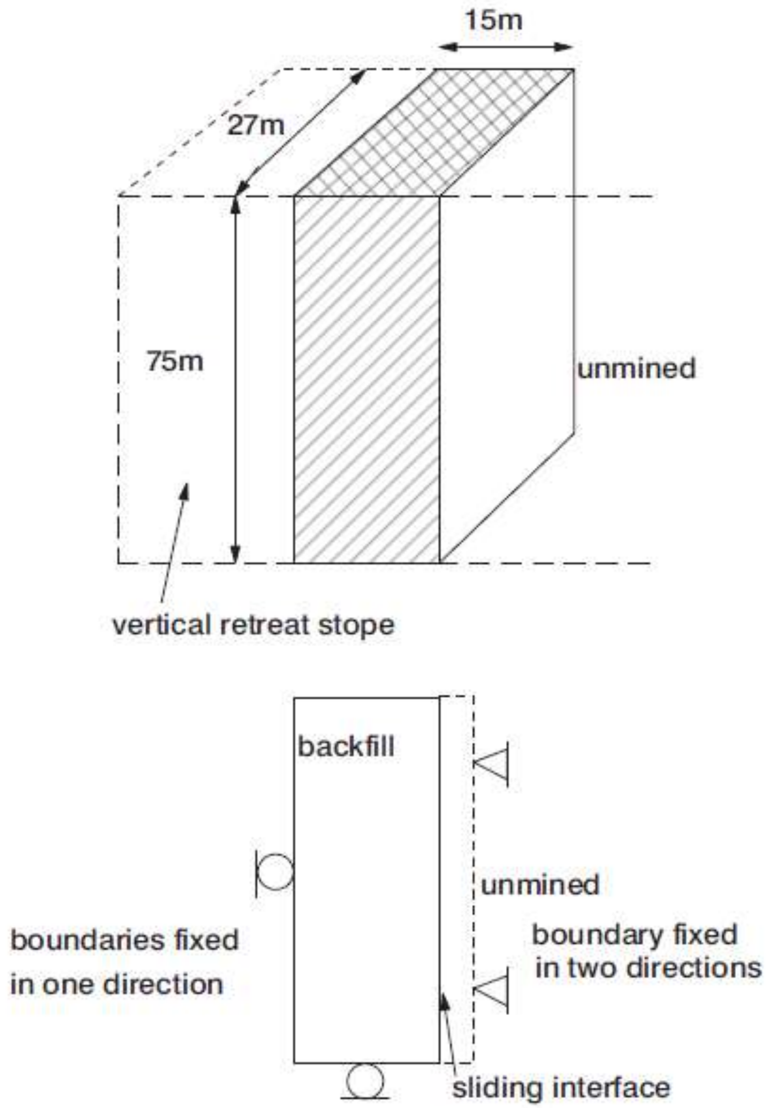
Example Application 3

Cemented Backfill Pillar Performance



Schematic illustrating true three-dimensional backfill pillar geometry and two-dimensional representation

Rock Properties – Mohr Coulomb Model

density	2700 kg/m^3
bulk modulus	30.56 GPa
shear modulus	22.92 GPa
friction angle	35.0^o
cohesion	10.0 MPa
tension limit	1.0 MPa

Backfill Pillar Properties – Mohr Coulomb Model

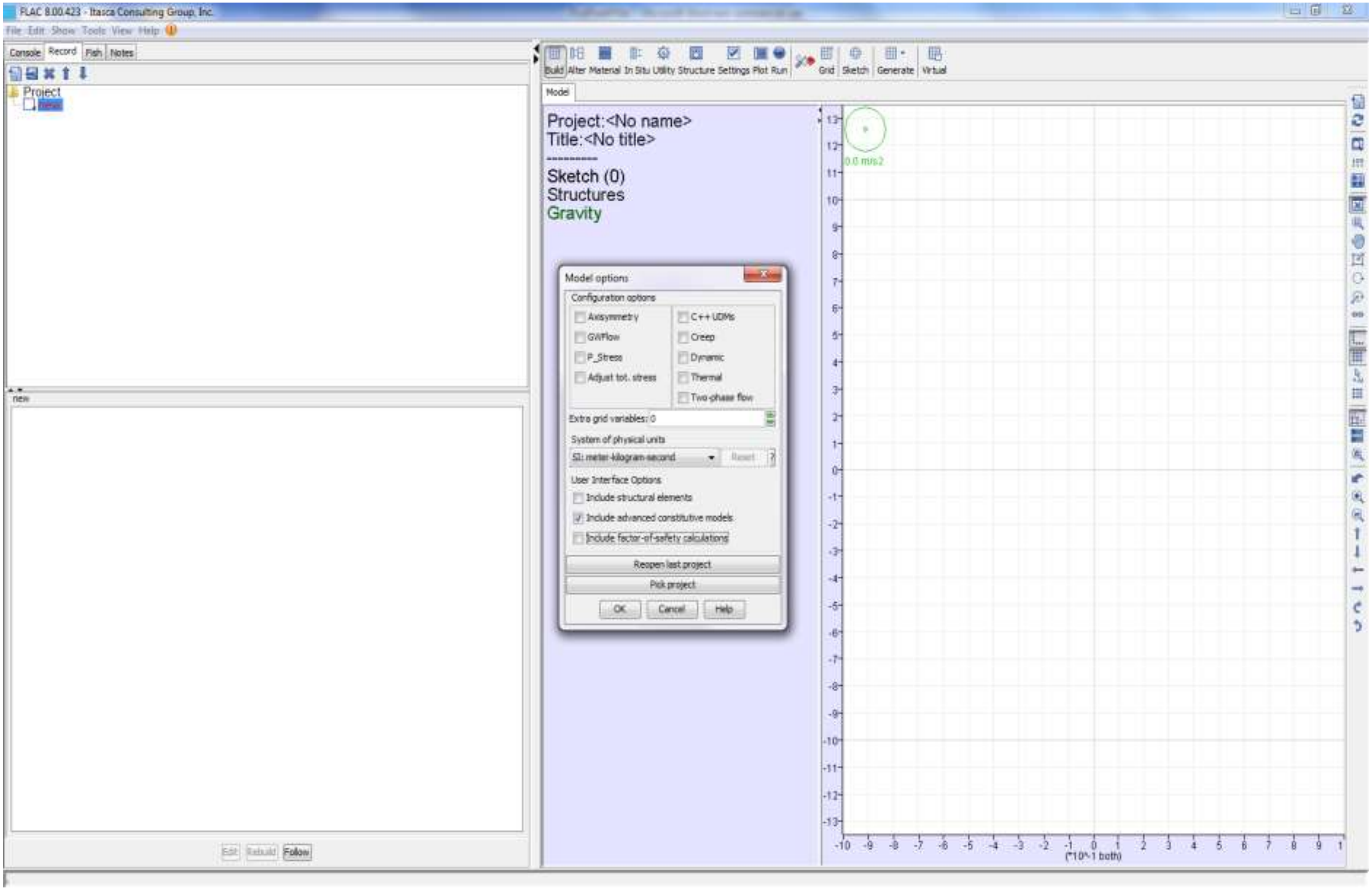
density	2100 kg/m^3
bulk modulus	110.0 MPa
shear modulus	37.0 MPa
friction angle	35.0^o
cohesion	0.1 MPa
tension limit	0.14 MPa

Modeling Procedure

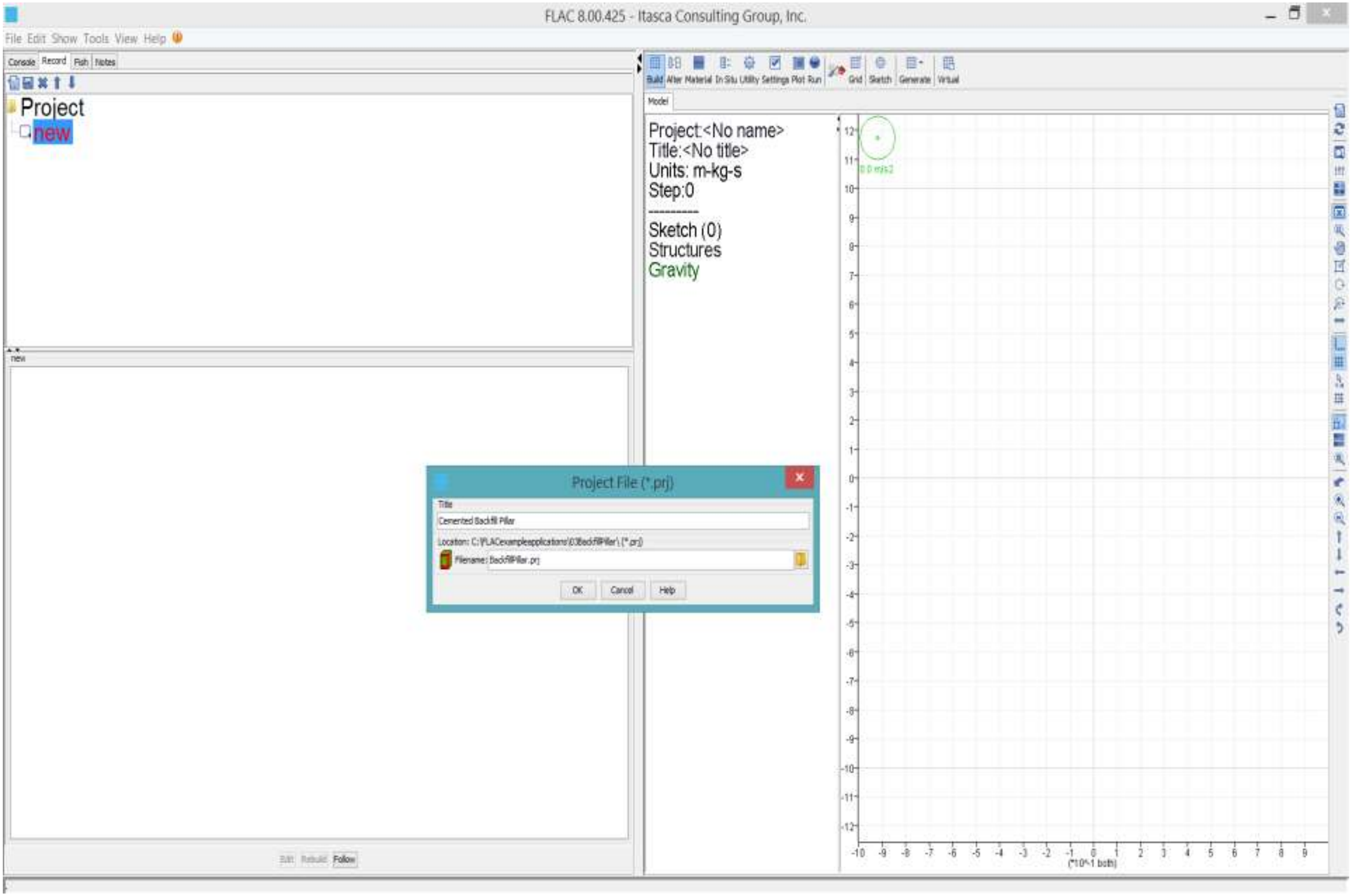
Step 1 Create backfill pillar model with interface between fill and rock. Assign material model and properties.

Step 2 Set the gravity and boundary conditions. Solve for the equilibrium state prior to retreat mining.

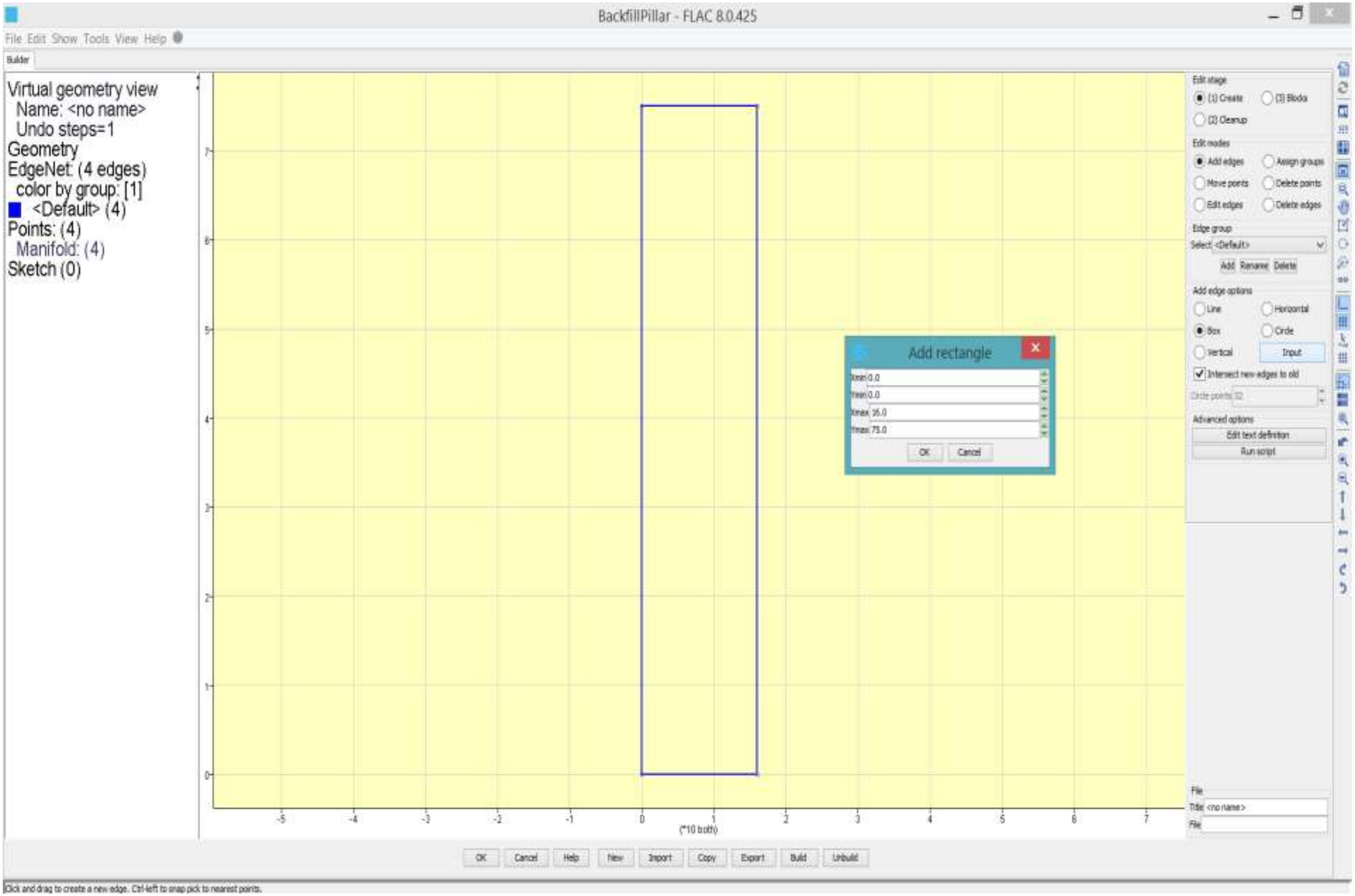
Step 3 Simulate vertical retreat mining by removing x-direction fix along left boundary in small increments to simulate 6 m blast height.



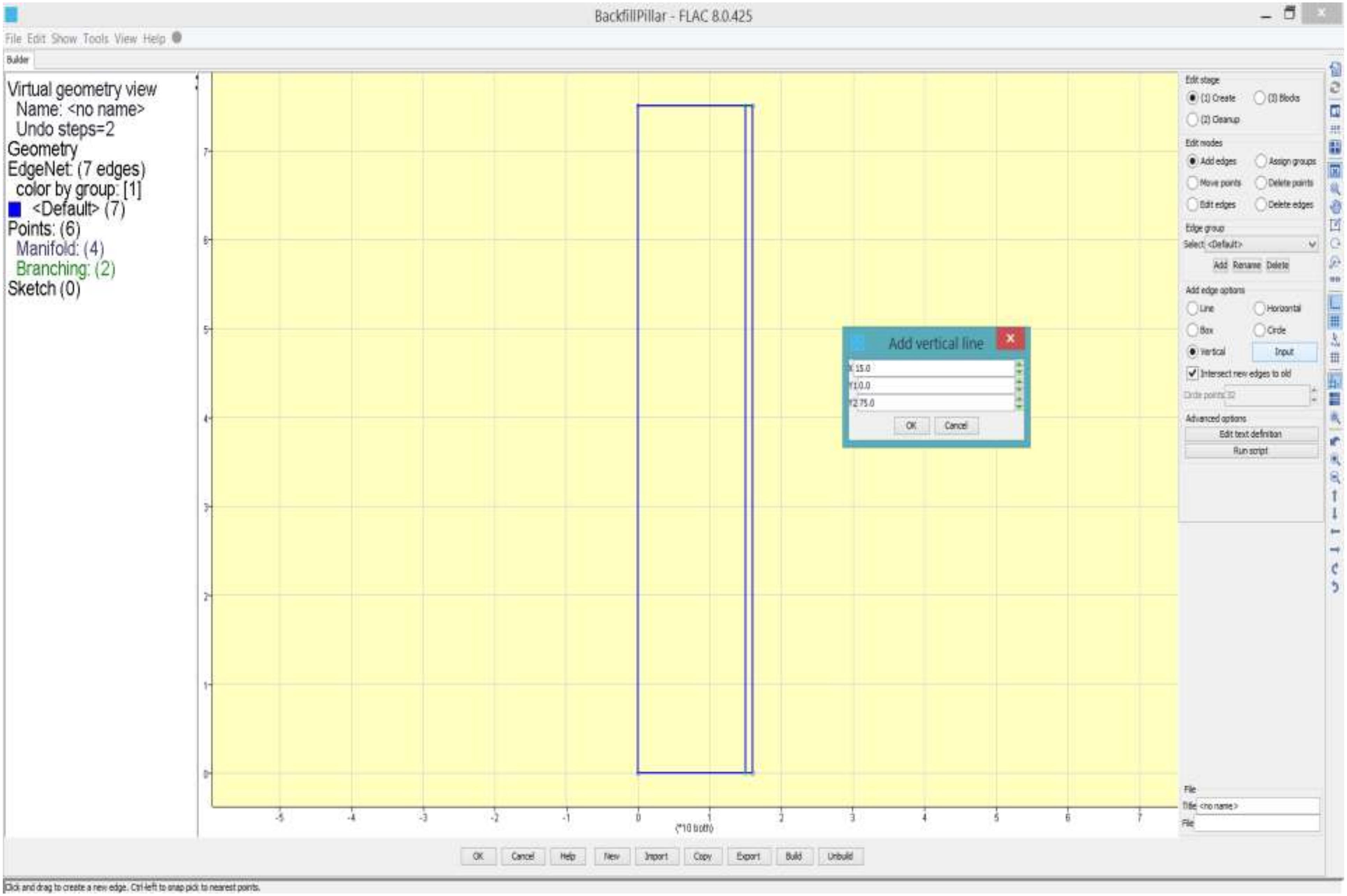
In the [Model options] dialog, select [Include advanced constitutive models] to allow changing material properties. The system of units are [SI: meter-kilogram-second].



A project title is assigned, and a project file **backfillpillar.prj** is created and stored in a working directory.



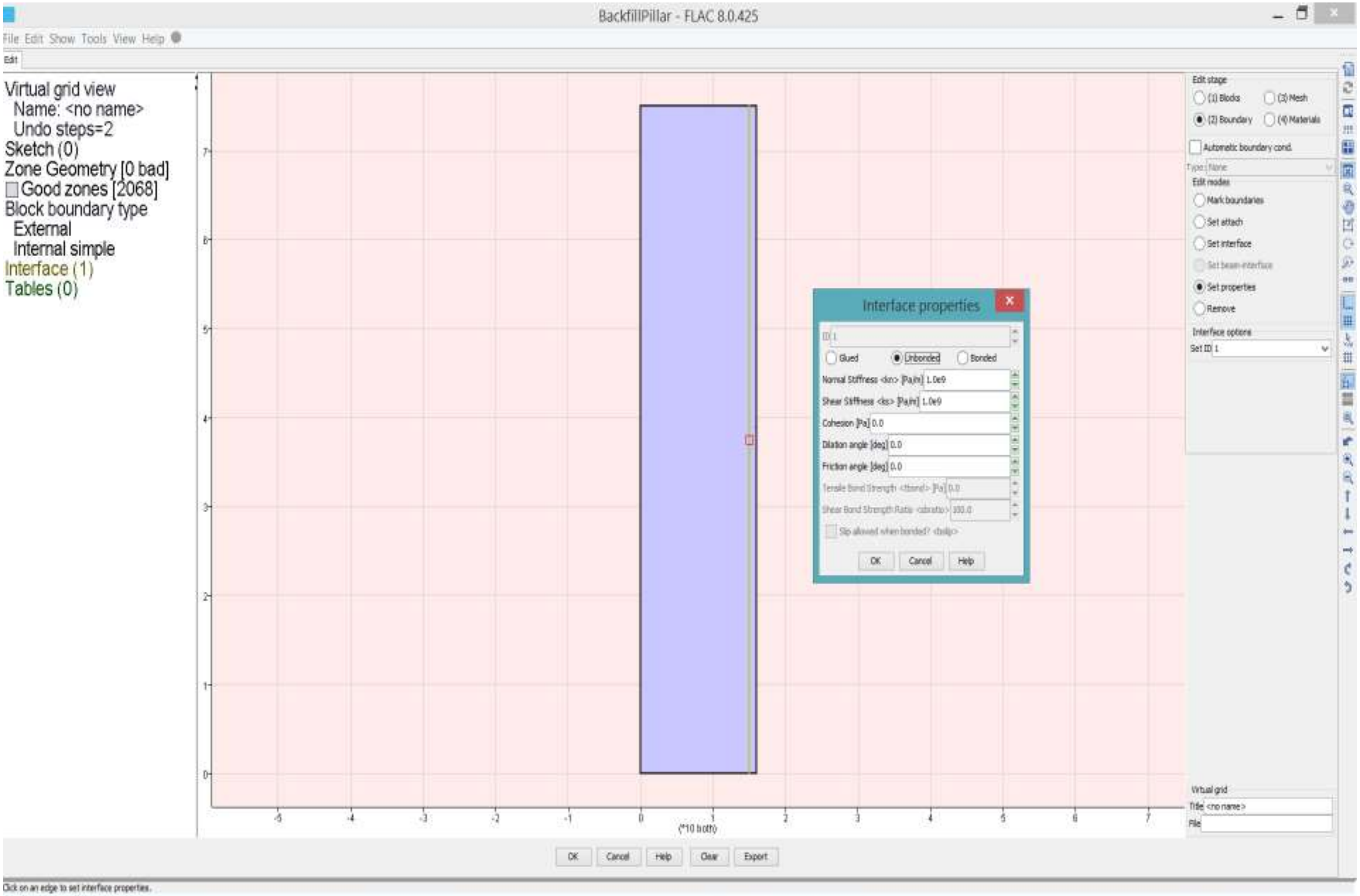
Step 1-1 In the [Geometry Builder] create a rectangular box with boundaries at Xmin = 0.0, Xmax = 16.0, Ymin = 0.0, Ymax = 75.0.



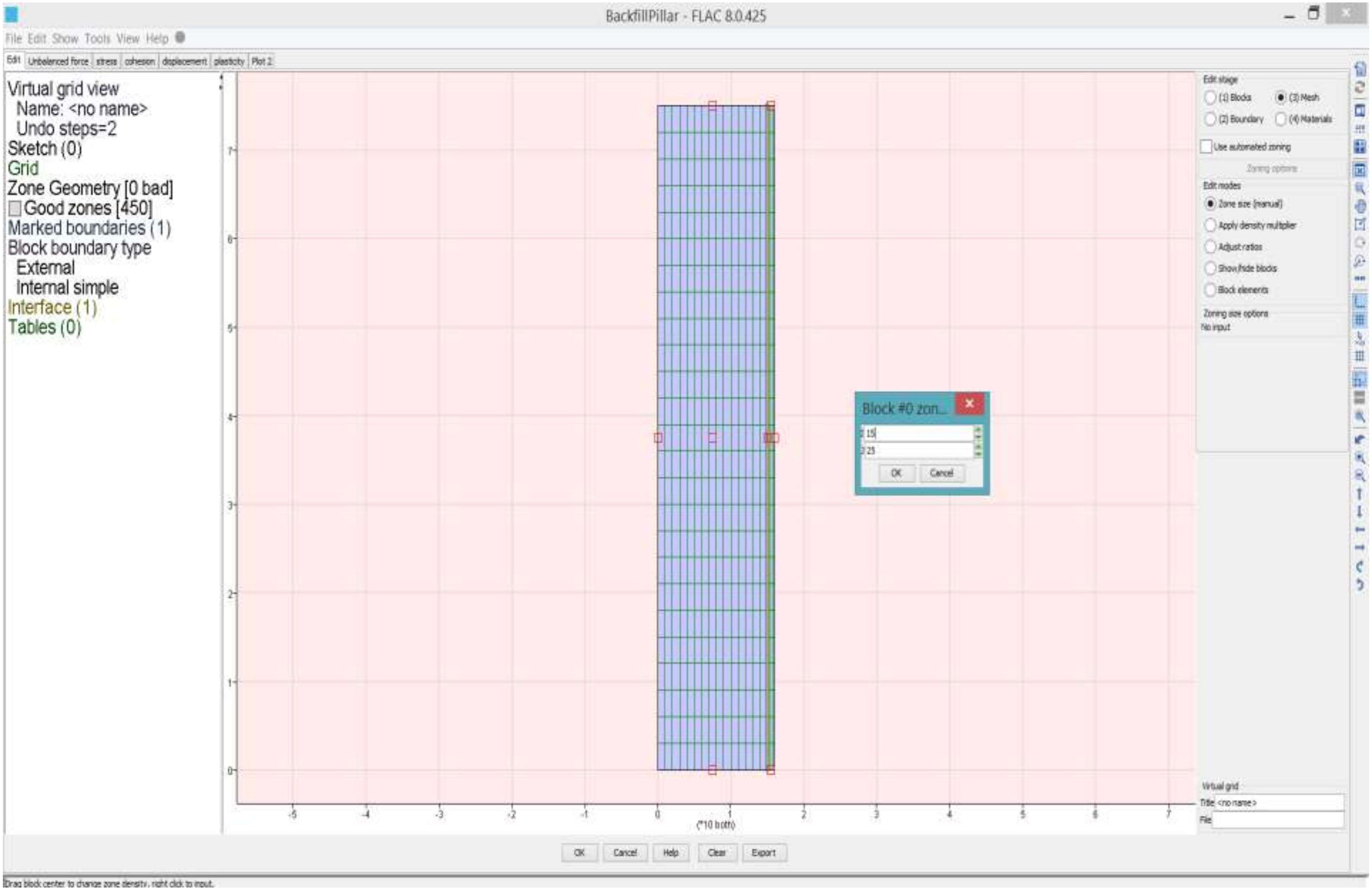
Step 1-2 Add a vertical line ay x = 15.0.

Step 1-3 In the [Blocks] edit stage check that the model contains two quad blocks. Press [OK] to exit the [Geometry builder].

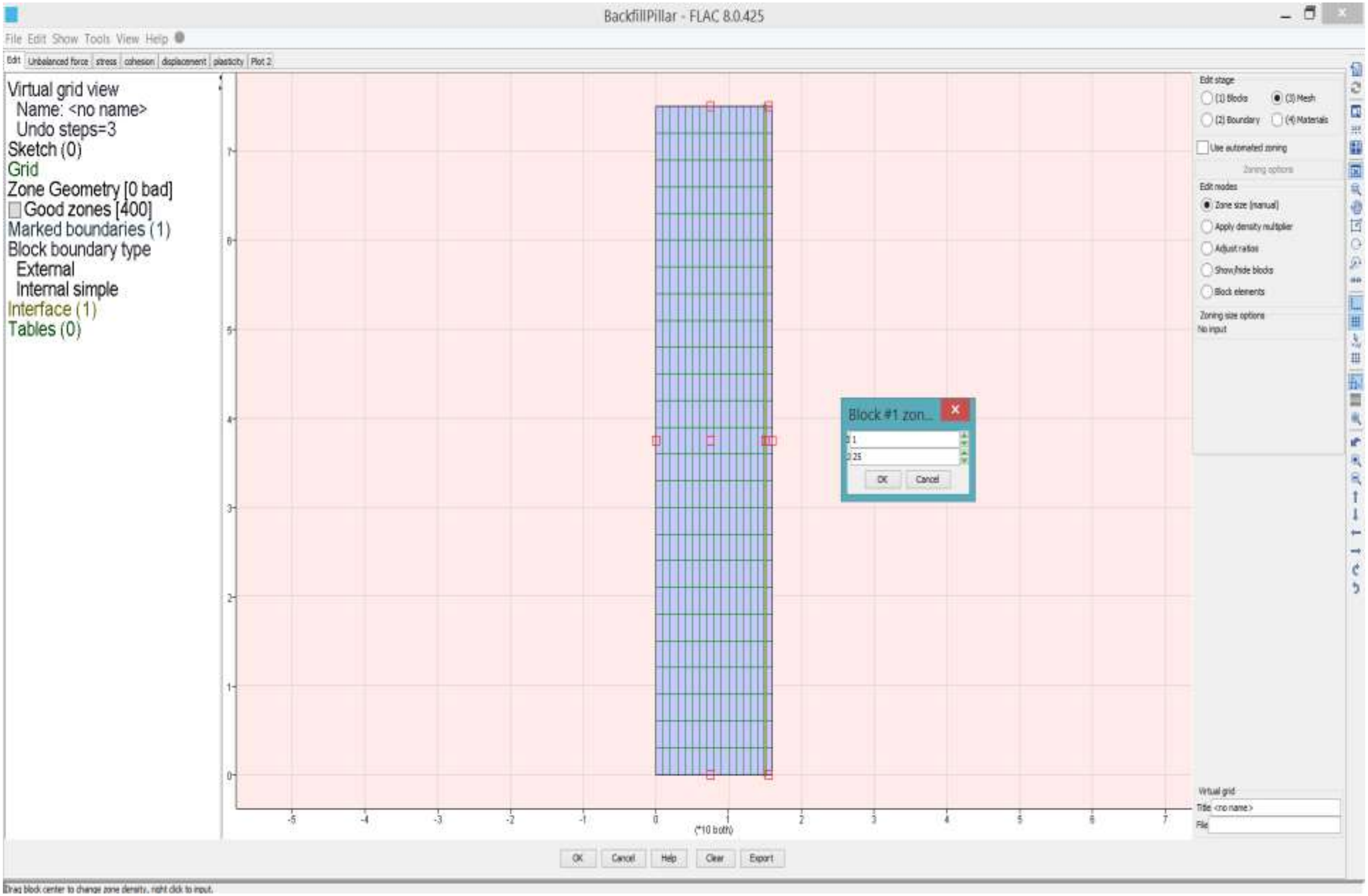
Step 1-4 Enter the [Virtual]/[Edit] tool and select the [Boundary] edit stage. Check [Set interface] and click on the line at x=15..



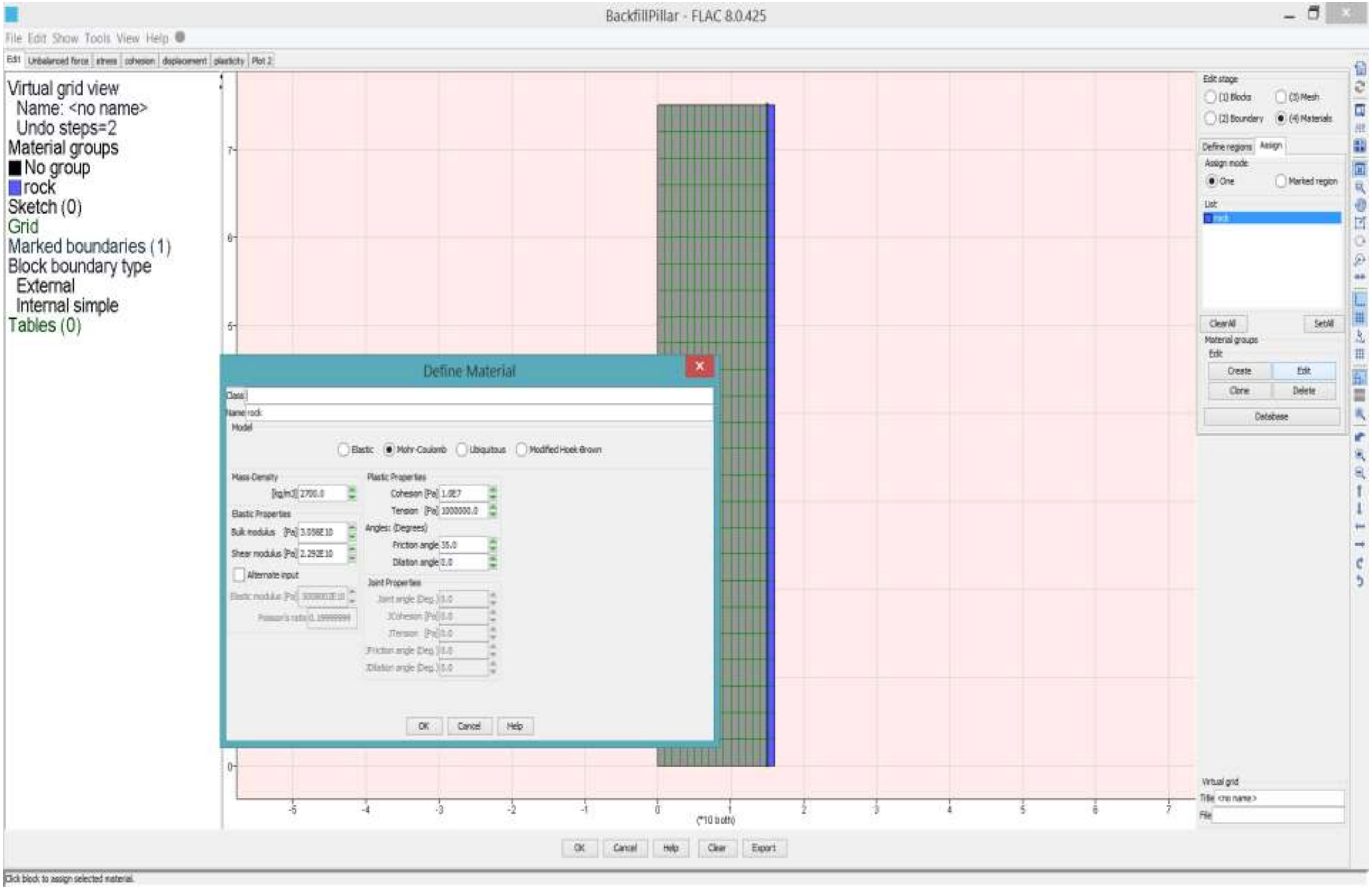
Step 1-5 Select [Set properties] and click on the interface to open the *Interface properties* dialog. Input the properties with zero friction initially.



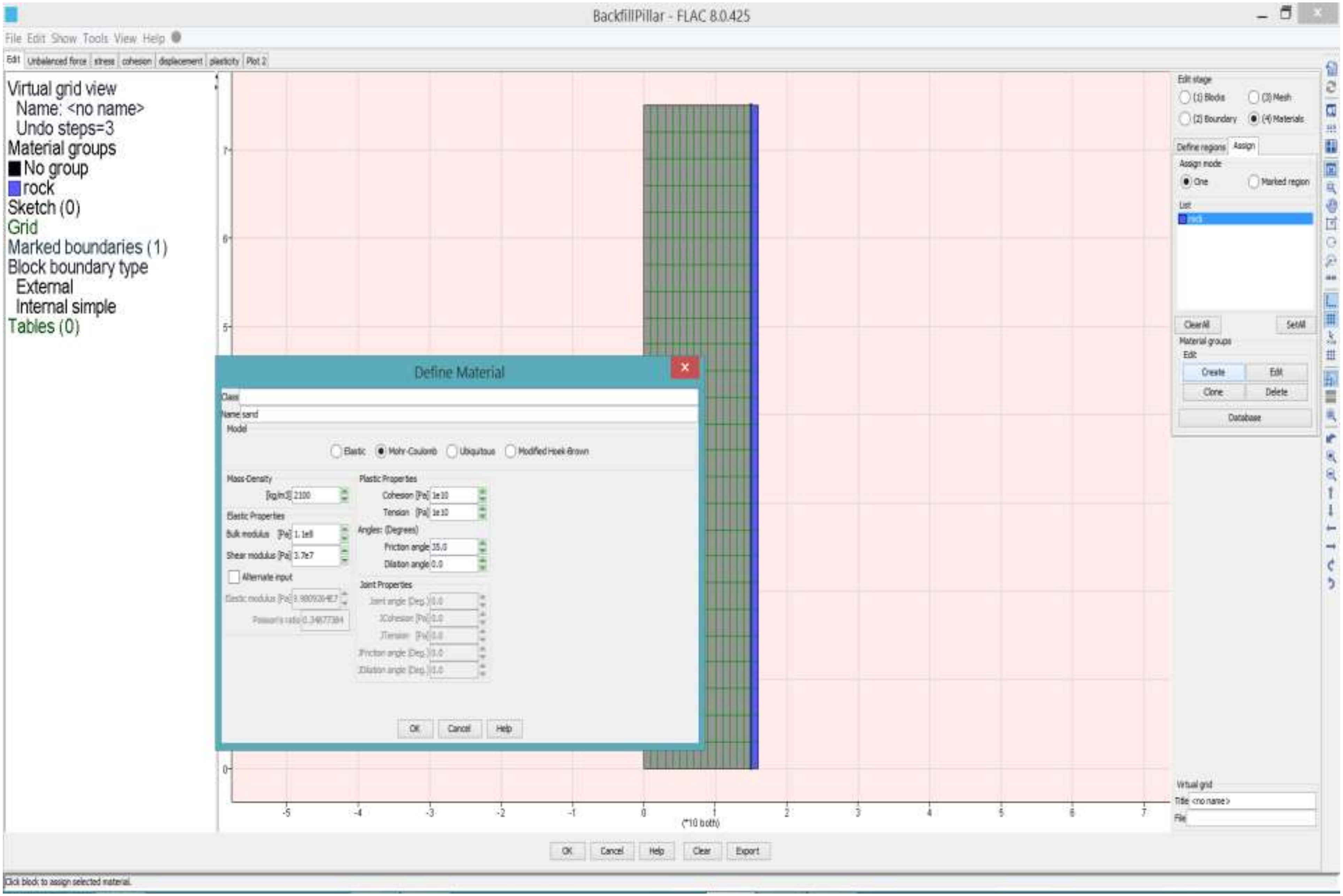
Step 1-6 Enter the [Mesh] edit stage and uncheck [Use automated zoning]. Input the mesh size for the backfill region manually, as shown.



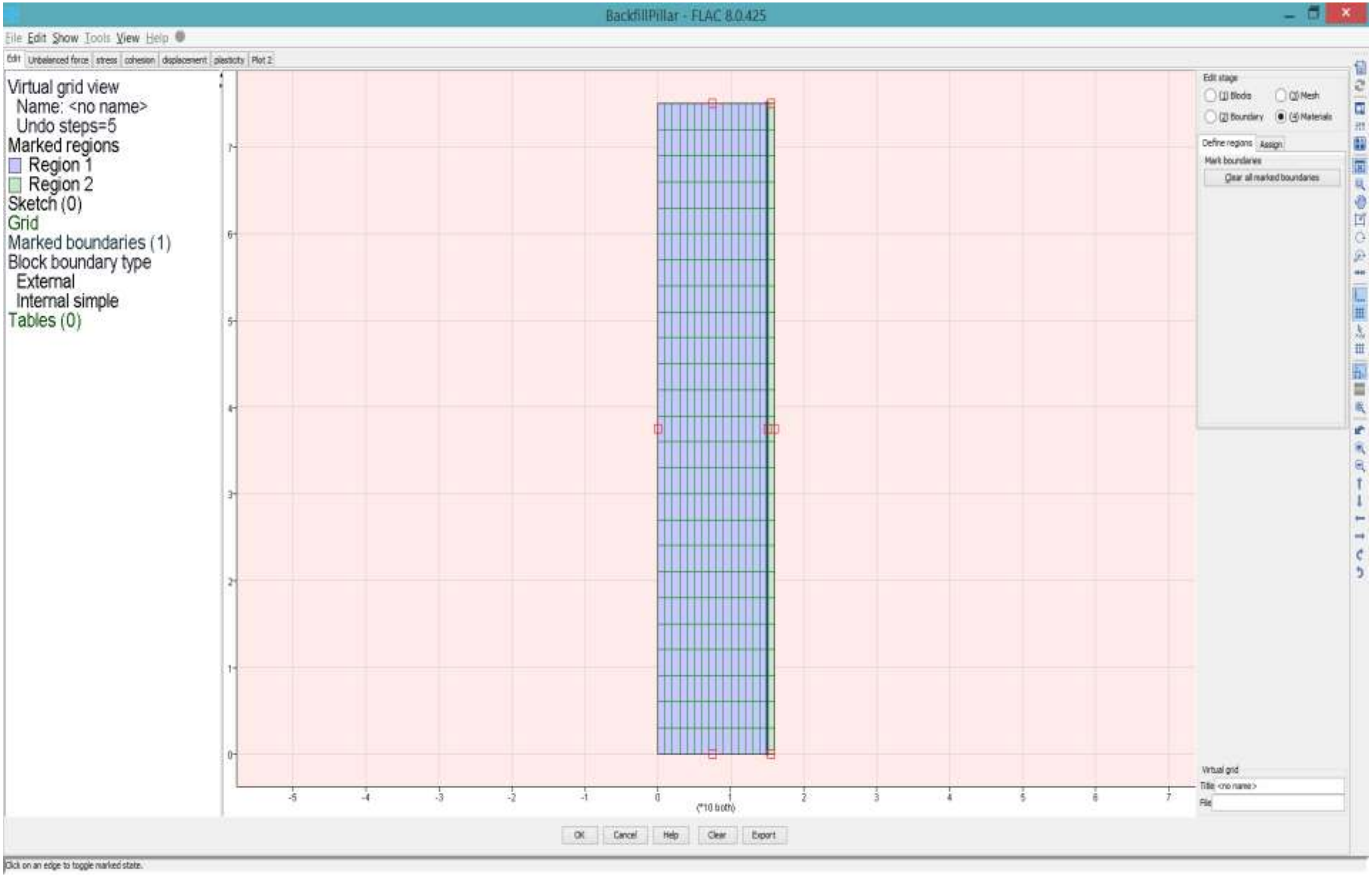
Step 1-7 Change the mesh size for the rock region as shown.



Step 1-8 Enter the [Material] edit stage, [Assign] tab and click on [Create] to open the Define material dialog. Create the **rock** material using the Mohr-Coulomb model and properties as shown.

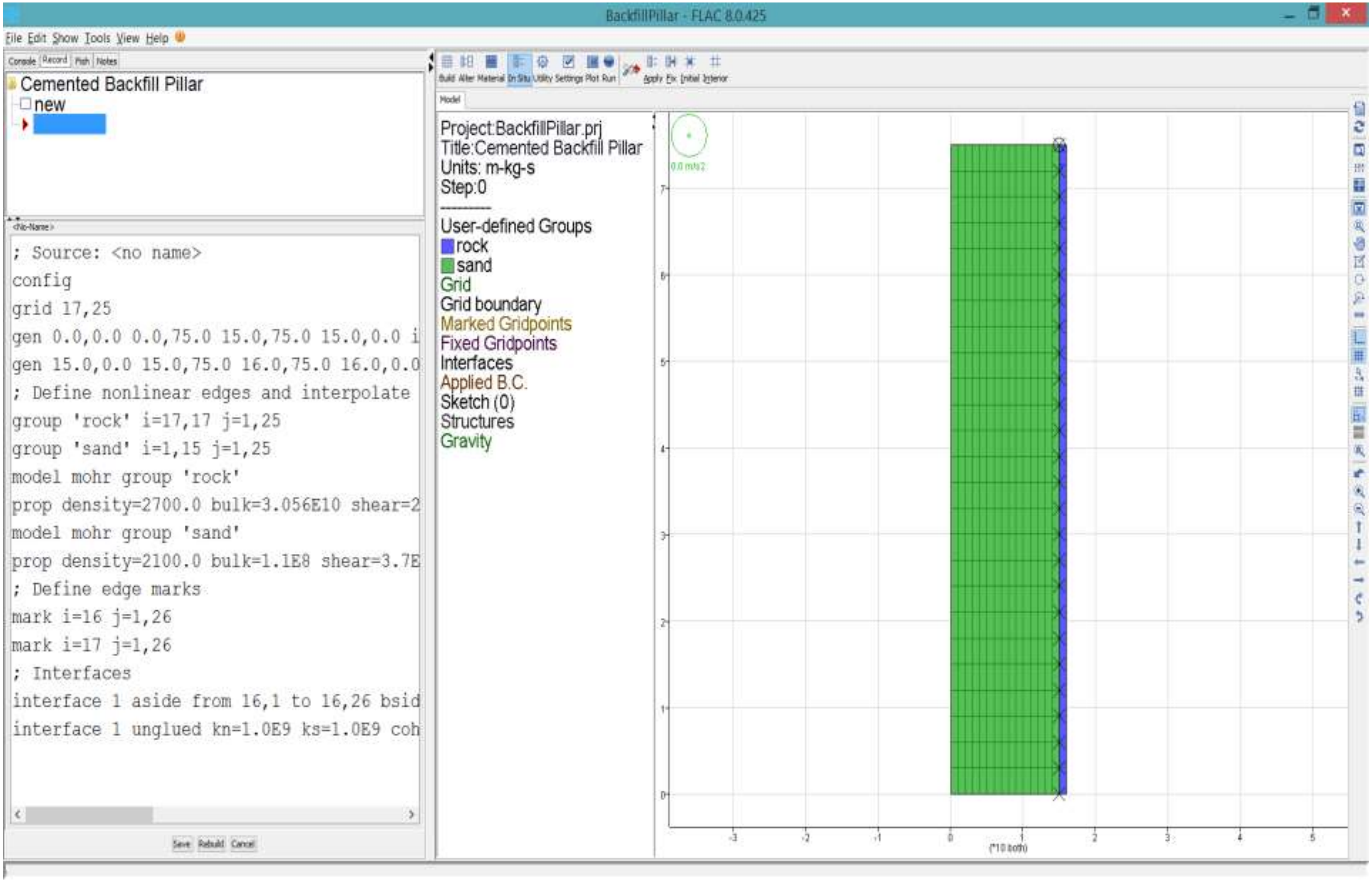


Step 1-9 Repeat Step 1-8 to create **sand** material with Mohr-Coulomb model and properties as shown.

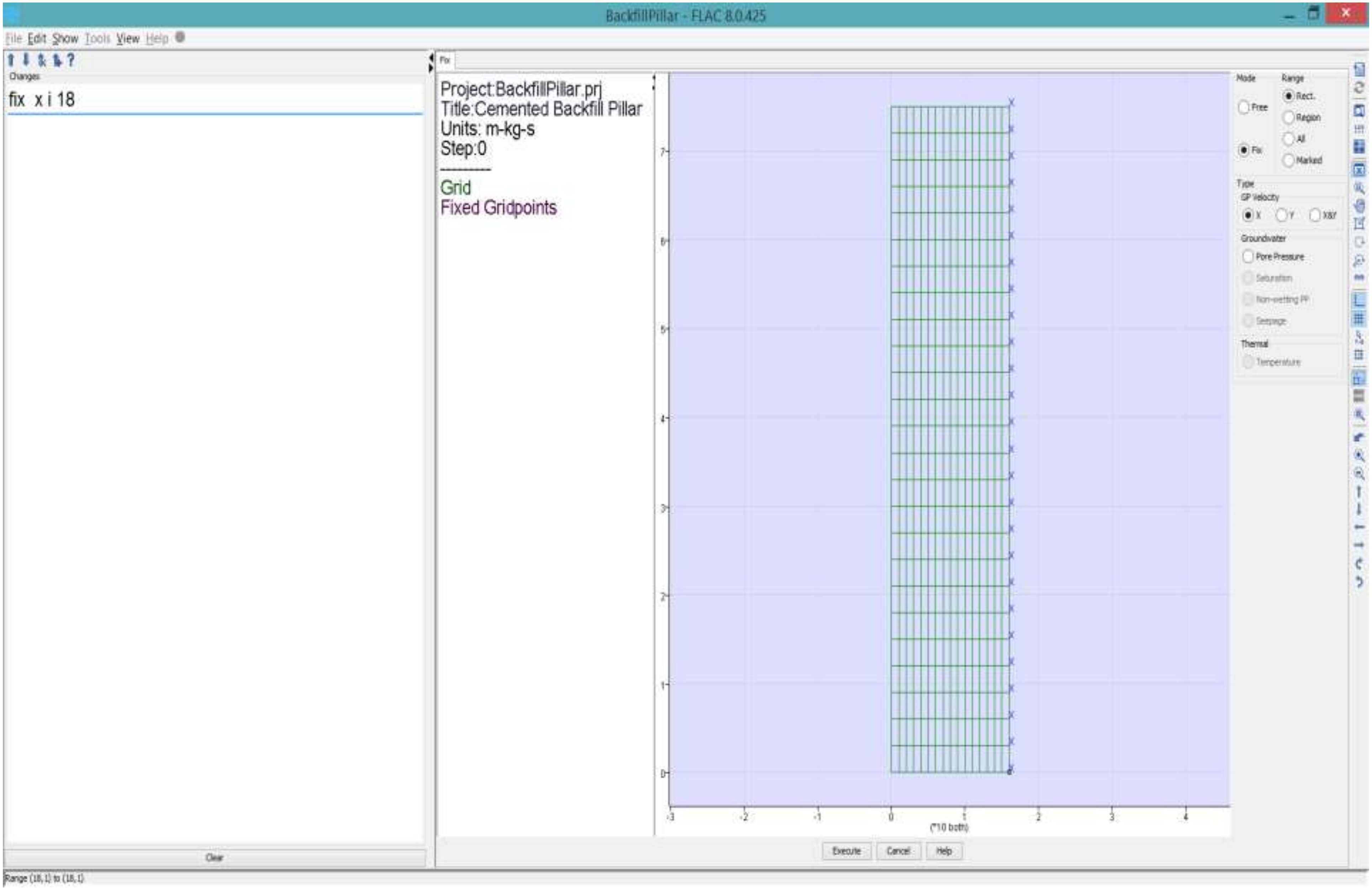


Step 1-10 Press [Define region] and click on the interface to create two regions in the model.

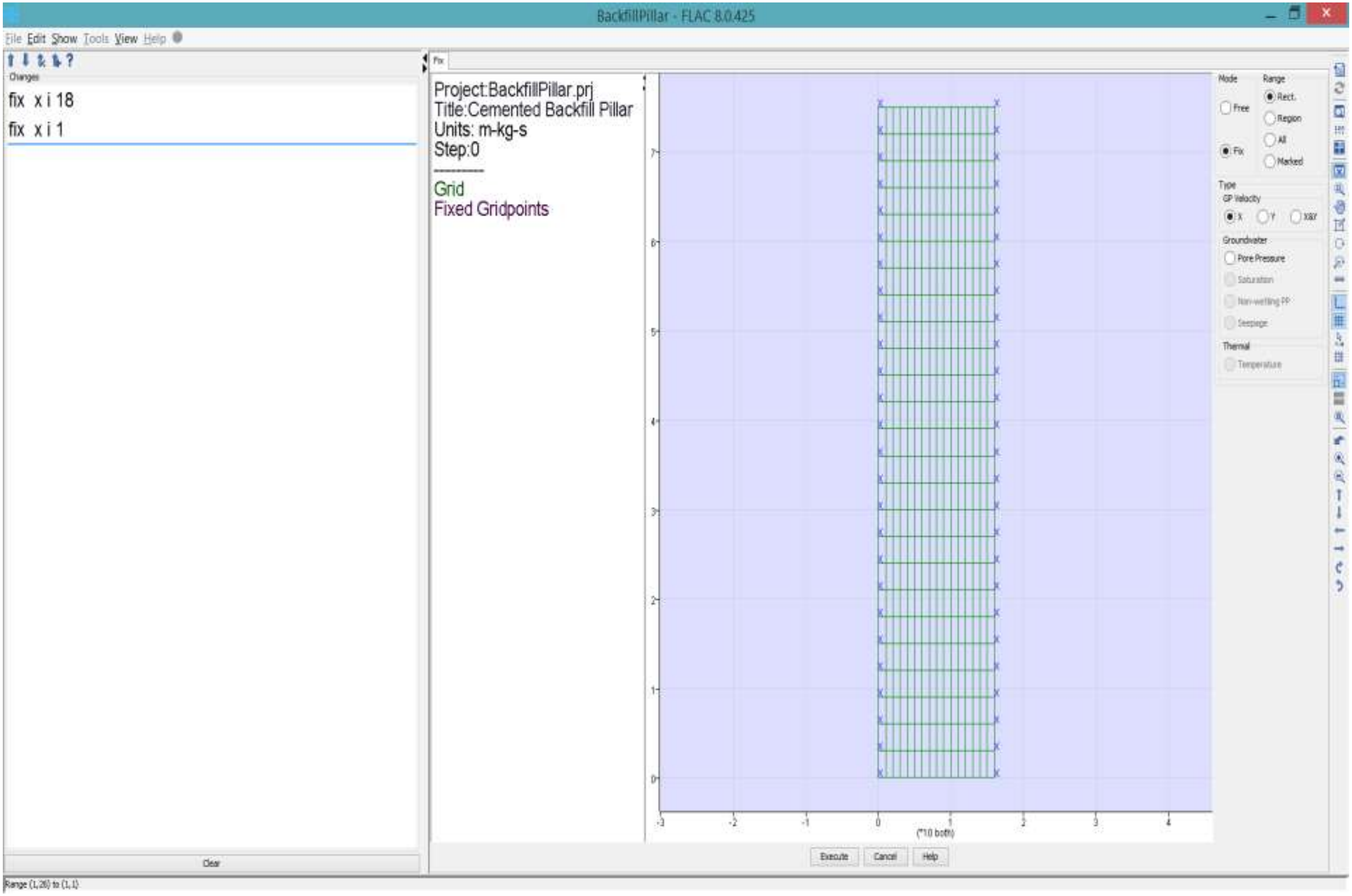
Step 1-11 Click on the [Assign] tab and [Marked region]. Select **rock** and click in Region 2 to assign the **rock** material. Select **sand** and click in Region 1 to assign the **sand** material. Press [OK] and then [Execute] to send commands to *FLAC*.



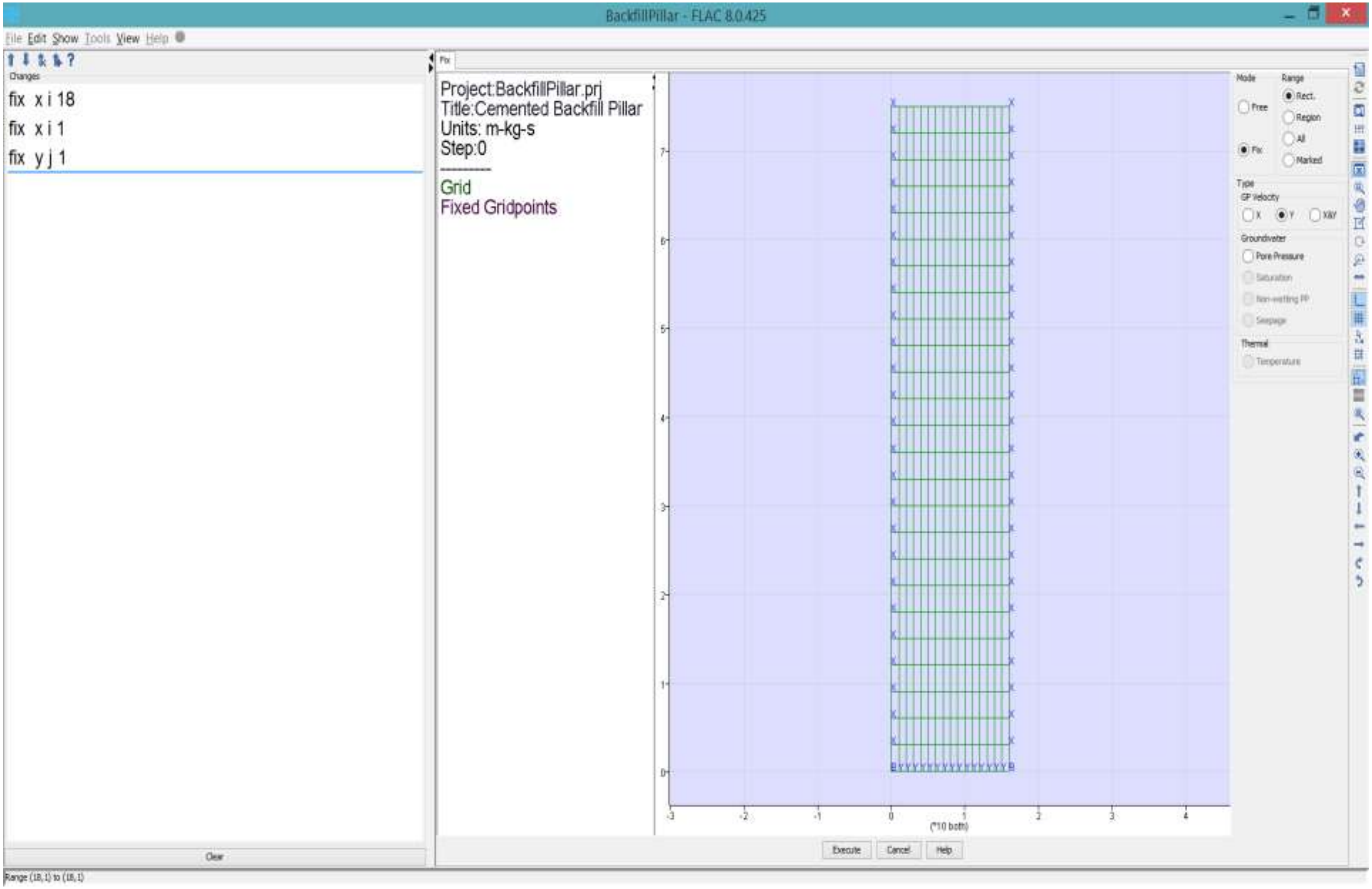
Step 1-12 The assignment of material models and the interface are shown in the *model view*, and the associated commands are listed in the *Record* pane.



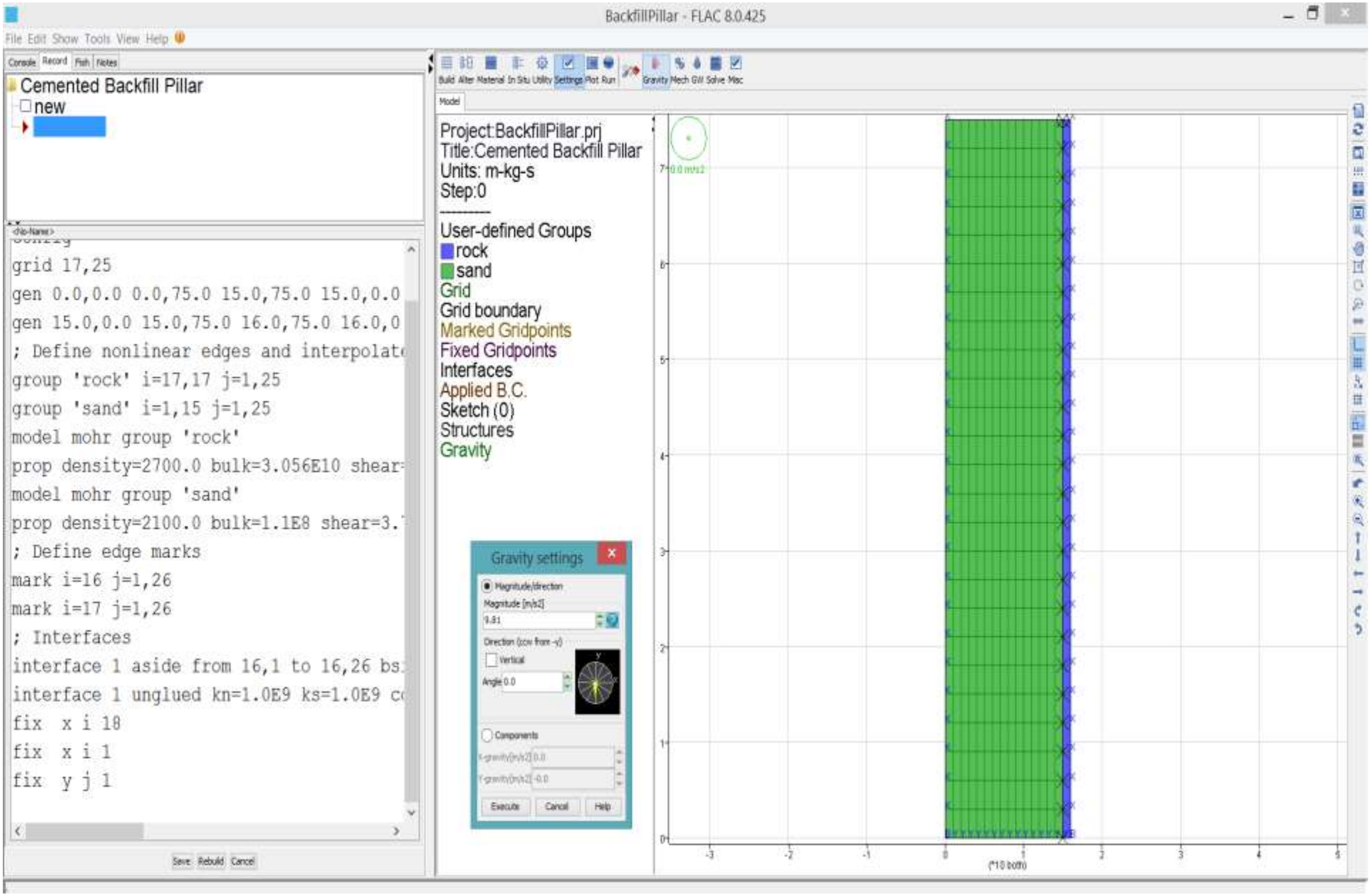
Step 2-1 Enter the [In Situ]/[Fix] tool. Select [Fix] and [X] GP Velocity Type. Drag the mouse along the right boundary to assign a roller boundary.



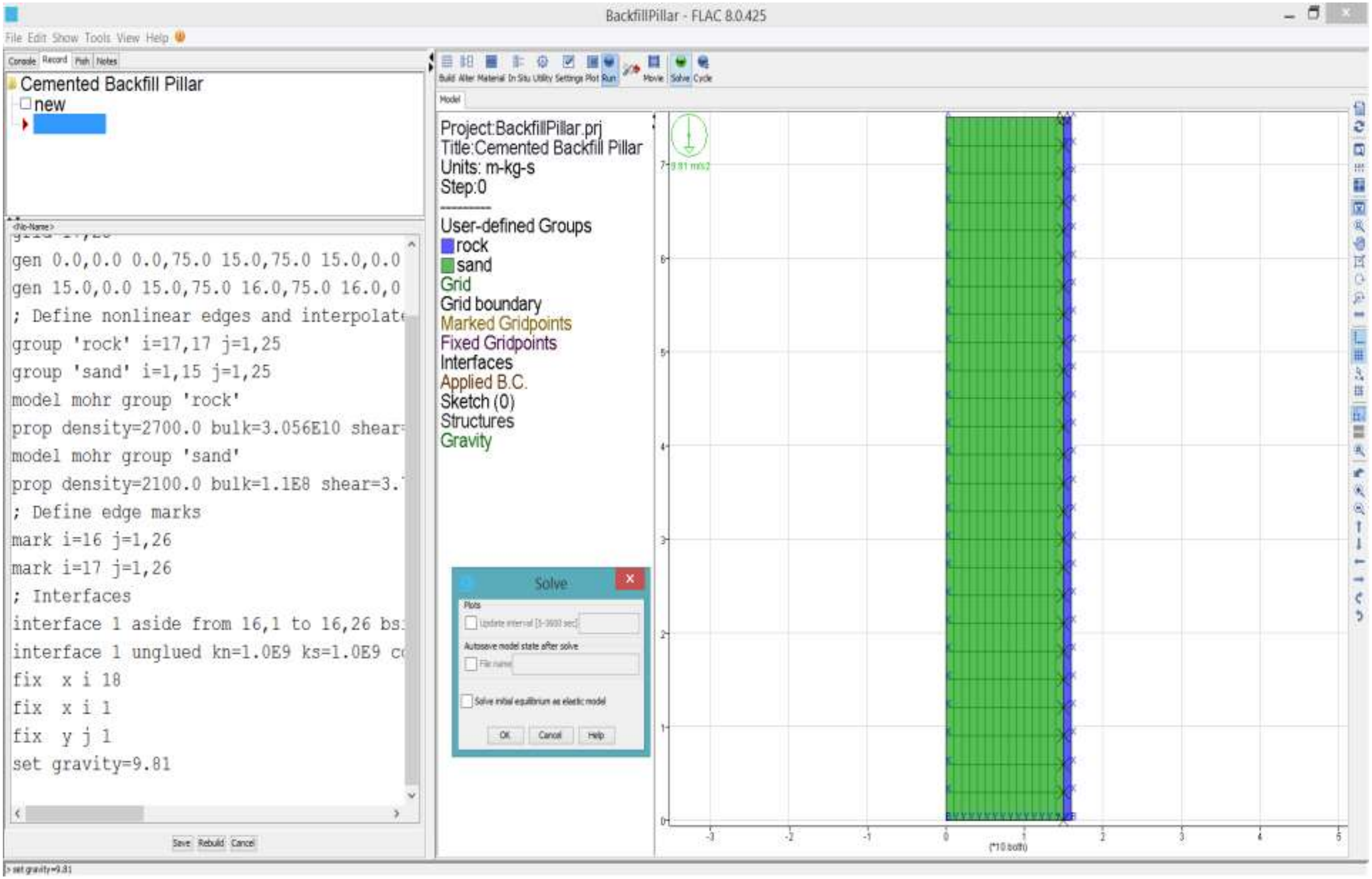
Step 2-2 Drag the mouse along the left boundary to assign a roller boundary.



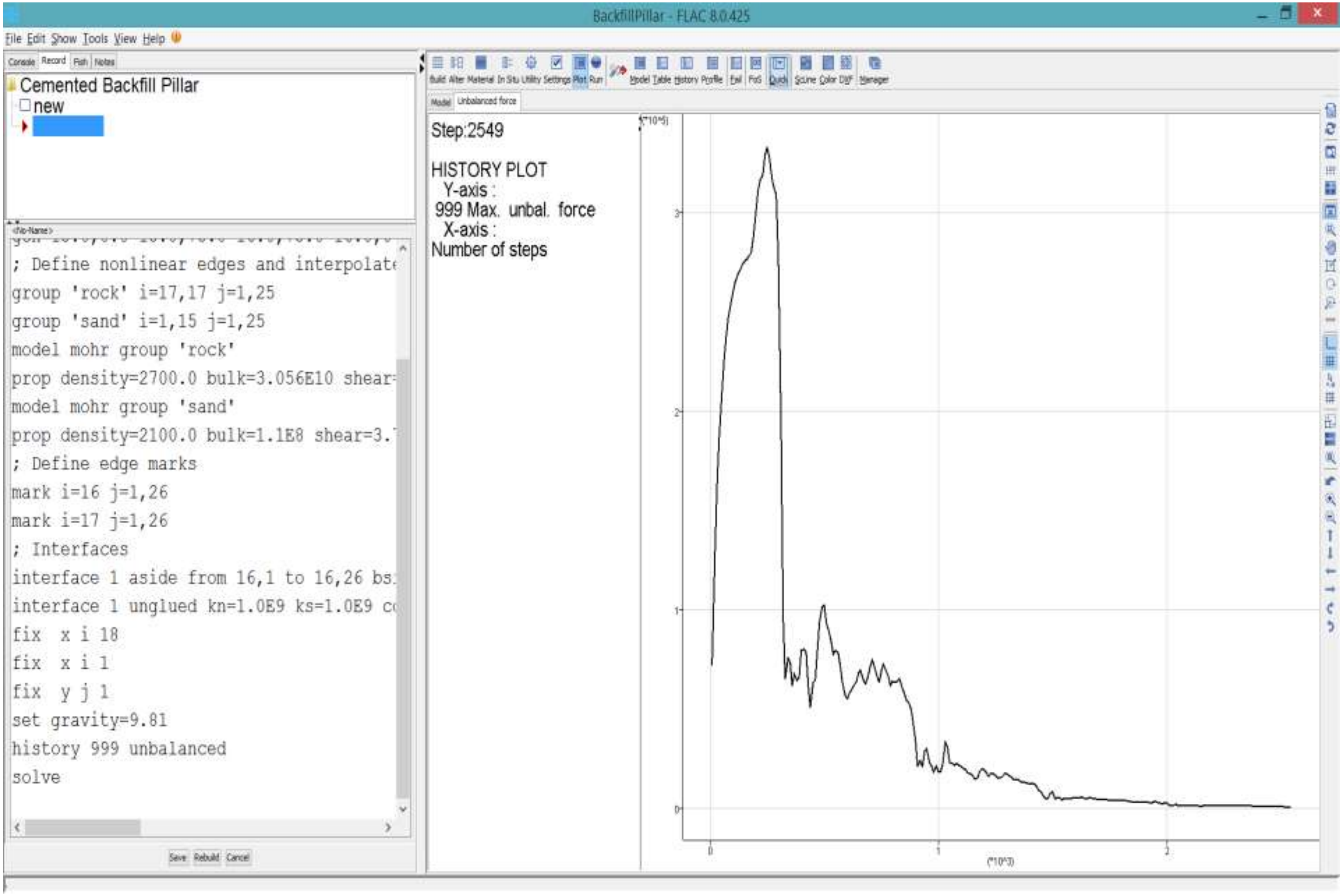
Step 2-3 Select [Fix] and [Y] GP Velocity Type. Drag the mouse along the bottom boundary to assign a roller boundary.



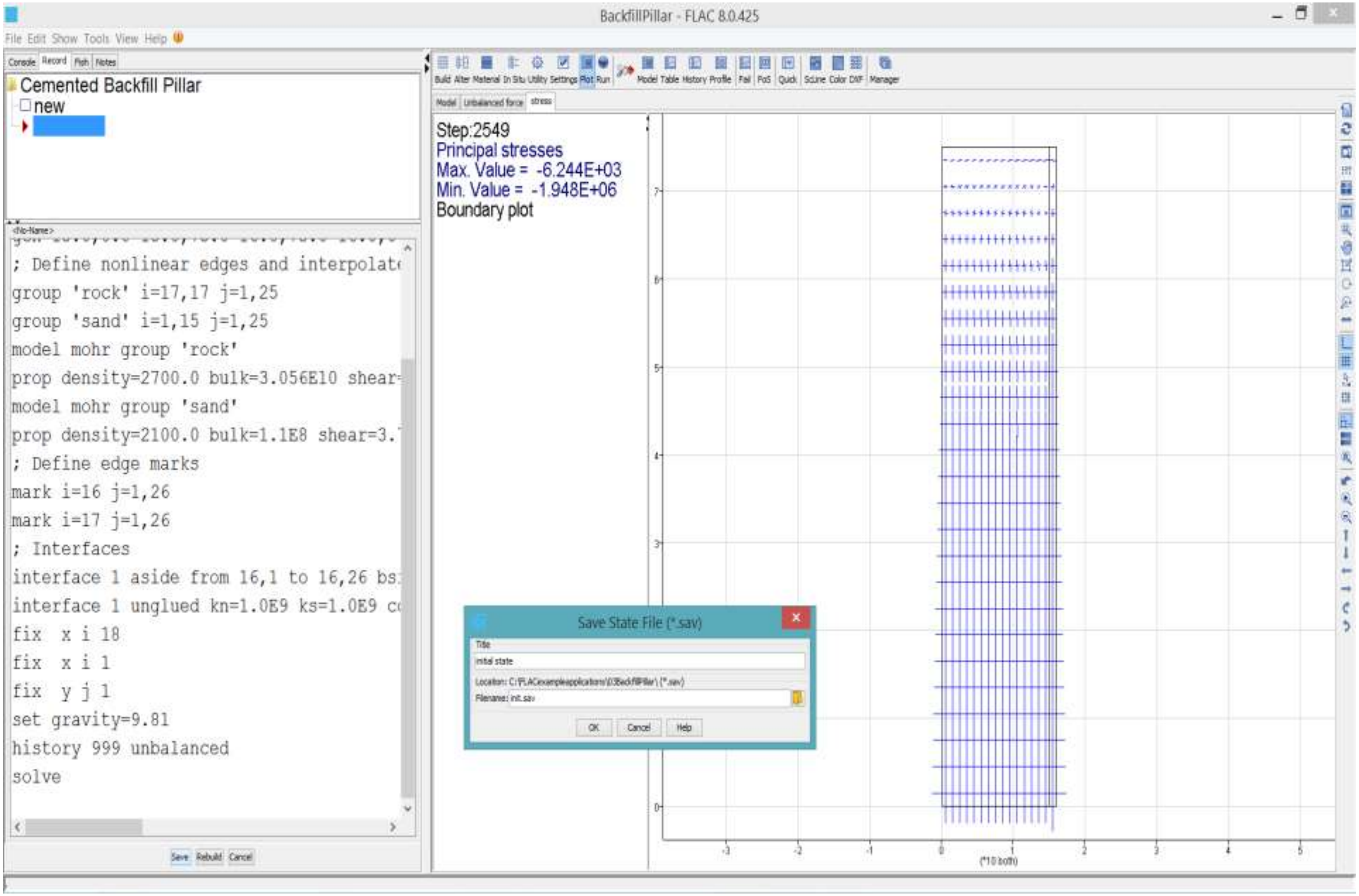
Step 2-4 Enter the [Settings/[Gravity] tool and assign the gravitational magnitude. Press [Execute] to send the command to *FLAC*.



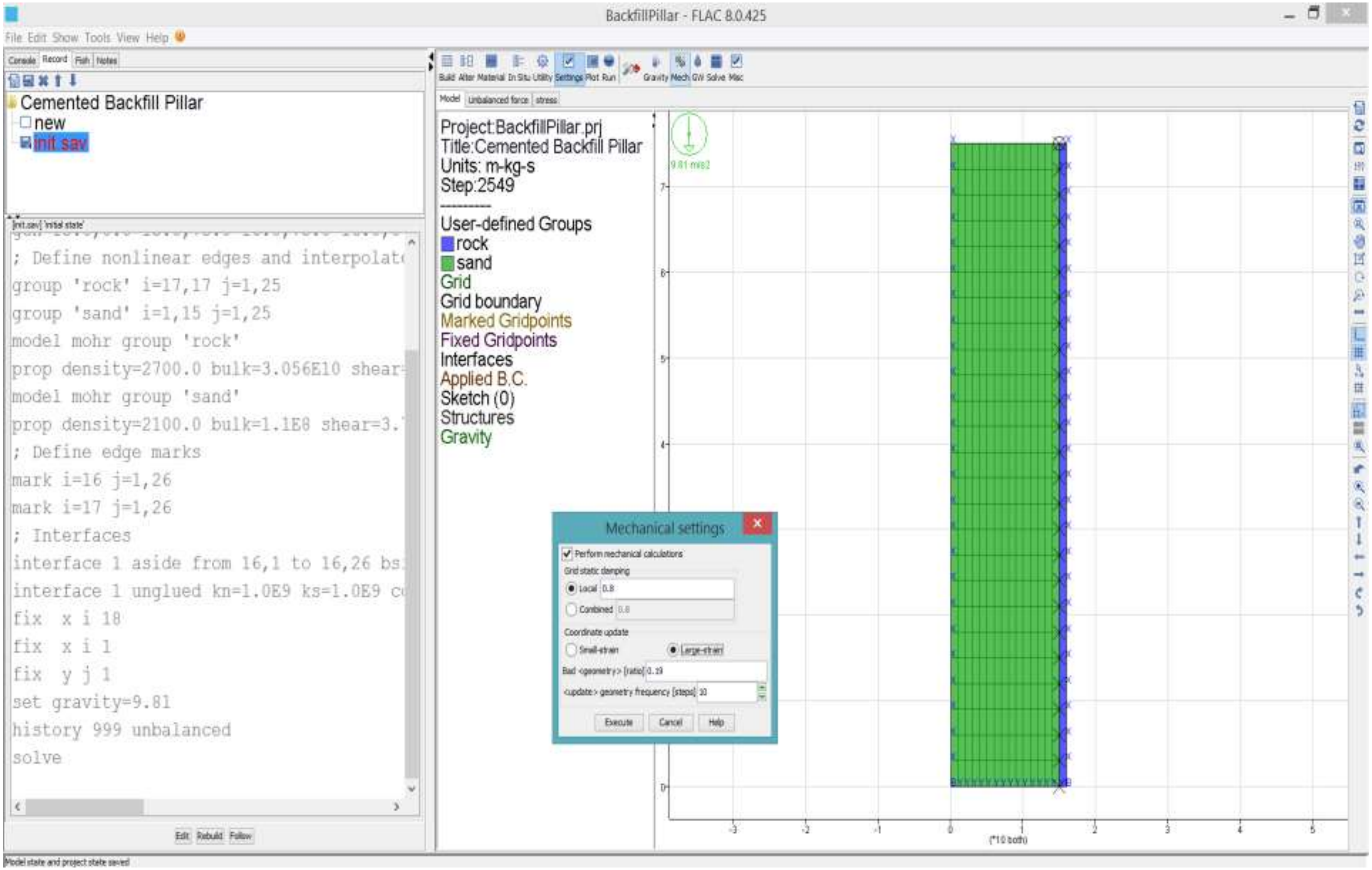
Step 2-5 Select [Run]/[Solve] and press [OK] to calculate the equilibrium state.



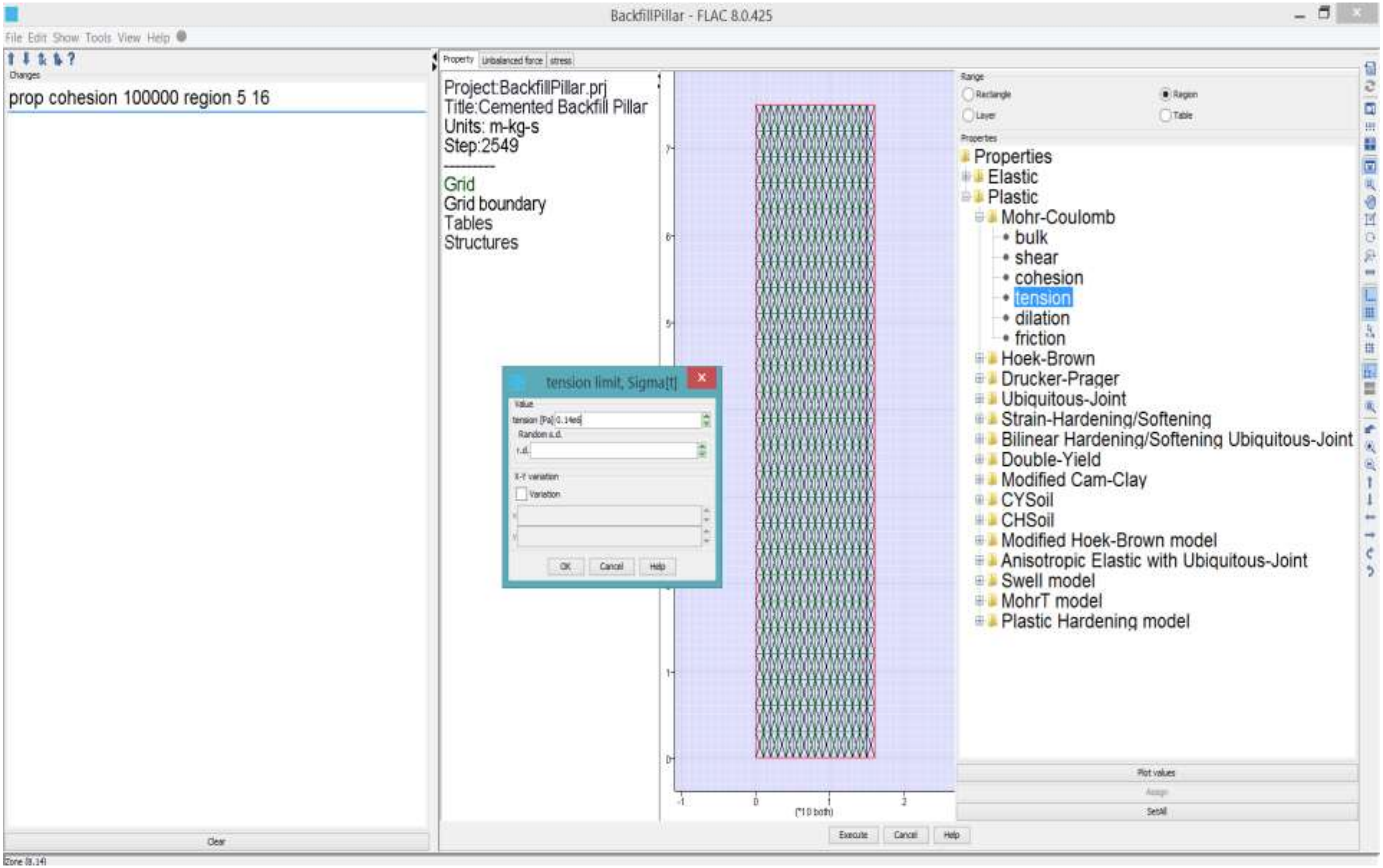
Step 2-6 The maximum unbalanced force history plot indicates that equilibrium is reached.



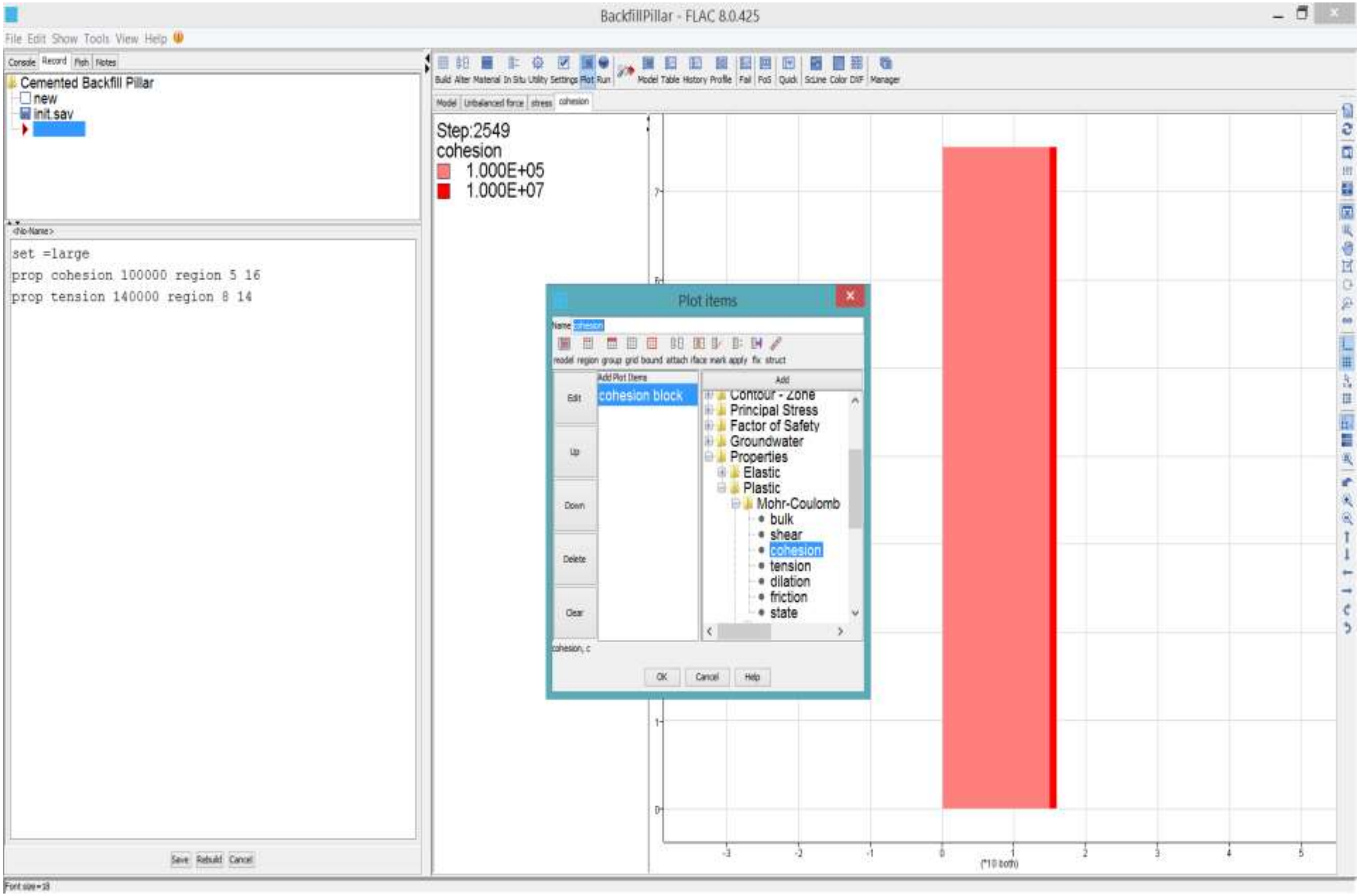
Step 2-7 Save the state as init.sav.



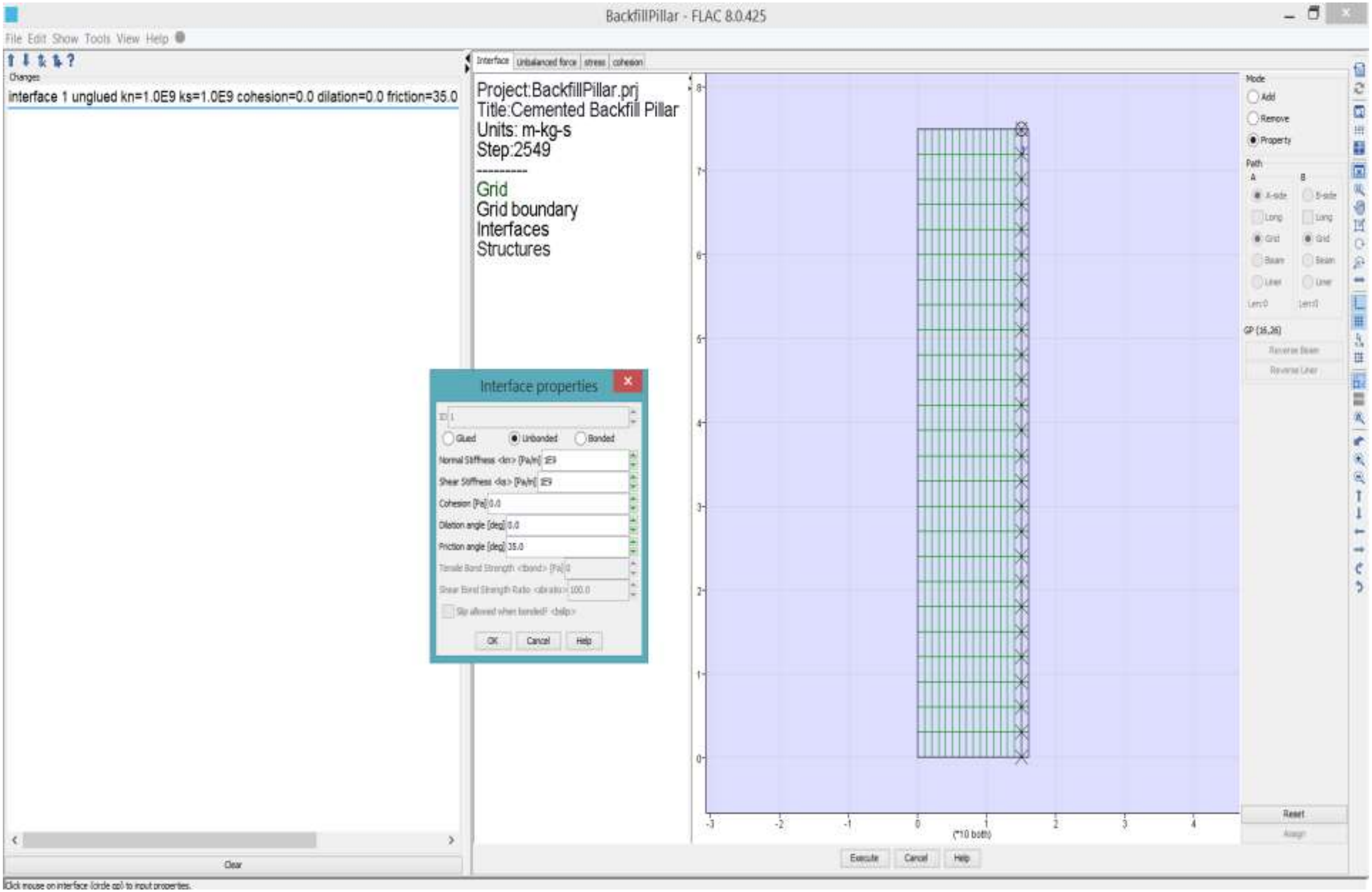
Step 3-1 Enter the [Settings]/[Mech] tool. Click on [Large-strain] to turn on the large strain calculation mode. Press [Execute] to send the command to *FLAC*.



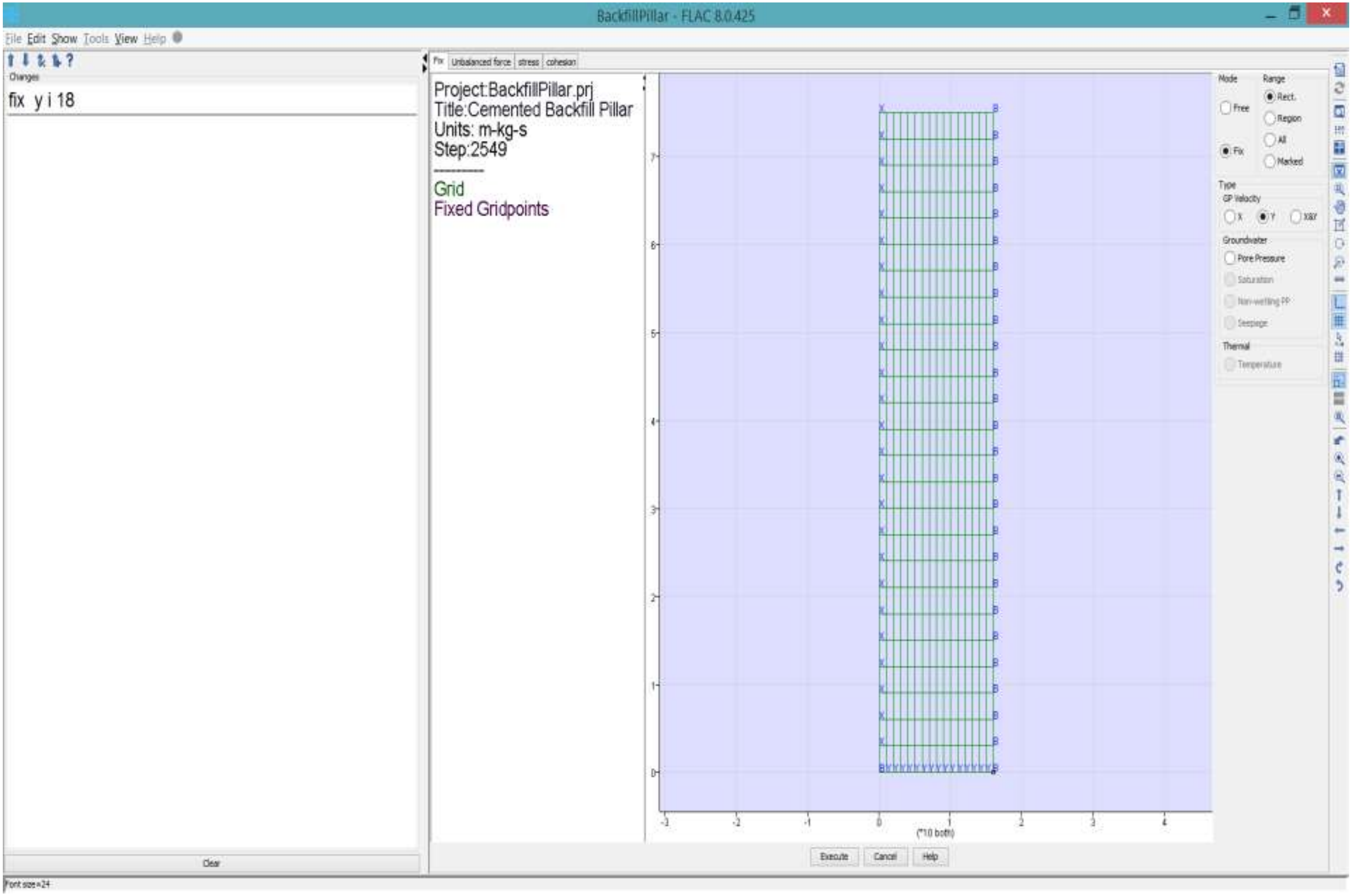
Step 3-2 Enter the [Material]/[ChangeProp] tool and select [Region]. In the []Properties] list, select [Mohr-Coulomb]/[cohesion] and click on a zone in the **sand** group to open the *cohesion* dialog. Change cohesion to 0.1 MPa and press [OK]. Repeat this procedure to change tension limit to 0.14 MPa. Press [Execute] to send the commands to *FLAC*.



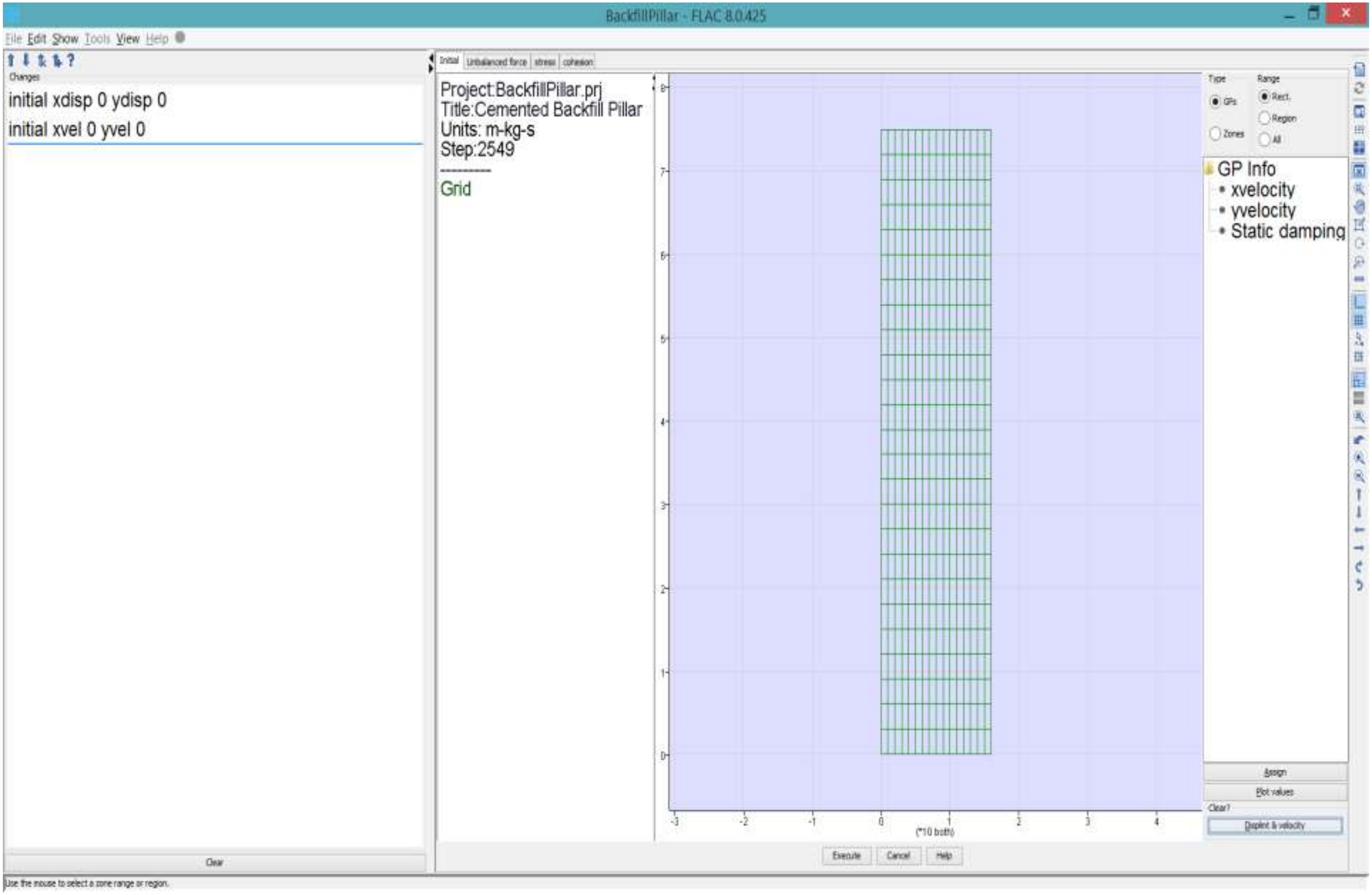
Step 3-3 Create a cohesion plot in the [Plot]/[Model] tool to check that cohesion is changed.



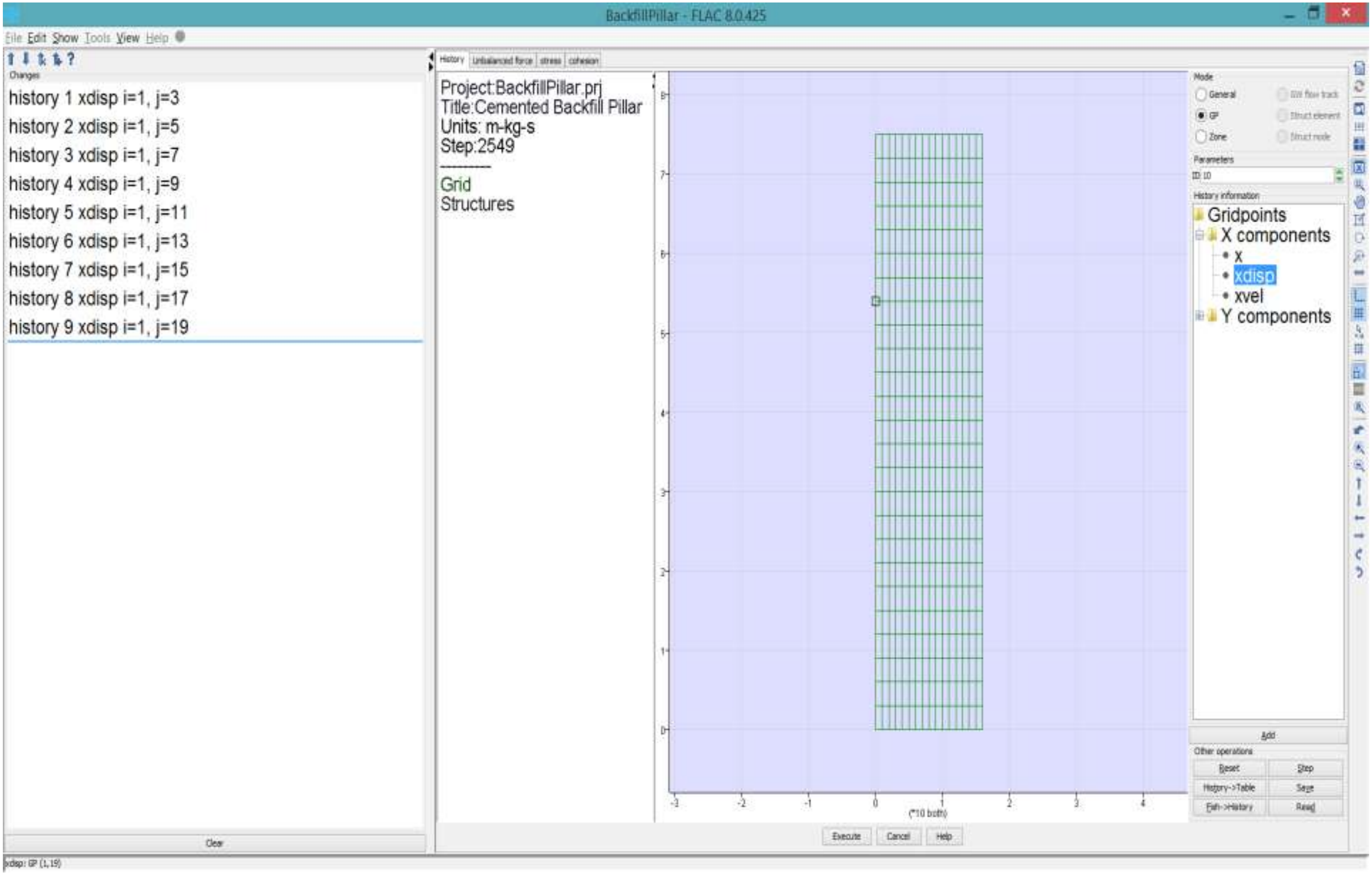
Step 3-4 Enter the [Alter]/[Interface] tool and select [Property]. Click on the interface to open the *Interface properties* dialog. Change friction angle to 35°.



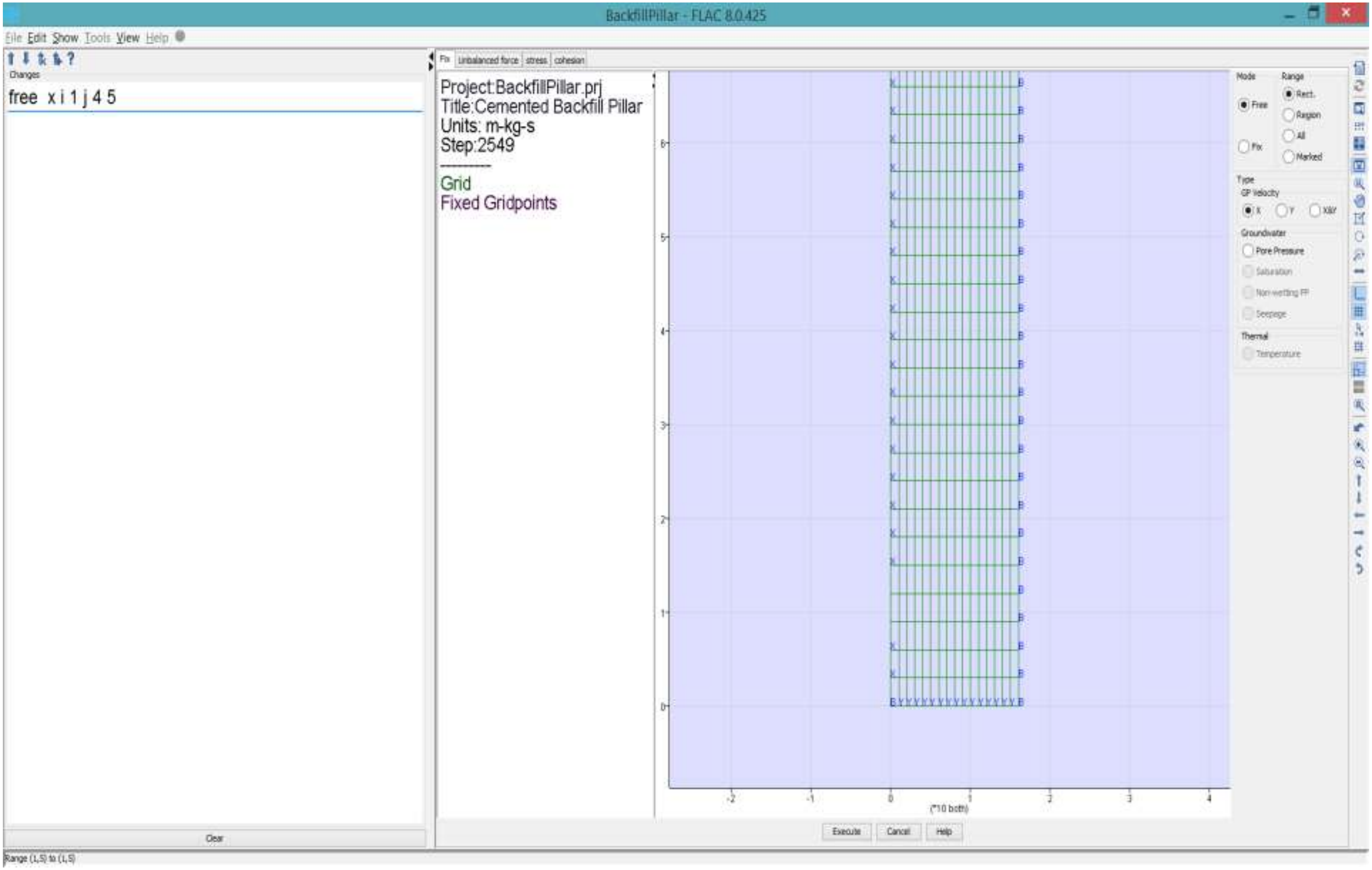
Step 3-5 Enter the [In Situ]/[Fix] tool and select [Fix and [Y]. Drag the mouse along the right boundary to fix the boundary



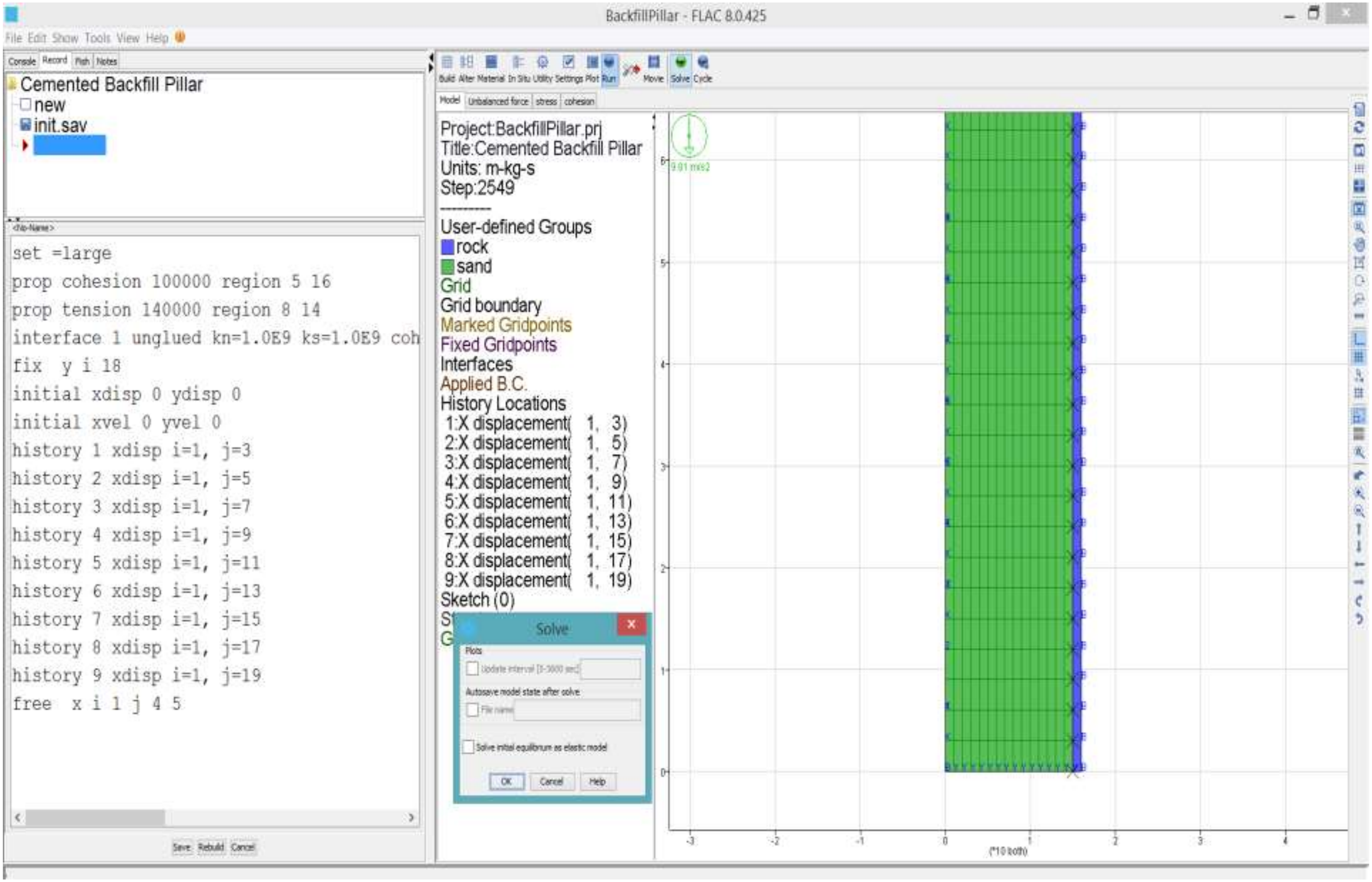
Step 3-6 Enter the [In Situ]/[Initial] tool and select [Clear? Dispmt & velocity] to initialize displacements and velocities.



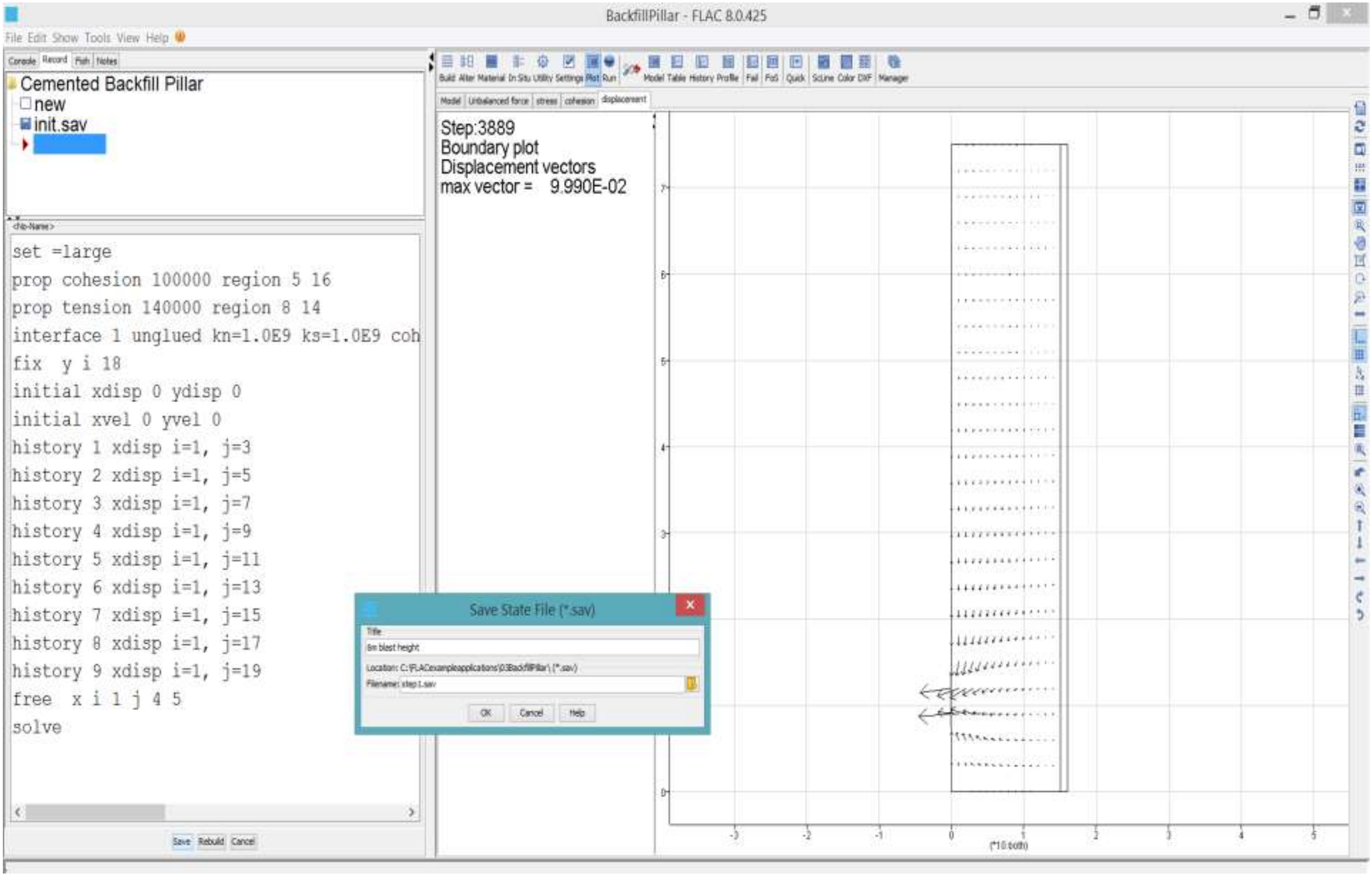
Step 3-7 Enter the [Utility]/[History] tool. Select [Gridpoints]/[xdisp] and click the mouse on the 9 gridpoints to monitor x-displacement, as shown above. Press [Execute] to send the commands to *FLAC*.



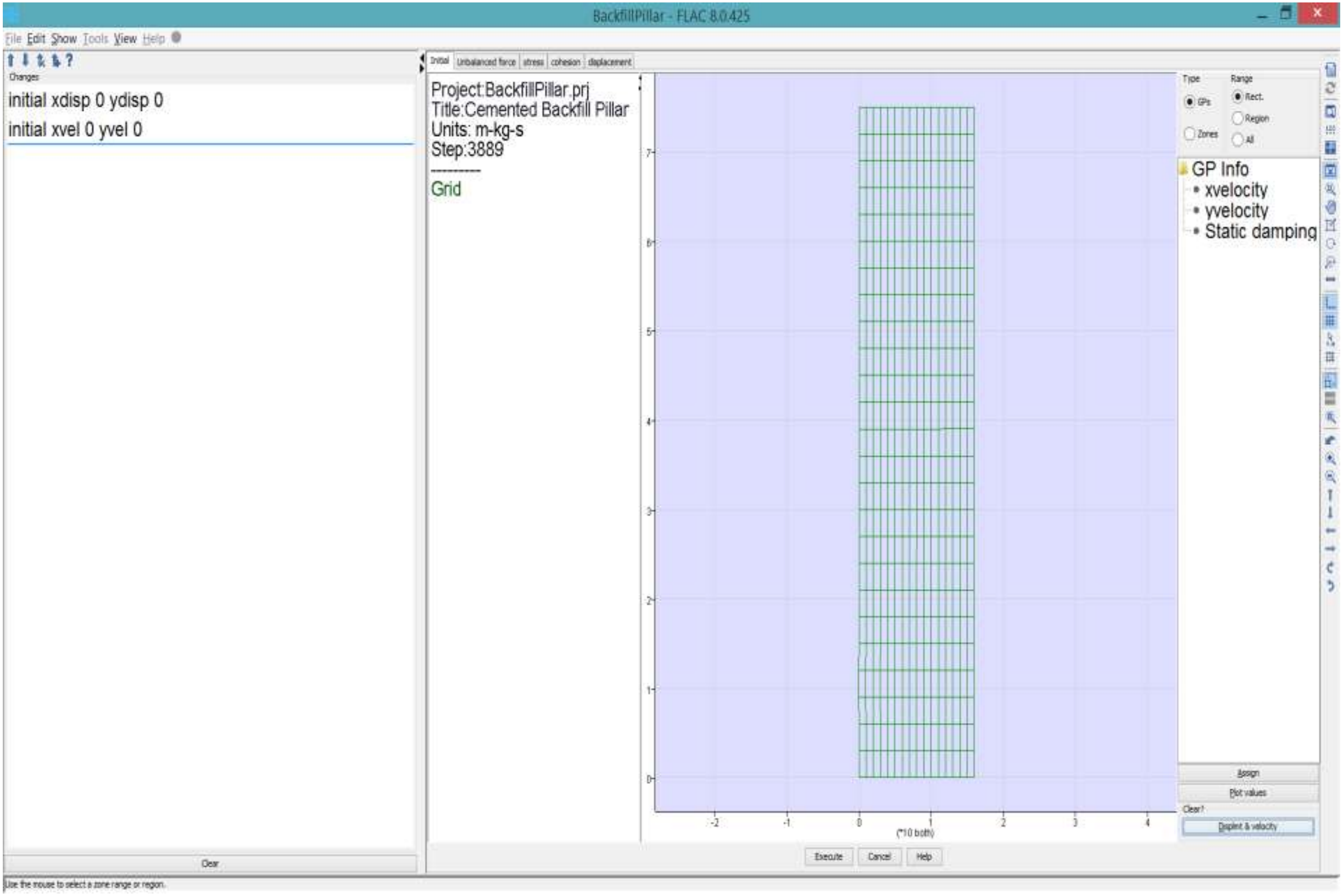
Step 3-8 Return to the [In Situ]/[Fix] tool. Select [Free] and click the mouse on gridpoints i=1, j=4 and i=1,j=5. Press [Execute] to send the command to *FLAC*.



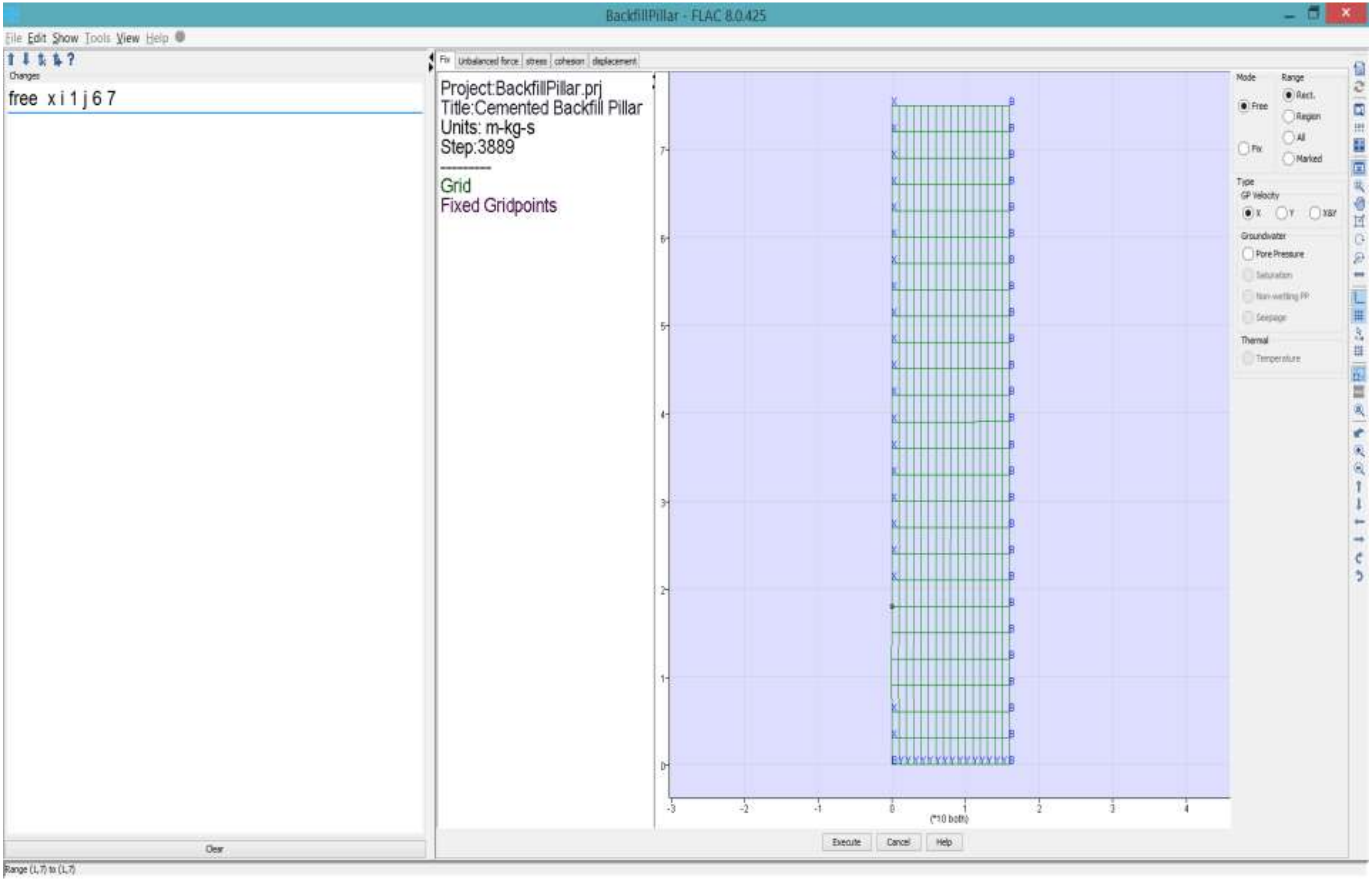
Step 3-9 Select [Run]/[Solve] and calculate the equilibrium state for the 6 m blast height.



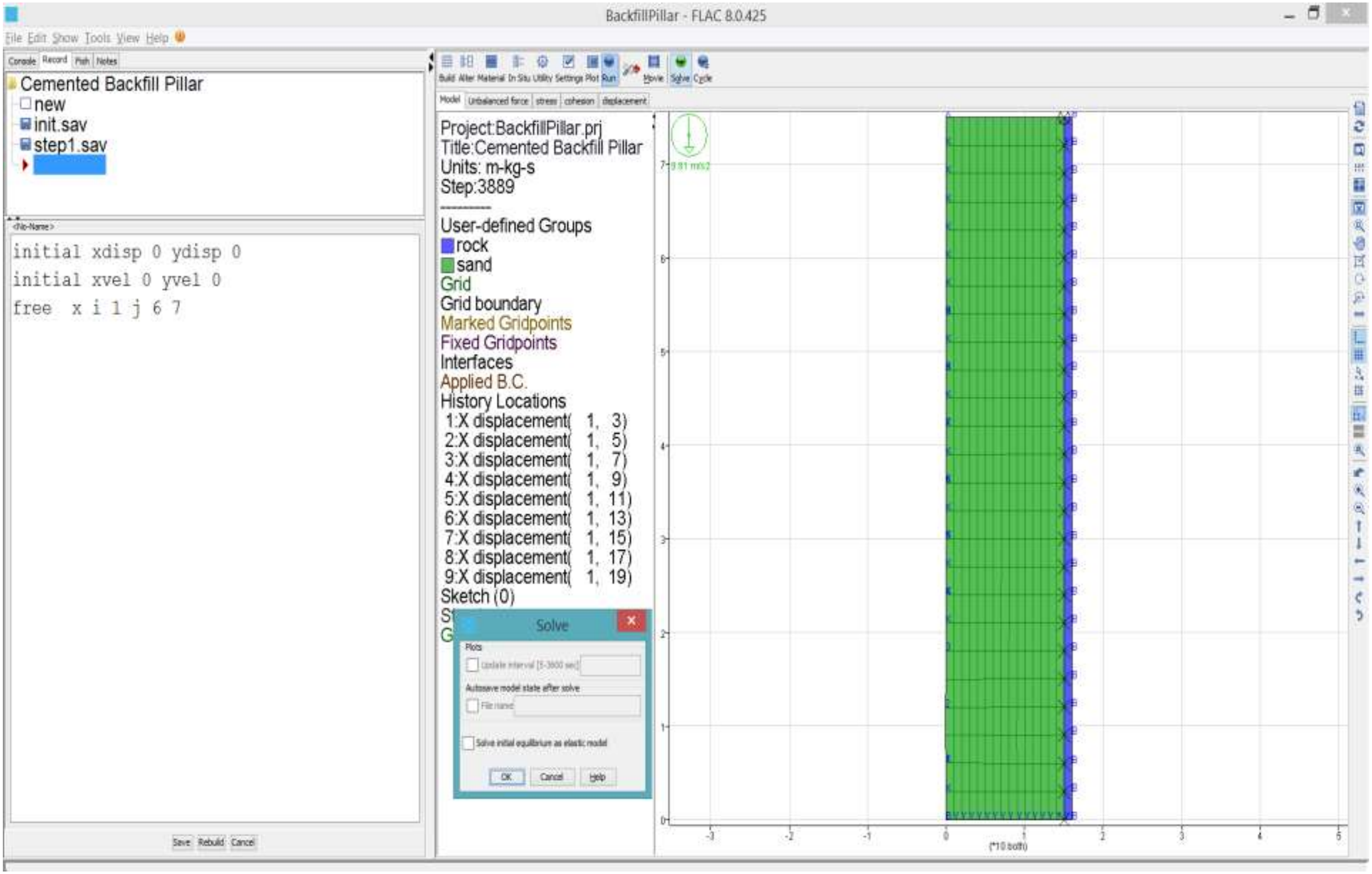
Step 3-10 Plot displacement vectors in the [Plot]/[Model] tool. Save the state as step1.sav.



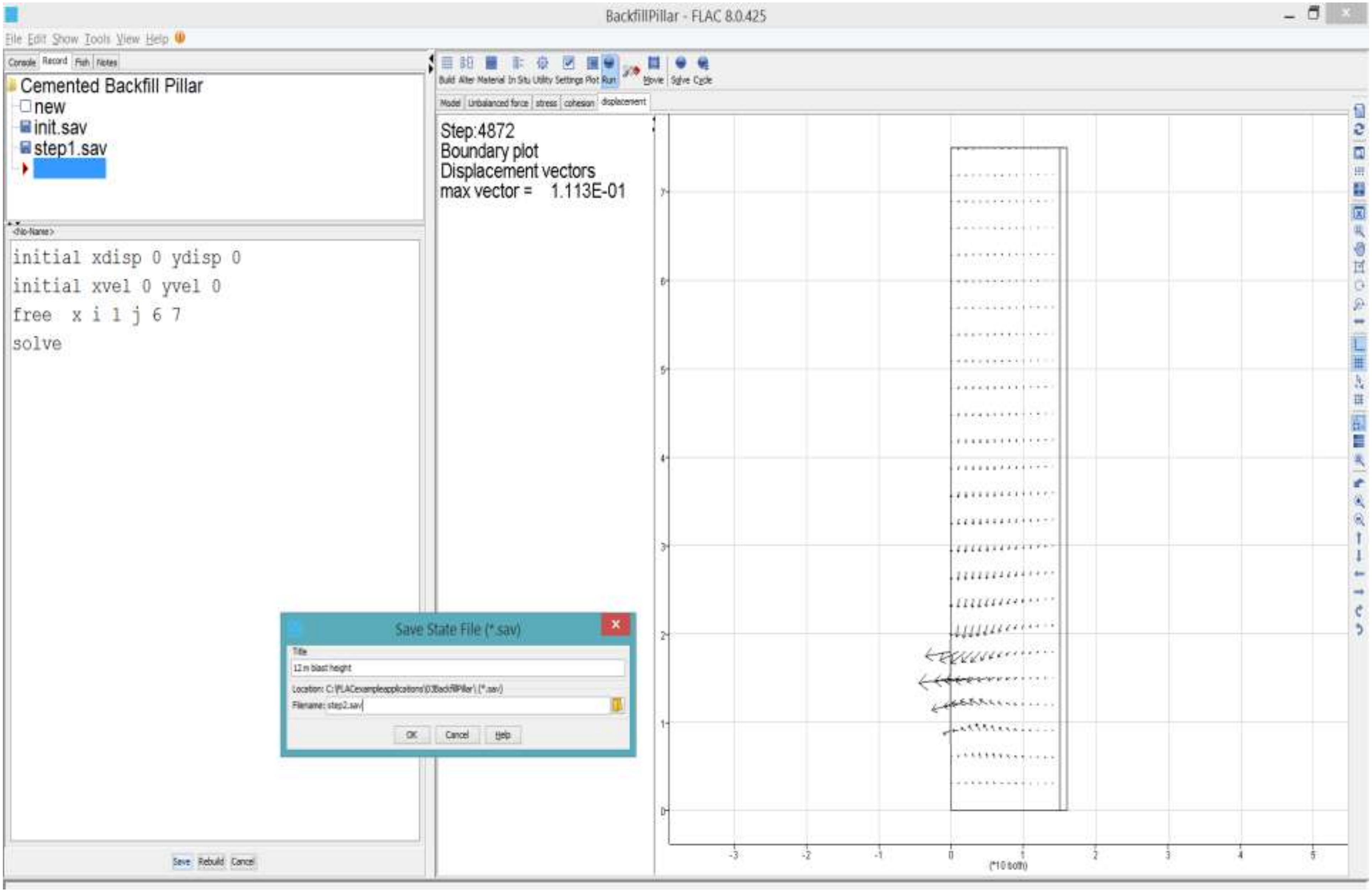
Step 3-11 Enter the [In Situ]/[Initial] tool and select [Clear? Dispmt & velocity] to initialize displacements and velocities.



Step 3-12 Return to the [In Situ]/[Fix] tool. Select [Free] and click the mouse on gridpoints i=1, j=6 and i=1,j=7. Press [Execute] to send the command to *FLAC*.

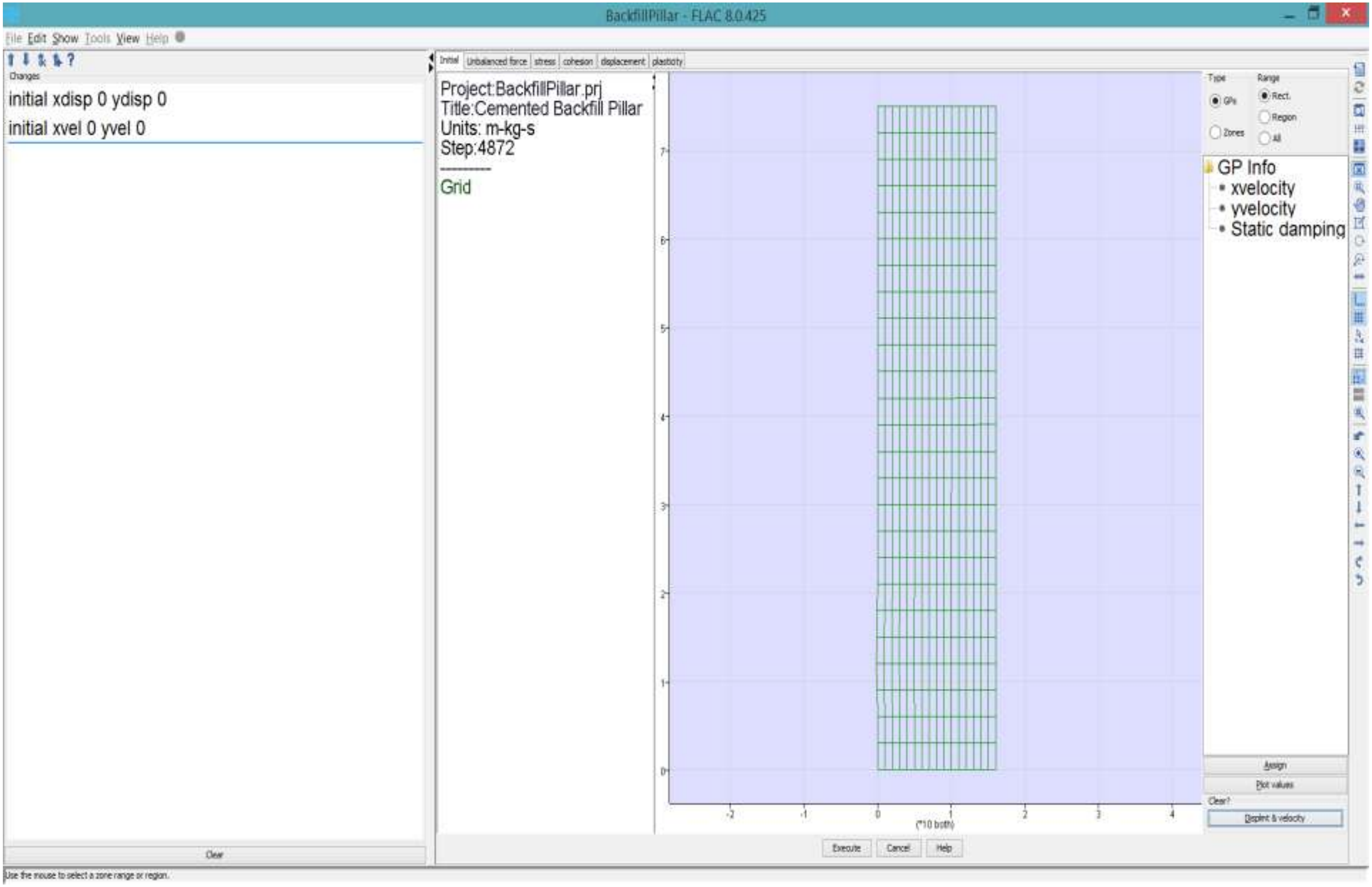


Step 3-13 Select [Run]/[Solve] and calculate the equilibrium state for the 12 m blast height.



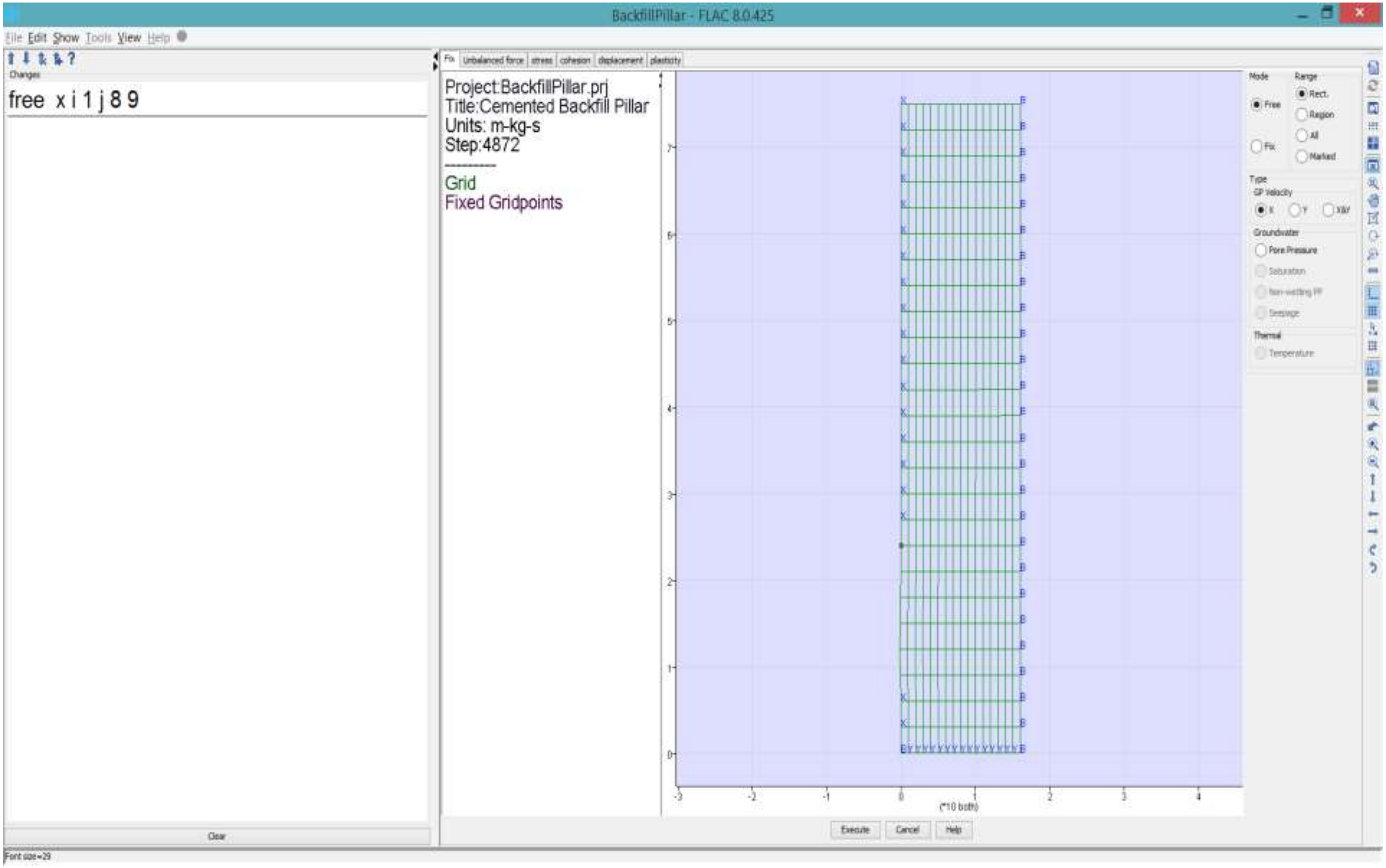
Step 3-14 Plot displacement vectors in the [Plot]/[Model] tool. Save the state as step2.sav.

.

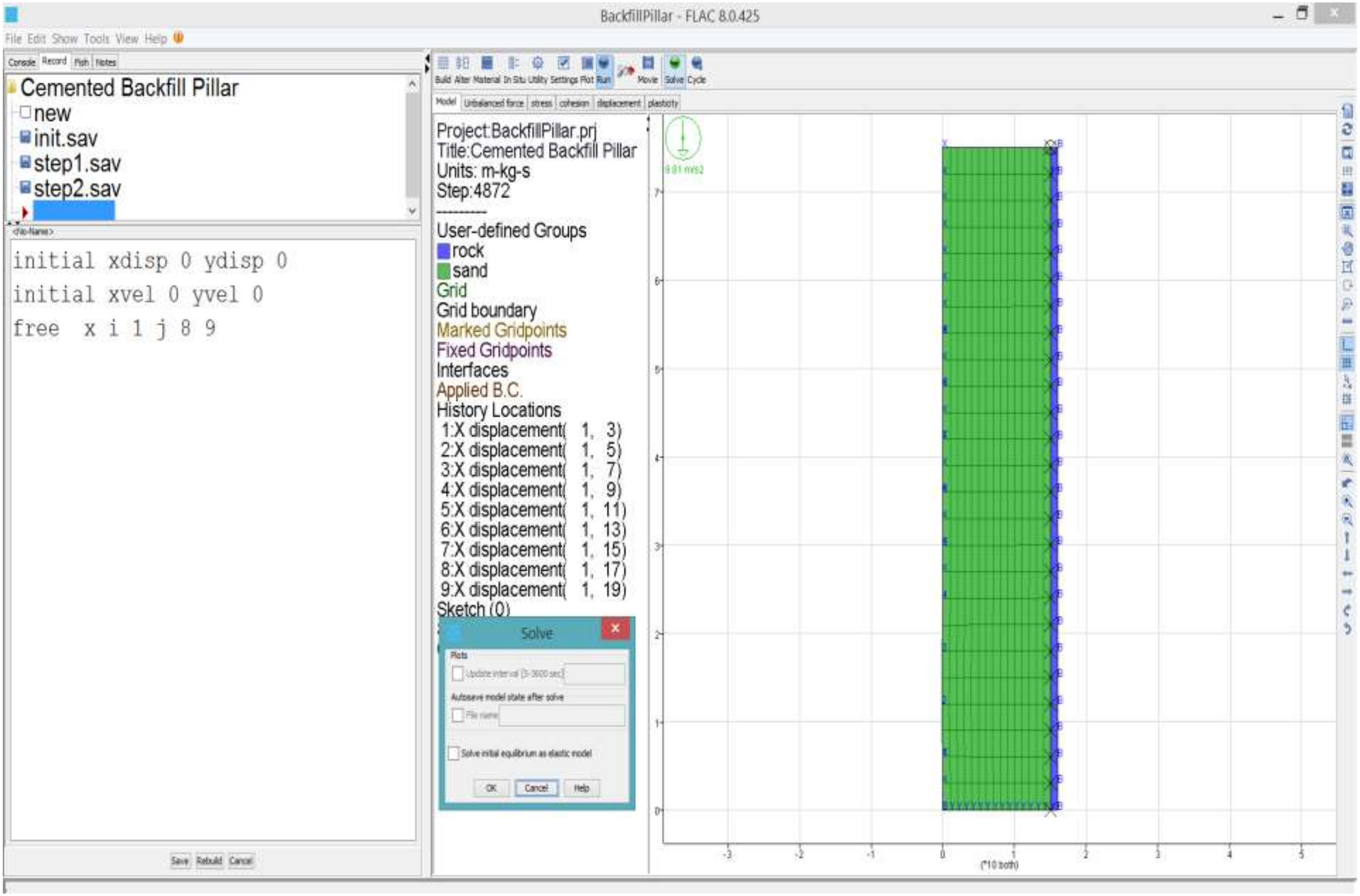


Step 3-15 Enter the [In Situ]/[Initial] tool and select [Clear? Dispmt & velocity] to initialize displacements and velocities.

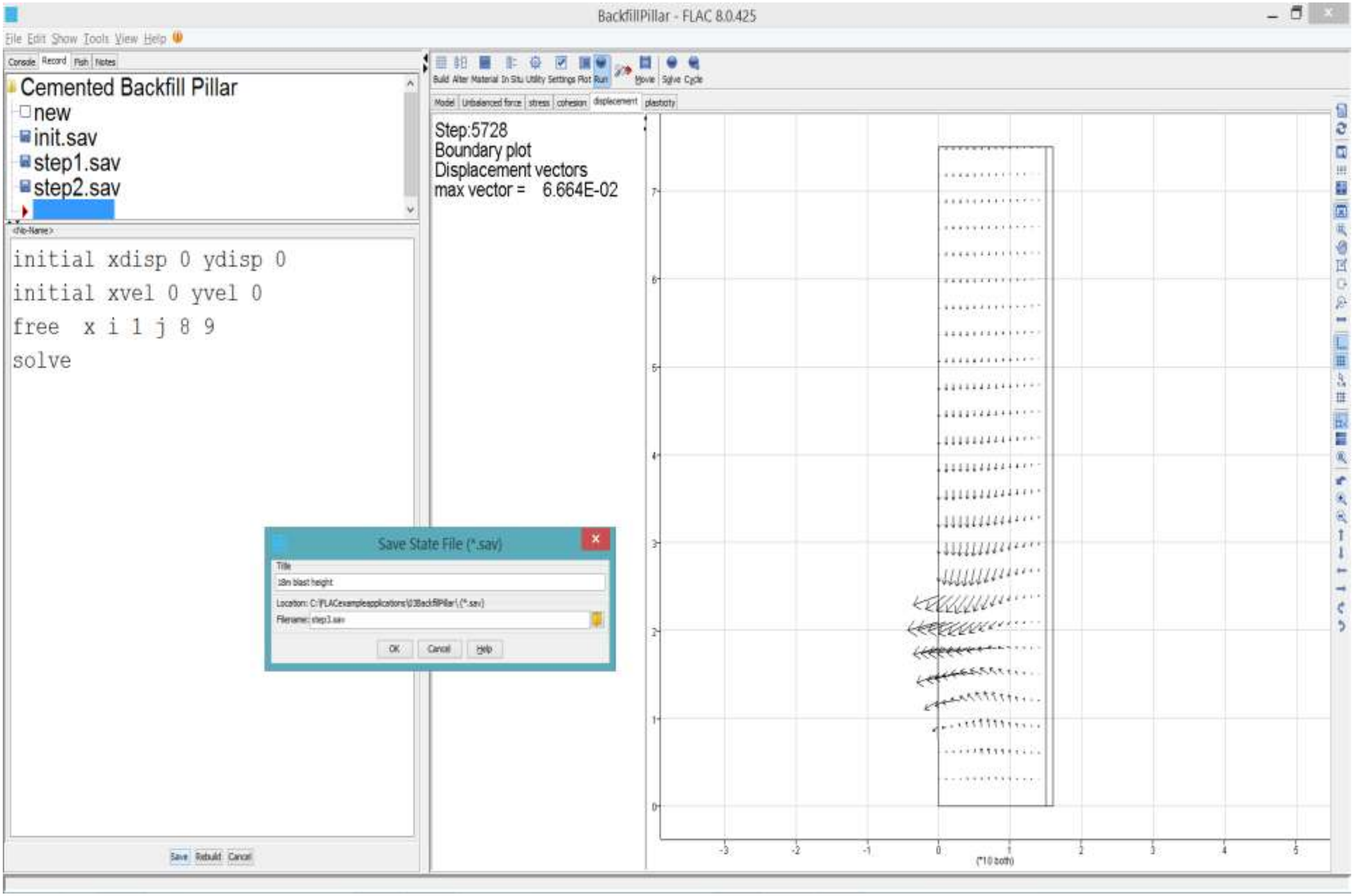
.



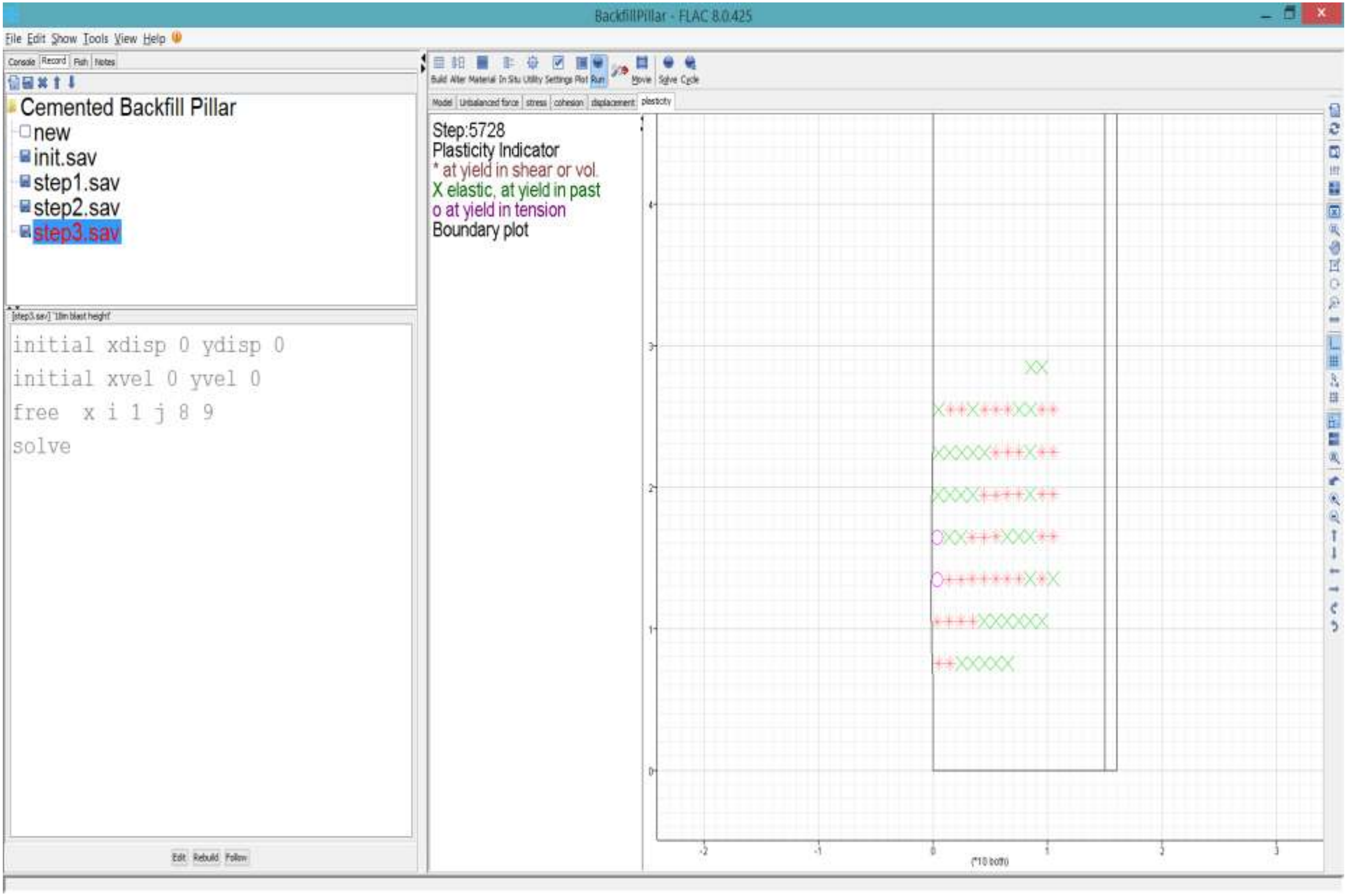
Step 3-16 Return to the [In Situ]/[Fix] tool. Select [Free] and click the mouse on gridpoints i=1, j=8 and i=1,j=9. Press [Execute] to send the command to *FLAC*.



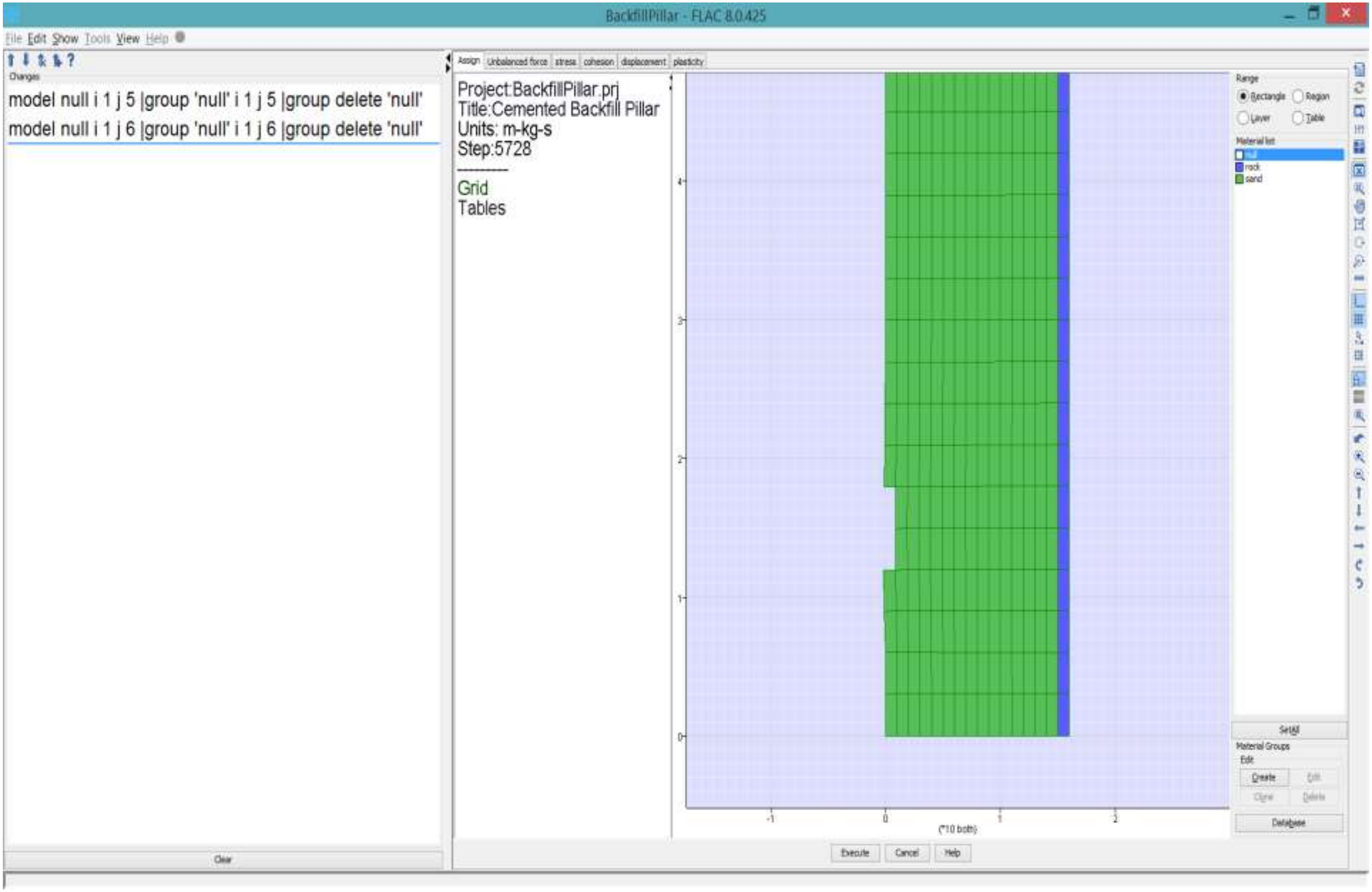
Step 3-17 Select [Run]/[Solve] and calculate the equilibrium state for the 18 m blast height.



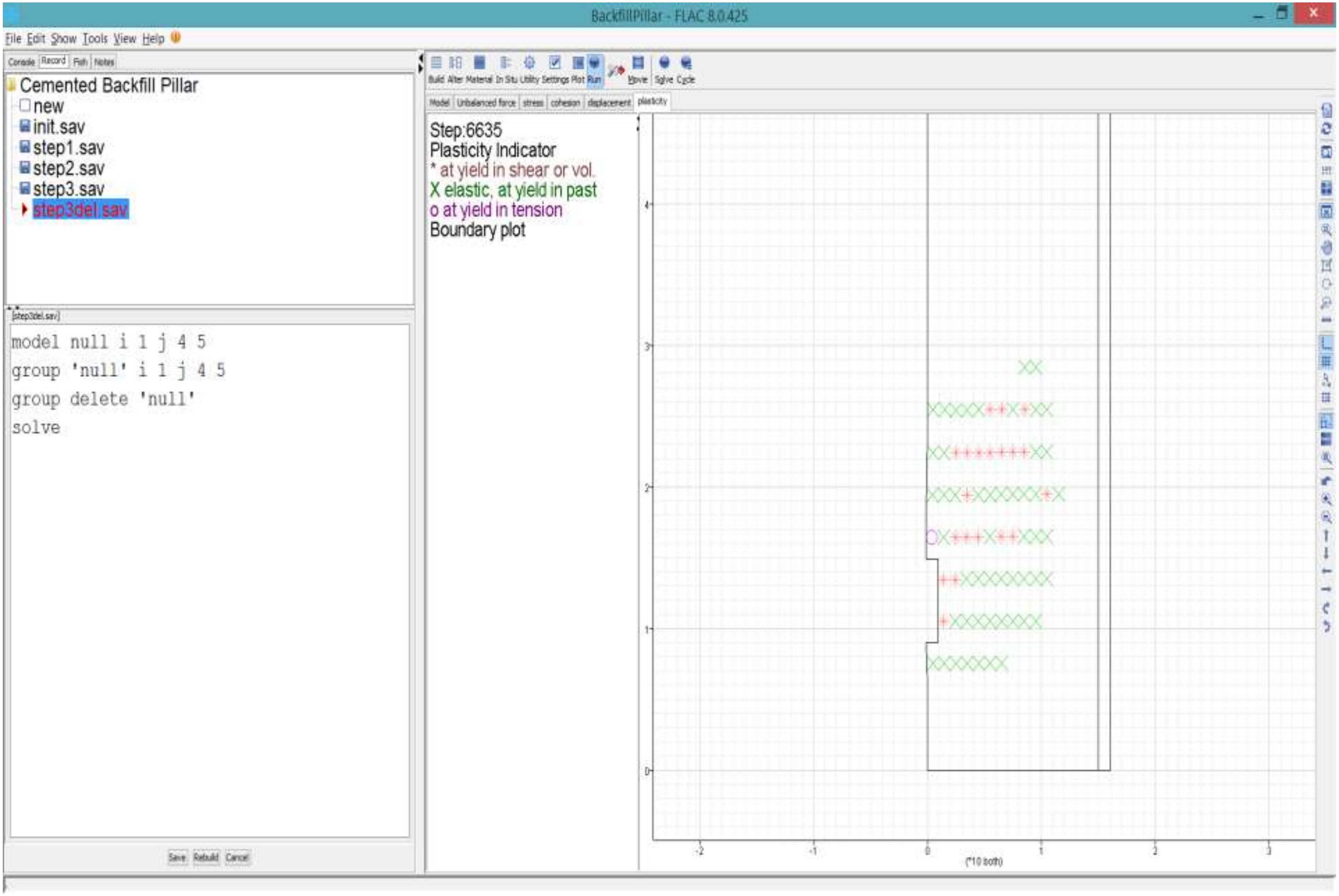
Step 3-18 Plot displacement vectors in the [Plot]/[Model] tool. Save the state as step3.sav.



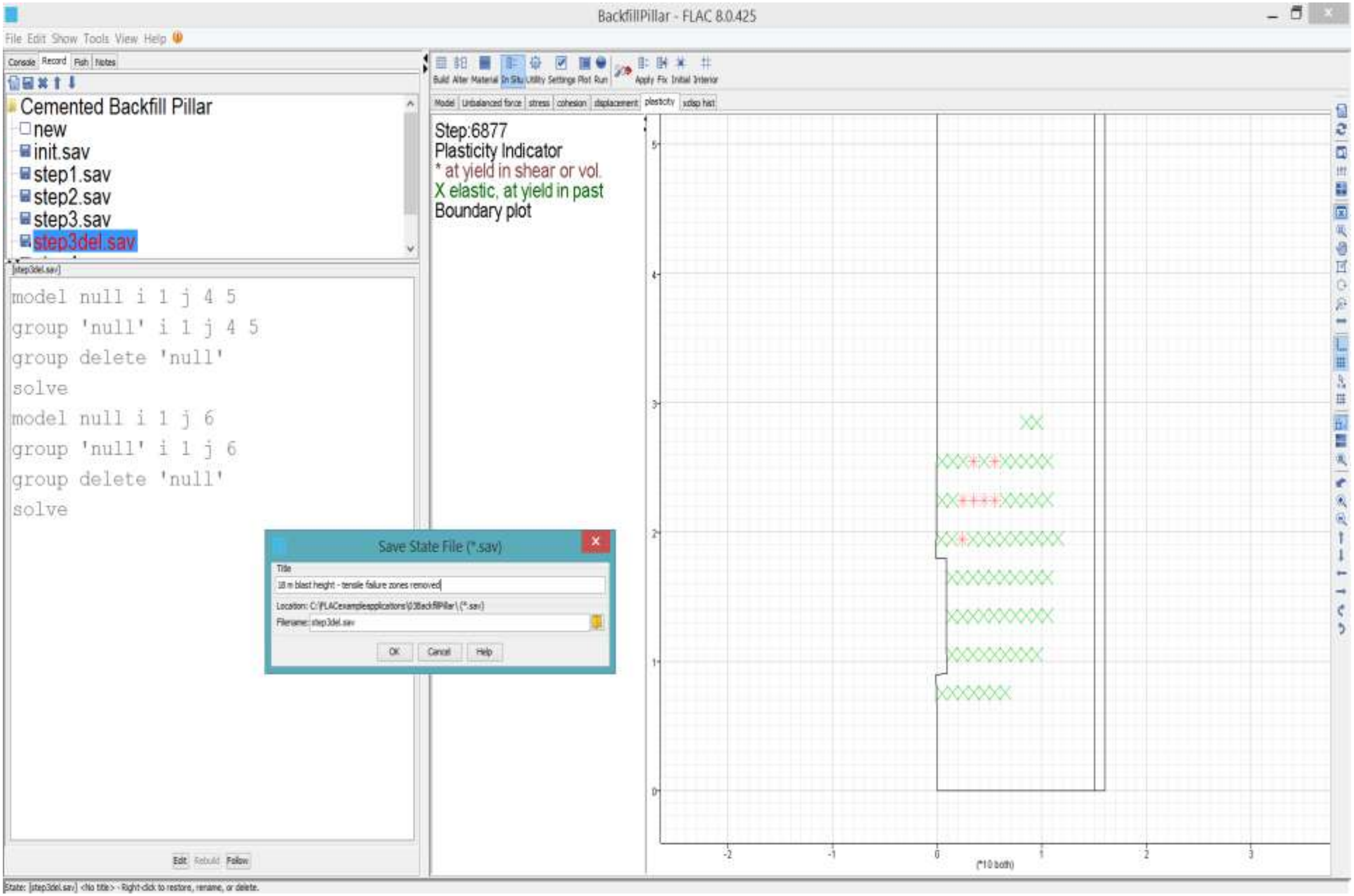
Step 3-19 Create a plasticity indicator plot in the [Plot]/[Model] tool. Note zones that fail in tension.



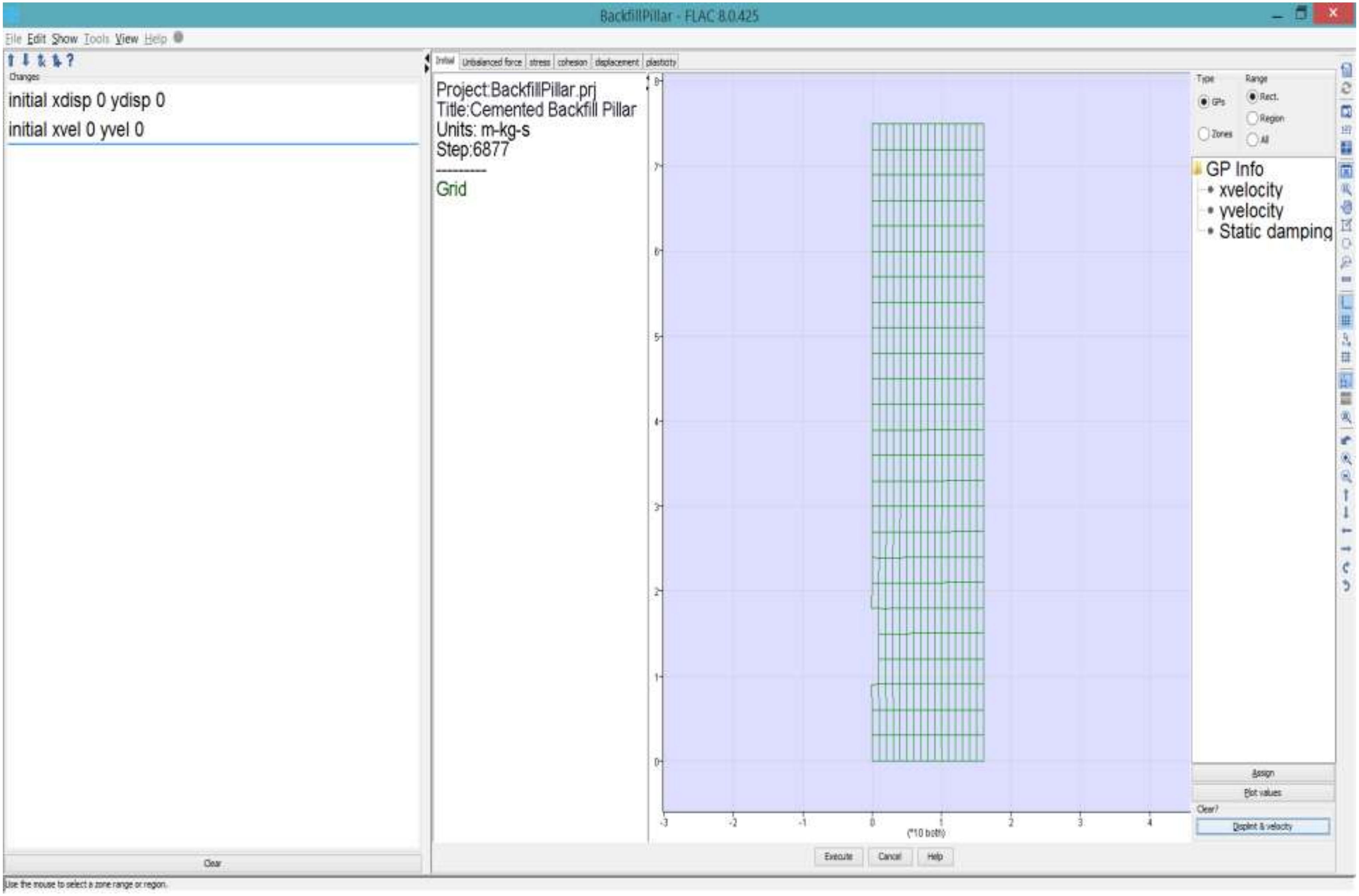
Step 3-20 Enter the [Material]/[Assign] tool. Select [null] and slick on the zones that have failed in tension. Press [Execute] to send the commands to *FLAC.*



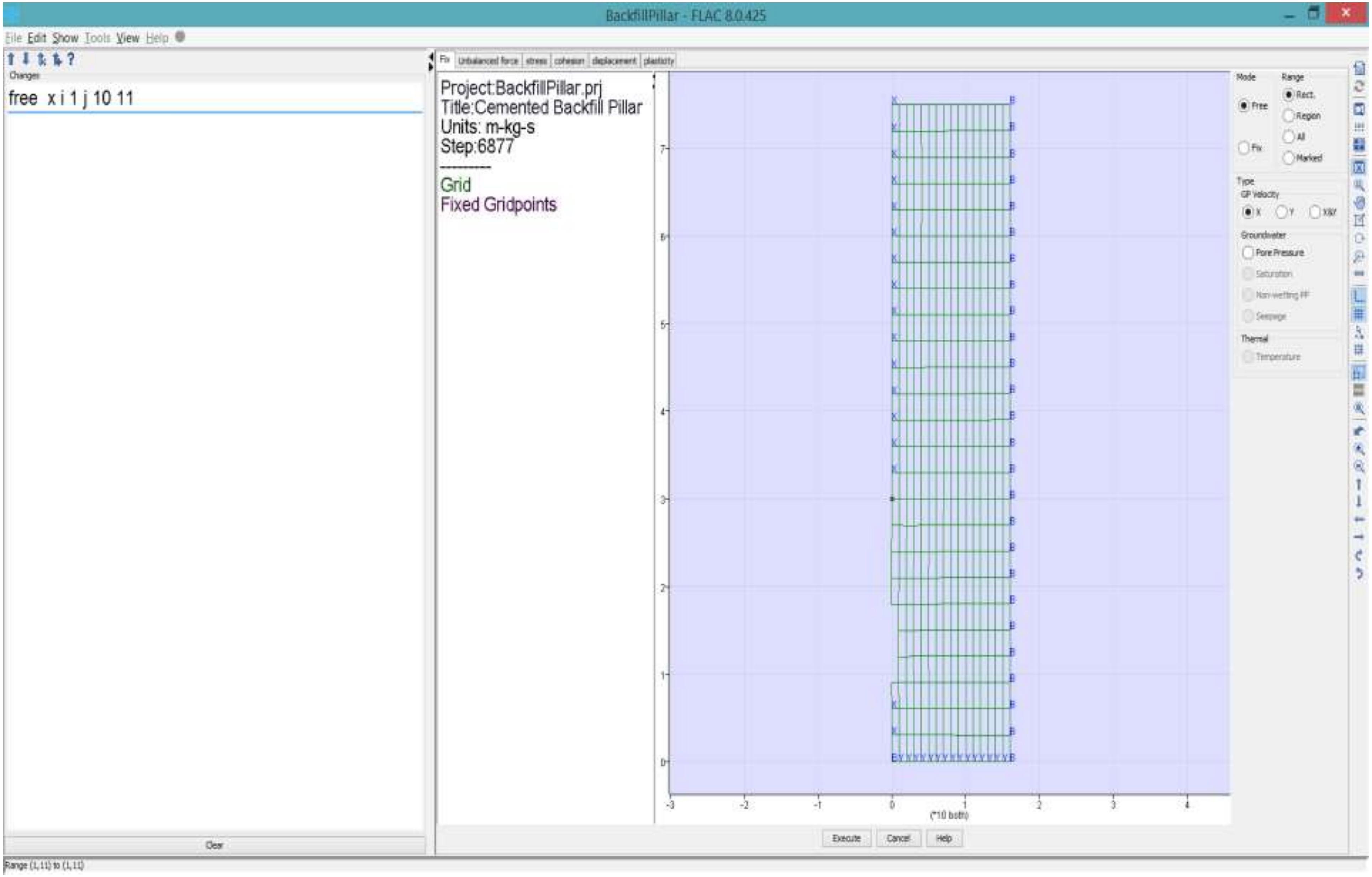
Step 3-21 Execute the [Run]/[Solve] tool and recheck the plasticity indicator plot for additional zones failing in tension.



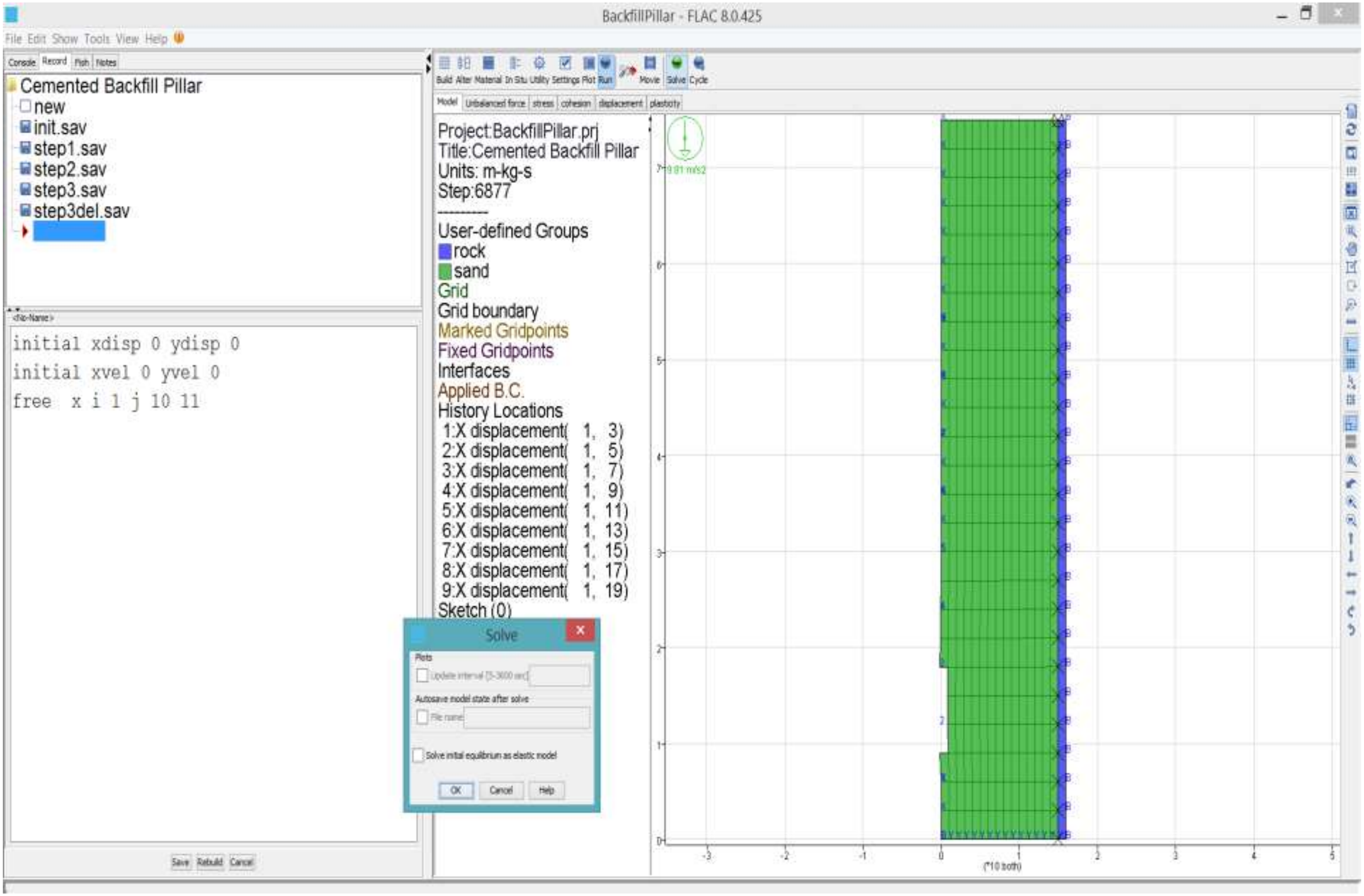
Step 3-22 Repeat thee [Run]/[Solve] command**.** No additional zones fail in tension. Save the state as step3del.sav.



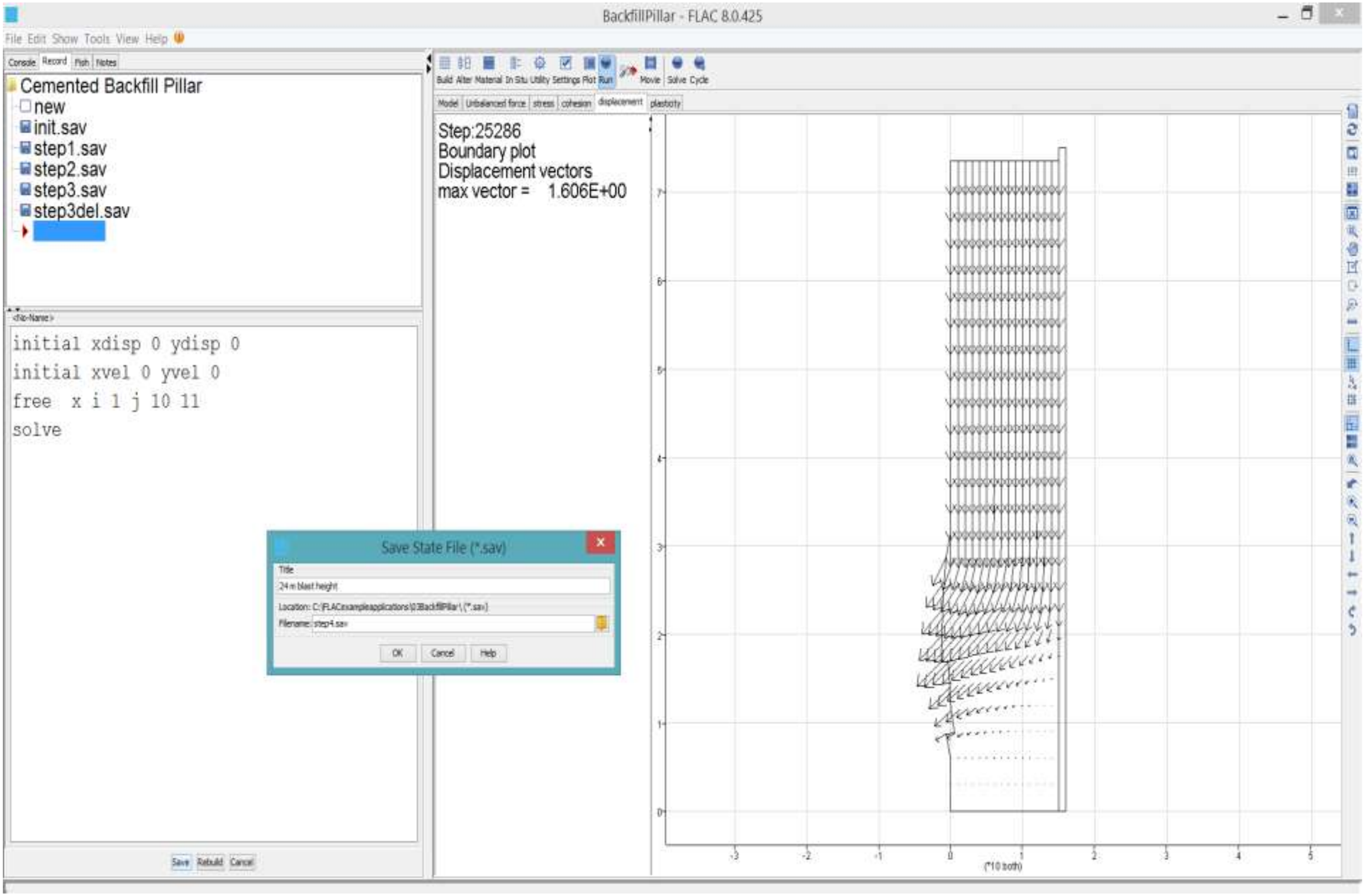
Step 3-23 Enter the [In Situ]/[Initial] tool and select [Clear? Dispmt & velocity] to initialize displacements and velocities.



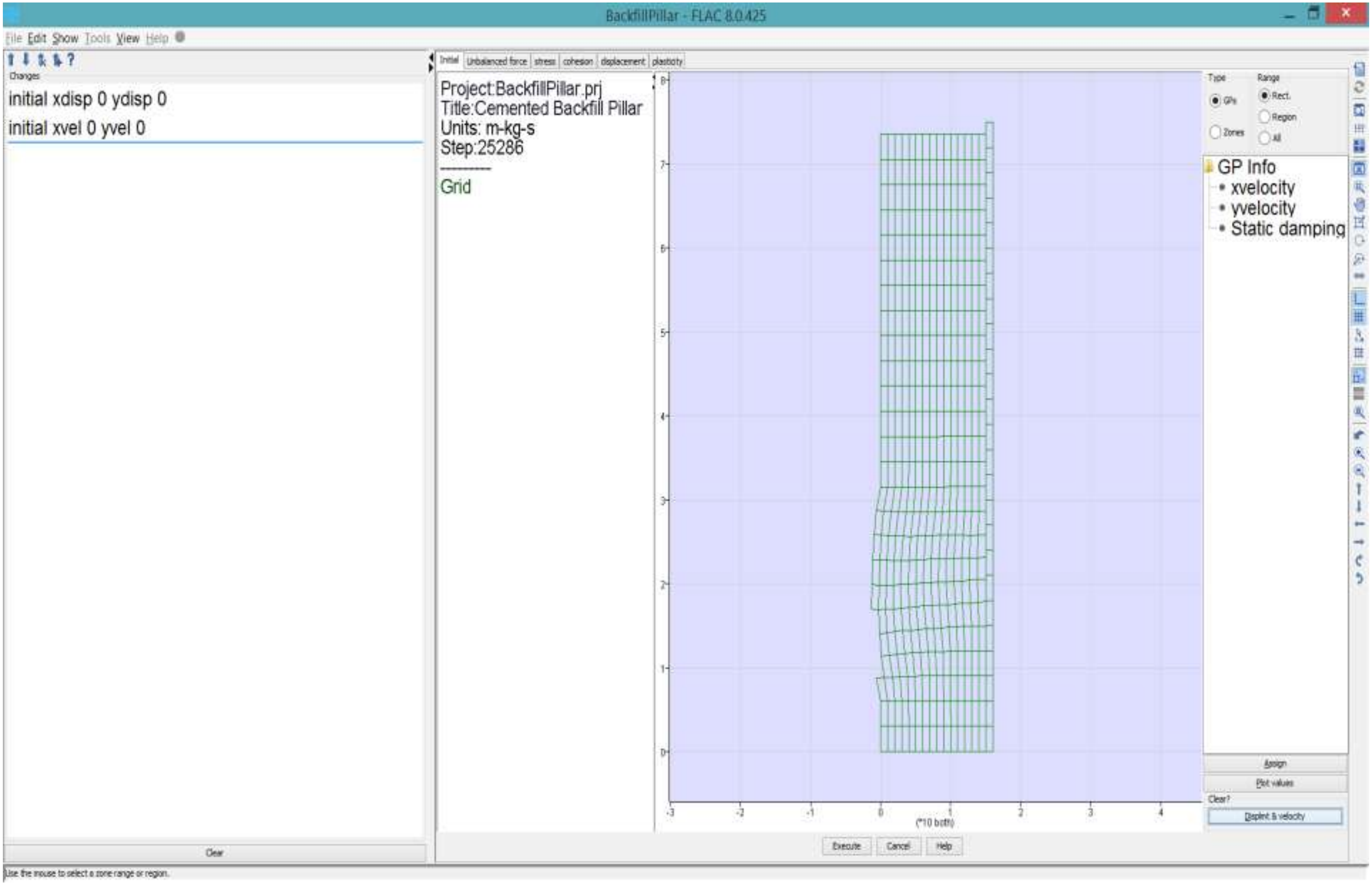
Step 3-24 Return to the [In Situ]/[Fix] tool. Select [Free] and click the mouse on gridpoints i=1, j=10 and i=1,j=11. Press [Execute] to send the command to *FLAC*.



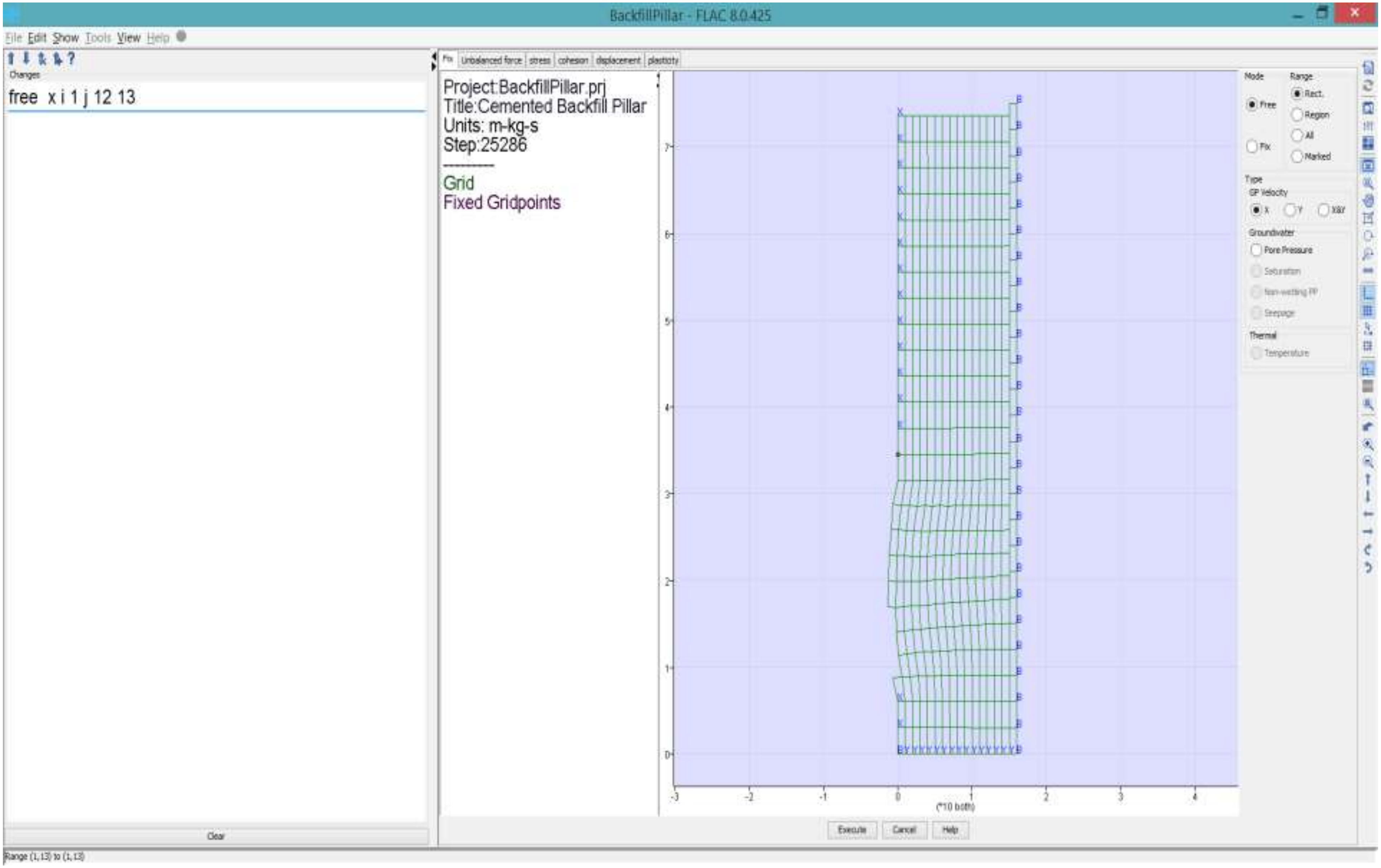
Step 3-25 Select [Run]/[Solve] and calculate the equilibrium state for the 24 m blast height.



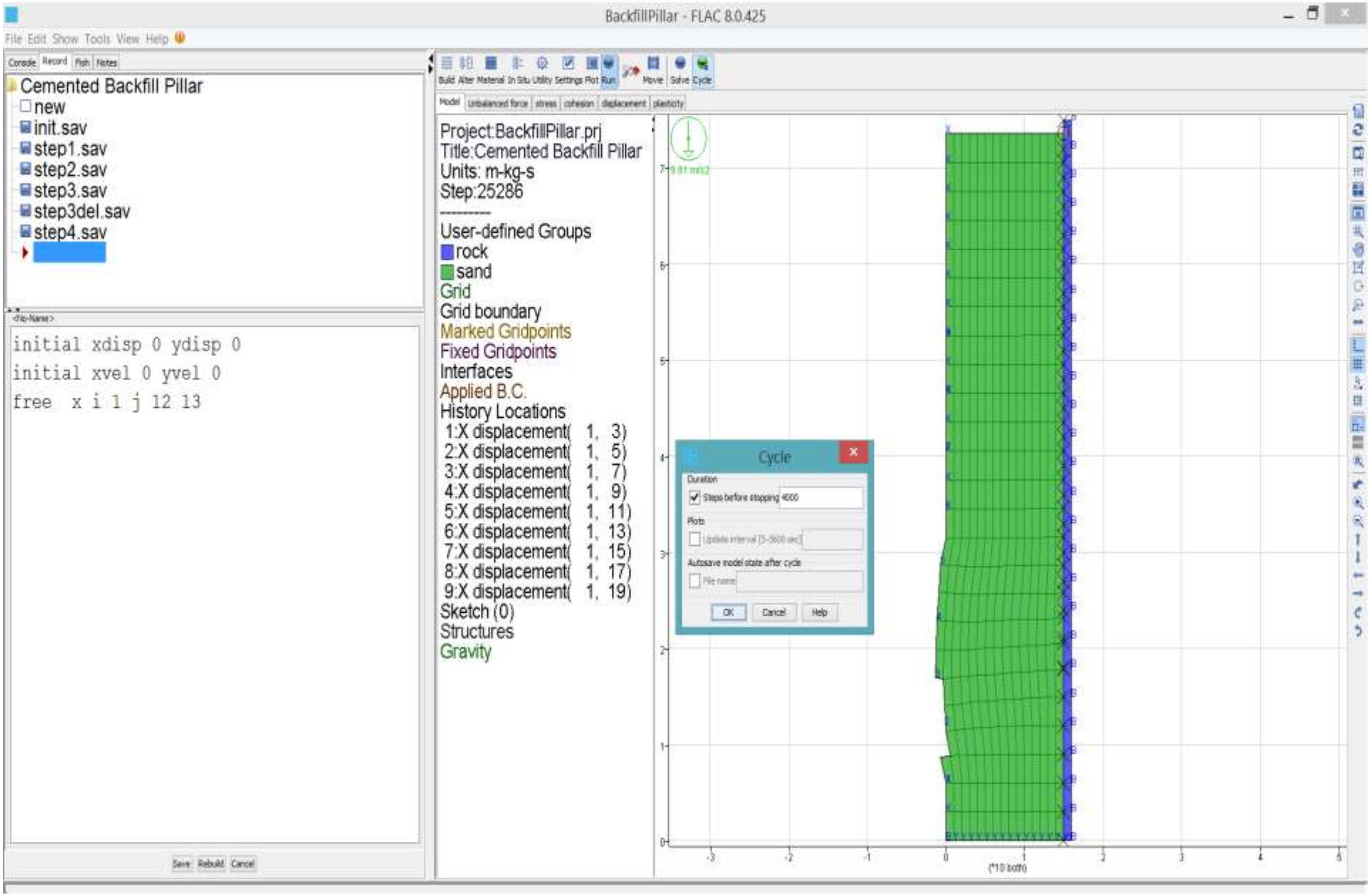
Step 3-26 Plot displacement vectors in the [Plot]/[Model] tool. Save the state as step4.sav.



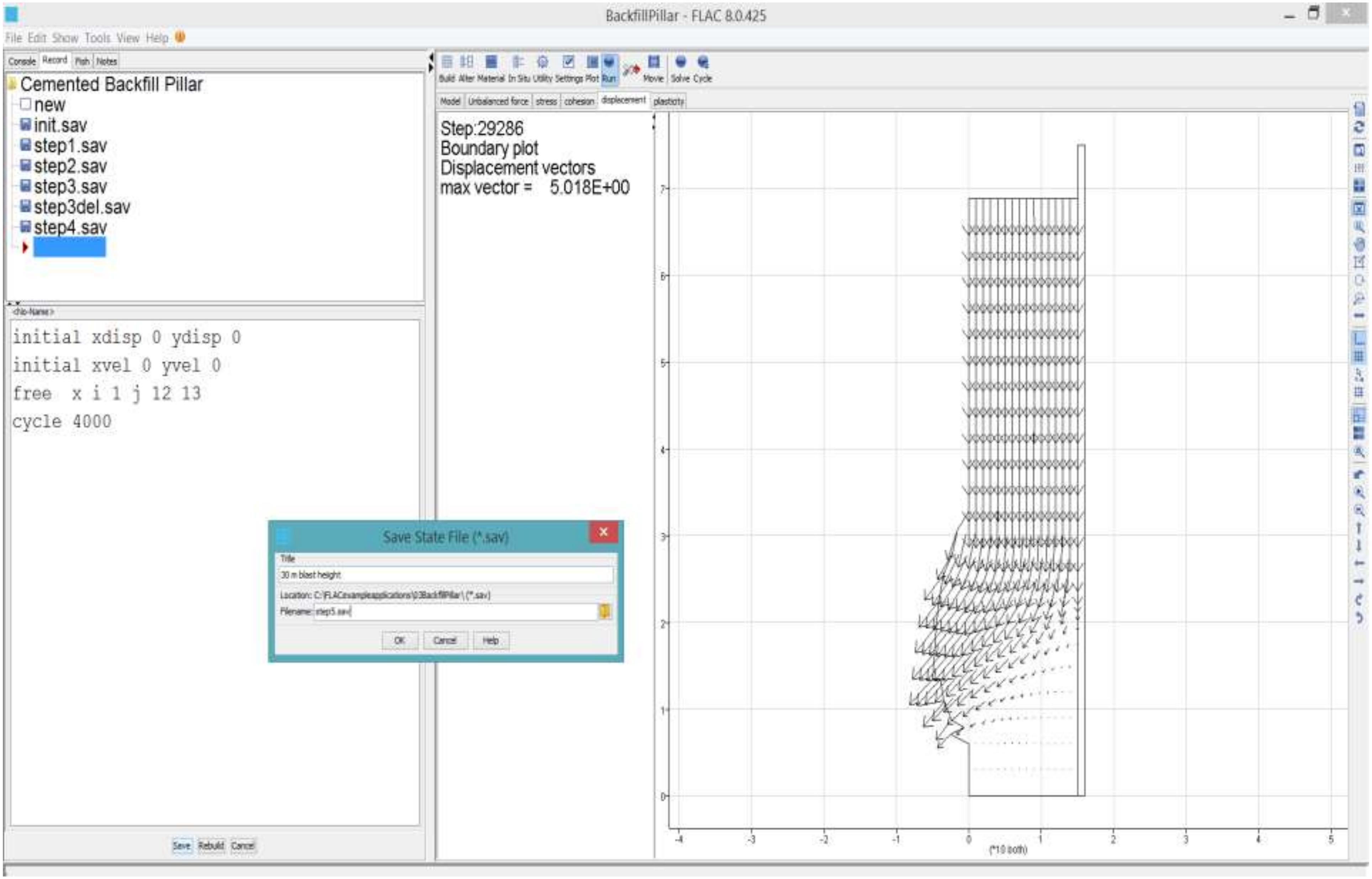
Step 3-27 Enter the [In Situ]/[Initial] tool and select [Clear? Dispmt & velocity] to initialize displacements and velocities.



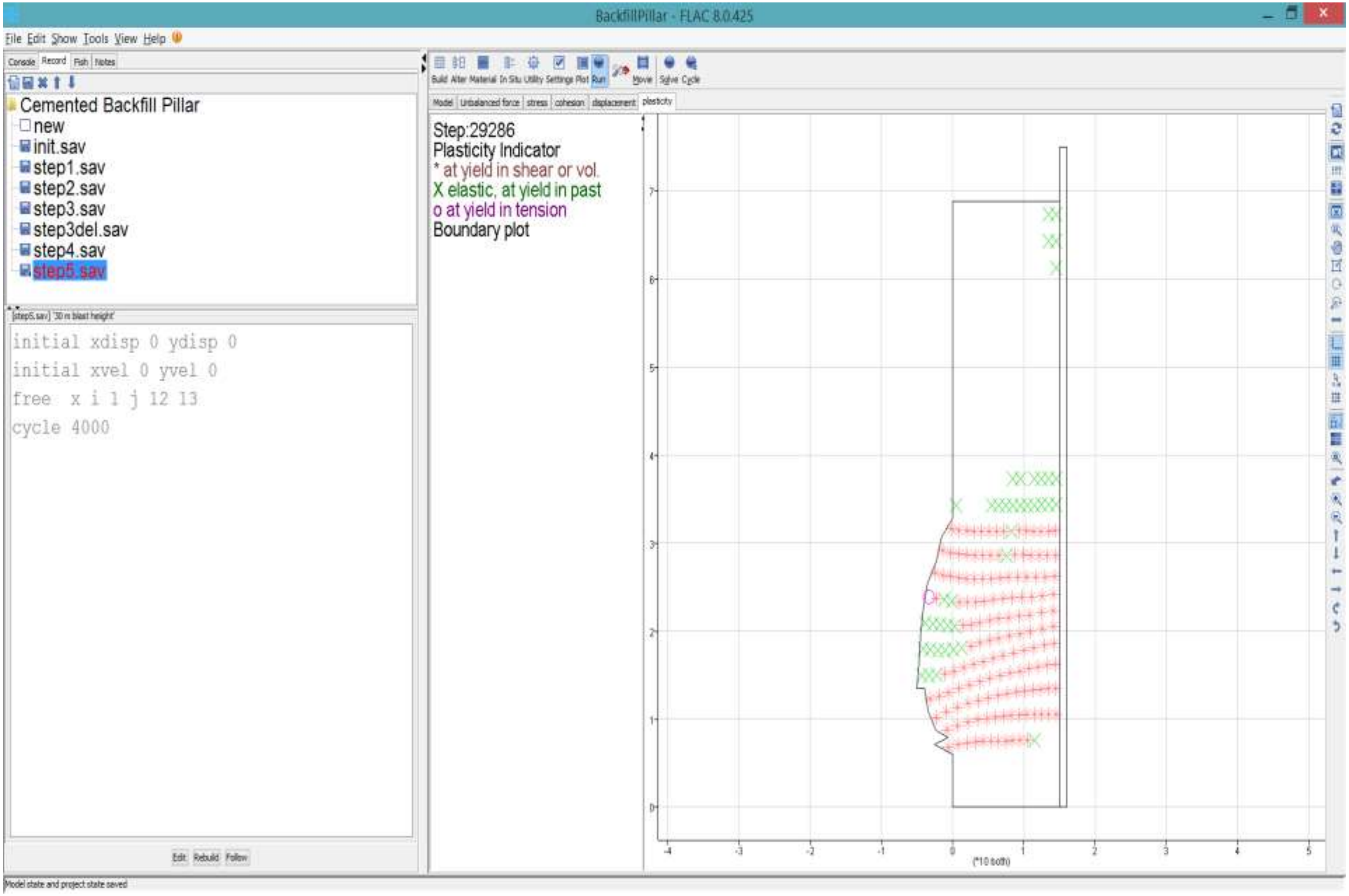
Step 3-28 Return to the [In Situ]/[Fix] tool. Select [Free] and click the mouse on gridpoints i=1, j=12 and i=1,j=13. Press [Execute] to send the command to *FLAC*.



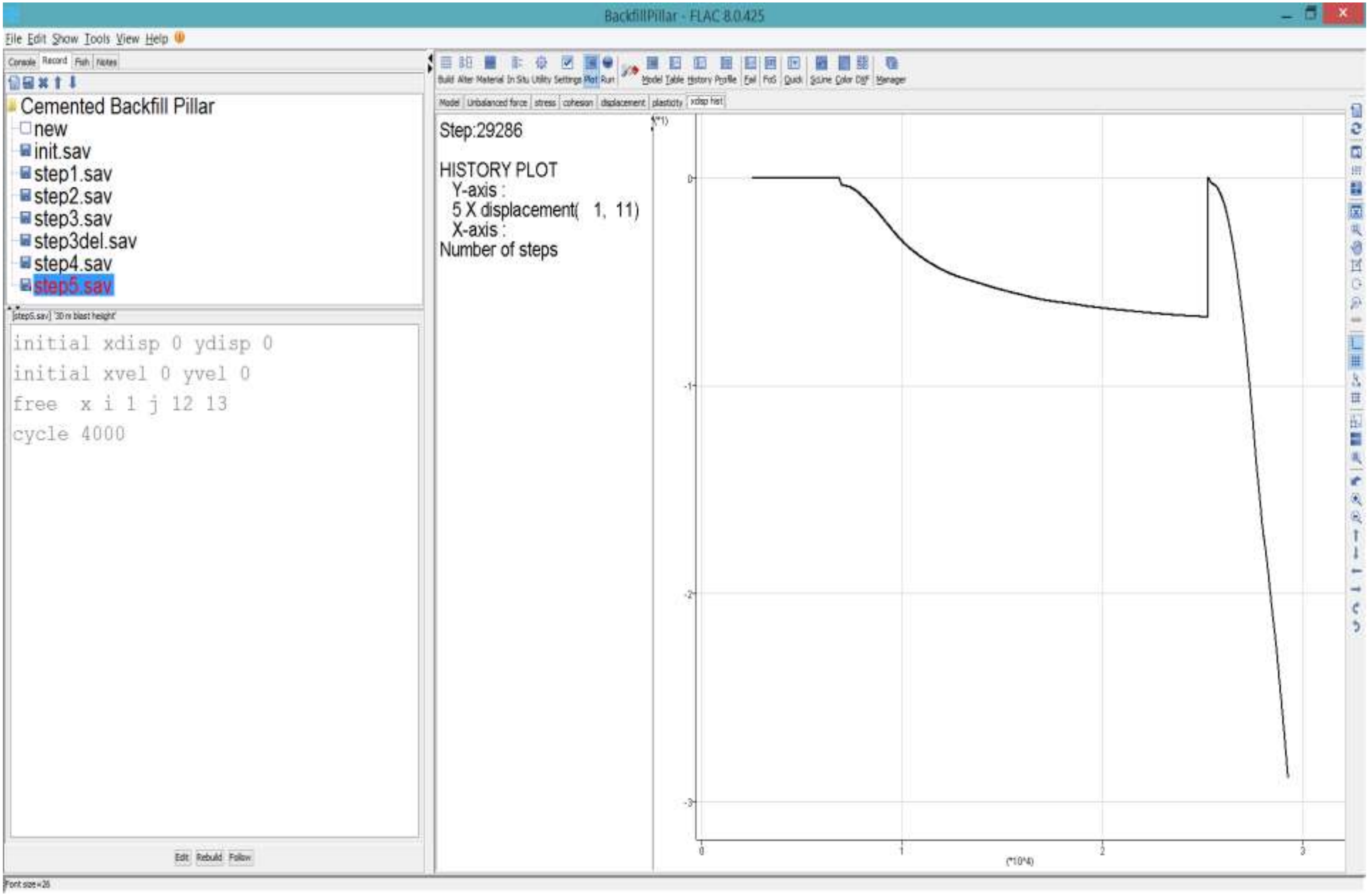
Step 3-29 Select [Run]/[Cycle] and perform a calculation for 4000 steps for the 30 m blast height.



Step 3-30 Plot displacement vectors in the [Plot]/[Model] tool. Save the state as step5.sav.



Step 3-31 Plot plasticity indicators in the [Plot]/[Model] tool.



Step 3-32 Enter the [Plot]/[History] tool. Plot the x-displacement history at the 30 m height on the pillar wall.