SMAP ® - 3D

Structure Medium Analysis Program

3-D Static, Consolidation and Dynamic
Analysis for Dry, Saturated and
Partially Saturated Soils
and Rock Mass

Example Problems

LICENSE AGREEMENT

LICENSE: COMTEC RESEARCH grants to Licensee a non-exclusive,non-transferable right to use the enclosed Computer Program only on a single computer. The use of the Computer Program is limited to the Licensee's own project. Licensee may not use the Computer Program to serve other engineering companies or individuals without prior written permission of COMTEC RESEARCH. Licensee may not distribute copies of the Computer Program or Documentation to others. Licensee may not rent, lease, or network the Computer Program without prior written permission of COMTEC RESEARCH.

TERM: The License is effective as long as the Licensee complies with the terms of this Agreement. The License will be terminated if the Licensee fails to comply with any term or condition of the Agreement. Upon such termination, the Licensee must return all copies of the Computer Program, Software Security Activator and Documentation to COMTEC RESEARCH within seven days.

COPYRIGHT: The Licensed Computer Program and its Documentation are copyrighted. Licensee agrees to include the appropriate copyright notice on all copies and partial copies.

USER SUPPORT: COMTEC RESEARCH will provide the Software Support for the Registered Users for a period of 90 days from the date of purchase. User support is limited to the investigation of problems associated with the correct operation of the Licensed Computer Program. The Licensee must return the Registration Card in order to register the Licensed Computer Program.

DISCLAIMER: COMTEC RESEARCH has spent considerable time and efforts in checking the enclosed Computer Program. However, no warranty is made with respect to the accuracy or reliability of the Computer Program. In no event will COMTEC RESEARCH be liable for incidental or consequential damages arising from the use of the Computer Program.

UPDATE POLICY: Update programs will be available to the Registered Licensee for a nominal fee. The Licensee must return all the Original Distribution Diskettes and Software Security Activator to receive the update programs.

GENERAL: The State of California Law and the U. S. Copyright Law will govern the validity of the Agreement. This Agreement may be modified only by a written consent between the parties. COMTEC RESEARCH, 12492 Greene Ave., Los Angeles, CA 90066, U.S.A

Contents

Introduction

Example Problems are mainly provided:

- To give you some guide in preparing input data.
- To demonstrate the validity of SMAP programs.

Section 2 describes methods of preparing Mesh Files which represent the geometry of structures to be analyzed.

Section 3 describes two different methods of running main- and post-processing programs.

Section 4 illustrates SMAP-3D main example problems as summarized in Table 1.1. First 9 problems are presented to demonstrate the accuracy and validity of SMAP-3D main- processing program.

Section 5 illustrates Group Mesh examples. Group Mesh Generator is a two dimensional CAD program specially designed to build group mesh which can be used to generate finite element mesh with the aid of program ADDRGN-2D.

Section 6 illustrates Block Mesh examples. Block Mesh Generator is a three dimensional CAD program specially designed to build block mesh which can be used to generate finite element mesh with the aid of program PRESMAP-GP.

Section 7 illustrates PRESMAP examples which are used to generate two and three dimensional Mesh Files.

Section 8 illustrates ADDRGN examples which are used to combine or modify existing Mesh Files. ADDGRN-2D has a powerful mesh generation feature as demonstrated in sub section 8.1.3.

Section 9 illustrates SUPPLEMENT examples which are useful to prepare input data for pre- and main-processing programs.

Section 10 illustrates LOAD examples which are used to generate external nodal loads in two and three dimensional coordinate systems.

Section 11 illustrates XY Graph examples. XY Graph is a two dimensional graph consisting of lines connecting each pair of data points, which can be plotted by PLOT-XY or Excel.

Table 1.1 List of SMAP-3D example problem

Problem Number	Project File Name	Run Time Pent. III 850	Description
1	VP1.dat	0.01 min.	Undrained uniaxial strain compression. Check: • Static • Fully coupled two-phase medium
2	VP2.dat	0.03	Terzaghi's linear consolidation Check: • Consolidation • Gravity load
	VP2-1.dat	0.10	Using linear wedge element
3	VP3.dat	0.37	Planar compression wave propagation Check: • Dynamic two-phase response
	VP3-1.dat	0.13	Using transmitting boundary
4	VP4.dat	0.35	Circular tunnel in Drucker-Prager medium Check: • 3-D elasto-plastic matrix of Generalized Hoek and Brown Model
	VP4-1.dat		Using element surface load
	VP4-2.dat		Using linear wedge element
5	VP5.dat	0.15	Laminated beam with slip interface Check: • Joint element • Joint model
	VP5-1.dat	0.98	Thin layer joint element, NM=4 Joint thickness by CARD 5.3.2.4.11

Table 1.1 List of SMAP-3D example problem, continued

Problem Number	Project File Name	Run Time Pent. III 850	Description
6	VP6.dat	0.02 min.	Gibson's construction pore pressure Check: • Consolidation • Variable time step • Moving boundary
	VP6-1.dat		Using linear wedge element
7	VP7.dat	0.01	Drained triaxial compression test Check: • Modified Cam Clay Model • Drained triaxial compression path
8	VP8.dat	0.01	Undrained plane strain comp. test. Check: • Modified Cam Clay Model • Undrained plane compression path
9	VP9.dat	0.01	Volumetric creep in isotropic undrained test. Check: • Modified Cam Clay Model • Volumetric creep
10	VP10.dat	0.01	Space truss analysis
11	VP11.dat	0.01	Fixed end beam analysis
12	VP12.dat	0.01	Beam dynamic analysis
13	VP13.dat	0.85	William's toggled beam analysis
14	VP14.dat	0.02	Plane strain tunnel analysis
15	VP15.dat	0.01	Hemispherical shell
	VP15-1.dat		Using triangular shell element
16	VP16.dat	0.02	Simply supported plate analysis

Table 1.1 List of SMAP-3D example problem, continued

Problem Number	Project File Name	Run Time Pent. III 850	Description
17	VP17.dat	0.01 min.	Heated beam modeled by shell
	VP17-1.dat		Heated beam modeled by beam
	VP17-2.dat		Heated beam modeled by continuum
18	VP18.dat	0.01	Thin pipe subjected to internal pressure
	VP18-1.dat		Single precision with FACBD = $1x10^6$
19	VP19.dat	24.12	Preload consolidation & excavation
20	VP20.dat	16.93	Seismic tunnel analysis
21	VP21.dat	0.01	Frames with hinge connection Modeled by beam element
	VP21-1.dat		Modeled by shell element
22	VP22.dat		Embedded rebars with slip
23	VP23.dat		Pseudo dynamic embankment fill
24	VP24.dat		Plane strain tunnel in jointed continuum
25	VP25.dat		Spring analysis
26	VP26.dat		Nonlinear truss analysis
27	VP27.dat		SDOF System To Ground Acceleration
28	VP28.dat		Frames with Rotational Spring Connection
29	VP29.dat		Reinforced Concrete Beam
30	VP30.dat		Reinforced Concrete Cylinder
31	VP31.dat		Plate Modal Analysis
32	VP32.dat		Seismic Response Analysis
33	VP33.dat		Silo Lining Analysis
34	VP34.dat		Liquefaction Analysis with PM4Sand

Pre-Processing Programs

Pre-Processing programs are mainly used to generate Mesh File described in Section 4.3 of SMAP-3D User's Manual. The Mesh File represents the geometry of the structure to be analyzed. This file contains information about nodal coordinates, element indexes, material property numbers, and boundary codes. In SMAP-3D, you may generate such Mesh Files using the following methods:

Method 1

First, generate 2D Mesh File representing a typical two dimensional section using Group Mesh Generator, Block Mesh Generator, or 2D PRESMAP. Modify this 2D Mesh File using ADDRGN-2D if you need to do it. And then extend the 2D mesh into 3D mesh using GEN-3D.

1. Generate 2D Mesh File

   GROUP MESH GENERATOR
   BLOCK MESH GENERATOR
   PRESMAP-2D NATM-2D
   CIRCLE-2D PRESMAP-GP

2. Modify 2D Mesh File

   ADDRGN-2D

3. Extend into 3D Mesh File

   GEN-3D

Method 2

Generate 3D Mesh Files using Block Mesh Generator or 3D PRESMAP. Then combine or modify these 3D Mesh Files using ADDRGN-3D if you need to do it.

1. Generate 3D Mesh File

   BLOCK MESH GENERATOR
   PRESMAP-3D CROSS-3D
   PRESMAP-GP

2. Combine or modify 3D Mesh File

   ADDRGN-3D

Above two methods can be combined to make a final 3D Mesh File representing the structure to be analyzed.

To view the Mesh Files, you can use PLOT-3D by selecting following order:
Plot → Mesh → F. E. Mesh → Open

Boundary codes can affect analysis result significantly so that it is strongly recommended for you to double check those codes to avoid solving wrong problems.

Main- and Post-Processing Programs

Main-Processing program reads Mesh and Main Files as input and performs static, consolidation, or dynamic analysis. Post-Processing programs read Post File along with analysis results from Main-Processing program and then produce graphical output.

Mesh Files can be generated using Pre-Processing programs as outlined in the previous Section 2. Main and Post Files can be created according to Section 4.4 and 4.5, respectively, in SMAP-3D User's Manual. Normally, they can copy existing Main or Post Files which are similar to the problem to be analyzed and modify those files using Text Editor.

Main- and Post-Processing programs can be executed using the following methods:

Method 1

Prepare Mesh, Main, and Post Files. Run EXECUTE menu to get analysis results. And run PLOT menu to view graphical output of analysis results.

1. Prepare All Input Files

   Mesh, Main and Post Files

2. Get Analysis Results

   RUN → SMAP → EXECUTE

3. View Graphical Output

   PLOT → RESULT → PLOT-XY, PLOT-2D, PLOT-3D

Method 2

Prepare Mesh, Main, and Blank Post Files. Run EXECUTE menu to get analysis results. Now, prepare Post File according to Section 4.5 in SMAP-3D User's Manual. Run PRE EXECUTE menu to obtain intermediate plotting information files. And then run PLOT menu to view graphical output of analysis results. Note that Blank Post File consists of following 3 lines:

```
┌  0,  1,  2
│  0
└  0,  4.5
```

1. Prepare Mesh and Main Files

Mesh, Main and Blank Post Files

2. Get Analysis Results

RUN → SMAP → EXECUTE Menu

3. Prepare Post File

Post File in Section 4.5 of User's Manual

4. Get Plotting Information Files

RUN → SMAP → PreEXECUTE

5. View Graphical Output

PLOT → RESULT → PLOT-XY, PLOT-2D, PLOT-3D

Method 2 is particularly useful when you are running large problems which take long execution time. You have to care in preparing Card Group 10 in Main File since Post File can only address those data requested in Card Group 10. You can repeat Steps 3 and 4 as long as your Post File addresses the output data within the range specified in Card Group 10 in Main File.

Post-Processing programs are mainly used to show graphical output of the analysis results.

PLOT-XY reads Card Group 12 in Post File and plots time histories of stresses, strains, and displacements. Once you run PLOT-XY , you will obtain intermediate plotting information file (PLOTXY.Lin). PLOTXY.Lin file can be modified as it will be described in Section 11 of SMAP Examples.

PLOT-2D reads Card Group 11 in Post File and plots two dimensional snap shots. Once you run PLOT-2D in PLOT menu, you will obtain intermediate plotting information file (PLOT2D.DAT).

PLOT-3D does not need any Post File.
This program plots following three dimensional snapshots:

- Finite element mesh
- Deformed shape
- Principal stress distribution
- Section forces in beam element
- Extreme fiber stresses/strains in beam elements (2D)
- Axial force/stress/strain in truss element
- Contours of stresses, strains and factor of safety
- 3D iso surface of stresses and strains

SMAP-3D
Example Problem

SMAP-3D is the main-processing program which computes static, consolidation and dynamic response of three-dimensional problems. Input parameters of SMAP-3D are described in detail in Section 4 of SMAP-3D User's Manual.

Running SMAP-3D is described in Section 3.2.1 of User's Manual and can be selected in the following order:

RUN → SMAP → EXECUTE

Manual procedure to run SMAP-3D is outlined in Section 3.5 of User's Manual. Once you finished execution of SMAP-3D, you can obtain graphical outputs by selecting:

PLOT → RESULT → PLOT-XY, PLOT-2D, or PLOT-3D

PLOT Menu is described in Section 3.3 of SMAP-3D User's Manual.

Table 1.1 in Section 1 shows the summary of SMAP-3D example problems. First nine example problems are the verification problems. The main objective of these verification problems is to demonstrate the accuracy and validity of SMAP-3D.

You can access all input files of example problems in the directory:

C:\Smap\Smap3D\Example\Smap

For each example problem, brief problem descriptions and partial graphical outputs will be presented in this section.

4.1 Undrained Uniaxial Strain Compression

The problem concerns fully coupled undrained uniaxial strain response of saturated porous linear elastic medium as shown in Figure 4.1.

Finite element mesh in Figure 4.2 is generated by Block Mesh Generator as explained in detail in Section 6.1 in SMAP-3D Example Problem.

The exact solution for the undrained stress response is given by Blouin and Kim, 1984.

$$\pi_o = \sigma_v \frac{1}{1 + \beta_m} \tag{4.1}$$

$$\beta_m = \frac{K_g^2 M_s + K_m K_s^2 - M_s K_m K_s - K_g K_m K_s}{K_m K_g (K_g - K_s)} \tag{4.2}$$

Where

σ_v	Applied total vertical stress
π_o	Pore water pressure
K_s	Bulk modulus of skeleton
G_s	Shear modulus of skeleton
M_s	Constrained modulus of skeleton ($M_s = K_s + 4G_s /3$)
n	Porosity
K_g	Bulk modulus of grain
K_w	Bulk modulus of water
K_m	Mixture modulus $K_m = K_g \; K_w / \{K_w + n [K_g - K_w]\}$

The following material properties are used for computing undrained uniaxial strain response:

K_g = 3.5210×10^6 t/m^2
K_w = 0.2042×10^6 t/m^2
E = 0.7042×10^6 t/m^2
ν = 0
n = 0.3
G_s = 2.674

K_s = 0.2347×10^6 t/m^2
G_s = 0.3521×10^6 t/m^2

The exact ratio of pore water pressure (π_o) to applied total vertical stress (σ_v) is obtained from equations 4.1 and 4.2

$$\pi_o / \sigma_v = 0.4592$$

and the exact ratio of effective vertical stress (σ_v') to applied total vertical stress (σ_v) is given by

$$\sigma'_v / \sigma_v = 0.5408$$

Figure 4.3 shows predicted undrained uniaxial stress response compared with an exact solution. As shown in Figure 4.3, the predicted response by program SMAP-3D is identical to the exact solution.

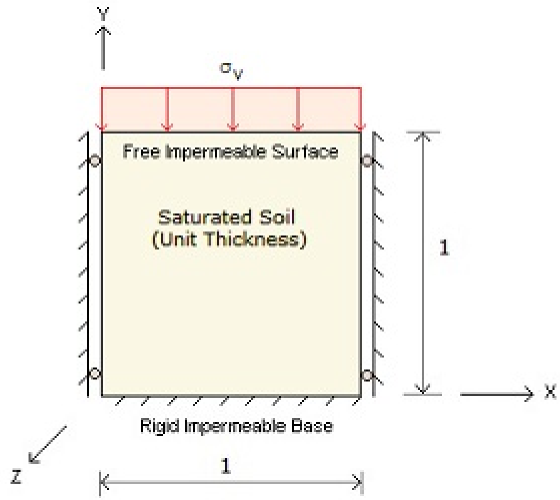


Figure 4.1 A cubic element subjected to undrained uniaxial strain loading

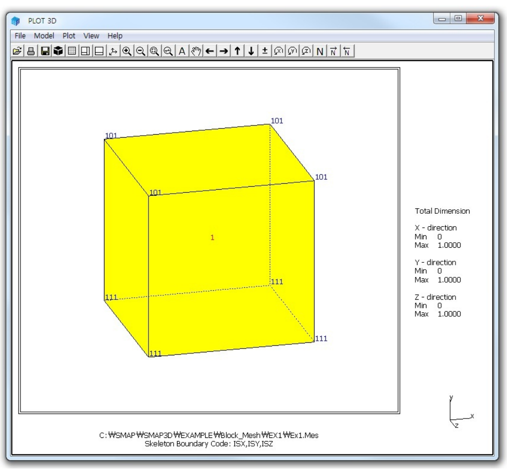


Figure 4.2 Finite element mesh

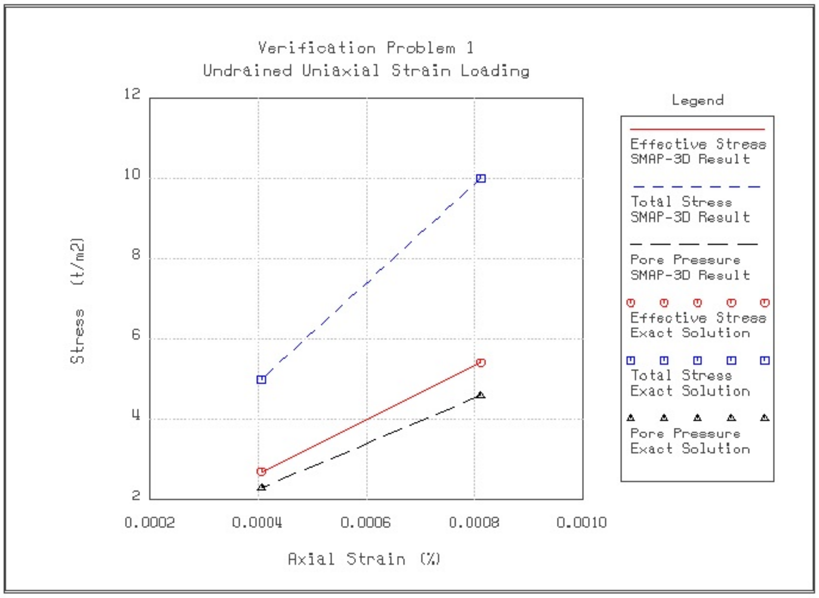


Figure 4.3 Computed undrained stress response compared with exact solution

4.2 Terzaghi's Linear Consolidation

The problem concerns Terzaghi's linear consolidation with initial triangular distribution of excess pore water pressures. As initial conditions, it is assumed that soil is liquefied and pore water takes all the weight. The exact solution for the excess pore water pressure (π_e) is given by

$$\pi_e = \sum_{m=1,3}^{\infty} \left(\frac{8\,\gamma'\,H}{m^2\,\pi^2} \right) \left(\mathrm{Sin}\,\frac{m\,\pi}{2} \right) \left(\mathrm{Sin}\,\frac{m\,\pi}{2\,H}\,y \right) e^{-\frac{m^2\pi^2}{4}T} \tag{4.3}$$

where

H Thickness of soil deposit.
  Top is free surface, bottom is rigid impermeable base.

y Distance from the free surface.

$\gamma' = \gamma - \gamma_w$
  γ is the total unit weight and
  γ_w is the unit weight of pore water.

And the time factor (T) is given by

$$T = \frac{k\,M\,t}{\gamma_w\,H^2}$$

where

t Time
k Coefficient of permeability
M Constrained modulus

To simulate numerically, following material parameters are assumed:

n = 0.3 Porosity
G_s = 2.7 Specific gravity of grain
γ_w = 1.0 t/m^3
γ = $\gamma_w\,(G_s\,(1-n) + n) = 2.19\ t/m^3$
γ' = 1.19 t/m^3

$E = 1{,}000 \ t/m^2$

$\nu = 0.3$

$M = (1-\nu) E / ((1+\nu)(1-2\nu)) = 1{,}346 \ t/m^2$

$k = 0.001$ m/day

$H = 10$ m

Figure 4.4 shows finite element mesh consisting of 20 elements used for this example problem.

Figure 4.5 shows profiles of pore water pressures at T = 0.05 and 0.5. And Figure 4.6 shows profiles of effective vertical stresses at T = 0.05 and 0.5. SMAP-3D calculations are very close to the exact solution.

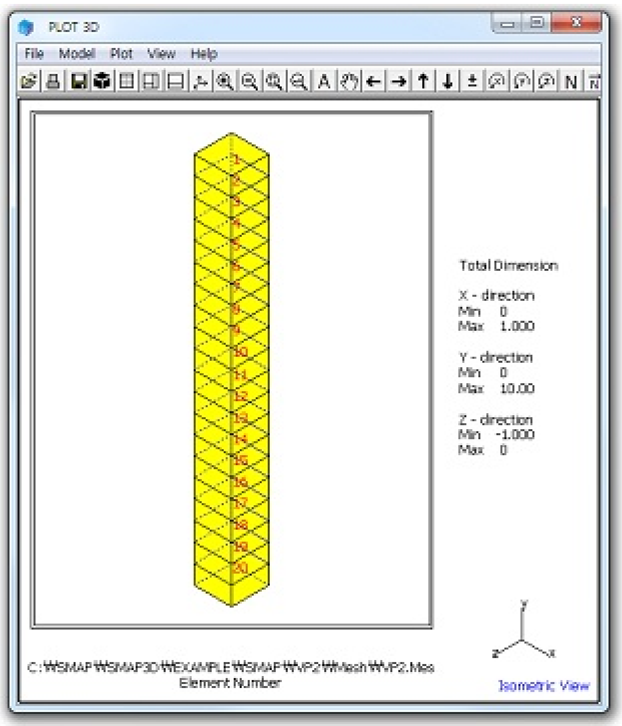


Figure 4.4 Finite element mesh

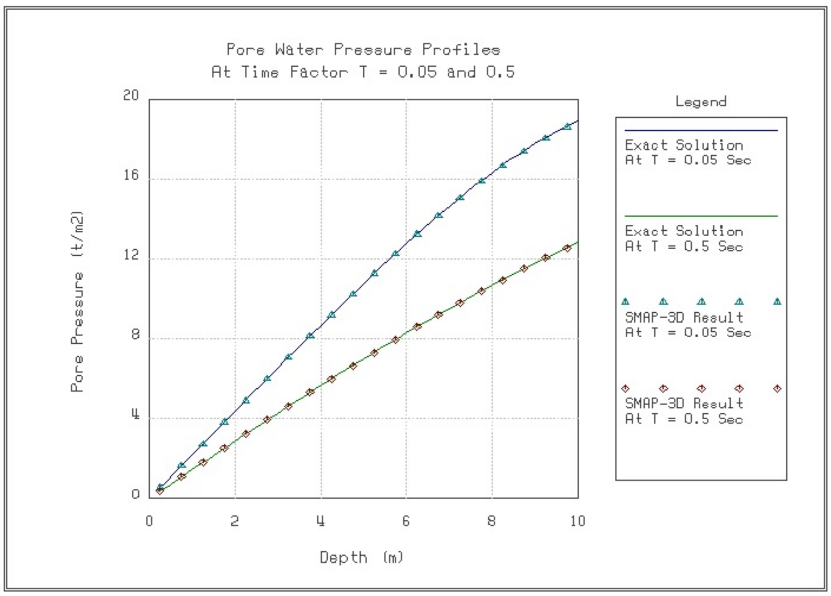


Figure 4.5 Pore water pressure profiles at T = 0.05 and 0.5

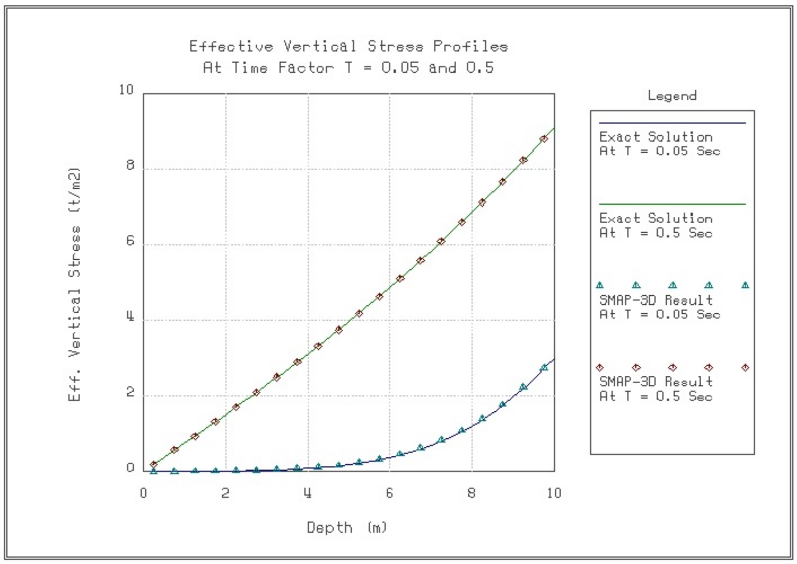


Figure 4.6 Effective vertical stress profiles at T = 0.05 and 0.5

4.3 Planar Compression Wave Propagation

The problem is to check overall two-phase dynamic equations implemented in the program SMAP-3D. A vertically propagating planar compression wave through idealized saturated soil is considered. The input loading, as shown in Figure 4.8, is a short rise time triangular pulse with a peak stress of 3,521 t/m^2 and a positive phase duration of 10 msec. The loading pulse is applied to the saturated sand having the properties listed in Figure 4.8. The load is applied to an impermeable boundary at the ground surface.

Figure 4.7 shows finite element mesh consisting of 200 elements.

Computed profiles of pore water pressure and effective vertical stress at 20 msec are shown in Figures 4.9 and 4.10, respectively. The closed-form solution for this problem is not available. So, the same problem has been solved by the existing two-dimensional version of TPDAP-II for direct comparison. These TPDAP-II results are not shown in Figures 4.9 and 4.10, but they are identical to the SMAP-3D results.

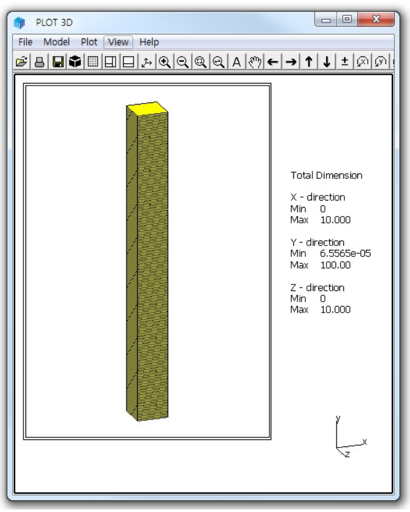
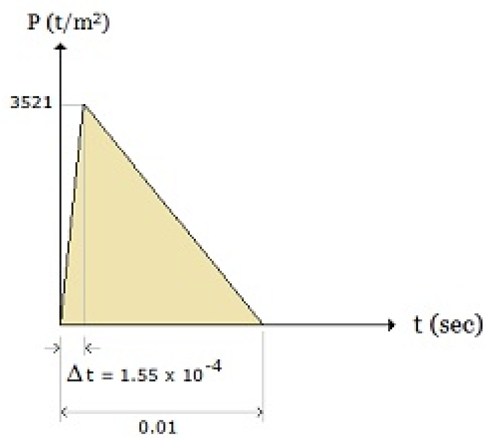


Figure 4.7 Finite element mesh



Assumed Material Properties

Pore Water	
Bulk Modulus	0.2042×10^6 t/m^2
Solid Grains	
Bulk Modulus	3.521×10^6 t/m^2
Specific Gravity	2.67
Drained Skeleton Properties	
Bulk Modulus	2113 t/m^2
Constrained Modulus	4225 t/m^2
Poisson's Ratio	0.20
Porosity	0.35
Permeability	2.54×10^{-5} m/s

Figure 4.8 Loading time history and material properties used in planar compression wave propagation through saturated soil

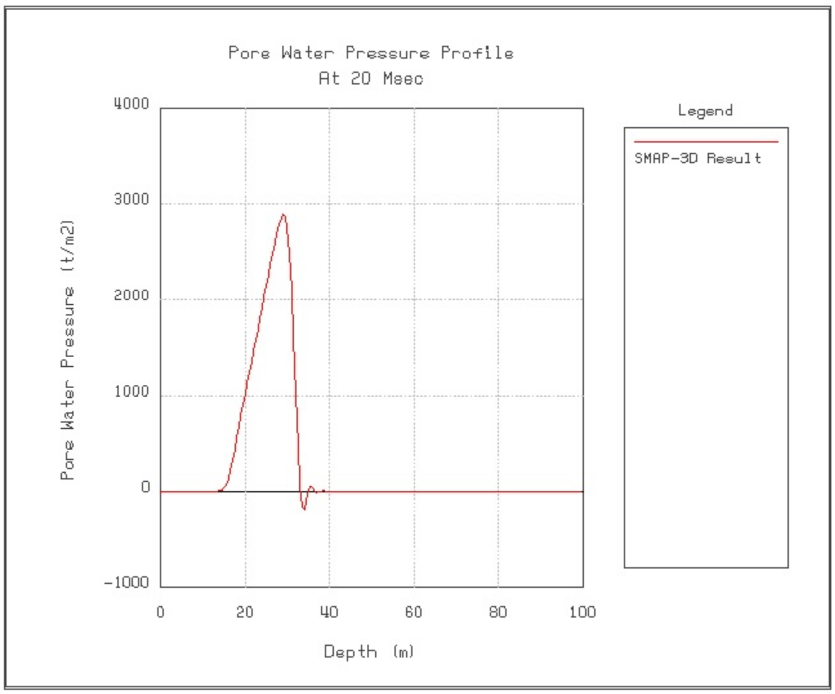


Figure 4.9 Profiles of pore water pressure at 20 msec

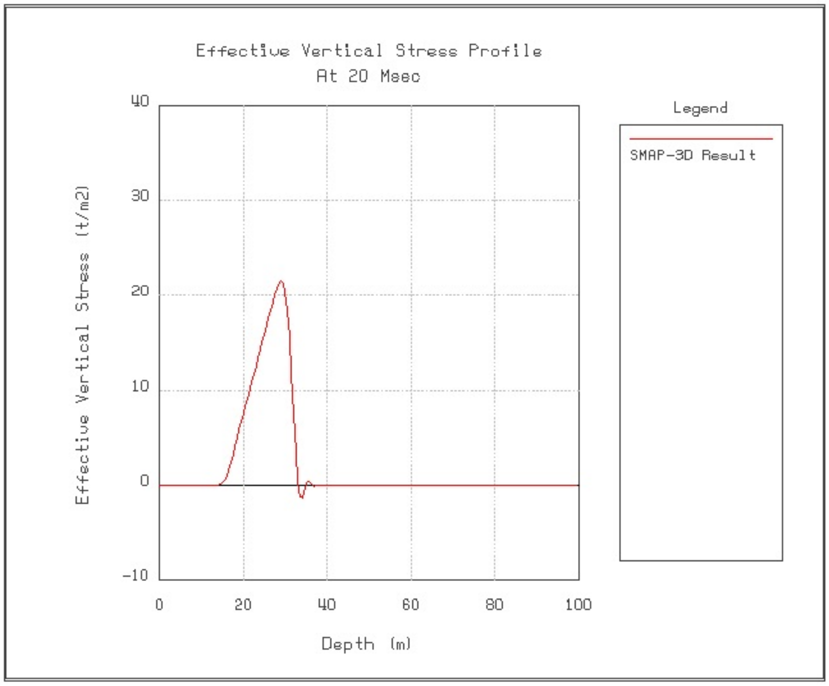


Figure 4.10 Profiles of effective vertical stress at 20 msec

4.4 Circular Tunnel in Drucker-Prager Medium

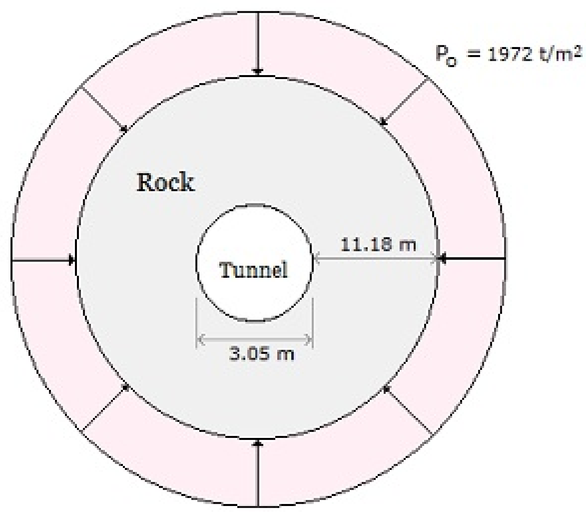
The problem is to check the implementation of the 3-dimensional formulation of elasto-plastic matrix derived for the Generalized Hoek and Brown Model. In this problem, the plane strain response of a tunnel subjected to axisymmetric loading as calculated using SMAP-3D is compared to a semi-analytical solution developed by Piepenburg, Kim and Davister (1986).

Figure 4.11 shows a schematic section view of 3.05m (10 feet) diameter circular tunnel subjected to a hydrostatic loading of 1972 t/m^2 (2800 psi). The surrounding rock is assumed to be linear elastic beneath the failure surface and to follow the Drucker-Prager plasticity model upon reaching the failure surface. The elastic and strength properties of the rock are listed in Figure 4.11.

By symmetry, only a quadrant of tunnel cross section is modeled as shown in Figure 4.12. Along the axis of tunnel (in z-direction), three elements (sections) are used so that the internal section can have unconstrained full 3 degrees of freedom per each node.
This is to check the uniform response of the integrated three dimensional grids though problem is essentially one dimensional axisymmetric.

Figure 4.13 shows tunnel displacement contour. Figure 4.14 shows stresses along the 4.5° from the X-axis in Section 2. And Figure 4.15 shows stresses along the 85.5° from the X-axis in Section 2. As we see, both deformations and stresses are uniform along the tunnel tangential direction. The computed tunnel radial displacement (0.896 Cm) is very close to the semi-analytical solution (0.89 Cm). The computed stress profiles agree well with the semi-analytical solution in both the plastic and elastic zones of deformation surrounding tunnel.

It should be noted that the stresses plotted in Figures 4.14 and 4.15 are in X, Y and Z coordinates so that for exact comparison, these stresses should have transformed to radial and tangential coordinate system.

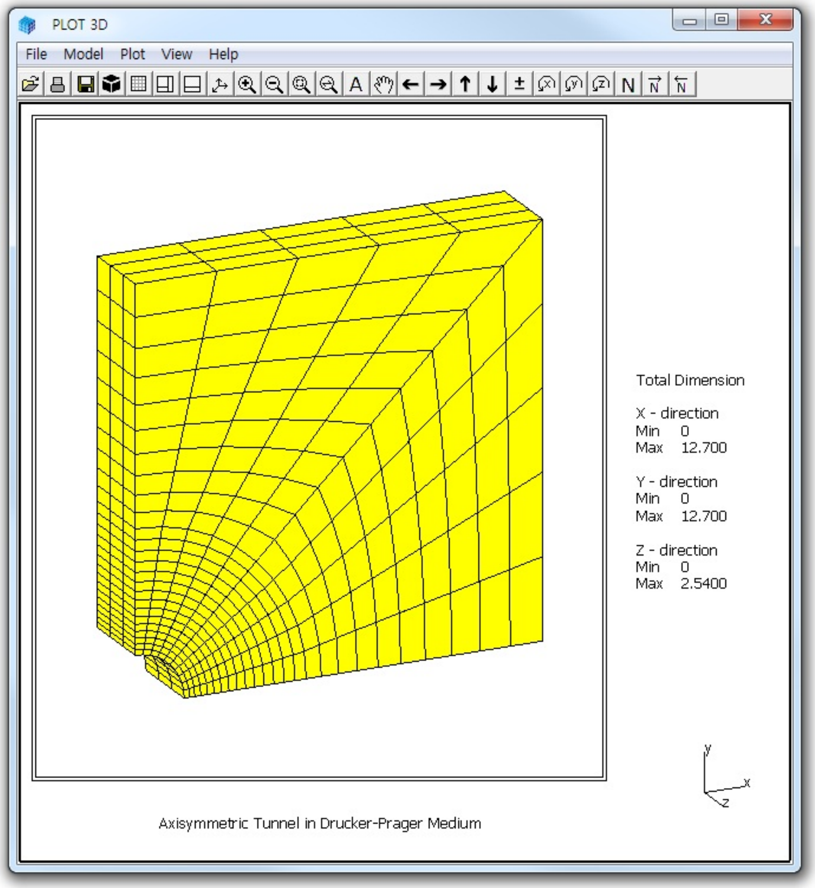


Material Model: Drucker-Prager Model

Rock Properties:

E	=	810,000 t/m² Young's Modulus
ν	=	0.33 Poisson's Ratio
σ_c	=	1,268 t/m² (1800 psi) Unconfined Strength
ϕ	=	18° Friction Angle

Figure 4.11 Circular tunnel subjected to axisymmetric loading



4.12 Finite element mesh

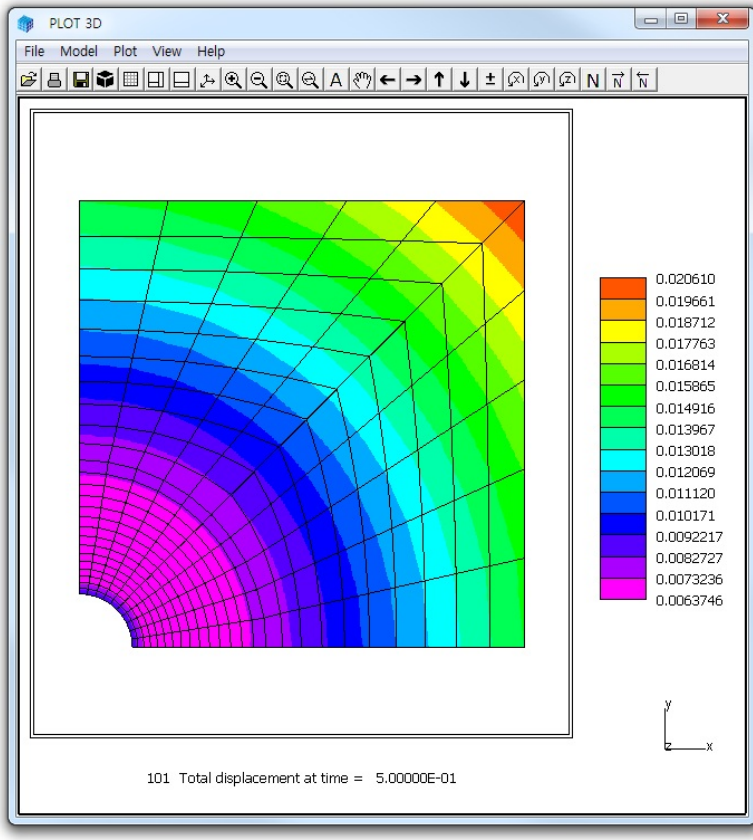


Figure 4.13 Tunnel displacement contour

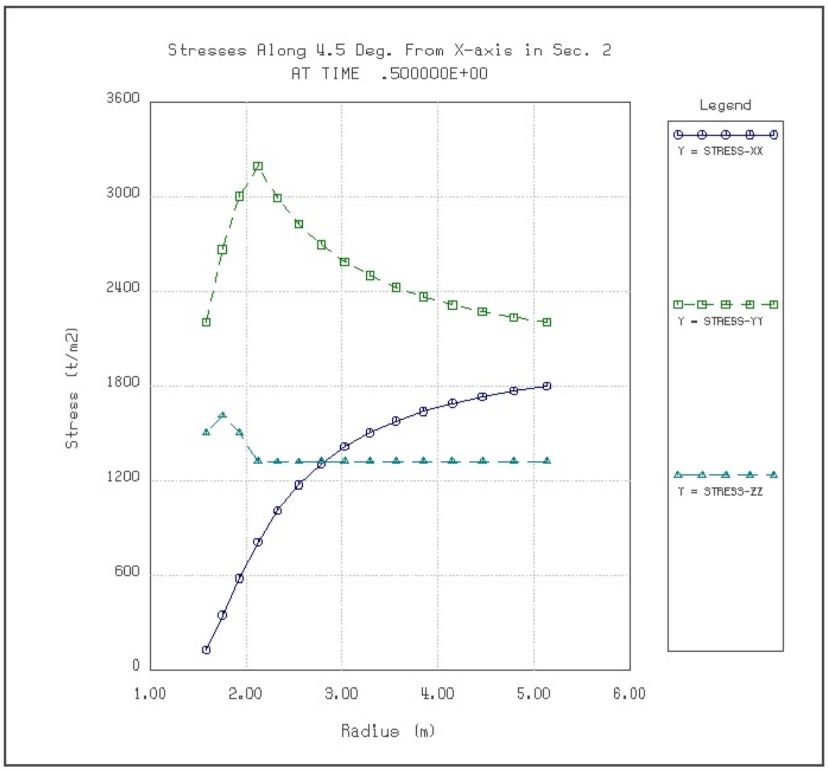


Figure 4.14 Stresses along 4 degree from X-axis in Section 2

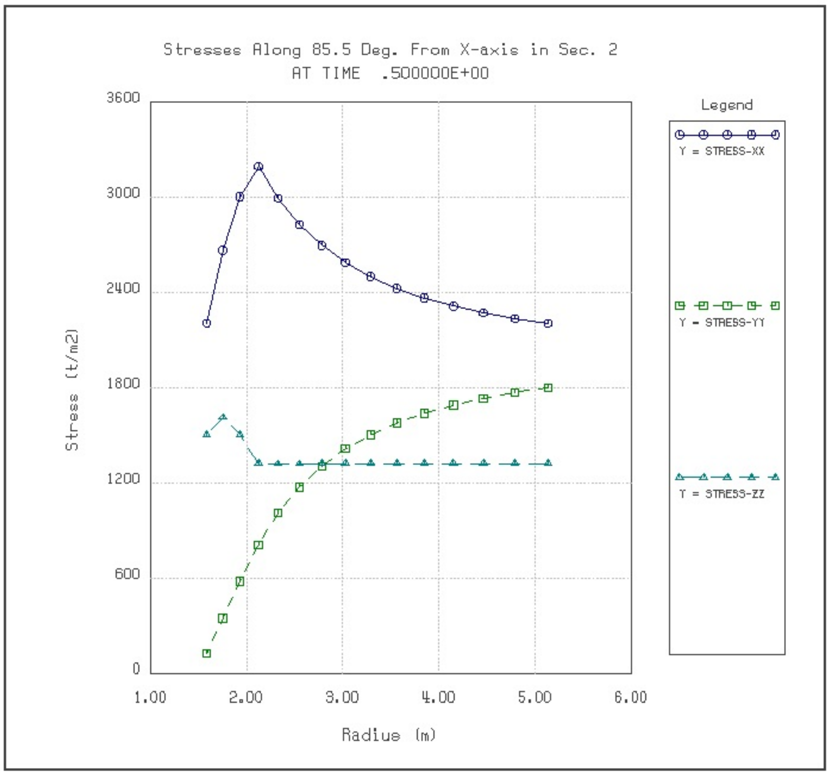


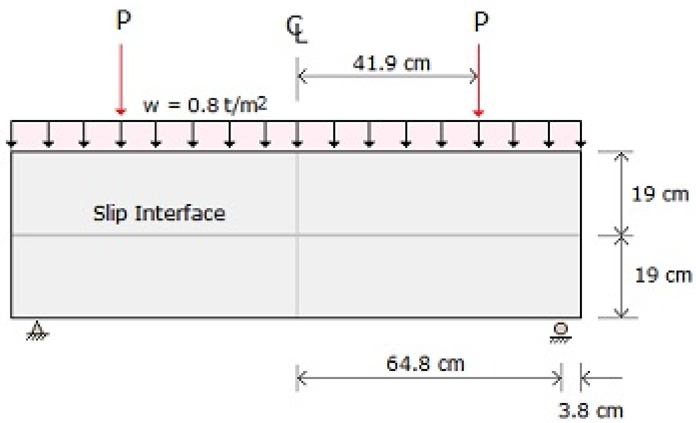
Figure 4.15 Stresses along 85.5 degree from X-axis in Section 2

4.5 Laminated Beam with Slip Interface

The problem is to check the joint element and the nonlinear joint model described in Section 3.6 in theory. Figure 4.16 shows the schematic view of a laminated simply supported beam subjected to uniform and concentrated transverse loads along with the material properties of the beam and the interface.

By symmetry, only the right half of the beam is modeled by 60 continuum elements and 10 joint elements as shown in Figures 4.17 and 18. Element numbers from 61 to 70 are joint elements which represent the slip interface. Joint face is designated along the line from nodes 4 to 144. Thus, nodal coordinates along the other side of joint face are used mainly for visual presentation of joint elements. That is, program SMAP-3D resets internally the nodal coordinates of nodes from 157 to 176 equal to the nodal coordinates of the joint face (nodes from 4 to 144). Then joint thickness (t=0.00254 cm) is specified through the material properties of the joint model.

In Figure 4.19, the midspan deflections by SMAP-3D are compared to the closed-form solution derived from beam theory (Agbabian Associates, 1981). Overall, SMAP-3D results show good agreement with the closed-form solution, especially when the sliding occurs along the interface. It should be noted that there are some differences between the beam and continuum theories, to which slight overestimation by SMAP-3D may be attributed.



Beam Properties	E	=	2.635×10^6 t/m^2
	ν	=	0.1
Interface Properties			
	C	=	4.93 t/m^2
	ϕ	=	0
	t	=	0.00254 cm
Transverse Loads	P	=	0.03 to 17 ton
	w	=	0.8 t/m^2

Figure 4.16 Laminated beam subjected to uniform and concentrated transverse loads

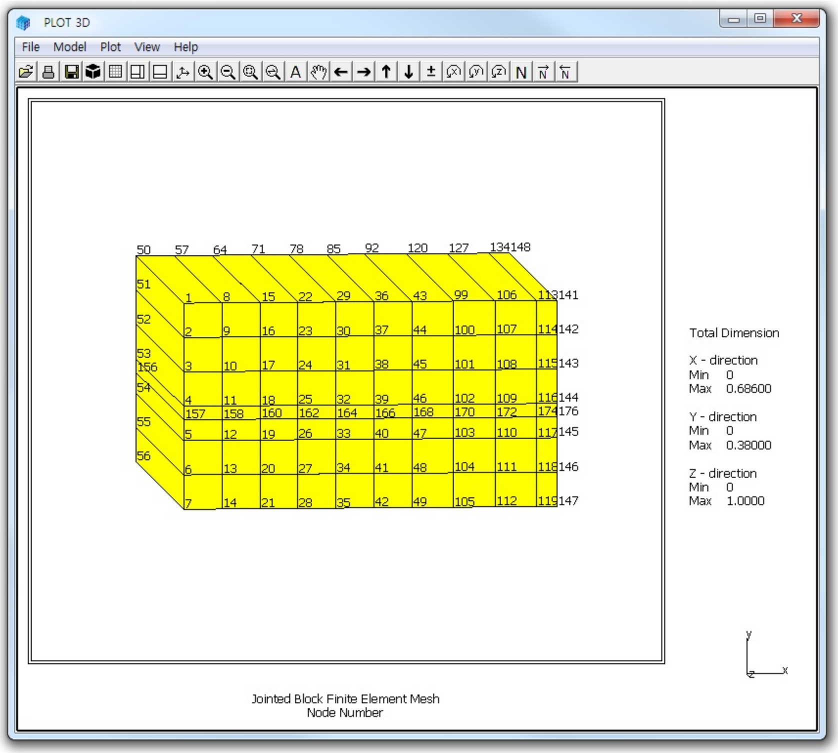


Figure 4.17 Node numbers of laminated beam

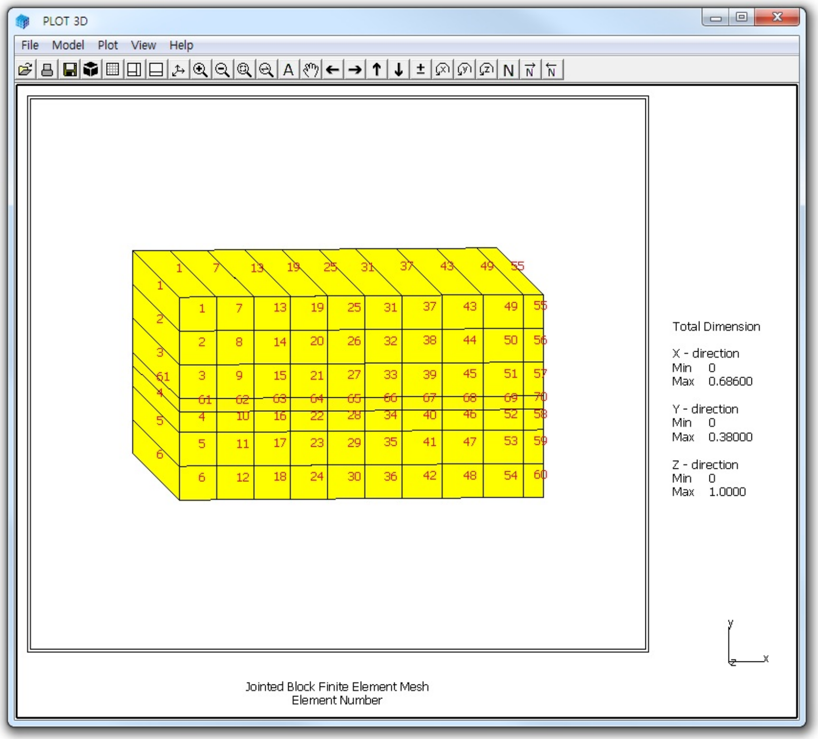


Figure 4.18 Element numbers of laminated beam

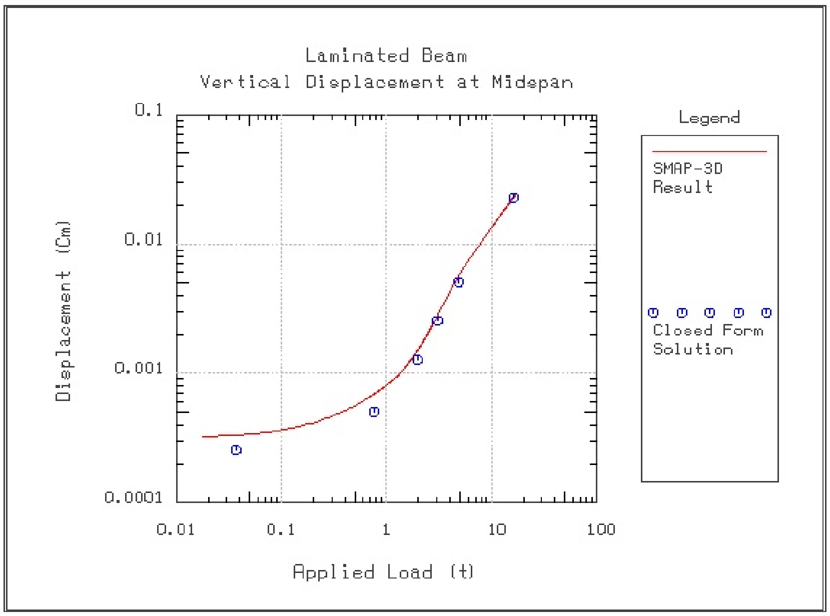


Figure 4.19 Vertical displacement at midspan

4.6 Gibson's Construction Pore Pressure

The problem is to check variable time steps and moving boundary during the construction of the fully saturated fill at constant rate. This problem, as schematically outlined in Figure 4.21, has been analytically solved by Gibson (1958).

$$\pi_e = \gamma' \, m\, t - \gamma' \, (\pi \, C_v \, t)^{-1/2} \cdot$$

$$\exp\frac{-x^2}{4C_v t} \int_0^\infty \left(\xi \tanh\frac{m\xi}{2C_v} \cosh\frac{x\xi}{2C_v t} \exp - \frac{\xi^2}{4C_v t} \right) d\xi \qquad (4.4)$$

π_e Excess pore pressure

C_v Coefficient of consolidation

t Time

All other parameters in Equation 4.4 are described in Figure 4.21.

The saturated fill has been modeled using 36 equally spaced laterally confined 3D continuum elements as shown in Figure 4.20.

Computation is performed until the height of fill reaches to 18 meters at time t = 60 days. Each time when new element is placed, dissipation of fill is followed according to the variable time steps listed in Table 4.1.

Table 4.1 Variable time steps applied for each lift

Sequence	$\Delta t/(\Delta h/m)$
Beginning	0.001
	0.106
	0.106
Intermediate	0.160
	0.160
	0.234
End	0.234

where Δt is time step and Δh thickness of current top layer.

Following input parameters are used to compute profiles of pore pressure.

E = 1000 t/m^2
ν = 0.3
G_s = 2.7
γ_w = 1.0 t/m^3
n = 0.6
k = 0.001 m/day
h = 18 m
t = 60.03 days

T = 4
m = 0.3 m/day
M_s = 1346.15 t/m^2
C_v = 1.3462 m^2/day
γ' = 0.68 t/m^3

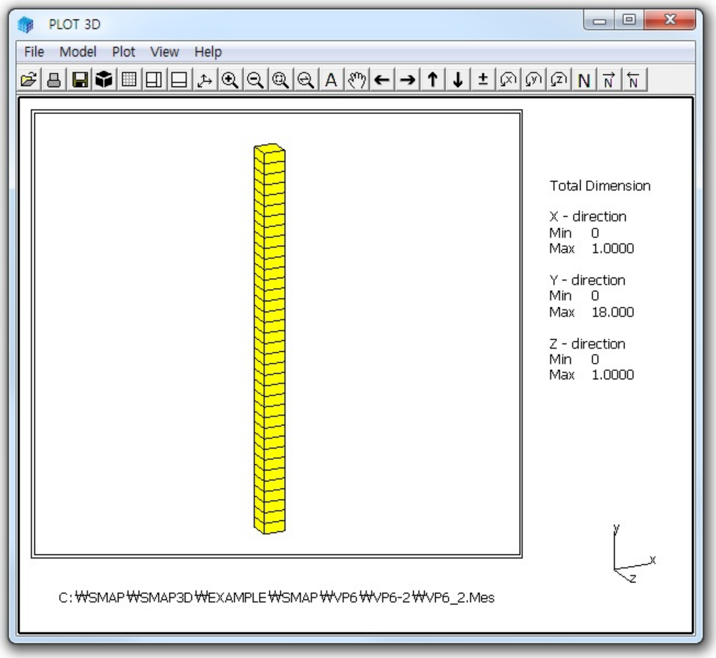
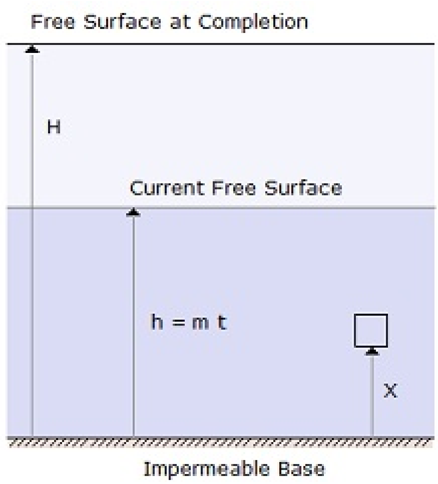


Figure 4.20 Finite element mesh



H	=	Height of complete fill
h	=	Current height of fill
T	=	$m^2\ t\ /\ C_v$ Time factor
m	=	Constant rate of increase of fill height
t	=	Current time
C_v	=	$M_s\ k\ /\ \gamma_w$ Coefficient of consolidation
k	=	Coefficient of permeability
γ_w	=	Unit weight of water
M_s	=	$(1-\nu)\ E\ /\ ((1+\nu)(1-2\nu))$ Constrained modulus
E	=	Young's modulus
ν	=	Poisson's ratio
γ'	=	$\gamma - \gamma_w$ Effective unit weight
γ	=	$G_s\ \gamma_w\ (1-n) + \gamma_w\ n$ Wet unit weight
G_s	=	Specific gravity of solid grain
n	=	Porosity
ξ	=	$X\ /\ h$ Normalized depth
π_{en}	=	$\pi_e\ /\ (\gamma_w\ X)$ Normalized excess pressure
π_e	=	$\pi_w - \gamma_w\ X$ Excess pore pressure
π_w	=	Pore water pressure

Figure 4.21 Fully saturated fill constructed at constant rate

Figure 4.22 shows the normalized excess pore pressure profiles at time factor T = 4. It has been normalized by the height of current fill.

As you see, the computed results of SMAP-3D are very close to Gibson's exact solution.

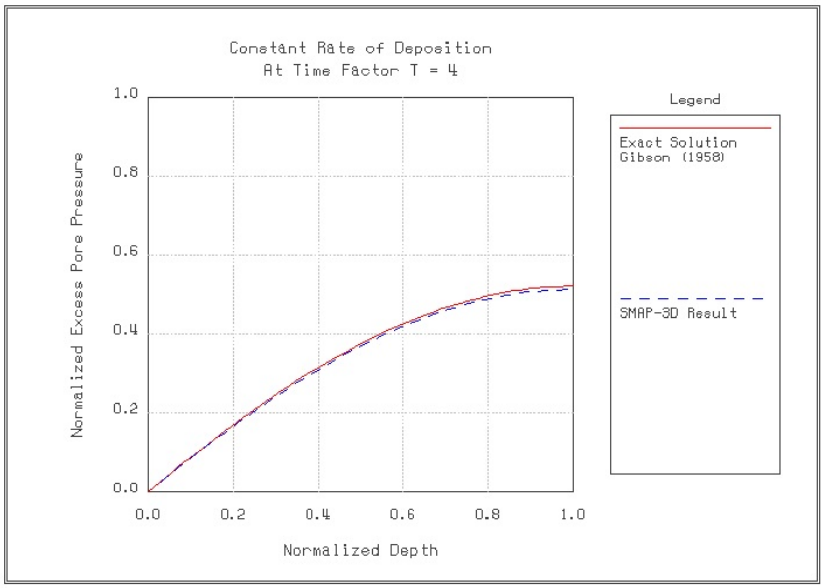


Figure 4.22 Normalized excess pressure profiles at T = 4

4.7 Drained Triaxial Compression Test

The problem is to check the implemented algorithm of the Modified Cam Clay Model in drained triaxial compression mode. The problem is to model the experimental test used by Karshenas and Ghaboussi.

The sample is modeled by a single cubic element with unit length as shown in Figure 4.23. The sample is artificial soil which is composed of 90% $C0_3C_a$ and 10% kaolinite. The material parameters tabulated in Figure 4.24 are those determined by Karshenas and Ghaboussi.

Both computed and measured values are plotted as a function of axial strain in Figure 4.25 for deviatoric stresses and in Figure 4.26 for volumetric strains. As you see, the SMAP-3D results reflect well the overall behavior of test results for the normally consolidated clay.

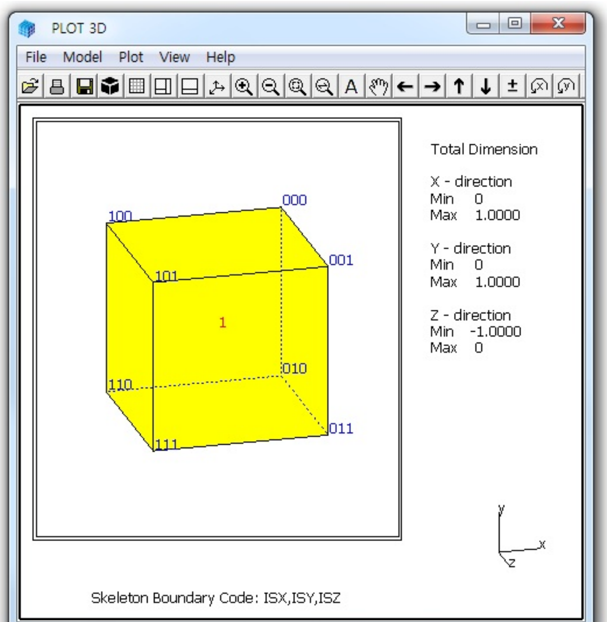
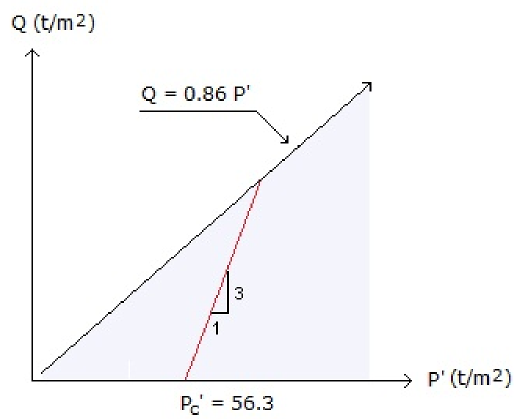


Figure 4.23 Finite element mesh



Material Parameters for Modified Cam Clay Model

Pre-consolidated Pressure	P_c'	=	56.3 t/m^2
Initial Elastic moduli	B_o	=	2540 t/m^2
	G_o	=	1530 t/m^2
Failure Parameter	M	=	0.86
Deformation Parameter	e_o	=	1.0
	ν	=	0.249
	C_c	=	0.2892
	C_r	=	0.1022

Figure 4.24 Drained triaxial compression test

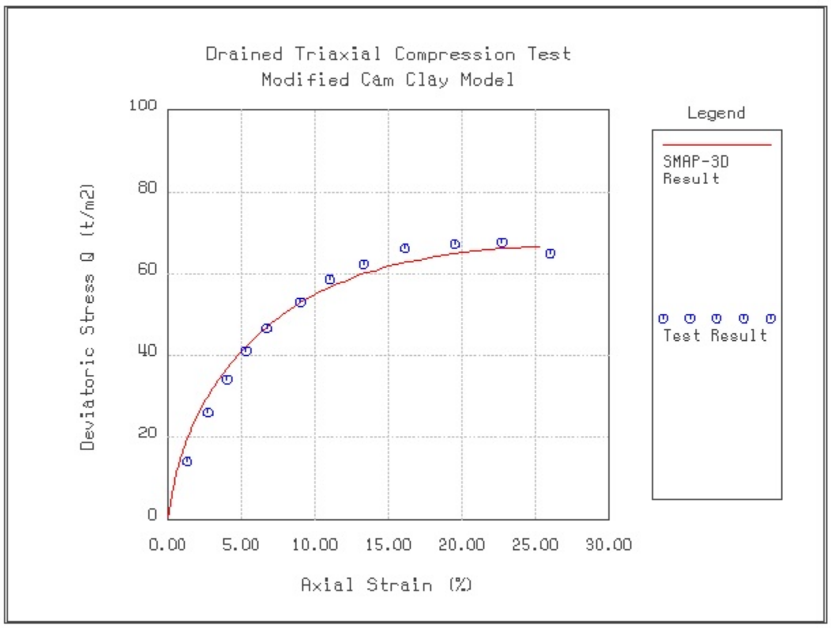


Figure 4.25 Deviatoric stress vs. axial strain for drained triaxial compression test

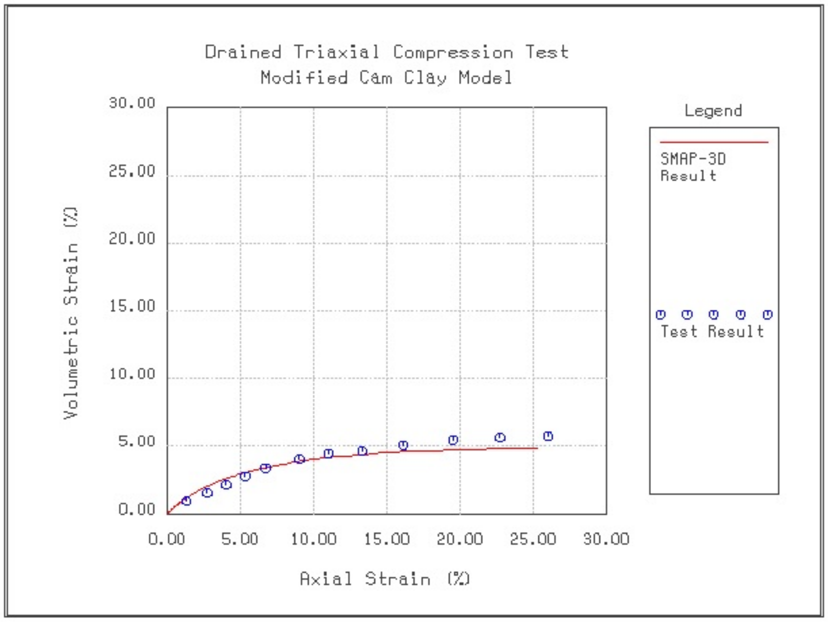


Figure 4.26 Volumetric strain vs. axial strain for drained triaxial compression test

4.8 Undrained Plane Strain Compression Test

The problem is to check the implemented algorithms of Modified Cam Clay Model in undrained plane strain compression stress path. The following analytical solution for this problem has been presented by Kim (1982).

Three components of the effective principal stresses are directly obtained from the specified value of axial strain increment.

$$d\sigma_x' = g_x \, d\varepsilon_x \qquad d\sigma_y' = g_y \, d\varepsilon_y \qquad d\sigma_z' = g_z \, d\varepsilon_z \tag{4.5}$$

$$\sigma_x' = \int d\sigma_x' \qquad \sigma_y' = \int d\sigma_y' \qquad \sigma_z' = \int d\sigma_z' \tag{4.6}$$

where

$$g_x = (b-a) - f\,[3a_o b + (a-b)\,a_x]$$

$$g_y = (a-b) - f\,[3a_o b + (a-b)\,a_y]$$

$$g_z = - f\,[3a_o b + (a-b)\,a_z]$$

$$f = \frac{(a-b)\,(a_y - a_x)}{(a-b)\,(a_x^2 + a_y^2 + a_z^2) + q\,a_0^2\,b + \beta\,M^2 P'\,P_o'\,(2P' - P_o')}$$

$$a = \frac{6.9\,(1+e_o)\,(1-\nu)}{C_r\,(1+\nu)}\,P' \qquad b = \frac{6.9\,(1+e_o)\,\nu}{C_r\,(1+\nu)}\,P'$$

$$a_x' = a_o + 3(\sigma_x' - P') \qquad a_y' = a_o + 3(\sigma_y' - P') \qquad a_z' = a_o + 3(\sigma_z' - P')$$

$$\beta = \frac{2.3\,(1+e_o)}{(C_c - C_r)} \qquad a_o = \frac{2}{3} M^2 (P' - \frac{1}{2} P_o')$$

$$P_o' = P_c' \exp(\beta\,\varepsilon_v^p)$$

Note that the initial stress conditions in Equation 4.6 should be imposed on the basis of the stress-strain state at the end of K_o-consolidated condition.

To perform numerical and analytical solutions, following K_o initial stresses and material parameters are assumed:

Initial stresses:

$\sigma_x' = 0.764$ t/m^2 $\sigma_y' = 1.472$ t/m^2 $\sigma_z' = 0.764$ t/m^2

Material Parameters:

$e_o = 1.339$ $C_c = 0.508$ $C_r = 0.254$ $M = 1.1137$ $\nu = 0.4$

The sample is modeled by a single cubic element with unit length as shown in Figure 4.27.

Figure 4.28 shows effective stresses normalized by preconsolidation pressure and plotted as a function of axial strain. It seems that the SMAP-3D results are very close to the analytical solution. It is interesting to note that the effective stress (σ_x') in x direction where total stress remains constant is decreasing while other effective stresses (σ_y' and σ_z') change very little.

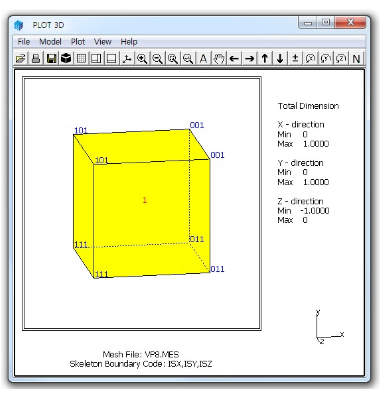


Figure 4.27 Finite element mesh

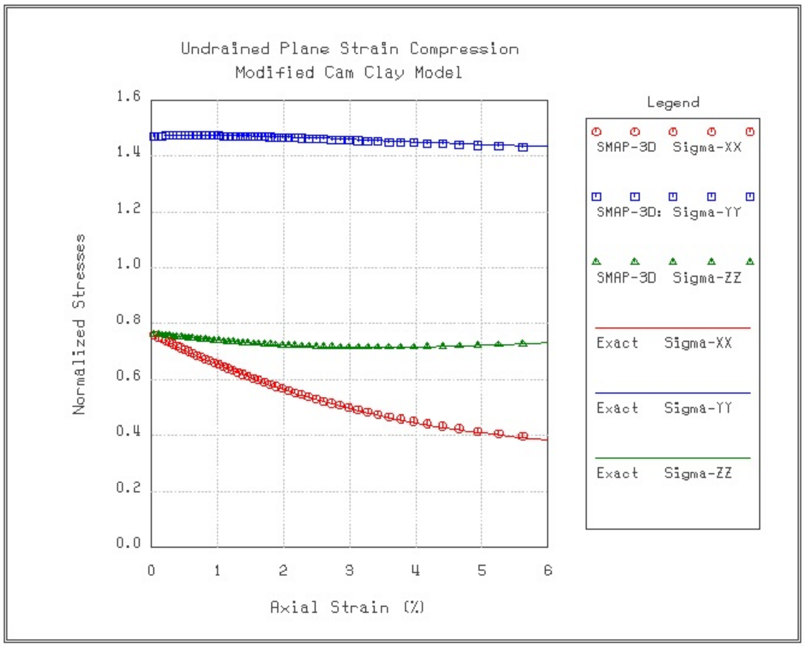


Figure 4.28 Effective stresses as a function of axial strain in K_o consolidated undrained plane strain compression test

4.9 Volumetric Creep in Isotropically Undrained Test

The problem is to check volumetric creep behavior in isotropically undrained test. The closed-form solution for this problem has been presented by Borja (1992).

$$P' = P_o \left[1 + \frac{C_c}{C_r} \left(\frac{t}{t_o} - 1 \right) \right]^{-\frac{C_r}{C_c}} \qquad \pi = P_o - P' \qquad (4.7)$$

Note that effective mean pressure (P') was P_o at initial time (t_o) but decreases with time (t) while total mean pressure (P_o) remains constant during the volumetric creep. Consequently, the excess pore pressure (π) increases with time.

The sample is modeled by a single cubic element with unit length as shown in Figure 4.29.

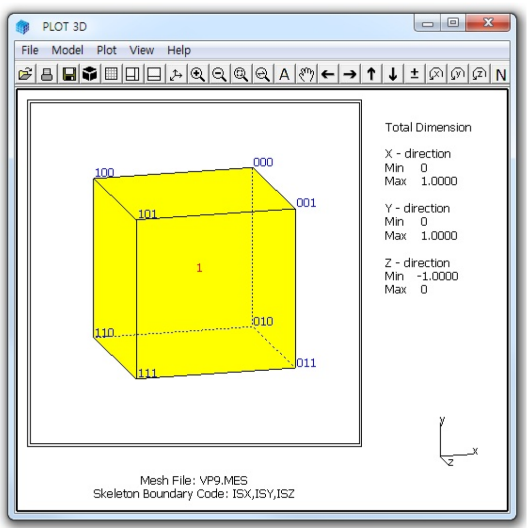


Figure 4.29 Finite element mesh

To conduct numerical calculation, the following initial conditions and material parameters are assumed:

e_o = 1.339 t_o = 1 day P_o = 1 t/m^2
C_c = 0.508 C_r = 0.254 C_α = 0.0374

Figure 4.30 shows variation of effective mean pressure and excess pore pressure as a function of time while total mean pressure remains constant. SMAP-3D results are almost identical to the closed-form solution.

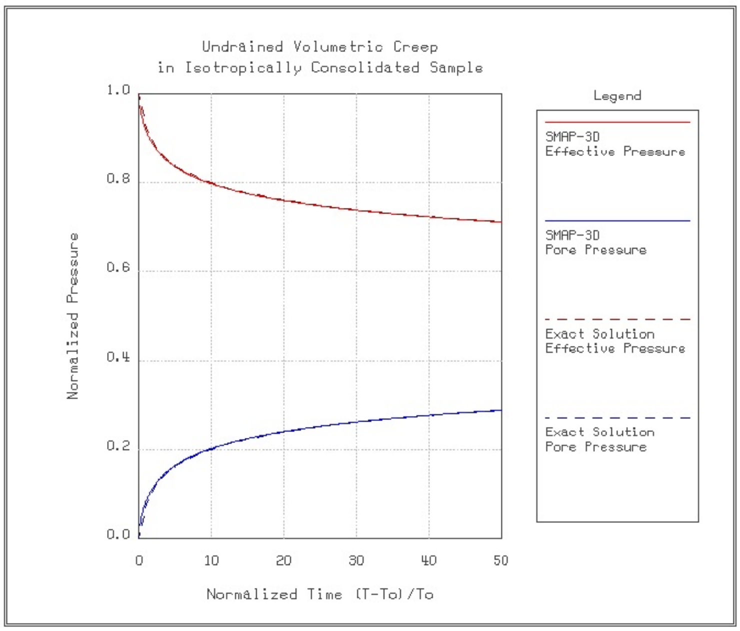


Figure 4.30 Volumetric creep in isotropically undrained test

4.10 Space Truss Analysis

This example problem is to solve the static response of space truss as shown in Figure 4.31. Block mesh example 5 illustrates how to generate this mesh. This space structure is subjected to a horizontal load along the negative z direction.

Graphical outputs are shown in Figure 4.32 for member axial forces and in Figure 4.33 for deformed shape of the structure. Note that the computed member forces are exact compared to the closed form solution.

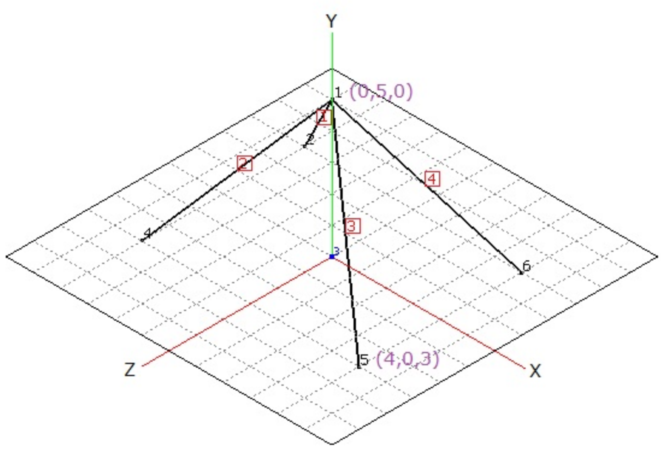


Figure 4.31 Schematic section view of space truss

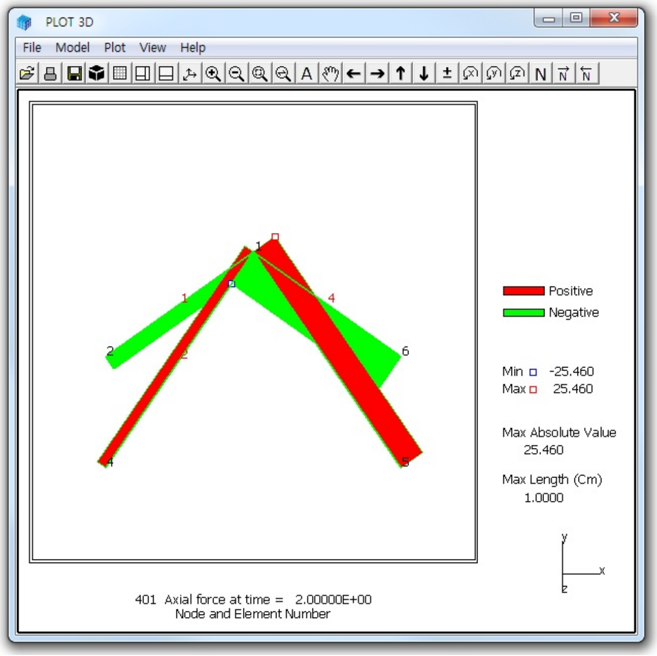


Figure 4.32 Member axial forces

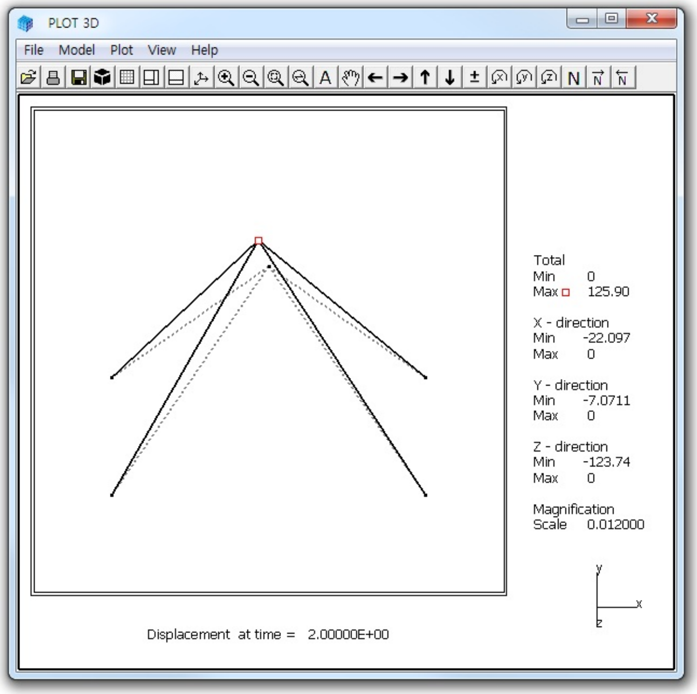


Figure 4.33 Deformed shape of space truss

4.11 Fixed End Beam Analysis

This example problem is to solve fixed end beam subjected to a concentrated load at mid span as schematically shown in Figure 4.34.

The exact solution for this beam is given below

$$\delta_{max} = \frac{PL^3}{192\,EI} = 0.01046 \text{ m} \qquad M_{max} = \frac{PL}{8} = 12.5 \text{ t-m}$$

$E = 21 \times 10^6 \text{ t/m}^2$ $\nu = 0.3$ $L = 10 \text{ m}$

$A = 0.008412 \text{ m}^2$ $I = 2.37 \times 10^{-4} \text{ m}^4$

δ_{max} = Maximum deflection at mid span

M_{max} = Maximum bending moment at mid span

The problem has been modeled by 20 beam elements as shown in Figure 4.35. Graphical outputs are plotted in Figures 4.36 and 4.37 for deformed shape and bending moment diagram, respectively. Both computed mid span deflection and maximum bending moment are the same as those of the exact solution.

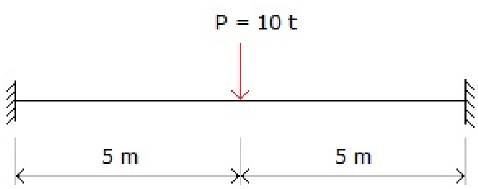


Figure 4.34 Fixed end beam subjected to concentrated load

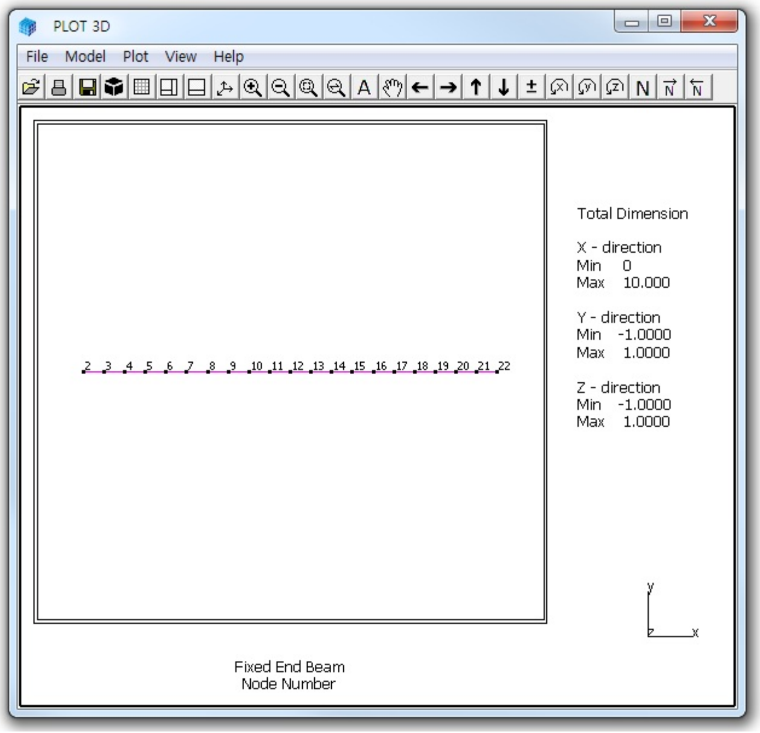


Figure 4.35 Beam node numbers

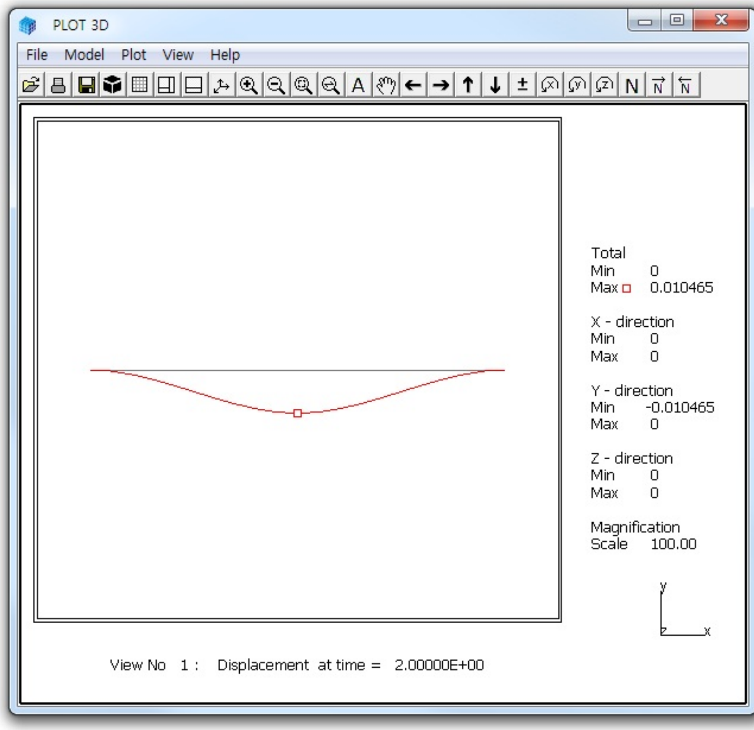


Figure 4.36 Beam deformed shape

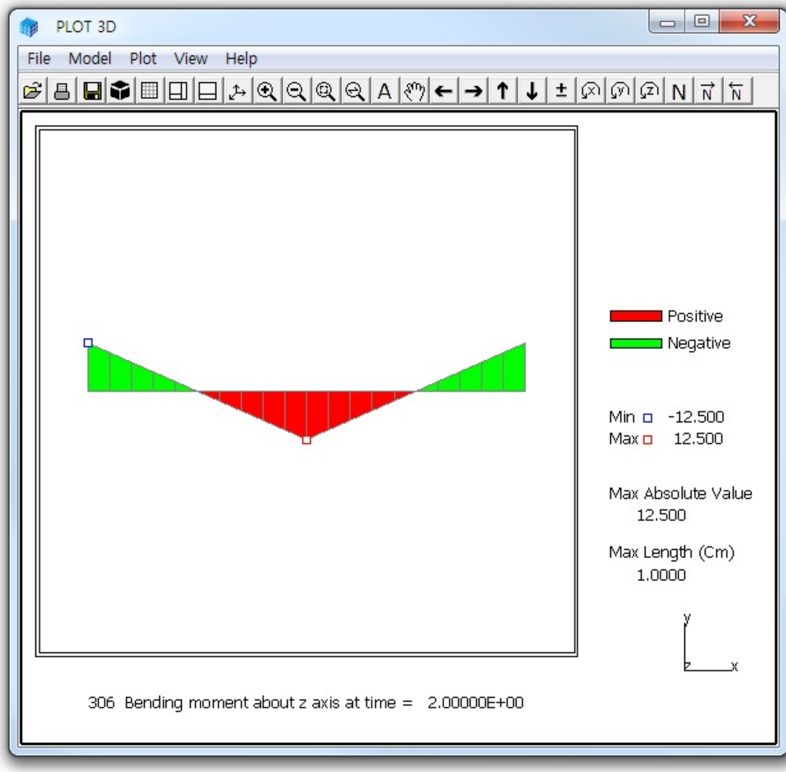


Figure 4.37 Bending moment diagram

4.12 Beam Dynamic Analysis

This example problem is to solve dynamic response of a simply supported beam subjected to a concentrated impact load at mid span. The exact solution for the deflection is given by:

$$\delta = \frac{2\, I_o\, L}{\pi^2 \sqrt{m\, E\, I}} \sum_{n=1,2}^{\infty} \frac{1}{n^2} \operatorname{Sin} \frac{n\, \pi\, x}{L} \operatorname{Sin} \frac{n\, \pi}{2} \operatorname{Sin} \omega_n t$$

$$\omega_n = n^2\, \omega_1 \qquad \omega_1 = \frac{\pi^2}{L^2} \sqrt{\frac{E\, I}{m}} \qquad m = \rho\, A$$

ρ	Mass density
A	Cross section area
L	Length of beam
I	Moment of inertia
I_o	Impulse
E	Young's modulus
x	Distance from beam support
t	Time

Numerical analysis for this simply supported beam shown in Figure 4.38 has been performed using the following parameters:

I_o = 0.1 t-sec

ρ = 0.786 t-s^2/m^4 L = 10 m

A = 0.008412 m^2 I = 2.37 x 10^{-4} m^4

E = 21 x 10^6 t/m^2 ν = 0.3

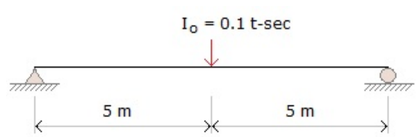


Figure 4.38 Simply supported beam subjected to impact load

The problem is modeled by 20 beam elements as shown in Fig. 4.39.
And impact load is simulated by the initial velocity applied at mid span.

Figure 4.40 shows the deformed shape at time t = 0.1 second.
Figure 4.41 shows time history plot of deflection at mid span.
SMAP-3D results agree well with the exact solution.

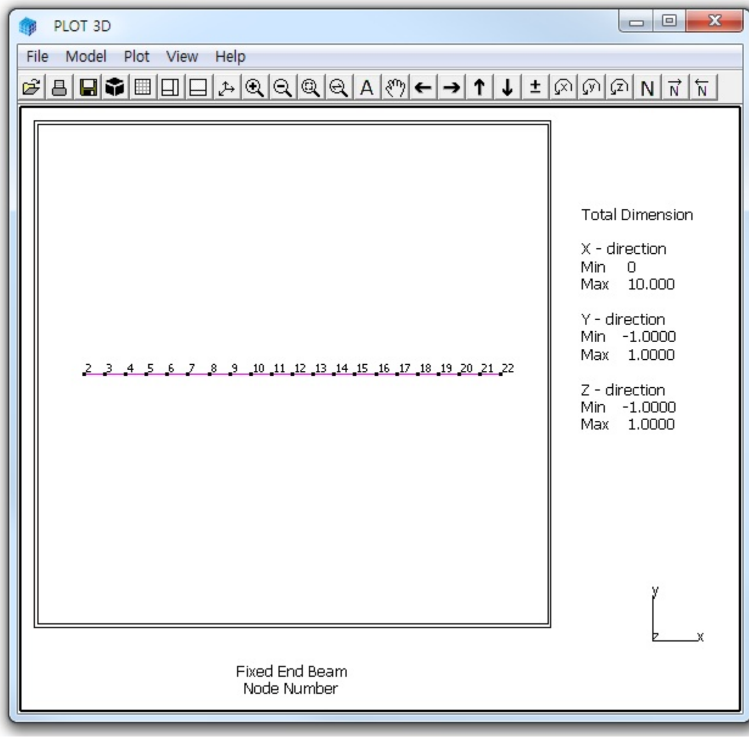


Figure 4.39 Beam node numbers

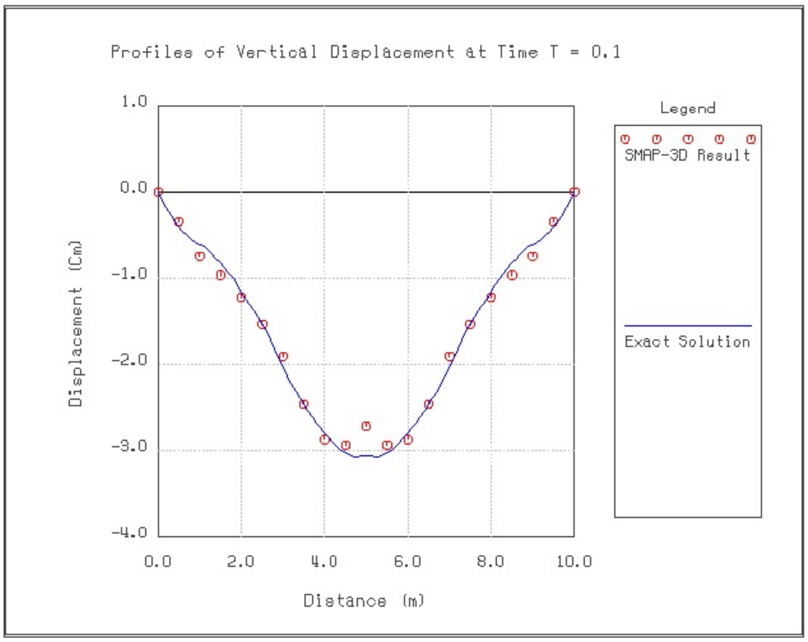


Figure 4.40 Beam deformed shape at 100 msec

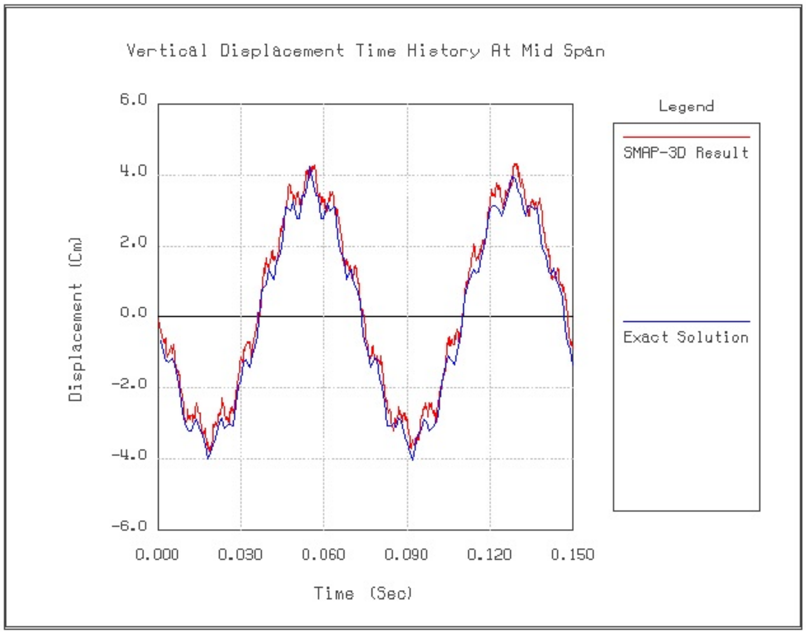


Figure 4.41 Time history of deflection at mid span

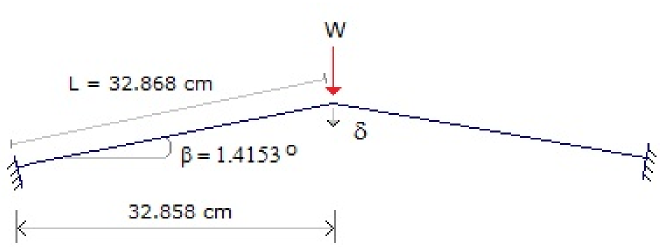
4.13 William's Toggled Beam Analysis

This classic problem of a rigidly jointed toggle is selected to verify the geometric nonlinear behavior of the continuum element.

For the toggle shown in Figure 4.42 the closed form solution as well as experimental results was obtained by Williams (Williams, F.W., An Approach to the Nonlinear Behavior of the Members of a Rigidly Jointed Plane Framework with Finite Deflections, Quarterly Journal of Mechanics and Applied Mathematics, Vol. 17, London, UK, 1964, pp. 451-469)

This toggled structure is modeled by 400 continuum finite elements: 100 elements along the beam axis, 4 elements across the depth, and only 1 element through the thickness.

Figures 4.43 and 4.44 show the load-deflection response at mid span and deformed shape at applied load of 16 kg, respectively.
SMAP-3D results are very close to the Williams' closed form solution.



$E = 7.238 \times 10^6 \ t/m^2$

$I = 3.746 \times 10^{-2} \ cm^4$ $\quad A = 1.18 \ cm^2$

Figure 4.42 William's toggled beam (Not Scaled)

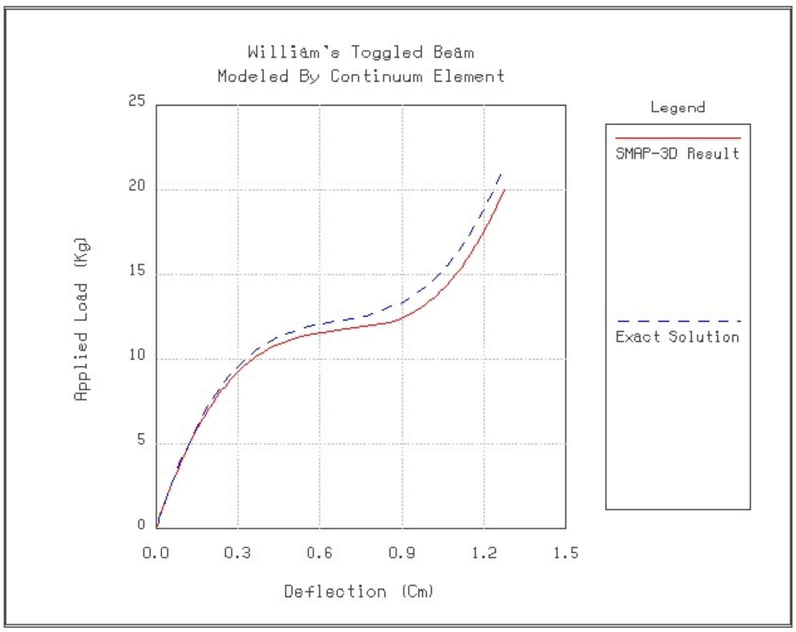


Figure 4.43 Load-deflection curve at mid span

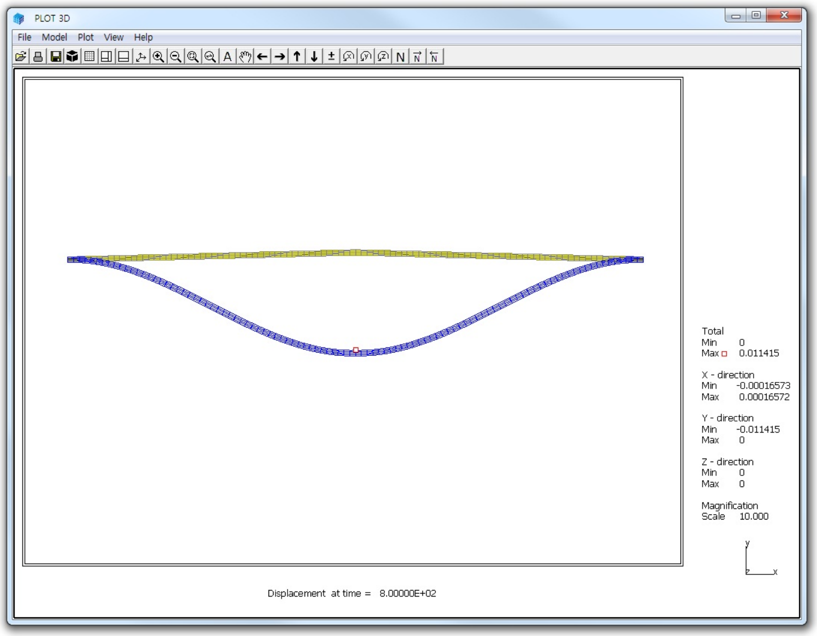


Figure 4.44 Deformed shape at applied load of 16 kg

4.14 Plane Strain Tunnel Analysis

The objective of this problem is to verify generation of in situ stresses and interaction of a tunnel liner with the surrounding soils. This example problem has been presented in SMAP-S2. Figure 4.45 shows schematic tunnel section view and material properties of soil and steel liner.

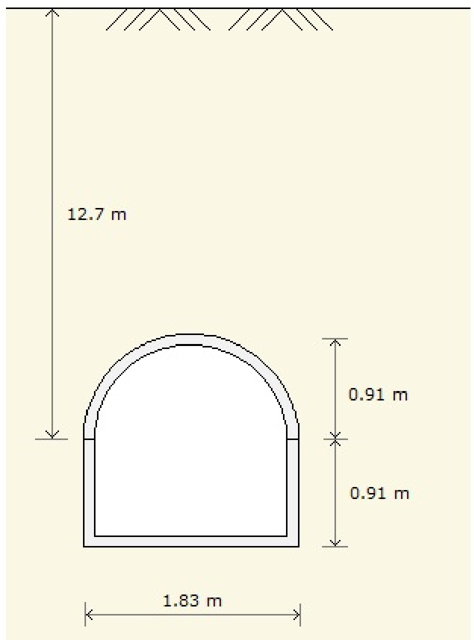
Figure 4.46 shows Finite element mesh. By symmetry, only the right half of the tunnel is modeled. Tunnel liner is modeled by shell elements as shown in Figure 4.47. Block mesh example 4 illustrates how to generate this mesh.

The first two load steps were used to generate in situ stresses. Tunnel excavation and liner installation were simulated by deactivating soil elements within the tunnel and activating liner elements at the third load step.

Graphical results are presented in the following order:

- Figure 4.48 Tunnel deformed shape
- Figure 4.49 Tunnel liner bending moment
- Figure 4.50 Tunnel liner axial stress
- Figure 4.51 Principal stress vector
- Figure 4.52 Major principal stress distribution
- Figure 4.53 Minor principal stress distribution

SMAP-3D results are almost identical to SMAP-S2 results



Soil		Steel Liner	
E	= 7,030 t/m^2	E	= 20.4 x10^6 t/m^2
ν	= 0.23	ν	= 0.3
K_o	= 0.3	t	= 2.54 cm (Thickness)

Figure 4.45 Schematic tunnel section view (Not Scaled)

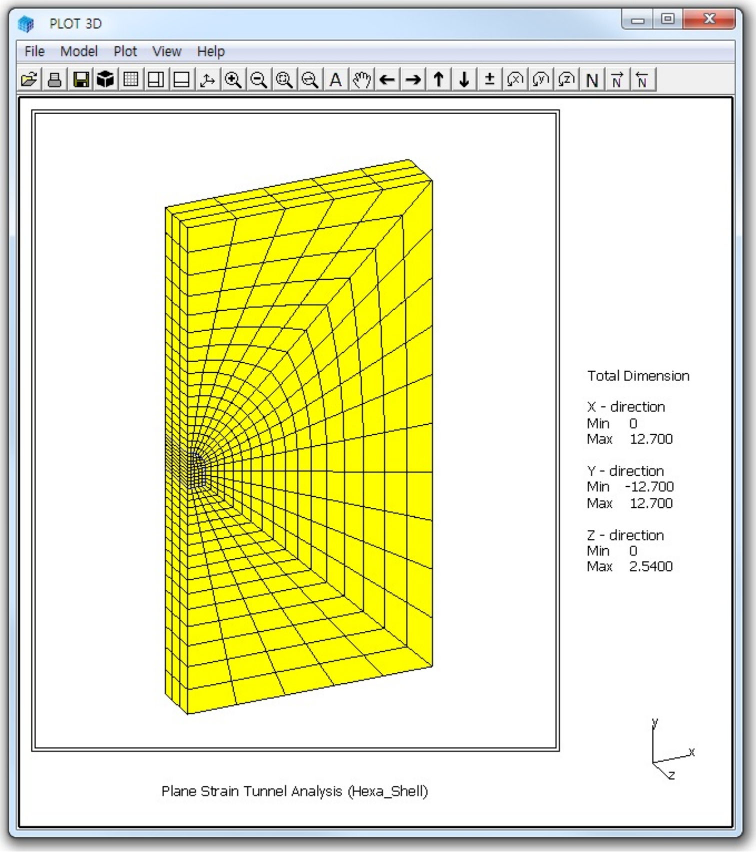


Figure 4.46 Finite element mesh

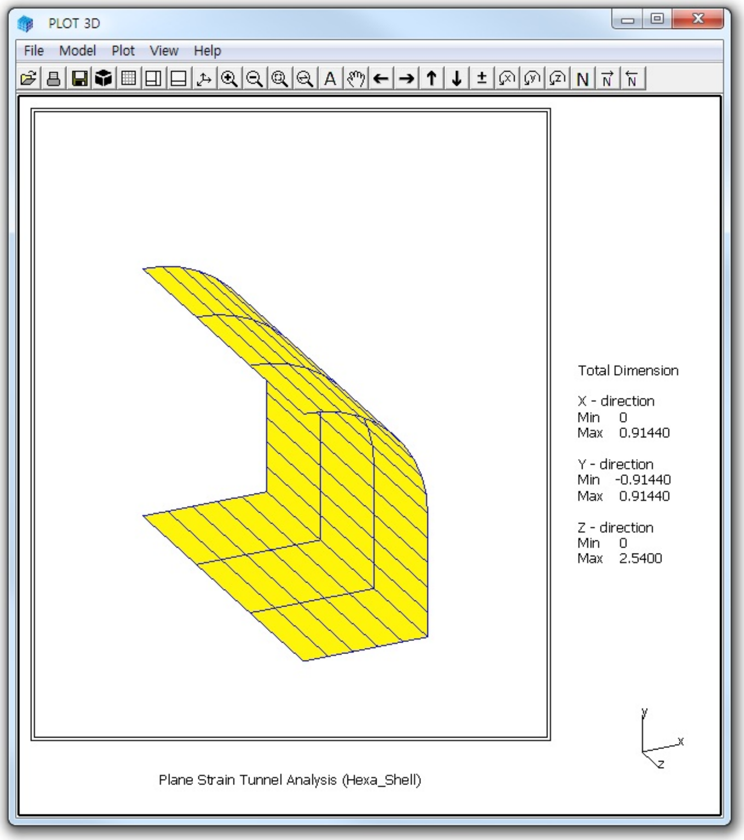


Figure 4.47 Finite element mesh for liner

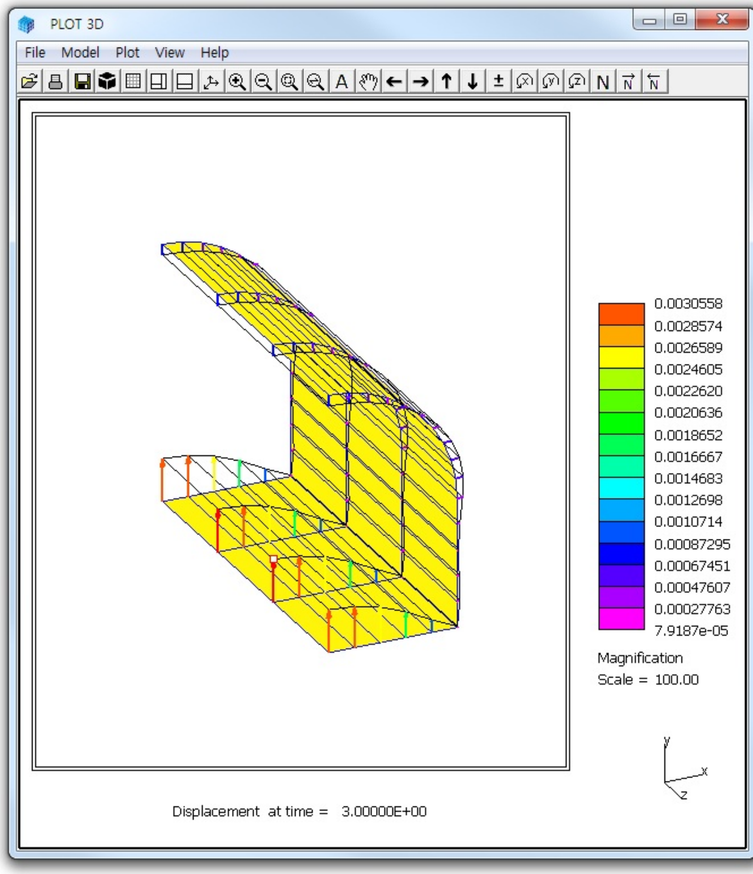


Figure 4.48 Tunnel deformed shape

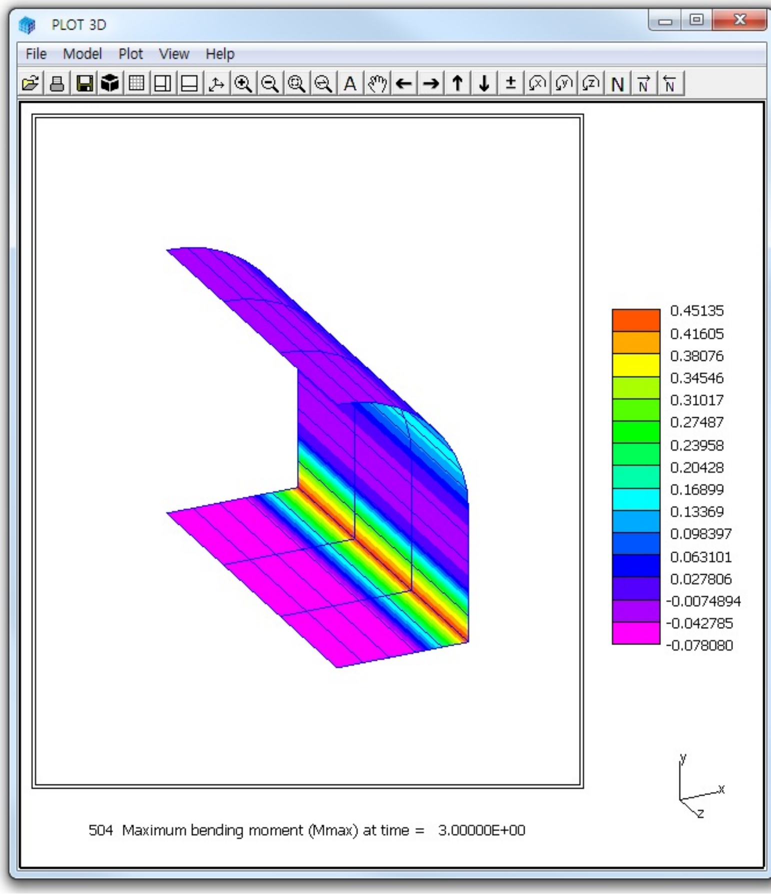


Figure 4.49 Tunnel liner bending moment

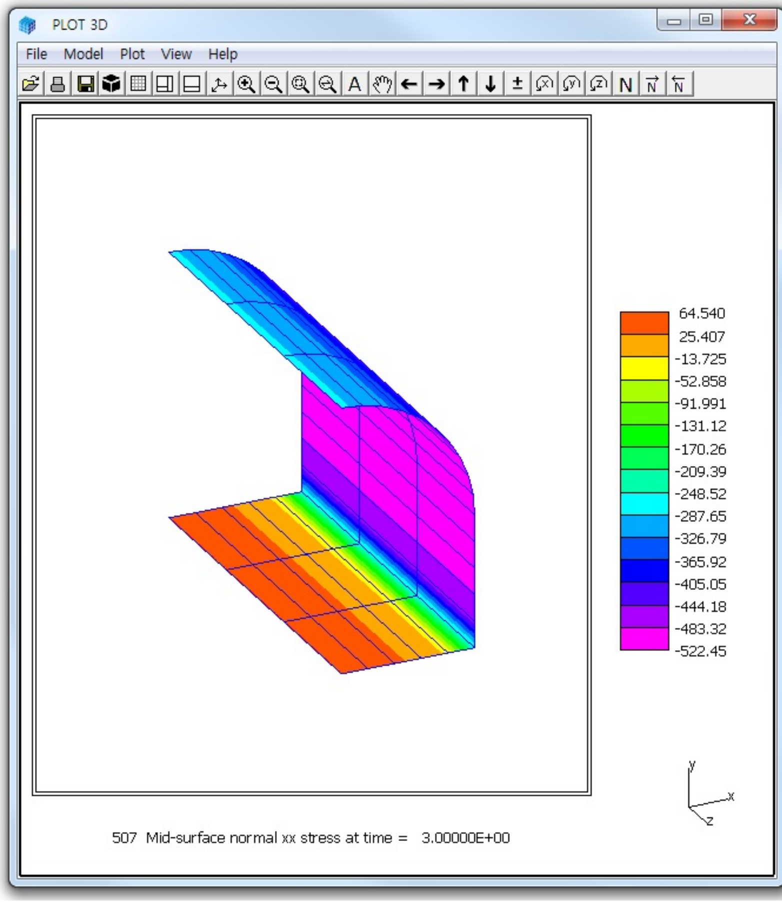


Figure 4.50 Tunnel liner axial stress

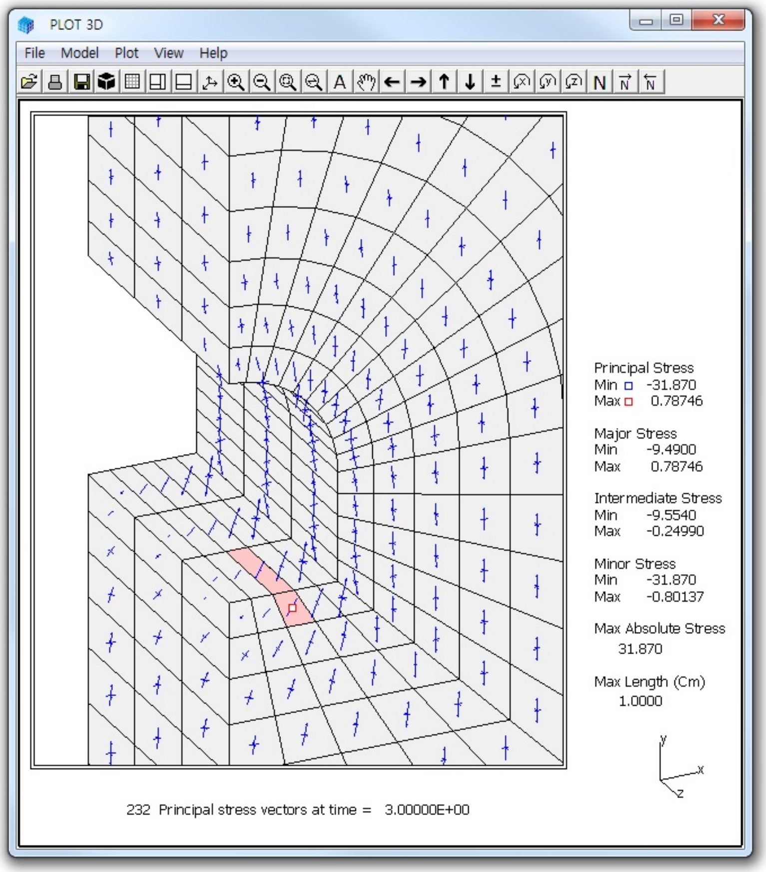


Figure 4.51 Principal stress vector

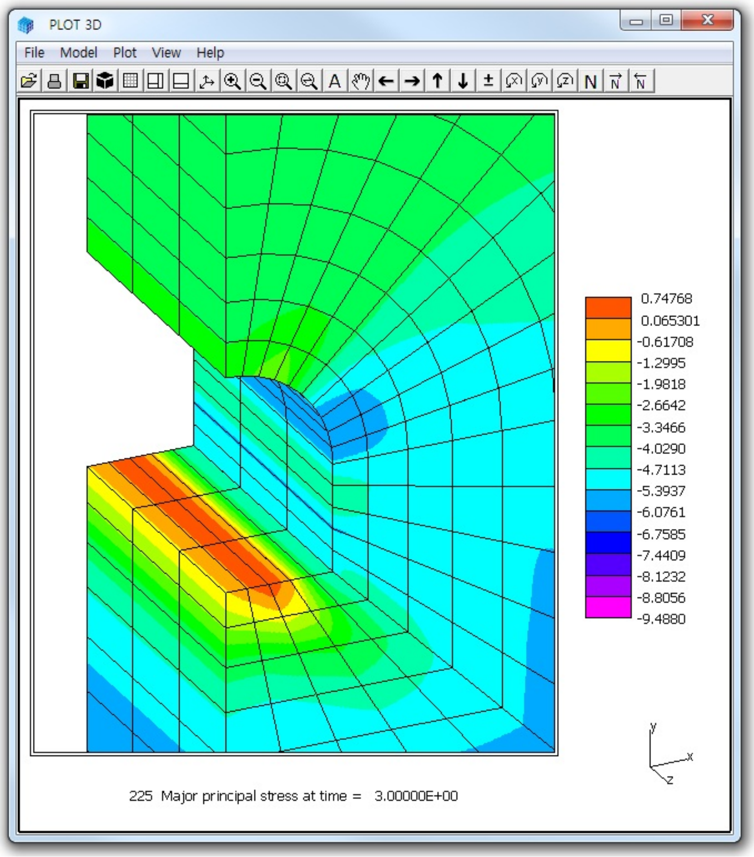


Figure 4.52 Major principal stress distribution

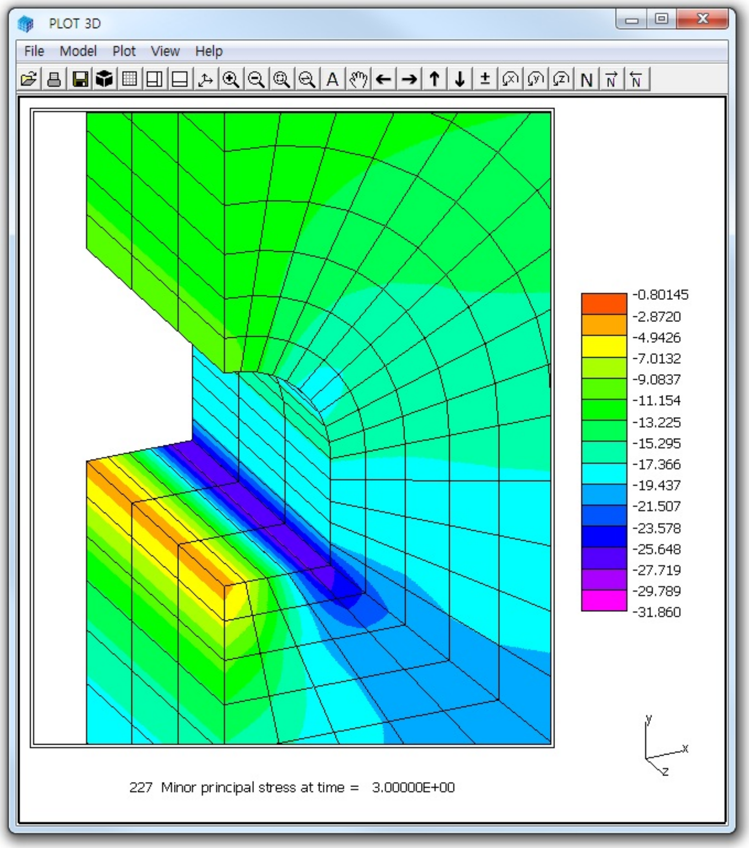


Figure 4.53 Minor principal stress distribution

4.15 Hemispherical Shell

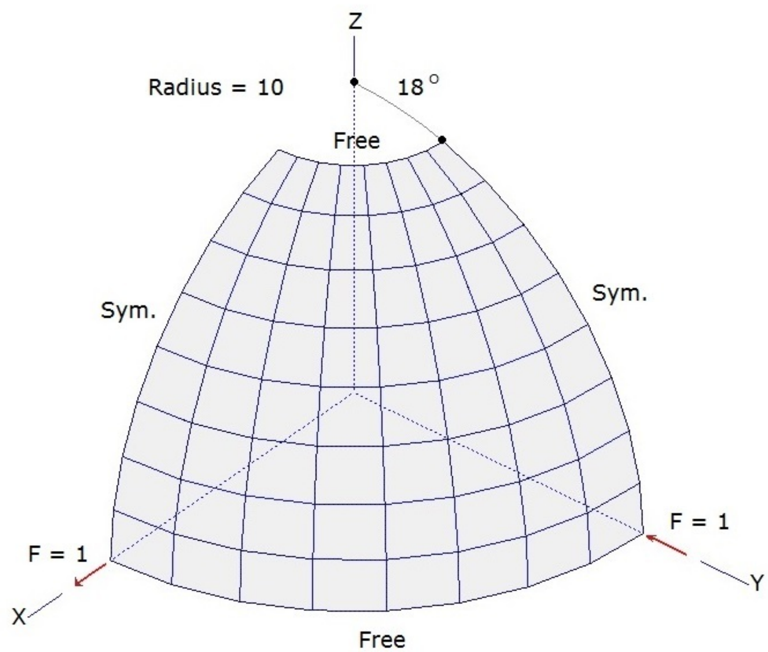
This classic problem of a hemispherical shell with 18˚ hole is selected to verify accuracy of the membrane and bending performance of shell element.

The theoretical solution for this problem was presented by R. H. MacNeal and R. L. Harder (A proposed standard set of problems to test finite element accuracy, Finite Element Anal. Des., 1, 3-20, 1985).

Figure 4.54 shows finite element mesh, material properties, loading and boundary conditions. By symmetry, only a quadrant of the shell is modeled. Block mesh example 3 illustrates how to generate this mesh.

Graphical results are presented in the following order:
Figure 4.55 Deformed shape
Figure 4.56 Maximum bending moment

SMAP-3D result gives excellent results for the displacement at the point of load in the direction of load as compared below:
Theoretical solution = 0.094
SMAP-3D result = 0.0944



$E = 6.825 \times 10^7$ $\nu = 0.3$ Thickness = 0.04

Figure 4.54 Hemispherical shell subjected to pinched point loads

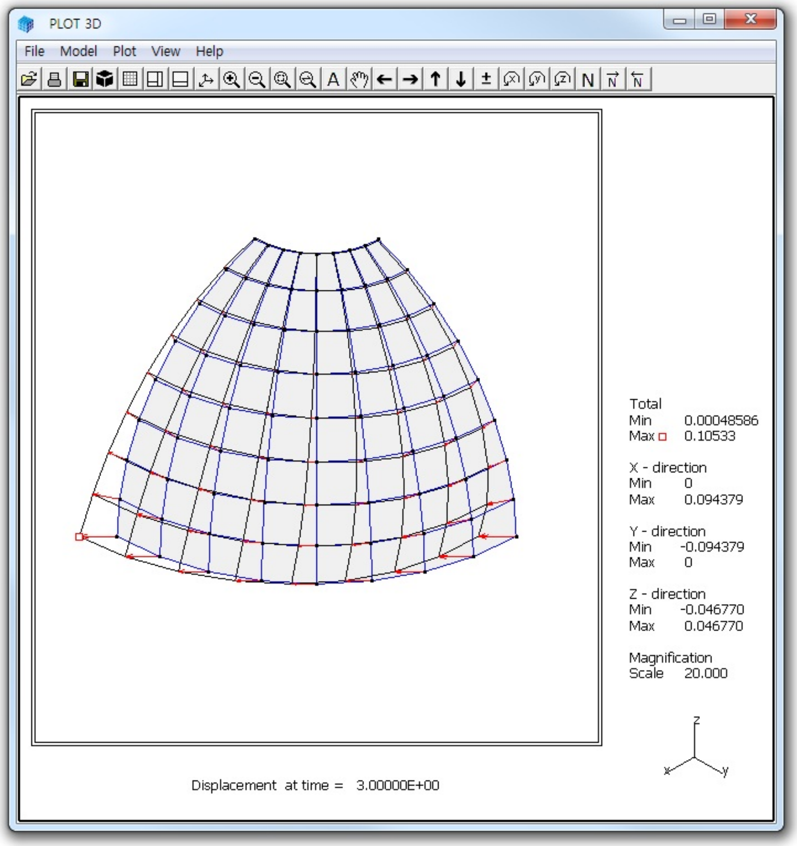


Figure 4.55 Deformed shape

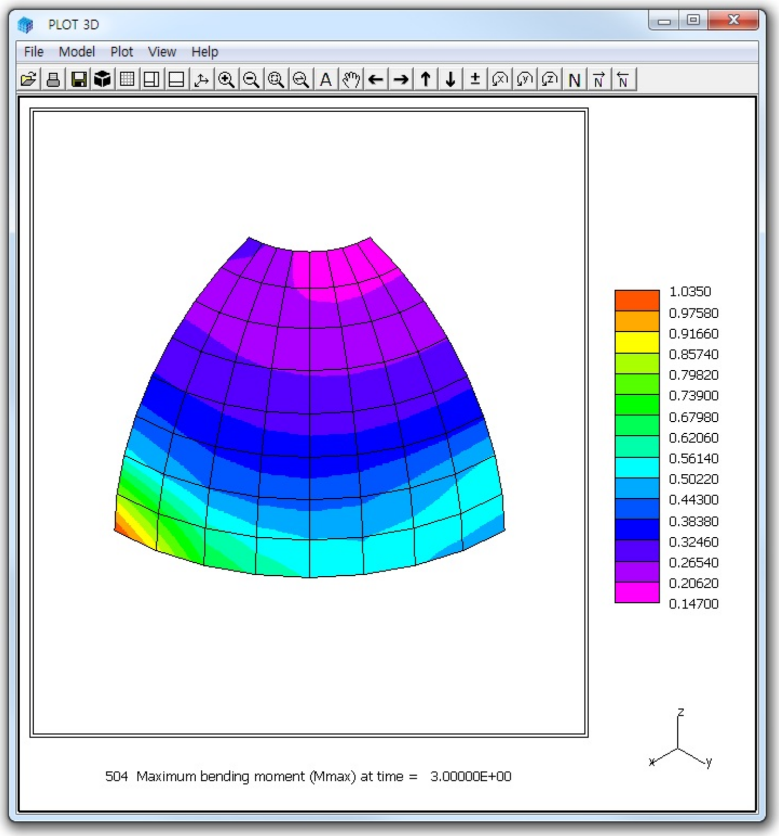


Figure 4.56 Maximum bending moment

4.16 Simply Supported Plate Analysis

A simply supported rectangular plate, shown in Figure 4.57, is selected to verify the dynamic response of shell element. By symmetry, only a quarter of the plate is modeled. The plate is subjected to a concentrated step load at center.

The computed displacement time history at plate center is shown in Figure 4.58 along with static results. SMAP-3D solution shows good results with such a relatively coarse mesh:

Static vertical displacement at plate center

Kirhhoff theory = 0.925 cm
SMAP-3D result = 0.942 cm

Period of the first mode

Kirhhoff theory = 0.2366 sec
SMAP-3D result = 0.237 sec
(Estimated from Figure 4.58)

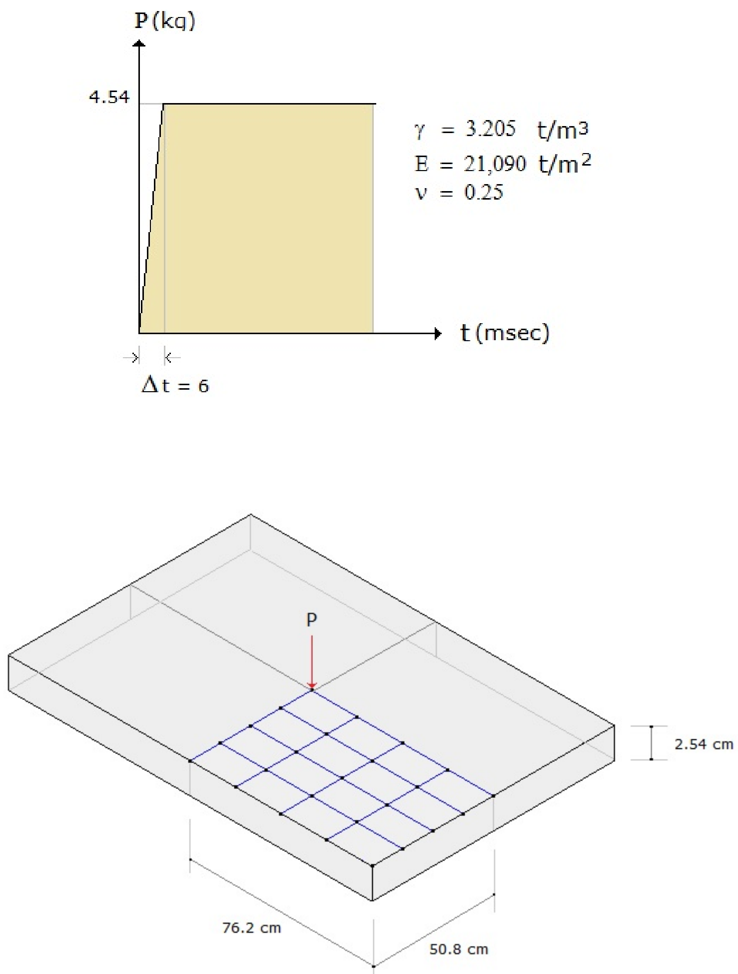


Figure 4.57 Simply supported plate (Not Scaled)

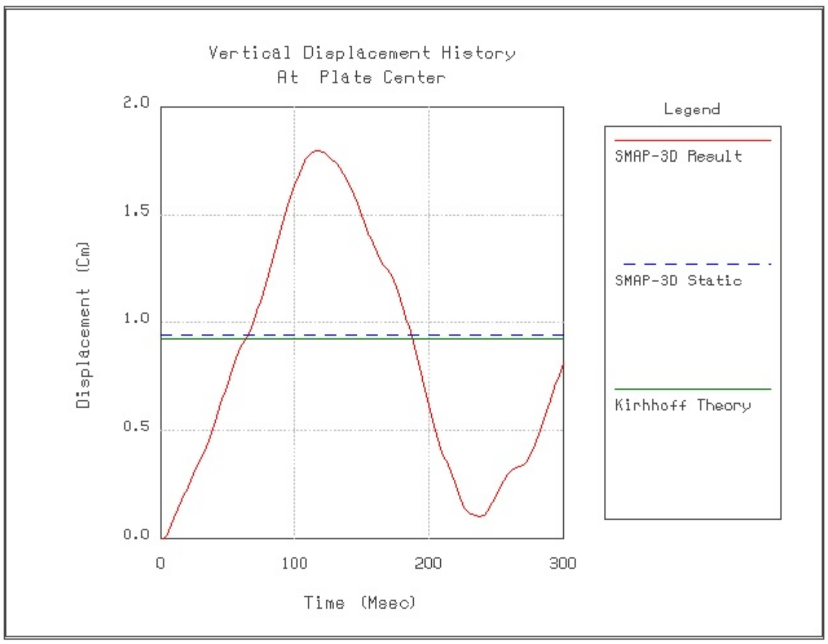


Figure 4.58 Vertical displacement time history at plate center

4.17 Heated Beam Analysis

A Simply supported plain concrete beam, shown schematically in Figure 4.59, is subjected to linear temperature increase through depth.

The temperature of the top surface of the beam is increased from -30° C to 50° C while temperature of the bottom surface remains constant at -30° C. Consequently, it is expected that the top surface expands relative to the bottom surface and the beam deflects upwards.

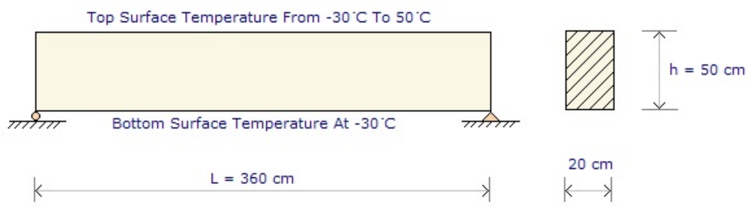


Figure 4.59 Heated beam subjected to temperature difference

By symmetry, only right half of the beam is modeled using a total of 22 shell elements as shown in Figure 4.60.

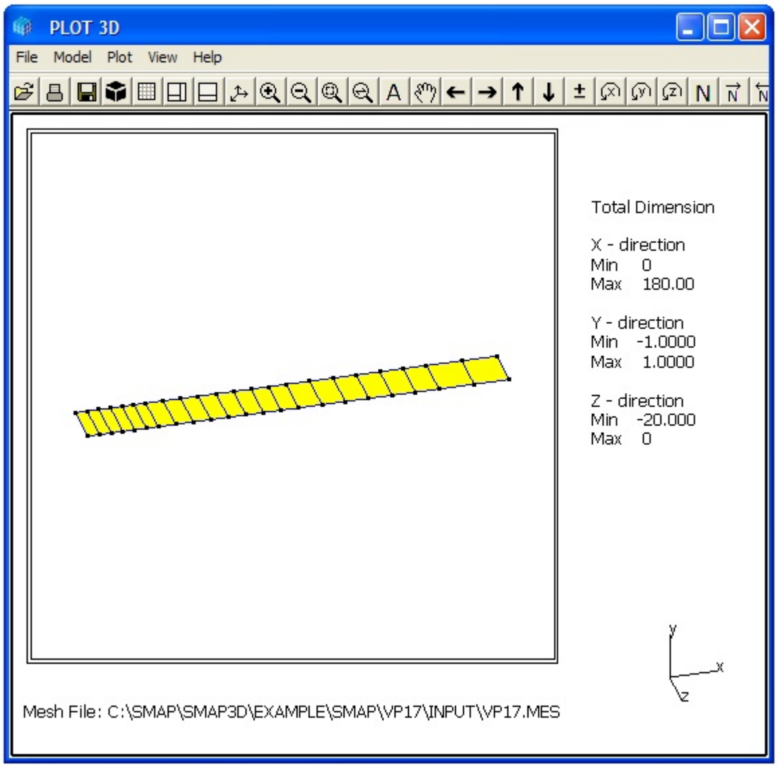


Figure 4.60 Finite element mesh

Material Properties are assumed as:

$\alpha = 3.2 \times 10^{-5}\ ^{\circ}C^{-1}$ $E = 2.7 \times 10^{5}$ kg/cm^2 $\nu = 0.15$

Theoretical Maximum Deflection is given as:

$\delta_{max} = \alpha L^2 (T_{top} - T_{bottom})/(8\ h) = 0.8294$ cm

Figure 4.61 shows beam deflections. SMAP-3D result gives excellent results for vertical displacement at center of the beam.

Theoretical solution = 0.8294 cm

SMAP-3D solution = 0.8276 cm

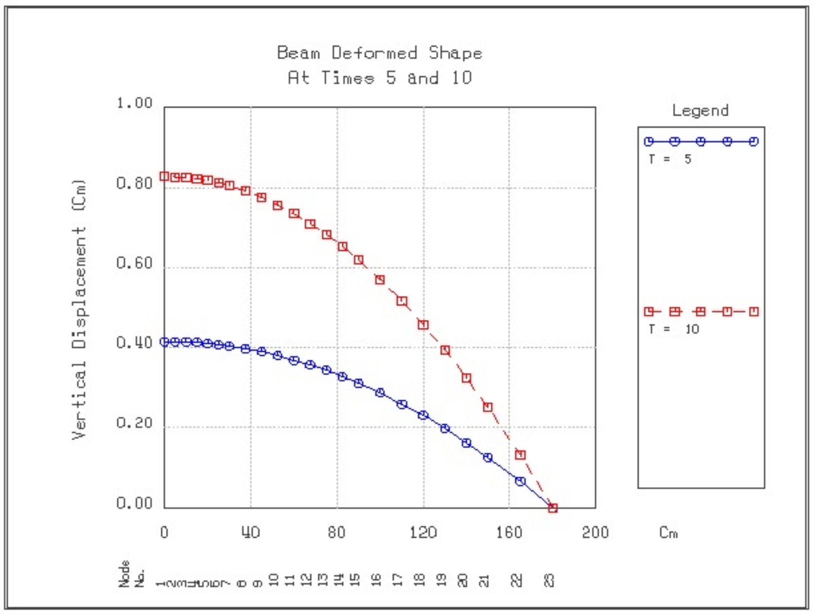


Figure 4.61 Beam deformed shapes

4.18 Thin Pipe Subjected To Internal Pressure

A very thin steel pipe, with radius of 20 cm and thickness of 0.003 cm, is subjected to the internal pressure of 0.2 kg/cm^2. The pipe is assumed to be in plain strain condition in the axial direction. Theoretically, the pipe is radially expanding due to the in-plane (membrane) deformations.

A total of 32 Shell elements is used to model the circular pipe as shown in Figure 4.62. A constant internal pressure is regarded as the hydrostatic pressure acting on the inner surface of Shell element.

Since the bending stiffness of the pipe is proportional to the third power of the pipe thickness while the in-plane stiffness is linearly proportional to the pipe thickness, the bending stiffness in such a very thin pipe would be much smaller than in-plane stiffness.

Thus, even a very small force associated with the bending degrees of freedoms may induce unrealistically large displacement. To improve the accuracy of displacement result, bending stiffness is multiplied by a factor of 100000.

The theoretical elastic solution gives the following radial displacement (u_r) and the hoop stress (σ_θ):

$$u_r = \frac{P \cdot r^2}{E \cdot t}(1 - v^2) \qquad \sigma_\theta = \frac{P \cdot r}{t}$$

where

E	Young's modulus	v	Poisson's ratio
t	Thickness of pipe	r	Radius of pipe
p	Internal pressure		

Numerical parameters are assumed as:

E	=	2.0×10^6 kg/cm^2	v = 0.3
t	=	0.003 cm	r = 20 cm
p	=	0.2 kg/cm^2	

Figure 4.63 shows deformed shape of pipe and Figure 4.64 shows hoop stress developed in the pipe.

SMAP-3D results are very close to theoretical solutions as listed below:
Theoretical solution: u_r = 0.01213 cm σ_θ = 1,333 kg/cm^2
SMAP-3D result: u_r = 0.01207 cm σ_θ = 1,327 kg/cm^2

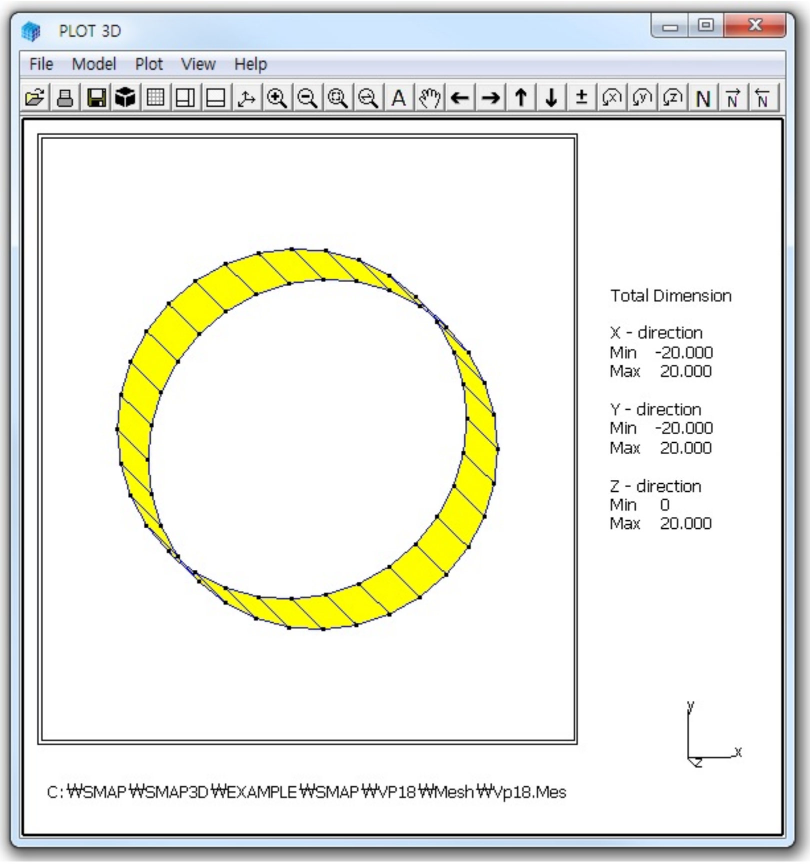


Figure 4.62 Finite element mesh

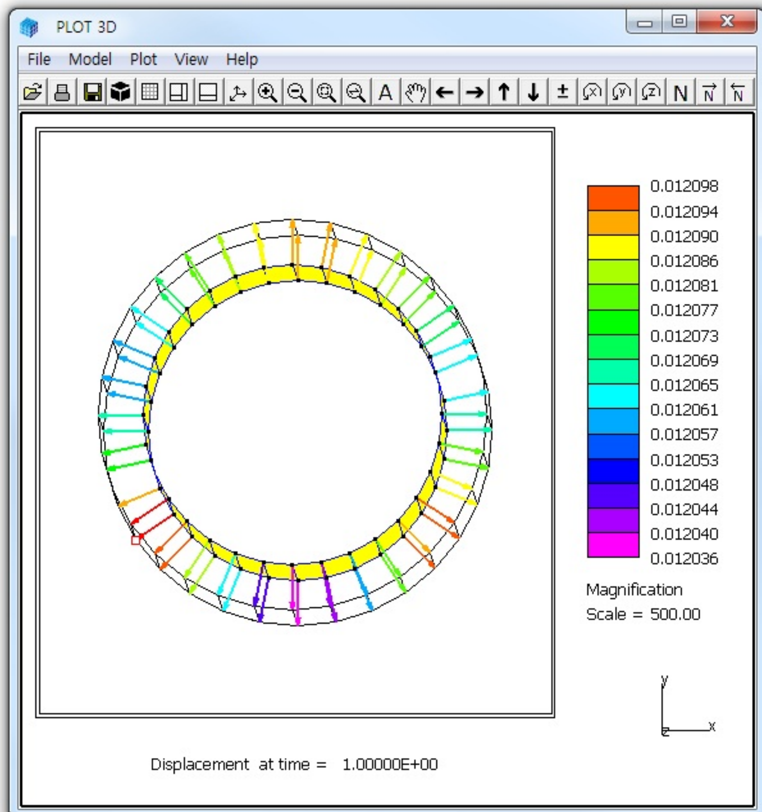


Figure 4.63 Pipe deformed shape

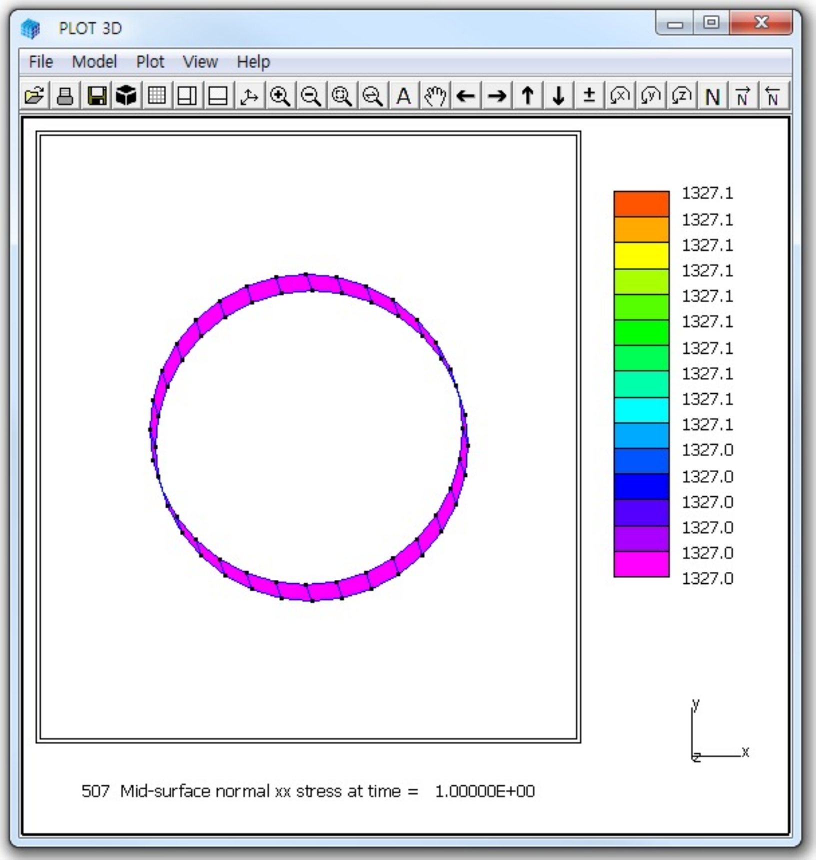


Figure 4.64 Pipe hoop stress

4.19 Preload Consolidation and Excavation

This example problem is to illustrate the analysis of the slope to be constructed under sea water. The in situ soil consists of about 40 meters of soft clay layer overlying hard soil layers.

Figure 4.65 shows schematically four stages of preloading embankment construction followed by excavation up to 17.6 meters below sea level.
Before preloading embankment, material zones 4, 5, 7, 8, 12 and 13 shown in Figure 4.66 are to be improved by drain methods (sand drain and PDB). In situ and improved soil properties are listed in Table 4.1.

The rate of embankment construction and excavation is shown schematically in Figure 4.67 along with computational steps used for SMAP-3D analysis.

Finite element meshes used for the analysis are shown:
- Figure 4.68 Finite element mesh
- Figure 4.69 Finite element mesh around preload
- Figure 4.70 Finite element mesh at completion
- Figure 4.71 Finite element mesh around slope

A total of 2330 elements is used to model a sequence of embankment construction and excavation.

Computed results at 152 days after completion of excavation are plotted by PLOT-3D in the following order:
- Figure 4.72 Deformed shape around slope
- Figure 4.73 Horizontal displacement distribution
- Figure 4.74 Pore pressure distribution
- Figure 4.75 Effective mean pressure distribution
- Figure 4.76 Deviatoric stress distribution

The horizontal contour lines of the hydrostatic water pressure in Figure 4.74 indicates that there will be no further consolidation settlement at 152 days after completion of excavation. Figure 4.76 shows that deviatoric stresses are concentrated around the base of the slope. Looking at both effective mean pressure (p') and deviatoric stress (q), the value of stress ratio (q/p′) is less than one at locations approximately 3 meters away from the surface of slope.

Figure 4.77 shows the location of selected elements where time histories of stresses and stress path are plotted. These selected elements are located within 10 meters from the surface of slope.

Computed results of time history of stresses are plotted by PLOT-XY in the following order:

Figure 4.78 Stress time history at element 120
Figure 4.79 Stress path at element 120

It should be noted that first 2000 days are used to generate in situ k_0 stresses. During embankment construction, excess pore water pressures develop mostly immediately after placement and then dissipate with time while effective stresses develop gradually. During excavation, effective stresses undergo unloading stress paths which will end up with higher horizontal stresses in over consolidated soil condition and pore water pressures drop rapidly and then get gradually back to the hydrostatic water pressure level as the dissipation length is shorter.

It is worth noting that the effective mean pressures decrease slightly while deviatoric stresses increase during the short period of placement of preloading fills. This is due to the fact that the compressive plastic volumetric strains develop while the total volumetric strains remain nearly constant since very little excess pore pressure dissipations are expected in such a short period.

Examining all the stress path plots, elements 120, 299, 477, 655 and 833 lie on the failure surface and elements 300 and 478 are slightly below the failure surface. Noting that elements 120, 299, 477, 655 and 833 are located within 2 meters from the surface of slope and elements 300 and 478 are located within 4 meters from the surface of slope, it is expected that soil failure would occur around the slope base within approximately 3 meters from the surface of slope. It may require redesign of the slope or accompany engineered structures for the slope to stay in safe.

Table 4.1 Material model parameters

Elastic Model Parameters

Material Number	Porosity (%)	Specific Gravity	k (m/day)	E (t/m^2)	v	Remark
1	42	2.7	0.0864	600	0.33	Dry
2	42	2.7	0.0864	600	0.33	Dry
3	42	2.7	0.0864	600	0.33	Saturated
6	44	2.7	0.0864	1400	0.33	Saturated
14	99.9	2.7	10.0	10.0	0.2	Water

Modified Cam-Clay Model Parameters

Material Number	Porosity (%)	Specific Gravity	k (m/day)	e_o	C_c	C_r	M
4	59.1	2.72	* 0.0274	1.49	0.55	0.077	1.2
5	61.0	2.72	* 0.0274	1.57	0.70	0.098	1.2
7	59.1	2.72	* 0.0274	1.49	0.55	0.077	1.2
8	61.0	2.72	* 0.0274	1.57	0.70	0.098	1.2
9	59.1	2.72	4.32×10^{-5}	1.49	0.55	0.077	1.2
10	61.0	2.72	4.32×10^{-5}	1.57	0.70	0.098	1.2
11	61.0	2.72	4.32×10^{-5}	1.62	0.80	0.112	1.2
12	61.0	2.72	* 0.0274	1.62	0.80	0.112	1.2
13	61.0	2.72	* 0.0274	1.62	0.80	0.112	1.2

(*) Soil permeability improved by sand drain or PDB

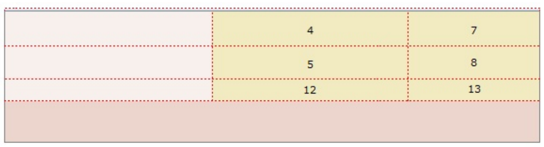
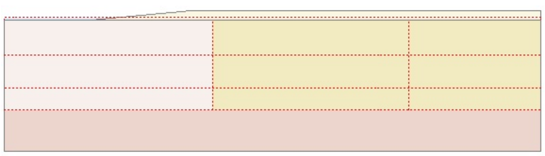
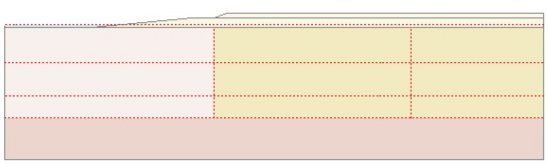
Construction State	Description
Step 101 (2000 days)  	In Situ State Sand Drain: Material 4, 5, 12 PDB: Material 7, 8, 13
Step 104 (+ 15 days) 	Completion of Stage 1 Embankment
Step 165 (+ 321 days) 	Completion of Stage 2 Embankment

Figure 4.65 Construction sequence

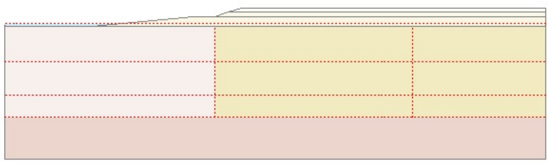
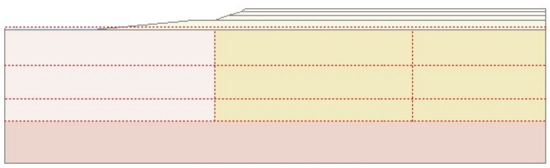
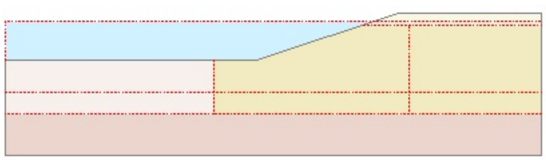
Construction State	Description
Step 216 (+ 627 days) 	Completion of Stage 3 Embankment
Step 267 (+ 933 days) 	Completion of Stage 4 Embankment
Step 333 (+ 1265 days) 	Completion of Final Excavation (Dredging)

Figure 4.65 Construction sequence (Continued)

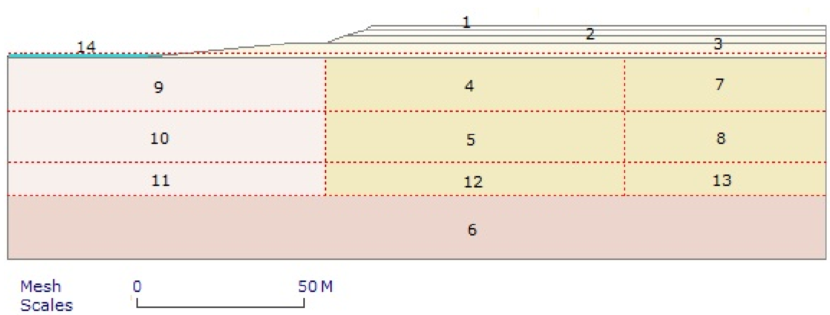


Figure 4.66 Material number

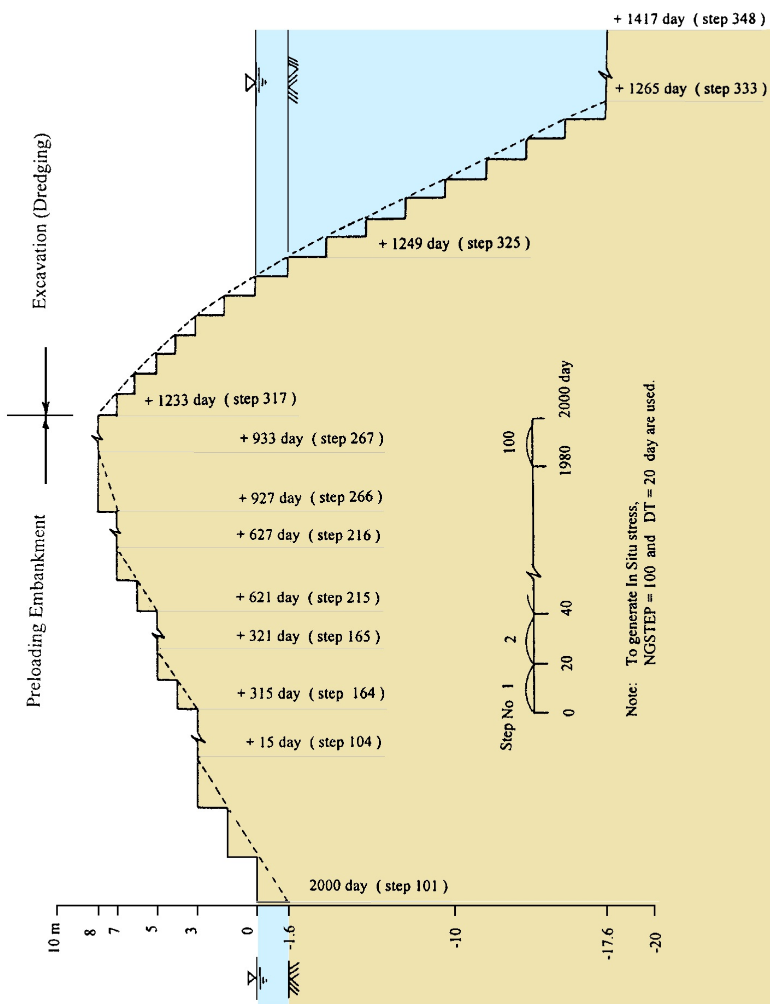


Figure 4.67 Construction rate

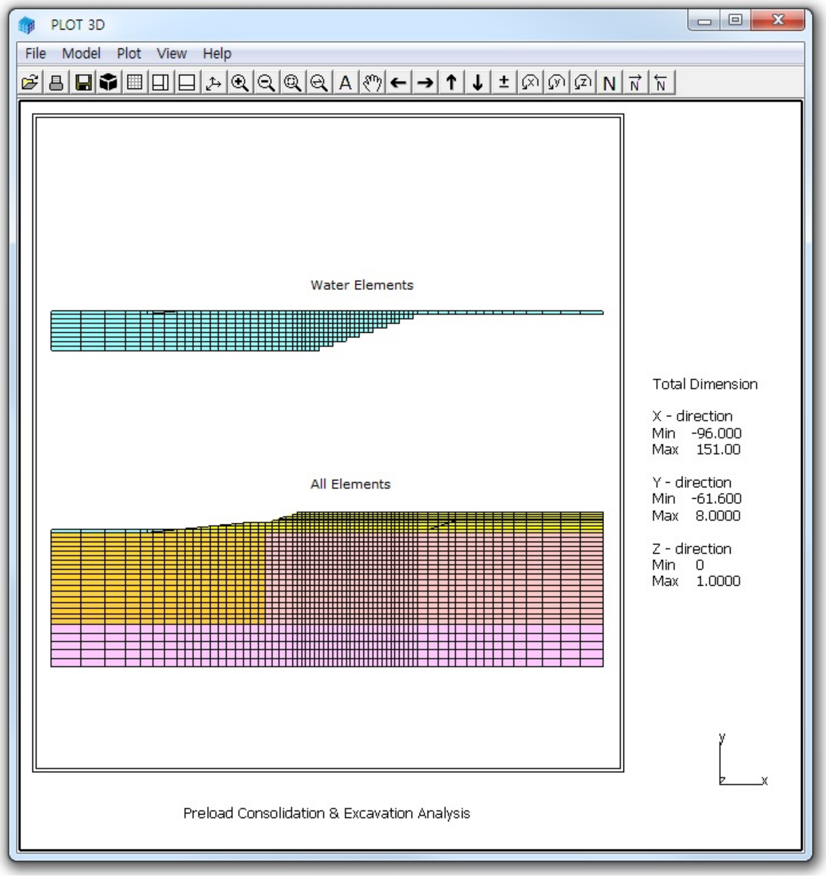


Figure 4.68 Finite element mesh

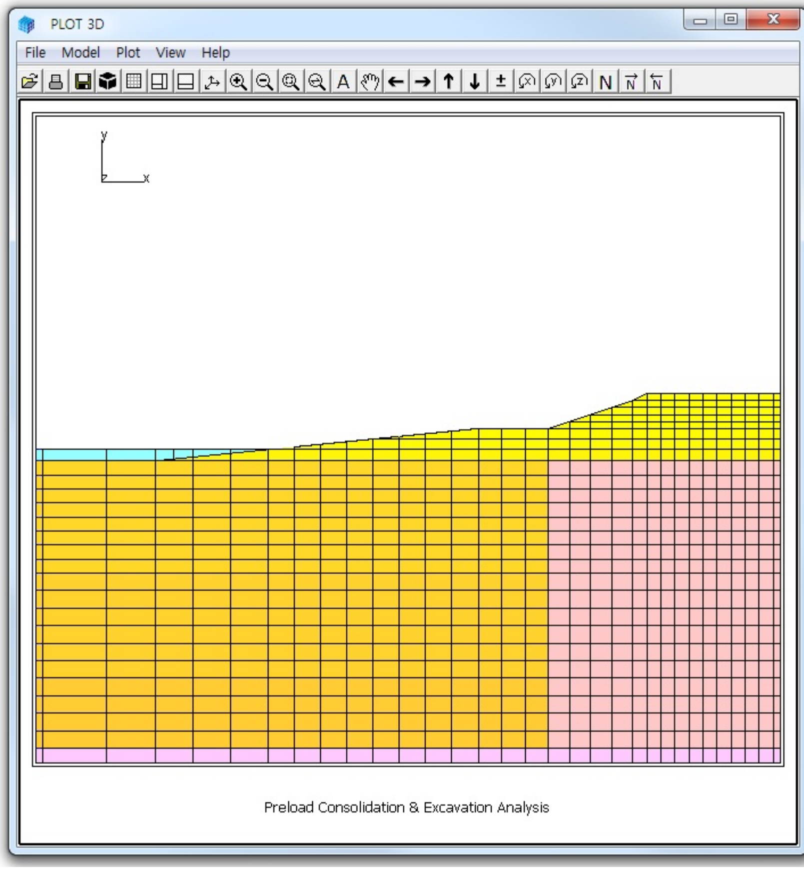


Figure 4.69 Finite element mesh around preload

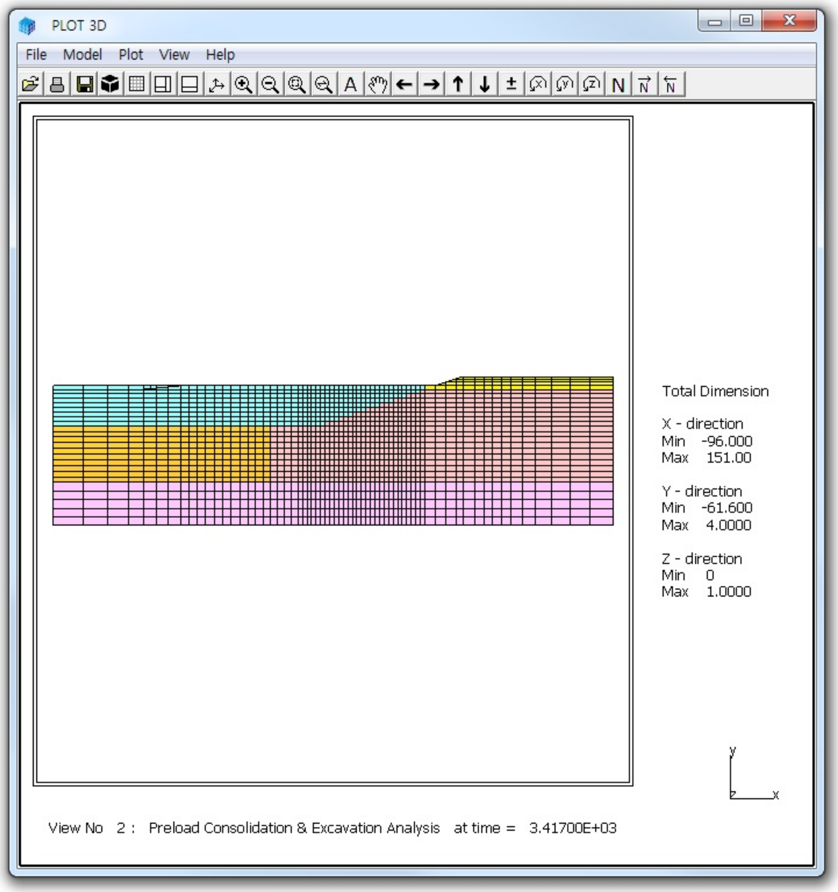


Figure 4.70 Finite element mesh at completion

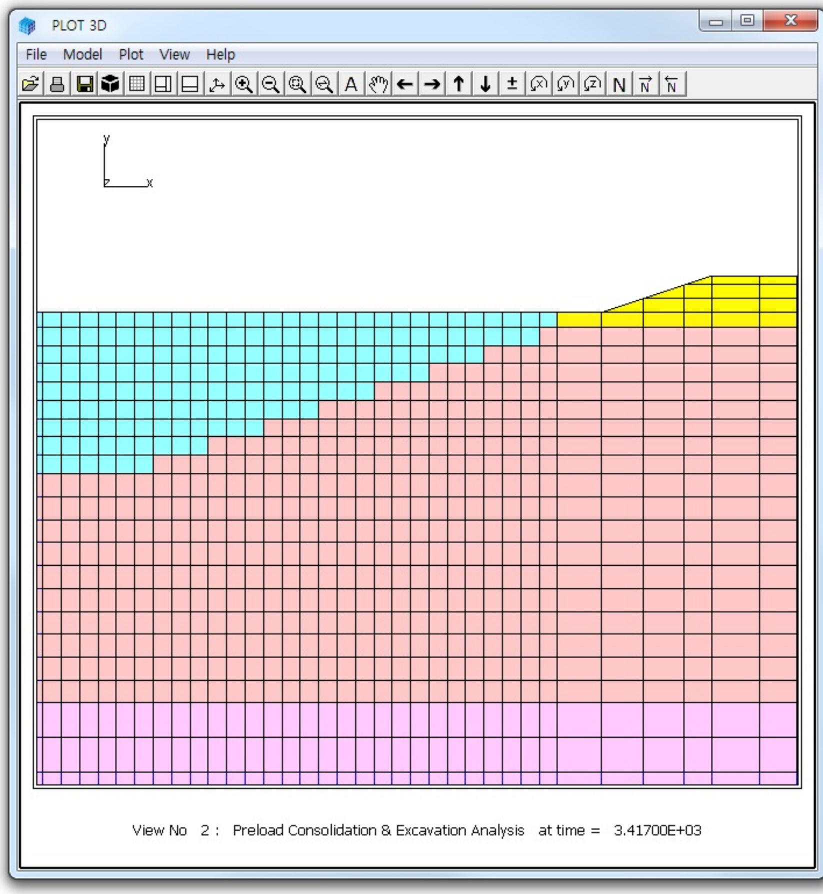


Figure 4.71 Finite element mesh around slope

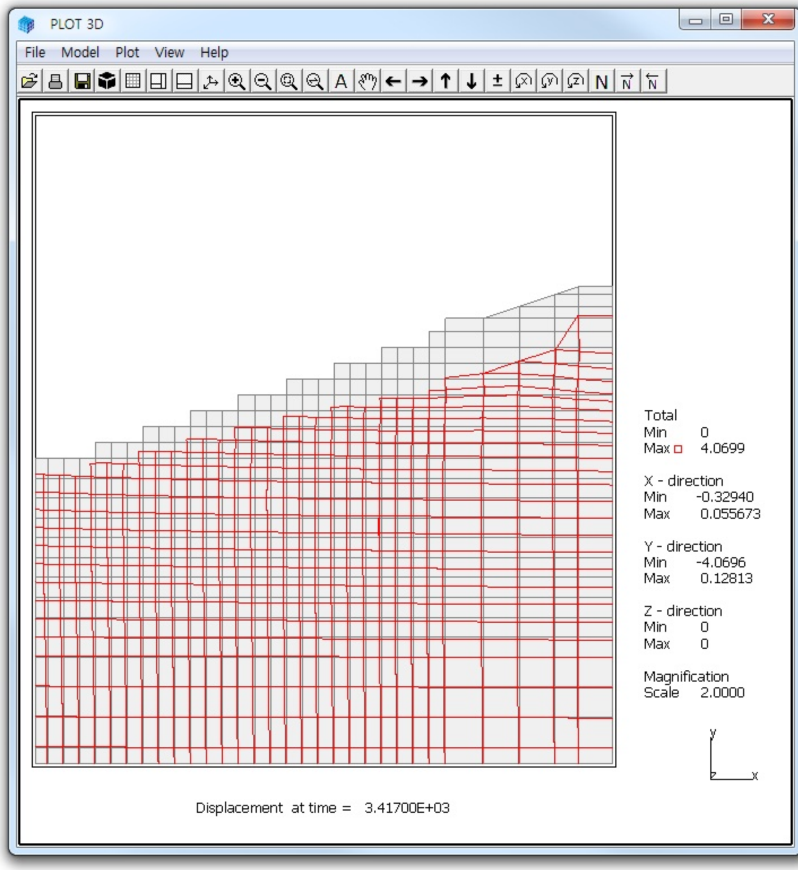


Figure 4.72 Deformed shape around slope

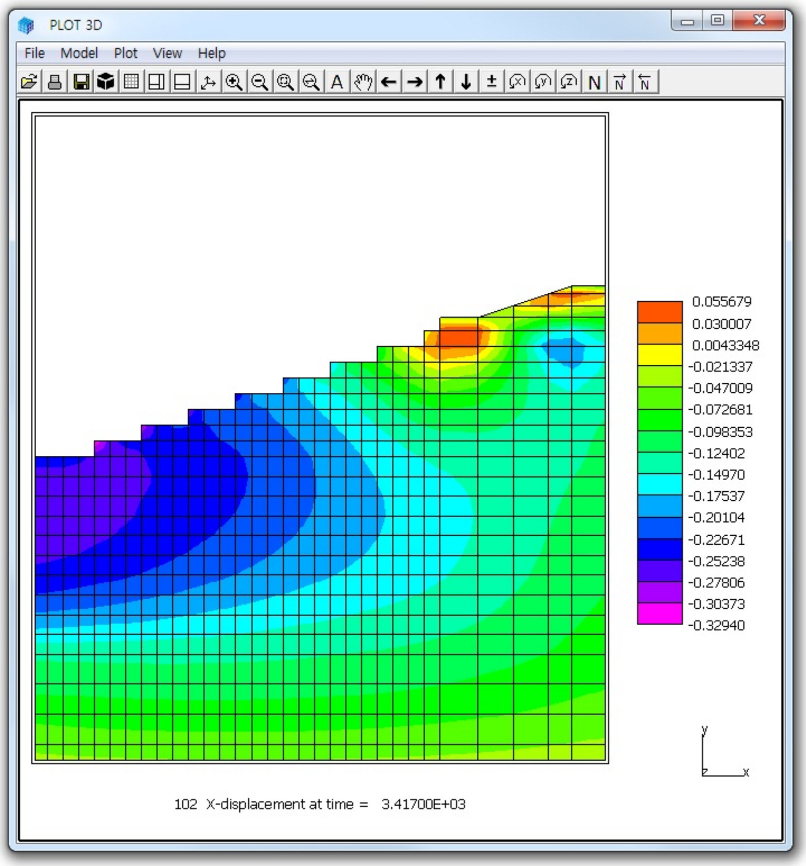


Figure 4.73 Horizontal displacement distribution

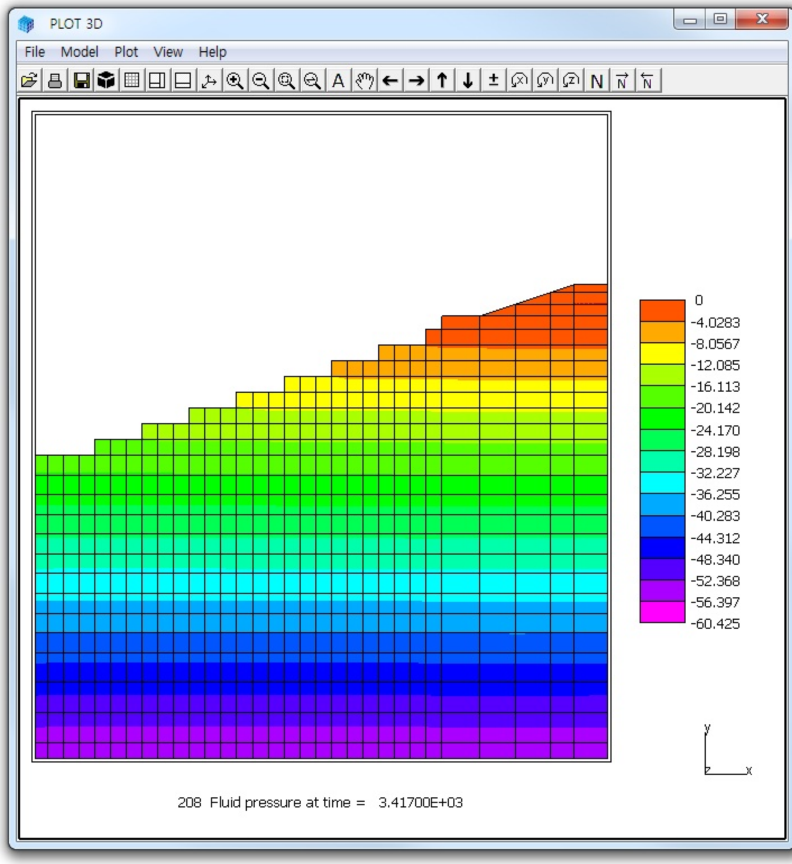


Figure 4.74 Pore pressure distribution

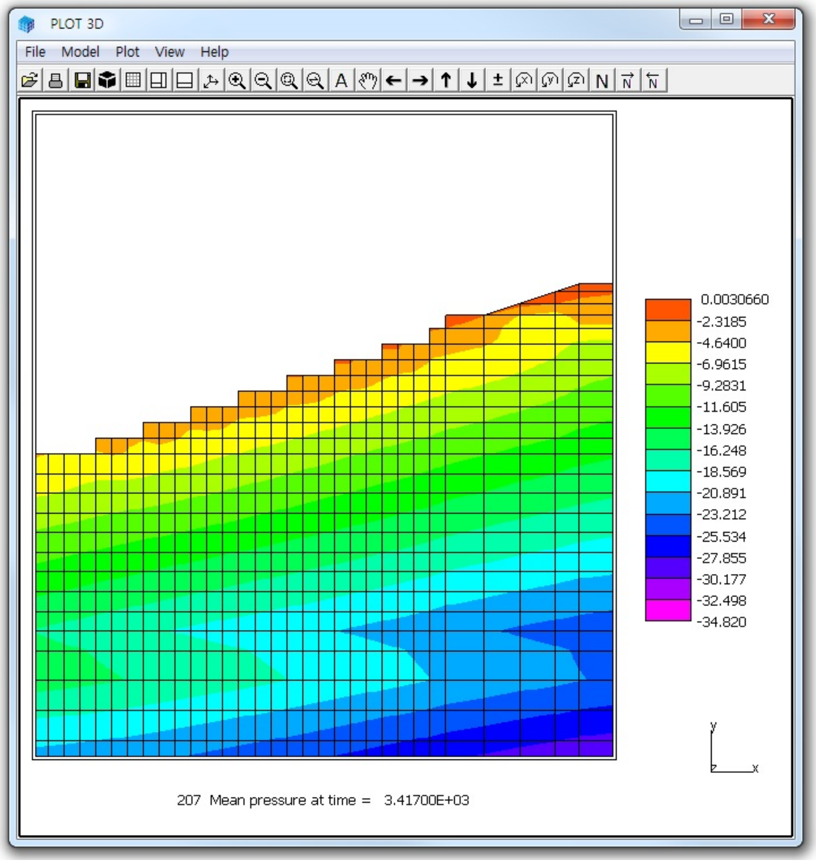


Figure 4.75 Effective mean pressure distribution

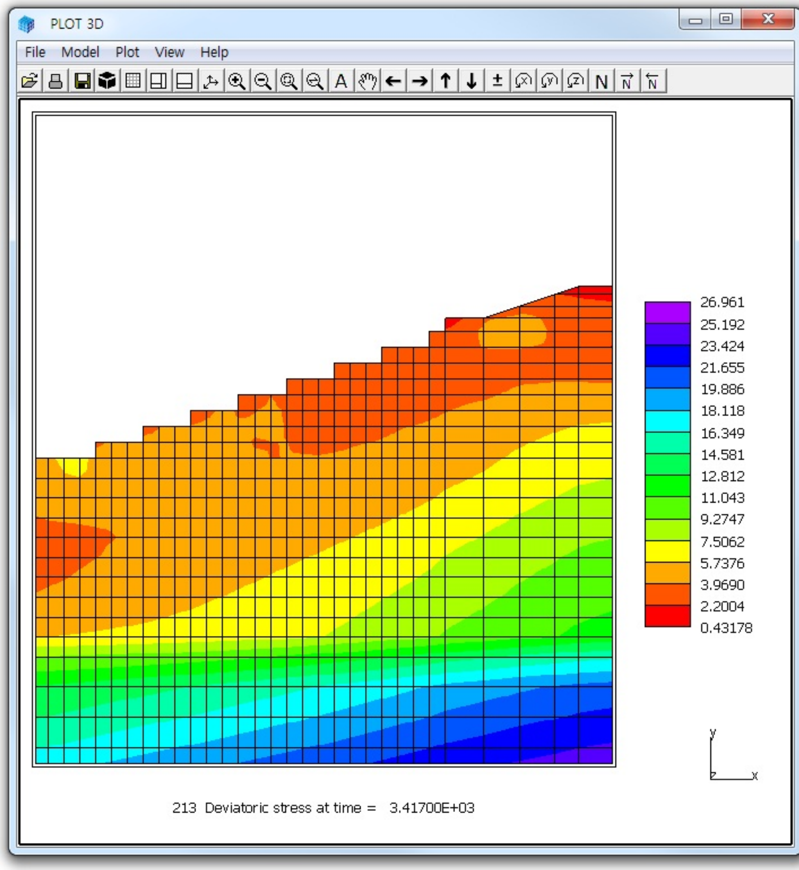


Figure 4.76 Deviatoric stress distribution

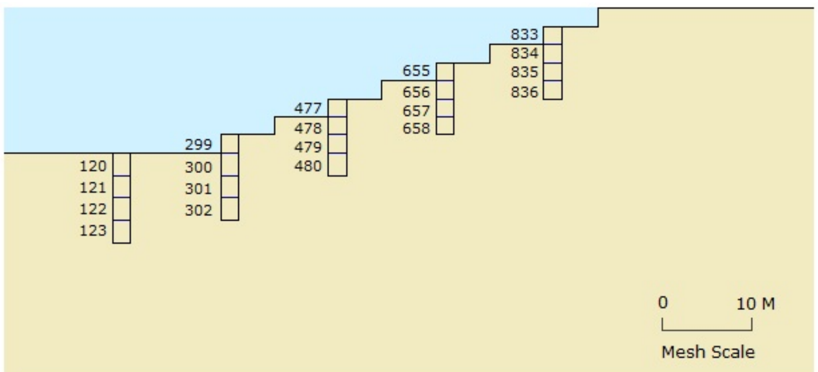


Figure 4.77 Element locations for time history plot

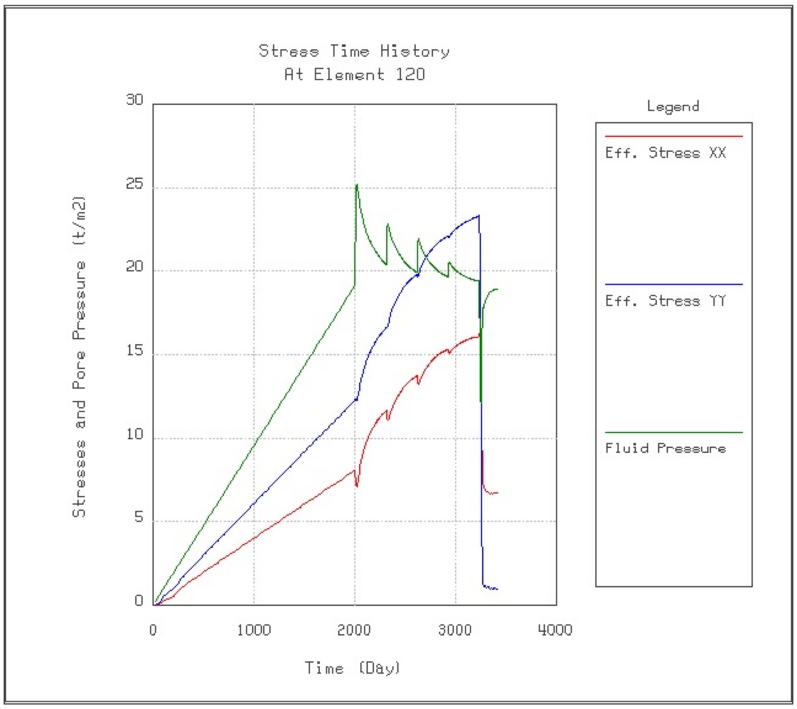


Figure 4.78 Stress time history at element 120

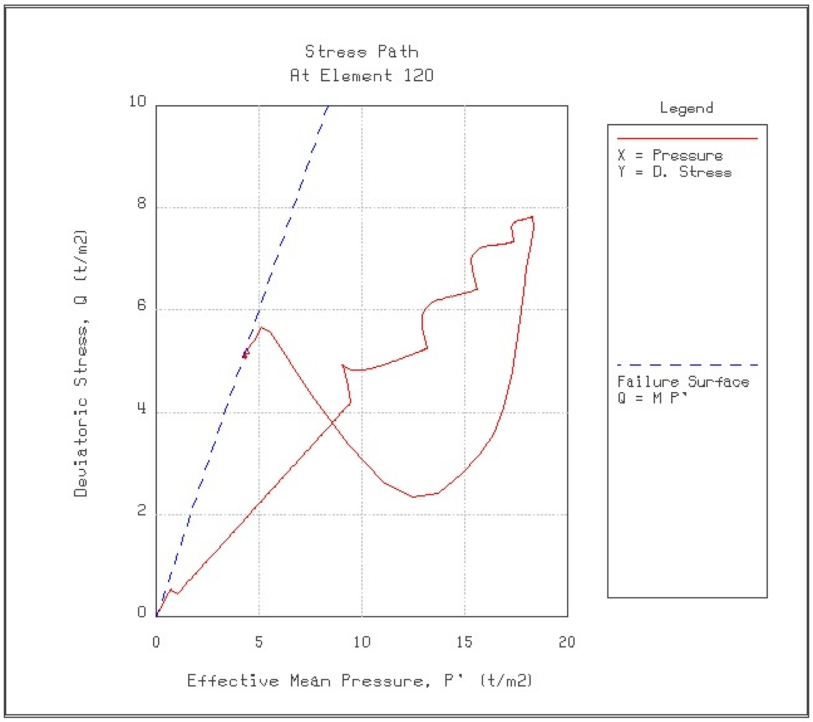


Figure 4.79 Stress path at element 120

4.20 Seismic Tunnel Analysis

This example problem is to analyze a typical NATM tunnel subjected to earthquake loading. The tunnel is located about 22 meters below ground surface as shown in Figure 4.80. Figure 4.81 shows detailed tunnel cross section. Material properties are listed in Table 4.2.

This example problem consists of static and dynamic analyses for the typical horseshoe tunnel constructed by NATM method.

The static part (Steps 1 thru 9) of the analyses as shown in Figure 4.82 is the same as the example problem 2 in TUNA Plus User's Manual except the followings:

- Top core excavation followed by lower core excavation.
- Lining modeled by Shell element with plain concrete.

The dynamic part starting from Step 10 as in Figure 4.83 is performed by applying following boundary conditions and base acceleration:

- Left and right sides of boundary are horizontal roller and bottom of mesh is fixed.
- As horizontal base acceleration, N-S component of the El Centro earthquake is applied with scaled maximum acceleration of 0.2g.

Figure 4.84 shows key location selected for displacement time history plot. Numbers shown in the figure represent node numbers. Figure 4.85 thru 4.87 show finite element meshes used for the analysis.

Figure 4.88 shows tunnel deformed shape at 5 seconds after the onset of earthquake loading. Figures 4.89 and 4.90 show top and bottom surface extreme fiber stresses at 5 seconds after onset of earthquake loading.

The graphical outputs of inner (bottom) and outer (top) extreme fiber stresses of the lining show the maximum compressive stress of 119.9 t/m^2 and the maximum tensile stress of 31.88 t/m^2 at 5 seconds after onset of earthquake loading. Such maximum extreme fiber stresses are far below the strength of the typical plain concrete.

Figure 4.91 shows ground surface horizontal displacement time histories at selected locations: Nodes 609, 837, and 2020. As it can be seen, horizontal ground surface displacements are influenced very little due to the presence of the tunnel.

Figures 4.92 and 4.93 show springline horizontal displacement time histories at the right and left sides of the tunnel, respectively. Each figure shows two adjacent nodes: inner and outer nodes which are separated by interface element as shown in Figures 4.84 and 4.87.

Compared with ground surface, displacements at tunnel springlines are much less amplified. Overall, tunnel lining is moving with the surrounding rock mass but the outgoing lining displacements are limited to the adjacent rock mass displacements. In other words, at those locations where lining is in contact with the adjacent rock mass, the outgoing lining displacements do not exceed the rock mass displacements.

Table 4.2 Material property

Material Type	γ (t/m³)	K_o	E (t/m²)	ν	φ deg.	C (t/m²)	T (t/m²)
Weathered Soil	1.90	0.50	2.00×10^{3}	0.33	30	3	20
Weathered Rock	1.90	0.43	5.000×10^{3}	0.30	35	30	30
Soft Rock	2.40	0.33	2.00×10^{4}	0.25	40	70	40
Hard Rock	2.55	0.25	2.00×10^{5}	0.20	45	100	50
Shotcrete (Soft)	2.40		0.50×10^{6}	0.20	30	500	100
Shotcrete (Hard)	2.40		1.50×10^{6}	0.20	30	500	100
Rock Bolt			2.10×10^{7}				
Reinforced Concrete Lining	2.50		2.10×10^{6}	0.20	30	500	300
Reinforcing Bar			2.10×10^{7}	0.20			
Interface Joint			2.00×10^{5}		5	0.001	0.02

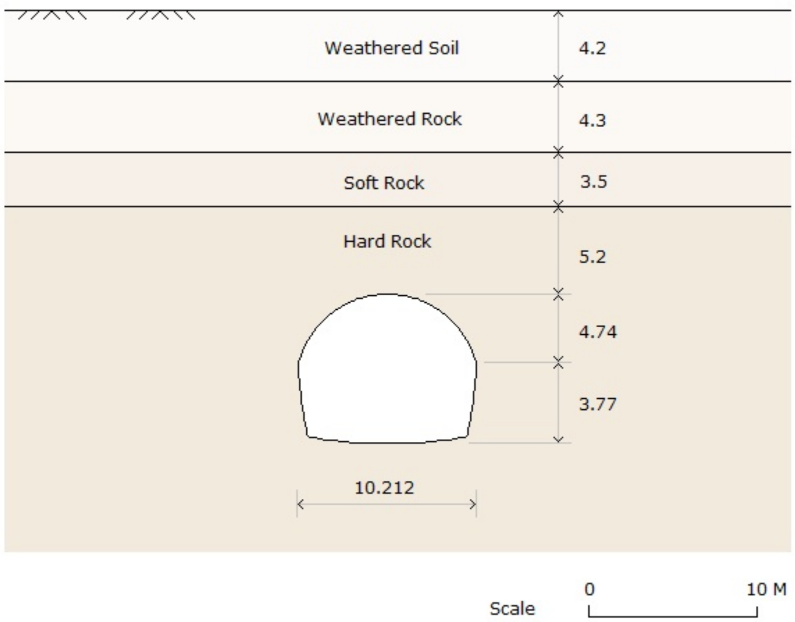
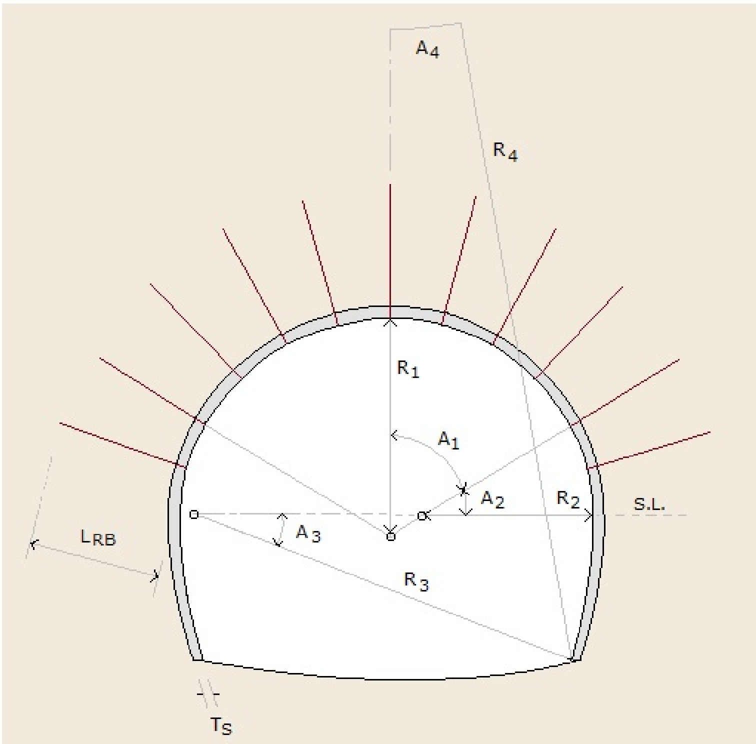


Figure 4.80 Geological profile



R_1	=	7.24 m	A_1	=	60 deg.
R_2	=	6.24 m	A_2	=	30 deg.
R_3	=	11.86 m	A_3	=	21.781 deg.
R_4	=	25.86 m			

Number of Rock Bolts (NUMRB)	=	11
Length of Rock Bolts (LRB)	=	3.0 m
Spacing of Rock Bolts (TSPACING)	=	1.2 m
Thickness of Shotcrete (TS)	=	20 cm
Thickness of Liner (TL)	=	40 cm

Figure 4.81 Tunnel cross section

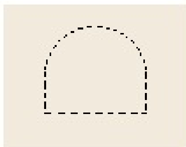
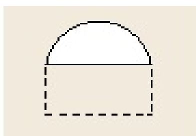
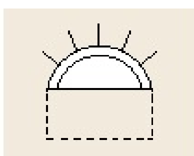
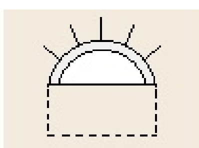
Step	Construction State	Descriptions	
1,2		In Situ K_o State	
3		50 % Stress Relief	Upper Core Excavation
4		75 % Stress Relief Soft Shotcrete Rock Bolt	
5		100 % Stress Relief Hard Shotcrete Rock Bolt	

Figure 4.82 Construction sequence, static part

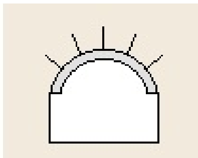
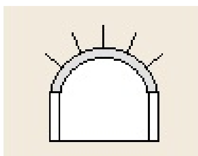
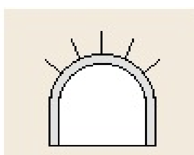
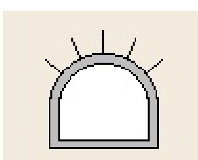
<table>
<tr><th>Step</th><th>Construction State</th><th colspan="2">Descriptions</th></tr>
<tr><td>6</td><td></td><td>50% Stress Relief</td><td rowspan="3">Lower Core Excavation</td></tr>
<tr><td>7</td><td></td><td>75% Stress Relief
Soft Shotcrete</td></tr>
<tr><td>8</td><td></td><td>100% Stress Relief
Hard Shotcrete</td></tr>
<tr><td>9</td><td></td><td colspan="2">Lining Subjected to: Weight</td></tr>
</table>

Figure 4.82 Construction sequence, static part (Continued)

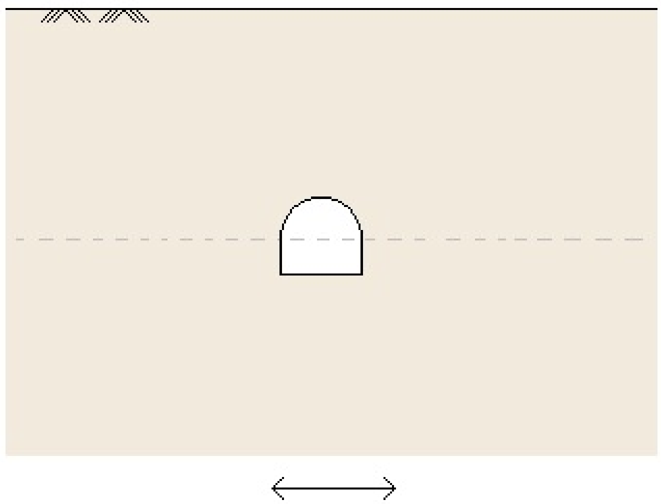


Figure 4.83 Seismic load, dynamic part

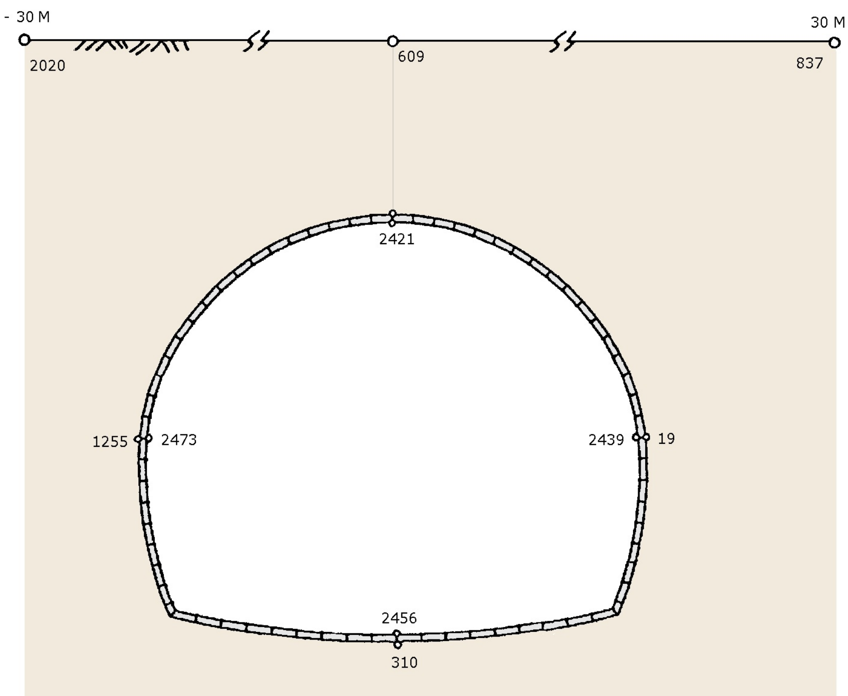


Figure 4.84 Key locations for displacement time history plot

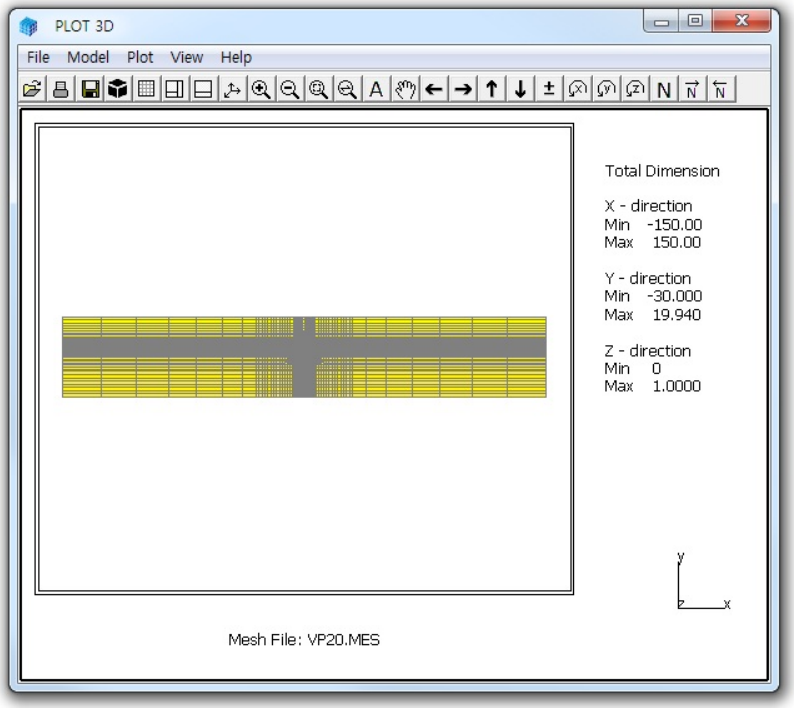


Figure 4.85 Finite element mesh

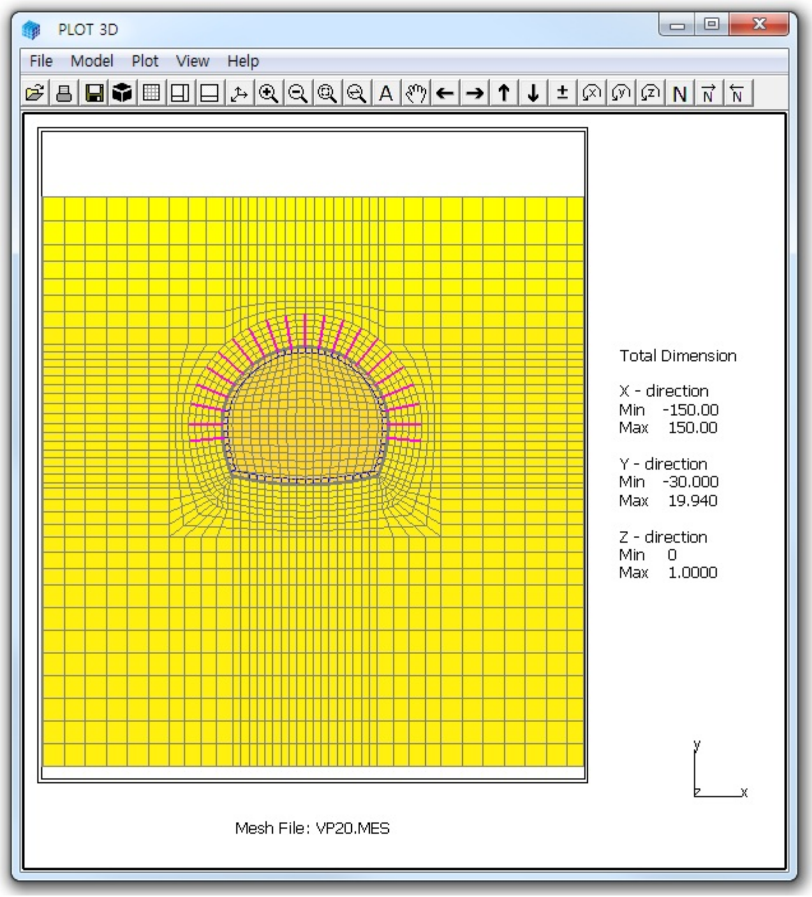


Figure 4.86 Finite element mesh around tunnel

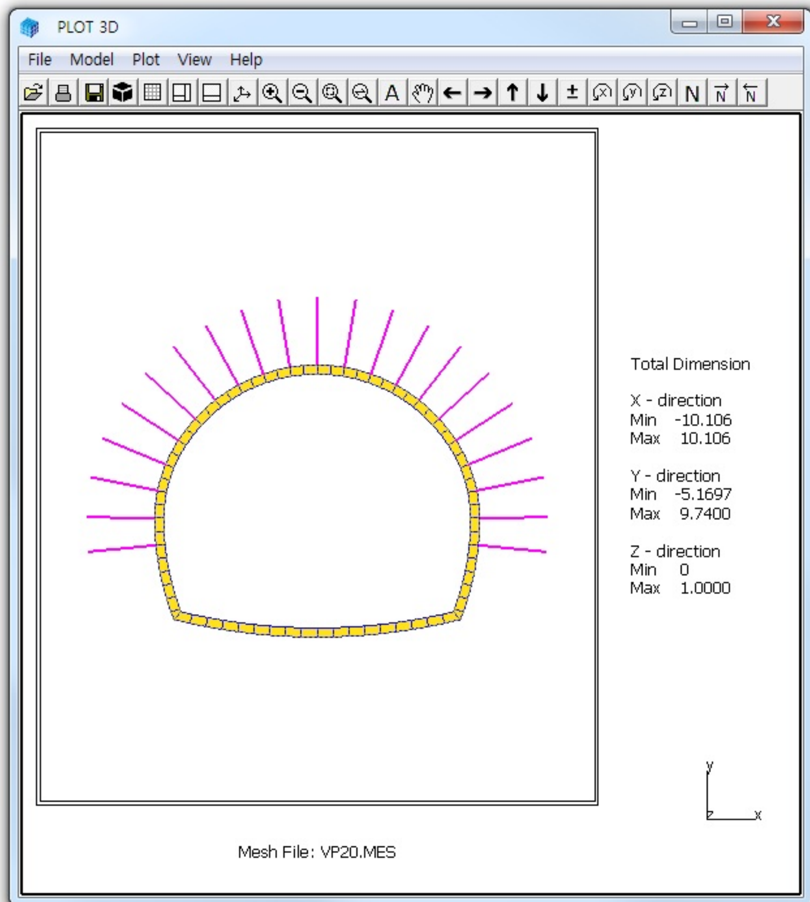


Figure 4.87 Interface between shotcrete and lining

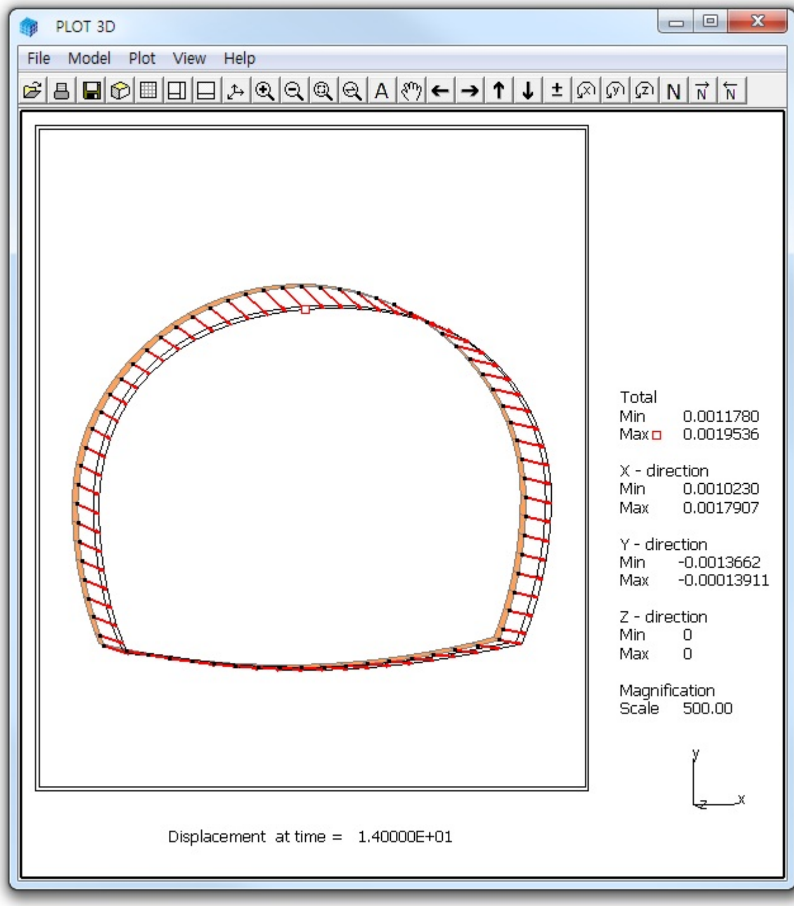


Figure 4.88 Tunnel deformed shape at t=14 sec

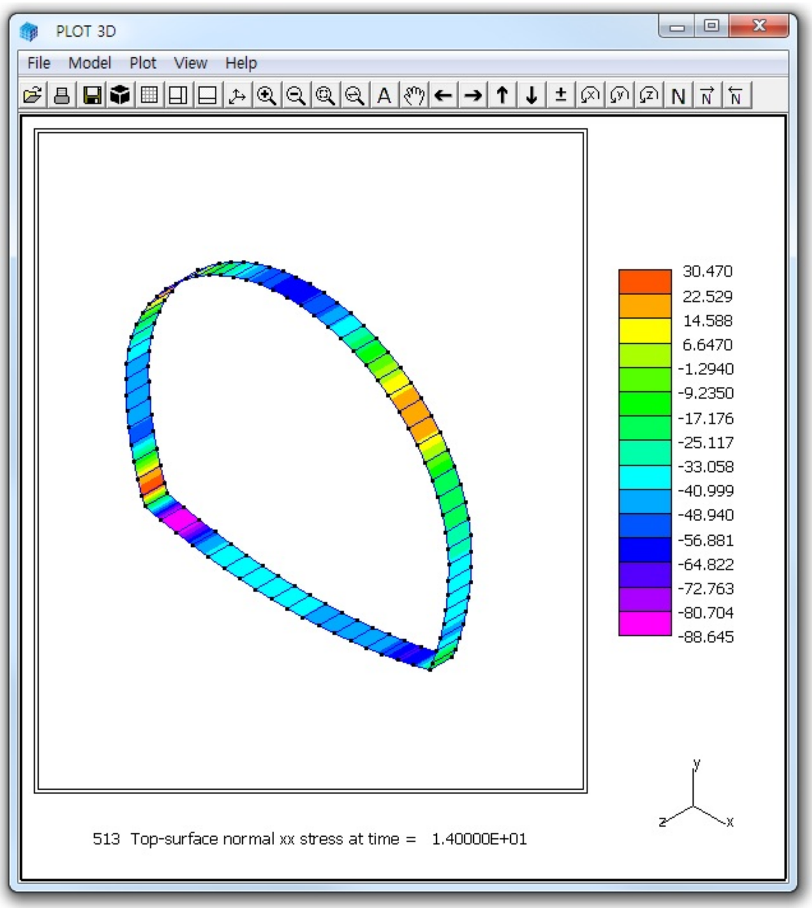


Figure 4.89 Top surface extreme fiber stress at t=14 sec

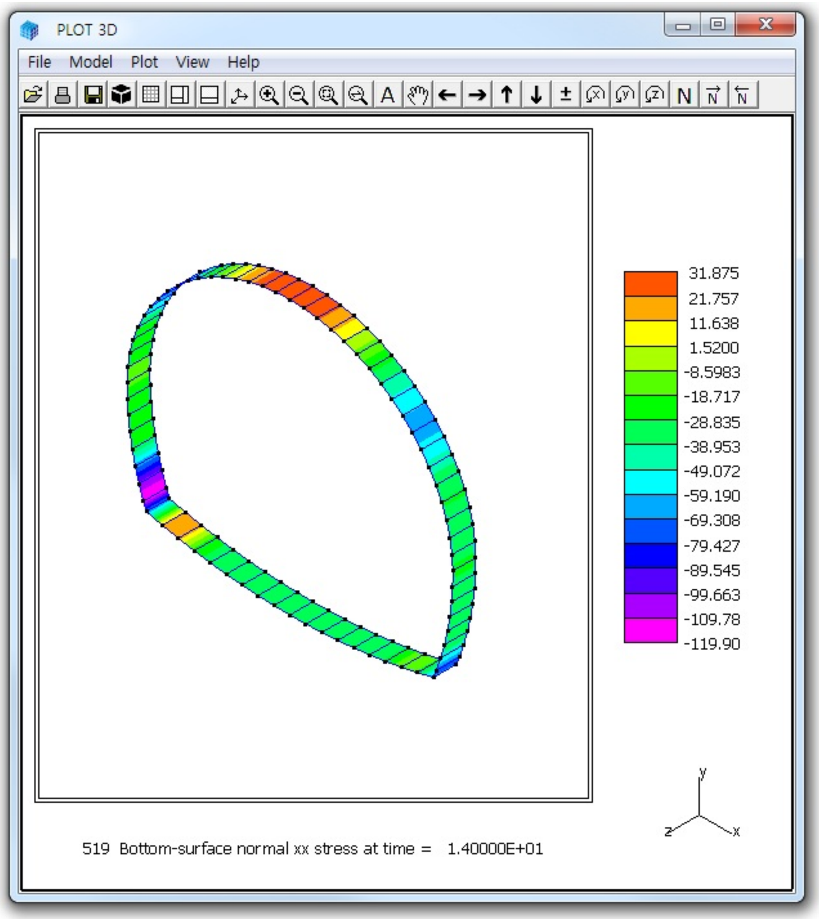


Figure 4.90 Bottom surface extreme fiber stress at t=14 sec

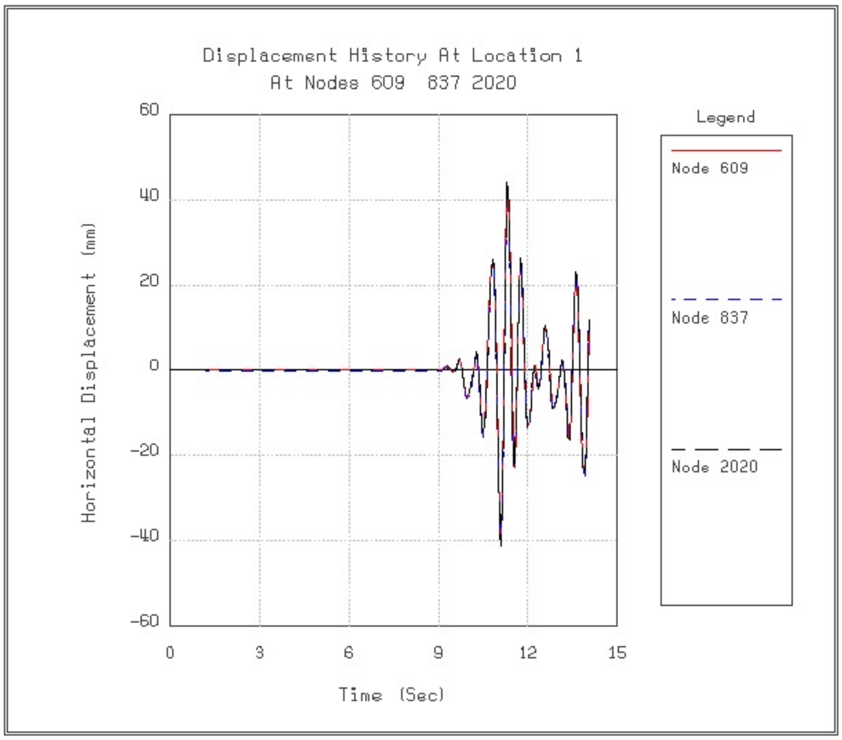


Figure 4.91 Horizontal displacement at nodes 609, 837, and 2020

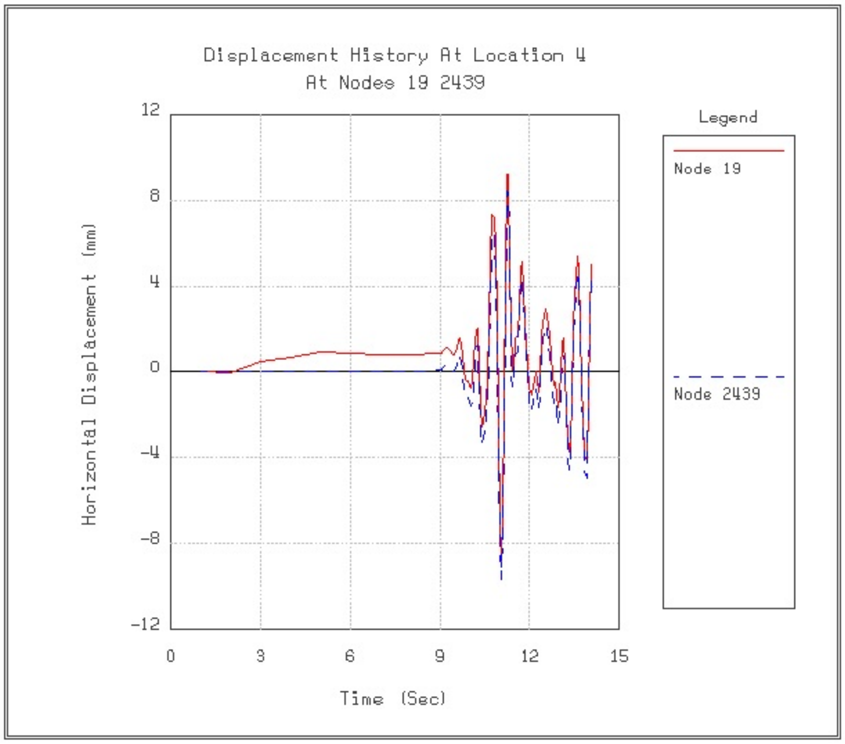


Figure 4.92 Horizontal displacement at nodes 19 and 2439

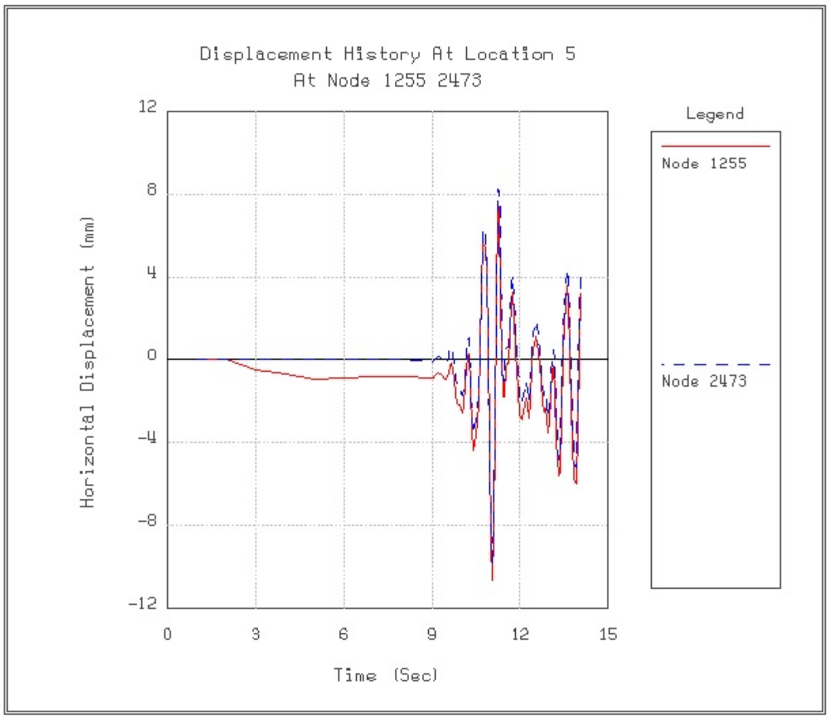


Figure 4.93 Horizontal displacement at nodes 1255 and 2473

4.21 Frames with Hinge Connection

This example problem is to solve symmetric plane frame members subjected to a vertical concentrated load at the hinge connecting both frames as shown is Figure 4.94.

The exact solutions for this frame structures without shear deformation are given below:

$$\delta = \frac{P}{EA/L + 3EI/L^3} \qquad M_{max} = \frac{PL/\sqrt{2}}{1 + AL^2/3I}$$

where

- δ Maximum deflection at the center
- M_{max} Maximum moment at fixed end

Two SMAP-3D calculations are performed using the geometrical and material parameters listed in Figure 4.94.

Frames modeled by 10 beam elements:

- Figure 4.95 Beam element with material number
- Figure 4.96 Beam deformed shape
- Figure 4.97 Beam bending moment diagram

Frames modeled by 40 shell elements:

- Figure 4.98 Shell element with material number
- Figure 4.99 Shell deformed shape
- Figure 4.100 Shell bending moment diagram

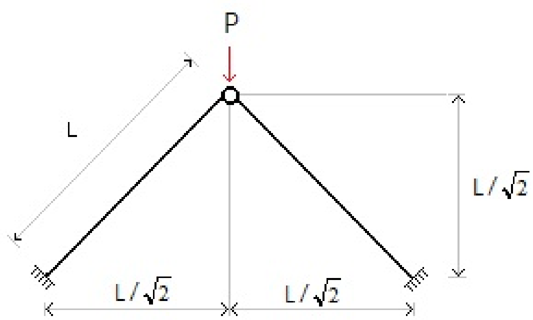
SMAP-3D results show good agreement with the exact solutions.

Maximum deflection at the center (δ)

- Exact solution = 0.01768 cm
- SMAP-3D (Beam) = 0.01767 cm
- SMAP-3D (Shell) = 0.01767 cm

Maximum moment at fixed end (M_{max})

- Exact solution = 0.1000 t-m
- SMAP-3D (Beam) = 0.1000 t-m
- SMAP-3D (Shell) = 0.1003 t-m



P = 100 t   L = 7.071 m

E = 20 x 10^6 t/m^2 v = 0.0

A = 0.2 m^2 I = 0.000667 m^4

Figure 4.94 Frames with hinge connection

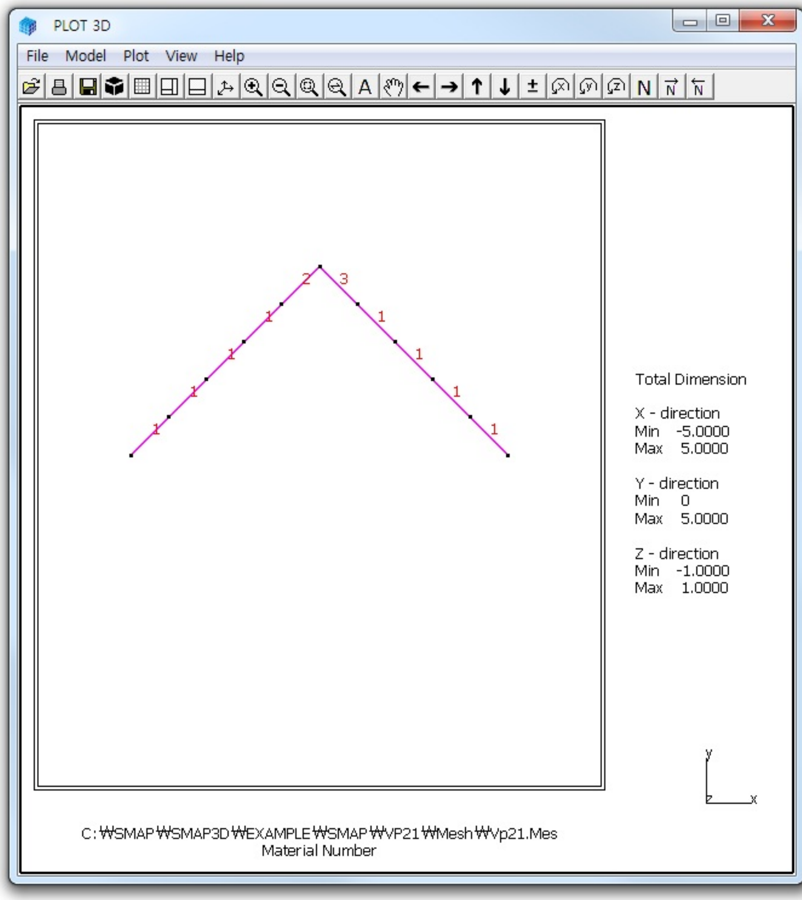


Figure 4.95 Beam element with material number

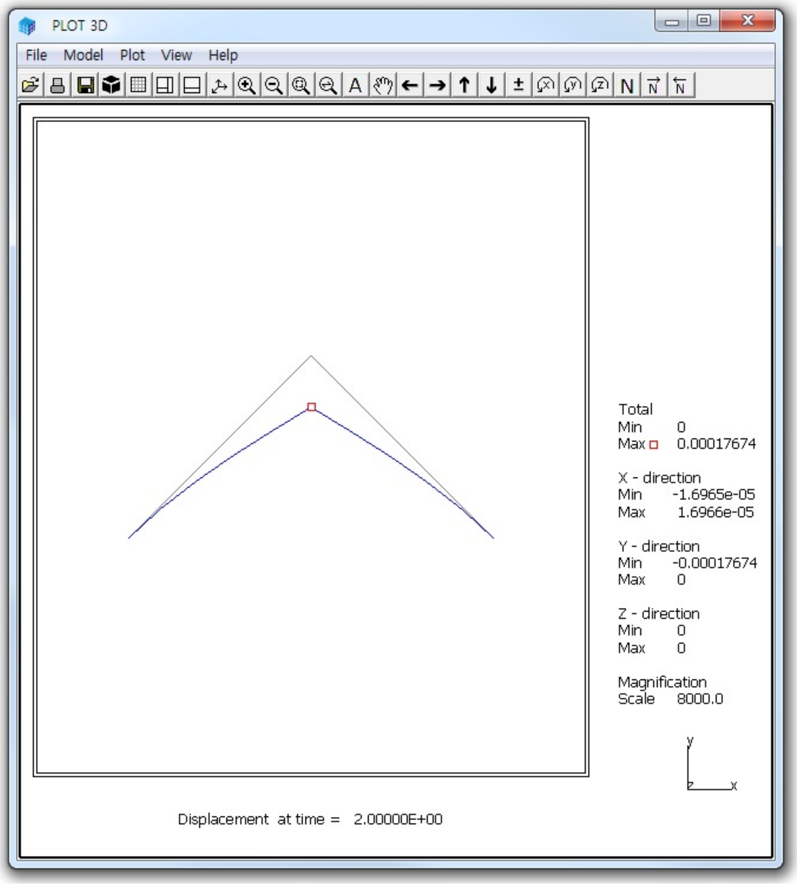


Figure 4.96 Beam deformed shape

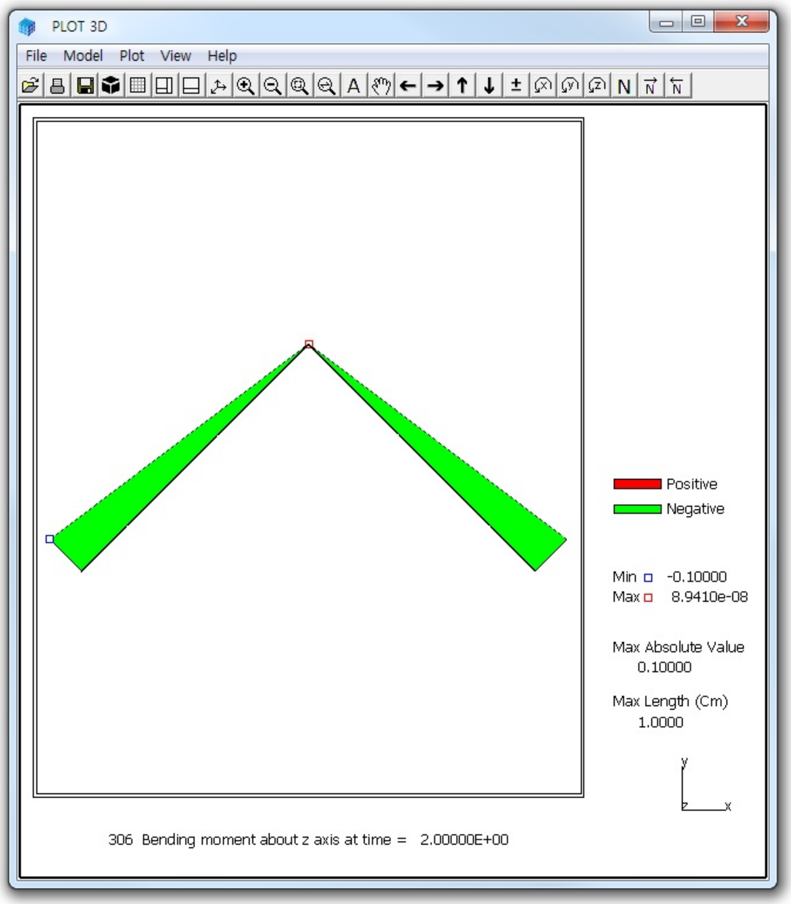


Figure 4.97 Beam bending moment diagram

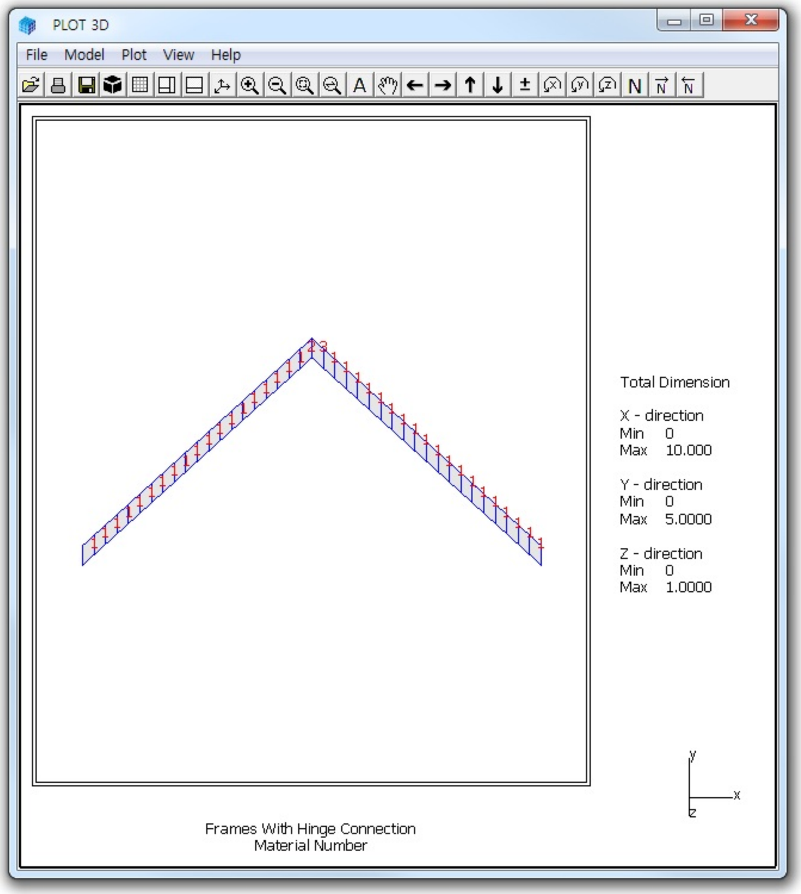


Figure 4.98 Shell element with material number

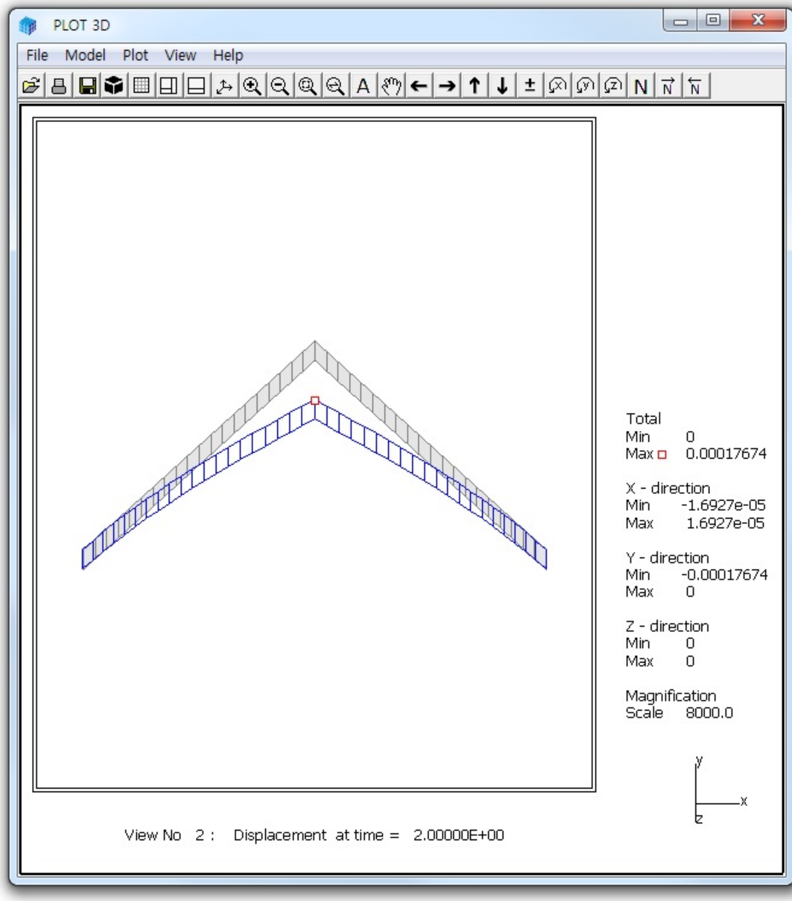


Figure 4.99 Shell deformed shape

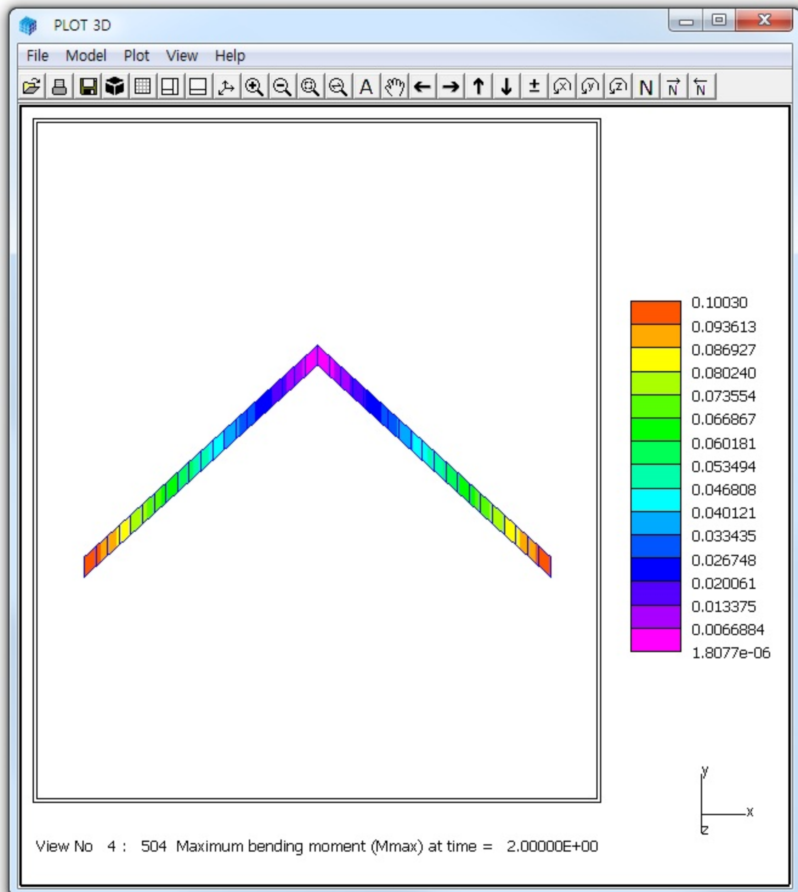


Figure 4.100 Shell bending moment diagram

4.22 Embedded Rebars with Slip

This example problem is to verify the implementation of the embedded reinforcing bars (rebars) with interface shear (slip) between rebars and surrounding concrete. Figure 4.101 shows a simply supported reinforced concrete beam subjected to a concentrated load at midspan. To simplify the problem, it was assumed that both reinforcing bars and concrete are linearly elastic while the interface shear is elastic - perfectly plastic with a limiting constant cohesion.

The exact beam solution without shear deformation is given below:

Maximum deflection at the center without rebars,

$$\delta = \frac{P \cdot L^3}{48 \; E_c \cdot I_c} = 1.190 \text{ Cm}$$

Maximum deflection at the center with rebars,

$$\delta = \frac{P \cdot L^3}{48 \; E_c \cdot I_t} = 1.040 \text{ Cm}$$

By symmetry, only left half of the beam is modeled using 60 continuum elements for concrete and 2 embedded truss elements for reinforcing bars as shown in Figure 4.102. It should be noted that the end points of embedded truss elements do not belong to the corner nodes of continuum elements.

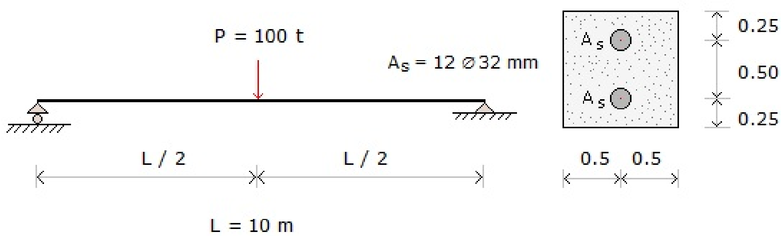
The computed center deflections are compared with the exact beam solution as shown in Table 4.3. SMAP-3D results approach to the upper bound beam solution at lower cohesion and the lower bound beam solution at higher cohesion. At the intermediate cohesion, however, the computed deflection is in between upper and lower bound beam solutions, indicating some resistance from slip strength.

Figures 4.103 and 4.104 show the deformed shape and the axial stress distribution, respectively, from SMAP-3D result at the intermediate cohesion of 5 t/m^2.

Table 4.3 Computed center deflections

Cmax (t/m^2)	SMAP-3D Result	Exact Beam Solution
0.1	1.1746 Cm	1.190 Cm (without rebar)
5.0	1.0990 Cm	
280	1.0379 Cm	1.040 Cm (with rebar)

Cmax : Interface Cohesion



E_c = 2.1×10^6 t/m^2
v_c = 0.2
E_s = 2.1×10^7 t/m^2
I_c = 0.0833 m^4
I_s = $2\ (E_s / E_c)\ A_s\ (0.025)^2$ = 0.012063 m^4
I_t = $I_c + I_s$ = 0.0954 m^4

Property of interface between rebar and concrete

G = 0.875×10^6 t/m^2 (shear modulus)
r_b = 0.016 m (radius of rebar)
t_g = 0.002 m (thickness of interface)
K_{se} = $G / (\ (r_b + t_g)\ \ln(1 + t_g / r_b)\ ) = K_s$(each)
K_s = 12 K_s(each) = 4.956×10^9 t/m^3

Figure 4.101 Embedded rebars with slip

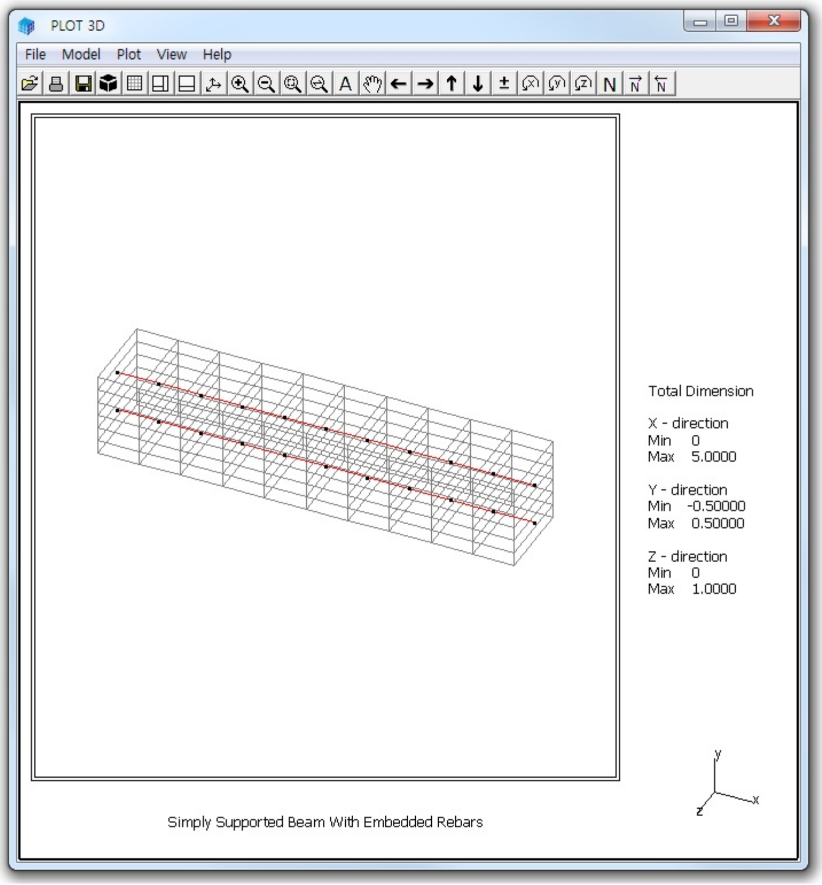


Figure 4.102 Finite element mesh

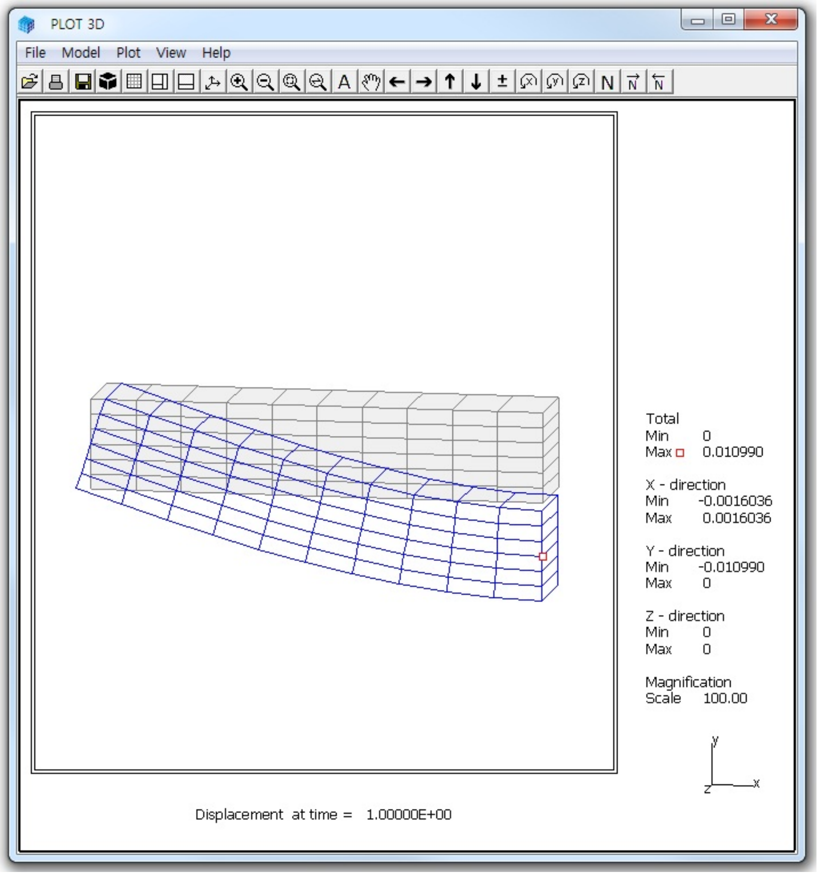


Figure 4.103 Deformed shape

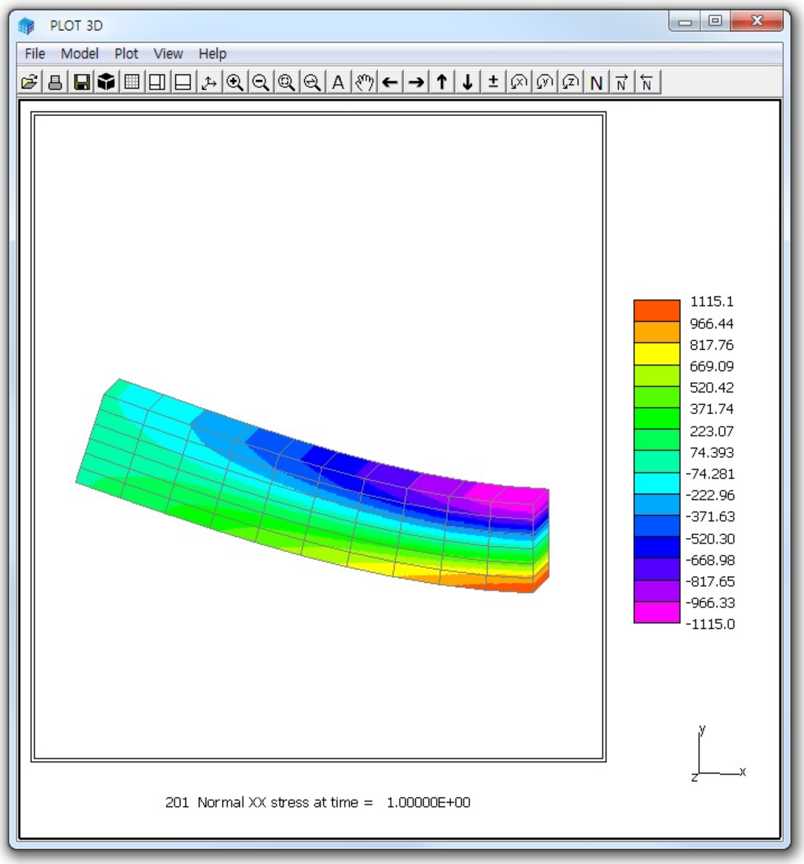


Figure 4.104 Axial stress distribution

4.23 Pseudo-Dynamic Embankment Fill Analysis

This example problem is to solve the response of an embankment fill subjected to pseudo-dynamic earthquake load as schematically shown in Figure 4.105.

As listed in Table 4.4, the sequence of construction consists of 5 steps. The first two steps are used to compute in situ Ko state with water table at GL-25. At step 3, water table is raised up to GL-5. At step 4, embankment fill is completed. At final step 5, pseudo-dynamic earthquake load is applied to the embankment fill.

Material properties are listed in Table 4.5.

The change of water table is modeled by adding Intensity times Distribution Factor to the Y component of unit gravity load (FRY). Intensity history number and distribution factor are specified in Card Group 9.1.2.

The pseudo-dynamic earthquake load is modeled by adding Intensity times Distribution Factor to the X component of unit gravity load (FRX).

Figure 4.106 shows the finite element mesh used for the analysis. Figures 4.107 and 108 show deformed shape and vertical stress distribution, respectively, at final step 5 where pseudo-dynamic earthquake load is applied to the embankment fill.

Computed vertical stress at GL-23 is reduced by 18 t/m^2 due to the water table at GL-5. The reduction of vertical stress is associated with the water head of 18 m at GL-23.

Horizontal displacement of 1.16 Cm is obtained at the top surface of embankment fill due to the pseudo dynamic load. Exact solution for this problem is not available. However, SMAP-S2 and SMAP-2D analyses show the same results.

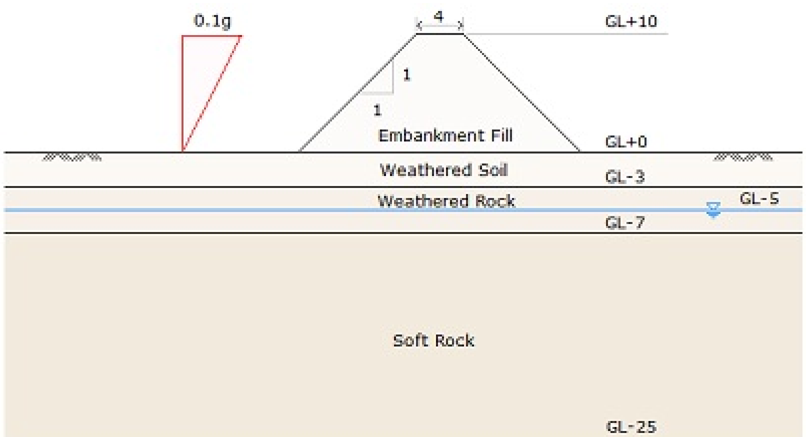


Figure 4.105 Schematic section view of embankment

Table 4.4 Construction sequence

Step	Description
1, 2	In Situ Ko state with water table at GL-25
3	In Situ Ko state with water table at GL-5
4	Completion of embankment fill
5	Embankment fill subjected to pseudo-dynamic load

Table 4.5 Material property

Material Type	γ (t/m^3)	K_o	E (t/m^2)	v	φ deg.	C (t/m^2)	T (t/m^2)
Weathered Soil	1.90	0.50	2.0×10^3	0.33	30	3	20
Weathered Rock	1.90	0.43	5.0×10^3	0.30	35	30	30
Soft Rock	2.40	0.33	2.0×10^4	0.25	40	70	40
Embankment Fill	2.00	0.50	3.0×10^3	0.33	30	3	20

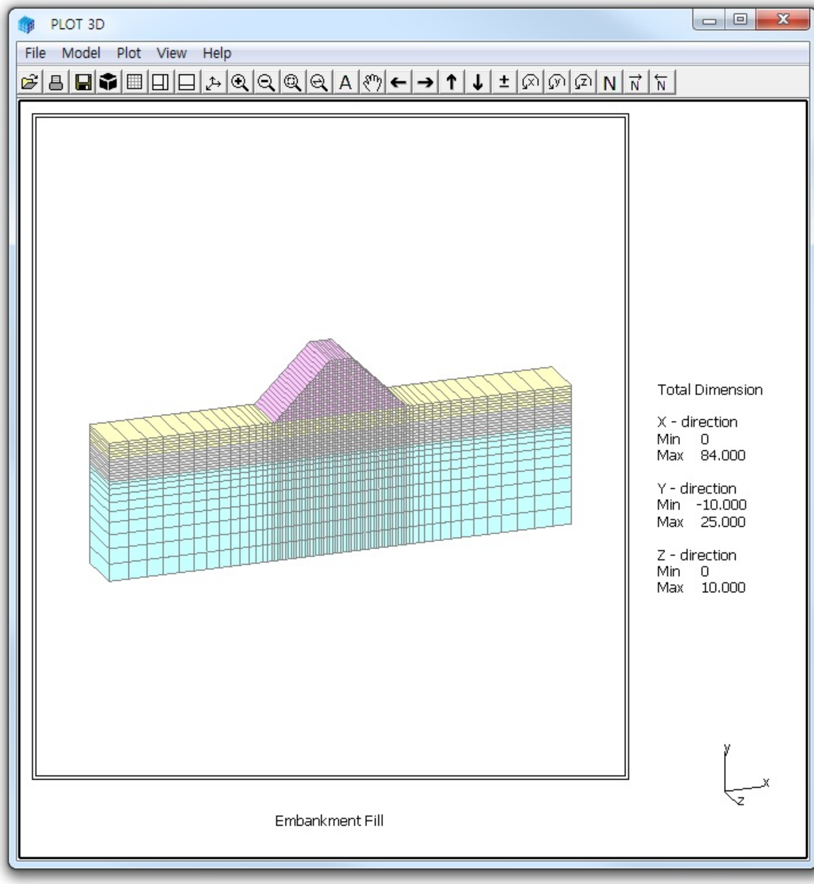


Figure 4.106 Finite element mesh

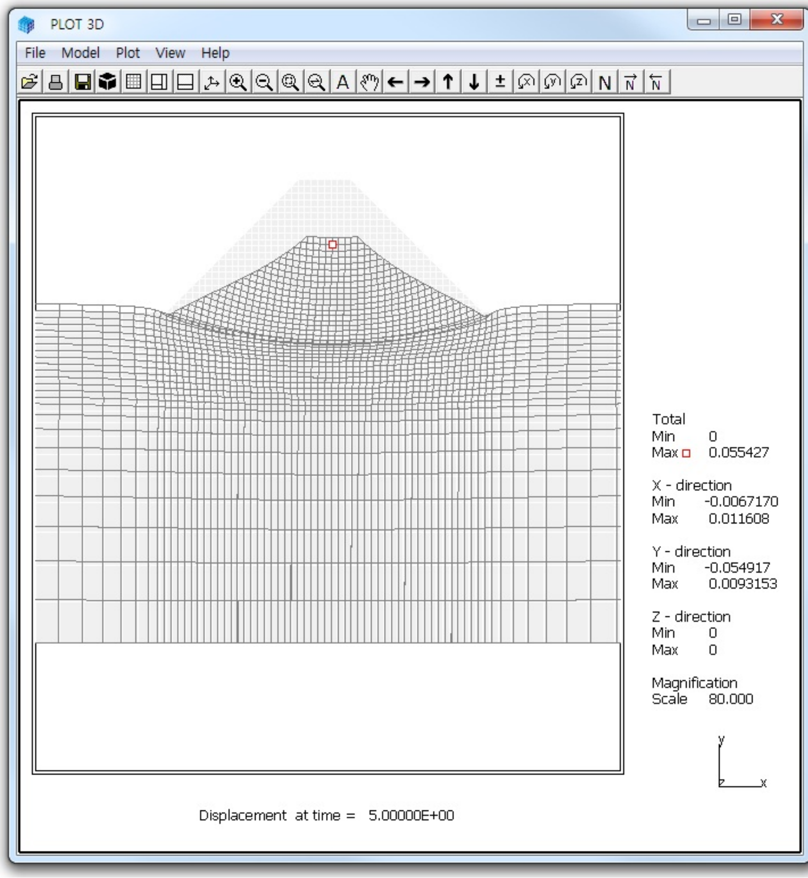


Figure 4.107 Deformed shape

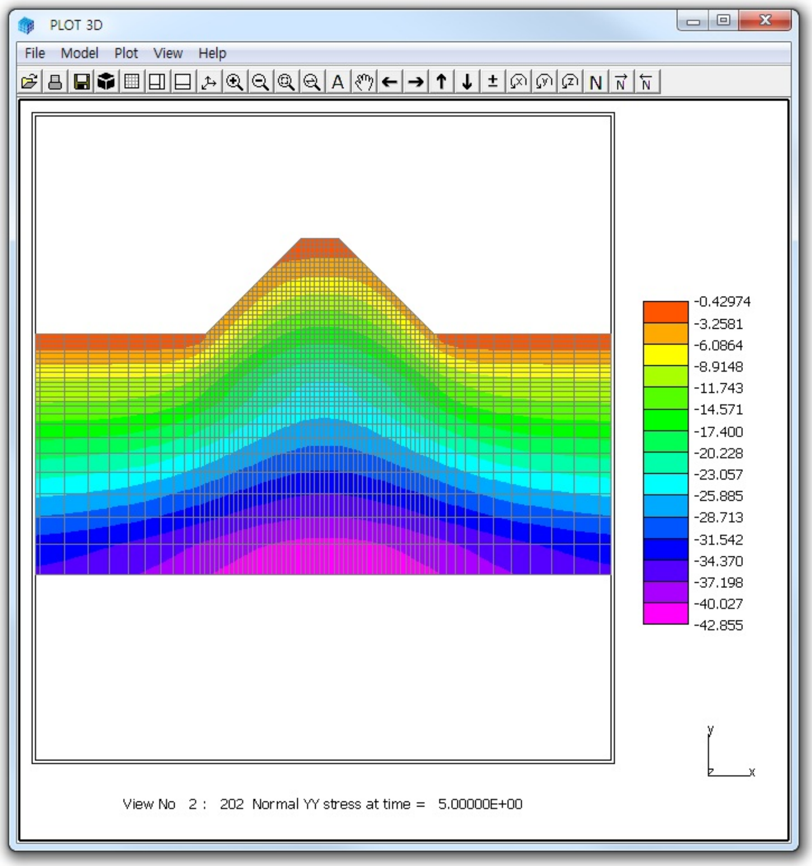


Figure 4.108 Vertical stress distribution

4.24 Plane Strain Tunnel in Jointed Continuum

This example problem is to verify the jointed continuum mesh generated by JOINT-3D pre-processing program. Jointed continuum analysis is similar to the discrete element analysis. For the jointed continuum analysis, each continuum finite element is surrounded by joint elements.

The main advantages of using such joint elements are to allow slippage along the joint when reaching shear strength and debonding normal to joint face when exceeding tensile strength.

This example is identical to the Example Problem 14 except that the tunnel is located in the jointed continuum. The jointed continuum mesh is generated by JOINT-3D program with the input file Joint.inp. Refer to JOINT-3D User's Manual.

Figure 4.109 shows the finite element mesh consisting of the jointed continuum around tunnel.

To compare with continuum model (Example Problem 14), two analyses are performed with Elastic and Plastic Joint Models.
The Elastic Joint Model assumes strong joint properties so that it essentially represents continuum model. The Plastic Joint Model assumes lower shear and tensile strengths so that it allows slippage and debonding along the joints.

Results are listed in the following order:

- Figure 4.110 Deformed shape for Elastic Joint
- Figure 4.111 Principal stress vector for Elastic Joint
- Figure 4.112 Bending moment for Elastic Joint
- Figure 4.113 Deformed shape for Plastic Joint
- Figure 4.114 Principal stress vector for Plastic Joint
- Figure 4.115 Bending moment for Plastic Joint

In general, rersults of the Elastic Joint Model are close to those of conventional continuum analysis in Example Problem 14.

On the other hand, Plastic Joint Model shows considerable amount of slippage below bottom corner of tunnel as in Figures 4.113 and 4.114. Stress distributions are quite different from Elastic Joint Model.

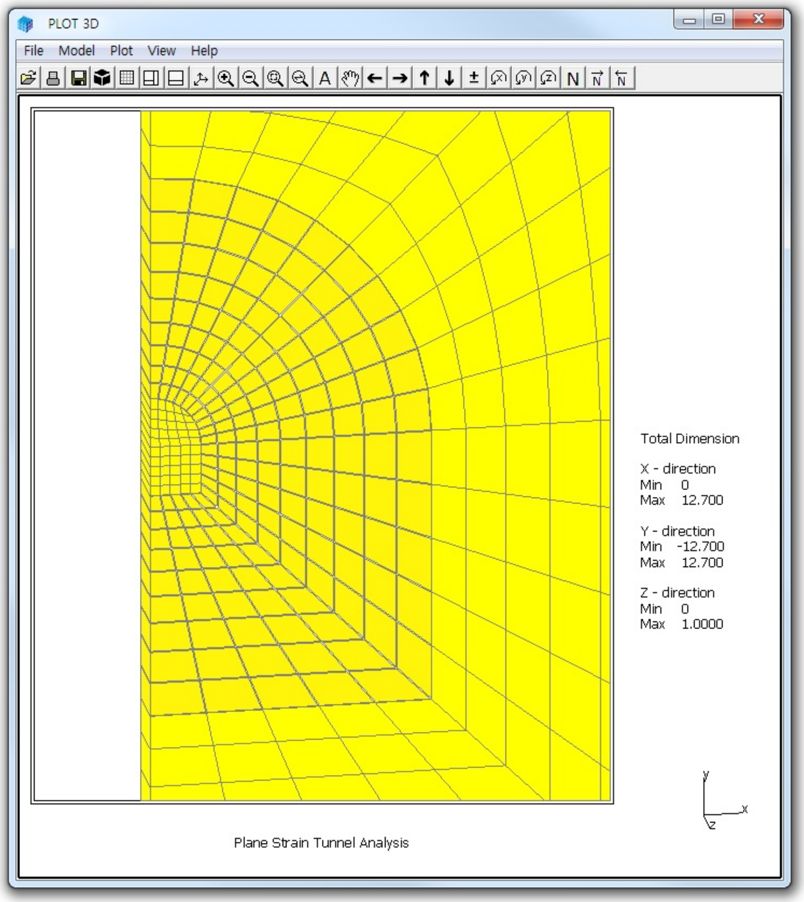


Figure 4.109 Finite element mesh around tunnel

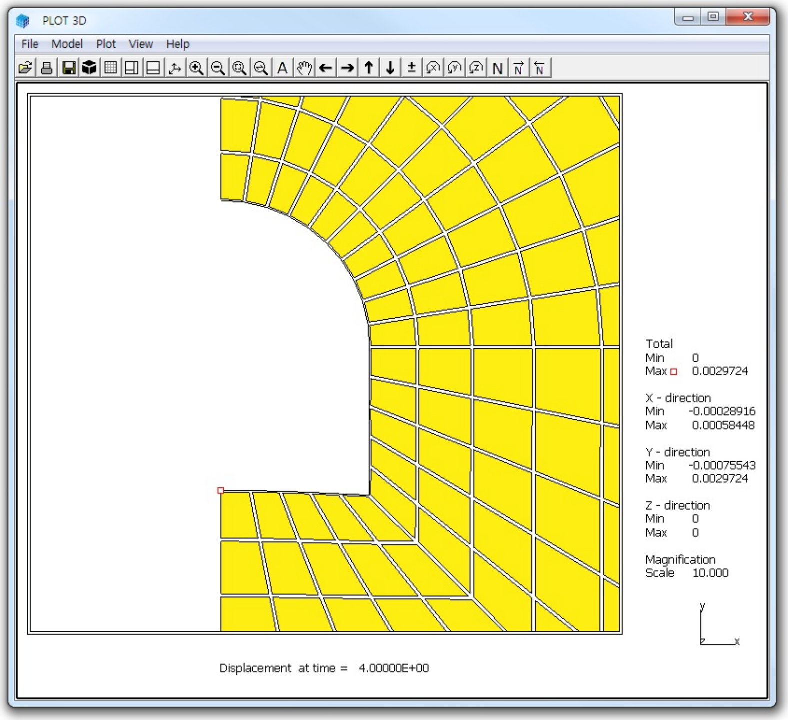


Figure 4.110 Deformed shape for Elastic Joint

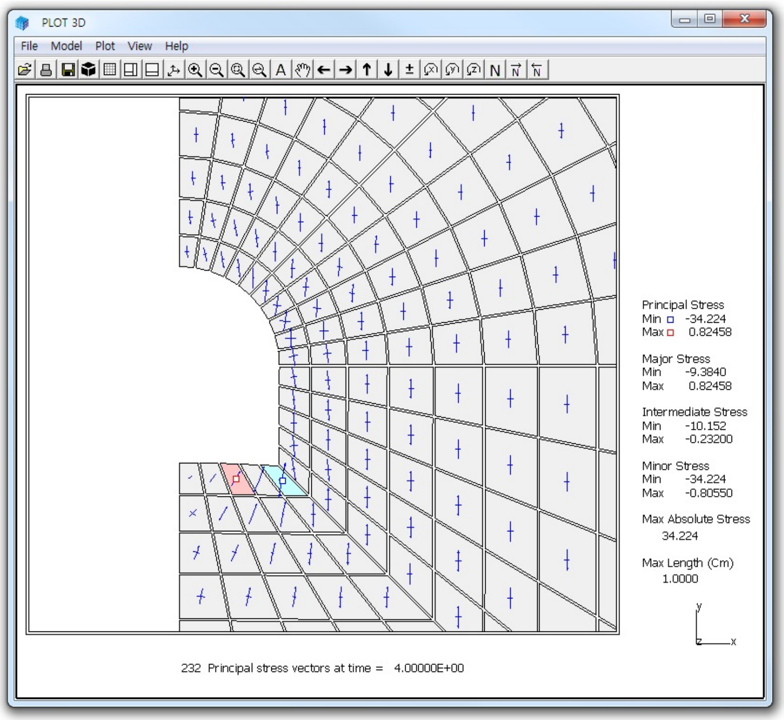


Figure 4.111 Principal stress vector for Elastic Joint

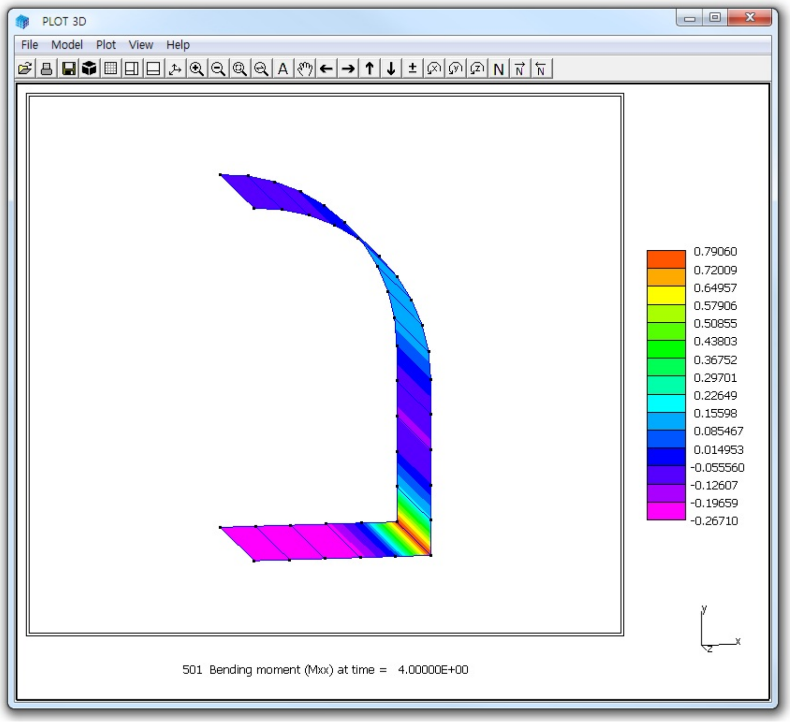


Figure 4.112 Bending moment for Elastic Joint

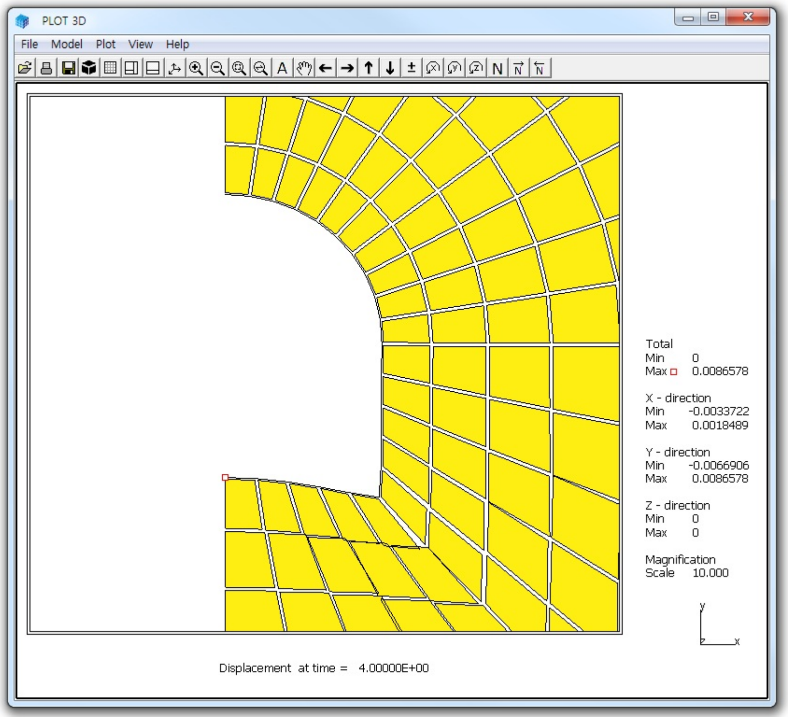


Figure 4.113 Deformed shape for Plastic Joint

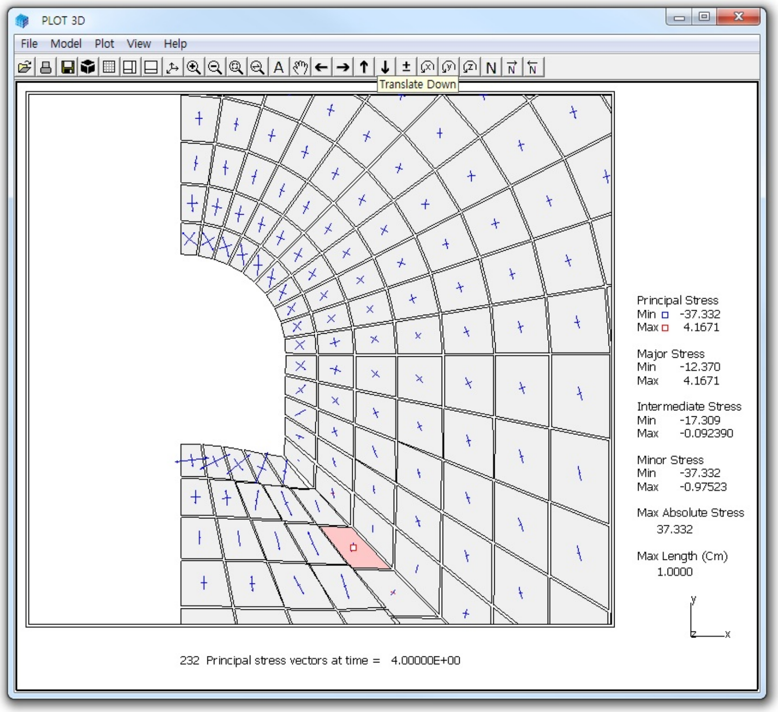


Figure 4.114 Principal stress vector for Plastic Joint

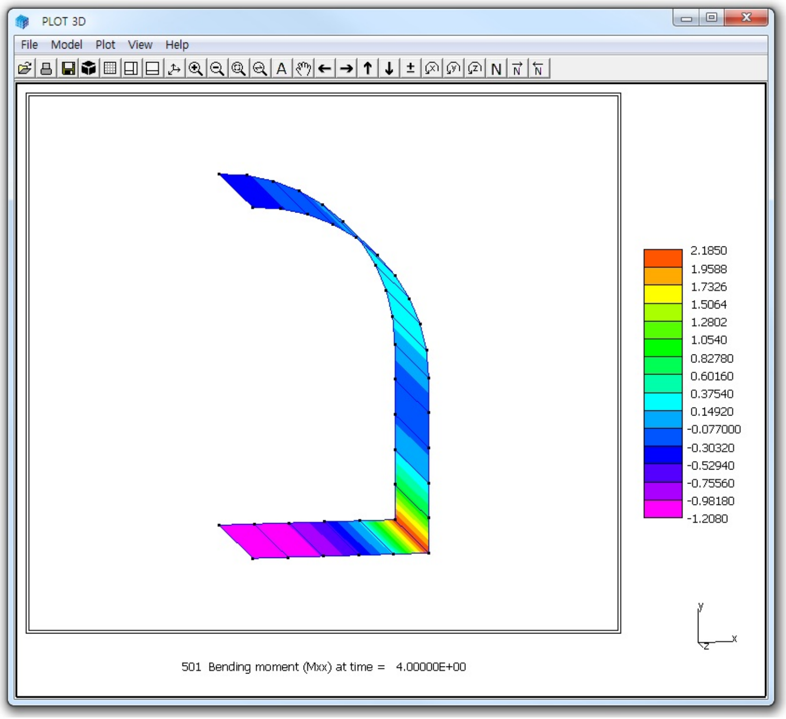


Figure 4.115 Bending moment for Plastic Joint

4.25 Spring Analysis

This example problem is to show how to model springs using special features in beam element in Card 6.4.1 of SMAP-3D User's Manual.

The example is composed of two truss members connected by horizontal and vertical springs as shown in Figure 4.116. The structure is subjected to external horizontal and vertical nodal forces.

Figure 4.117 shows the finite element mesh consisting of two beam elements and two truss elements. Beam element 1 and 2 are used to model vertical and horizontal spring, respectively. When you specify MR = 11 or -11 in Card 6.4.1, beam axial stiffness (E A/L) represents axial spring constant (Ks).

For the material properties, dimensions and loads in Figure 4.116, the exact solution gives following displacements and truss axial forces:

Horizontal	Displacement = 0.04		
Vertical	Displacement = 0.02		
Horizontal	Truss	Axial	Force = 40 (Compression)
Vertical	Truss	Axial	Force = 20 (Tension)

SMAP-3D results show exact as shown in Figures 4.118, and 4.119 for displacements and truss axial forces, respectively.

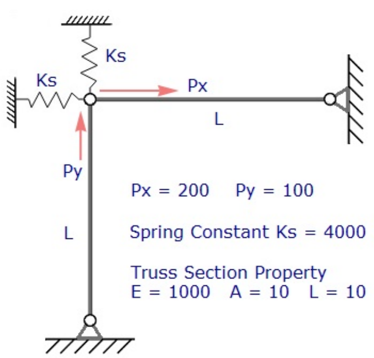


Figure 4.116 Truss members connected by springs

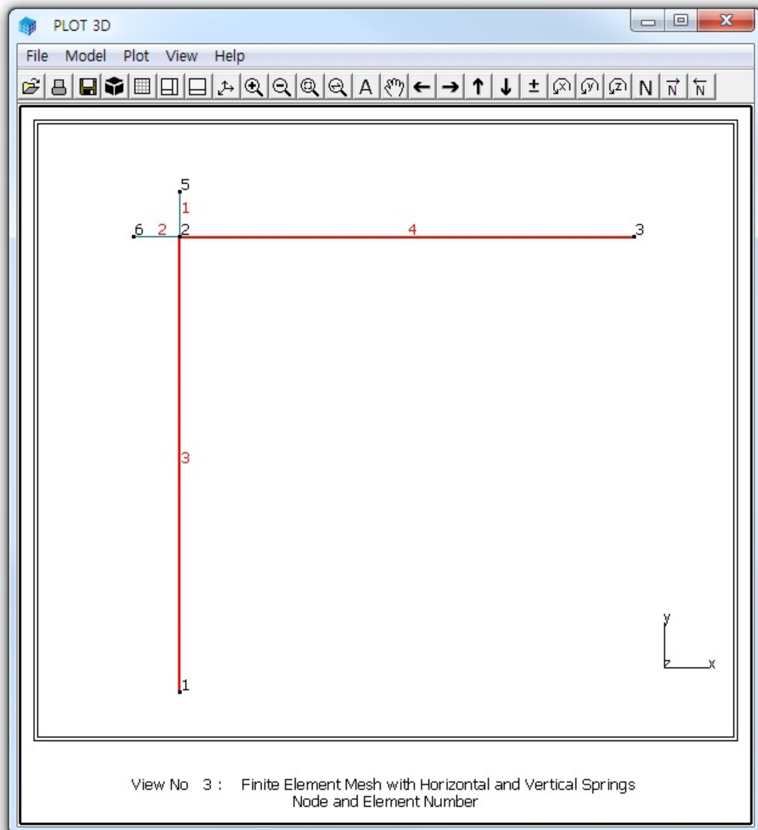


Figure 4.117 Finite element mesh for Example Problem 25

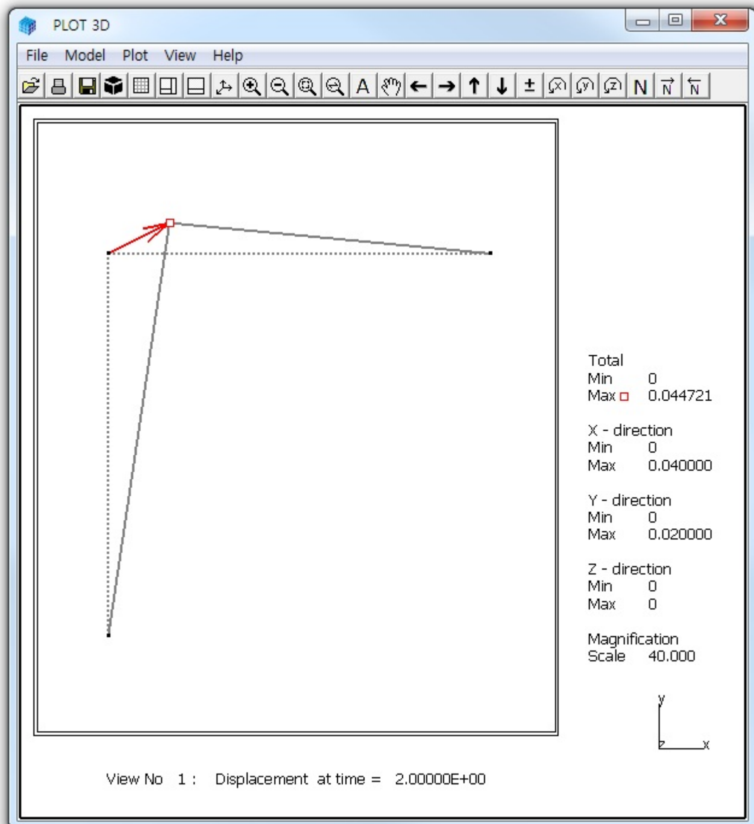


Figure 4.118 Deformed shape for Example Problem 25

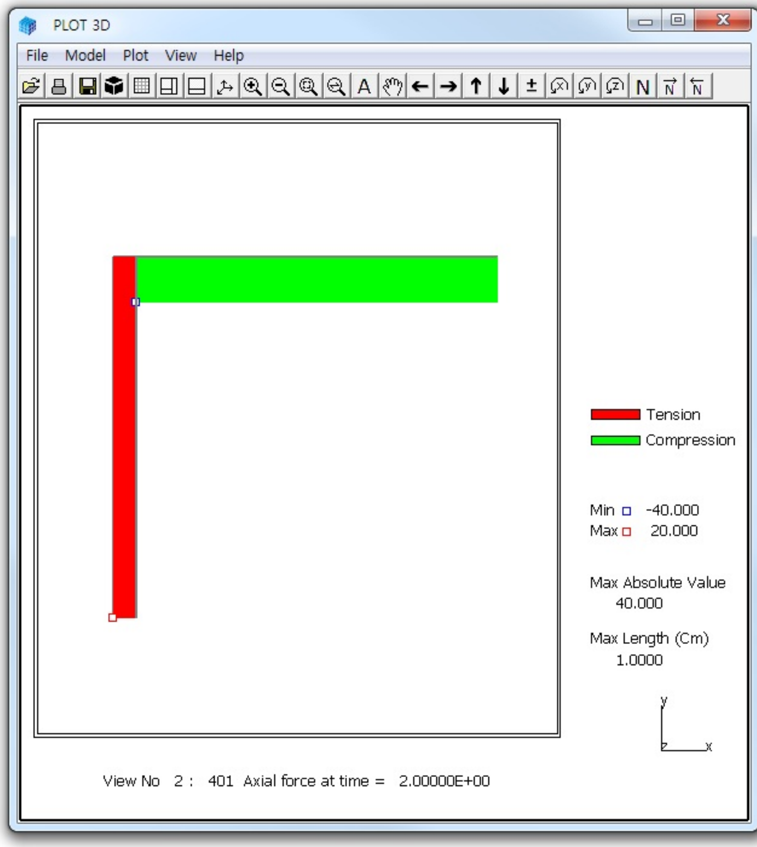


Figure 4.119 Truss axial force for Example Problem 25

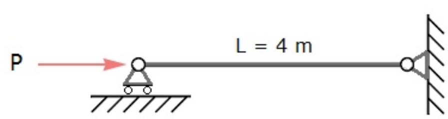
4.26 Nonlinear Truss Analysis

Truss elements in SMAP can consider nonlinear behavior such as yielding and post buckling as schematically illustrated in Figure 4.121. Following examples are to show how to use such material parameters in truss element in Card 7.4.3 of SMAP-3D User's Manual.

Figure 4.120 shows a horizontal truss element subjected to axial force. A typical I-section (400x150@720kN/m) is assumed for truss member with material and cross section properties as listed in the figure.

Six different cases are performed:
1. Buckling and Tension Yielding (Figure 4.122)
2. Compression and Tension Yielding (Figure 4.123)
3. Tension Yielding for No Compression Member (Figure 4.124)
4. Compression Yielding for No Tension Member (Figure 4.125)
5. Buckling for No Tension Member (Figure 4.126)
6. Initial Stress (See Case 6 at the end of example)

Compression resistance is not allowed for No Compression Member such as cable and tension resistance is not allowed for No Tension Member such as strut. A linear elastic truss element is added to prevent the structure from being unstable when plastic yielding. Both compression and tension yield strengths are increased more than 12 times in order to make an exaggerated graphical presentation associated with load and unload.



I-Section (400x150@720 kN/m)

I_x = 2400 Cm^4 I_y = 887 Cm^4
E = 21000 kN/Cm^2 A = 91.73 Cm^2
σ_y = 32.5 kN/Cm^2 σ_{cr} = 12.52 kN/Cm^2

Figure 4.120 Truss member subjected to axial force

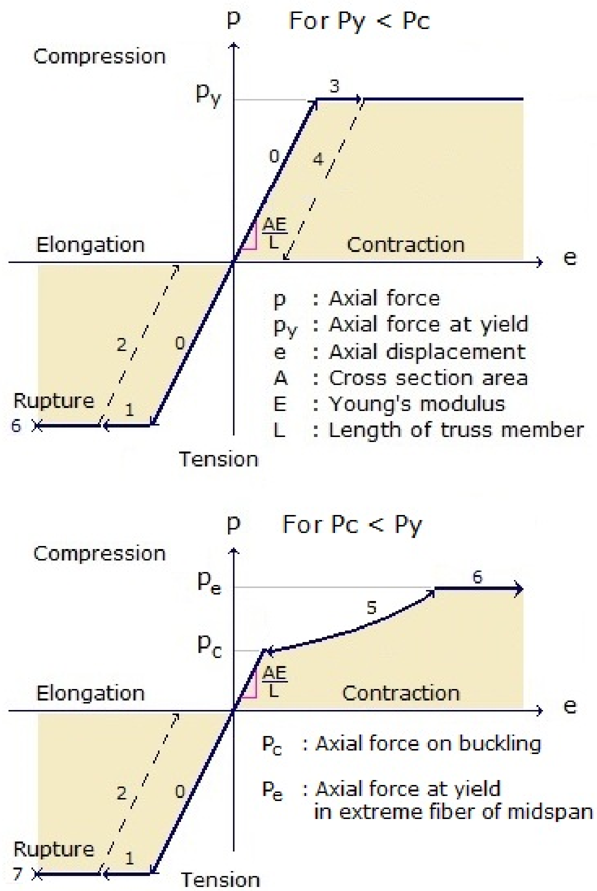


Figure 4.121 Nonlinear force displacement curve

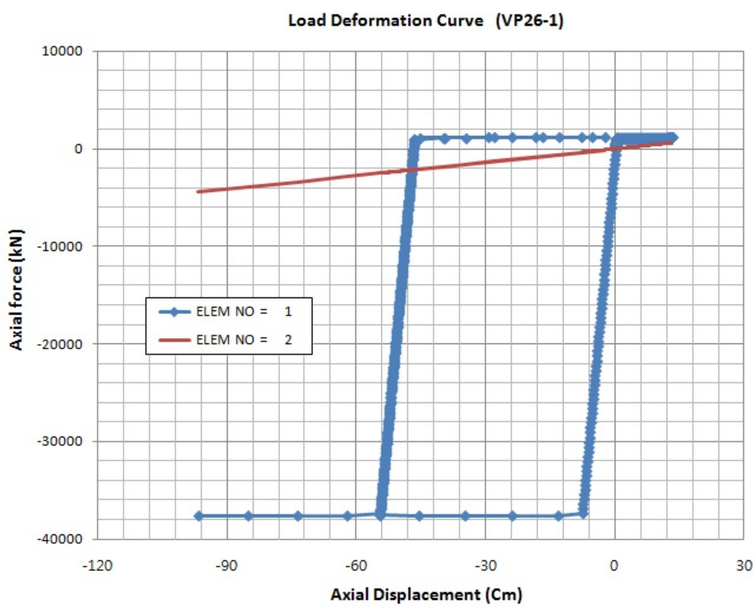


Figure 4.122 Buckling and Tension Yielding (VP26-1)

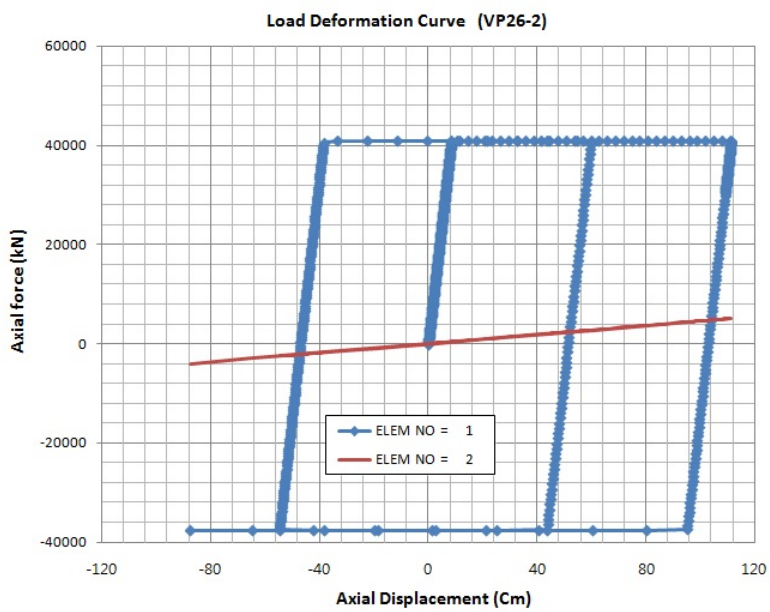


Figure 4.123 Compression and Tension Yielding (VP26-2)

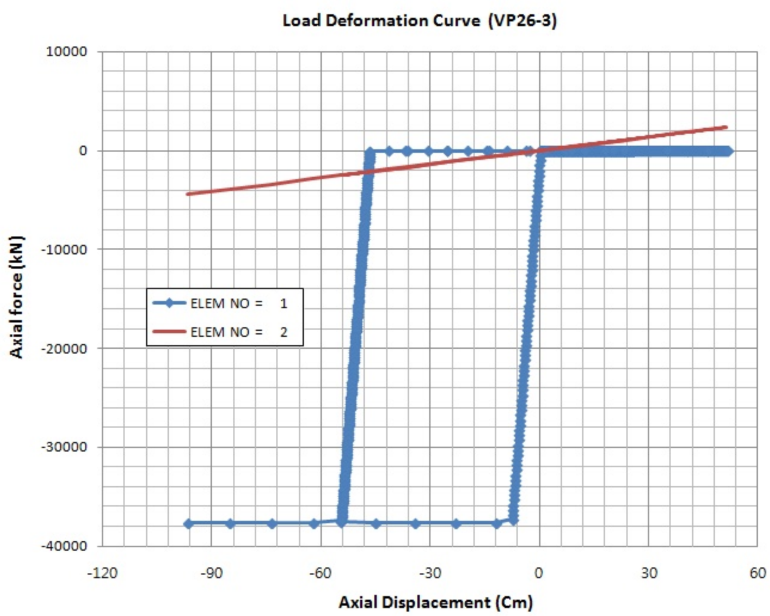


Fig 4.124 Tension Yielding for No Compression Member (VP26-3)

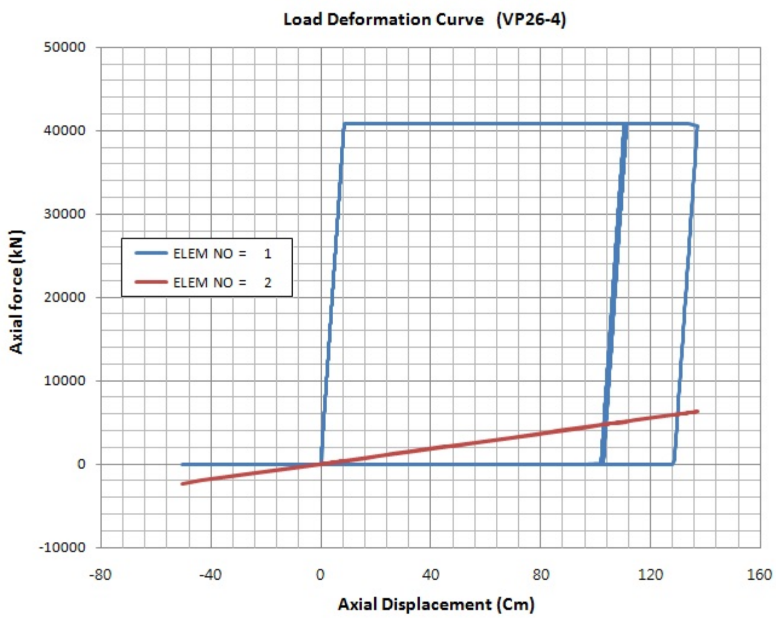


Fig 4.125 Compression Yielding for No Tension Member (VP26-4)

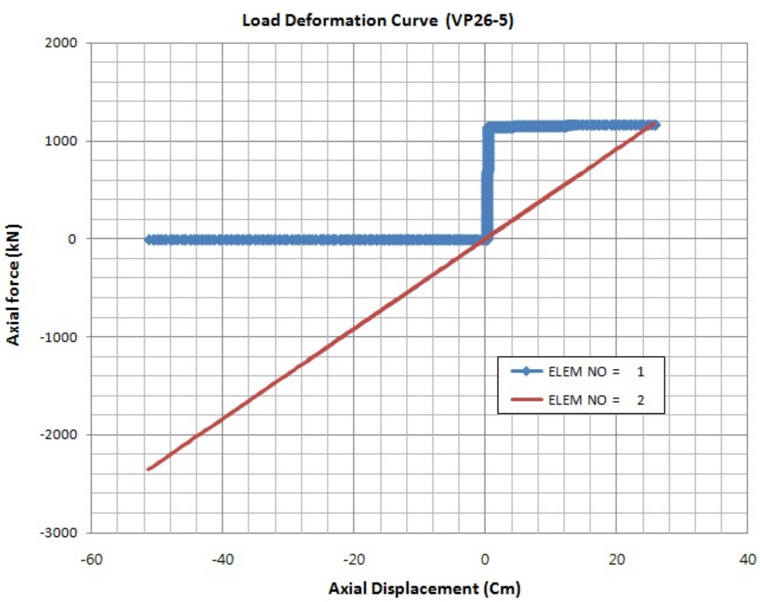


Figure 4.126 Buckling for No Tension Member (VP26-5)

Case 6 Initial Stress

For this example, following parameters are used:
$L = 400$ Cm $E_1 = 21000$ kN/Cm^2 $E_2 = 1000$ kN/Cm^2

To check Initial Stress, Member 1 is assumed to have initial compressive stress ($\sigma_i = -10$ kN/Cm^2) with the corresponding initial strain ($\varepsilon_i = \sigma_i / E_1 = -0.00047619$).

Thus the original length of Member 1 at stress free
$Lo = L / (1 + \varepsilon_i) = 400 / (1 - 0.00047619) = 400.19057$ Cm

Now, when Members 1 and 2 are connected,

$$\sigma_1 \cdot A + \sigma_2 \cdot A = P = 0 \quad \text{i.e.} \quad \sigma_2 = -\sigma_1 \qquad (1)$$

$$\sigma_2 = E_2 \cdot \varepsilon_2 \qquad (2)$$

$$\begin{aligned} \varepsilon_1 &= ((L + \Delta L\ \ ) - Lo) / Lo \\ &= ((L + \varepsilon_2 \cdot L) - Lo) / Lo \\ &= (L / Lo) \cdot (1 + \varepsilon_2) - 1 \qquad (3) \end{aligned}$$

$$\begin{aligned} \sigma_1 &= E_1 \cdot \varepsilon_1 \\ &= (E_1 \cdot L / Lo) \cdot (1 + \varepsilon_2) - E_1 \qquad (4) \end{aligned}$$

Substituting (2) and (4) into (1),

$$\begin{aligned} \varepsilon_2 &= E_1 (1 - L / Lo) / (E_2 + E_1 \cdot L / Lo) \qquad (5) \\ &= 0.00045475 \end{aligned}$$

From (3)

$$\varepsilon_1 = -0.000021654$$

And from (2) and (1)

$\sigma_1 = -0.45475$ kN/Cm^2 (Compression)
$\sigma_2 = 0.45475$ kN/Cm^2 (Tension)

SMAP results show exact solution.

4.27 SDOF System To Ground Acceleration

A single Truss element is used to model axial spring subjected to sinusoidal ground acceleration as schematically shown in Figure 4.127. Mass is lumped at the node in the right side of truss member.

Following parameters are assumed:

L = 120 inch A = 1 in^2 $E = 30 \times 10^6$ psi

ρ = (1/1.2) $lb\text{-}s^2/in^4$ a = 200 in/s^2 ω = 40 rad/s

c = 500 lb-s/in

Lumped mass at right node:

$m = \rho A L = (1/1.2)(1)(120) = 100$ $lb\text{-}s^2/in$

Equivalent spring constant:

$k = E A / L = (30 \times 10^6)(1)/(120) = 250{,}000$ lb/in

Natural frequency:

$\omega_n = (k/m)^{1/2} = (250{,}000/100)^{1/2} = 50$ rad/s

Critical damping ratio: $\xi = c / (2 m \omega_n) = 0.05$

Damped natural frequency : $\omega_d = \omega_n \sqrt{1 - \xi^2}$

Frequency ratio: $\beta = \omega / \omega_n = 40/50 = 0.8$

For systems with viscously damped single degree of freedom, the relative displacement is given by

$$\bar{x}(t) = e^{-\xi \omega_n t}(A \cos \omega_d t + B \sin \omega_d t) + C \sin \omega t + D \cos \omega t$$

The constants C and D are given by

$$C = \frac{m a}{k} \frac{1 - \beta^2}{(1 - \beta^2)^2 + (2 \xi \beta)^2} \qquad D = \frac{m a}{k} \frac{-2 \xi \beta}{(1 - \beta^2)^2 + (2 \xi \beta)^2}$$

Assuming initial conditions at rest, constants A and B are given by

$$A = -D \qquad B = -\left(\frac{\omega}{\omega_d}\right) C - \xi \left(\frac{\omega_n}{\omega_d}\right) D$$

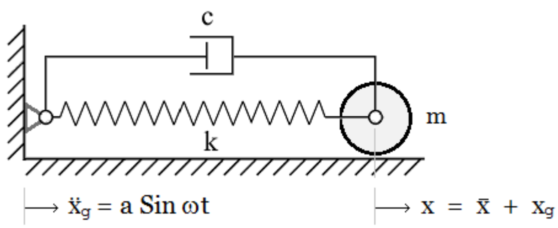


Figure 4.127 SDOF system subjected to ground acceleration

Figure 4.128 shows time history of computed relative displacements. SMAP results are almost identical to the exact solution.

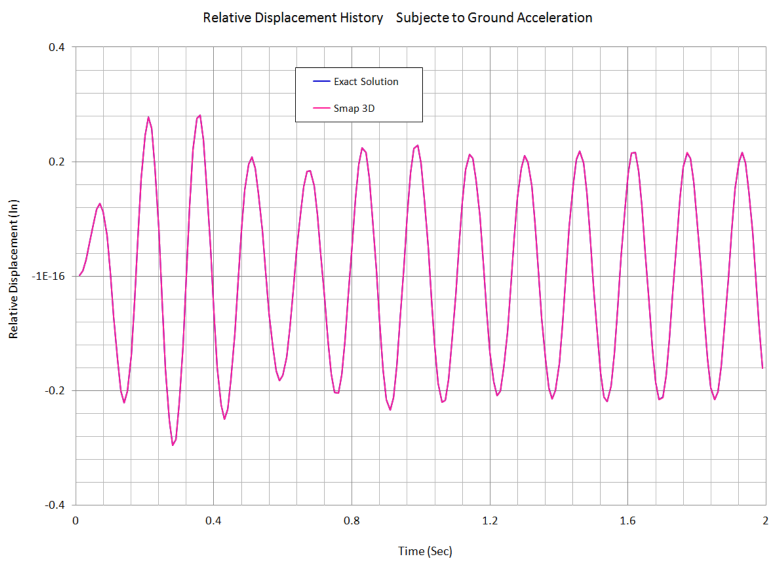


Fig 4.128 Time history of relative displacements

4.28 Frames with Rotational Spring Connection

This example is the same as Example problem 21 except that it is connected by rotational spring and subjected to both moment and horizontal force at the connection as shown in Figure 4.129.

The rotational spring is modeled by the simple Joint Spring Element which can consider axial, shear, torsional and flexural resistances. For this example, the Joint Spring properties are assumed very rigid in all deformation modes except the rotation about z-axis.

Five analyses are performed to see the influence of connection:

1. Rigid connection
2. Hinge connection
3. Rotational spring connection, rigid Kr = $1x10^{6}$ t-m/rad
4. Rotational spring connection, very flexible Kr = $1x10^{-3}$ t-m/rad
5. Rotational spring connection, somewhat rigid Kr = $1x10^{4}$ t-m/rad

Computed results are summarized in detail in Joint_Spring_3D.pdf. It approaches to rigid connection when the rotational spring is rigid and hinge connection when the spring constant is very flexible.

Figures 4.130 to 4.134 show finite element mesh, deformed shape, thrust, shear and bending moment distributions, respectively, for the rotational spring connection with Kr = $1x10^{4}$ t-m/rad.

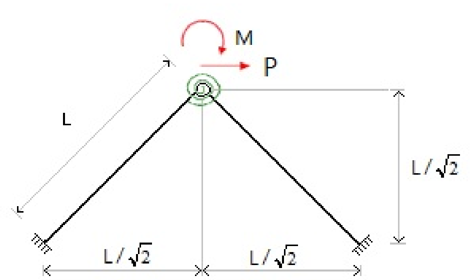


Fig 4.129 Frames with rotational spring connection

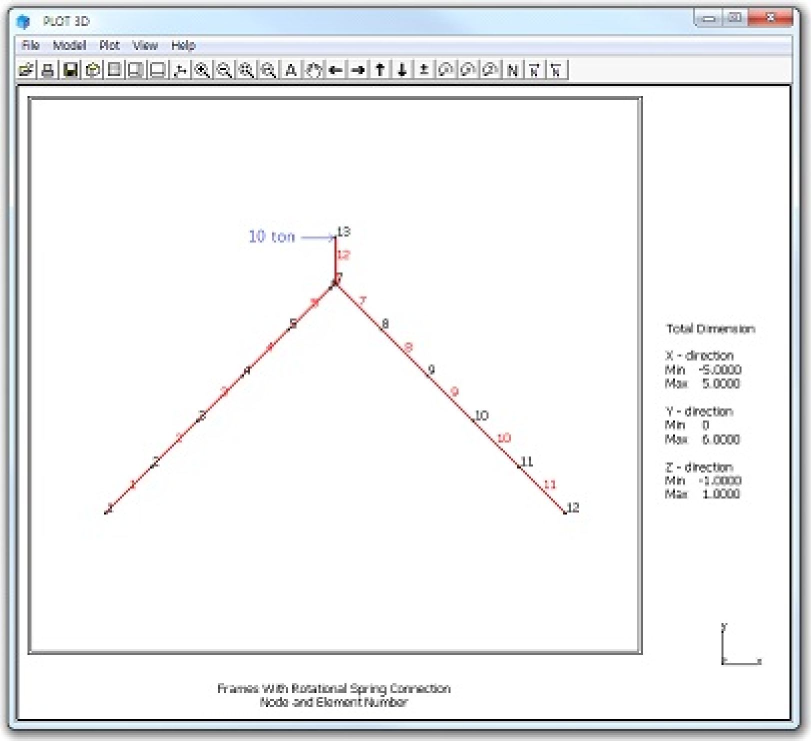


Figure 4.130 Finite element mesh

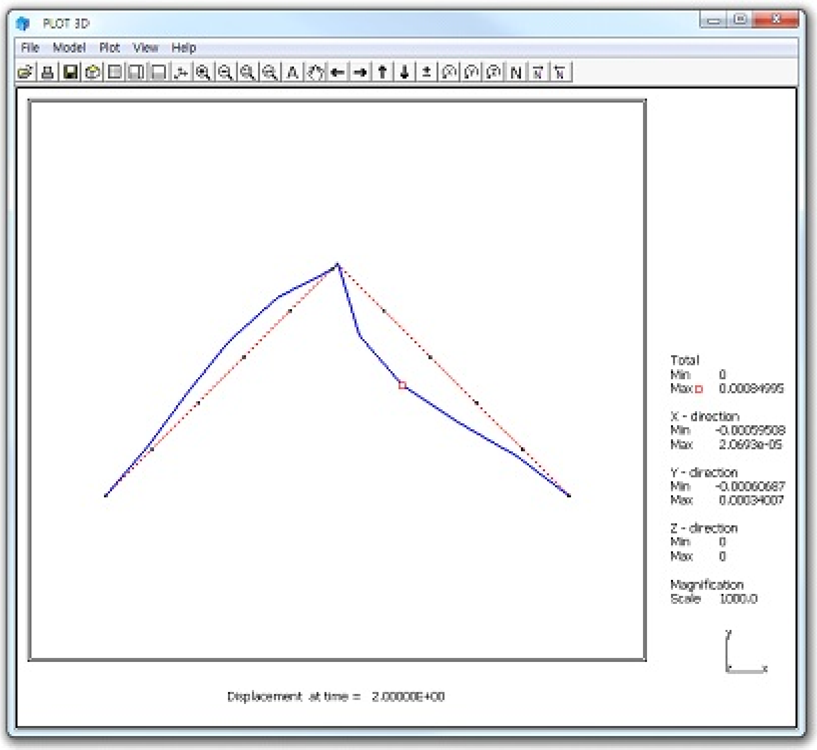


Figure 4.131 Deformed shape

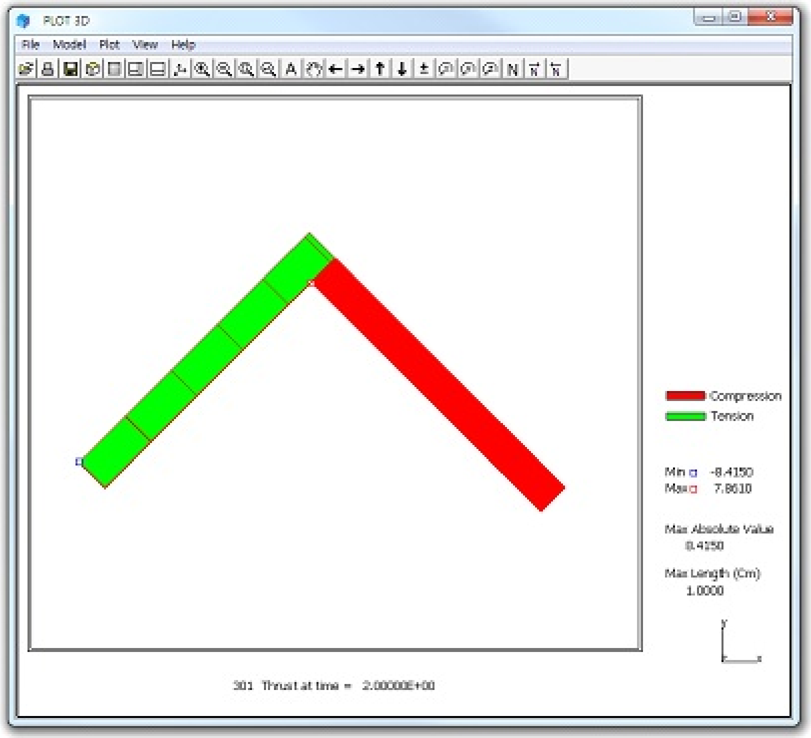


Figure 4.132 Thrust distribution

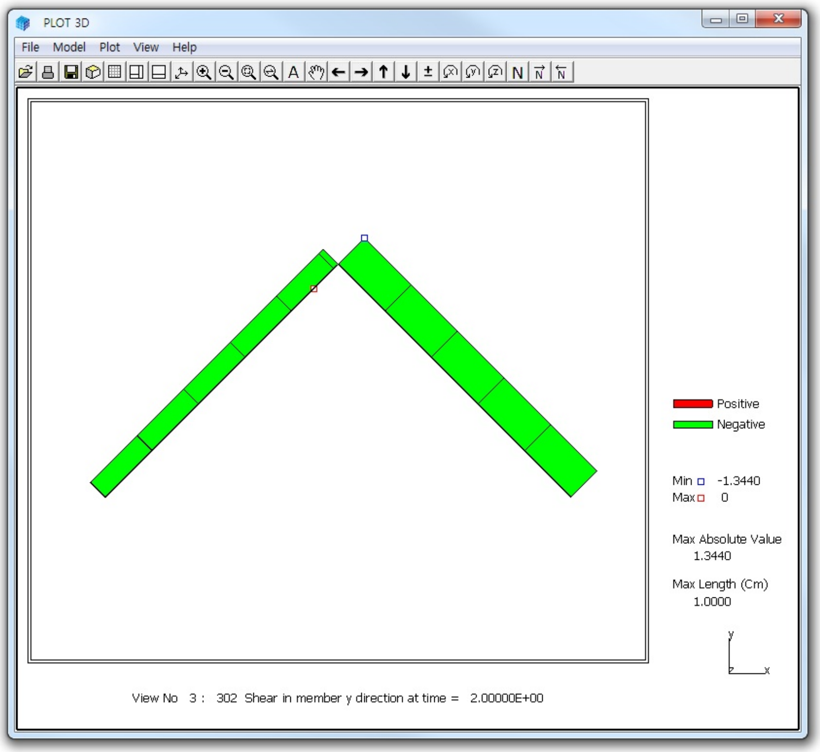


Figure 4.133 Shear distribution

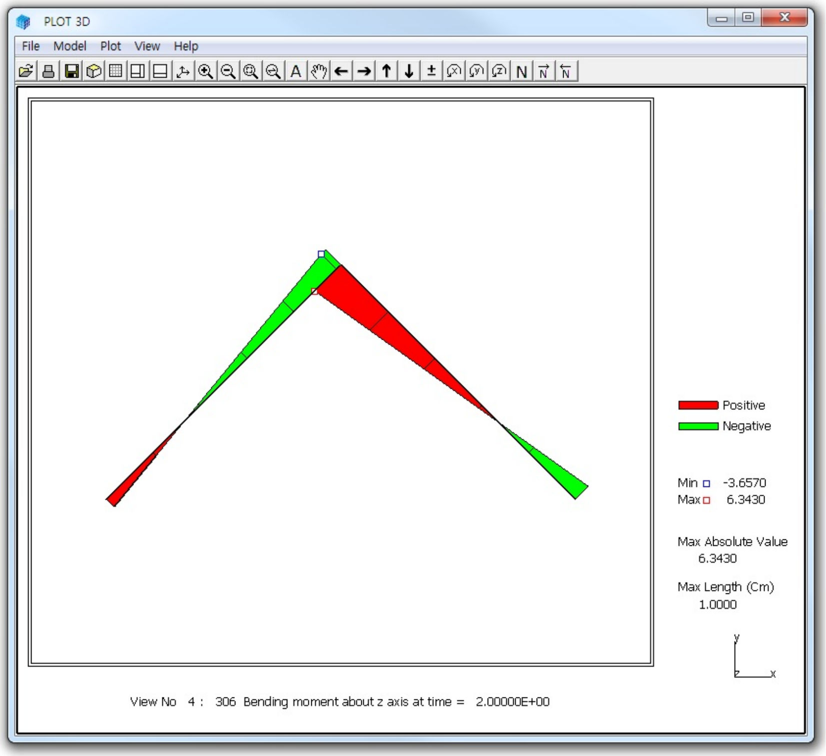


Figure 4.134 Bending moment distribution

4.29 Reinforced Concrete Beam

This example problem is to verify the implementation of reinforcing bars (rebars) into quadrilateral shell element. This example is the same as Example problem 22 except that it is modeled by reinforced shell element. Figure 4.135 shows a simply supported reinforced concrete beam subjected to a concentrated load at midspan.
To simplify the problem, it was assumed that both reinforcing bars and concrete are linearly elastic.

The exact beam solution without shear deformation is given below:

Maximum deflection at the center without rebars,

$$\delta = \frac{P \cdot L^3}{48 \ E_c \cdot I_c} = 1.190 \text{ Cm}$$

Maximum deflection at the center with rebars,

$$\delta = \frac{P \cdot L^3}{48 \ E_c \cdot I_t} = 1.040 \text{ Cm}$$

By symmetry, only left half of the beam is modeled using 10 reinforced shell elements.

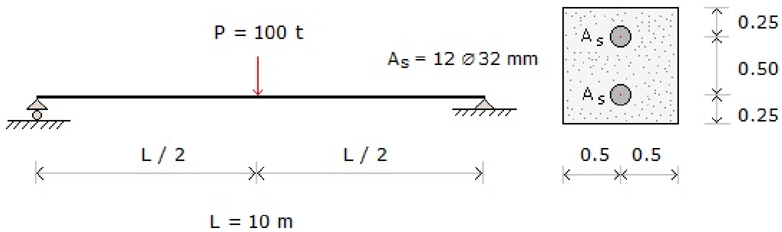
The computed center deflections are compared with the exact beam solution as shown in Table 4.6. SMAP-3D results are very close to the exact beam solutions.

Computed results are shown in the following order:

Figure 4.136	Deformed shape
Figure 4.137	Bending moment
Figure 4.138, 4.139	Top and bottom surface axial stress
Figure 4.140, 4.141	Top and bottom reinforing bar axial stress

Table 4.6 Computed center deflections

Reinforcement	SMAP-3D Shell Element	Exact Beam Solution
Plain Concrete	1.1812 Cm	1.190 Cm (without rebar)
Reinforced Concrete	1.0329 Cm	1.040 Cm (with rebar)



E_c = 2.1×10^6 t/m^2

v_c = 0.2

E_s = 2.1×10^7 t/m^2

I_c = 0.0833 m^4

I_s = $2\,(E_s / E_c)\,A_s\,(0.025)^2$ = 0.012063 m^4

I_t = $I_c + I_s$ = 0.0954 m^4

Shell Element Input Parameter for Reinforcing Bars

NRBX = 1, NRBY = 0, d1 = d2 = 0.25 m

As1 = As2 = 0.00965 m^2

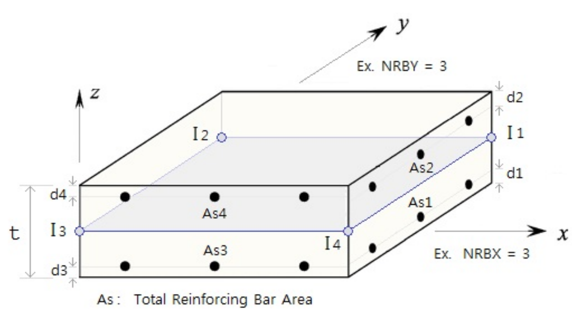


Figure 4.135 Reinforced concrete beam

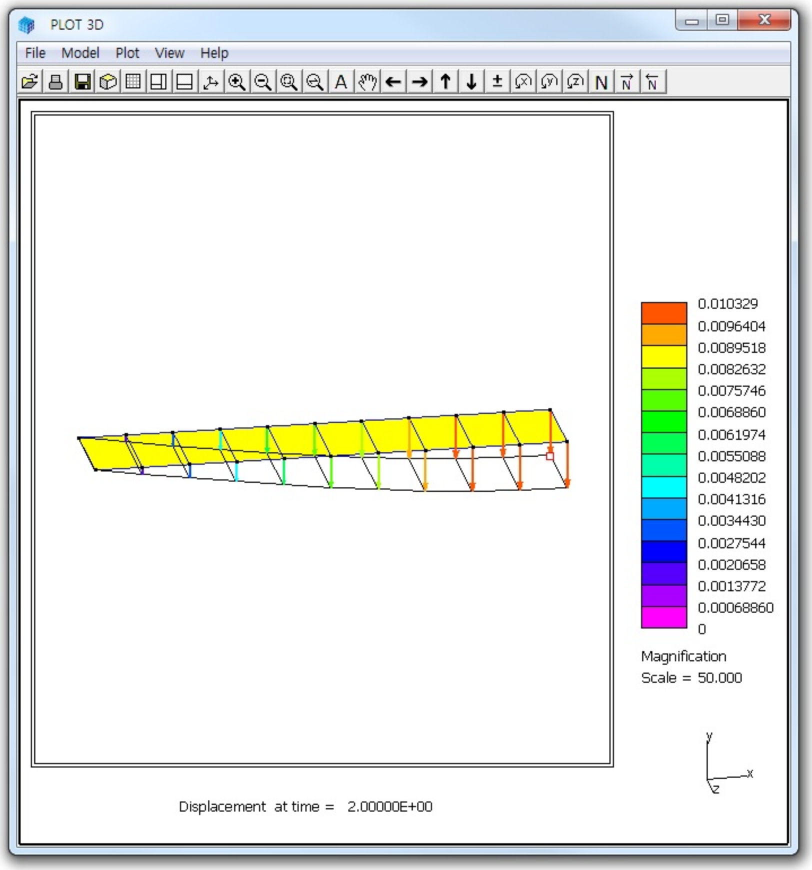


Figure 4.136 Deformed shape

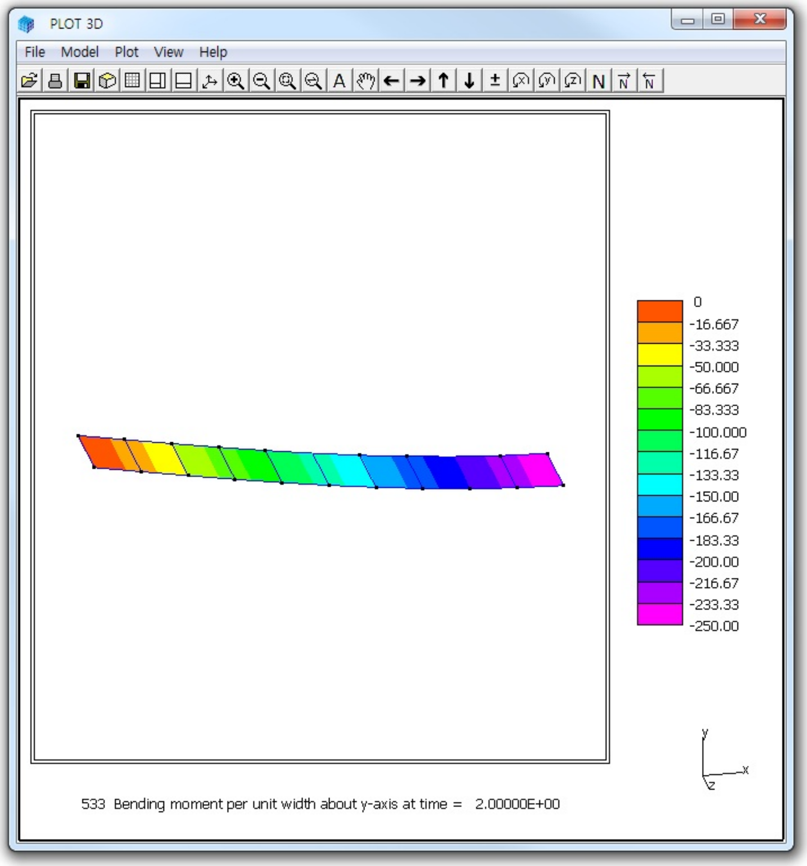


Figure 4.137 Bending moment about y-axis

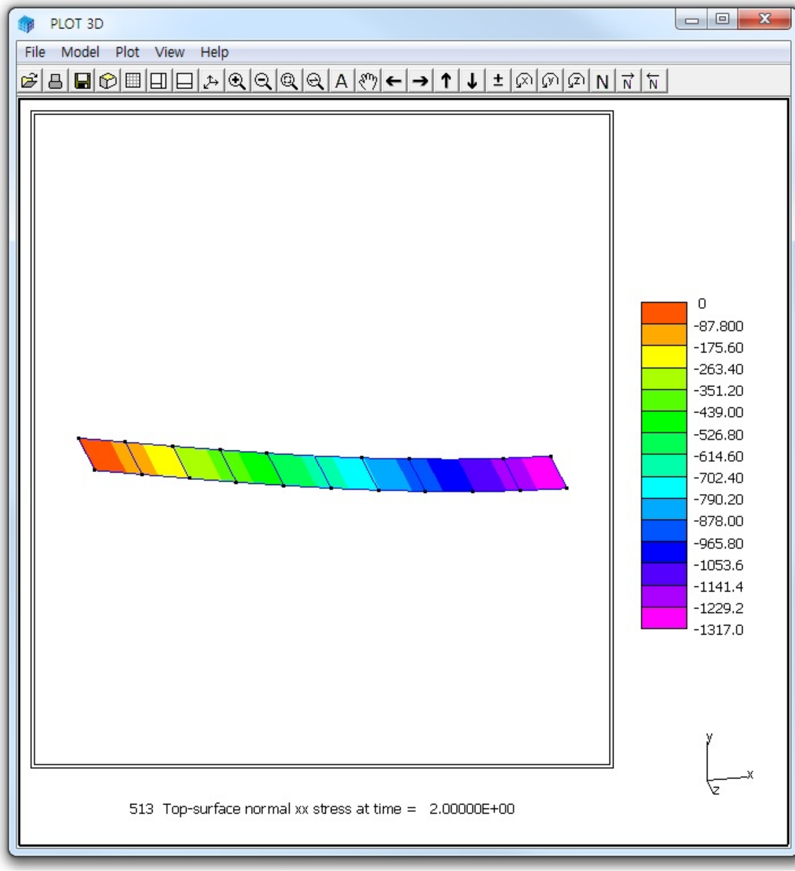


Figure 4.138 Top surface axial stress

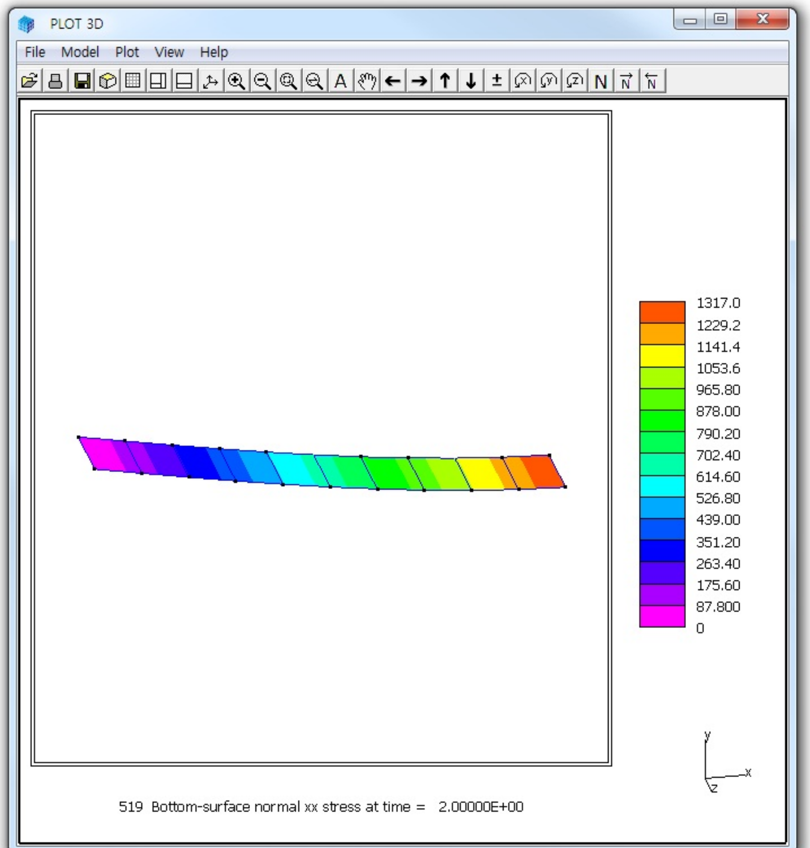


Figure 4.139 Bottom surface axial stress

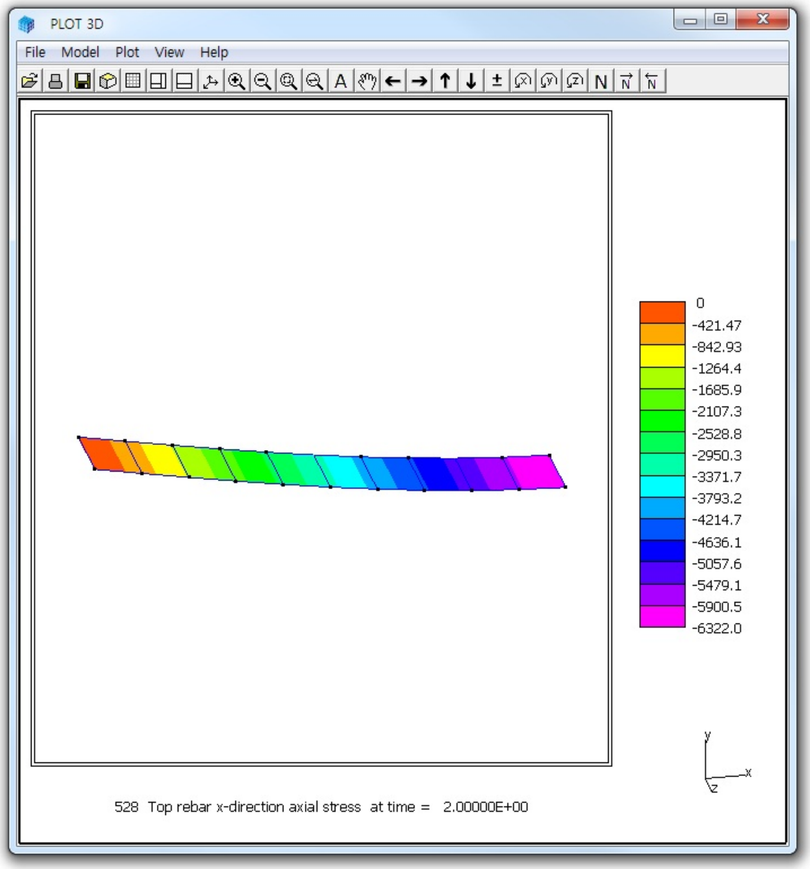


Figure 4.140 Top reinforcing bar axial stress

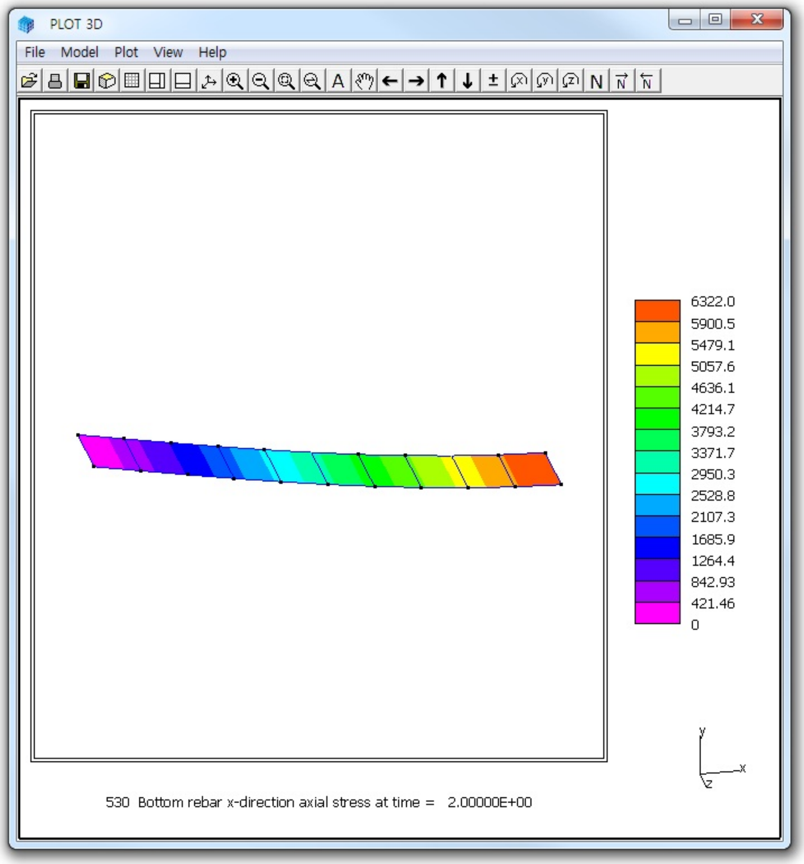


Figure 4.141 Bottom reinforcing bar axial stress

4.30 Reinforced Concrete Cylinder

This example is to check the reinforced concrete cylinder subjected to uniformly distributed radial line loads as shown in Figure 4.142.
This example is an axially symmetric problem since both the structure and the external load are axially symmetric.

The exact solution for unreinforced cylinder can be obtained from the reference: Timoshenko and Woinowsky-Krieger, Theory of Plates and Shells, 2nd Edition, McGraw-Hill International Series, 28th Printing 1989.

This exact solution is further modified here such that it includes both axial (meridian) and hoop (circumferential) reinforcements as listed in the file Reinforced_Cylinder_3D.pdf.

Four cases are performed with different reinforcements:

1. Concrete without reinforcements
2. Concrete with hoop reinforcements
3. Concrete with axial & hoop reinforcements, $\nu_c = 0.15$
4. Concrete with axial & hoop reinforcements, $\nu_c = 0.0$

Note that the analytical solutions represent exact solutions except the case 3 where it is an approximate closed-form solution.

As in Figure 4.143, the structure is modeled by quadrilateral shell elements which have capability of modeling two way reinforcements.

Overall, SMAP-3D results are very close to the exact solutions.
Refer to the following two files for detailed graphical outputs:
Reinforced_Cylinder_3D.pdf and Smap-3D_Vp30.pdf.

SMAP-3D results for case 3 are compared with closed-form solutions:
Figure 4.144 Radial displacement profile
Figure 4.145 Meridian bending moment profile

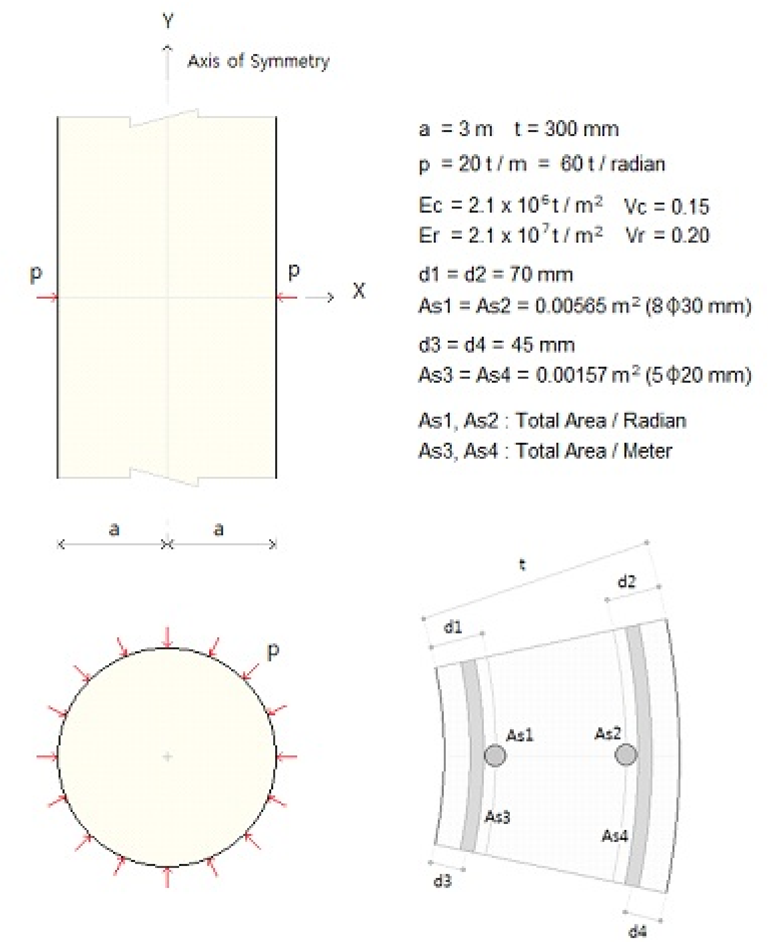


Figure 4.142 Reinforced cylinder section view

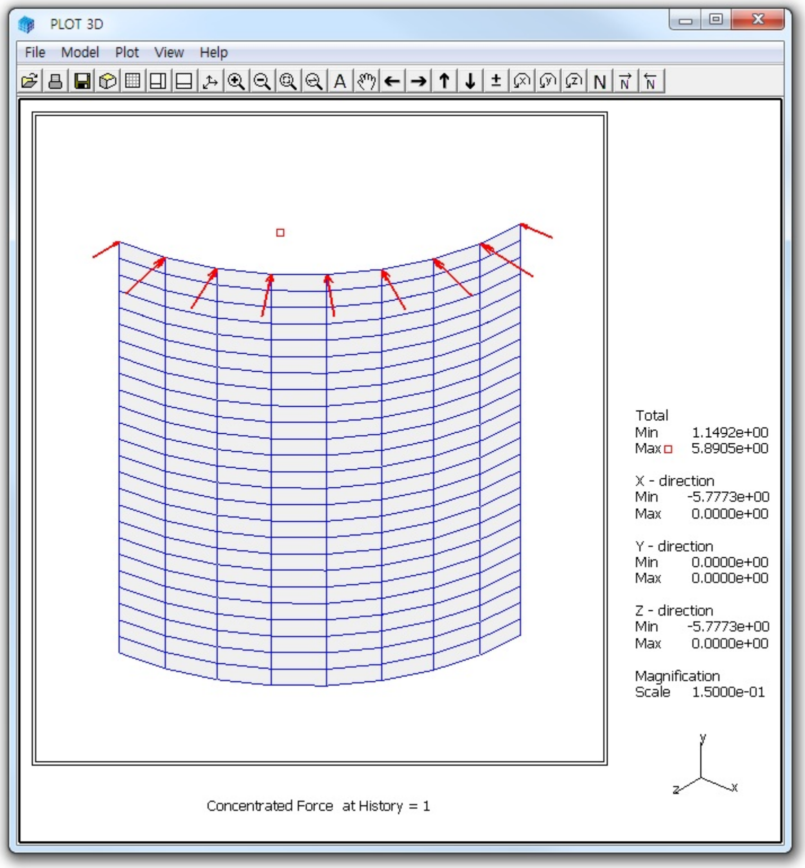


Figure 4.143 Finite element mesh with applied load

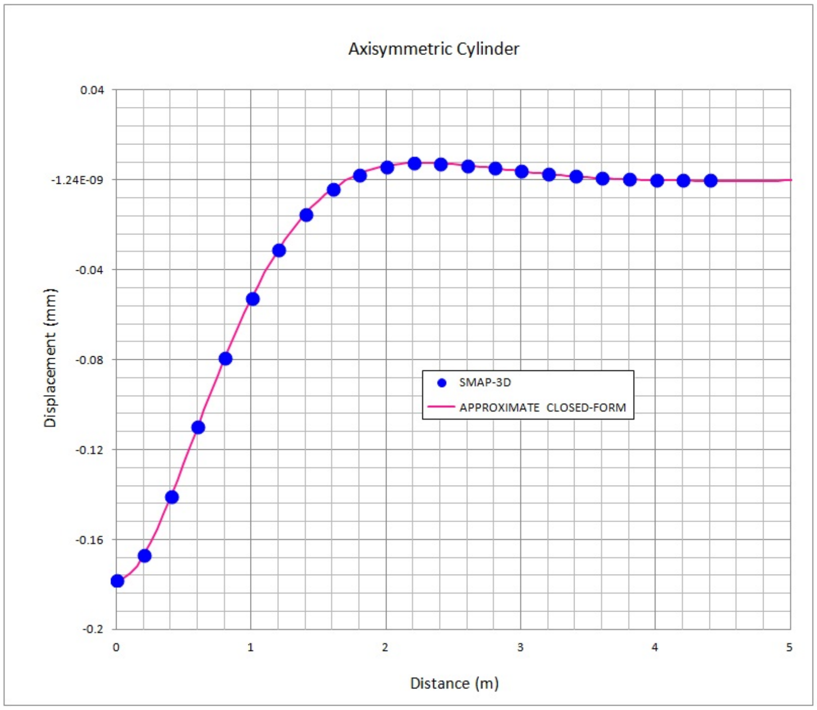


Figure 4.144 Radial displacement profile

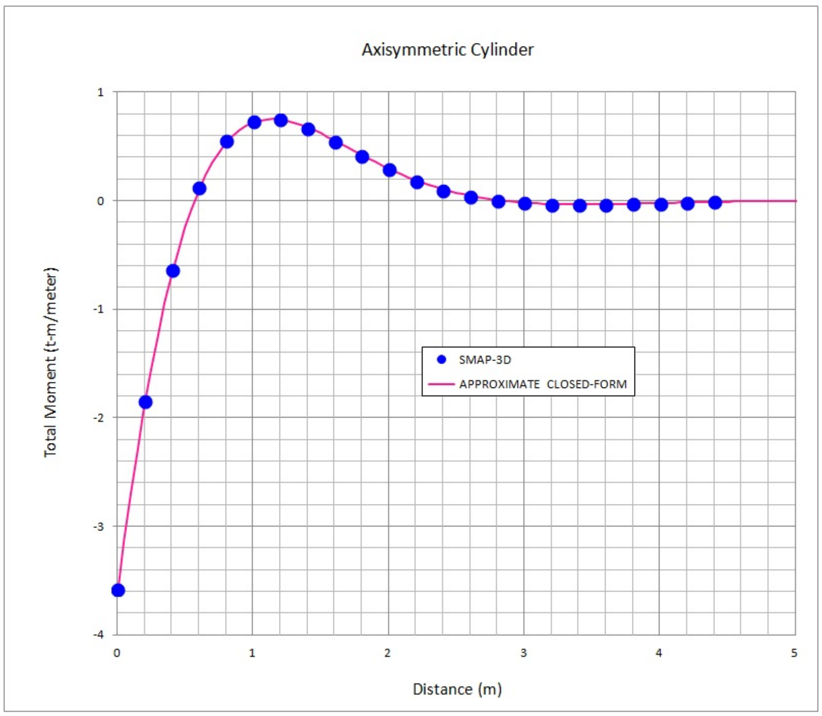


Figure 4.145 Meridian bending moment profile

4.31 Plate Modal Analysis

A simply supported rectangular plate, shown in Figure 4.146, is selected to verify the Modal Superposition method for the dynamic response. By symmetry, only a quarter of the plate is modeled. The plate is subjected to a concentrated step load at center. This problem is identical to the Verification Problem 4.16 which was solved by Direct Integration method.

The closed form solution of natural frequencies of simply supported rectangular plate is given by Kirchhoff plate theory:

$$\omega_{mn} = \sqrt{\frac{D}{\rho h}} \left[\left(\frac{m\pi}{a}\right)^2 + \left(\frac{n\pi}{b}\right)^2 \right] \qquad D = \frac{E h^3}{12(1 - v^2)}$$

$\rho = 0.0003$ lb-s^2 / in^4 $\quad v = 0.25 \quad h = 1$ in

$E = 3\times10^4$ lb / in^2 $\quad a = 60$ in $\quad b = 40$ in

Table 4.7 summarizes the computed natural frequencies along with closed form solution. Both shell and continuum modal analyses predict pretty well natural frequencies of the simply supported rectangular plate.

Figure 4.147 shows the contours of the first three modes solved by shell modal analysis.

Figure 4.148 shows deflection time history at plate center as predicted by modal superposition method using only first 6 mode shapes.
To verify the computed response of the modal superposition method, step-by-step solution by direction integration with the same shell element mesh which was used in shell modal superposition is included. SMAP-3D modal superposition solutions predict very closely the direct integration solution.

Table 4.7 Computed natural frequencies (rad/s)

Mode No	Kirchhoff Plate Theory	Shell 4 Node Quad 16x24 Mesh	Continuum 8 Node Hexa* 8x12 Mesh
1	26.565	26.544	26.412
2	91.955	91.729	91.356
3	173.693	172.992	173.411

Notes:

1. Computed frequencies represent natural frequencies associated with symmetric boundary conditions.
   $\omega_1 = \omega_{11}$, $\omega_2 = \omega_{31}$, $\omega_3 = \omega_{13}$

2. All modal analyses used Subspace Iteration method with lumped mass to compute natural frequencies.

3. Shell modal analysis used 16x24 mesh consisting of 4 node quadrilateral shell elements.

4. Continuum modal analysis used 8x12 mesh consisting of 8 node hexahedral continuum elements with 3 incompatible extra degrees of freedom* (IEDOF =1).

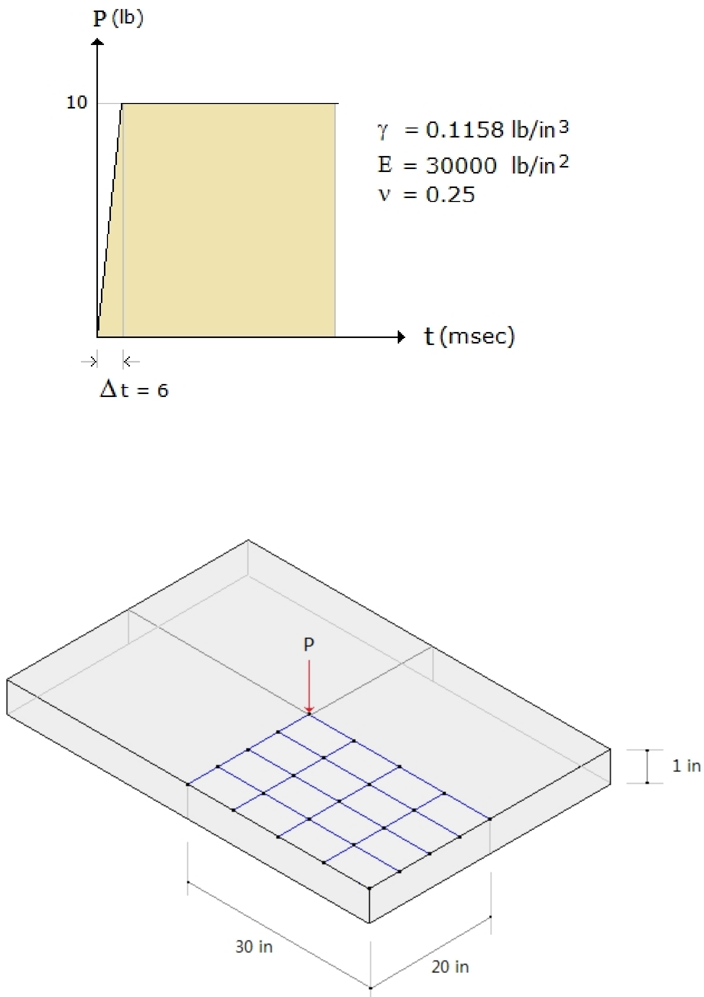


Figure 4.146 Simply supported plate (Not Scaled)

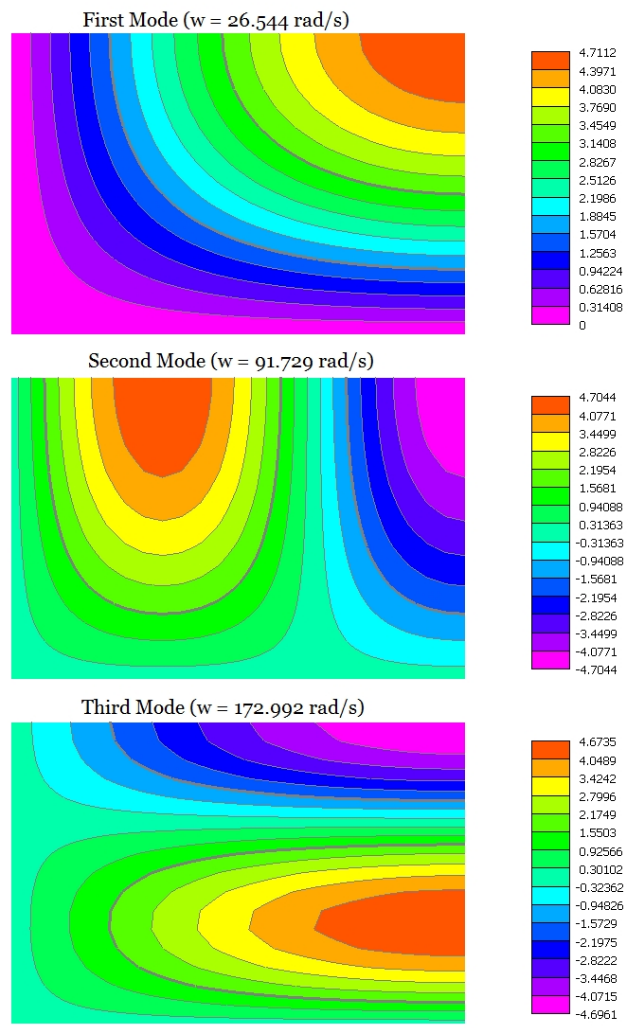


Figure 4.147 First three modes solved by Shell (16x24)

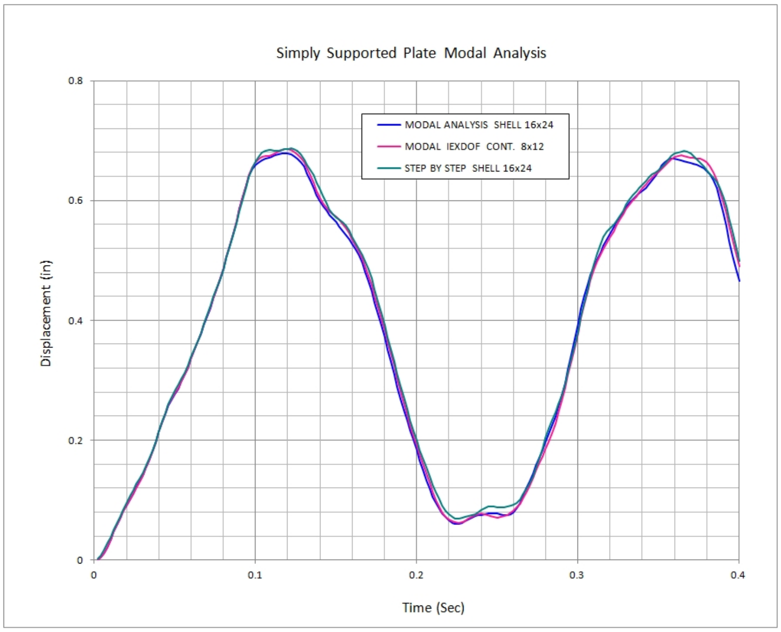


Figure 4.148 Deflection time history at plate center

4.32 Seismic Response Analysis

This example is to solve the free-field seismic response of the linearly viscous elastic soil profile, shown in Figure 4.149 along with material properties, subjected to earthquake excitations from the bedrock.

This problem is the same as the sample problem in SHAKE91 (Idriss and Sun, 1992). A 45.72 m (150 ft) soil profile is subjected to Diamond Heights earthquake in 1989 as outcrop to the elastic half space. The earthquake is scaled to peak acceleration of 0.1g. Scaled earthquake time history and its spectral acceleration are shown in Figures 4.150 and 4.151, respectively. The predominant period of the earthquake is about 0.4 second as shown in the response spectrum.

To mitigate frequency dependency, Rayleigh mass and stiffness proportional damping constants (a, b) are computed in the equation:

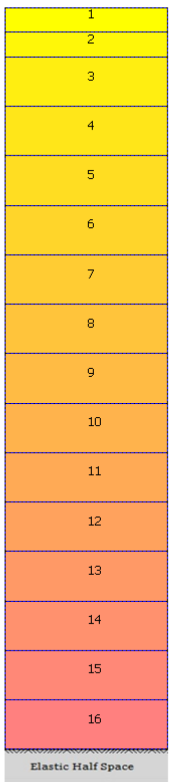
$$a = 2\,\beta\,\omega_1\,\omega_i \,/\, (\,\omega_1 + \omega_i\,) \qquad b = 2\,\beta \,/\, (\,\omega_1 + \omega_i\,)$$

where ω_1 represents for fundamental natural circular frequency of soil profile, ω_i for predominant circular frequency of the input earthquake motion and β for critical damping ratio in an element.

Figure 4.152 shows computed acceleration time histories on the ground surface and Figure 4.153 shows the same accelerations between 10 and 12 seconds where strong motions occur. SMAP-3D solutions predict very closely the closed-form frequency domain SHAKE91 solution.

Figure 4.154 shows spectral accelerations with 5% structural damping on the ground surface and Figure 4.155 shows the same accelerations between 0.1 and 1 seconds. SMAP-3D solutions are very close to SHAKE91 solution.

It should be noted that both base shear and base acceleration options for earthquake load produce exactly the same results as presented in the reference (S. H. Kim and K. J. Kim, 2024).



γ (kN/m³)	G (MPa)	β (%)
19.63	185.963	5
19.63	150.608	5
19.63	150.608	5
19.63	167.878	5
19.63	185.963	5
19.63	185.963	5
19.63	225.042	5
19.63	225.042	5
20.42	326.859	5
20.42	326.859	5
20.42	379.120	5
20.42	379.120	5
20.42	435.255	5
20.42	435.255	5
20.42	495.264	5
20.42	626.675	5
21.99	3333.116	1

Total Dimension

X - direction
Min 0
Max 10.000

Y - direction
Min 0
Max 45.720

Figure 4.149 Finite element meshes and material properties

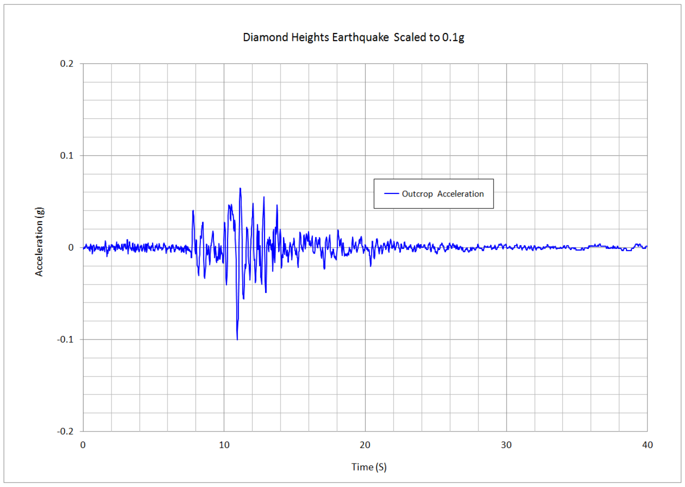


Figure 4.150 Diamond Heights acceleration time history

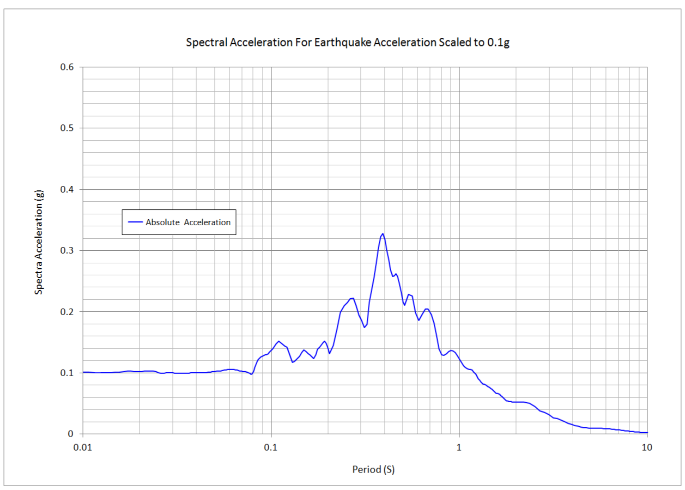


Figure 4.151 Spectral acceleration for input earthquake

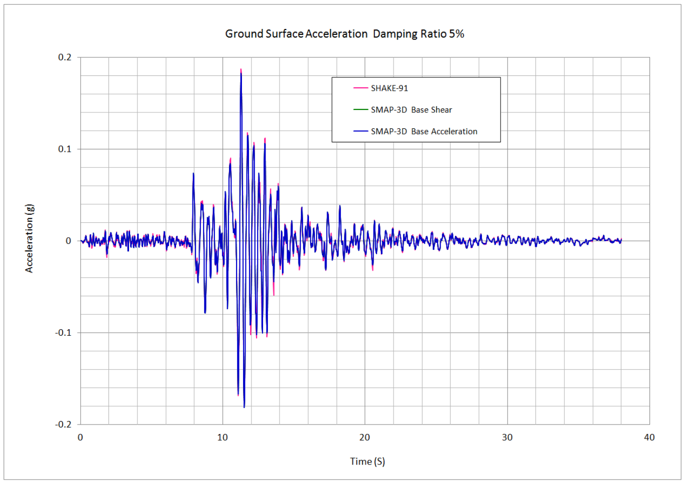


Figure 4.152 Ground surface accelerations

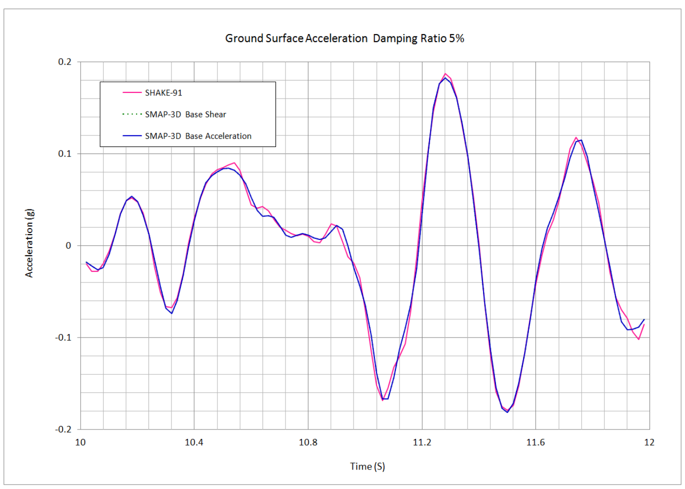


Figure 4.153 Ground surface accelerations between 10 and 12 sec.

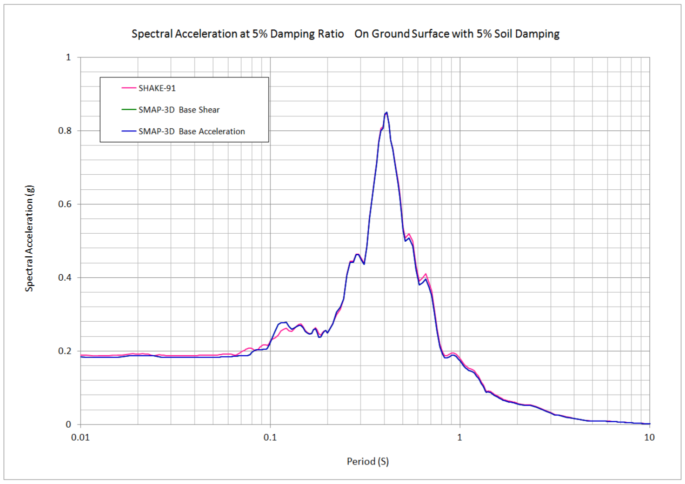


Figure 4.154 Spectral accelerations on ground surface

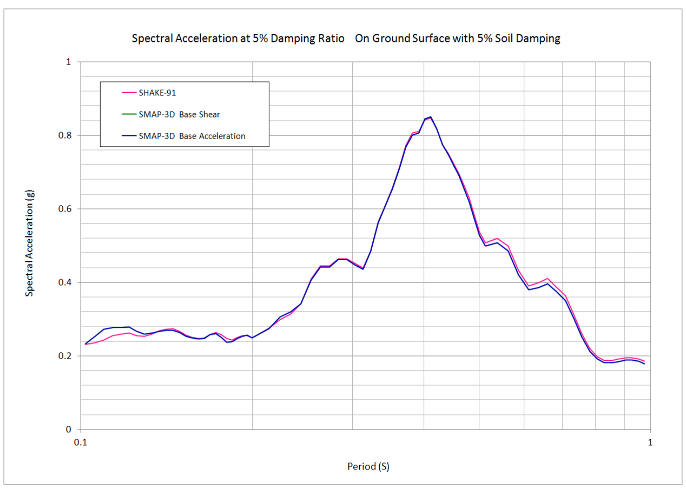


Figure 4.155 Spectral accelerations between 0.1 and 1 sec.

4.33 Silo Lining Analysis

This example is to solve the lining stresses developed in underground silo subjected to residual water pressure. This silo structure in Gyeongju, South Korea, was constructed to store the low-and-intermediate-level radioactive waste.

Figures 4.156 and 4.157 show finite element meshes and close-up view around silo, respectively. This 3 dimensional model consists of 65,598 continuum, 792 joint, 1,584 shell elements and 71,867 nodes. Program used thin shell elements to model reinforced concrete lining.

Table 4.8 lists material properties and Figure 4.158 shows schematic view of detailed silo lining structure. Table 4.9 lists lining thickness and reinforcement. Figure 4.159 shows silo lining material numbers. Table 4.10 shows schematically the sequence of silo construction including residual water pressure applied at step 5. Figure 4.160 shows key locations along the silo lining.

The following is a partial listing of graphical outputs at load step 5 when lining is subjected to residual water pressure head of 17.47m:

Figure 4.161	Deformed shape of silo lining
Figure 4.162	Dome deflection along A-B
Figure 4.163	Storage wall radial displacement along C-D
Figure 4.164	Dome lining inner hoop stress along A-B
Figure 4.165	Dome outer rebar meridian stress along A-B
Figure 4.166	Storage wall lining inner hoop stress along C-D
Figure 4.167	Storage wall outer rebar meridian stress along C-D

SMAP-3D results are compared with SMAP-2D results to verify the validity of the solution. As shown, SMAP-3D results are very close to SMAP-2D results. It seems that the reinforced concrete lining is in safe condition under the applied residual water pressure head of 17.47m.

Note: It takes about 5 hours of run time in the following computer:
64 Bit Windows 11, 8 Core i7-11700F CPU, 16 GB of DDR4 Ram.

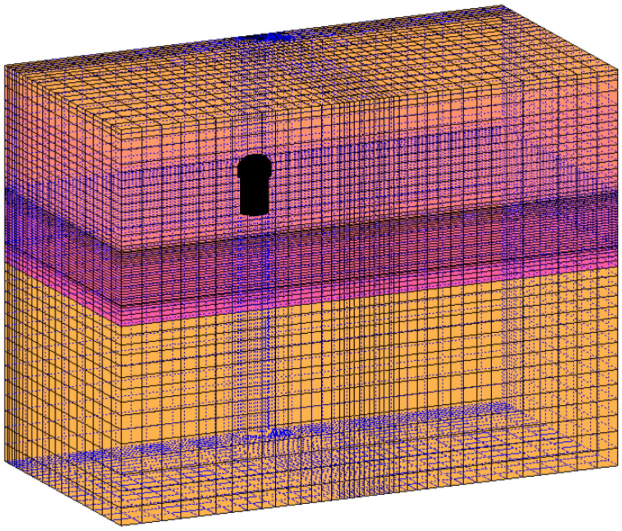


Figure 4.156 Finite element meshes

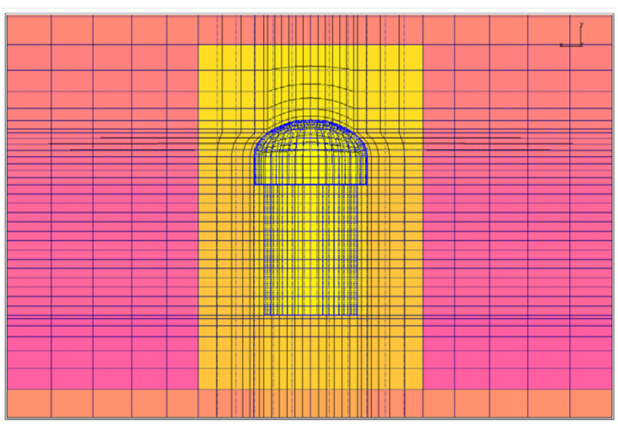


Figure 4.157 Finite element meshes around silo

Table 4.8 Material properties

Ground Layer	Unit weight (KN/m^3)	Young's modulus (MPa)	Poisson's ratio	Internal Friction Angle
Soil Layer	18.56	0.124×10^4	0.33	30°
Weathering Rock	20.52	0.342×10^4	0.30	38°
Rock	26.28	8.260×10^4	0.27	43°
Shotcrete	23.0	24,500	0.167	-
Concrete	23.5	29,500	0.167	-
Rebar	-	210,000	0.25	-

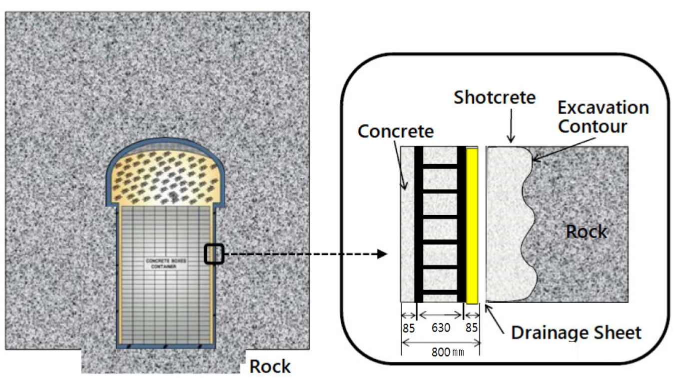


Figure 4.158 Schematic view of detailed silo lining structure

Table 4.9 Silo lining thickness and reinforcement

Material Number	Thickness (Meter)	Steel Ratio (%)		Location
		Hoop	Meridian	
1	1.211	0.85	0.85	Dome Crown
4	1.246	0.83	0.83	Dome Crown
5	1.279	0.81	0.81	Dome Crown
6	1.328	0.78	0.78	Dome Crown
7	1.398	0.74	0.74	Dome Crown
8	1.475	0.70	0.70	Dome Crown
9	1.547	0.67	0.67	Dome Crown
10	1.594	0.65	0.65	Dome Crown
11	1.600	0.65	0.65	Dome Wall
12	1.200	0.86	0.86	Dome Bottom
13	0.800	1.29	1.29	Storage Wall
14	1.200	0.86	0.86	Storage Bottom
15	1.200	0.86	0.86	Storage Bottom

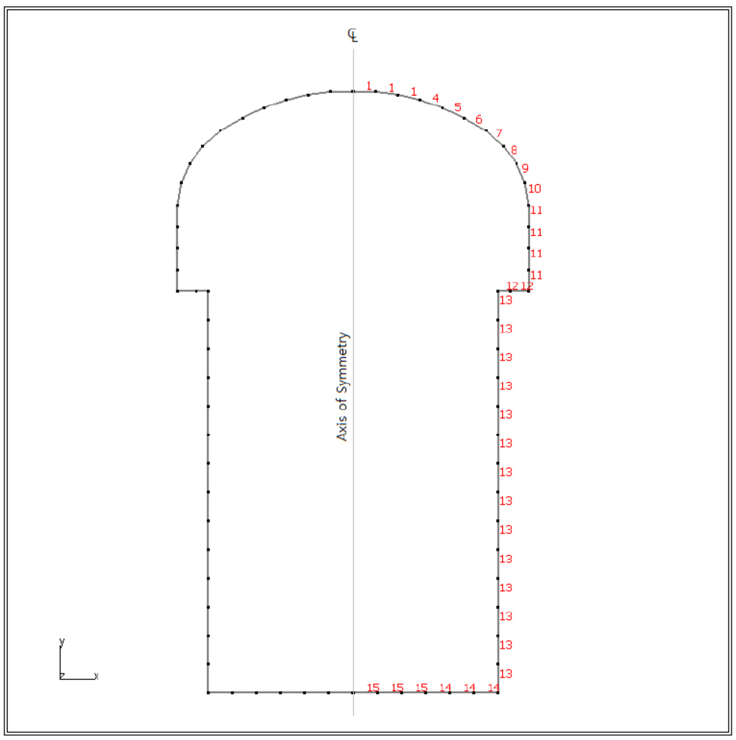
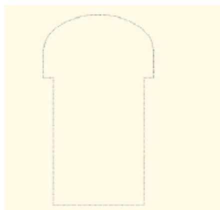
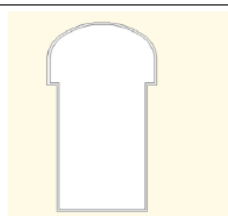
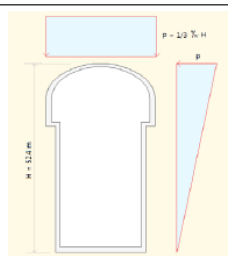


Figure 4.159 Silo lining material numbers

Table 4.10 Construction sequence

Step	Construction State	Descriptions
1,2		In Situ Ko State
3		Excavate Silo and install Shotcrete of 50cm Thickness
4		Install Reinforced Concrete Lining with its Own Self Weight
5		Lining is Subjected to Residual Water Pressure Head of 17.47m

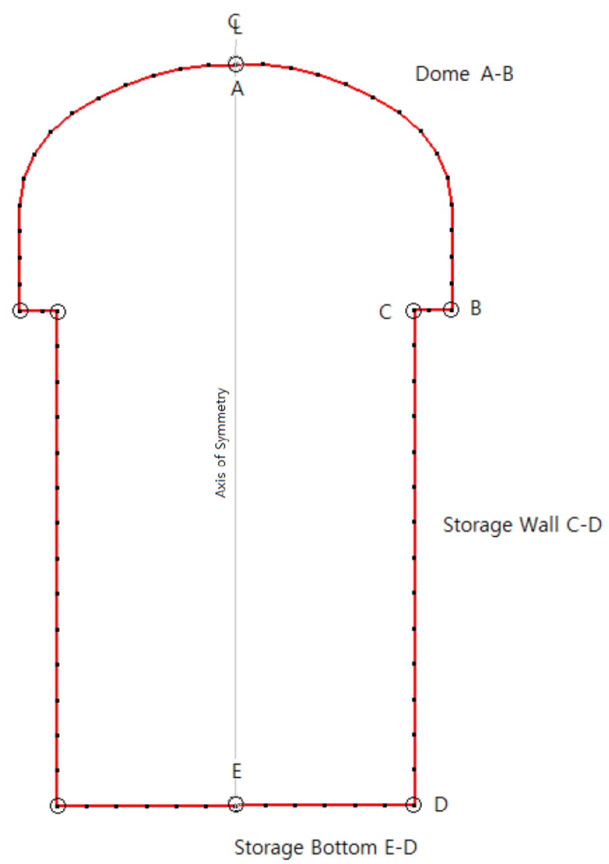


Figure 4.160 Key locations along silo lining

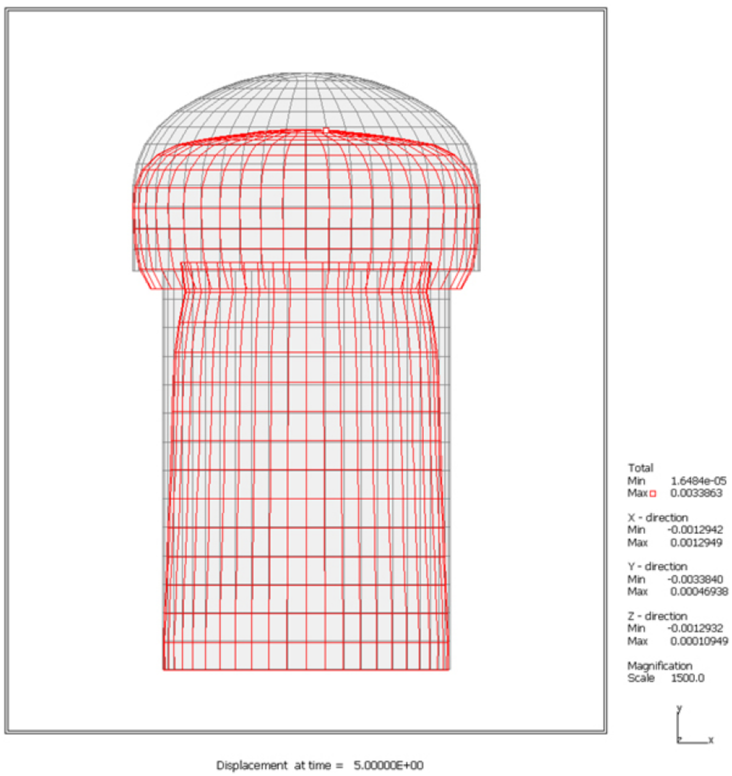


Figure 4.161 Deformed shape of silo lining

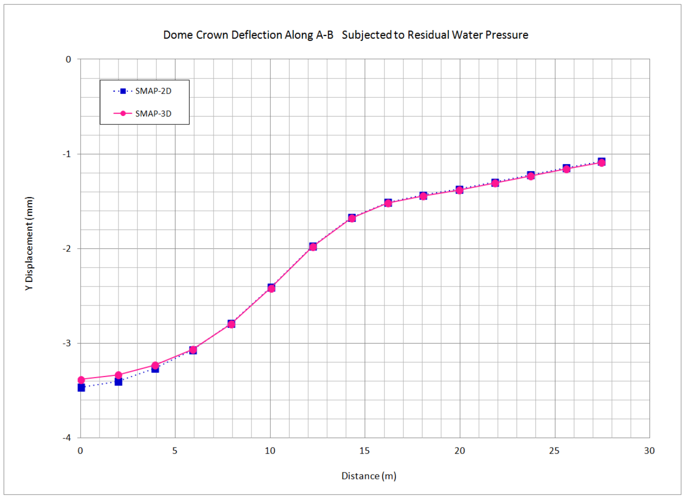


Figure 4.162 Dome deflection along A-B

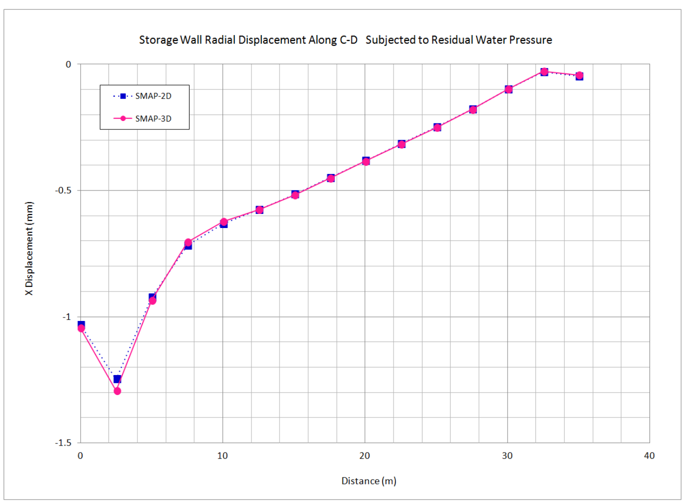


Figure 4.163 Storage wall radial displacement along C-D

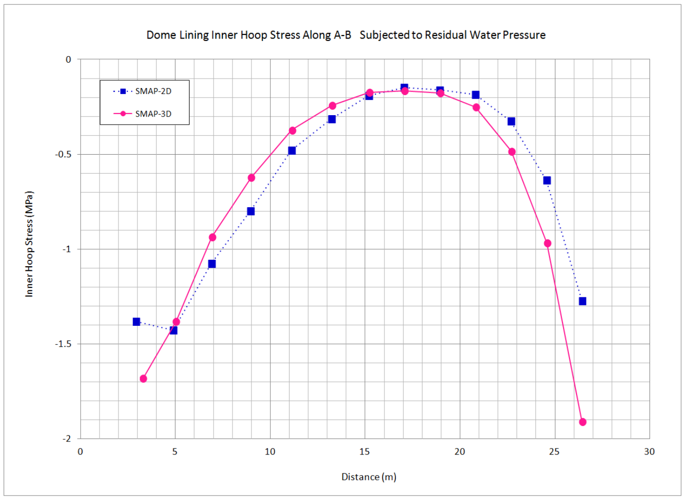


Figure 4.164 Dome lining inner hoop stress along A-B

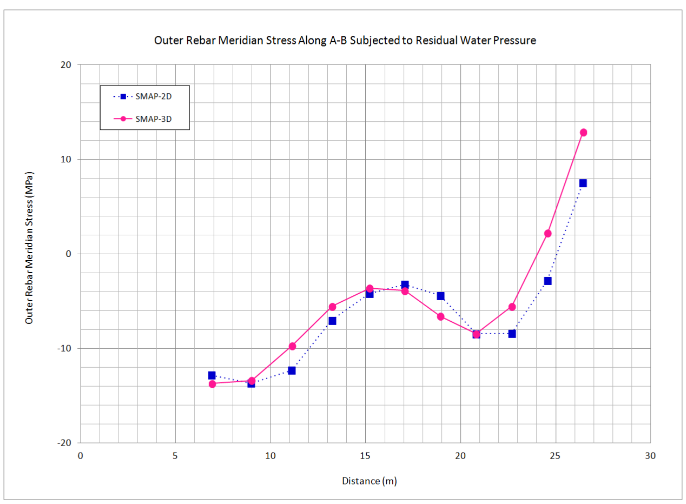


Figure 4.165 Dome outer rebar meridian stress along A-B

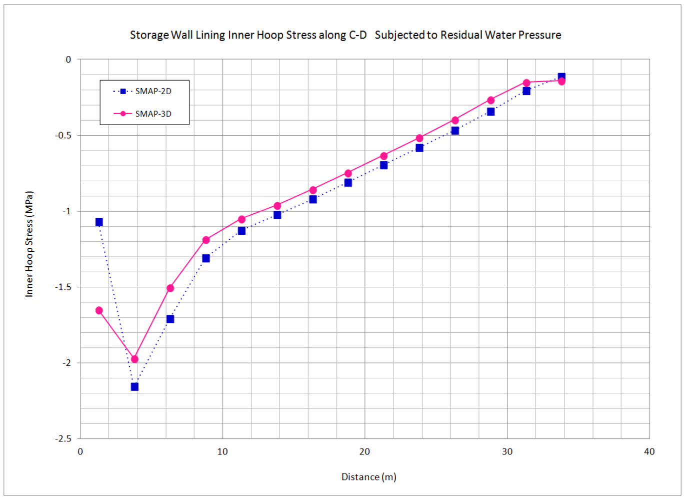


Figure 4.166 Storage wall lining inner hoop stress along C-D

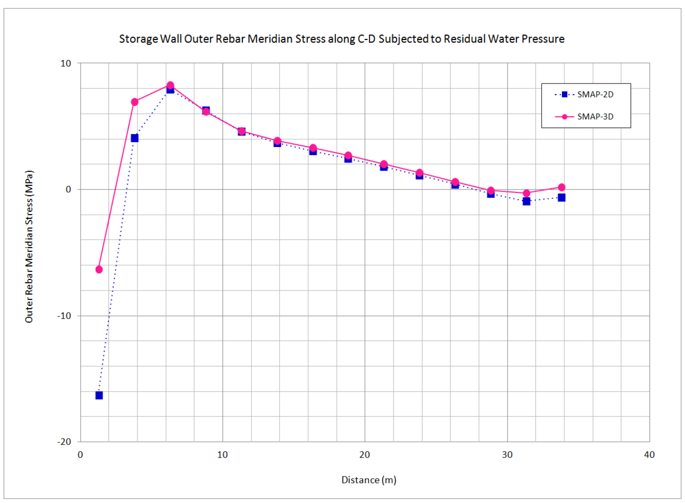


Figure 4.167 Storage wall outer rebar meridian stress along C-D

4.34 Liquefaction Analysis with PM4Sand

It should be noted that PM4Sand in SMAP-3D works only for plane strain condition. It does not work for general 3 dimensional condition.

The main objective of this example is to verify PM4Sand model implemented in SMAP-3D finite element program. The PM4Sand model (Boulanger and Ziotopoulou, 2017) is the effective stress material model which is calibrated in the finite difference program FLAC 8.0 (Itasca 2016) for the plane strain condition.

As first step, several different stress paths for a single element are considered to verify implementation; including drained and undrained conditions, monotonic and cyclic loadings, and isotropic and K_O initial conditions. Figure 4.168 shows isotropic consolidated drained cyclic direct simple shear test. All other results are summarized in the file; Single Element Stress-Strain Response of PM4Sand Model.pdf

This analysis is to solve the free-field seismic response of the soil profile, shown in Figure 4.169 along with material properties, subjected to earthquake excitation from the bedrock.

This problem is the same as the problem in the report (Chen and Arduino, 2021). A 6 m soil profile is subjected to Loma Prieta earthquake in 1989 (RSN766) as outcrop to the elastic half space. Earthquake time history with peak acceleration 0.37g and its spectral acceleration are shown in Figures 4.170 and 4.171, respectively.

Figures 4.172 and 4.173 show computed profiles of peak ground accelerations and maximum shear strains, respectively, compared with SHAKE 91 and DEEP SOIL. Note that this linear elastic analysis is performed to check the initial stresses and boundary conditions prior to liquefaction analysis by scaling down peak acceleration to 0.02g.

Results of liquefaction analysis are presented in the following:

Figure 4.174	Maximum acceleration profile (PGA)
Figure 4.175	Maximum displacement profile
Figure 4.176	Maximum shear strain profile
Figure 4.177	Maximum r_u profile
	r_u = Excess Pore Pressure / Initial Effective Ver. Stress

Overall, PM4Sand in SMAP-3D is performing very well in predicting the stress-strain responses compared to the calibrated FLAC results.

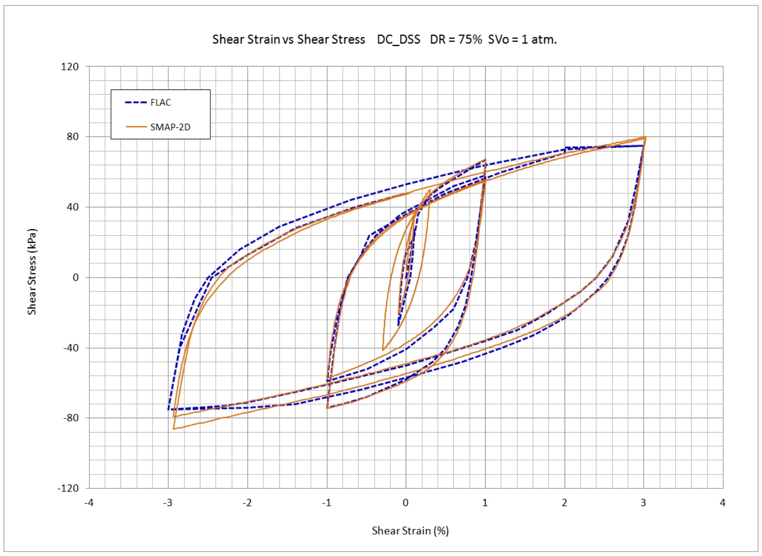
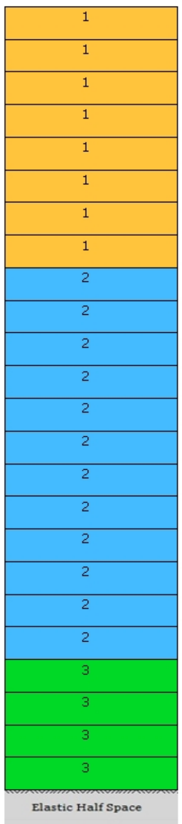


Figure 4.168 Isotropic consolidated drained cyclic direct simple shear test



No	γ (kN/m³)	G (MPa)	β (%)
1	15.8	Dry n = 39.66% PM4Sand Dr = 0.466 Go = 468.3 hpo = 0.463	3.55
2	19.7	Saturated n = 39.59% PM4Sand Dr = 0.466 Go = 584.1 hpo = 0.45	3.55
3	19.7	Saturated n = 39.59% Elastic G = 66.54	3.55
	19.7	Elastic G = 66.54	

Total Dimension

Y - direction
Min -6.0000
Max 0

Figure 4.169 Finite element meshes and material properties

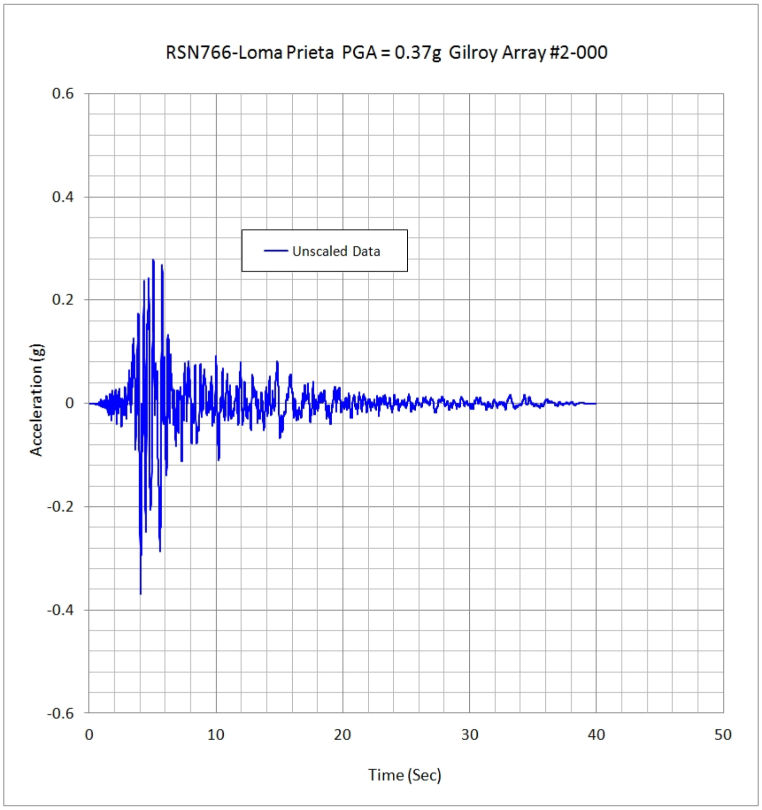


Figure 4.170 Loma Prieta (RSN766) acceleration time history

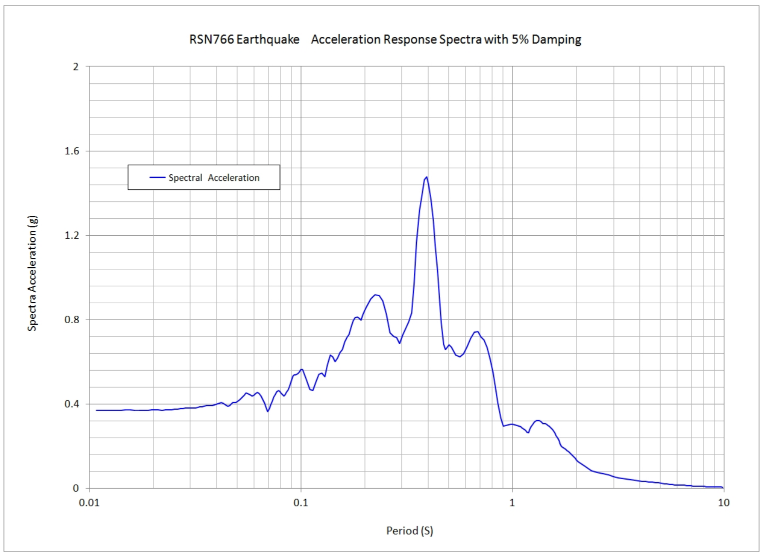


Figure 4.171 Spectral acceleration for input earthquake (RSN766)

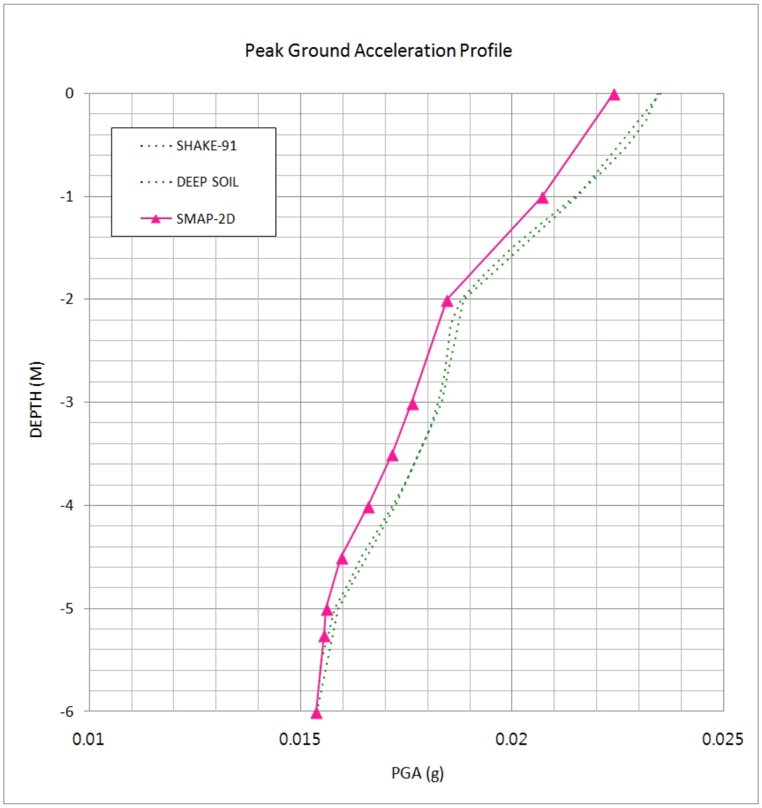


Figure 4.172 Peak ground acceleration profile, Elastic analysis

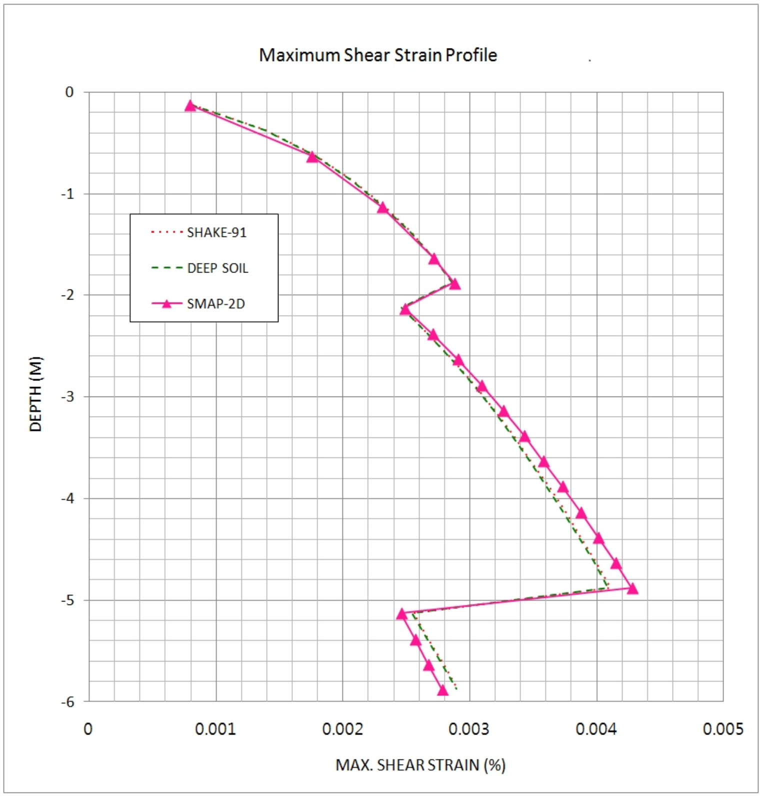


Figure 4.173 Maximum shear strain profile, Elastic analysis

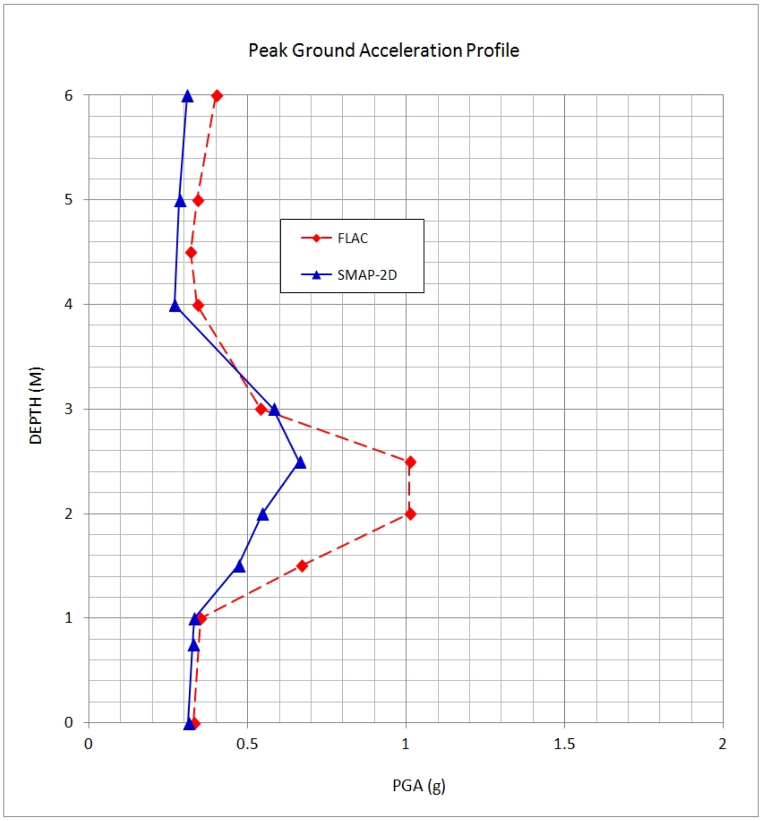


Figure 4.174 Maximum acceleration profile, Liquefaction analysis

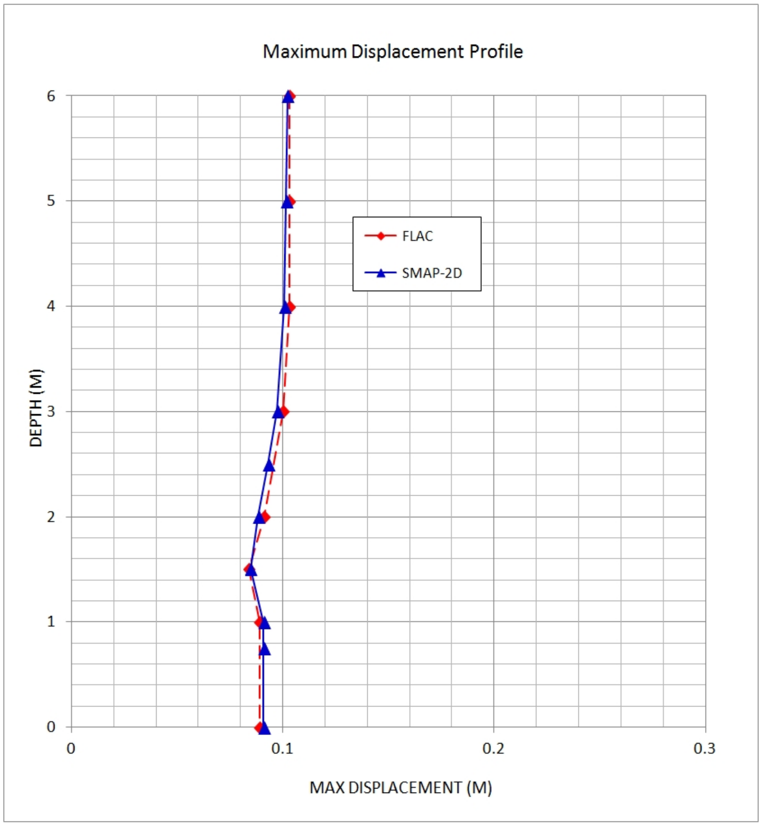


Figure 4.175 Maximum displacement profile, Liquefaction analysis

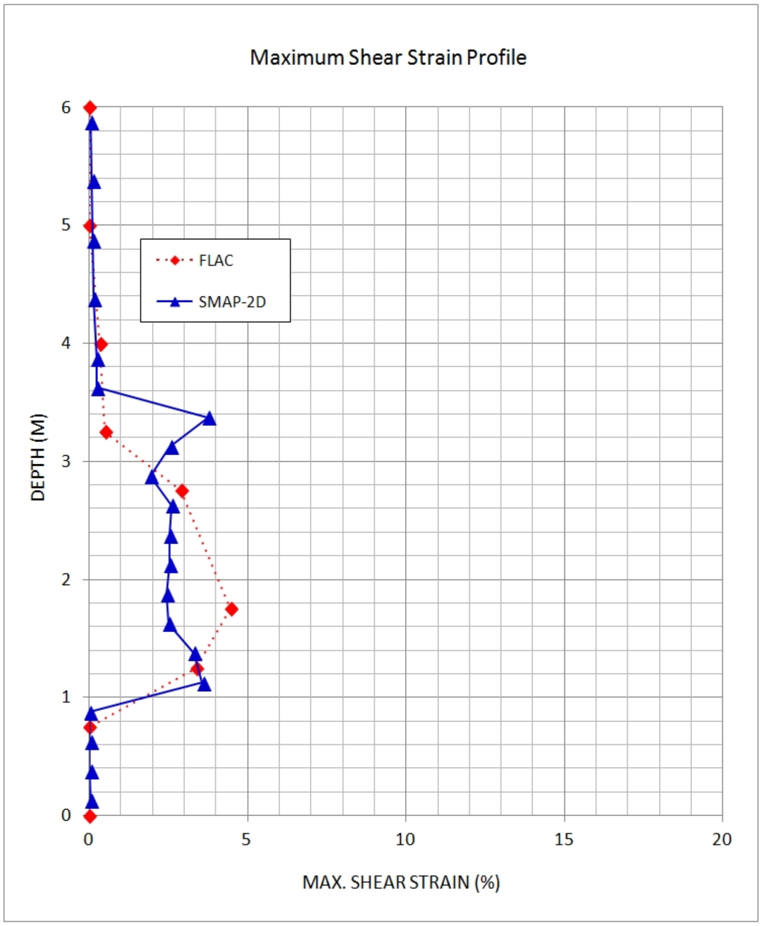


Figure 4.176 Maximum shear strain profile. Liquefaction analysis

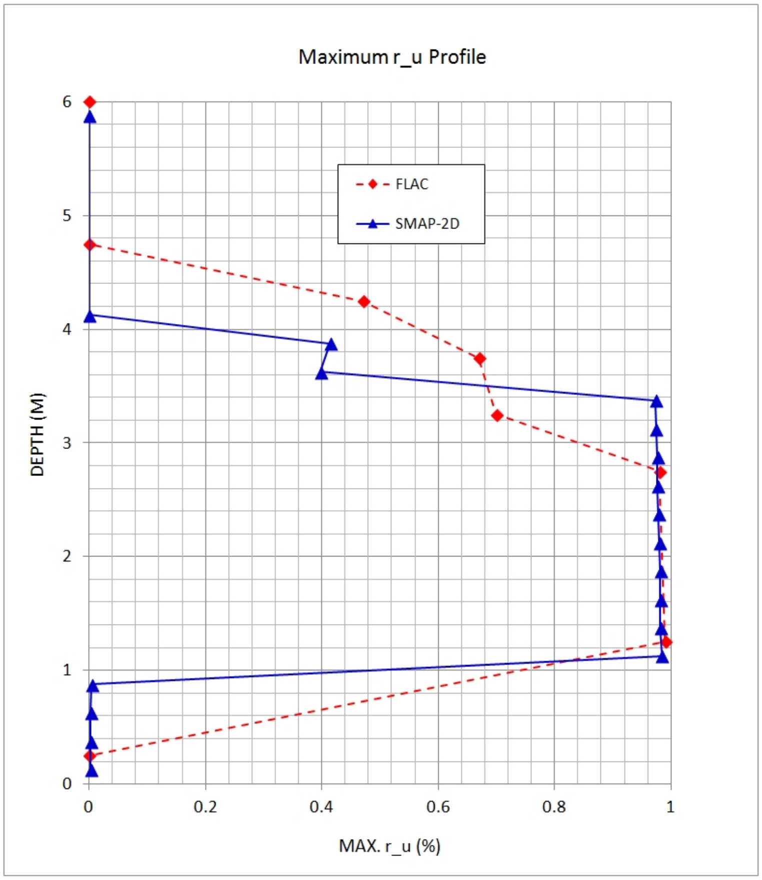


Figure 4.177 Maximum r_u profile, Liquefaction analysis

Group Mesh
Example Problem

Group Mesh Generator is a two-dimensional CAD program specially designed to build group mesh which can be used to generate finite element mesh with the aid of program ADDRGN-2D. Group Mesh User's Manual describes all the basic functions associated with group mesh generation and modifications.

Six example problems are presented:

1. Arch Tunnel
   Shows step by step procedure to create and modify group meshes.
2. NATM Tunnel
   Builds group mesh for typical NATM tunnel.
3. Excavation
   Builds group mesh for typical multi-step excavations performed near the existing structure.
4. Buried Pipe
   Builds group mesh for typical pipe buried in the trench followed by multi-step embankment lifts.
5. Arch Warehouse
   Builds group mesh for typical arch warehouse structure.
6. Finite Element Mesh Modification
   Illustrates how to modify existing finite element meshes using Mesh Generator.

5.1 Arch Tunnel

The main objective of this first example is to show the step by step procedure to create and modify group meshes.

This example has the following three parts:

Part 1 : Creating Arch Tunnel (Figure 5.1)

- Create group mesh
- Set built-in base mesh
- Draw arch tunnel
- Plot finite element mesh

Part 2 : Adding Rock Bolts (Figure 5.2)

- Open the group mesh file in part 1
- Add three rock bolts
- Plot finite element mesh

Part 3 : Adding Utility Tunnel (Figure 5.3)

- Open the group mesh file in part 2
- Remove the first rock bolt
- Change the second rock bolt length
- Replace the third rock bolt by utility tunnel
- Plot finite element mesh

Table 5.1 shows the construction sequence.

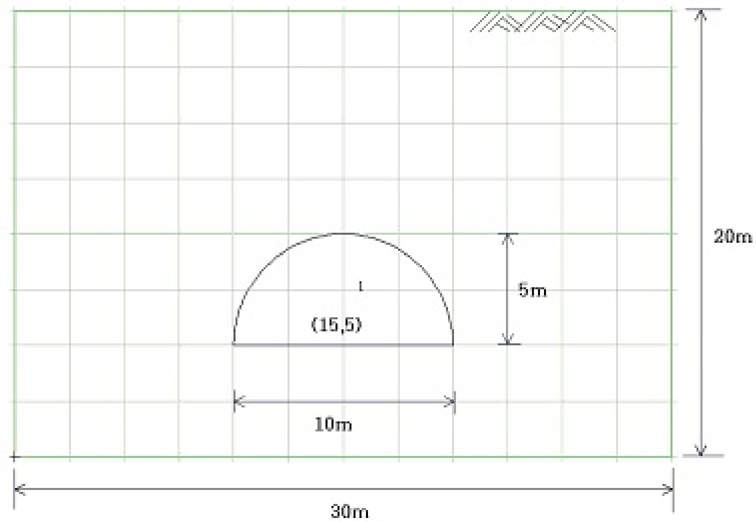


Figure 5.1 Arch tunnel (Part 1)

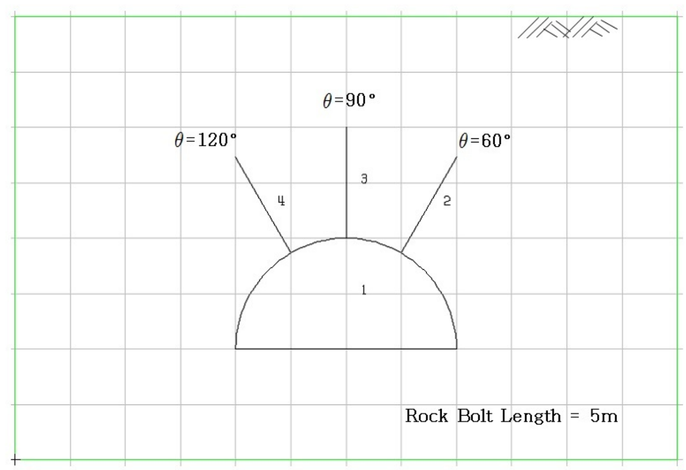


Figure 5.2 Arch tunnel with rock bolts (Part 2)

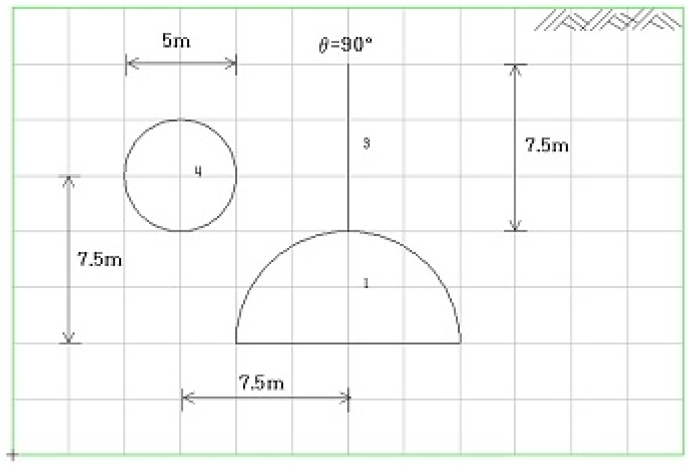


Figure 5.3 Arch tunnel with utility tunnel (Part 3)

Step No	Description
1, 2	In-Situ Stress
3	Arch Tunnel Excavation & Lining Installation
4	Rock Bolt Installation
5	Utility Tunnel Construction

Table 5.1 Construction sequence

5.1.1 Part 1: Creating Arch Tunnel

Part 1 consists of the following main actions:

- Create group mesh
- Set built-in base mesh
- Draw arch tunnel
- Plot finite element mesh

Step 1: Group Mesh Generator (New)

Access Group Mesh Generator by selecting the following menu items in SMAP (Figure 5.4):
Run → Mesh Generator → Group Mesh → New

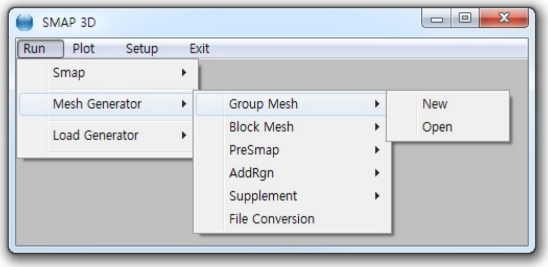


Figure 5.4 Accessing group mesh generator (New)

Step 2: Group Input (New)

Select Built-in Base Mesh in Figure 5.5.
Click OK.

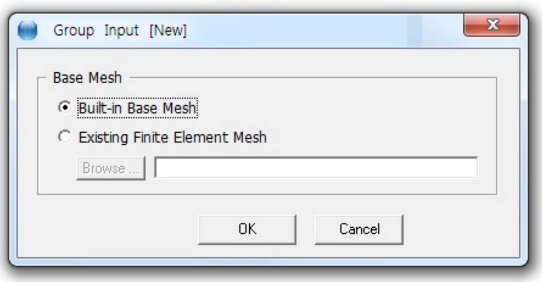


Figure 5.5 Group input (New)

Step 3: Group Menu and Dialog

Click Group menu in PLOT-2D as shown in Figure 5.6.

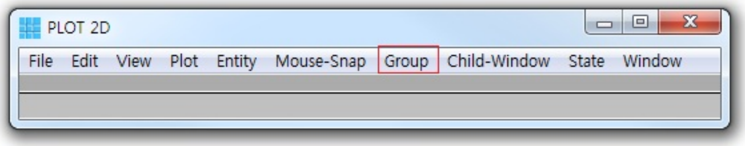


Figure 5.6 Group menu

Group dialog in Figure 5.7 is displayed with initial default values.

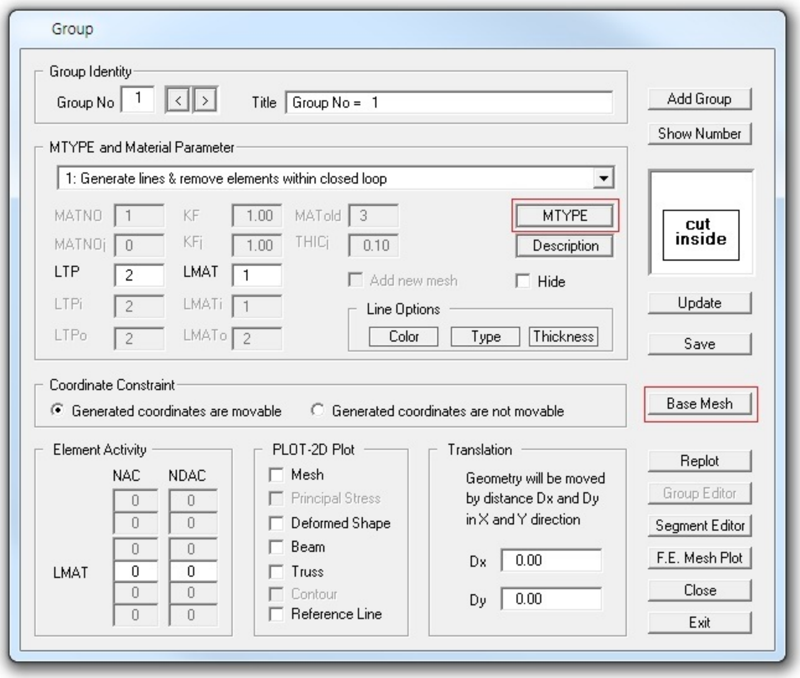


Figure 5.7 Group dialog with initial default values

Step 4: Built-in Base Mesh

Click Base Mesh button in Group dialog.

Fill in input fields for Built-in Base Mesh as shown in Figure 5.8.

Click OK.

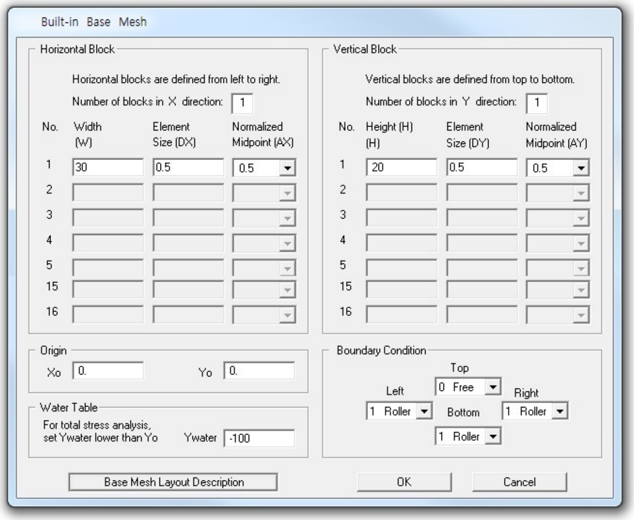


Figure 5.8 Built-in base mesh dialog

Figure 5.9 shows Base Mesh with dimensions of 30m x 20m on drawing board in PLOT-2D.

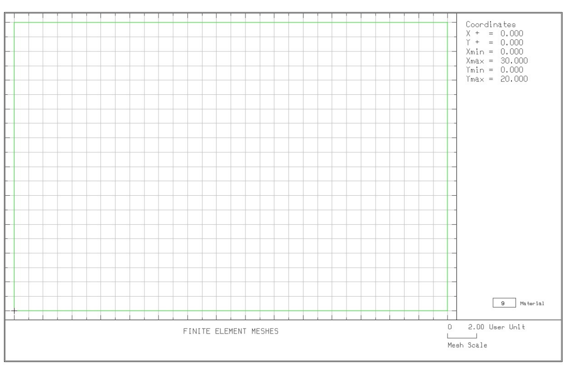


Figure 5.9 Base mesh on drawing board

Step 5: MTYPE

Click MTYPE button in Group dialog.
Select MTYPE=3 in MTYPE dialog in Figure 5.10.
Click OK.

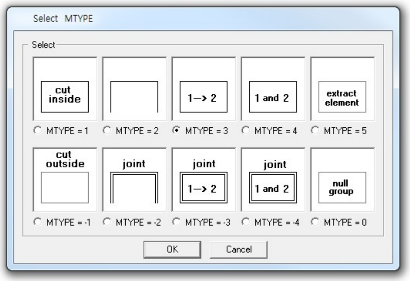


Figure 5.10 MTYPE dialog

Fill in input fields for Group dialog as shown in Figure 5.11.

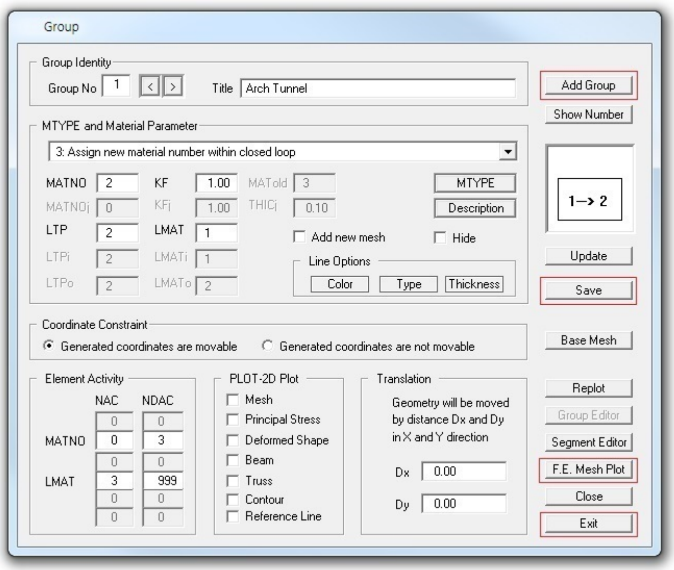


Figure 5.11 Group dialog with MTYPE = 3

Step 6: Mouse Snap

Click Mouse-Snap menu in PLOT-2D.
Select Snap to Grid in Figure 5.12. Click OK.

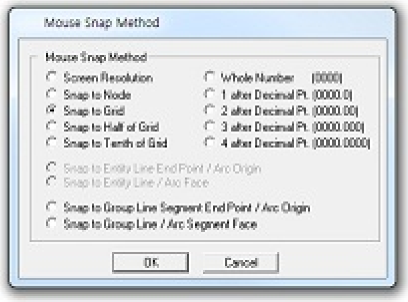


Figure 5.12
Mouse snap dialog

Step 7: Add Group

Click Add Group button in Group dialog.

Table 5.2 summarizes group parameters used for arch tunnel.

Group No	MTYPE	Description	Element Type	Mat. Np.	Element Activity	
					NAC	NDAC
1	3	Core	Cont.	MATNO=2	0	3
		Lining	Beam (LPT=2)	LMAT=1	3	999

Group No	Seg. No	Line Segment				Arc Segment						IEND
		Beginning Point		Ending Point		Origin		Radius and Angle				
		X	Y	X	Y	X_o	Y_o	R_X	R_Y	Θ_b	Θ_e	
1	1	10	5	20	5							2
	2					15	5	5	5	0	180	2

Table 5.2 Group parameters for arch tunnel

Step 8: Line Segment

Click Draw button in Line Segment dialog in Figure 5.13.

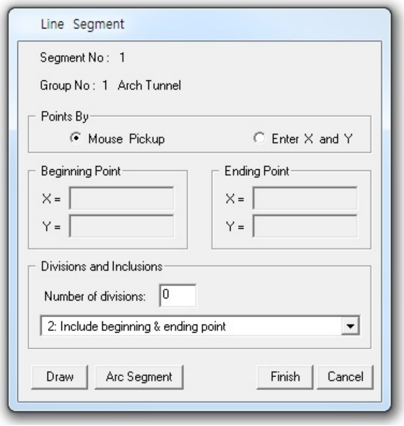


Figure 5.13 Line segment dialog

Click the mouse where the line begins and then click the mouse where the line ends as shown in Figure 5.14.

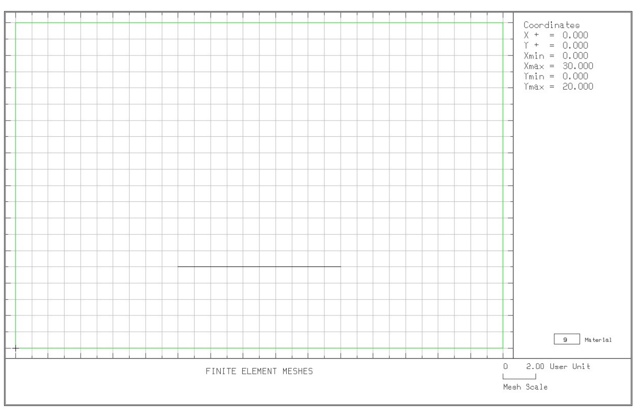


Figure 5.14 Line segment on drawing board

Step 9: Arc Segment

Click Arc Segment button in Line Segment dialog.
Fill in input fields for Arc Segment as shown in Figure 5.15.
Click Draw.

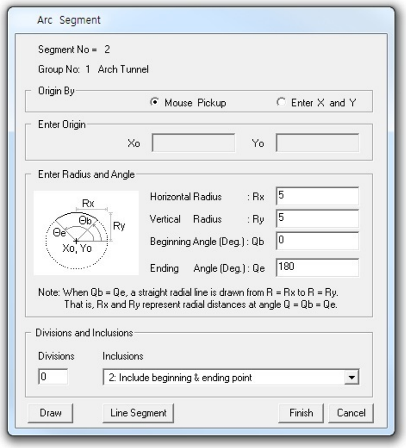


Figure 5.15 Arc segment dialog

Press down and hold mouse button on the drawing board.
Drag the mouse to the location of arc origin and then release the mouse button as shown in Figure 5.16.
Click Finish.

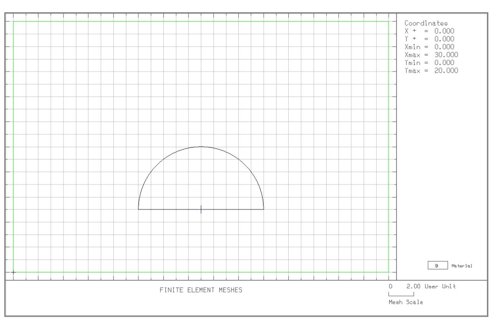


Figure 5.16 Arc segment on drawing board

Step 10: Save

Click Save button in Group dialog.
Group.Meg is saved as shown in Figure 5.17.

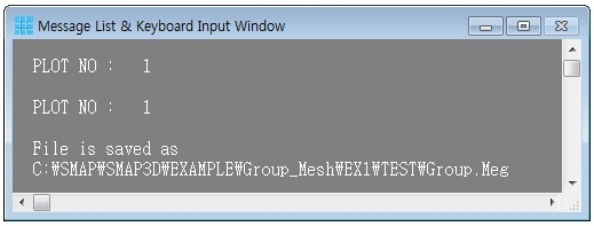


Figure 5.17 Message for file save

Step 11: Finite Element Mesh

Click F.E. Mesh Plot button in Group dialog.
Click Yes in Figure 5.18.

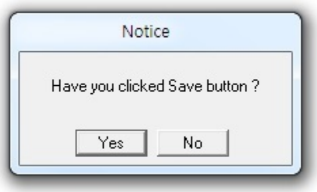


Figure 5.18 Notice for finite element mesh plot

Please Wait... message in Figure 5.19 is shown on the screen while generating finite element mesh plot.

Figure 5.19 Notice while generating finite element mesh plot

Once finished, finite element mesh file is generated as Group.Mes in the directory Plot_Mesh as shown in Figure 5.20 along with finite element mesh plot in Figure 5.21.

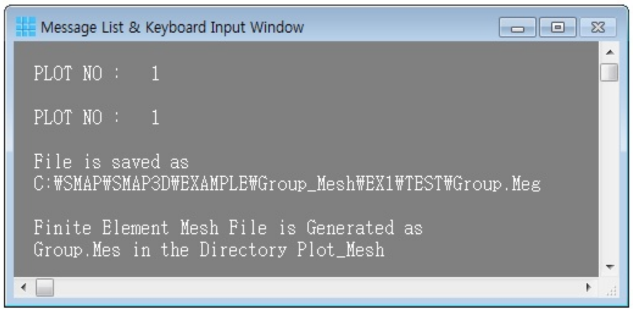


Figure 5.20 Message for finite element mesh file

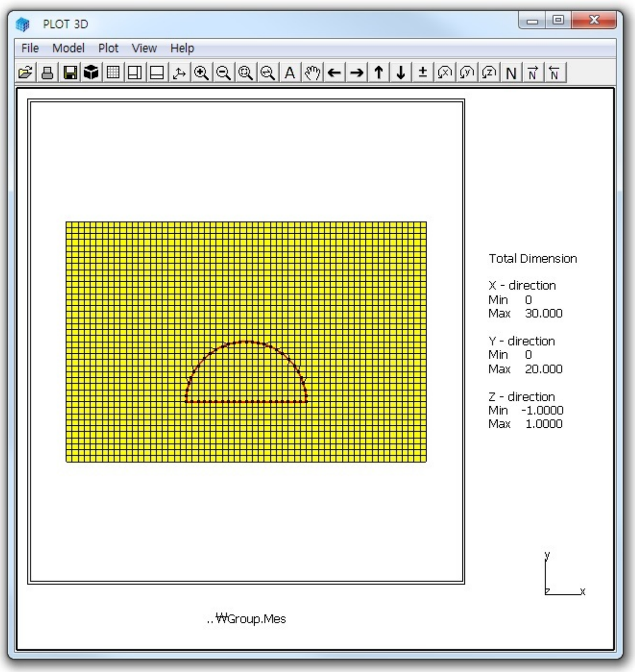


Figure 5.21 Finite element mesh plot

Step 12: Exit

Click Exit button in Group dialog.

Click OK in Exit dialog as shown in Figure 5.22.

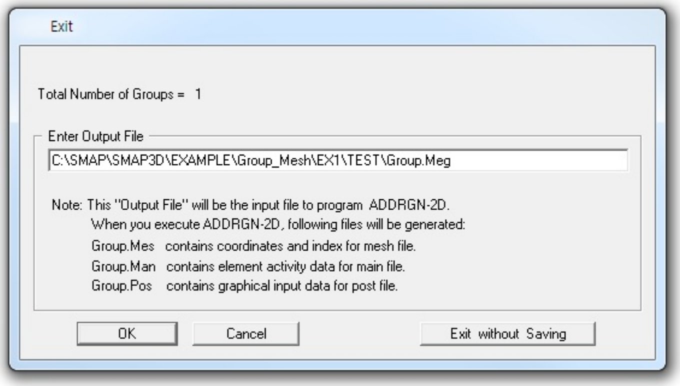


Figure 5.22 Exit dialog

5.1.2 Part 2: Adding Rock Bolts

Part 2 consists of the following main actions:

- Open the group mesh file in part 1
- Add three rock bolts
- Plot finite element mesh

Step 13: Group Mesh Generator (Open)

Access Group Mesh Generator by selecting the following menu items in SMAP (Figure 5.4):
Run → Mesh Generator → Group Mesh → Open

Step 14: Group Input (Open)

File open dialog will be displayed as in Figure 5.23.
Select group mesh file Group.Meg in Part 1 and click Open.

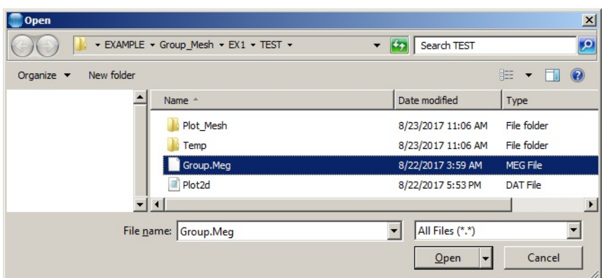


Figure 5.23 File open dialog

Step 15: Group Menu and Dialog

Click Group menu in PLOT-2D as shown in Figure 5.6.
Group dialog for Group No 2 is displayed with initial default values.

Step 16: MTYPE

Click MTYPE button in Group dialog.
Select MTYPE=2 in MTYPE dialog in Figure 5.10.
Click OK.

Step 17: Group No 2 for Rock Bolt 1

Table 5.3 summarizes group parameters for rock bolts.
Rock bolt is modeled by a straight radial line in Arc Segment.

Group No	Bolt No	MTYPE	Elem. Type (LTP)	Mat. No (LMAT)	Element Activity		Radius and Angle				IEND
					NAC	NDAC	R_X	R_Y	Θ_b	Θ_e	
2	Bolt-1	2	Truss (3)	1	4	999	5	10	60	60	-2
3	Bolt-2	2	Truss (3)	1	4	999	5	10	90	90	-2
4	Bolt-3	2	Truss (3)	1	4	999	5	10	120	120	-2

Table 5.3 Group parameters for rock bolts

Group No 2 represents Rock Bolt 1 with a length of 5m at 60 degrees.
Fill in input fields for Group dialog as shown in Figure 5.24.

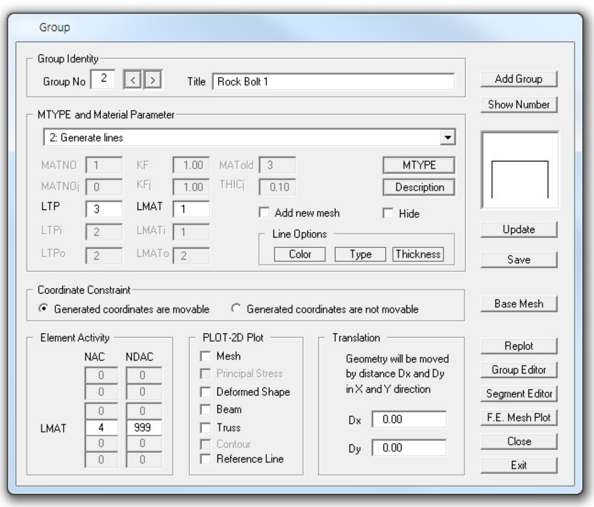


Figure 5.24 Group dialog for Rock Bolt 1

Step 18: Mouse Snap

Click Mouse-Snap menu in PLOT-2D.
Select Snap to Grid in Figure 5.12.
Click OK.

Step 19: Add Group

Click Add Group button in Group dialog.

Step 20: Arc Segment

Click Arc Segment button in Line Segment dialog.
Fill in input fields for Arc Segment as shown in Figure 5.25.
Click Draw.

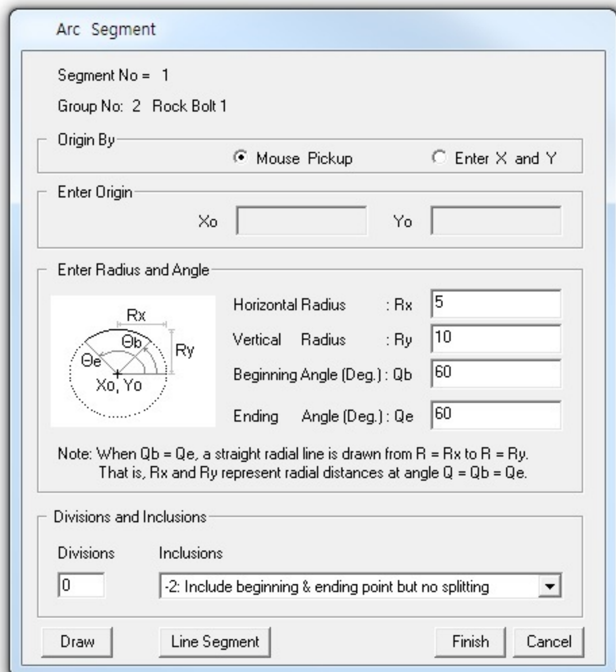


Figure 5.25 Arc segment dialog for Rock Bolt 1

Press down and hold mouse button on the drawing board.
Drag the mouse to the location of arc origin and then release the mouse button as shown in Figure 5.26. Click Finish.

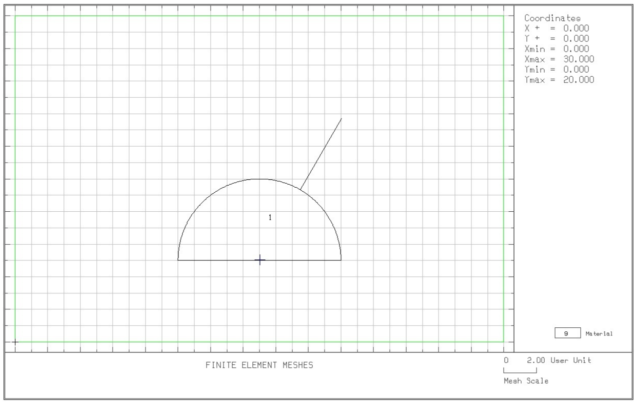


Figure 5.26 Rock Bolt 1 on drawing board

Step 21: Group No 3 & 4 for Rock Bolt 2 & 3

Repeat Steps 16 through 20 to add rock bolts at 90 and 120 degrees.
All three rock bolts are shown on drawing board in Figure 5.27.
Click Save button in Group dialog.

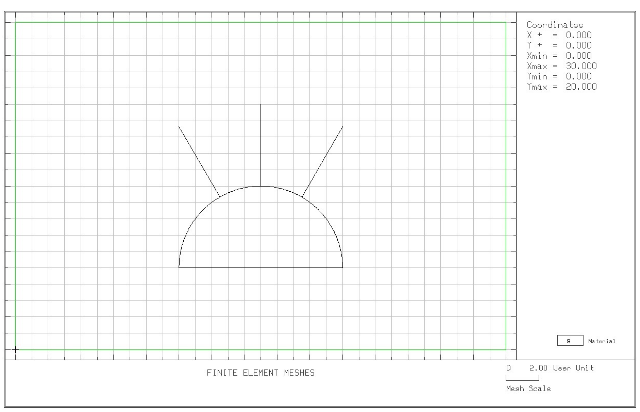


Figure 5.27 All three rock bolts on drawing board

Step 22: Finite Element Mesh

Click F.E. Mesh Plot button in Group dialog.
Follow the same procedure as in Steps 10 and 11.
Finite element meshes are shown in Figure 5.28

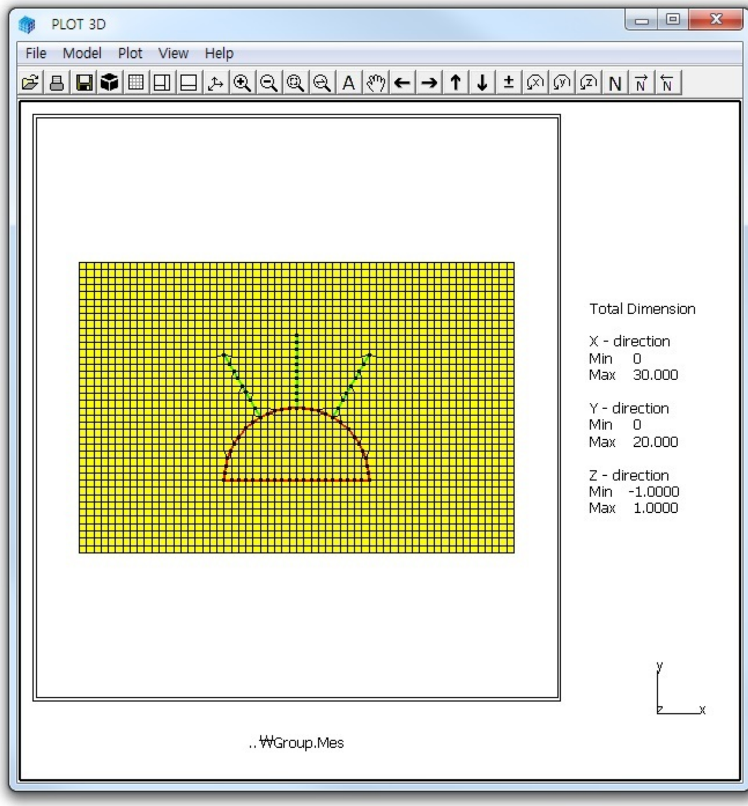


Figure 5.28 Finite element mesh plot

Step 23: Exit

Click Exit button in Group dialog.
Click OK in Exit dialog as in Figure 5.22.

5.1.3 Part 3: Adding Utility Tunnel

Part 3 consists of the following main actions:

- Open the group mesh file in part 2
- Remove the first rock bolt
- Change the second rock bolt length
- Replace the third rock bolt by utility tunnel
- Plot finite element mesh

Step 24: Open Group Mesh File in Part 2

Follow Steps 13 through 15 to open Group dialog for Group No 2.

Step 25: Remove Rock Bolt 1

Select Group No 2 in Group dialog.
Click MTYPE button in Group dialog.
Select MTYPE=0 in MTYPE dialog in Figure 5.10.
Click OK.

Click Update and then Replot buttons in Group dialog.
A new plot with the Group No 2 missing is displayed in Figure 5.29

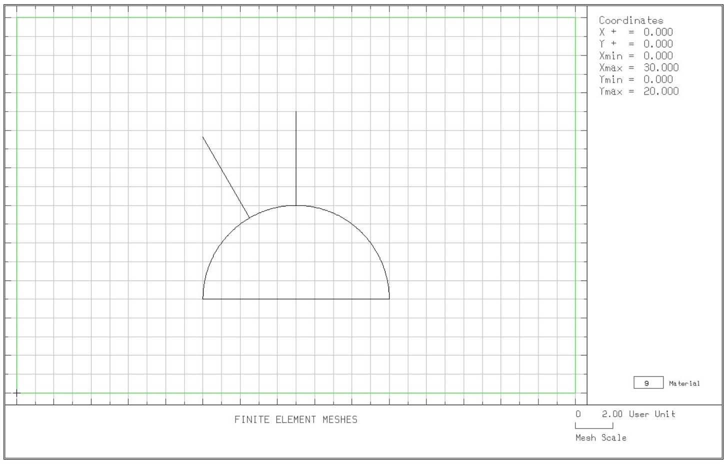


Figure 5.29 Rock Bolt 1 removed on drawing board

Step 26: Change Length of Rock Bolt 2

Select Group No 3 in Group dialog.
Click Edit Group button in Group dialog.

Click Edit button in Edit Segment dialog in Figure 5.30.

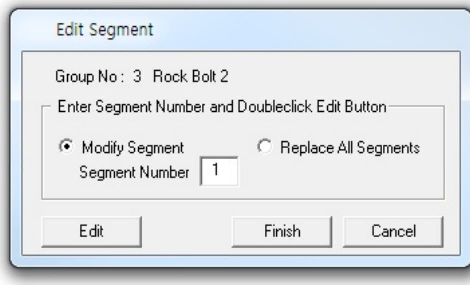


Figure 5.30
Edit segment dialog
for Group No 3

Fill in input fields for Arc Segment dialog as shown in Figure 5.31.
Click Draw and then Finish in Arc Segment dialog.
Click Finish in Edit Segment dialog.

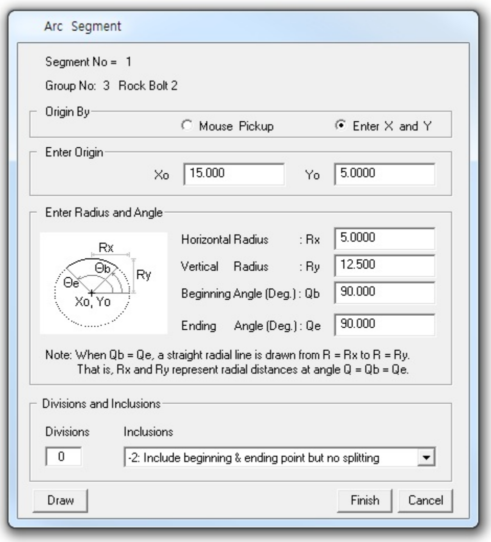


Figure 5.31
Arc segment dialog with
rock bolt length modified

Figure 5.32 shows a new plot with longer Rock Bolt 2.

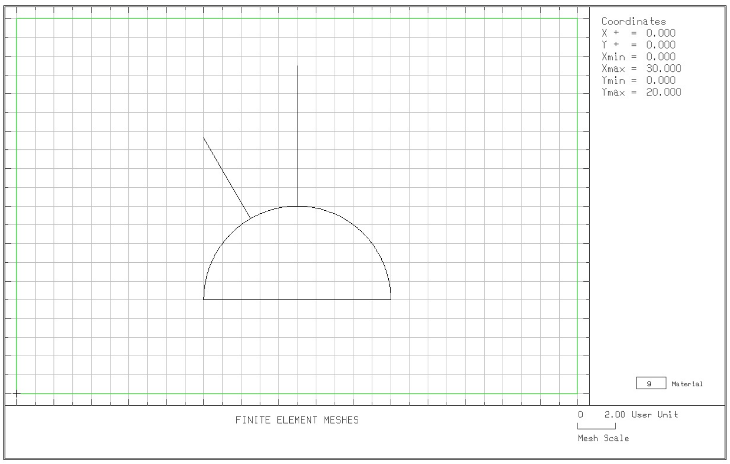


Figure 5.32 Longer Rock Bolt 2 on drawing board

Step 27: Replace Rock Bolt 3 by Utility Tunnel

Select Group No 4 in Group dialog.

Click MTYPE button in Group dialog.
Select MTYPE=1 in MTYPE dialog in Figure 5.10.
Click OK.

Fill in input fields for Group dialog as shown in Figure 5.33.
Click Edit Group.

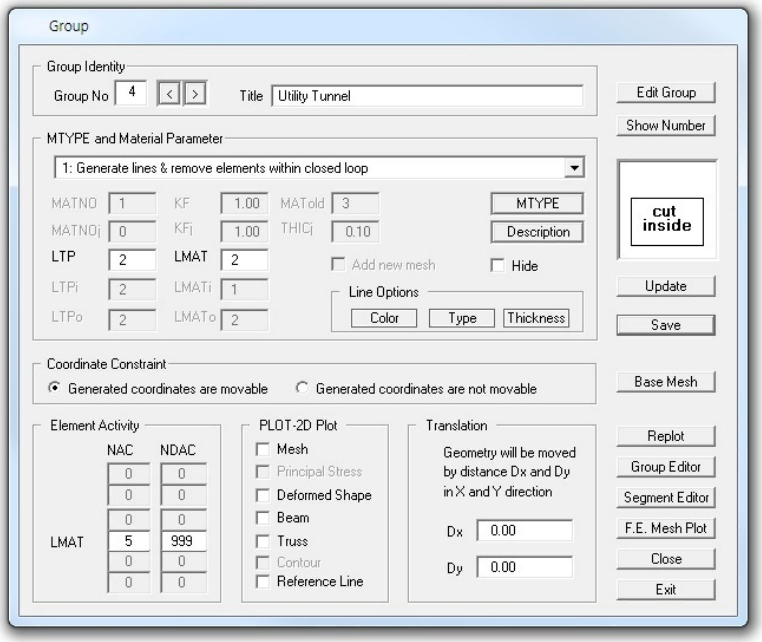


Figure 5.33 Group dialog for Utility Tunnel

Select Replace All Segments in Edit Segment dialog in Figure 5.34
Click Edit.

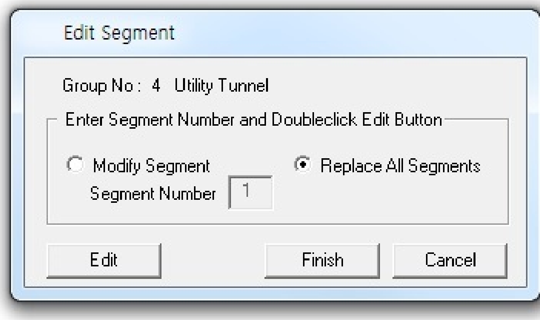


Figure 5.34 Edit segment dialog for Group No 4

Warning message is displayed as shown in Figure 5.35.
Click OK.

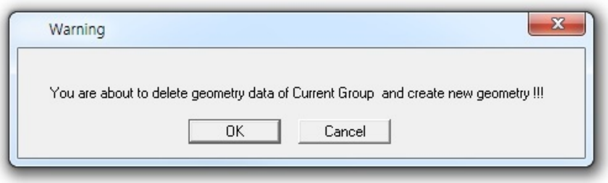


Figure 5.35 Warning message

Fill in input fields for Arc Segment dialog as shown in Figure 5.36.
Click Draw and then Finish in Arc Segment dialog.
Click Finish in Edit Segment dialog in Figure 5.34.

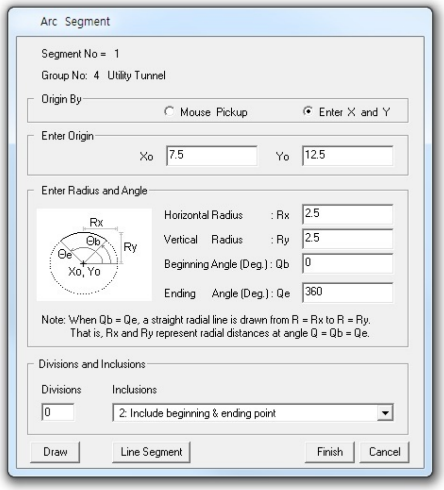


Figure 5.36 Arc segment dialog for Utility Tunnel

Click Update and then Replot buttons in Group dialog.
Figure 5.37 shows a new plot with Utility Tunnel on drawing board.

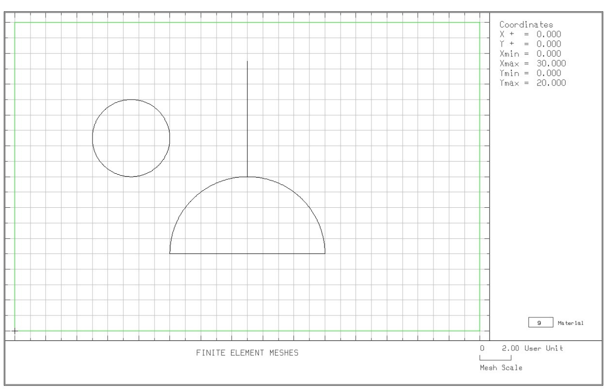


Figure 5.37 Arch and Utility Tunnels on drawing board

Step 28: Finite Element Mesh

Click Save and F.E. Mesh Plot button in Group dialog.
Follow the same procedure as in Steps 10 and 11.
Finite element meshes are shown in Figure 5.38

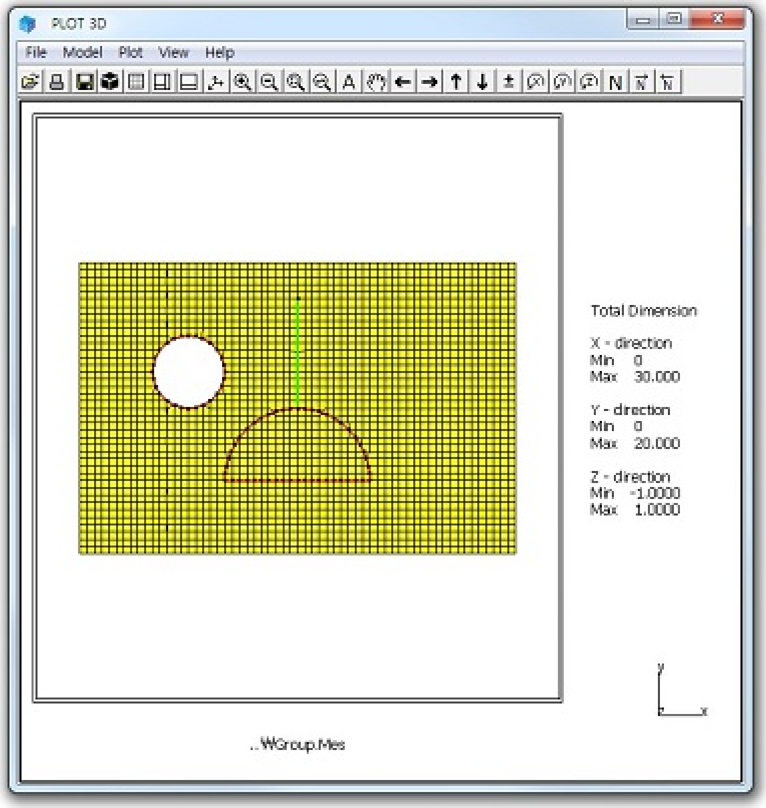


Figure 5.38 Finite element mesh plot

Step 29: Exit

Click Exit button in Group dialog.
Click OK in Exit dialog as in Figure 5.22.

5.2 NATM Tunnel

This example illustrates how to build group meshes for typical NATM (New Austrian Tunneling Method) tunnel.

5.2.1 Overview

The cross section of NATM tunnel consists of rock bolts, shotcrete, reinforced concrete liner, and core as schematically shown in Figure 5.39.

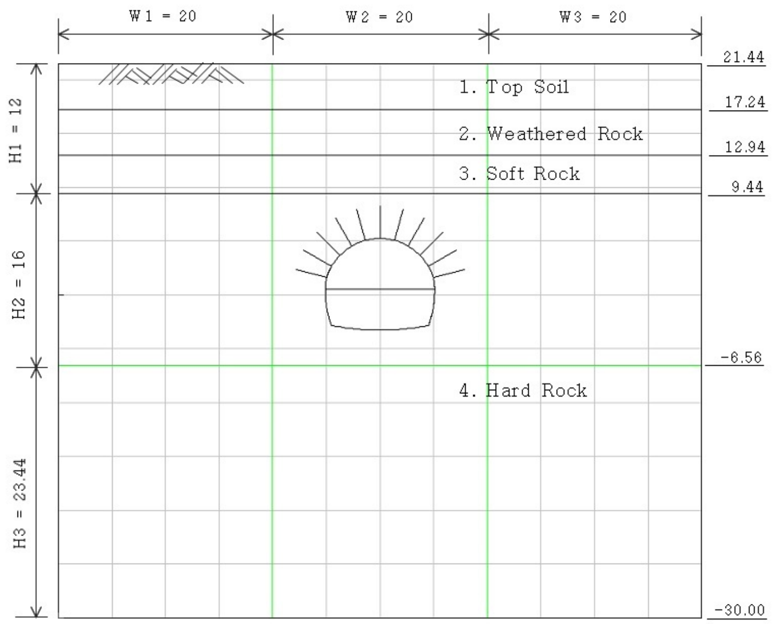


Figure 5.39 Tunnel cross section

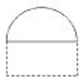
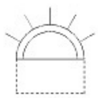
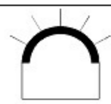
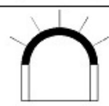
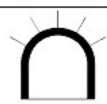
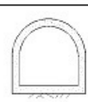
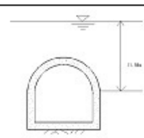
Step	Construction State	Description	
1, 2		In Situ Ko State	
3		50 % Stress Relief	Upper Core Excavation
4		75 % Stress Relief Soft Shotcrete, Rock Bolt	
5		100 % Stress Relief Hard Shotcrete, Rock Bolt	
6		50 % Stress Relief	Lower Core Excavation
7		75 % Stress Relief, Soft Shotcrete	
8		100 % Stress Relief, Hard Shotcrete	
9		Lining Subjected to : Weight	
12		Lining Subjected to : Weight + Water Pressure	

Table 5.4 Construction sequence

A total of 21 groups are used to model NATM tunnel as schematically shown in Figure 5.40: 4 for in situ geological profile, 11 for rock bolts, 1 for lining, 3 for shotcrete, and 2 for core.

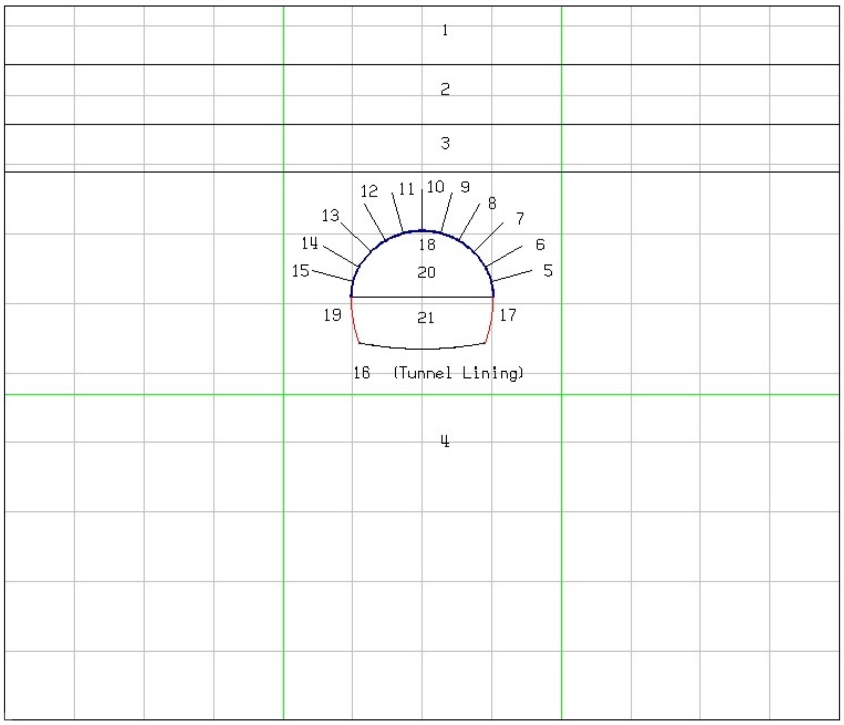


Figure 5.40 Group section view

Table 5.5 summarizes key parameters of groups.

Group	Name	MTYPE	NAC	NDAC	MATNO / LTP / LMAT / IEND
1	Top Soil	3			1 / 0 / 0 / 2
2	Weathered Rock	3			2 / 0 / 0 / 2
3	Soft Rock	3			3 / 0 / 0 / 2
4	Hard Rock	3			4 / 0 / 0 / 2
5	Rock Bolt-1	2	4	999	0 / 3 / 1 / -2
6	Rock Bolt-2	2	4	999	0 / 3 / 1 / -2
7	Rock Bolt-3	2	4	999	0 / 3 / 1 / -2
8	Rock Bolt-4	2	4	999	0 / 3 / 1 / -2
9	Rock Bolt-5	2	4	999	0 / 3 / 1 / -2
10	Rock Bolt-6	2	4	999	0 / 3 / 1 / -2
11	Rock Bolt-7	2	4	999	0 / 3 / 1 / -2
12	Rock Bolt-8	2	4	999	0 / 3 / 1 / -2
13	Rock Bolt-9	2	4	999	0 / 3 / 1 / -2
14	Rock Bolt-10	2	4	999	0 / 3 / 1 / -2
15	Rock Bolt-11	2	4	999	0 / 3 / 1 / -2
16	Tunneling Lining	-2	9	999	MATNOj = 7, LTPi = 0, LTPo = 2 LMATo = 2, IEND = 2
17	Shotcrete Right Lower	2	7	999	0 / 2 / 1 / 3
18	Shotcrete Upper	2	4	999	0 / 2 / 1 / 3
19	Shotcrete Left Lower	2	7	999	0 / 2 / 1 / 3
20	Upper Core	3	0	5	5 / 0 / 0 / 3
21	Lower Core	3	0	8	6 / 0 / 0 / 3

Table 5.5 Group key parameters

5.2.2 Base Mesh

Built-in Base Mesh dialog is shown in Figure 5.41 with input data for blocks and boundary condition. Element size is more refined at the center block considering relatively high stress change here due to tunnel construction.

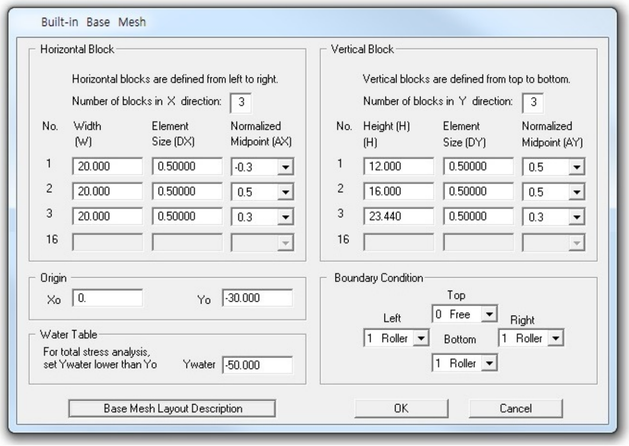


Figure 5.41 Built-in base mesh dialog

Figure 5.42 shows base mesh plot on drawing board.

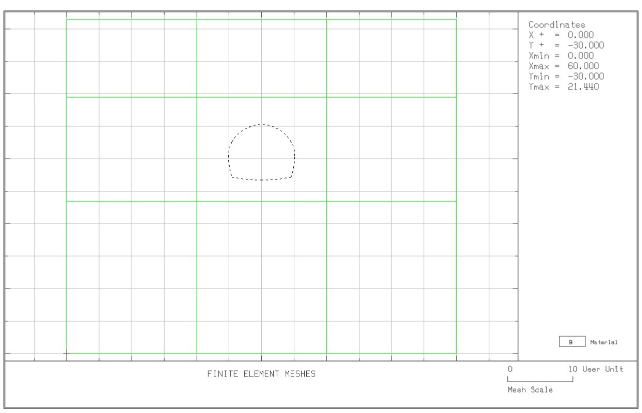


Figure 5.42 Base mesh plot on drawing board

5.2.3 Groups

Group meshes are divided into five parts:

- Geological profile
- Rock bolt
- Lining
- Shotcrete
- Core

Final finite element meshes are most influenced by group order and IEND.

5.2.3.1 Geological Profile

In situ geological profile consists of four layers: top soil, weathered rock, soft rock, and hard rock. Table 5.6 lists key parameters of these groups.

Group	Profile	MTYPE	Elem.	MATNO	Seg.	Beginning Point		Ending Point		IEND
						X	Y	X	Y	
1	Top Soil	3	Cont	1	1	0	17.24	60	17.24	2
					2	60	17.24	60	21.44	2
					3	60	21.44	0	21.44	2
					4	0	21.44	0	17.24	2
2	Weathered Rock	3	Cont	2	1	0	12.94	60	12.94	2
					2	60	12.94	60	17.24	2
					3	60	17.24	0	17.24	2
					4	0	17.24	0	12.94	2
3	Soft Rock	3	Cont	3	1	0	9.44	60	9.44	2
					2	60	9.44	60	12.94	2
					3	60	12.94	0	12.94	2
					4	0	12.94	0	9.44	2
4	Hard Rock	3	Cont	4	1	0	-30	60	-30	2
					2	60	-30	60	9.44	2
					3	60	9.44	0	9.44	2
					4	0	9.44	0	-30	2

Table 5.6 Key parameters for geological profile

Figure 5.43 shows Group dialog for top soil layer.
Group dialogs for the other layers are very similar to this group 1.
It is a good idea to click Save button occasionally in case of system down.

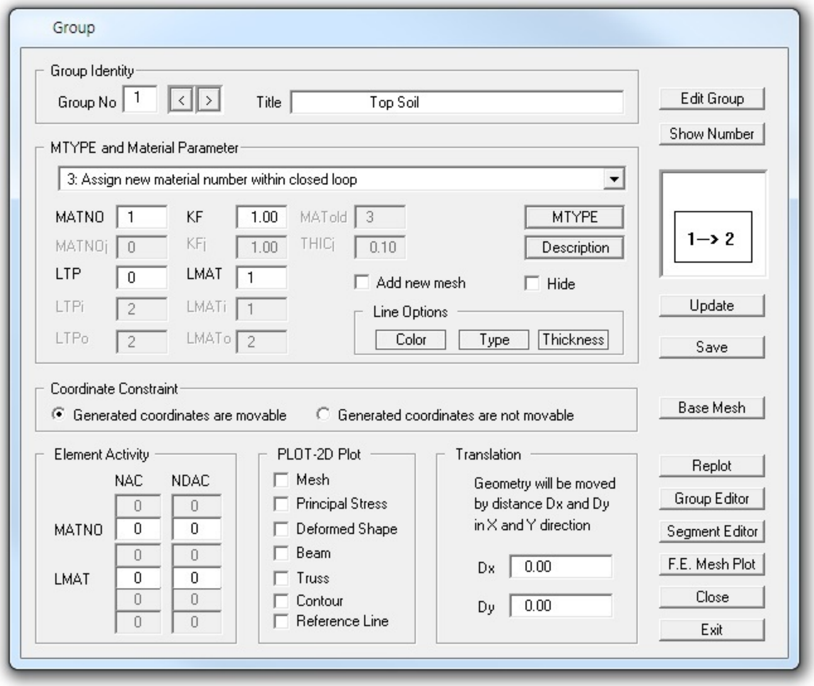


Figure 5.43 Group dialog for top soil layer

5.2.3.2 Rock Bolt

There are eleven rock bolts above the tunnel crown as schematically shown in Figure 5.44. Table 5.7 lists key parameters of these groups.

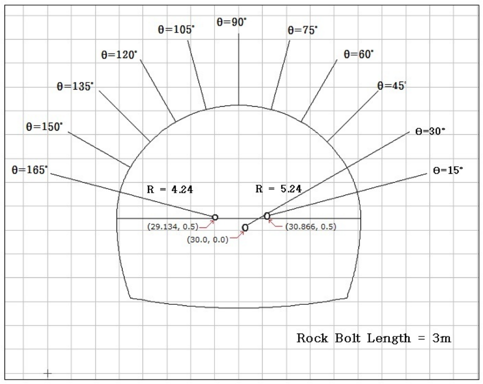


Figure 5.44 Rock bolt layout

Group	Name	NAC/NDAC	Origin		Radius & Angle				MTYPE/LTP/LMAT/IEND
			X_o	Y_o	R_X	R_Y	Θ_b	Θ_e	
5	Bolt-1	4 / 999	30.866	0.5	4.24	7.24	15	15	2 / 3 / 1 / -2
6	Bolt-2	4 / 999	30	0	5.24	8.24	30	30	2 / 3 / 1 / -2
7	Bolt-3	4 / 999	30	0	5.24	8.24	45	45	2 / 3 / 1 / -2
8	Bolt-4	4 / 999	30	0	5.24	8.24	60	60	2 / 3 / 1 / -2
9	Bolt-5	4 / 999	30	0	5.24	8.24	75	75	2 / 3 / 1 / -2
10	Bolt-6	4 / 999	30	0	5.24	8.24	90	90	2 / 3 / 1 / -2
11	Bolt-7	4 / 999	30	0	5.24	8.24	105	105	2 / 3 / 1 / -2
12	Bolt-8	4 / 999	30	0	5.24	8.24	120	120	2 / 3 / 1 / -2
13	Bolt-9	4 / 999	30	0	5.24	8.24	135	135	2 / 3 / 1 / -2
14	Bolt-10	4 / 999	30	0	5.24	8.24	150	150	2 / 3 / 1 / -2
15	Bolt-11	4 / 999	29.134	0.5	4.24	7.24	165	165	2 / 3 / 1 / -2

Table 5.7 Key parameters for rock bolt

Figure 5.45 shows Group dialog for the first rock bolt at 15 degrees. Group dialogs for other rock bolts are very similar to this group 5.

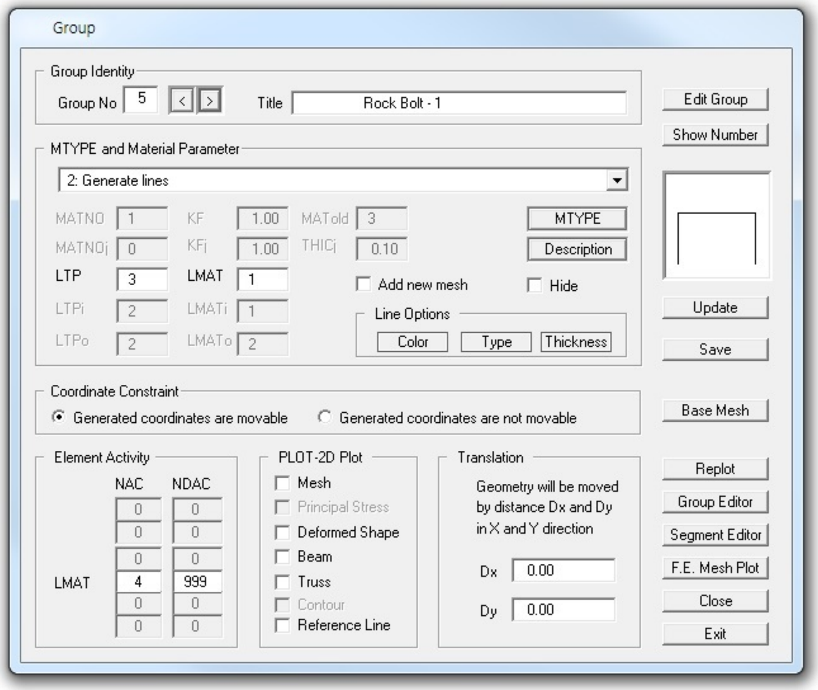


Figure 5.45 Group dialog for rock bolt at 15 degrees

5.2.3.3 Lining

Lining is the reinforced concrete liner which is modeled by beam elements. Seven segments are used to model lining as shown in Figure 5.46.

The interface between lining and shotcrete is modeled by joint element as shown in Figure 5.47. It should be noted that MTYPE = -2 in this group includes both lining and joint elements.

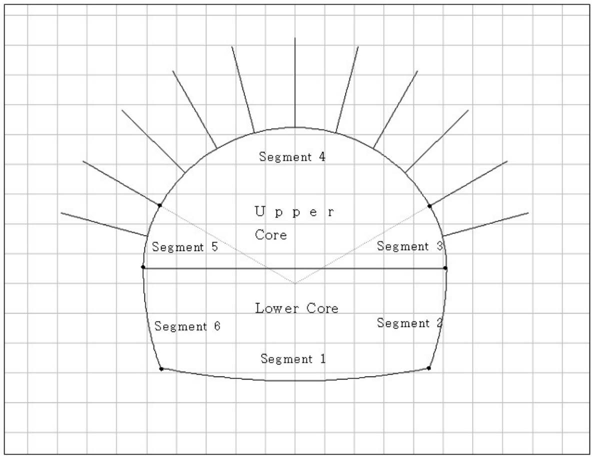


Figure 5.46 Lining segments

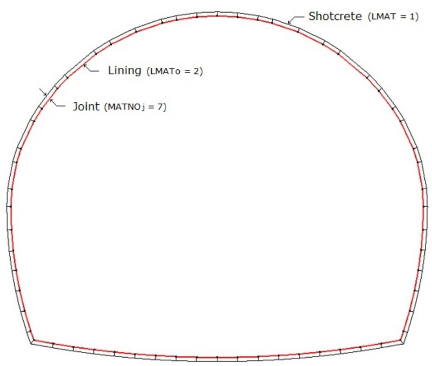


Figure 5.47 Interface joint element

Table 5.8 lists key parameters of this group.

	Element Type	Material No	Element Activity	
			NAC	NDAC
Interface	Joint	MATNOj = 7	9	999
Lining	Beam (LTPo = 2)	LMATo = 2	9	999

Group	Name	MTYPE	Seg.	Origin		Radius & Angle				IEND
				X_o	Y_o	R_X	R_Y	Θ_b	Θ_e	
16	Tunnel Lining	-2	1	30	20.59	23.86	23.86	270	280.94	2
			2	25.25	0.5	9.86	9.86	-19.78	0	2
			3	30.866	0.5	4.24	4.24	0	30	2
			4	30	0	5.24	5.24	30	150	2
			5	29.134	0.5	4.24	4.24	150	180	2
			6	34.75	0.5	9.86	9.86	-180	-160.22	2
			7	30	20.59	23.86	23.86	259.06	270	2

Table 5.8 Key parameters for lining and joint elements

Figure 5.48 shows Group dialog for tunnel lining.

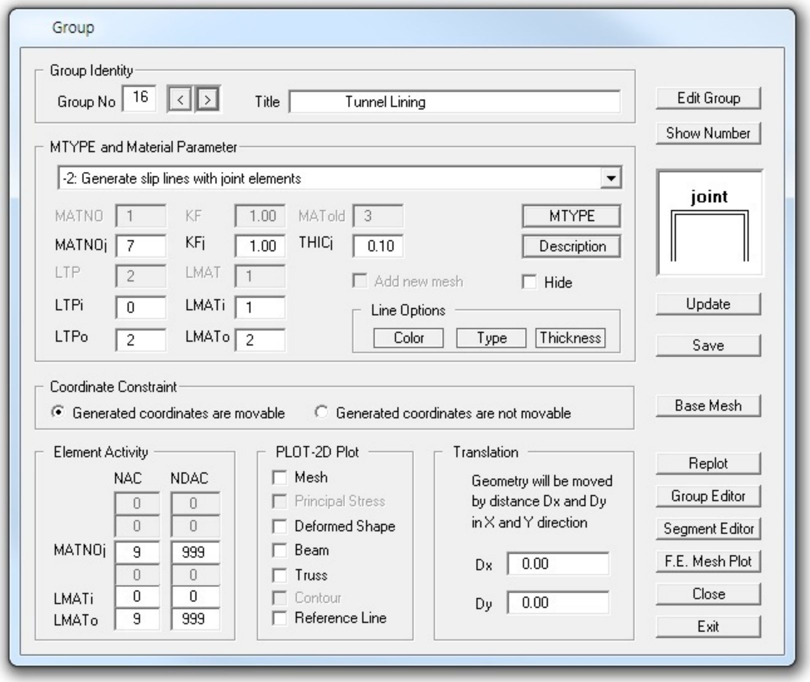
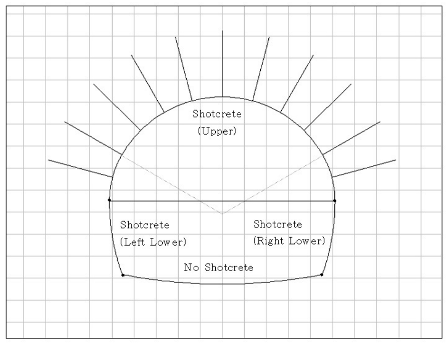


Figure 5.48 Group dialog for tunnel lining

5.2.3.4 Shotcrete

Shotcrete is applied to upper tunnel wall right after excavation of upper core and lower tunnel walls right after excavation of lower core as shown in Figure 5.49. But shotcrete is not applied at tunnel invert.
Table 5.9 lists key parameters of these groups.

Figure 5.49
Shotcrete cross section



Group	Name	MTYPE	LTP	LMAT	Element Activity	
					NAC	NDAC
17	Shotcrete: Right Lower	2	2	1	7	999
18	Shotcrete: Upper	2	2	1	4	999
19	Shotcrete: Left Lower	2	2	1	7	999

Group	Name	MTYPE	Seg	Origin		Radius & Angle				IEND
				X_o	Y_o	R_X	R_Y	Θ_b	Θ_e	
17	Shotcrete Right Lower	2	1	25.25	0.5	9.86	9.86	-19.78	0	3
18	Shotcrete Upper	2	1	30.866	0.5	4.24	4.24	0	30	3
			2	30	0	5.24	5.24	30	150	3
			3	29.134	0.5	4.24	4.24	150	180	3
19	Shotcrete Left Lower	2	1	34.75	0.5	9.86	9.86	-180	-160.22	3

Table 5.9 Key parameters for shotcrete elements

Figure 5.50 shows Group dialog for the upper shotcrete.
Group dialogs for other lower shotcrete are very similar to this group 18.

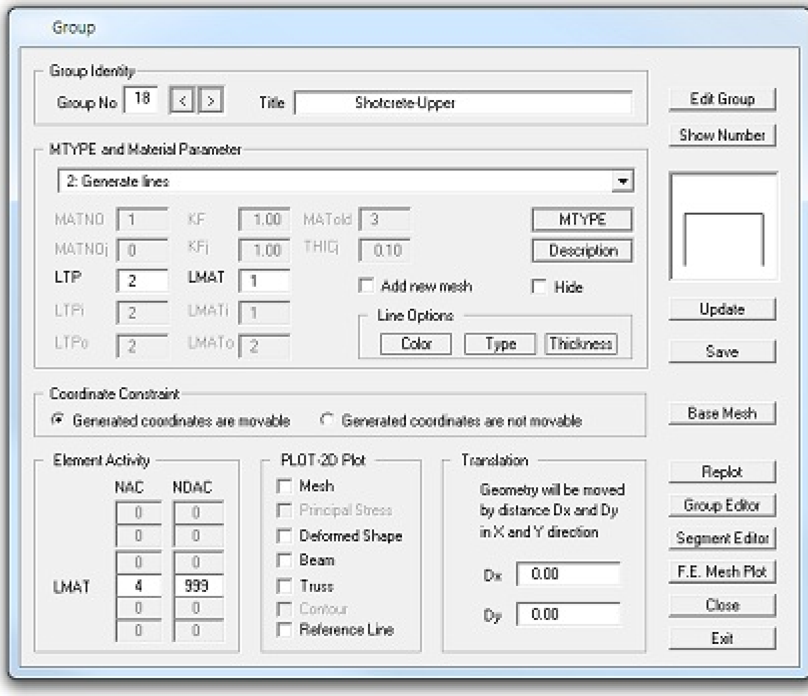


Figure 5.50 Group dialog for upper shotcrete

5.2.3.5 Core

Core is divided into upper and lower parts as in Figure 5.46 considering the order of excavation. Table 5.10 lists key parameters of these groups.

Group	Name	MTYPE	Element	MATNO	Element Activity	
					NAC	NDAC
20	Upper Core	3	Cont.	5	0	5
21	Lower Core	3	Cont.	6	0	8

Group	Seg	Line Segment				Arc Segment						IEND
		Beginning Pt.		Ending Pt.		Origin		Radius & Angle				
		X	Y	X	Y	X_o	Y_o	R_X	R_Y	Θ_b	Θ_e	
20	1	24.894	0.5	35.106	0.5							3
	2					30.866	0.5	4.24	4.24	0	30	3
	3					30	0	5.24	5.24	30	150	3
	4					29.134	0.5	4.24	4.24	150	180	3
21	1					30	20.59	23.86	23.86	259.06	280.94	3
	2					25.25	0.5	9.86	9.86	-19.78	0	3
	3	35.106	0.5	24.894	0.5							3
	4					34.75	0.5	9.86	9.86	-180	-160.22	3

Table 5.10 Key parameters for core elements

Figure 5.51 shows Group dialog for the upper core.
Group dialog for the other lower core is very similar to this group 20.

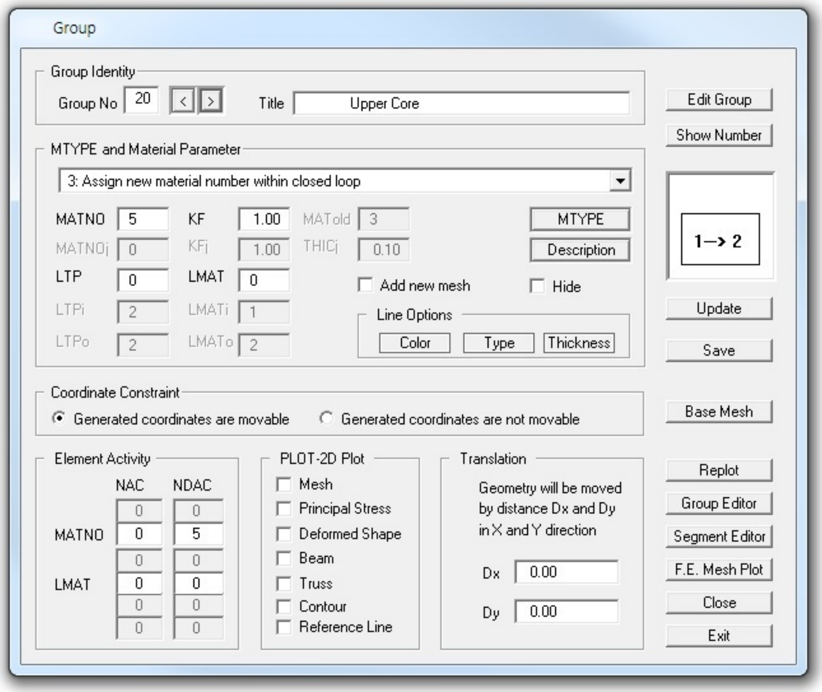


Figure 5.51 Group dialog for upper core

5.2.4 Finite Element Mesh Plot

Figure 5.52 shows finite element meshes generated from group meshes. Finite element meshes around tunnel are shown in Figure 5.53.

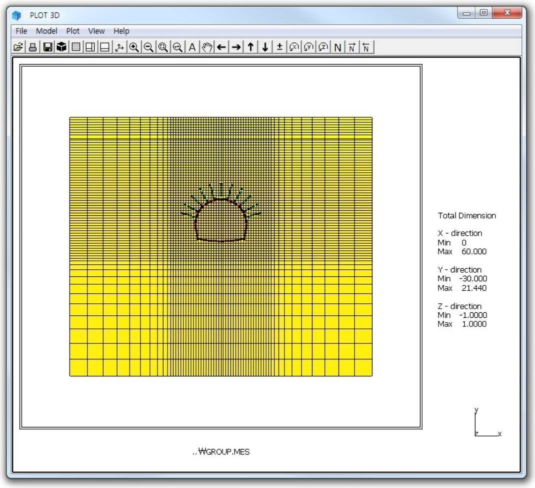


Figure 5.52 Finite element meshes for NATM tunnel

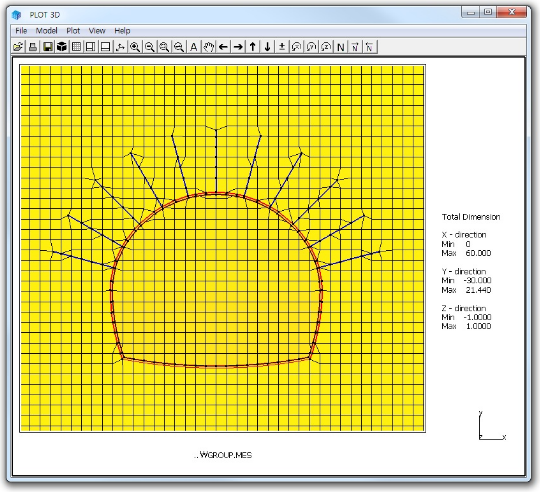


Figure 5.53 Finite element meshes around tunnel

5.3 Excavation

This example illustrates how to build group meshes for typical multi-step excavations performed near the existing box structure.

5.3.1 Overview

The cross section of this excavation problem consists of box structure, SCE-wall, anchors, and excavation zones as shown in Figure 5.54.

Cross section near the box structure is shown in detail in Figure 5.55.

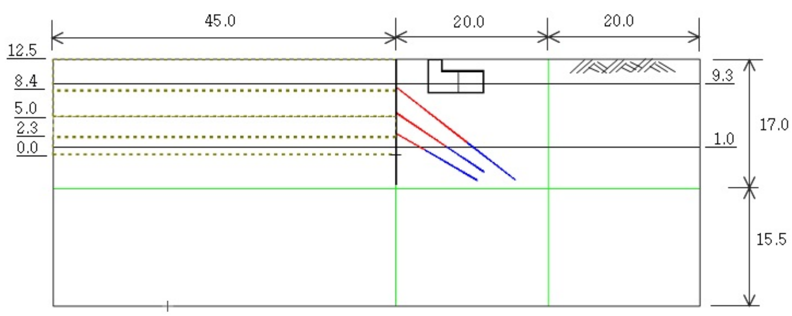


Figure 5.54 Schematic section of excavation problem

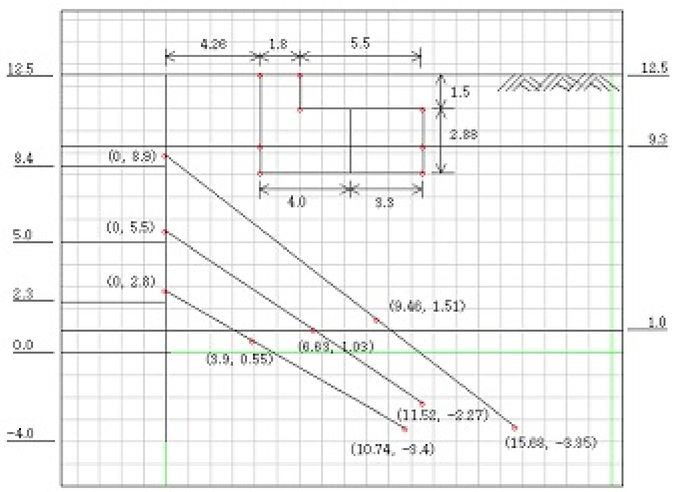


Figure 5.55 Cross section near box structure

Table 5.11 shows the construction sequence associated with multi-step excavations.

Step	Description
1,2	In situ stress
3	Box Excavation and Frame Construction
4	First Excavation (Y = 8.4 m)
5	First Anchor Installation
6	Second Excavation (Y = 5.0 m)
7	Second Anchor Installation
8	Third Excavation (Y = 2.3 m)
9	Third Anchor Installation
10	Fourth Excavation (Y = 0.0 m)

Table 5.11 Construction sequence

A total of 17 groups are used to model this excavation problem as schematically shown in Figure 5.56: 3 for in situ geological profile, 3 for box structure, 1 for SCE-wall, 4 for excavations, and 6 for anchors.

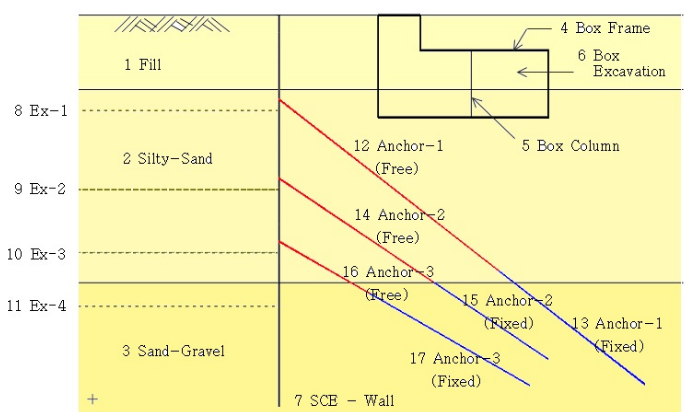


Figure 5.56 Group section view

Table 5.12 summarizes key parameters of groups.

Group	Name	MTYPE	NAC	NDAC	MATNO / LTP / LMAT / IEND
1	Fill	3	0	0	1 / 0 / 0 / 2
2	Silty-Sand	3	0	0	2 / 0 / 0 / 2
3	Sand-Gravel	3	0	0	3 / 0 / 0 / 2
4	Box Frame	2	3	999	0 / 2 / 2 / 2
5	Box Column	2	3	999	0 / 2 / 3 / 2
6	Box Excavation	3	0	3	0 / 0 / 0 / 3
7	SCE-Wall	2	4	999	0 / 2 / 1 / 2
8	Excavation-1	3	0	4	0 / 0 / 0 / 2
9	Excavation-2	3	0	6	0 / 0 / 0 / 2
10	Excavation-3	3	0	8	0 / 0 / 0 / 2
11	Excavation-4	3	0	10	0 / 0 / 0 / 2
12	Anchor-1 Free	2	5	999	0 / 3 / 1 / 0
13	Anchor-1 Fixed	2	5	999	0 / 3 / 2 / -2
14	Anchor-2 Free	2	7	999	0 / 3 / 3 / 0
15	Anchor-2 Fixed	2	7	999	0 / 3 / 4 / -2
16	Anchor-3 Free	2	9	999	0 / 3 / 5 / 0
17	Anchor-3 Fixed	2	9	999	0 / 3 / 6 / -2

Table 5.12 Group key parameters

5.3.2 Base Mesh

Built-in Base Mesh dialog is shown in Figure 5.57 with input data for blocks and boundary condition. Element size is more refined at the top center block considering relatively high stress change here due to excavation.

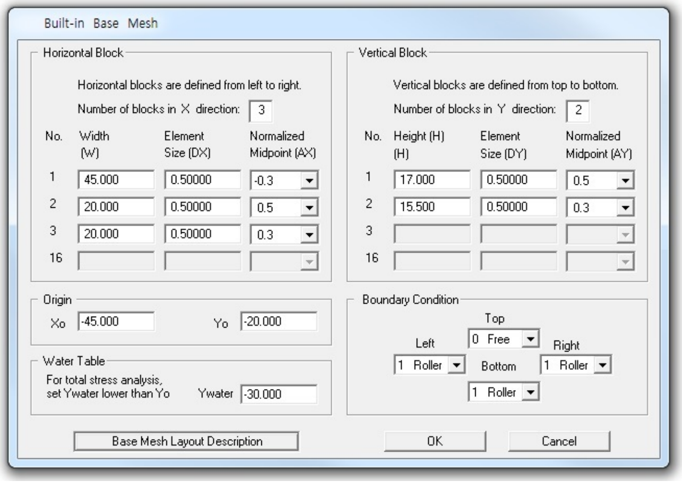


Figure 5.57 Built-in base mesh dialog

Figure 5.58 shows base mesh plot on drawing board.

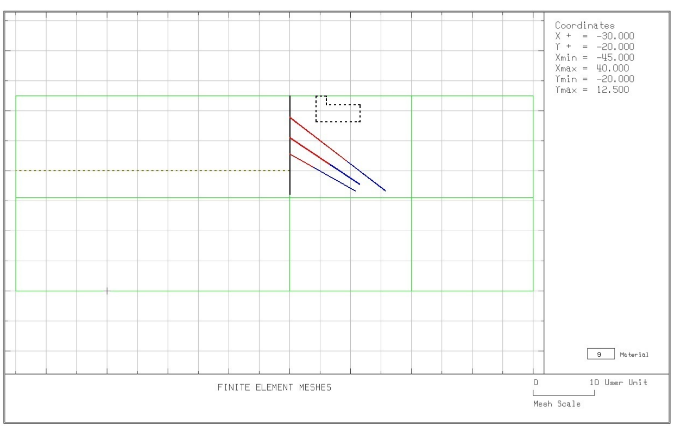


Figure 5.58 Base mesh plot on drawing board

5.3.3 Groups

Group meshes are divided into five parts:

- Geological profile
- Box structure
- SCE-Wall
- Excavation
- Anchor

It should be noted that the final finite element meshes are most influenced by group order and IEND.

5.3.3.1 Geological Profile

In situ geological profile consists of three layers: fill, silty-sand, and sand-gravel. Table 5.13 lists key parameters of these groups

Group	Profile	MTYPE	Elem.	MATNO	Seg.	Beginning Point		Ending Point		IEND
						X	Y	X	Y	
1	Fill	3	Cont	1	1	-45	9.3	40	9.3	2
					2	40	9.3	40	12.5	2
					3	40	12.5	-45	12.5	2
					4	-40	12.5	-45	9.3	2
2	Silty-Sand	3	Cont	2	1	-45	1	40	1	2
					2	40	1	40	9.3	2
					3	40	9.3	-45	9.3	2
					4	-45	9.3	-45	1	2
3	Sand-Gravel	3	Cont	3	1	-45	-20	40	-20	2
					2	40	-20	40	1	2
					3	40	1	-45	1	2
					4	-45	1	-45	-20	2

Table 5.13 Key parameters for geological profile

Figure 5.59 shows Group dialog for top fill.
Group dialogs for the other layers are very similar to this group 1.

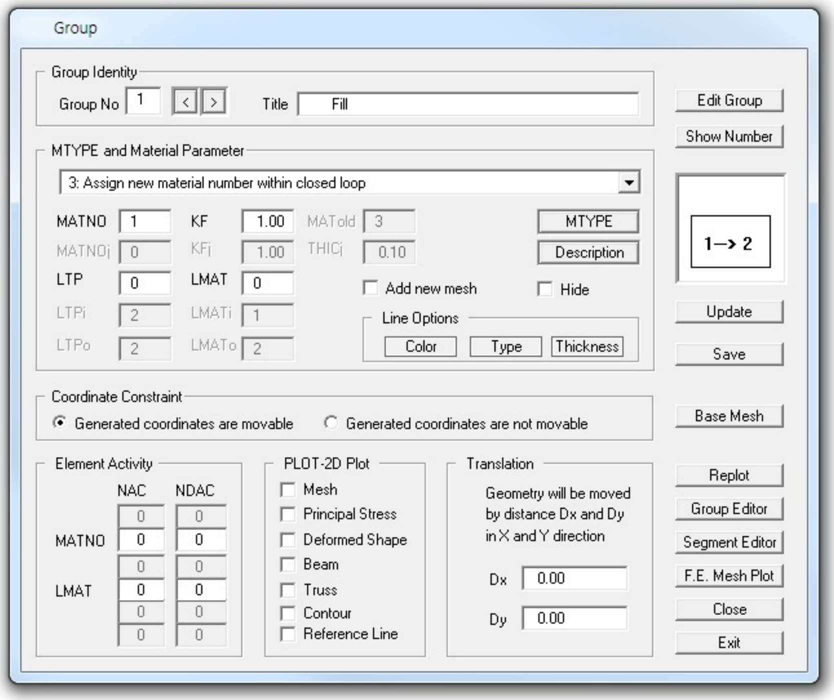


Figure 5.59 Group dialog for top fill

5.3.3.2 Box Structure

Box structure consists of frame, column, and excavation as schematically shown in Figure 5.56. Table 5.14 lists key parameters of these groups.

Group	Name	MTYPE	LTP	LMAT	Element Activity		Seg	Beginning Point		Ending Point		IEND
					NAC	NDAC		X	Y	X	Y	
4	Frame	2	2	2	3	999	1	4.26	8.12	11.56	8.12	2
							2	11.56	8.12	11.56	11	2
							3	11.56	11	6.06	11	2
							4	6.06	11	6.06	12.5	2
							5	6.06	12.5	4.26	12.5	2
							6	4.26	12.5	4.26	8.12	2
5	Column	2	2	3	3	999	1	8.26	11	8.26	8.12	2

Group	Name	MTYPE	Elem	MATNO	Element Activity		Seg	Beginning Point		Ending Point		IEND
					NAC	NDAC		X	Y	X	Y	
6	Excavation	3	Cont	0	0	3	1	4.26	8.12	11.56	8.12	2
							2	11.56	8.12	11.56	11	2
							3	11.56	11	6.06	11	2
							4	6.06	11	6.06	12.5	2
							5	6.06	12.5	4.26	12.5	2
							6	4.26	12.5	4.26	8.12	2

Table 5.14 Key parameters for box structure

Figure 5.60 shows Group dialog for the box frame.
Group dialog for box column is very similar to this group 4.

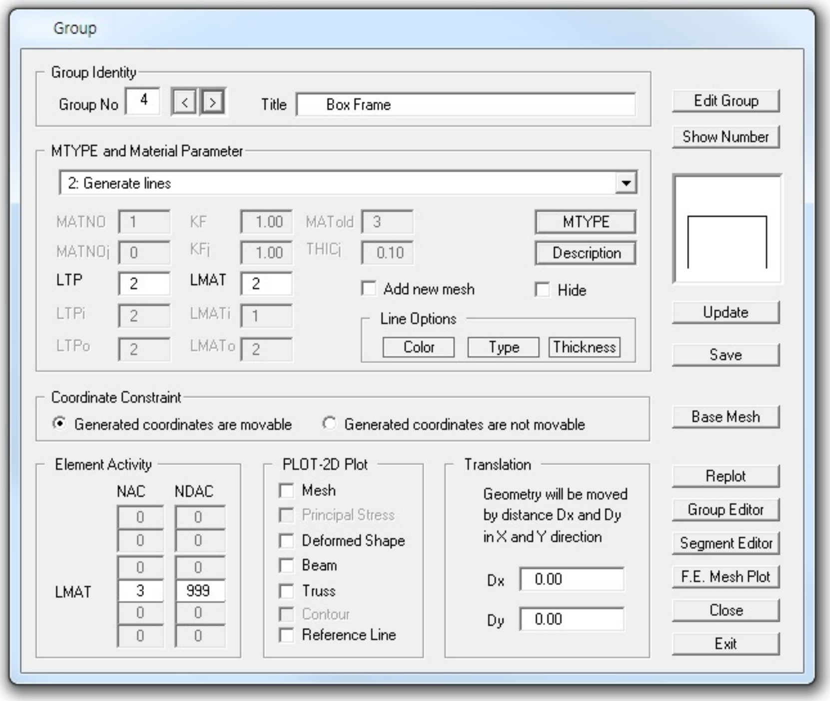


Figure 5.60 Group dialog for box frame

Figure 5.61 shows Group dialog for the box excavation.

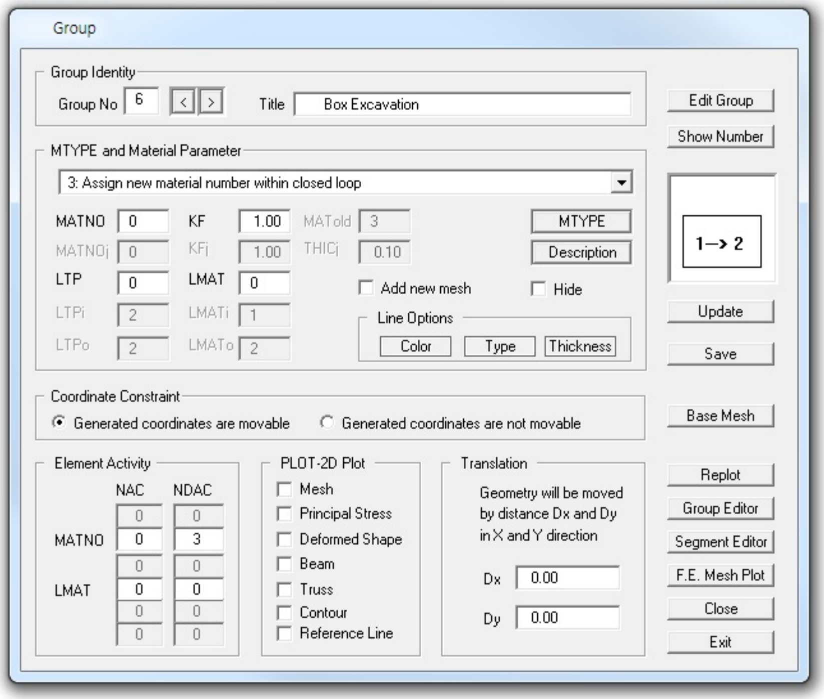


Figure 5.61 Group dialog for box excavation

5.3.3.3 SCE-Wall

SCE-Wall is the structure to prevent ground movement due to excavations and is supported by anchors as schematically shown in Figure 5.56.
Table 5.15 lists key parameters of this group.

Group	Name	MTYPE	LTP	LMAT	Element Activity		Seg	Beginning Point		Ending Point		IEND
					NAC	NDAC		X	Y	X	Y	
7	SCE-Wall	2	2	1	4	999	1	0	12.5	0	-4	2

Table 5.15 Key parameters for SCE-wall

Figure 5.62 shows Group dialog for SCE-wall.

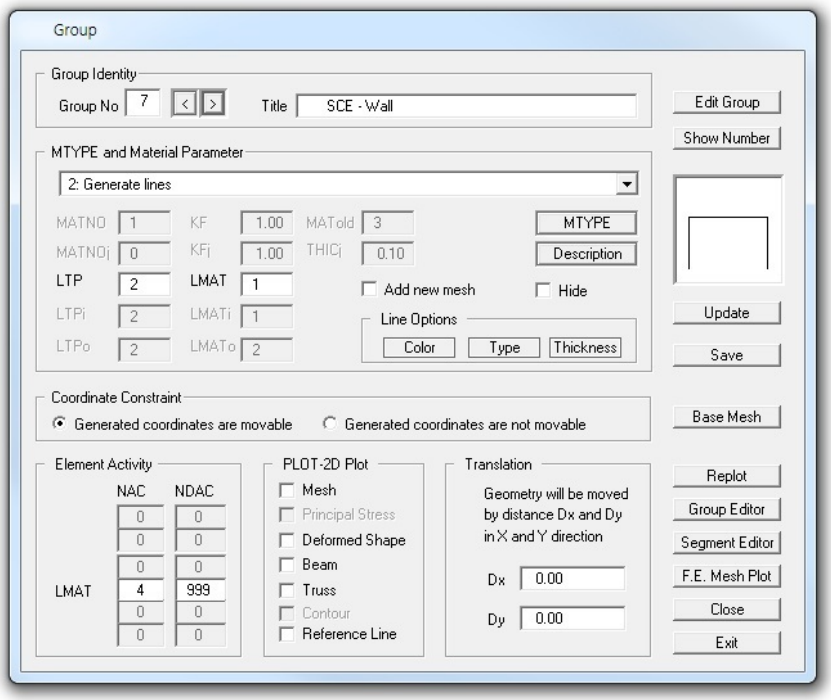


Figure 5.62 Group dialog for SCE-wall

5.3.3.4 Excavation

Excavations are conducted through four stages as schematically shown in Figure 5.56. Table 5.16 lists key parameters of these groups.

Group	Name	MTYPE	Elem	MATNO / NAC / NDAC	Seg.	Beginning Point		Ending Point		IEND
						X	Y	X	Y	
8	Excavation-1	3	Cont	0 / 0 / 4	1	-45	8.4	0.0	8.4	2
					2	0	8.4	0	12.5	2
					3	0	12.5	-45	12.5	2
					4	-45	12.5	-45	8.4	2
9	Excavation-2	3	Cont	0 / 0 / 6	1	-45	5	0	5	2
					2	0	5	0	8.4	2
					3	0	8.4	-45	8.4	2
					4	-45	8.4	-45	5	2
10	Excavation-3	3	Cont	0 / 0 / 8	1	-45	2.3	0	2.3	2
					2	0	2.3	0	5	2
					3	0	5	-45	5	2
					4	-45	5	-45	2.3	2
11	Excavation-4	3	Cont	0 / 0 / 10	1	-45	0	0	0	2
					2	0	0	0	2.3	2
					3	0	2.3	-45	2.3	2
					4	-45	2.3	-45	0	2

Table 5.16 Key parameters for excavation

Figure 5.63 shows Group dialog for the first excavation.
Group dialogs for the other excavations are very similar to this group 8.

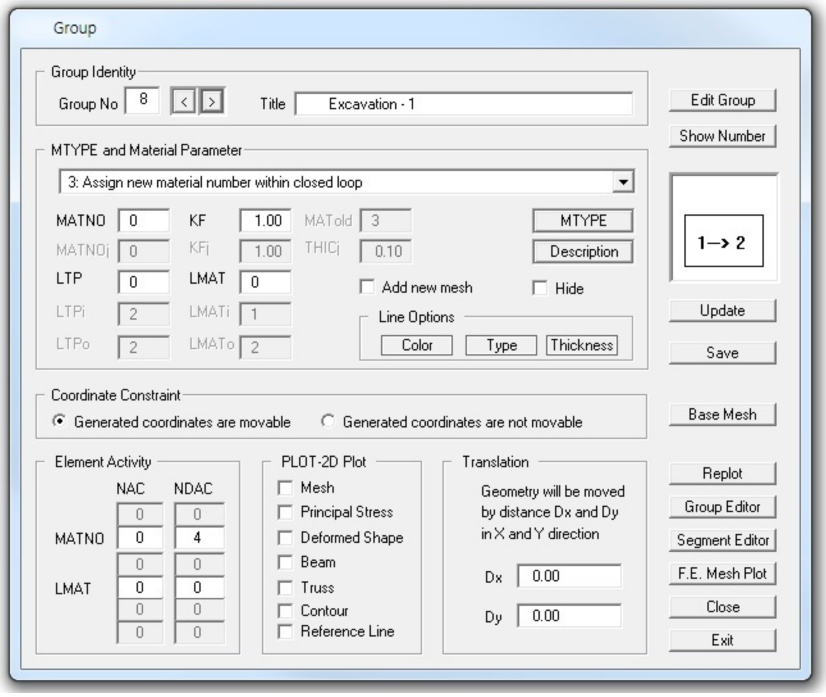


Figure 5.63 Group dialog for the first excavation

5.3.3.5 Anchor

Three anchors are used to support SCE-wall as schematically shown in Figure 5.56. Each anchor consists of two parts: free and fixed length. Table 5.17 lists key parameters of these groups.

Group	Name	MTYPE / LTP / LMAT / NAC / NDAC	Seg.	Beginning Point		Ending Point		NDIV	IEND
				X	Y	X	Y		
12	Anchor-1 Free	2 / 3 / 1 / 5 / 999	1	0	8.9	9.46	1.51	1	0
13	Anchor-1 Fixed	2 / 3 / 2 / 5 / 999	1	9.46	1.51	15.68	-3.35	0	-2
14	Anchor-2 Free	2 / 3 / 3 / 7 / 999	1	0	5.5	6.63	1.03	1	0
15	Anchor-2 Fixed	2 / 3 / 4 / 7 / 999	1	6.63	1.03	11.52	-2.27	0	-2
16	Anchor-3 Free	2 / 3 / 5 / 9 / 999	1	0	2.8	3.9	0.55	1	0
17	Anchor-3 Fixed	2 / 3 / 6 / 9 / 999	1	3.9	0.55	10.74	-3.4	0	-2

Table 5.17 Key parameters for anchor

Figure 5.64 shows Group dialog for the first anchor (free part). Group dialogs for other anchors are very similar to this group 12.

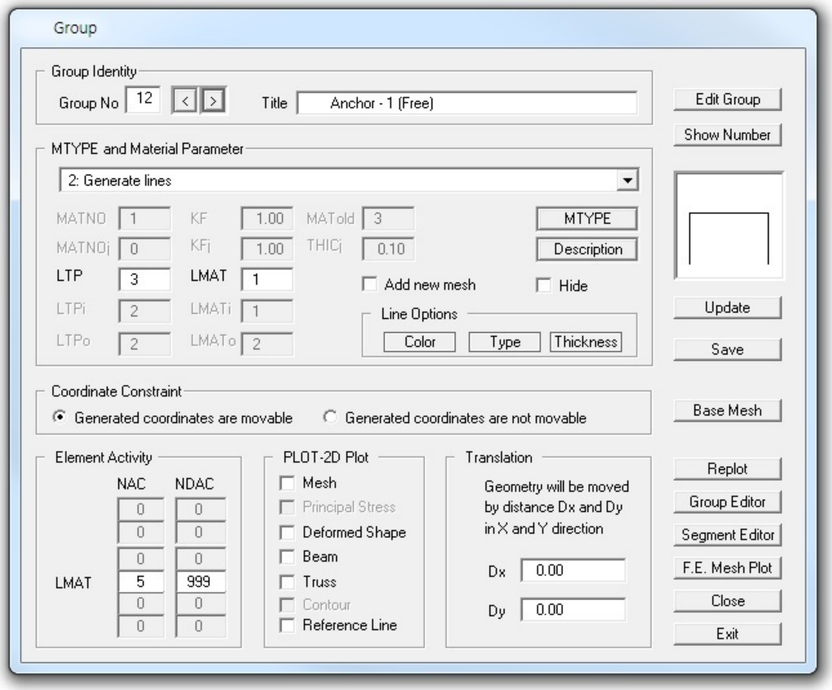


Figure 5.64 Group dialog for the first anchor (free part)

5.3.4 Finite Element Mesh Plot

Figure 5.65 shows finite element meshes generated from group meshes. Finite element meshes near box structure are shown in Figure 5.66.

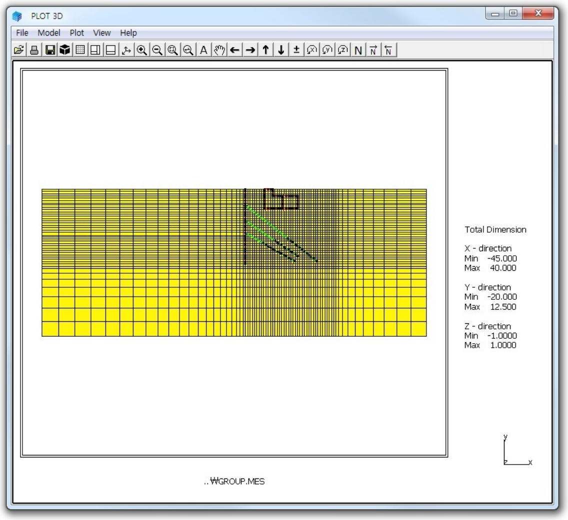


Figure 5.65
Finite element meshes

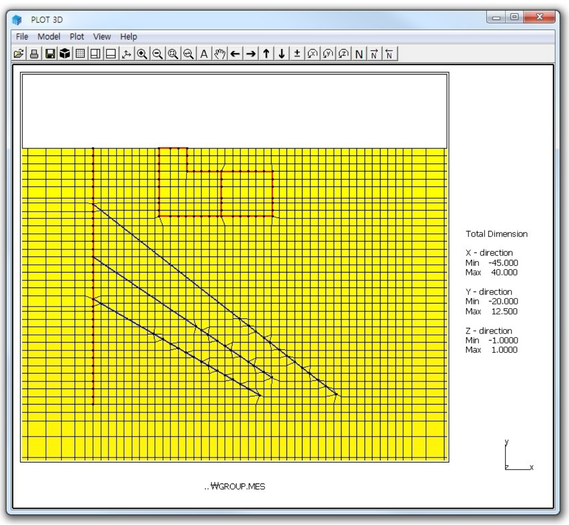


Figure 5.66
Finite element meshes near box structure

5.4 Buried Pipe

This example illustrates how to build group meshes for typical pipe buried in the trench followed by multi-step embankment lifts.

5.4.1 Overview

The cross section of this buried pipe consists of natural soil, bedding, steel pipe, backfill, and lifts as shown in Figure 5.67.

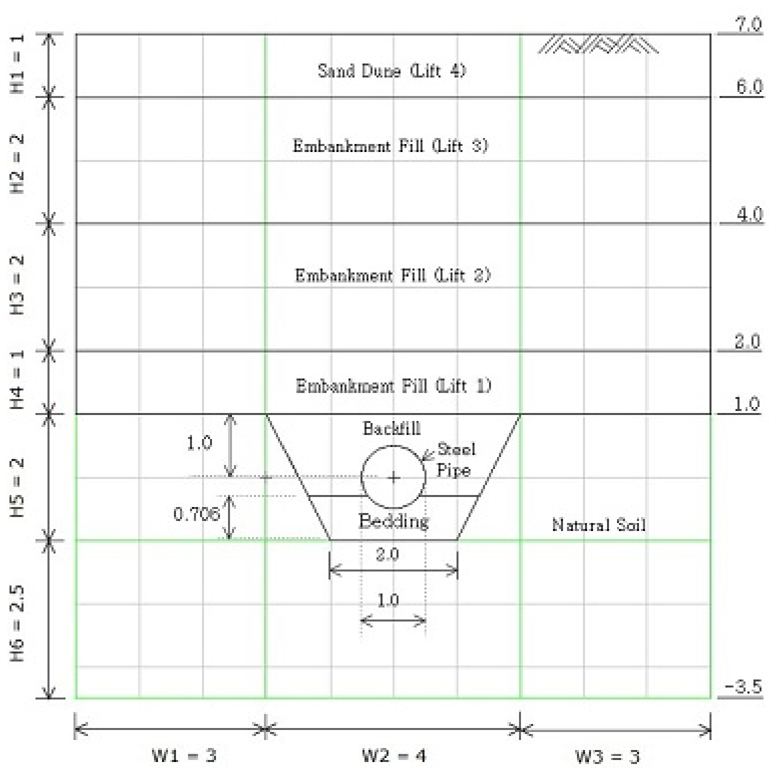


Figure 5.67 Schematic section of buried pipe

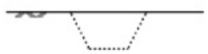
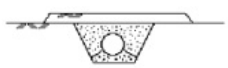
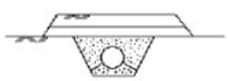
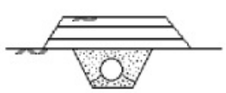
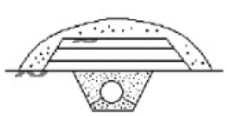
Step	Construction Sequence	Description	Element Activity
1,2		In situ K_o state	Active elements: Natural soil within trench
3		Excavate trench	Deactive elements: Natural soil within trench
4		Place bedding	Active elements: Compacted sand for bedding
5		Place steel pipe Fill the backfill	Active elements: Steel pipe Compacted sand for backfill
6		Place first lift of embankment fill	Active elements: First lift of embankment fill
7		Place second lift of embankment fill	Active elements: Second lift of embankment fill
8		Place third lift of embankment fill	Active elements: Third lift of embankment fill
9		Place fourth lift of sand done	Active elements: Fourth lift of sand done

Table 5.18 Construction sequence

A total of 9 groups are used to model this buried pipe as schematically shown in Figure 5.68: 1 for natural soil, 1 for excavation, 2 for compacted sands, 1 for steel pipe, and 4 for lifts.

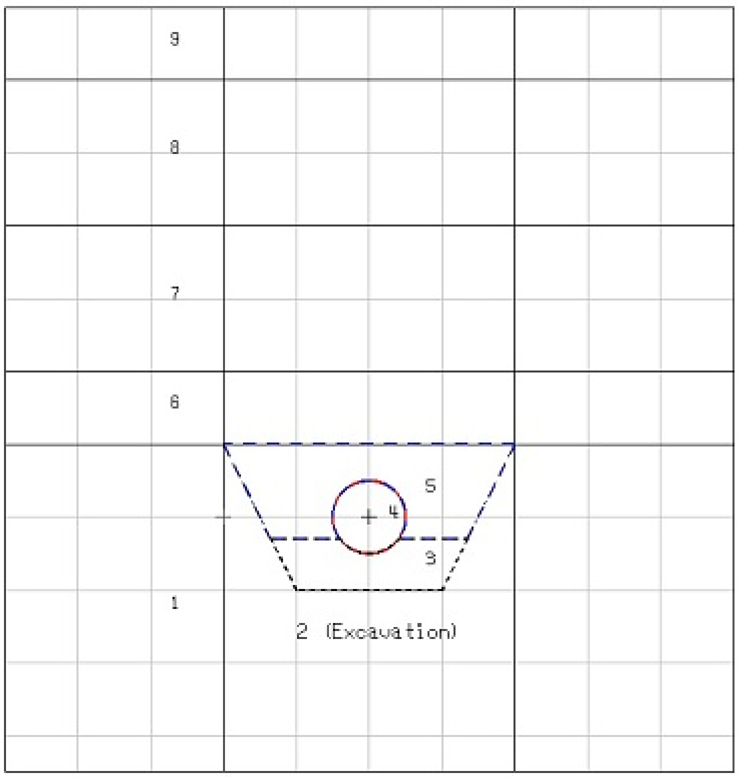


Figure 5.68 Group section view

Table 5.19 summarizes key parameters of groups.

Group	Name	MTYPE	NAC	NDAC	MATNO / LTP / LMAT / IEND
1	Natural Soil	3	0	0	1 / 0 / 0 / 2
2	Excavation	3	0	3	1 / 0 / 0 / 2
3	Bedding	3	4	999	2 / 0 / 0 / 2
4	Steel Pipe	2	5	999	0 / 2 / 1 / 2
5	Backfill	3	5	999	3 / 0 / 0 / 2
6	Lift-1	3	6	999	4 / 0 / 0 / 2
7	Lift-2	3	7	999	5 / 0 / 0 / 2
8	Lift-3	3	8	999	6 / 0 / 0 / 2
9	Lift-4	3	9	999	7 / 0 / 0 / 2

Table 5.19 Group key parameters

5.4.2 Base Mesh

Built-in Base Mesh dialog is shown in Figure 5.69 with input data for blocks and boundary condition. Element size is more refined at the block in trench considering relatively high stress change here due to pipe construction. Figure 5.70 shows base mesh plot on drawing board.

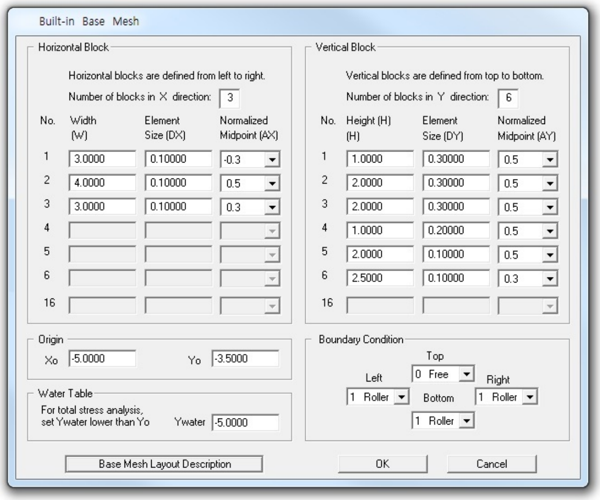


Figure 5.69 Built-in base mesh dialog

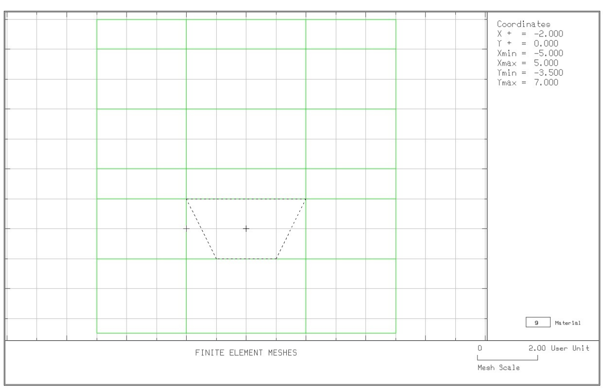


Figure 5.70 Base mesh plot on drawing board

5.4.3 Groups

Group meshes are divided into three parts:

- Natural soil and excavation
- Pipe construction
- Lift

It should be noted that the final finite element meshes are most influenced by group order and IEND.

5.4.3.1 Natural Soil and Excavation

Excavation is performed in natural soil to make trench.
Table 5.20 lists key parameters of these groups

Group	Name	MTYPE	Elem	MATNO / NAC / NDAC	Seg.	Beginning Point		Ending Point		IEND
						X	Y	X	Y	
1	Natural Soil	3	Cont	1 / 0 / 0	1	-5	-3.5	5	-3.5	2
					2	5	-3.5	5	1	2
					3	5	1	-5	1	2
					4	-5	1	-5	-3.5	2
2	Excavation	3	Cont	1 / 0 / 3	1	-1	-1	1	-1	2
					2	1	-1	2	1	2
					3	2	1	-2	1	2
					4	-2	1	-1	-1	2

Table 5.20 Key parameters for natural soil and excavation

Figure 5.71 shows Group dialog for natural soil.

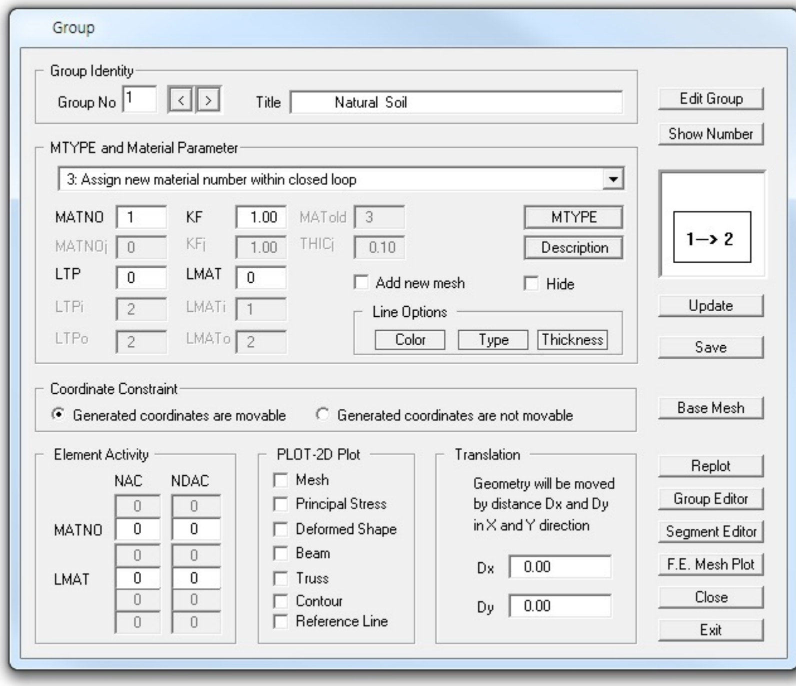


Figure 5.71 Group dialog for natural soil

Figure 5.72 shows Group dialog for excavation.

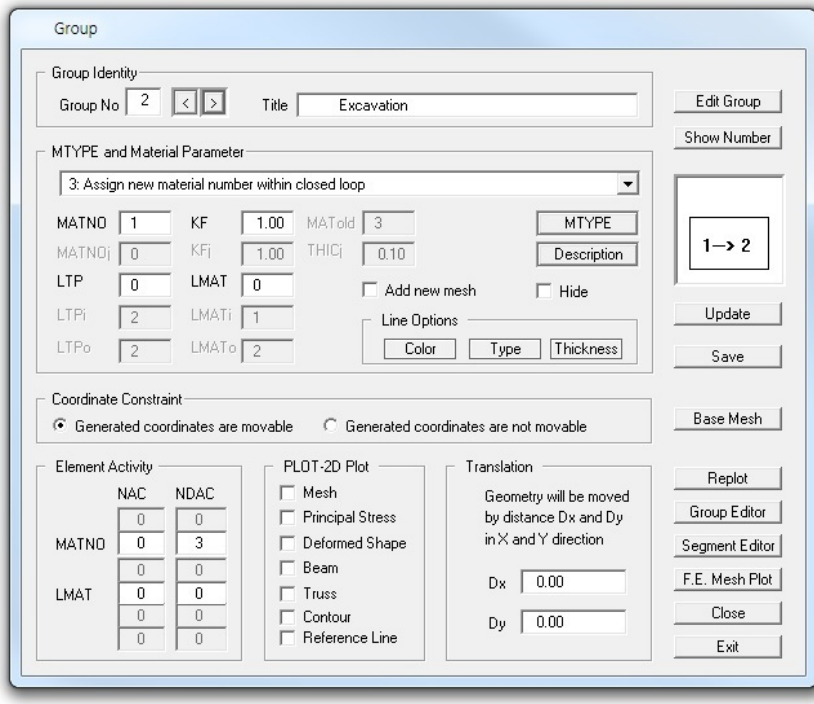


Figure 5.72 Group dialog for excavation

5.4.3.2 Pipe Construction

Pipe construction consists of bedding, steel pipe, and backfill as shown in Figure 5.67. Table 5.21 lists key parameters of these groups

Group	Name	MTYPE	Add New Mesh	Element	MATNO / LMAT	Element Activity	
						NAC	NDAC
3	Bedding	3	Checked	Cont.	2 / 0	4	999
4	Steel Pipe	2		Beam	0 / 1	5	999
5	Backfill	3	Checked	Cont.	3 / 0	5	999

Group	Seg	Line Segment				Arc Segment						IEND
		Beginning Point		Ending Point		Origin		Radius & Angle				
		X	Y	X	Y	X_o	Y_o	R_X	R_Y	Θ_b	Θ_e	
3	1	-1	-1	1	-1							2
	2	1	-1	1.353	-0.294							2
	3	1.353	-0.294	0.4045	-0.294							2
	4					0	0	0.5	0.5	-36	-144	2
	5	-0.4045	-0.294	-1.353	-0.294							2
	6	-1.353	-0.294	-1	-1							2
4	1					0	0	0.5	0.5	0	360	2
5	1	2	1	-2	1							2
	2	-2	1	-1.353	-0.294							2
	3	-1.353	-0.294	-0.4045	-0.294							2
	4					0	0	0.5	0.5	216	-36	2
	5	0.4045	-0.294	1.353	-0.294							2
	6	1.353	-0.294	2	1							2

Table 5.21 Key parameters for pipe construction

Figure 5.73 shows Group dialog for bedding.
Group dialog for backfill is very similar to this group 3.

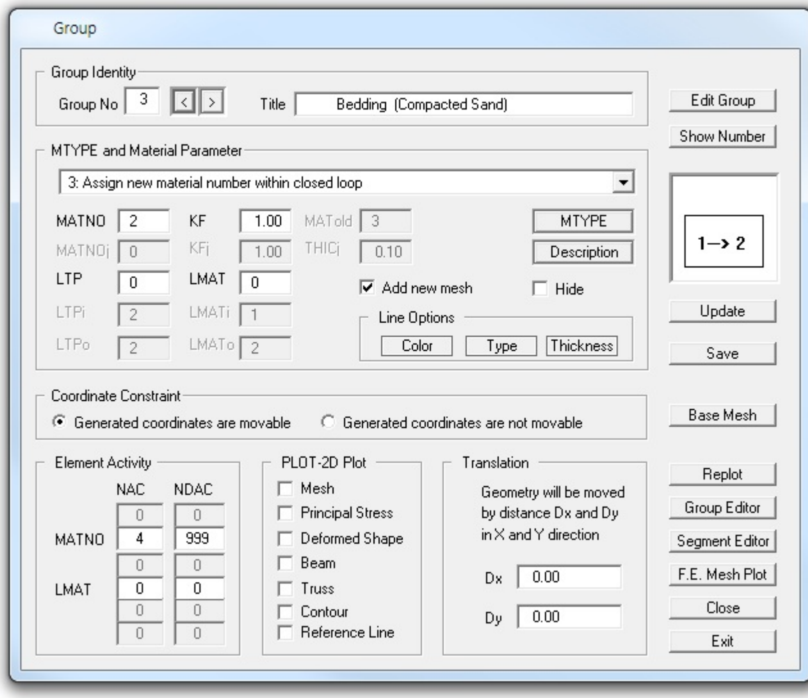


Figure 5.73 Group dialog for bedding

Figure 5.74 shows Group dialog for steel pipe.

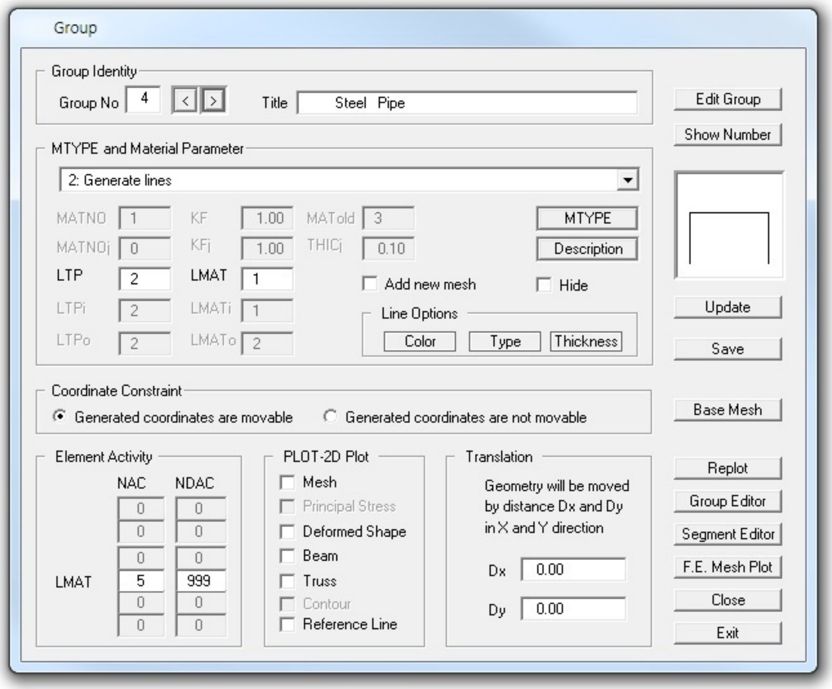


Figure 5.74 Group dialog for steel pipe

5.4.3.3 Lift

Embankment lifts are placed through four steps as shown in Figure 5.67. Table 5.22 lists key parameters of these groups

Group	Name	MTYPE	Element	MATNO / NAC / NDAC	Seg.	Beginning Point		Ending Point		IEND
						X	Y	X	Y	
6	Lift-1	3	Cont	4 / 6 / 999	1	-5	1	5	1	2
					2	5	1	5	2	2
					3	5	2	-5	2	2
					4	-5	2	-5	1	2
7	Lift-2	3	Cont	5 / 7 / 999	1	-5	2	5	2	2
					2	5	2	5	4	2
					3	5	4	-5	4	2
					4	-5	4	-5	2	2
8	Lift-3	3	Cont	6 / 8 / 999	1	-5	4	5	4	2
					2	5	4	5	6	2
					3	5	6	-5	6	2
					4	-5	6	-5	4	2
9	Lift-4	3	Cont	7 / 9 / 999	1	-5	6	5	6	2
					2	5	6	5	7	2
					3	5	7	-5	7	2
					4	-5	7	-5	6	2

Table 5.22 Key parameters for lift

Figure 5.75 shows Group dialog for the first lift.
Group dialogs for other lifts are very similar to this group 6.

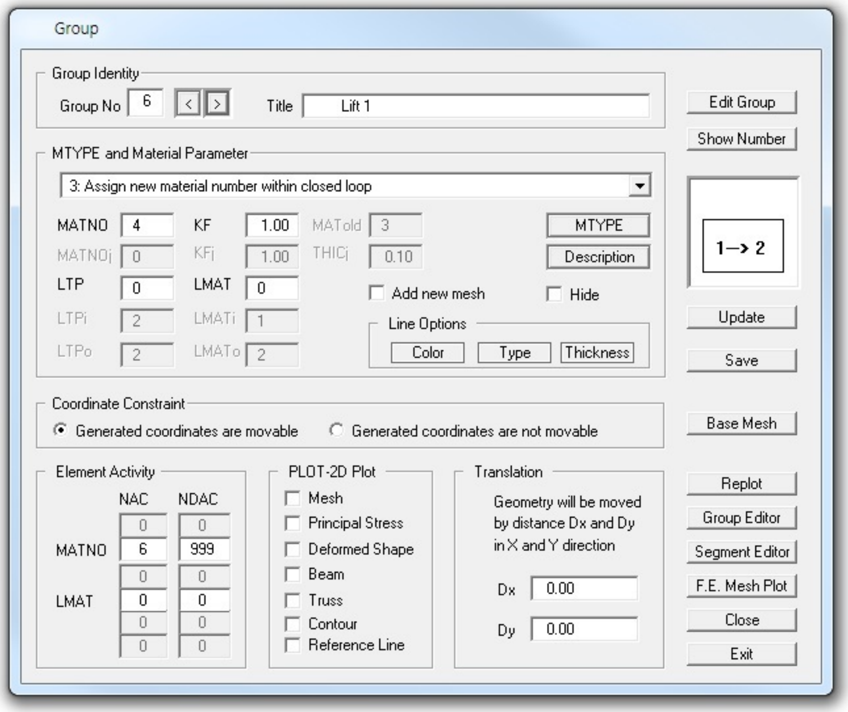


Figure 5.75 Group dialog for first lift

5.4.4 Finite Element Mesh Plot

Figure 5.76 shows finite element meshes generated from group meshes. Finite element meshes near buried pipe are shown in Figure 5.77.

Figure 5.76
Finite element meshes

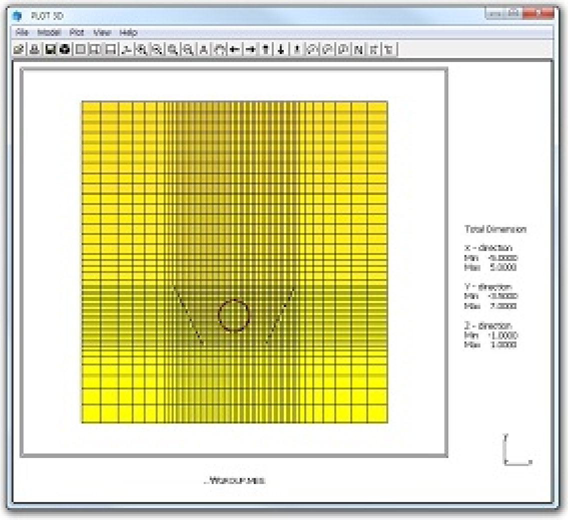
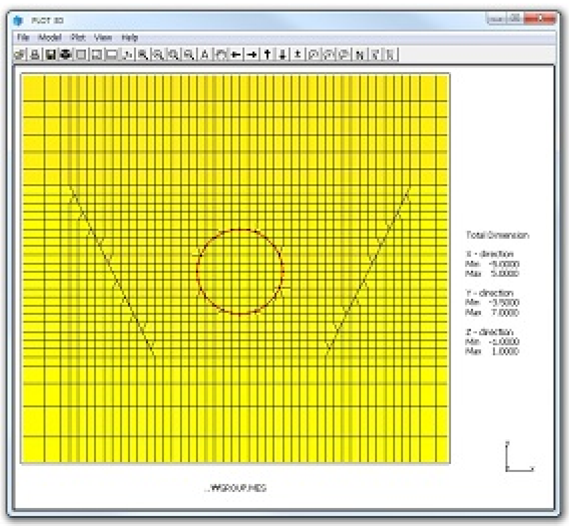


Figure 5.77
Finite element meshes near buried pipe



5.5 Arch Warehouse

This example illustrates how to build group meshes for typical arch warehouse structure.

5.5.1 Overview

The cross section of this arch warehouse consists of soil layer, foundations, and arch frame as shown in Figure 5.78.
Construction sequence is listed in Table 5.23.

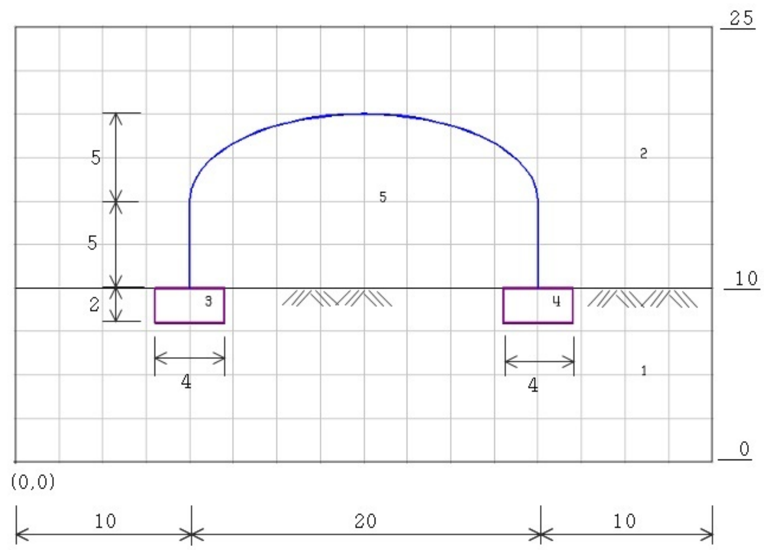


Figure 5.78 Schematic section of arch warehouse

Step	Description
1,2	In situ stress
3	Excavate trench & place foundation
4	Construct steel arch frame

Table 5.23 Construction sequence

A total of 5 groups are used to model this arch warehouse as schematically shown in Figure 5.79: 1 for soil layer, 1 for above ground, 2 for foundations, and 1 for arch frame. Table 5.24 summarizes key parameters of groups.

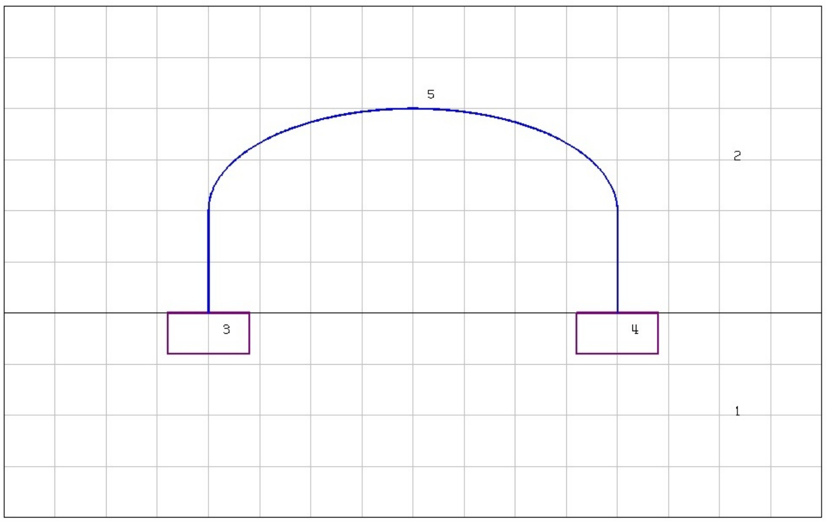


Figure 5.79 Group section view

Group	Name		MTYPE	NAC / NDAC	MAT_{OLD} / MATNO / LTP / LMAT / IEND
1	Soil Layer		3	0 / 0	1 / 0 / 0 / 0 / 2
2	Above Ground		1	0 / 0	0 / 0 / 0 / 0 / 0
3	Left Foundation	MAT_{OLD}	4	0 / 3	2 / 3 / 0 / 0 / 2
		MATNO		3 / 999	
4	Right Foundation	MAT_{OLD}	4	0 / 3	2 / 3 / 0 / 0 / 2
		MATNO		3 / 999	
5	Arch Frame		2	4 / 999	0 / 0 / 2 / 1 / 2 (Checked Add new mesh)

Table 5.24 Group key parameters

5.5.2 Base Mesh

Built-in Base Mesh dialog is shown in Figure 5.80 with input data for blocks and boundary condition. Figure 5.81 shows base mesh plot on drawing board.

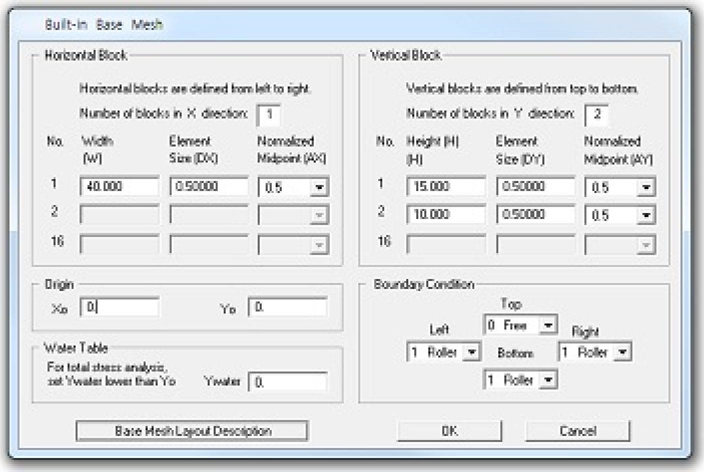


Figure 5.80 Built-in base mesh dialog

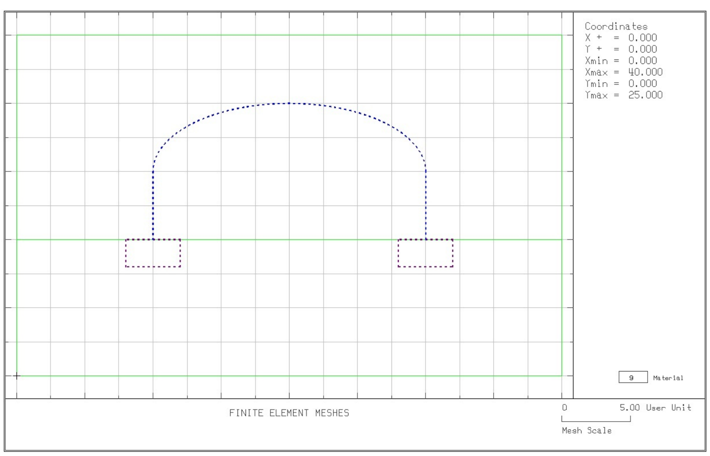


Figure 5.81 Base mesh plot on drawing board

5.5.3 Groups

Group meshes are divided into three parts:

- Soil layer and above ground
- Foundation
- Arch frame

It should be noted that the final finite element meshes are most influenced by group order and IEND.

5.5.3.1 Soil Layer and Above Ground

Above Ground represents upper block of base mesh which will vanish. Table 5.25 lists key parameters of these groups

Group	Name	MTYPE	Elem	MATNO / NAC / NDAC	Seg.	Beginning Point		Ending Point		IEND
						X	Y	X	Y	
1	Soil Layer	3	Cont	1 / 0 / 0	1	0	0	40	0	2
					2	40	0	40	10	2
					3	40	10	0	10	2
					4	0	10	0	0	2
2	Above Ground	1	Cont	0 / 0 / 0	1	0	10	40	10	2
					2	40	10	40	25	2
					3	40	25	0	25	2
					4	0	25	0	10	2

Table 5.25 Key parameters for soil layer and above ground

Figure 5.82 shows Group dialog for soil layer.

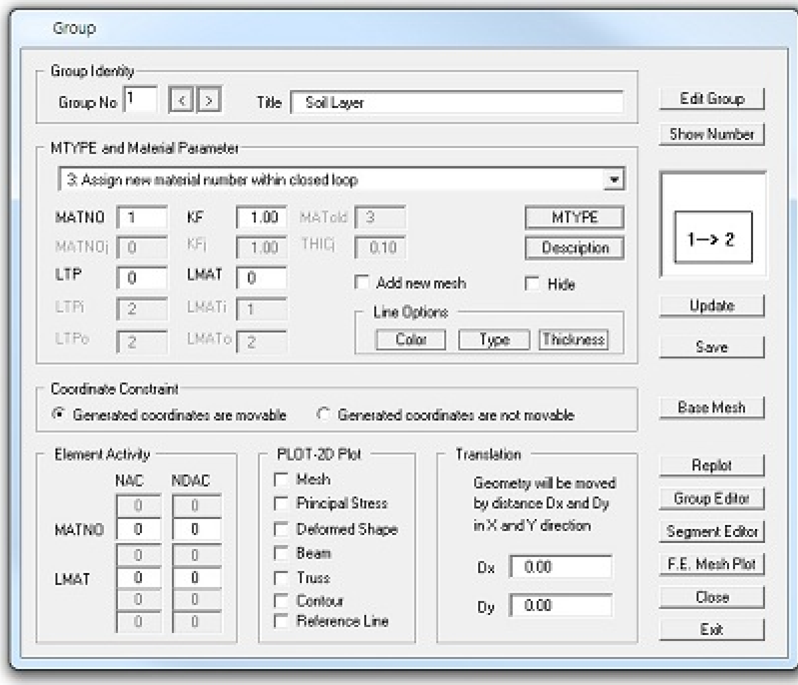


Figure 5.82 Group dialog for soil layer

Figure 5.83 shows Group dialog for above ground.

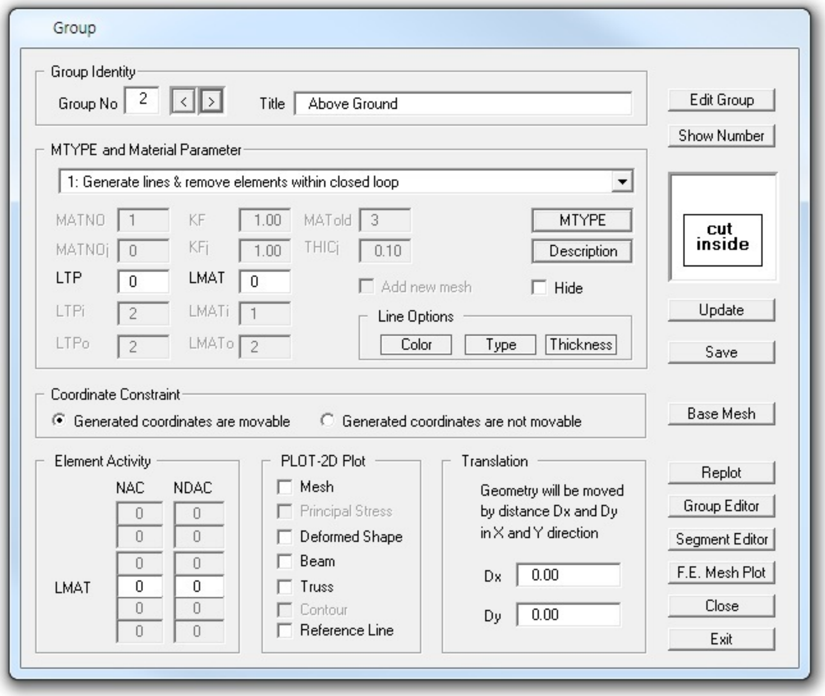


Figure 5.83 Group dialog for above ground

5.5.3.2 Foundation

Each foundation group includes both in situ soils and concrete block such that in situ soils are replaced by concrete block when foundation is built. Table 5.26 lists key parameters of these groups.

Group	Name		NAC / NDAC	MTYPE Elem	Seg.	Beginning Point		Ending Point		IEND
						X	Y	X	Y	
3	Left Foundation	MAT_{OLD}=2	0 / 3	4 Cont	1	8	8	12	8	2
					2	12	8	12	10	2
		MATNO=3	3 / 999		3	12	10	8	10	2
					4	8	10	8	8	2
4	Right Foundation	MAT_{OLD}=2	0 / 3	4 Cont	1	28	8	32	8	2
					2	32	8	32	10	2
		MATNO=3	3 / 999		3	32	10	28	10	2
					4	28	10	28	8	2

Table 5.26 Key parameters for foundation

Figure 5.84 shows Group dialog for left foundation.
Group dialog for right foundation is very similar to this group 3.

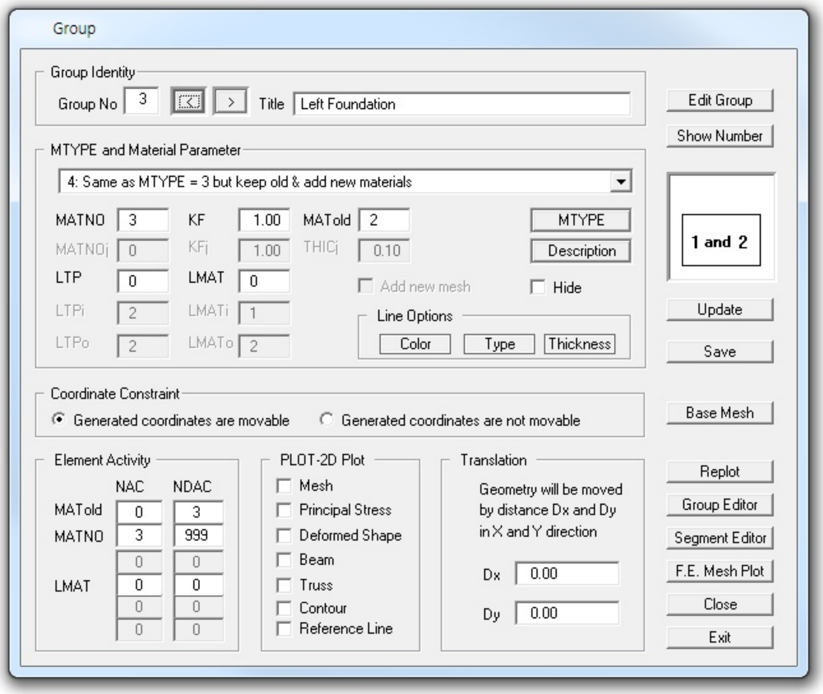


Figure 5.84 Group dialog for left foundation

5.5.3.3 Arch Frame

Arch Frame is the only structure in the upper block of base mesh since the Above Ground group generates void space. Table 5.27 lists key parameters of this group.

Group	Name	MTYPE	Element	LTP / LMAT	Element Activity	
					NAC	NDAC
5	Arch Frame	2	Beam	2 / 1	4	999

Group	Seg	Line Segment				Arc Segment						NDIV	IEND
		Begin. Pt.		Ending Pt.		Origin		Radius & Angle					
		X	Y	X	Y	X_o	Y_o	R_X	R_Y	Θ_b	Θ_e		
5	1	30	10	30	15							5	2
	2					20	15	10	5	0	180	20	2
	3	10	15	10	10							5	2

Table 5.27 Key parameters for arch frame

Figure 5.85 shows Group dialog for the arch frame. It should be noted that Add new mesh be checked.

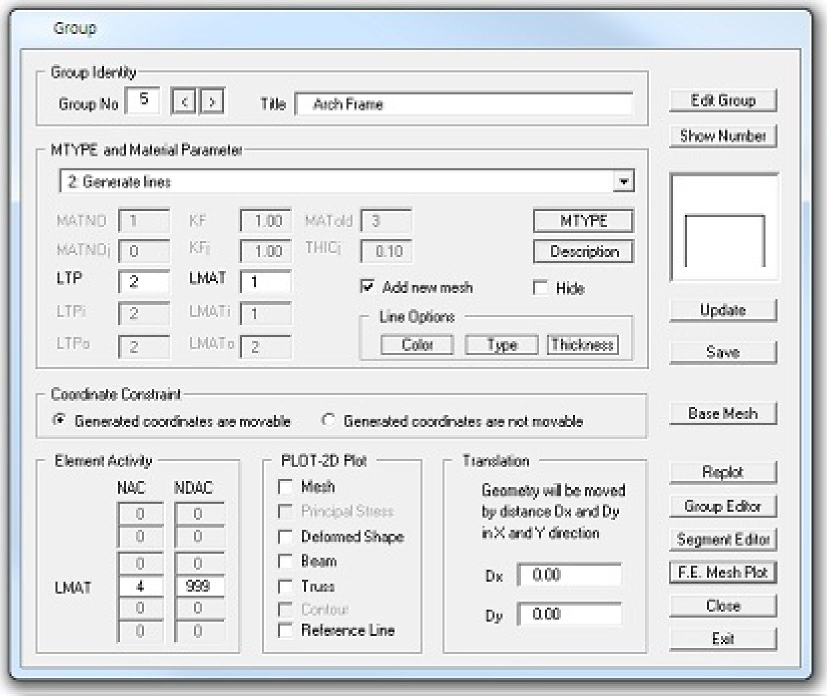


Figure 5.85 Group dialog for arch frame

5.5.4 Finite Element Mesh Plot

Figure 5.86 shows finite element meshes generated from group meshes.

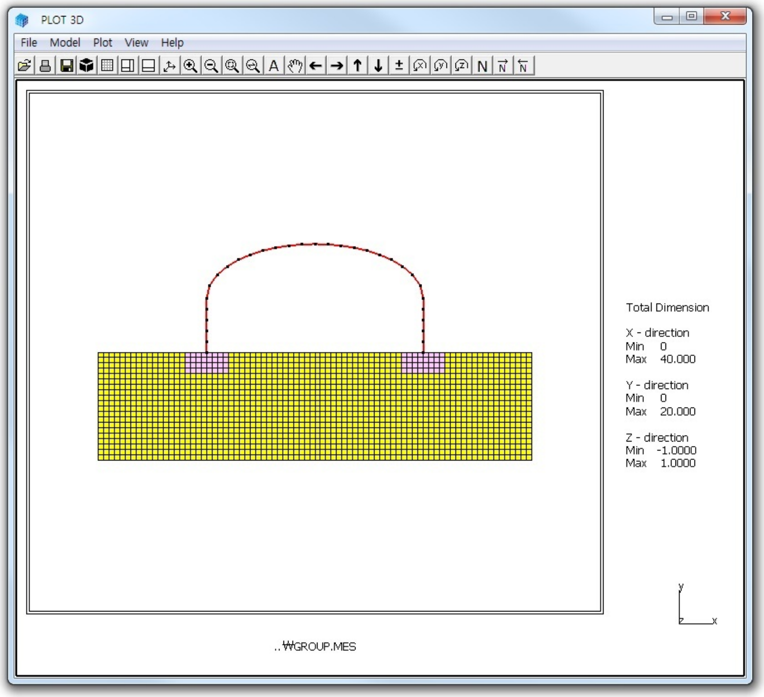


Figure 5.86 Finite element meshes

5.6 Finite Element Mesh Modification

This example illustrates how to modify existing finite element meshes using Mesh Generator.

5.6.1 Overview

When you open input file, Mesh Generator reads the extension of the input file name and it assumes that the input file is the finite element mesh file if the extension is .Mes.

Editing finite element meshes has three parts: Nodal Boundary, Nodal Coordinate and Element Material. These editing modes can be accessed from Mesh menu in PLOT-2D as shown in Figure 5.87.

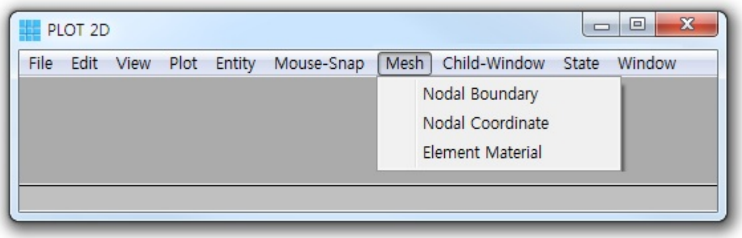


Figure 5.87 Menu for editing finite element mesh

It should be noted that once you edited the finite element meshes, modified finite element mesh is saved as MeshFile.Mes in the current working directory. The original input mesh file is not changed.

Figure 5.88 shows existing finite element mesh with six layers of natural soils. The top layer of this existing mesh is to be replaced by sand embankment with reduced width as schematically shown in Figure 5.89.

This modification involves following three works:

- Change top surface nodal coordinates
- Change top surface nodal boundaries
- Change top layer element materials

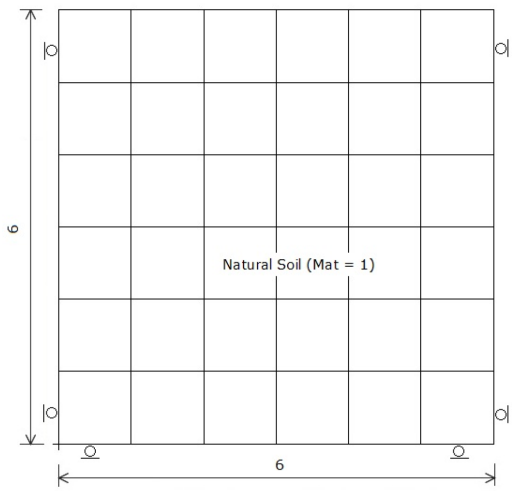


Figure 5.88 Existing finite element mesh

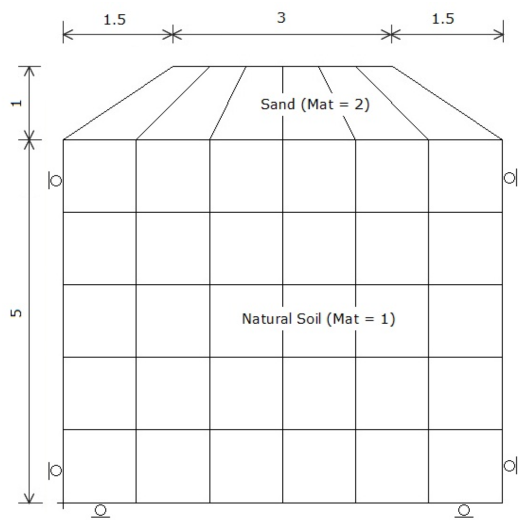


Figure 5.89 Modified finite element mesh

5.6.2 Change Top Surface Nodal Coordinates

Click Nodal Coordinate from the Mesh menu, then Edit Coordinate dialog in Figure 5.90 is displayed.

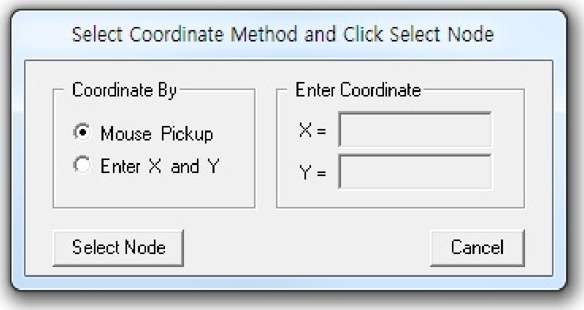


Figure 5.90 Edit coordinate dialog

For this example, Snap to Half of Grid in Figure 5.91 is the most convenient method for Mouse Pickup.

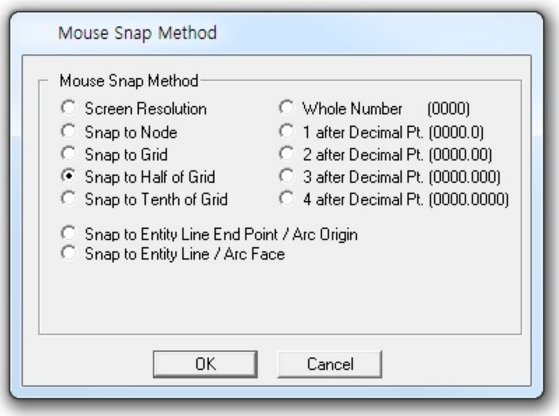


Figure 5.91 Mouse snap method

Click Select Node in Figure 5.90.
When you select the node by Mouse Right Click, the selected node is marked as an open circle on the drawing board as in Figure 5.92.

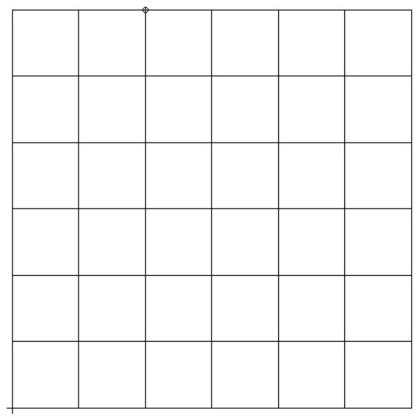


Figure 5.92 First selected node on drawing board

Now, move the first selected node by using drag-and-drop of Mouse Left Button as shown in Figure 5.93.

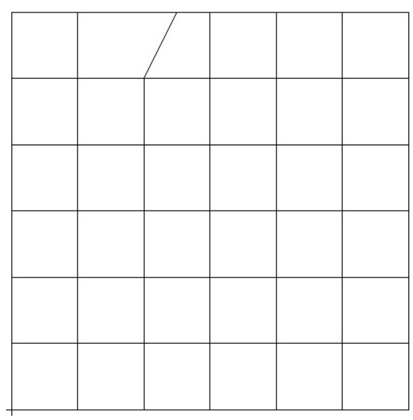


Figure 5.93 New position of first selected node

Select the next node by Mouse Right Click as shown in Figure 5.94.

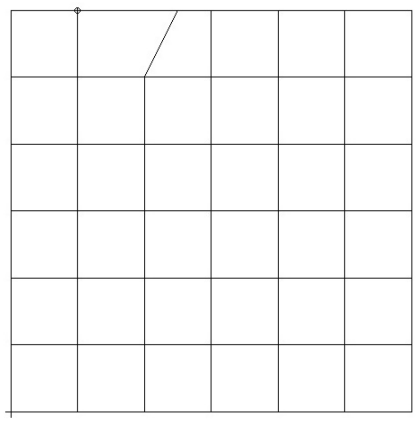


Figure 5.94 Second selected node on drawing board

Now, move the second selected node by using drag-and-drop of Mouse Left Button as shown in Figure 5.95.

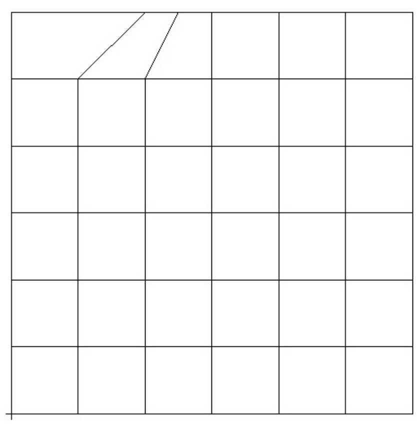


Figure 5.95 New position of second selected node

Repeat the same procedure for all other nodes on the top surface. Once finished, click Finish button in Figure 5.96.

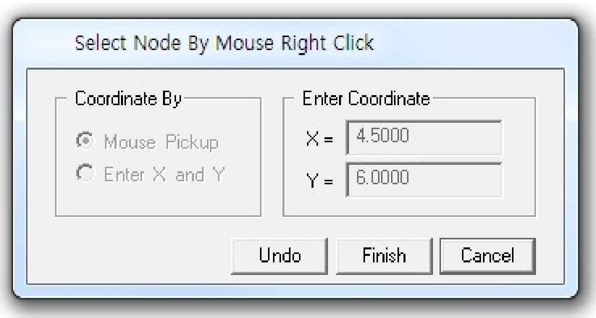


Figure 5.96 Edit coordinate dialog

Figure 5.97 shows final finite element mesh on the drawing board.

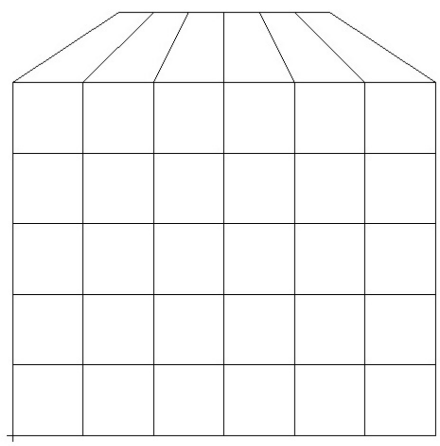
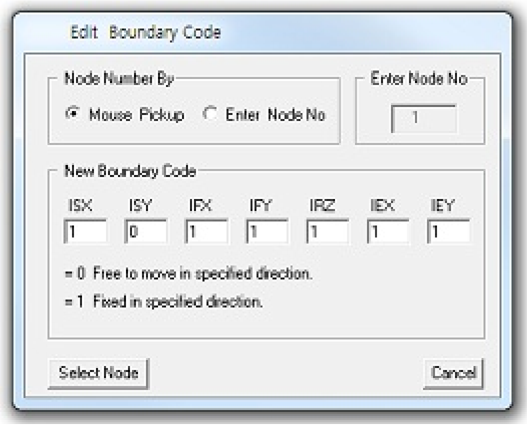


Figure 5.97 Final finite element mesh

5.6.3 Change Top Surface Nodal Boundaries

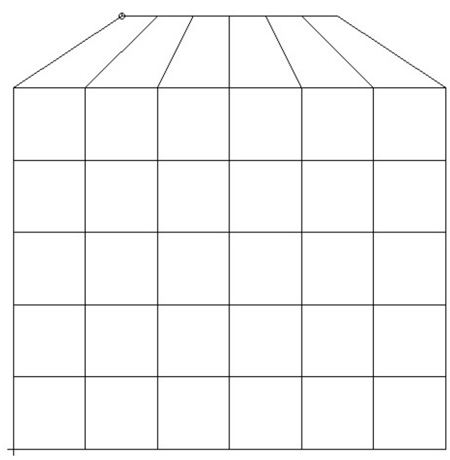
Click Nodal Boundary from the Mesh menu, then Edit Boundary Code dialog in Figure 5.98 is displayed.

Figure 5.98
Edit boundary dialog



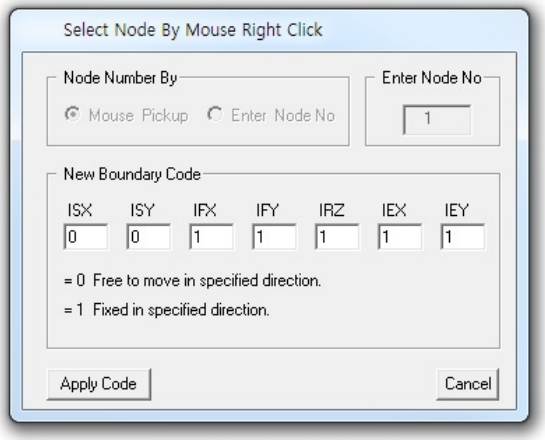
Click Select Node in Figure 5.98.
When you select the node by Mouse Right Click, the selected node is marked as an open circle on the drawing board as in Figure 5.99.

Figure 5.99
Selected node on drawing board



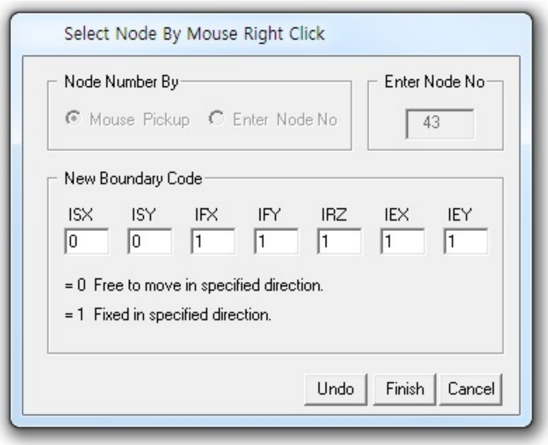
Change the boundary codes as in Figure 5.100 so that the top left node can be free to move in both horizontal and vertical directions and then click Apply Code button.

Figure 5.100
Modified boundary code
for top left node



In the same way, select the top right node, modify boundary codes, and click Apply Code. Since all boundary codes are modified, click Finish button in Figure 5.101.

Figure 5.101
Modified boundary code
for top right node



Click General View from the View menu. Select Skeleton Boundary Code in General View Options dialog as shown in Figure 5.102 and then click OK button. Modified skeleton boundary codes are shown in Figure 5.103.

Figure 5.102
General view
for skeleton boundary code

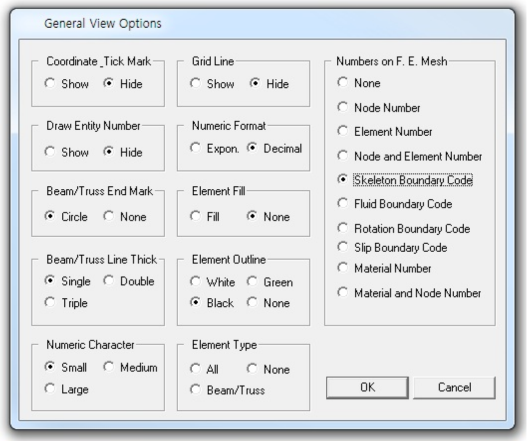
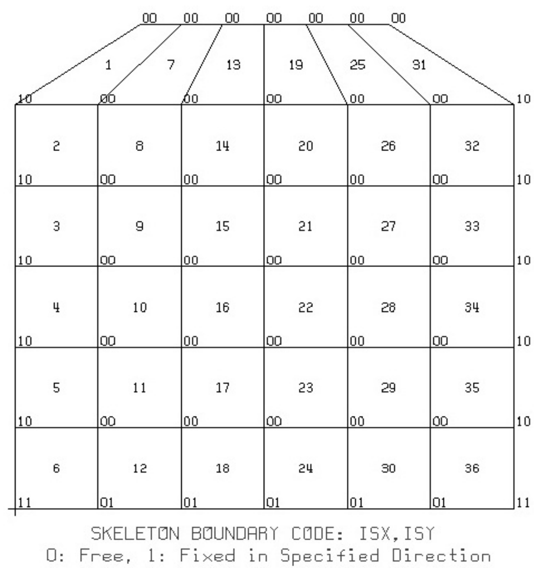


Figure 5.103
Modified skeleton
boundary code plot

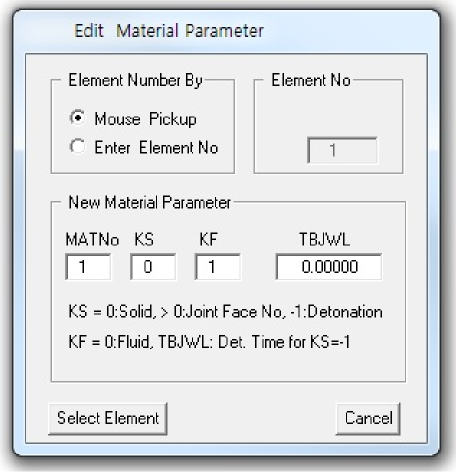


5.6.4 Change Top Layer Element Materials

Click Element Material from the Mesh menu, then Edit Material Parameter dialog in Figure 5.104 is displayed.

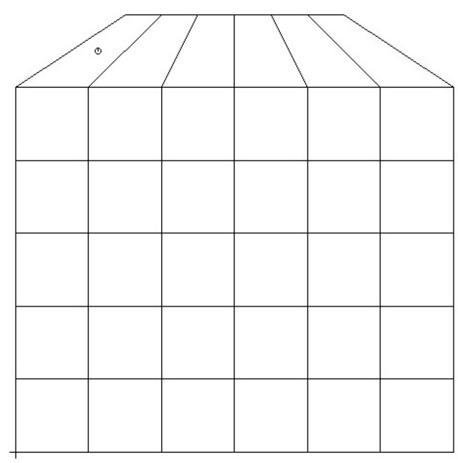
Figure 5.104

Edit element material dialog



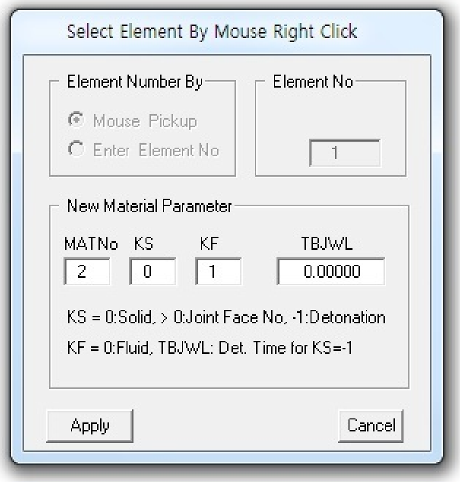
Click Select Element button.
Click the element on the top layer by Mouse Right Click.
Selected element is marked as an open circle as shown in Figure 5.105.

Figure 5.105
Selected element on drawing board



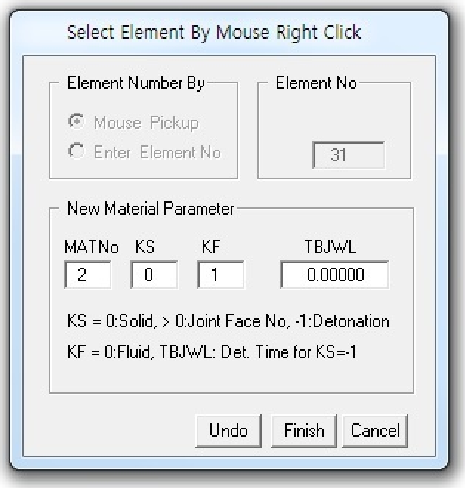
Change the material number as shown in Figure 5.106 and then click Apply button.

Figure 5.106
Modified material number for element 1



Repeat the same procedure for the other elements on the top layer. Once finished, click Finish button in Figure 5.107.

Figure 5.107
Modified material number for element 31



Click General View from the View menu. Select Material Number in General View Options dialog as shown in Figure 5.108 and then click OK button. Modified material number is shown in Figure 5.109.

Figure 5.108
General view
for material number

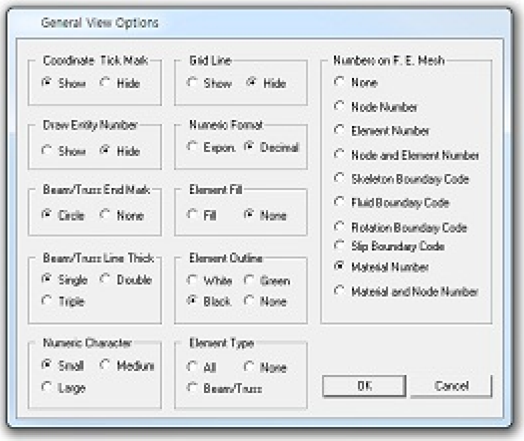
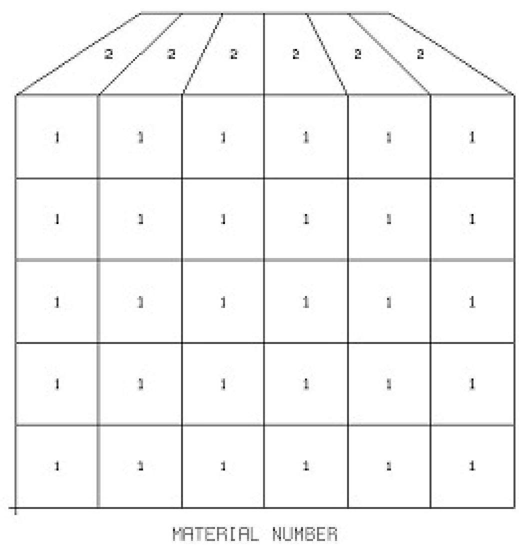


Figure 5.109
Modified material number
plot



Block Mesh
Example Problem

Block Mesh Generator is a three-dimensional CAD program specially designed to build block mesh which can be used to generate finite element mesh with the aid of program PRESMAP-GP. Block Mesh User's Manual describes all the basic functions associated with block mesh generation and modifications.

Five example problems are presented:

1. Single Element
   Shows step by step procedure to create block mesh.
2. Cube Foundation
   Builds block mesh for cube foundation.
3. Hemispherical Shell
   Builds block mesh for hemispherical shell subjected to concentrated loads.
4. Horseshoe Tunnel
   Builds block mesh for typical horseshoe tunnel with reinforced concrete lining.
5. Space Truss
   Builds directly finite element mesh for space truss.

6.1 Single Element

The main objective of this first example is to show the step by step procedure to create block mesh.

This example is to build single cube element in Figure 6.1 by using block mesh generator. This single element is subjected to undrained uniaxial strain loading.

This example involves following seven main steps:

1. Access block mesh generator
2. Set work plane
3. Build cube entity
4. Build hexahedron block
5. Edit block boundary code
6. View skeleton boundary code
7. Plot finite element mesh

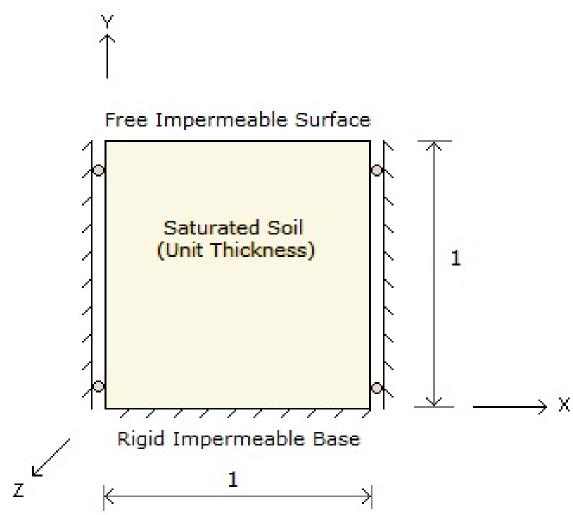


Figure 6.1 Single element in uniaxial strain condition

Step 1: Access Block Mesh Generator (New)

Access Block Mesh Generator by following menu items in SMAP
Run → Mesh Generator → Block Mesh → New

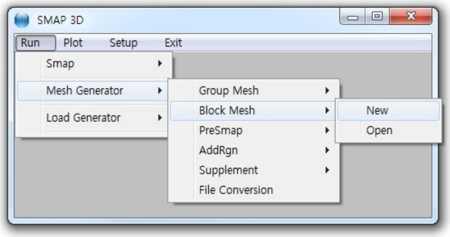


Figure 6.2 Accessing block mesh generator

Step 2: Set Work Plane

Prebuilt Work Plane is displayed on drawing board along with Work Plane Editor dialog. Modify NDx and Wx in Figure 6.3 and click Update.

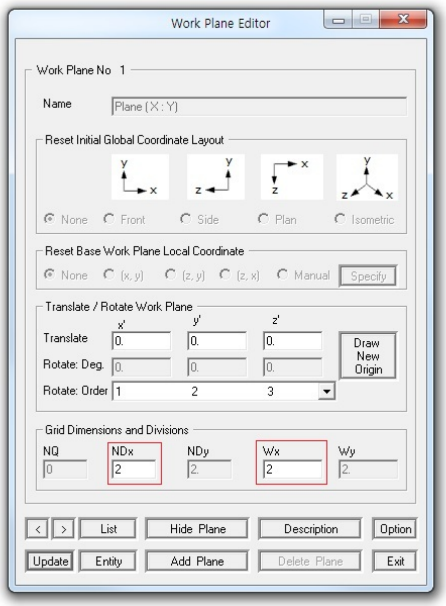


Figure 6.3 Work plane editor

Step 3: Build Cube Entity

1. Click Entity button in Figure 6.3.
2. Entity Editor dialog is displayed as in Figure 6.4.

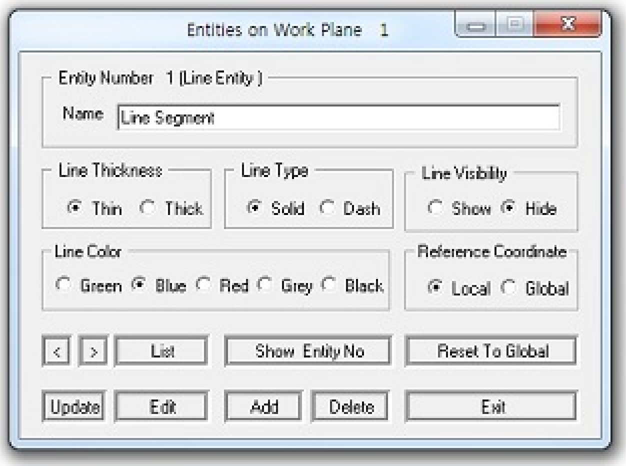


Figure 6.4 Entity editor

3. Click Add button in Figure 6.4.
4. Select Cube entity and click OK button in Figure 6.5.

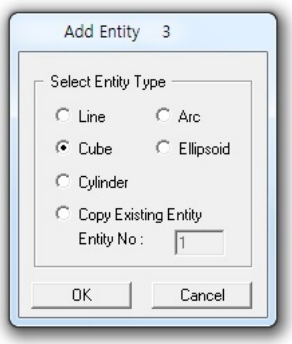


Figure 6.5 Entity type selection

5. Modify input fields of Lx, Ly, and Lz as shown in Figure 6.6.
6. Click Draw Cube Entity button.

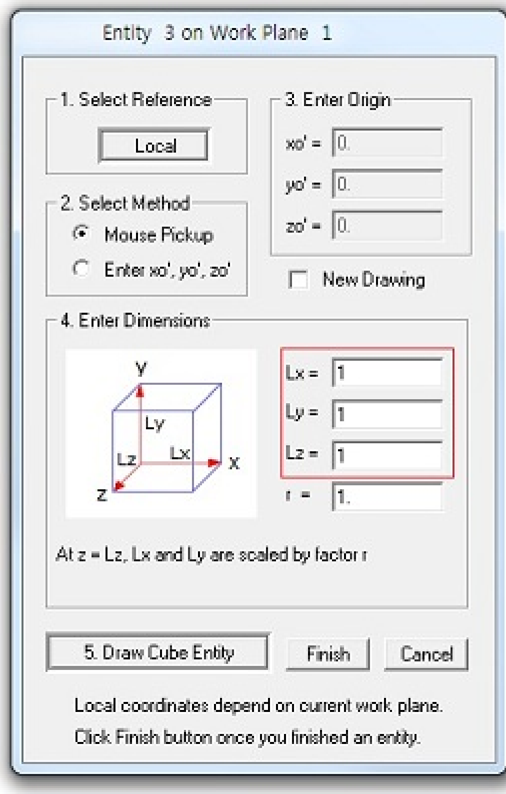


Figure 6.6 Cube entity

7. Coordinates on Work Plane dialog is displayed as in Figure 6.7.
8. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.8.

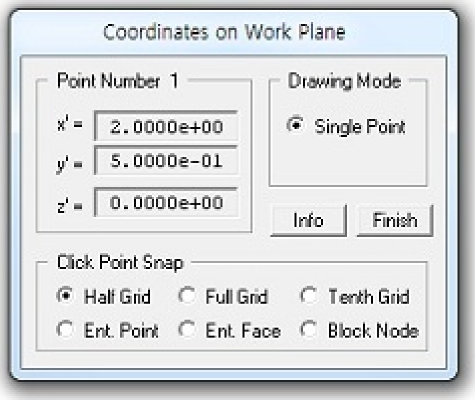


Figure 6.7 Coordinates on work plane

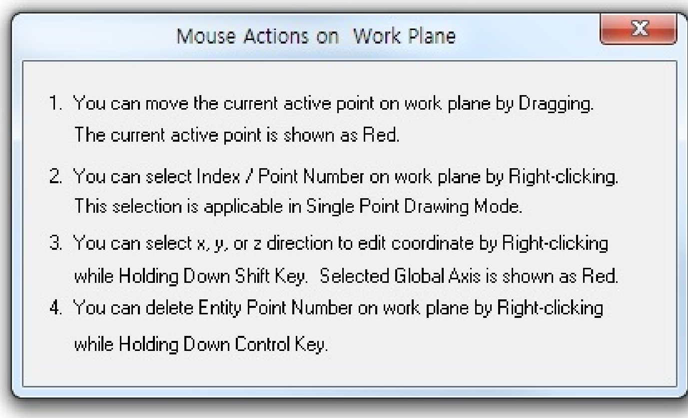


Figure 6.8 Mouse actions on work plane

9. Click Axis toolbar as shown in Figure 6.9.

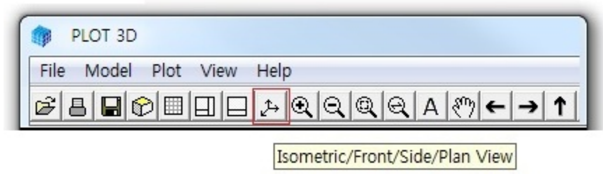


Figure 6.9 Axis toolbar

10. Click Mouse at the origin of coordinates.
11. Cube entity is shown on isometric work plane in Figure 6.10.

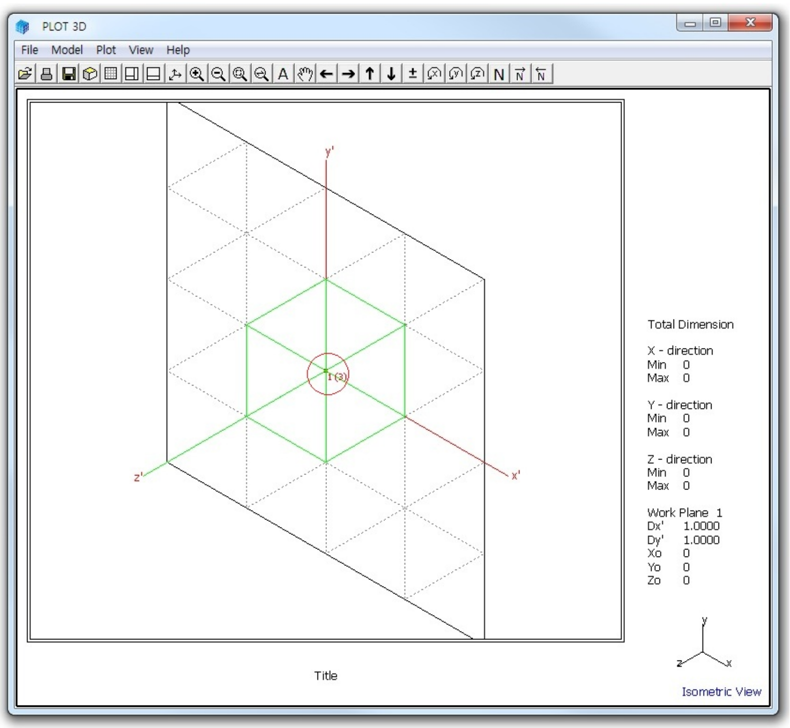


Figure 6.10 Cube entity on isometric work plane

12. Click Finish in Figure 6.7.
13. Click Finish in Figure 6.6.
14. Select Global for Reference Coordinate in Figure 6.11.
15. Click Reset To Global and then Exit buttons in Figure 6.11.

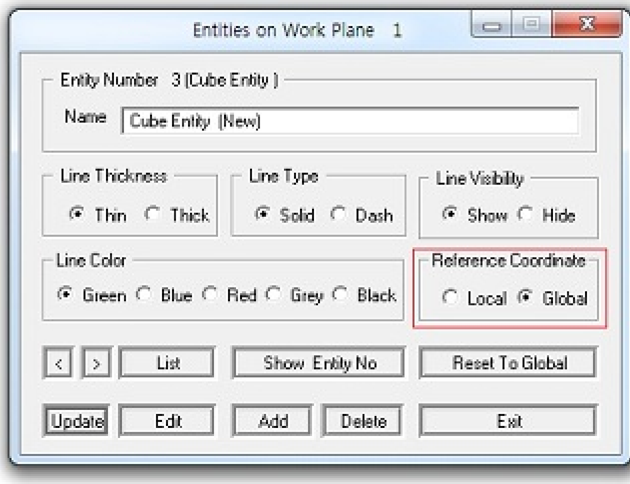


Figure 6.11 Entity editor

Step 4: Build Hexahedron Block

1. Click Block Editor toolbar in Figure 6.12.

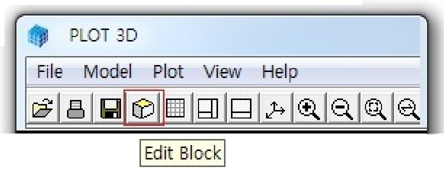


Figure 6.12 Block editor toolbar

2. Select Hexa for block type and click OK in Figure 6.13.

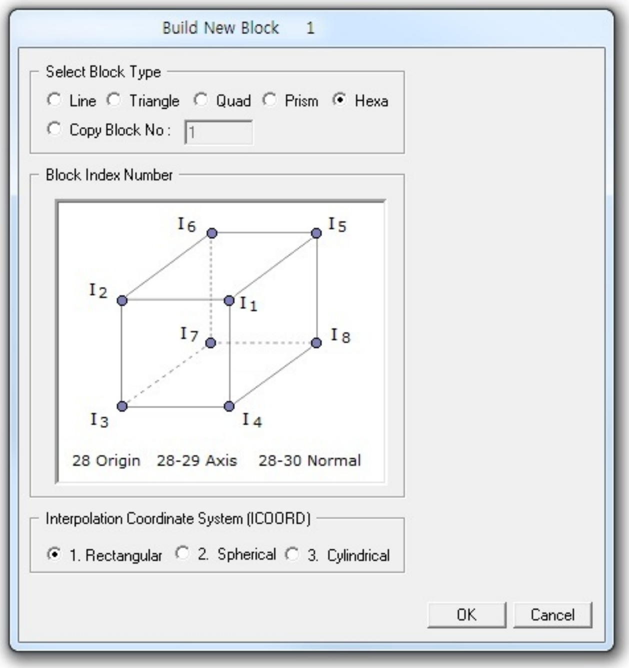


Figure 6.13 Block type selection

3. Click Draw Index Number in Figure 6.14.
4. Coordinates on Work Plane dialog is displayed in Figure 6.15.

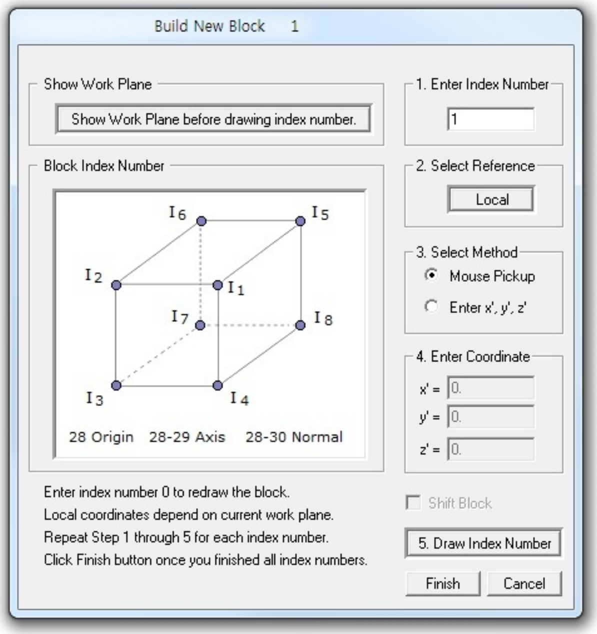


Figure 6.14 Hexa block

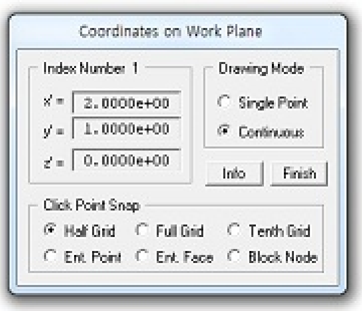


Figure 6.15 Coordinates on work plane

5. Translate work plane as in Figure 6.16 and click Update button.

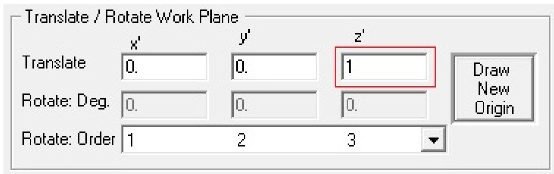


Figure 6.16 Work plane translation (z' = 1)

6. Click the points for index numbers on front surface as in Fig. 6.17.

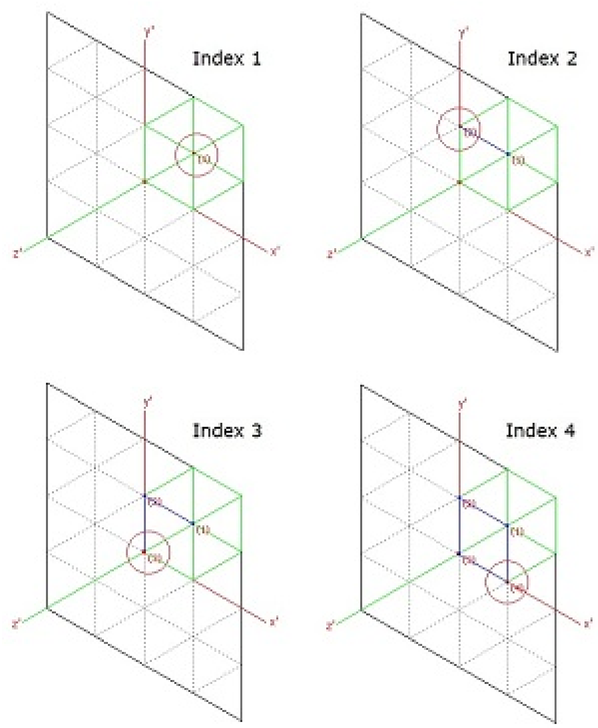


Figure 6.17 Index numbers on front surface

7. Translate work plane as in Figure 6.18 and click Update button.

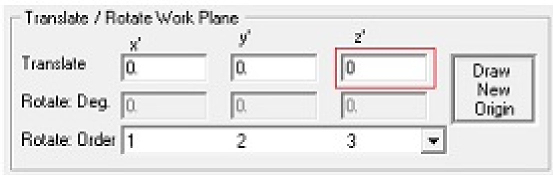


Figure 6.18 Work plane translation (z' = 0)

8. Click the points for index numbers on back surface as in Figure 6.19.

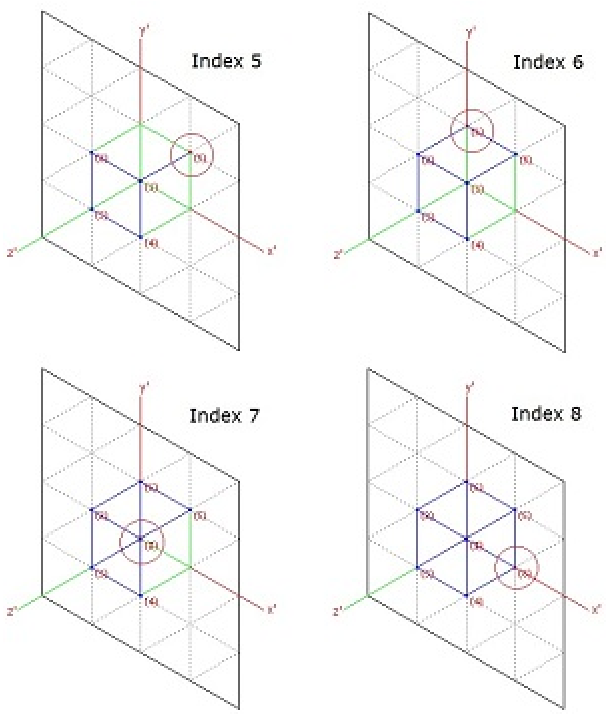


Figure 6.19 Index numbers on back surface

Now, the geometry of hexahedron block is completed.

9. Click Finish in Figure 6.20 and then click Finish in Figure 6.14.

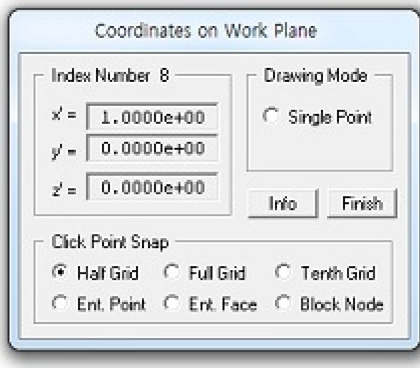


Figure 6.20 Coordinates on work plane

10. Get back to Work Plane Editor dialog and click Entity.

11. Select Entity Number 3, Hide for line visibility, click Update, and click Exit in Figure 6.21.

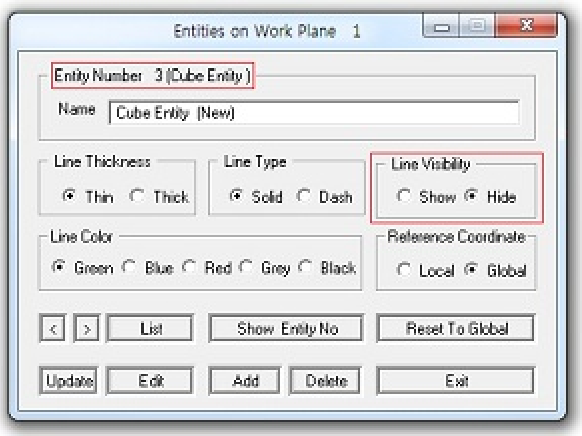


Figure 6.21 Entity editor

12. Modify Title and Material & Element Generation Parameters in Block Editor as shown in Figure 6.22.
13. Click Save and type in file name as EX1.

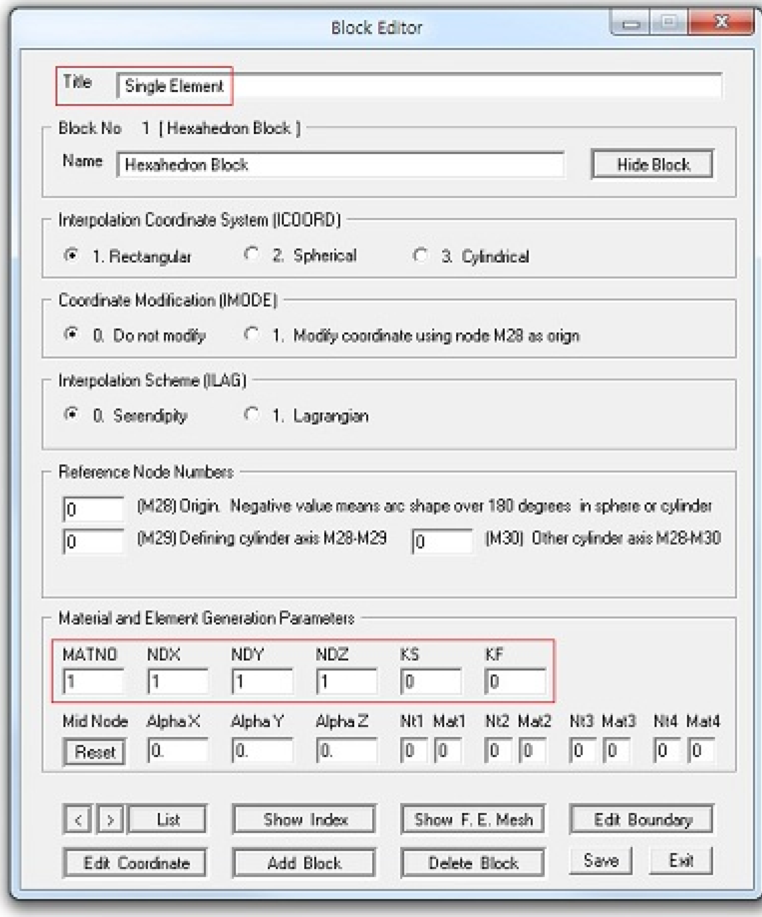


Figure 6.22 Block editor

Step 5: Edit Block Boundary Code

1. Click Edit Boundary in Figure 6.22.
2. Set the boundary codes as shown in Figure 6.23.
3. Click IBTYPE button to see description of boundary type in Fig. 6.24.
4. Click Update and then OK buttons.

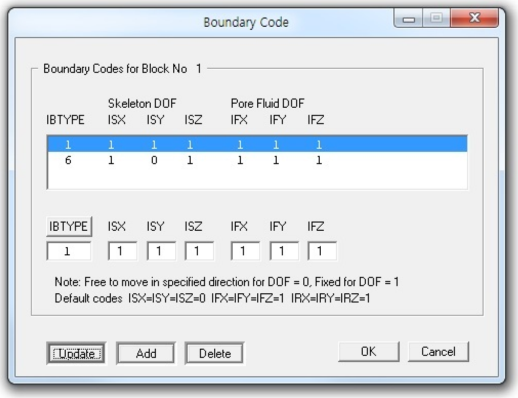


Figure 6.23 Boundary code editor

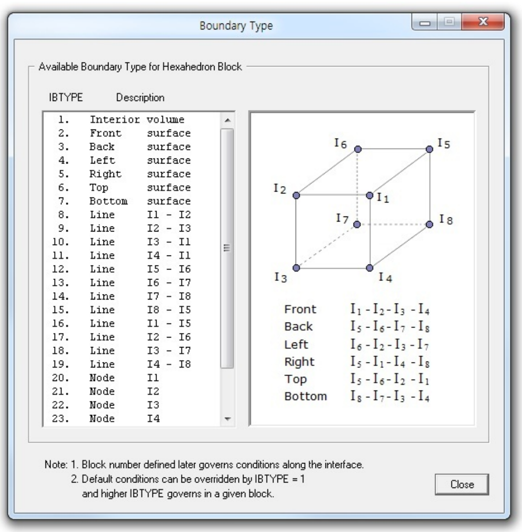


Figure 6.24 Boundary type for hexa block

Step 6: View Skeleton Boundary Code

1. Select View → General in PLOT-3D menu.
2. Select Skeleton Boundary Code and click OK in Figure 6.25.
3. Click Save in Figure 6.22.

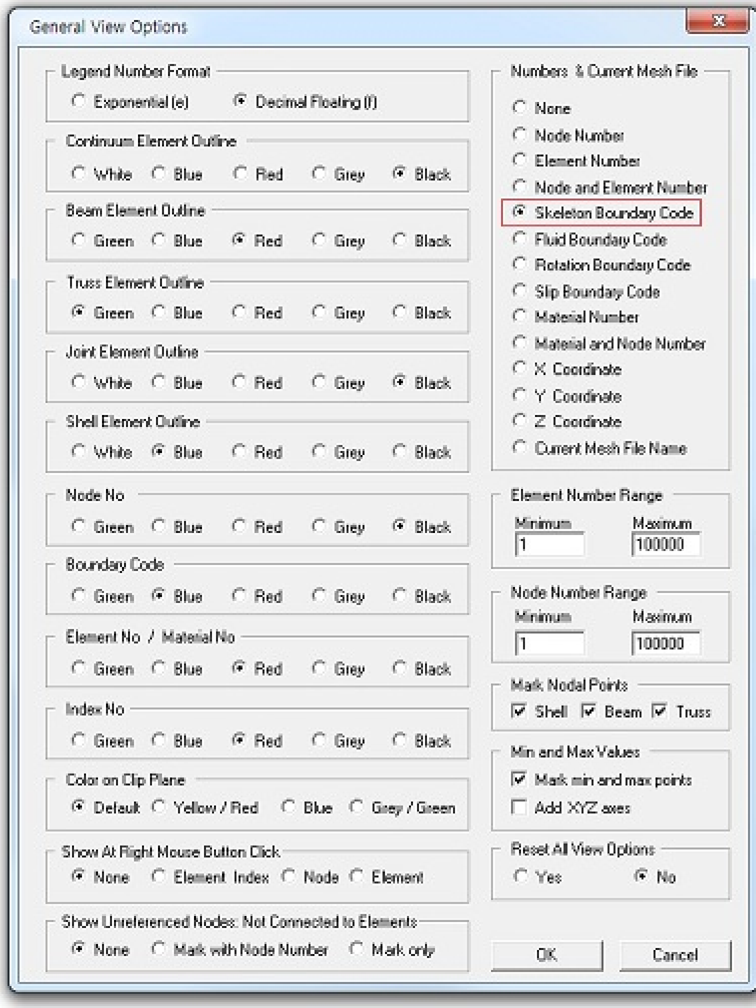


Figure 6.25 General view options

4. Click Show Numbers toolbar as shown in Figure 6.26.

Figure 6.26 Show numbers toolbar

5. Skeleton boundary codes are shown in Figure 6.27.

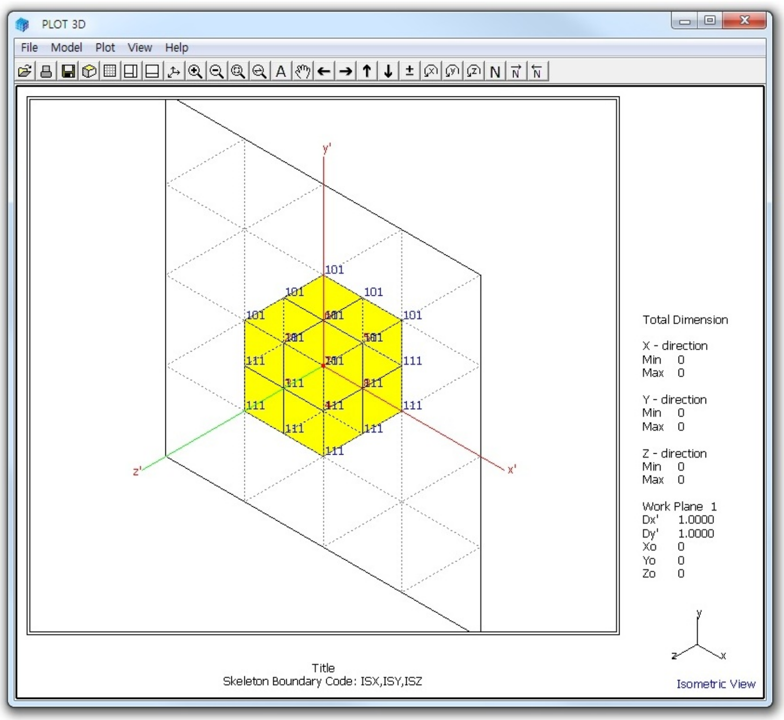


Figure 6.27 Skeleton boundary codes on drawing board

Step 7: Plot Finite Element Mesh

1. Click Show F. E. Mesh in Figure 6.22.
2. Rotate the finite element mesh as shown in Figure 6.28.

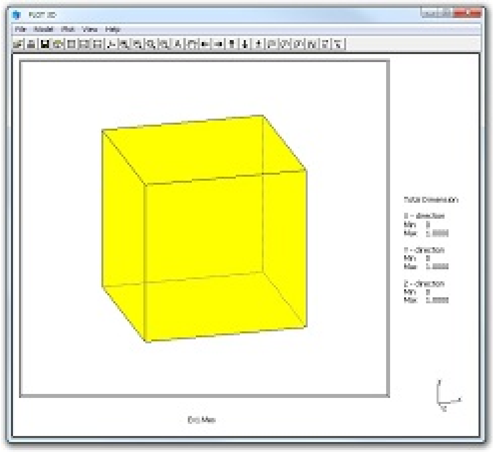


Figure 6.28 Finite element mesh

3. Follow same procedure to plot skeleton boundary codes in Step 6.
4. Figure 6.29 shows skeleton boundary codes for finite element mesh.

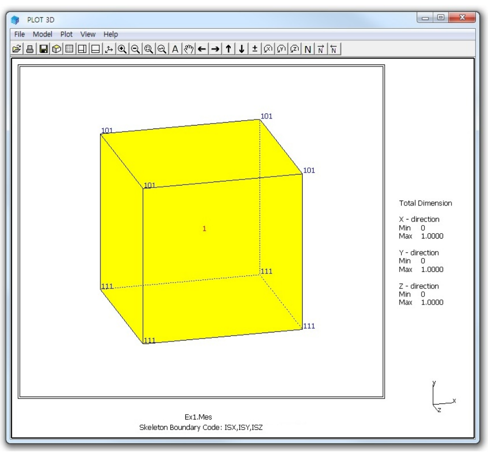


Figure 6.29 Skeleton boundary codes

6.2 Cube Foundation

This example illustrates how to build block mesh for cube foundation. Cube foundation has the dimensions of 100 x 100 x 100 units with all roller boundaries except free on top surface.

This example has the following two parts:

Part 1: Creating Cube Foundation (Figure 6.30)

- Access block mesh generator (New)
- Set work plane
- Build hexahedron block
- Edit block boundary
- Set global boundary
- View skeleton boundary code
- Plot finite element mesh

Part 2: Modifying Cube Foundation (Figure 6.31)

- Access block mesh generator (Open)
- Modify element generation parameters
- Plot finite element mesh

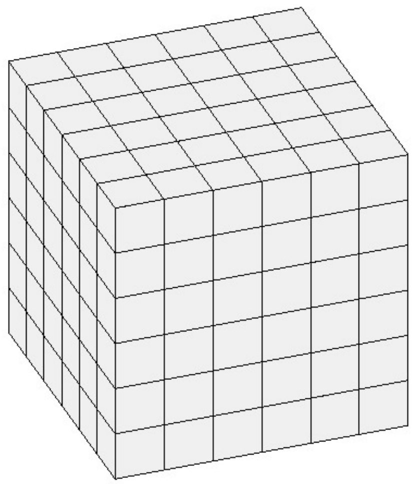


Figure 6.30 Cube foundation with constant element size

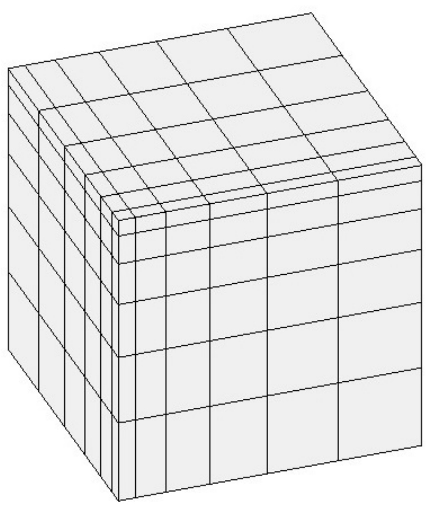


Figure 6.31 Cube foundation with variable element size

6.2.1 Part 1: Creating Cube Foundation

Part 1 consists of the following seven main steps:

1. Access block mesh generator (New)
2. Set work plane
3. Build hexahedron block
4. Edit block boundary
5. Set global boundary
6. View skeleton boundary code
7. Plot finite element mesh

Step 1: Access Block Mesh Generator (New)

Access Block Mesh Generator by selecting the following menu items in SMAP (Figure 6.2):

Run → Mesh Generator → Block Mesh → New

Step 2: Set Work Plane

Prebuilt Work Plane is displayed on drawing board along with Work Plane Editor dialog. Modify NDx and Wx in Figure 6.32 and click Update button.

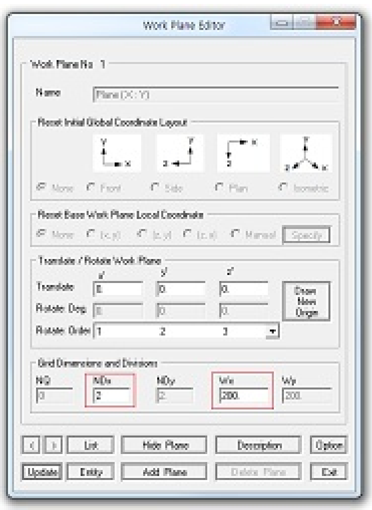


Figure 6.32 Work plane editor

Step 3: Build Hexahedron Block

Follow the same procedure as in Step 4 in the first example.
1. Click Axis toolbar as shown in Figure 6.9.
2. Click Block Editor toolbar in Figure 6.12.
3. Select Hexa for block type and click OK in Figure 6.13.
4. Click Draw Index Number in Figure 6.14.
5. Coordinates on Work Plane dialog is displayed as in Figure 6.15.

Index Numbers on Front Surface

6. Translate work plane as in Figure 6.33 and click Update button.
7. Click the points for index numbers on front surface as in Fig. 6.34.

Index Numbers on Back Surface

8. Translate work plane as in Figure 6.35 and click Update button.
9. Click the points for index numbers on back surface as in Figure 6.36.

Now, the geometry of hexahedron block is completed.
10. Click Finish in Figure 6.20.
11. Click Finish in Figure 6.14.

12. Modify Title and Material & Element Generation Parameters in Block Editor dialog as shown in Figure 6.37.

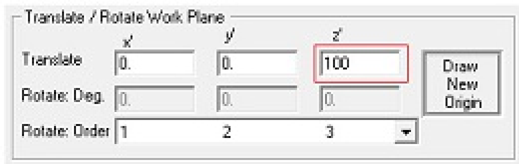


Figure 6.33 Work plane translation (z' = 100)

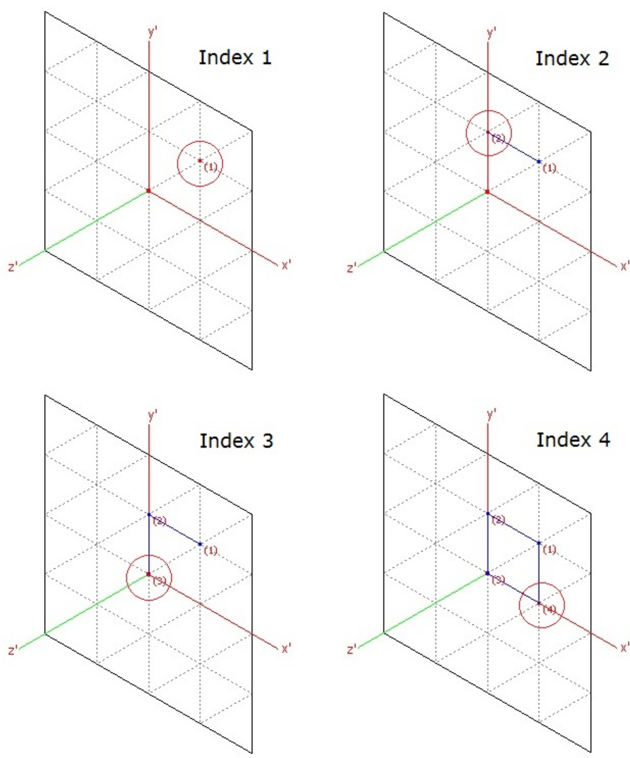


Figure 6.34 Index numbers on front surface

Figure 6.35 Work plane translation (z' = 0)

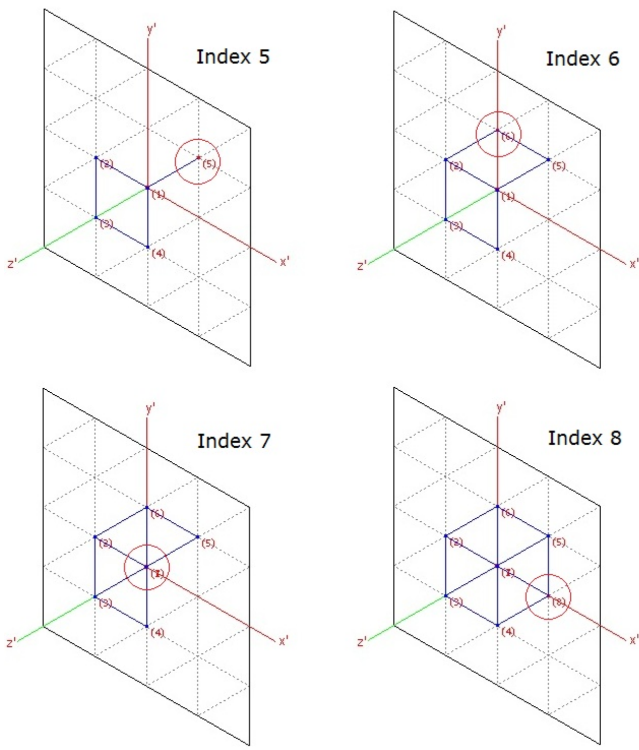


Figure 6.36 Index numbers on back surface

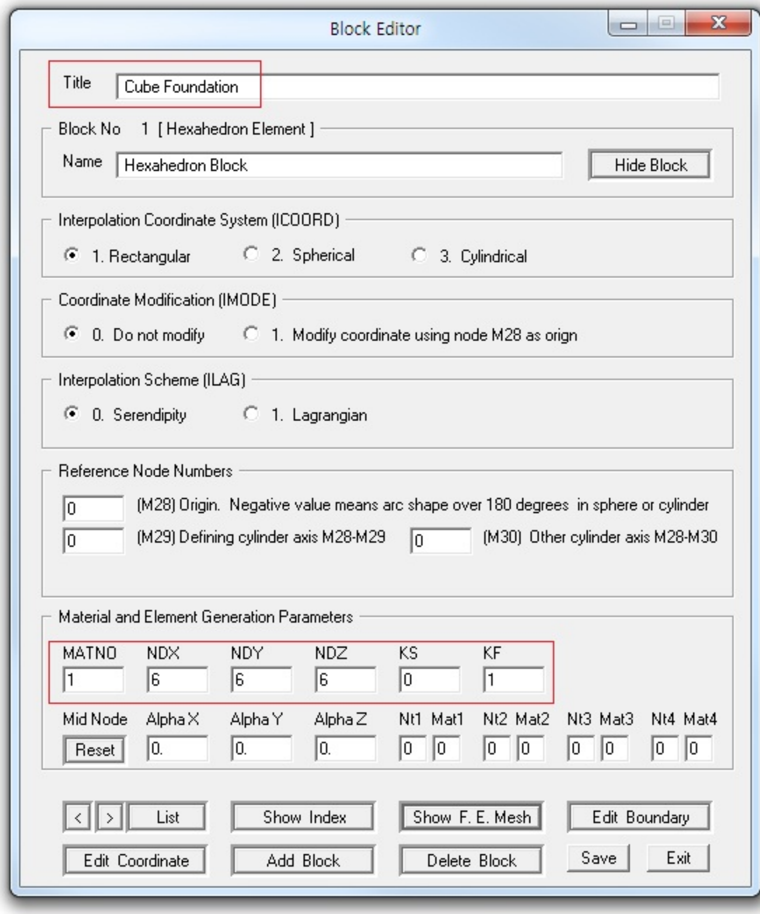


Figure 6.37 Block editor

Step 4: Edit Block Boundary Code

1. Click Edit Boundary in Figure 6.37.
2. Set the boundary codes as shown in Figure 6.38.
3. Click IBTYPE button to see description of boundary type in Fig. 6.39.
4. Click Update and then OK buttons in Figure 6.38.

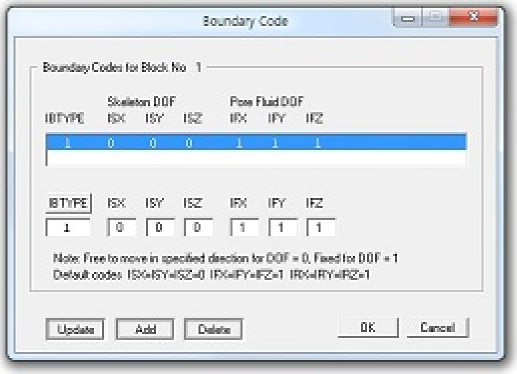


Figure 6.38 Boundary code editor

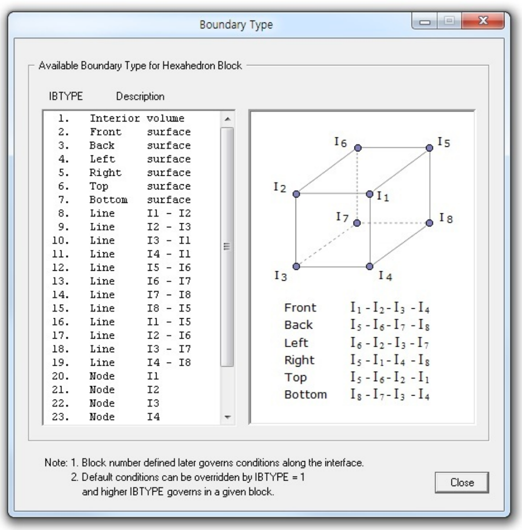


Figure 6.39 Boundary type for hexa block

Step 5: Set Global Boundary Code

1. Select Model → Edit Global Boundary in Figure 6.40.

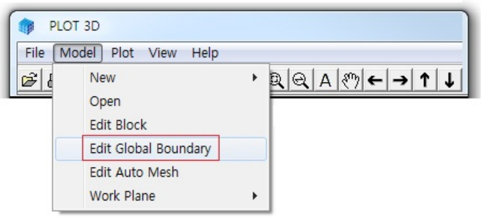


Figure 6.40 Edit global boundary menu

2. Set the boundary codes as shown in Figure 6.41.
3. Select Yes override block boundary.
4. Click Save and type in file name as EX2.

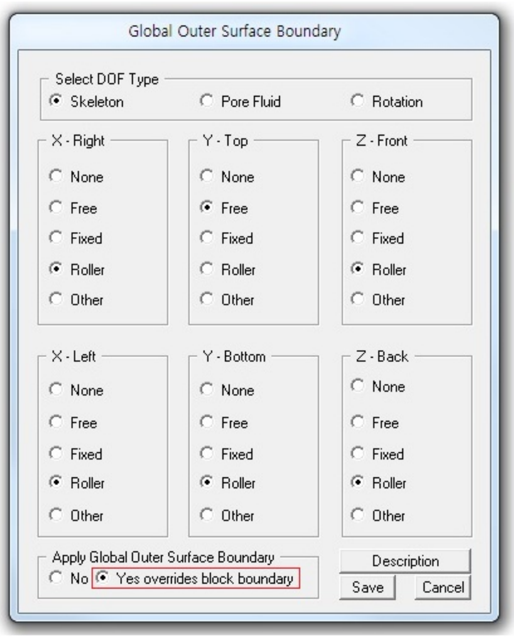


Figure 6.41 Global outer surface boundary

Step 6: View Skeleton Boundary Code

1. Select View → General in PLOT-3D menu.
2. Select Skeleton Boundary Code and click OK in Figure 6.25.
3. Click Show Numbers toolbar as shown in Figure 6.26.
4. Skeleton boundary codes are shown in Figure 6.42.

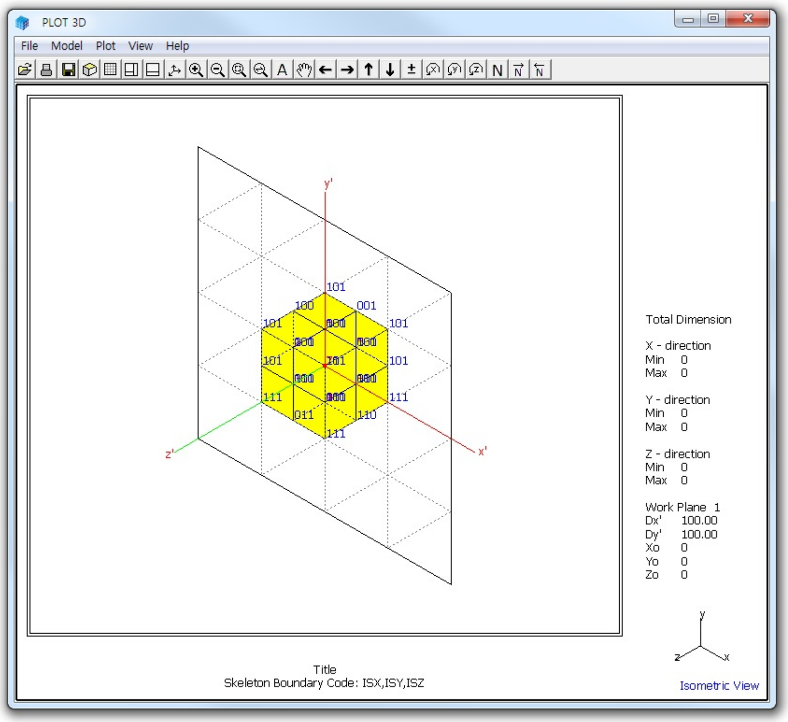


Figure 6.42 Skeleton boundary codes on drawing board

Step 7: Plot Finite Element Mesh

1. Click Show F. E. Mesh in Figure 6.37.
2. Rotate the finite element mesh as shown in Figure 6.43.

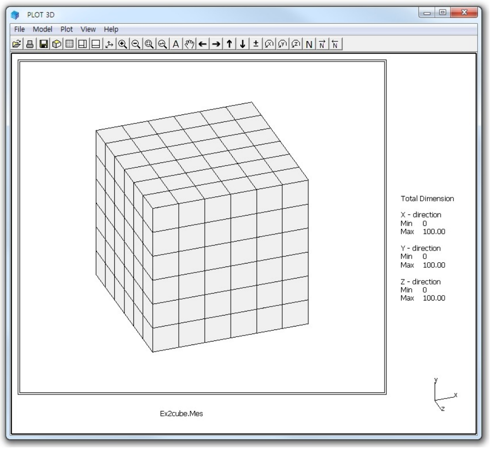


Figure 6.43 Finite element mesh

3. Follow same procedure to plot skeleton boundary codes in Step 6.
4. Figure 6.44 shows skeleton boundary codes for finite element mesh.

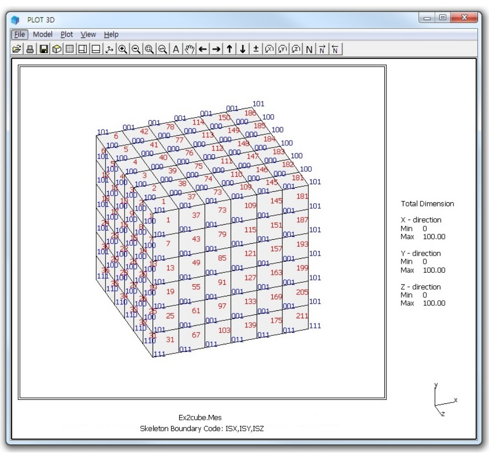


Figure 6.44 Skeleton boundary codes

6.2.2 Part 2: Modifying Cube Foundation

Part 2 consists of the following three main steps:
1. Access block mesh generator (Open)
2. Modify element generation parameters
3. Plot finite element mesh

Step 8: Access Block Mesh Generator (Open)

1. Access Block Mesh Generator by selecting the following menu items in SMAP (Figure 6.2):
   Run → Mesh Generator → Block Mesh → Open

2. Click Browse button in Open Input File dialog in Figure 6.45.
3. Select the input file EX2.Meb generated in Part 1 .

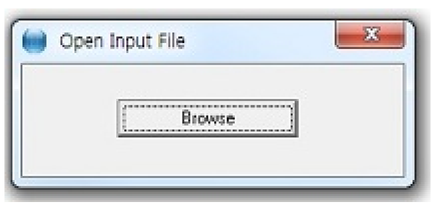


Figure 6.45 Open input file

Step 9: Modify Element Generation Parameters

1. Click Block Editor toolbar in Figure 6.12.
2. Modify Alpha X, Alpha Y, Alpha Z as in Figure 6.46.
3. Click Reset.
4. Click Save.

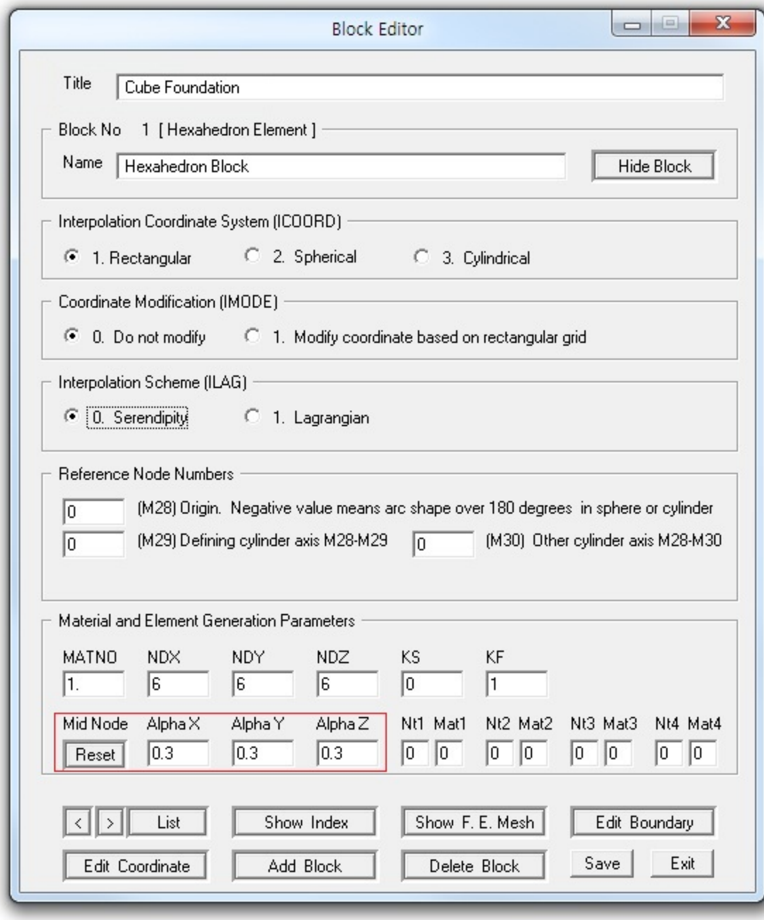


Figure 6.46 Block editor

5. Click Work Plane toolbar and then click Show Plane button.
6. Click Axis toolbar in Figure 6.9.
7. Block mesh is shown in Figure 6.47.

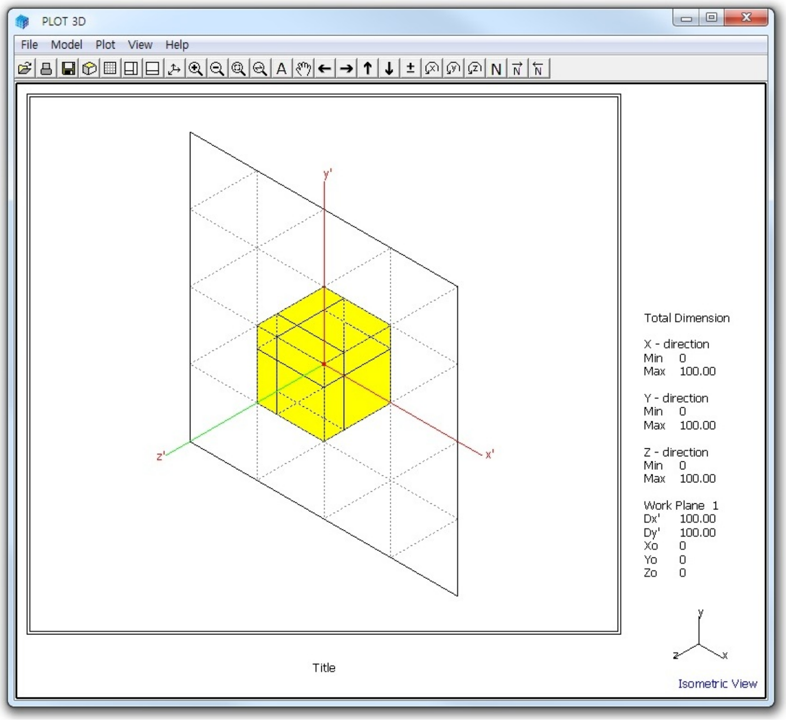


Figure 6.47 Block mesh on drawing board

Step 7: Plot Finite Element Mesh

1. Click Show F. E. Mesh in Figure 6.46.
2. Rotate the finite element mesh as shown in Figure 6.48.

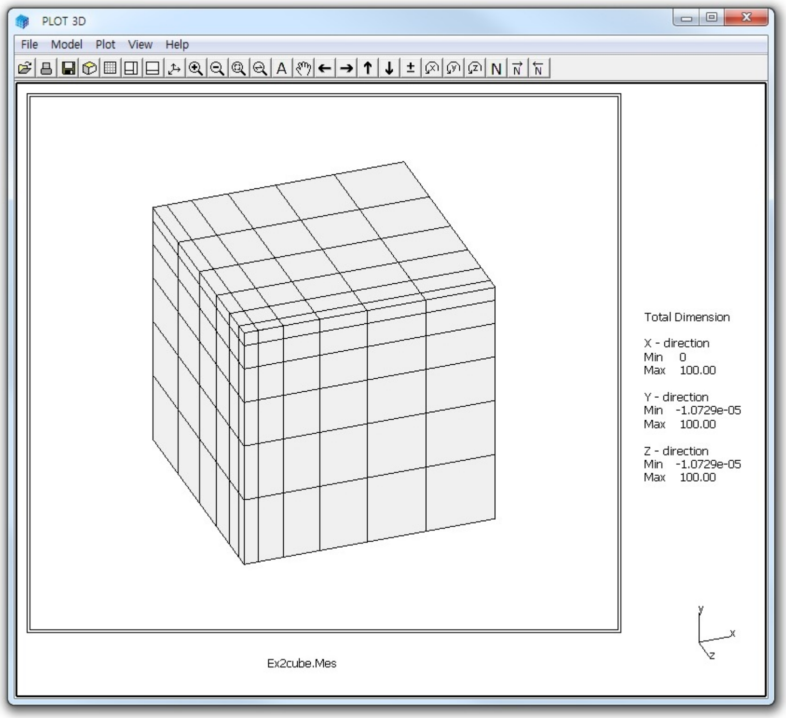


Figure 6.48 Finite element mesh

6.3 Hemispherical Shell

This example illustrates how to build block mesh for hemispherical shell subjected to concentrated loads as schematically shown in Figure 6.49.

This example involves following seven main steps:

1. Access block mesh generator
2. Set work plane
3. Build arc entities
4. Build quad block
5. Edit block boundary code
6. View skeleton and rotation boundary codes
7. Plot finite element mesh

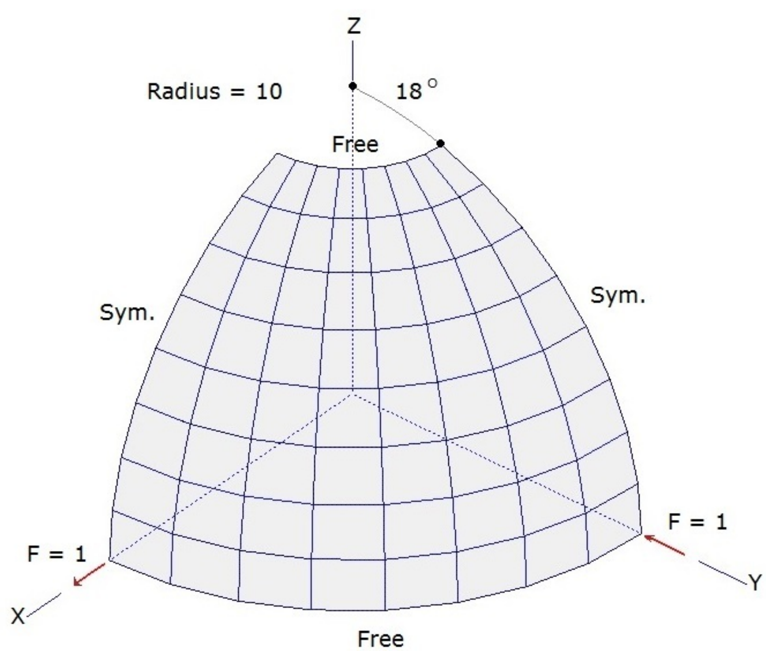


Figure 6.49 Hemispherical shell subjected to concentrated loads

Step 1: Access Block Mesh Generator (New)

Access Block Mesh Generator by selecting the following menu items in SMAP (Figure 6.2):
Run → Mesh Generator → Block Mesh → New

Step 2: Set Work Plane

1. Select Work Plane No 4 and set parameters for Grid Dimension and Division as shown in Figure 6.50.

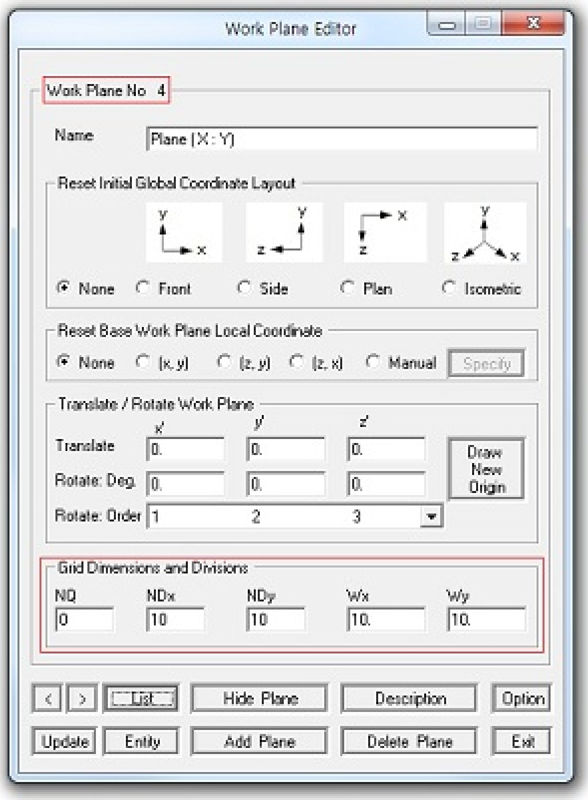


Figure 6.50 Work plane editor

2. Select Model → Work Plane → Isometric Z-axis in Figure 6.51.
3. Figure 6.52 shows work plane with Isometric Z-axis.

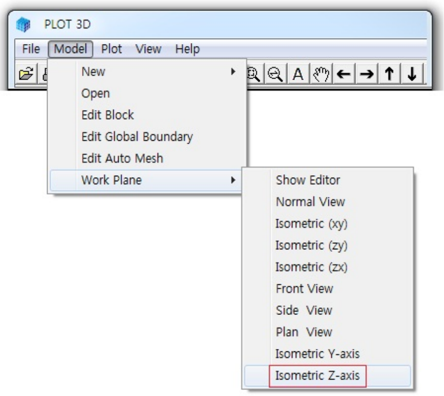


Figure 6.51 Isometric Z-axis menu

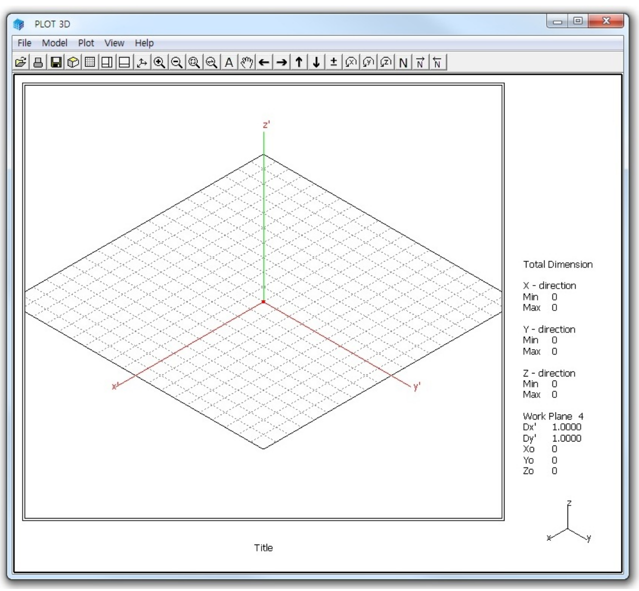


Figure 6.52 Work plane with Isometric Z-axis

Step 3: Build Arc Entities

Click Entity in Figure 6.50.

Arc Entity on YZ plane

1. Click Add in Entity Editor dialog in Figure 6.53.

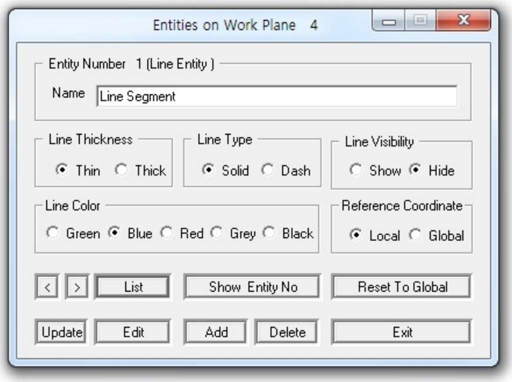


Figure 6.53 Entity editor

2. Select Arc in Entity Type Selection dialog in Figure 6.54.
3. Click OK.

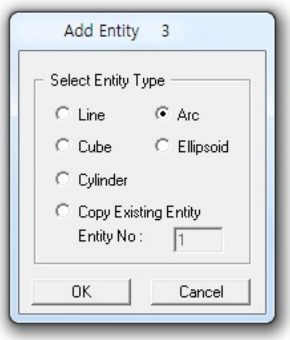


Figure 6.54 Entity type selection

4. Type in dimensions of arc entity as shown in Figure 6.55.
5. Click Draw Arc Entity.

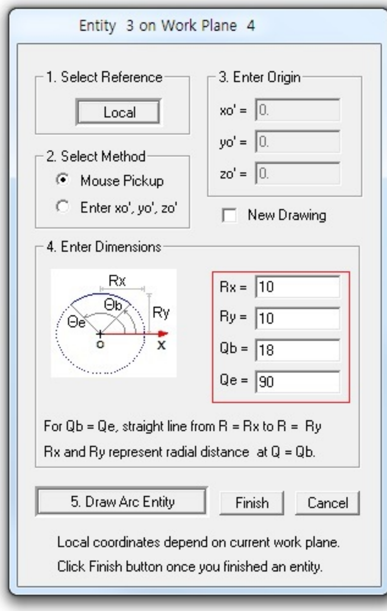


Figure 6.55 Arc entity

6. Figure 6.56 shows Coordinates on Work Plane dialog.

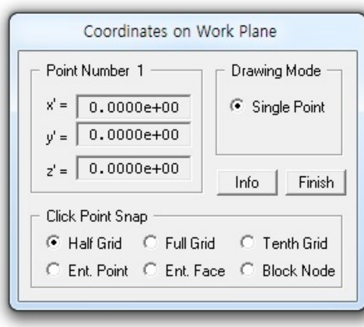


Figure 6.56 Coordinates on work plane

7. Rotate work plane as in Figure 6.57 and click Update button.

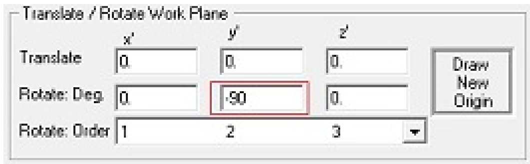


Figure 6.57 Work plane rotation

8. Click Mouse at the origin of coordinates as shown in Figure 6.58.

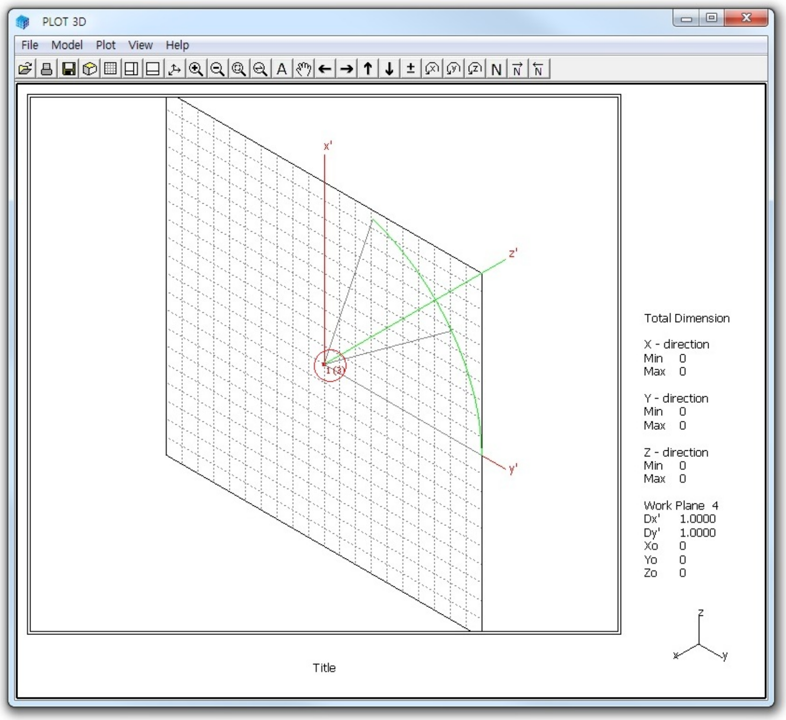


Figure 6.58 Arc entity on YZ plane

9. Click Finish in Figure 6.56.
10. Click Finish in Figure 6.55.
11. Click Global for Reference Coordinate in Figure 6.59.
12. Click Reset To Global.

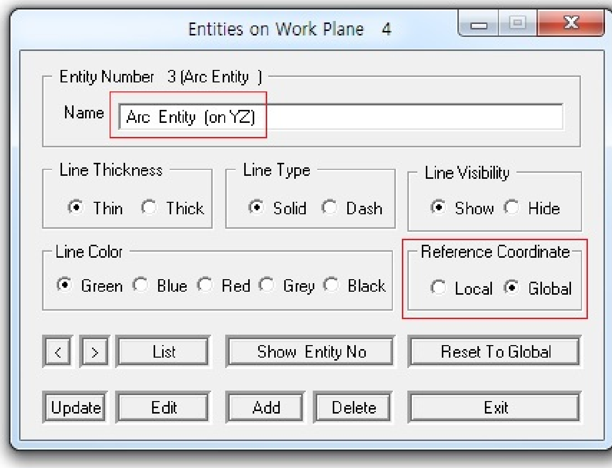


Figure 6.59 Entity editor

Arc Entity on XZ plane

Follow the same procedure as for Arc Entity on YZ plane.

1. Click Add in Entity Editor dialog in Figure 6.59.
2. Select Arc in Entity Type Selection dialog in Figure 6.54.
3. Click OK.

4. Type in dimensions of arc entity as shown in Figure 6.60.
5. Click Draw Arc Entity.
6. Coordinates on Work Plane dialog in Figure 6.56 is shown.

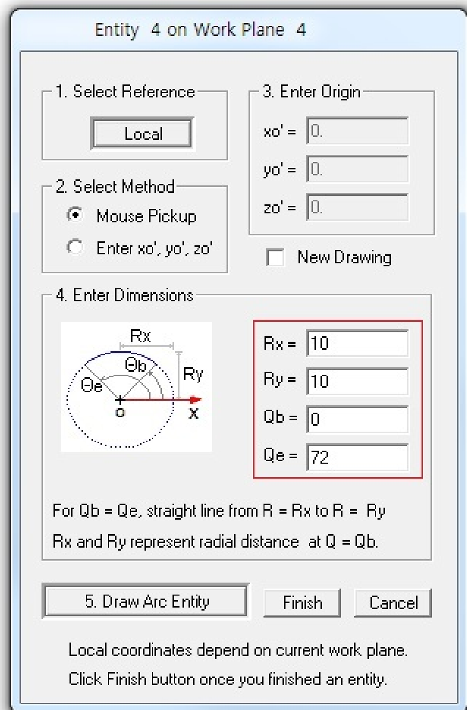


Figure 6.60 Arc entity

7. Rotate work plane as in Figure 6.61 and click Update button.

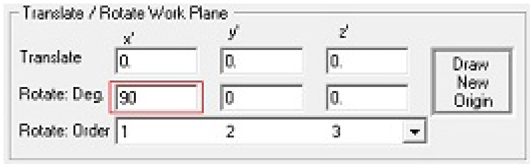


Figure 6.61 Work plane rotation

8. Click Mouse at the origin of coordinates as shown in Figure 6.62.

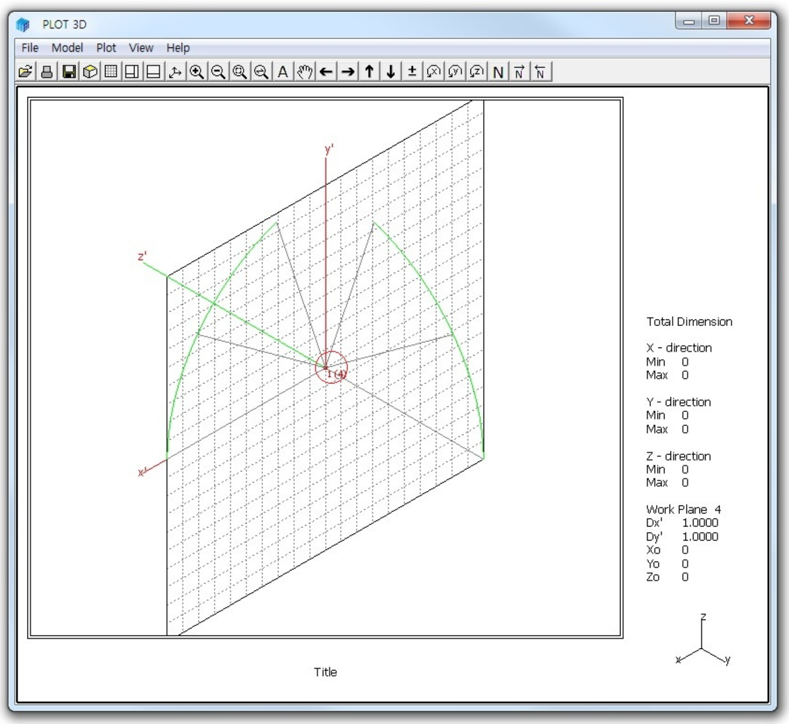


Figure 6.62 Arc entity on XZ plane

9. Click Finish in Figure 6.56.
10. Click Finish in Figure 6.60.
11. Click Global for Reference Coordinate in Figure 6.63.
12. Click Reset To Global and then Exit buttons in Figure 6.63.

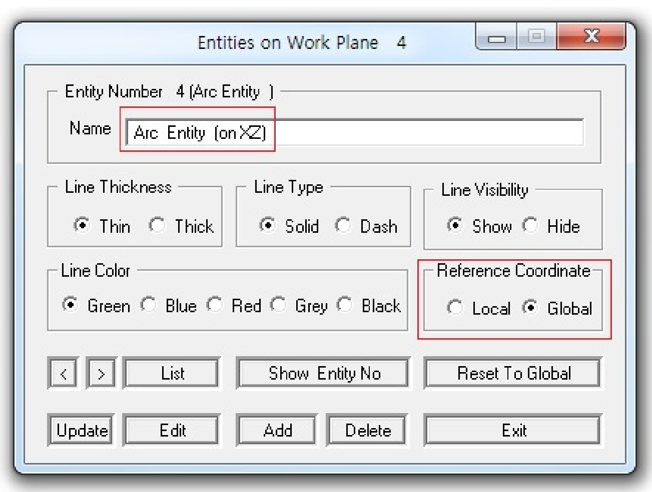


Figure 6.63 Entity editor

Step 4: Build Quad Block

1. Click Block Editor toolbar in Figure 6.64.

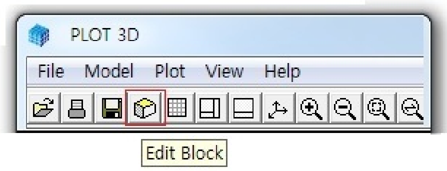


Figure 6.64 Block editor toolbar

2. Select Quad for block type in Figure 6.65.
3. Select Spherical for Interpolation Coordinate System.
4. Select Origin for Reference Point.
5. Click OK.

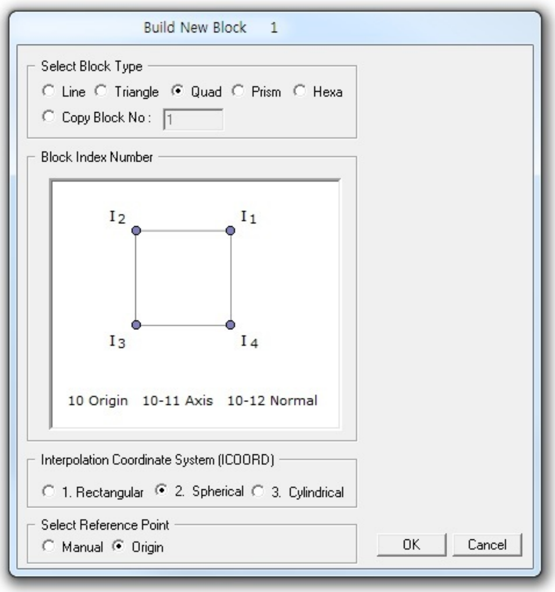


Figure 6.65 Block type selection

6. Click Draw Index Number in Figure 6.66.
7. Select Ent. Point for Click Point Snap as in Figure 6.67.

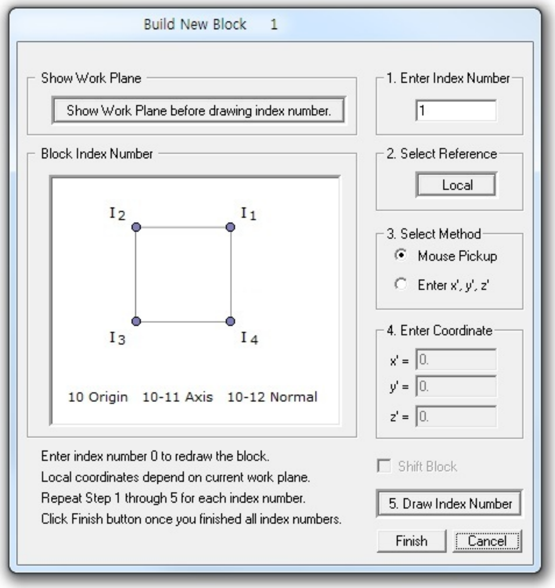


Figure 6.66 Quad block

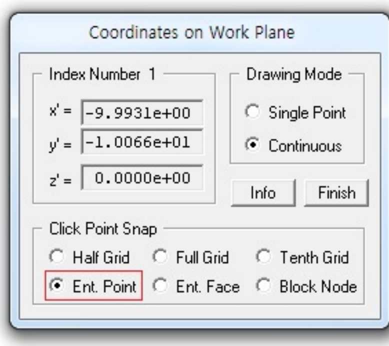


Figure 6.67 Coordinates on work plane

8. Rotate work plane as in Figure 6.68 and click Update button.

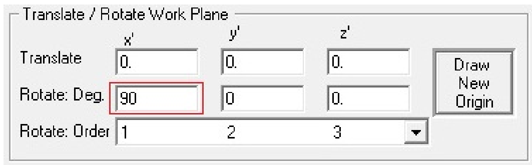


Figure 6.68 Work plane rotation

9. Click the points for index numbers 1 and 2 as in Figure 6.69.

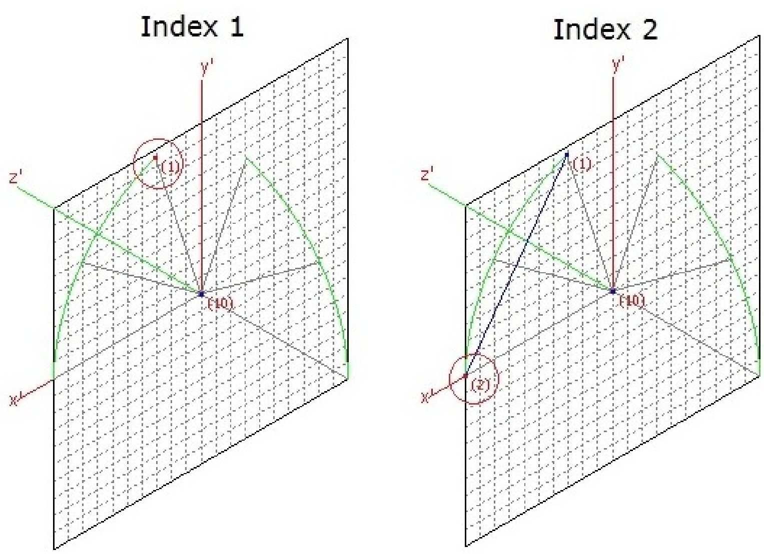


Figure 6.69 Index numbers 1 and 2 on XZ plane

10. Rotate work plane as in Figure 6.70 and click Update button.

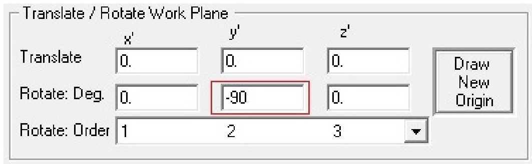


Figure 6.70 Work plane rotation

11. Click the points for index numbers 3 and 4 as in Figure 6.71.

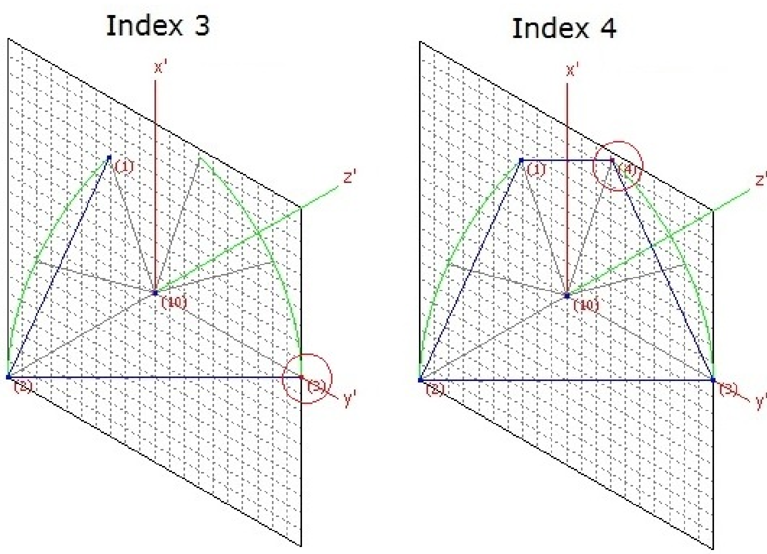


Figure 6.71 Index numbers 3 and 4 on YZ plane

Now, the geometry of quad block is completed.

12. Click Finish in Figure 6.67 and then click Finish in Figure 6.66.
13. Figure 6.72 shows completed quad block with index numbers.

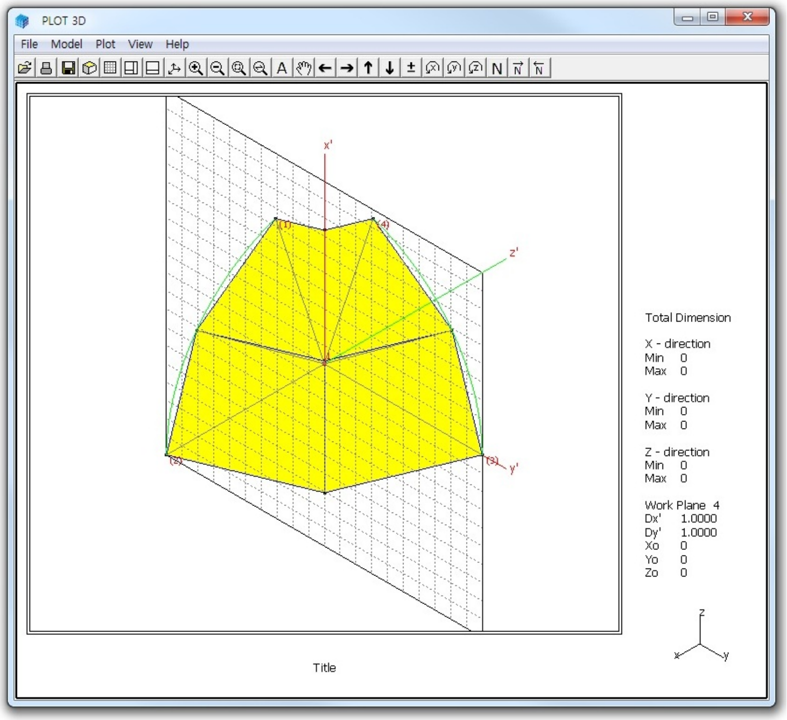


Figure 6.72 Completed quad block

14. Modify Title and Material & Element Generation Parameters in Block Editor dialog as shown in Figure 6.73.

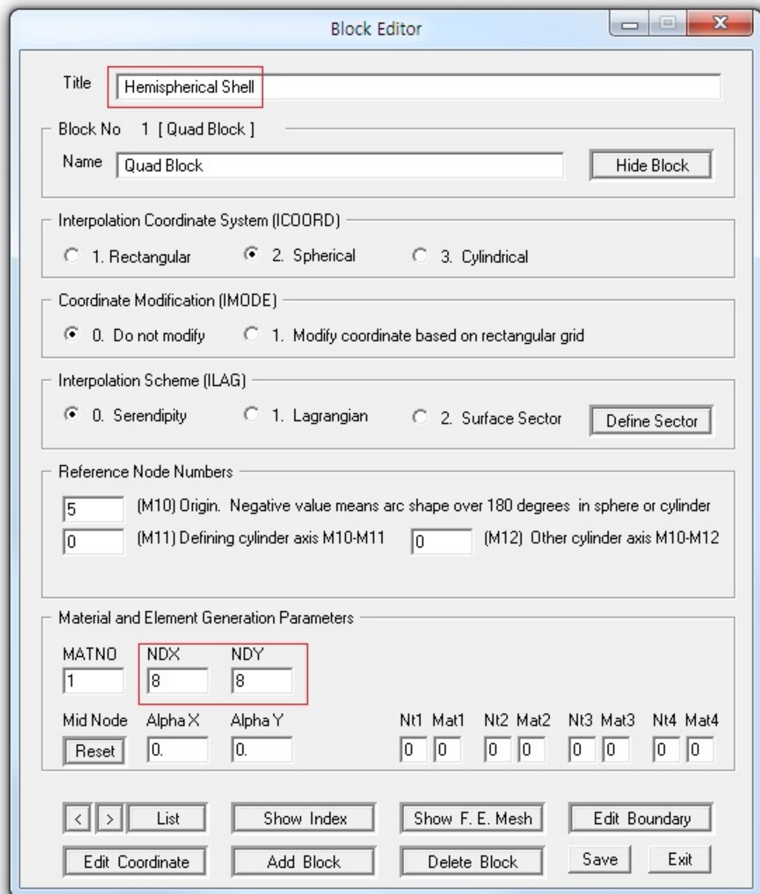


Figure 6.73 Block editor

Step 5: Edit Block Boundary Code

1. Click Edit Boundary in Figure 6.73.
2. Set the boundary codes as shown in Figure 6.74.
3. Click IBTYPE button to see description of boundary type in Fig. 6.75.
4. Click Update and then OK buttons.
5. Click Save in Figure 6.73 and type in file name as EX3.

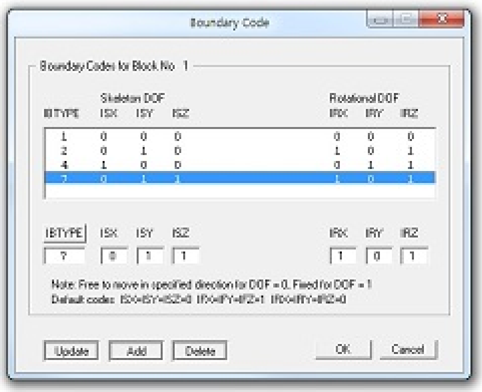


Figure 6.74 Boundary code editor

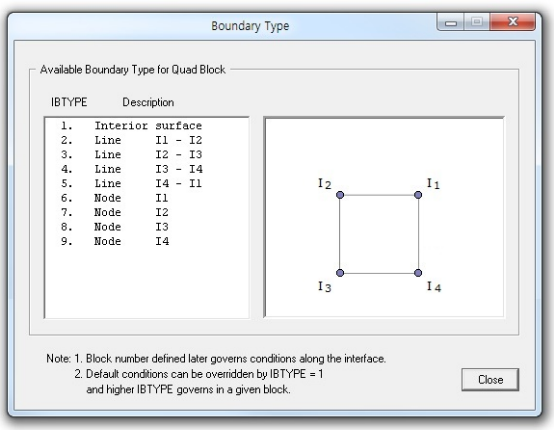


Figure 6.75 Boundary type for quad block

Step 6: View Skeleton and Rotation Boundary Code

1. Select View → General in PLOT-3D menu.
2. Select Skeleton Boundary Code and click OK in Figure 6.25.
3. Click Show Numbers toolbar as shown in Figure 6.26.
4. Skeleton boundary codes are shown in Figure 6.76.

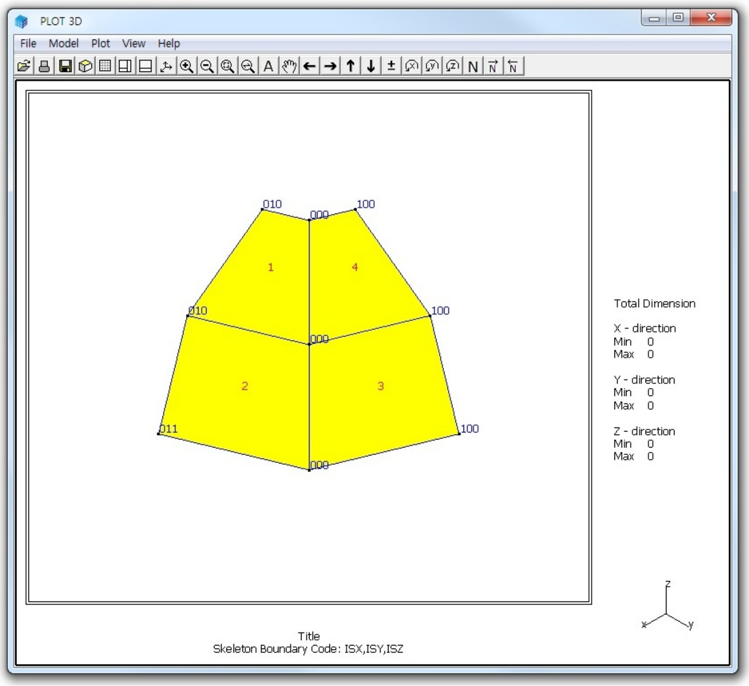


Figure 6.76 Skeleton boundary codes

5. Click Next Numbers toolbar twice in Figure 6.77.

Figure 6.77 Next numbers toolbar

6. Rotation boundary codes are shown in Figure 6.78.

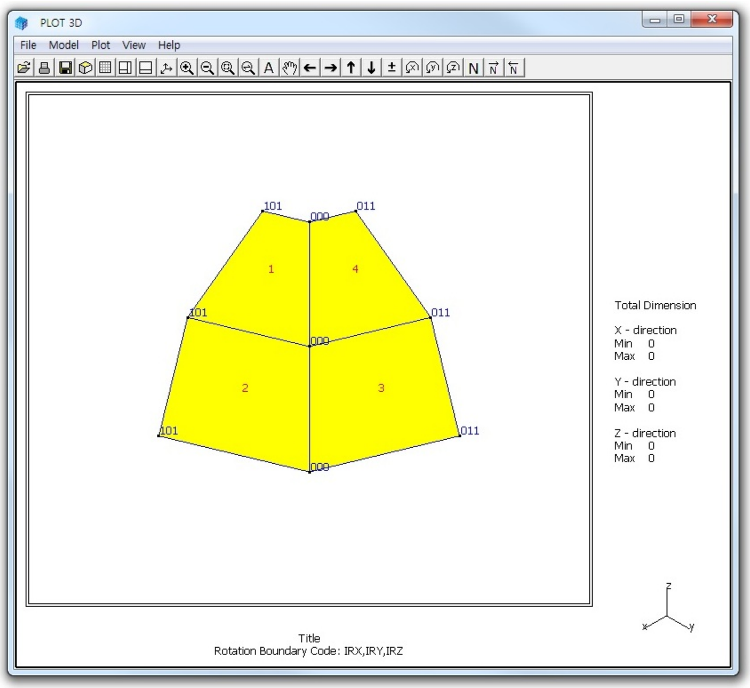


Figure 6.78 Rotation boundary codes

Step 7: Plot Finite Element Mesh

1. Click Show F. E. Mesh in Figure 6.73.
2. Select Model → Work Plane → Isometric Z-axis in Figure 6.51.
3. Finite element meshes are shown in Figure 6.79.

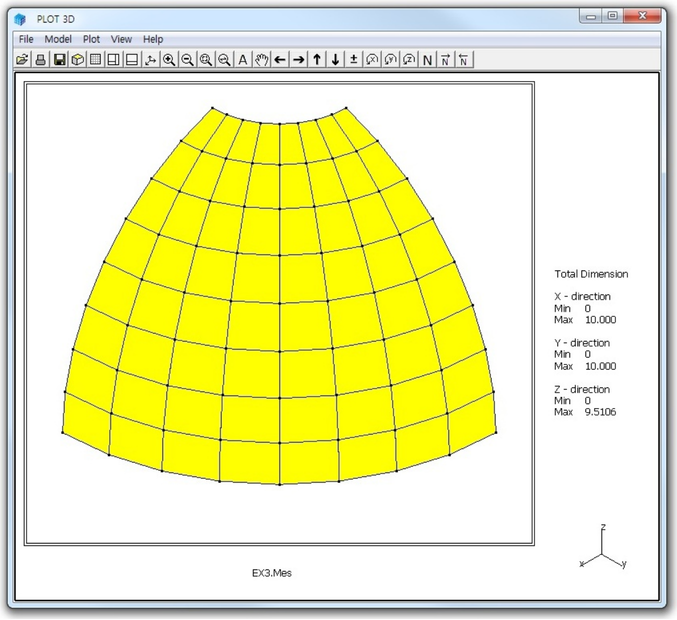


Figure 6.79 Finite element mesh

4. Follow the same procedure to plot boundary codes as in Step 6.
5. Skeleton and rotation boundary codes are shown in Figures 6.80 and 6.81, respectively.

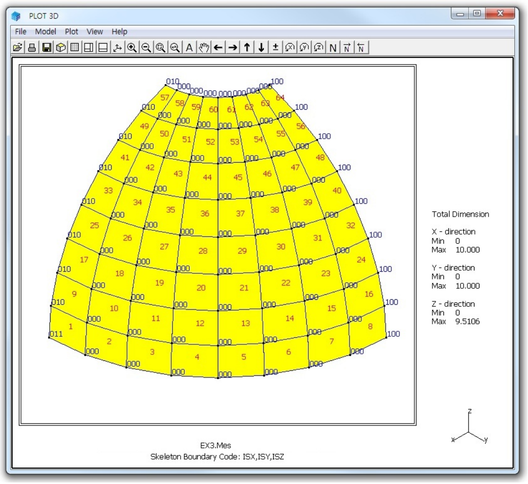


Figure 6.80 Skeleton boundary codes

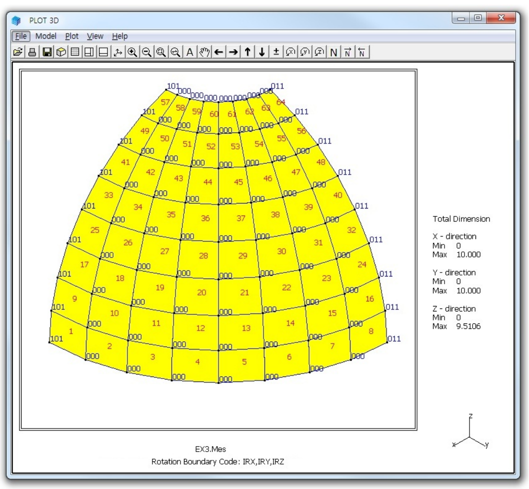


Figure 6.81 Rotation boundary codes

6.4 Horseshoe Tunnel

This example illustrates how to build block mesh for horseshoe tunnel with reinforced concrete lining as schematically shown in Figure 6.82.

This example involves following eight main steps:

1. Access block mesh generator
2. Set work plane
3. Build entities
4. Add work plane
5. Build blocks
6. Set global boundary
7. View selected material
8. Plot finite element mesh

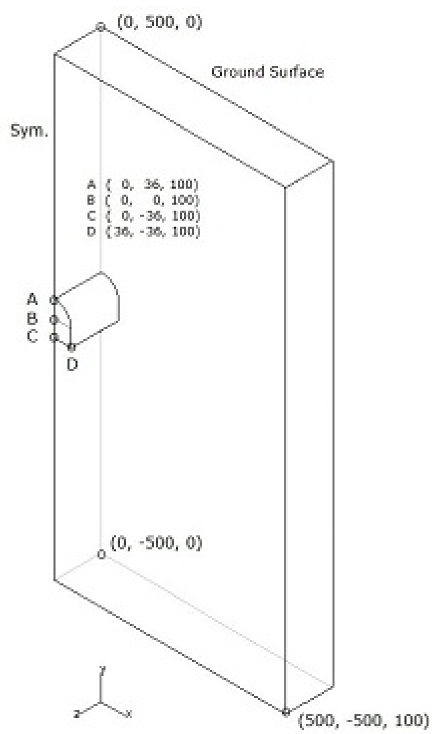


Figure 6.82 Schematic horseshoe tunnel section

Step 1: Access Block Mesh Generator (New)

Access Block Mesh Generator by selecting the following menu items in SMAP (Figure 6.2):

Run → Mesh Generator → Block Mesh → New

Step 2: Set Work Plane

1. Select Work Plane No 4 as shown in Figure 6.83.
2. Select Isometric for Reset Initial Global Coordinate Layout.
3. Set parameters for Grid Dimensions and Divisions.
4. Click Description to see layout of NQ = 8 in Figure 6.84.
5. Click Update.
6. Figure 6.85 shows isometric view of work plane.

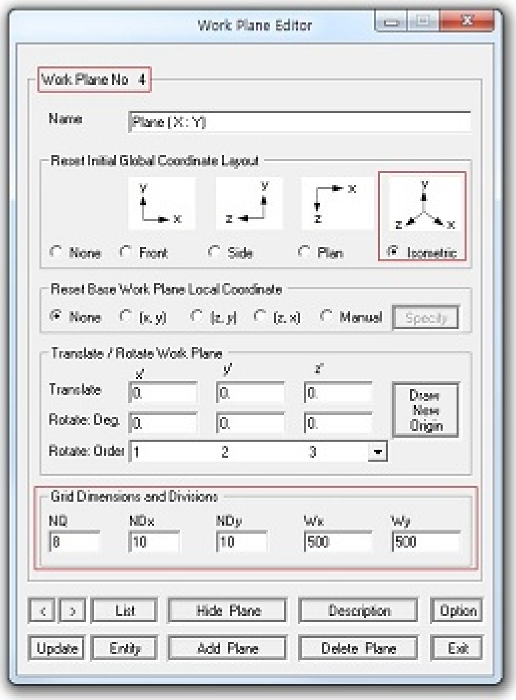


Figure 6.83 Work plane editor

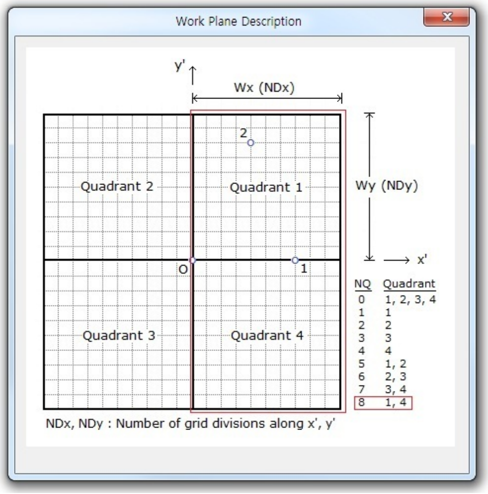


Figure 6.84 Work plane description (NQ=8)

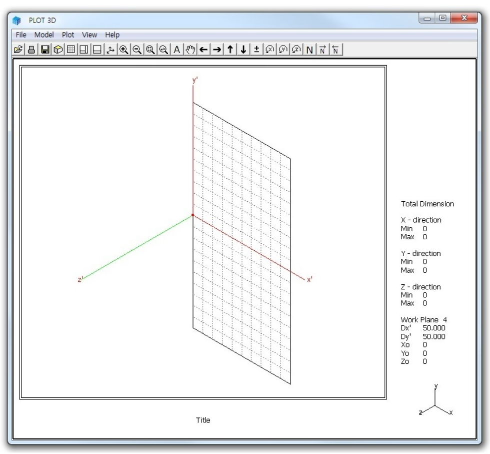


Figure 6.85 Work plane on drawing board

Step 3: Build Entities

Following five entities are used to make it easier to build blocks
- Cylinder entity for Upper Core
- Cube entity for Lower Core
- Cylinder entity for Around Upper Core
- Cube entity for Around Lower Core
- Cube entity for Outer Boundary

Upper Core by Cylinder Entity

1. Click Entity in Figure 6.83.
2. Click Add in Entity Editor dialog in Figure 6.88.
3. Click Cylinder in Figure 6.86 and click OK.
4. Set the geometric parameters as in Figure 6.87.
5. Click Draw Cylinder Entity and then click Finish.
6. Set option parameters as in Figure 6.88 and click Reset To Global.
7. Cylinder entity for upper core is shown in Figure 6.89.

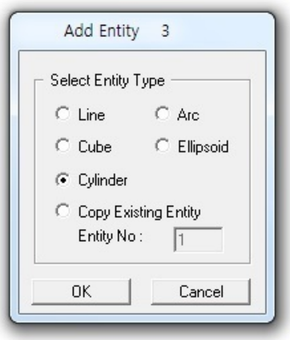


Figure 6.86 Entity type selection

Other Entities

8. Follow the same procedure as for upper core.

Lower Core:	Figures 6.90 - 6.92
Around Upper Core:	Figures 6.93 - 6.95
Around Lower Core:	Figures 6.96 - 6.98
Outer Boundary:	Figures 6.99 - 6.101

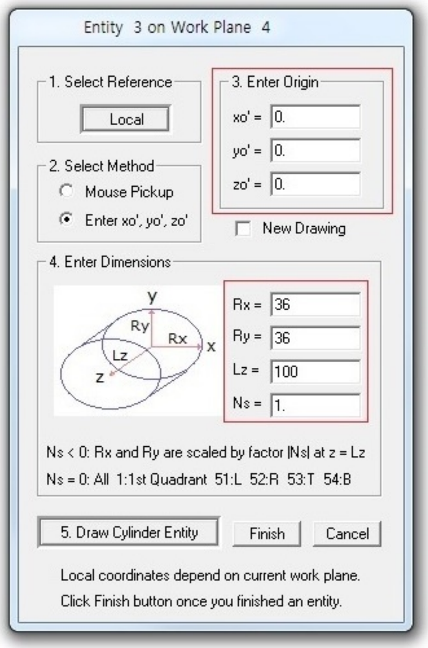


Figure 6.87 Cylinder entity for upper core

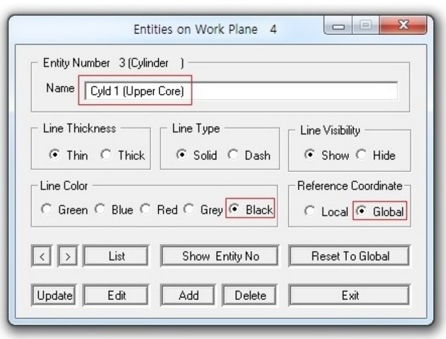


Figure 6.88 Entity editor

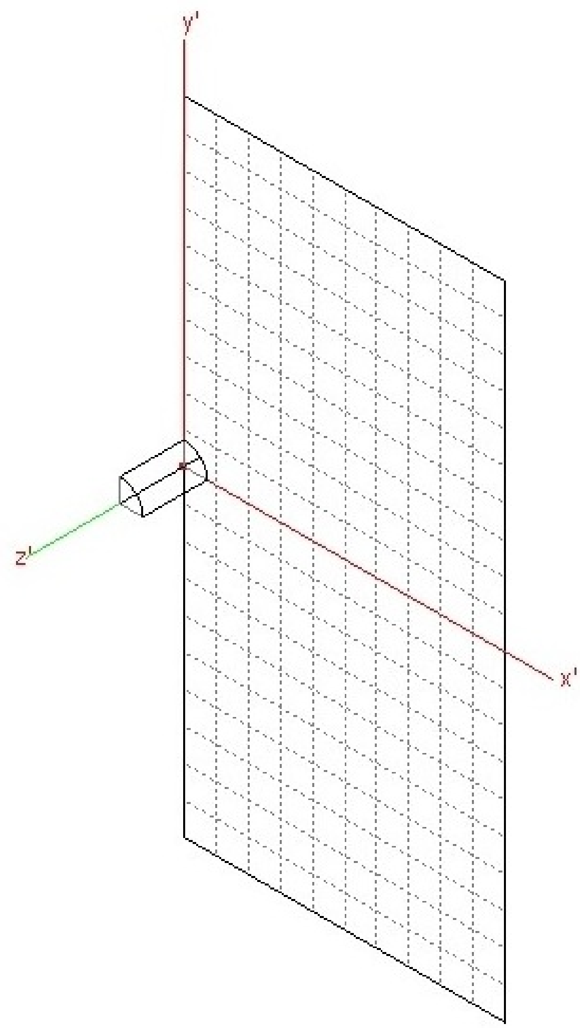


Figure 6.89 Upper core on drawing board

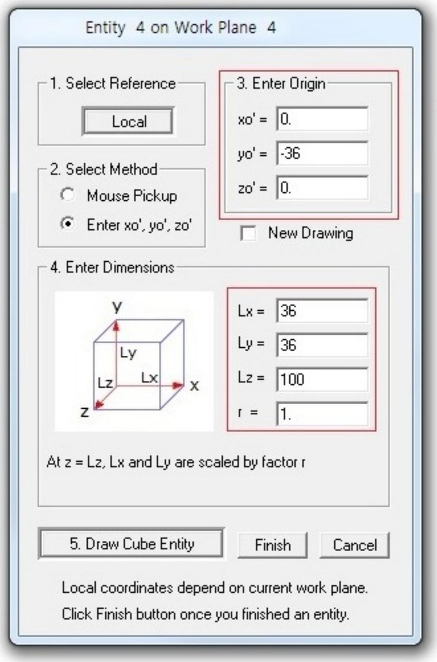


Figure 6.90 Cube entity for lower core

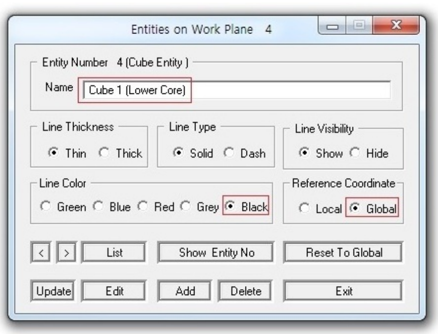


Figure 6.91 Entity editor

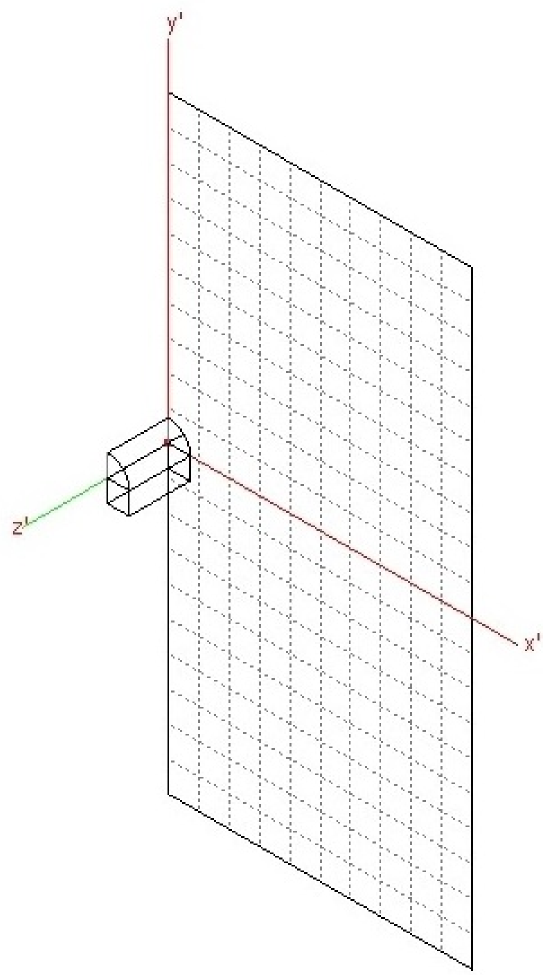


Figure 6.92 Lower core on drawing board

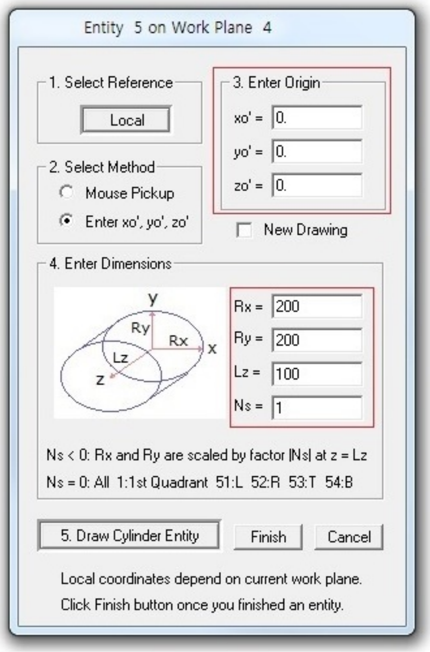


Figure 6.93 Cylinder entity for around upper core

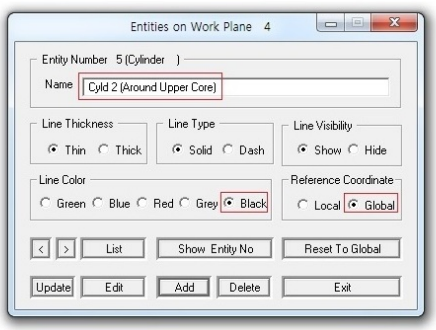


Figure 6.94 Entity editor

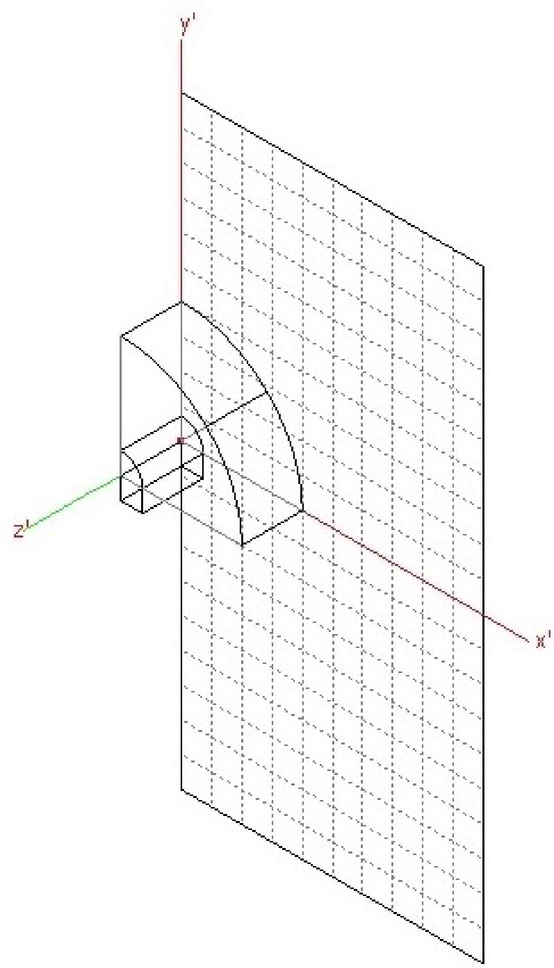


Figure 6.95 Around upper core on drawing board

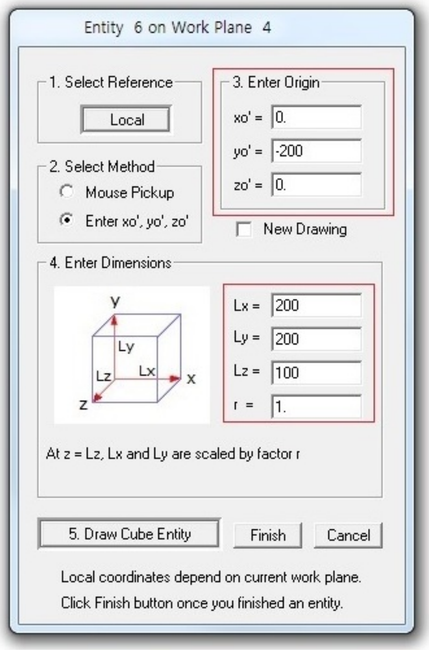


Figure 6.96 Cube entity for around lower core

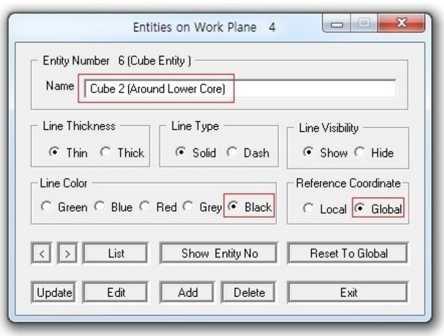


Figure 6.97 Entity editor

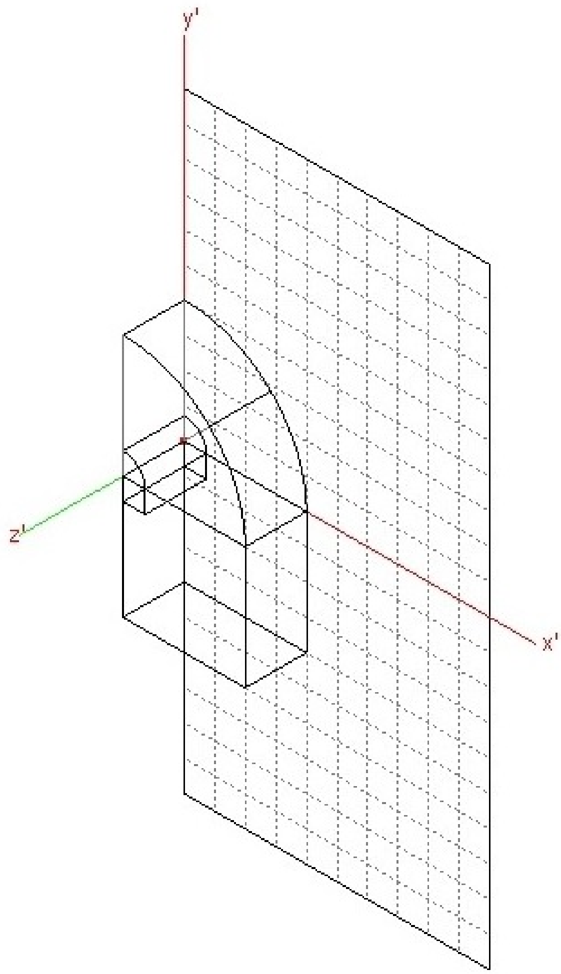


Figure 6.98 Around lower core on drawing board

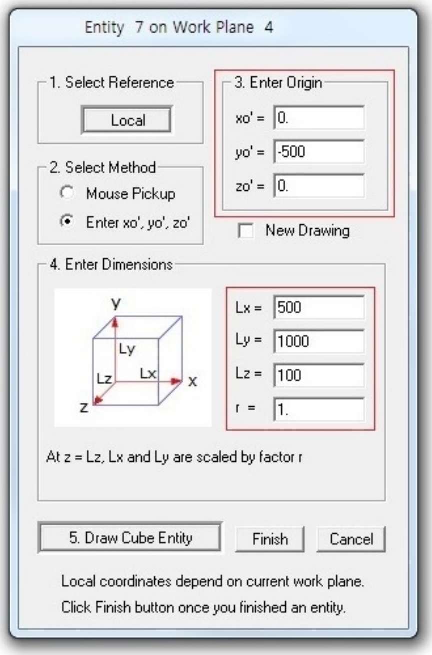


Figure 6.99 Cube entity for outer boundary

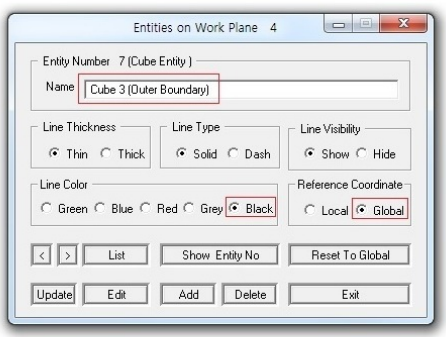


Figure 6.100 Entity editor

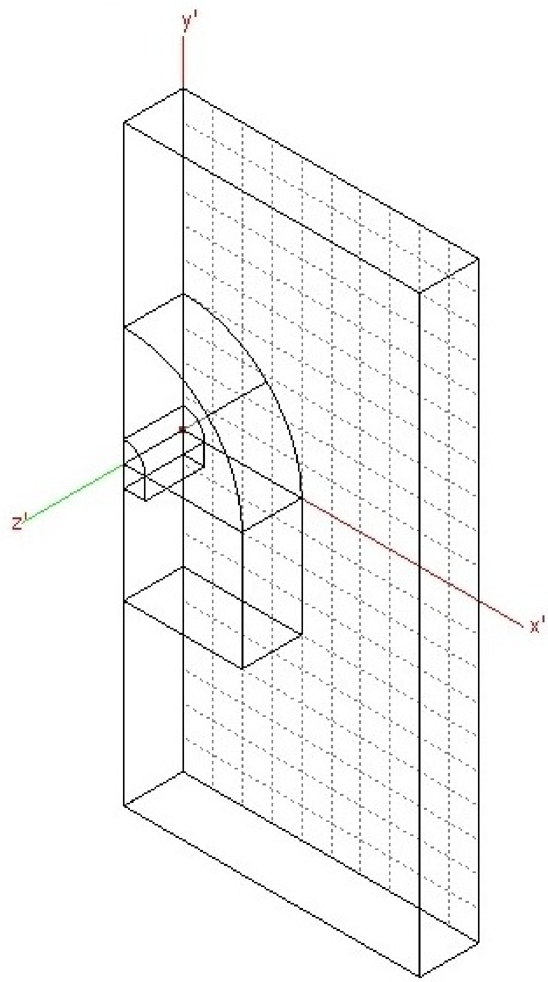


Figure 6.101 Outer boundary on drawing board

9. Available entities on work plane 4 are listed as shown in Figure 6.102 by clicking List button in the Entity Editor dialog.
10. Click OK in Figure 6.102.
11. Click Exit in Figure 6.100.

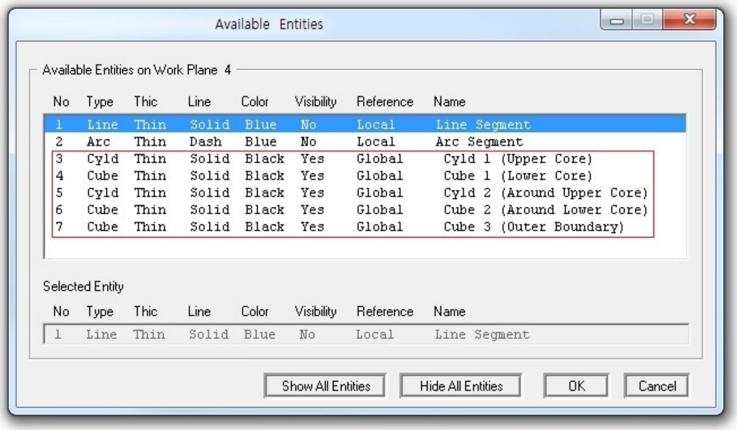


Figure 6.102 Available entities on work plane 4

Step 4: Add Work Plane

At Step 2, we set Work Plane No 4 which represents back surface.
At Step 3, we built 5 entities on this Work Plane No 4.

Here, we want to add new Work Plane No 5 in the following way:

- Copy Work Plane No 4 along with entities on it.
- Add this copied one as new Work Plane No 5.
- Modify such that it represents front surface.

Once we have this new Work Plane No 5, it will be much easier to build blocks since front and back surfaces of work planes can be accessed simply by one click of Back or Next button on Coordinates on Work Plane dialog in Figure 6.103.

Perform the following four steps:

1. Select Work Plane No 4 in Work Plane Editor dialog in Figure 6.83
2. Click Add Plane button in Figure 6.83
3. Modify Name and Translation as in Figure 6.104
4. Click Update in Figure 6.104

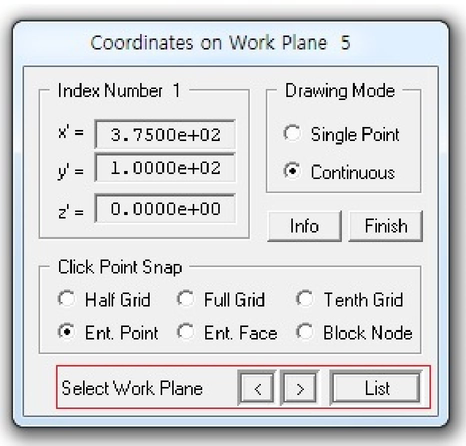


Figure 6.103 Coordinates on work plane

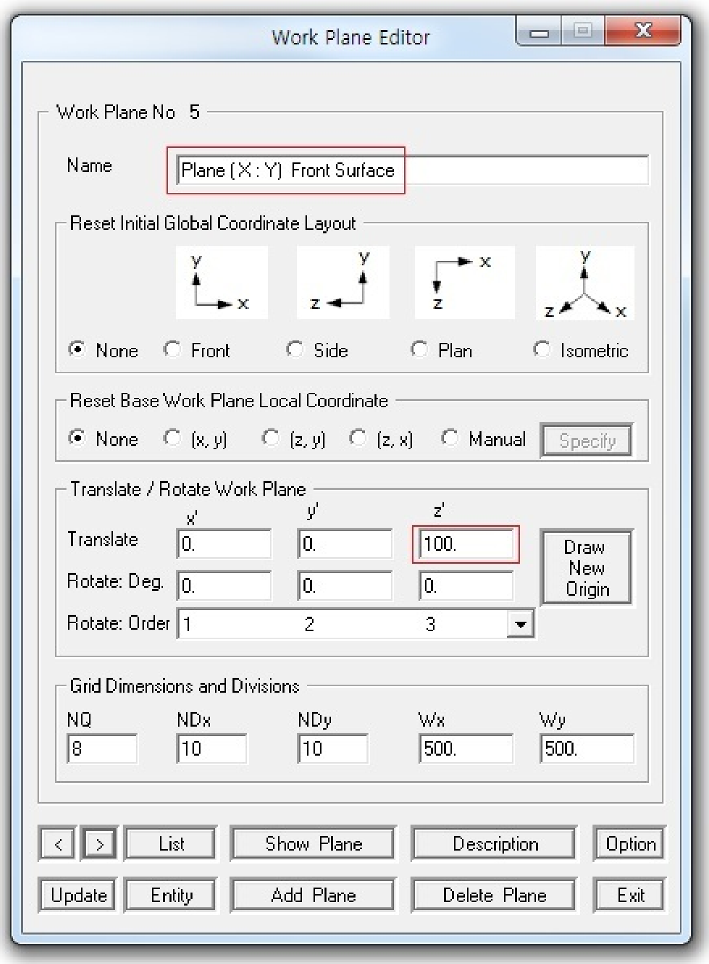


Figure 6.104 Work Plane No 5 representing front surface

Step 5: Build Blocks

Fourteen blocks are used to model the geometry of horseshoe tunnel as shown in Figures 6.105 and 6.106.

- 8 blocks for surrounding medium
- 2 blocks for tunnel core
- 4 blocks for tunnel lining as shell elements

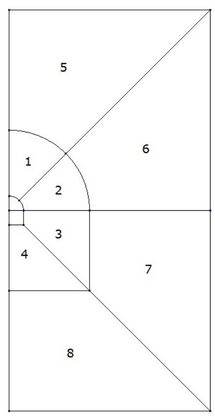


Figure 6.105 Block numbers for surrounding medium

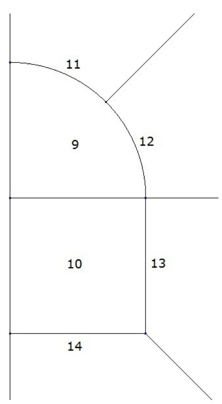


Figure 6.106 Block numbers for tunnel core and lining

Building Block No 1

1. Click Zoom toolbar to turn on in Figure 6.107.
2. Click the mouse at top-left point and release at bottom-right point as shown in Figure 6.108.
3. Figure 6.109 shows zoomed area around upper core.
4. Click Zoom toolbar again to turn off.
5. Click Block Editor toolbar in Figure 6.110.
6. Select Hexa, Cylindrical, and Z axis as shown in Figure 6.111.
7. Click OK.
8. Click Draw Index Number in Figure 6.112.
9. Select Ent. Point for Click Point Snap as in Figure 6.103.

Figure 6.107 Zoom toolbar

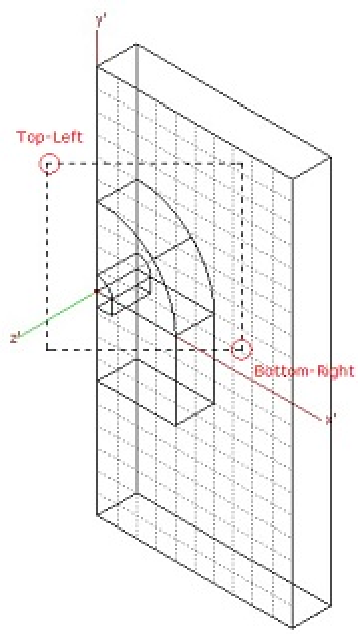


Figure 6.108 Zoom area selection

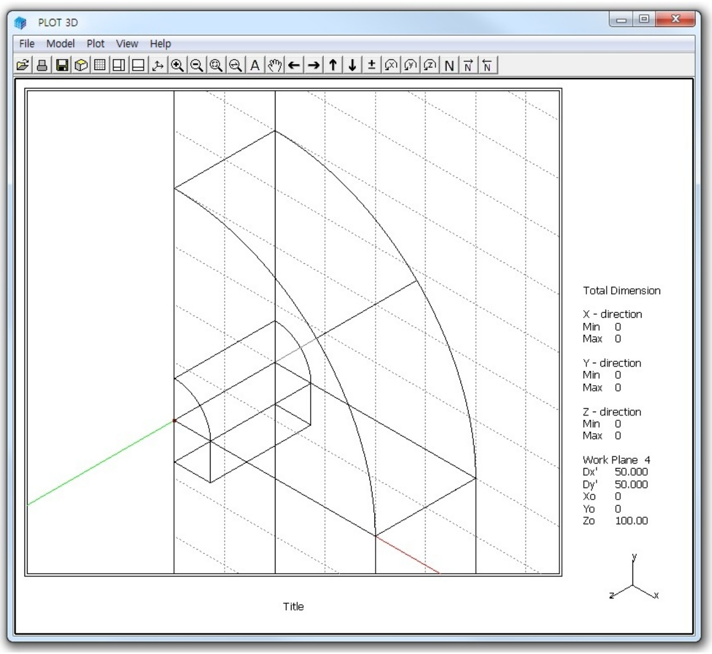


Figure 6.109 Zoomed area around upper core

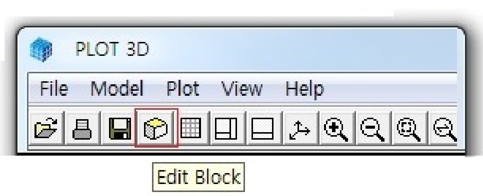


Figure 6.110 Block editor toolbar

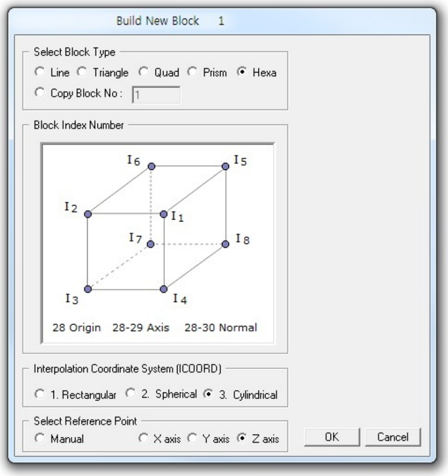


Figure 6.111 Block type selection

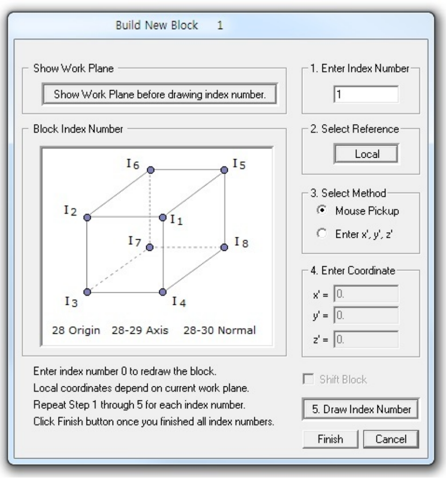


Figure 6.112 Hexa block

10. Select Work Plane No 5 for front surface using Back / Next button in Figure 6.103.
11. Click the points for index numbers on front surface as in Fig 6.113.

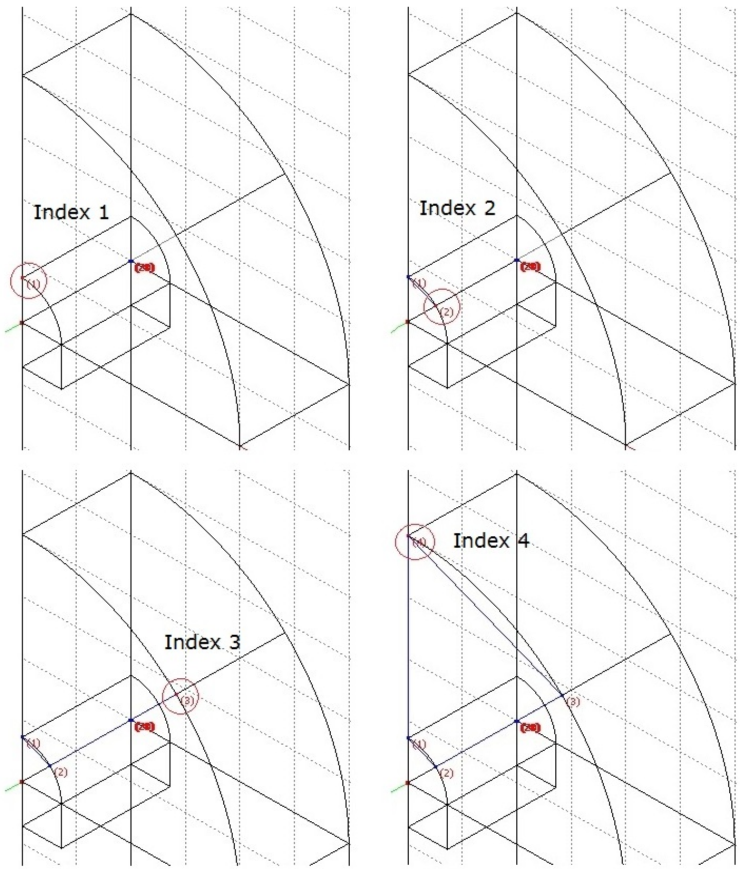


Figure 6.113 Index numbers on front surface

12. Select Work Plane No 4 for back surface using Back / Next button in Figure 6.103.
13. Click the points for index numbers on back surface as in Fig 6.114.

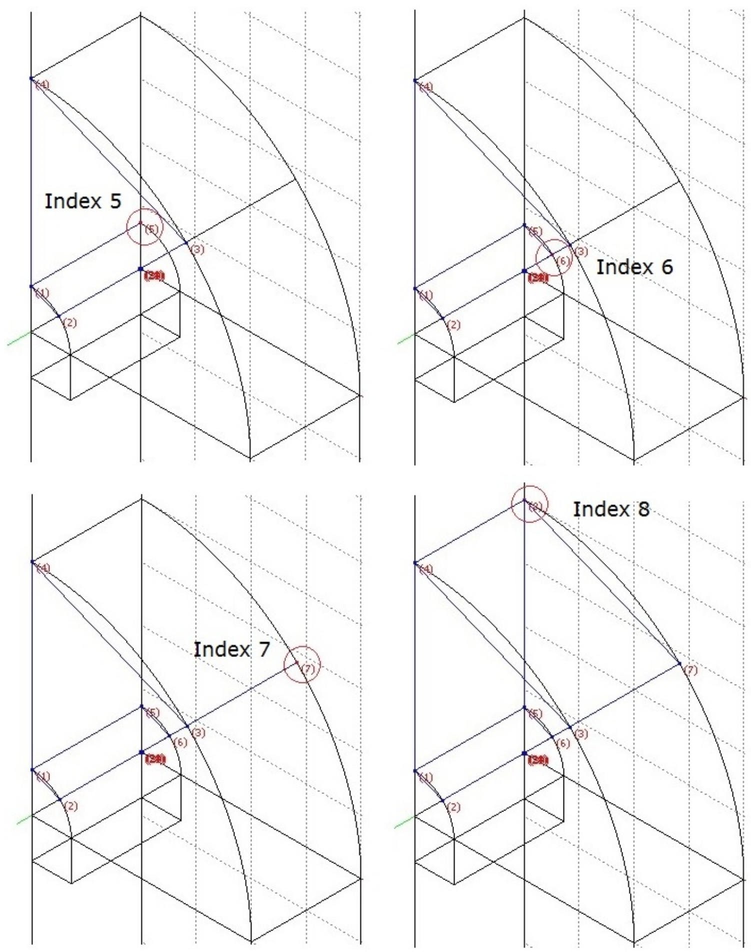


Figure 6.114 Index numbers on back surface

Now, the geometry of the first hexahedron block is completed.

14. Click Finish in Figure 6.103 and then click Finish in Figure 6.112.
15. Modify Title, Block Name and Material & Element Generation Parameters in Block Editor as shown in Figure 6.115.
16. Click Reset button.

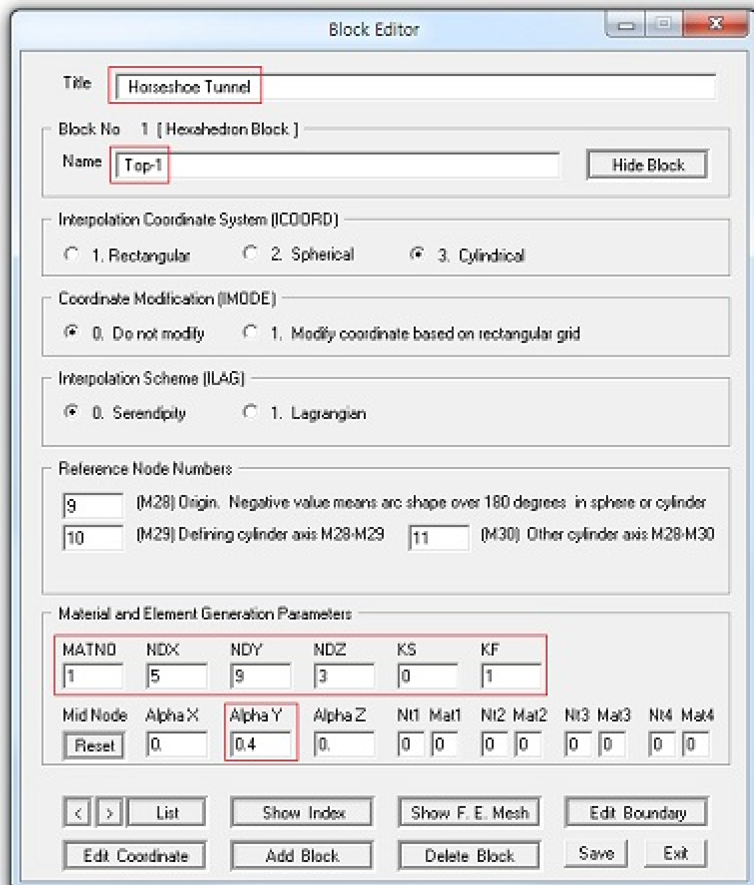


Figure 6.115 Block No 1

17. Figure 6.116 shows Block No 1 on drawing board.

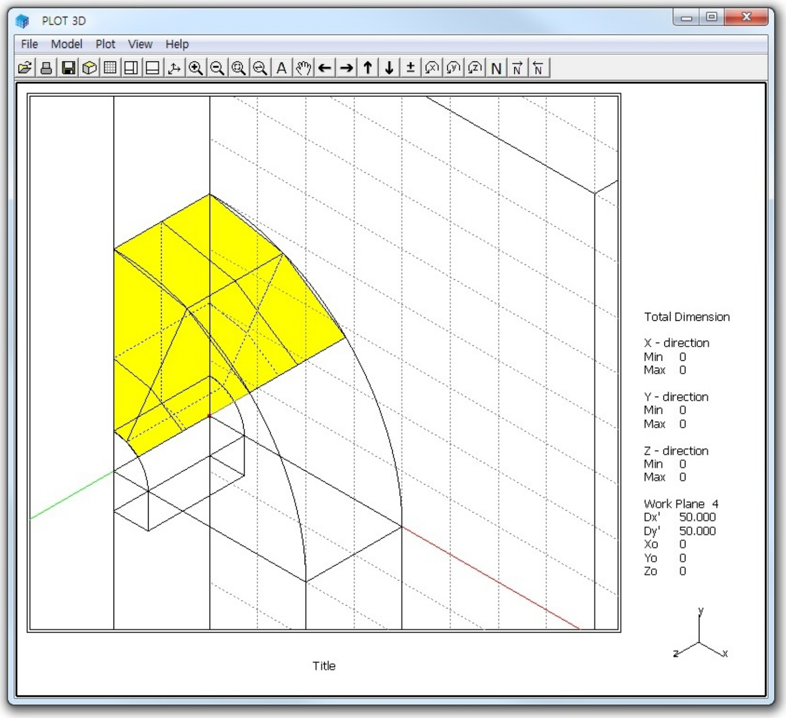


Figure 6.116 Block No 1 on drawing board

Building Other Blocks

18. Follow the same procedure as for Block No 1.

Block No 2	(Side-1):	Figures 6.117 - 6.118
Block No 3	(Side-2):	Figures 6.119 - 6.120
Block No 4	(Bottom-1):	Figures 6.121 - 6.122
Block No 5	(Top-2):	Figures 6.123 - 6.124
Block No 6	(Side-3):	Figures 6.125 - 6.126
Block No 7	(Side-4):	Figures 6.127 - 6.128
Block No 8	(Bottom-2):	Figures 6.129 - 6.130
Block No 9	(Core-1):	Figures 6.131 - 6.132
Block No 10	(Core-2):	Figures 6.133 - 6.134
Block No 11	(Liner-1):	Figures 6.135 - 6.136
Block No 12	(Liner-2):	Figures 6.137 - 6.138
Block No 13	(Liner-3):	Figures 6.139 - 6.140
Block No 14	(Liner-4):	Figures 6.141 - 6.142

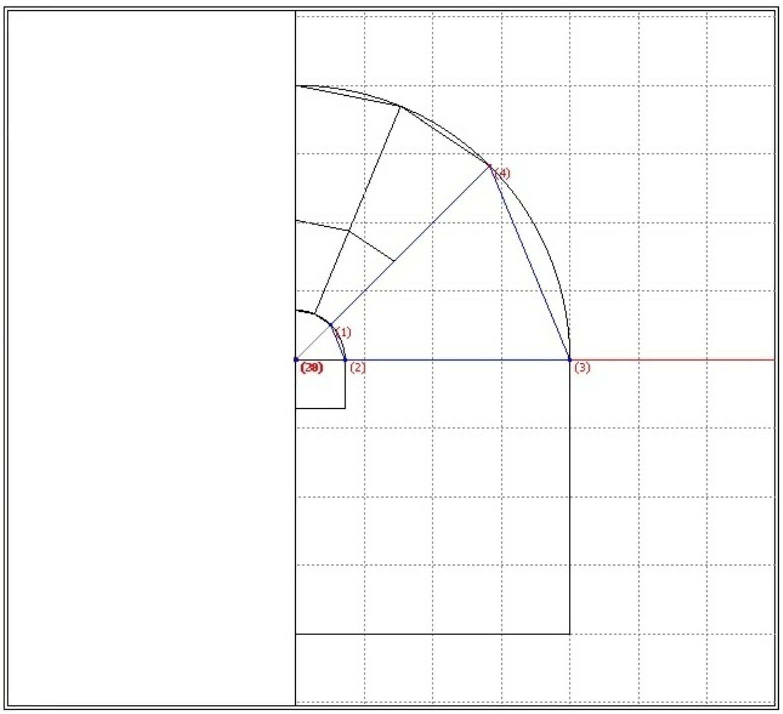


Figure 6.117 Index numbers on front surface (Block No 2)

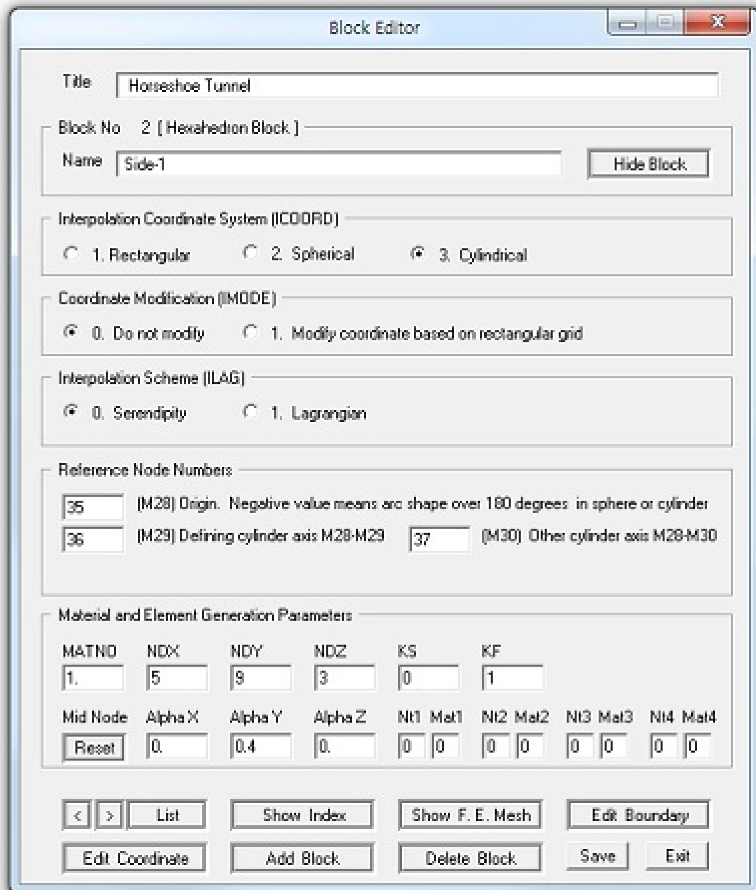


Figure 6.118 Block No 2 (Side-1)

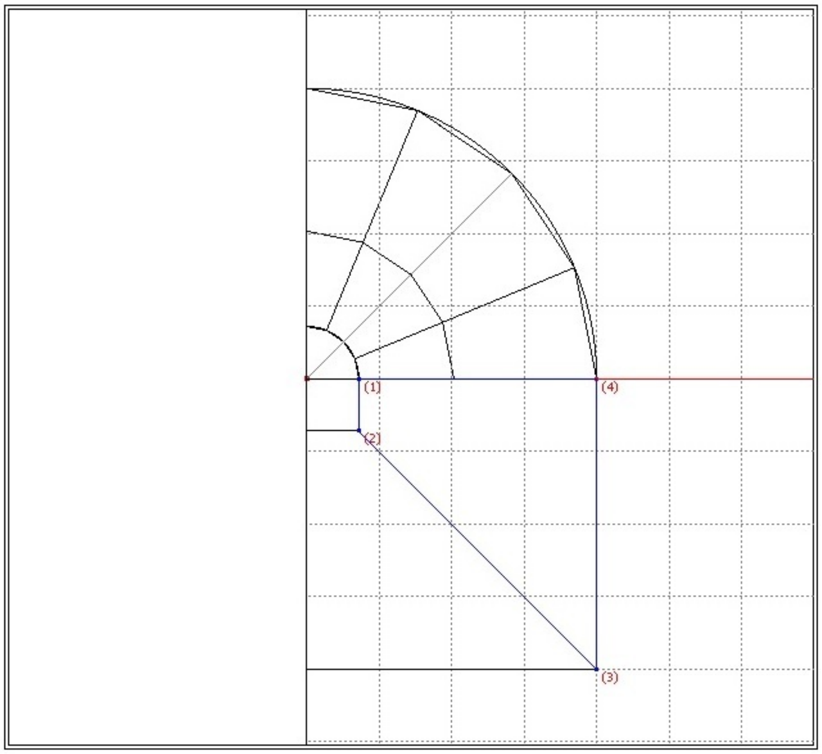


Figure 6.119 Index numbers on front surface (Block No 3)

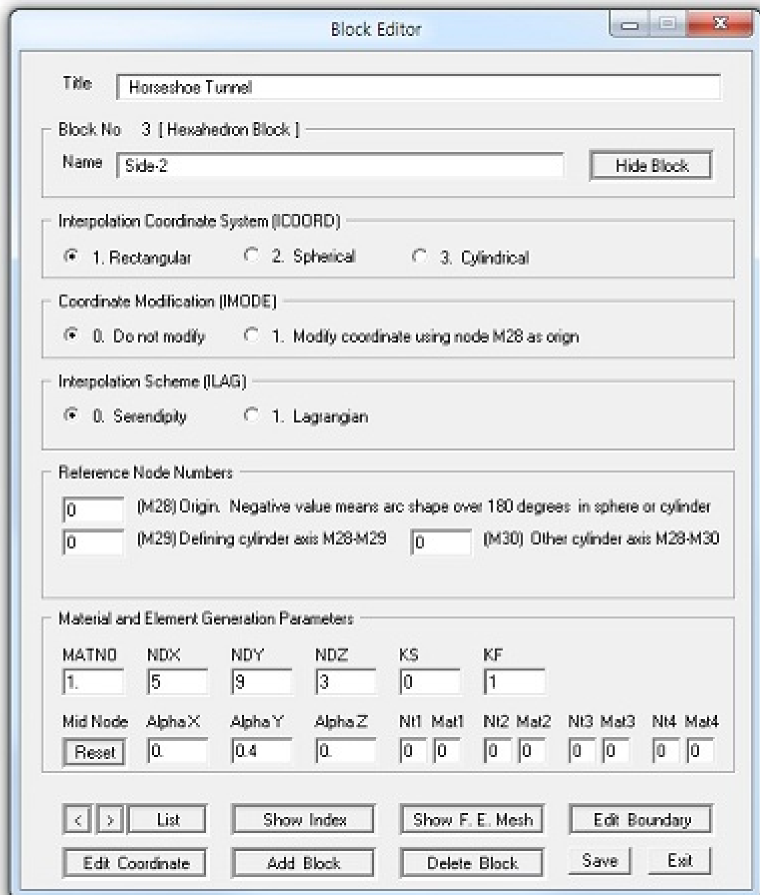


Figure 6.120 Block No 3 (Side-2)

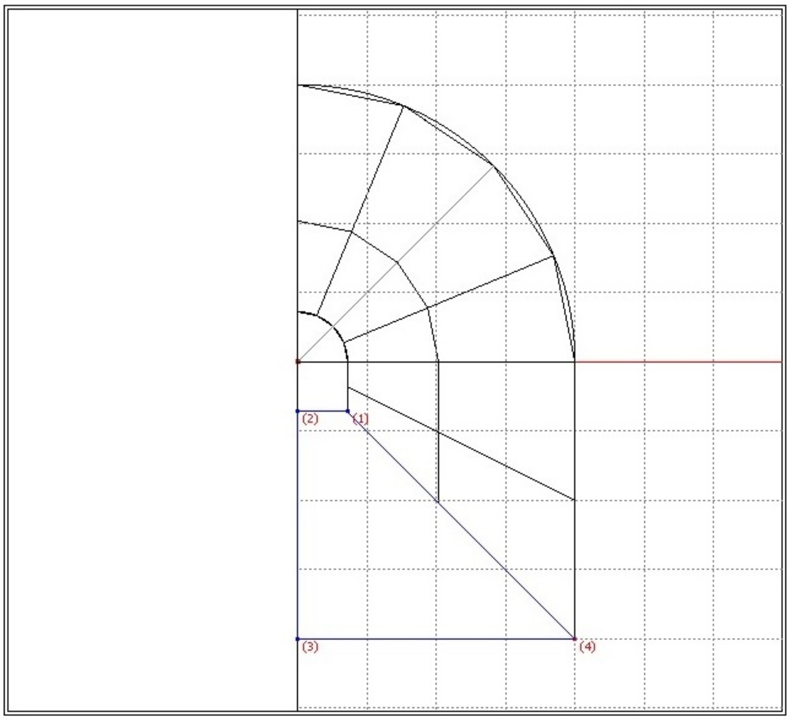


Figure 6.121 Index numbers on front surface (Block No 4)

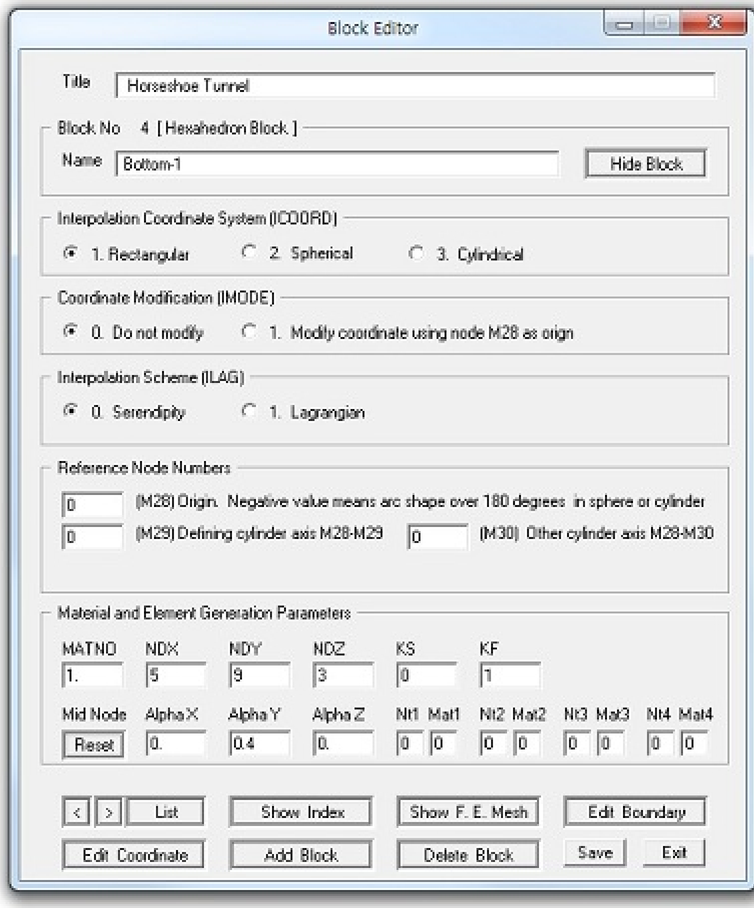


Figure 6.122 Block No 4 (Bottom-1)

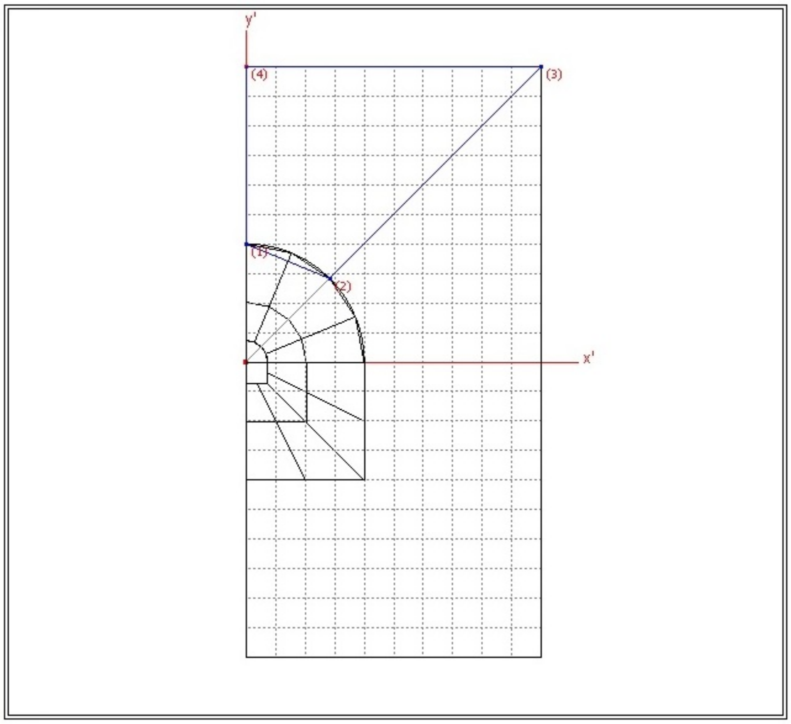


Figure 6.123 Index numbers on front surface (Block No 5)

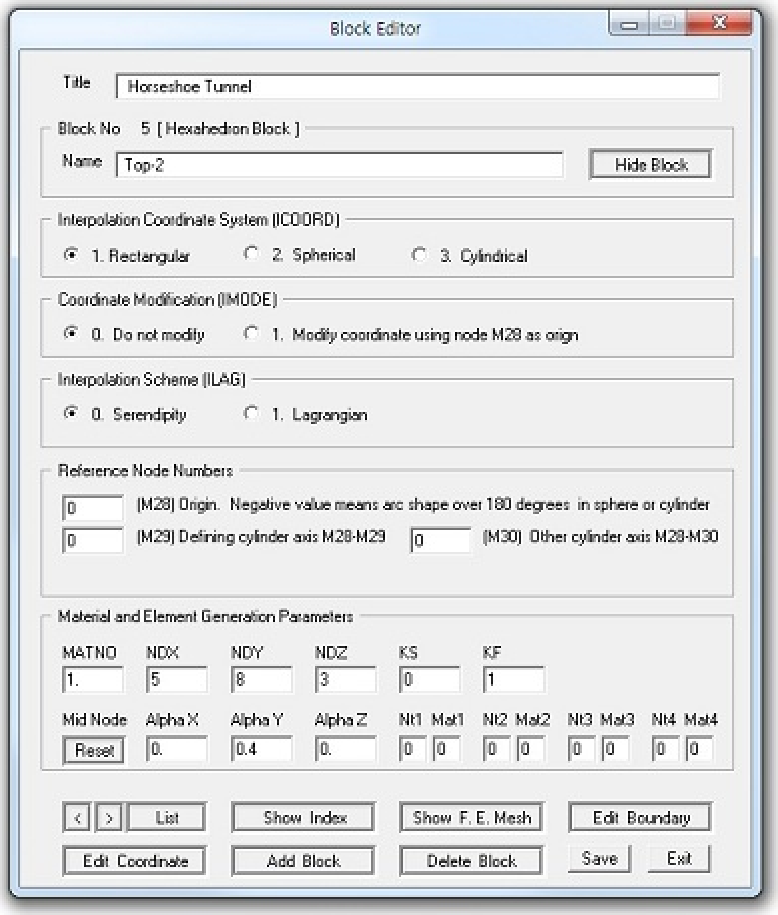


Figure 6.124 Block No 5 (Top-2)

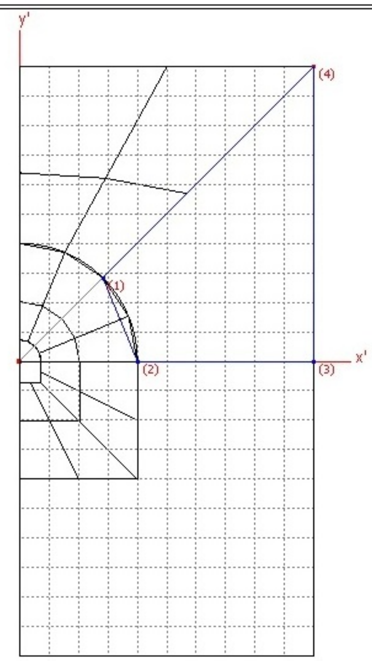


Figure 6.125 Index numbers on front surface (Block No 6)

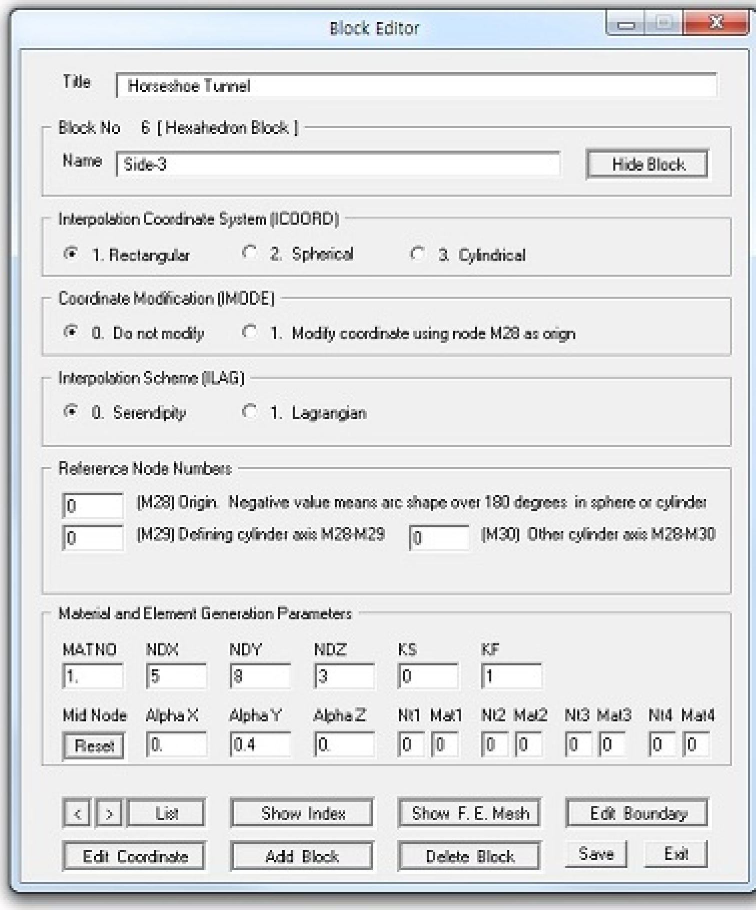


Figure 6.126 Block No 6 (Side-3)

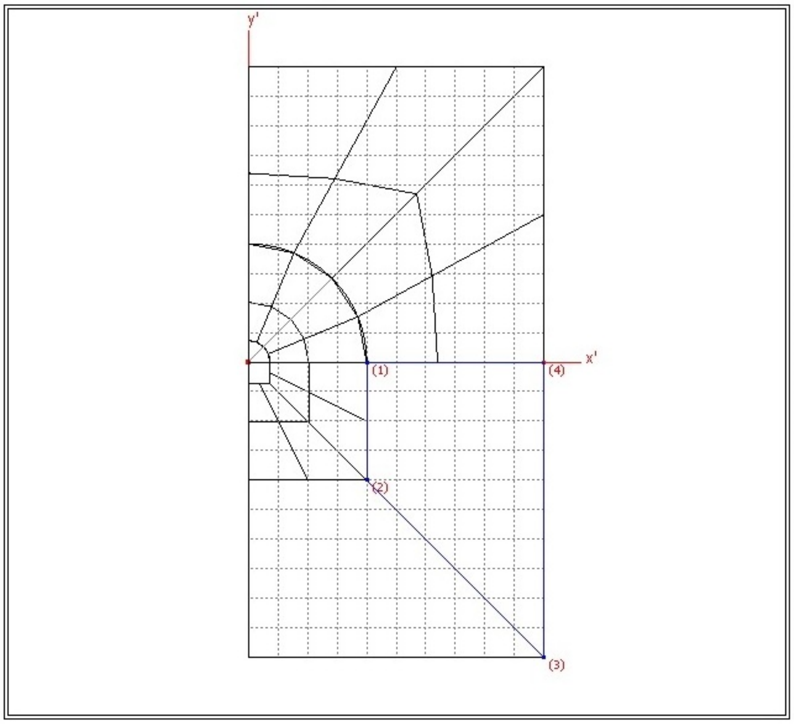


Figure 6.127 Index numbers on front surface (Block No 7)

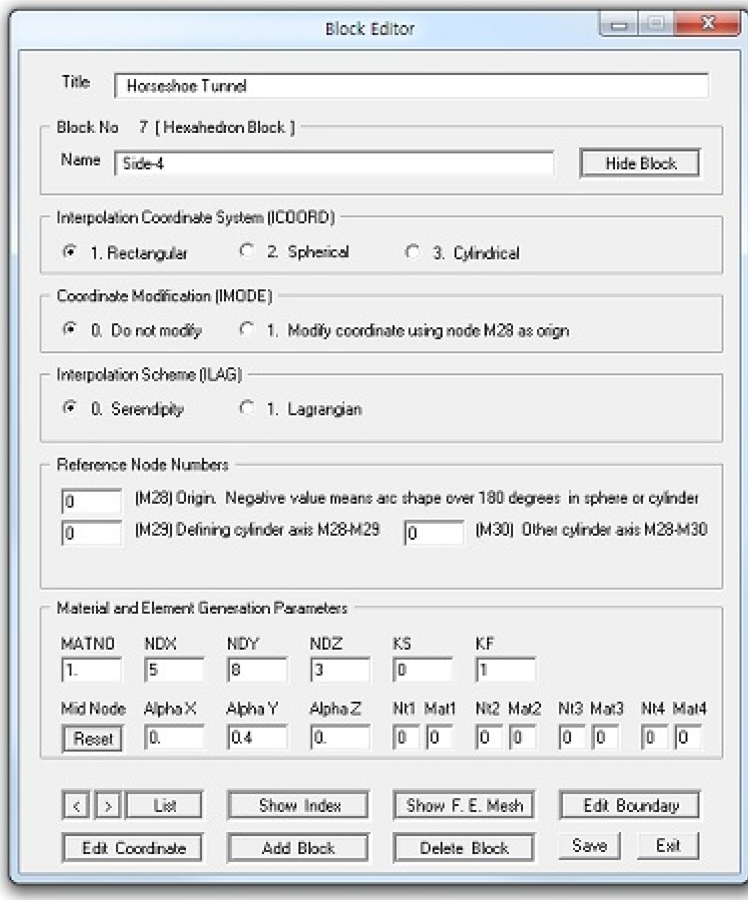


Figure 6.128 Block No 7 (Side-4)

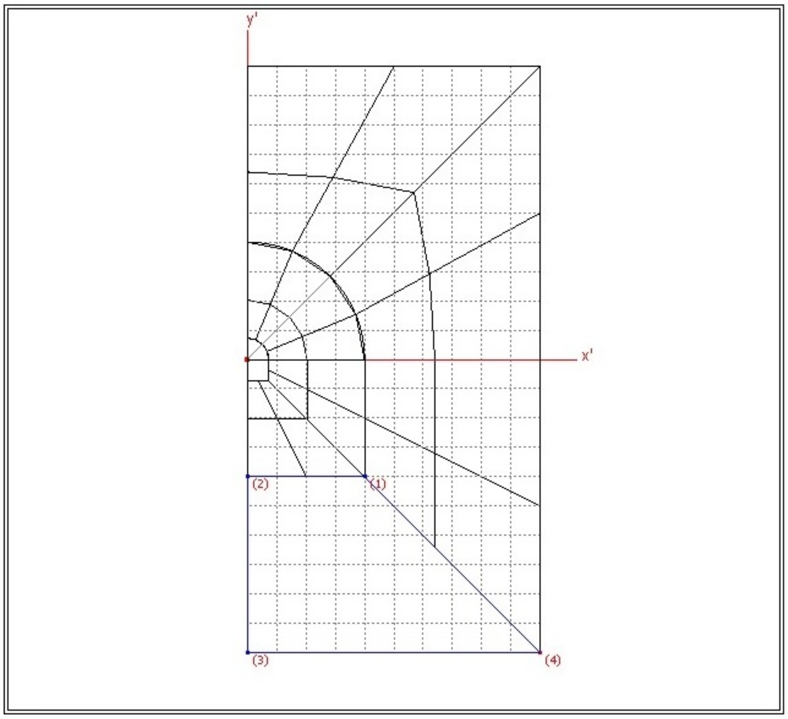


Figure 6.129 Index numbers on front surface (Block No 8)

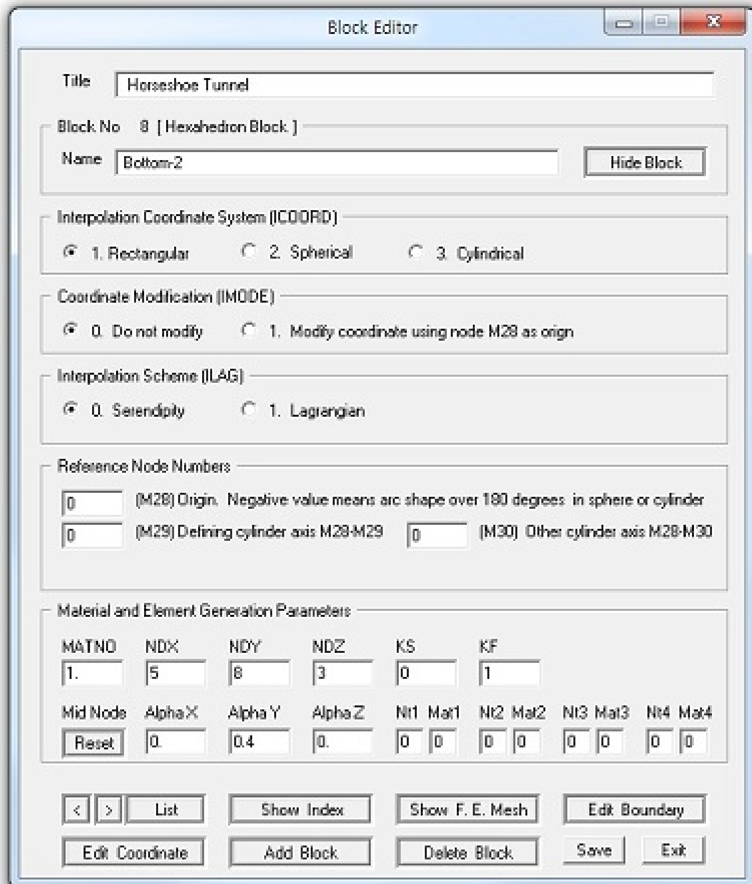


Figure 6.130 Block No 8 (Bottom-2)

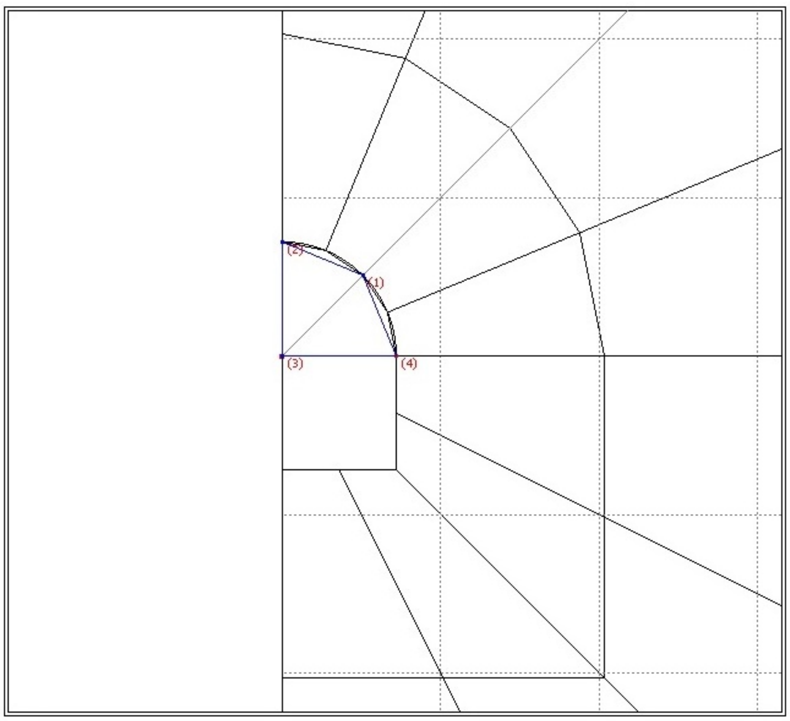


Figure 6.131 Index numbers on front surface (Block No 9)

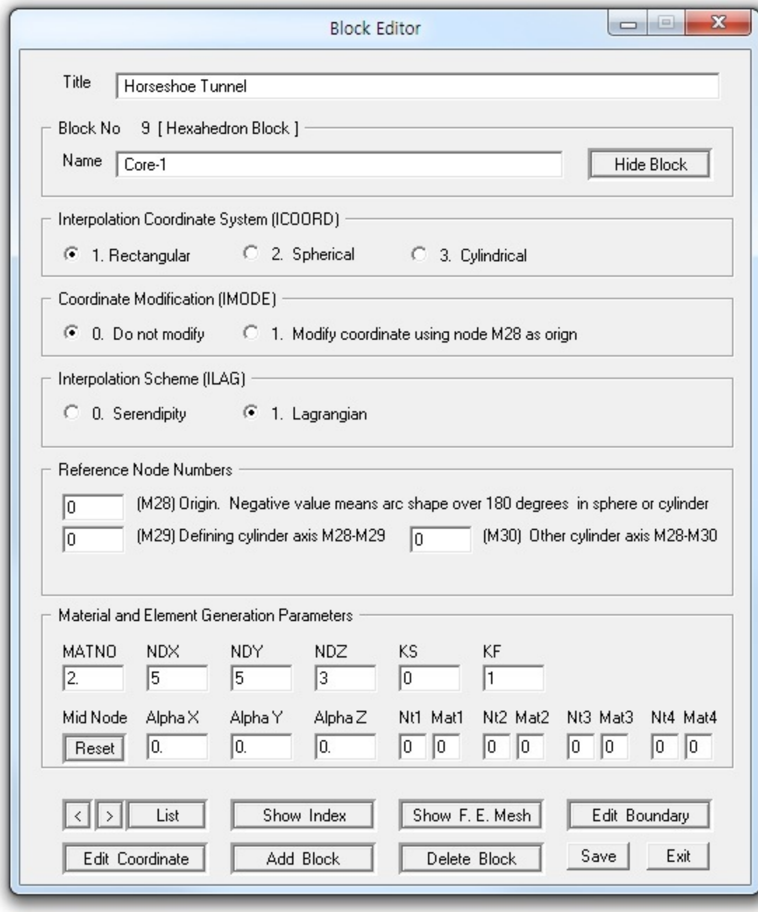


Figure 6.132 Block No 9 (Core-1)

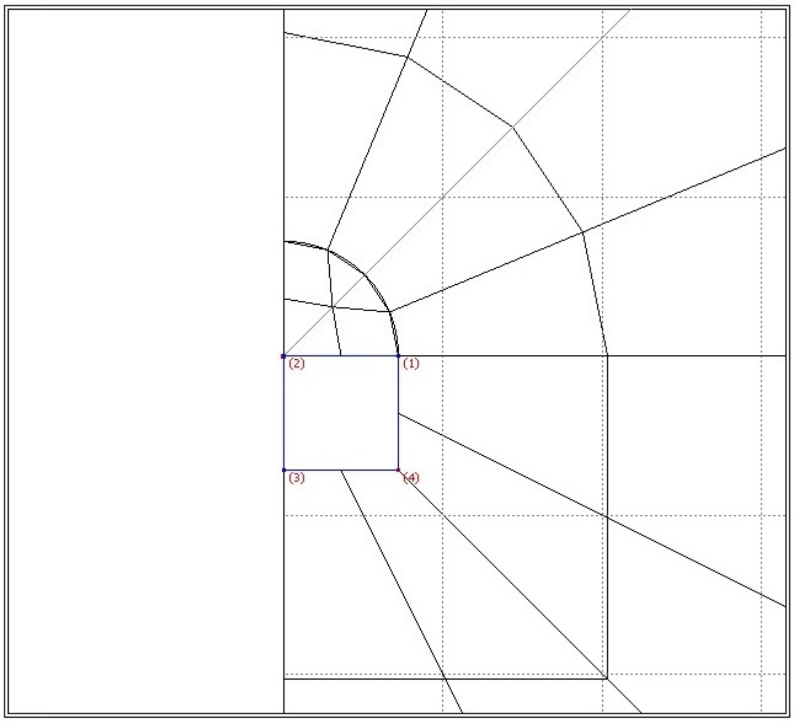


Figure 6.133 Index numbers on front surface (Block No 10)

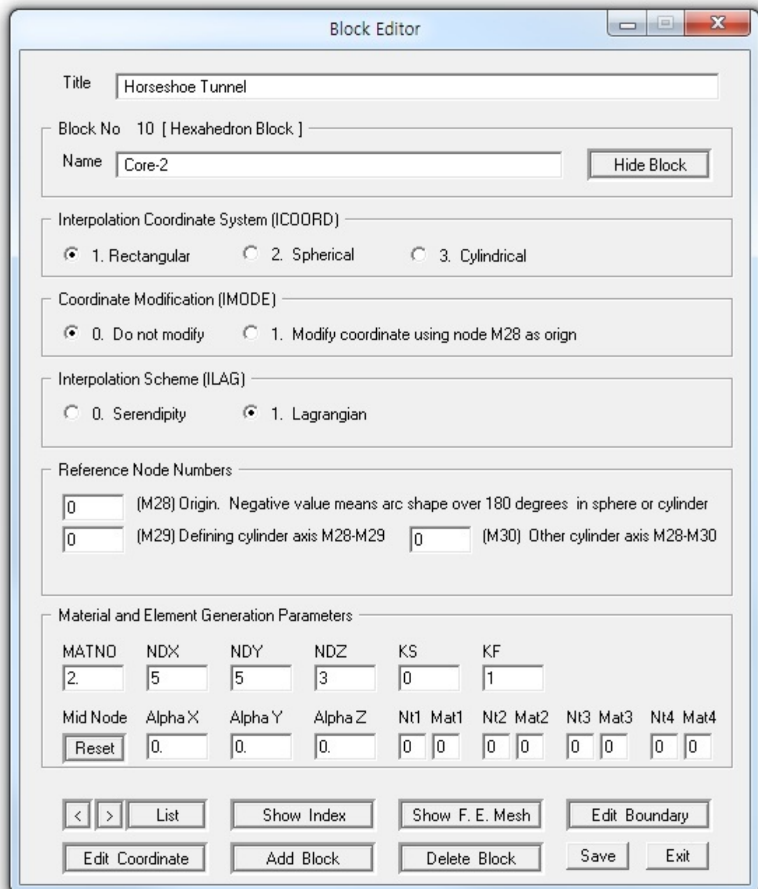


Figure 6.134 Block No 10 (Core-2)

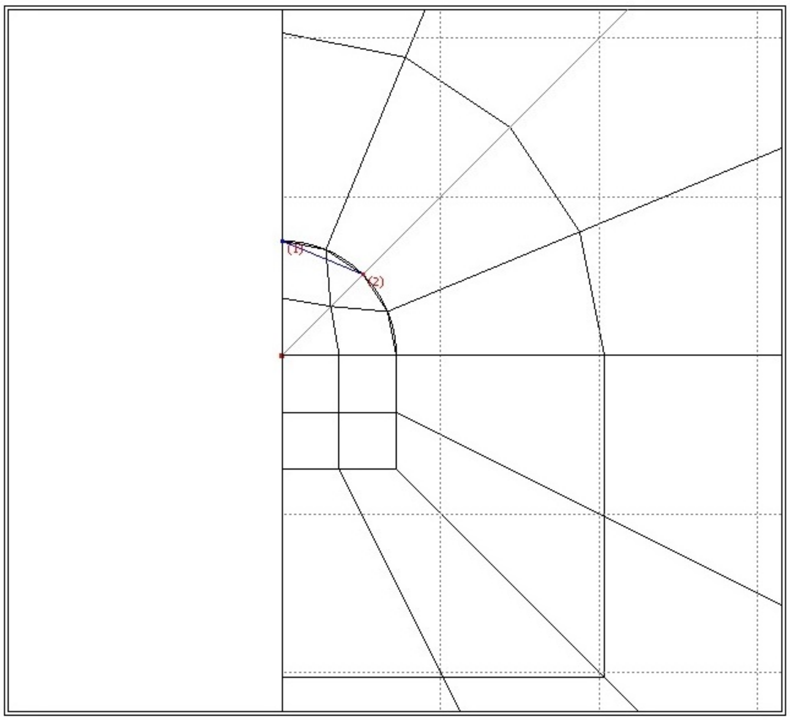


Figure 6.135 Index numbers on front surface (Block No 11)

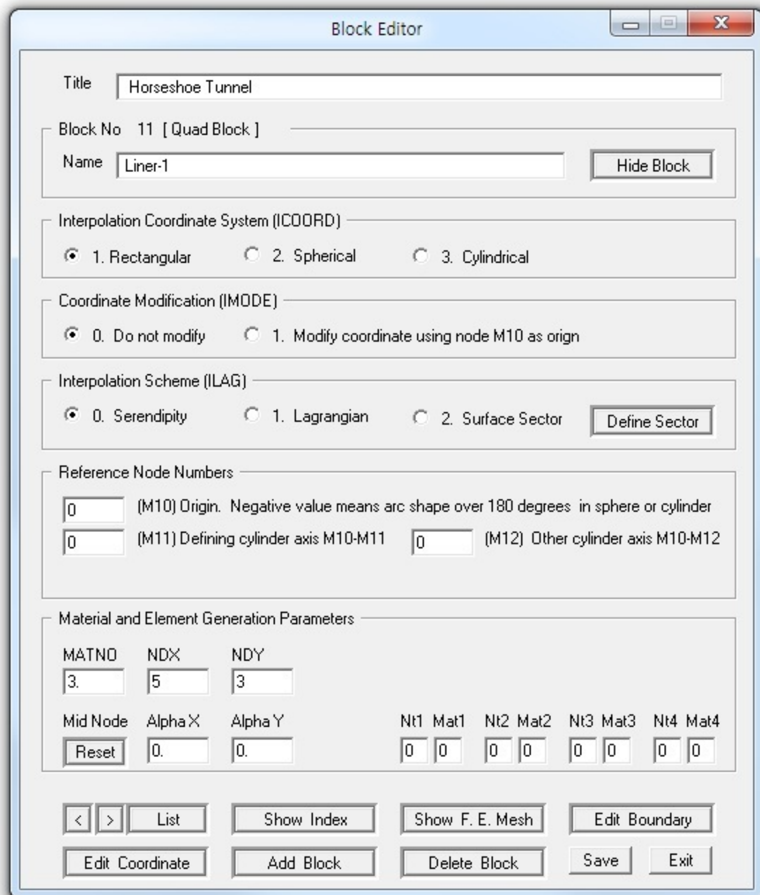


Figure 6.136 Block No 11 (Liner-1)

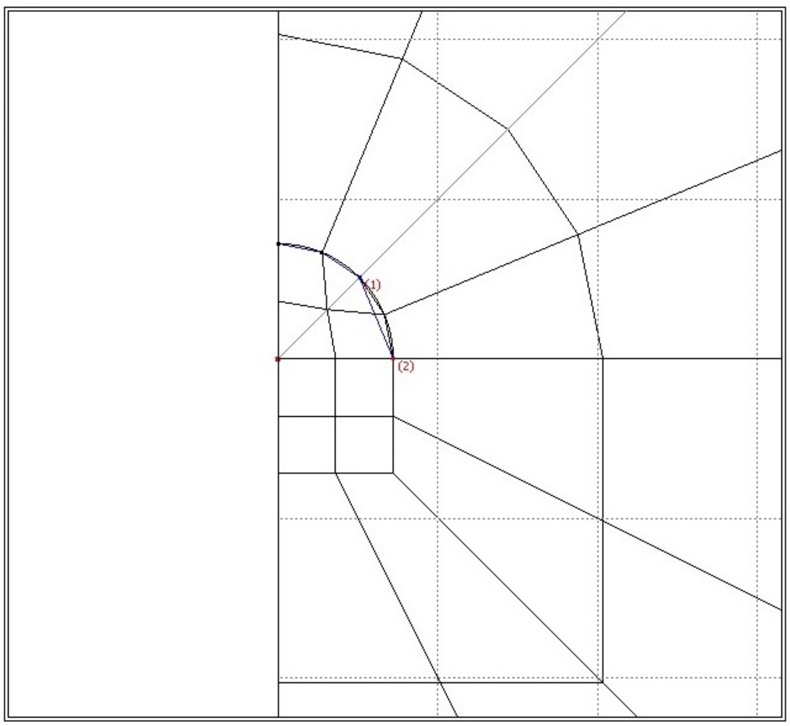


Figure 6.137 Index numbers on front surface (Block No 12)

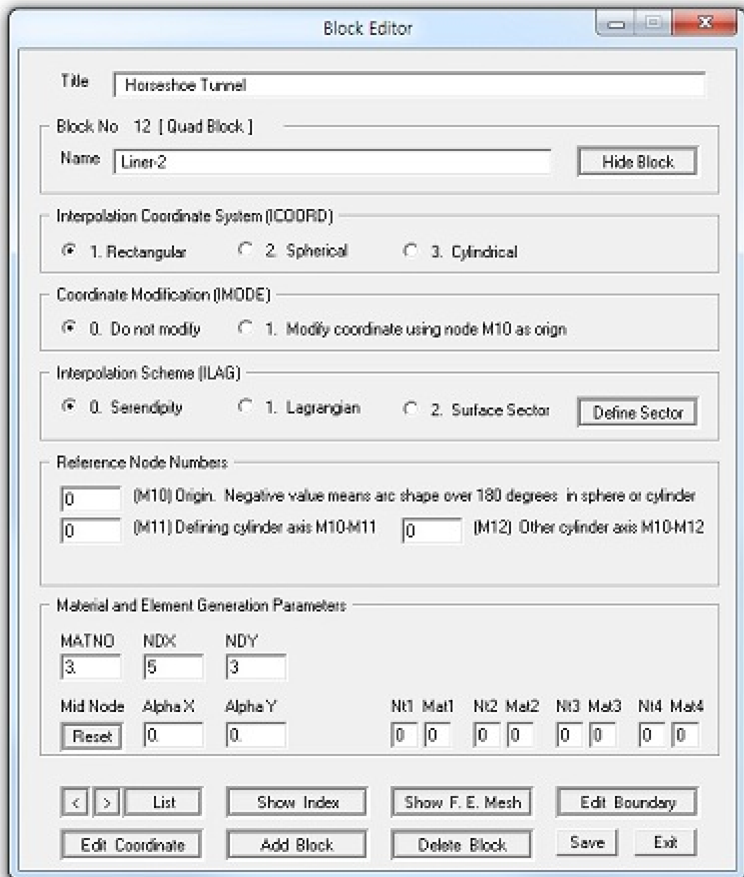


Figure 6.138 Block No 12 (Liner-2)

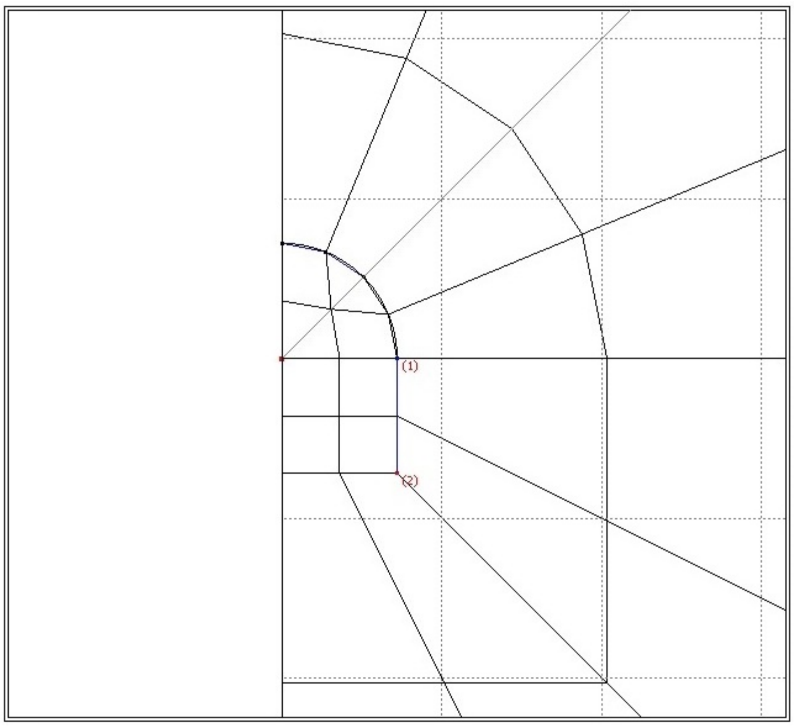


Figure 6.139 Index numbers on front surface (Block No 13)

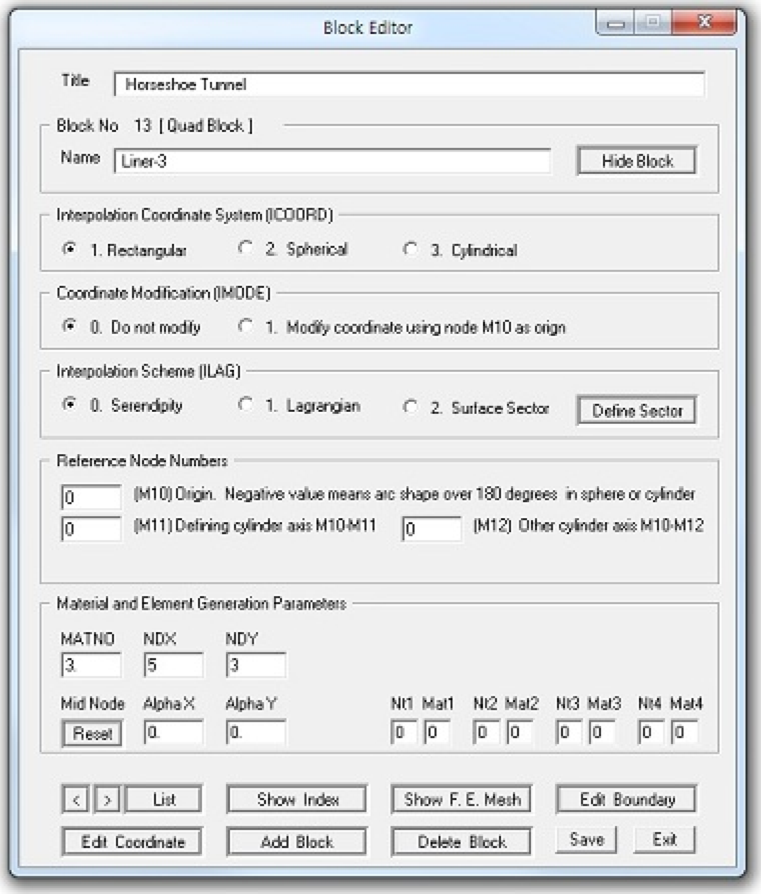


Figure 6.140 Block No 13 (Liner-3)

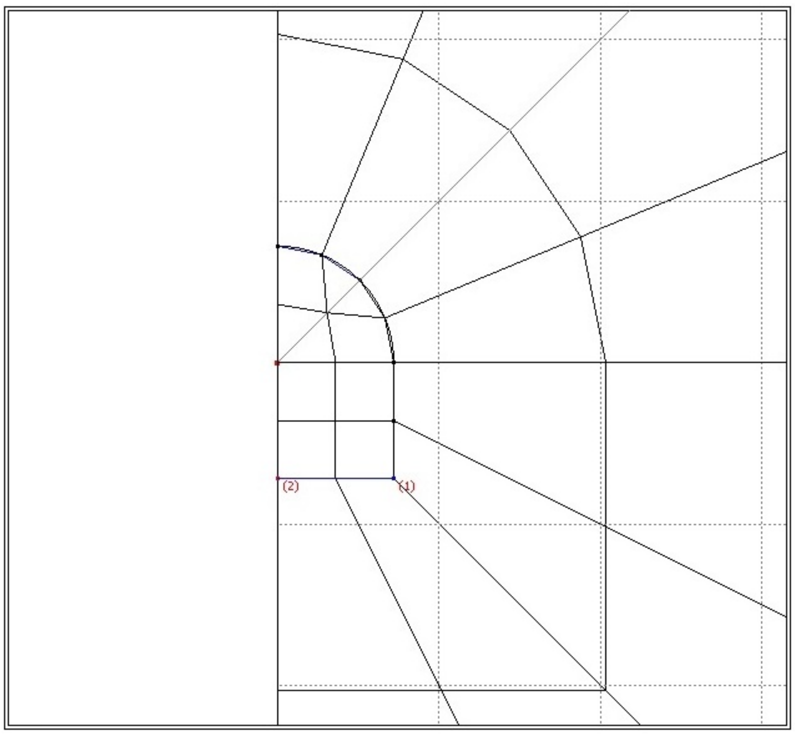


Figure 6.141 Index numbers on front surface (Block No 14)

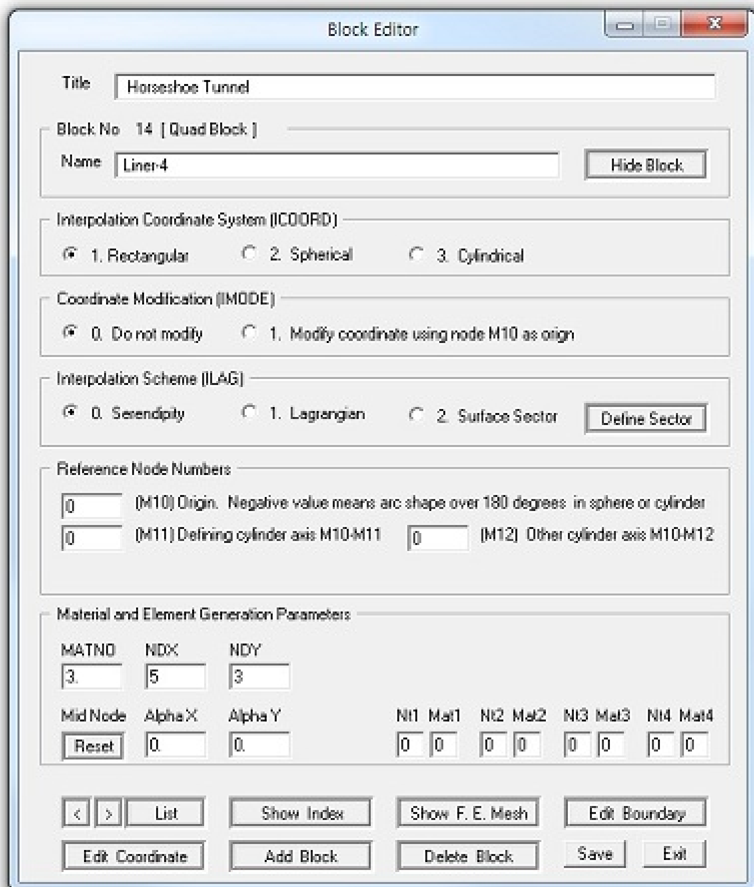


Figure 6.142 Block No 14 (Liner-4)

19. All blocks are listed as shown in Figure 6.143 by clicking List button in the Block Editor dialog.
20. Click OK.

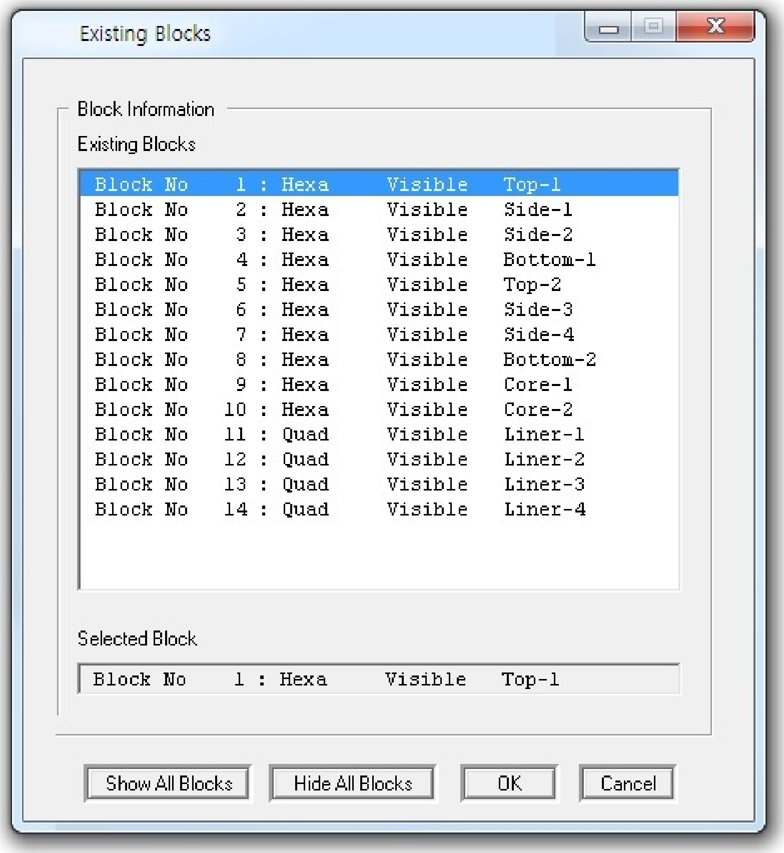


Figure 6.143 Listing of all of the blocks

Step 6: Set Global Boundary

1. Select Model → Edit Global Boundary in Figure 6.144.

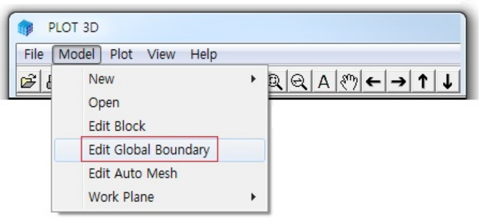


Figure 6.144 Edit global boundary menu

2. Set the boundary codes as shown in Figures 6.145 and 6.146.
3. Select Yes override block boundary.
4. Click Save and type in file name as EX4.

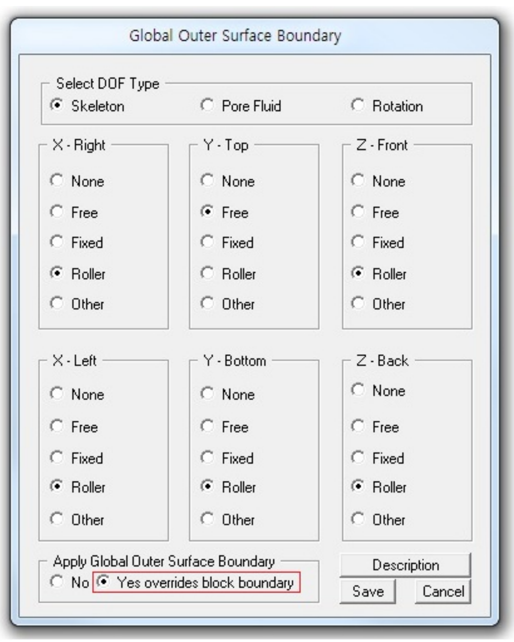


Figure 6.145 Global outer surface boundary (Skeleton)

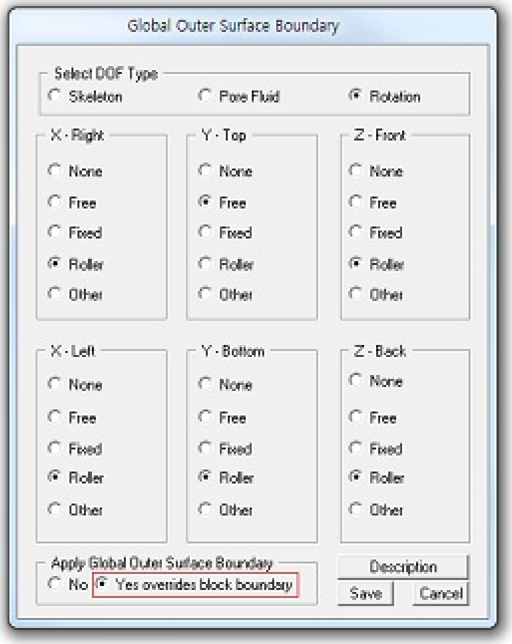


Figure 6.146 Global outer surface boundary (Rotation)

Step 7: View Selected Material

1. Select View → Mesh in PLOT-3D menu.
2. Select Only Selected One for Material Selection in Figure 6.147.
3. Click Number 3 in Available list.
4. Click OK.

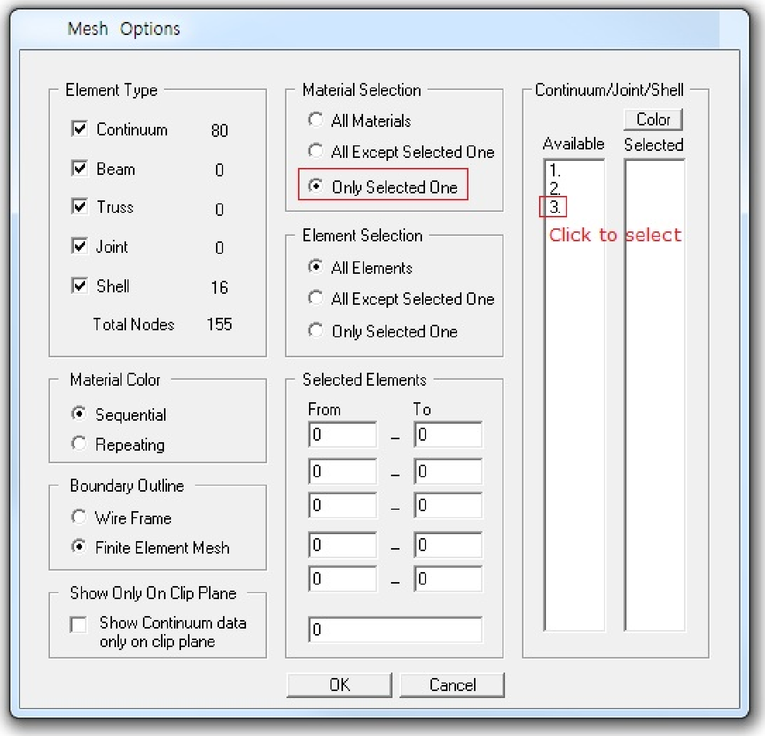


Figure 6.147 Mesh options

5. Figure 6.148 shows selected block meshes with material number 3 which represents tunnel lining.

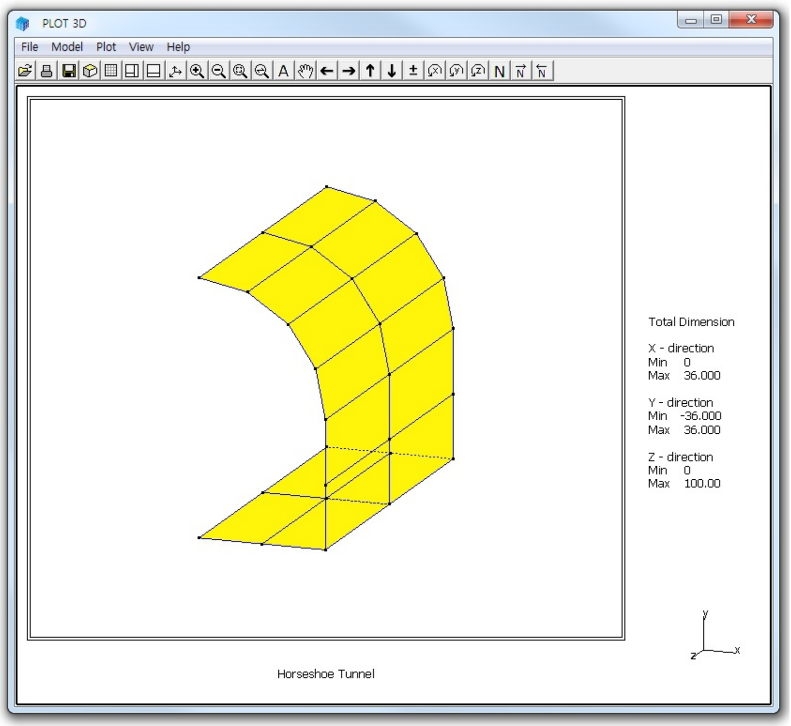


Figure 6.148 Block meshes representing tunnel lining

Step 8: Plot Finite Element Mesh

1. Click Show F. E. Mesh in Block Editor dialog.
2. Select Model → Work Plane → Isometric Y-axis in PLOT-3D menu.
3. Figure 6.149 shows finite element mesh.

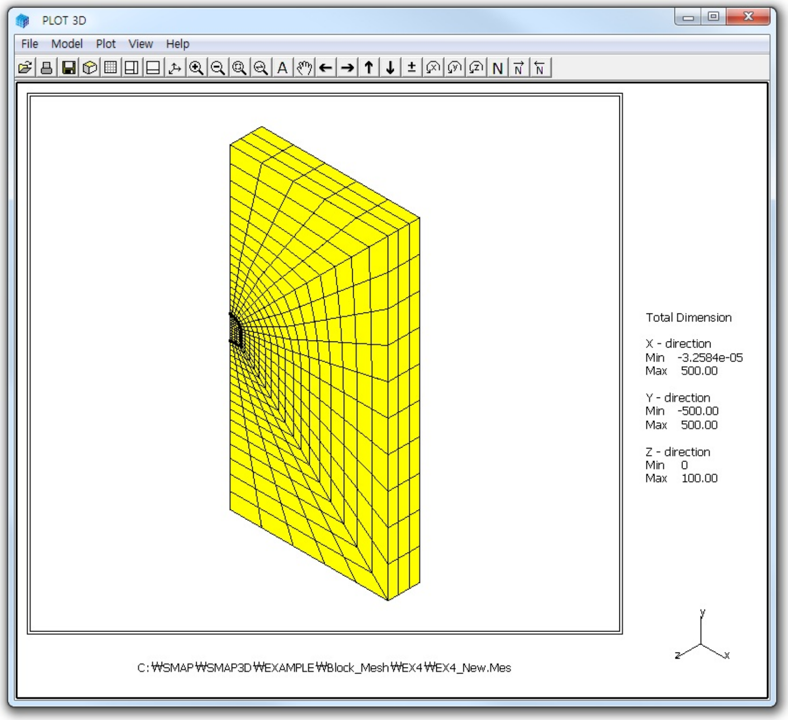


Figure 6.149 Finite element mesh

4. Follow the same procedure to view selected material as in Step 6.
5. Figure 6.150 shows finite element mesh with material number 3 which represents tunnel lining.

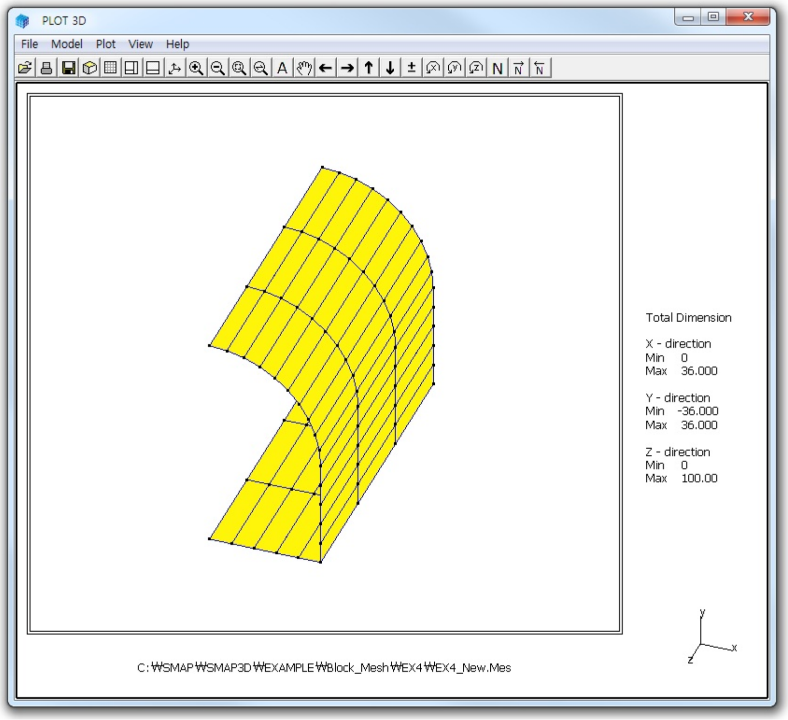


Figure 6.150 Finite element mesh representing tunnel lining

6.5 Space Truss

This example illustrates how to build directly finite element meshes for the space truss as schematically shown in Figure 6.151.

This example involves following nine main steps:

1. Access finite element mesh generator
2. Set work plane
3. Build cube entity
4. Add work plane
5. Build truss elements
6. Edit mesh title
7. Plot node and element numbers
8. Edit boundary codes
9. Plot skeleton boundary codes

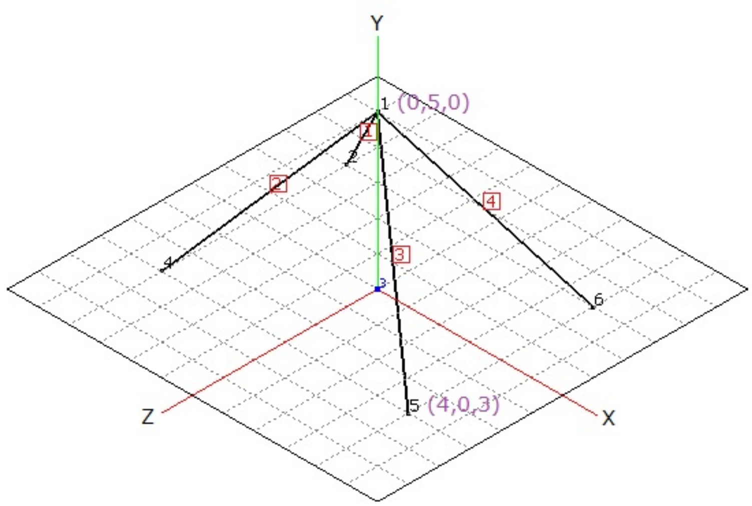


Figure 6.151 Space truss consisting of four elements

Step 1: Access Finite Element Mesh Generator

Access Finite Element Mesh Generator by selecting the following menu items in SMAP as in Figure 6.152:

Plot → Mesh → F. E. Mesh → New

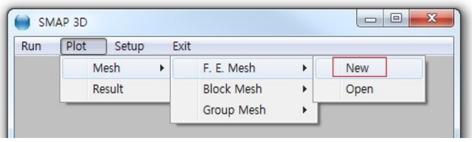


Figure 6.152 Access finite element mesh generator

Step 2: Set Work Plane

1. Select Work Plane No 4 and set parameters as shown in Fig. 6.153.
2. Click Update button.

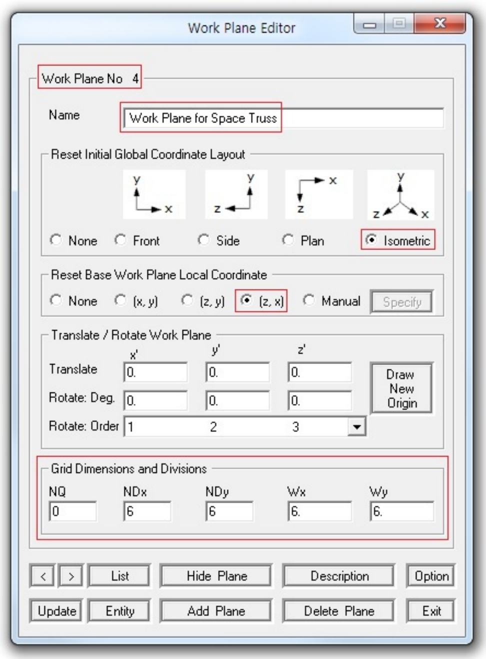


Figure 6.153 Work plane editor

3. Figure 6.154 shows work plane with Isometric Y-axis.

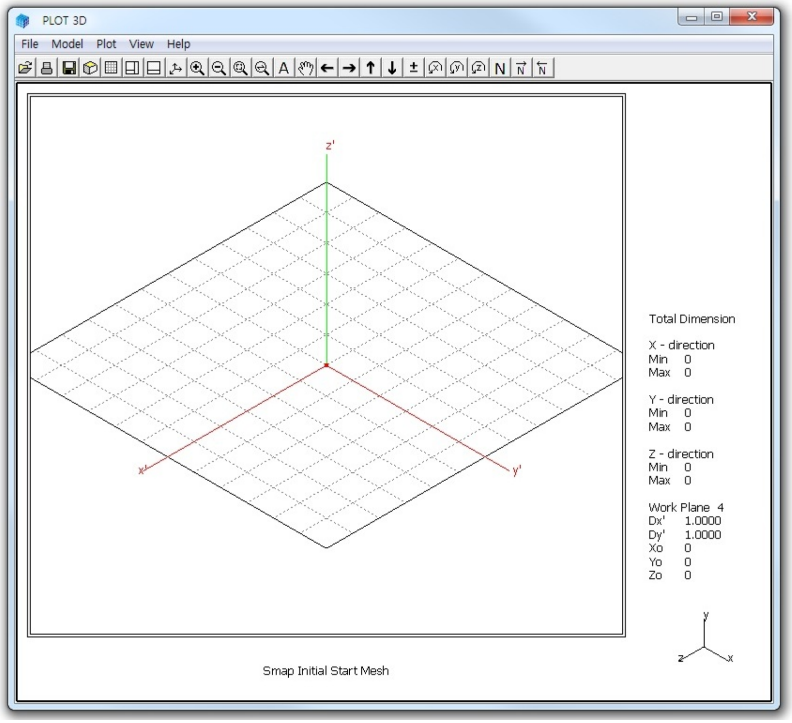


Figure 6.154 Work plane with Isometric Y-axis

Step 3: Build Cube Entity

1. Click Entity in Figure 6.153.
2. Click Add in Entity Editor dialog in Figure 6.155.

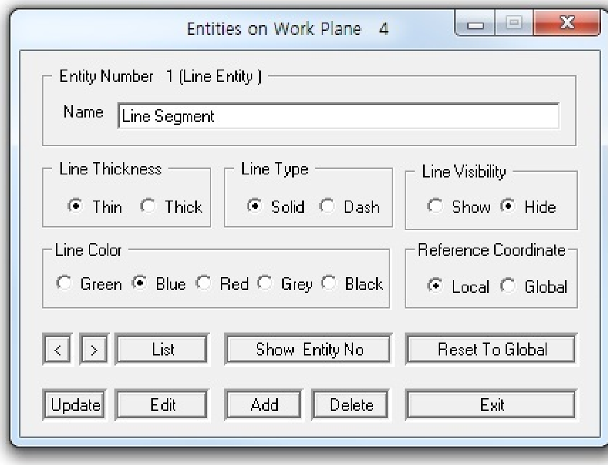


Figure 6.155 Entity editor

3. Select Cube in Entity Type Selection dialog in Figure 6.156.
4. Click OK.

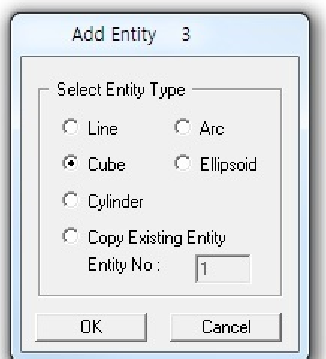


Figure 6.156 Entity type selection

5. Set geometric parameters of cube entity as shown in Figure 6.157.
6. Click Draw Cube Entity.
7. Click Finish.

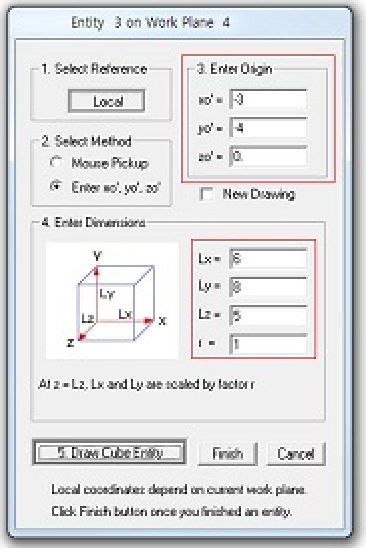


Figure 6.157 Cube entity

8. Set parameters of cube entity as shown in Figure 6.158.
9. Click Reset To Global and then click Exit.

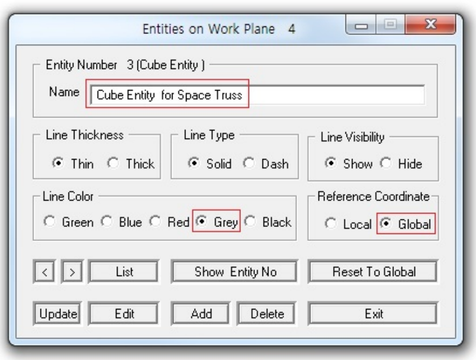


Figure 6.158 Entity editor

10. Figure 6.159 shows cube entity on drawing board.

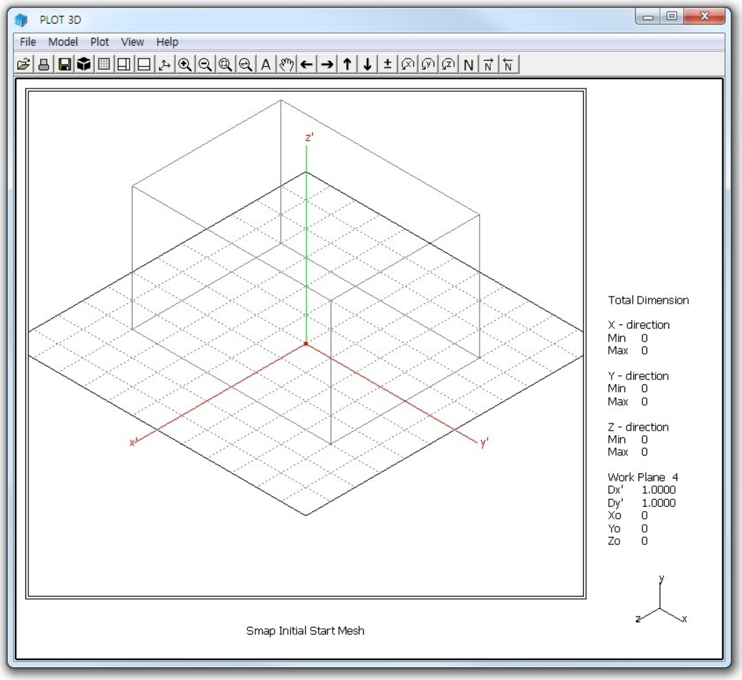


Figure 6.159 Cube entity on drawing board

Step 4: Add Work Plane

At Step 2, we set Work Plane No 4 which represents bottom surface. At Step 3, we built cube entity on this Work Plane No 4.

Here, we want to add new Work Plane No 5 in the following way:

- Copy Work Plane No 4 along with cube entity on it.
- Add this copied one as new Work Plane No 5.
- Modify such that it represents top surface.

Once we have this new Work Plane No 5, it will be much easier to build blocks since top and bottom surfaces of work planes can be accessed simply by one click of Back or Next button on Coordinates on Work Plane dialog in Figure 6.160.

Perform the following four steps:

1. Select Work Plane No 4 in Work Plane Editor dialog in Figure 6.153
2. Click Add Plane button in Figure 6.153
3. Modify Name and Translation as in Figure 6.161
4. Click Update in Figure 6.161

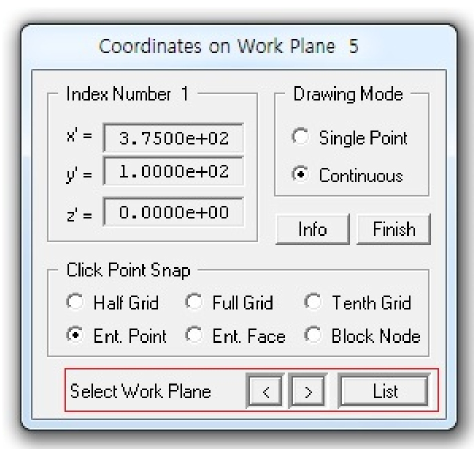


Figure 6.160 Coordinates on work plane

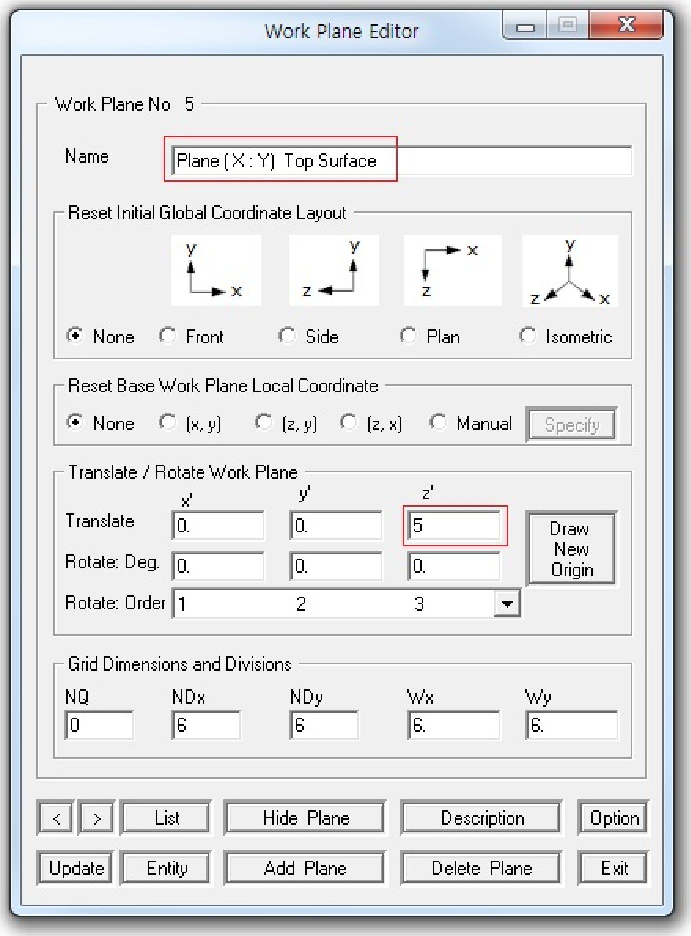


Figure 6.161 Work Plane No 5 representing top surface

Step 5: Build Truss Elements

Build Element 1

1. Click Edit Element toolbar in Figure 6.162.

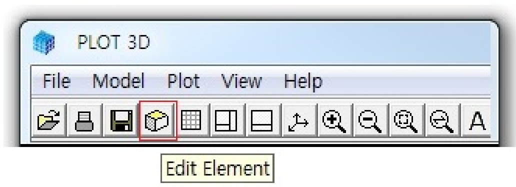


Figure 6.162 Edit element toolbar

Draw Index Numbers For Element 1

2. Select Line for element type in Figure 6.163.
3. Click OK.

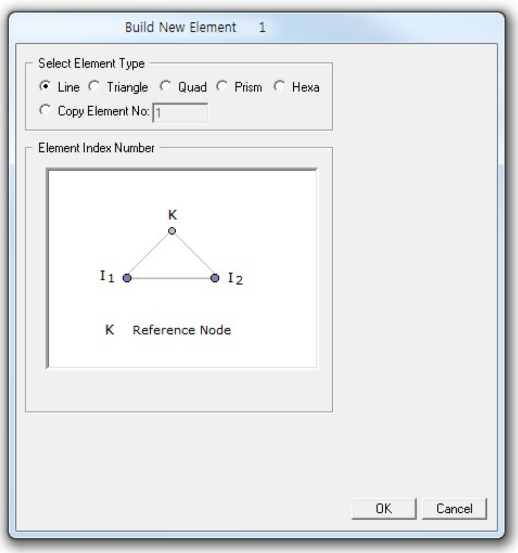


Figure 6.163 Element type selection

4. Type in 1 for MatNo and 1 for truss element as in Figure 6.164.
5. Click Draw Index Number.

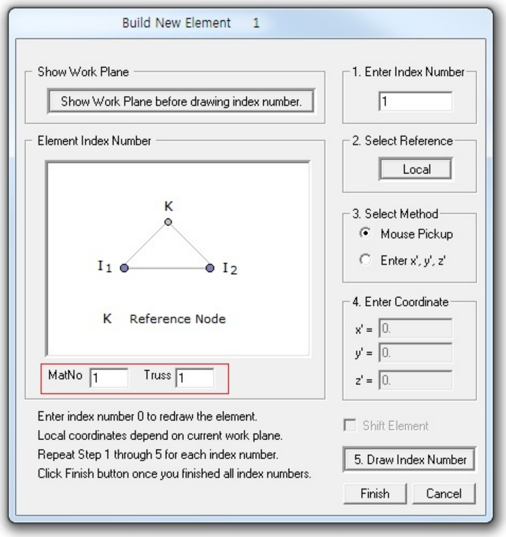


Figure 6.164 Line element

6. Select Full Grid for Click Point Snap as in Figure 6.165.

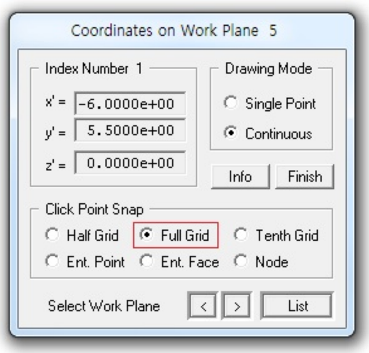


Figure 6.165 Coordinates on work plane

7. Select Work Plane No 5 for top surface using Back / Next button in Figure 6.165.
8. Click the point for index number 1 as in Figure 6.166.

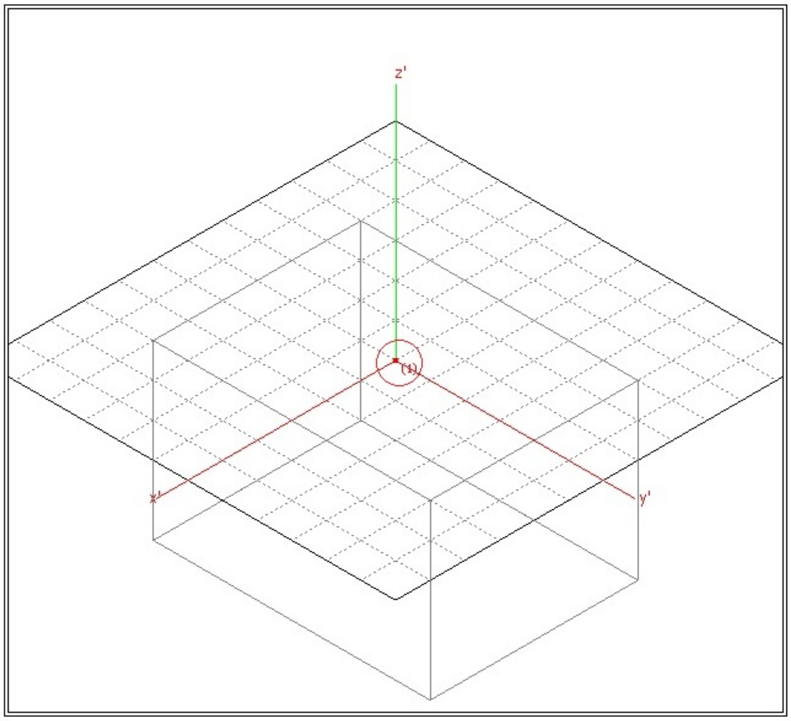


Figure 6.166 Index number 1 for Element 1

9. Select Work Plane No 4 for bottom surface using Back / Next button in Figure 6.165.
10. Click the points for index numbers 2 and 3 as in Figure 6.167.
11. Click Finish in Figure 6.165 and then click Finish in Figure 6.164.

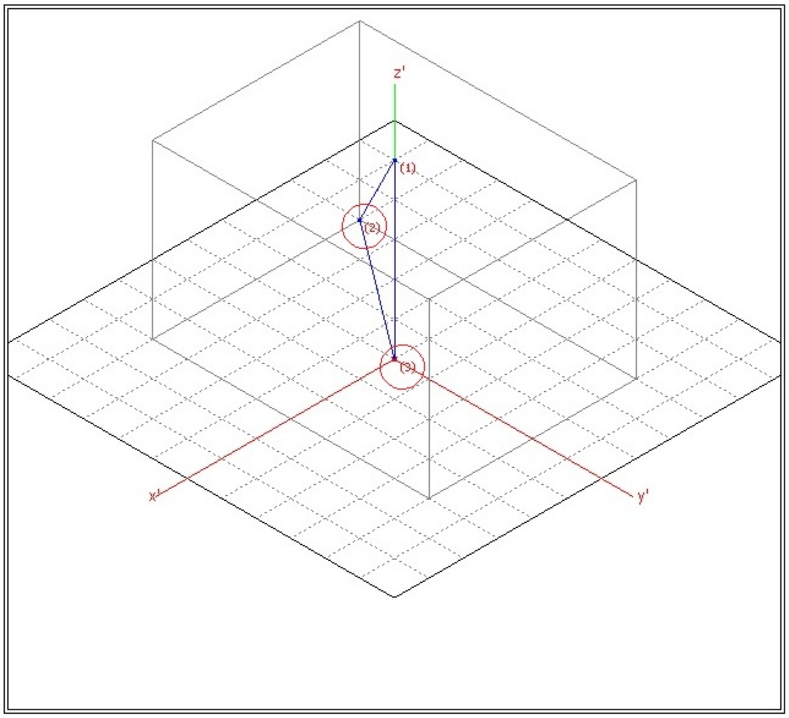


Figure 6.167 Index numbers 2 and 3 for Element 1

Build Element 2

12. Select Model → Edit Element in Figure 6.168.

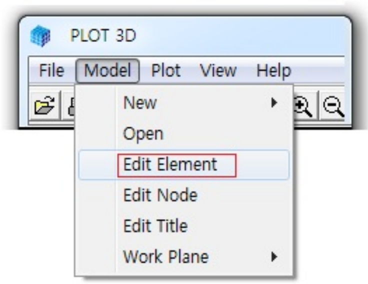


Figure 6.168 Edit element menu

13. Popup menu in Figure 6.169 is displayed by Shift + Right click.
14. Click Add menu.

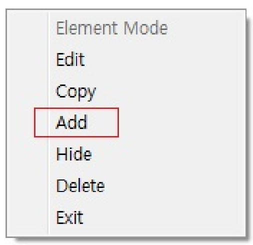


Figure 6.169 Popup menu for edit element

Draw Index Numbers For Element 2

15. Repeat steps 2 through 11 for Element 2 with MatNo = 2.
16. Figure 6.170 shows index numbers for Element 2.

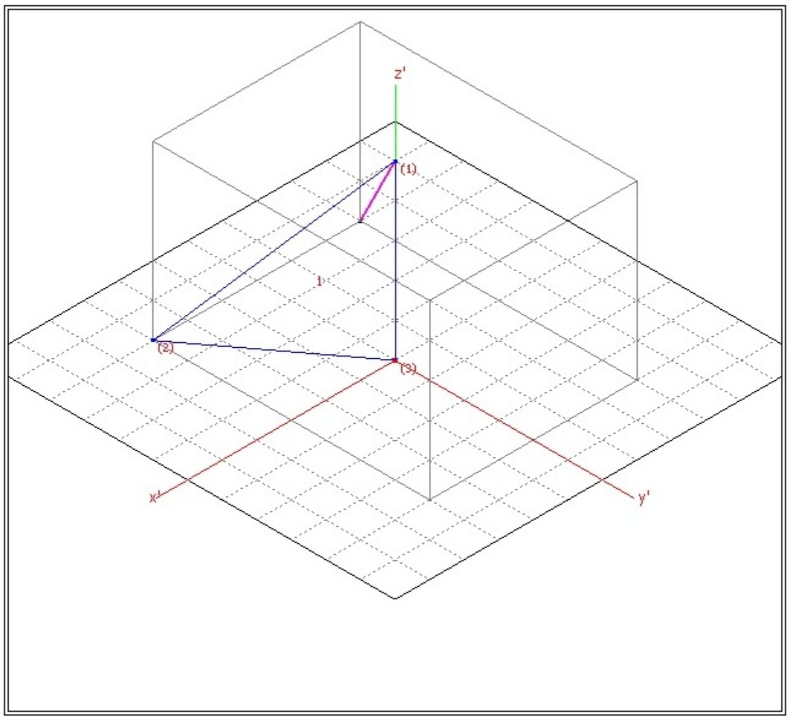


Figure 6.170 Index numbers for Element 2

Build Element 3

17. Get Popup menu in Figure 6.169 by Shift + Right click.
18. Click Add menu.

Draw Index Numbers For Element 3

19. Repeat steps 2 through 11 for Element 3 with MatNo = 3.
20. Figure 6.171 shows index numbers for Element 3.

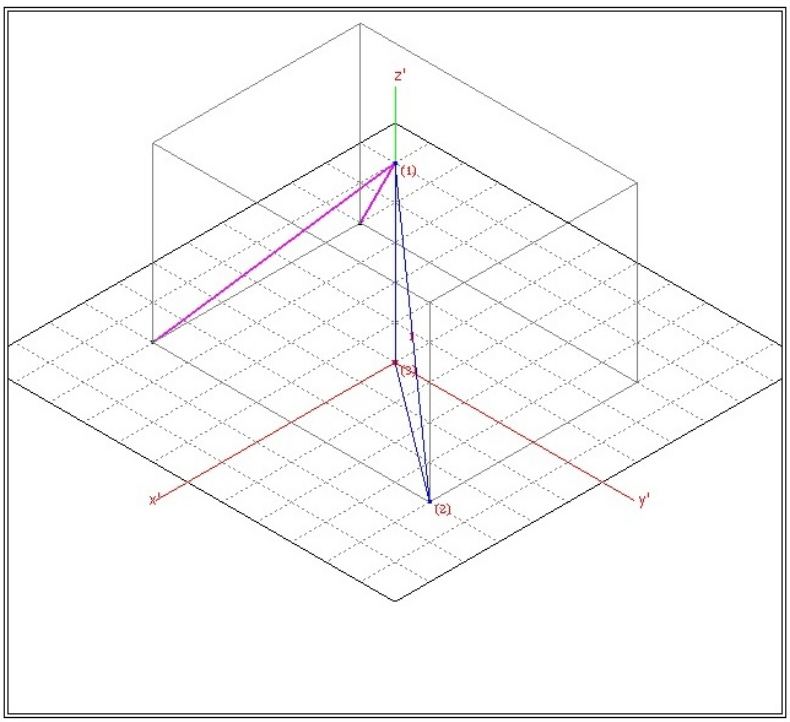


Figure 6.171 Index numbers for Element 3

Build Element 4

21. Get Popup menu in Figure 6.169 by Shift + Right click.
22. Click Add menu.

Draw Index Numbers For Element 4

23. Repeat steps 2 through 11 for Element 4 with MatNo = 4.
24. Figure 6.172 shows index numbers for Element 4.

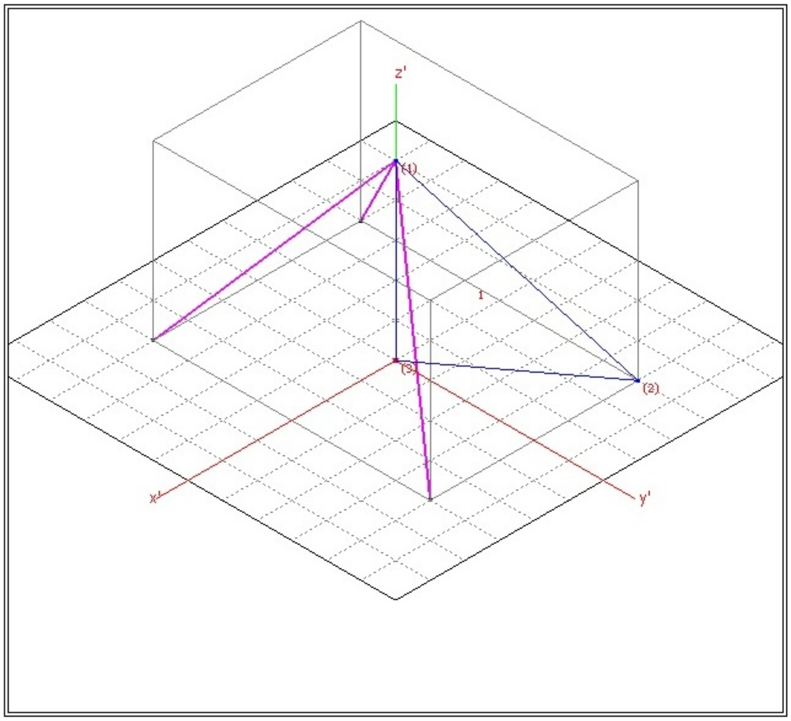


Figure 6.172 Index numbers for Element 4

Step 6: Edit Mesh Title

1. Select Model → Edit Title in Figure 6.168.
2. Type in new title in Mesh Title Editor dialog in Figure 6.173.
3. Click OK.

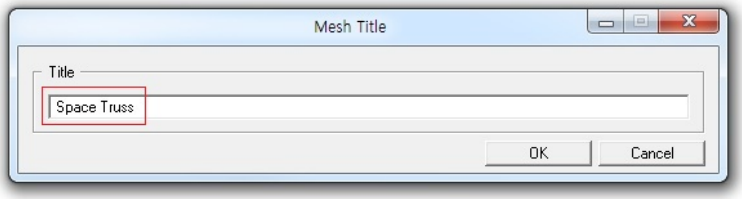


Figure 6.173 Mesh title editor

4. Click Save toolbar in Figure 6.174 and type file name as EX5.

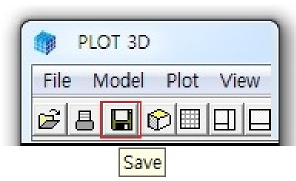


Figure 6.174 Save file toolbar

Step 7: Plot Node and Element Numbers

1. Click Show Numbers toolbar in Figure 6.175.

Figure 6.175 Show numbers toolbar

3. Click Next Numbers toolbar three times in Figure 6.176.

Figure 6.176 Next numbers toolbar

4. Fig 6.177 shows finite element mesh with node & element numbers.

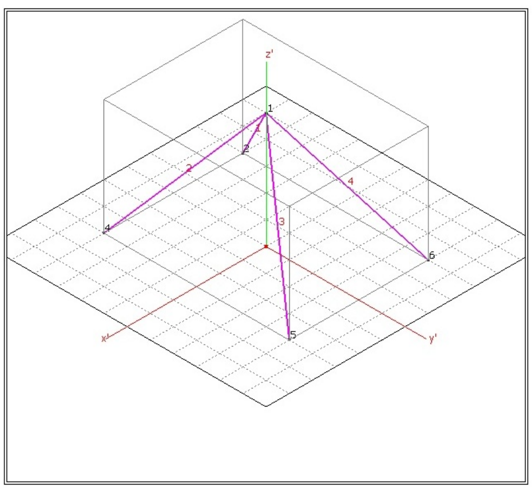


Figure 6.177 Node & element numbers

Step 8: Edit Boundary Codes

1. Select Model → Edit Node in Figure 6.178.

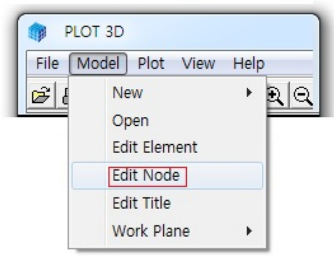


Figure 6.178 Edit node menu

2. Popup menu in Figure 6.179 is displayed by Shift + Right click.
3. Click Boundary menu.

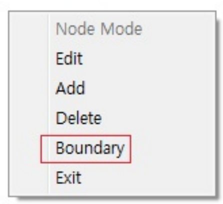


Figure 6.179 Popup menu for edit node

4. Set the boundary codes for Node 1 as shown in Figure 6.180.
5. Click Update button.

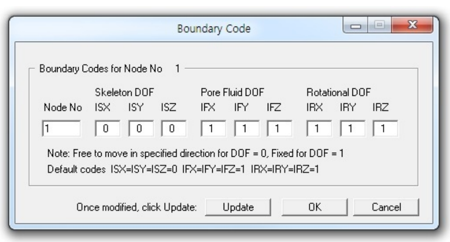


Figure 6.180 Boundary codes for Node 1

6. Set the boundary codes for Node 2 as shown in Figure 6.181.
7. Click Update button.

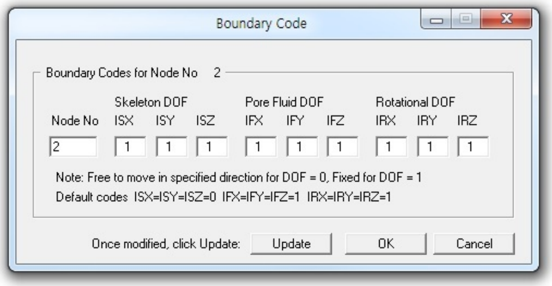


Figure 6.181 Boundary codes for Node 2

8. Repeat steps 6 and 7 for Nodes 3, 4, 5 and 6.
9. Click OK button.
10. Click Save toolbar in Figure 6.174.

Step 9: Plot Skeleton Boundary Codes

1. Click Show Numbers toolbar in Figure 6.175.
2. Click Next Numbers toolbar in Figure 6.176 until skeleton boundary.
3. Figure 6.182 shows skeleton boundary codes for the space truss.

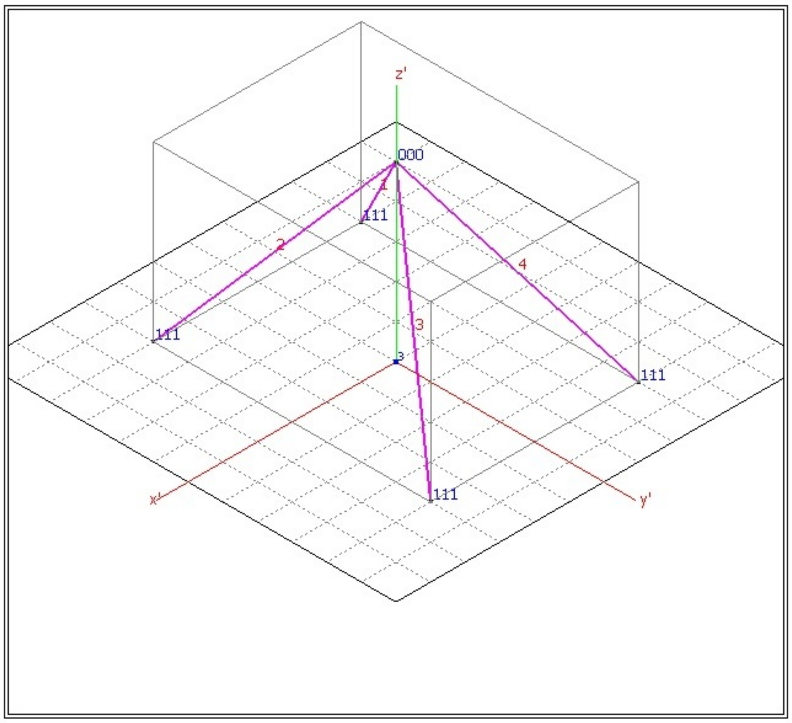


Figure 6.182 Skeleton boundary codes for space truss