13 Dewatered Construction of a Braced Excavation

13.1 Problem Statement

A braced excavation is constructed in saturated ground. The excavation is dewatered during construction and is supported by diaphragm walls that are braced at the top by horizontal struts. The purpose of the *FLAC* analysis is to evaluate (1) the deformation of the ground adjacent to the walls and at the bottom of the excavation, and (2) the performance of the walls and struts throughout the construction stages. The analysis starts from the stage after the walls have been constructed, but prior to any excavation. Dewatering, excavation and installation of struts are simulated in separate construction stages. In addition, three different material models (the Mohr-Coulomb model, the Cysoil model and the Plastic Hardening [PH] model) are used to represent the behavior of the soils and compare the difference in deformational response produced by these models when subjected to this construction sequence.

In practice, the construction may involve several stages of dewatering, excavation and adding of support. For simplicity, in this example, only three construction stages are analyzed: (1) dewatering to a 20 m depth in the region to be excavated; (2) excavation to a 2 m depth; and (3) installation of a horizontal strut and excavation to a 10 m depth. Additional excavation stages can readily be incorporated in the *FLAC* analysis, as required.

Figure 13.1 shows the geometry for this example. The excavation is 20 m wide and the final depth is 10 m. The diaphragm walls extend to a 30 m depth and are braced at the top by horizontal struts at a 2 m interval. The ground consists of two soil layers: a 20 m thick soft clay underlain by a stiff sand layer that extends to a great depth. The initial water table is at the ground surface. The ratio of horizontal effective stress to vertical effective stress is 0.5 at the site.

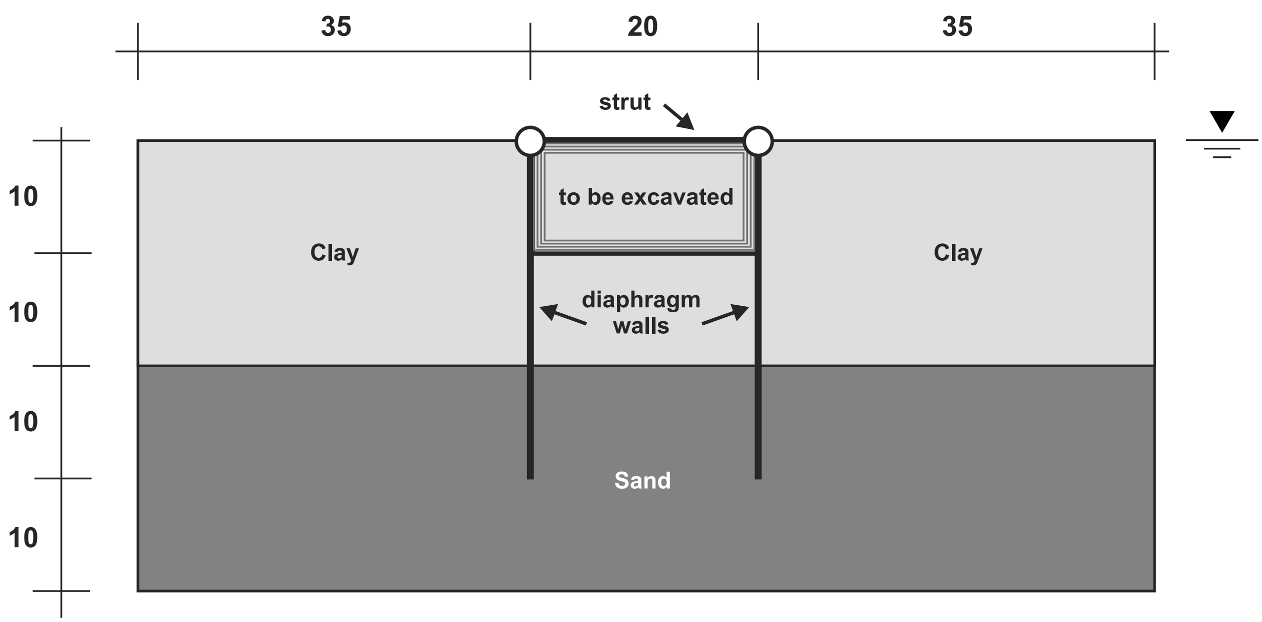

Figure 13.1 Geometry for braced excavation example

The properties selected in this example to simulate the behavior of the diaphragm wall and the struts are listed in Tables 13.1 and 13.2. The thickness of the diaphragm wall varies, and an equivalent thickness is estimated to be 1.26 m. Note that for the two-dimensional *FLAC* analysis, the Young's modulus of the wall should be divided by $(1 - \nu^2)$ to convert the plane-stress formulation for the structural elements to the plane-strain condition of a continuous wall. Thus, a value of 5.95 GPa is input to *FLAC* for the wall elastic modulus.

The strut properties are listed in Table 13.2. The spacing of the struts is 2 m. A simple way to simulate the three-dimensional effect of the strut spacing in the *FLAC* model is with linear scaling of the material properties of the struts by dividing by the strut spacing. (See Section 1.9.4 in **Structural Elements**.) For this example, by using elastic beam elements, it is only necessary to scale the elastic modulus and the density of the struts. This is done in *FLAC* automatically when the parameter **spacing** is specified.

The soil/wall interface is relatively smooth. The interface friction angle is 12.5° and the interface cohesion is 2500 Pa.

The hydraulic conductivity of the soils is assumed to be a constant value of 10^{-6} m/sec (which corresponds to approximately 10^{-10} m^2/(Pa-sec) for the mobility coefficient). The porosity of the sand and clay is assumed to be 0.3.

Table 13.1 Properties of the diaphragm wall

Equivalent thickness (m)	1.26
Density (kg/m^3)	2000
Young's modulus (GPa)	5.712
Poisson's ratio	0.2
Moment of inertia (m^4)	0.167

Table 13.2 Properties of the strut

Cross-sectional area (m^2)	1.0
Spacing (m)	2.0
Density (kg/m^3)	3000
Young's modulus (GPa)	4.0
Moment of inertia (m^4)	0.083

13.1.1 Deformational Behavior of the Soils

The deformation of the soils during excavation is of particular interest in this example, specifically the heave at the bottom of the excavation and the surface settlement adjacent to the excavation wall. Three different material models (the Mohr-Coulomb model, the Cysoil model and the PH model) are used to illustrate the effect of the material model on the calculated deformational response of the soil. It is noted that for uniform elastic properties, the linear elastic/perfectly plastic Mohr-Coulomb model may predict unrealistically large deformations in soils subjected to loading and unloading, such as heave induced at the bottom of excavations. A more realistic calculation may be obtained with the nonlinear elastic/plastic Cysoil and PH models.

The material properties chosen for this example are similar to those in Example 4 of the Plaxis Tutorial Manual (2002), which are properties for the Plaxis hardening soil model described in Section 10.7 of the Plaxis Material Models Manual (2002). The Plaxis hardening soil properties (adapted from Table 10.2 of the Plaxis Material Models Manual 2002 for this exercise) are listed in Table 13.3. The following changes were made for this exercise: values for E_{50}^{ref} are set to 25% of E_{ur}^{ref}, the cohesion property, C, is set to zero, and power m is set to 0.55.

The Mohr-Coulomb properties adapted from Example 4 of the Plaxis Tutorial Manual (2002) are drained material properties and are listed in Table 13.4. For this exercise, the cohesion property is set to zero.

The PH model properties, in general, correspond directly to the properties listed in Table 13.3. See Section 1.6.13 in **Constitutive Models** for a description of the PH model.

The formulation of the Cysoil model has components in common with the Plaxis hardening soil model. See Section 1.6.11 in **Constitutive Models** for a description of the Cysoil model. A connection, as shown in Table 13.5, is proposed between hardening Cysoil properties and Plaxis hardening soil properties (see Section 1.6.11.4 in **Constitutive Models**). Note, however, that among other things, differences exist in the hardening and dilatancy laws. (See the Plaxis Material Models Manual for a description.) Thus, the model responses should not be expected to be identical.

The Cysoil properties are listed in Table 13.6. Note that for this exercise, the bulk modulus exponent, m_k, is set to 0.99, the dimensionless parameter, α, is set to 1.0, and the over consolidation ratio, ocr, is set to 1.0 (normally consolidated material).

Both Cysoil and PH models also have stress-dependent properties. The Cysoil properties cap-pressure, p_c, mobilized friction angle, ϕ_m, upper limit of elastic shear modulus, G_{upper} and accumulated plastic shear strain, γ^p, are stress-dependent properties and are set using *FISH* function "CY.FIS" after the stress state is initialized in the model. The equations that describe the relation between each of these properties and the stress state are given in Section 1.6.11.4 in **Constitutive Models**.

The PH model properties, **sig1**, **sig2** and **sig3**, are set using *FISH* function "SETEFFSTRESS.FIS".

The in-situ stress state is initialized in the model for the Cysoil material and the PH material using *FISH* function "ININV.FIS" (see Section 13.2.2). This function is executed before "CY.FIS" when the model is composed of Cysoil material, and before "SETEFFSTRESS.FIS" when the model is composed of PH material.

Table 13.3 Plaxis hardening soil properties*

Hardening-Soil Property	Sand layer	Clay layer
Dry density, ρ (kg/m^3)	1700	1600
p_{ref} (MPa)	0.1	0.1
E_{50}^{ref} (MPa)	22.5	6.0
E_{ur}^{ref} (MPa)	90	24
E_{oed}^{ref} (MPa)	30	4
Poisson's ratio, ν_{ur}	0.2	0.2
Cohesion, C	0	0
Friction angle, ϕ (degrees)	32	25
Dilation angle, ψ (degrees)	2	0
Power, m	0.55	0.55
Failure ratio, R_f	0.9	0.9

* adapted from Table 10.12 of the Plaxis Material Models Manual (2002)

Table 13.4 Mohr-Coulomb drained properties for sand and clay layers*

	Sand layer	Clay layer
Dry density (kg/m^3)	1700	1600
Young's modulus (MPa)	40.0	10.0
Poisson's ratio	0.3	0.35
Cohesion (Pa)	0	0
Friction angle (degrees)	32	25
Dilation angle (degrees)	2	0

* adapted from Table 4.1 of the Plaxis Tutorial Manual (2002)

Table 13.5 Relation between Cysoil and Plaxis hardening soil properties

Plaxis Hardening-Soil	Hardening Cysoil
E_{50}^{ref}	–
E_{ur}^{ref}	$G_{ref}^{e} = \frac{1-2\nu_{ur}}{2(1-\nu_{ur})}\frac{E_{oed}^{ref}}{P_{ref}}$
E_{oed}^{ref}	–
–	$R = \frac{E_{ur}^{ref}}{E_{oed}^{ref}}\frac{1-\nu_{ur}}{(1+\nu_{ur})(1-2\nu_{ur})} - 1$
Cohesion, C	zero
Friction angle, ϕ	ϕ_f
Dilation angle, ψ	ψ_f
Poisson's ratio, ν_{ur}	ν_{ur}
Tensile strength	zero
Failure ratio, R_f	idem

Table 13.6 Hardening Cysoil properties

	Sand layer	Clay layer
Dry density, ρ (kg/m^3)	1700	1600
Ultimate friction angle, ϕ_f (degrees)	32	25
Ultimate dilation angle, ψ_f (degrees)	2	0
Multiplier, R	2.333	5.667
G_{ref}^{e}	112.5	45.0
Reference pressure, p_{ref} (MPa)	0.1	0.1
Poisson's ratio, ν_{ur}	0.2	0.2
Cohesion, C	0	0
Power, m_k	0.99	0.99
Failure ratio, R_f	0.9	0.9
Over consolidation ratio	1.0	1.0
Cap-yield surface parameter, α	1.0	1.0
Calibration factor, β	1.0	1.0

13.2 Modeling Procedure

A recommended procedure to simulate this type of problem with *FLAC* is illustrated by performing the analysis in six steps:

Step 1: Generate the model grid and assign material models, material properties and boundary conditions to represent the physical system.

Step 2: Determine the initial in-situ stress state of the ground prior to construction.

Step 3: Determine the initial in-situ stress state of the ground with the diaphragm wall installed.

Step 4: Lower the water level within the region to be excavated to a depth of 20 m below the ground surface.

Step 5: Excavate to a depth of 2 m.

Step 6: Install the horizontal struts at the top of the wall, and then excavate to a depth of 10 m.

When starting this project, the groundwater flow option, the adjust total stress option*, structural elements and advanced constitutive models are activated from the *Model options* dialog. The SI system of units is also selected for this example. The dialog is shown in Figure 13.2.

We set up a project file to save the model state at various stages of the simulation. We click on ? in the *Project File (*.prj)* dialog to select a directory in which to save the project file. We assign a title to our project and save the project as "EXCAVATE.PRJ". (Note that the ".PRJ" extension is assigned automatically.) This project file contains the project tree and allows direct access to all of the save (".SAV") files that we will create for the different stages of the analysis. We can stop working on the project at any stage, save it and reopen it at a later time simply by opening the project file (from the FILE / OPEN PROJECT menu item); the entire project and associated SAV files will be accessible in the graphical interface. A record of the *FLAC* commands used to create this model can also be saved separately using the FILE / EXPORT RECORD menu item.

* The automatic adjustment of total stresses for external pore-pressure change is selected because we will use this facility for the dewatering stage. See Section 1.10.7 in **Fluid-Mechanical Interaction**.

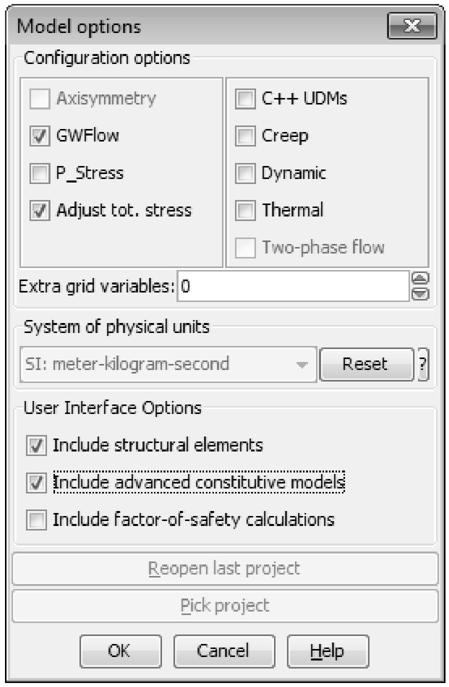

Figure 13.2 Model options selected for braced excavation example

13.2.1 Model Generation

We begin the analysis by building the model grid using the BUILD tool in the one-step (simple grid) generation mode. The braced excavation is a common form of retaining structure used in geotechnical engineering. We can find this type of geometry in the grid library available from the BUILD/GENERATE/LIBRARY tool. We click on the "Retaining wall, 2 interfaces" library item to access this grid type. The *Grid Library* dialog for this tool is shown in Figure 13.3.

We click OK to begin manipulating this grid to fit our problem geometry. Note that it is only necessary to consider half of the problem region shown in Figure 13.1 because of the symmetric geometry. The grid corners are selected to correspond to the right half of the excavation, with the origin of axes at the centerline of the excavation. The automatic zoning option is selected, and 45 zones are specified in the horizontal direction in the *Zoning Options* dialog. This produces a uniform mesh density with a zone size of 1 m. By using this library item, the wall is created automatically as a set of beam elements connected to the grid on both sides by interfaces. Note that the boundary conditions (roller boundaries on the sides and pinned boundary at the bottom) can also be selected in this tool by checking the AUTOMATIC BOUNDARY COND.? box. The *FLAC* commands are automatically generated to create this grid when the EXECUTE button is pressed. The grid created for the braced excavation example is shown in Figure 13.4. The figure also shows the location of the wall beams and identifies the two interfaces by ID numbers 1 and 2.

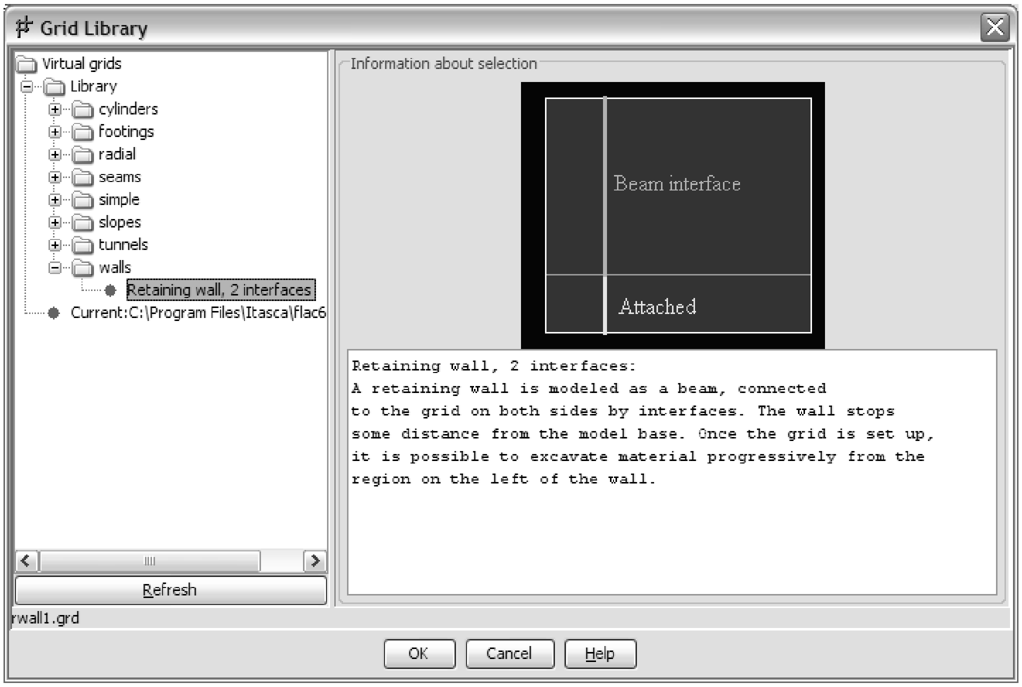

Figure 13.3 Grid Library tool for retaining wall grid

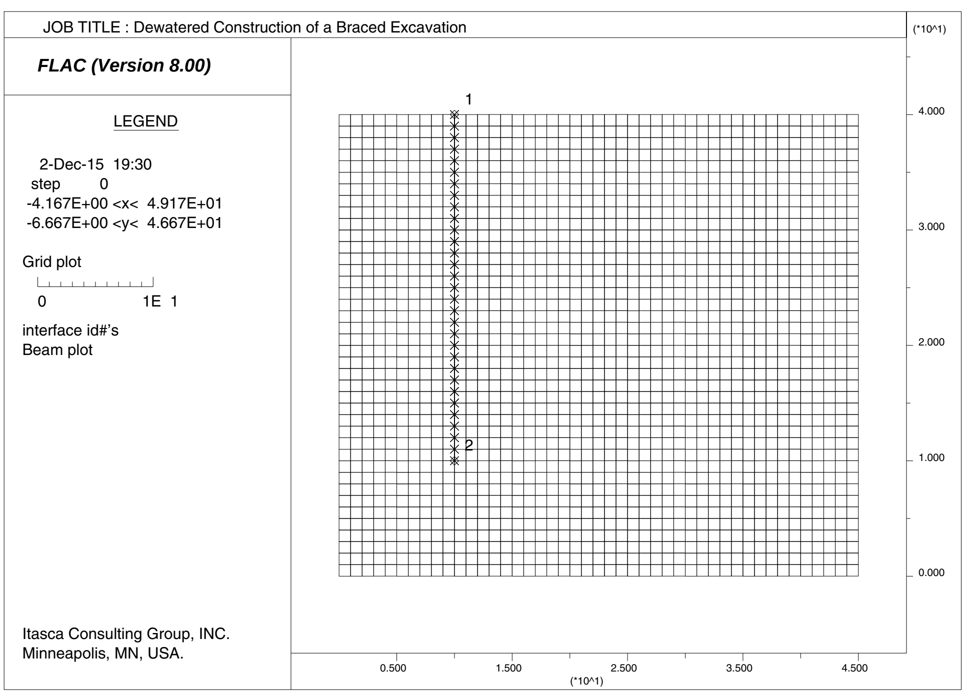

Figure 13.4 Grid created for braced excavation

We can now assign material properties for the diaphram wall and the soil/wall interface. We need to specify the material properties for the diaphragm wall because the initial model grid includes the structural elements representing the wall. We click on the STRUCTURE/SEPROP tool, and then click on one of the beam elements to open the *Beam Element Properties* dialog. The dialog is divided into two panes, as shown in Figures 13.5 and 13.6. We enter the area, Young's modulus and moment of inertia from Table 13.1. (The area is 1.26 m^2 because the two-dimensional model assumes a 1 m dimension out of the analysis plane.) We do not assign the density of the wall at this stage because we will first calculate the equilibrium stress state before the wall is constructed; this is done by neglecting the weight of the wall.

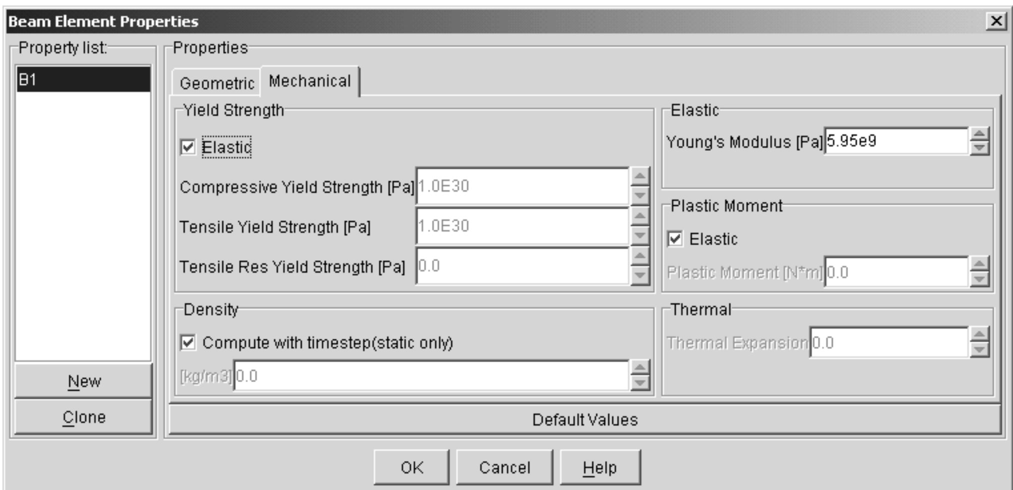

Figure 13.5 Beam properties assigned for the diaphragm wall – mechanical properties

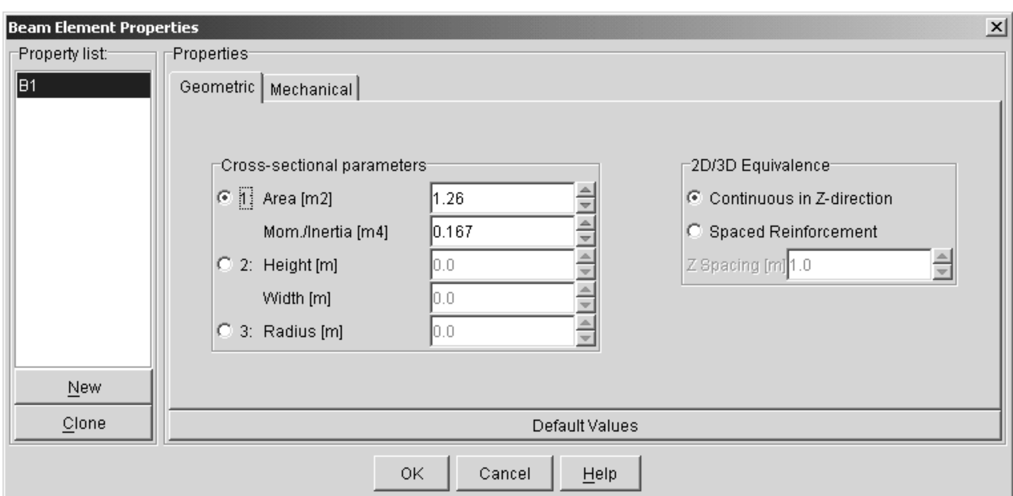

Figure 13.6 Beam properties assigned for the diaphragm wall – geometric properties

The soil/wall interface properties are prescribed using the Alter/Interface tool. We click on the Property radio button in this pane and then click on the circled number at one end of the interface highlighted in the model plot. An *Interface properties* dialog will appear, as shown in Figure 13.7. We select the Unbonded button and then enter the interface properties.

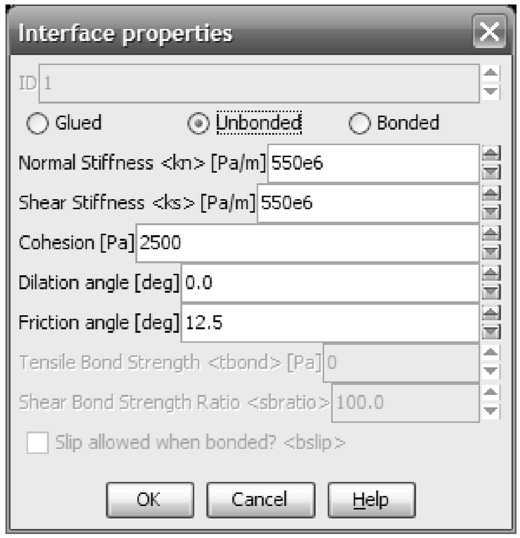

Figure 13.7 Interface properties dialog

It is usually reasonable to select the interface normal- and shear-stiffness properties such that the stiffness is approximately ten times the equivalent stiffness of the stiffest neighboring zone. By doing this, the deformability at the interface will have minimal influence on both the compliance of the total model and the calculation speed. The equivalent stiffness of a zone normal to the interface is

$$\max\left[\frac{\left(K+\frac{4}{3}G\right)}{\Delta z_{\min}}\right] \tag{13.1}$$

where K & G are the bulk and shear moduli, respectively; and
$\Delta z_{\min}$ is the smallest width of an adjoining zone in the normal direction.

The max [ ] notation indicates that the maximum value over all zones adjacent to the interface is to be used (e.g., there may be several materials adjoining the interface).

In this example, the smallest grid width adjacent to the interface is 1 m, and the maximum equivalent stiffness is approximately 55 MPa. Therefore, we select a representative value of 550 MPa/m for the normal and shear stiffnesses.

The groundwater properties porosity and permeability are assigned in the Material/GWProp tool. We click on the SetAll button to open the *Model Groundwater properties* dialog to enter these properties.

Note that the "permeability" required by *FLAC* is actually the *mobility coefficient* (i.e., the coefficient of the pore pressure term in Darcy's law – see Section 1.8.1 in **Fluid-Mechanical Interaction**). When we click OK, these properties are assigned to all zones in the model.

We specify gravity using the SETTINGS/GRAVITY tool. We select a gravitational magnitude of 10.0 m/sec^2 to simplify this example. We also assign the water density at this point using the SETTINGS/GW tool. We set the water density to 1000 kg/m^3. The fluid flow calculation mode is turned off.

We save the model state at this stage by clicking on the SAVE button at the bottom of the *Record* pane. We name the saved state "EXC_GRID.SAV"; a new "branch" with this name appears in the project tree shown in the *Record* pane. See Figure 13.8. Note that the commands associated with this branch have now been grayed out. If we find we have made a mistake or wish to modify these commands, we can press the EDIT button at the bottom of this pane. We can then edit the commands in this pane and re-execute them in *FLAC* by pressing the REBUILD button at the bottom of the pane. The state must be saved again if modifications are made.

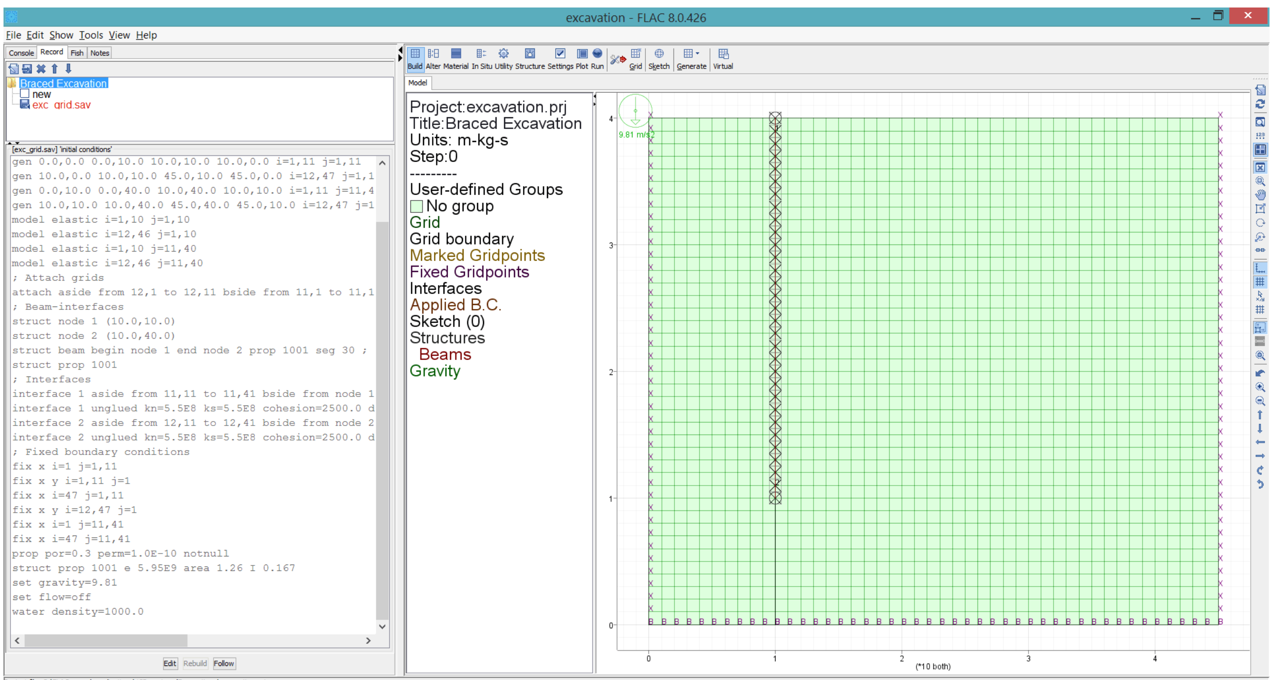

Figure 13.8 Project tree at completion of step 1

13.2.2 Soil Material Model Assignment

Three parallel branches are created in the project to evaluate the behavior of the soils. In the first branch, the clay and sand are defined as Mohr-Coulomb materials; in the second, the soils are defined as Cysoil materials; and in the third, the soils are defined as PH material.

We create the first branch by continuing from save state “EXC_GRID.SAV”. We can enter Mohr-Coulomb properties either by using the MATERIAL/ASSIGN tool or the MATERIAL/MODEL tool. For this example, we use the MATERIAL/MODEL tool because the properties of the Cysoil and PH models can only be entered with this tool. We select the Mohr-Coulomb material model from the models list, assign a group name “sand_mc,” select the rectangular range and drag the mouse over the bottom half of the model grid. The *Model mohr properties:* dialog opens, as shown in Figure 13.9. We enter the properties for sand from Table 13.4. We repeat this procedure for the clay properties with the group name “clay_mc,” and then press EXECUTE to send these commands to *FLAC*.

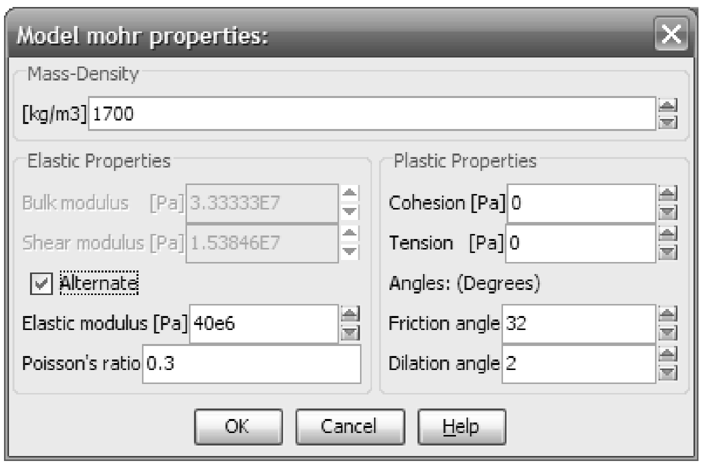

Figure 13.9 Model mohr properties dialog with sand properties

Before saving the state with Mohr-Coulomb materials, the static equilibrium is specified for the initial pre-excavation stress state with the water table at the ground surface. The pore pressure and total (and effective) stress distributions must be compatible at the initial state. This is accomplished by using the *FISH* function “ININV.FIS” provided in the *FISH* library. (See Section 3 in the ***FISH* volume**.) We call in this *FISH* function from the *FISH* library by clicking on the UTILITY/FISHLIB tool. We click on the *Library/Groundwater/ininv* menu item in the *FISH/Library* dialog, and press OK. This opens the *Fish Call Input* dialog, as shown in Figure 13.10. We enter the phreatic surface height (***wth*** = 40) and the K_o ratios (***k0x*** = 0.5 and ***k0z*** = 0.5) in the dialog, and press OK. The *FISH* function will then be called into *FLAC* and executed. The pore pressure distribution and total stress adjustment are calculated automatically. After the *FISH* function is executed, the **SOLVE elastic** command is given from the RUN/SOLVE tool. Some calculation stepping is performed to ensure the model is in force equilibrium. The initial pore pressure distribution is shown in Figure 13.11.

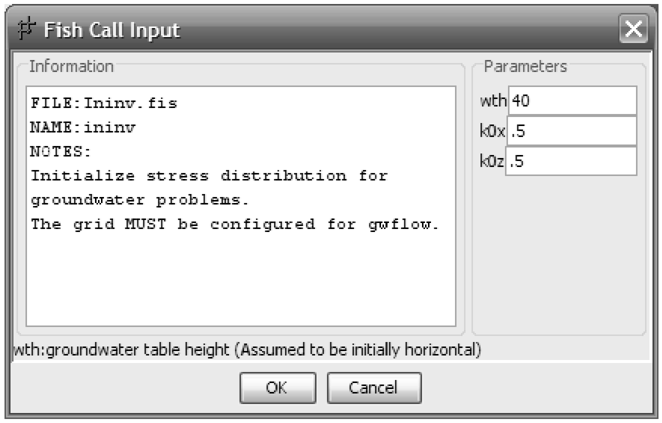

Figure 13.10 Fish Call Input input dialog

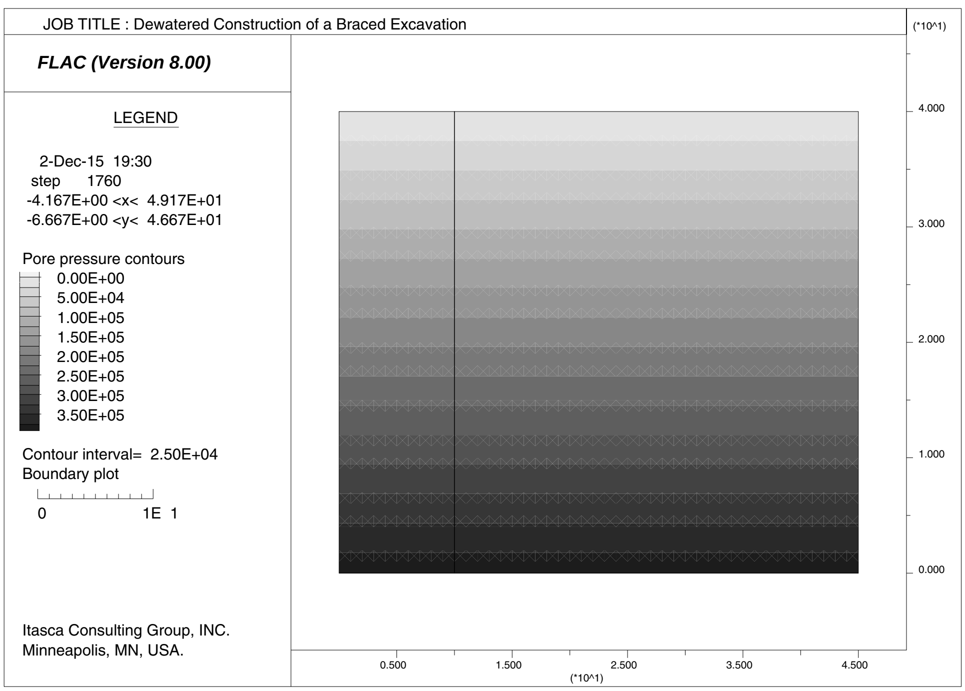

Figure 13.11 Initial pore pressure distribution – Mohr-Coulomb material

The state of the model with Mohr-Coulomb material, and at the initial stress state prior to construction, is saved as “EXC01MC.SAV”.

In order to create the second branch with Cysoil properties, we double-click the left mouse button on the branch named “EXC_GRID.SAV” in order to move back to the state before the material model is assigned. Now we reenter the MATERIAL/MODEL tool. This time we select the Cysoil material and follow the same procedure previously used to enter properties for the Mohr-Coulomb model, but now for a “sand_cy” group and a “clay_cy” group. The *Model cysoil properties* dialog opens, as shown in Figure 13.12.

The Cysoil properties given in Table 13.6 are input in this dialog. Note that when this dialog is executed, the second branch is automatically created in the project tree.

The stress-dependent properties can now be prescribed using “CY.FIS”. This function can be opened and executed from the *FISH* editor, which is accessed from the FISH tab in the *Resource* pane. Figure 13.13 displays the *FISH* editor with “CY.FIS” loaded for execution. The model equilibrium state is calculated in the same manner as for the Mohr-Coulomb material.

The model with Cysoil material and the initial stress state specified is saved as “EXC01CY.SAV”. Figure 13.14 shows the project tree at this stage.

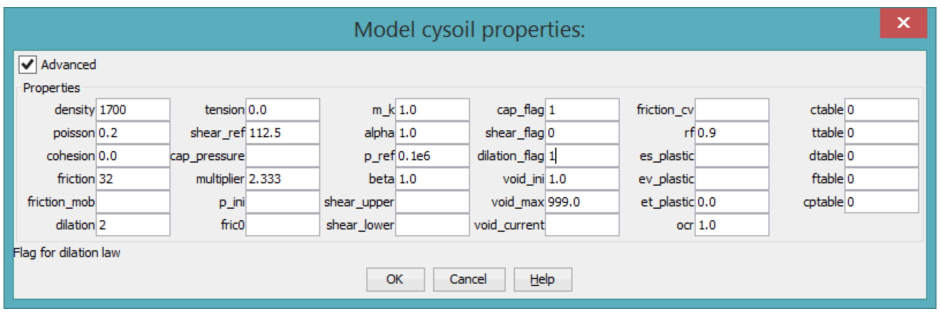

Figure 13.12 Model cysoil properties dialog with sand properties

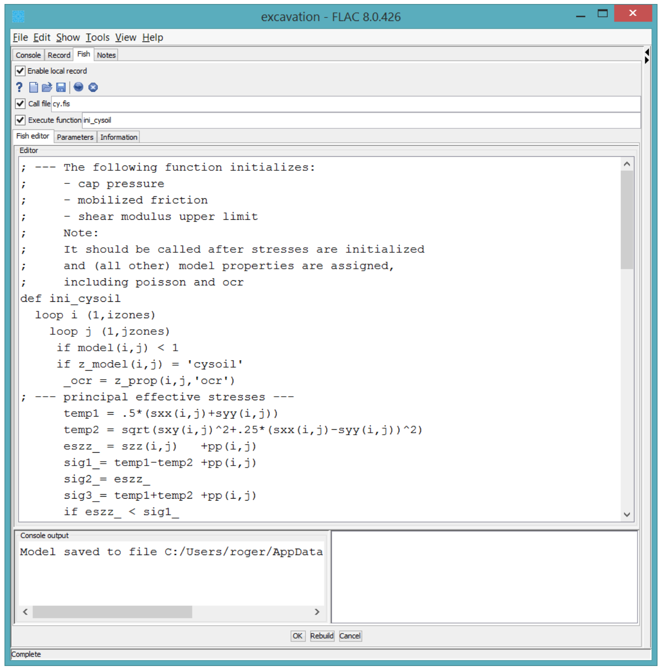

Figure 13.13 "CY.FIS" function loaded into FISH editor

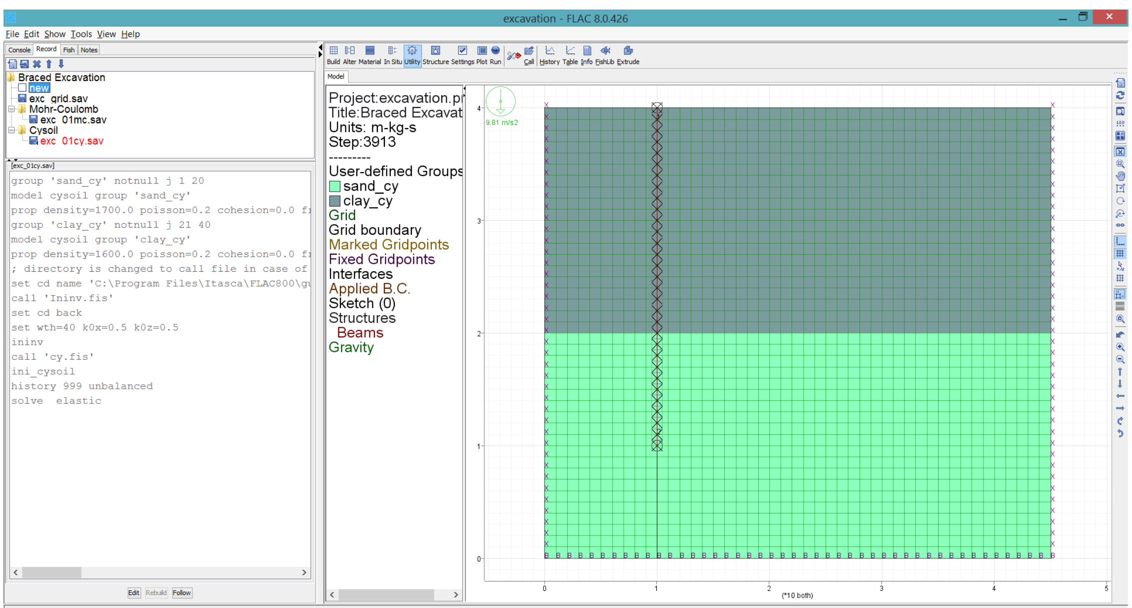

Figure 13.14 Project tree at completion of step 2 showing the Mohr-Coulomb material branch and Cysoil material branch

The same procedure is followed to create the third branch for the PH material. The PH model is also accessed from the MATERIAL/MODEL tool. The group names "sand_ph" and "clay_ph" are assigned, and the properties are input in the *Model p-hardening properties* dialog. Figure 13.15 shows the

PH model properties input in the dialog for sand_ph. Note that the cap yielding surface parameter, α, and the hardening modulus for cap pressure, Hc, are calculated internally by default. In order to provide a closer comparison to the Cysoil model, the internal calculation for the cap yielding surface parameter, α, is over-written and manually set to 1.0. The hardening modulus for cap pressure, Hc, is a function of α. Hc is recalculated for sand_ph and clay_ph using the equation in Section 1.6.13.2 in **Constitutive Models**. The commands **PROP alpha=1.0 hc=3.1e7 group 'sand_ph'** and **PROP alpha=1.0 hc=2.3e6 group 'clay_ph'** are added to overwrite the calculated properties.

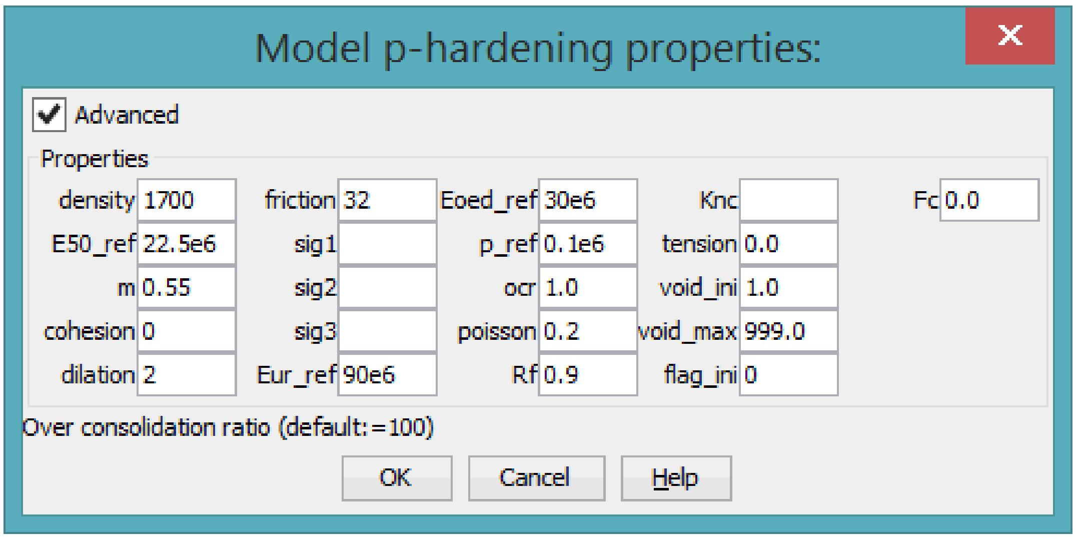

Figure 13.15 Model PH properties dialog with sand properties

The in-situ stress state is initialized using "ININV.FIS" and, this time, *FISH* function "SETEFF-STRESS.FIS" is executed from the *FISH* editor to set the stress-dependent PH properties. The state is stepped to equilibrium and saved as "EXC01PH.SAV".

13.2.3 Install Diaphragm Wall

The next stage of the analysis is the installation of the diaphragm wall. This is simulated by adding the weight of the wall in the model. We begin at "EXC01MC.SAV" for the Mohr-Coulomb material, and include the weight of the wall by specifying a mass density for the beam elements. We use the STRUCTURE/SEPROP tool to enter the density in the *Beam Elements Properties* dialog. We press RUN/SOLVE again to find the equilibrium state with the wall weight included. We save this state as "EXC02MC.SAV". Figure 13.16 shows the total vertical stress distribution at this stage.

The same procedure is repeated for the Cysoil material beginning at "EXC01CY.SAV". The total vertical stress distribution at equilibrium for this stage is shown in Figure 13.17. There is a slight difference in stress distribution around the wall in the Cysoil material compared to the wall in Mohr-Coulomb material. This can be attributed to the difference between the stress-dependent stiffness properties of the Cysoil material and the constant, uniform stiffness properties of the interfaces adjacent to the wall. Although the difference is minor, a stress-dependent variation of interface stiffnesses could be applied using *FISH* in order to provide a closer representation. The equilibrium state for Cysoil material with the wall in place is saved as "EXC02CY.SAV".

The procedure is repeated again for the PH material beginning at "EXC01PH.SAV". The vertical stress distribution is shown in Figure 13.18. The stress distribution is similar to that for the Cysoil material.

13.2.4 Dewater to a Depth of 20 m

For the dewatering stage we assume, for simplicity, that the water level is dropped instantaneously within the excavation region.* We start from "EXC02MC.SAV" and set the saturation and pore pressure to zero using the In Situ/Initial tool. We click on the *GP Info/Groundwater/saturation* menu item and drag the mouse over the gridpoints within the dewatered region ($0 \le x \le 10$, $20 \le y \le 40$). The affected gridpoints will be highlighted. We click on Assign to open the dialog to specify a zero saturation value for these gridpoints. We click OK to create this command, then click on the *GP Info/Groundwater/pp* menu item and press Assign. A dialog will open to assign a zero pore pressure to the same region. Figure 13.19 shows the In Situ/Initial tool with the affected region selected from $0 \le x \le 10$, $20 \le y \le 40$.

The saturation and pore pressure are also fixed at zero in the dewatered region. This is accomplished with the In Situ/Fix tool. The mouse is dragged over the affected gridpoints with the fixed saturation mode selected, and then with the fixed pore-pressure mode selected. Figure 13.20 shows the tool with the affected region highlighted.

The total stress is adjusted automatically when we impose this change in the pore pressures. This is a result of selecting the Adjust Tot. Stress box in the *Model options* dialog. We can check that this adjustment to total stress has been made by plotting effective stresses before and after these commands are issued; the effective stresses are unchanged in the model when the instantaneous pore-pressure change is imposed.

* For a more realistic solution, *FLAC* can calculate the gradual lowering of the phreatic surface and change of the stress state due to pumping.

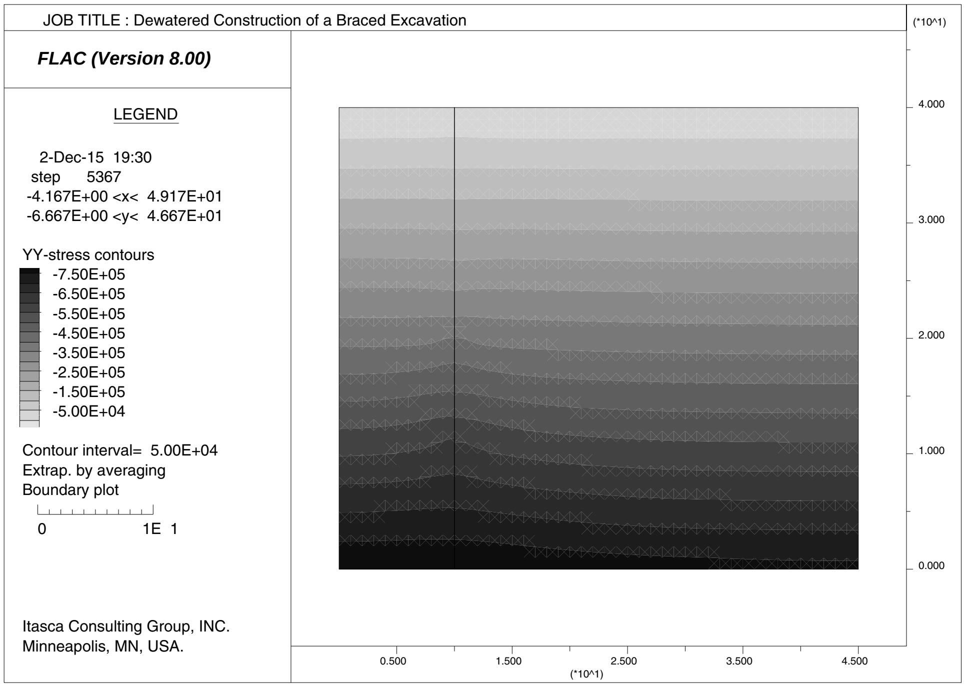

Figure 13.16 Total vertical stress contours for initial saturated state with weight of wall included – Mohr-Coulomb material

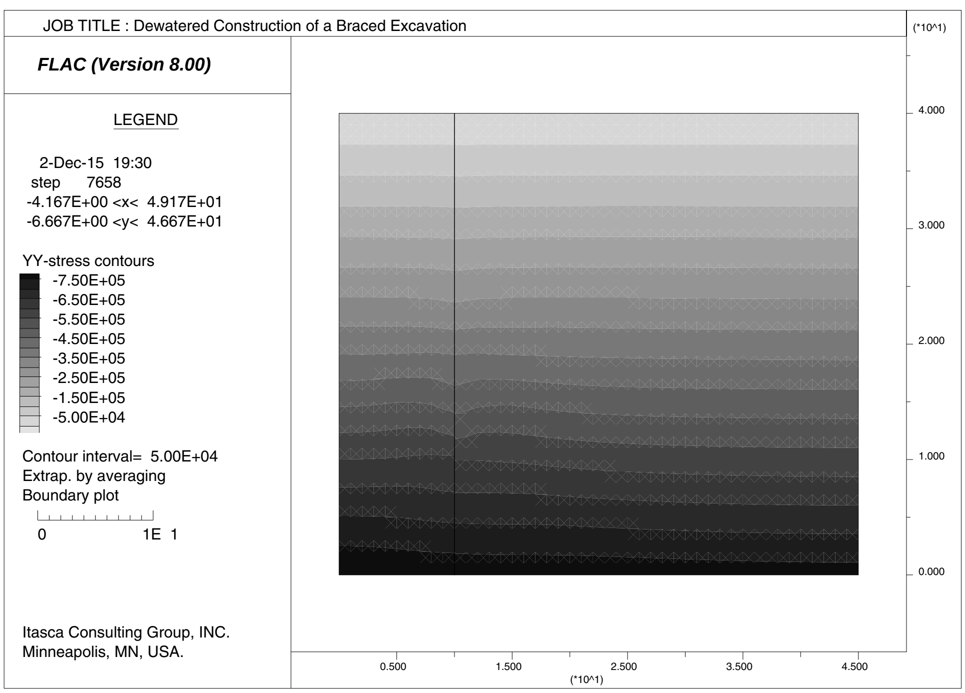

Figure 13.17 Total vertical stress contours for initial saturated state with weight of wall included – Cysoil material

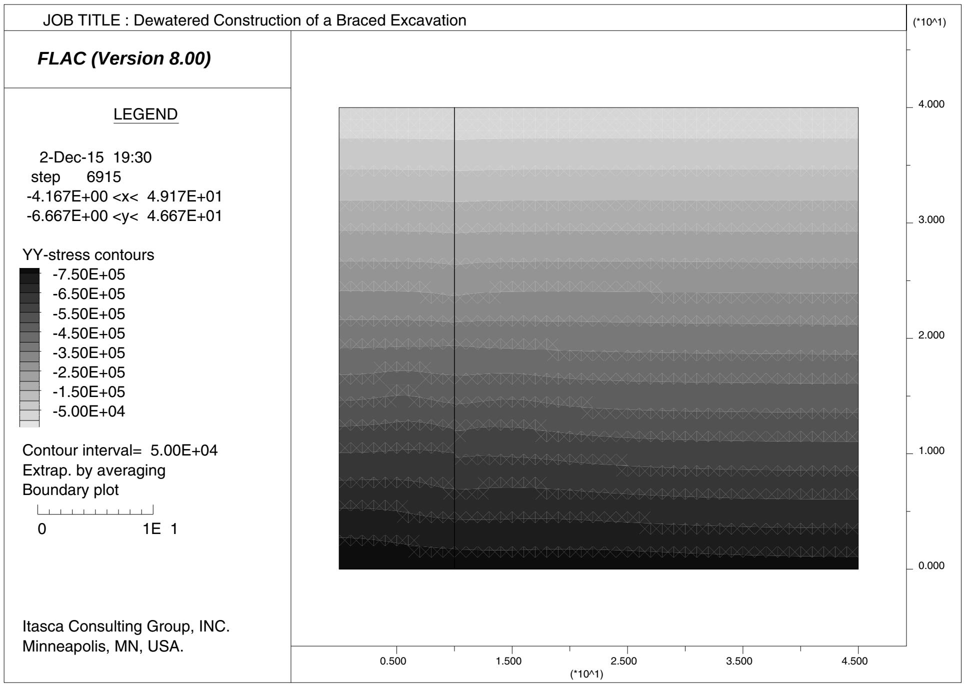

Figure 13.18 Total vertical stress contours for initial saturated state with weight of wall included – PH material

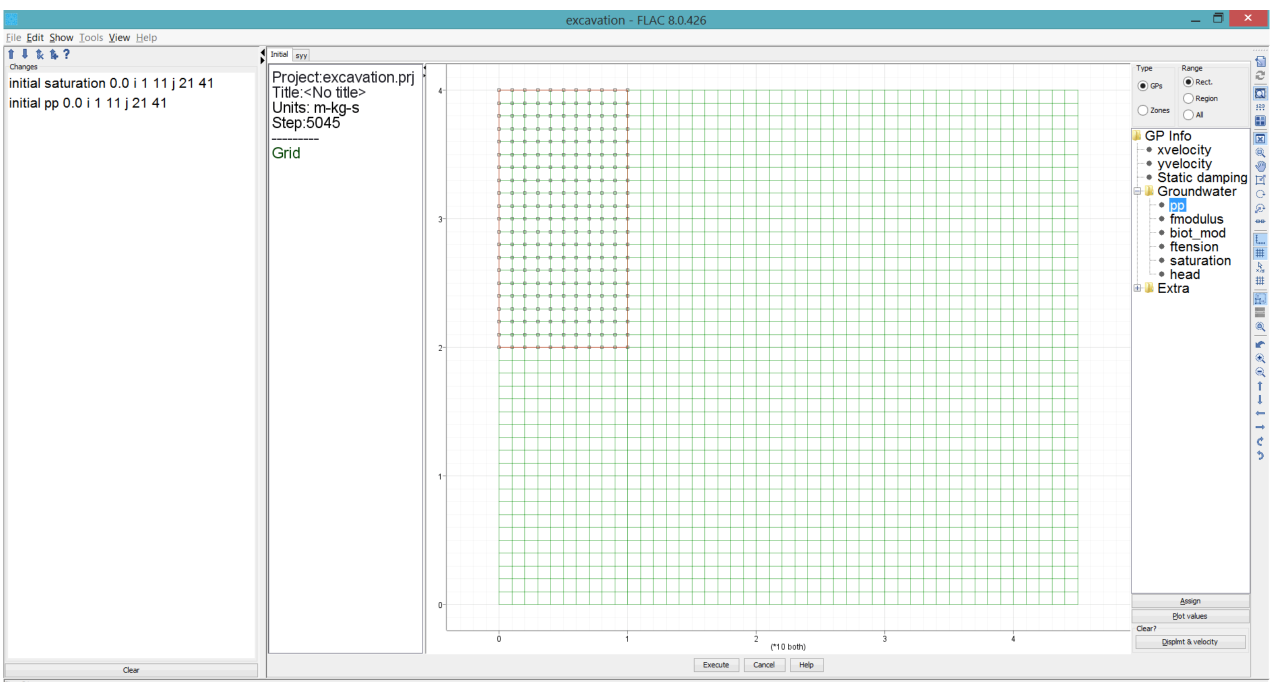

Figure 13.19 IN SITU/INITIAL *tool with saturation and pore pressure set to zero in dewatered region*

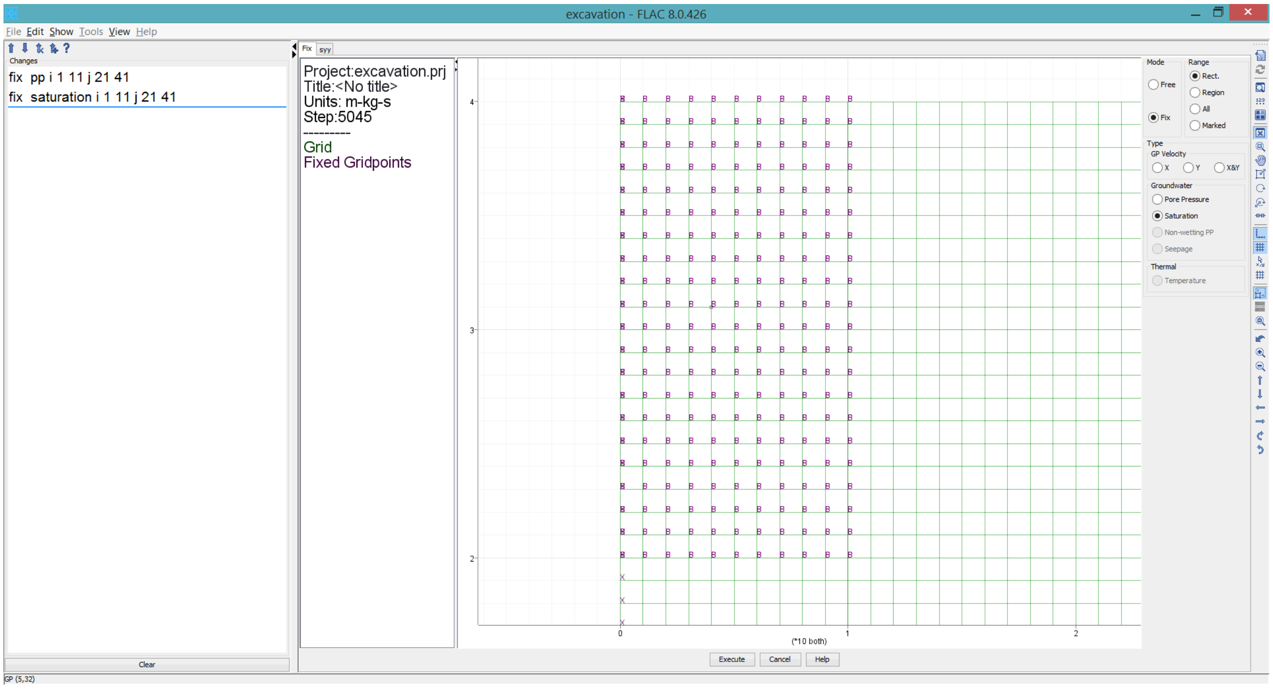

Figure 13.20 In Situ/Fix ***tool with saturation and pore pressure fixed in dewatered region***

We can now solve for the coupled response that results from the dewatering. In the Settings/GW tool, we set groundwater flow on and the water bulk modulus to 10,000 Pa. We need to specify the water bulk modulus for this calculation, so we specify this low value in order to speed convergence to steady-state flow. We can do this because we are not interested in the transient behavior. (Note that there is a lower limit for the water bulk modulus to satisfy numerical stability – see Section 1.4.2.1 in **Fluid-Mechanical Interaction**.) This is an unsaturated flow analysis, so we can also use the *fast-unsaturated flow* scheme to speed the calculation to steady state. (See Section 1.4.1 in **Fluid-Mechanical Interaction**.) We check the <FUNSAT>FAST UNSATURATED FLOW CALCULATION? box to turn on this scheme.

We use the In situ/Fix tool to fix pore pressures along the right side and saturation along the top of the model. The material outside the excavation is assumed to remain fully saturated. We initialize the displacements in the model to zero so that we can monitor the displacement change that occurs due only to the dewatering. Press the Displmt & Velocity button in the In Situ/Initial tool to initialize displacements and velocities. Then click on Run/Solve to solve for the equilibrium state with dewatering.

The steady-state pore pressure distribution after dewatering is shown in Figure 13.21. Figure 13.22 plots the displacement vectors at equilibrium. This indicates the amount of settlement induced by the dewatering: approximately 8 cm. We save the model state as “EXC03MC.SAV” for the Mohr-Coulomb material.

We repeat this dewatering procedure for the Cysoil material. The final pore pressure distribution is nearly identical to that for the Mohr-Coulomb material, as shown in Figure 13.21. The displacements induced by dewatering, however, are greater (approximately 62 cm), as shown in Figure 13.23. The model with Cysoil material is saved at this stage as “EXC03CY.SAV”.

When this stage is repeated with PH material, the pore pressure distribution is also identical to that for Mohr-Coulomb material. This time the displacements induced by dewatering reach approximately 57 cm, as shown in Figure 13.24. This stage is saved as “EX03PH.SAV”.

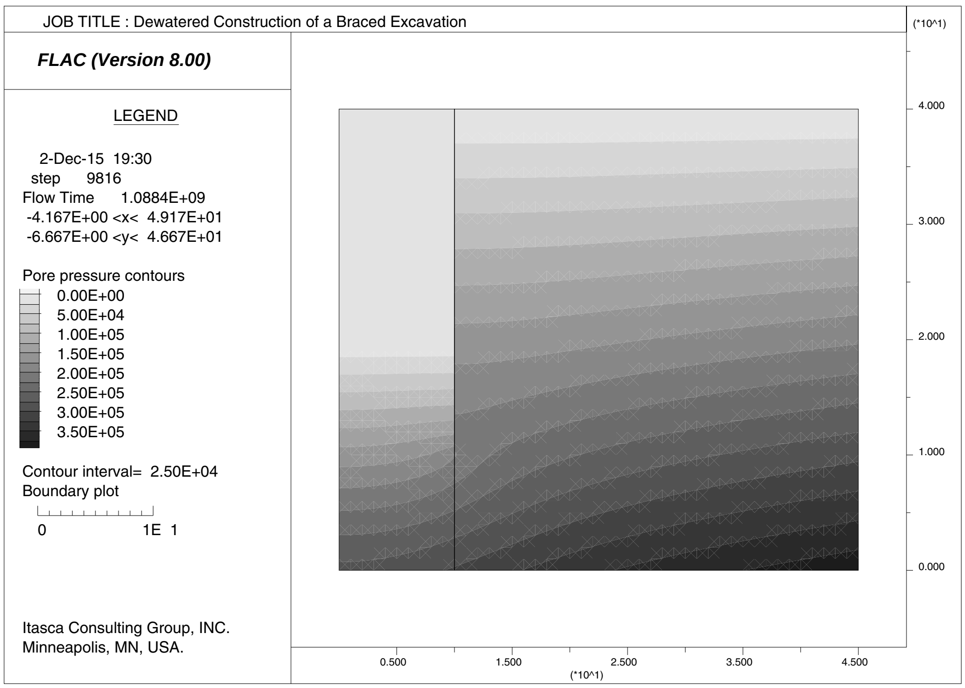

Figure 13.21 Pore pressure distribution following dewatering – Mohr-Coulomb material

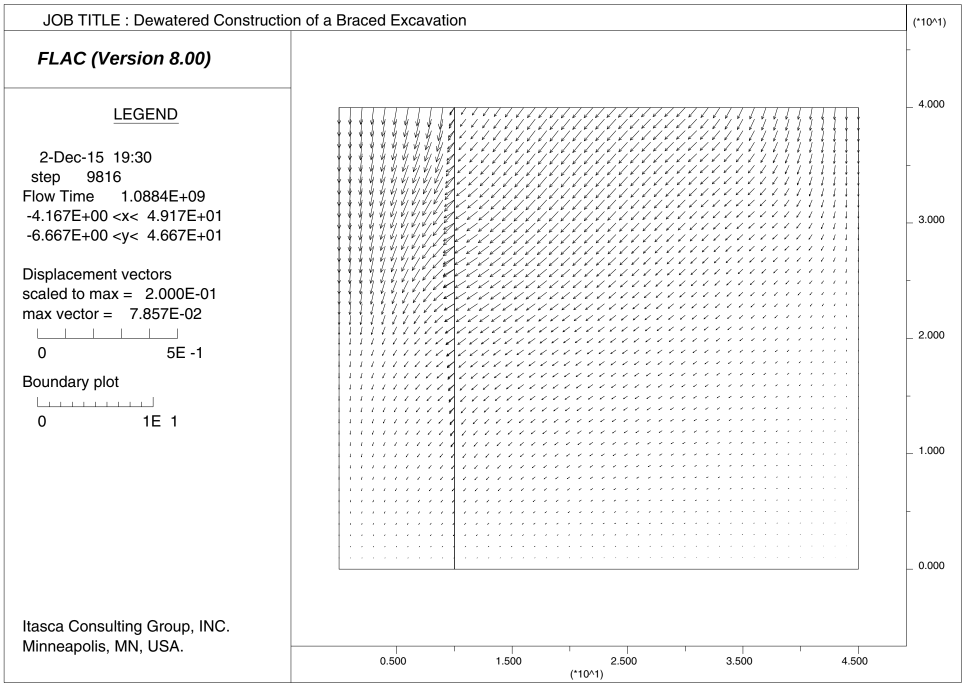

Figure 13.22 Displacements induced by dewatering – Mohr-Coulomb material

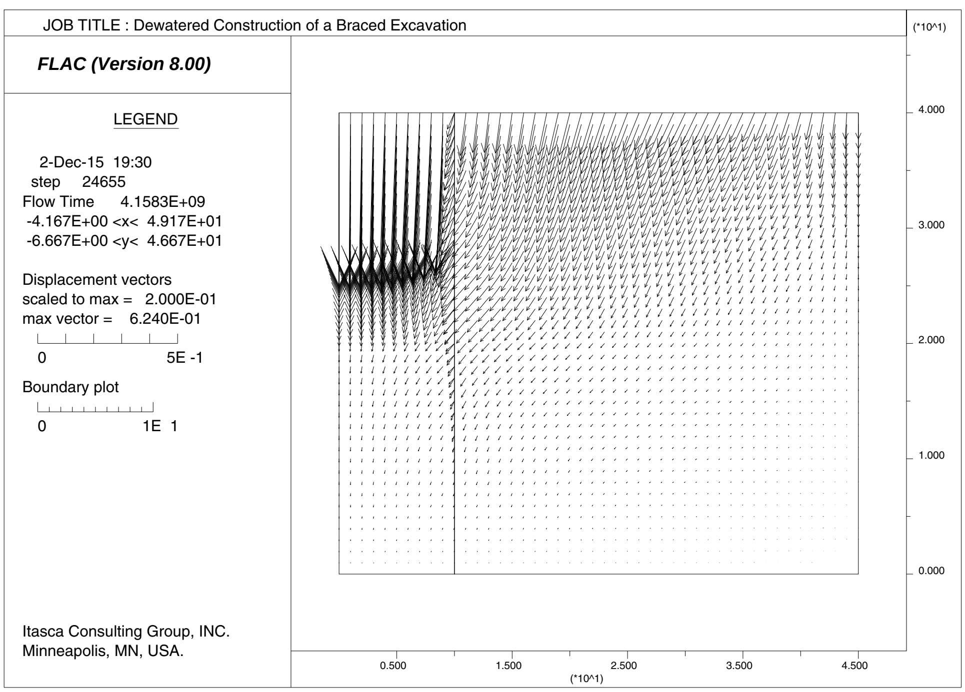

Figure 13.23 Displacements induced by dewatering – Cysoil material

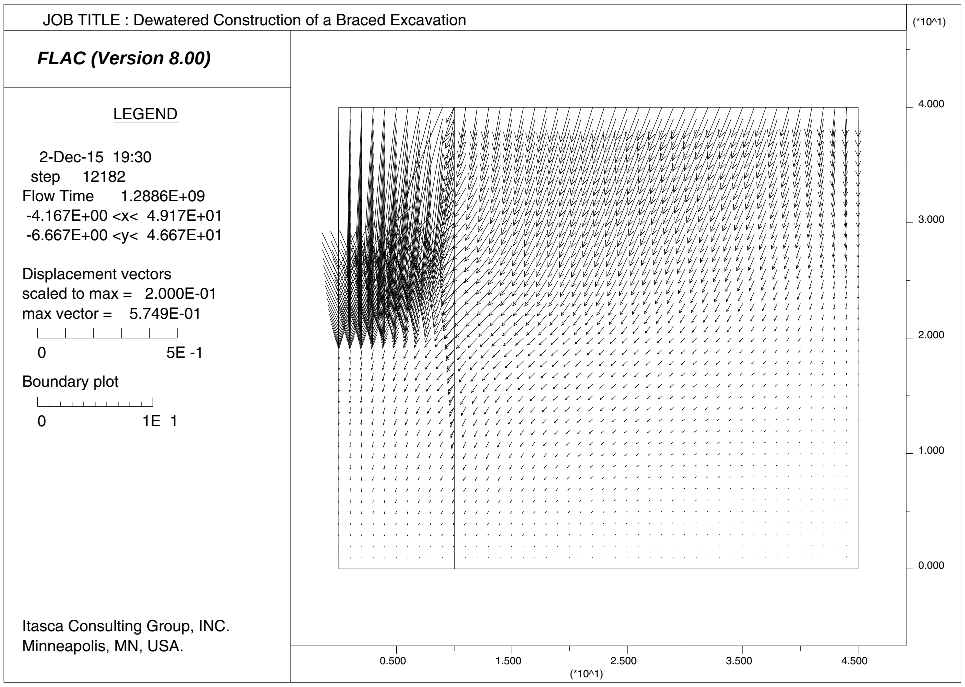

Figure 13.24 Displacemnts induced by dewatering – PH material

13.2.5 Excavate to 2 m Depth

We are now ready to begin the excavation. We start from "EXC03MC.SAV", set flow off, and set the water bulk modulus to zero for this mechanical-only calculation. We again initialize the displacements, using the DISPLMT & VELOCITY button, in order to evaluate the deformation induced by the excavation.

We use the MATERIALASSIGN tool to perform the excavation. We excavate by assigning the null model to the material to be removed. We click on the zones in the region $0 \le x \le 10$, $38 \le y \le 40$. These zones are then removed from the model plot, and the corresponding **MODEL null** commands are created for sending to *FLAC*. See Figure 13.25.

We press RUN/SOLVE to calculate the equilibrium state with this first excavation. This is the long-term response (with water bulk modulus set to zero). We save this state as "EXC04MC.SAV".

The displacements induced by this excavation are illustrated in Figure 13.26. A maximum heave of roughly 4.2 cm occurs at the bottom of the excavation. We can also calculate the response of the wall. For example, the moment distribution in the wall after the first excavation is shown in Figure 13.29. Note that various results for the wall response (e.g., wall displacements, axial forces, shear forces) can be plotted using the *Plot items* dialog in the PLOT/MODEL tool.

This step is now repeated with Cysoil material starting from "EXC03CY.SAV". The displacements induced by this excavation stage are plotted in Figure 13.27 and indicate a maximum heave of roughly 2.2 cm. The moment distribution for this case is plotted in Figure 13.30; the maximum value is approximately 1.8 times that for the wall in Mohr-Coulomb material.

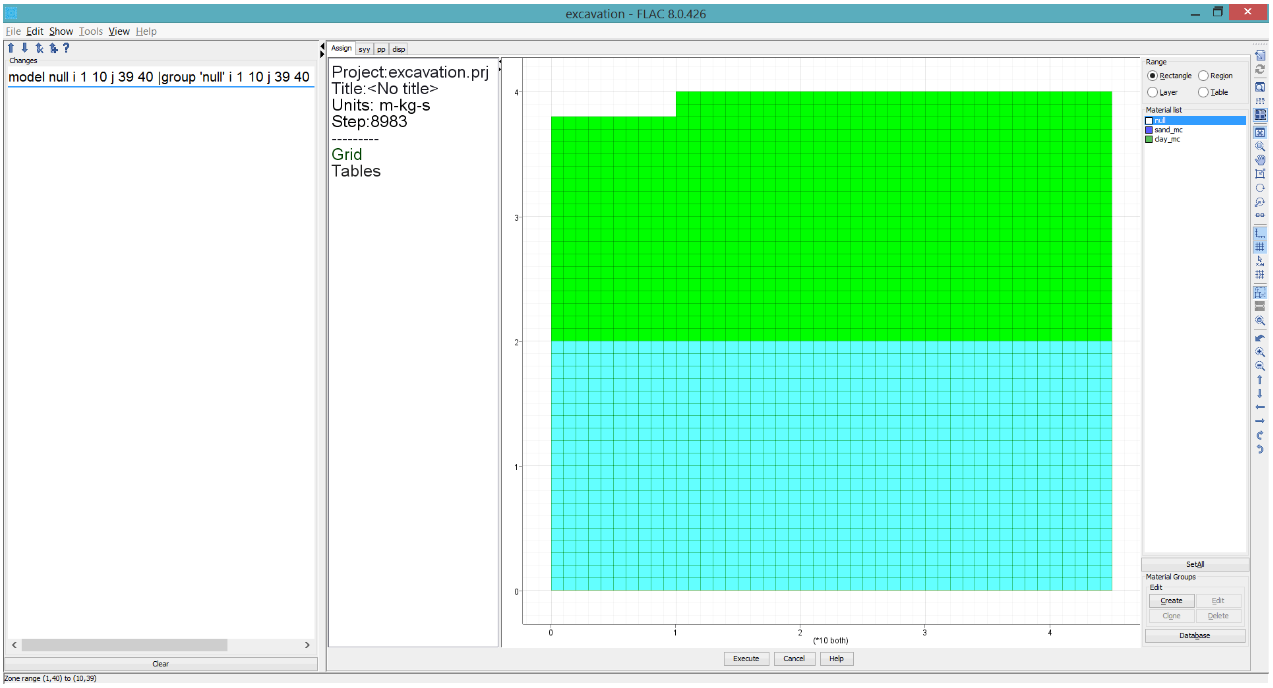

Figure 13.25 Excavated zones in the MATERIAL ASSIGN *tool*

This step is repeated with the PH material starting with “EX04PH.SAV”. The displacements are plotted in Figure 13.28 and the wall moments in Figure 13.31.

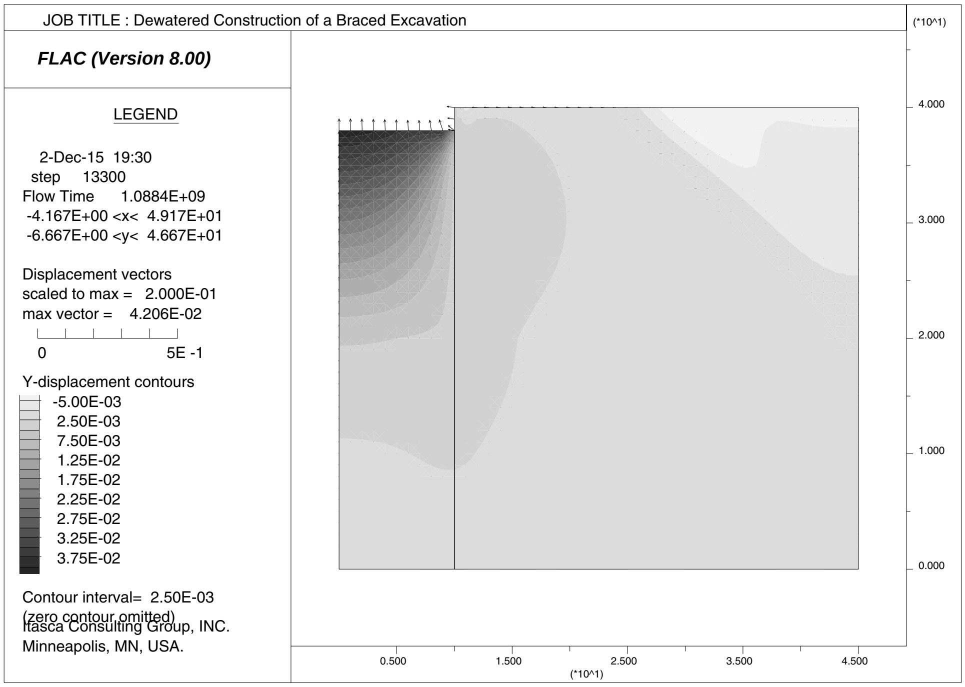

Figure 13.26 Displacements induced by excavation to 2 m depth – Mohr-Coulomb material

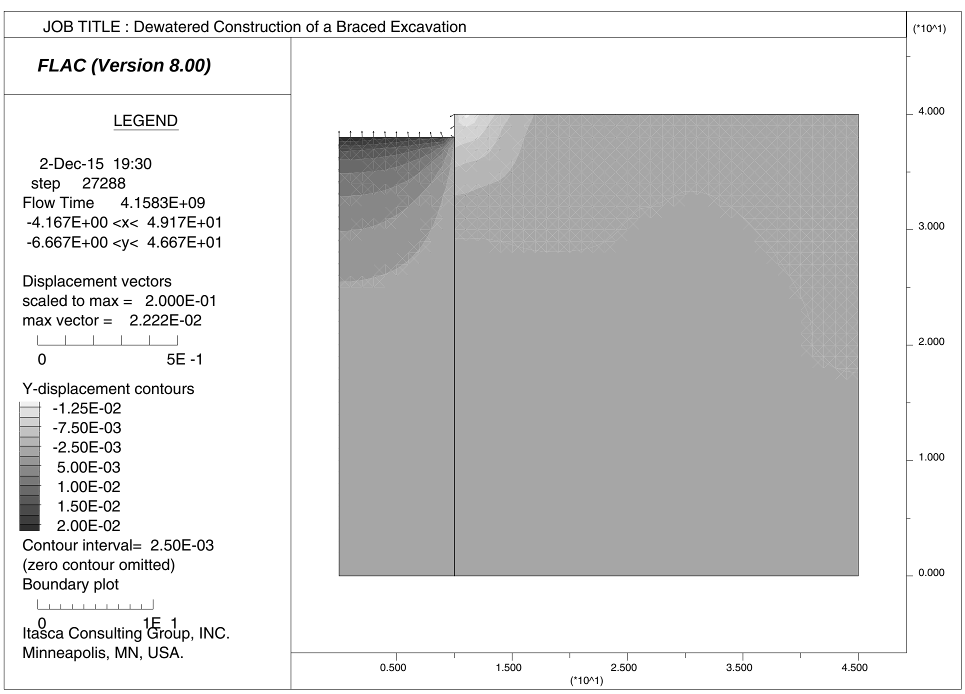

Figure 13.27 Displacements induced by excavation to 2 m depth – Cysoil material

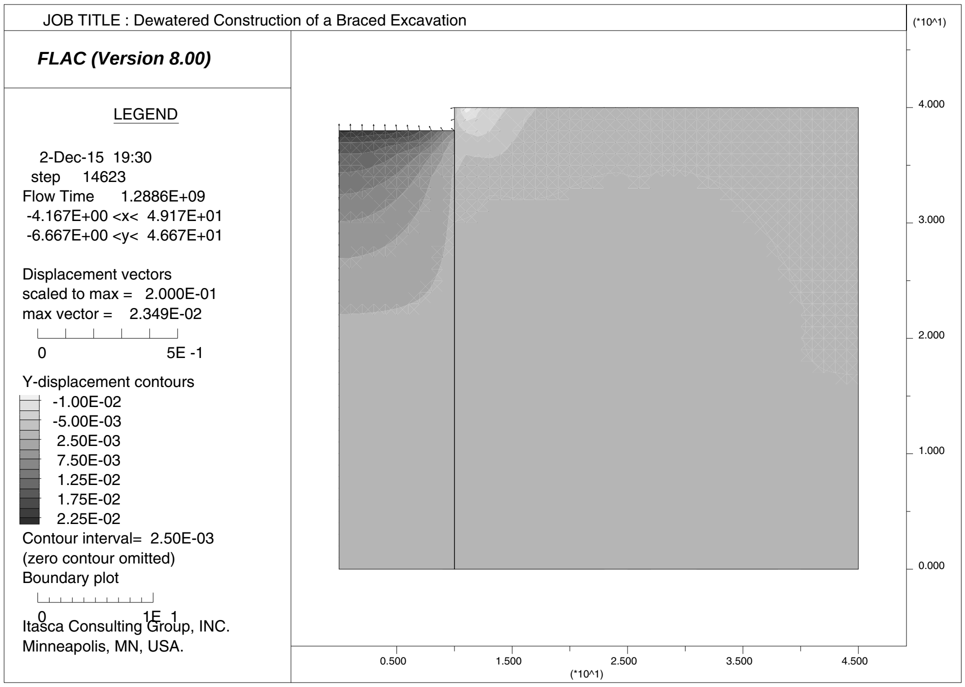

Figure 13.28 Displacements induced by excavation to 2 m depth – PH material

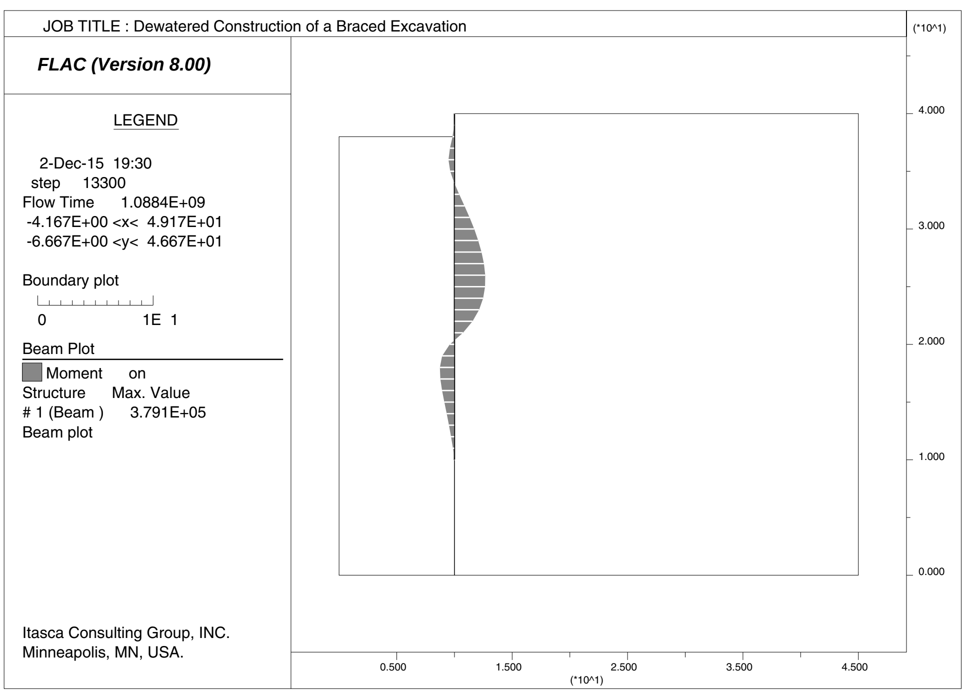

Figure 13.29 Moment distribution in wall after excavation to 2 m depth – Mohr-Coulomb material

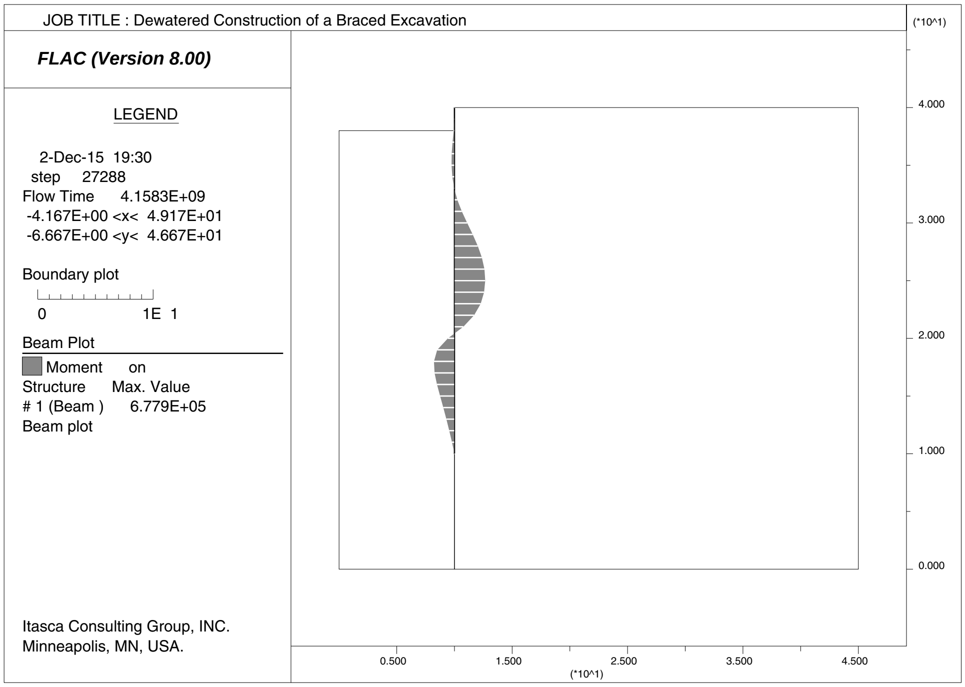

Figure 13.30 Moment distribution in wall after excavation to 2 m depth – Cysoil material

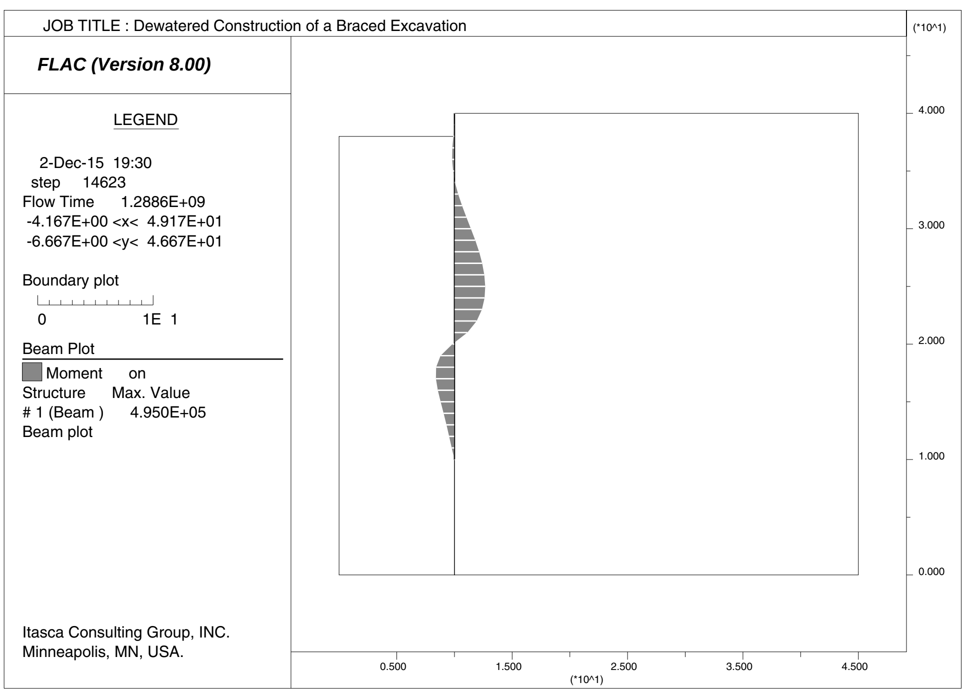

Figure 13.31 Moment distribution in wall after excavation to 2 m depth – PH material

13.2.6 Install Strut and Excavate to 10 m Depth

For the final excavation step, we install a horizontal strut at the top of the wall and then excavate to a 10 m depth. We use the STRUCTUREBEAM tool to install the strut. We press the ADD radio button in the *Modes* menu, then move the mouse on the model view to one end position of the strut, and hold the left button and move the mouse to the other end position. A line indicating the location of the strut will be drawn. We can position the strut more precisely by right-clicking the mouse over each end location. A dialog opens and we enter the endpoint coordinates. Note that the left node is free. We position the right node at the same location as the top node of the wall, and consequently, the right node is slaved to the existing node of the wall.* Figure 13.32 shows the position of the strut in the STRUCTUREBEAM tool.

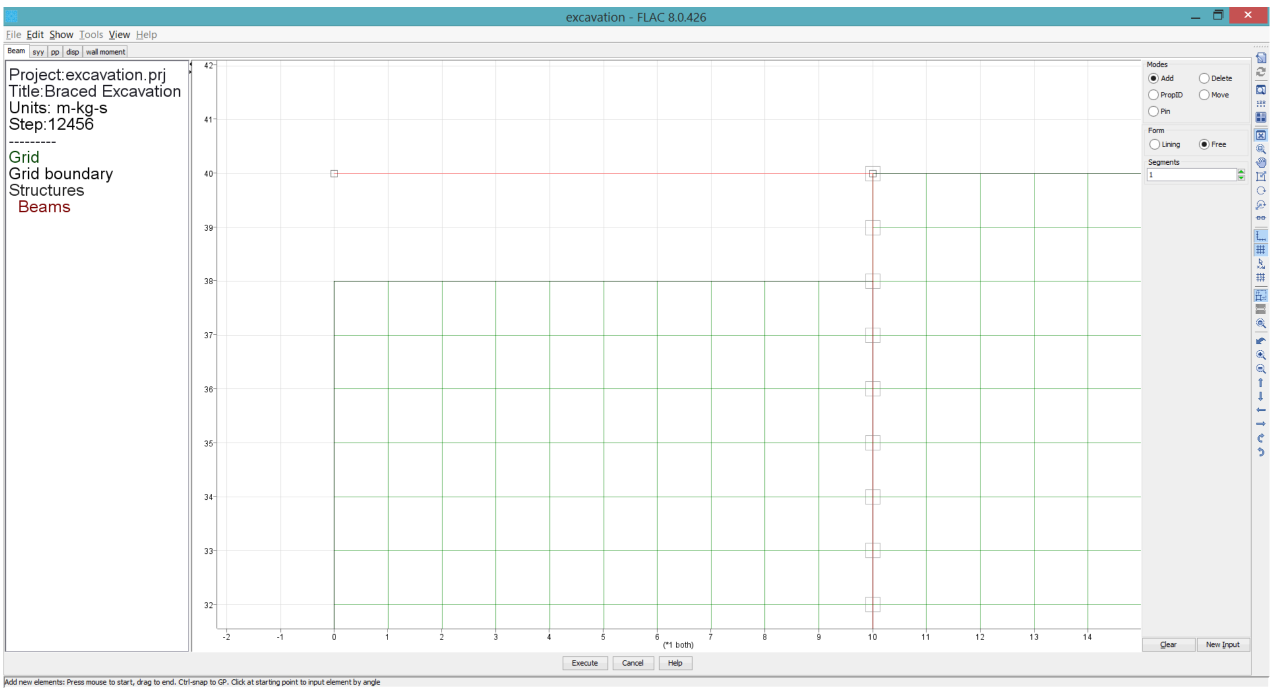

Figure 13.32 Positioning the horizontal strut using the STRUCTUREBEAM tool

The strut is not rigidly connected to the wall in this exercise. We define a pin connection (which permits free rotation at the strut/wall connection) by selecting the PIN radio button in the *Modes* list and clicking the mouse over the connecting node. An arrow is drawn at the node, denoting this as a pin connection (see Figure 13.33).

* When the mouse is positioned to create a new node at the same location as an existing node, the new node is automatically slaved to the existing node. If two separate (non-slaved) nodes are required at the same position, first offset the mouse slightly to create the new node, and then reposition the new node at the same location as the existing node using the MOVE mode.

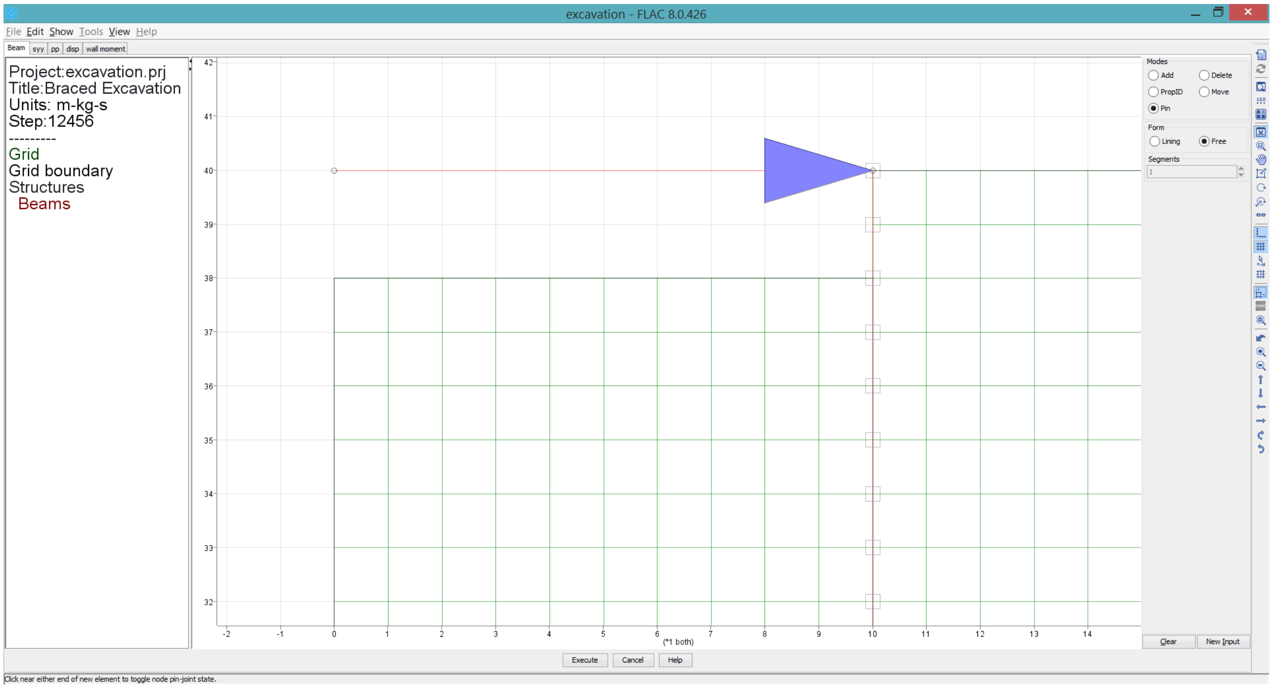

Figure 13.33 Selecting a pin connection in the STRUCTURE BEAM *tool*

We also prescribe a different material property number to the strut in the BEAM tool so that we can assign the strut properties. We click on the PROPID radio button in the *Modes* list, and the identification number *B1* appears over the beam elements in the model plot. We click on the strut element, and a dialog opens to allow us to rename the property ID to *B2*.

We now press EXECUTE to send these commands to *FLAC* to create the strut, pin the strut to the wall and assign the property number. Two nodes (32 and 33) are created, connected as a single beam element and assigned property number 1002. A pin connection is defined between node 33 and wall node 2.

We enter the STRUCTURE/NODE tool to assign fixity conditions for the strut. Node 33 is located along the centerline of the excavation. We click on this node to open a *Node:33* dialog, as shown in Figure 13.34. We fix this node from movement in the x-direction, and from rotating (which are appropriate conditions for a node located along a line of symmetry), by clicking on the X-VELOCITY and ROTATION check boxes in the dialog. We click OK and then EXECUTE to send the node condition commands to *FLAC*.

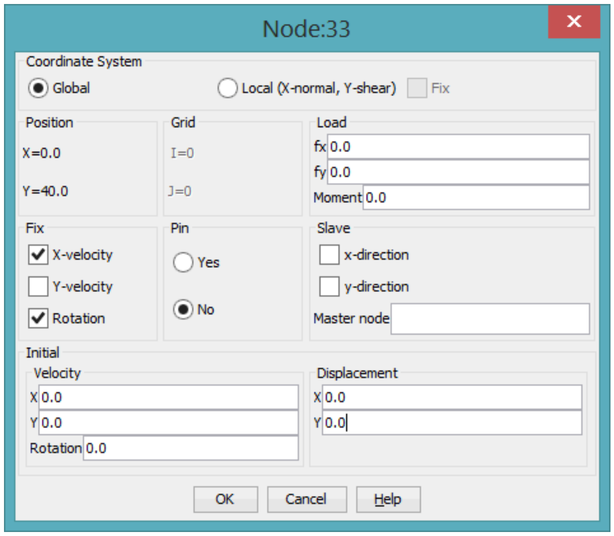

Figure 13.34 Node 33 dialog in the* STRUCTURE NODE *tool

We assign the strut properties using the STRUCTURE/SEPROP tool. We click on the strut element in this tool and open the *Beam Element Properties* dialog (as we did previously for the wall properties) to enter the strut properties as listed in Table 13.2.

We are now ready to perform the second excavation step. We use the MATERIAL ASSIGN tool and change the zones within the range $0 \le x \le 10$, $30 \le y \le 38$ to null material. We press RUN/SOLVE to calculate the equilibrium state with this second excavation. We save this state as “EXC05MC.SAV”.

The total displacements induced by the excavation to the 10 m depth is illustrated in Figure 13.35; the moment distribution in the wall and axial force in the strut are shown in Figure 13.38.

This stage is repeated for Cysoil material and for PH material. The total displacements are plotted in Figure 13.36 for Cysoil material and in Figure 13.37 for PH material for comparison to the Mohr-Coulomb material. The maximum displacement at the bottom of the excavation is greater for the Mohr-Coulomb material than for both Cysoil and PH materials. Also, the extent of the excavation-induced displacement is more confined than that for the Mohr-Coulomb material; compare the y-displacement contour plot in Figures 13.36 and 13.37 to that in Figure 13.35.

The wall moments and strut load for Cysoil material are plotted in Figure 13.39 and for the PH material in Figure 13.40. The values are roughly the same as those for Mohr-Coulomb material.

The surface settlement and the wall deflection are also compared for the three soil types. Figure 13.41 plots surface settlement and shows roughly the same settlement profiles for all three material models. Figure 13.42 plots the wall deflection. The Mohr-Coulomb material results in higher deflections. Deflections resulting from Cysoil and PH materials match quite closely.

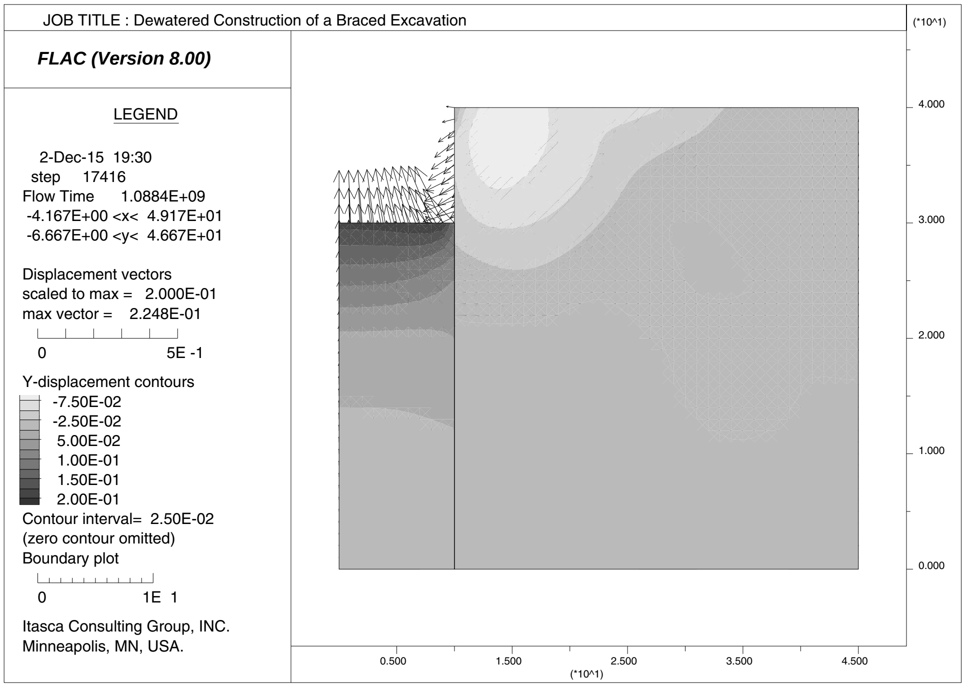

Figure 13.35 Displacements induced by excavation to 10 m depth – Mohr-Coulomb material

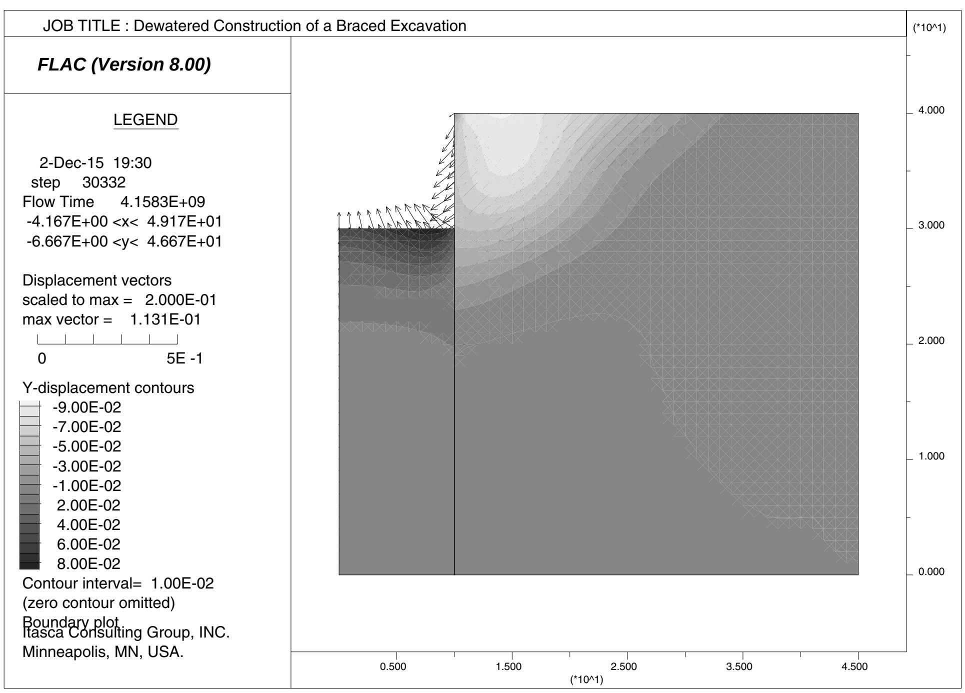

Figure 13.36 Displacements induced by excavation to 10 m depth – Cysoil material

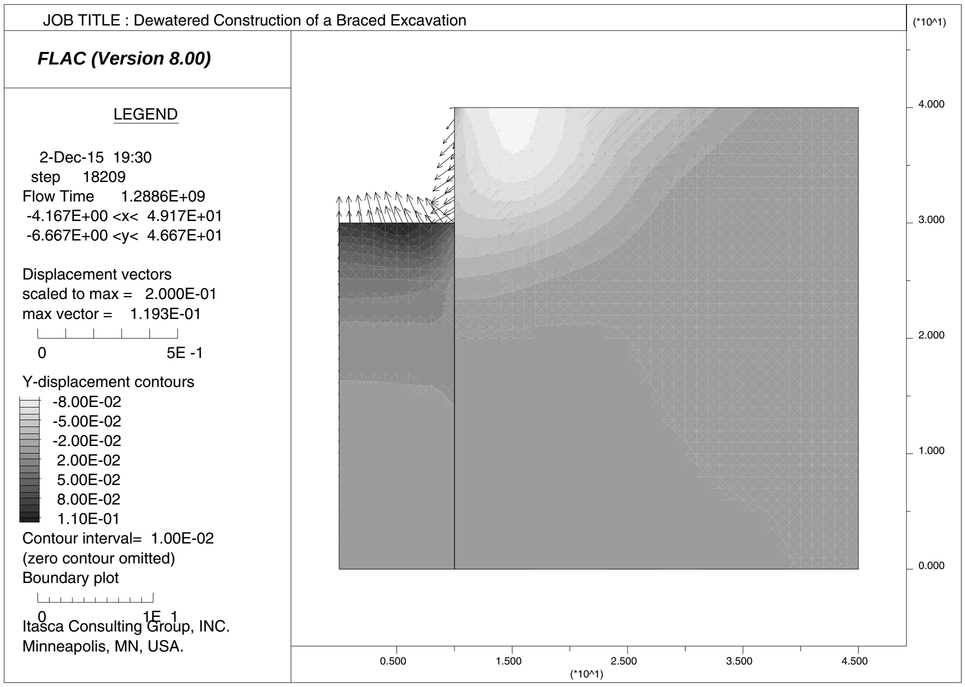

Figure 13.37 Displacements induced by excavation to 10 m depth – PH material

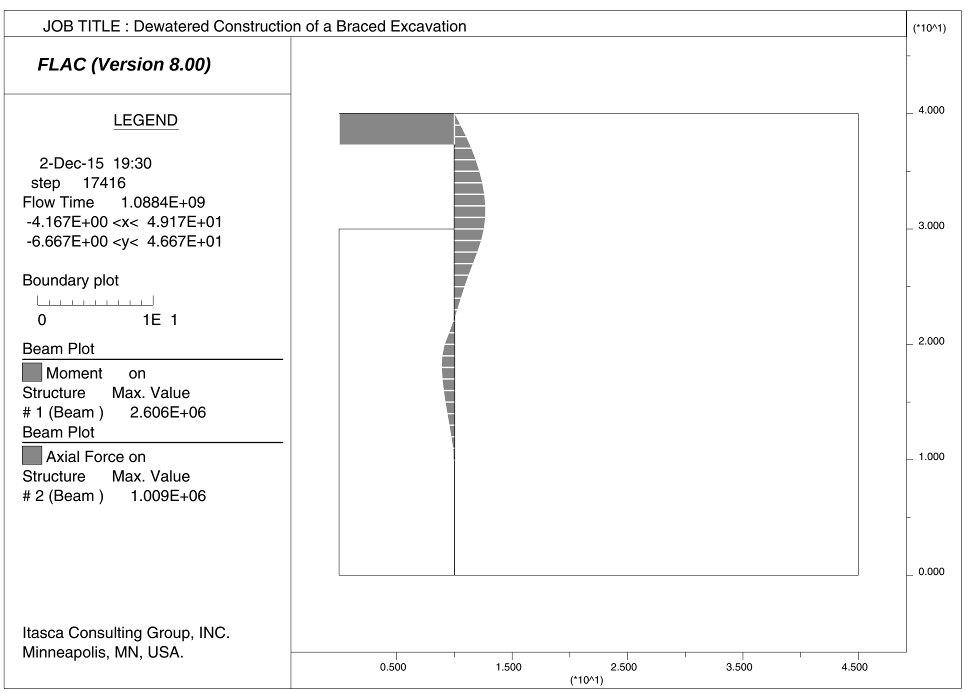

Figure 13.38 Moment distribution in wall and axial force in strut after excavation to 10 m depth – Mohr-Coulomb material

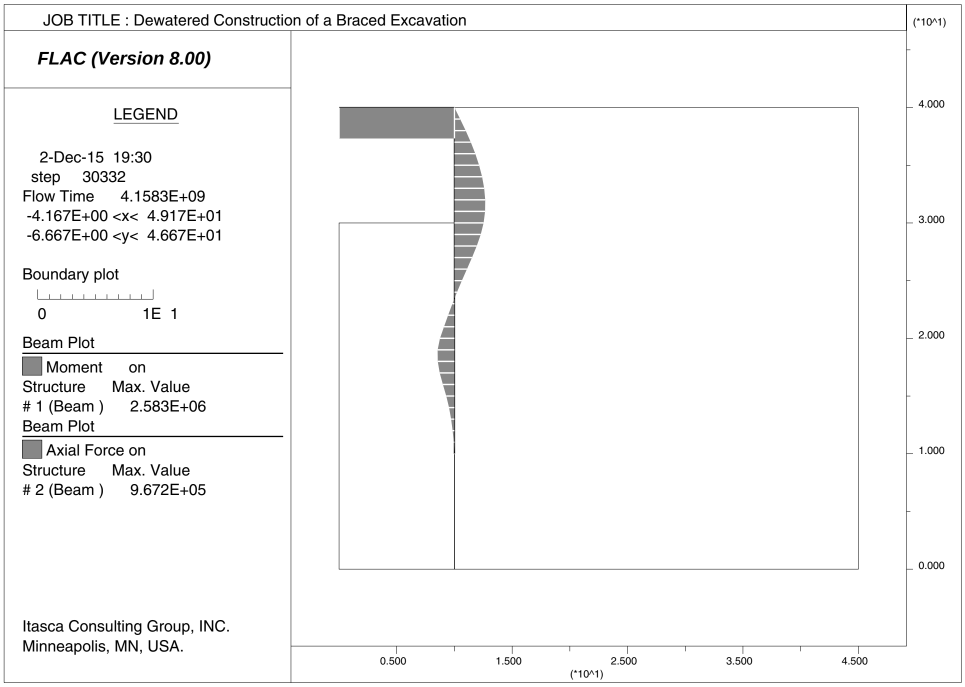

Figure 13.39 Moment distribution in wall and axial force in strut after excavation to 10 m depth – Cysoil material

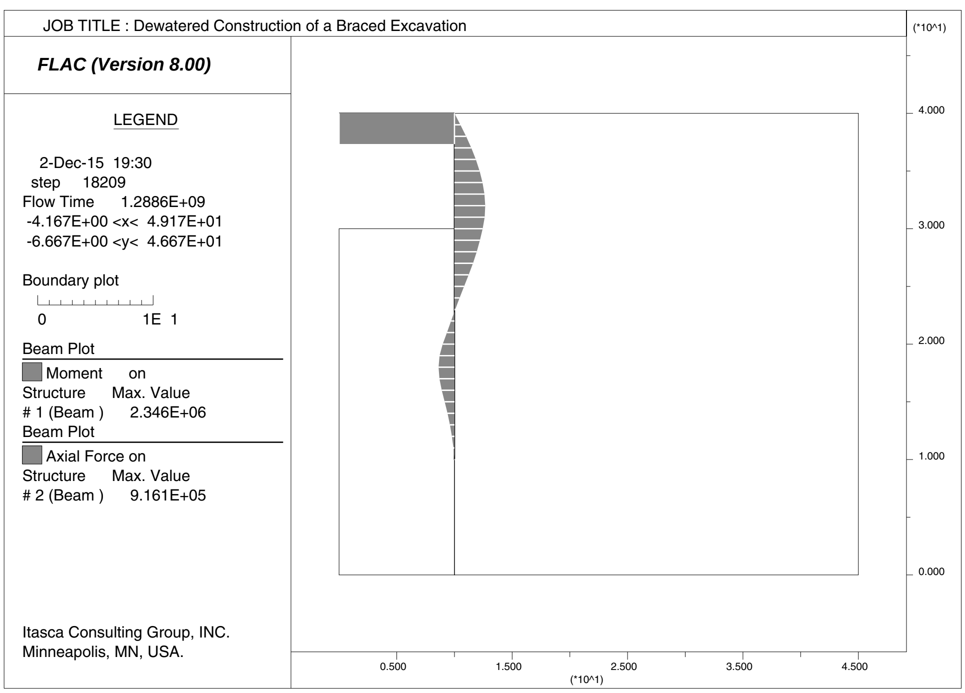

Figure 13.40 Moment distribution in wall and axial force in strut after excavation to 10 m depth – PH material

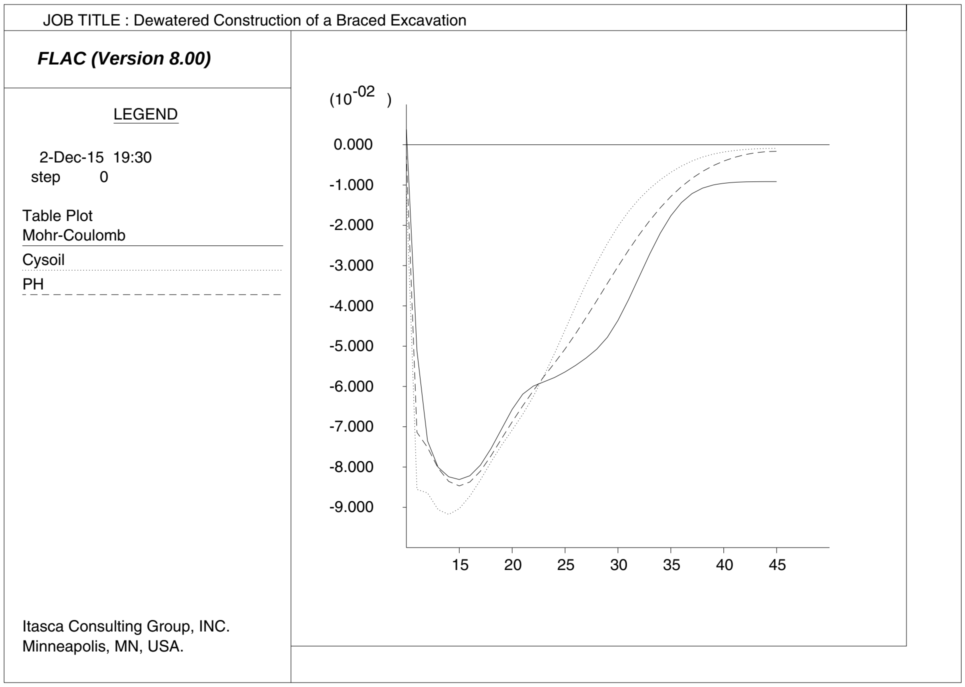

Figure 13.41 Surface settlement profiles

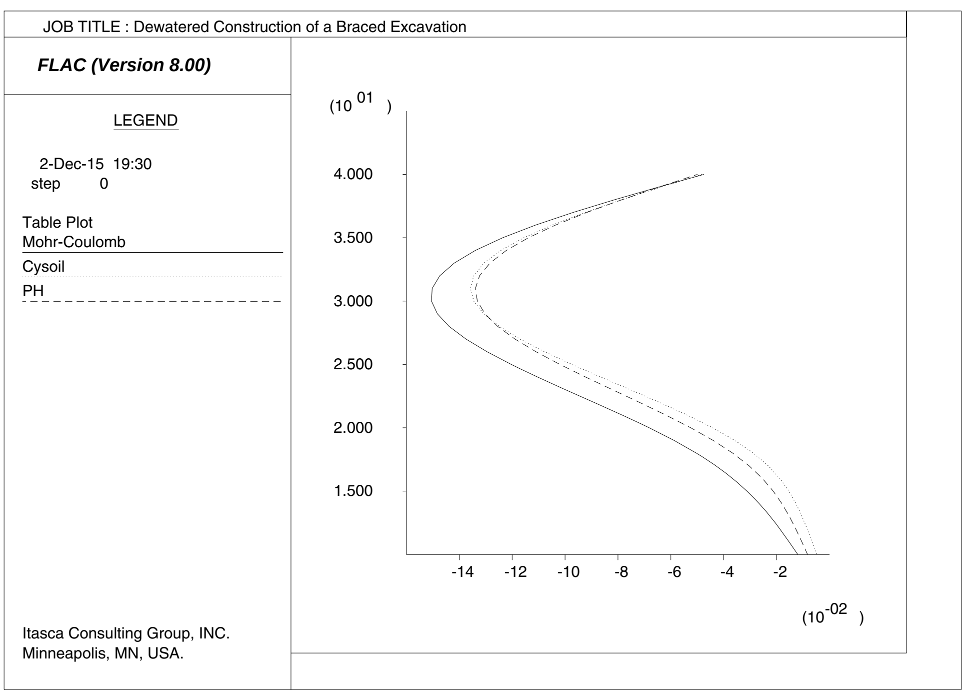

Figure 13.42 Wall deflection

13.3 Observations

This example illustrates the effect of the material model on soil deformational response for unloading problems such as the construction on a braced excavation. The heave that occurs at the bottom of the excavation is considerably greater for the construction in Mohr-Coulomb material than it is for Cysoil material or PH material. This is primarily attributed to the stress-dependent elastic moduli and stiffer unloading response of the Cysoil and PH materials. This is evident from the comparison of the displacement contour plots in Figure 13.35 for the Mohr-Coulomb material, and Figure 13.36 for the Cysoil material and Figure 13.37 for PH material.

This model example was derived for comparison to the excavation example with Mohr-Coulomb material and hardening soil material given in the Plaxis Material Models Manual (2002). A qualitative agreement with those results is shown here. This example illustrates a procedure to match properties between the Cysoil and PH materials. Also, see Section 17 for a comparison between the Cysoil model, the PH model and the hardening soil model in a field benchmark study.

13.4 References

Plaxis BV. ***PLAXIS Version 8, Material Models Manual***. R.B.J. Brinkgreve, ed. Delft: Plaxis (2002).

Plaxis BV. ***PLAXIS Version 8, Tutorial Manual***. R.B.J. Brinkgreve, ed. Delft: Plaxis (2002).