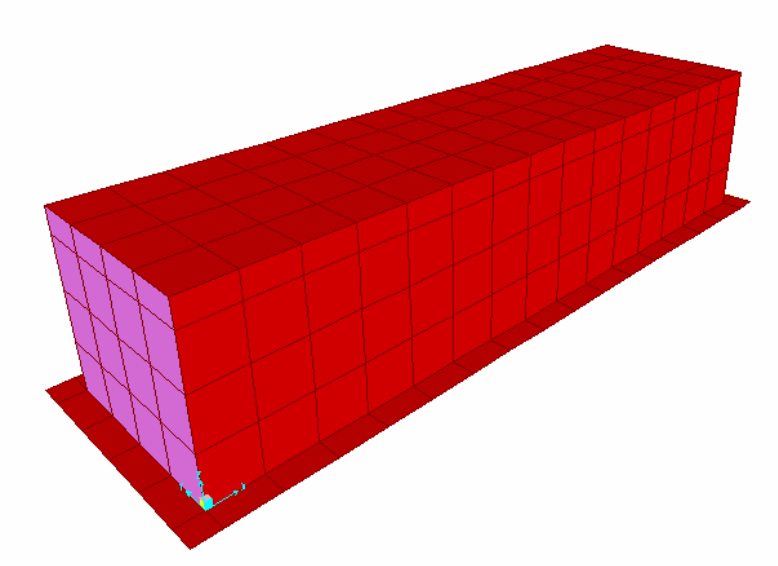
Analysis of Underground Water Tank

Using SAP2000

(Metric Units)

ACECOMS, AIT



Analysis of Underground Water Tank

Using SAP2000

Table of Content

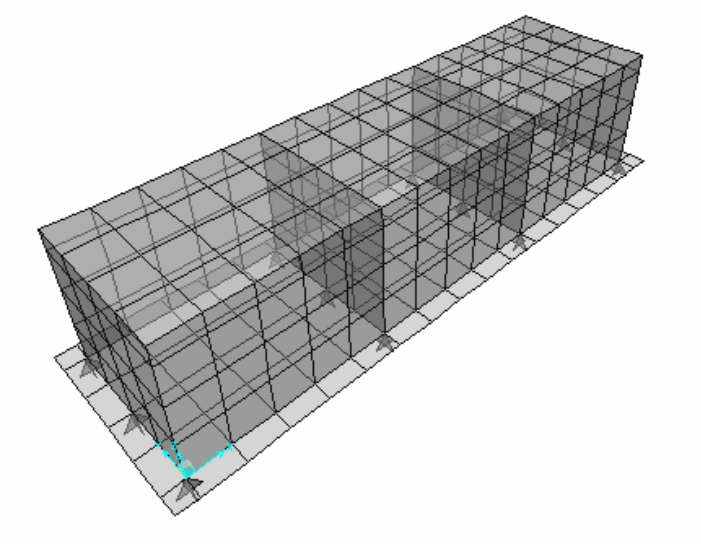
Objective

To demonstrate and practice step-by-step on the modeling and static analysis of underground water tank

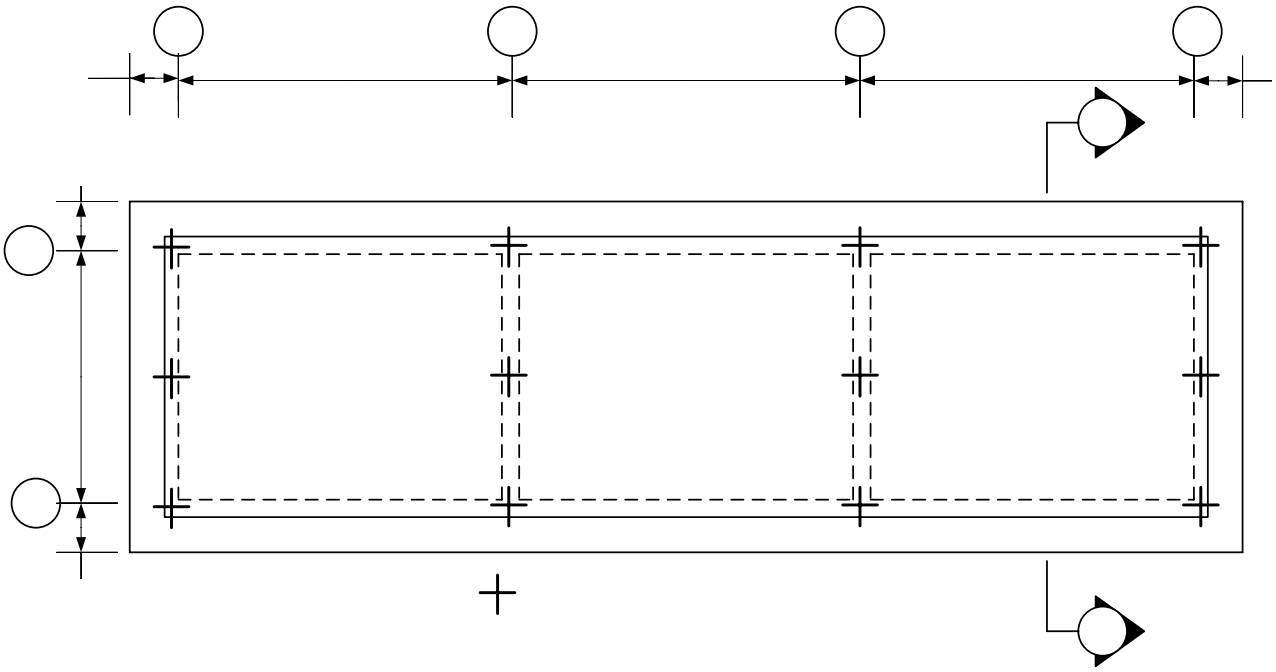
Problem

Carry out static analysis of underground water tank with the following details

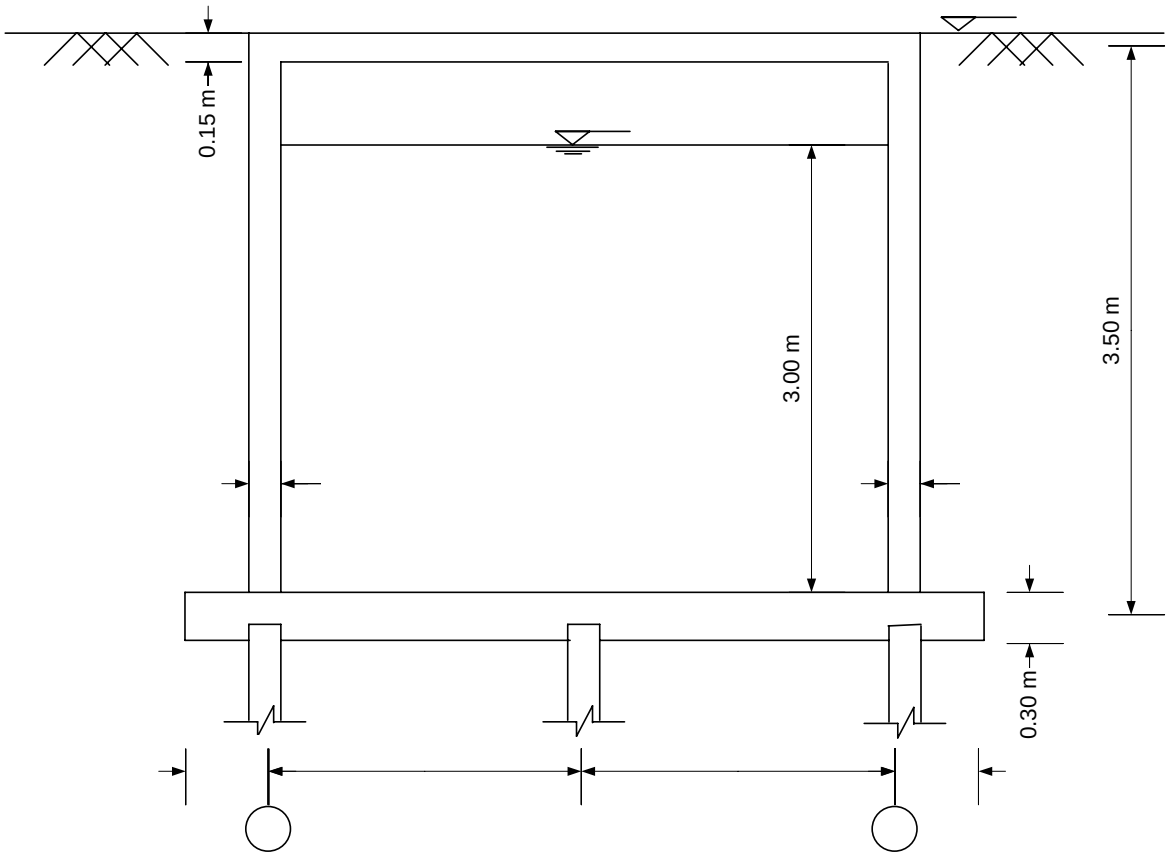
3D View



Plan View



Section View



Material Properties (Use Default Material)

Compressive strength of concrete (f'_c)	=	281	kg/cm^2
Young Modulus of concrete (E_c)	=	253,105	kg/cm^2
Unit Weight of Concrete	=	2,403	kg/m^3
Unit Weight of Water	=	1,000	kg/m^3
Unit Weight of Soil	=	1,800	kg/m^3
Coefficient of Active Earth Pressure (Ka)	=	0.6	
(Assuming the back fill is cohesive)			

Modulus of Subgrade Reaction (K_S) Properties

Direction	Value ($T/m^2/m$)
Vertical (z)	1200
Horizontal (x and y)	500

Shell Section Properties

Location	Section
Top Slabs	15 cm
Bottom Slabs	30 cm
Walls	25 cm

Load Cases and Details

Load Name	Load Type	Details
DEAD	Dead Load	Self Weight: Calculate automatically using Self Weight Multiplier in SAP2000
LIVE	Live Load	250 kg/m^2 on Top Slabs
TANK1	Weight & Pressure of Fluid	Weight and Pressure of Water on Walls and Bottom Slabs in Tank 1
TANK2	Weight & Pressure of Fluid	Weight and Pressure of Water on Walls and Bottom Slabs in Tank 2
TANK3	Weight & Pressure of Fluid	Weight and Pressure of Water on Walls and Bottom Slabs in Tank 3
SOIL	Weight & Pressure of Soil	Soil Pressure on All Side Walls and Cantilever Part of Base Slab (Not including any surcharge from service vehicles)
UPLIFT	Pressure of Water	Uplift Pressure on Bottom Slabs (3.5 T/m^2: Upward)

Load Combinations (without Sanitary Durability Coefficient)

Load Comb. No.	Details
1	U = 1.4 DEAD + 1.7 WATER
2	U = 1.2 DEAD + 1.7 WATER + 1.6 LIVE +1.7 SOIL
3	U = 1.2 DEAD + 1.7 WATER + 1.6 LIVE +1.7 SOIL + 1.7 UPLIFT
4	U = 1.2 DEAD + 1.0 LIVE
5	U = 0.9 DEAD + 1.7 SOIL + 1.7 UPLIFT

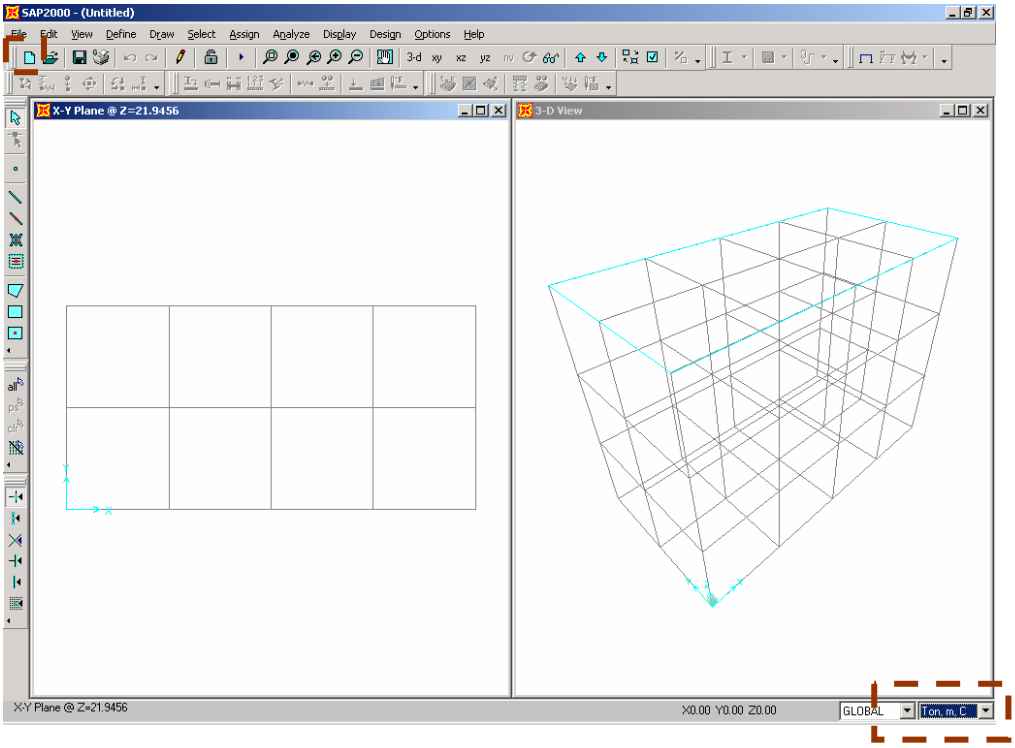
Sanitary Durability Coefficient
(Modified ACI Capacity Factors for Water Retaining Structure)

Reinforcement in Flexure	= 1.3 U
Direct Tension/Hoop Reinforcement	= 1.6 U
Excess Shear (for Stirrup)	= 1.3 U
Compression + Flexure	= 1.0 U

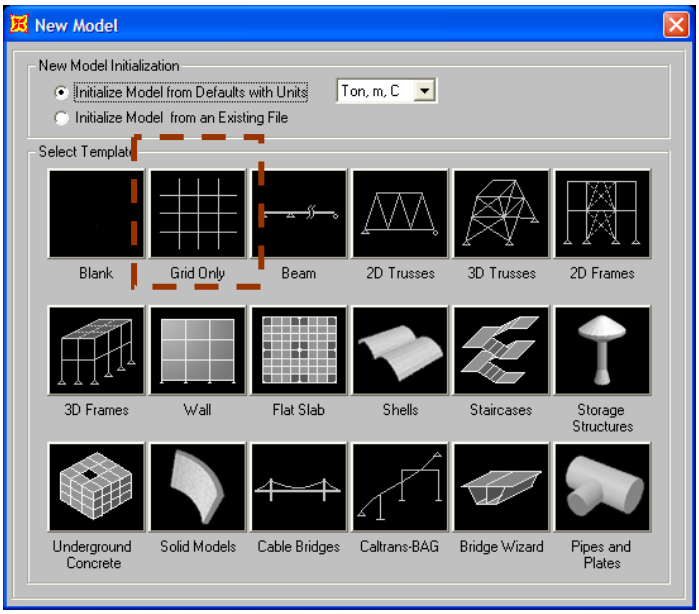
Step by Step

1. Start Model with Template

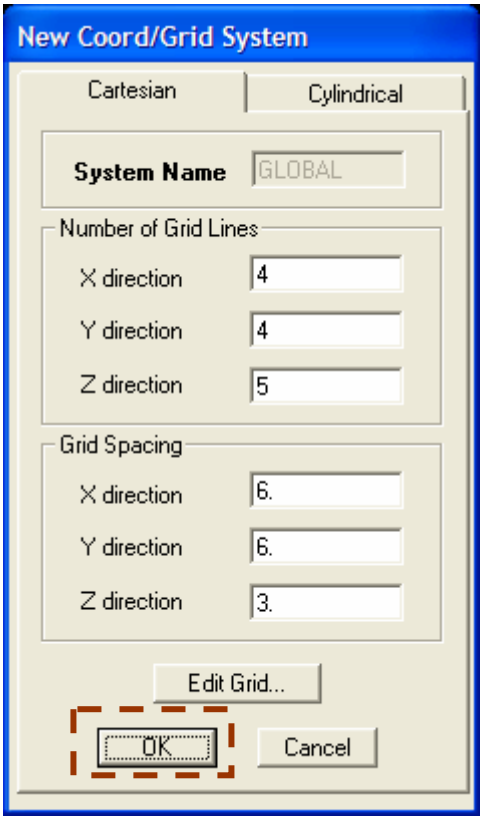
Step 1-1: Select Working Unit and Start New Model with Template
Start up screen of SAP2000, select working unit to be “ton-m” at drop-down menu on the bottom-right of screen and click on  to start new model with template



Step 1-2: Select Template
Select “Grid Only” by clicking on it



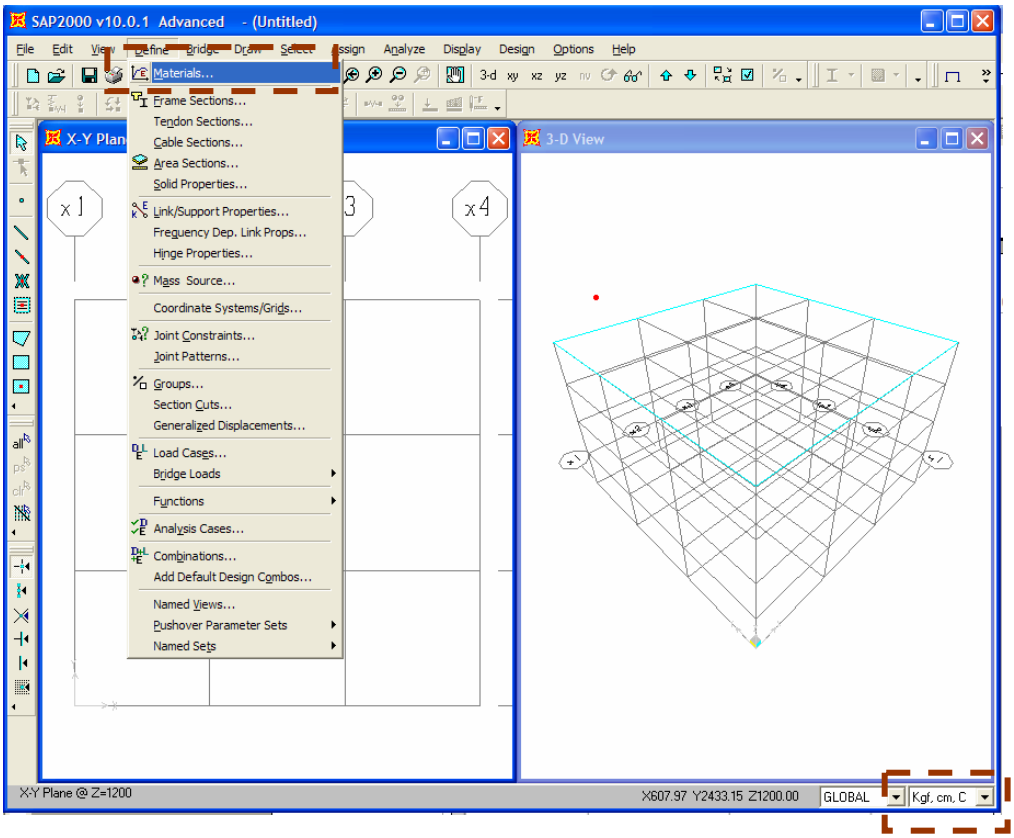
Step 1-3: Specify Grid System
Specify grid system as shown in above figure and click “OK”.



2. Define Material Properties

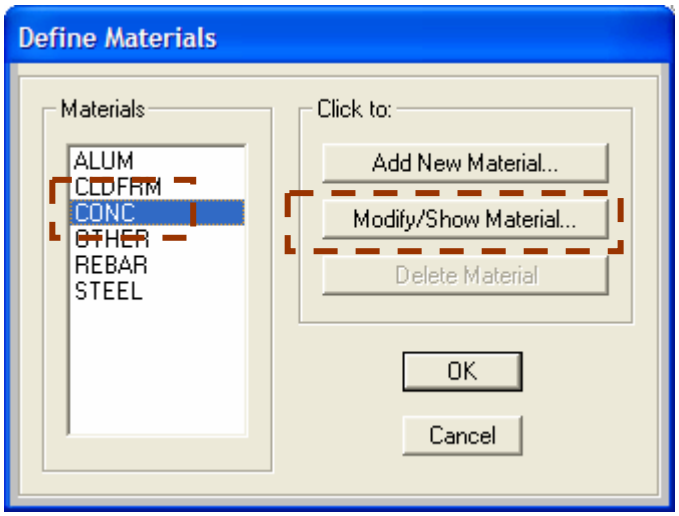
Step 2-1: Change Working Unit

Change working unit to “kg-cm”



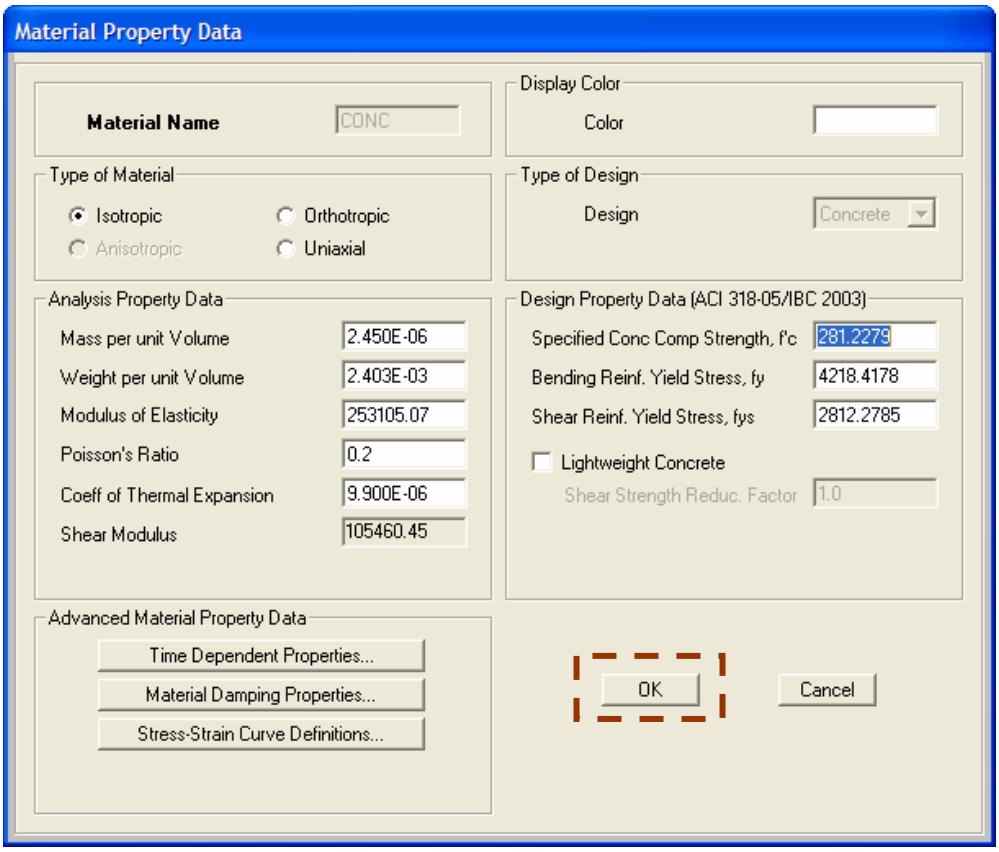
Step 2-2: Modify Material Properties

Go to **Define >> Materials...**, and select “CONC”, click on “Modify/ Show Material...”



Step 2-3: Accept the Default Values

Click “OK” to accept the default values. Next, also click “OK” in the preceding window (Define Materials).

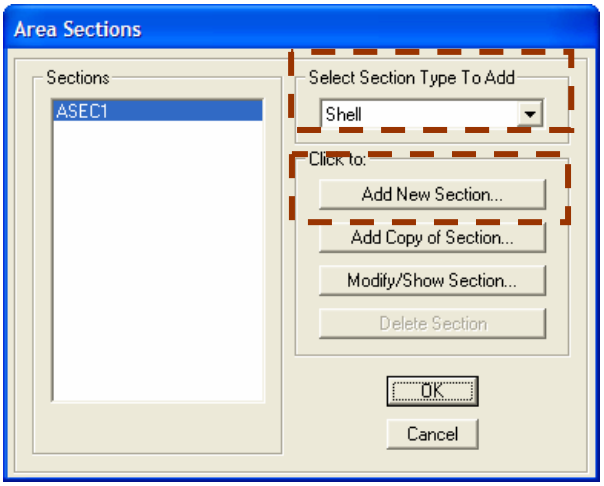


3. Define Shell Sections

Three shell sections (thick. = 15, 25 and 30 cm) are required in this model.

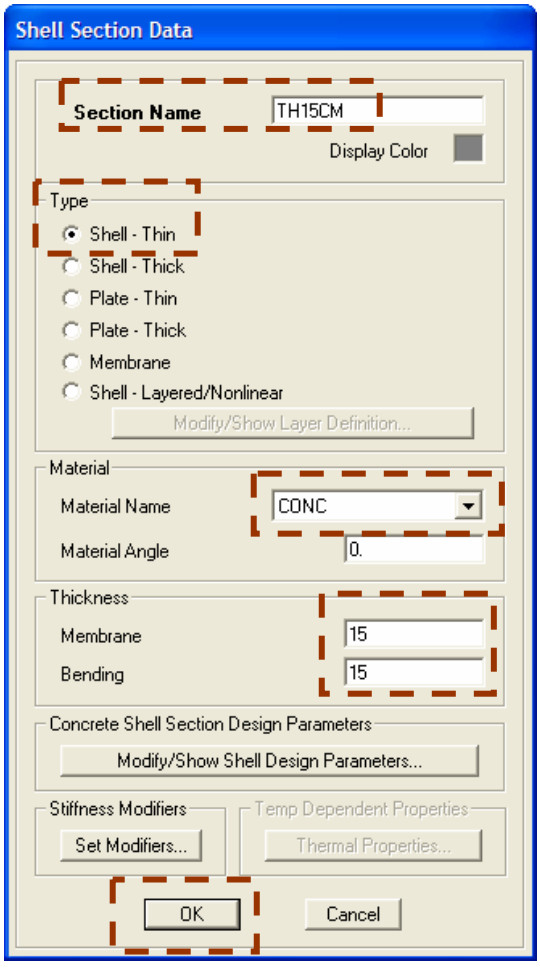
Step 3-1: Add New Shell Section

Go to **Define >> Area Section**, select "Shell" from drop-down menu and click on "Add New Section".



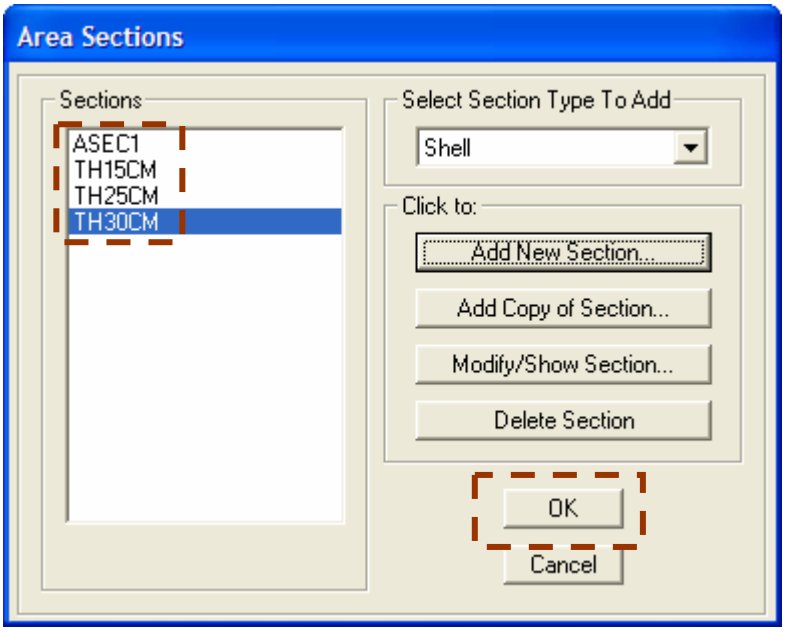
Step 3-2: Specify Shell Section Properties

Specify shell section properties as shown below for shell section thickness = 15 cm. Afterwards, click "OK".



Step 3-3: Add 2 New Shell Sections

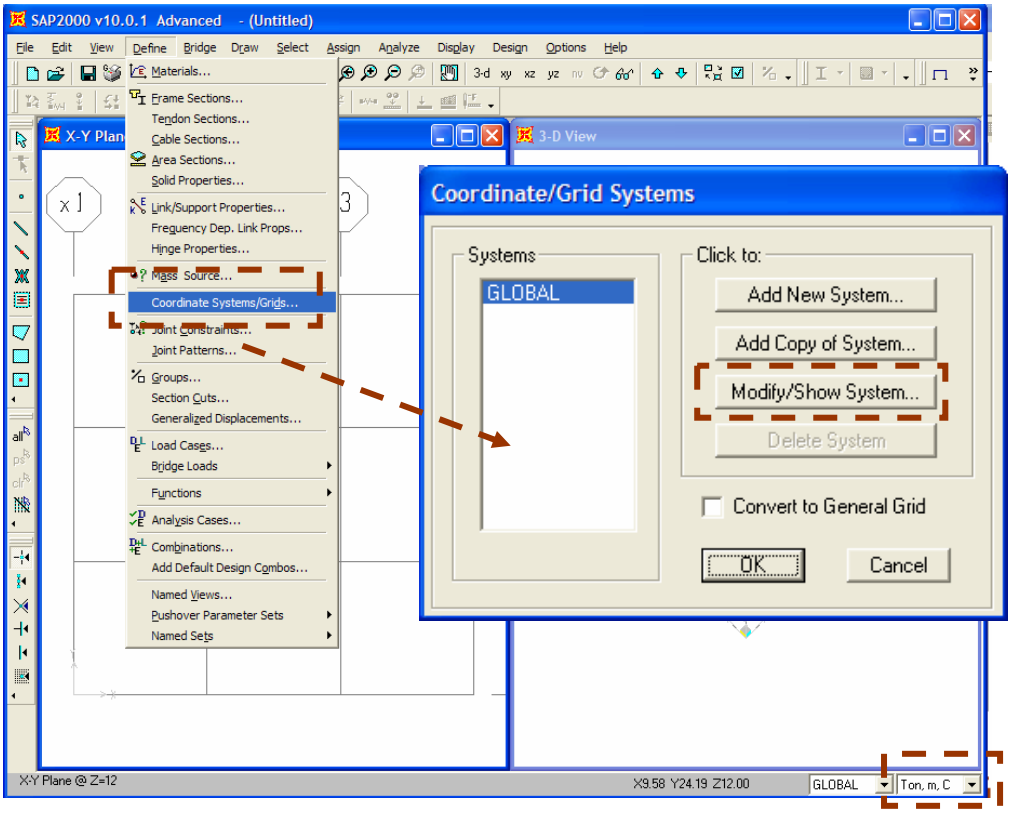
Repeat Step 3-1 and 3-2 to add two more shell sections with thicknesses of 25 and 30 cm, respectively. Next, click “OK”.



4. Modify Grid System

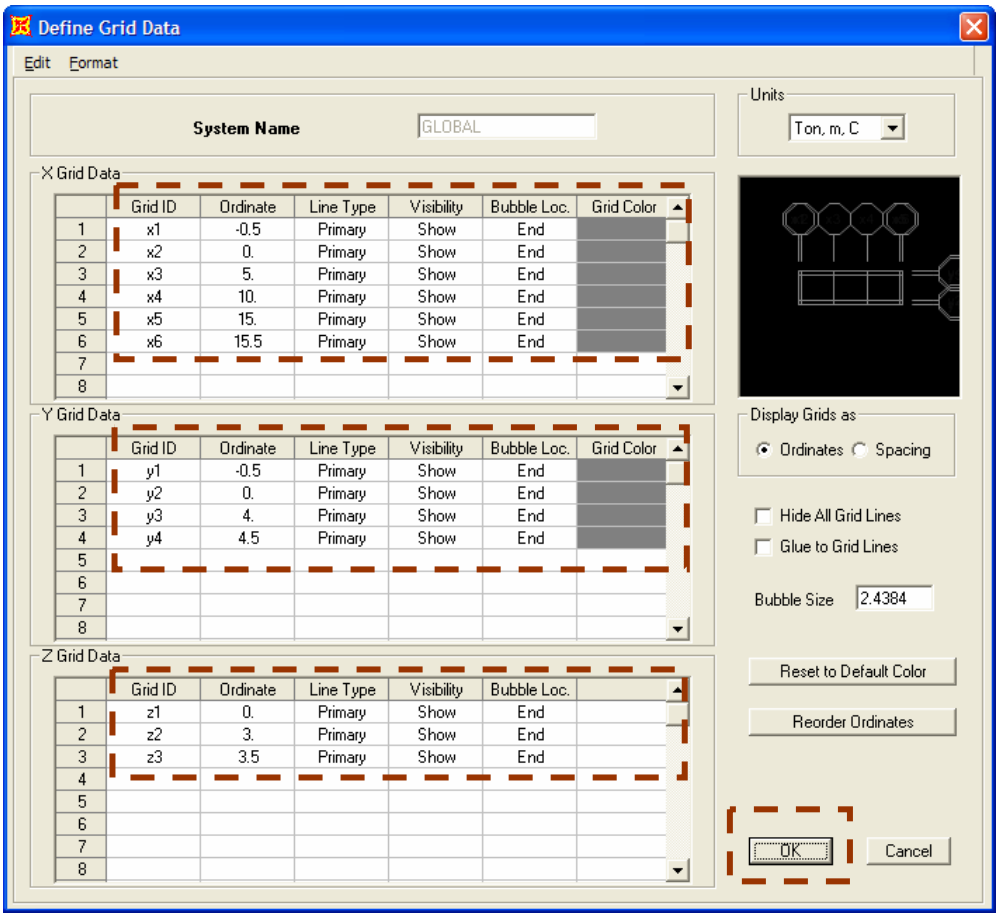
Step 4-1: Modify Grid System

Select working unit to be “Ton, m, C”, go to **Define >> Coordinate System/Grid** and click on “Modify/Show System...”



Step 4-2: Modify Grid System

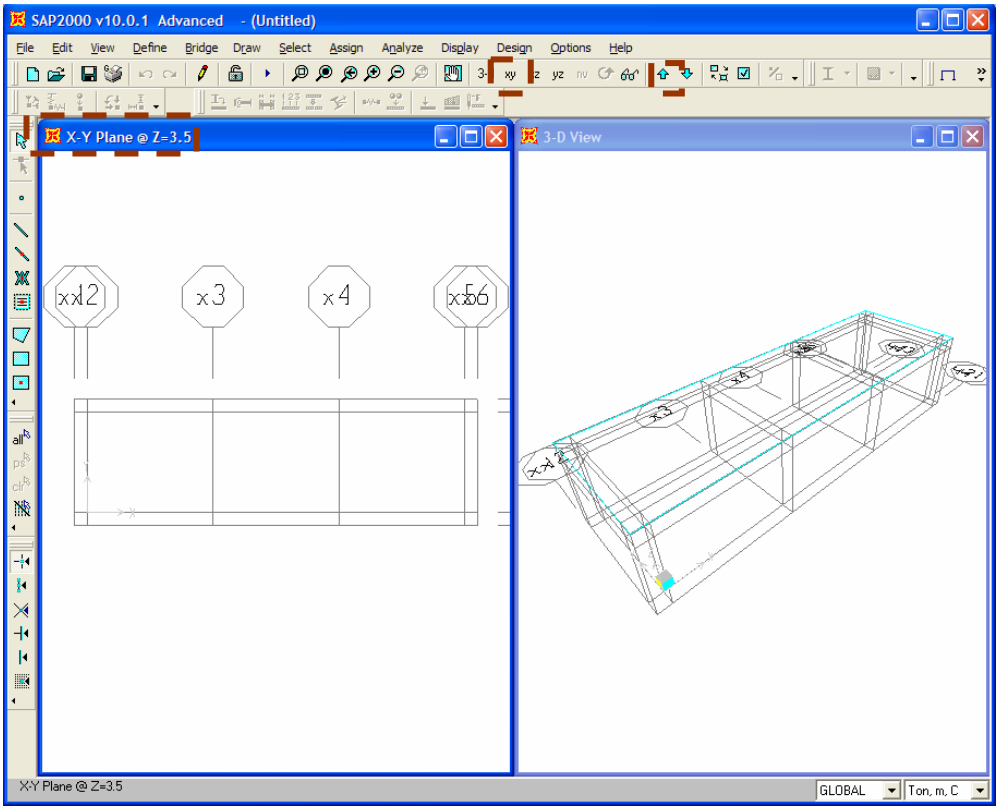
Modify grid system as shown below. Click “OK” button. Next, also click “OK” in the preceding window (Coordinate/ Grid Systems).



5. Draw Top Slabs

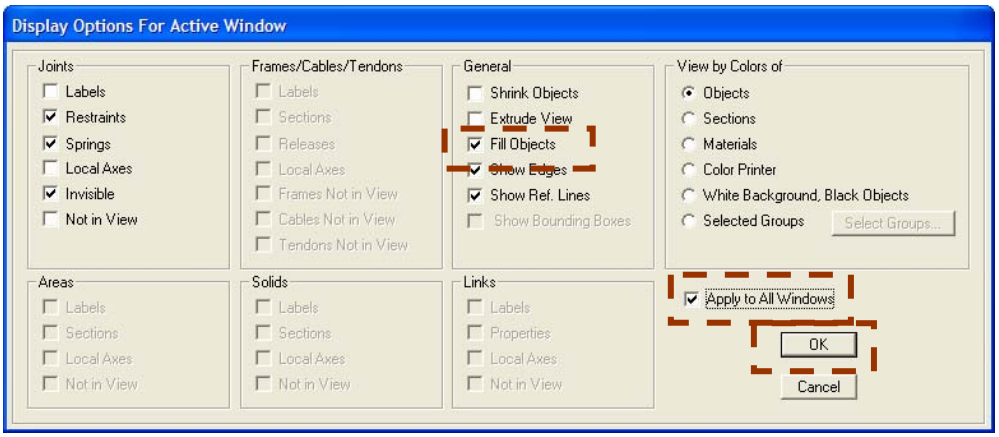
Step 5-1: Change Plan View to Top Slab Level

Select left window, click on *Set XY View* button to change view to plan view and and click on or to move to move plan level to Z = 3.5 m. Plan elevation (X-Y Plane) is displayed at top-left of window.



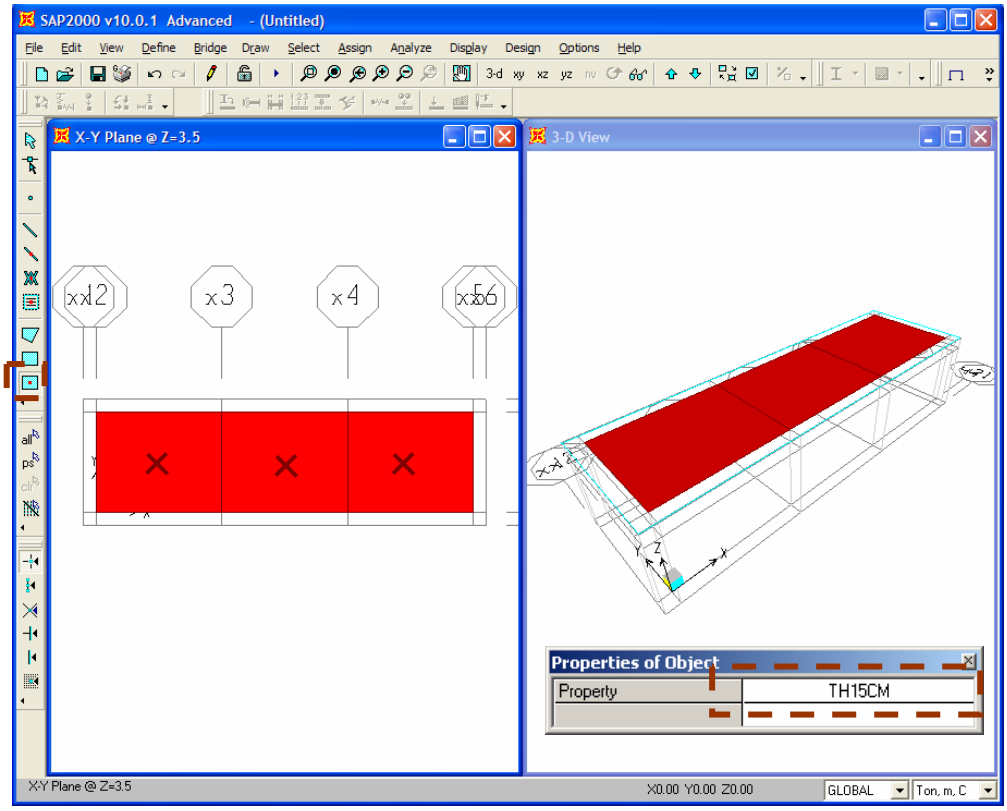
Step 5-2: Fill Object to Show Shell Element

Go to **View >> Set Display Options**, select “Fill Object” and “Apply to All Windows”. Next, click “OK”.



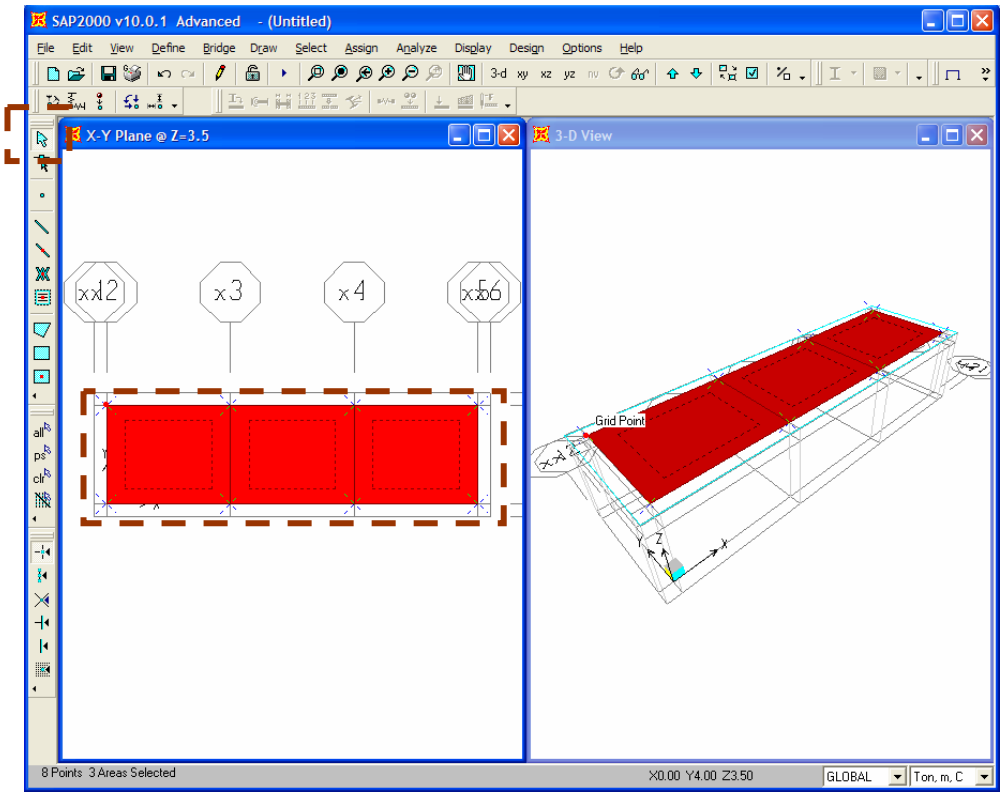
Step 5-3: Draw Roof Slab

Click on *Quick Draw Area Element* button, select “TH15CM” and click on roof slab panel area as shown in figure below one by one.



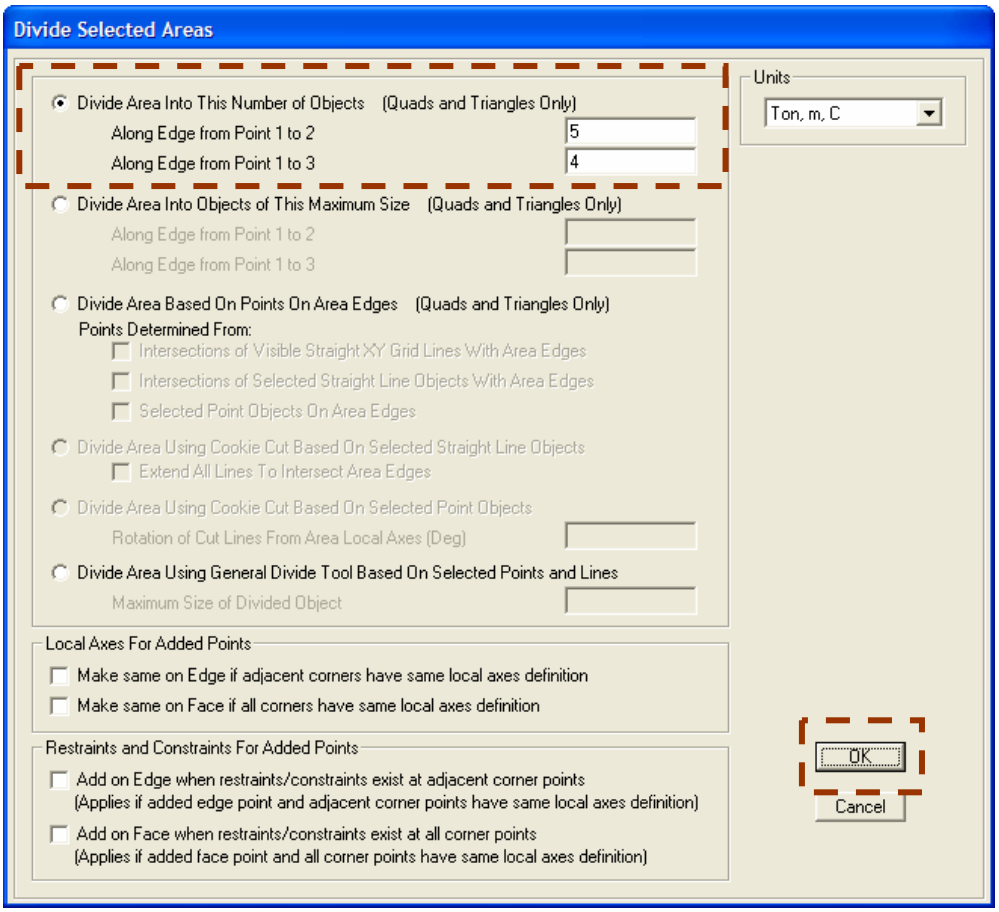
Step 5-4: Divide Roof Slab

Click on *Set Select Mode* button, select all roof slab panels, go to **Edit >> Divide Areas...**.



Step 5-5: Specify Slab Meshing Size

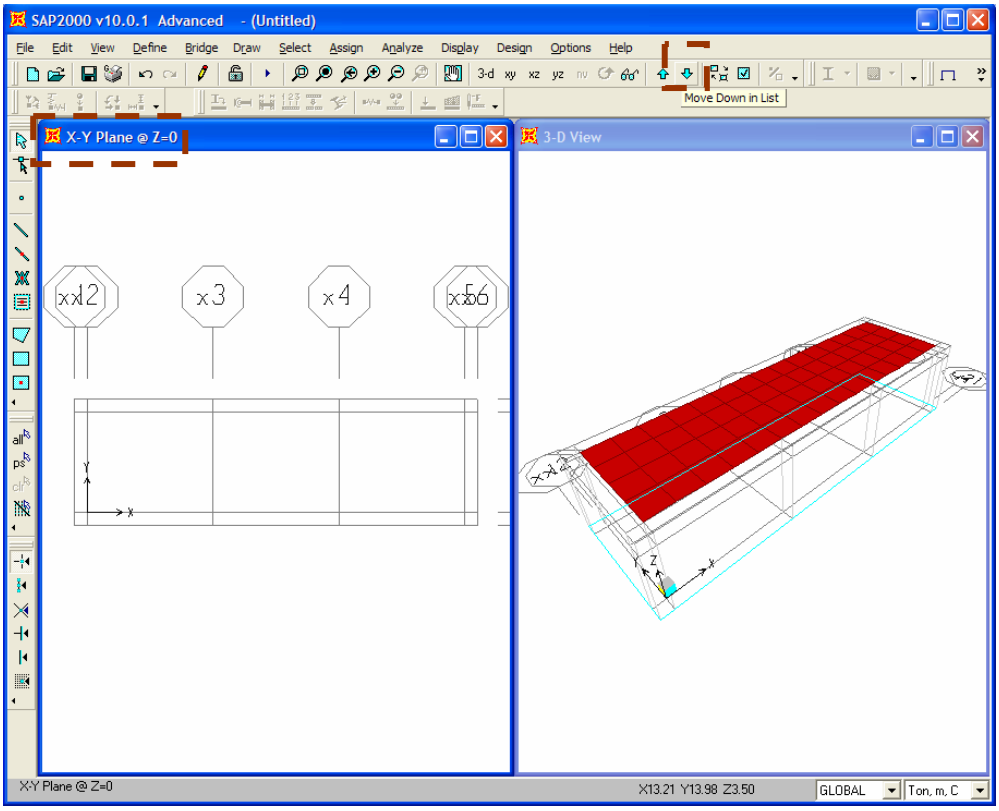
Specify parameters as shown below. Then, click “OK”.



6. Draw Bottom Slabs

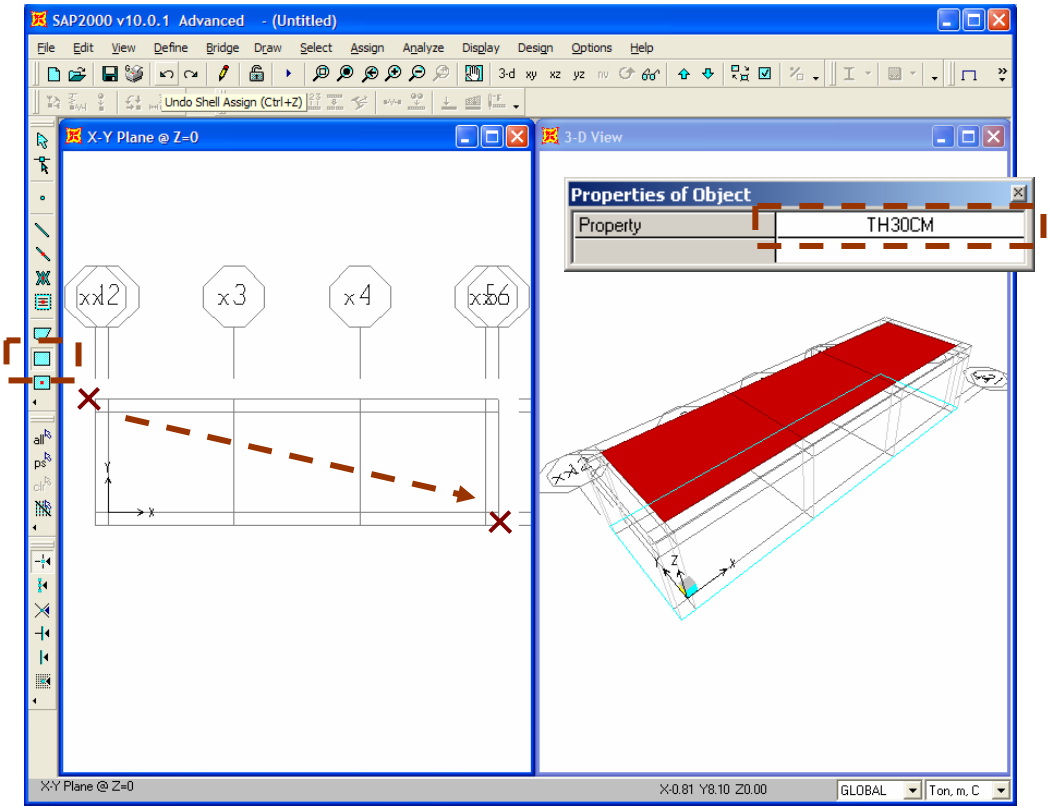
Step 6-1: Move Plan View to Bottom Level

Click on twice to move plan view to the level Z = 0.00.



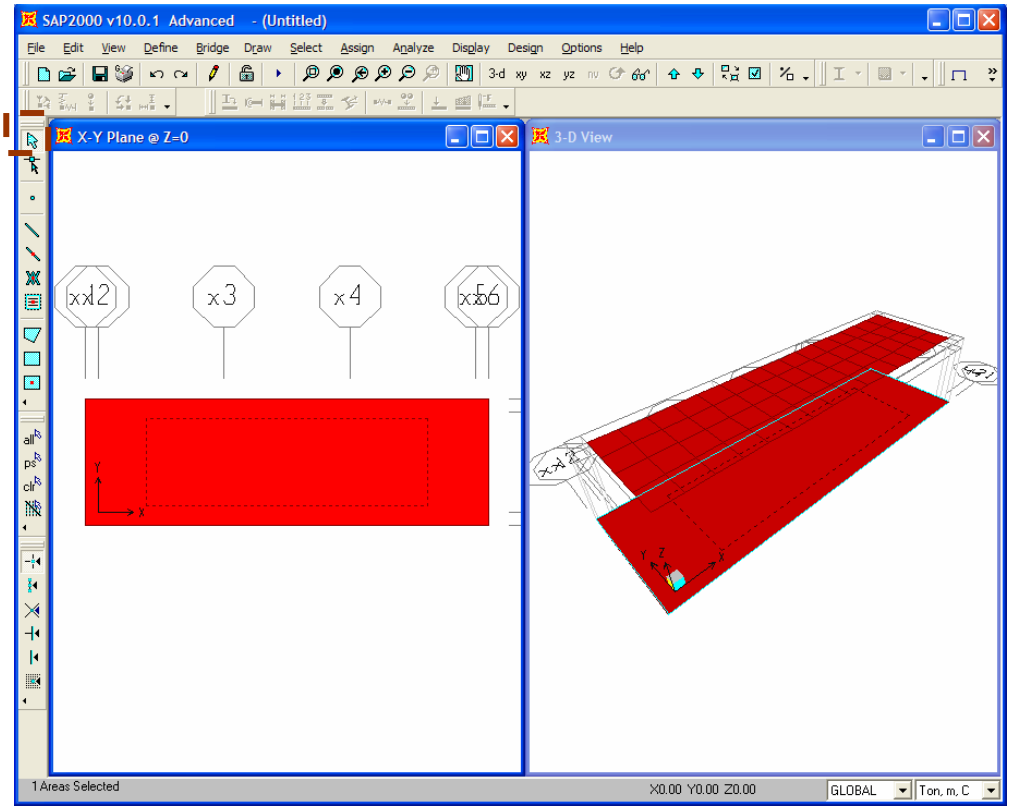
Step 6-2: Draw Bottom Slabs

Click on *Draw Rectangular Area Element* button, select “TH30CM” and click on two nodes at the corners of bottom slab as shown below.

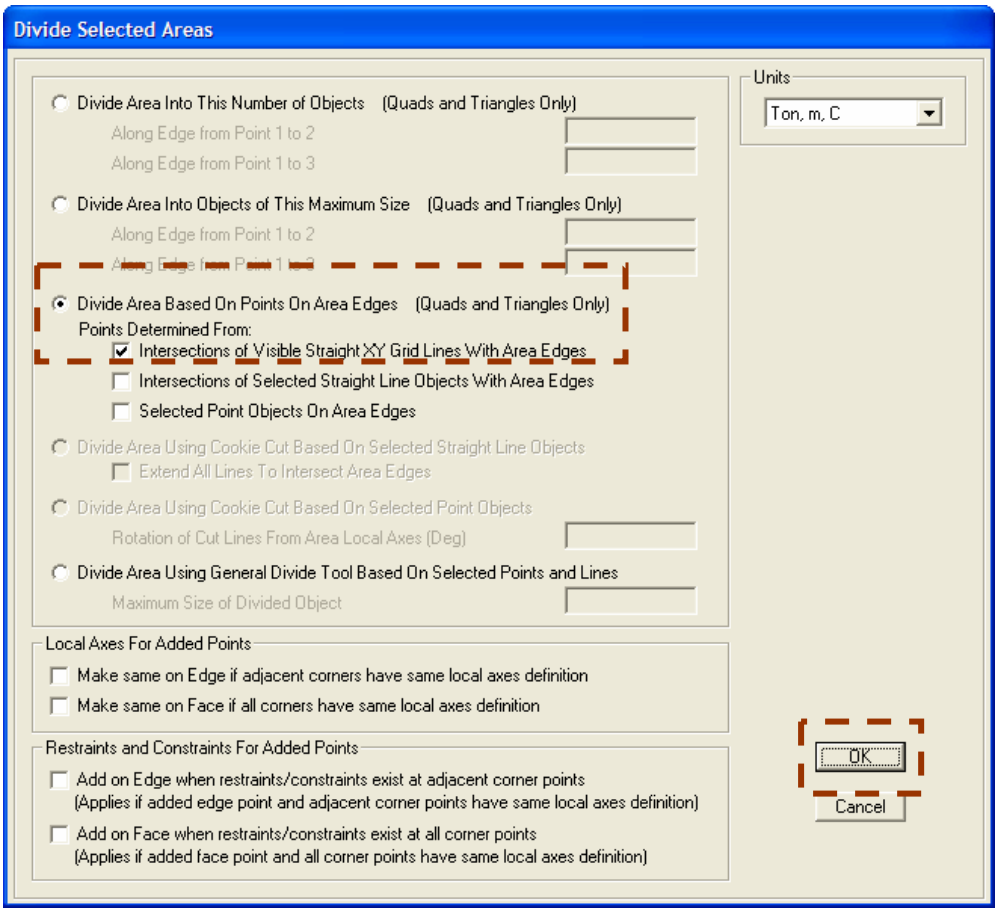


Step 6-3: Mesh Bottom Slabs

Select bottom slabs as shown in figure below and go to **Edit >> Divide Areas...**

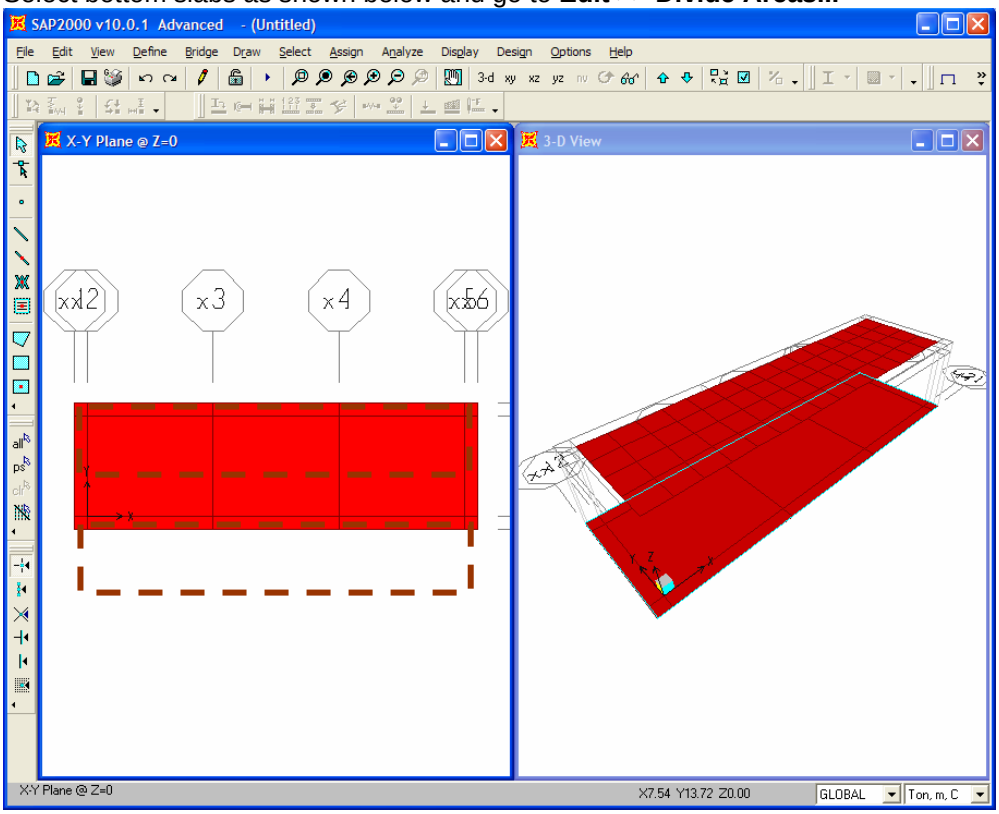


Specify parameters as shown below. Then, click “OK”.

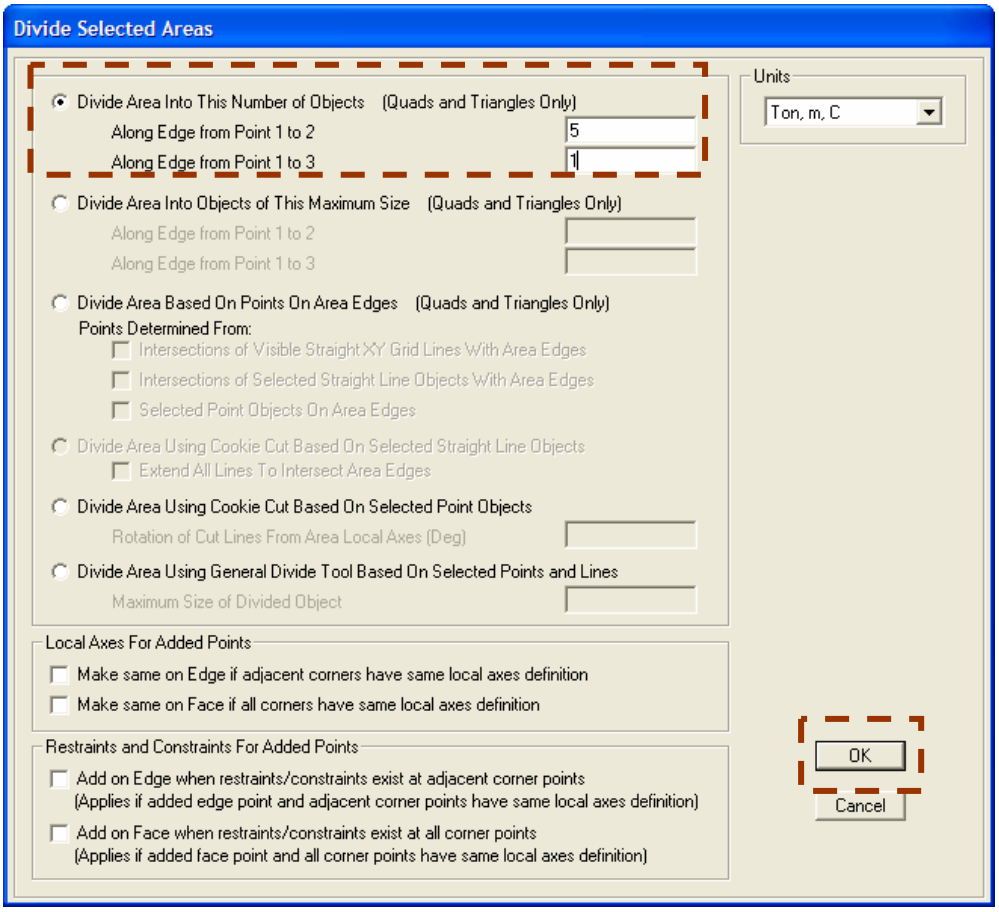


Step 6-4: Mesh Bottom Slabs

Select bottom slabs as shown below and go to **Edit >> Divide Areas...**

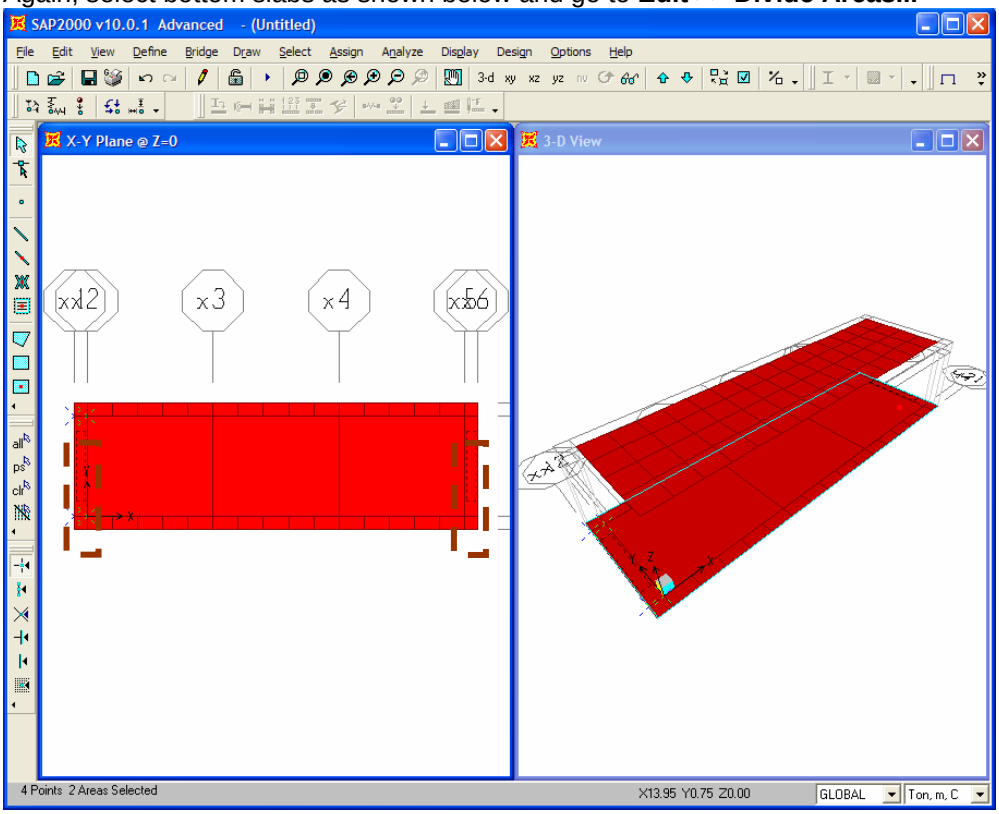


Specify parameters as shown below. Then, click “OK”.

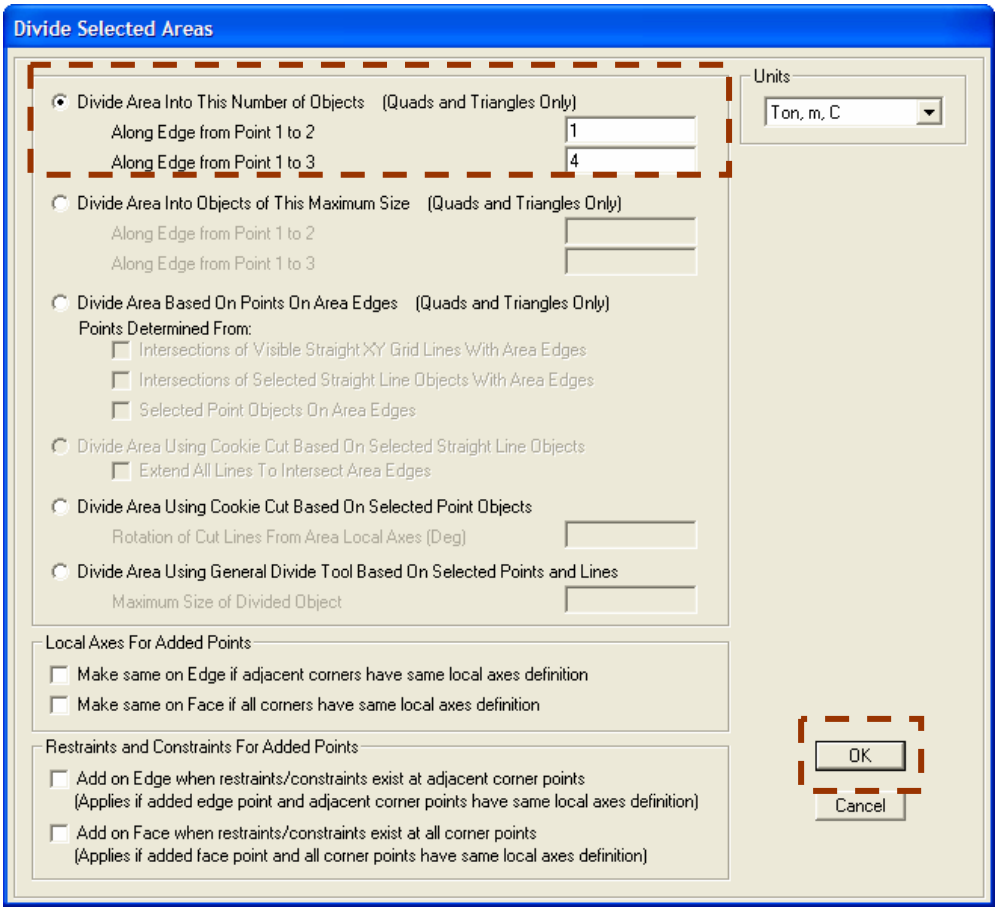


Step 6-5: Mesh Bottom Slabs

Again, select bottom slabs as shown below and go to **Edit >> Divide Areas...**

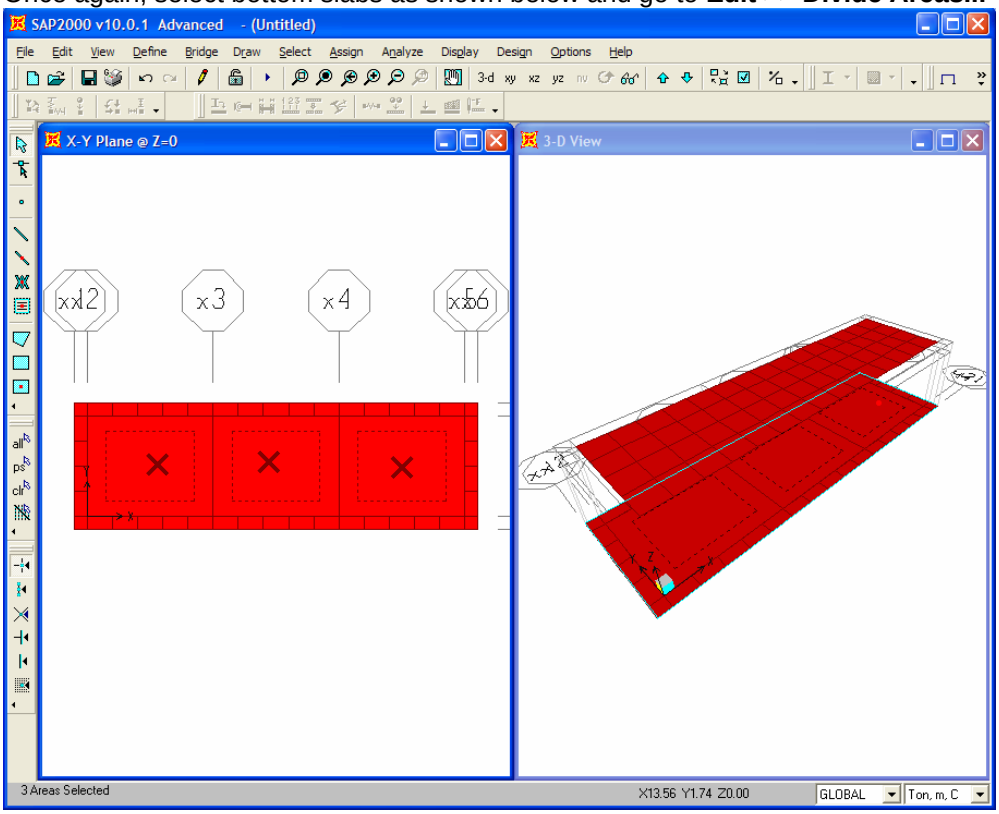


Specify parameters as shown below. Then, click “OK”.

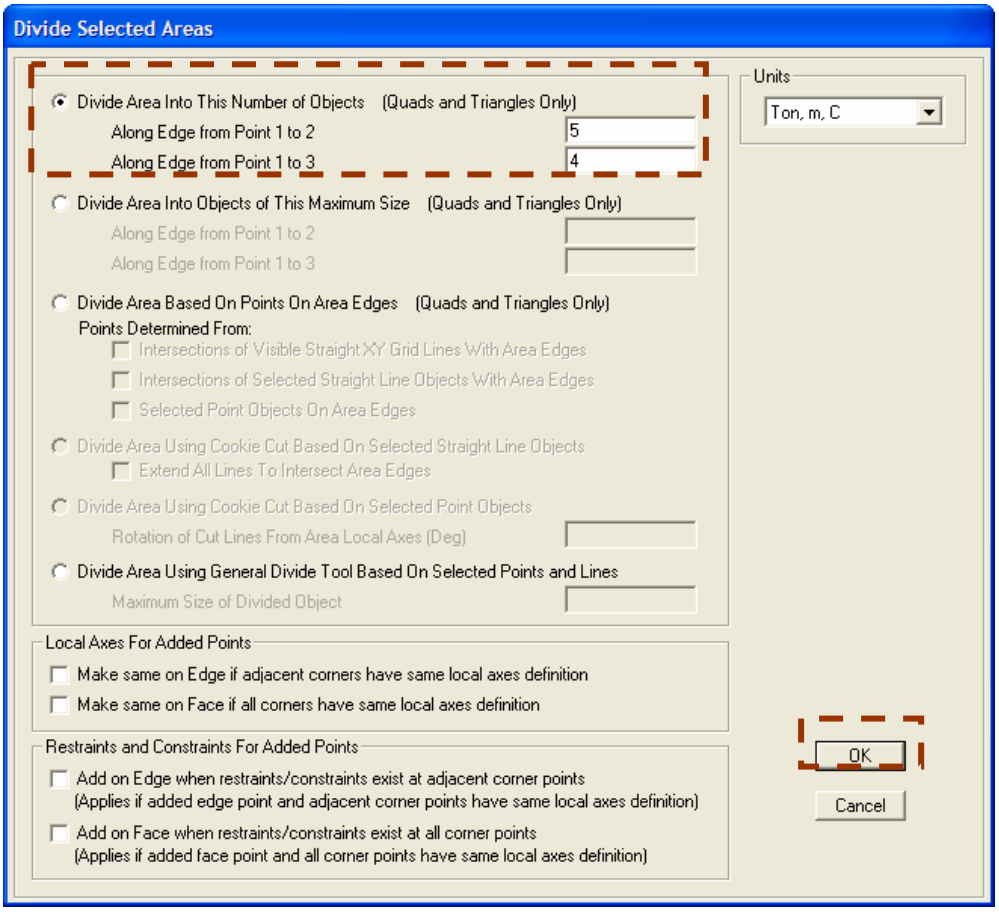


Step 6-6: Mesh Bottom Slabs

Once again, select bottom slabs as shown below and go to **Edit >> Divide Areas...**



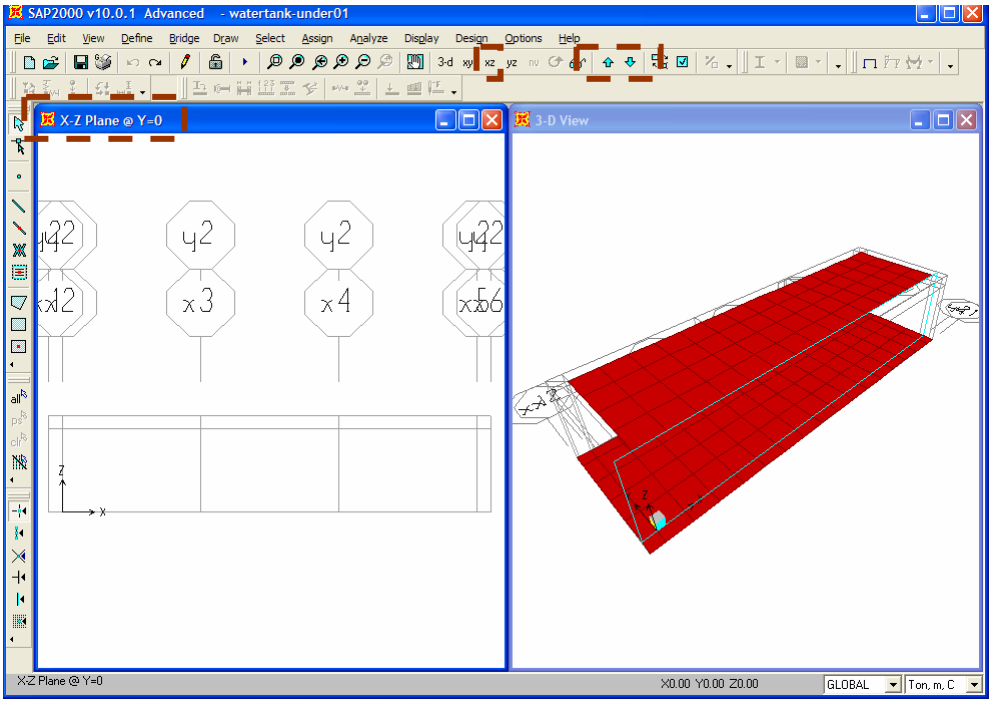
Specify parameters as shown below. Then, click “OK”.



7. Draw Wall Panels

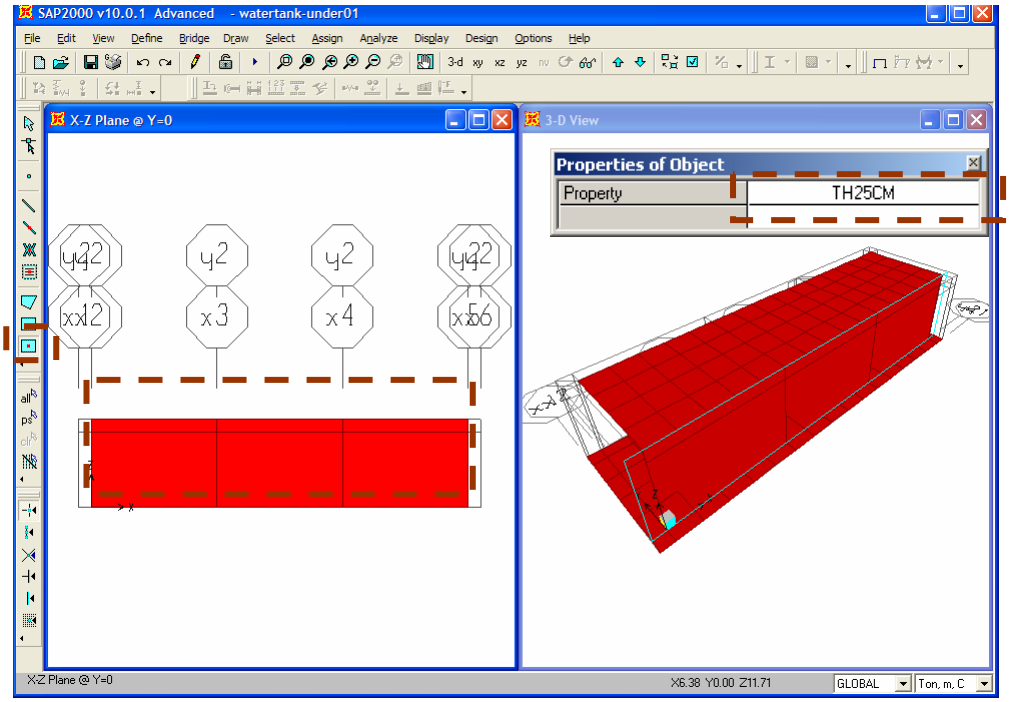
Step 7-1: Change View to Elevation View to XZ Plane

Select left window, click on *Set XZ View* button to change view and click on or to move elevation view to Y = 0 m.



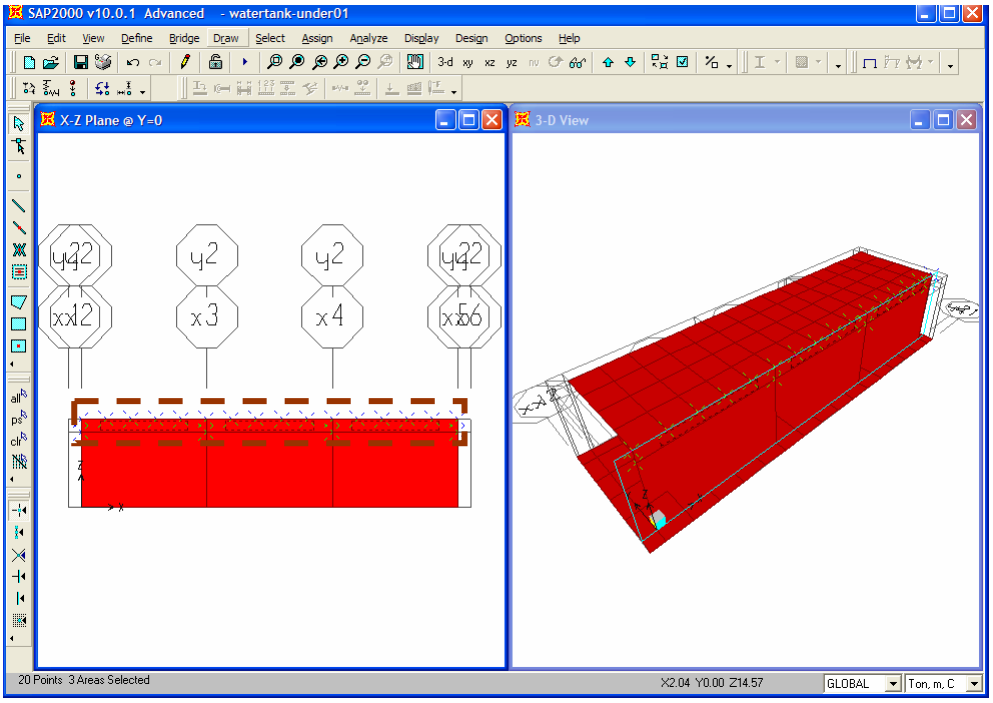
Step 7-2: Draw Wall Panels

Click on *Quick Draw Area Element* button, select “TH25CM” and draw area to cover wall panel area as shown in figure below.

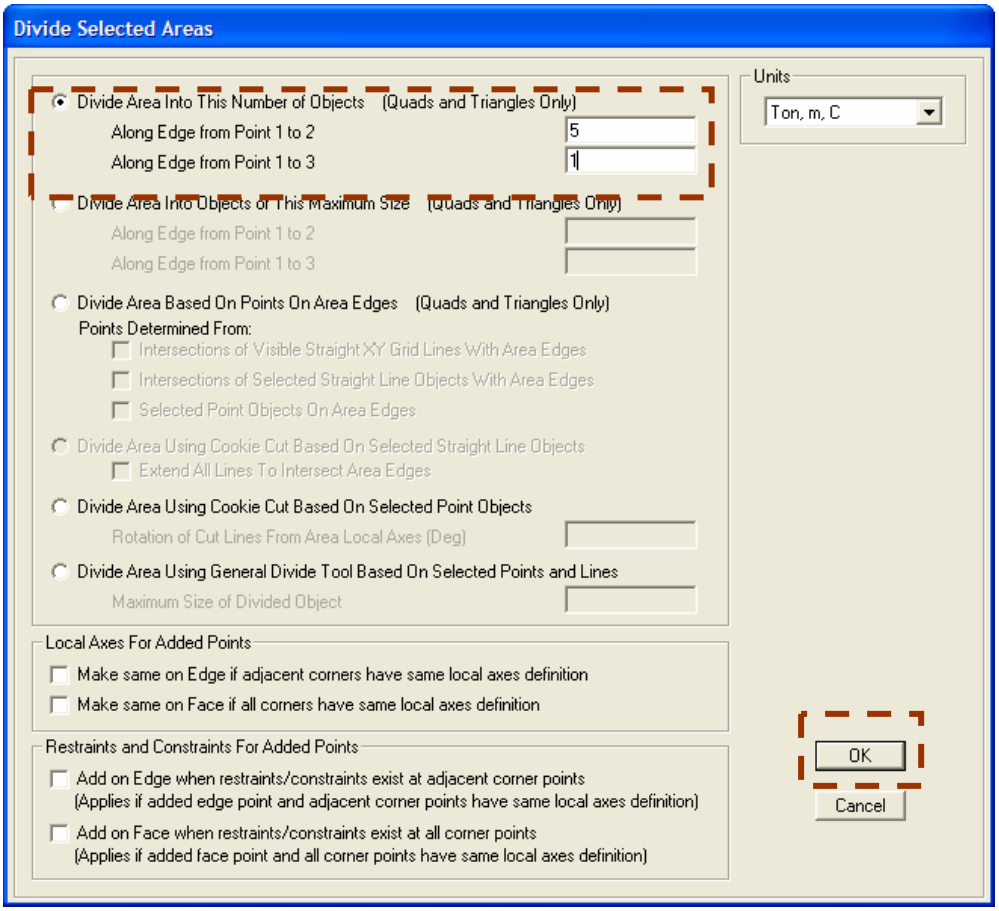


Step 7-3: Mesh Wall Panels

Click on , select the top part of the wall panel as shown in figure below and go to **Edit >> Divide Areas...**

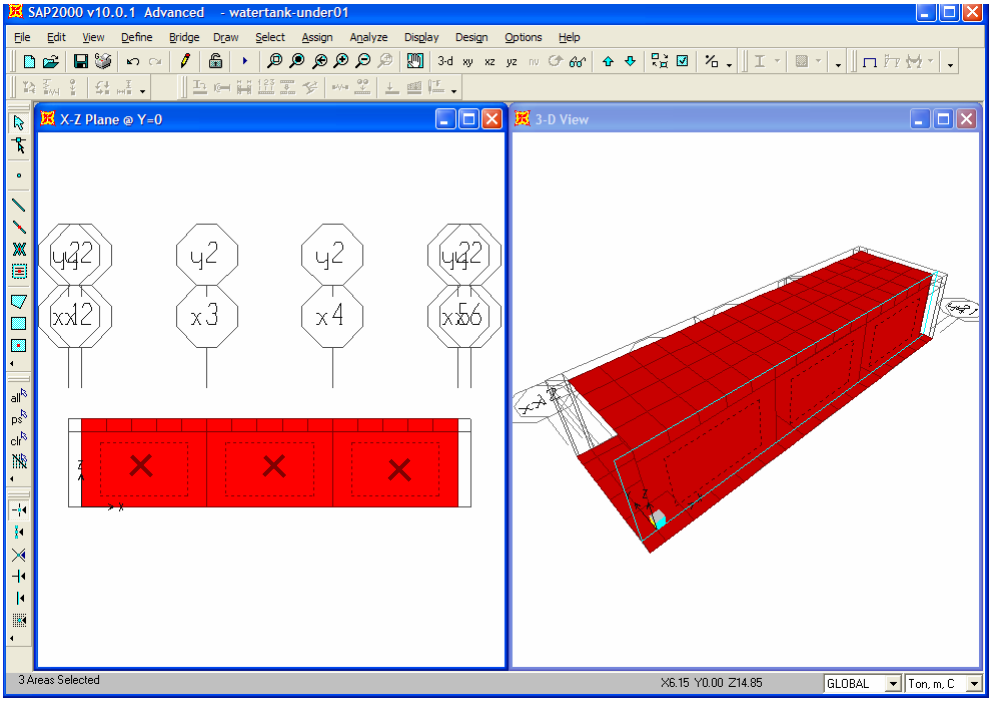


Specify parameters as shown below. Then, click “OK”.

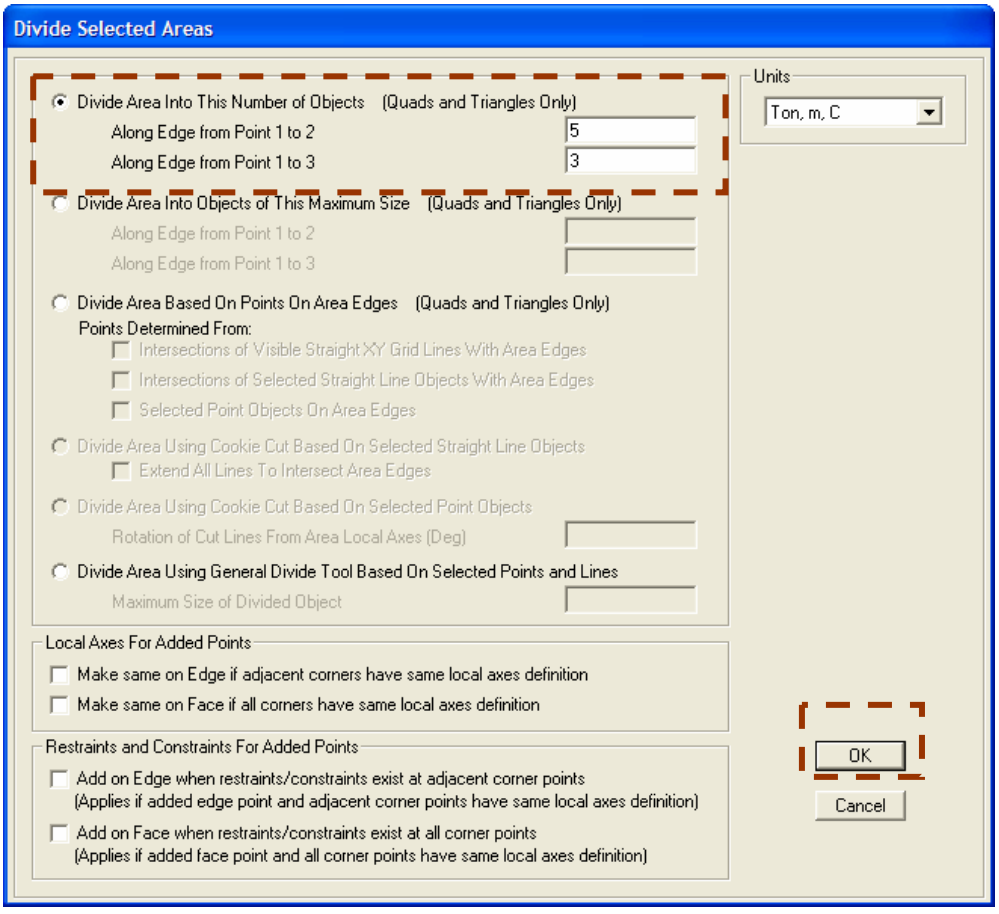


Step 7-4: Mesh Wall Panels

Select bottom wall panels as shown in above figure and go to **Edit >> Mesh Area**.

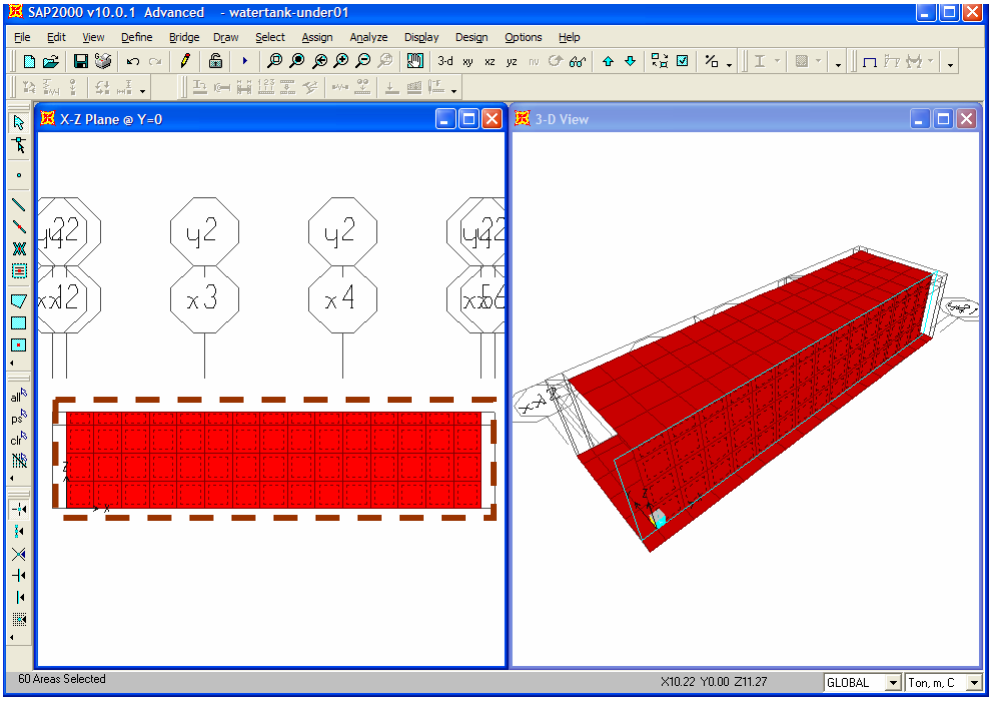


Specify parameters as shown below. Then, click “OK”.

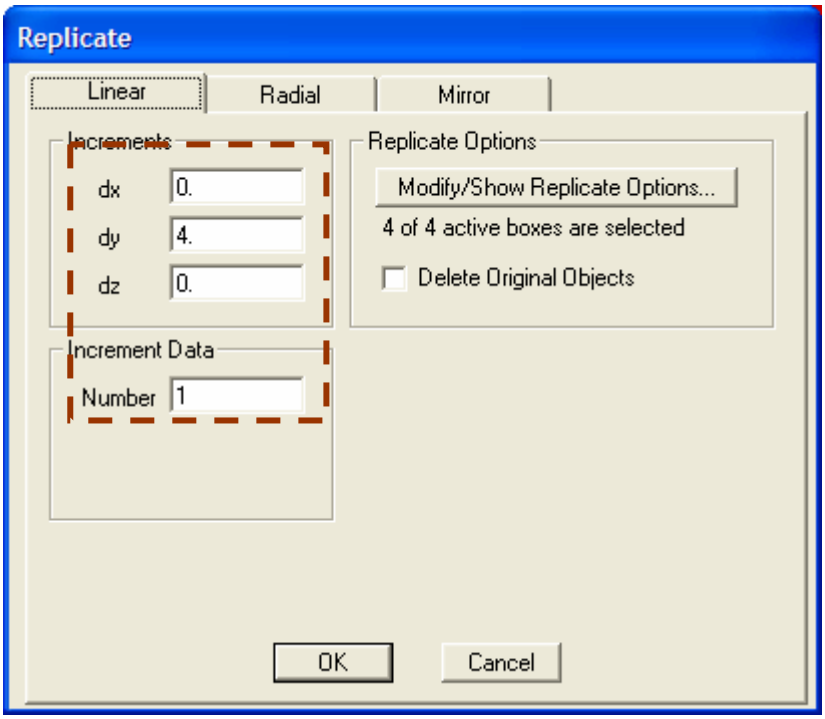


Step 7-5: Copy Wall Panels

Select all wall panels, go to **Edit >> Replicate** by drawing rectangular selection to cover all wall panels.

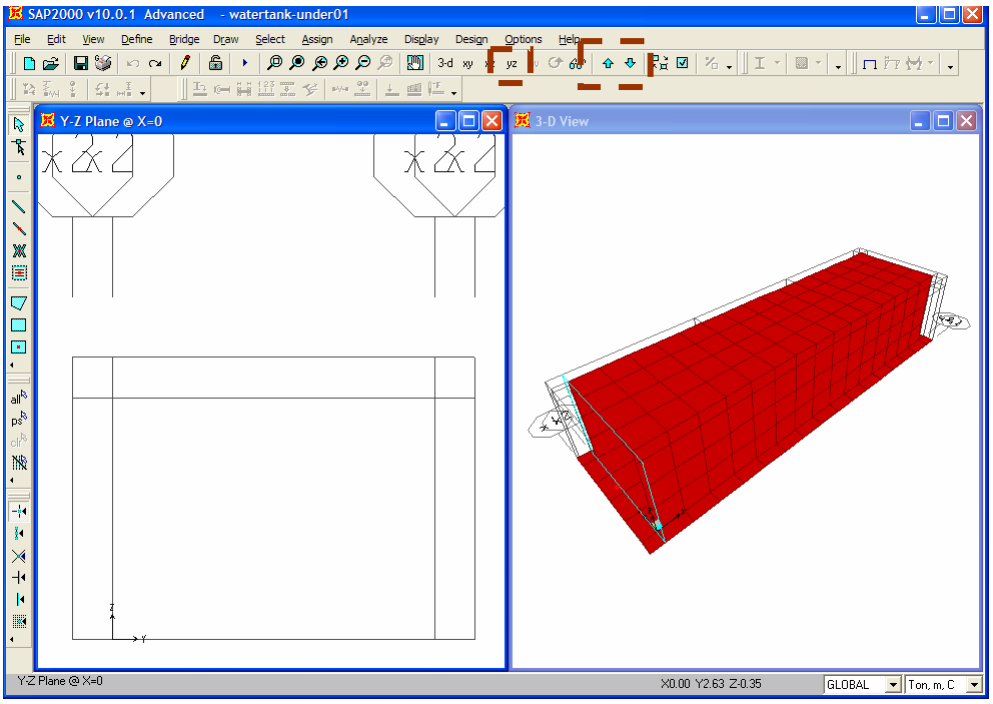


Specify parameters as shown below. Then, click “OK”.



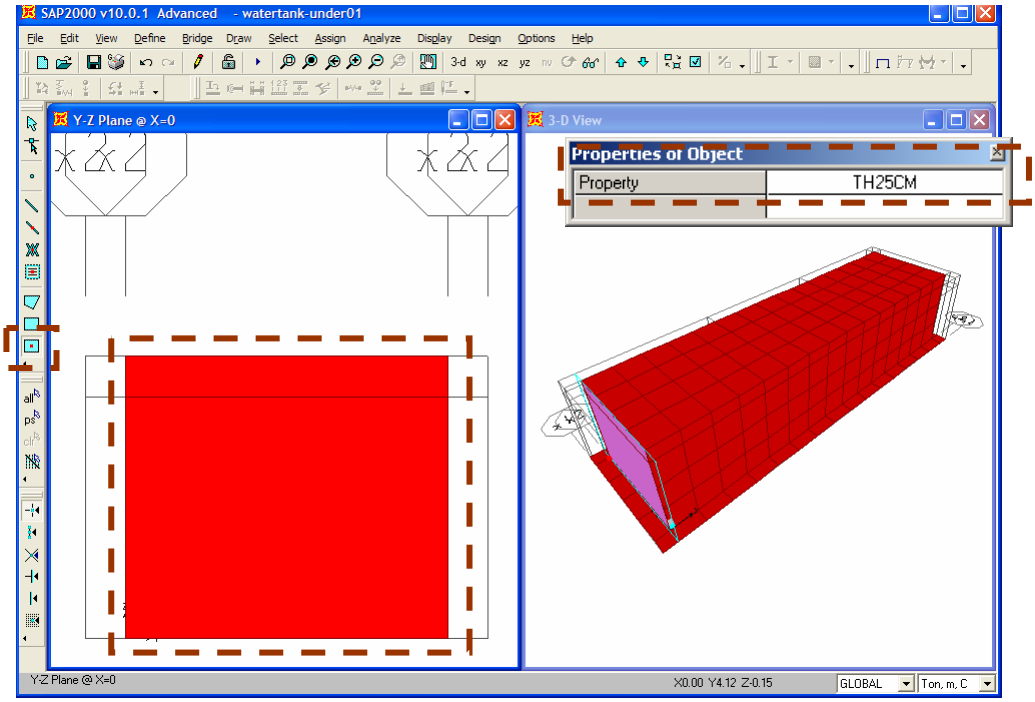
Step 7-6: Change View to Elevation View to YZ Plane

Select left window, click on *Set YZ View* button to change view and click on or to move elevation view to X = 0 m.



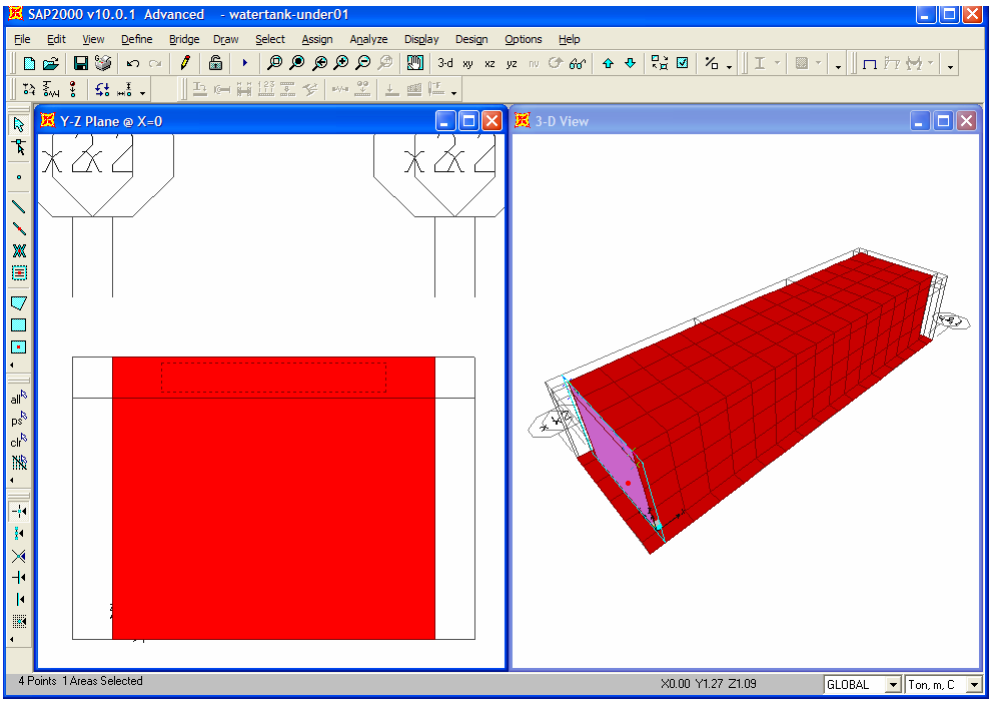
Step 7-7: Draw Wall Panels

Click on *Quick Draw Area Element* button, select "TH25CM" and draw rectangular to cover wall areas as shown in above figure.

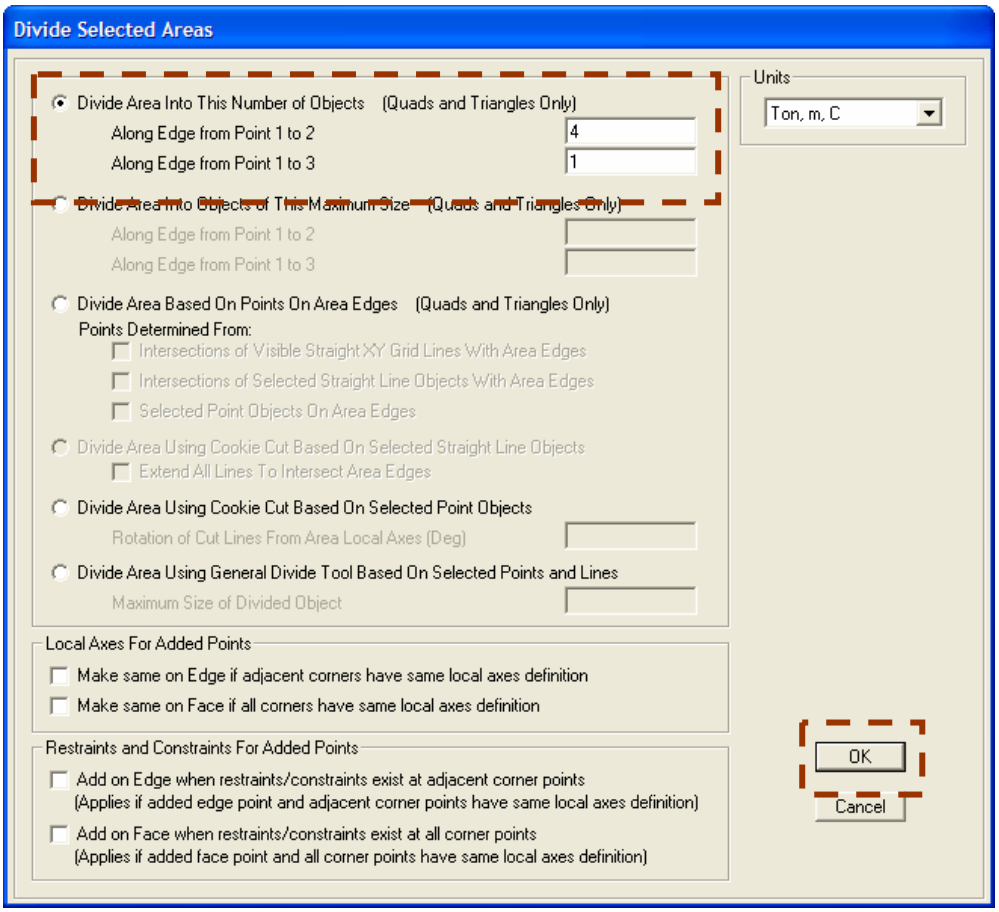


Step 7-8: Mesh Wall Panels

Click on *Set Select Mode* button, select top wall panels as shown in figure below and go to **Edit >> Divide Areas...**

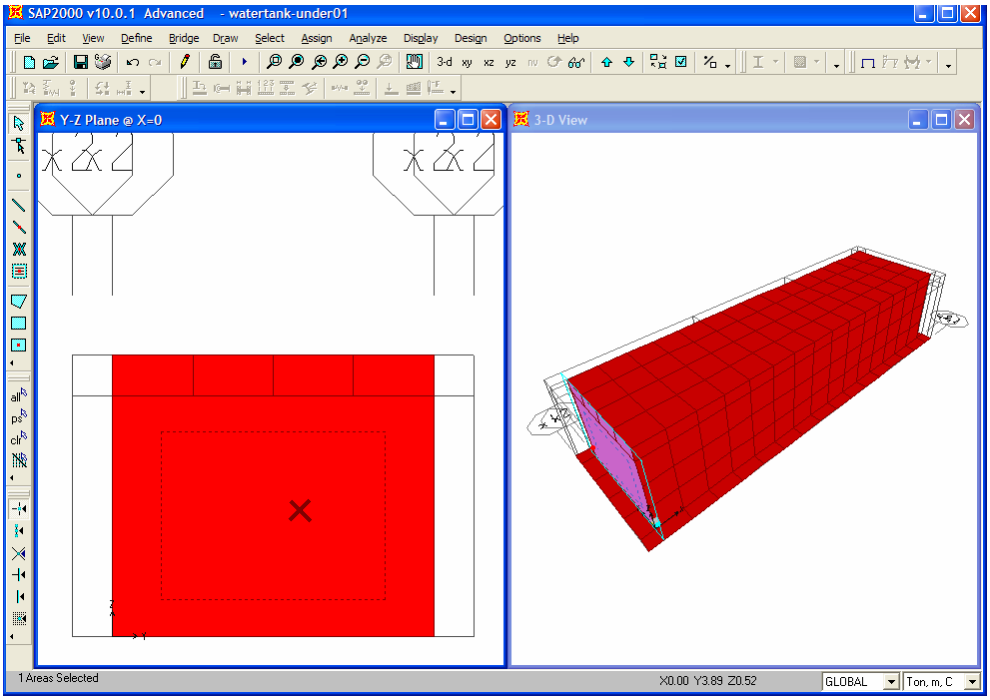


Specify parameters as shown below. Then, click “OK”.

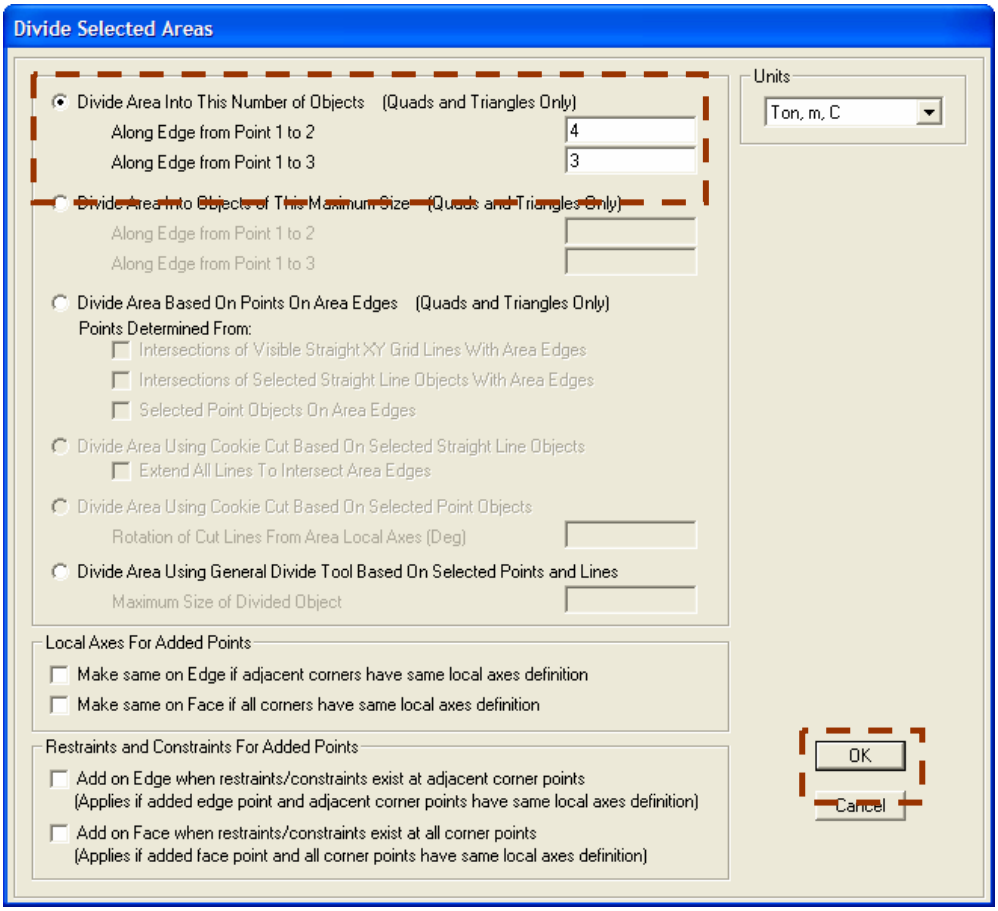


Step 7-9: Mesh Wall Panels

Select bottom wall panels as shown in figure below and go to **Edit >> Divide Areas...**

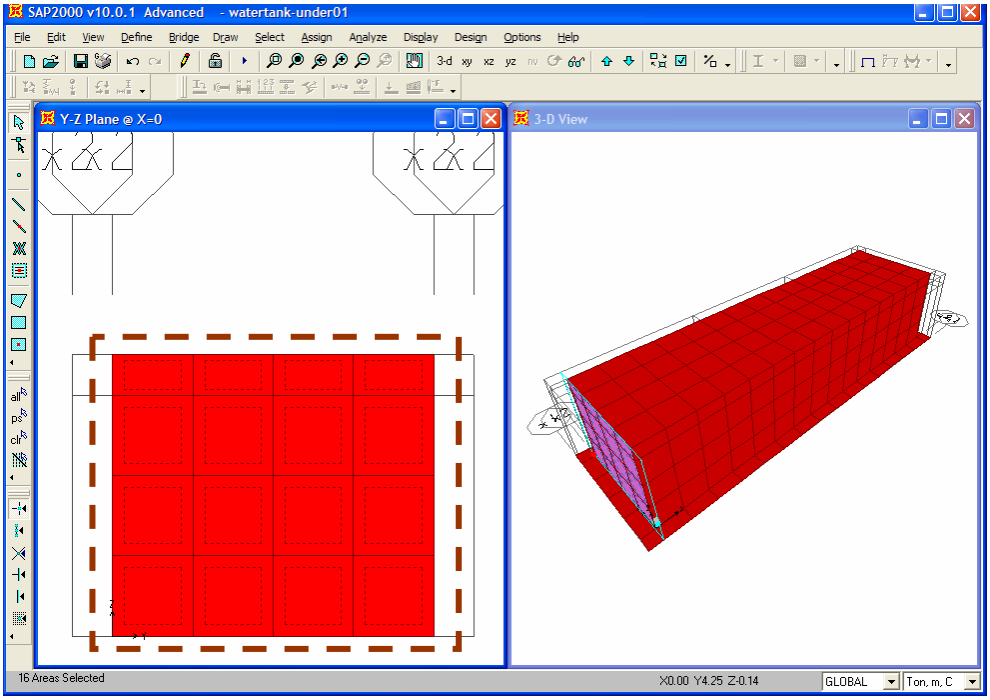


Specify parameters as shown below. Then, click “OK”.

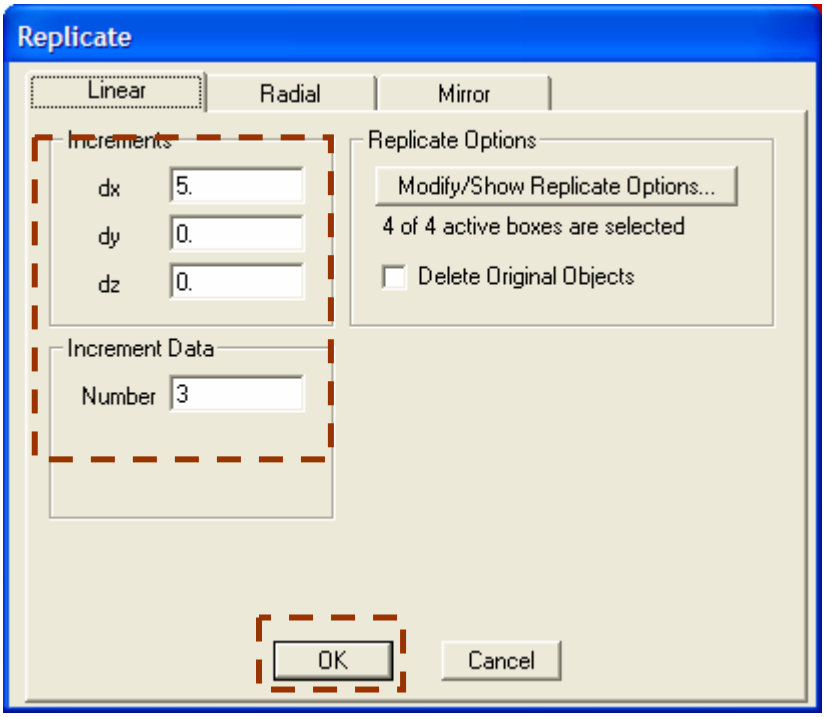


Step 7-10: Copy Wall Panels

Select all wall panels and go to **Edit >> Replicate**



Specify parameters as shown below. Then, click “OK”.

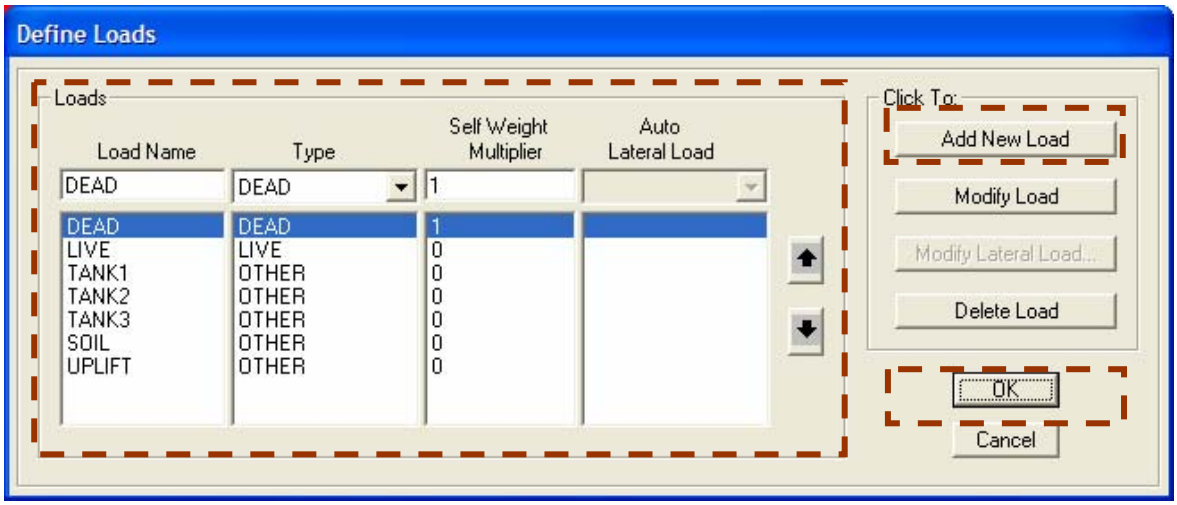


8. Define Load Cases

Water load in each tank will be assigned with difference load cases (TANK1, TANK2 and TANK3) for all possible loading condition.

Step 8-1: Add Load Case

Go to **Define >> Load Cases**. Then, specify load case name in text box “Load Case” and then click on “Add New Load” for each and every load case as shown in figure below

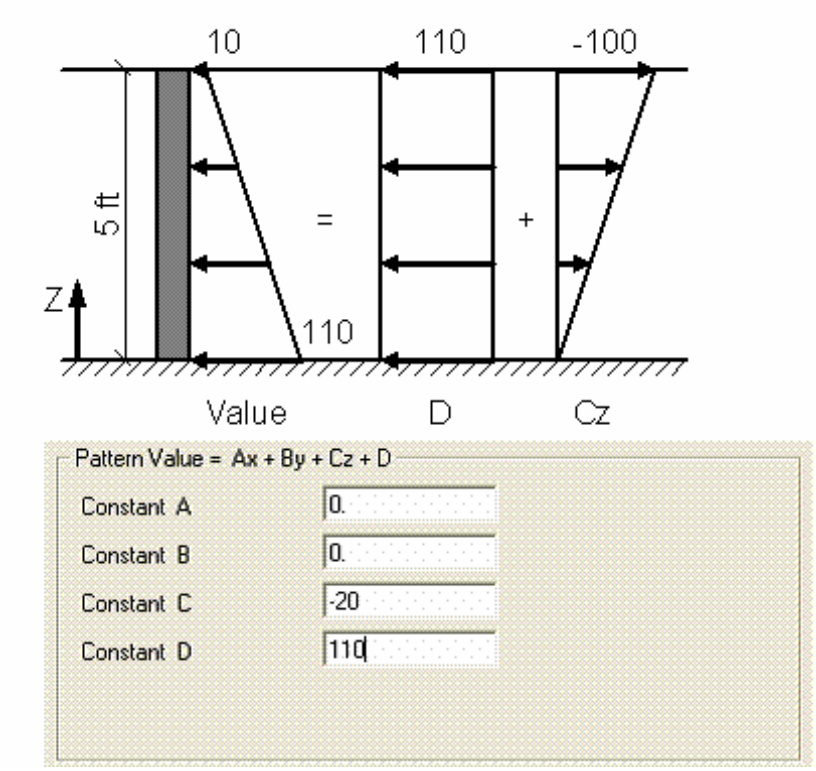


9. Define and Assign Joint Pattern

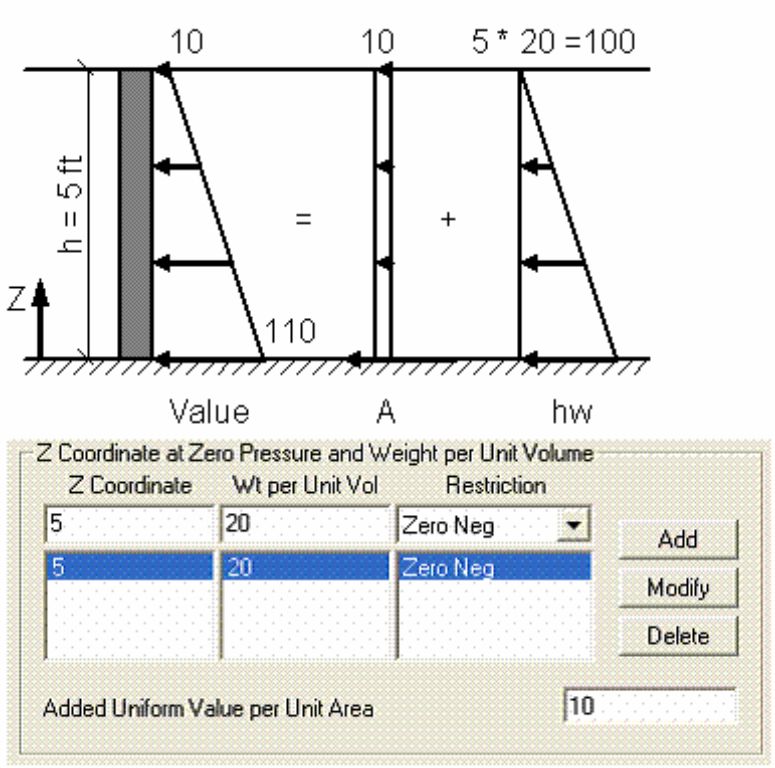
A joint pattern is simply a set of scalar values defined at the joints for assigning more complex distributions of temperature and pressure over the structure. Joint patterns by themselves create no loads on the structure. There are two types of joint pattern as described below.

a. **X Y, Z Multiplier Method.** Type values in the Constant A, B, C and D edit boxes. The pattern value at each selected joint is calculated as Ax+By+Cz+D, where x, y and z are the joint coordinates of the selected joint in the current coordinate system. The units of constants A, B, and C are 1/Length and constant D is unitless. The calculated pattern value is also unitless.

As an example of this assignment method, the following input defines the joint pattern values used to apply a varying soil pressure load on a wall.



b. **Z Coordinate at Zero Pressure and Weight Per Unit Volume Method.** Type Z Coordinate and weight per unit volume (Wt per unit Vol) values in the edit boxes and select a **Restriction** from the drop down list, then click the Add button. If needed, check the *Added Uniform Value per Unit Area* check box to specify an added uniform value per unit area. The specified Z coordinate is assumed to be in the current coordinate system. The pattern value at each selected joint is calculated as:



Zcoord	=	specified Z coordinate
z	=	Z coordinate of the selected joint in the current coordinate system
w	=	specified weight per unit volume
A	=	specified added value per unit area

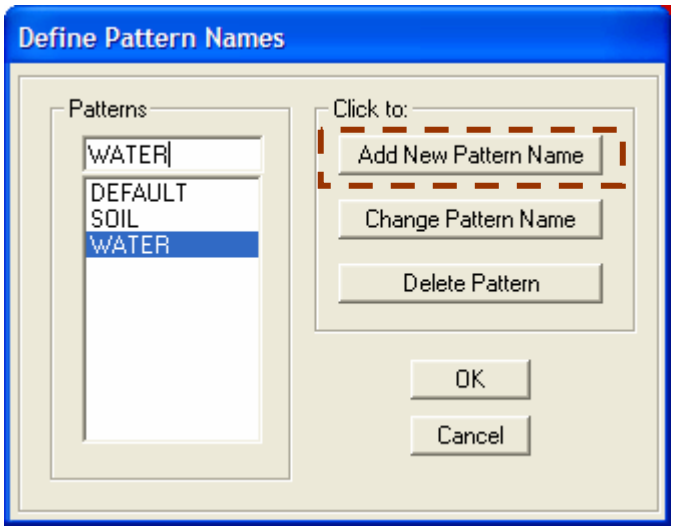
All appropriate unit conversion factors are used when calculating the pattern value; however, after the pattern value has been obtained, it is thereafter assumed to be unitless.

Note that positive and negative value restrictions can be specified individually for each set of Z coordinates and weight per unit volume in the summation as well as for the final calculated pattern value.

As an example of this assignment method, the following input defines the joint pattern values used to apply a varying soil pressure load on a wall.

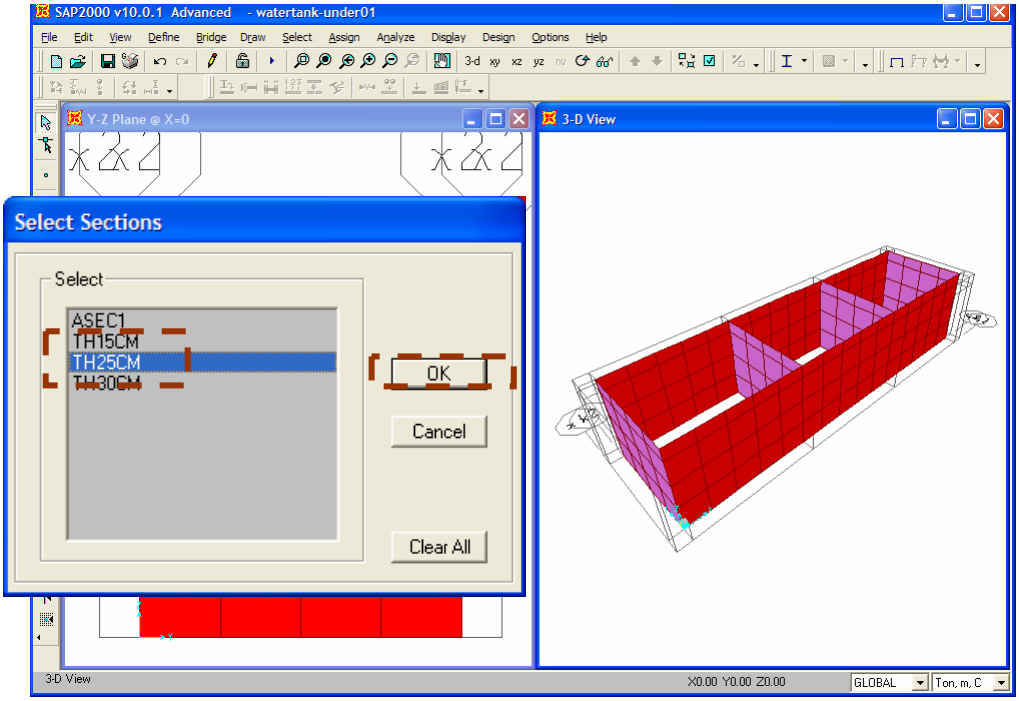
Step 9-1: Add New Pattern

Go to **Define >> Joint Patterns**, enter pattern name and click on “Add New Pattern Name to add 2 new joint patterns (“SOIL” and “WATER”)



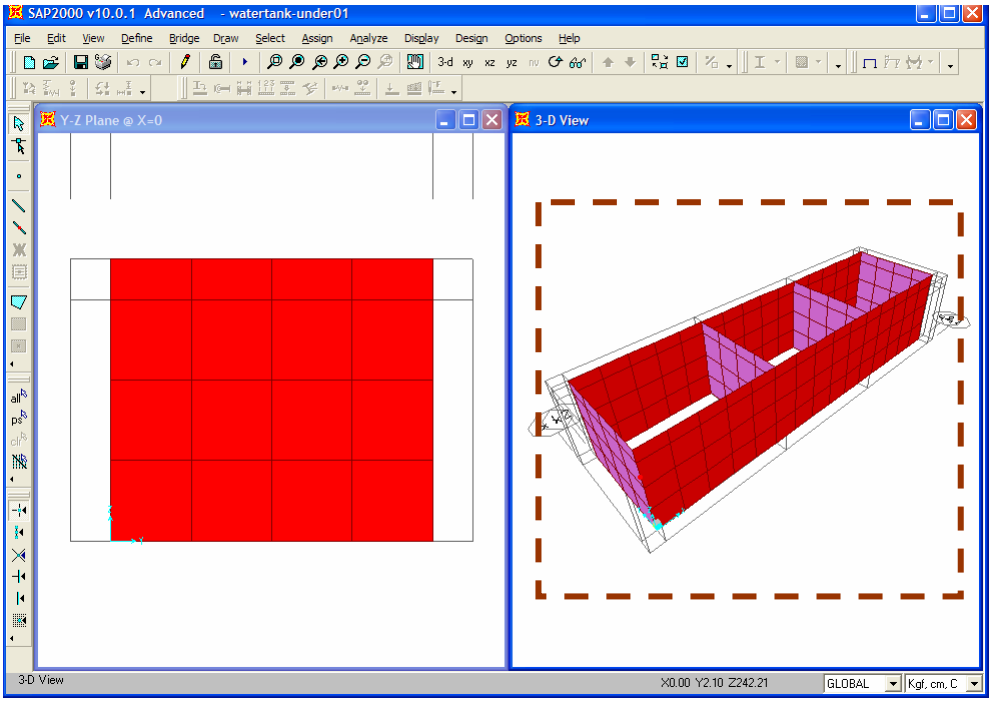
Step 9-2: Select Only Wall Panels

Activate 3D View Window by clicking on it, go to **Select >> Select >> Area Section** and select “TH25CM” from the list to select only wall panels. Go to **View >> Selection Only** to view only selected element (wall panels)



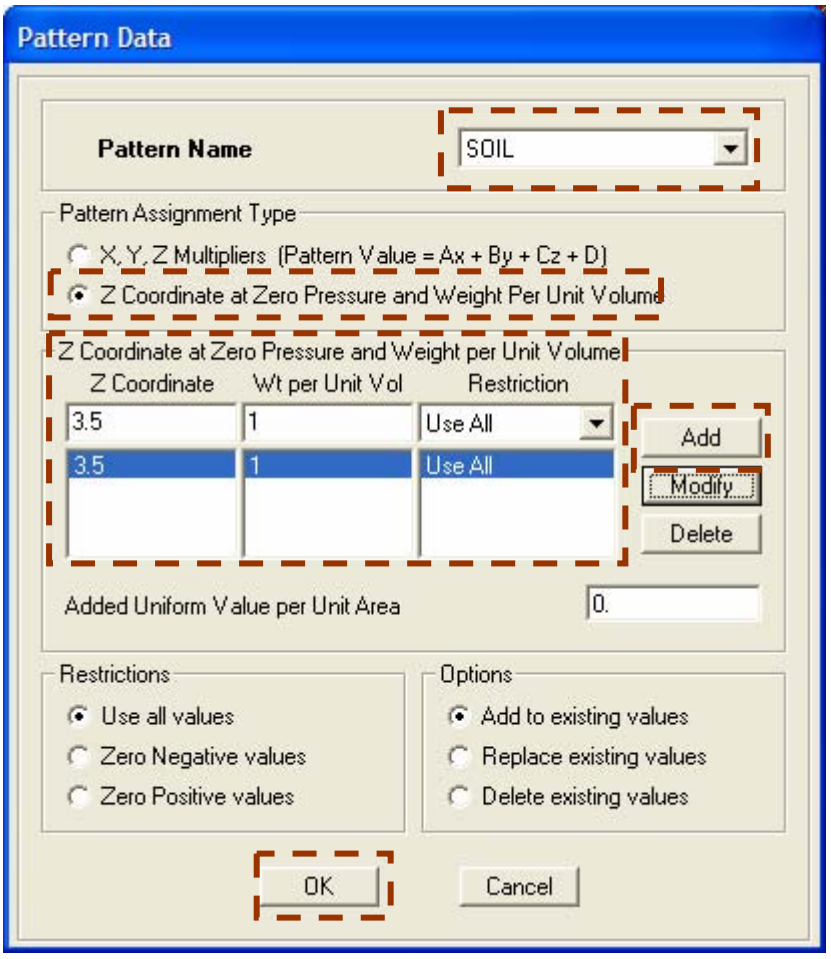
Step 9-3: View and Select Only Wall Panels

After viewing the selected elements only (wall panels), select all elements in 3D view.



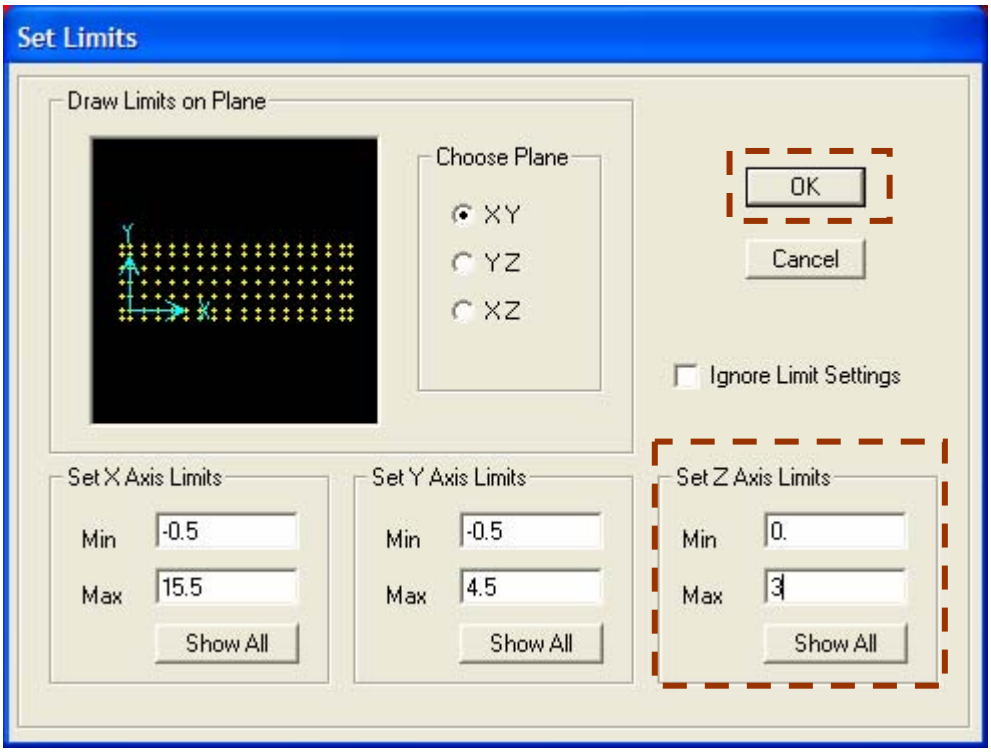
Step 9-4: Assign Joint Pattern ("SOIL")

Go to **Assign >> Joint Patterns** and specify parameters as shown in figure below. Afterwards, click "OK".



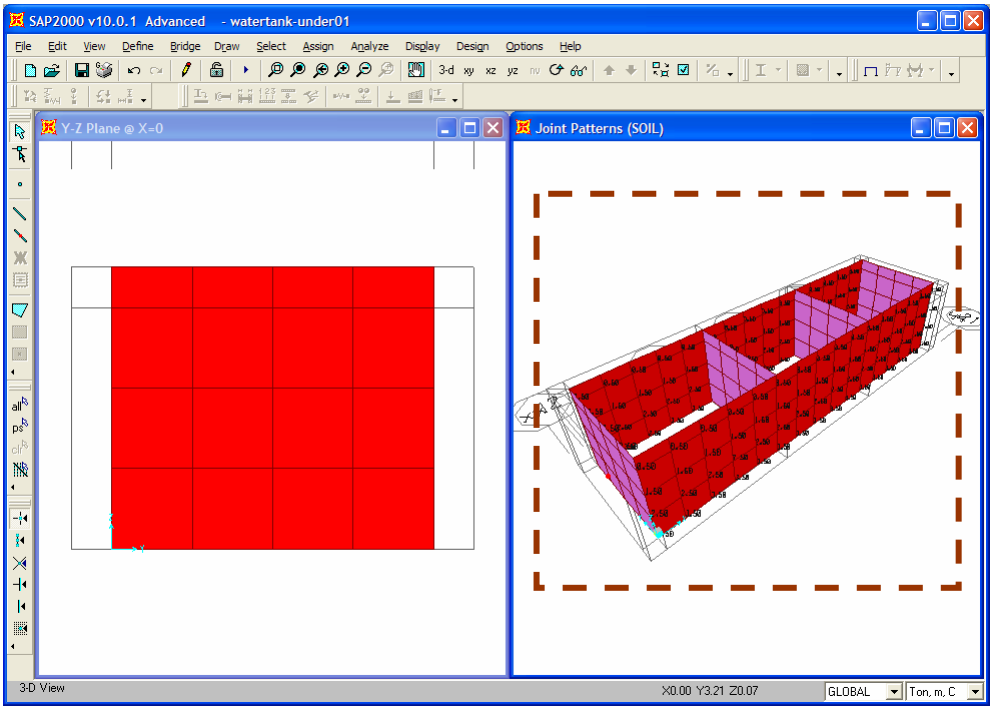
Step 9-5: Set View Limit

Go to **View >> Set Limit** and enter “Set Z Axis Limits” as shown in figure below. Afterwards, click “OK”.



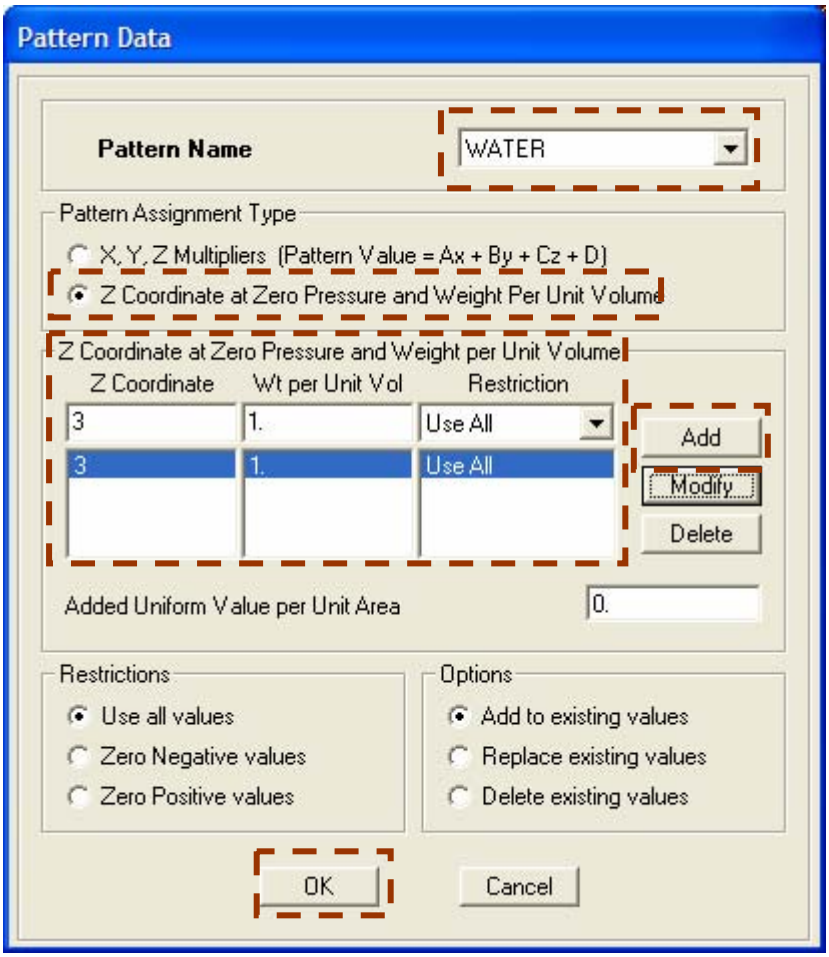
Step 9-6: Select Wall Panels

Select all shell elements in 3D view.



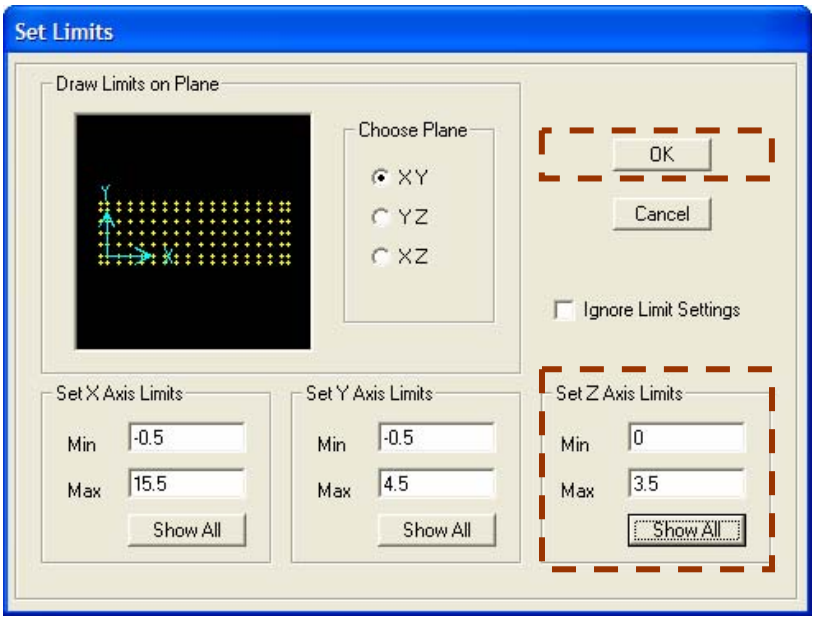
Step 9-7: Assign Joint Pattern ("SOIL")

Go to **Assign >> Joint Patterns** and specify parameters as shown in figure below.



Step 9-8: Clear Z Axis View Limit and Show All Elements

Specify parameters as shown in figure below and go to **View >> Show All**.



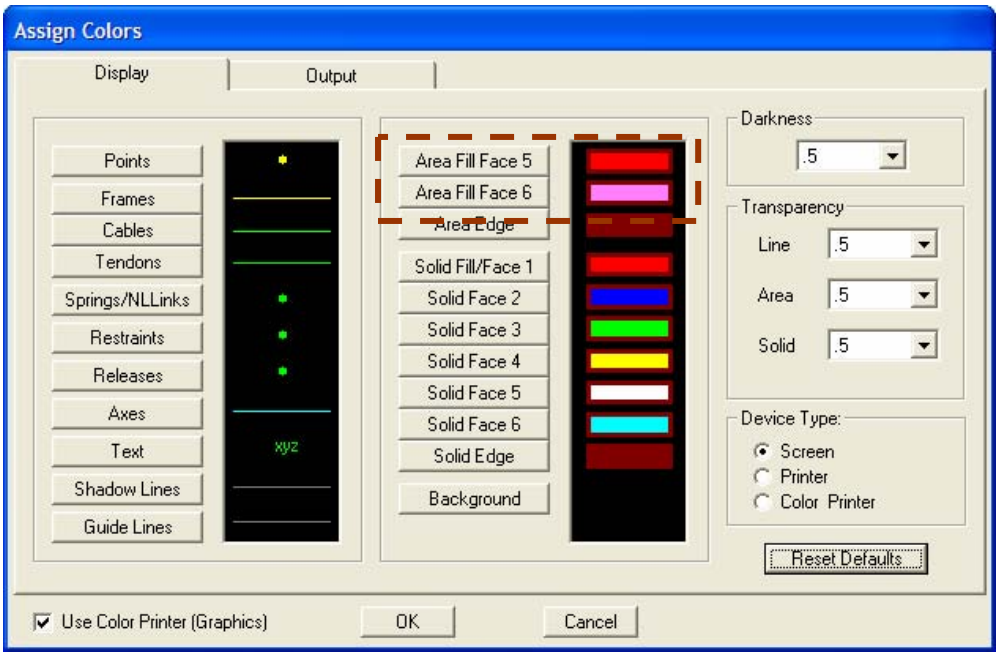
10. Assign Soil and Water Pressure on Wall Panels

Pressure loads are assigned as triangular load using joint pattern. The direction of pressure depends on side of shell element (Positive pressures are directed toward the interior of the element.). In SAP2000, 2 sides of shell element (top and bottom) are represented as face 5 and face 6. Face 5 of shell element is the positive side of local axis 3, shown in blue color arrow. Local axes 1, 2, and 3 are marked with red, white and blue. Water pressures are assigned in the load cases separately for each tank (TANK1, TANK2 and TANK3).

Note:

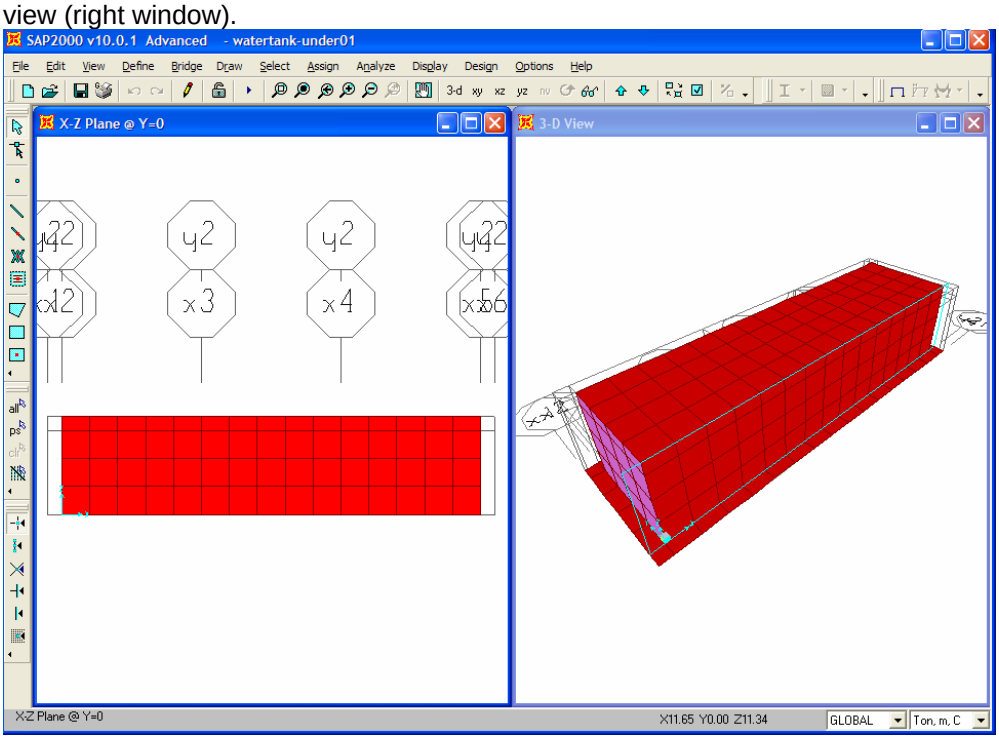
To view local axis of shell element: Go to **View >> Set Display Options** and or click on *Set Display Options* button and select "Local Axes" in Areas

To set color for face of shell element (Area): Go to **Options >> Color >> Display**



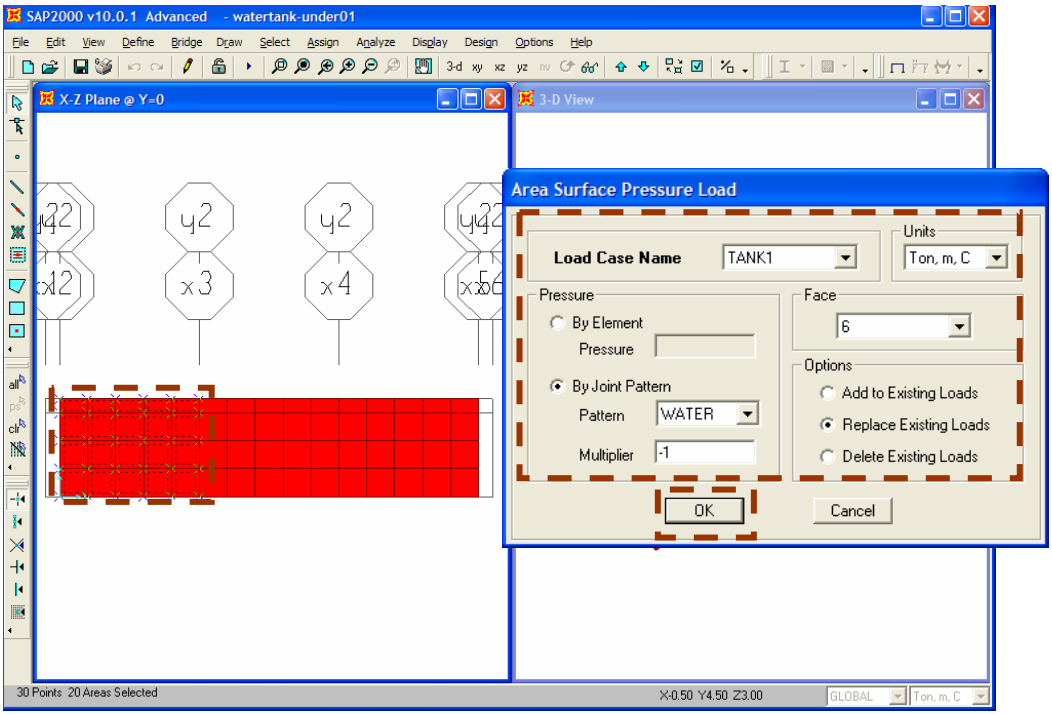
Step 10-1: Change to in X-Z Plane at Y = 0.00

Select left window, click on *Set XZ View* button and move elevation view to Y = 0 by clicking or . The location of this elevation is displayed in blue outline in 3D view (right window).



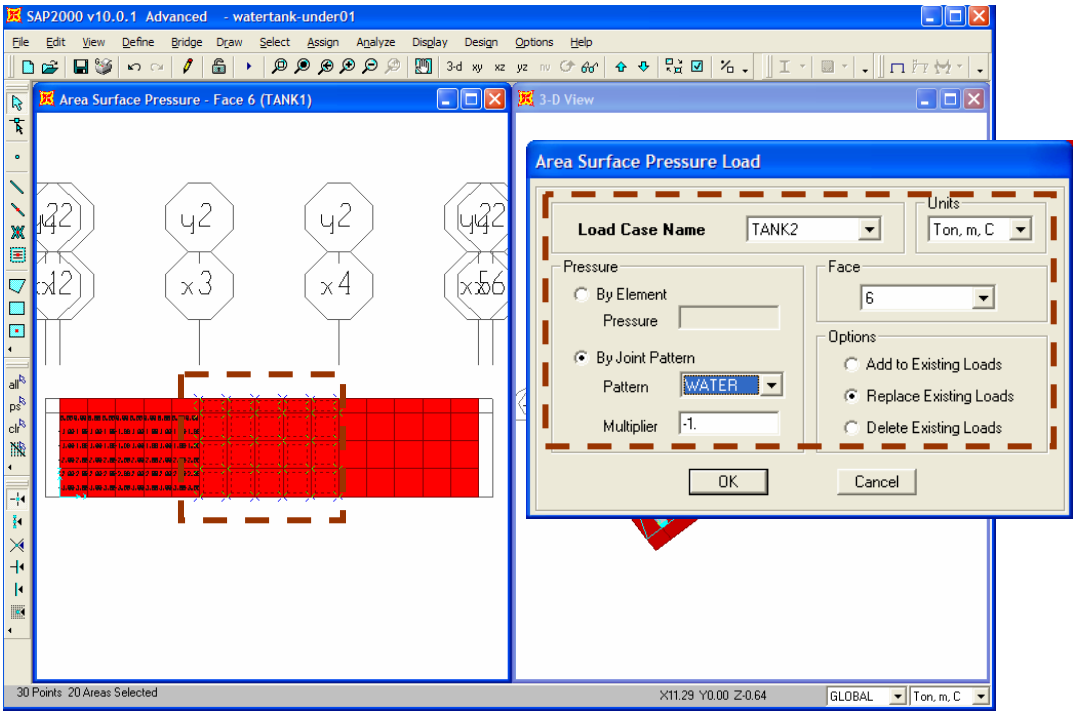
Step 10-2: Assign Water Pressure Load to Tank 1 Wall in XZ Plan at Y = 4.00

Select wall panels in tank 1, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below.



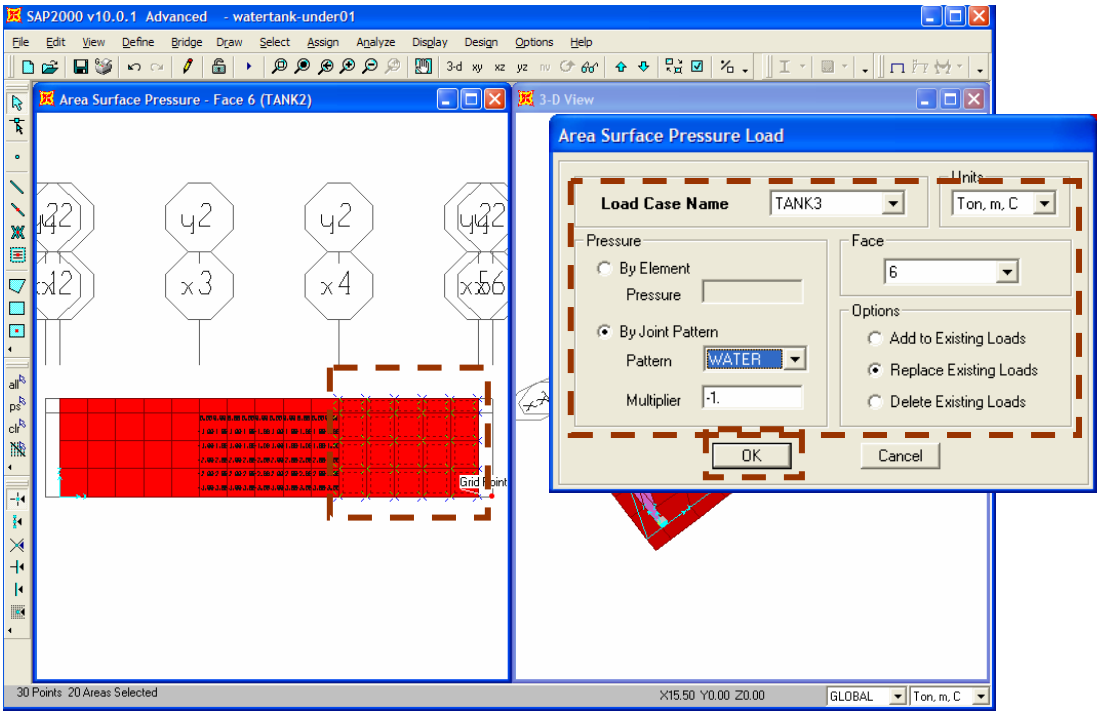
Step 10-3: Assign Water Pressure Load to Tank 2 Wall in XZ Plan at Y = 4.00

Select wall panels in tank 2, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below.



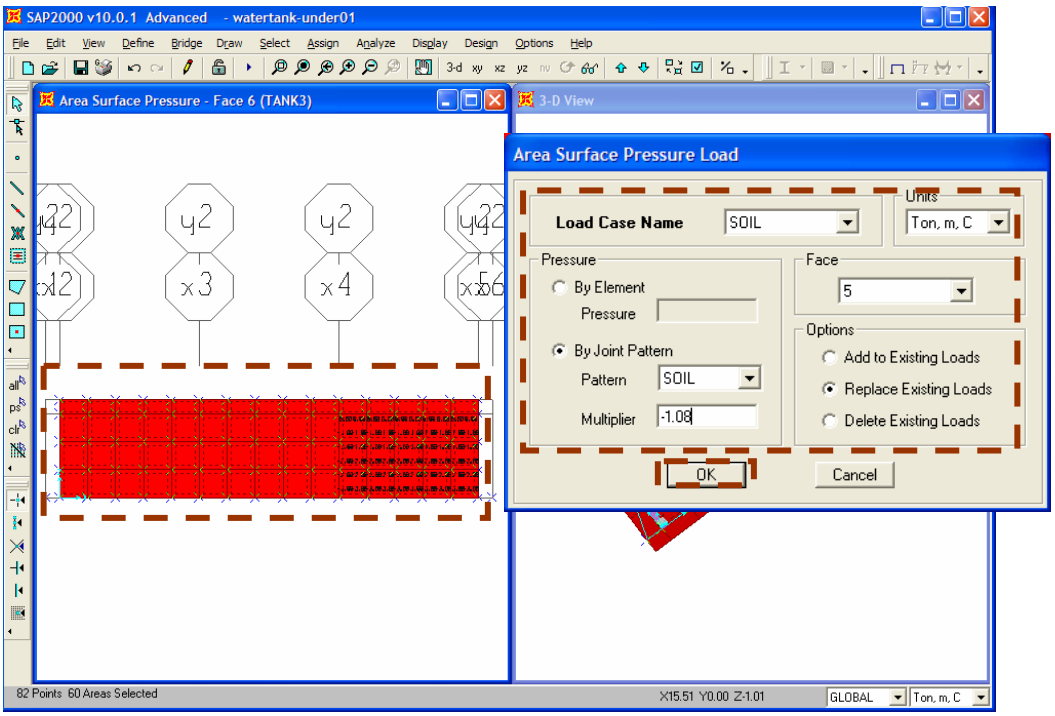
Step 10-4: Assign Water Pressure Load to Tank 3 Wall in XZ Plan at Y = 4.00

Select wall panels in tank 3, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below.



Step 10-5: Assign Soil Pressure to All Tank Wall in XZ Plan at Y = 4.00

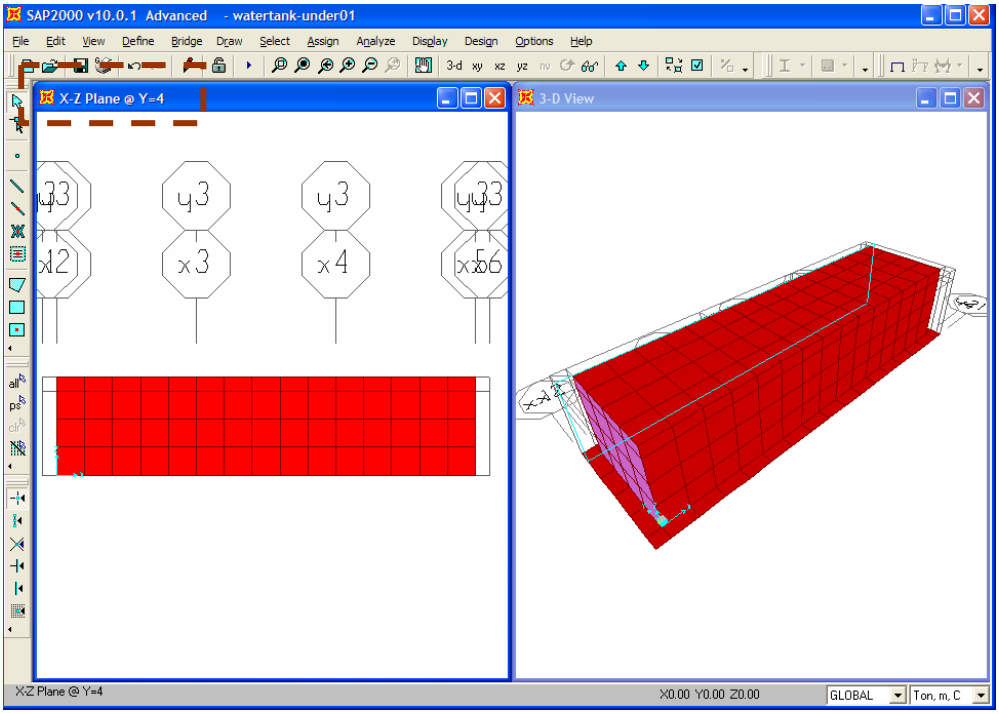
Soil pressure will be assigned to opposite face of water pressure (face 5). Select wall panels in all tanks, go to **Assign >> Area Load >> Surface Pressure (All)**, select "Add to Existing Loads" options and specify pressure parameters as shown in figure below (Multiplier = Ka x unit weight of soil = 0.6 x 1.8 = 1.08).



Note: For more details on coefficient of active lateral earth pressure (K_a), please refer the standard book on geotechnical engineering (for example: page 478 of "Foundation Analysis and Design" by Joseph E. Bowles)

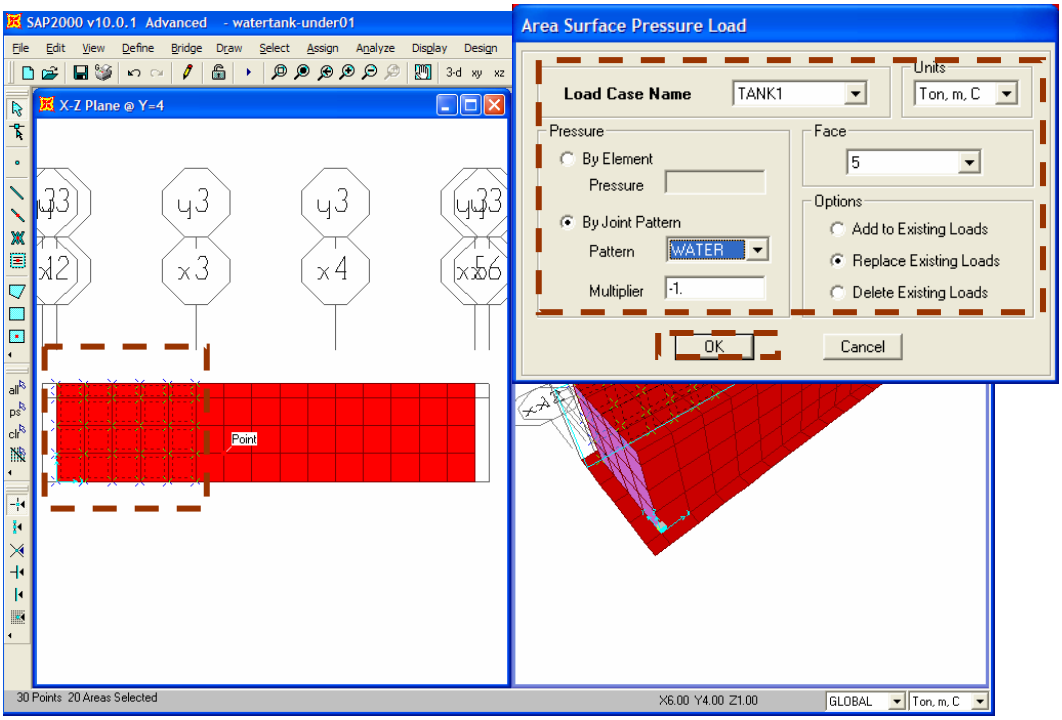
Step 10-6: Change View to XZ Plan to Y = 4.00

Move elevation view to Y = 4.00 by clicking  or  . In this elevation view, face 5 of wall panels faces outside of water tank then water pressure load will be assigned at face 6 (water tank) and soil pressure will be assigned at face 5 (outside tank) by using same parameters as assigned at elevation Y = 4.00.



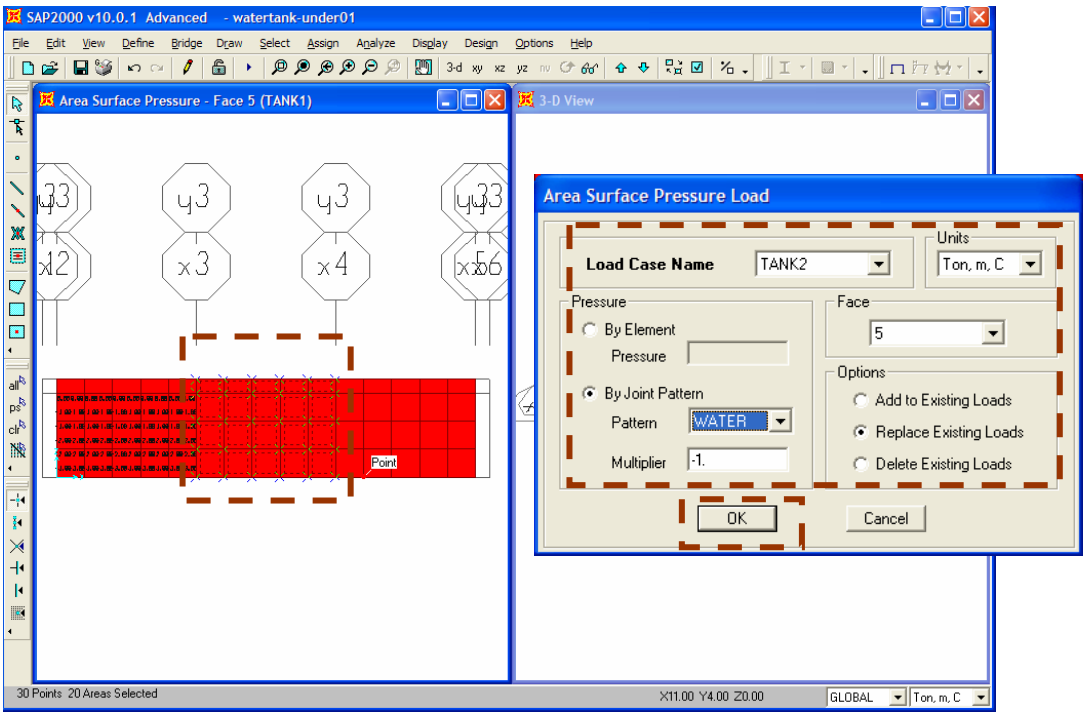
Step 10-7: Assign Water Pressure Load to Tank 1 Wall in XZ Plan at Y = 0.00

Select wall panels in tank 1, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below.



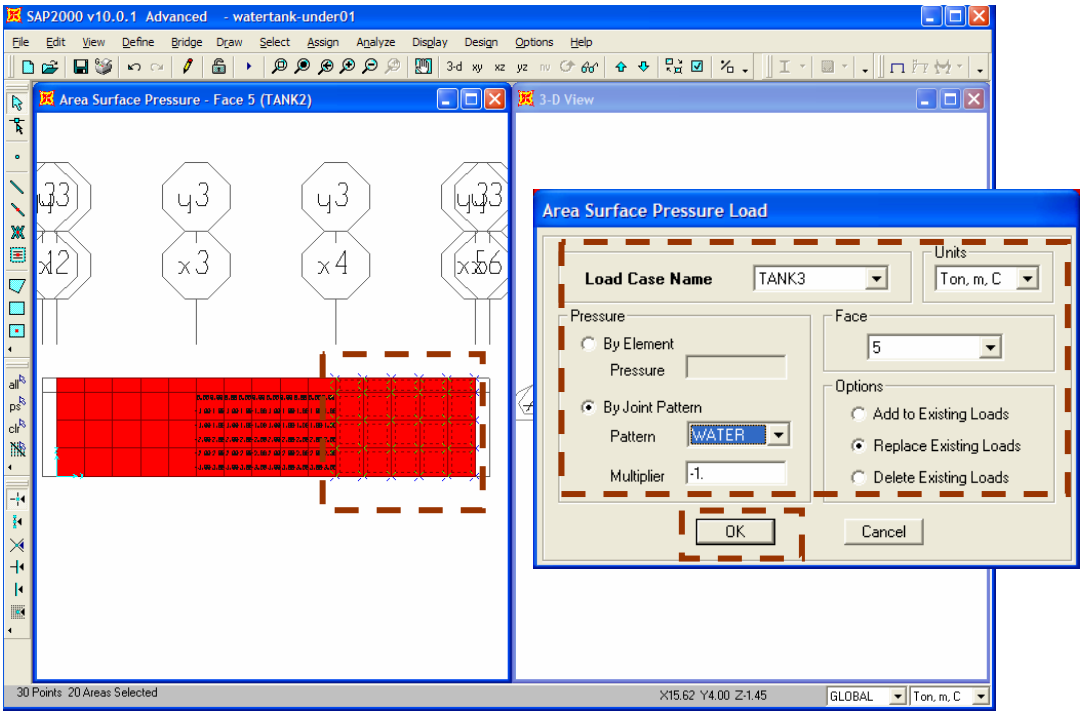
Step 10-8: Assign Water Pressure Load to Tank 2 Wall in XZ Plan at Y = 0.00

Select wall panels in tank 2, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below.



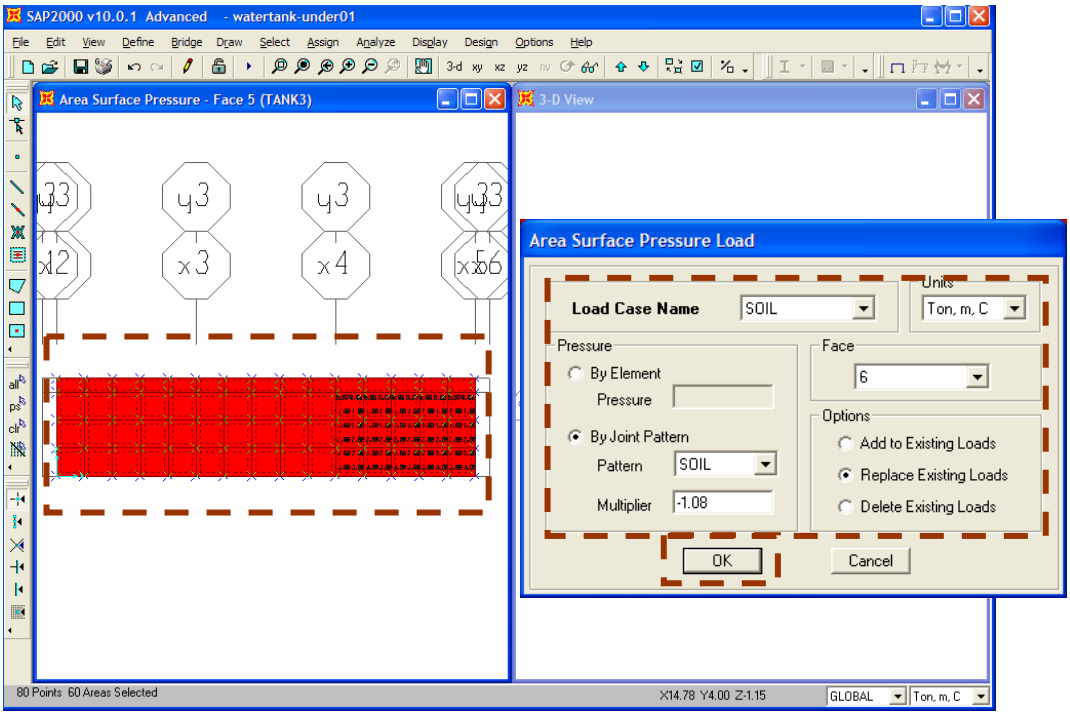
Step 10-9: Assign Water Pressure Load to Tank 3 Wall in XZ Plan at Y = 0.00

Select wall panels in tank 3, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below.



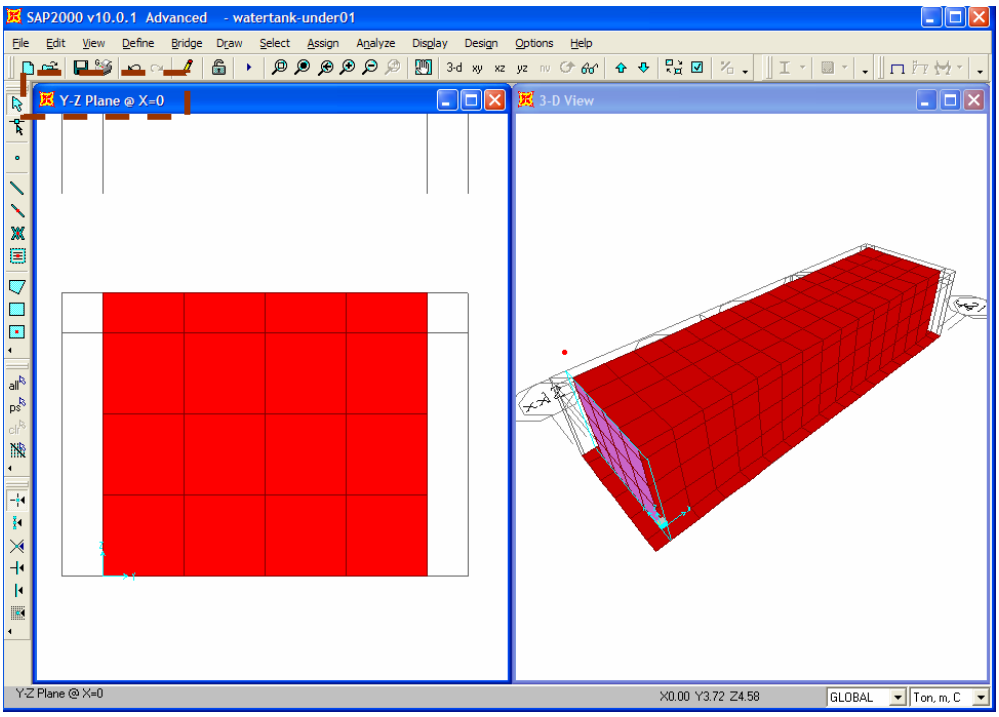
Step 10-10: Assign Soil Pressure to All Tank Wall in XZ Plan at Y = 0.00

Soil pressure will be assigned to opposite face of water pressure (face 5). Select wall panels in all tanks, go to **Assign >> Area Load >> Surface Pressure (All)**, select "Add to Existing Loads" options and specify pressure parameters as shown in figure below (Multiplier = Ka x unit weight of soil = 0.6 x 1.8 = 1.08).



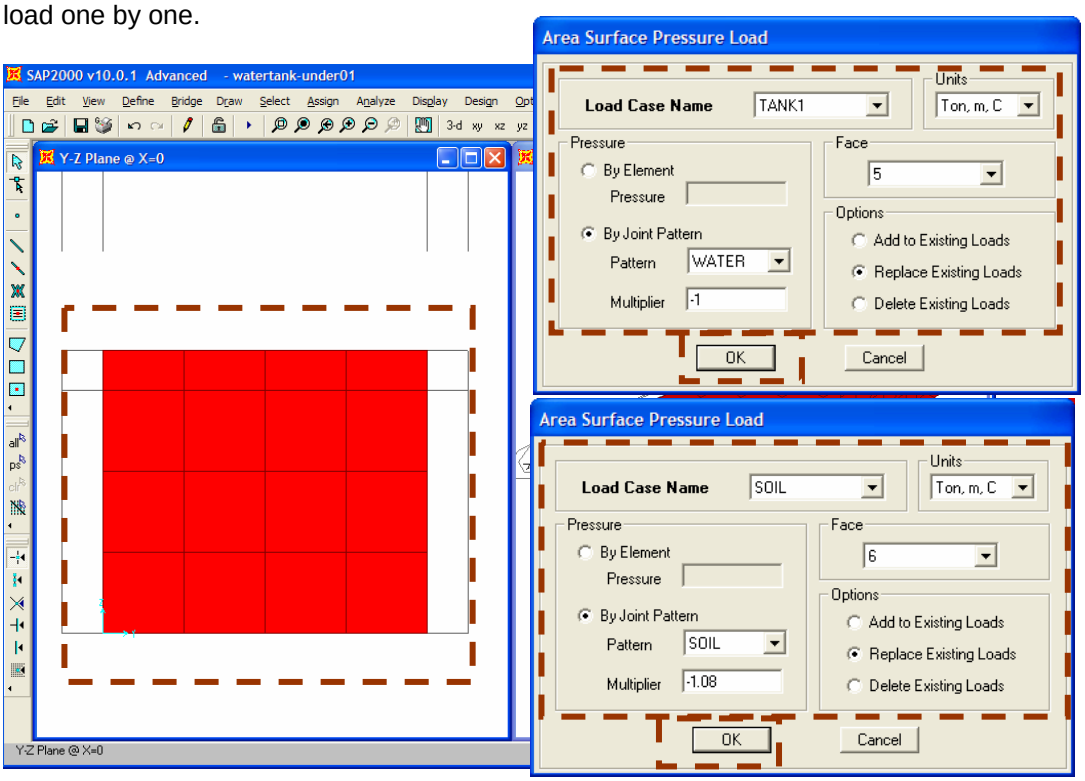
Step 10-11: Change to in YZ Plan at X = 0

Select left window, click on yz and move elevation view to X= 0 by clicking ↑ or ↓. In this elevation view, shows face 5 of wall panels is inside tank then water pressure will be assigned in face 5 and soil pressure will be assigned in face 6.



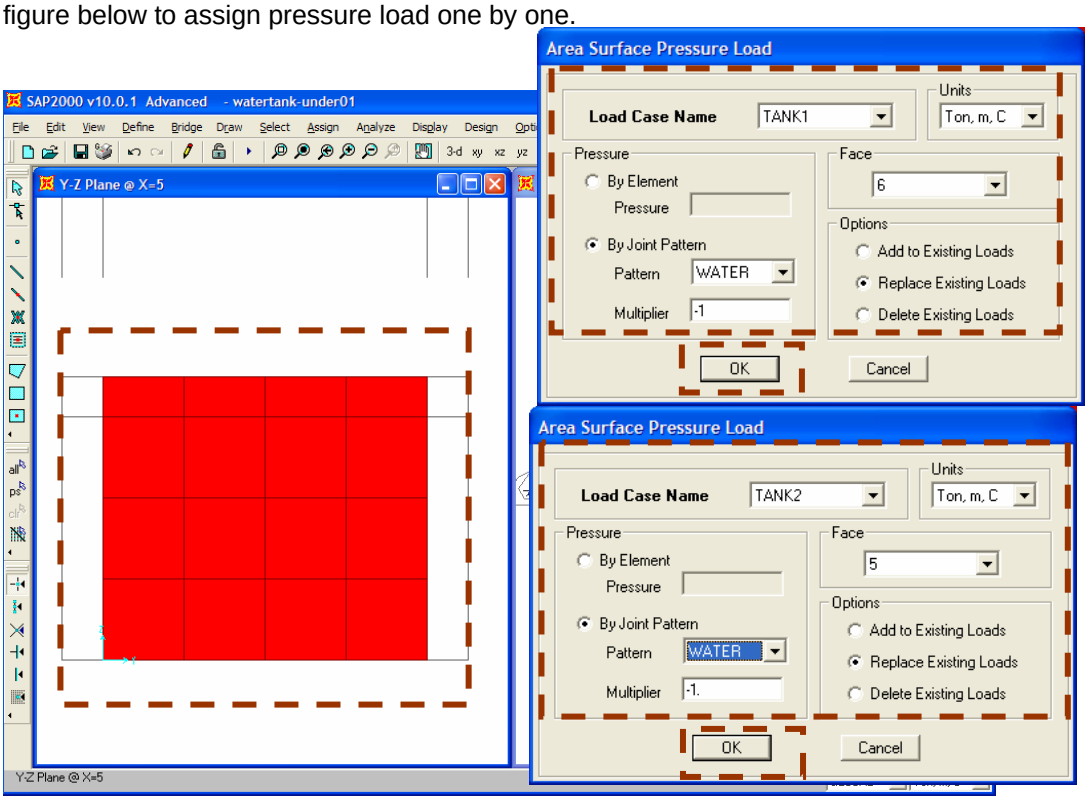
Step 10-12: Assign Water and Soil Pressure to Tank 1 Wall in XZ Plan at X = 0

Select wall panels in tank 1, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below to assign pressure load one by one.



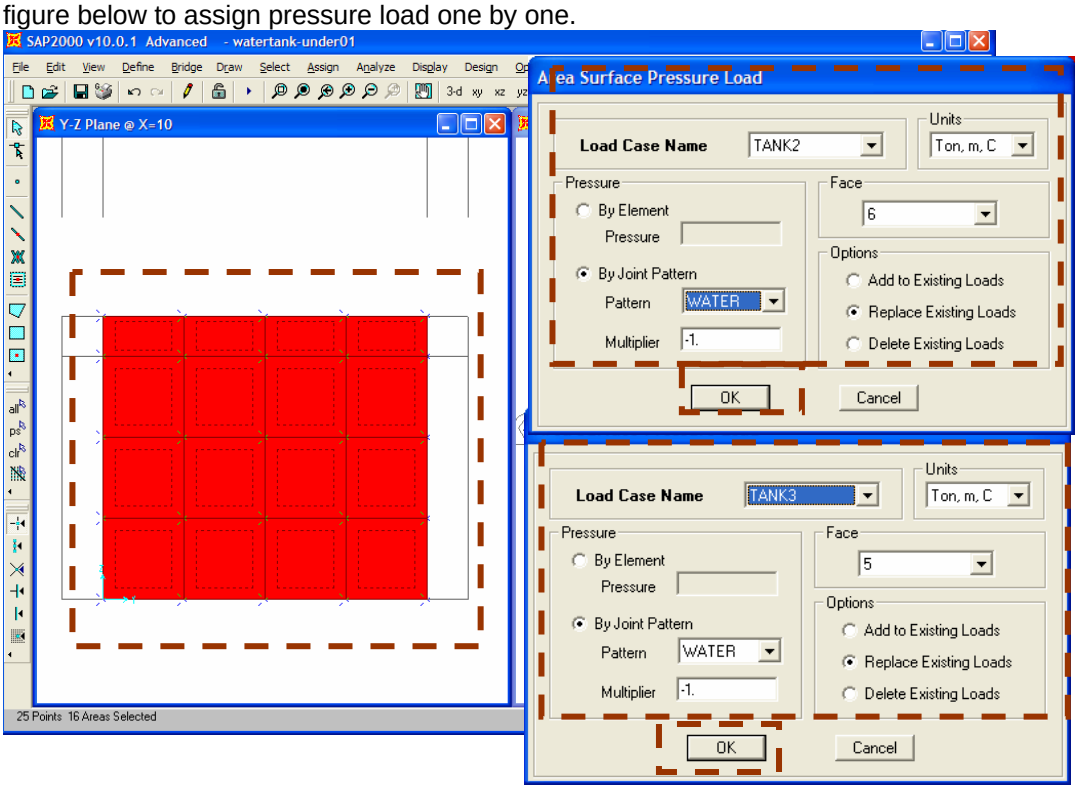
Step 10-13: Assign Water Pressure for Tank 1 and 2 in XZ Plan at X = 5

Click on  to move elevation view, select all wall panels, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below to assign pressure load one by one.



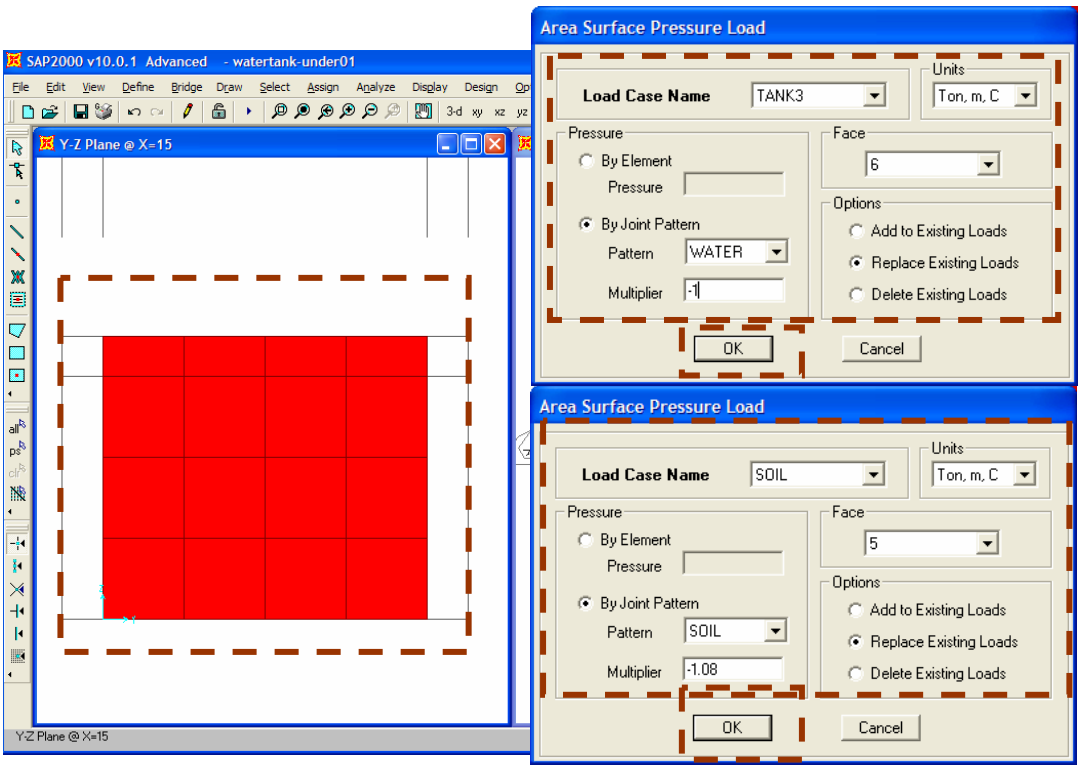
Step 10-14: Assign Water Pressure for Tank 2 and 3 in XZ Plan at X = 10

Click on  to move elevation view, select all wall panels, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below to assign pressure load one by one.



Step 10-15: Assign Water and Soil Pressure to Tank 3 Wall in XZ Plan at X = 15

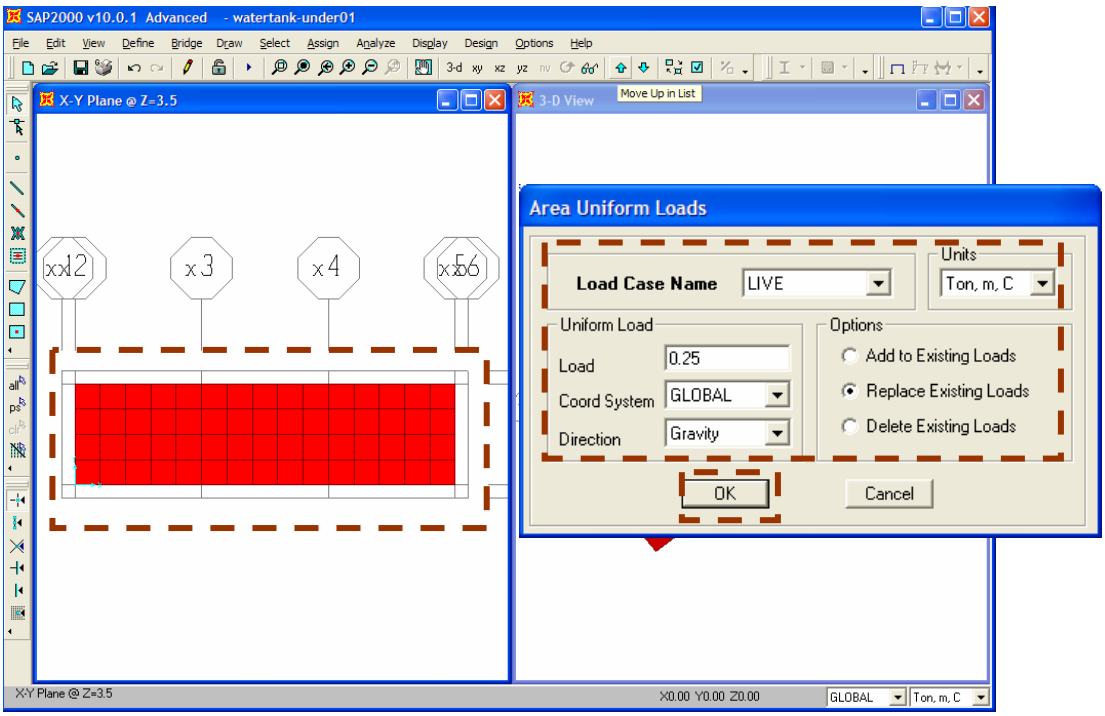
Click on  to move elevation view, select all wall panels, go to **Assign >> Area Load >> Surface Pressure (All)** and specify pressure parameters as shown in figure below to assign pressure load one by one.



11. Assign Load on Top and Bottom Slabs

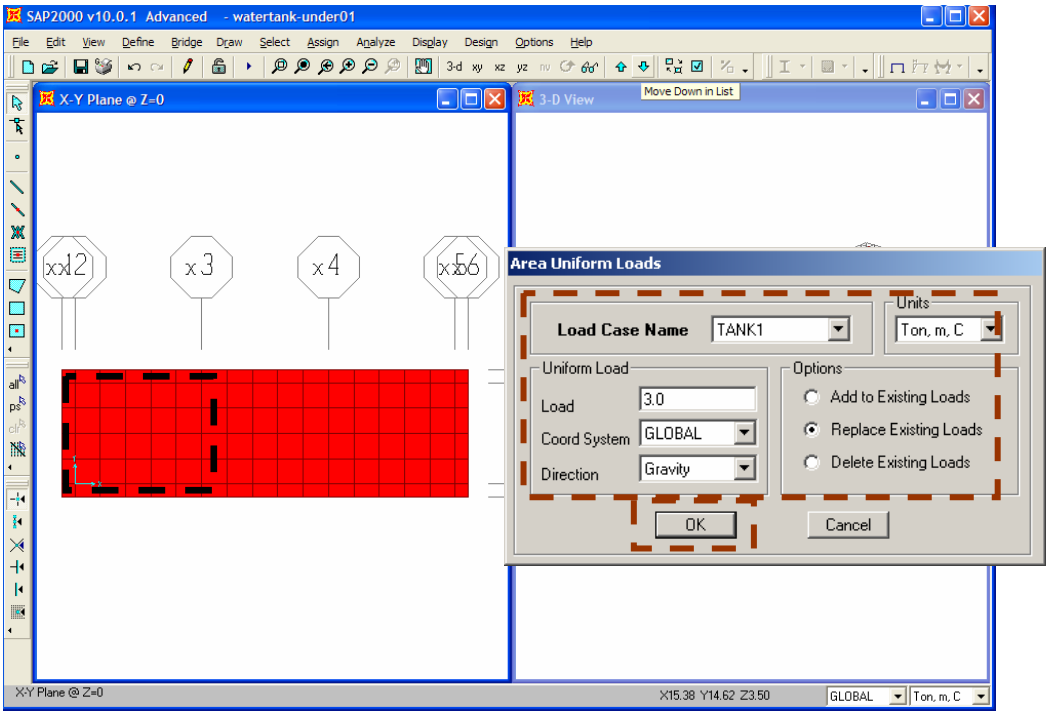
Step 11-1: Assign Live Load on Top Slabs (Z = 3.5)

Change view to XY plan at Z = 3.5 by clicking on *Set XY View* button and move plan level to Z = 3.50 by clicking on . Select all slab panels, go to **Assign >> Area Loads >> Uniform (Shell)** and specify parameters as shown in figure below.



Step 11-2: Assign Water Pressure in Bottom Slabs at Tank 1 (Z = 0.0)

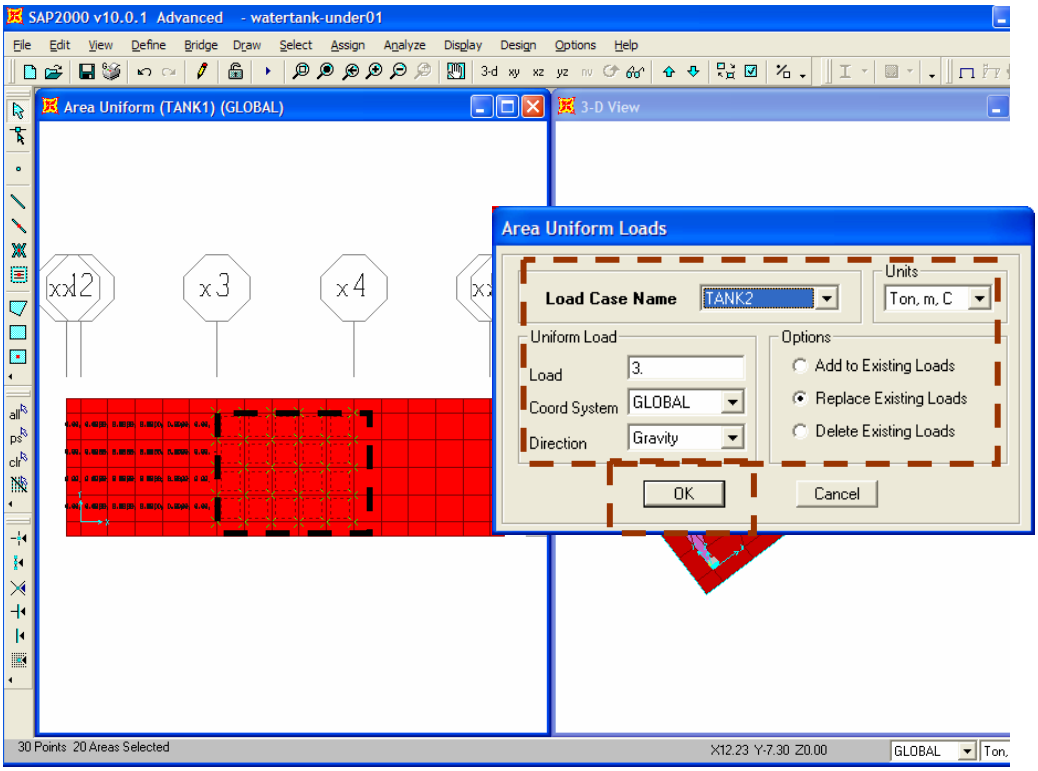
Click on to Z = 0, Select bottom slabs of tank 1, go to **Assign >> Area Loads >> Uniform (Shell)** and specify parameters as shown in figure below.



Note: To do slab selection easier, deactivate snap option by deselect *Points and Grid Intersections* button

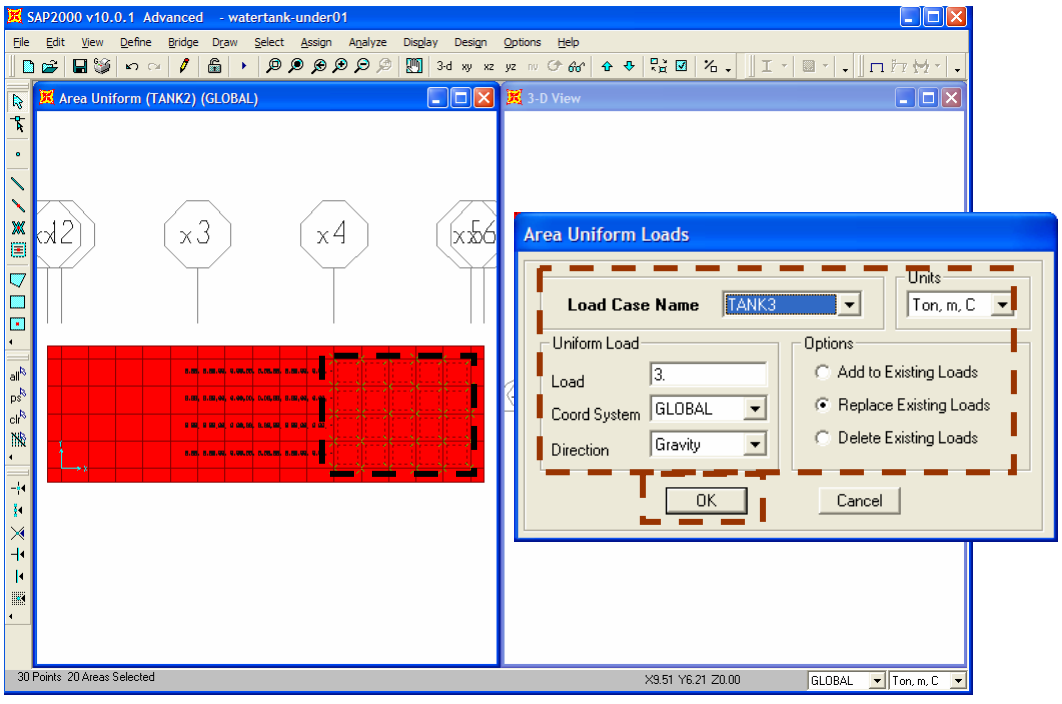
Step 11-3: Assign Water Pressure in Bottom Slabs at Tank 2 (Z = 0.0)

Select bottom slabs of tank 2, go to **Assign >> Area Loads >> Assign >> Area Loads >> Uniform (Shell)** and specify parameters as shown in figure below.



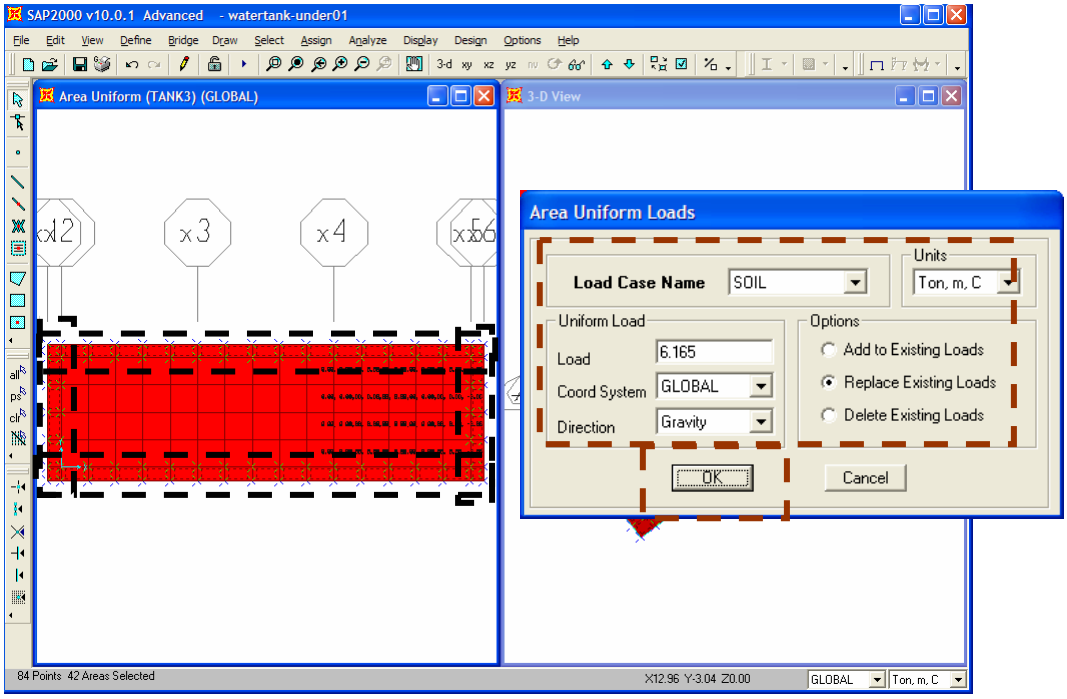
Step 11-4: Assign Water Pressure in Bottom Slabs at Tank 3 (Z = 0.0)

Select bottom slabs of tank 3, go to **Assign >> Area Loads >> Uniform (Shell)** and specify parameters as shown in figure below.



Step 11-5: Assign Soil Pressure on Cantilever Slabs (Z = 0.0)

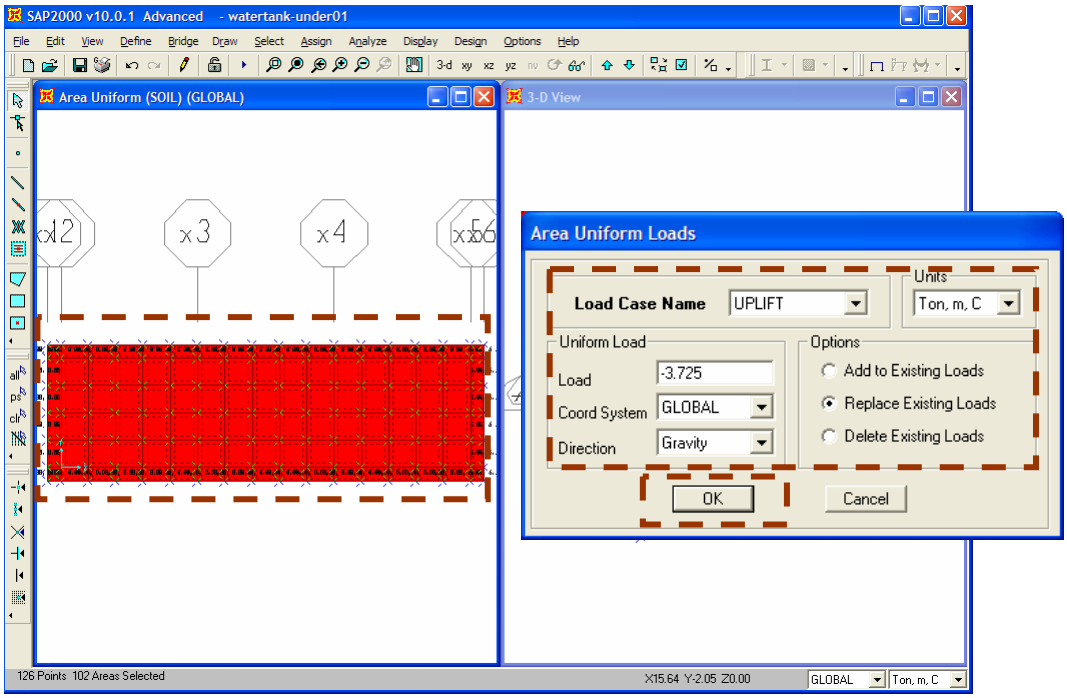
Select cantilever slabs, go to **Assign >> Area Loads >> Uniform (Shell)** and specify parameters as shown in figure below.



Note: Soil Pressure = (3.50 – 0.30/2 + 0.15/2) x 1.8 = 6.165 ton/m^2

Step 11-6: Assign Uplift Pressure in Bottom Slabs (Z = 0.0)

Select all slabs, go to **Assign >> Area Loads >> Uniform** and specify parameters as shown in above figure. Negative value is assigned for “Gravity” direction.



Note: Soil Pressure = 3.50 + 0.30/2 + 0.15/2 = 3.725 ton/m^2

12. Assign Spring Support

Vertical Spring (Z Direction)

Vertical spring support can be calculated from length and cross section of pile as shown in following details.

Area (A)	= 455	cm^2
Length (L)	= 20	m
Modulus of Concrete (E_c)	= 253,150	kg/cm^2
K (Vertical) = 2 x E x A / L	= 2 x 253,150 x 455 / 2000	
	= 115,183	kg/cm

Horizontal Spring (X and Y Direction)

Horizontal spring support will be assigned as line spring using dummy frame element (30 x 30 cm) and can be calculated from contact area between soil and foundation as shown in following detail

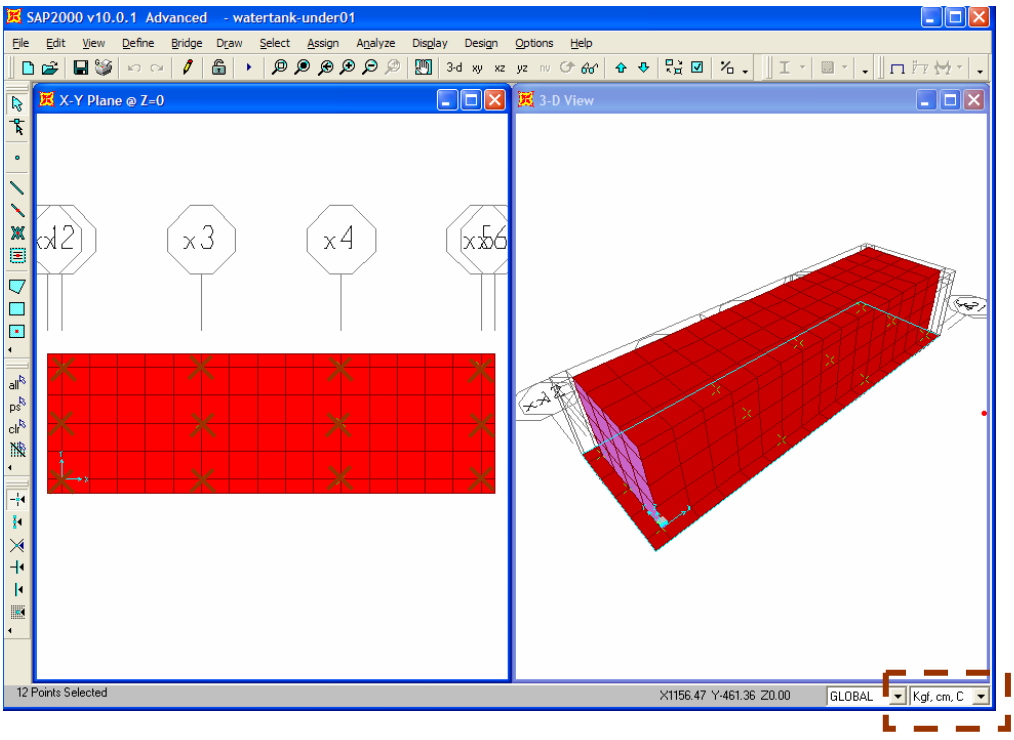
Foundation Depth (D)	= 0.30	m
Modulus of Subgrade Reaction (K_S)	= 500	$T/m^2/m$
K (Horizontal) = D x K_S	= 500 x 0.3	
	= 150	T/m/m

Note: 1. The soil restraints on side walls have not been included. If the lateral soil restraint and water pressure are applied together the moment on the side walls will be less than when tank has water without back fill.

2. For more details on modulus of subgrade reaction (K_S), please refer the standard book on geotechnical engineering (for example: page 405 of "Foundation Analysis and Design" by Joseph E. Bowles)

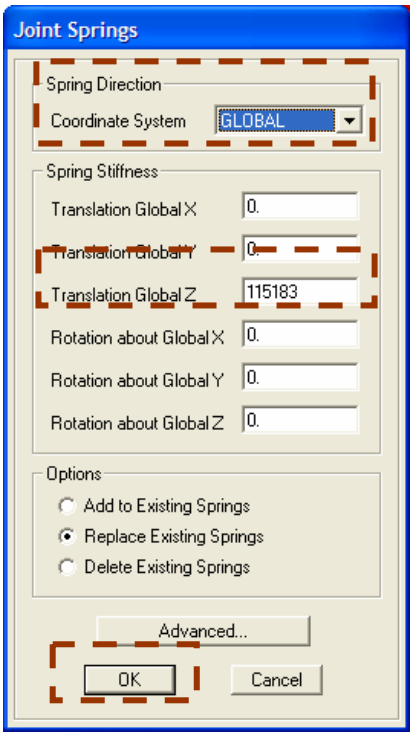
Step 12-1: Select Nodes at Pile Location to Assign Vertical Springs

Select working unit = "kg, cm", activate snap option by clicking on *Points and Grid Intersections* button and select support nodes by clicking on nodes at pile location as shown in above figure.



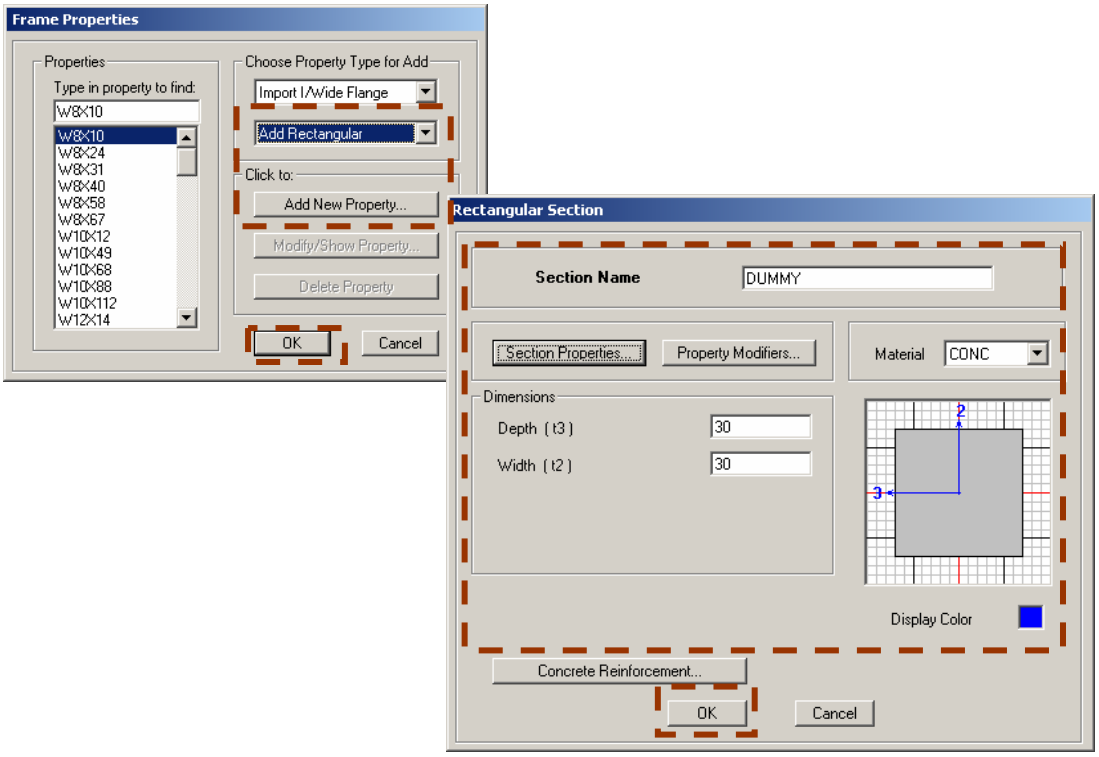
Step 12-2: Assign Vertical Springs

Go to **Assign >> Joint >> Springs** and specify parameters as shown in above figure.



Step 12-3: Add "DUMMY" Frame Section

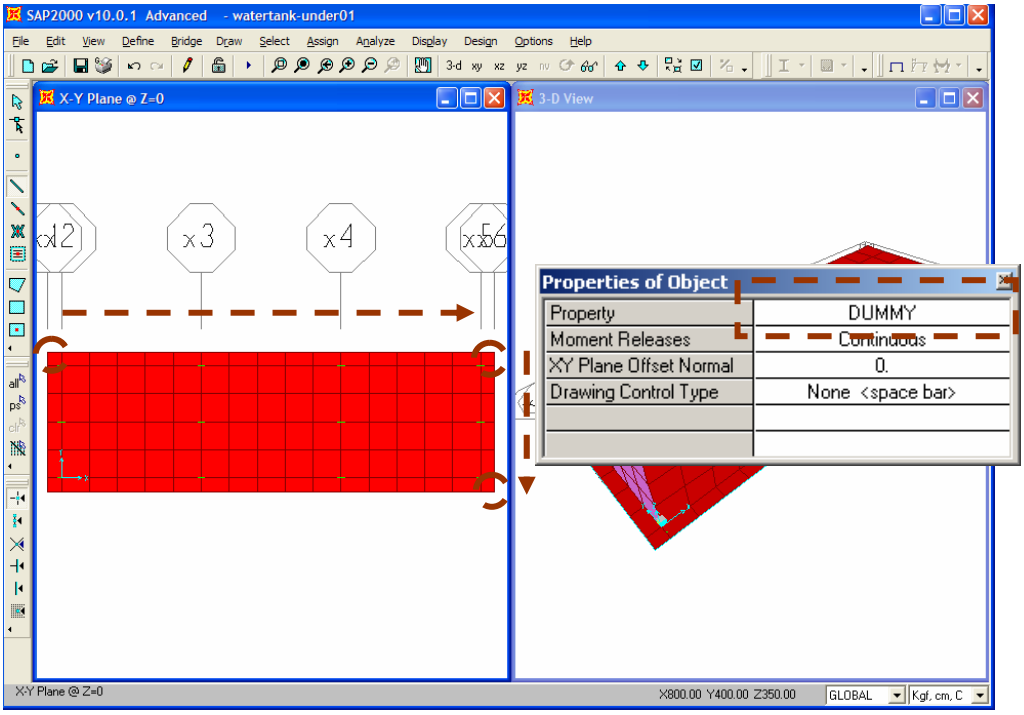
Go to **Define >> Frame Sections...**, select "Add Rectangular" from second drop-down menu. Click "Add New Property..." and specify parameters as shown in figure below.



Note: The beam does not exist in the tank in reality. The beam is defined only for the propose of applying lateral soil restraints. The dimensions of this beam are assumed based on the thickness of the base slab (30 cm).

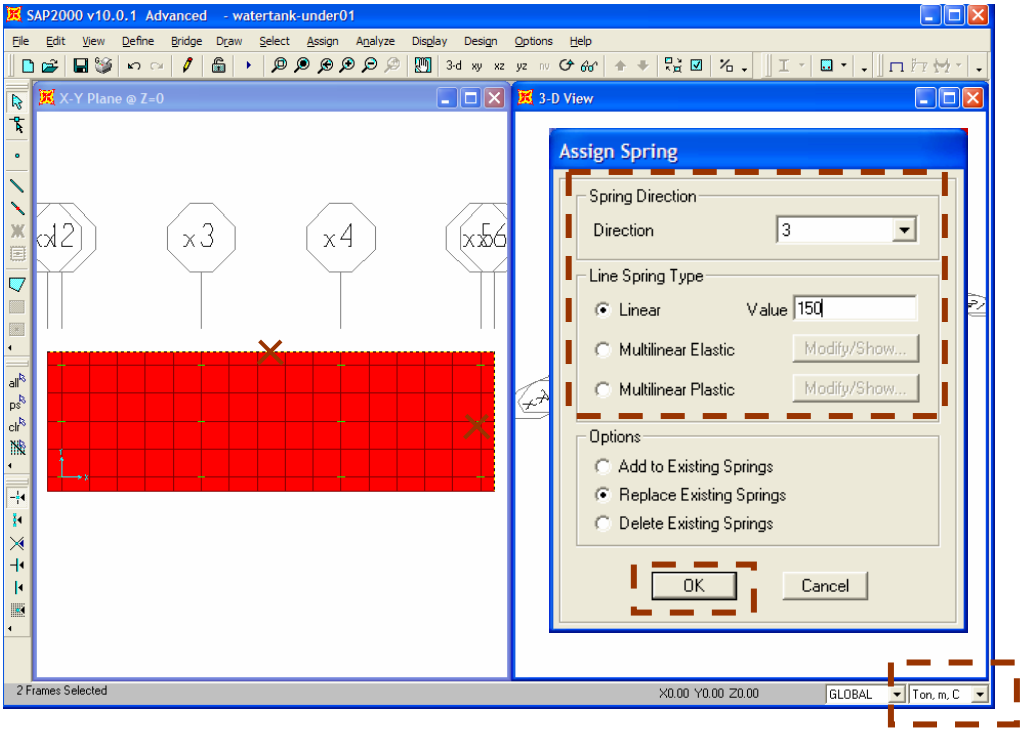
Step 12-4: Draw "DUMMY" Frame Section

Click on *Draw Frame/Cable Element* button, select "DUMMY" and draw frame section as shown in figure below.



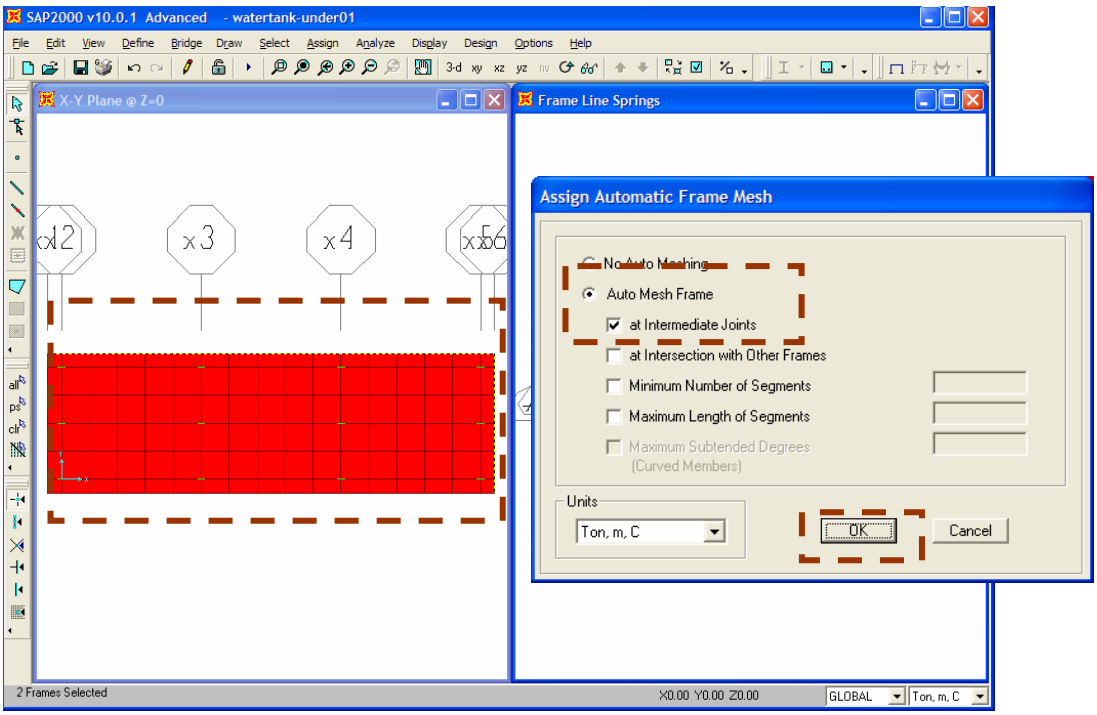
Step 12-5: Assign Horizontal Springs

Change working unit to "Ton, m", click on *Set Select Mode* button, select "DUMMY" frames as shown in above figure, go to **Assign >> Frame/Cable/Tendon >> Line Springs** and specify parameter as shown in figure below.



Step 12-6: Apply Automatic Frame Mesh to "DUMMY" Frames

Select all "DUMMY" frames by clicking on *Get Previous Selection* button, go to **Assign >> Frame/Cable/Tendon >> Automatic Frame Mesh** and specify parameters as shown in figure below.



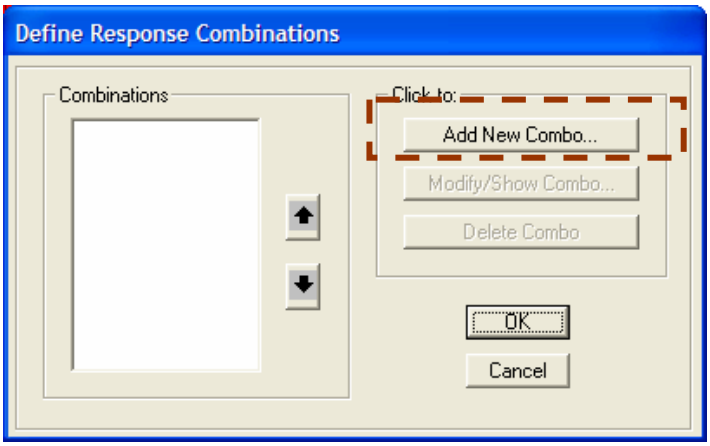
13. Define Load Combinations

Envelop option is used in load combination ("WATER") to find maximum and minimum results among of all possible full or empty water in each tank. Load combination details (without sanitary durability coefficient) are summarized in following table.

Load Combination Name	Combination Type	Combination Details
TANK120	Linear Add	TANK1 + TANK2
TANK023	Linear Add	TANK2 + TANK3
TANK103	Linear Add	TANK1 + TANK3
TANK123	Linear Add	TANK1 + TANK2 + TANK3
WATER	Envelope	TANK1, TANK2, TANK3, TANK120, TANK023, TANK103, TANK123
DESIGN1	Linear Add	1.4 DEAD + 1.7 WATER
DESIGN2	Linear Add	1.2 DEAD + 1.7 WATER + 1.6 LIVE +1.7 SOIL
DESIGN3	Linear Add	1.2 DEAD + 1.7 WATER + 1.6 LIVE +1.7 SOIL + 1.7 UPLIFT
DESIGN4	Linear Add	1.2 DEAD + 1.0 LIVE
DESIGN5	Linear Add	0.9 DEAD + 1.7 SOIL + 1.7 UPLIFT

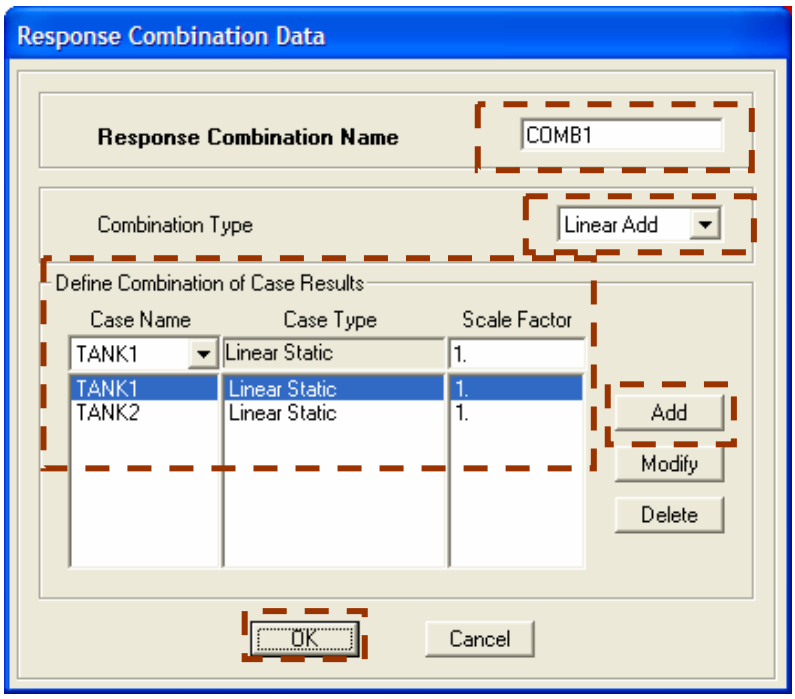
Step 13-1: Add New Load Combination

Go to **Define >> Combinations** and click on “Add New Combo”

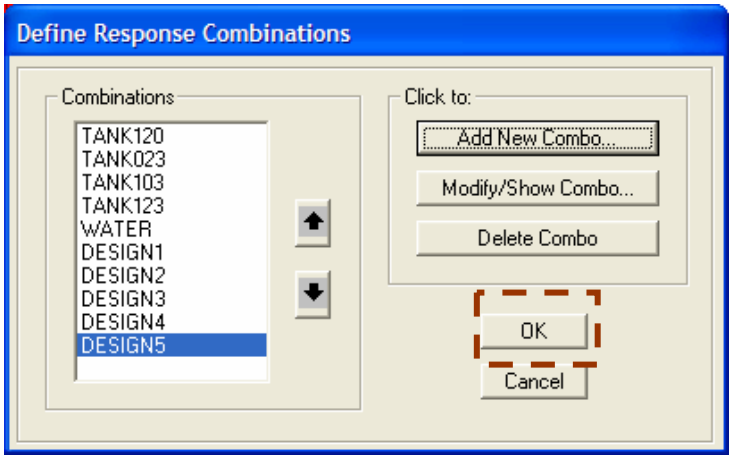


Step 13-2: Define Load Combination

Specify “Name”, select “Combination Type”, enter load and scale factor and click on “Add” to define load combination as shown in abovementioned table.



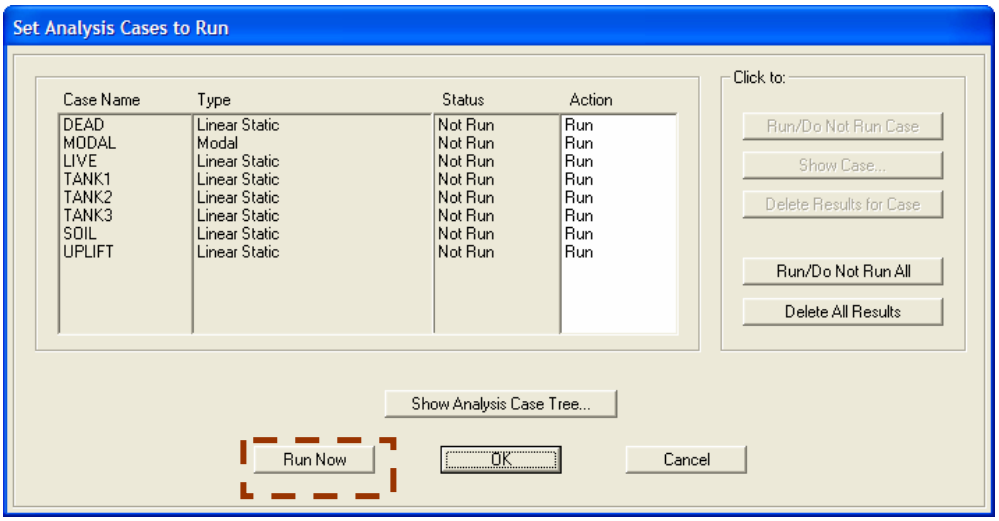
Repeat this step to enter load combination as shown in table from previous page.

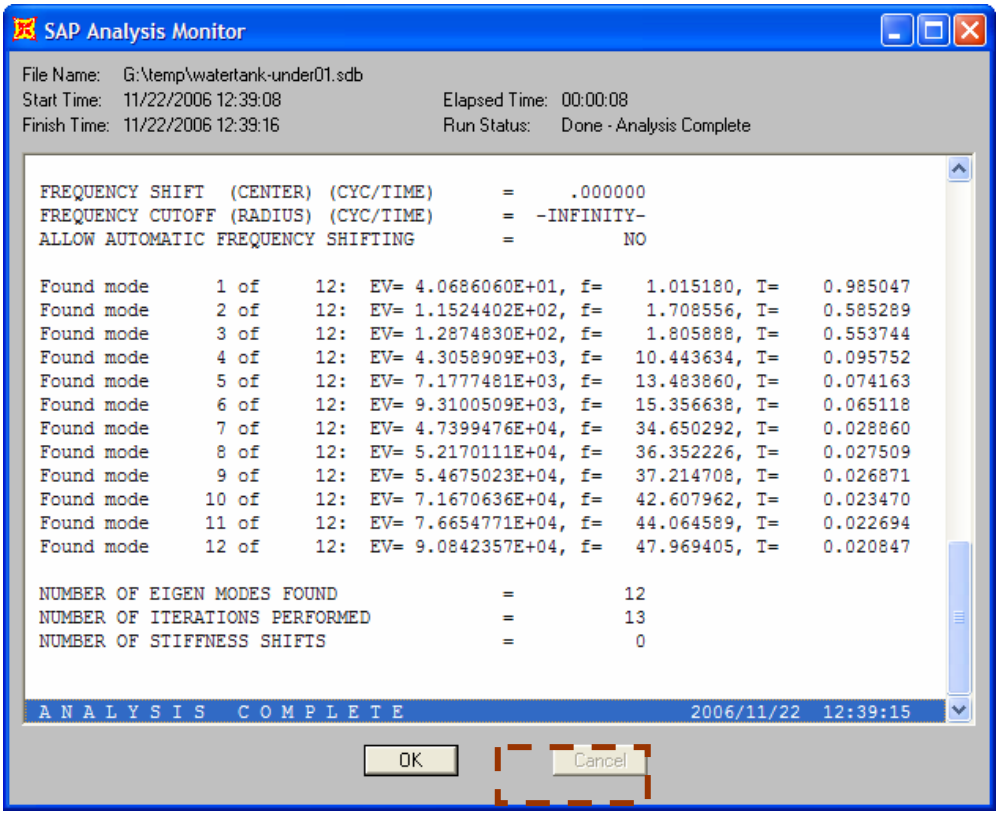


14. Run Analysis

Step 14-1: Start Analysis

Go to **Analyze > Run Analysis** or click on *Run Analysis* button and click on "Run Now" button to start analysis and wait until analysis process complete.





Step 14-2: Complete Analysis

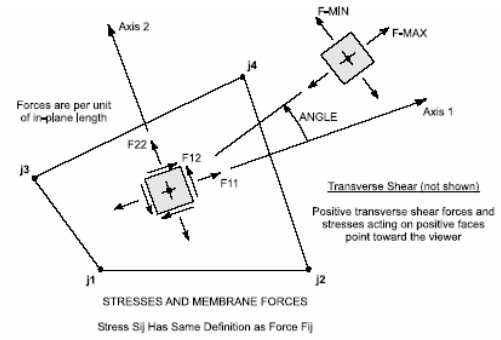
Wait until analysis status window display "ANALYSIS COMPLETE".

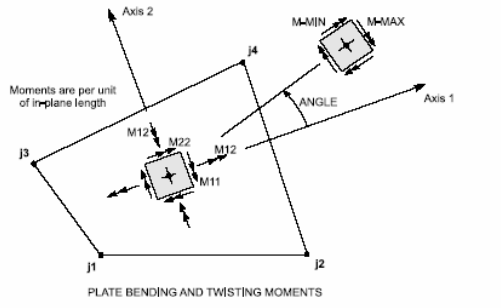
15. View Analysis Results

Forces and stresses in shell element for slabs, walls and footing can be displayed in color contour. To use these results for design, sanitary durability coefficients need to be applied as following details.

Reinforcement in Flexure	= 1.3 U
Direct Tension/Hoop Reinforcement	= 1.6 U
Excess Shear (for Stirrup)	= 1.3 U
Compression + Flexure	= 1.0 U

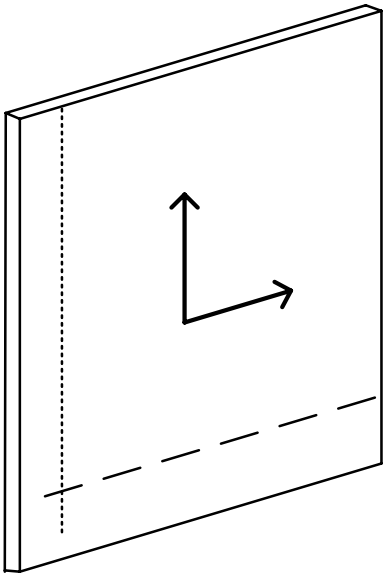
**U = Load Combination





The direction and sign convention of forces/ stresses in shell element are described in “SAP2000 Analysis Reference Manual” page 131 to page 151 that is available in PDF format (file name “SapRefer.pdf”) located at the “Manual” subfolder in folder that SAP2000 has been installed in your computer.

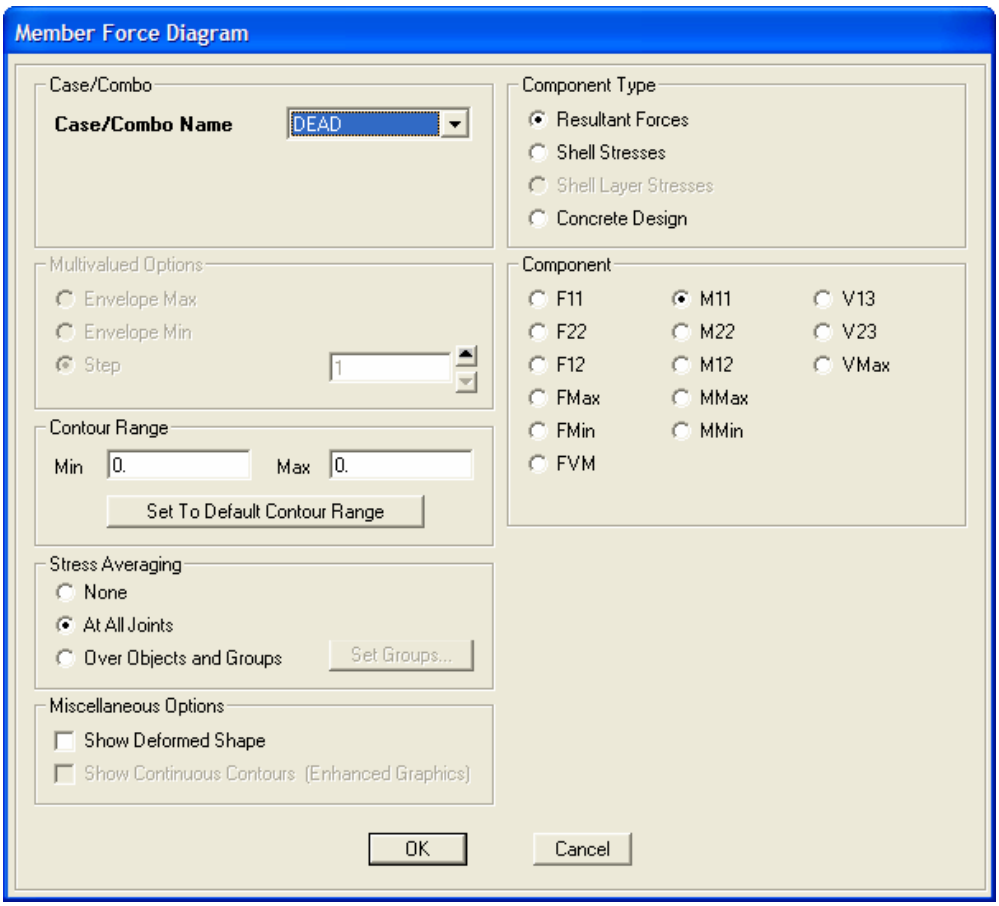
How to Compute Reinforcement from Finite Element Output



The direction of reinforcement for M11 and M 22 are as shown in left figure,

Step 15-1: Select Shell Analysis Results to View in Graphic

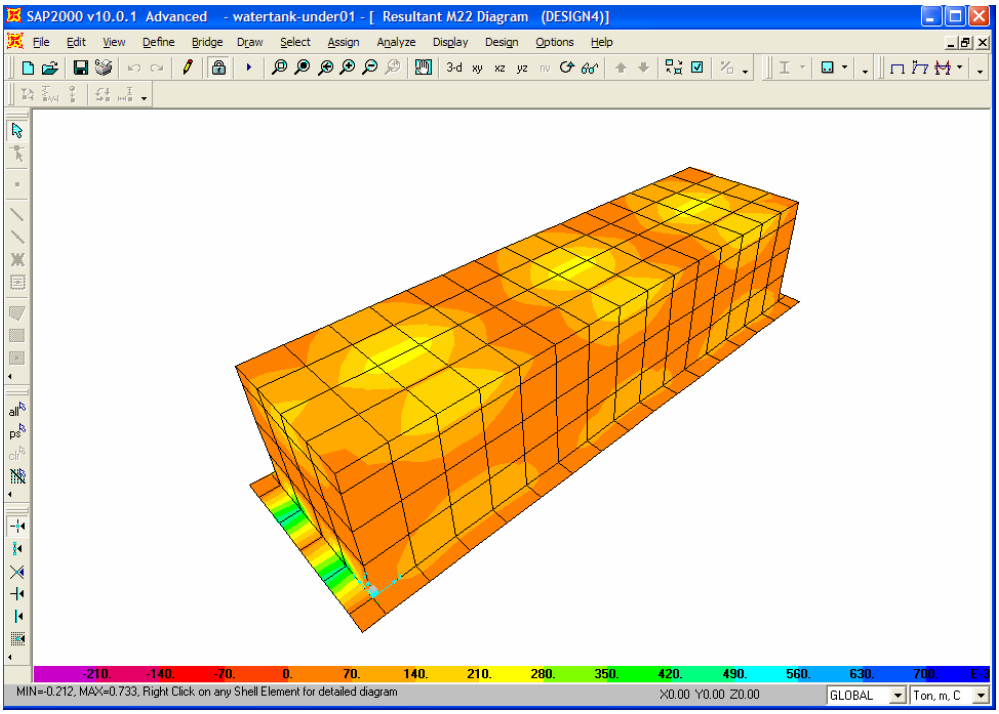
Go to **Display >> Forces/Stresses >> Shells** and select desired result parameters



Reinforcement Design for M22

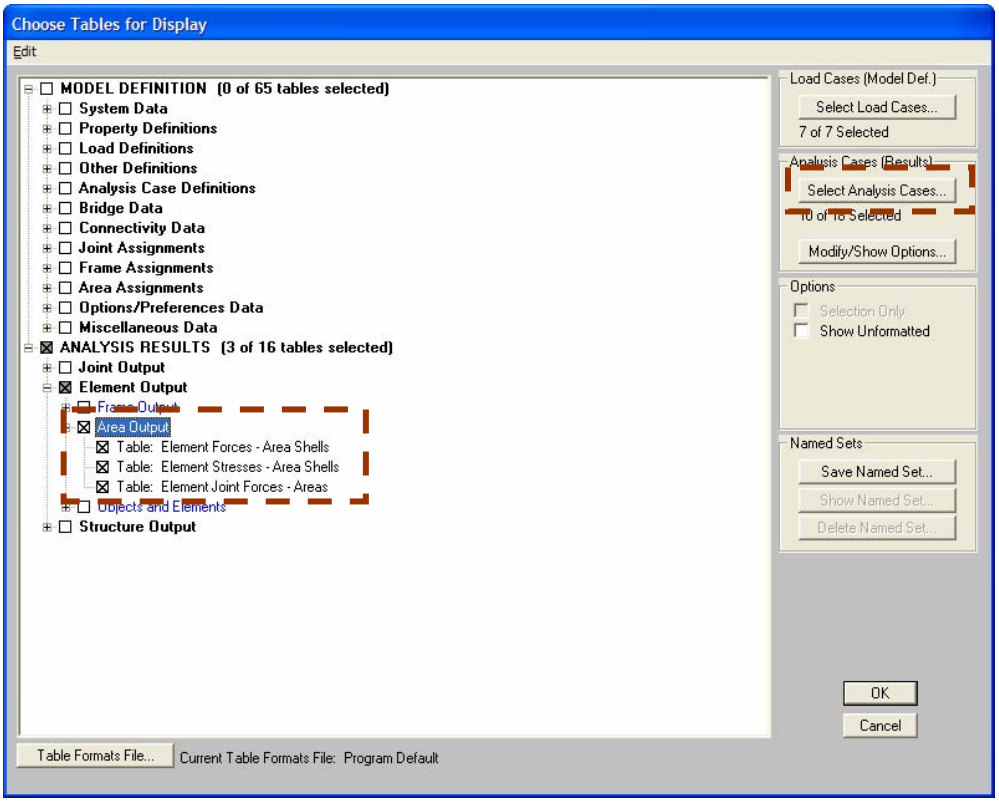
Step 15-2: View Shell Analysis Results in Contour

Forces/Stresses in shell elements are displayed in contour color. Color tab at the bottom of screen indicates the value range of the results.



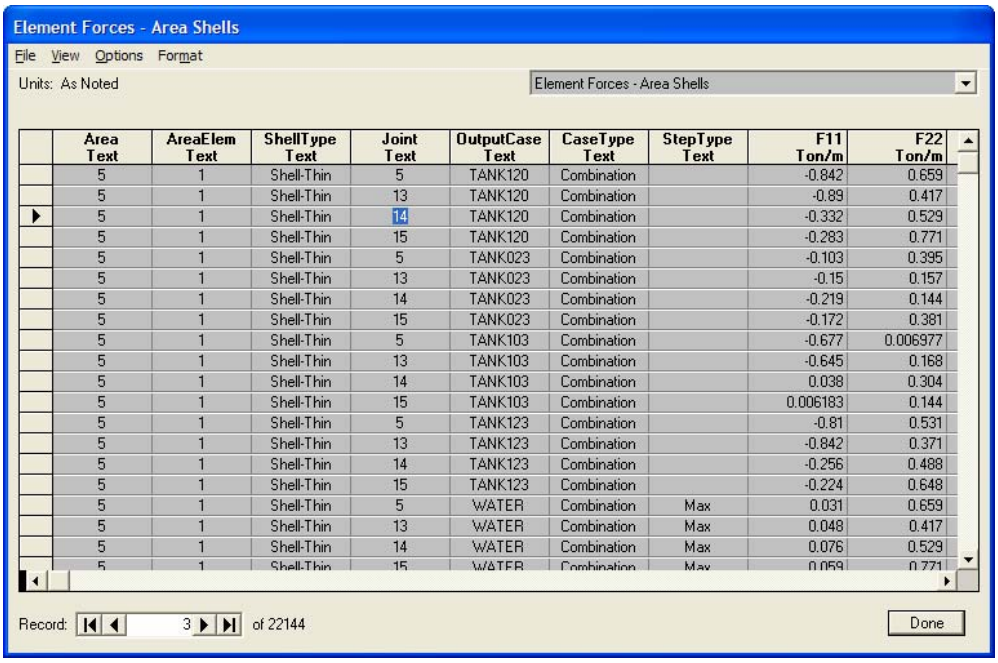
Step 15-3: Select Shell Analysis Results to View in Tabular Form

Forces/Stresses in shell elements are displayed in tabular form. Go to **Display >> Show Table**. Select “Area Output”, click on “Select Analysis Cases” to select desired load cases and combinations



Step 15-4: View Shell Analysis Results to View in Tabular Form

Shell analysis results are displayed in tabular form as shown in figure below.



Area Text	AreaElem Text	ShellType Text	Joint Text	OutputCase Text	CaseType Text	StepType Text	F11 Ton/m	F22 Ton/m
5	1	Shell-Thin	5	TANK120	Combination		-0.842	0.659
5	1	Shell-Thin	13	TANK120	Combination		-0.89	0.417
5	1	Shell-Thin	14	TANK120	Combination		-0.332	0.529
5	1	Shell-Thin	15	TANK120	Combination		-0.283	0.771
5	1	Shell-Thin	5	TANK023	Combination		-0.103	0.395
5	1	Shell-Thin	13	TANK023	Combination		-0.15	0.157
5	1	Shell-Thin	14	TANK023	Combination		-0.219	0.144
5	1	Shell-Thin	15	TANK023	Combination		-0.172	0.381
5	1	Shell-Thin	5	TANK103	Combination		-0.677	0.006977
5	1	Shell-Thin	13	TANK103	Combination		-0.645	0.168
5	1	Shell-Thin	14	TANK103	Combination		0.038	0.304
5	1	Shell-Thin	15	TANK103	Combination		0.006183	0.144
5	1	Shell-Thin	5	TANK123	Combination		-0.81	0.531
5	1	Shell-Thin	13	TANK123	Combination		-0.842	0.371
5	1	Shell-Thin	14	TANK123	Combination		-0.256	0.488
5	1	Shell-Thin	15	TANK123	Combination		-0.224	0.648
5	1	Shell-Thin	5	WATER	Combination	Max	0.031	0.659
5	1	Shell-Thin	13	WATER	Combination	Max	0.048	0.417
5	1	Shell-Thin	14	WATER	Combination	Max	0.076	0.529
5	1	Shell-Thin	15	WATER	Combination	Max	0.059	0.771