Appendix B

IMPLEMENTATION OF COUPLED CONSOLIDATION IN SAGE

B.1. INTRODUCTION

The purpose of this appendix is to describe the implementation of coupled consolidation into the finite element program SAGE. First, some features of SAGE are described in order to provide a background for understanding some of the implementation issues of the changes to SAGE. Next, the general features of the implementation of coupled consolidation in SAGE are described. In the interest of brevity many of the minor details are left out.

B.2. SAGE

The purpose of this section is to describe two things. The first is the basic structure of SAGE. The second is the use of degree of freedom mapping arrays in SAGE. A basic knowledge of these two things is required in order to understand the implementation of the changes described in this appendix.

STRUCTURE OF SAGE

SAGE is written in FORTRAN 77 in a very structured program style. The key features of the programming of SAGE are that subroutines are used to perform individual tasks and common blocks are not used (i.e. all arrays and variables are passed from subroutine to subroutine through argument lists). These features make the SAGE code easier to document, maintain, and modify. This section describes important elements of how SAGE performs an analysis.

SAGE is designed to perform incremental analyses. The load history of each problem analyzed is modeled with a series of construction steps. Each construction step is used to model a different stage in the history of a problem. The initial conditions for each construction step are the results of the calculations for the previous construction step. The finite element mesh and the boundary conditions for each construction step are defined in terms of changes from the previous construction step. For example, excavation is modeled by applying loads to simulate the removal of material, and by deactivating elements that were active in the previous step. Point loads, specified displacements, and filling (placement of 2-D elements) can also be modeled. The use of incremental analyses allows complex loading histories to be modeled.

Each construction step is divided into substeps. The number of substeps, n, for any construction step must be one or more. The changes in hydraulic boundary conditions and the load vector for the construction step are divided into n equal increments to obtain the incremental boundary conditions and loads for the substep finite element analyses. The finite element equations are assembled and solved for each substep using Newton-Raphson or modified Newton-Raphson iteration methods. Morrison (1995) described of the iteration algorithms used in SAGE. The results of each substep calculation are incremental displacements, stresses, and forces. The incremental values from a substep calculation are added to arrays that store the cumulative values of displacements, stresses, and forces.

DEGREE OF FREEDOM MAPPING ARRAYS

SAGE is designed to be a general-purpose finite element program for solving geotechnical problems. The creation of a general-purpose program requires great internal flexibility. Many bookkeeping arrays are used in SAGE to provide the required flexibility. One type of bookkeeping array is degree of freedom mapping arrays.

Degree of freedom mapping arrays are used in SAGE to simplify tasks like assembling the global stiffness matrix and applying boundary conditions to the assembled finite element equations. Several factors complicate assembling the global stiffness matrix and applying boundary conditions in SAGE. One complication is that the number of active nodes and elements can change from step to step in an analysis. A second complicating factor is that there are several different types of elements in SAGE. For instance, the rotation degree of freedom associated with the beam-bar element in SAGE is not present for any other element

Degree of freedom mapping arrays are very similar to arrays used to specify element connectivity. A connectivity array maps the local nodes of individual elements onto global node numbers. Degree of freedom mapping arrays are similar because they map local degrees of freedom for nodes to degrees of freedom for elements or the entire mesh. Hence, there are two types of degree of freedom mapping arrays used in SAGE. One type is a global degree of freedom mapping array. A global degree of freedom mapping array maps the local degrees of freedom for individual nodes onto global degrees of freedom. The second type of array is an element degree of freedom mapping array. An element degree of freedom mapping array maps the degrees of freedom for the local nodes of an element to the degrees of freedom for the element.

Many of the changes made to SAGE during the implementation of coupled consolidation were simplified by the flexibility that the degree of freedom mapping arrays provide.

B.3. COUPLED CONSOLIDATION

The need for coupled consolidation in SAGE is described in Chapter 3. This section describes how coupled consolidation is implemented in SAGE. First, some important details of the finite element formulation used are discussed.

Next, the manner in which each of the types of elements in SAGE was incorporated into the coupled consolidation capability is discussed. Only the most significant details of the implementation are discussed.

FINITE ELEMENT FORMULATION

The finite element formulation for coupled consolidation, or Biot consolidation, is derived in Appendix A and discussed in Chapter 3. Recall that the formulation is

$$\begin{bmatrix} [K] & [K_v]^T \\ [K_v] & \Delta t\theta[K_h] \end{bmatrix} \begin{Bmatrix} \{\Delta d\} \\ \{\gamma_w h\}\big|_{t+\Delta t} \end{Bmatrix} = \begin{Bmatrix} \{F^1\}\big|_{t+\Delta t} - \{F^1\}\big|_t + [K_v]^T\{\gamma_w h\}\big|_t \\ \Delta t\left(\{Q^1\}\big|_{t+\Delta t} - \{Q^1\}\big|_t\right) - \Delta t(1-\theta)[K_h]\{\gamma_w h\}\big|_t \end{Bmatrix}$$

Equation B.36

One thing not previously discussed about the formulation is the potential for round-off error. Round-off error can be a problem during solution of the finite element equations because of difference in the magnitudes of the terms in the $[K_h]$ and [K] matrices. Recall that

$$[K] = \int_\Omega [B]^T [D][B] d\Omega$$

Equation B.37

and

$$[K_h] = \int_\Omega [B_h]^T [\kappa][B_h] d\Omega$$

Equation B.38

The strain displacement matrix, $[B]$, and the gradient head matrix, $[B_h]$, are composed of terms of similar magnitudes. There can be a large difference in the terms in the stress-strain matrix, $[D]$, and the permeability matrix, $[\kappa]$, depending on the units used. The units for the stress-strain matrix are stress (force/area) and the units for permeability are (length/time). This difference is measured in orders of magnitude and can cause serious numerical round-off errors when Gaussian elimination techniques are used to solve the finite element equations. One way of dealing with the problem is to use units for the soil stiffness and permeability that result in their numerical values that are within the same order of magnitude. This is not a user-friendly solution, since it may require using inconvenient units. Another way of dealing with the numerical round-off problems is to use the following formulation.

$$\begin{bmatrix} [K] & \alpha[K_v]^T \\ \alpha[K_v] & \alpha^2\Delta t\theta[K_h] \end{bmatrix} \begin{Bmatrix} \{\Delta d\} \\ \left\{\dfrac{\gamma_w h}{\alpha}\right\} \end{Bmatrix}_{t+\Delta t} = \begin{Bmatrix} \{F\}\big|_{t+\Delta t} - \{F\}\big|_t + [K_v]^T\{\gamma_w h\}\big|_t \\ \alpha\left\{\Delta t\left(\{Q\}\big|_{t+\Delta t} - \{Q\}\big|_t\right) - \Delta t(1-\theta)[K_h]\{\gamma_w h\}\big|_t\right\} \end{Bmatrix}$$

Equation B.39

The purpose of multiplying terms in Equation B.39 by α and α^2 is to keep the resulting terms in the finite element equations in the same order of magnitude. Equation B.39 is the finite element formulation used in SAGE for coupled consolidation. In SAGE α is a user-specified input parameter. Using double precision variables and arrays in SAGE also helps to reduce round-off error problems. α can be estimated by taking the square root of the Young's Modulus, E, divided by the permeability, k, of a material ($\sqrt{E/k}$). This value will vary with material properties and with stress conditions for materials with stress dependent stiffness. The objective is to chose a value of α that will scale the transmissivity matrix terms to within one or two orders of magnitude of the stiffness matrix terms. The parameter ALPHAR in the input data file for a SAGE analysis corresponds to α.

The simplest manner to implement the time stepping required for coupled consolidation in SAGE was for each substep to occur over an increment of time. This means that any load or boundary conditions imposed in a construction step occur in a linear manner during the time corresponding to the construction step. Hence, all applied loads are ramp loads. For instance, if a force of 100 lbs. is specified at a node during a step with 10 substeps, 10 lbs. is applied in each substep. There is a difference in the application of displacement and hydraulic boundary conditions. A displacement boundary condition during a construction step signifies how much displacement occurs during that step. In contrast, when head is specified for a node during a construction step, it is the value of head at the node at the end of the step. For instance, suppose a head of 15 feet is assigned during a step to a node that had a head of 10 feet at the end of the previous step. If five substeps were used, SAGE would impose the head boundary condition so that the head would be 11 feet after the first step, 12 feet after the second step, and so on.

ELEMENT TYPES

The variety of element types available in SAGE presented one of the greatest challenges during the implementation of coupled consolidation. The main types of elements in SAGE are 2-D triangular and quadrilateral elements, interface elements, and beam-bar elements. The purpose of having all of these element types is to increase the modeling capability and flexibility of SAGE.

The finite element formulation for coupled consolidation only applies to continuum elements. Hence, special consideration had to be given to interface and beam-bar elements. The following paragraphs describe how the implementation of coupled consolidation affected each group of elements.

2-D Elements

2-D continuum elements are used in SAGE to model soil and structural materials such as concrete or steel. Since structural materials do not consolidate while soil does, there was a need to create consolidating and non-consolidating 2-D elements. The following paragraphs discuss the implementation of the consolidating and non-consolidating 2-D elements in SAGE.

Consolidating 2-D elements use the coupled finite element formulation presented previously in this appendix. Two basic changes were required to implement consolidating 2-D elements in SAGE. One was to add pore water head as a new nodal degree of freedom and choose shape functions for pore water head. The second was to generate the element stiffness matrices required for the coupled consolidation formulation. Once the shape functions for pore water head were chose, generation of the stiffness matrices required for coupled consolidation was accomplished by adding new subroutines and modifying some old subroutines. The following paragraph describes how the shape functions for nodal head were chosen.

The previous version of SAGE contained 3 and 6 node triangular elements and 4, 8, and 9 node quadrilateral elements. In coupled consolidation, displacements and nodal pore water head are interpolated over elements. Nodal head is directly related to pore water pressure, which is a stress, and the derivatives of nodal displacement are strain, which is related to effective stress through the stress-strain matrix. Choosing a shape function for nodal head of one degree lower than the shape function for displacement ensures that pore pressures are interpolated over the element in the same manner as the effective stress in the soil skeleton (Sandhu and Wilson; 1969). In other words, if quadratic shape functions are used for displacements, linear shape functions should be used for nodal pore water head. Therefore, coupled consolidation was implemented only for the 6 node triangular and 8 and 9 node quadrilateral elements. Nodal head is an active degree of freedom only at the vertex nodes of the triangular and quadrilateral elements. Figure B.1 shows the consolidation elements implemented into SAGE.

Non-consolidating 2-D elements do not allow pore water flow. This means that only the formulation for equilibrium needs to be satisfied for these elements. Hence, the element stiffness matrices $\left[K_v\right]$, $\left[K_v\right]^T$, and $\left[K_h\right]$ are identically zero and nodal head is not an active degree of freedom at any node in non-

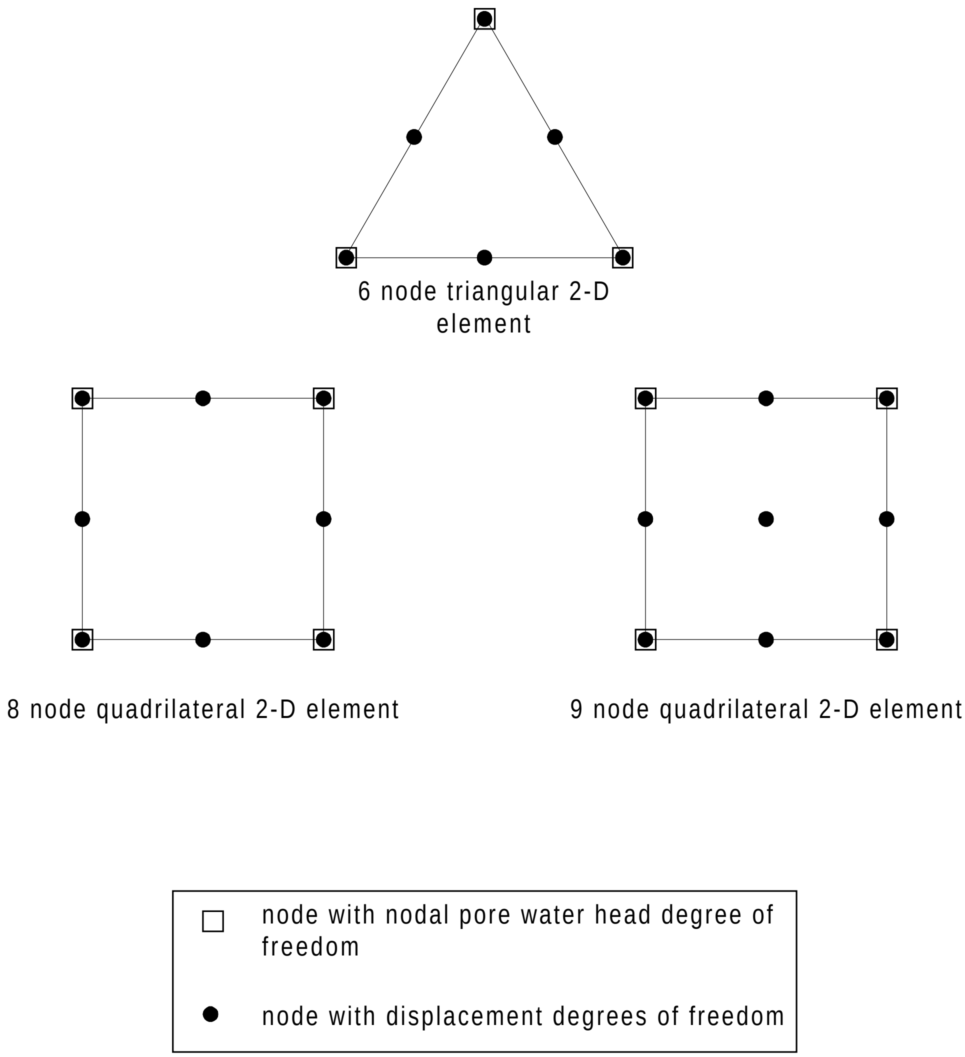

Figure B.1. 2-D consolidation elements in SAGE consolidating elements. These characteristics of non-consolidating 2-D elements match the characteristics of the 2-D elements implemented in the previous version of SAGE. Therefore, no significant work was required to implement non-consolidating 2-D elements. The only requirement was to distinguish non-consolidating 2-D elements from consolidating 2-D elements. Using different element type numbers for non-consolidating and consolidating 2-D elements in SAGE accomplished this.

Interface Elements

Zero-thickness interface elements are used in SAGE to model interfaces between dissimilar materials. Because these interface elements have zero thickness, they can not consolidate, or change volume. The issue of how interface elements deal with pore water flow has two logical alternatives. The two alternatives were implemented in SAGE as separate types of interface elements. These alternatives and their implementations are discussed below.

In the first alternative, permeable interface elements, have no effect on pore water flow. This is consistent with the zero thickness assumption. A zero thickness interface element cannot consolidate because it has no thickness, and therefore cannot change volume. The zero thickness interface element only represents a discontinuity for displacements (i.e. sliding is allowed) and therefore pore water is continuous across an interface.

An interface element with nodal head as an active degree of freedom was created to implement this first alternative, or permeable interface. The degree of freedom mapping arrays used in SAGE allow the degrees of freedom for nodal head to be identical for adjacent nodes. Adjacent nodes are two nodes that are on opposite sides of the interface, but occupy the same point in space. This means that the pore water head is identical for adjacent nodes on the interface element. This connection of adjacent nodes satisfies the continuity equations and means that

$[K_v]$ and $[K_h]$ matrices are zero for the element. Figure B.2 illustrates how permeable interface elements function in a coupled analysis. Note that the 6-node permeable interface element is compatible with the 2-D consolidation elements. The equilibrium equations for the permeable interface element were changed to include the force due to pore water pressure. This was accomplished by creating a $[K_v]^T$ matrix for the permeable interface element. Note that forces due to pore water pressure are simply transmitted through the interface, since the pore water head is identical for opposite nodes of the permeable interface element. This means that the forces due to displacements in the permeable interface element are due to effective stresses and not total stresses. If the $[K_v]^T$ matrix were omitted from the formulation for the permeable interface element, the stresses in the element would be total stresses.

The second alternative, impermeable interface elements, is needed for compatibility if an interface between non-consolidating 2-D elements is modeled. This alternative did not require much work, since the interface element type of the previous SAGE functions like an impermeable element without any changes. Element type numbers are used in SAGE to distinguish between permeable and impermeable interface elements.

Beam-bar elements

Beam-bar elements are 1-D finite elements used in SAGE to model sheet piles, struts, and anchor tendons. None of these consolidate. It is important to remember that beam-bar elements are 1-D elements, which can be used within a mesh of 2-D elements. In one sense, beam-bar elements are simply superimposed on top of the 2-D mesh. Beam-bar elements only contribute to the $[K]$ portion of the global stiffness matrix.

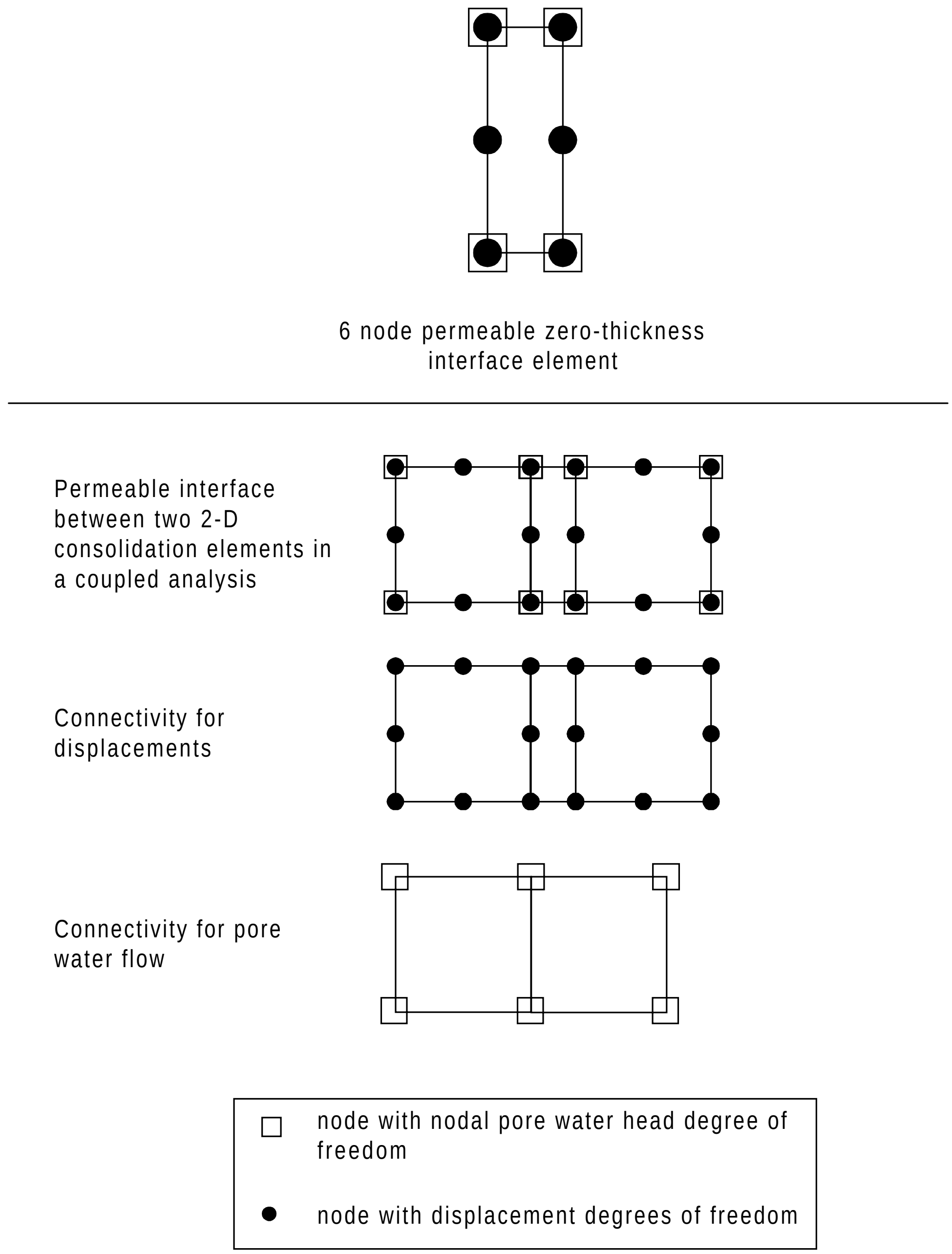

Figure B.2. Permeable zero-thickness interface element

Leaving the beam-bar elements implemented as in the previous version of SAGE would mean that beam-bar elements would not have any effect on pore fluid flow (they would always be permeable). For some modeling situations, this permeable beam-bar element is perfectly acceptable. For example, an anchor tendon has no effect on pore fluid flow. In other cases, for example, sheetpile walls, it is desirable to have the structural element modeled with the beam-bar element act as a barrier to pore fluid flow. This dictated the need to create a beam-bar element that would be a barrier to pore water flow (i.e. an impermeable beam-bar element).

Both impermeable and permeable beam-bar element types were implemented in SAGE. No significant work was required to implement the permeable beam-bar element. The first step in implementing the impermeable beam-bar element was to give it a distinct element type number. The second step to implement the impermeable element was to create a dummy node for each real node of the element. Each dummy node is assigned the same displacement and rotation degrees of freedom as the real node with which it is paired. Separate degrees of freedom are assigned to the dummy and real nodes for pore water head. The manipulation of the degrees of freedom is possible because of the degree of freedom mapping arrays. When an impermeable beam-bar element is placed in the mesh between two 2-D elements, one 2-D element shares the real nodes of the beam-bar element and the second 2-D element shares the dummy nodes of the beam-bar element. Figure B.3 illustrates how permeable and impermeable beam-bar elements function.

INITIAL CONDITIONS

Initial conditions must be established before a coupled consolidation analysis can begin. At the beginning of every SAGE analyses (before any calculations) all element stresses, nodal displacements, and nodal heads are set to zero. Many of

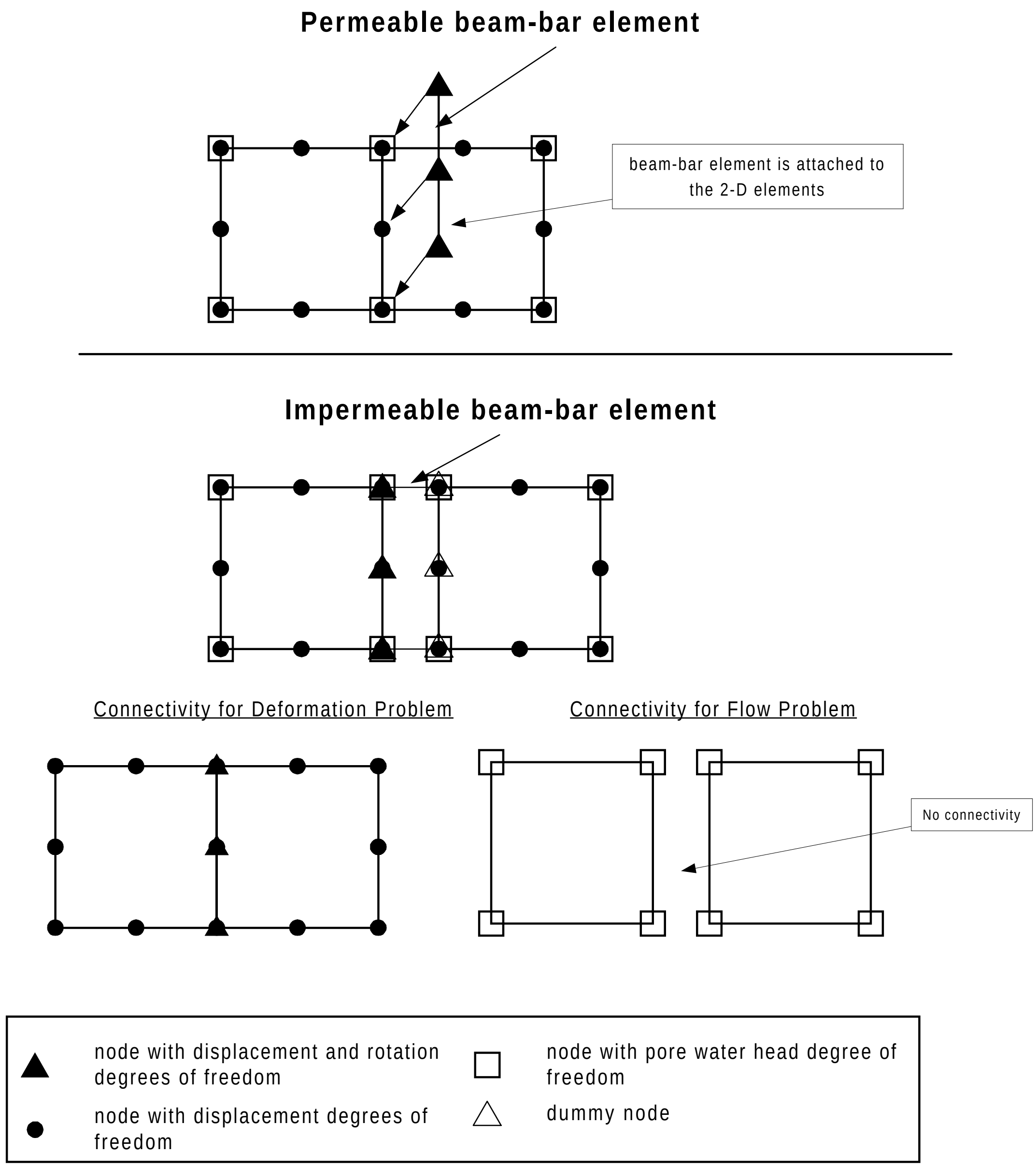

Figure B.3. Permeable and impermeable beam-bar elements the material models used in SAGE require non-zero initial stresses in order to function properly. Two approaches for obtaining initial conditions for a coupled analysis were implemented.

One approach is to assign initial stresses and nodal heads to every Gauss point and node before the analysis begins. Another way of accomplishing this is to allow the user to specify element stresses and nodal heads. This was implemented in SAGE.

The second approach for establishing initial conditions is to use the program to calculate initial stresses and nodal heads. This approach was implemented by setting aside a group of constructions steps that use the uncoupled finite element formulation. Stresses can be established in these initial construction steps using features such as gravity turn-on, fill placement, excavation, and imposition of forces. These features are described by Morrison (1995). In addition, two new features were implemented to facilitate calculation of initial conditions. One of these is a steady state pore fluid flow calculation module. The second is piezometric lines. Both are described in Chapter 4.