



## In this issue:

Boundary Conditions Simplified

### **Boundary Conditions Simplified**

There are three components to a Finite Element Analysis: geometry, soil material behavior (properties or constitutive models), and boundary conditions. This article will provide some clarification regarding how certain boundary conditions are specified and how they are used in the solution process. Examples will be taken from [SEEP/W](#) and [SIGMA/W](#) to illustrate some important concepts.

### **What the solver does**

SEEP/W solves a partial differential equation for finite element liquid water flow where the water flows in response to a hydraulic energy gradient (e.g., total head). The rate of flow is a function of the gradient and the hydraulic conductivity, which is the resistance the water experiences as it moves through the voids in the soil profile.

SIGMA/W solves a force equilibrium equation where the displacement of the soil depends on the force applied and the stiffness of the soil. While the equations can be complicated when viewed in their partial differential form, they are ultimately reduced to the following simple forms inside the solver:

$$[K]\{H\} = \{Q\} \text{ for seepage, and}$$
$$[K]\{D\} = \{F\} \text{ for stress/deformation.}$$

In these equations the K term represents the resistance to flow for seepage analyses, and the soil stiffness for stress/deformation analyses. In either case, the K values are material properties and are input by the user. In other words, the solver never computes the K value as its primary output objective. The H and D terms are the hydraulic heads and displacements at each node in the finite element mesh, and Q and F are the nodal flows and forces in SEEP/W and SIGMA/W respectively.

In order to solve the partial differential equations, the user must specify either head (H) or flow (Q) as a boundary condition in SEEP/W; or in SIGMA/W, either displacement (D) or force (F) must be specified.

For example, if in a seepage analysis you input a nodal hydraulic head boundary condition, then the solver will compute flow at that node. Conversely, if you input a nodal flow then the solver will compute the head that results from that flow. If you say nothing about a boundary condition at a node it is assumed to mean  $Q = 0$ , and the solver will compute the head. It is important to note that you can not control both head and flow at a single node. One is defined and the other computed.

A similar logic applies in the force/displacement equation solver. If you specify force, the solver computes displacement. If you say nothing about force or displacement at a node it is assumed that  $F = 0$  and the solver will compute the displacement.

## Boundary conditions

The solver program only has two options when it solves the seepage or displacement equations at all the nodes in the finite element mesh. What can be confusing is that the software provides several different ways to specify these two unknown variables.

In a seepage model, you can specify total head in the following ways:

- as a known value (H);
- as a water pressure head (P) which the solver then converts to total head,  $H = P + y$  where y is the elevation head or the y-coordinate of the node.

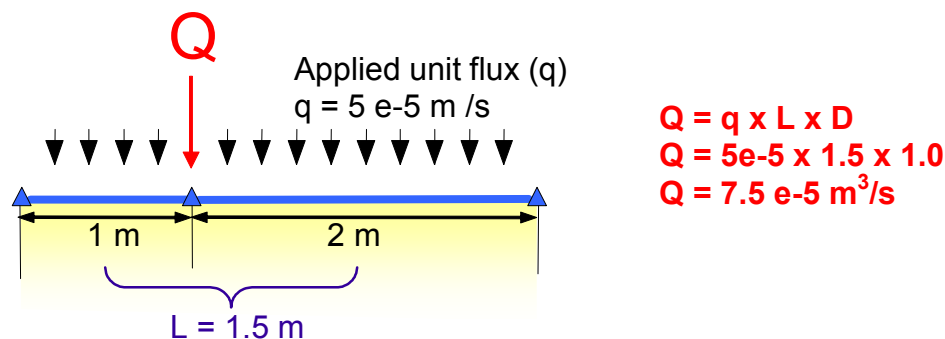
Alternatively, you can specify the volume flow at a node in the following ways:

- as a known volume flow value (Q);
- as a unit flux rate (q) applied over a specific area which the solver converts to  $Q = q \times \text{area}$ .

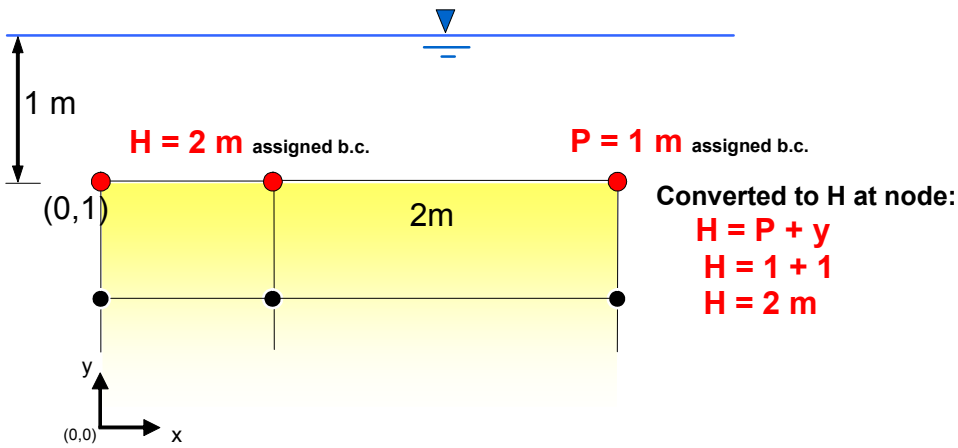
In a displacement model you can specify the displacement directly, or you can specify the force in the following two ways:

- as a known force value (F);
- as a surface pressure or stress (S) applied over a specific area which the solver converts to nodal forces using the relationship  $F = S / \text{area}$ .

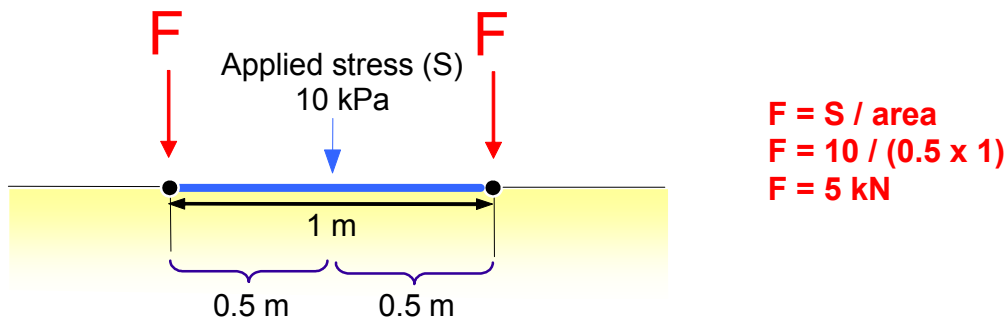
These concepts are more easily understood when applied to actual model cross-sections as shown in the images below.



**Figure 1: An edge unit flux and its nodal Q equivalent**



**Figure 2: A total head and its pressure head equivalent**



**Figure 3: An edge stress and its nodal force equivalent**

In all of these cases, when a boundary condition is applied to an element edge it is converted to an equivalent nodal value by the solver after taking into account the contributing element length over which the boundary condition was applied and assuming a unit depth. It is much easier for the solver to know the contributing area of each element than for you to do the conversions manually and apply nodal flows or forces.

The key point to remember is that the solver converts the boundary conditions to Head or Volume Flow in the seepage case, or Forces and Displacements in the stress/deformation case. The software allows you to specify these boundary conditions in several different ways, but ultimately all boundary conditions are reduced in order to solve the partial differential equations.

If you are unsure about how a particular boundary condition is going to be reduced, set up a simple model like those shown above and do a quick test to clarify your understanding.

Back issues of Direct Contact are available online in our [newsletter archive](#).