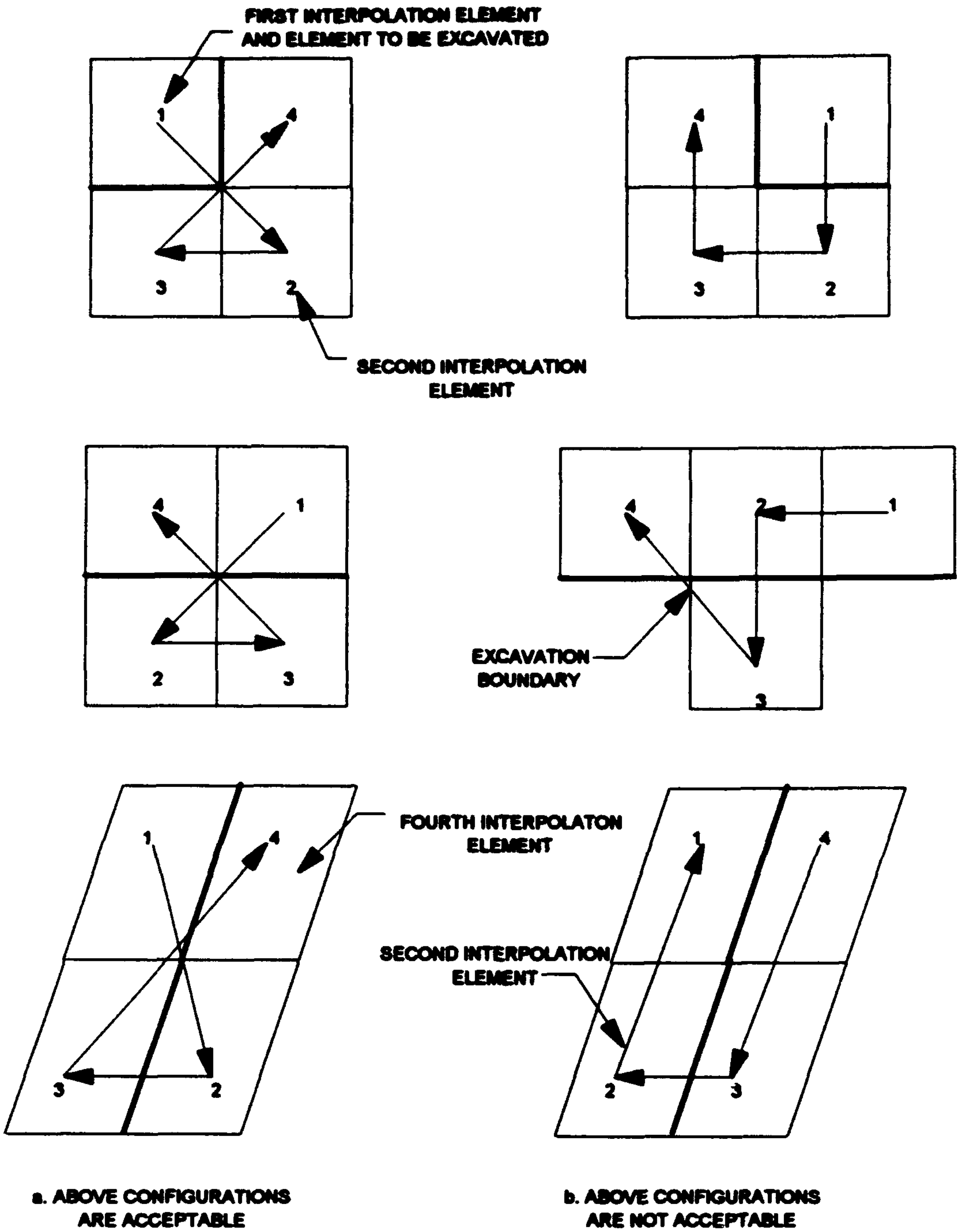


Figure A7. Illustration of numbering of interpolation elements for calculation of stresses at excavation boundary (Clough 1984)

At placement, the fill element(s) is assigned a low modulus and the surface displacements are set equal to zero. Stresses assigned to the newly place fill are based on $\sigma_x = K_0\sigma_y$, where σ_y is equal to the product of the unit weight of the fill element times the depth below the surface to the center of the element.

(1) FIRST CARD (CONTROL DATA) **FORMAT(5I5,D10.2)**

Column	Variable	Explanation
1-5	NLEL	Total number of elements to be placed including interface and structural elements.
6-10	NJ	Total number of elements to be placed less the number of inactive interface elements to be placed.
11-15	NONP	Number of nodal points within the placed layer(s) to be assigned zero displacements. This include all nodal points of the elements to be placed except those nodal points in common with an already existing element.
16-20	NCE	Number of structural elements to be placed.
21-25	NB1E	Number of backfill type 1 elements to be placed. (The number of NB2TYP elements is determined internally.)
26-35	HTB	New y-coordinate of the top of the place backfill.

(2) SECOND CARD (ELEMENT NUMBERS) **FORMAT (16I5)**

Element number of 16 placed elements can be supplied on one card. A maximum of 100 can be specified in one loading step. Element number of all placed elements, N =1 to NLEL, are to be supplied. The elements must be read in by material type according to the following sequence:

1. structural elements
2. backfill type 1 elements
3. backfill type 2 elements
4. interface elements to be activated
5. interface elements to be left inactive but to be placed between elements of like materials.

Column	Variable	Explanation
1-5	LEL(N,1)	Element number of first element to be placed
6-10	LEL(N,2)	Material number of the element

Information for the next 7 placed elements is supplied in the next 14 five-column fields.

(3) THIRD CARD (NODE NUMBERS) **FORMAT(16I5)**

Nodal point numbers of 16 nodes to be assigned zero displacement can be supplied on one card. A maximum of 50 can be specified in one loading step. Nodal point numbers, N = 1 to NONP, are to be specified in sequential order.

Column	Variable	Explanation
1-5	NP(N)	Nodal point number of first node assigned zero displacement.

Information for the next 15 nodal point is supplied in the next 15 five-column fields.

(c) SEEPAGE

These cards are supplied only if KCS(N,1), KCS(N,2), OR KCS(N,3) = 3. Input is handled by subroutine SEEP. Seepage loads are determined from changes in pore pressure specified as DP(N) on nodal point cards, or from the specified phreatic level changes.

(1) FIRST CARD (CONTROL DATA) **FORMAT(I5)**

Column	Variable	Explanation
1-5	NCODE	Option code specifying how seepage is input- = 0 If specified as DP(N) on nodal point cards (section 9). = 1 If to be calculated using the new phreatic surface input on the following cards.

(2) SECOND CARD (NUMBER OF PHREATIC SEGMENTS) **FORMAT(I5)**

This card is required only if NCODE = 1.

Column	Variable	Explanation
1-5	NWAT	Number of phreatic surface segment and points used to specify the new phreatic surface. NWAT must be greater than or equal to 2. The number of phreatic surface segments is equal to NWAT. The minimum value of NWAT is 30.

(3) THIRD CARD (PHREATIC LEVEL DATA) **FORMAT(6D10.2)**

This card is required only if NCODE = 1.

The end points of the phreatic surface segments delineating the new and old phreatic surfaces are specified as x-coordinates and must be the same as the x-coordinates of a nodal point (Figure A8). Both the present and new phreatic surfaces are assumed to be linear between the boundary x-coordinates. The left hand side of the mesh is always the first boundary x-coordinate specified. A boundary x-coordinate on the old phreatic surface will require, usually, specification of the same x-coordinate on the new phreatic surface.

Two end points (x-coordinates), with associated new and old phreatic levels (y-coordinates), are supplied a card each. All end points, N = 1 to NWAT, must be specified.

Column	Variable	Explanation
1-10	XW(N)	X-coordinate boundary the levels PREL(N) and FUEL(N) on the right hand side. (Must be the same as the x-coordinate of a nodal point.)
11-20	PREL(N)	Present level (y-coordinate) of the phreatic surface at XW(N).
21-30	FUEL(N)	New level (y-coordinate) of the phreatic surface at XW(N).

Information for the next end point is supplied in the next 3 ten-column fields.

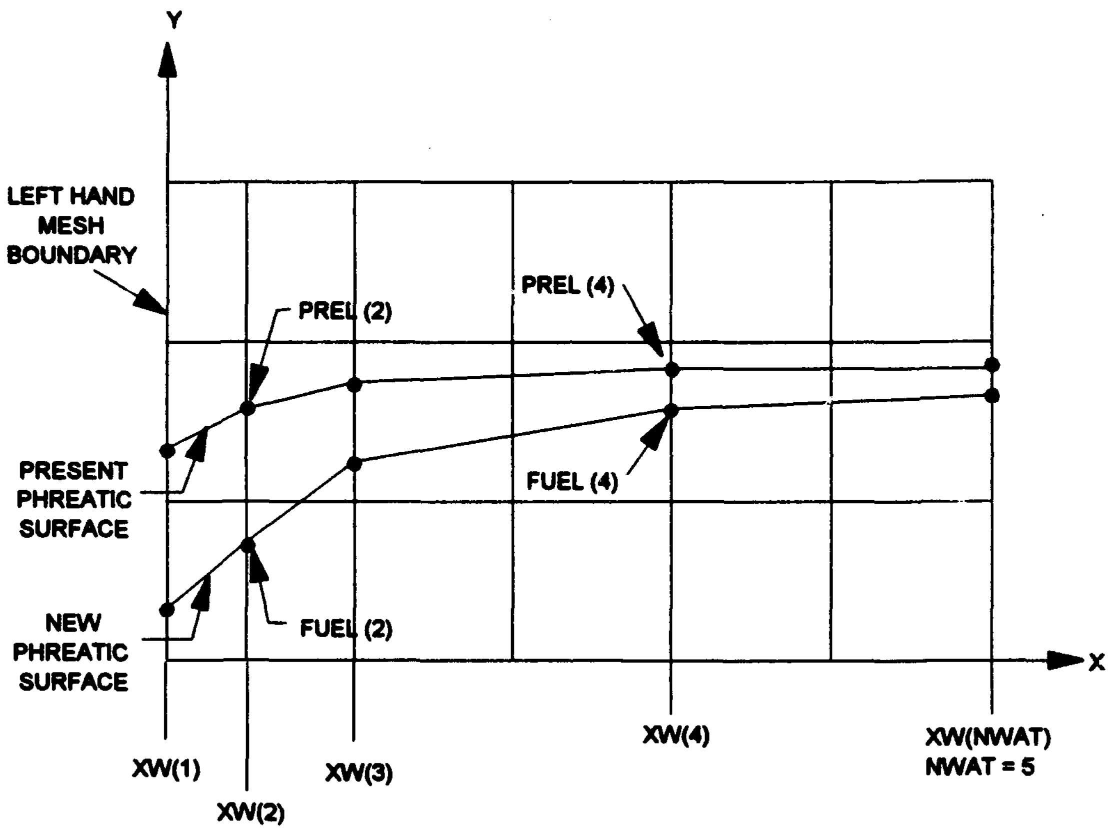


Figure A8. Example illustrating phreatic level data required for the seepage loading/construction case (Clough 1984)

(d) DELETION OF BAR ELEMENTS

These cards supplied only if KCS(N,1), KCS(,N,2), OR KCS(N,3) =4. Input is handled by the main program.

The deleted bar elements remain in the mesh but with zero stiffness. The force the bar element carried is applied to the free node or nodes at its ends.

This loading/construction mode cannot be specified in the same loading as fill or concrete placement.

(1) FIRST CARD (CONTROL DATA) **FORMAT(I5)**

Column	Variable	Explanation
1-5	NCARDS	Number of deleted bar elements. There is no limit other than the number of bar elements presently in the mesh.

(2) SECOND CARD (ELEMENT NUMBERS) **FORMAT(16I5)**

Column	Variable	Explanation
1-5	N	Element number of bar elements to be deleted.

Element numbers for the next 15 elements are supplied in the next 15 five-column fields.

(e) ADDITION OF BAR ELEMENTS

These cards are supplied only if KCS(N,1), KCS(N,2), KCS(B,3) = 5. Input is handled by the main program.

Information on the second card is the same as that explained for the Bar Element Card (section 12). The added bar elements are numbered sequentially form NUMBAR + 1, where NUMBAR is the number of bar elements in the mesh before the present loading step.

(1) FIRST CARD (CONTROL DATA) **FORMAT(I5)**

Column	Variable	Explanation
1-5	NCARDS	Number of bar elements to be added. Any number can be added in a loading step, however, the maximum number of bar elements that can be in the mesh (including inactive or deleted elements) is 15.

(2) SECOND CARD (ADDED BAR ELEMENT DATA)
FORMAT(4I5,2D10.7,2D10.1,D10.5)

Column	Variable	Explanation
1-5	N	Element number of added bar element.
6-10	IB(N,1)	Number of nodal point J
11-15	IB(N,2)	Number of nodal point I

Column	Variable	Explanation
16-20	IB(N,3)	Spring response type code.
21-30	BAR(N,1)	cos α
31-40	BAR(N,2)	sin α
41-50	BAR(N,3)	Prestress force in the bar element.
51-60	BAR(N,4)	Stiffness of bar element
61-70	BAR(N,5)	Displacement of bar element at activation

Note that in a restart analysis the added bar element(s) information must be included in the bar element connectivity cards (Section 12) and the value for NUMBAR in the data control card (Section 2a) must be updated.

(f) BOUNDARY PRESSURE LOADING

These cards are supplied only if KCS(N,1), KCS(N,2), OR KCS(N,3) = 6. Input is handled by subroutine SURFLD. Linear pressure distribution are assumed, based on the pressure specified for the nodal points.

Column	Variable	Explanation
1-5	NLDS	Number of loaded boundaries. There is no limit to the number of loaded boundaries that can be specified.
6-10	NOLDSX	Number of loaded boundaries for which the horizontal components of the nodal points load vectors are set equal to zero.
11-15	NOLDSY	Number of loaded boundary for which the vertical components of the nodal points load vectors are set equal to zero.

(2) SECOND CARD (LOADED BOUNDARY DATA) FORMAT(2I5,4D10.2)

Information for one loaded boundary is specified on each card. Nodes I and J are specified counterclockwise on an element (Figure A9). A normal compressive traction (pressure) is positive. A tangential traction (shear stress) is positive **when directed clockwise (node J to node I) as shown in Figure A9**. A total of N = 1 to (NLDS+NOLDSX+NOLDSY) loaded boundaries must be specified. NLDS cards are specified first, followed by NOLDSX cards and the NOLDSY cards.

Column	Variable	Explanation
1-5	I	Nodal point number of the first node of the loaded boundary.
6-10	J	Nodal point number of the second node of the loaded boundary.
11-20	WS1	Value of the normal pressure acting at node I.
21-30	WS2	Value of the normal pressure acting at node J.
31-40	WS3	Value of the shear stress at node I.
41-50	WS4	Value of the shear stress at node J

(g) TEMPERATURE LOADING

No cards are required for this loading/construction mode. Temperature Card (section 7). If KCS(N,1), KCS(,N,2), or KCS(N,3) = 7, then the values of DP(N) are acknowledged by the main program and processed as temperature changes. Note that DP(N) can also be used to input phreatic level changes for the seepage loading/construction mode. Thus, if seepage is specified as being input through values of DP(N), seepage and temperature loading cannot be included in the same loading step. Generally temperature loading requires a restart analysis, with the DP(N) values being changed to reflect the temperature changes prior to the analysis.

The temperature scale used (°C or °F) must correspond to the coefficient of thermal expansion designated on the Material Property Cards (section 5). Temperature changes are typically designated for structural materials only.

(h) CONCENTRATED FORCE OR DISPLACEMENT LOADING

These cards are supplied only if KCS(N,1), KCS(N,2), or KCS(N,3) = 8 (nodal displacements) or 9 (nodal loads). Input is handled by the main program. Refer to the Loading Information Card (Section 5) for instructions regarding loading/construction modes 8 and 9.

(1) FIRST CARD (CONTROL DATA) FORMAT(I5)

Column	Variable	Explanation
1-5	NUMNDE	Number of loaded or displaced nodes. There is no limit to the number of loaded or displaced nodes that can be specified.

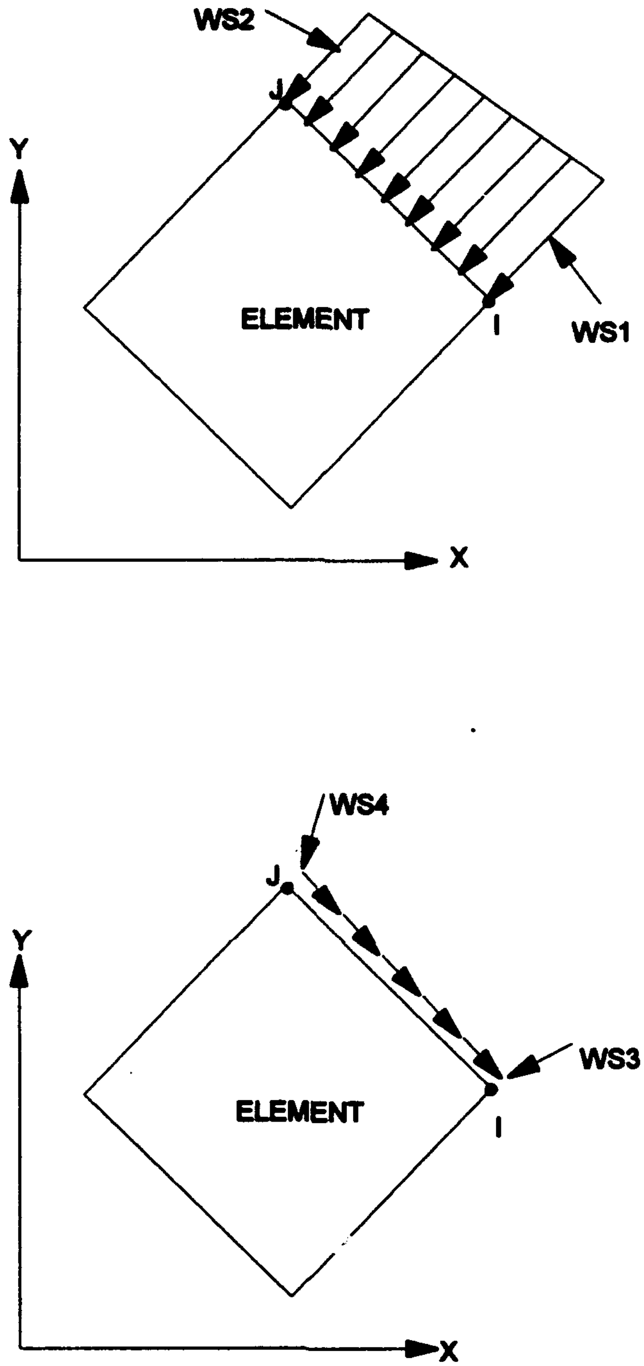


Figure A9. Example of input parameters for positive boundary pressure loading (Ebeling, Peters and Clough, 1992)

(2) SECOND CARD DISPLACEMENT DATA FORMAT(2(I5,2(1X,1D14.7)))
[LOAD DATA] FORMAT(2(I10,2D10.2))

Information for two loaded or displaced nodal points is supplied on each card. A total of N = 1 to NUMNDE nodes must be specified. Sign convention is positive to the right (positive x-direction)and positive up (positive y-direction). Nodal points specified as being loaded or displaced do not have to be in numerical order.

Column	Variable	Explanation
1-5	I	Node number of the first displaced or loaded node.
7-20	X1	Component of displacement or force in the x-direction at node I.
22-35	Y1	Component of displacement or force in the y-direction at node I.
36-40	J	Node number of the second displaced or loaded node.
42-55	X2	Component of displacement or force in the x-direction at node J.
57-70	Y2	Component of displacement or force in the y-direction at node J

If there is no second, or J node to be specified on the last card, then leave the second set of columns blank.

Displaced nodes are to be included in Boundary Condition Codes (Section 10)

(i) ELEMENT MATERIAL TYPE CHANGE

These cards are supplied only if KCS(N,1), KCS(N,2), or KCS(N,3) = 10. Input is handled by the main program.

The material type of the specified element is changed **before** the analysis of the loading step which specifies the change. The material type change includes modifying the values of modulus, E, and Poison's ratio, GUI, and zeroing the stresses, SIG(N,1). Thus, if a material type change is specified in conjunction with boundary loading, in the same loading step, the elements whose material type is changed will respond to the loading with new material properties.

As included, this loading/construction mode is intended to physically represent the grouting of an anchor. At a given step in the analysis, the material types of soil elements can be changed to represent the assumed linear elastic grout zone. If there is a need to change material types for any other reason, this can be done by stopping the analysis after the appropriate load step, modifying the material types on the Material Type Designation Card (section 14D), then restarting the analysis.

(1) FIRST CARD (CONTROL CARD) FORMAT(I5)

The maximum number of elements whose material type number can be changed in a load step is 120. The excavation and material type change loading/construction modes cannot be specified in the same loading step since the same variable, LUL(N,I), is used to input data for both.

Column	Variable	Explanation
1-5	NELCH	Number of elements whose material type number is being changed.

(2) SECOND CARD (ELEMENT DATA) FORMAT(16I5)

The element numbers and new material type numbers of 8 elements can be supplied on one card. A total of N=1 to NELCH elements must be specified.

Column	Variable	Explanation
1-5	LUL(N,1)	Element number of first element with a specified new material type number.
6-10	LUL(N,2)	New material type number of the specified element.

Information for the next seven elements is supplied in the next 7 pairs of five-column fields.

APPENDIX B: SEQUENCE OF OPERATION

1. The program SOILSTRUCT uses the direct stiffness method, $\{F\}=[K]\{u\}$, to solve for incremental nodal displacements $\{u\}$ resulting from incremental loads applied to the nodal points $\{F\}$. The local element stiffness matrices are first formulated. then assembled into the global stiffness matrix. Equivalent nodal loads due to construction or applied loadings are assembled in the incremental load vector, $\{F\}$. The computed incremental displacements are then used to compute the incremental change in stress acting at the center of the elements. The values of total stress are updated by the computed incremental changes in stress. The total stresses are then used to revise the elastic moduli used in the formulation of the element stiffnesses. These procedures are repeated for each iteration, and in turn for each substep of each load case during the analysis. The determination of initial gravity stresses is accomplished in a similar fashion and can be viewed as an initial load step, with the nodal loads equal to the body forces.

2. The following is a listing of the names of each of the 33 subroutines comprising the program SOILSTRUCT and a brief description of their purpose:

- **Main Program**. The main program serves to control the execution of SOILSTRUCT. It calls subroutines, prints input data, load case information, material properties, node and element data, and boundary conditions. The input data for excavation, seepage, embankment construction, and boundary loadings are printed in their respective subroutines. Calculated equivalent nodal loads due to installation or deletion of bar elements, and concentrated forces or displacements, are added in the main program.

- **DETNA**. Subroutine DETNA calculates the number of degrees of freedom, determines the location of the diagonal terms of the global stiffness matrix in the vector SN, and computes the required size of SN.

- **INITAL**. Subroutine INITAL calculates and prints initial stresses for a gravity turn-on analysis. This is done by sequential calls to STRSTF, OPTSOL, and STRESS. If a restart analysis is specified, INITAL reads the continuation data and initializes the material property, stress, and displacement arrays for the nodes and elements.

- **SUBSTP**. Subroutines SUBSTP controls the analysis of each load case when substeps are specified for that load case. SUBSTP divides the calculated equivalent nodal point loads, applied displacements, or temperature changes into the specified number of equal increments prior to performing the analysis.

- **STRSTF**. The terms of the global stiffness matrix are assembled in subroutine STRSTF by sequential calls to QUAD, JTSTF, BAREL and BEMSTF.

- **OPTSOL**. Subroutine OPTSOL solves the series of simultaneous equations using Crout reduction to obtain the incremental displacements.

- **STRESS**. Subroutine STRESS compute stresses and strains for the two-dimensional elements and print results. STRESS call MODCAL, BAREL, and JSTRES, used to update the modulus values for two-dimensional elements, interface stiffnesses, and bar stiffnesses for use in the next iteration or load case.

- **QUAD.** Subroutine QUAD computes the local element stiffness matrix and stress-strain matrix for two-dimensional elements, computes equivalent nodal point forces due to temperature changes of non-soil elements, and compute equivalent nodal point forces due to gravity forces.

- **BAREL**. Subroutine BAREL compute the stiffness of the bar elements, and updates the bar forces.

- **JTSTF**. Subroutine JTSTF computes the stiffness of the interface elements.

- **JSTRES**. Subroutine JSTRES computes the stresses and relative displacements for the interface elements, updates the interface stiffness values, and prints results.

- **MODCAL**. Subroutine MODCAL updates the modulus values assigned to the soil elements.

- **BUILD**. Subroutine BUILD computed the nodal point loads which are equivalent to the weight of the elements representing a newly placed embankment lift and establishes the initial stresses and material properties for these newly placed elements.

- **EXCAV**. Subroutine EXCAV calculates the stresses acting on an excavation boundary.

- **EQNDFO**. Subroutine EQNDFO converts the stresses calculated by EXCAV to equivalent nodal point forces, which are added to the incremental load vector by EXCAV.

- **SURFLD**. Subroutine SURFLD computes equivalent nodal point forces due to a boundary pressure loading applied along the face of an element and adds these computed forces to the incremental load vector.

- **SEEP**. Subroutine SEEP calculates equivalent nodal point forces due to changes in the phreatic surface and adds these forces to the incremental load vector. The nodal point forces are formulated based upon changes in pore water pressures.

- **AUXOUT**. Subroutine AUXOUT writes continuation data to a file for use in subsequent analyses.

- **PRNCIP**. Subroutine PRNCIP calculates principal stresses and the maximum shear strain for two-dimensional elements.

- **PRNTFD**. All non-zero values of the incremental load vector are printed by subroutine PRNTFD.

- **GETFIL**. Subroutine GETFIL initializes the execution of SOILSTRUCT by requesting the names of the input and output files and the corresponding opening and closing of the disc storage devices.

- **NOTENS**. Subroutine NOTENS checks for tension failure within interface elements, reduces excessive stresses within failed interface elements and removes failed elements by assigning zero stiffness.

- **UNBALS**. Subroutine UNBALS computes the unbalanced force for failed element and places these forces in the FD array as well as updates the element stresses.

- **PRNTF0**. Subroutine PRNTF0 prints the contents of the F0 array.

- **PRNTJT**. Subroutine PRNTJT prints the contents of the F0 array for all interface elements.

- **NEWMOD**. Subroutine NEWMOD assigns the new moduli to all elements.

- **INTERF**. Subroutine INTERF checks interface elements for possible tension failure at the center of the element and computes the fraction of applied forces resulting in zero overshoot normal force for that interface element. With σ_t = 0, application of the alpha method results in zero normal stress at the center line of the interface element.

- **ALFA2D**. Subroutine ALFA2D locates the 2-D soil elements whose stress levels are greater than 1.00 and computes the fraction of the increment in stresses that result in a stress level just equal to 1.00.

- **COLAPS**. Subroutine COLAPS checks to if excessive movement has occurred.

- **BEMSTF**. Subroutine BEMSTF calculates the equivalent stiffness matrix of the boundary element system.
- **MATRX**. Subroutine MATRX calculates the matrix of influence coefficient of the boundary element system.
- **FUNC**. Subroutine FUNC computes the boundary integral necessary in the computation of the matrix of influence coefficients.
- **MELAN**. Subroutine MELAN computes the stresses and displacements in the Melan fundamental solution.
- **SETCON**. Subroutine SETCON computes the constants used in the computation of the Melan fundamental solution.
- **NMATRX**. Subroutine NMATRX computes the interpolation matrix N for converting nodal tractions to nodal forces.
- **STFSYM**. Subroutine STFSYM extracts the symmetric part of the boundary element stiffness matrix.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
March 1994

3. REPORT TYPE AND DATES COVERED
Final report

4. TITLE AND SUBTITLE
User's Guide for the Incremental Construction, Soil-Structure Interaction Program SOILSTRUCT with Far-Field Boundary Elements

5. FUNDING NUMBERS

6. AUTHOR(S)
T. Kuppusamy, Mark A. Zarco
Robert M. Ebeling

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Charles Via Department of Civil Engineering, Virginia Polytechnic Institute and State University, Blacksburg, VA 24061 and U.S. Army Engineer Waterways Experiment Station, 3909 Halls Ferry Road, Vicksburg, MS 39180-6199.

8. PERFORMING ORGANIZATION REPORT NUMBER
Technical Report
ITL-94-2

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
U.S. Army Corps of Engineers, Washington DC 20314-1000

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES
Available from National Technical Information Service, 5285 Port Royal Road Springfield, VA 22161.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*
This report describes the computer program SOILSTRUCT, a two-dimensional, plane strain, finite element program used in the incremental construction, soil-structure interaction analysis of earth retaining structures. The initial version of the program was developed by Professors G. W. Clough and J. M. Duncan (1969) for use in the analysis of Port Allen and Old River U-frame locks. The program has been enhanced over the last 20 years by Professor Clough and coworkers.

In this version of SOILSTRUCT, a substructure method for coupling the boundary element method (BEM) with the finite element method (FEM) is incorporated in order to solve soil-structure interaction problems more accurately and efficiently so that the nonlinear effects can be included in the near field by FEM, while the far field is simulated by BEM. Linear boundary elements based on the Melan fundamental solution are coupled with the QM5 finite elements.

SOILSTRUCT has the capability to simulate incremental construction which may include embankment construction or backfilling, dewatering, excavation, installation of a strut or tie back anchor support system, removal of the same system, and the placement of concrete or other construction material. In addition, SOILSTRUCT has the capability to include the modeling of the interface region between the soil backfill and the structure, using interface elements.

14. SUBJECT TERMS
Boundary Element Method
Finite Element Method
Soil-Structure Interaction - Computer Programs, SOILSTRUCT

15. NUMBER OF PAGES
81

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED		

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Technical Report K-78-1	List of Computer Programs for Computer-Aided Structural Engineering	Feb 1978
Instruction Report O-79-2	User's Guide: Computer Program with Interactive Graphics for Analysis of Plane Frame Structures (CFRAME)	Mar 1979
Technical Report K-80-1	Survey of Bridge-Oriented Design Software	Jan 1980
Technical Report K-80-2	Evaluation of Computer Programs for the Design/Analysis of Highway and Railway Bridges	Jan 1980
Instruction Report K-80-1	User's Guide: Computer Program for Design/Review of Curvilinear Conduits/Culverts (CURCON)	Feb 1980
Instruction Report K-80-3	A Three-Dimensional Finite Element Data Edit Program	Mar 1980
Instruction Report K-80-4	A Three-Dimensional Stability Analysis/Design Program (3DSAD)	
	Report 1: General Geometry Module	Jun 1980
	Report 3: General Analysis Module (CGAM)	Jun 1982
	Report 4: Special-Purpose Modules for Dams (CDAMS)	Aug 1983
Instruction Report K-80-6	Basic User's Guide: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Dec 1980
Instruction Report K-80-7	User's Reference Manual: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Dec 1980
Technical Report K-80-4	Documentation of Finite Element Analyses	
	Report 1: Longview Outlet Works Conduit	Dec 1980
	Report 2: Anchored Wall Monolith, Bay Springs Lock	Dec 1980
Technical Report K-80-5	Basic Pile Group Behavior	Dec 1980
Instruction Report K-81-2	User's Guide: Computer Program for Design and Analysis of Sheet Pile Walls by Classical Methods (CSHTWAL)	
	Report 1: Computational Processes	Feb 1981
	Report 2: Interactive Graphics Options	Mar 1981
Instruction Report K-81-3	Validation Report: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Feb 1981
Instruction Report K-81-4	User's Guide: Computer Program for Design and Analysis of Cast-in-Place Tunnel Linings (NEWTUN)	Mar 1981
Instruction Report K-81-6	User's Guide: Computer Program for Optimum Nonlinear Dynamic Design of Reinforced Concrete Slabs Under Blast Loading (CBARCS)	Mar 1981
Instruction Report K-81-7	User's Guide: Computer Program for Design or Investigation of Orthogonal Culverts (CORTCUL)	Mar 1981
Instruction Report K-81-9	User's Guide: Computer Program for Three-Dimensional Analysis of Building Systems (CTABS80)	Aug 1981
Technical Report K-81-2	Theoretical Basis for CTABS80: A Computer Program for Three-Dimensional Analysis of Building Systems	Sep 1981
Instruction Report K-82-6	User's Guide: Computer Program for Analysis of Beam-Column Structures with Nonlinear Supports (CBEAMC)	Jun 1982

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Instruction Report K-82-7	User's Guide: Computer Program for Bearing Capacity Analysis of Shallow Foundations (CBEAR)	Jun 1982
Instruction Report K-83-1	User's Guide: Computer Program with Interactive Graphics for Analysis of Plane Frame Structures (CFRAME)	Jan 1983
Instruction Report K-83-2	User's Guide: Computer Program for Generation of Engineering Geometry (SKETCH)	Jun 1983
Instruction Report K-83-5	User's Guide: Computer Program to Calculate Shear, Moment, and Thrust (CSMT) from Stress Results of a Two-Dimensional Finite Element Analysis	Jul 1983
Technical Report K-83-1	Basic Pile Group Behavior	Sep 1983
Technical Report K-83-3	Reference Manual: Computer Graphics Program for Generation of Engineering Geometry (SKETCH)	Sep 1983
Technical Report K-83-4	Case Study of Six Major General-Purpose Finite Element Programs	Oct 1983
Instruction Report K-84-2	User's Guide: Computer Program for Optimum Dynamic Design of Nonlinear Metal Plates Under Blast Loading (CSDOOR)	Jan 1984
Instruction Report K-84-7	User's Guide: Computer Program for Determining Induced Stresses and Consolidation Settlements (CSETT)	Aug 1984
Instruction Report K-84-8	Seepage Analysis of Confined Flow Problems by the Method of Fragments (CFRAG)	Sep 1984
Instruction Report K-84-11	User's Guide for Computer Program CGFAG, Concrete General Flexure Analysis with Graphics	Sep 1984
Technical Report K-84-3	Computer-Aided Drafting and Design for Corps Structural Engineers	Oct 1984
Technical Report ATC-86-5	Decision Logic Table Formulation of ACI 318-77, Building Code Requirements for Reinforced Concrete for Automated Constraint Processing, Volumes I and II	Jun 1986
Technical Report ITL-87-2	A Case Committee Study of Finite Element Analysis of Concrete Flat Slabs	Jan 1987
Instruction Report ITL-87-1	User's Guide: Computer Program for Two-Dimensional Analysis of U-Frame Structures (CUFRAM)	Apr 1987
Instruction Report ITL-87-2	User's Guide: For Concrete Strength Investigation and Design (CASTR) in Accordance with ACI 318-83	May 1987
Technical Report ITL-87-6	Finite-Element Method Package for Solving Steady-State Seepage Problems	May 1987
Instruction Report ITL-87-3	User's Guide: A Three Dimensional Stability Analysis/Design Program (3DSAD) Module	Jun 1987
	Report 1: Revision 1: General Geometry	Jun 1987
	Report 2: General Loads Module	Sep 1989
	Report 6: Free-Body Module	Sep 1989

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Instruction Report ITL-87-4	User's Guide: 2-D Frame Analysis Link Program (LINK2D)	Jun 1987
Technical Report ITL-87-4	Finite Element Studies of a Horizontally Framed Miter Gate Report 1: Initial and Refined Finite Element Models (Phases A, B, and C), Volumes I and II Report 2: Simplified Frame Model (Phase D) Report 3: Alternate Configuration Miter Gate Finite Element Studies–Open Section Report 4: Alternate Configuration Miter Gate Finite Element Studies–Closed Sections Report 5: Alternate Configuration Miter Gate Finite Element Studies–Additional Closed Sections Report 6: Elastic Buckling of Girders in Horizontally Framed Miter Gates Report 7: A.:lication and Summary	Aug 1987
Instruction Report GL-87-1	User's Guide: UTEXAS2 Slope-Stability Package; Volume I, User's Manual	Aug 1987
Instruction Report ITL-87-5	Sliding Stability of Concrete Structures (CSLIDE)	Oct 1987
Instruction Report ITL-87-6	Criteria Specifications for and Validation of a Computer Program for the Design or Investigation of Horizontally Framed Miter Gates (CMITER)	Dec 1987
Technical Report ITL-87-8	Procedure for Static Analysis of Gravity Dams Using the Finite Element Method – Phase 1a	Jan 1988
Instruction Report ITL-88-1	User's Guide: Computer Program for Analysis of Planar Grid Structures (CGRID)	Feb 1988
Technical Report ITL-88-1	Development of Design Formulas for Ribbed Mat Foundations on Expansive Soils	Apr 1988
Technical Report ITL-88-2	User's Guide: Pile Group Graphics Display (CPGG) Post-processor to CPGA Program	Apr 1988
Instruction Report ITL-88-2	User's Guide for Design and Investigation of Horizontally Framed Miter Gates (CMITER)	Jun 1988
Instruction Report ITL-88-4	User's Guide for Revised Computer Program to Calculate Shear, Moment, and Thrust (CSMT)	Sep 1988
Instruction Report GL-87-1	User's Guide: UTEXAS2 Slope-Stability Package; Volume II, Theory	Feb 1989
Technical Report ITL-89-3	User's Guide: Pile Group Analysis (CPGA) Computer Group	Jul 1989
Technical Report ITL-89-4	CBASIN–Structural Design of Saint Anthony Falls Stilling Basins According to Corps of Engineers Criteria for Hydraulic Structures; Computer Program X0098	Aug 1989

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Technical Report ITL-89-5	CCHAN–Structural Design of Rectangular Channels According to Corps of Engineers Criteria for Hydraulic Structures; Computer Program X0097	Aug 1989
Technical Report ITL-89-6	The Response-Spectrum Dynamic Analysis of Gravity Dams Using the Finite Element Method; Phase II	Aug 1989
Contract Report ITL-89-1	State of the Art on Expert Systems Applications in Design, Construction, and Maintenance of Structures	Sep 1989
Instruction Report ITL-90-1	User's Guide: Computer Program for Design and Analysis of Sheet Pile Walls by Classical Methods (CWALSHT)	Feb 1990
Technical Report ITL-90-3	Investigation and Design of U-Frame Structures Using Program CUFRBC Volume A: Program Criteria and Documentation Volume B: User's Guide for Basins Volume C: User's Guide for Channels	May 1990
Instruction Report ITL-90-6	User's Guide: Computer Program for Two-Dimensional Analysis of U-Frame or W-Frame Structures (CWFRAM)	Sep 1990
Instruction Report ITL-90-2	User's Guide: Pile Group–Concrete Pile Analysis Program (CPGC) Preprocessor to CPGA Program	Jun 1990
Technical Report ITL-91-3	Application of Finite Element, Grid Generation, and Scientific Visualization Techniques to 2-D and 3-D Seepage and Groundwater Modeling	Sep 1990
Instruction Report ITL-91-1	User's Guide: Computer Program for Design and Analysis of Sheet-Pile Walls by Classical Methods (CWALSHT) Including Rowe's Moment Reduction	Oct 1991
Instruction Report ITL-87-2 (Revised)	User's Guide for Concrete Strength Investigation and Design (CASTR) in Accordance with ACI 318-89	Mar 1992
Technical Report ITL-92-2	Fiinite Element Modeling of Welded Thick Plates for Bonneville Navigation Lock	May 1992
Technical Report ITL-92-4	Introduction to the Computation of Response Spectrum for Earthquake Loading	Jun 1992
Instruction Report ITL-92-3	Concept Design Example, Computer Aided Structural Modeling (CASM) Report 1: Scheme A Report 2: Scheme B Report 3: Scheme C	 Jun 1992 Jun 1992 Jun 1992
Instruction Report ITL-92-4	User's Guide: Computer-Aided Structural Modeling (CASM) - Version 3.00	Apr 1992
Instruction Report ITL-92-5	Tutorial Guide: Computer-Aided Structural Modeling (CASM) - Version 3.00	Apr 1992

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Concluded)

	Title	Date
Contract Report ITL-92-1	Optimization of Steel Pile Foundations Using Optimality Criteria	Jun 1992
Technical Report ITL-92-7	Refined Stress Analysis of Melvin Price Locks and Dam	Sep 1992
Contract Report ITL-92-2	Knowledge-Based Expert System for Selection and Design of Retaining Structures	Sep 1992
Contract Report ITL-92-3	Evaluation of Thermal and Incremental Construction Effects for Monoliths AL-3 and AL-5 of the Melvin Price Locks and Dam	Sep 1992
Instruction Report GL-87-1	User's Guide: UTEXAS3 Slope-Stability Package; Volume IV, User's Manual	Nov 1992
Technical Report ITL-92-11	The Seismic Design of Waterfront Retaining Structures	Nov 1992
Technical Report ITL-92-12	Computer-Aided, Field-Verified Structural Evaluation	
	Report 1: Development of Computer Modeling Techniques for Miter Lock Gates	Nov 1992
	Report 2: Field Test and Analysis Correlation at John Hollis Bankhead Lock and Dam	Dec 1992
	Report 3: Field Test and Analysis Correlation of a Vertically Framed Miter Gate at Emsworth Lock and Dam	Dec 1993
Instruction Report GL-87-1	User's Guide: UTEXAS3 Slope-Stability Package; Volume III, Example Problems	Dec 1992
Technical Report ITL-93-1	Theoretical Manual for Analysis of Arch Dams	Jul 1993
Technical Report ITL-93-2	Steel Structures for Civil Works, General Considerations for Design and Rehabilitation	Aug 1993
Technical Report ITL-93-3	Soil-Structure Interaction Study of Red River Lock and Dam No. 1 Subjected to Sediment Loading	Sep 1993
Instruction Report ITL-93-3	User's Manual—ADAP, Graphics-Based Dam Analysis Program	Aug 1993
Instruction Report ITL-93-4	Load and Resistance Factor Design for Steel Miter Gates	Oct 1993
Technical Report ITL-94-2	User's Guide for the Incremental Construction, Soil-Structure Interaction Program SOILSTRUCT with Far-Field Boundary Elements	Mar 1994