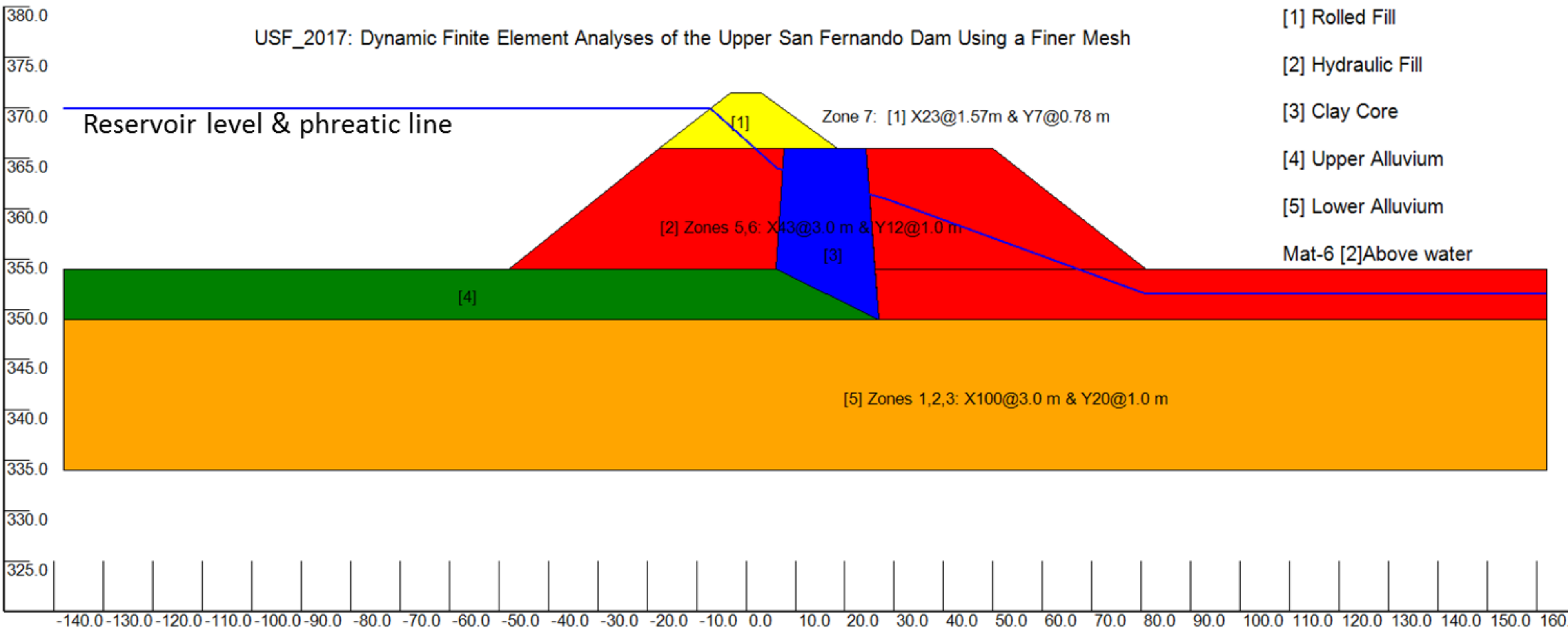
STEP-BY-STEP PROCEDURES FOR:

- making the finite element model of the **Upper San Fernando Dam**

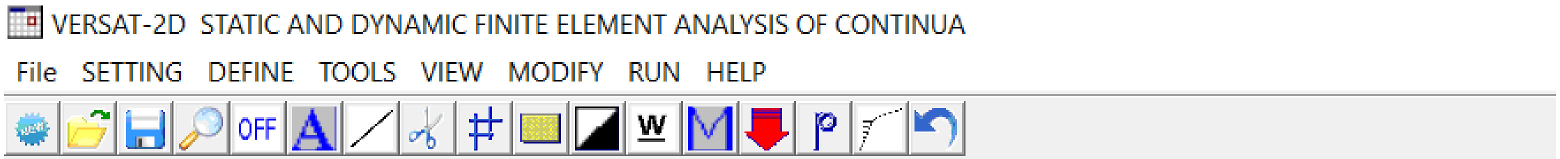
Step 0: Sketch on paper for five soil units



1. Make finite elements for 3 areas & SORT:

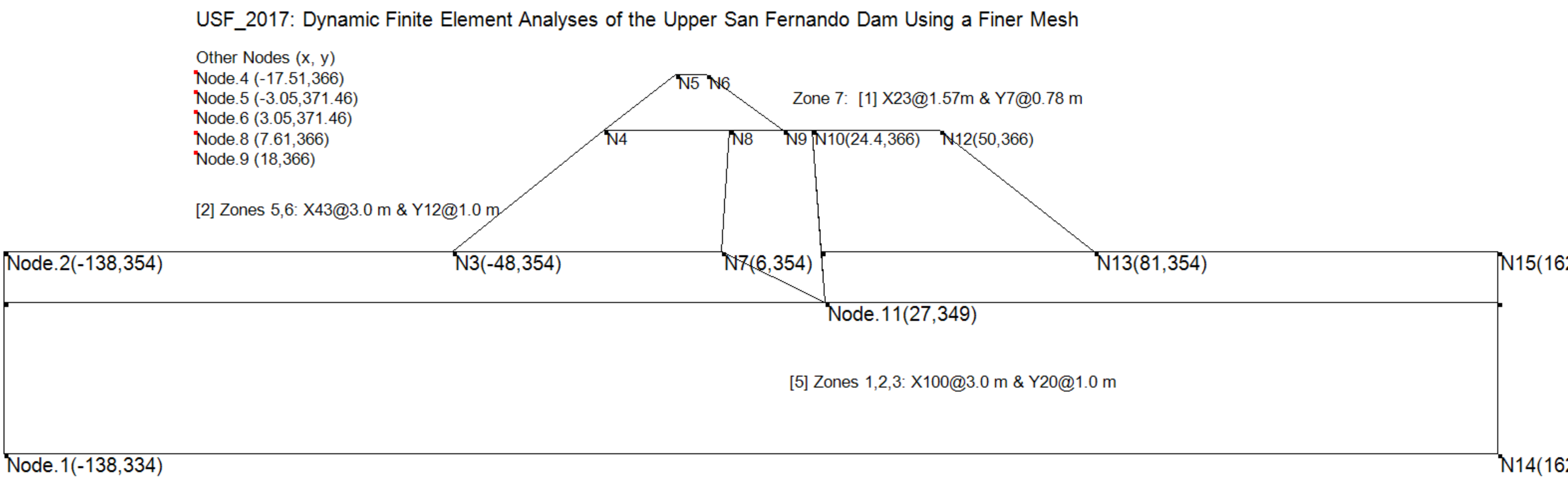
[download versat-2d_2024.03MINI_ak-2.4M](#)
(run password="**gwu**")

(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:

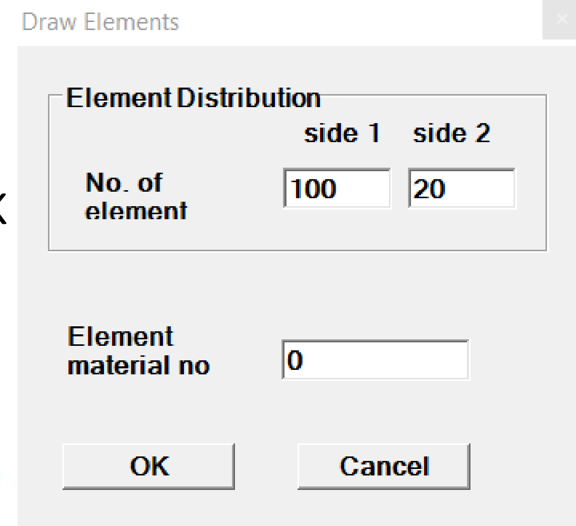
(4). Under **File**, load model data "*USF_1_nodes.sta*"

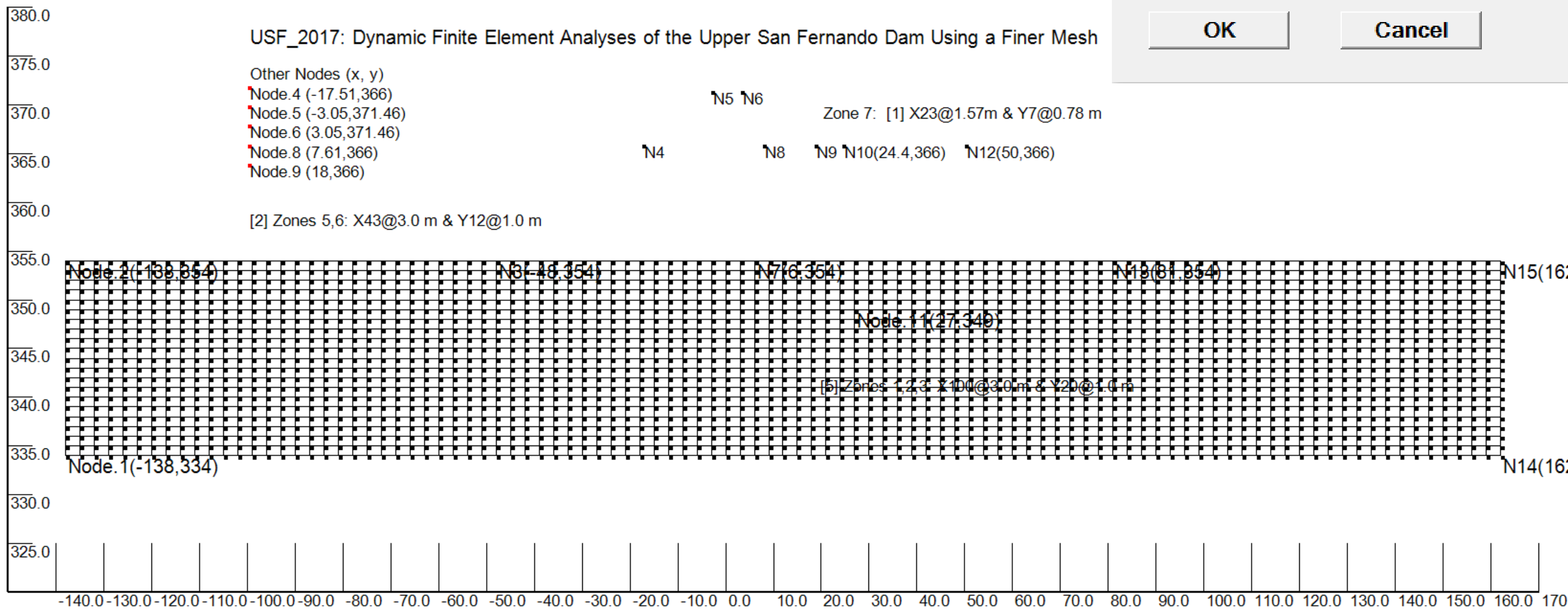
(5). Make Area 1: 100 x 20 by:

(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





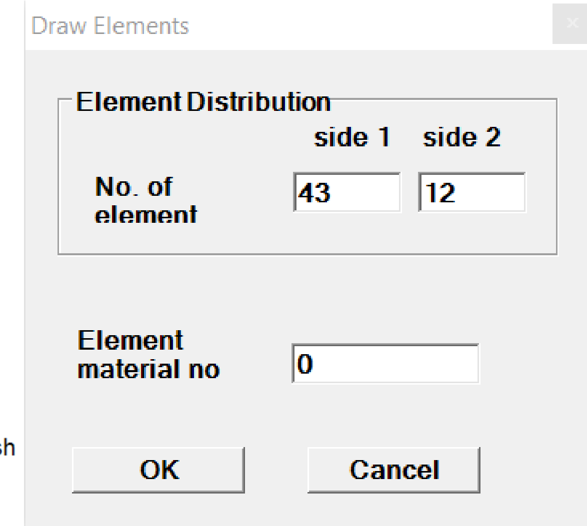
1. Make finite elements for 3 areas & SORT:

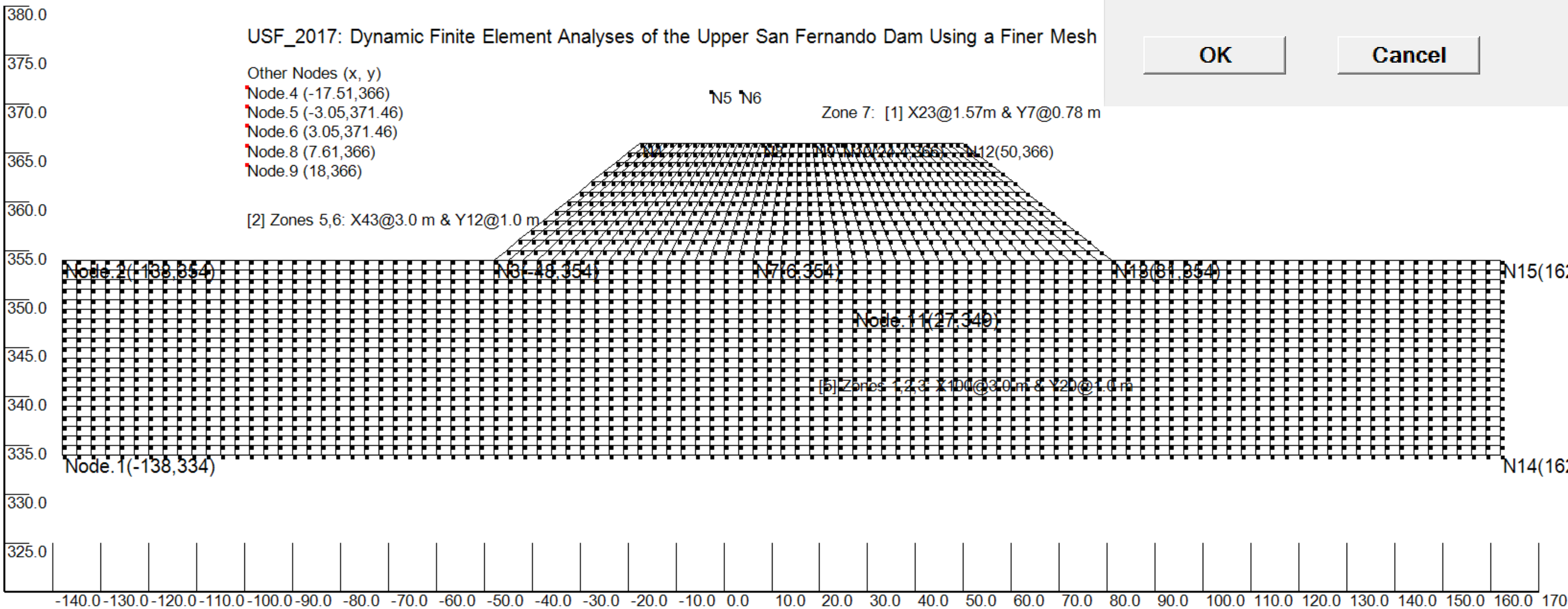
(6). Make Area 2: 43 x 12 by:

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

Snap sequentially nodes:

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





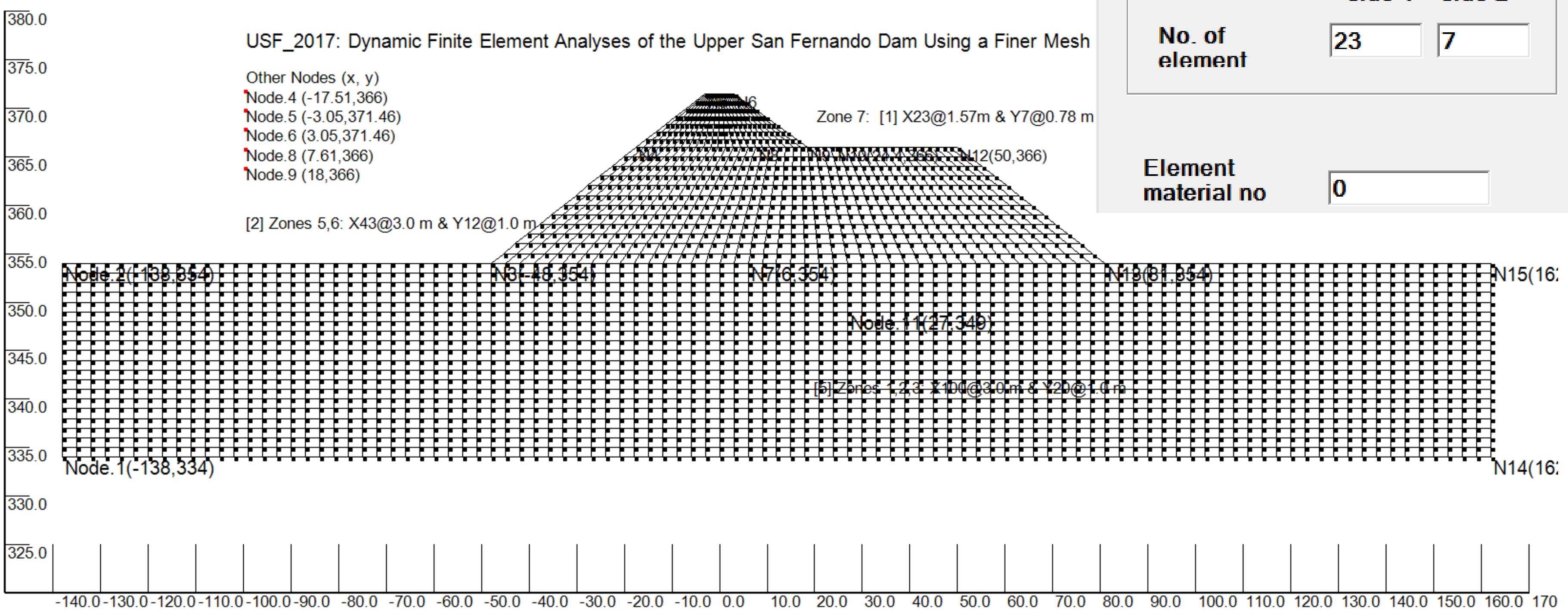
1. Make finite elements for 3 areas & SORT

(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



2818 nodes 2677 element(s) position x=-160.000 y=381.532 STATIC ON USF_1: Sketch a Plot for Five Soil Units

(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;

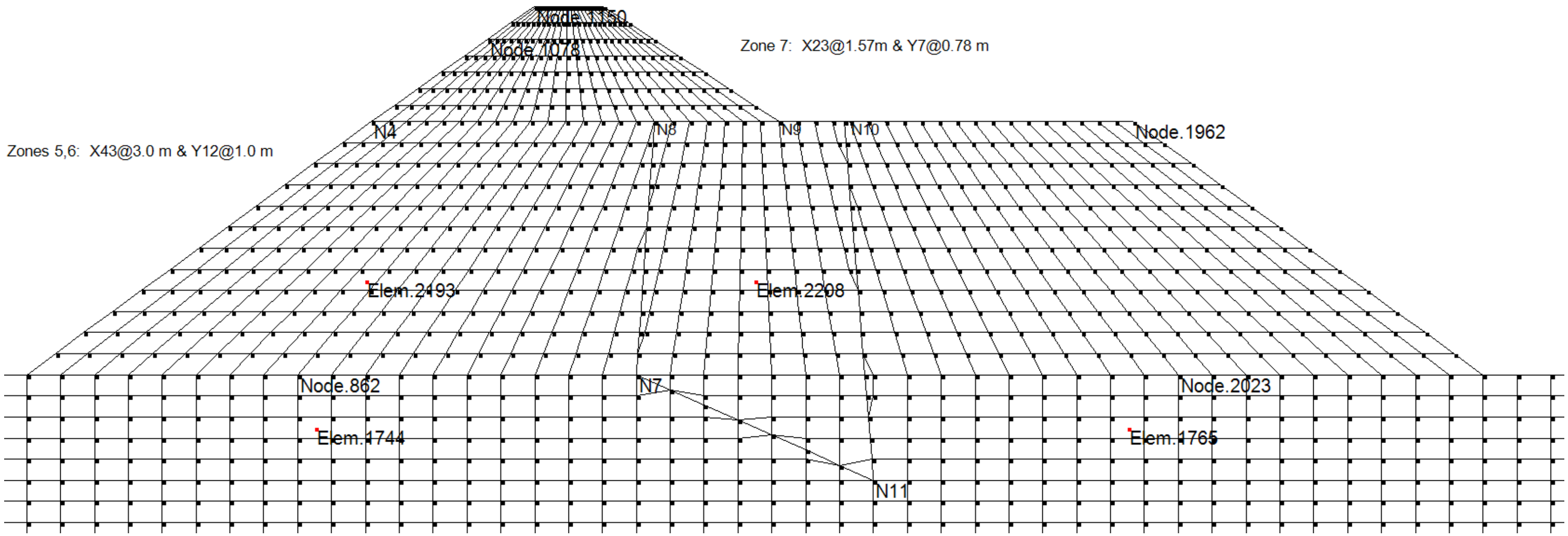
(3). again, draw a line from nodes N7 to N8;

(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!



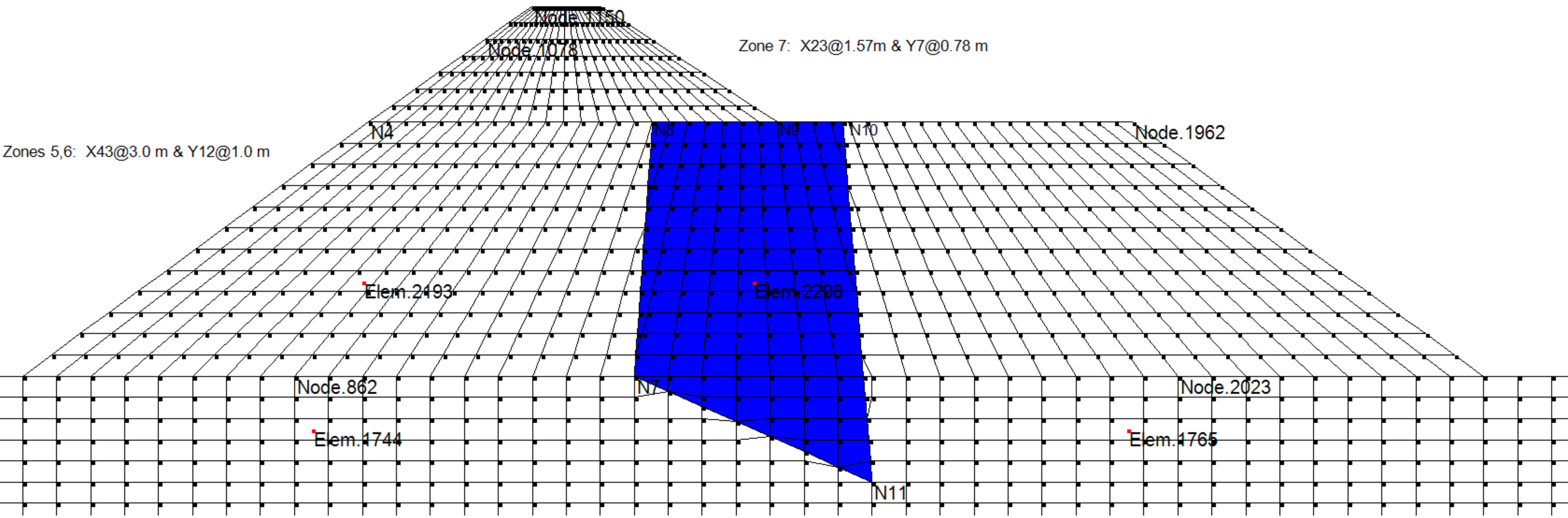
(6). under **File**, SAVE model data to "*USF_3_mesh.sta*" for future use (if required).

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)

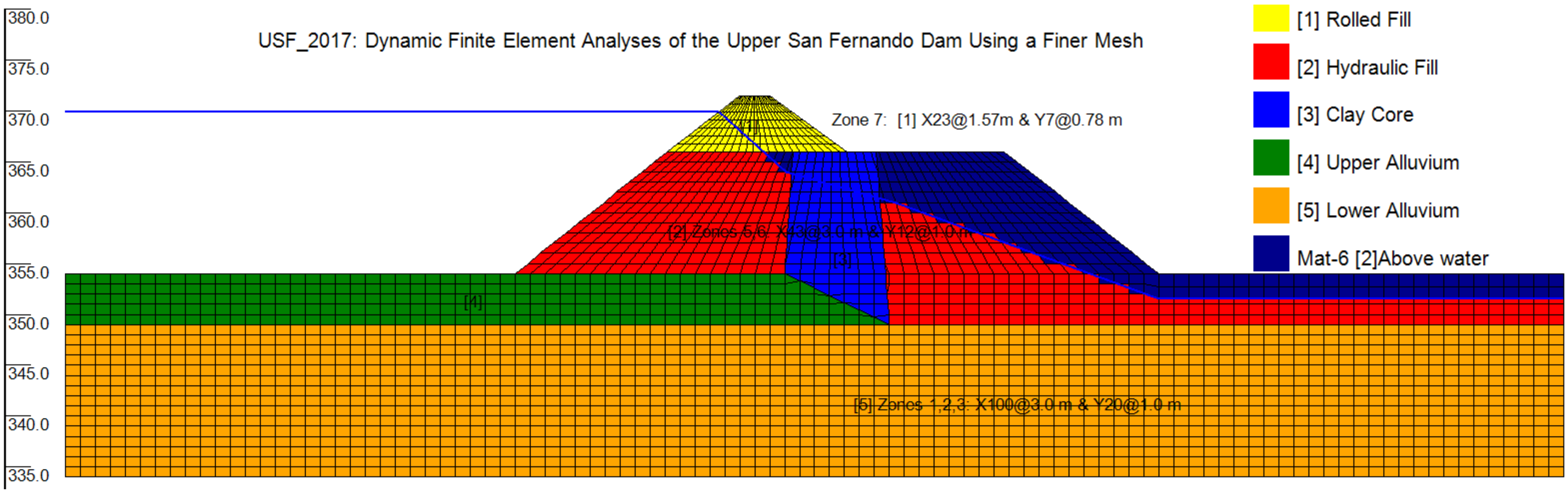
(2). do the same for all other soil zones

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



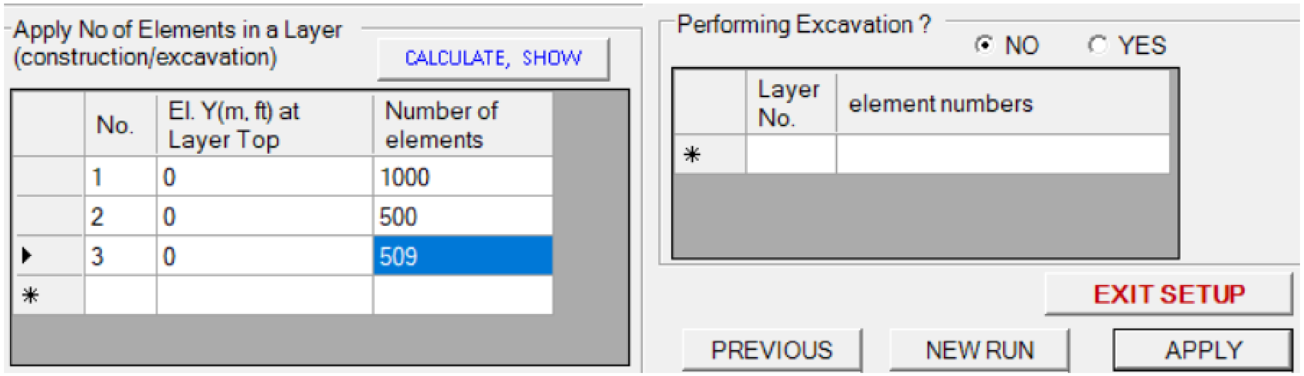
3. Assign soil unit no. for all zones; Define soil parameters, Adjust D/S layer thickness, Set RUNs (layers, water tables, etc.), boundary, water loads (in RUN4)

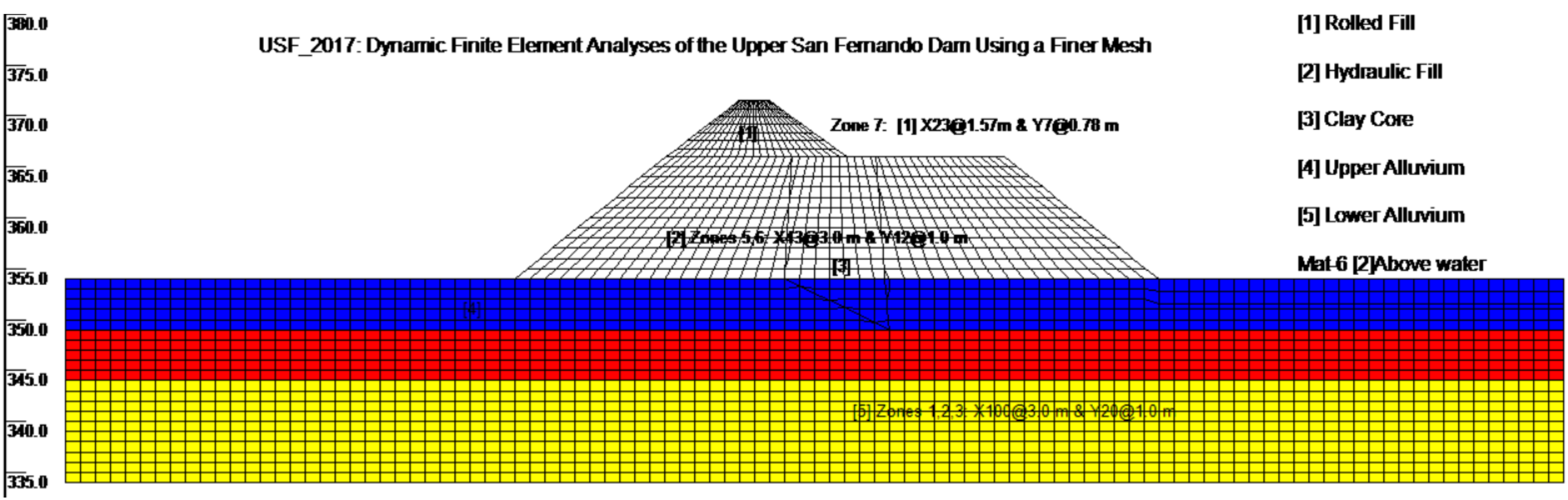
(4). do the same for all other soil zones (note: Use No. 6 for hydraulic fill above water) under **SETTING** => load "*USF_Model.log*" and refresh view (with *Show Material color* on)



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)

(5). 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





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)(5).

(6) 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).

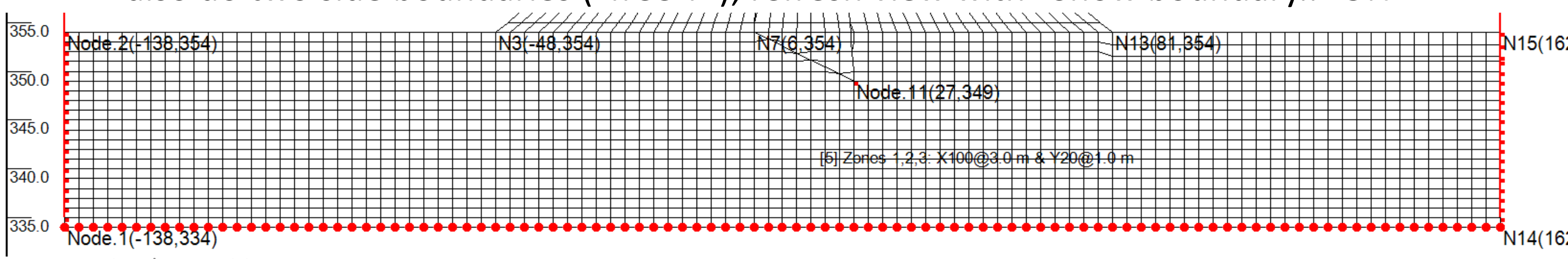
Soil unit	Soil material	Unit weight (kN/m^3)	Strength parameters: c' (kPa)	Strength parameters: ϕ' (°)	Stiffness parameters*: K_{2max}	Stiffness parameters*: K_g	Stiffness parameters*: K_b
1	Rolled fill	22.0	124.5	25	52	1128	2821
2	Hydraulic fill	19.2	0	37	30	651	1630
3	Clay core	19.2	0	37	—[†]	651	1630
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.

(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

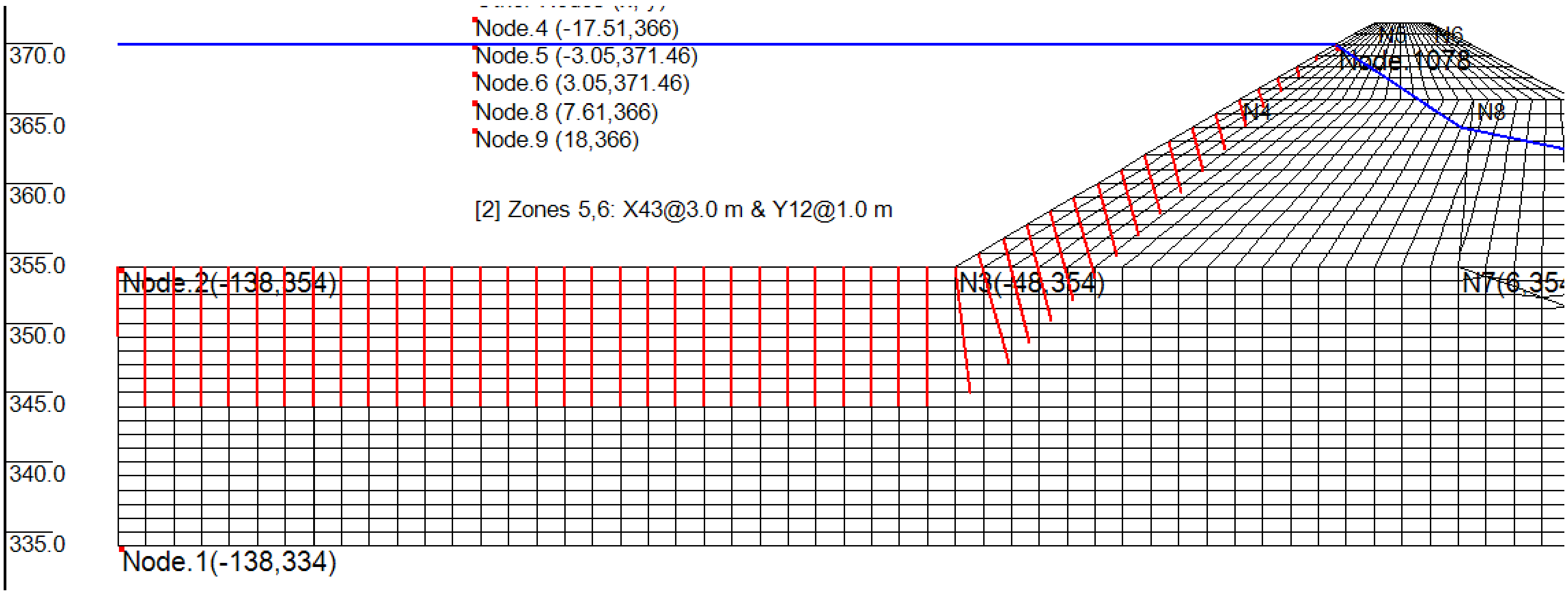
note: pressure values = 9.81 x water head from reservoir to the surface.

(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

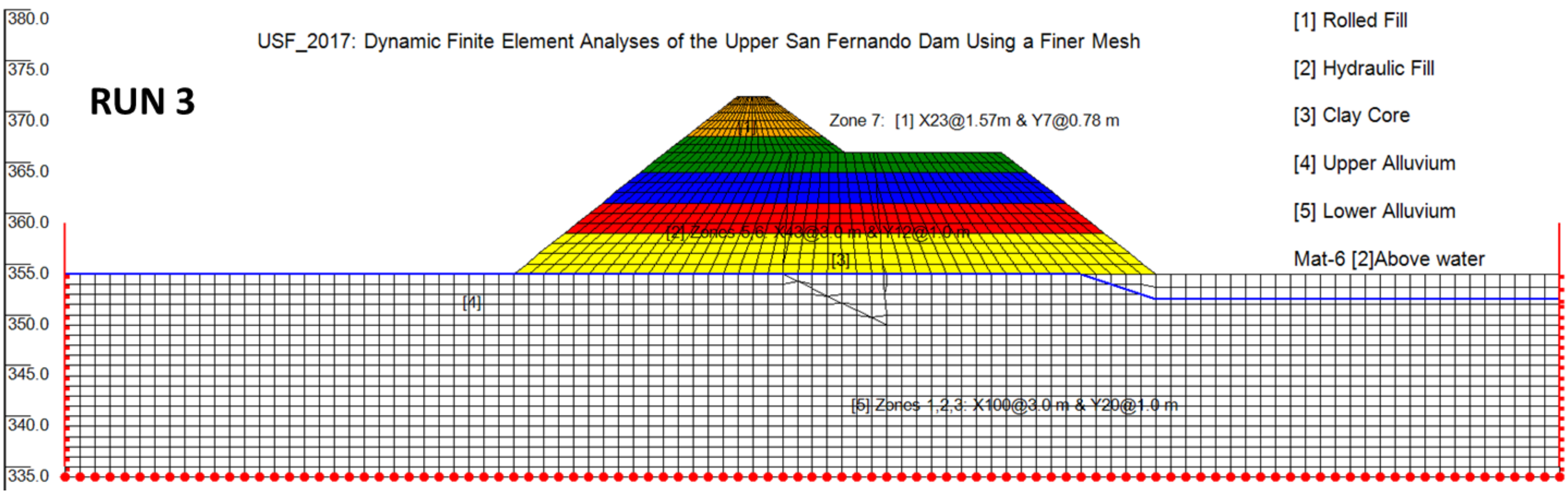
4. Final Check & RUN

(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

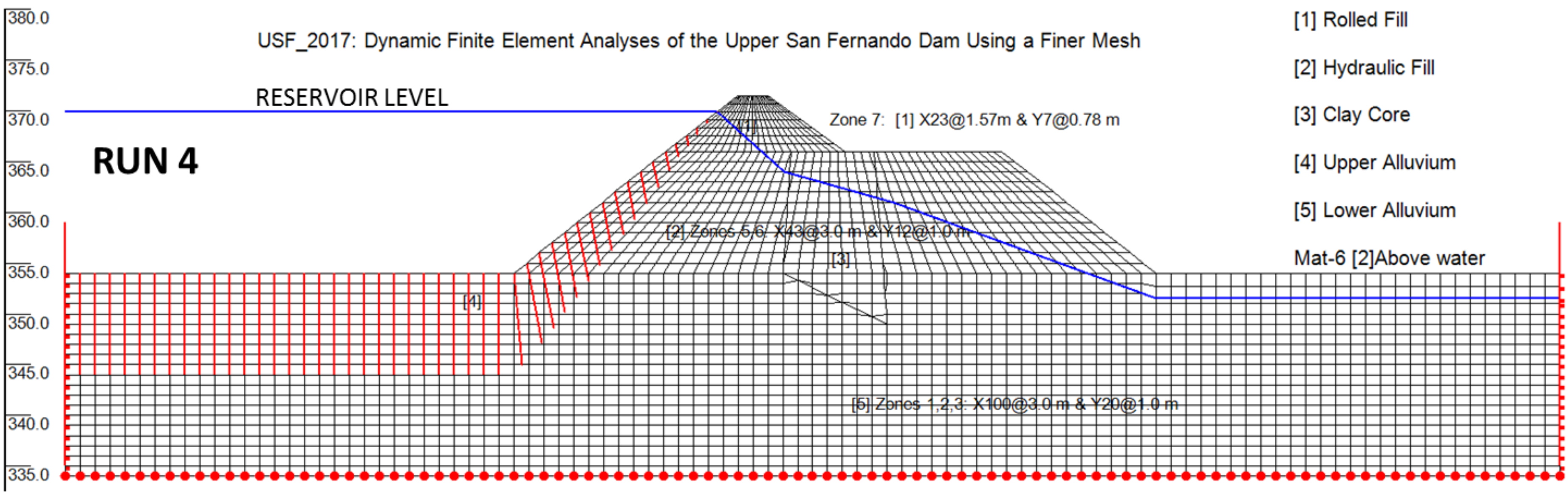
Note: RUN2 only applies the water table (blue line in the figure below) in 4 increments.



4. Final Check && RUN

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

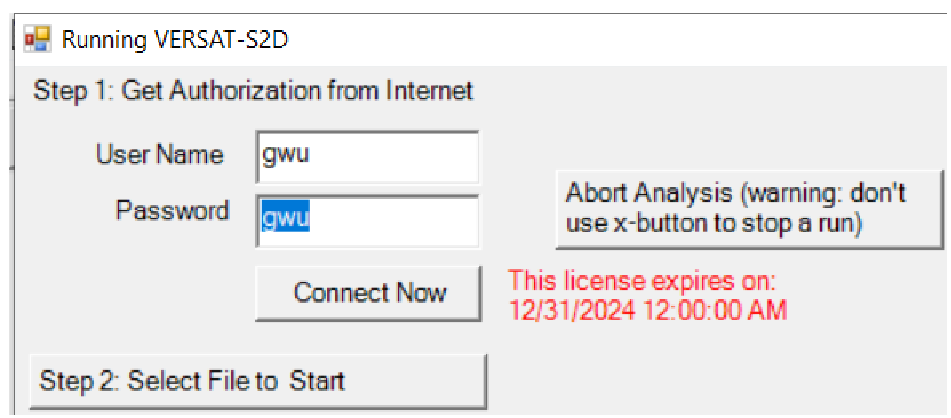
(3). Check soil parameters under RUN1 (they will be carried forward, unless reassigned!!)



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

(a) Enter PW and Connect Now

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



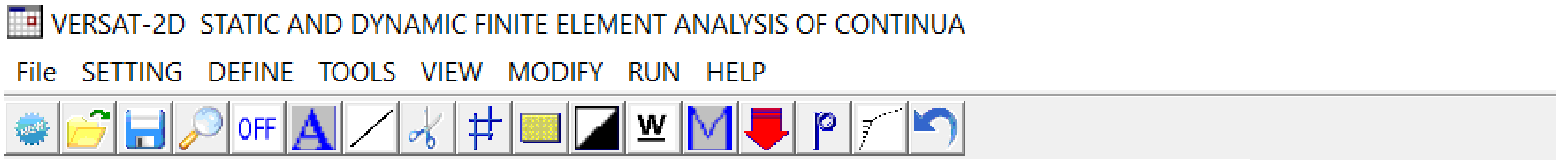
(5). results from static analysis ..

5. Setup the dynamic analysis:

5. Setup the dynamic analysis:

(1) Download VERSAT-2D

(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.

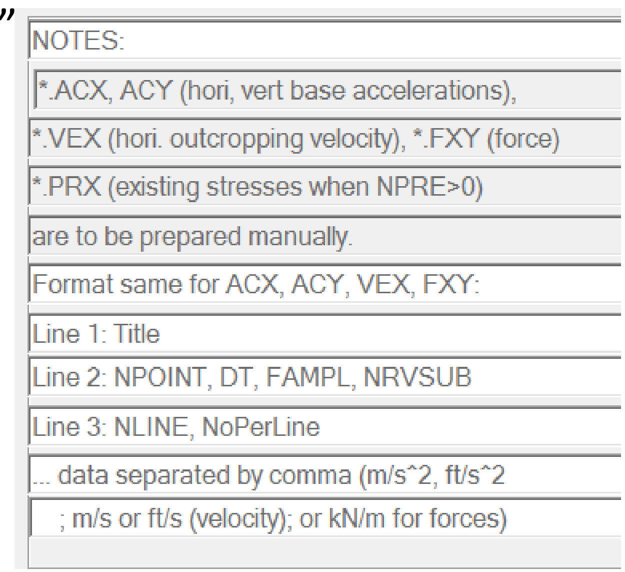
(8). Get ready to run (create a new folder "Dynamic" for dynamic analysis):

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

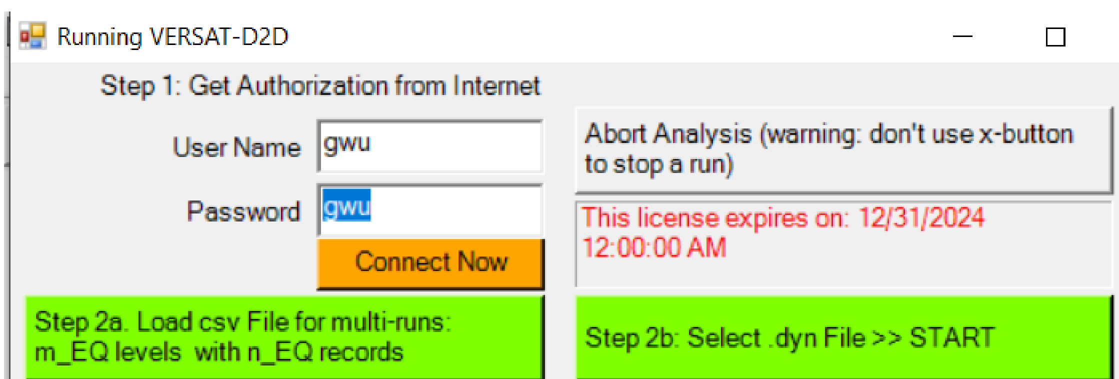
(b). copy "*USF_4_FINAL.pr4*" to "*USF_4_FINAL.PRX*" – copy results from static run

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

(this is already done for you !!)

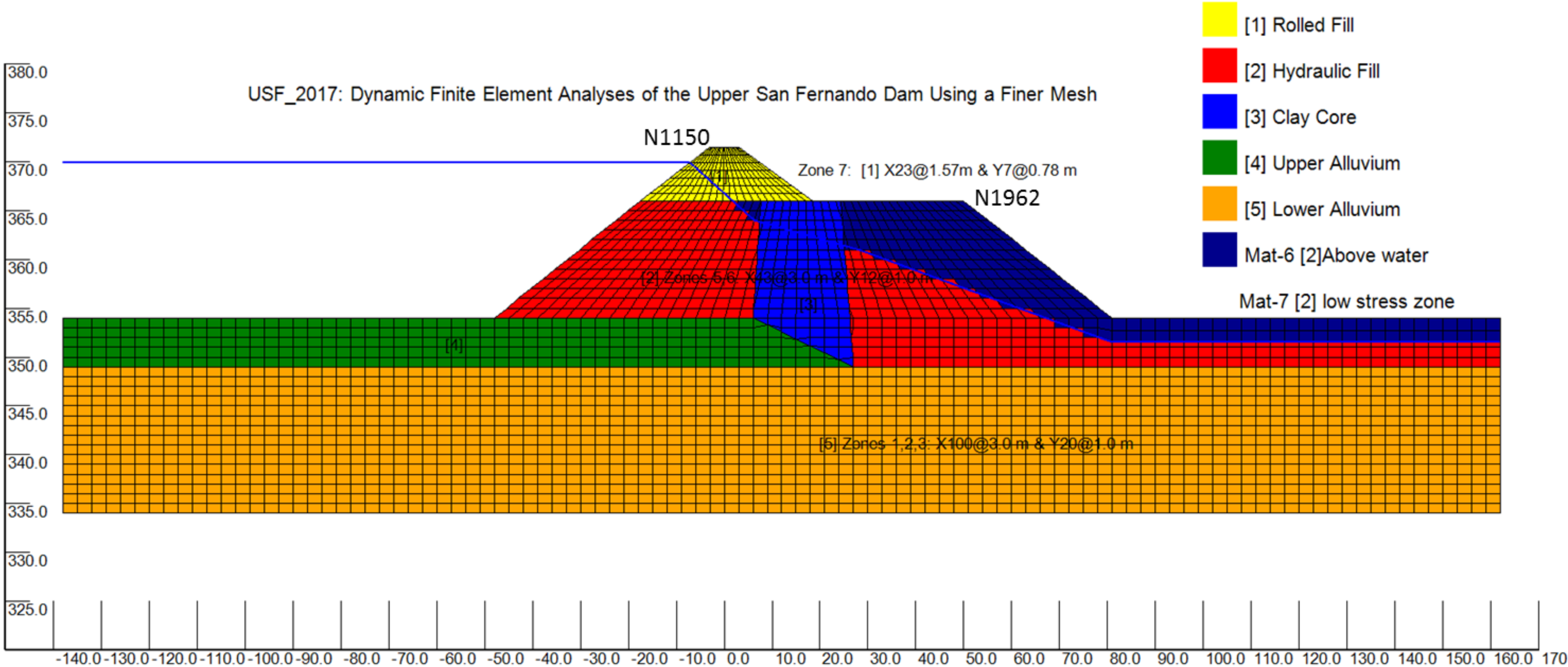


(9). Run **versat-d2d**



February 2017 Analysis Model

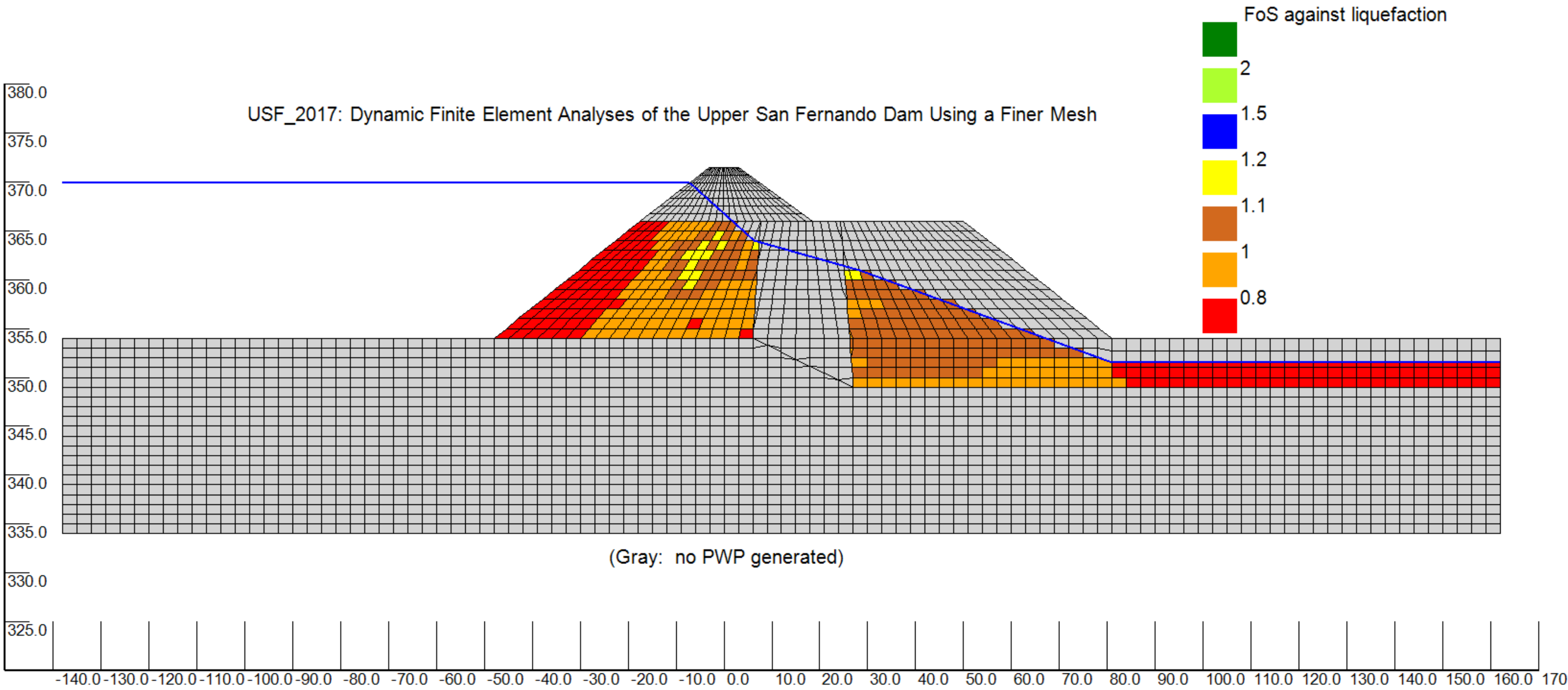
- 2835 Nodes and 2704 Elements   Note: 678 nodes and 625 elements used in 2001 model in Wu (2001)
- subjected to 1971 San Fernando EQ Pacoima Record (PGA 0.6g) (Wu 2001)



Note: Feb. 2017 Computed displacements at Node points:
N1150 (0.77 m,-0.52 m); N1962(2.72m, -0.40m) with Seed's PWP Model;
N1150 (0.42 m,-0.44 m); N1962(2.54m, -0.50m) using Wu(2001) PWP Model:

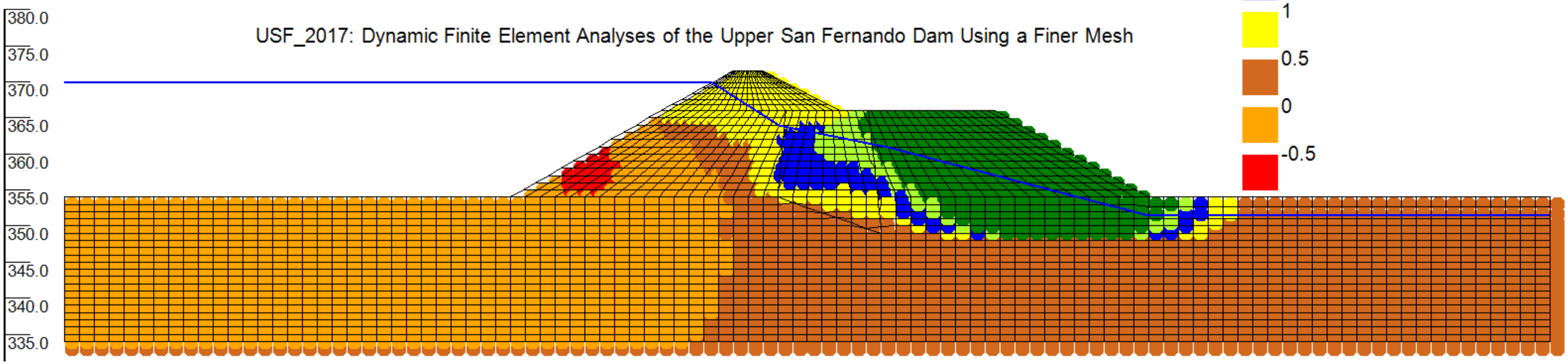
February 2017 Analysis Results:

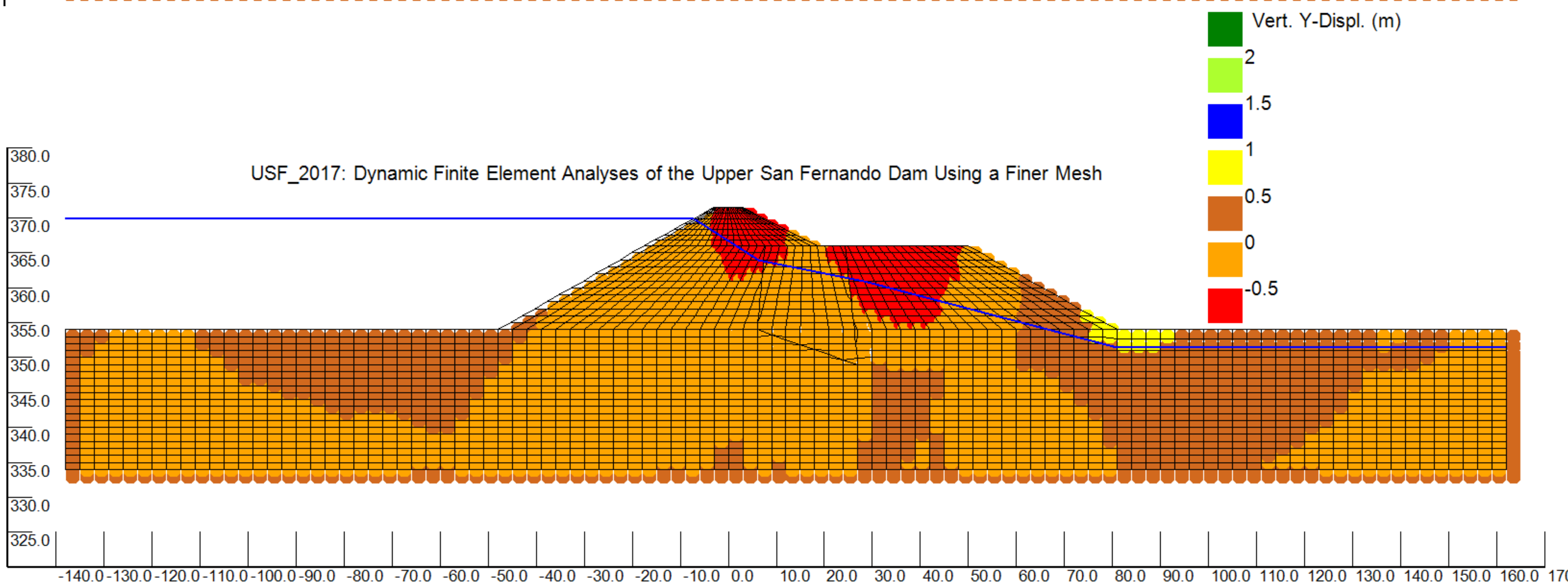
- Factor of Safety Against Liquefaction using Seed's PWP model



February 2017 Analysis Results:

Horizontal (X) and Vert (Y) ground displacements (m)





February 2017 Analysis Results:

Computed Deformed Ground (RED) on original ground (black) with Seed's PWP model