VERSAT - S2D & VERSAT - D2D

Version 2019.10.3 with PSPA & $[C]_{a.b}$ modules

STATIC AND DYNAMIC 2-DIMENSIONAL FINITE ELEMENT ANALYSIS OF CONTINUA

- **Microsoft .NET Framework 3.5**
- **Microsoft Windows XP, & Windows 7 & Windows 10**

Volume 2: USER MANUAL

Website: http://www.wutecgeo.com

LIMITATION OF LIABILITY

The following terms and conditions with regard to limitation of liability must be accepted to proceed with the use of VERSAT-2D.

TABLE OF CONTENTS

Page

1. INTRODUCTION

VERSAT-2D is a software package consisting of three computer programs, namely, VERSAT-2D Processor (the Processor), VERSAT-S2D and VERSAT-D2D.

It is noted that these three components of VERSAT-2D function independently. Interactions among them take place through data files saved in a Windows Explorer file folder. A brief description for each program is provided below.

VERSAT-2D Processor (the Processor) is a Windows based graphic interface program. It serves as a pre and post processor for VERSAT-S2D and VERSAT-D2D. The program is used to generate a finite element mesh, define soil zones, assign material properties, define boundary conditions, assign pressure loads, and generate input data for VERSAT-S2D & VERSAT-D2D. The program can also display and plot results from analyses such as stresses, displacements, accelerations, pore-water pressures, and a deformed mesh.

VERSAT-S2D is a computer program for static 2D plane-strain finite element analyses of stresses, deformations, and soil-structure interactions. The static analyses can be conducted using stress-strain constitutive relationships from linear elastic model to elasto-plastic models, i.e., Mohr-Coulomb model and Von-Mises model. This program can also be used to compute or determine static pre-existing stresses for use in a subsequent dynamic finite element analysis. Main features of VERSAT-S2D are:

- Linear elastic model
- Von-Mises model for CLAY type
- Mohr-Coulomb model for CLAY, SILT and SAND types
- Stress level dependent stiffness parameters
- External load applications
- Staged construction by adding layers
- Staged excavation by removing layers
- Pore water pressure application
- Calculation of stresses and deformations caused by strain-softening of soils
- Simulation of sheet pile wall and anchors
- Updated Lagrangian analysis
- Factors of safety calculation
- Gravity on and off
- Calculation of pre-existing stresses for use in a dynamic analysis using VERSAT-D2D
- 4-node, 6-node and 8-node solid elements to represent soils

- 2-node line elements to represent sheet pile walls (beam) or anchors (bar/truss)
- Use of any consistent units and sign conventions

VERSAT-D2D is a computer program for dynamic 2D plane-strain finite element analyses of earth structures subjected to dynamic loads from earthquakes, machine vibration, waves or ice actions. The dynamic analyses can be conducted using linear, or nonlinear, or nonlinear effective stress method of analysis. The program can be used to study soil liquefaction, earthquake induced deformation and dynamic soil-structure interaction such as pile-supported bridges. Main features of VERSAT-D2D are:

- Application of horizontal, or horizontal and vertical, ground accelerations at a rigid base
- Application of horizontal outcropping ground velocities at a viscous/elastic base
- Application of a load-time-history at any nodal points
- Global force equilibrium enforced at all time
- Linear elastic model
- Non-linear hyperbolic stress-strain model for SAND type and CLAY type
- Non-linear strain-softening model for SILT type
- Stress level dependent stiffness parameters
- Mohr-Coulomb failure criterion
- **VERSAT-SAND Model** for liquefaction analysis of sandy soils
- **VERSAT-SILT Model** for liquefaction analysis of silty soils (*new since 2016*)
- **E**ffective **S**tress **M**ethod of analysis, including the effect of earthquake induced pore water pressure on soil strength and on soil stiffness
- Calculation of ground deformations caused by soil liquefaction
- Calculation of factor of safety against soil liquefaction or strain-softening
- Simulation of sheet pile wall and anchors
- Updated Lagrangian analysis
- Gravity off for **VERSAT-1D module**
- Free-field stress boundary
- 4-node solid elements to represent soils
- 2-node line elements to represent sheet pile walls (beam) or anchors (bar/truss)
- Use of any consistent units and sign conventions
- Probabilistic Seismic Performance Analysis (PSPA) including subduction Interface (e.g., Magnitude 9 or M9) and Non-Interface (e.g., M7) earthquakes (new since 2018)
- Local viscous damping (a, b) for stiff structures

FLOW CHART TO ILLUSTRATE TYPICAL STEPS IN A DYNAMIC ANALYSIS:

Step 1:

VERSAT-2D Processor

- Generate a finite element mesh (2D)
- Define soil zones and material parameters
- Define structural elements and parameters
- Define boundary conditions, apply pressure loads
- Generate input data for VERSAT-S2D or VERSAT-D2D

Step 2:

VERSAT-S2D

- Conduct static stress analyses
- Conduct static deformation analyses
- Conduct static pore water pressure applications
- Conduct static soil-structure interaction analyses
- Determine pre-existing stresses

Step 3:

VERSAT–D2D

- Conduct dynamic linear analyses with or without gravity
- Conduct dynamic nonlinear analyses of earth structures subjected to dynamic loads from earthquakes, machine vibration, waves or ice actions
- Conduct dynamic nonlinear effective stresses analyses to determine soil liquefaction (SAND & SILT) and earthquake induced deformations
- Conduct dynamic analyses of soil-structure interaction such as pile-supported bridges

Step 4:

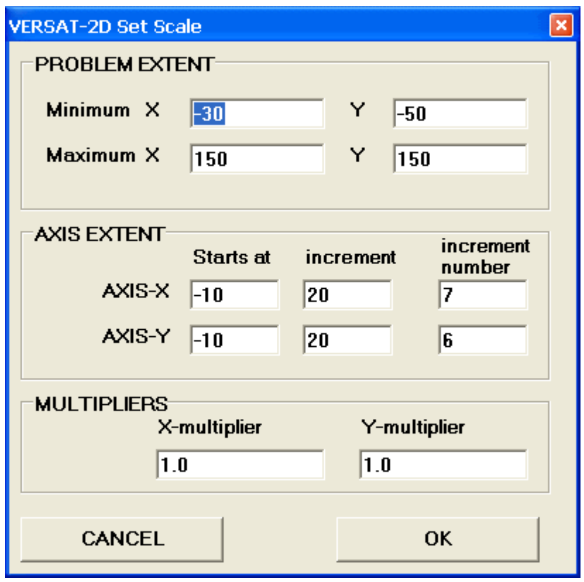
VERSAT-2D Processor

- View and print finite element mesh including node, element numbers
- View and print soil material zones (Color printer required)
- View and print results of stresses or displacements (peak and instant)
- View and print acceleration values (peak and instant), if applicable
- View and print analysis results of shear strains (peak and instant) or pore water pressure or factor of safety against liquefaction
- View and print deformed mesh
- Save graphics as image files (.emf, or .gif, or .jpeg etc)

2. PREPARING DATA FOR A NEW PROBLEM

2.1. Create a Setting File for Model Scale, Axis Scale and Others

1. The model scale (problem extent) and axis details (axis extent) are defined using 'S*et scale'* under **SETTING**. The Processor window only shows the part of a model within the X and Y extent defined herein.
2. The defined model scale and axis details are saved using '*Save Setting'* under **SETTING**. A setting file can be retrieved next time using '*Load Setting'* under **SETTING**. A setting file has a file type (extension) of log.



3. In addition to the Problem Extent and Axis Extent, a setting file also saves the following:
    a. Font size of text and numbers (node and element etc) shown on the model
    b. Position, font size and content of all texts added through using '*Draw text' command* under **TOOLS**.

    Note: Setting file is a text file. Therefore, all items in a setting file (font sizes, text position, color ranges, etc) can be edited outside the Processor and then re-loaded into the Processor to take effect.
4. The x-multiplier and y-multiplier is used to scale up or scale down a model by multiplying the X and Y coordinates by x-multiplier and y-multiplier, respectively. For examples, the multipliers can be used to convert the model between different units. Normally they are kept as one.

2.2. Create a Finite Element Mesh

The following commands may be used in creating a finite element mesh:

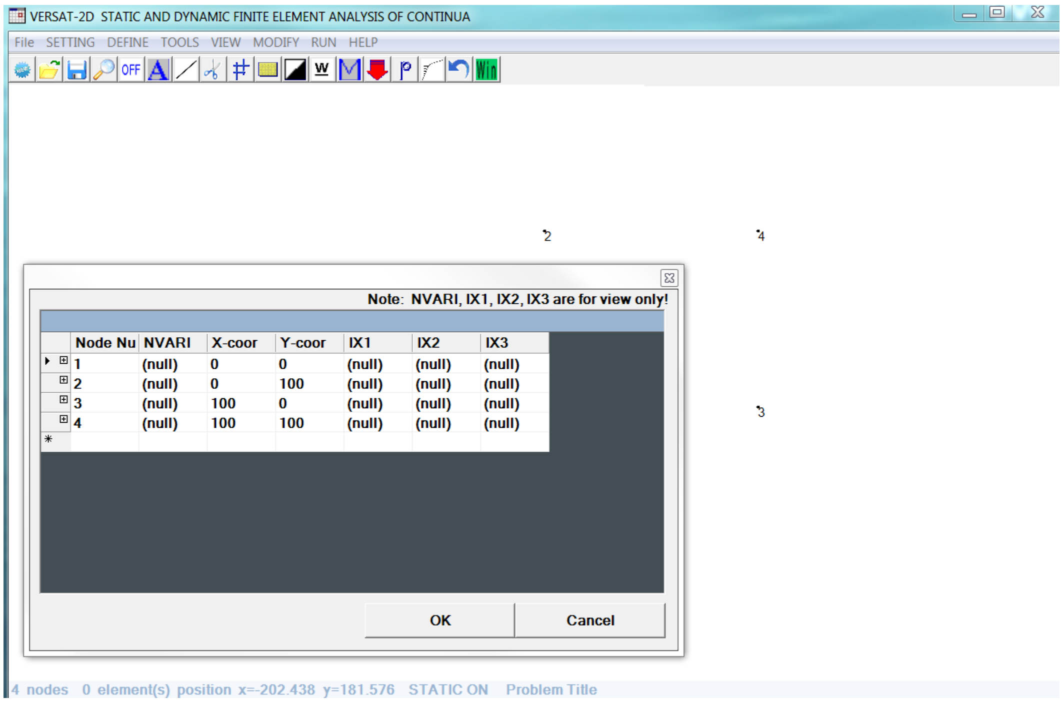
- Select '*New*' under **FILE** to start a new problem.
- Use '*Finite element node/grid points*' under **DEFINE** to pre-define the four nodes with their exact X and Y coordinates.
- Choose '*Draw finite element grid*' under **TOOLS** to create finite elements. Enter number of elements for each grid side and press OK. Then click on screen for 1^{st}, 2^{nd}, 3^{rd}, and 4^{th} nodes that form the finite element grid.
- Use '*Draw line'* under **TOOLS** to create a boundary within a finite element mesh. It is required to organize the mesh after this modification.
- Choose 'C*ut/remove finite elements*' under **TOOLS** to remove finite elements that are not needed. Then click on screen to select four points. The nodes and elements within a block enclosed by the four points will be removed. It is required to organize the mesh after this modification.
- To organize a modified mesh: choose '*Clear duplicate nodes/elements'* under **MODIFY, t**hen perform '*Sort node(..)*/*element(..)*' under **MODIFY**. This process renumbers nodes and elements of the model.
- If needed, use '*Finite element node/grid points'* under **DEFINE** to change X and/or Y coordinates of a node.
- If needed, use '*move grid line'* under **TOOLS** to move a grid line within a finite element mesh.

An example for creating a finite element mesh follows:

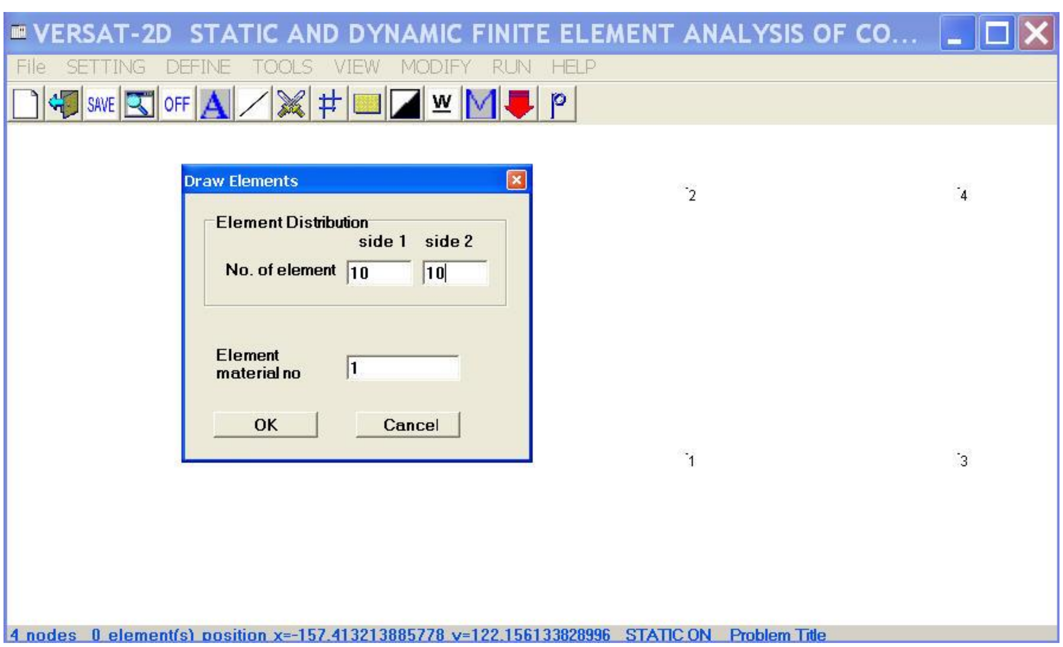
[Very important notes: During the course of mesh construction, it is recommended that data be saved, using '*Save Data*' under **FILE**, as frequently as possible. You may find it necessary to save data, from various stages in a mesh construction, using different names such as con1, con2, con3, con4, etc., because the Processor does not provide "*undo*" functions.]

Step 1 Use '*Finite element node/grid points*' under **DEFINE** to enter Node No. and (X, Y) Coordinates of the four nodes (Note: NVARI, IX1, IX2, IX3 not required; they will be defined by later)

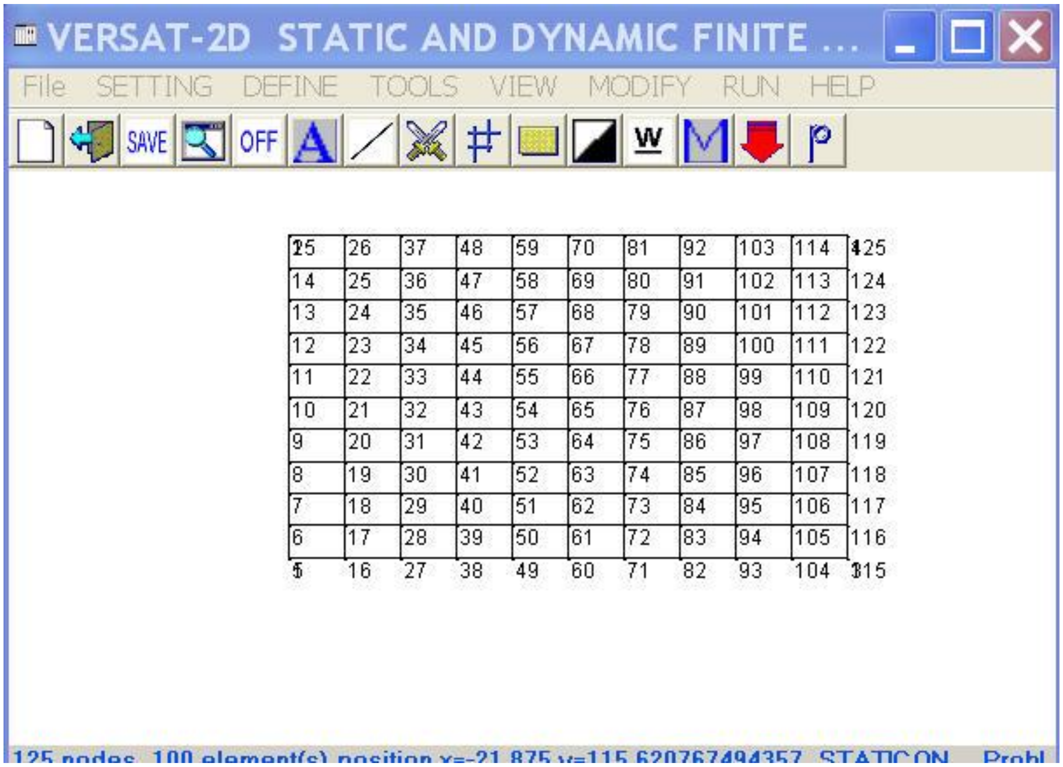
This screen shot of the main Window is from: VERSION 2019.05.08
(Some Windows in this User Manual are for an earlier version)



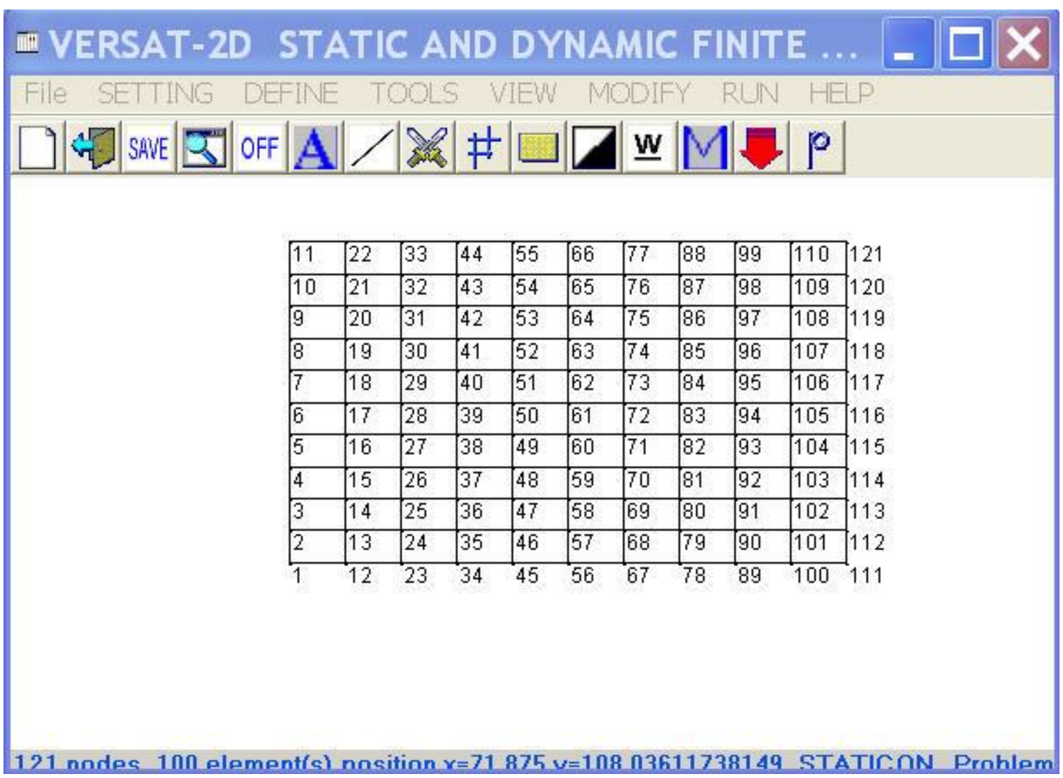
Step 2 Use '*Draw finite element grid*' under **TOOLS** to create a 10x10 finite element grid



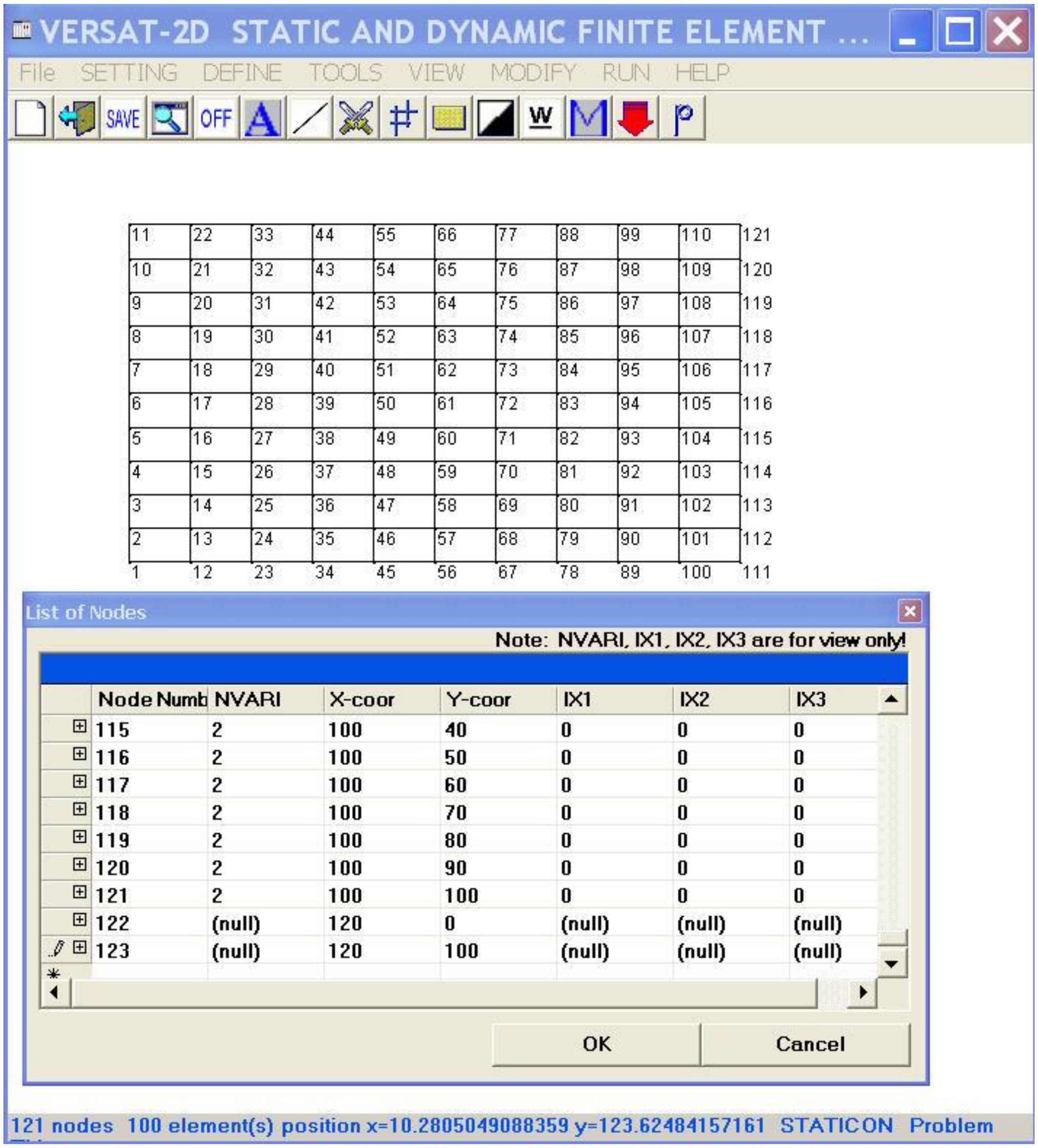
Step 3 OK and then click on nodes 1, 3, 4 and 2 to create a 10x10 grid



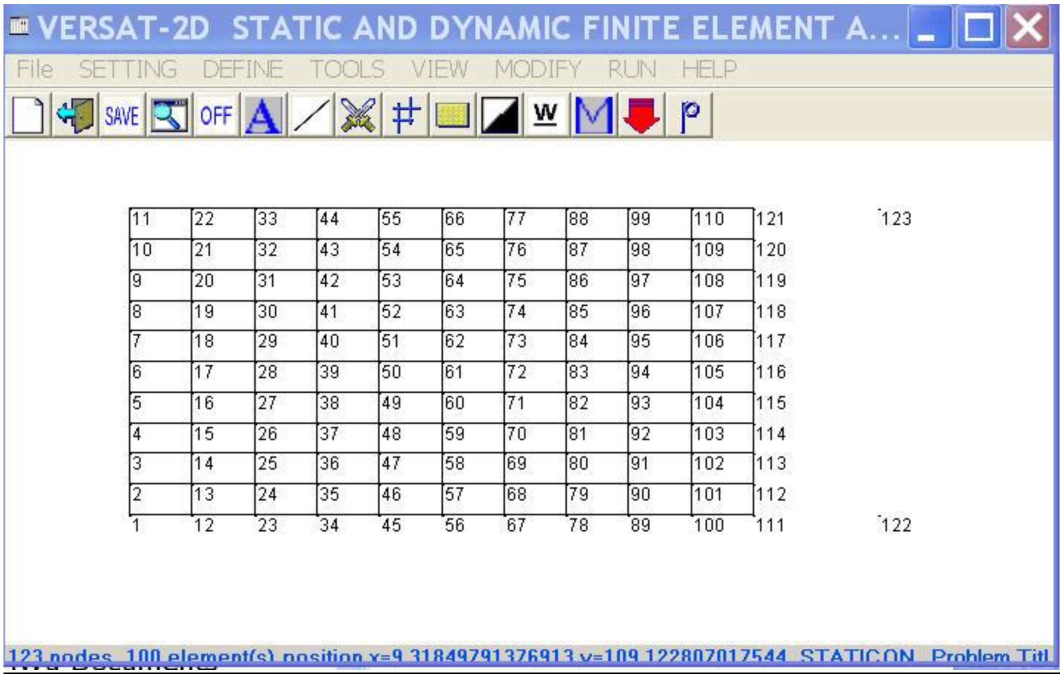
Step 4 Use '*Clear duplicate nodes/elements'* **then** '*Sort node(..)/element(..)*' under **MODIFY** to re-organize the mesh



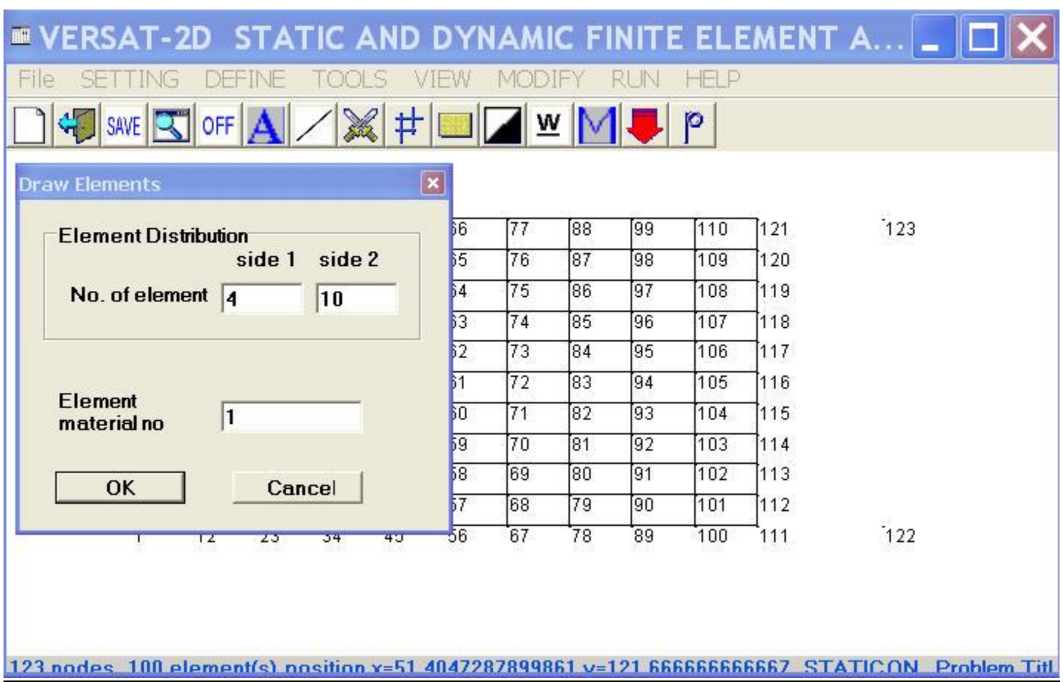
Step 5 Add more nodes using '*Finite element node/grid points*' under **DEFINE**



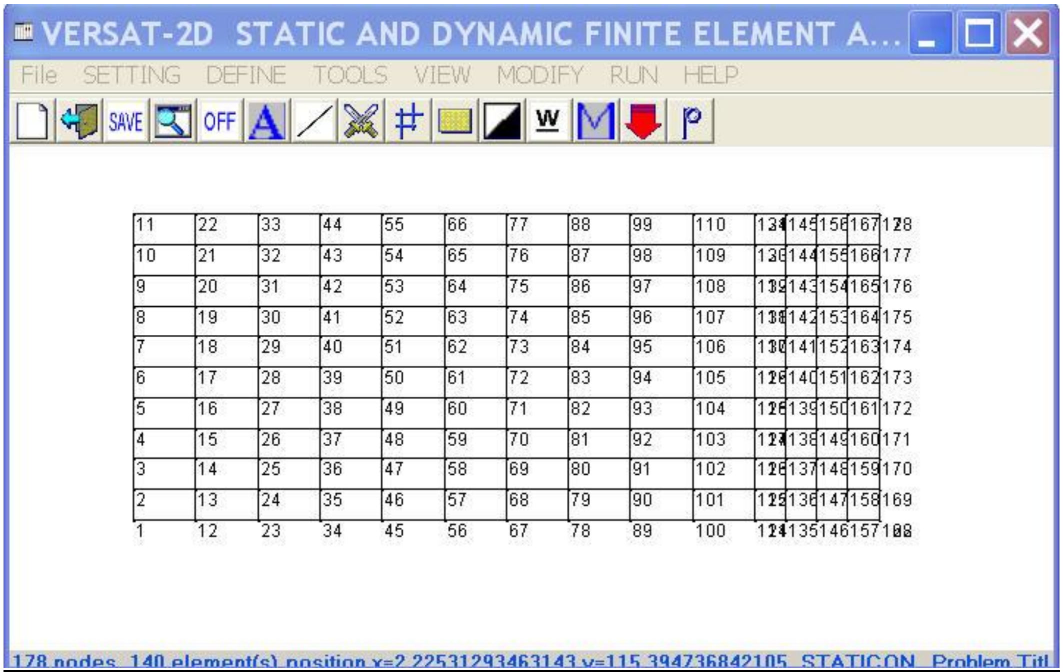
Step 6 OK to show nodes 122 and 123



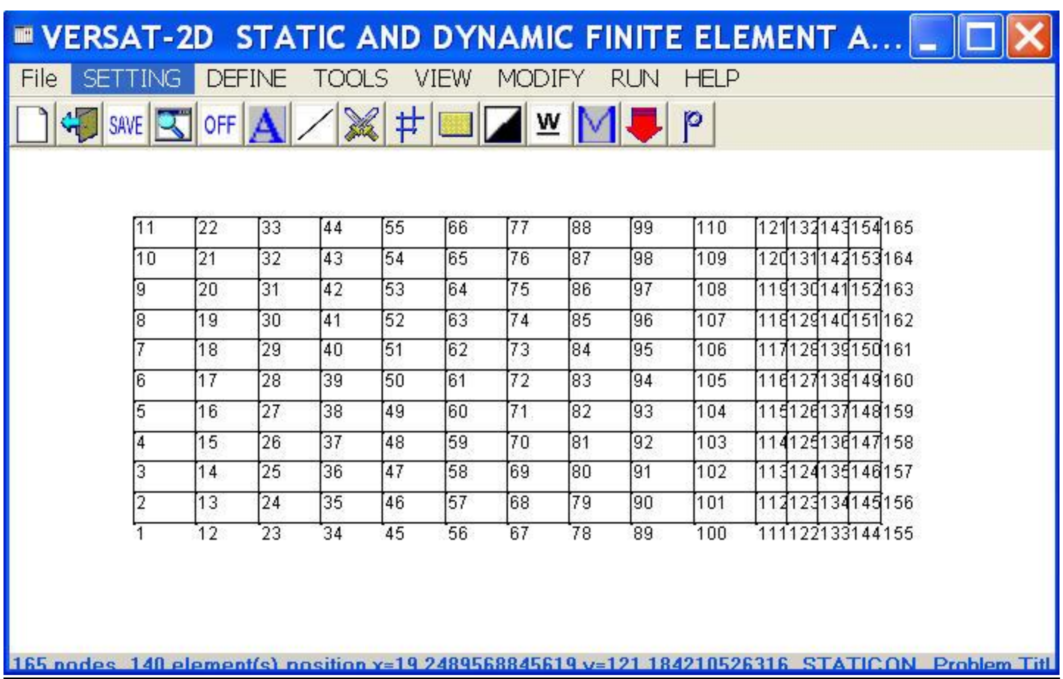
Step 7 Use '*Draw finite element grid*' under **TOOLS** to create a 4x10 finite element grid



Step 8 OK and then click on nodes 111,122, 123 and 121 to create a 4x10 grid

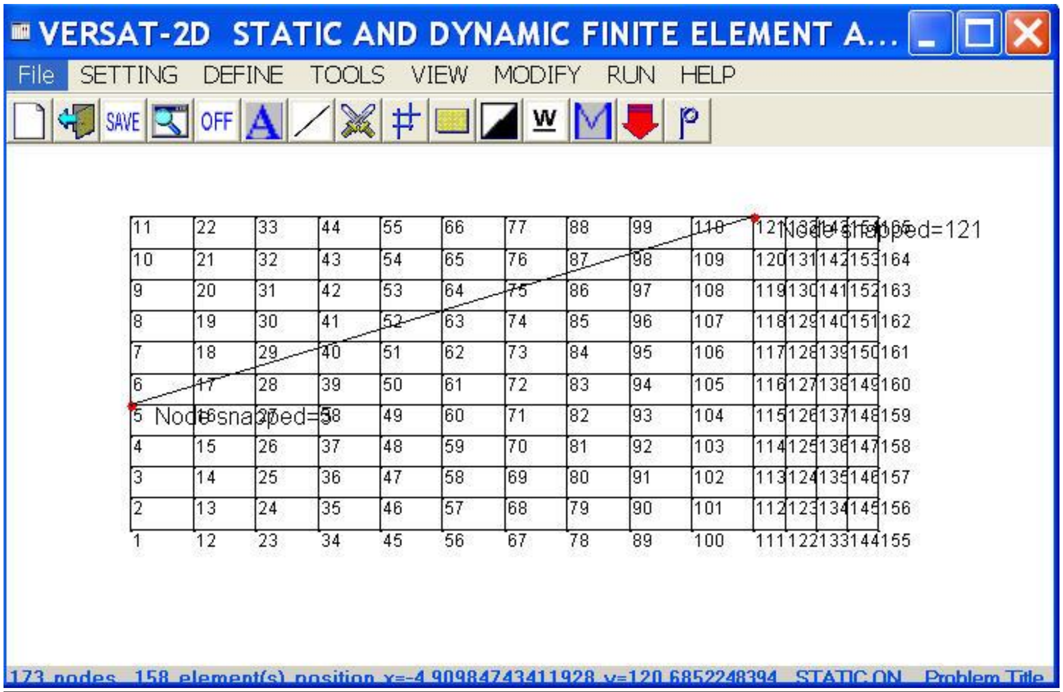


Step 9 Use '*Clear duplicate nodes/elements'* **then** '*Sort node(..)/element(..)*' under **MODIFY** to re-organize the mesh

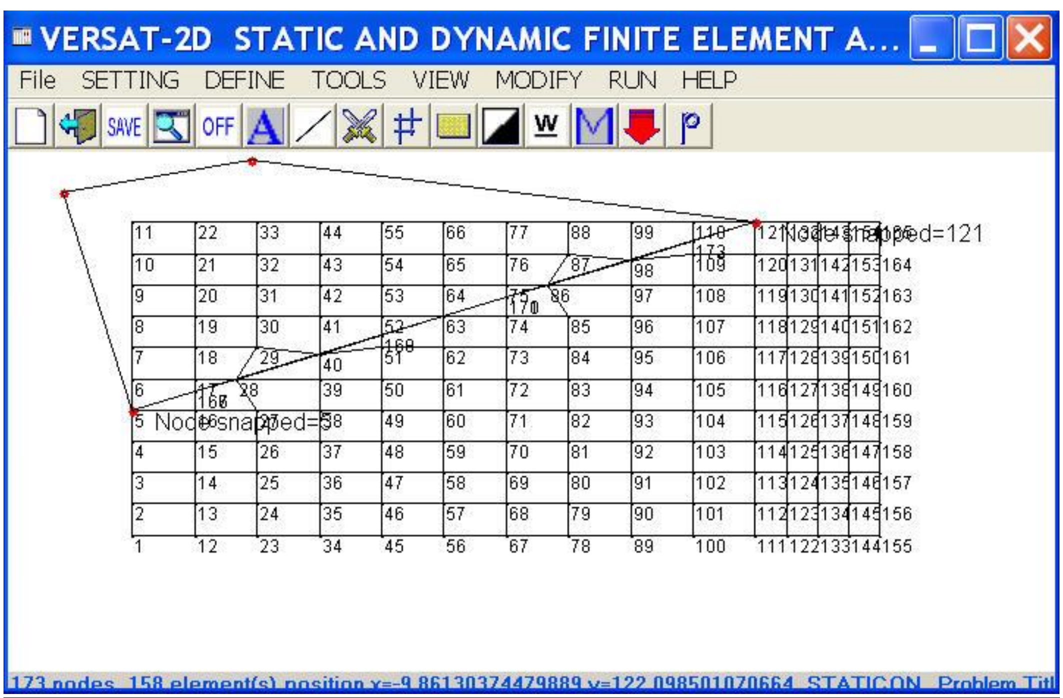


Note: Suggest saving the data as demo1.sta

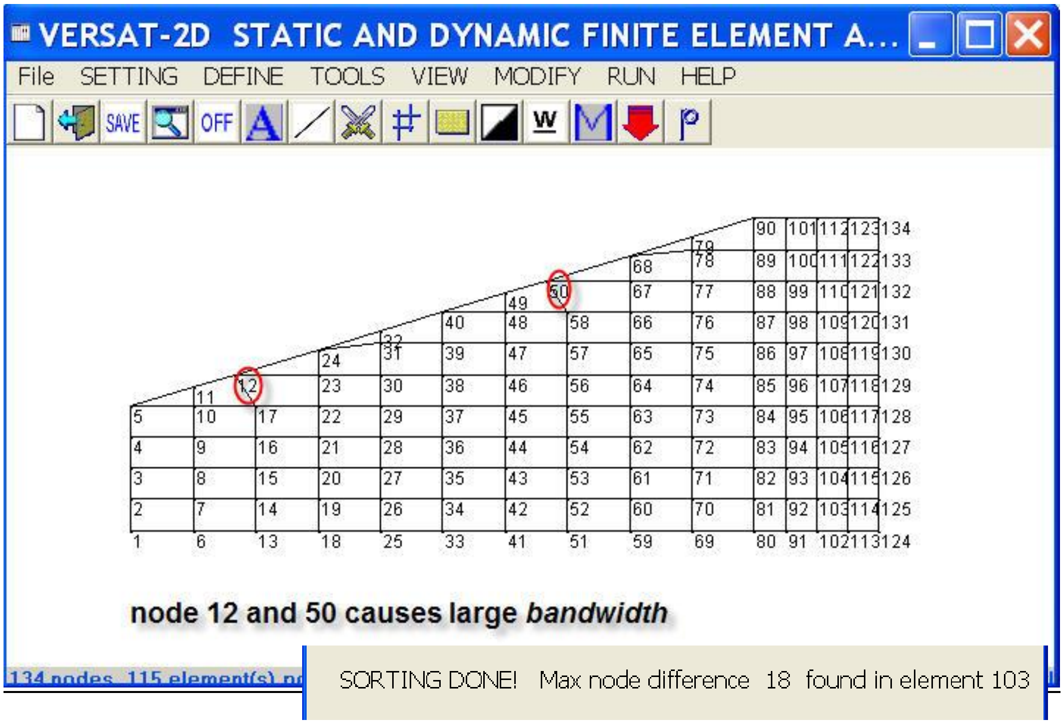
Step 10 Choose '*Draw line'* under **TOOLS** then click on Node 5 and Node 121



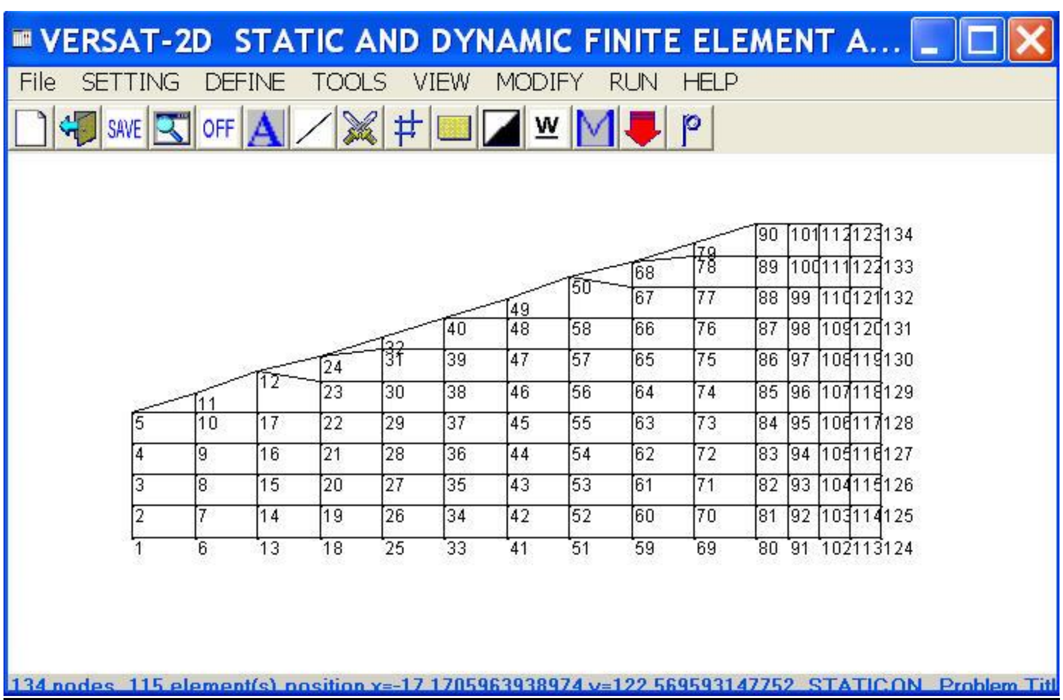
Step 11 Choose *'Cut/remove finite elements*' under **TOOLS** then click four points



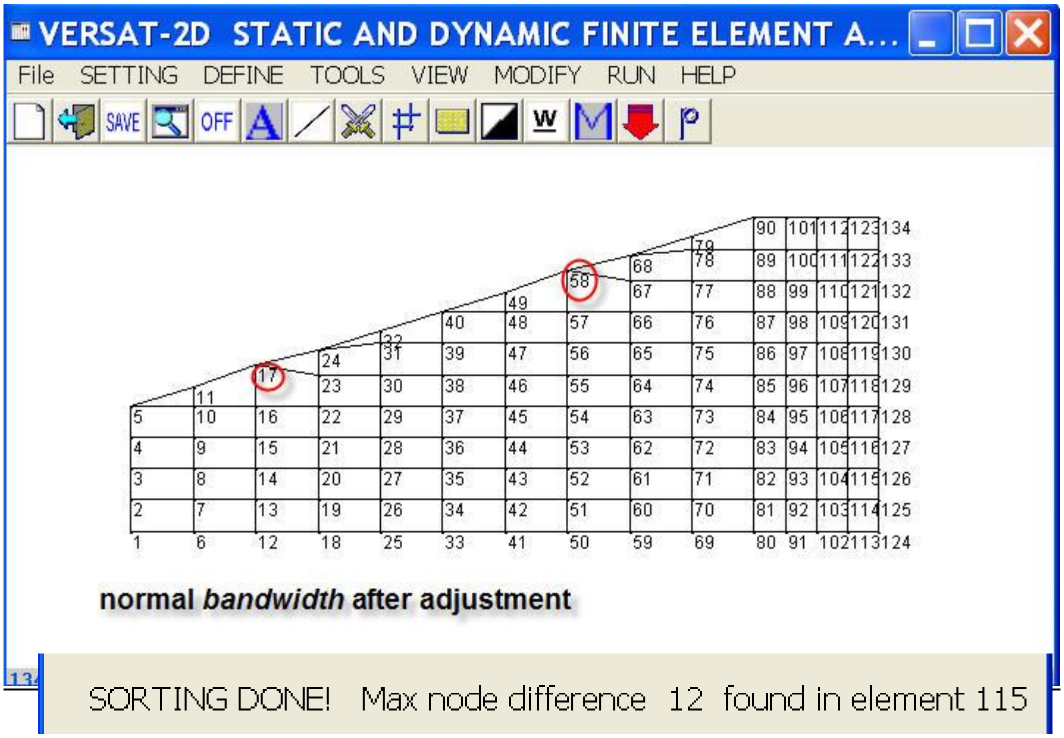
Step 12 Use '*Clear duplicate nodes/elements'* **then** '*Sort node(..)/element(..)*' under **MODIFY**



Step 13 Use '*Finite element node/grid points*' under **DEFINE** to adjust X and Y coordinates of node 12 and node 50



Step 14 Perform '*Clear duplicate nodes/elements'* **then** '*Sort node(..)/element(..)*' under **MODIFY**



Notes:

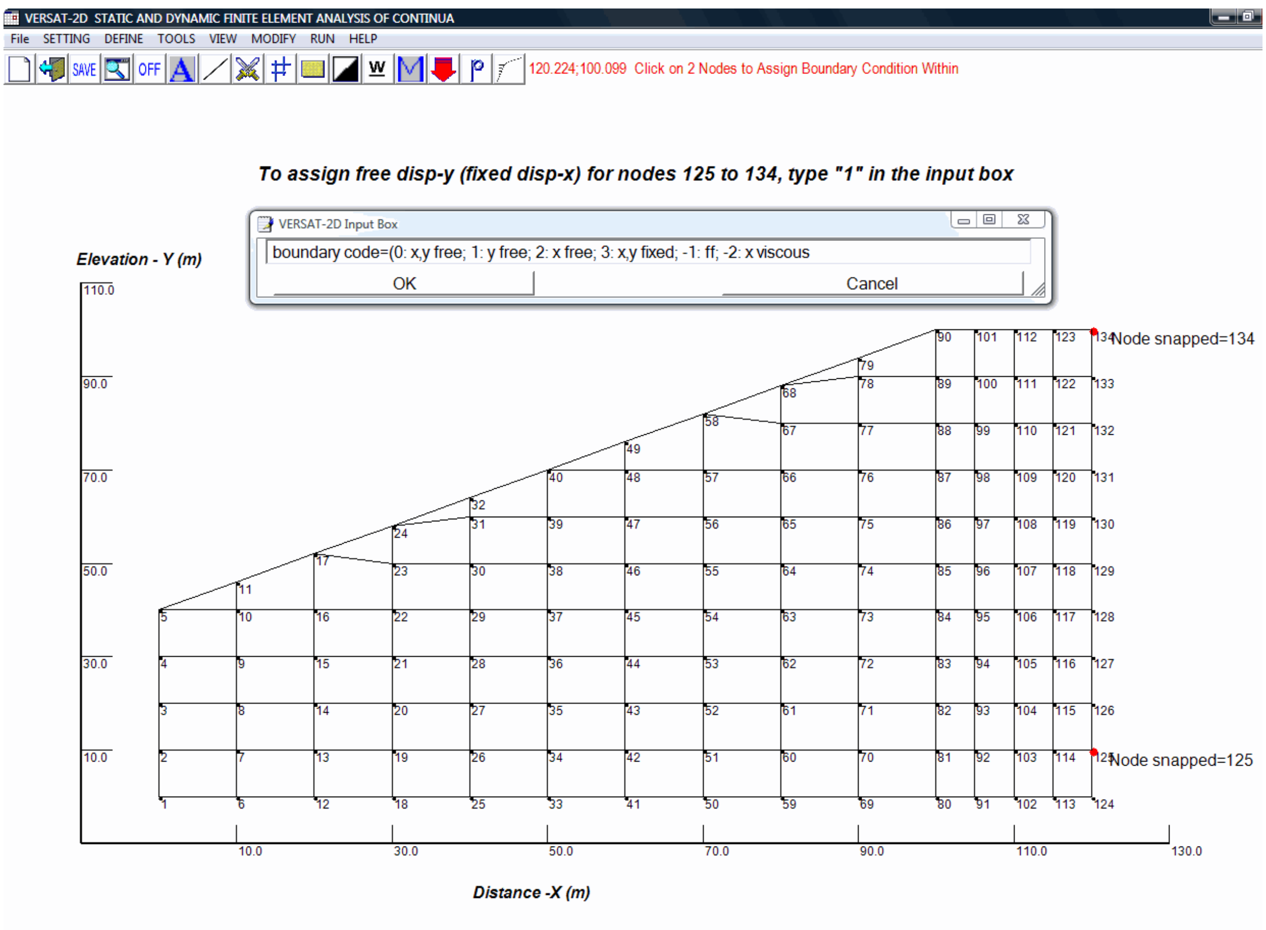
- Suggest saving the data as demo2.sta
- The purpose of adjusting coordinates of node 12 and 50 (shown in Step 12) is to reduce the band width of the model. This is done by relocating node 12 and node 50 so their X-coordinates are equal to or greater than those of node 17 and node 58 (also shown in Step 12), respectively. The model after the node adjustment is shown in Step 14. This process reduces the maximum node difference in the model from 18 to 12.
- This kind of node adjustment may reduce a problem size to half. As an example, a finite element grid of 6000 elements (120 × 50) should have a band width of approximately 100 and a problem size of approximately 12,000,000. However, the problem size could be doubled when the band width is 200 due to inadequate node numbering. As a result, computing time in a dynamic time history analysis may be increased from 4 hours to 8 hours for one analysis.

2.3. Set Boundary Conditions

Continue on the example:

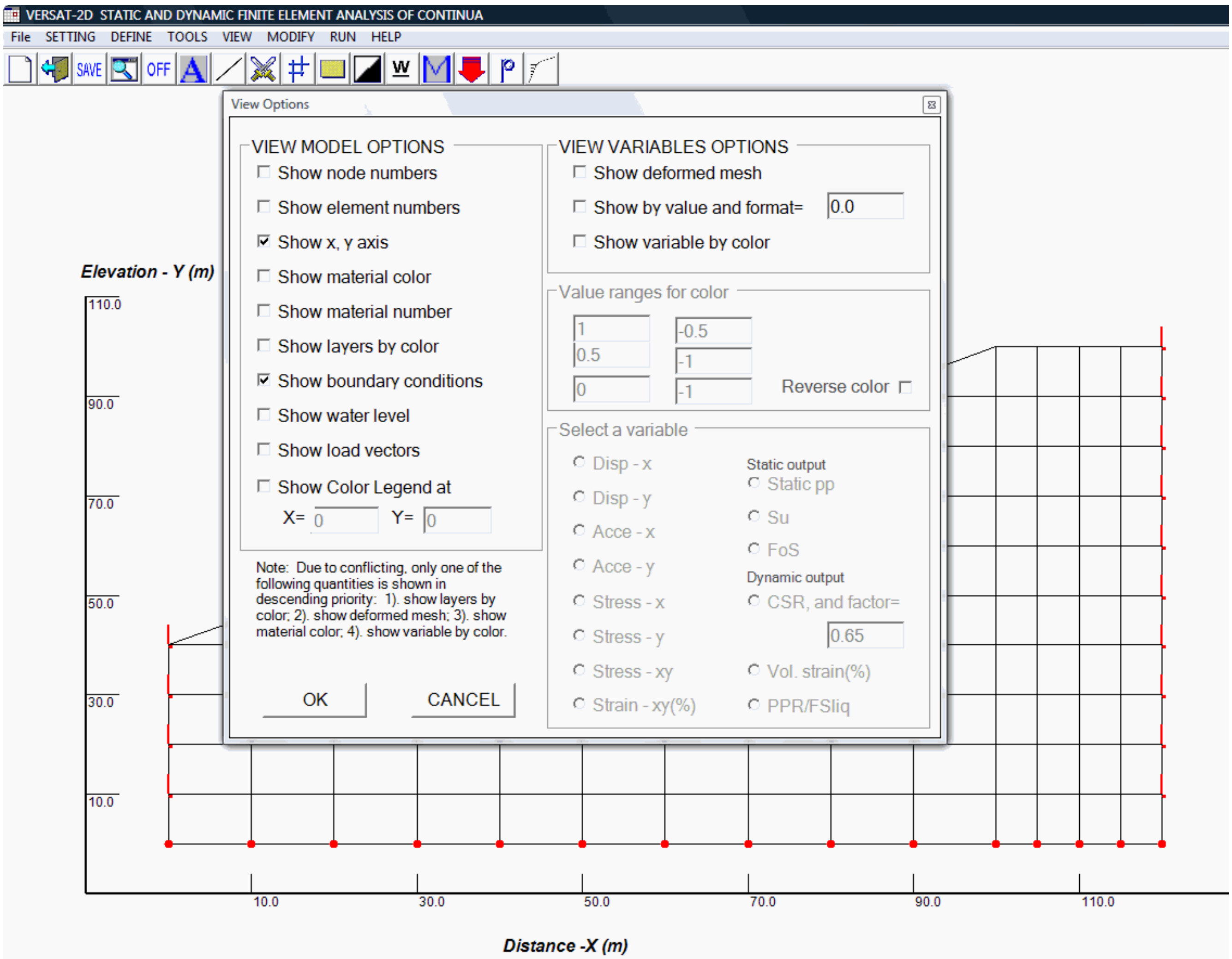
Step 1: Choose '*free all boundary'* under **MODIFY**

Step 2: Choose '*assign boundary conditions'* under **TOOLS** to define boundary conditions. For left boundary, click node 2 and 5. For the bottom, click node 1 and 124. For right boundary, click node 125 and 134 and then assign boundary code as appropriate. All nodes on the segment within the two points will be assigned to a specified boundary condition.



Step 3: Choose '*model view options*' under **VIEW**

- Check "*show boundary condition"*
- Check "*Show x, y axis"*
- Uncheck all others
- Click OK



Note: The boundary conditions of the model are shown above in red (solid circle = fixed/zero displacement in X and Y directions; vertical line=free vertical displacement; horizontal line=free horizontal displacement).

2.4. Apply Distributed Loads

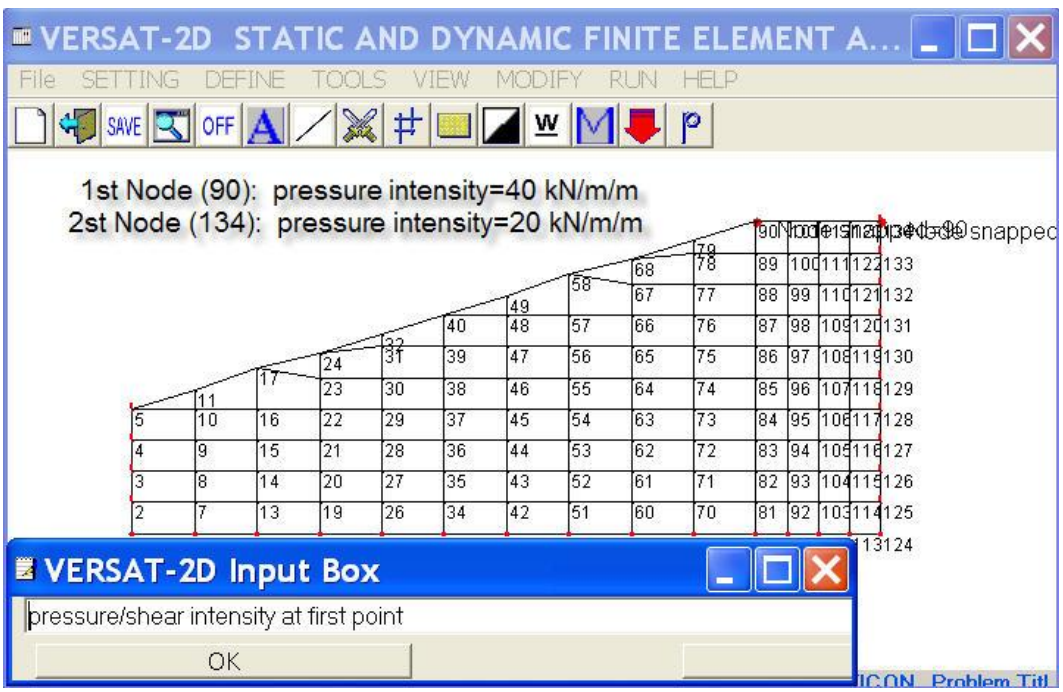
This command is used to apply uniform or non-uniform distributed loads on a surface, such as structural loads on a footing and water pressures on a submerged surface.

Continue on the example:

Step 1: Bring back the model with node numbers [i.e., check “show node numbers”]; then choose ‘*apply distributed load*’ under **TOOLS,** click nodes 90 and 134

Step 2: Enter ‘pressure/shear intensity’ at the 1st and 2nd nodes [40 and 20 respectively], then enter ‘inclination angle’ of the pressure [0° for pressure normal to the surface].

Note: Use an inclination angle of 90° if pressure is parallel to the surface. The pressure intensity between the two nodes is computed by linear interpolation.

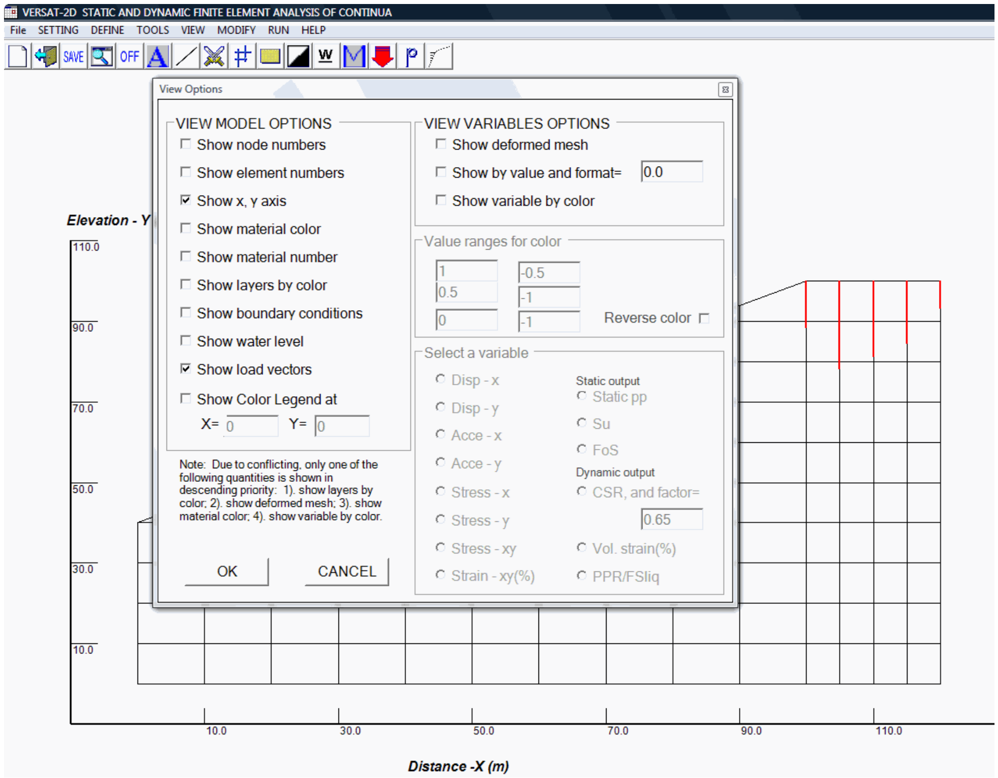


Note: Refer to Section 2.7 for applying distributed loads on a submerged surface with the use of “ywt0 > 0 Option”. The option allows an automatic calculation of water loads on the surface from a pre-defined water level (e.g., the reservoir level).

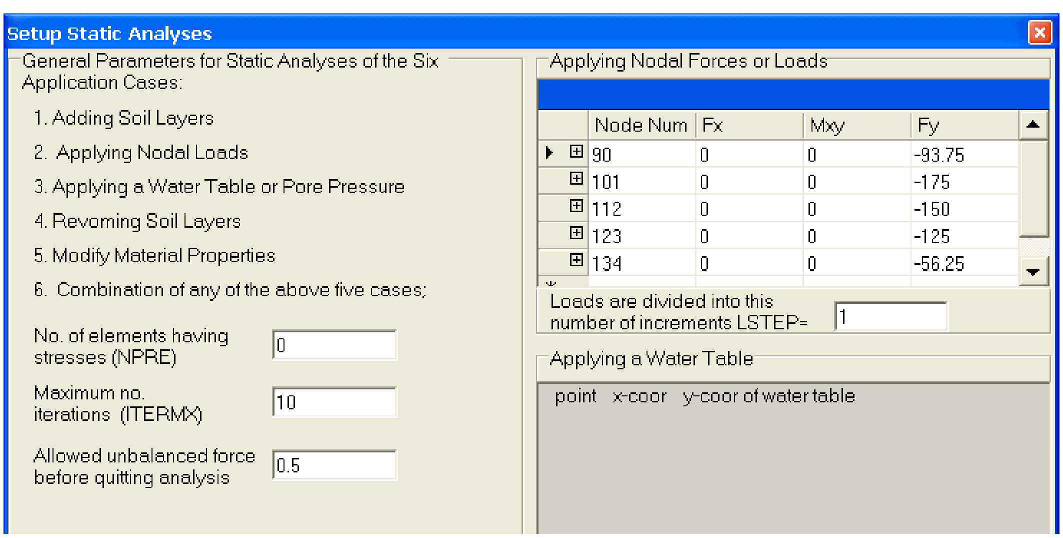
Step 3: Choose '*model view options'* under **VIEW**

- check "*show load vectors*"
- check "show x, y axis"
- uncheck all others and click OK

Note: Force vectors are shown in red lines, starting from the nodal points.



Step 4: Choose "setup static analysis/setup window" under **DEFINE**. The forces/loads on nodes 90, 101, 112, 123 and 134 are calculated by the program and shown in this window for review [click "Exit Setup" to close this window].

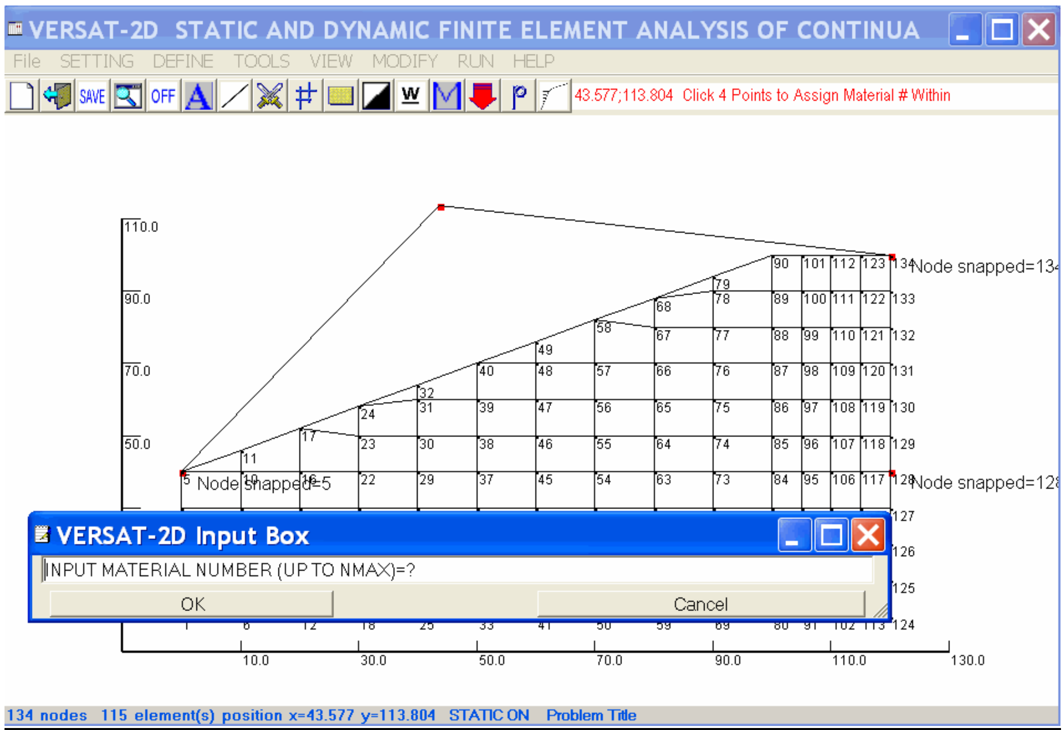


2.5. Assign Soil Zones

Continue on the example:

Step 1: Bring back the model with node numbers [i.e., check "show node numbers"]; then choose '*assign soil zones*' under **TOOLS**. Click four points as shown below.

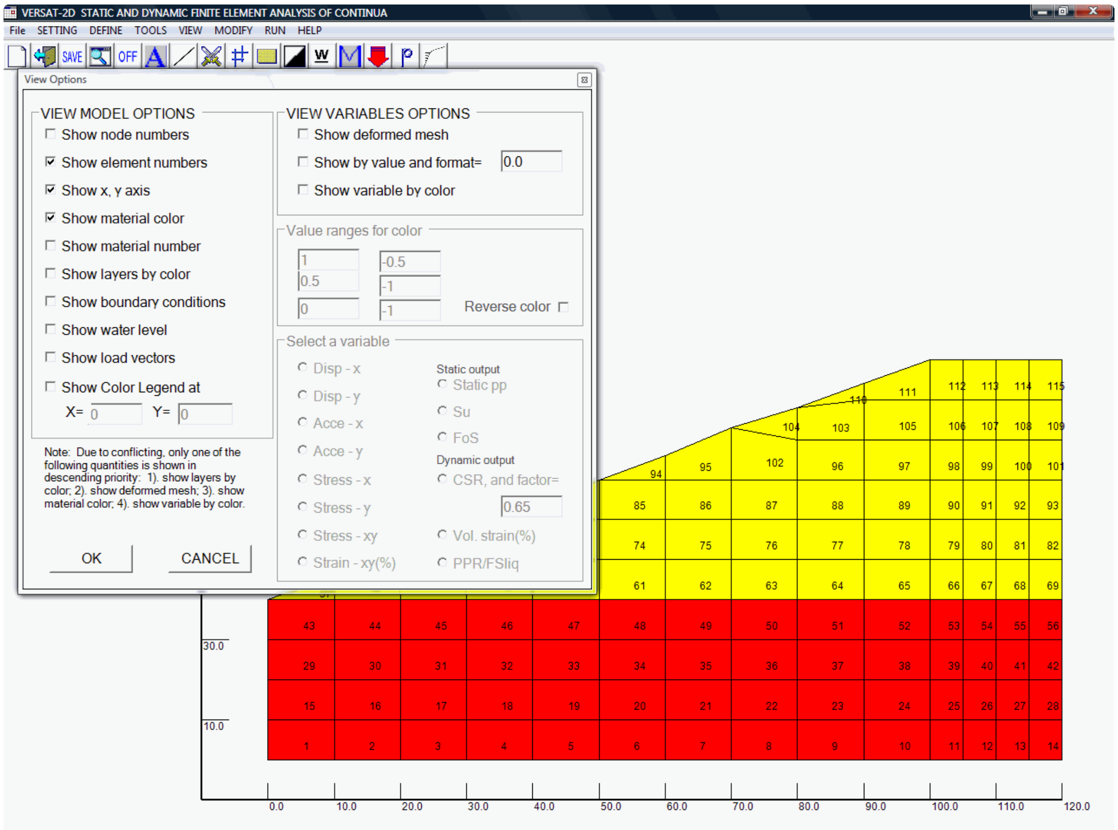
Step 2: Enter soil material number in the Input Box [1 for soil zone 1] and click OK. Elements inside the area bounded by the four points are assigned to Material #1.



Step 3: Repeat above steps but assign Material #2 to the red zone shown in the next figure

Step 4: Choose '*model view options*' under **VIEW**

- check "show element number"
- check "show x, y axis"
- check "show material color"
- uncheck all others
- click OK



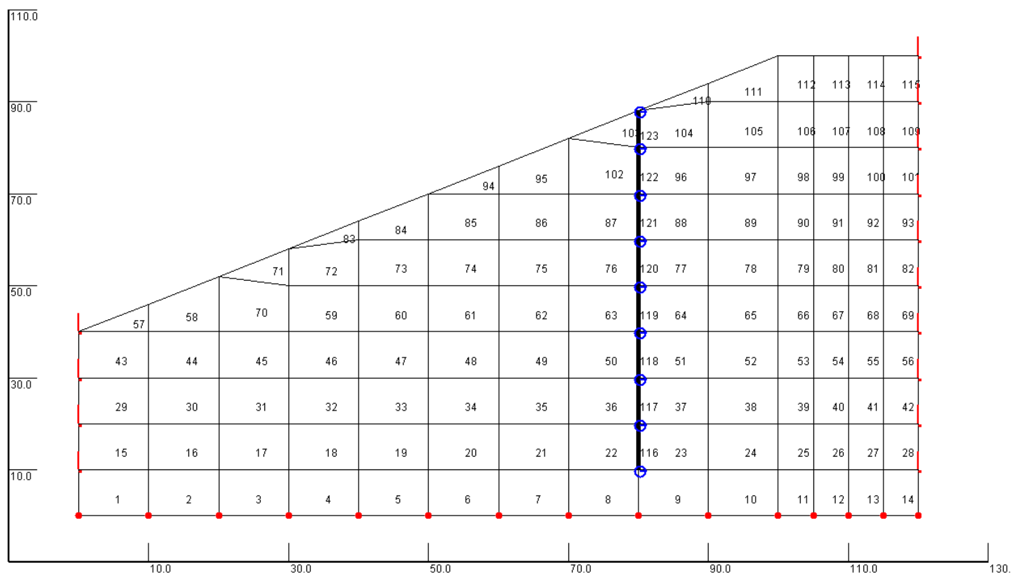
Notes:

- Suggest saving the data as demo3.sta
- The color codes for material zones are always the same as follow:
  - Yellow for Material #1;
  - Red for Material #2;
  - Blue for Material #3
  - Green for Material #4
  - Orange for Material #5
  - Dark blue for Material #6
  - Brown for Material #7
  - and more

2.6. Use of Beam or Truss Element[1]

Define beam or truss elements: choose '*Assign beam/bar elements'* under **TOOLS**, then click on screen to select two nodal points.

- This action does not add nodes, but add elements.
- All nodal points on the line segment within the two nodes are assigned as beam or bending elements when the "*beam*" option is chosen so, or
- One truss element is added to the model to connect the two nodes when the "*truss*" option is chosen.
- Nodes on a beam element are assigned rotational degree-of-freedom in addition to the two translational degree-of-freedoms, as shown in blue circles.
- It is recommended that this action be performed at the end of a mesh construction.



Note: This figure on beam element is shown for display only. The beam elements are not included in the example that is presented earlier and later.

[1] The beginner who is not so familiar with VERSAT-2D finite element setup should avoid using BEAM elements. It is recommended that BEAM elements be invoked only by advanced users.

2.7. Use of ywt0 > 0 Option[2]

Water loads on submerged surface (such as the upstream slope of a dam with a reservoir) can change when the submerged surface undergoes significant amount of ground deformations. In a large strain static analysis or in a dynamic analysis for earthquake loading, it is sometime necessary to update the water loads on the submerged surface to reflect the change of geometry resulting from ground deformations. The water loads can be automatically updated according to the new geometry by using the "ywt0 > 0 option".

The use of "ywt0>0 option" will result in

1. All nodal loads and static pore water pressures (PWP) be set to zero
2. In every step of analysis, new water loads on submerged surface nodes be computed using the assigned water level "ywt0". Note: Nodes in the "Applying Nodal Forces/Loads" Window are used in this calculation, and these nodes must be from left to right on one surface in the model.
3. In every step of analysis, static PWP in elements will be re-calculated using the assigned water table and the deformed mesh. Therefore, the static PWP must be defined using Method 1 with a water table, not using Method 2 with values (Technical Manual Section 2.7).

<u>Computing loads from water pressure</u>:

Choose "*Verify or use water loads from ywt0*" under **TOOLS**. This option allows that water loads on a submerged ground surface be calculated by the program automatically using a defined water level of "*ywt0*", such as in a reservoir. The parameter "*ywt0*" is specified in a setup window in Figure 3.1 (Section 3.1 bullet 7) for a static analysis and in Figure 4.1 (Section 4.3 bullet 9) for a dynamic analysis. The detailed instructions regarding its usage are also provided in the VERSAT-2D Processor when the option is invoked.

[2] The beginner who is not so familiar with VERSAT-2D finite element setup should avoid using 'ywt0 > 0 option'. It is recommended that this option be invoked only by advanced users.

3. SETUP A STATIC ANALYSIS

3.1. First Static Run – Run 1

A static analysis is setup using **DEFINE**. Choose '*General parameters*' to define key parameters that control the analysis including

- An option for gravity ON [default] or OFF
- constants of gravity acceleration, unit weight of water and atmospheric pressure
- an option of linear or nonlinear [default] static analysis, and
- an option for small strain [default] or large strain (updating mesh)

The default constants are for metric units.

Choose '*Setup static analysis'* and then '*Setup Window*' to start a setup window as shown in Figure 3-1. An input file for a static analysis can contain one or multiple static runs. The first of these static runs is called "*Run 1*". A static run may include one or a combination of the following parameters or load applications:

1. Add Soil Layers [add gravity force]:

- A static run can start with NPRE elements that already have stresses [default NPRE=0].
- Gravity forces have already been applied to these [NPRE] elements having stresses.
- A sub window "*Apply No of Elements in a Layer*" is used to add one layer or multiple layers of elements to NPRE to which gravity forces are applied. A static run can contain multiple load applications, i.e., multiple layers.
- Gravity is applied or turned on layer by layer, but one layer each time. Assuming the number of elements in the layer is NADD, the total number of elements included in this load application should be $NPRE_{updated}$[=NPRE+NADD]. All other elements with an element number greater than $NPRE_{updated}$ are automatically excluded in this load application.
- At the end of this load application, elements in this layer are then included in NPRE elements (i.e., elements having stresses) and NPRE is increased automatically.
- A static run can also contain no layer of elements to be added to NPRE. In this case, gravity forces are applied to all elements in the model and NPRE is deemed to be equal to the total number of elements of the model. Thus, the sub window "*Apply No of Elements in a Layer*" is left blank.
- In addition to being used in "*Run 1*", this load application "*Add Soil Layers*" can also be used in subsequent static runs until the updated NPRE reaches the total number of elements of the model.
- This load application "*Add Soil Layers*" is void when the option for gravity is set OFF.

2. Apply a Water Table:
    - A water table or a phreatic surface is defined in a sub window "*Applying a Water Table*". Points are added manually by clicking the "*Add*" button and entering X and Y coordinate of a point. A water table connects all points in the sub window consecutively.
    - A water table or a phreatic surface, once defined, will remain unchanged until they are replaced/updated by another one defined in a subsequent static run.

    Note: Choose '*define water level or pore pressures'* under **TOOLS** to: (1) compute pore water pressures from a pre-defined water level, or (2) assign constant pore water pressures or pore pressure ratios within a soil zone [see Volume I: Technical Manual for details of applications]. Detailed instructions are provided in the Processor when this operation is invoked.

    - This window may be left blank if a water table does not exist.
    - LWSTEP=1 should be used
3. Apply nodal forces or loads:
    - In addition to using '*Apply distributed load*' under **TOOLS** as described in Section 2.4, individual loads can also be added or edited in a sub window "*Applying Nodal Forces or Loads*".
    - Loads should not be applied to elements that are not included in the current model as defined by $NPRE_{updated}$
    - LSTEP=1 should be used.
4. Perform Excavation:
    - This is a reverse process of adding soil layers described above.
    - The option in the sub window "*Performing Excavation?*" is set to YES.
    - The number of elements to be excavated or removed (NEXC) is entered in the sub window "*Apply No of Elements in a Layer*". Only one layer of elements should be defined in this sub window. Multiple layers of elements should not be used for this application.
    - The element number of these NEXC elements is entered in the sub window "*Performing Excavation?*"
5. Modify Material Parameters
    - One set of material parameters are used for one static run.
    - A set of material parameters will remain unchanged and effective until they are replaced/updated by another set of parameters defined in a subsequent static run
    - When initiating a new static run, a user has the option to define a new set of parameters which can be modified from the current set of parameters.
    - Modifying soil parameters from strong to weak such as strength reduction due to soil liquefaction can cause deformations in sloped grounds.

6. Review maximum no. of iterations (ITERMX): A static load application terminates when ITERMX is reached or the requirement for "allowed unbalanced force…" is satisfied.
7. Review the water level parameter "*ywt0*" [default=0 for function not used]: For static runs, this parameter is used only when the "large-strain" option is chosen for the analysis. See Section 2.7 for details of the option.

Continue on the example in Step 4 of Section 2.5:

- Start "*Setup static analysis*" and '*Setup Window*'
- Delete the nodal forces within the sub window "*Applying Nodal Forces or Loads*". They are added later in static Run 2.
- Click "*Add a layer*" in sub window "*Apply No of Elements in a Layer*", add two new layers with 28 elements each.
- Click "Apply" and "Exit Setup"

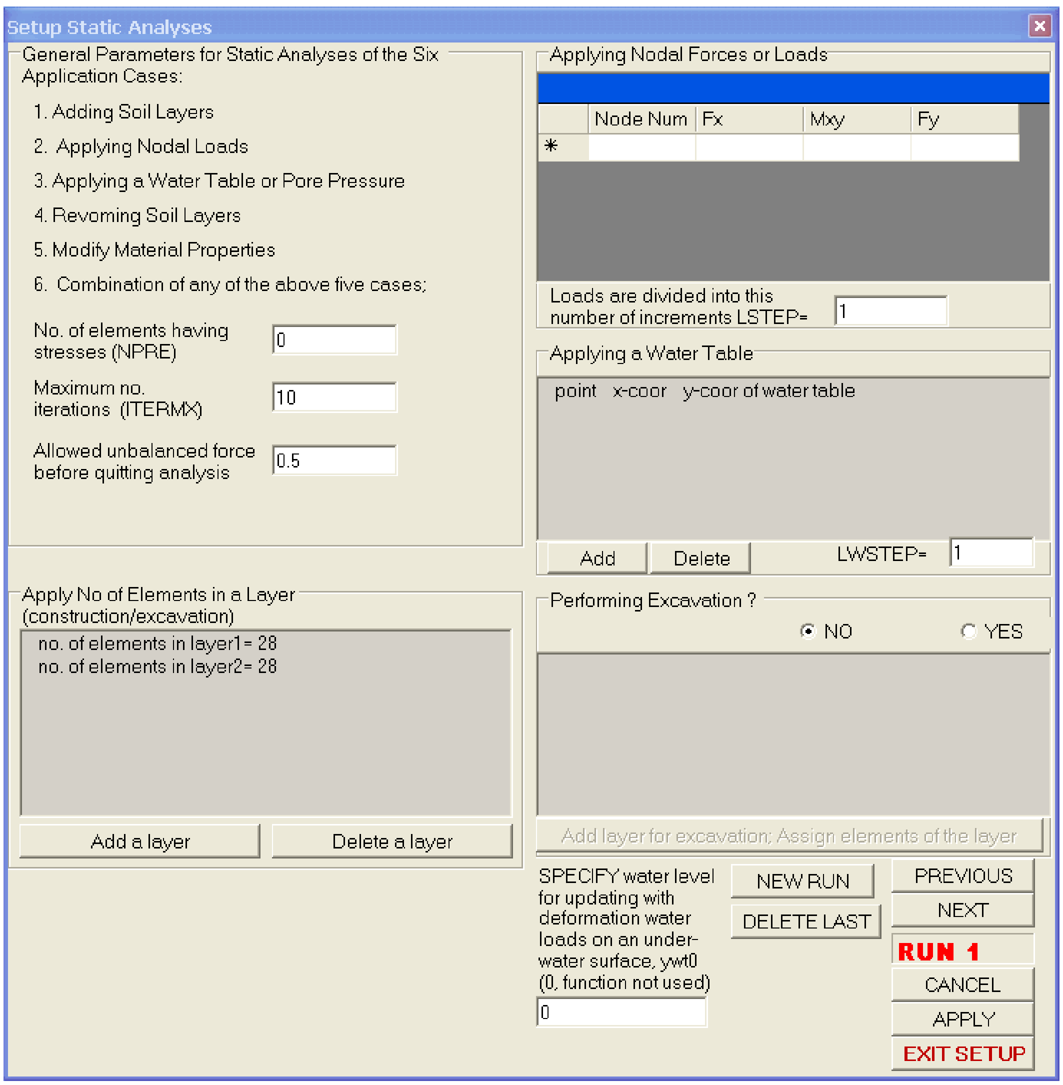
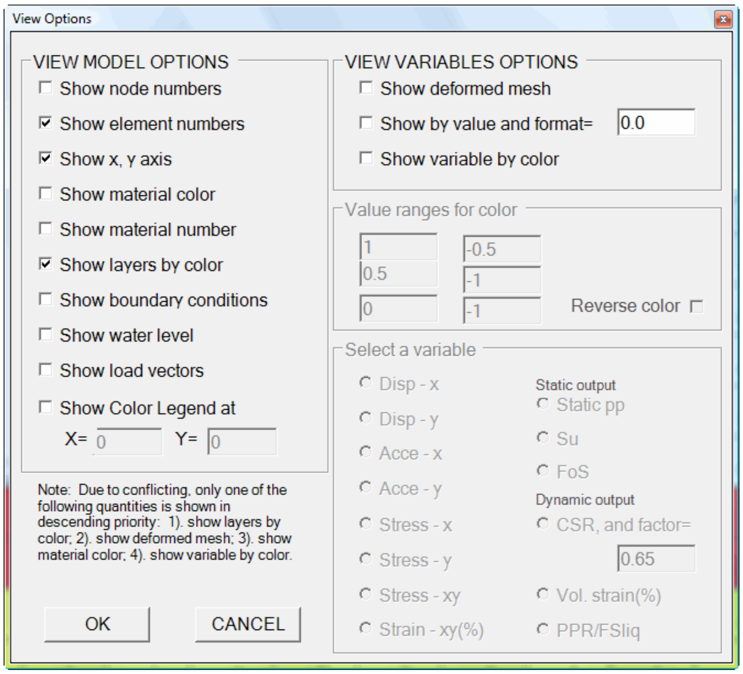


Figure 3.1 A static setup window

Choose '*model view options*' under **VIEW**

- ➢ check "show element number"
- ➢ check "show x, y axis"
- ➢ check "show layers by color"
- ➢ uncheck all others
- ➢ click OK



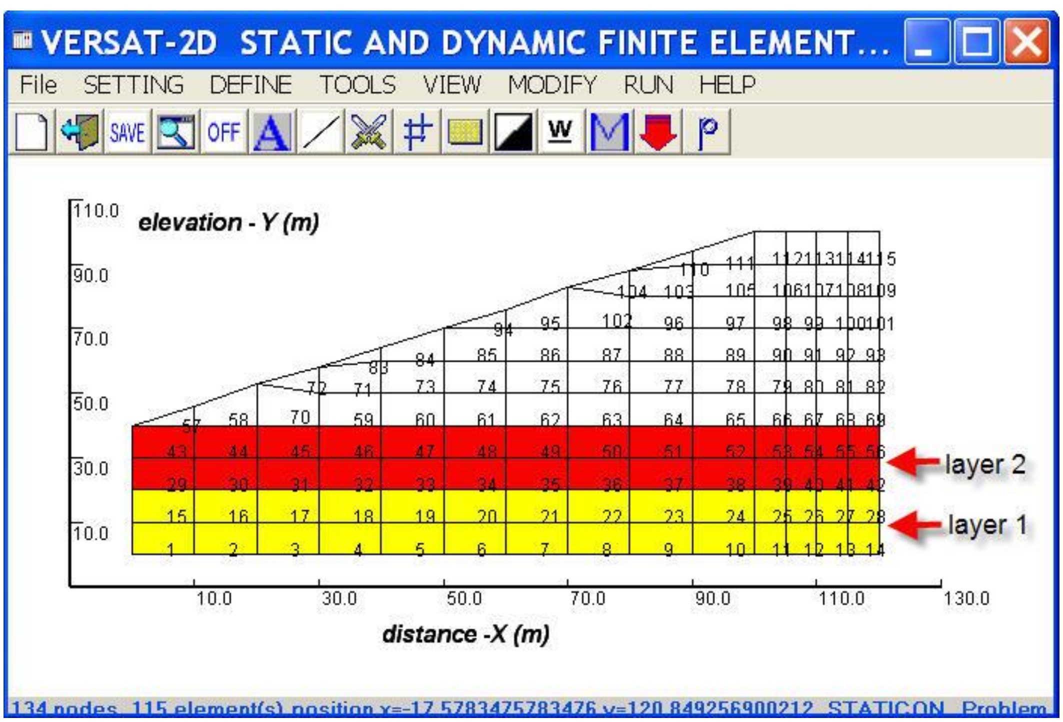


Figure 3.2 A window showing layers in color

3.2. Define Soil and Structure Parameters

Continue on the example:

- Choose '*Input material parameters'* under **DEFINE**
- Enter "1" in material number box
- Select "*Sand*" in the sub window "Select a Material Type"
- Edit the parameter boxes as needed [default values are shown herein]
- Click "*Add/modify a material*" button
- Enter "2" in material number box
- Select "*Clay*" in the sub window "Select a Material Type"
- Edit the parameter boxes as needed [default values are shown herein]
- Click again "*Add/modify a material*" button
- Click "*APPLY ALL*" button to save and exit this window.

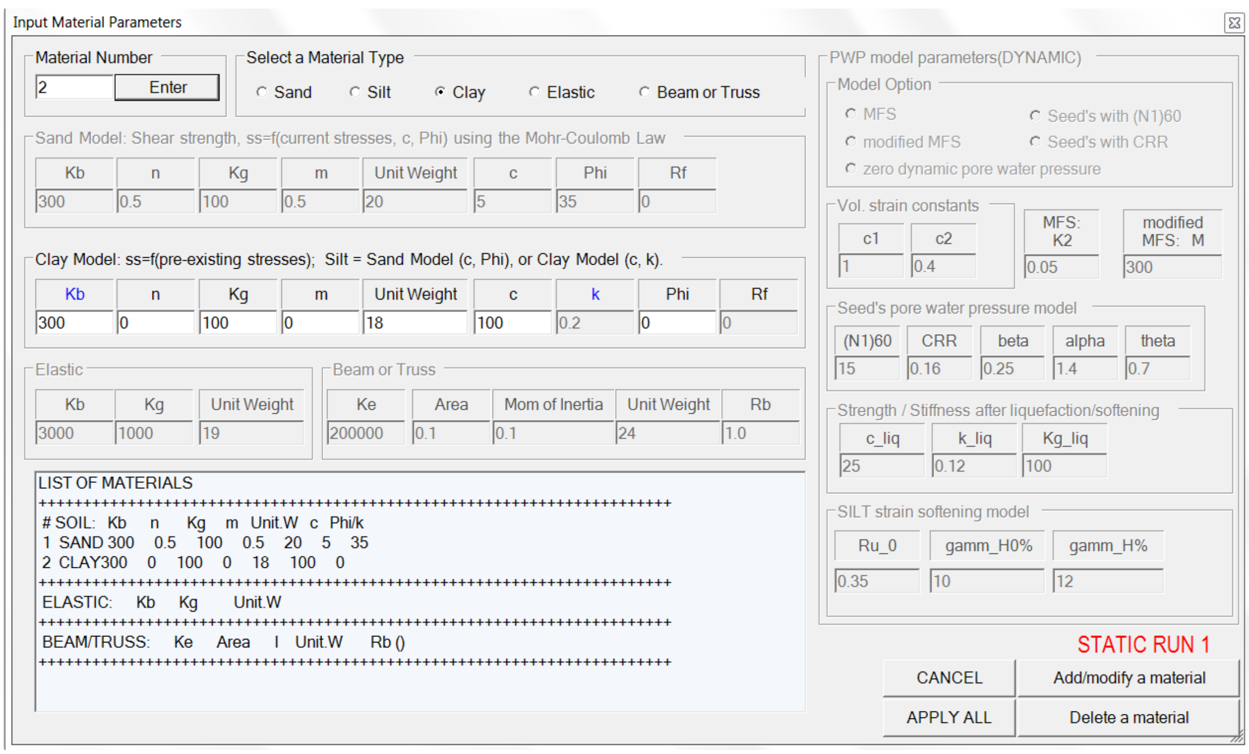


Figure 3.3 A window showing material parameters defined in Run 1

3.3. Second and More Static Runs

A new static run is normally required when nodal loads, or the water table, or soil properties are changed. These quantities are unchangeable within a static run.

Continue on the example to setup a second static run – Run 2:

- Start "*Setup static analysis*" and '*Setup Window*' again.
- Press "*NEW RUN*" button and "*YES*" to initiate a new static run
- "*Copy Soil Parameters from RUN 1?*" and "No" to not redefine soil parameters. If changes on parameters are required, then answer "YES" to copy and modify.
- Click "*Add a layer*" in the sub window "*Apply No of Elements in a Layer*", add one new layer with 59 elements as shown in Figure 3.4

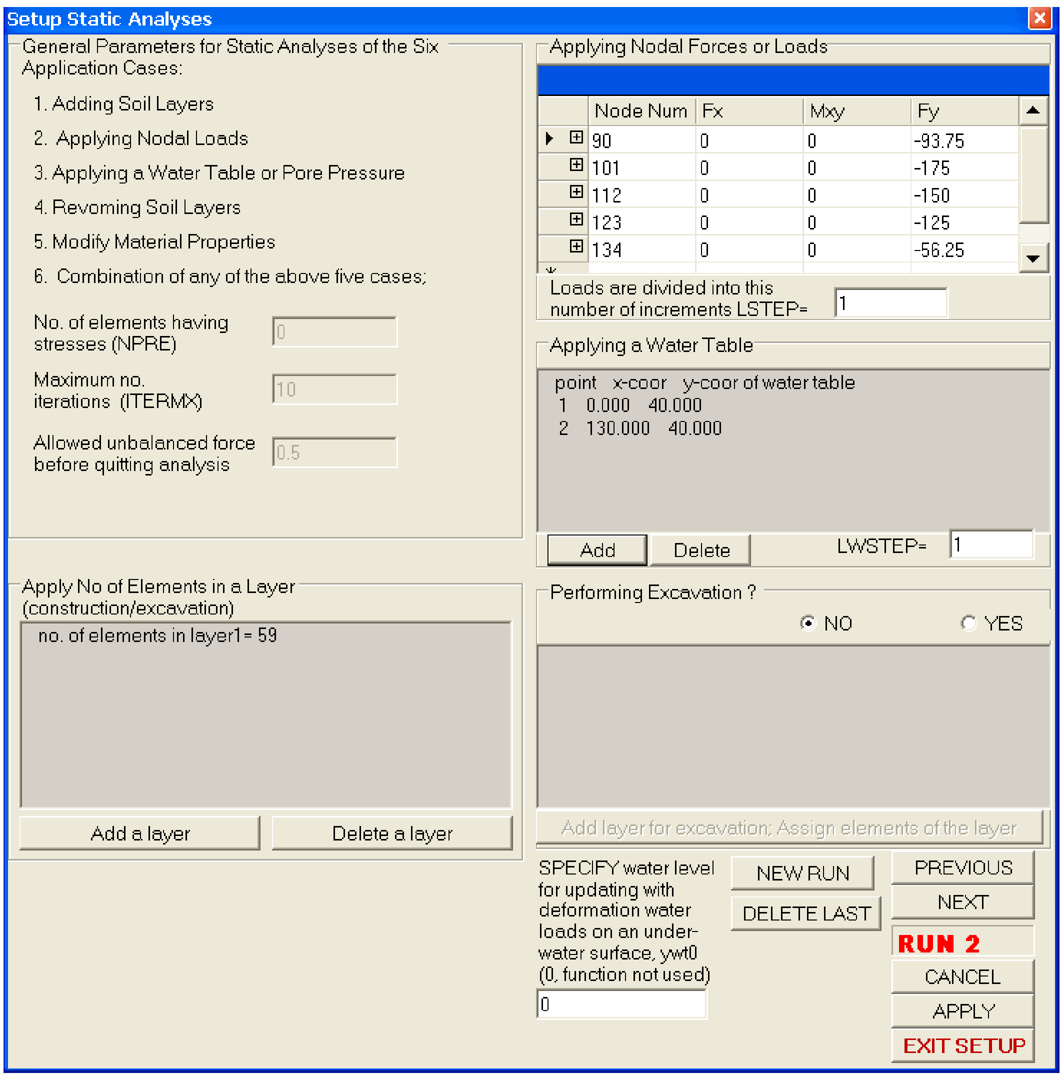


Figure 3.4 A static setup window for Run 2

- Click "*Add*" in the sub window "*Apply a Water Table*", add two points to define a water table as shown in Figure 3.4.
- Click "*Apply*" and then "*Exit Setup*".
- Follow steps in Section 2.4 to apply non-uniform distributed loads on the surface from node 90 to node 134 [Section 2.4 is there for instruction purpose].
- Go back to "*Setup static analysis"* and '*Setup Window*' again. The window as shown in Figure 3.4 should contain the loads for nodes 90, 101, 112, 123 and 134.
- Refresh the model using "*Show layers by color*" and "*Show load vectors*" as shown in Figure 3.5

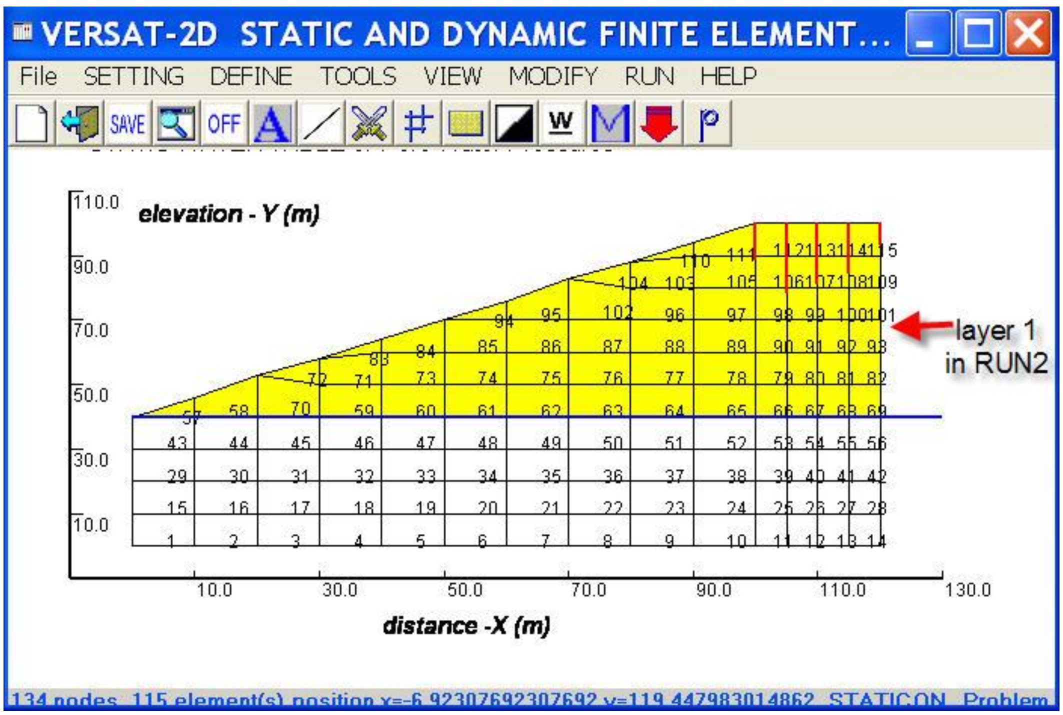


Figure 3.5 A window showing layers and loads for Run 2

Note: A static run can be deleted by pressing the "*DELETE LAST"* button. Only one static run with the highest run number is deleted at a time.

3.4. Save Data and Start A Static Analysis

Continue on the example:

- Save the data as *Run2.STA* using '*Save Data'* under **FILE**.
- Choose '*Run versat-s2d*' under **RUN**
- Enter the User Name and Password
- Press "*Connect Now*' (see Figure 3.6)
- Select the input file "run*2.sta*" to run after an authorization is obtained

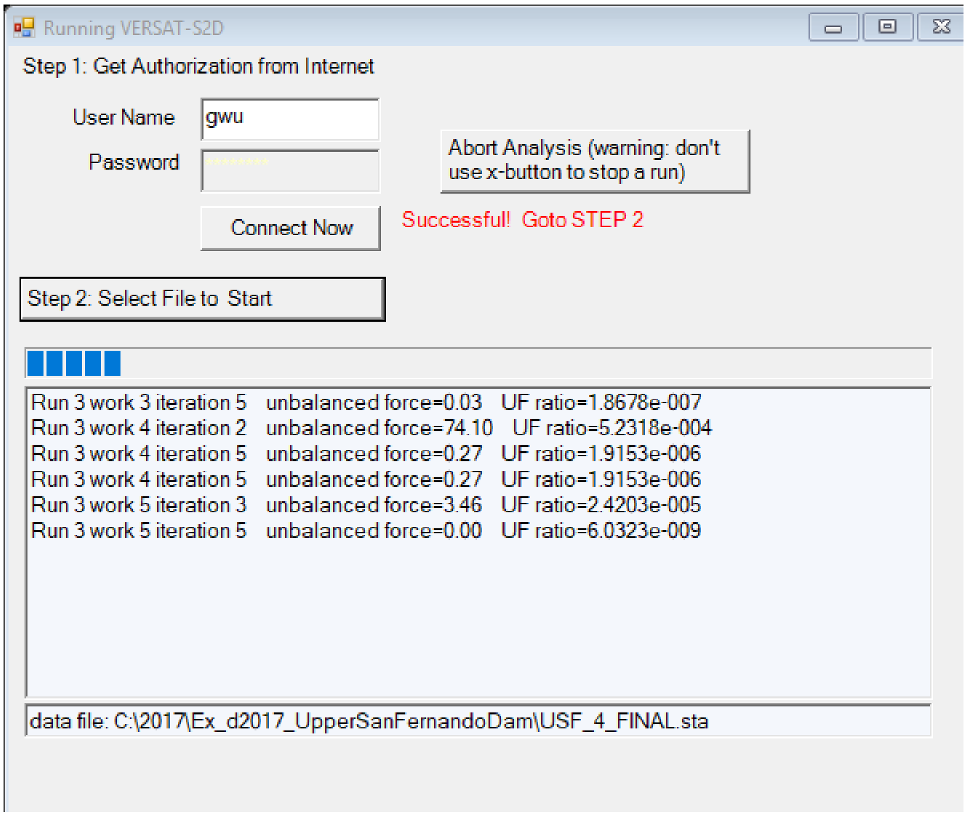


Figure 3.6 A window showing '*Run versat-s2d*' under **RUN**

Note:

- Results of this static run include a file named "run*2.pr4*". This file is needed in a subsequent dynamic time-history analysis.
- An Internet connection is required in order to run any analyses using VERSAT-2D. However, it is not required for data preparation.

3.5 Input Files for Static Run

The input files for a VERSAT-2D static analysis consist of 1 file or 2 files:

1. Filename.STA (main input file prepared in VERSAT-2D Processor)
2. Filename.PRX (pre-existing stresses, and it is needed when NPRE > 0)

4. SETUP A DYNAMIC ANALYSIS

4.1. Turn On Dynamic

In general, a dynamic analysis can start only when gravity-induced static stresses within a finite element model are determined in a static analysis. As such, the dynamic analysis model can be constructed using the same finite element model as the one used for that static stress analysis.

Continue on the example in Section 3 and prepare data for a dynamic run:

1. Restart the Processor
2. Load the setting file (see Section 2.1) using "*Load Setting*" under **SETTING**
3. Load the data file "*run2.sta*" (see Section 3.4) using "*Load Data*" under **FILE**
4. Click '*Dynamic On'* under **SETTING** and select "*Yes*"

Note: "*DYNAMIC ON*" should be shown at the bottom of the Processor.

4.2. Key Parameters for a Dynamic Analysis

1. Choose "*General parameters*" under **DEFINE**
2. Gravity on/off: On [default]
   - Choose Gravity Off [enter 1 in the box] to run dynamic analysis of a 1D soil column
3. Gravity acceleration: 9.81 m/s^2 [default] for metric unit
   - Enter "-9.81" in this box to use a sine input instead of a time-history input. The use of a sine input is demonstrated in an example file called *ex_d2.dyn*.
4. Method of analysis:
   - [default] - Non-linear analysis (ICHANG=1)
   - Select "Non-linear Effective Stress" (ICHANG=2): to include dynamic pore water pressures in the calculation of soil strengths and ground displacements (see Section 3.5 of the Technical Manual for details)
   - Select "ICHANG=3 for Advanced User": This option is available since v.2019.5.8. Its functionality is the same as "Non-linear Effective Stress Analysis" or ICHANG=2. See Section 4.11 in the User Manual for further explanations.

Note: Geometry mesh is always updated with deformation (large-strain option is always ON) in the dynamic analysis.

4.3. Setup a Dynamic Analysis

Continue on the example and name the input file of the dynamic analysis as "*run3.dyn*":

1. Choose '*Setup dynamic analyses*' to start a setup for dynamic analysis.
2. Enter "*115*" for NPRE. When the dynamic analysis starts, the program automatically looks for an input file named "*run3.prx*" to read in the stresses of these "*115*" elements. It is required that the file "*run2.pr4*" [see Section 3.4] be renamed manually as "*run3.prx*" prior to this dynamic analysis. Otherwise, the program stops because of incomplete input files.
3. Select "*Hori Base Acceleration*" [default] under a sub window "*Options for input motions/forces (NBF)*". When the dynamic analysis starts, the program automatically looks for an input file named "*run3.ACX*" to read in time-history data of accelerations at the rigid base (the input ground motions). It is required that the file "*run3.ACX*" be created prior to the dynamic analysis. Otherwise, the program stops because of incomplete input files.
   - The data format for "*run3.ACX*" is shown in the left and lower area of the setup window, as shown in Figure 4.1, where *NPOINT* is the total number of acceleration data to be used in the analysis, *DT* is the time interval of the accelerations, FAMPL is a linear scaling factor by which the data are multiplied; and
   - NRVSUB is number of sub time step: 0 for no modification to the ground motion data and time interval provided in the file run3.ACX, 1 for inserting one point, 2 for inserting two points, 3 for inserting three points, and so on, to two consecutive data. All sub time steps are created by linear interpolation of acceleration data and time interval DT.
   - *NLINE* is number of record lines, and *NoPerLine* is number of data per record line. Data points must be in CSV format or comma delimited.
   - The data for ground motions are in the same unit as the gravity acceleration [m/s^2 for metric unit].
   - It is also noted:
     - When the option "*Hori + Vert Base Accelerations*" is chosen, another input file "*run3.ACY*" is required. This file provides data for vertical ground accelerations at the base of the model. It is noted that the time interval (*DT)* must be same for "*run3.ACX*" and "*run3.ACY*"; otherwise DT from run3.ACX is used for run3.ACY.
     - When the option "*Forces at Nodal Points*" is chosen, the input force time history is provided in the file "run3.FXY".
     - When the option "Hori. Outcropping Velocity" is chosen, the input velocity time history is provided in the file "run3.VEX".
     - The data format is same for "*run3.ACY*", or for "*run3.ACX*", or for "run3.FXY", or for "run3.VEX". Refer to "*NicoM_1c.ACX*" in the examples library [copy "*NicoM_1c.ACX*" to "*run3.ACX*"].

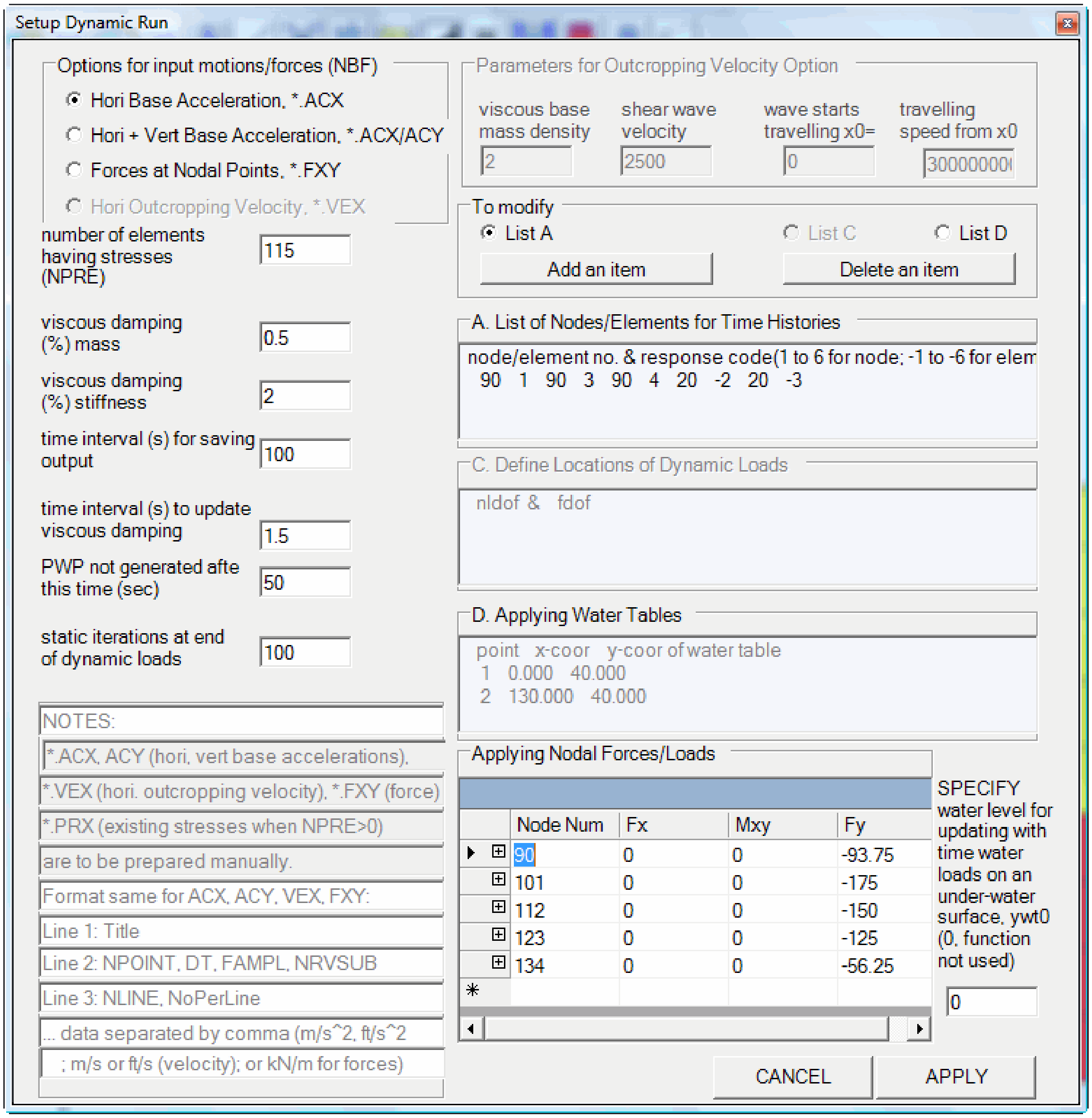


Figure 4.1 A setup window for a dynamic analysis

4. Review, modify as needed, the following parameters:

- viscous damping (%) of mass [=λ_m, see Section 3.2 of the Technical Manual, default=0.5]
- viscous damping (%) of stiffness [=λ_k, also see Section 3.2, default=2]
- time interval (s) for saving output [default=100 sec]: All recordable quantities such as accelerations, displacements, shear stress, and pore water pressure are printed at this specified time interval (in sec) in the output file (Filename.oud) and in the plotting files (Filename.dis, Filename.sig).
- time interval (s) to update viscous damping [default=1.5 sec]: In a nonlinear analysis, the frequencies of the model vary with time of shaking. The viscous damping constants (a, b in Section 3.2 of the Technical Manual) are updated at the specified time interval (in sec).

- PWP not generated after this time (sec) [default=50 sec]: In a non-linear effective stress analysis and within this specified time of shaking, dynamic pore water pressures are calculated and added to stresses of soil elements and are included in the force equilibrium. Beyond this time, dynamic pore water pressures are kept the same as at this time, i.e., constant with time. Enter a large number (e.g, 999) when this restriction is not required.
- Static iteration at end of dynamic loads [default=100]: At the end of shaking, static equilibrium analyses are carried out. All quantities related to vibrations such as accelerations and velocities are set to zero in this post-dynamic static analysis.

5. The sub window "*To modify*" allows changes to List A, C (enabled when the model is subjected to dynamic loads instead of ground shaking, not used in example "run3.dyn") and D. While the option "*List A*" is chosen and on, time history response of nodes and elements can be requested through the sub window "*List of Nodes/Elements for Time Histories*" as follows:

   - Click on "*Add an item*" and enter "*node or element no & response code"*
   - Repeat above for each pair of "*node or element no & response code"* until the required number of response points are entered.

   Notes: The requested time history data are saved using the file name of the input data and a file extension (or file type) of CSV such as *NicoM_1c.csv, run3.csv etc.* The time history data are compatible with Microsoft Excel.

   Response codes for saving node response data (and also in PSPA output) are:

   1 = X-displacement

   2 = X-velocity

   3 = X-acceleration

   4 = Y-displacement

   5 = Y-velocity

   6 = Y-acceleration

   7 = X-acceleration at the base

   8 = Y-acceleration at the base

   Response codes for saving element response data (and also in PSPA output) are:

   -1 = Stress-X (σ_x) or bending moment at centre of a beam element

   -2 = Shear Strain (in %)(γ_{xy}) or axial force of a beam/truss element

   -3 = Shear Stress (τ_{xy}) or shear force of a beam element

   -4 = Pore Pressure Ratio (PPR) or bending moment at J-node of a beam element

   -5 = Stress-Y (σ_y)

   -6 = Normal strain X (in %) (ε_x) (*In PSPA output*: peak Gamm_max in 1st set, DynStressRatio in 2nd set)

-7 = Normal strain Y (in %) (ε_y)

-8 = Volumetric strain (%)

6. The option "*List B*" is disabled in Version 2011.
7. While the option "*List C*" under the sub window "*To modify*" is chosen and on, location and magnitude of dynamic loads are entered by clicking "*Add an item*". The input box requires two parameters: a) *nldof* = degree-of-freedom number (printed in *run3.oug*) at which the load is applied, and b). *fdof* = linear scaling factor for *nldof* by which the input loads in *run3.eq1* are multiplied. This option is not used in this example.
8. While the option "*List D*" under the sub window "*To modify*" is chosen and on, a water table can be defined by clicking "*Add an item*". A water table already defined in a static analysis is maintained and transferred into a dynamic analysis when "*Dynamic On*" is turned on.
9. Review the water level parameter "*ywt0*" [default=0 for function not used]: An input box is located above the "*APPLY*" button in Figure 4.1. This function is invoked by entering a positive value (commonly the reservoir elevation) in the input box. Be very cautious in initiating this function. Its usage is recommended when submerged ground surface is expected to have large deformations under loading or ground shaking.

   Refer to Section 2.7 for details of requirements and its implication of 'ywt0 > 0 option'.

4.4. Assign Boundary Conditions for Dynamic Analysis

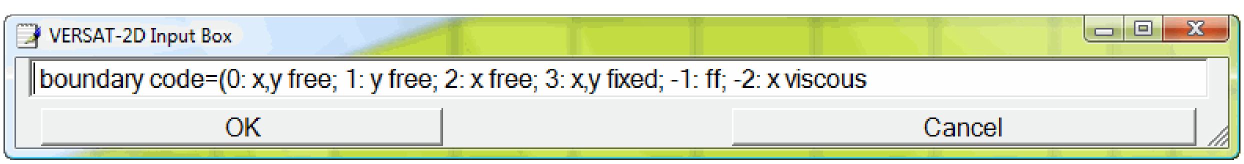
In addition to the boundary conditions used in a static analysis as described in Section 2.3, boundary conditions used in dynamic analysis also include the free-field stress boundary and the viscous boundary.

Free-field Stress Boundary

The free-field stress boundary is described in Section 3.12 of VERSAT-2D Technical Manual. This boundary condition should only be used in a dynamic analysis and only for side boundaries, and it is assigned by typing boundary code "-1" in the input box in Figure 4.2 (see Section 2.3).

Viscous Boundary

The viscous boundary must be, and is only, applied when "Hori Outcropping Velocity" is chosen as the option for input ground motion in Figure 4.1. In order to use this option, the finite element model should have a horizontal base with a viscous boundary. The viscous boundary condition is assigned by typing boundary code "-2" in the input box in Figure 4.2 (see Section 2.3).



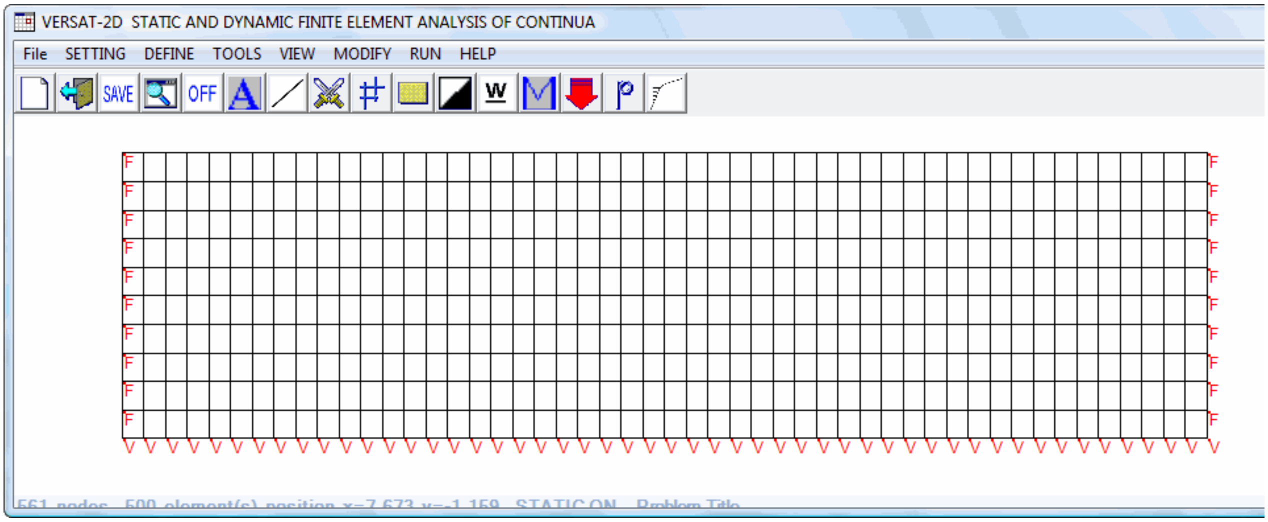


Figure 4.2 A finite element model assigned with free-field stress boundary on the sides and viscous boundary at the base

4.5. Define Soil and Structure Parameters for a Dynamic Analysis

Continue on the example:

1. choose "*Input material parameters*" under **DEFINE** to define material parameters for dynamic analysis as shown in Figure 4.3.
2. Enter "1" in material number box
3. Select "*Sand*" in the sub window "Select a Material Type". Note that the sub window "*PWP model parameters (DYNAMIC)*" is now enabled for PWP model and its parameters.
4. Edit the parameter boxes as needed [default values are shown herein, except R_f=1000]
5. Click "*Add/modify a material*" button
6. Enter "2" in material number box
7. Select "*Clay*" in the sub window "Select a Material Type"
8. Edit the parameter boxes as needed [default values are shown herein]
9. Click again "*Add/modify a material*" button
10. Click "*APPLY ALL*" button to save and exit this window.

Notes:

- ➢ Click on a parameter box in blue to see further explanations on the parameter
- ➢ See Section 3.4 of the Technical Manual for details of R_f
- ➢ See Section 3.5 of the Technical Manual for details of various PWP models.

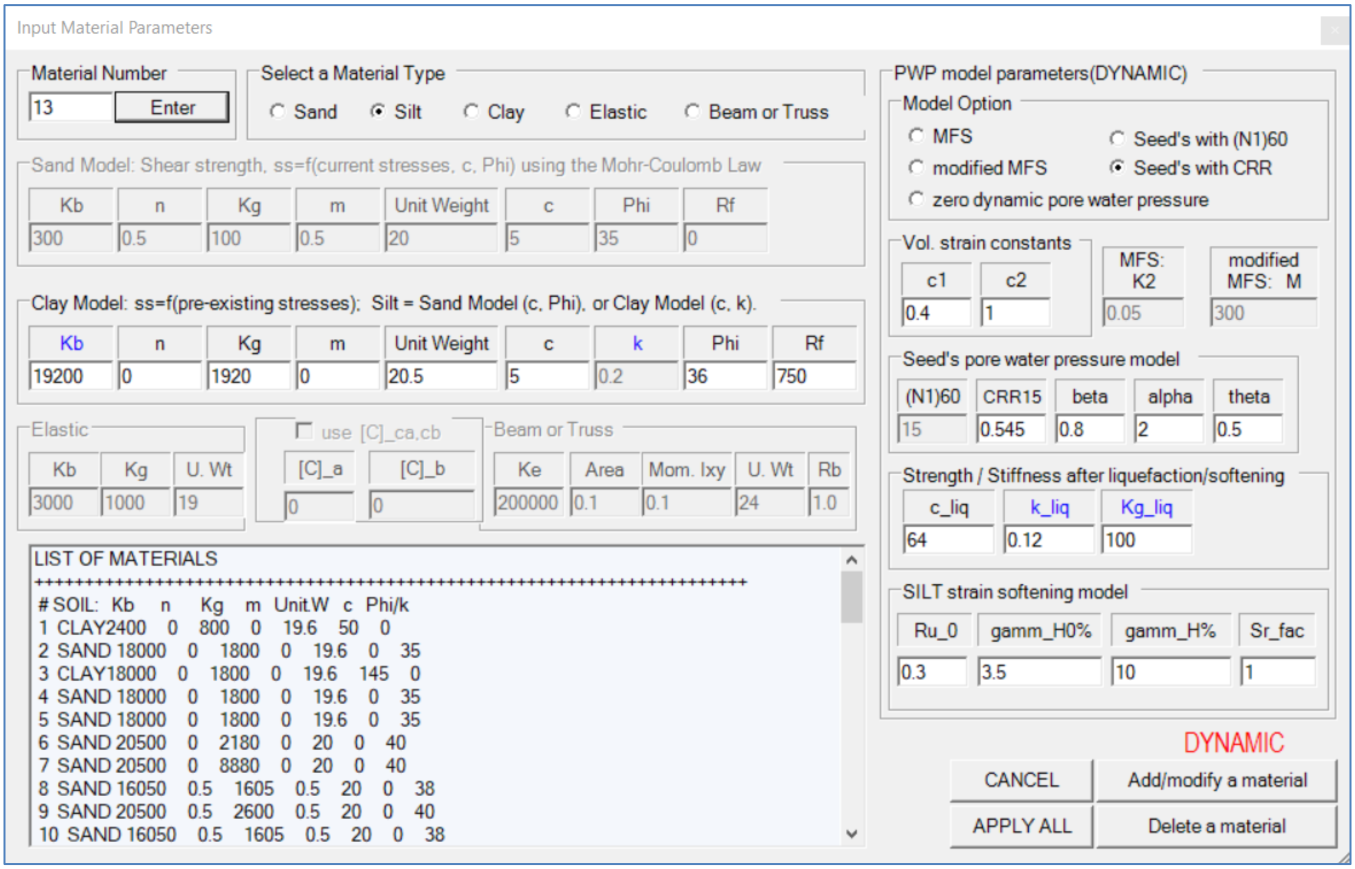


Figure 4.3 A window showing input parameters for a dynamic analysis (v.2019.10.3 and later)

4.6. Save Data and Start A Dynamic Analysis

Continue on the example:

- Save the data as *run3.dyn* using '*Save Data'* under **FILE**.
- Choose '*Run versat-d2d*' under **RUN**
- Enter the User Name and Password
- Press "*Connect Now*'
- Select the input file "run3.*dyn*" to run after "*Successful! Goto STEP 2*" in red is obtained as shown in Figure 4.4.

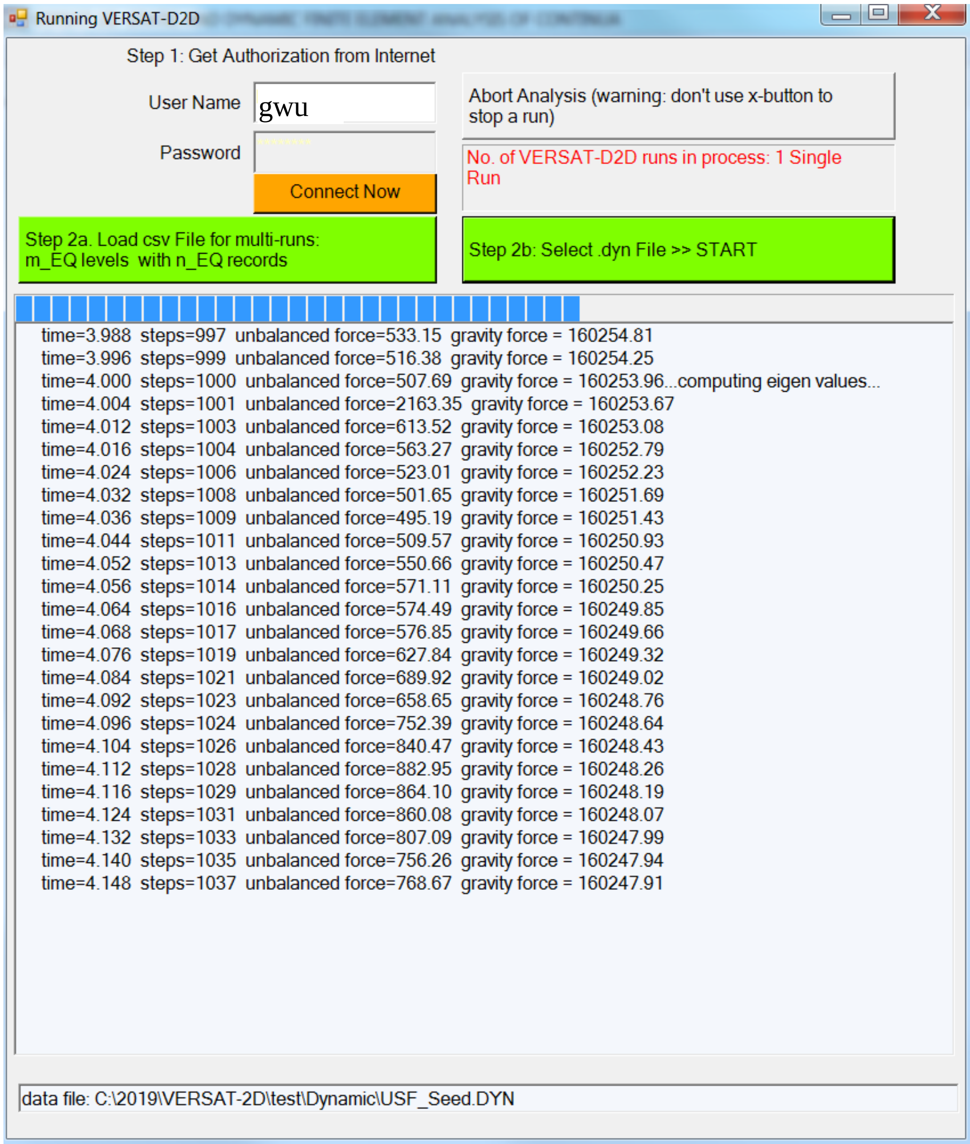


Figure 4.4 A window (v.2019.5.8 and later) showing a dynamic run in progress: A Single Run

4.7. Dynamic Analysis of One Dimensional Soil Column

An example of this application is provided in the example library under "*example_1D*".

1. Determine static stresses: Static stresses are computed from a static analysis of the one dimensional (1D) soil column. An example 1D soil column is shown in Figure 4.5.
   - For the static analysis, the boundary conditions of nodes along the two sides are "*free in vertical displacement*", i.e., boundary fixity = 1 (see Section 2.3).
   - The static stresses are required in order to compute the stiffness and shear strength parameters of the 1D soil column in a dynamic time-history analysis.

2. Follow steps in Sections 4.1 through 4.6 for a dynamic analysis of the 1D soil column with the following special treatments:
   - For the dynamic analysis, the boundary conditions of nodes along the two sides are "*free in horizontal displacement*", i.e., boundary fixity = 2 (see Section 2.3).
   - Specify "*Gravity OFF*" in "*General parameters*" under **DEFINE**. Refer Section 4.2 for this option.

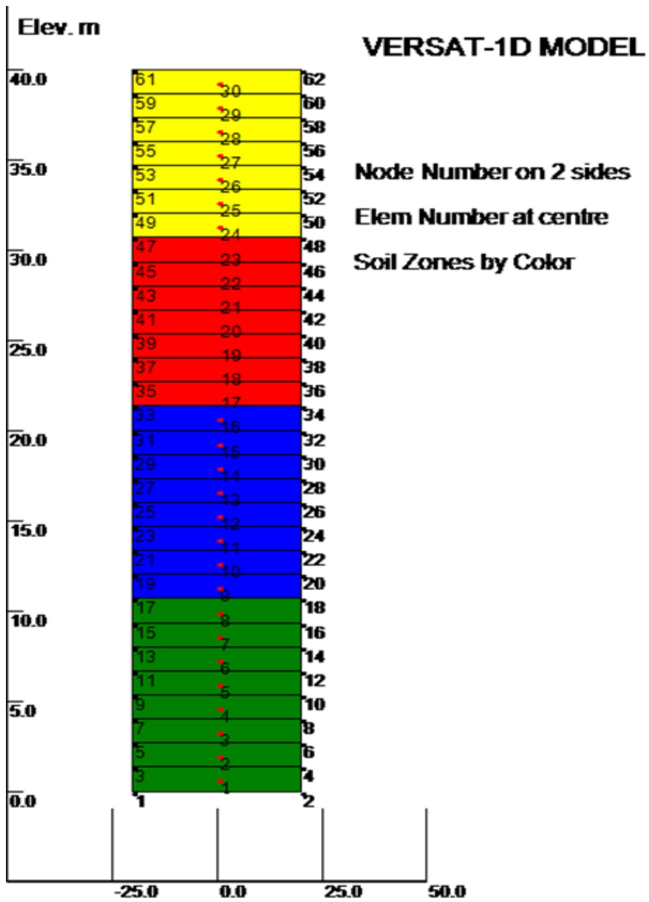


Figure 4.5 An example 1D soil column showing soil zones, a water table and boundary conditions for a static analysis

4.8 Input Files for Dynamic Run

The input files for a VERSAT-2D dynamic analysis consist of a minimum of 3 files:

1. Filename.DYN (main input file)
2. Filename.PRX (pre-existing stresses)
3. Filename.ACX (acceleration data for horizontal or X motion)
4. Filename.ACY (acceleration data for vertical or Y motion)

In above, "Filename.DYN" is prepared using VERSAT-2D Processor, i.e., the main window of the program; "Filename.PRX" is renamed from an output file of the static stress analysis (See Section 4.3 bullet 2); "Filename.ACX" is explained next, in Section 4.9 below. The 4th file, "Filename.ACY" is required when the vertical acceleration is also applied as input motion. The ACY file has the same format as ACX file.

The elastic base (instead of a rigid base) with "Outcrop Velocity" input option can be selected (See Section 4.1 of the Technical Manual, Section 4.3 bullet 3 and Figure 4.1 of this User Manual). In this case, "Filename.ACX" is replaced by the following:

5. Filename.VEX (outcrop velocity data for horizontal or X motion)

Instead of applying ground motions, forces at nodal points can be applied and selected (See Section 4.3 bullet 3 and Figure 4.1 of this User Manual). In this case, "Filename.ACX" is replaced by the following:

6 Filename.FXY (nodal forces to be applied at the specified nodes)

4.9 Input File Format for Filename.ACX, .ACY, .VEX, and .FXY

See Notes in Figure 4.1 and Section 4.3 bullet 3 of this User Manual for details. All these input files use the same format and contain 3 lines of heading and then data values:

- Line 1 Title (Record name, date, component, etc.)
- Line 2: NPOINT, DT, FAMPL, NRVSUB
- Line 3: NLINE, NoPerLine (total number of lines from Line 4, and data points per line)
- Line 4 data values separated by comma

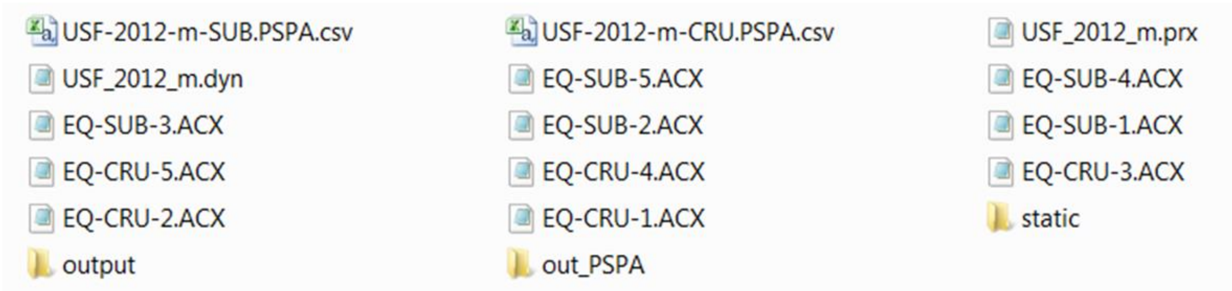
In above, NPOINT is number of data to be used in analysis, NPOINT <= NLINE*NoPerLine;

DT is the time (sec) increment for the data; FAMPL is scale factor to be applied to the data in analysis; NRVSUB is the number of sub time step (0, 1, 2, 3, 4, or higher integer). The time increment used in analysis "dt" is equal to DT/(1+NRVSUB), and data are added using a linear function in between "dt".

The data for ACX, ACY should carry the same unit as gravity acceleration, e.g., m/s^2; the corresponding data for VEX should be in "m/s", and FXY data should carry the unit of "kN".

4.10 Input and output files for PSPA Run

An additional input file, *"USF-2012-m-SUB.PSPA.csv"* *or* *"USF-2012-m-CRU.PSPA.csv"*, is required to conduct the PSPA multiple analyses; this file is loaded by selecting Step 2a in the VERSAT-D2D Run window. See below a list of files in "*VERSAT-2D_2019_Examples\Ex_PSPA-runs*":

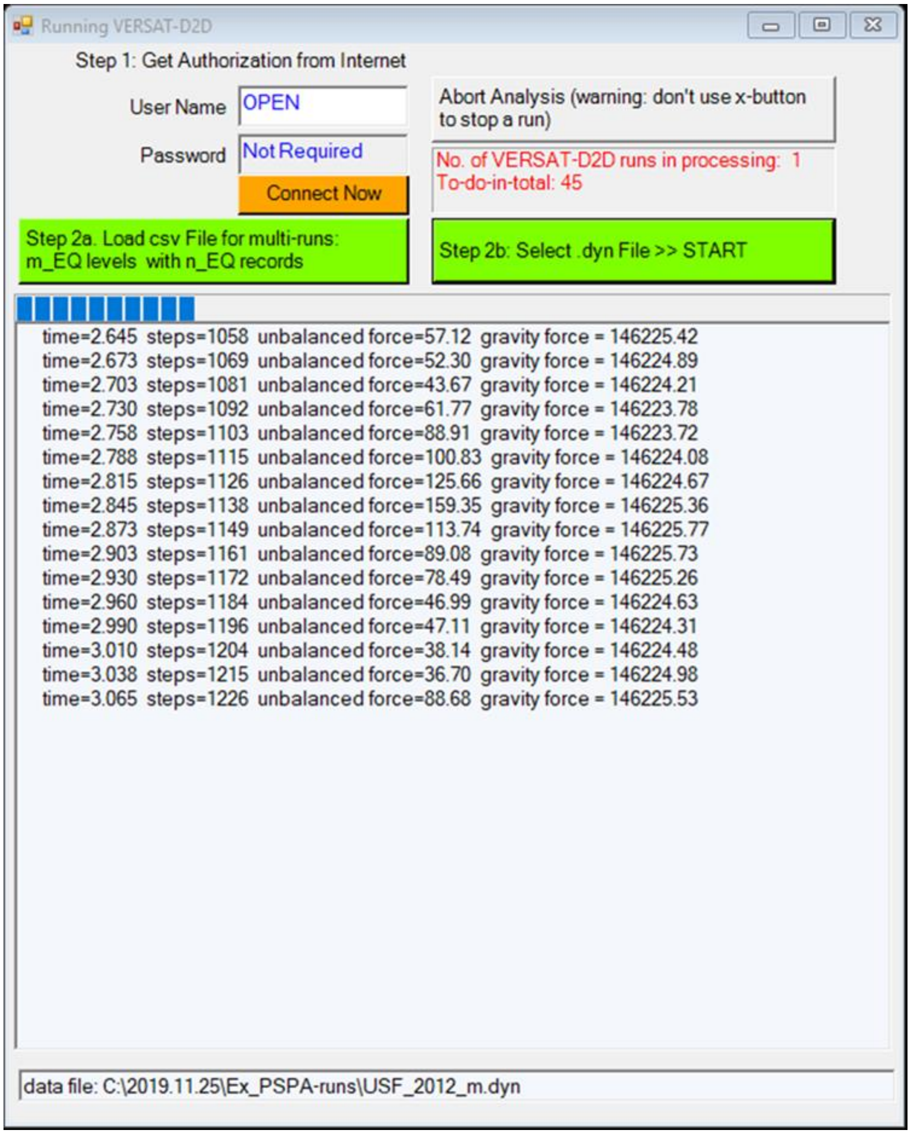


See below the content in the file "*USF-2012-m-CRU.PSPA.csv*" for detailed input format for setting up this input file. The highlighted areas are required and read; the rest are for information only.

	A	B	C	D	E
1	9	9 prob-level x 5 EQ = 45 runs			
2	Prob.Levl	SF_x	SF_y	AEP	Return-Yr
3	P1	0.46	0.46	0.001	1000
4	P2	0.6	0.6	0.000555556	1800
5	P3	0.74	0.74	0.00040404	2475
6	P4	0.88	0.88	0.000266667	3750
7	P5	1	1	0.0002	5000
8	P6	1.15	1.15	0.000125	8000
9	P7	1.32	1.32	0.0001	10000
10	P8	1.53	1.53	6.67E-05	15000
11	P9	1.69	1.69	0.00005	20000
12	5	N-points	dt	Comments	
13	EQ-CRU-1	902	0.02	EQ-CRU-1	
14	EQ-CRU-2	902	0.02	EQ-CRU-2	
15	EQ-CRU-3	902	0.02	EQ-CRU-3	
16	EQ-CRU-4	902	0.02	EQ-CRU-4	
17	EQ-CRU-5	902	0.02	EQ-CRU-5	

An additional output file will be generated by the program after completing each of the probability run. This file is named as "*Filename.Prob-level.record.csv", such as "USF-2012-m-CRU.PSPA.P9.EQ-CRU-5.csv*", which contains response data (peak and residual values) for the nodes and elements where time history response has been requested, as per Section 4.3 bullet 2 in this User Manual. The data are accumulated or appended to the end for each record; the output file from the last record in the series is used to generate the response hazard curves using the probabilistic approach (Wu 2017, 2018).

See below an image of the VERSAT-D2D Run window when executing multiple PSPA runs:



4.11 Use ICHANG=3 to Input Soil Strength by Element

The option is invoked by selecting ICHANG=3 in the "General parameter" window in VERSAT-2D Processor. When this option is selected, the data in the file "*Filename.soil-strength.CSV*" are read in by the program. In the current version, only three quantities (*Su, crr15, Su_liq)* in this file are used by the program to replace (supersede) the corresponding quantities calculated from the material zone parameters. This option is only valid in a dynamic analysis, i.e., not applicable to a static analysis. The functionality is the same as "Non-linear Effective Stress Analysis" or ICHANG=2.

This file "*Filename.soil-strength.CSV"* is also an output file when the advanced option of "ICHANG=3" is not selected; however, the same file becomes an input file when this option is selected. By this arrangement, the user has the choice of updating the three quantities for the desired soil elements based on in-situ stresses, which can be read in the file "*Filename.OUG*".

5 INTERPRETING RESULTS OF A STATIC ANALYSIS

5.1 Output Quantities

Now let's use "*ex_d10*" from the library of Examples (Ex_d10_UpperSanFernandoDam) as an example for Sections 5 and 6.

The main output file from a static analysis carries an extension of "*OUT*" such as *ex_d10.out*. The quantities in the main output file include '*node disp-x rot. disp-y elem sig-x(mx0) sig-y(ta) tau-xy(sh.) gamm_xy%(mi) pp su fos sig-m*'. The meanings of these quantities are explained as follows:

1. *node:* list of node number in this column;
2. *disp-x*: list of displacement in X-direction (incremental if imsh=0; cumulative if imsh=1).
3. *rot.:* rotation at this node (applicable for a beam node only);
4. *disp-y:* list of displacement in Y-direction;
5. *elem:* list of element number in this column;
6. *sig-x(mx0):* effective stress in horizontal (X) direction (σ_x) for a soil element; If this is a beam element, *mx0* is the bending moment at the centre of the element;
7. *sig-y(ta):* effective stress in vertical (Y) direction (σ_y); If this is a beam element, *ta* is the axial force in the element;
8. *tau-xy(sh.):* shear stress in the XOY plane (τ_{xy}); If this is a beam element, *sh.* is the shear force of the element;
9. *gamm_xy(%)(mi)*: shear strain (in percentage) in XOY plane. If this is a beam element, *mi* is the bending moment at the first node of the element;
10. *pp:* static pore water pressure in the element;
11. *su:* shear strength of the soil element;
12. *fos:* factor of safety against a shear failure;
13. *sig-m:* effective mean normal stress (σ_m).

To present the results using the Processor, nodal displacements are duplicated in an output file with an extension of "*DIS*" such as *ex_d10.dis*, and element quantities are duplicated in an output file with an extension of "*SIG*" such as *ex_d10.sig*.

The geometry output file carries an extension of "*OUG"*, such as *ex_d10.oug*, and contains the geometry input data including node information (X and Y coordinates, degrees of freedom) and element information (node composition, element type, material number, and pore water pressures).

The stress output file carries an extension of "*PR4*", such as *ex_d10.pr4*, and contains stresses at every Gauss point of the finite element model including structural elements. The stresses contained in this file are referred as the pre-existing stresses[3]. In a subsequent static or dynamic analysis, this file should be renamed or copied as an input file with an extension of "*PRX*".

5.2 Display Results of a Static Analysis Using the Processor

(see Appendix A for a plotting example)

1. Select **FILE** and choose *LOAD DATA* to load an input data file such as *ex_d10.sta*.
2. Select **SETTING** and choose *LOAD SETTING* to load a problem specific setting file such as *ex_d10.log*. This will allow the geometry of the model to be shown properly on screen.
3. Select **FILE** and choose *LOAD OUTPUT* to load the output files containing displacements and stresses such as *ex_d10.dis* and *ex_d10.sig*.
4. Select **VIEW** and choose "*MODEL VIEW OPTIONS"* to select the type of information you want to show on screen including node numbers, element numbers, material colors, boundary conditions, displacements, stresses, and others.

Note: The finite element model for *ex_d10.sta* is shown in Figure 5.1. More details on showing results of analyses are provided in Section 6.2.

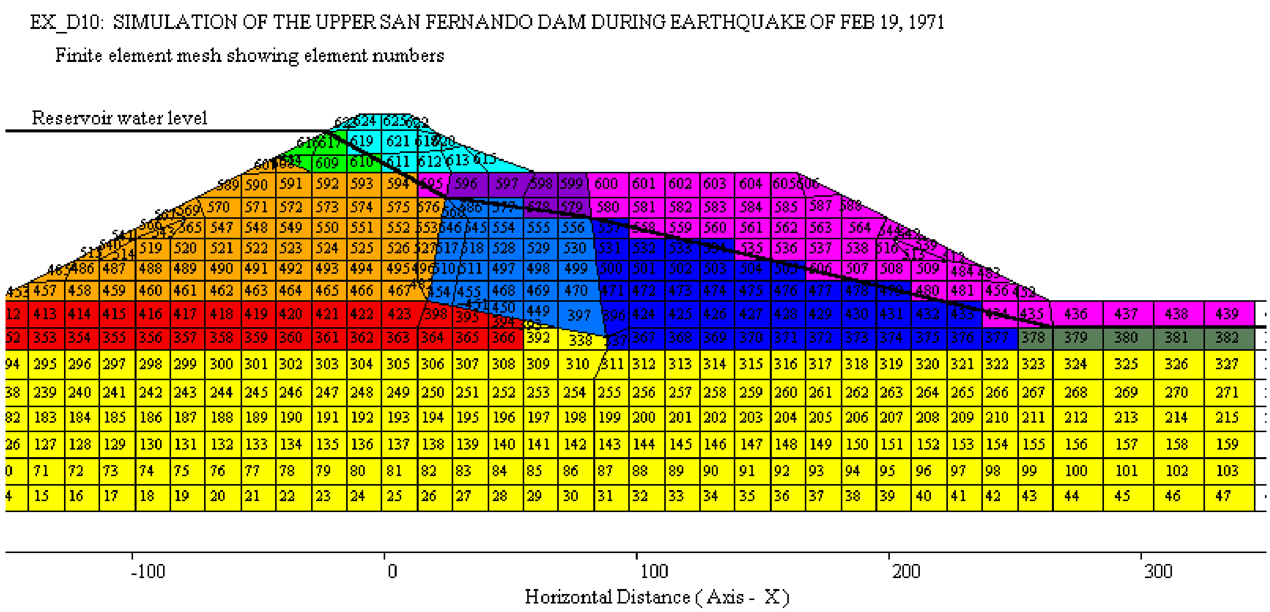


Figure 5.1 Finite element model showing element number and soil zones for ex_d10.sta

[3] In VERSAT-2D static and dynamic analyses, pre-existing stresses are always obtained from an input file with an extension of PRX, which contains model stresses computed from a previous static analysis and saved (in an output file with an extension of PR4) for a subsequent static analysis or for a dynamic analysis. The PR4 file is then renamed as PRX file.

6 INTERPRETING RESULTS OF A DYNAMIC ANALYSIS

6.1 Output Quantities

The main output file from a dynamic analysis carries an extension of "*OUD*" such as *ex_d10b.oud*. The quantities in the main output file include '*node disp-x disp-y acc-x(g) acc-y(g) elem sig-x(mx0) sig-y(ta) tauxy(sh.) gamm_xy%(mj) PEAKgamm_max(%) vol(%) ppr(FSliq)*'. The meanings of these quantities are explained as follows:

1. *node:* list of node number in this column;
2. *disp-x*: list of displacement in X-direction (instant at time t; or maximum);
3. *disp-y:* list of displacement in Y-direction;
4. *acce-x*: list of acceleration in X-direction (instant at time t; or maximum);
5. *acce-y:* list of acceleration in Y-direction;
6. *elem:* list of element number in this column;
7. *sig-x(mx0):* effective stress in horizontal (X) direction (σ_x) for a soil element; If this is a beam element, *mx0* is the bending moment at the centre of the element;
8. *sig-y(ta):* effective stress in vertical (Y) direction (σ_y); If this is a beam element, *ta* is the axial force in the element;
9. *tauxy(sh.):* shear stress (including static shear stress) in the XOY plane (τ_{xy}); If this is a beam element, *sh.* is the shear force of the element;
10. *gamm_xy%(mj):* shear strain (in percentage) in the XOY plane (γ_{xy}); If this is a beam element, *mj* is not defined;
11. **(a)** *PEAKgamm_max(%)*: peak dynamic maximum shear strain of the element up to current time, γ_{max} = square root $[\gamma_{xy}^2 + (\varepsilon_x - \varepsilon_y)^2]$. Output with group of quantities for "***state at the end of earthquake…",*** i.e., 1st set in the PSPA output

    (b) *DynStressRatio* = tauxy_dyn/σ_{v0}': the ratio of peak dynamic shear stress (not including static shear stress) over the initial effective vertical stress, e.g. CSR=0.65•*DynStressRatio***.** Output with group of quantities for **"peak dynamic response between …"**, 2nd set in PSPA.
12. *vol(%):* volumetric strain in percentage (%) caused by dynamic loads;
13. *ppr(FSliq):* dynamic pore water pressure ratio (PPR) or factor of safety against liquefaction (FSliq).

To present the results using the Processor, nodal displacements are duplicated in an output file with an extension of "*DIS*" such as *ex_d10b.dis*, and element quantities are duplicated in an output file with an extension of "*SIG*" such as *ex_d10b.sig*.

The geometry output file carries an extension of "*OUG"*, such as *ex_d10b.oug*, and contains the geometry input data including node information (X and Y coordinates, degrees of freedom) and element information (node composition, element type, material number, and pore water pressures).

6.2 Display Results of a Dynamic Analysis Using the Processor

1. Select **SETTING** and choose *DYNAMIC ON* to turn on dynamic option.
2. Select **FILE** and choose *LOAD DATA* to load an input data file such as *ex_d10b.dyn*.
3. Select **SETTING** and choose *LOAD SETTING* to load a problem specific setting file such as *ex_d10.log*. This will allow the geometry of the model to be shown properly on screen.
4. Select **FILE** and choose *LOAD OUTPUT* to load the output files containing displacements and stresses such as *ex_d10b.dis* and *ex_d10b.sig*. It is noted that the Processor can only load ***one*** set of results (displacements, stresses etc) to display. Therefore, the first set of results must be deleted from a DIS file and a SIG file in order to load the second set of results.
5. Select VIEW and choose "*MODEL VIEW OPTIONS*" to select the type of information you want to show on screen including node numbers, element numbers, material colors, boundary conditions, displacements, stresses, and others.

Notes:

- An example of ground displacements at the end of shaking is shown in Figure 6.1: Ex_d9_Bridge Crossing with Piles\ dynamic analysis\ ex_d9_NicoM_1c.*dyn* as input data.
- The data entered in the windows in Figure 6.1, such as "*Value range for color*", can be saved in a setting file. The setting file can be retrieved later. Details on setting file are provided in Section 2.1.

6.3 Retrieving Time-History Response

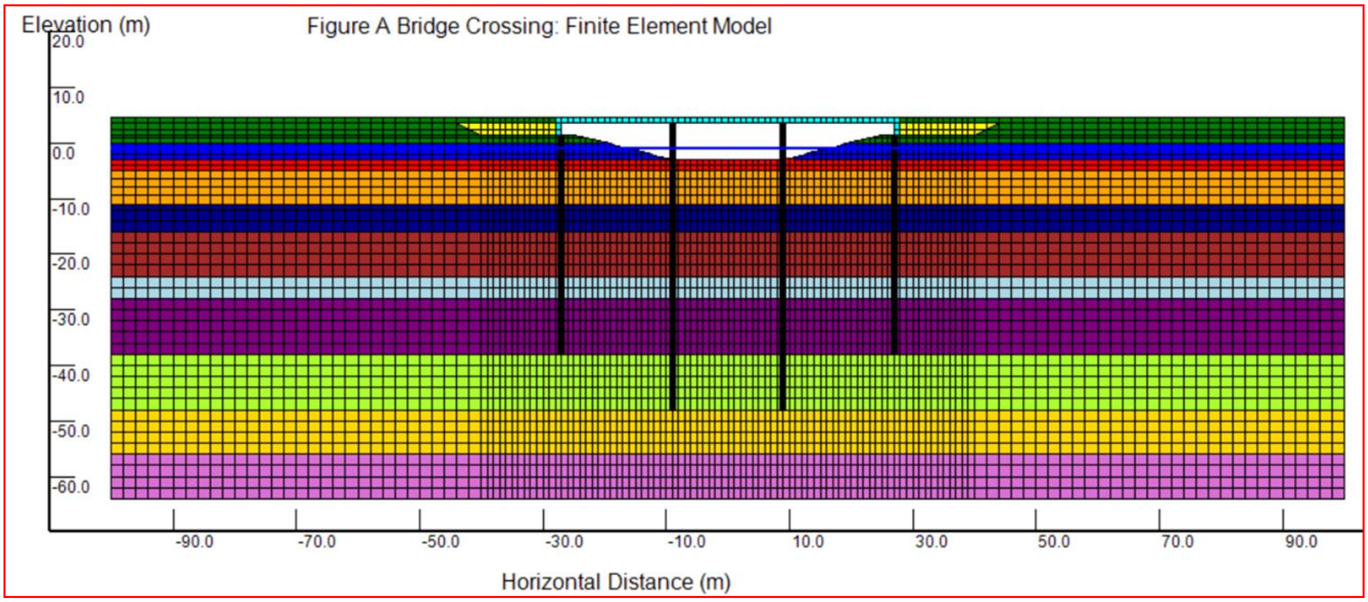
The time history data are saved in a file with an extension of CSV, such as ex_d10b.csv. Bullet 5 of Section 4.3 provides details on how to obtain time history data for acceleration and displacement for nodes, and stress, strain and pore water pressure data for elements.

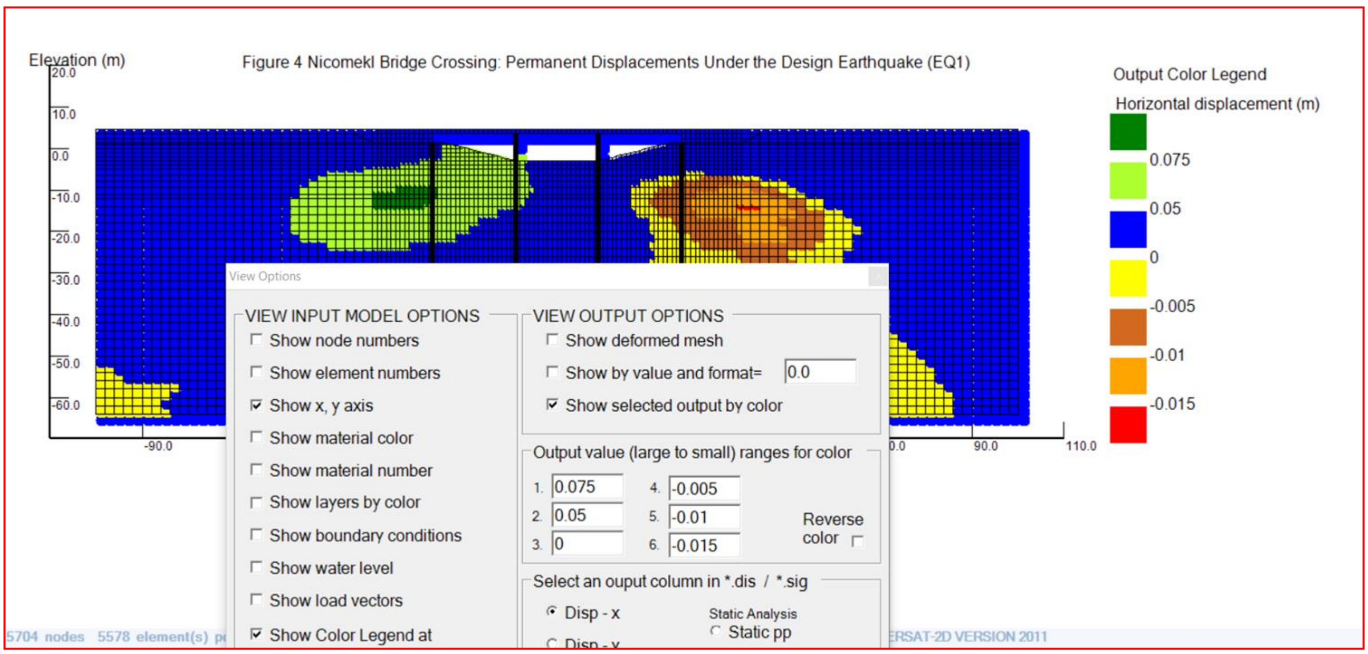
6.4 Regarding Nodal Response for Outcropping Velocity Option

When the Outcropping Velocity Option is used, absolute values of displacement and acceleration at all nodes in the finite element model are computed directly by the program, and therefore all reported quantities (including instant values, maximum values and time history data) on displacements and accelerations should be interpreted as absolute values[4].

[4] When the Acceleration Option is used, the reported quantities on displacements and velocities are relative to the model base.

Figure 6.1 Display finite element model (material in color) and horizontal displacements (Disp-x) in color



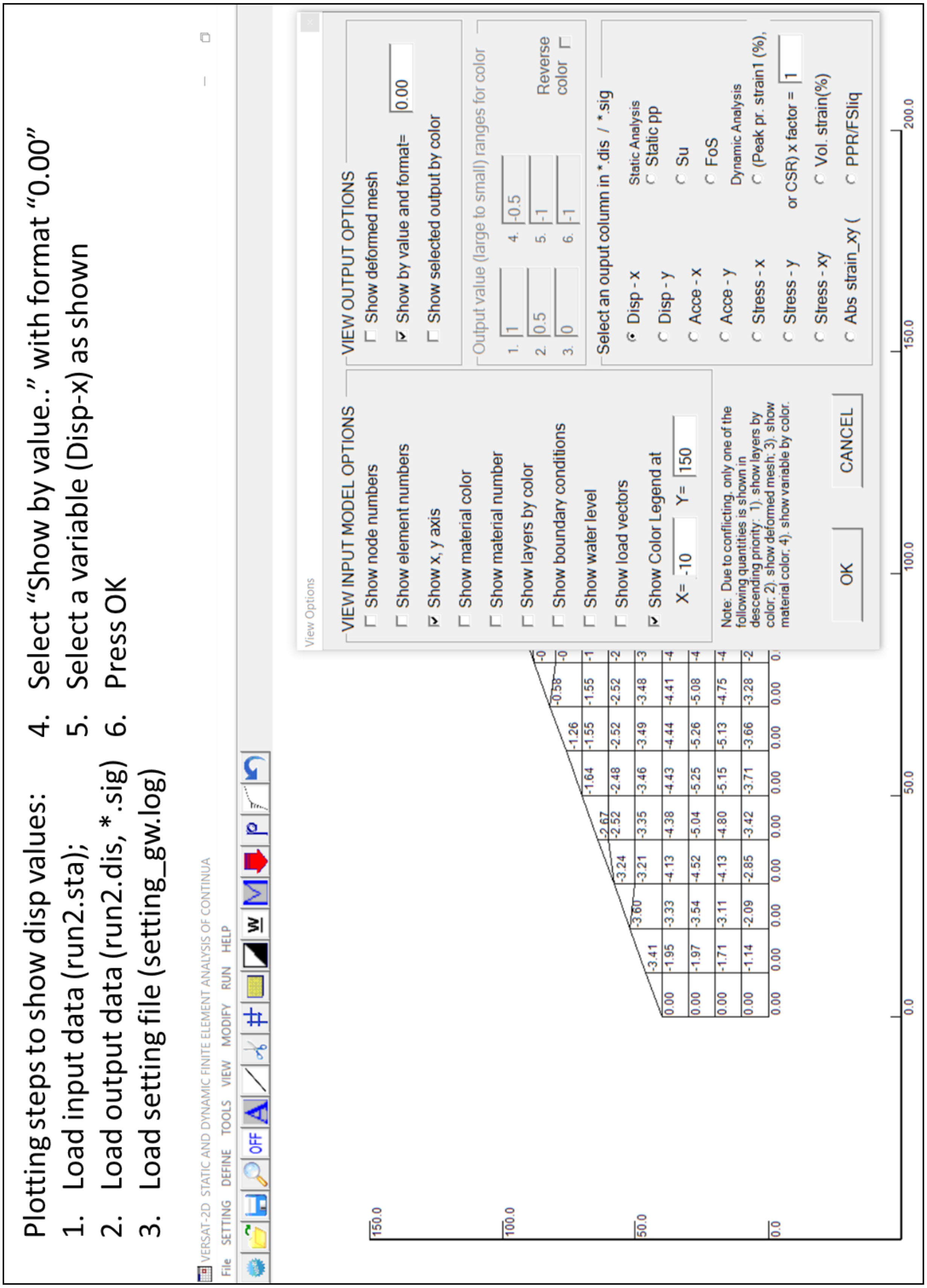


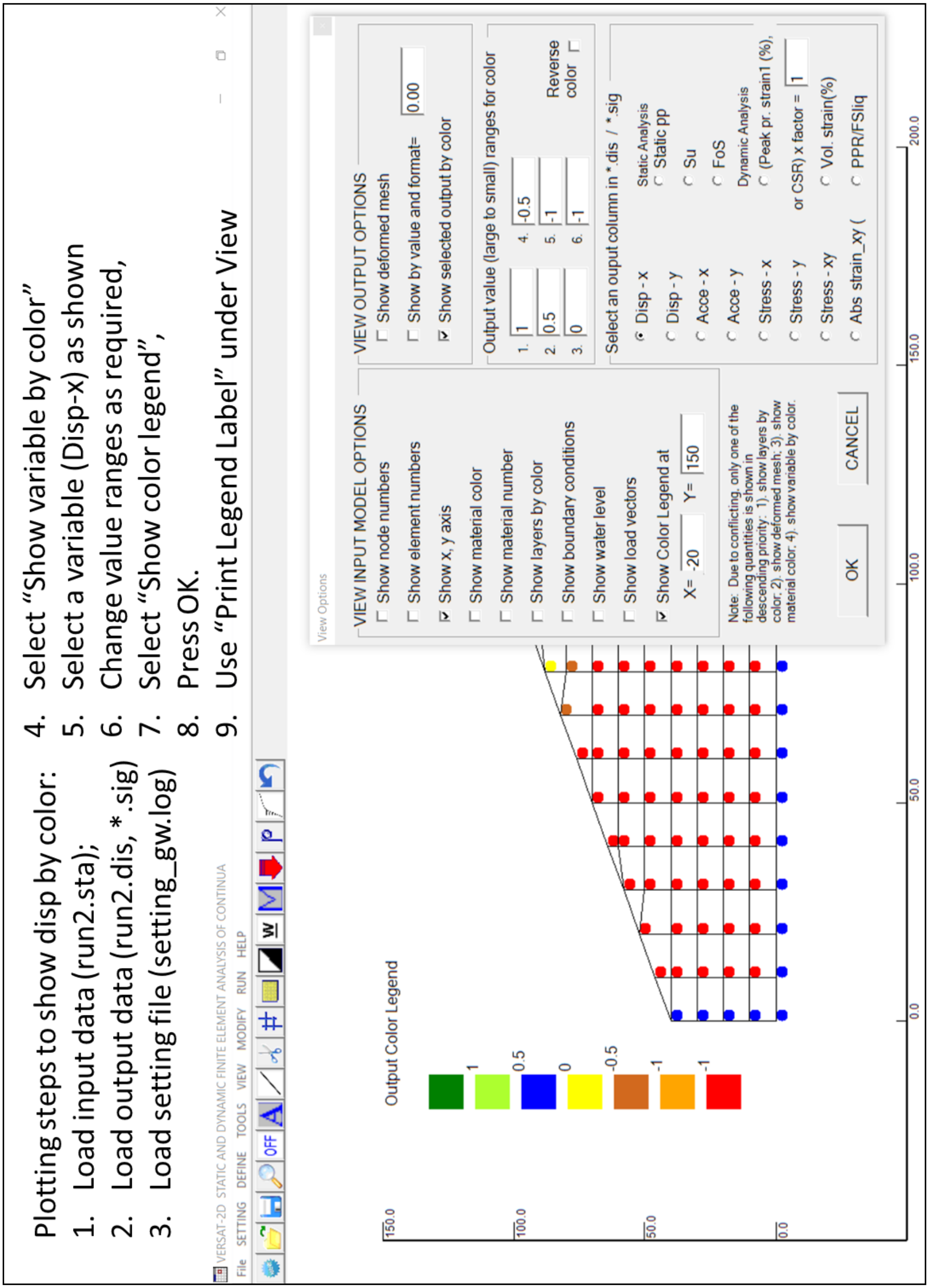
APPENDIX A: EXAMPLES FOR PLOTTING

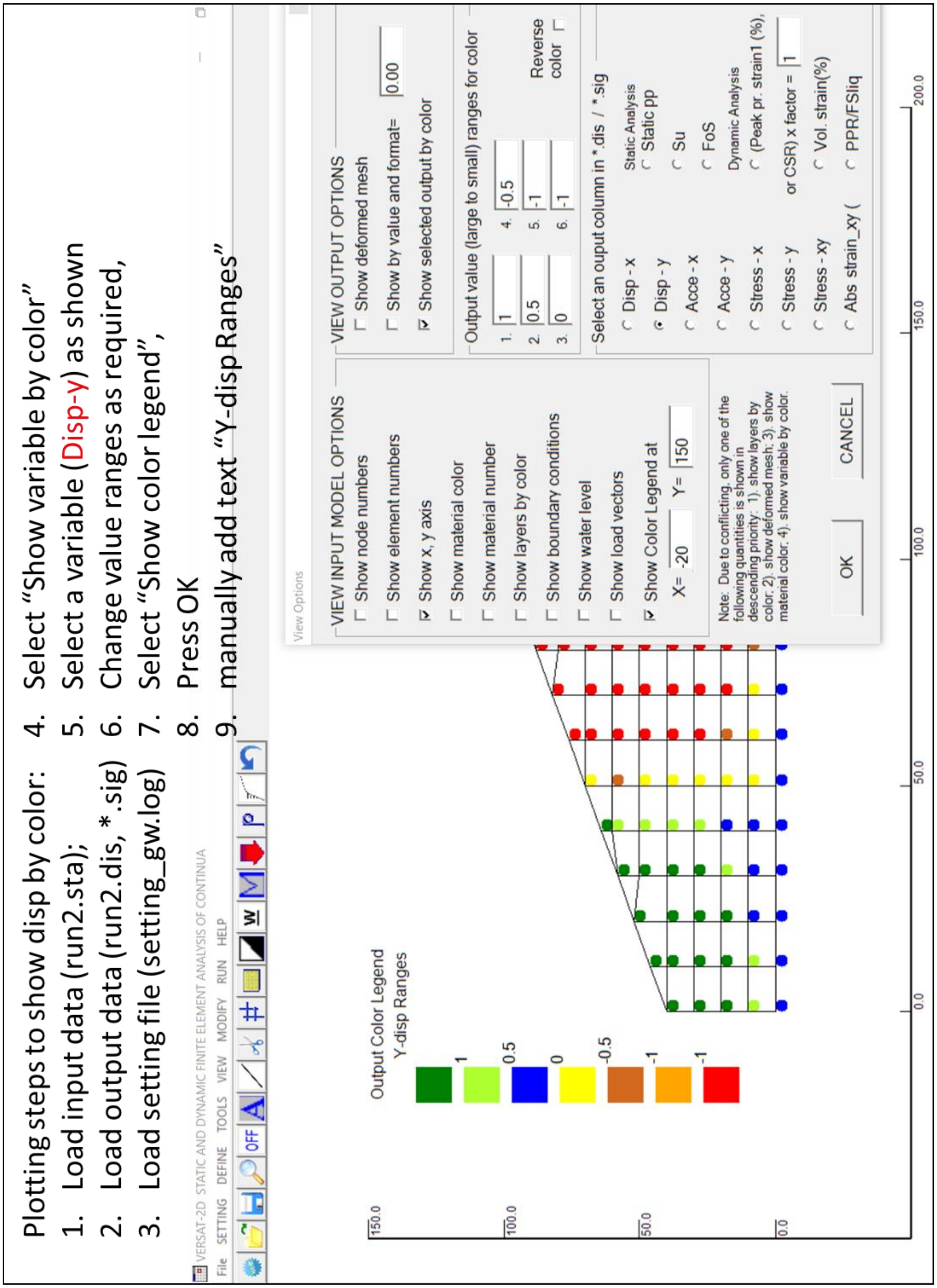
displacements

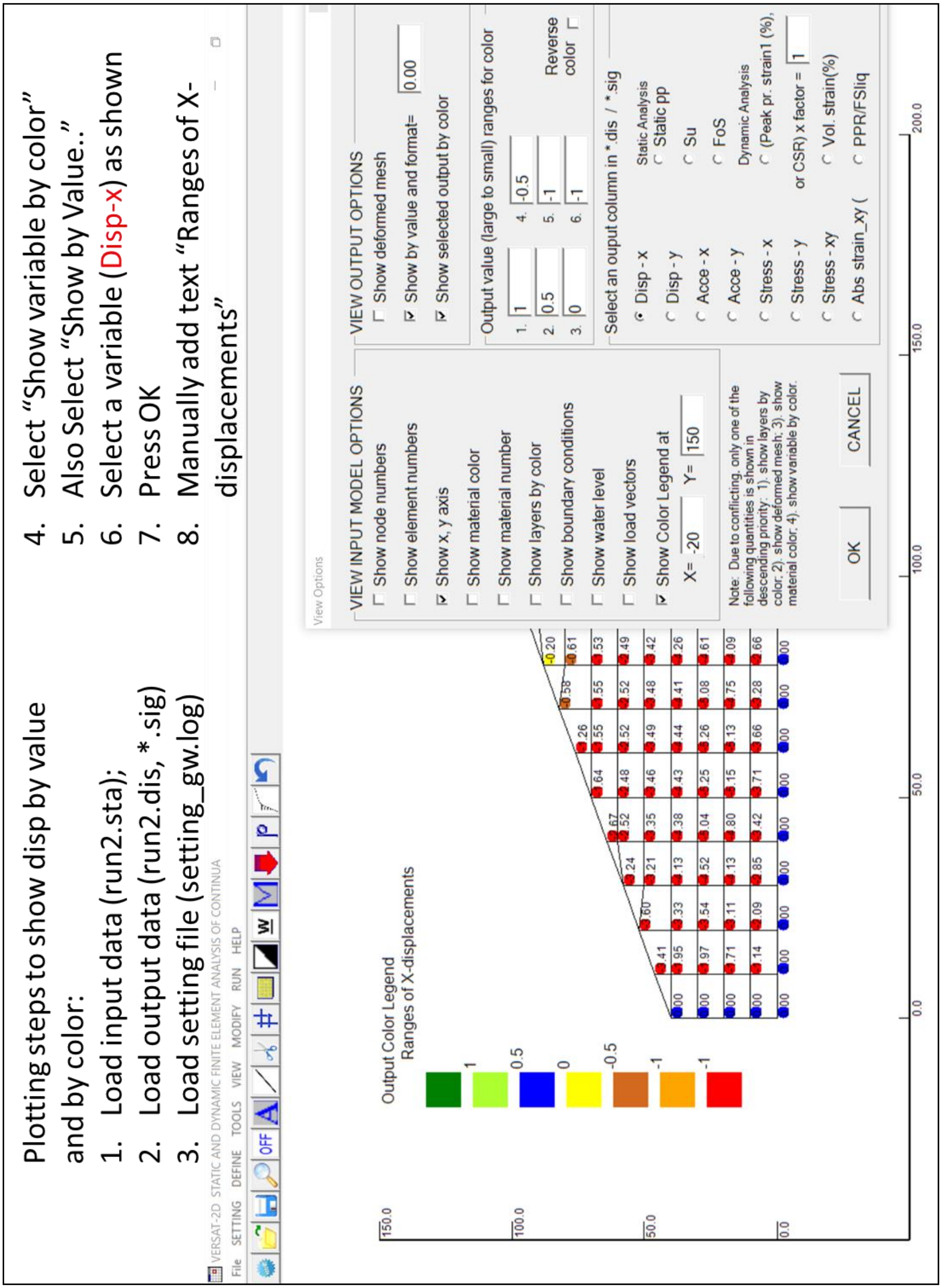
by values or by color

December 8, 2016


Plotting steps to show disp values:
1. Load input data (run2.sta);
2. Load output data (run2.dis, *.sig)
3. Load setting file (setting_gw.log)
4. Select "Show by value.." with format "0.00"
5. Select a variable (Disp-x) as shown
6. Press OK
VERSAT-2D STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS OF CONTINUA
File SETTING DEFINE TOOLS VIEW MODIFY RUN HELP
View Options
VIEW INPUT MODEL OPTIONS
Show node numbers
Show element numbers
Show x, y axis
Show material color
Show material number
Show layers by color
Show boundary conditions
Show water level
Show load vectors
Show Color Legend at
X= -10 Y= 150
Note: Due to conflicting, only one of the following quantities is shown in descending priority: 1). show layers by color; 2). show deformed mesh; 3). show material color; 4). show variable by color.
OK
CANCEL
VIEW OUTPUT OPTIONS
Show deformed mesh
Show by value and format= 0.00
Show selected output by color
Output value (large to small) ranges for color
1. 1
2. 0.5
3. 0
4. -0.5
5. -1
6. -1
Reverse color
Select an ouput column in *.dis / *.sig
Disp - x
Disp - y
Acce - x
Acce - y
Stress - x
Stress - y
Stress - xy
Abs strain_xy (
Static Analysis
Static pp
Su
FoS
Dynamic Analysis
(Peak pr. strain1 (%),
or CSR) x factor = 1
Vol. strain(%)
PPR/FSliq
150.0
100.0
50.0
0.0
0.0
50.0
100.0
150.0
200.0


Plotting steps to show disp by color:
1. Load input data (run2.sta);
2. Load output data (run2.dis, *.sig)
3. Load setting file (setting_gw.log)
4. Select "Show variable by color"
5. Select a variable (Disp-x) as shown
6. Change value ranges as required,
7. Select "Show color legend",
8. Press OK.
9. Use "Print Legend Label" under View
VERSAT-2D STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS OF CONTINUA
File SETTING DEFINE TOOLS VIEW MODIFY RUN HELP
Output Color Legend
1
0.5
0
-0.5
-1
-1
View Options
VIEW INPUT MODEL OPTIONS
Show node numbers
Show element numbers
Show x, y axis
Show material color
Show material number
Show layers by color
Show boundary conditions
Show water level
Show load vectors
Show Color Legend at
X= -20 Y= 150
Note: Due to conflicting, only one of the following quantities is shown in descending priority: 1). show layers by color; 2). show deformed mesh; 3). show material color; 4). show variable by color.
OK
CANCEL
VIEW OUTPUT OPTIONS
Show deformed mesh
Show by value and format= 0.00
Show selected output by color
Output value (large to small) ranges for color
1. 1
2. 0.5
3. 0
4. -0.5
5. -1
6. -1
Reverse color
Select an ouput column in *.dis / *.sig
Disp - x
Disp - y
Acce - x
Acce - y
Stress - x
Stress - y
Stress - xy
Abs strain_xy (
Static Analysis
Static pp
Su
FoS
Dynamic Analysis
(Peak pr. strain1 (%),
or CSR) x factor = 1
Vol. strain(%)
PPR/FSliq
150.0
100.0
50.0
0.0
0.0
50.0
100.0
150.0
200.0


Plotting steps to show disp by color:
1. Load input data (run2.sta);
2. Load output data (run2.dis, *.sig)
3. Load setting file (setting_gw.log)
4. Select "Show variable by color"
5. Select a variable (Disp-y) as shown
6. Change value ranges as required,
7. Select "Show color legend",
8. Press OK
9. manually add text "Y-disp Ranges"
VERSAT-2D STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS OF CONTINUA
File SETTING DEFINE TOOLS VIEW MODIFY RUN HELP
Output Color Legend
Y-disp Ranges
1
0.5
0
-0.5
-1
-1
View Options
VIEW INPUT MODEL OPTIONS
Show node numbers
Show element numbers
Show x, y axis
Show material color
Show material number
Show layers by color
Show boundary conditions
Show water level
Show load vectors
Show Color Legend at
X= -20 Y= 150
Note: Due to conflicting, only one of the following quantities is shown in descending priority: 1). show layers by color; 2). show deformed mesh; 3). show material color; 4). show variable by color.
OK
CANCEL
VIEW OUTPUT OPTIONS
Show deformed mesh
Show by value and format= 0.00
Show selected output by color
Output value (large to small) ranges for color
1. 1
2. 0.5
3. 0
4. -0.5
5. -1
6. -1
Reverse color
Select an ouput column in *.dis / *.sig
Disp - x
Disp - y
Acce - x
Acce - y
Stress - x
Stress - y
Stress - xy
Abs strain_xy (
Static Analysis
Static pp
Su
FoS
Dynamic Analysis
(Peak pr. strain1 (%),
or CSR) x factor = 1
Vol. strain(%)
PPR/FSliq
150.0
100.0
50.0
0.0
50.0
100.0
150.0
200.0


Plotting steps to show disp by value and by color:
1. Load input data (run2.sta);
2. Load output data (run2.dis, *.sig)
3. Load setting file (setting_gw.log)
4. Select "Show variable by color"
5. Also Select "Show by Value.."
6. Select a variable (Disp-x) as shown
7. Press OK
8. Manually add text "Ranges of X-displacements"
VERSAT-2D STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS OF CONTINUA
File SETTING DEFINE TOOLS VIEW MODIFY RUN HELP
Output Color Legend
Ranges of X-displacements
View Options
VIEW INPUT MODEL OPTIONS
Show node numbers
Show element numbers
Show x, y axis
Show material color
Show material number
Show layers by color
Show boundary conditions
Show water level
Show load vectors
Show Color Legend at
X= -20 Y= 150
Note: Due to conflicting, only one of the following quantities is shown in descending priority: 1). show layers by color; 2). show deformed mesh; 3). show material color; 4). show variable by color.
OK
CANCEL
VIEW OUTPUT OPTIONS
Show deformed mesh
Show by value and format= 0.00
Show selected output by color
Output value (large to small) ranges for color
1. 1
2. 0.5
3. 0
4. -0.5
5. -1
6. -1
Reverse color
Select an ouput column in *.dis / *.sig
Disp - x
Disp - y
Acce - x
Acce - y
Stress - x
Stress - y
Stress - xy
Abs strain_xy (
Static Analysis
Static pp
Su
FoS
Dynamic Analysis
(Peak pr. strain1 (%),
or CSR) x factor = 1
Vol. strain(%)
PPR/FSliq

APPENDIX B: VERSAT-2D Model Preparation for Upper San Fernando Dam

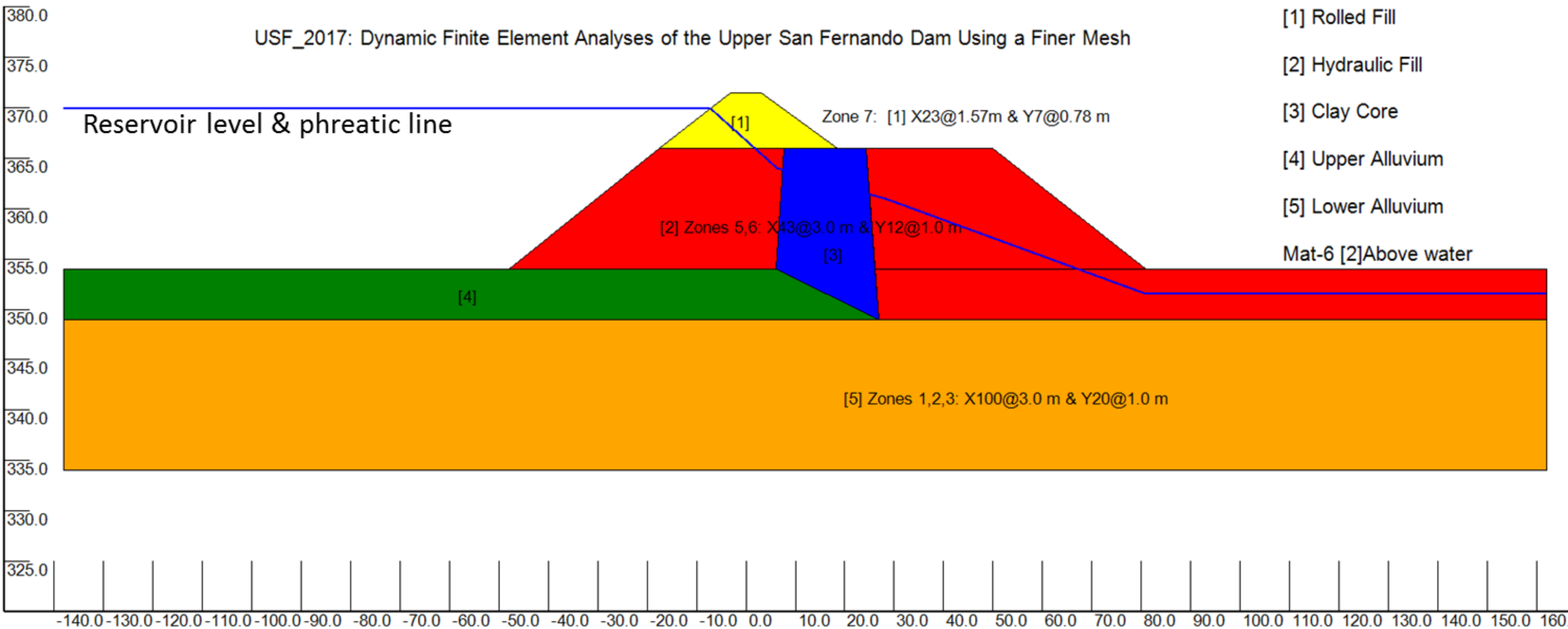
(see Volume 1 Technical Manual for Results of Analyses)

March 5, 2017

ATTACHMENTS:

Procedures for making the finite element model used in the dynamic analysis of the Upper San Fernando Dam;

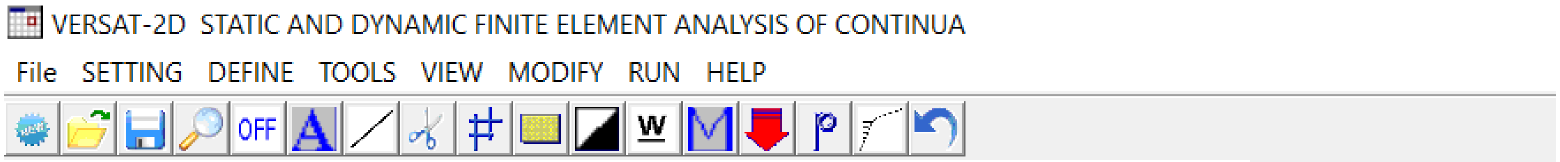
Step 0: Sketch on paper for five soil units



1. Make finite elements for 3 areas & SORT:

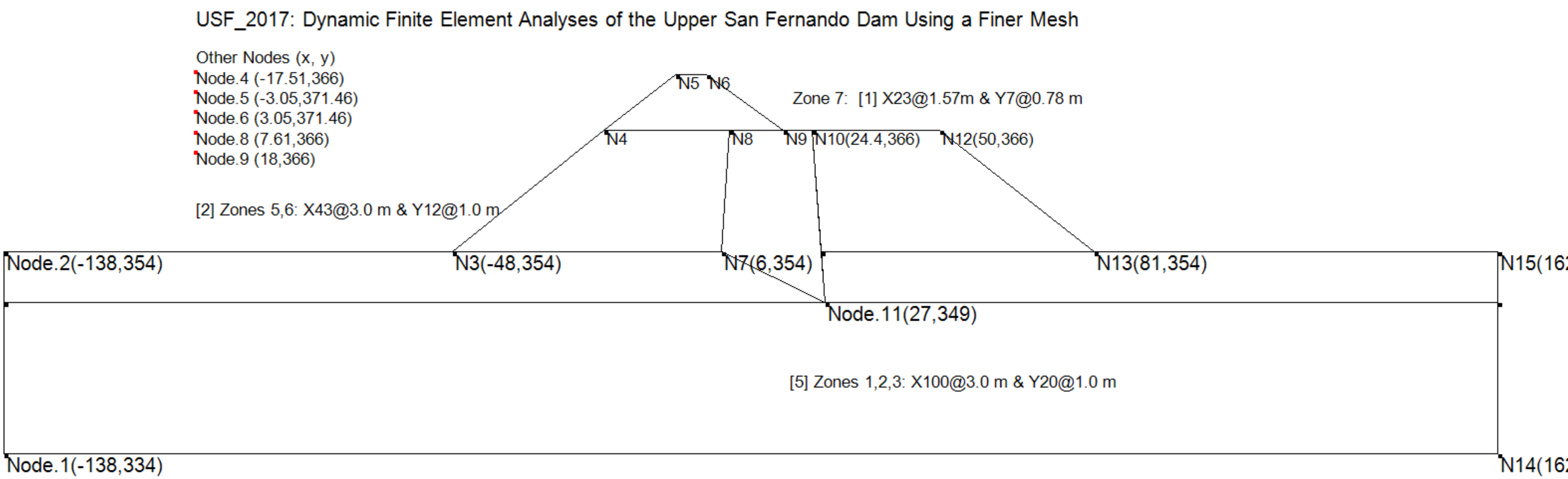
(download VERSAT-2D from: http://www.wutecgeo.com/documents/VERSAT-2D_2016gwu.zip

(1). Start VERSAT-2D Processor by "Accept" terms:



(2). Change view option: **VIEW** => *Draw Marker for Node/Elem => Show Marker Only*

(3). under **SETTING**, load "USF_m_1.log" & under **File**, load data "*USF_1_view_only.sta*"



Note: The colored soil units (in previous slide) can be shown by doing the following:

VIEW *Model View Options => Show x, y axis; Show Material color*

1. Make finite elements for 3 areas & SORT:

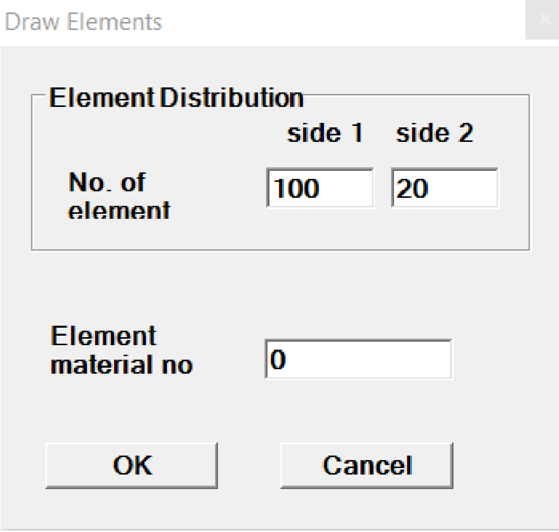
(4). Under **File**, load model data "*USF_1_nodes.sta*"

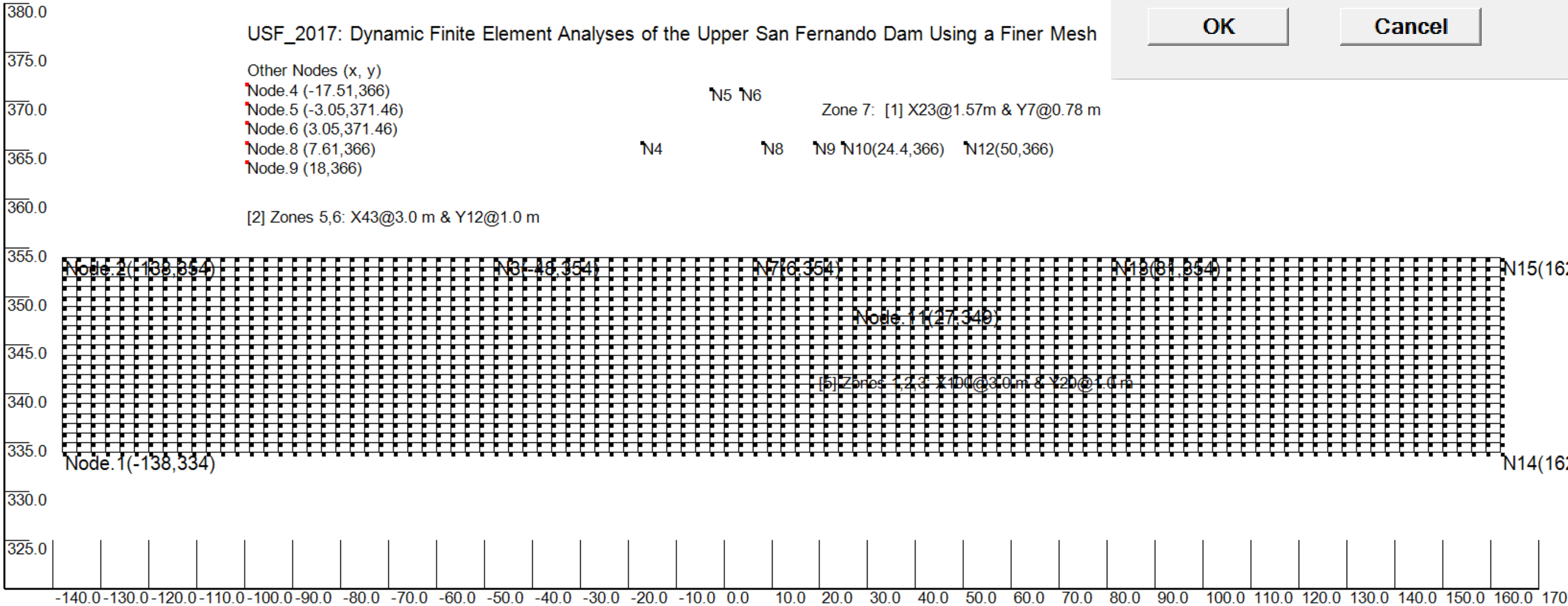
(5). Make Area 1: 100 x 20 by:

(a).**TOOLS** => Draw finite element grid => enter: 100, 20, 0 => OK

(b) Snap sequentially nodes:

Node1, N14, N15 and Node2 to create the grids below





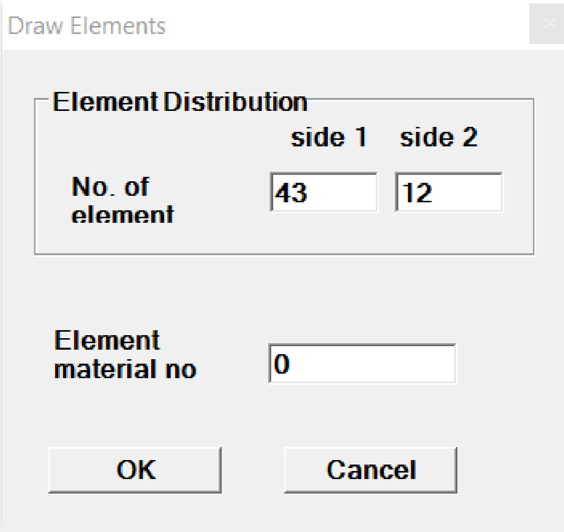
1. Make finite elements for 3 areas & SORT:

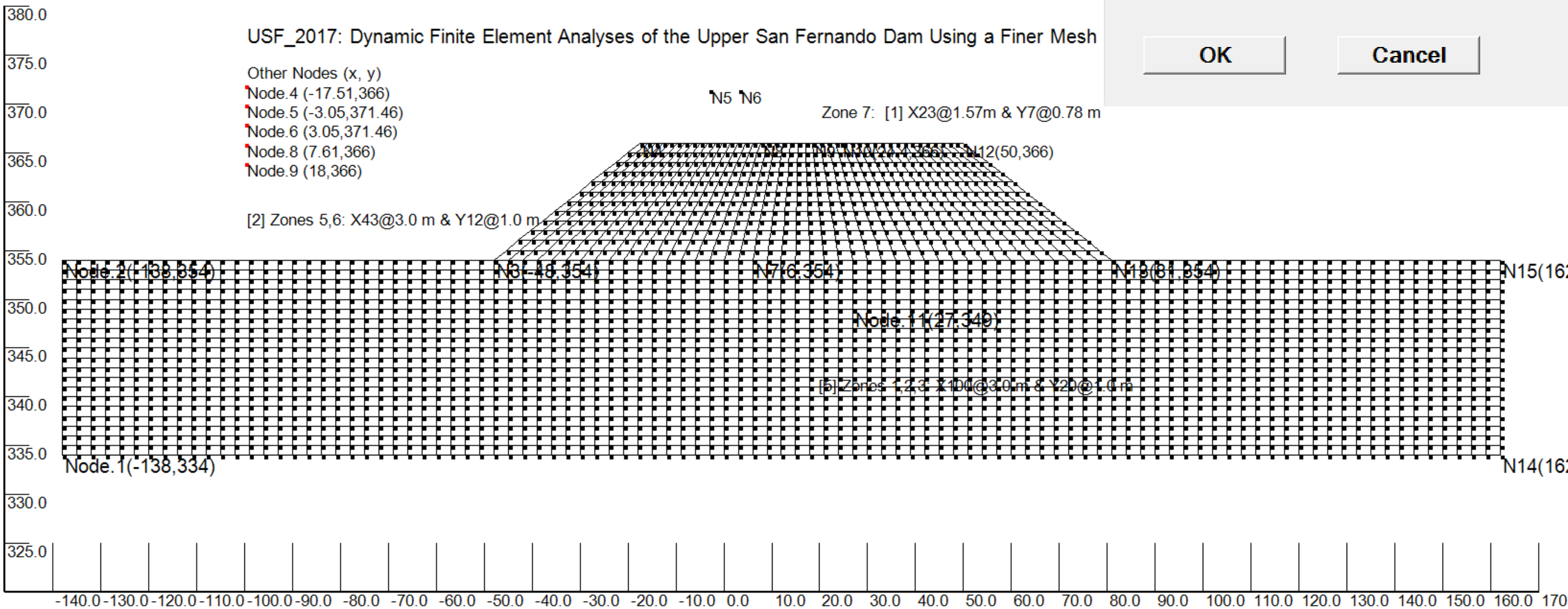
(6). Make Area 2: 43 x 12 by:

TOOLS => Draw finite element grid => enter: 43, 12, 0 ; OK

Snap sequentially nodes:

N3, N13, N12 and N4 to create the grids below





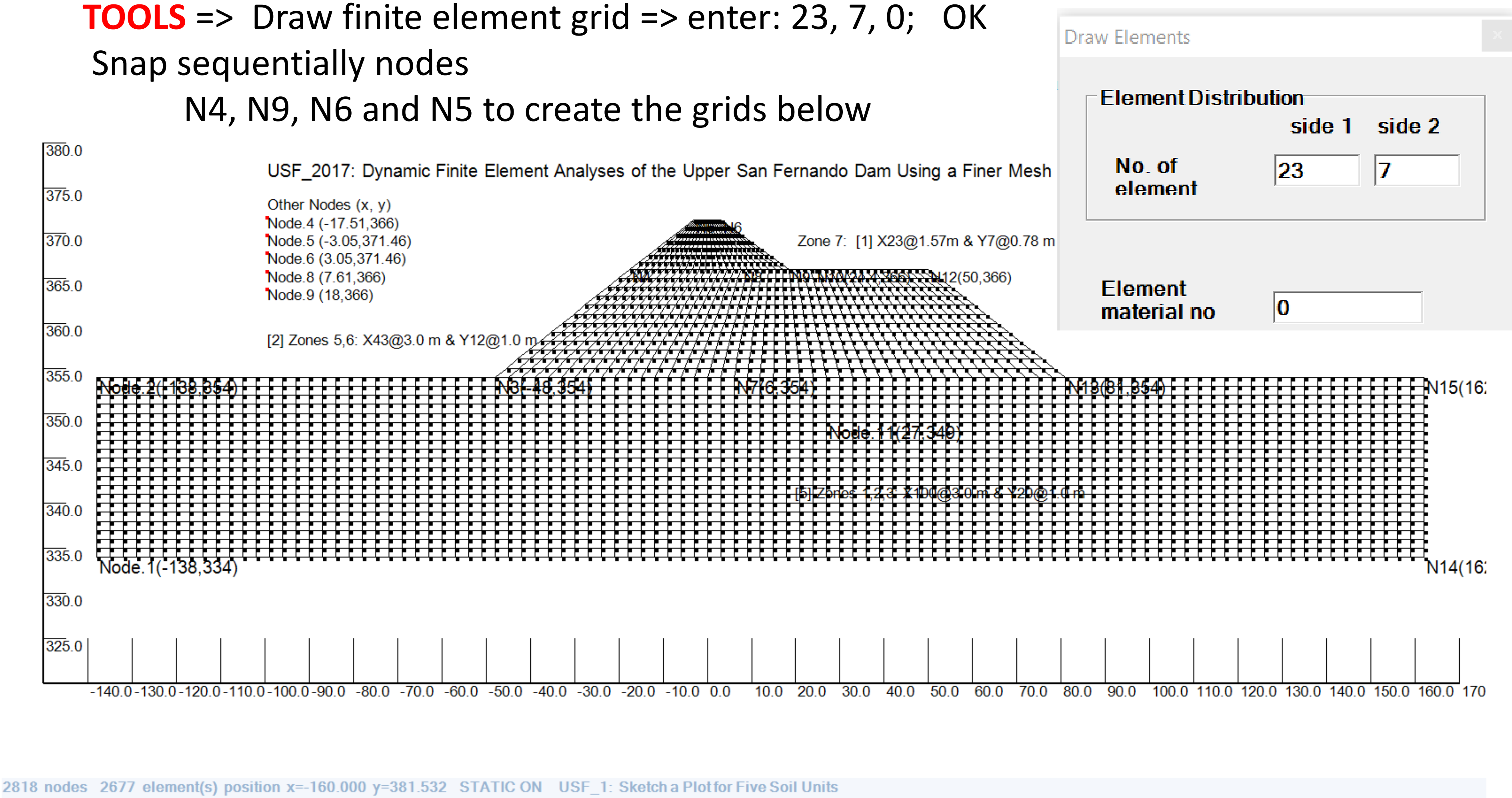
1. Make finite elements for 3 areas & SORT

(7). Make Area 3: 23 x 7 by: [under **SETTING**, load "USF_upper_dam.log" for a larger view]

TOOLS => Draw finite element grid => enter: 23, 7, 0; OK

Snap sequentially nodes

N4, N9, N6 and N5 to create the grids below



(8). sort the Mesh under **MODIFY** => Clear duplicate nodes => Sort nodes (v)/element(h)

(9). Under **File** =>SAVE Model Data as "*USF_2.sta*"; so it can be reloaded if needed.

2. Make zone boundaries forming the Clay Core & SORT!

(1). under **SETTING**, load "*USF_upper_dam.log*"

(2). under **TOOLS** => draw a line.. ; snap nodes N7 and N11 to create a line;

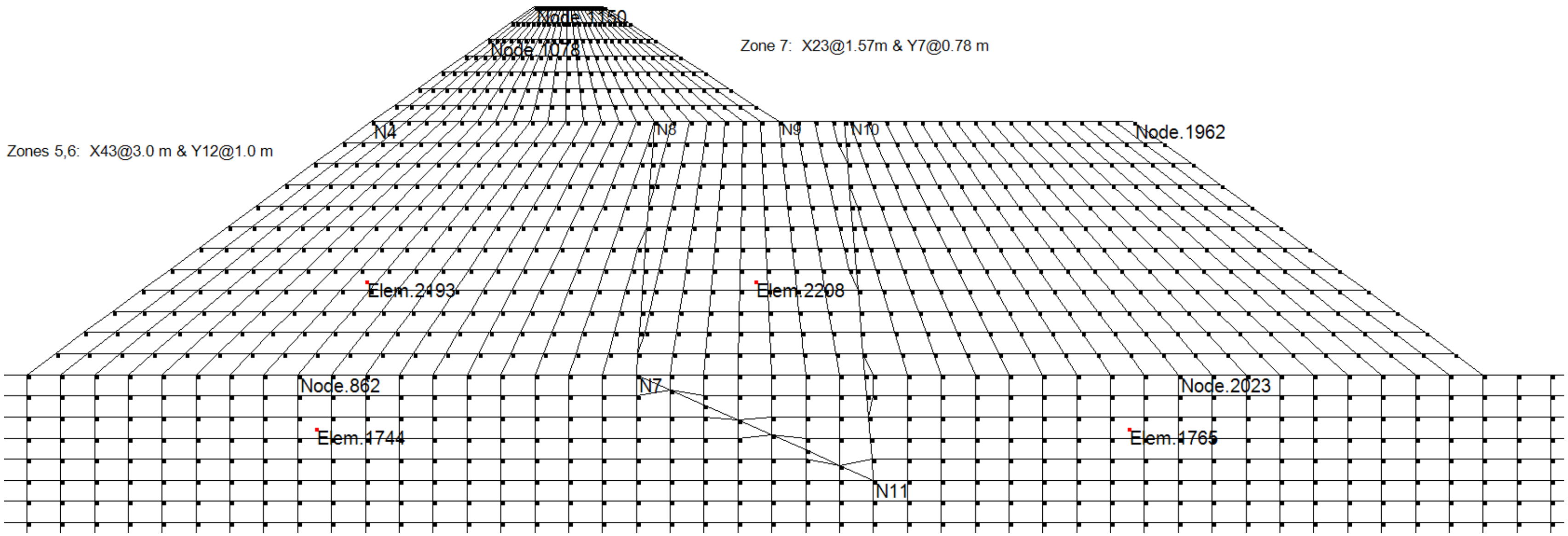
(3). again, draw a line from nodes N7 to N8;

(4). again, draw a line from nodes N10 to N11;

(5). sort the Mesh under **MODIFY** => Clear duplicate nodes => Sort nodes (v)/element(h);

Total 2835 nodes, 2704 elements ;

mesh creation completed!



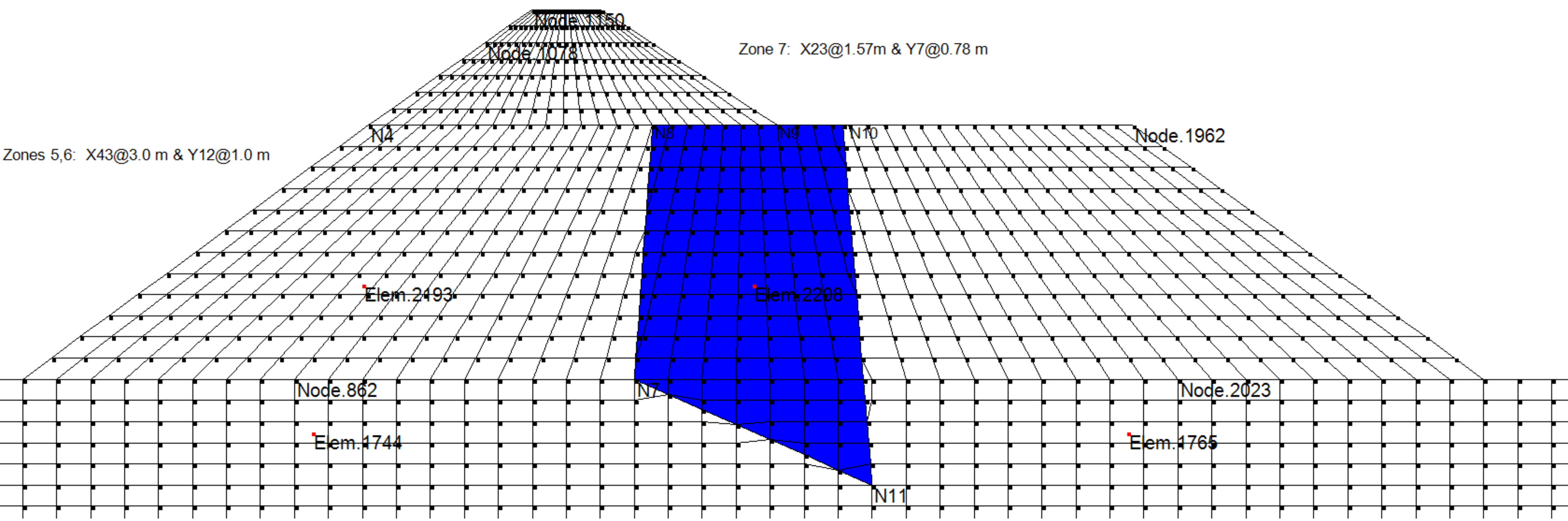
(6). under **File**, SAVE model data to "*USF_3_mesh.sta*" for future use (if required).

3. Assign soil unit no. for all zones; Define soil parameters, Adjust D/S layer thickness, Set RUNs (layers, water tables, etc.), boundary, water loads (in RUN4)

(1). under **TOOLS** => Assign soil zones; snap nodes N7, N11, N10, N8; type 3 (Clay Core) for the input box; refresh view to see blue zone (shown below)

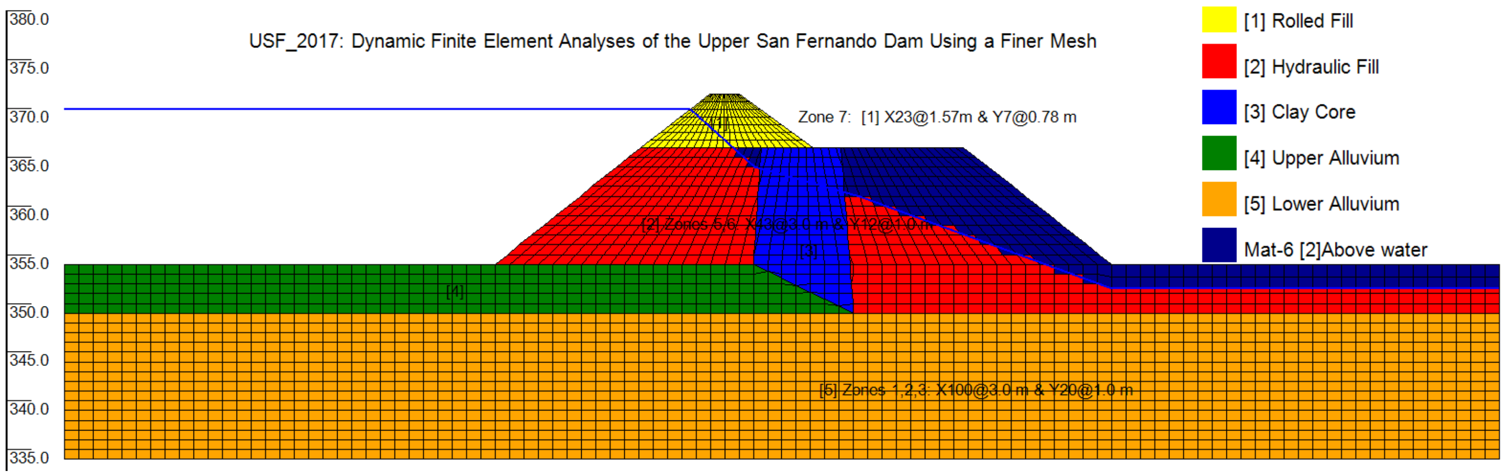
(2). do the same for all other soil zones

(3). under **File**, SAVE model data to "*USF_4_temp.sta*" for recovery, if required.



3. Assign soil unit no. for all zones; Define soil parameters, Adjust D/S layer thickness, Set RUNs (layers, water tables, etc.), boundary, water loads (in RUN4)

(4). do the same for all other soil zones (note: Use No. 6 for hydraulic fill above water) under **SETTING** => load "*USF_Model.log*" and refresh view (with *Show Material color* on)



(5). Under **DEFINE** => Input material parameters, as per table below [APPLY ALL]

(use Kg/3 and Kb/3 for static analysis; use the same parameters for No. 2 and No. 6)

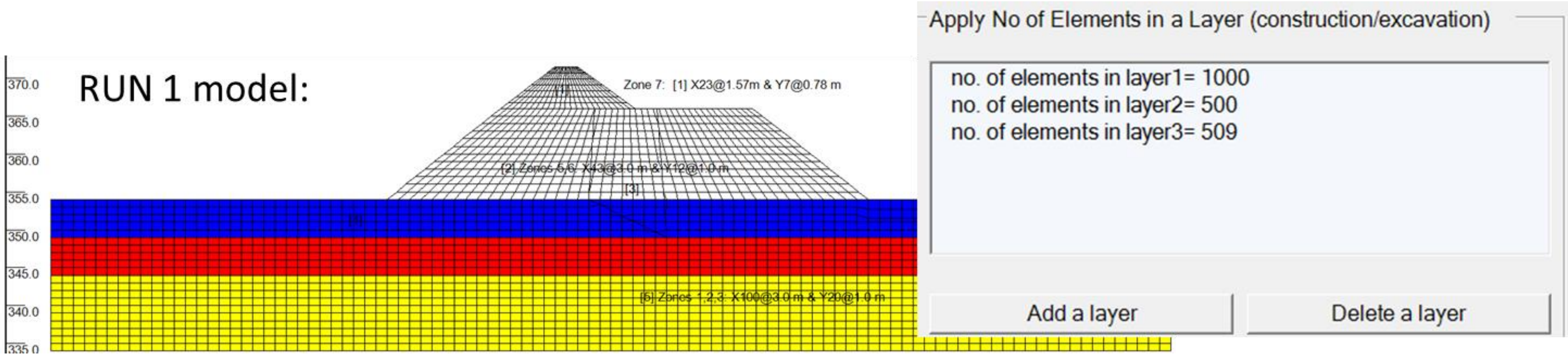
Table 1. Soil stiffness and strength parameters of the Upper San Fernando Dam (Seed et al. 1973).

Soil unit	Soil material	Unit weight (kN/m³)	Strength parameters: c' (kPa)	ϕ' (°)	Stiffness parameters*: K_{2max}	K_g	K_b
1	Rolled fill	22.0	124.5	25	52	1128	2821
2	Hydraulic fill	19.2	0	37	30	651	1630
3	Clay core	19.2	0	37	—†	651	1630
4	Upper alluvium	20.3	0	37	40	868	2170
5	Lower alluvium	20.3	0	37	110	2387	6000

*Modulus exponents ($m = n = 0.5$) were used for all soil units.

3. Assign soil unit no. for all zones; Define soil parameters, Adjust D/S layer thickness, Set RUNs (layers, water tables, etc.), boundary, water loads (in RUN4)

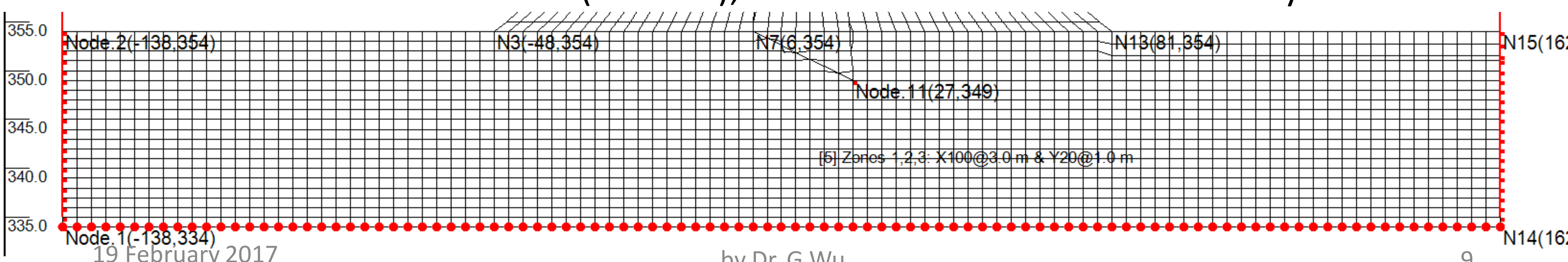
(6). RUN1: under **DEFINE =>** setup static analysis => Add a layer: 1000 elements; repeat for 500, 509; APPLY and EXIT SETUP; refresh view with "Show layers by color" ON



(7). under **DEFINE =>** setup static analysis => click "NEW RUN" to setup RUN 2, 3, and 4 (see USF_4-FINAL.sta for more details on sequence of static analysis, RUN 1, 2, 3, &4)

(8). re-load setting file: "*USF_m_1.log*"

Under **TOOLS** => Assign boundary …; snap two nodes at the base to assign "fixed"; also do two side boundaries ("free Y"); refresh view with "Show boundary.." ON



3. Assign soil unit no. for all zones; Define soil parameters, Adjust D/S layer thickness, Set RUNs (layers, water tables, etc.), boundary, water loads (in RUN4)

(9). In RUN4, apply water loads on the model surface under the reservoir

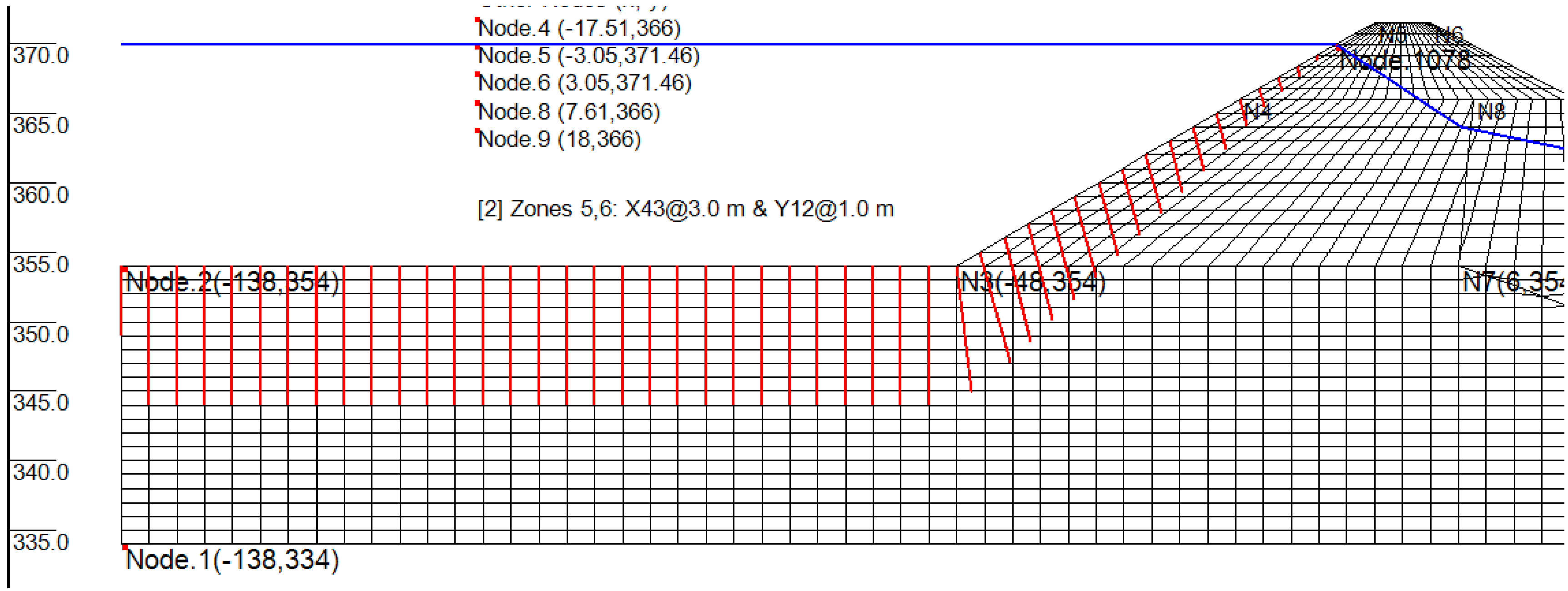
note: pressure values = 9.81 x water head from reservoir to the surface.

(a). under **TOOLS=>** Apply distributed load, snaps nodes N2, N3, and enter two pressure values of “156.0” that normal to the surface;

(b). also for N3, N4 with “156.0” and “39.1” as values;

(c). also for N4, N1078 with “39.1” and “0” as values.

Refresh view with “show load vectors” ON



(10) under **File**, SAVE model data to “*USF_4_FINAL.sta*”; ready for RUN

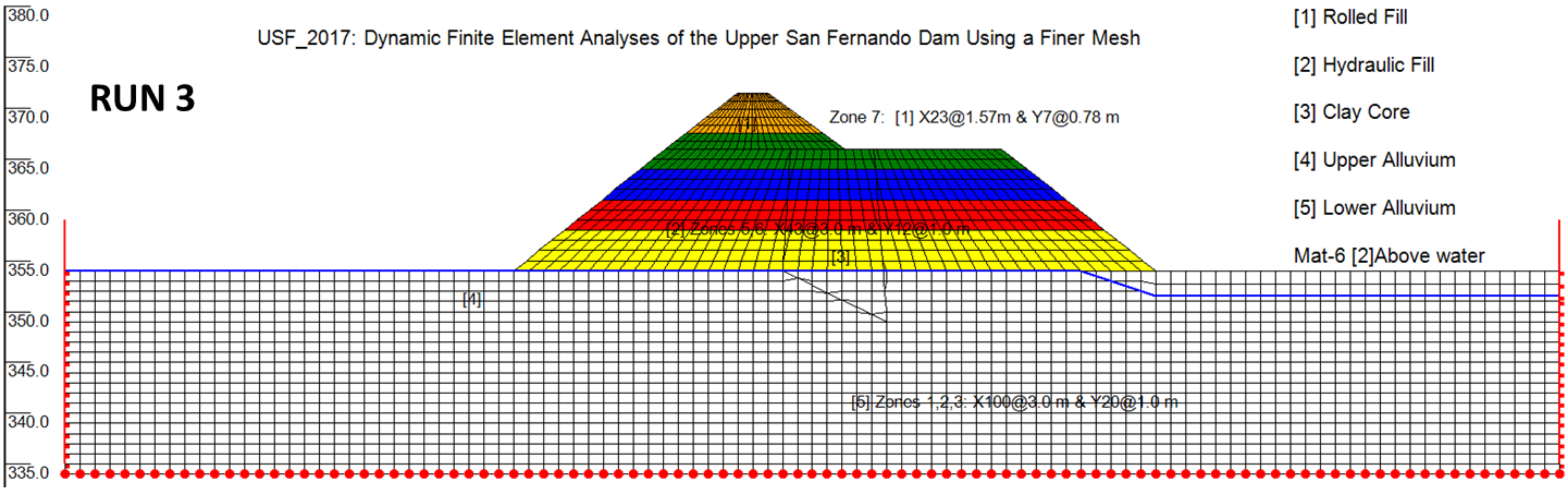
4. Final Check & RUN (run password: "gwu2015"; not required for preparing a model)

(1). Check model RUN3:

re-load setting file: "*USF_Model.log*" ;

refresh view with "Show layers by color" "Show boundary..", "Show load vectors" and "Show water leel" "Show x, y axis" ON

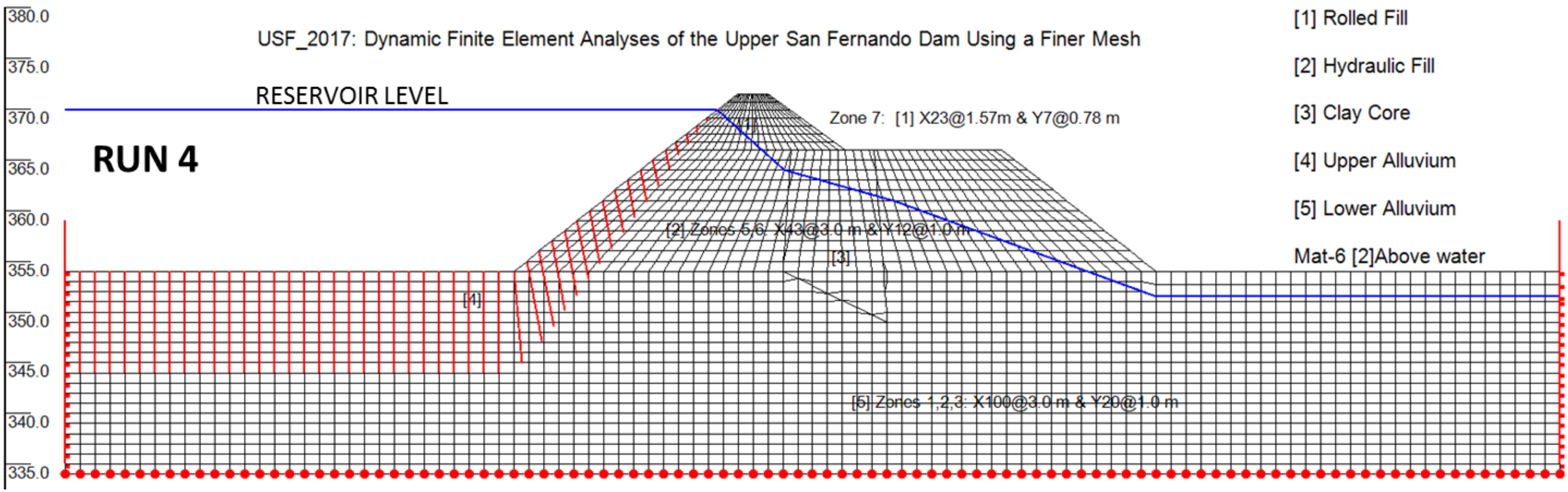
Note: RUN2 only applies the water table (blue line in the figure below) in 4 increments.



4. Final Check && RUN (run password: "gwu2015"; not required for preparing a model)

(2). Check RUN4 (apply new water table in blue and water loads, both in 6 increments)

(3). Check soil parameters under RUN1 (they will be carried forward, unless reassigned!!)

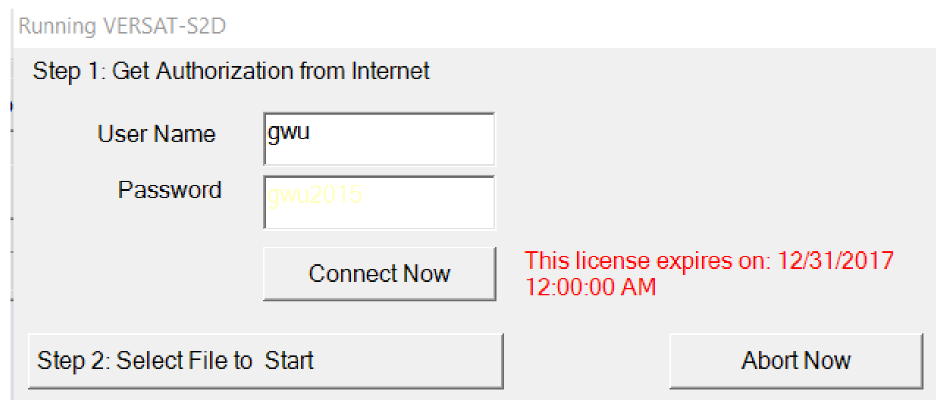


(4). Run **versat-s2d** with 4 RUNs

(a) Enter PW and Connect Now

(b). Step 2 Select file "*USF_4_FINAL*"

(5). results from static analysis ..

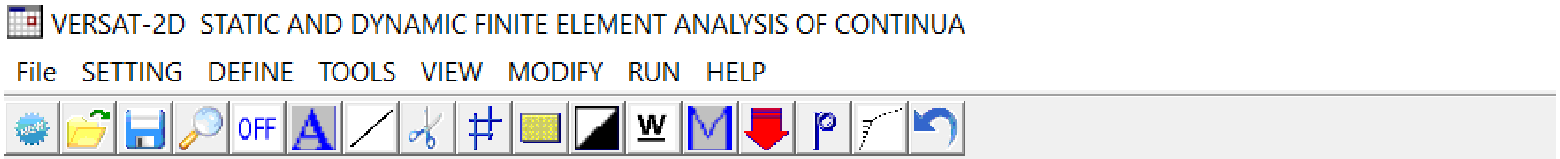


5. Setup the dynamic analysis:

5. Setup the dynamic analysis:

(1). ([download VERSAT-2D](http://www.wutecgeo.com/documents/VERSAT-2D_2016gwu.zip)from: http://www.wutecgeo.com/documents/VERSAT-2D_2016gwu.zip

(2). Start VERSAT-2D Processor by "Accept" terms:



(3). under **SETTING**, load "USF_model.log" & under **File**, load data "*USF_4_FINAL.sta*"

(4). under **SETTING** : Dynamic on; click "YES"

(5). under **DEFINE** : General parameters, click "nonlinear effective stress...", & new title

(6). under **DEFINE** : Setup dynamic analysis: (a) enter "2704" under NPRE; and (b) enter numbers for node/elem time histories (TH). – leave blank if TH not wanted. "APPLY"

(7). Under **DEFINE**: Input material parameters:

(a). update Kg, Kb with values listed on Table 4 (Slide No. 12),

(b). input PWP parameters in [2] hydraulic fill (using values for No. 2a in Table 4)

(c). APPLY ALL, reload this window and check!

Table 4. Pore-water pressure parameters and residual strengths used in Seed et al. (1976) pore-water pressure model.

Material No.	Soil description	Equivalent $(N_1)_{60}$	CRR	α	θ	Residual strength (kPa)*	K_cLIQ
2a	Upstream hydraulic fill	14	0.154	3.0	0.1	23.0 (480)	400
2b	Downstream hydraulic fill	14	0.154	3.0	0.1	23.0 (480)	400
2c	Hydraulic fill in the downstream free field	14	0.154	3.0	0.1	14.4 (300)	400

* Pounds per square feet in parentheses.

5. Setup the dynamic analysis:

(7). (b). If Wu(2001) PWP is used, then use the values in Table 3 below:

Table 3. Pore-water pressure parameters and residual strengths used in the modified MFS model.

Material No.	Soil description	C_1	C_2	M	Residual strength (kPa)*	K_cLIQ	Equivalent $(N_1)_{60}$
2a	Upstream hydraulic fill	0.32	1.25	320	23.0 (480)	400	14
2b	Downstream hydraulic fill	0.32	1.25	320	23.0 (480)	400	14
2c	Hydraulic fill in the downstream free field	0.32	1.25	320	14.4 (300)	400	14

* Pounds per square feet in parentheses.

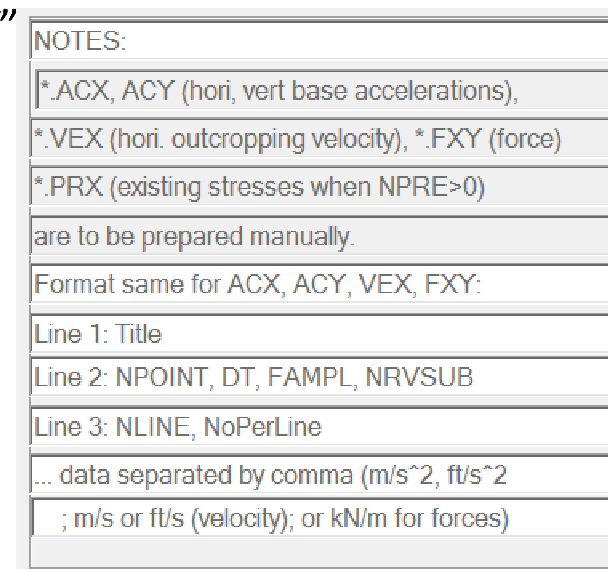
(8). Get ready to run (create a new folder "Dynamic" for dynamic analysis):

(a). under **File**, SAVE model data as "*USF_Seed.DYN*" – input master file to run in (9).

(b). copy "*USF_4_FINAL.pr4*" to "*USF_4_FINAL.PRX*" – copy results from static run

(c). manually prepare (using notepad) "*USF_4_FINAL.ACX*"

(this is already done for you !!)



(9). Run **versat-d2d** (run password: "gwu2015")