SMAP ® - 2D

Structure Medium Analysis Program

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

User's Manual Version 7.06

COMTEC RESEARCH

Printed in the United States of America.

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

1.1 Overview

SMAP-2D is an advanced two-dimensional finite element computer program developed for the geometric and material nonlinear structure-medium interaction analysis. The program can be a powerful tool for the geomechanical analysis since it can solve static, consolidation and dynamic problems in dry, partially saturated or fully saturated soils and porous rock mass. The program has been designed to integrate the pre-, main-, and post-processors as shown at the end of this Section.

1.2 Features

Features of SMAP-2D include:

- Two-dimensional plane strain, plane stress or axisymmetric isoparametric continuum element.
  - Models soils, rocks and concrete media
  - Allows yielding and tension cut-off
  - Models dry, saturated and partially saturated porous media
- Joint element
  - Models faults, joints, and interface
  - Allows sliding and debonding

- Beam element
  - Models rectangular, tee-shape, and I-shape reinforced concrete or composite beams.
  - Allows cracking and crushing of concrete and yielding of the reinforcing bar.
- Truss element
  - Models rock bolts and anchor bar
  - Allows yielding, buckling and post-buckling
- External loads
  - Pressure time history
  - Displacement/Velocity/Acceleration time history
  - Initial velocity
  - Gravity load
  - Base acceleration time history
- Special boundary condition
  - Skew boundary
  - Transmitting boundary
- Simulation of a sequence of excavation and construction
- Nonlinear material model
  - Von Mises model
  - Mohr-Coulomb model
  - In Situ Rock model
  - JWL Explosive Source model
  - Modified Cam-Clay model
  - Hyperbolic model
  - Engineering model
  - User defined model
- Large deformations
  - Use updated Lagrangian
- One-dimensional spherical model

1.3 Applications

Applications of SMAP-2D include:

- Dynamic analysis
  - Wave propagation
  - Ground motions due to tunnel blasting
  - Blast-induced liquefaction
  - Earthquake analysis
- Consolidation analysis
  - Foundation settlement
  - Earth dam stability during construction
  - Ground water flow through tunnel liner
- Rock-structure interaction analysis
  - Underground power plant chamber
  - Lined or unlined shafts and tunnels subjected to internal water pressures as well as external earth pressures.
- Shallow and deep foundation analysis
- Slope stability analysis
- Framed structural analysis

Overview of SMAP-2D Program Structure

USER INPUT User prepares Mesh, Main, and Post Files according to SMAP-2D User's Manual as described in Section 4.

PRESMAP Pre-processors to automatically generate Mesh File which contains nodal coordinates, boundary constraints, and element indexes.

SMAP-2D Main-processor executing Mesh and Main Files to compute displacements, stresses and strains. Output files include:

CONTSS.DAT	Stresses/strains in continuum
BEAMSF.DAT	Section forces in beam
BEAMSS.DAT	Stresses/strains in beam
RBARSS.DAT	Reinforcing bar stresses/strains
TRUSS.DAT	Stresses/strains in truss
DISPLT.DAT	Nodal displacements, velocities and accelerations.

PLOT-XY
PLOT-2D
PLOT-3D

Post-processors executing Post File for graphical output:

- Finite element mesh
- Deformed shape
- Section forces in beam elements
- Extreme fiber stresses/strains in beam elements
- Axial force/stress/strain in truss element
- Contours of stresses and factor of safety
- Time histories of displacements/stresses/strains

Installing SMAP -2D

2.1 Minimum System Requirements

✓ Windows 64 bit operating system
✓ Intel Pentium 4 or AMD processors
✓ 4 GB Ram with 30 GB free space in Drive C
✓ SVGA monitor

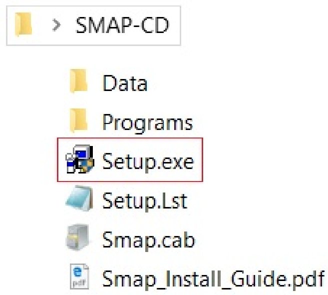
2.2 Installation Procedure

1. Uninstall if there are pre-existing SMAP programs.
   To uninstall SMAP programs, remove following program using Add/Remove in Control Panel:
      SMAP
   Delete following files if they are existing:
      C:\Program Files\Smap
      C:\Windows\Setup1.exe
   Rename or delete following folders if they are existing:
      C:\SMAP
      C:\SmapKey

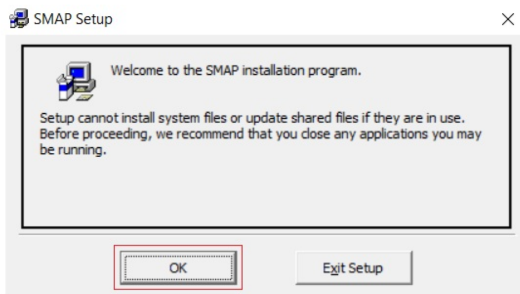
2. Download SMAP-CD.exe from the Download section of www.ComtecResearch.com

3. Run SMAP-CD.exe
   SMAP-CD folder will be created with SMAP installation programs

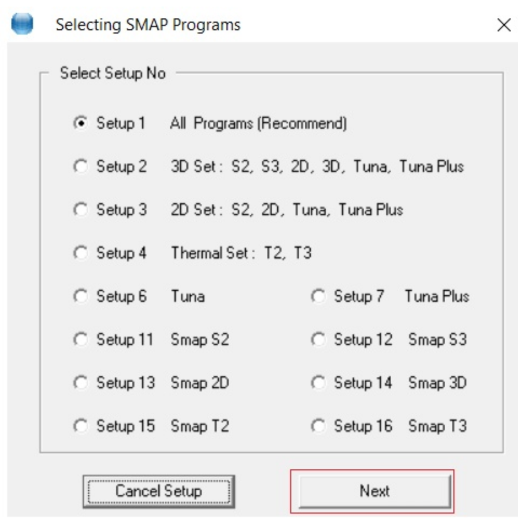
4. Double-click **Setup.exe**



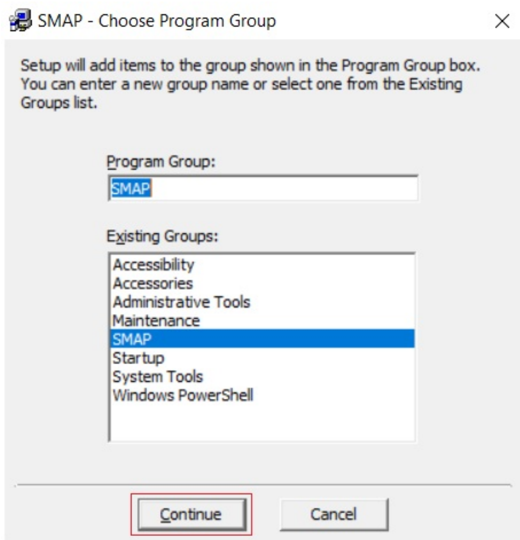
5. Click **OK**



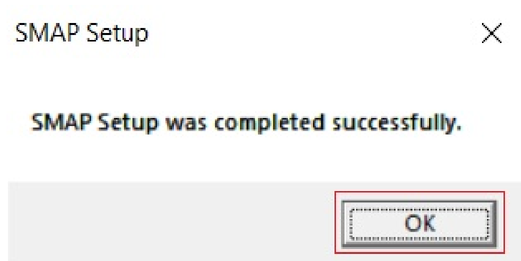
6. Click **Next**
   It will take few minutes.
   Wait until next step.



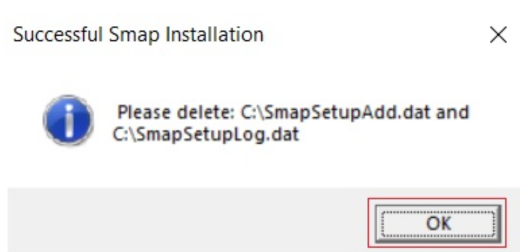
7. Click **Continue**



8. Click **OK**



9. Click **OK**



Note:

Following two log files will be generated once finished:
C:\SmapSetupAdd.dat
C:\SmapSetupLog.dat

If Smap Installation is successful, delete these two files.

If Smap Installation is not successful,
follow the instruction in SmapSetupAdd.dat.

If you still have problems with Smap Installation,
send these two files to info@ComtecResearch.com

Running Programs

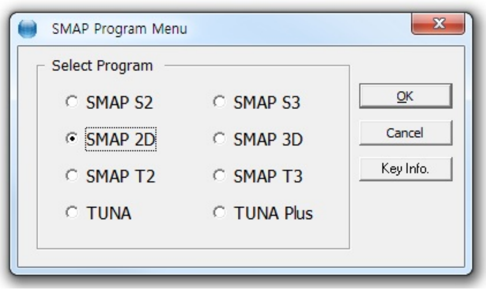
3.1 Introduction

Generally, SMAP-2D consists of pre-, main-, and post-processing programs. Pre-processing programs are mainly used to automatically generate Mesh Files which will contain nodal coordinates, boundary conditions, and element indexes. Main-processing program of SMAP-2D is the one which computes static, consolidation and dynamic response of two-dimensional problems. Post-processing programs are used to show graphically the results from the main-processing program.

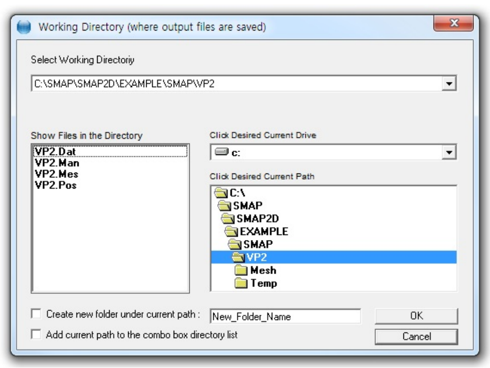
Accessing SMAP-2D Programs

1. When it is the first time, you copy Smap.exe in C:\Ct\Ctmenu and setup a Shortcut to SMAP Icon on your computer desktop. Then You simply double-click SMAP Shortcut.

2. Select **SMAP-2D** radio button and then click **OK** button.

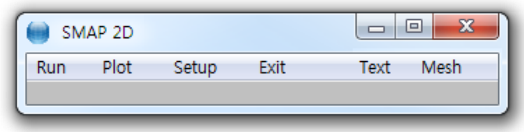


3. Next, you need to select **Working Directory**. Working Directory should be the existing directory where all the output files are saved. It is a good idea to have all your input files for the current project in this Working Directory. Click the disk drive, double-click the directory, and then **OK** button. Note that when you select **Working Directory**, a sub directory **Temp** is created automatically. All intermediate scratch files are saved in this sub directory **Temp**.

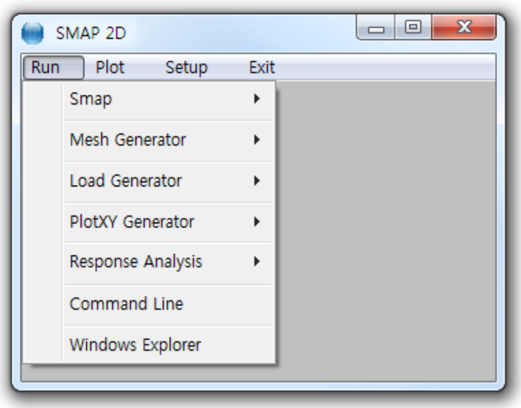


SMAP-2D Menu

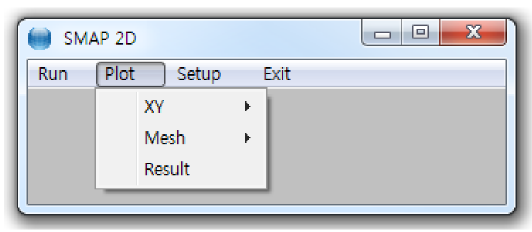
SMAP-2D provides following Main Menus; Run, Plot, Setup, Exit, Text and Mesh.



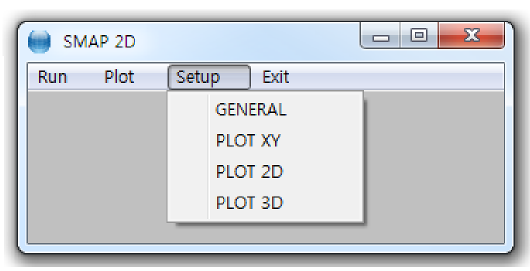
RUN executes main- and pre-processing programs and has following Sub Menus;
Smap,
Mesh Generator,
Load Generator,
PlotXY Generator,
Response Analysis,
Command Line and
Windows Explorer.



PLOT executes XY, Mesh, and Result. Result is associated with post-processing programs to show graphically the computed results.



SETUP is mainly used to set plotting control parameters for PLOT-XY, PLOT-2D, and PLOT-3D and has the following Sub Menus; General, PLOT-XY, PLOT-2D and PLOT-3D.



EXIT is used to end SMAP-2D.

TEXT is used to edit Text files.

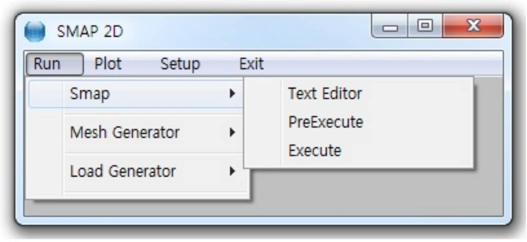
MESH is used to plot F. E. Mesh files.

3.2 RUN Menu

3.2.1 SMAP

Once you have prepared the input files (Mesh, Main, and Post) according to the SMAP-2D User's Manual in Section 4, you are ready to execute SMAP-2D main-processing program.

SMAP Menu has the following Sub Menus; Text Editor, PreExecute, and Execute.



TEXT EDITOR is used to create or modify the input file using Notepad.

PRE EXECUTE is used either to check the input file or to generate plotting information files. **PRE EXECUTE** is especially useful when you want to check input data to see whether there is any input error. It is also useful when you have finished **EXECUTE** but you want to add or modify the Post File for plot. In this case, you edit the Post File as you want, run **PRE EXECUTE**, and then run post-processing programs in **PLOT** menu.

EXECUTE executes SMAP-2D main-processing program.

SMAP-2D Output Files

Once you execute SMAP-2D, generally you can obtain following output files:

CONTSS.DAT	Contains stresses/strains in continuum element
BEAMSF.DAT	Contains section forces in beam element
BEAMSS.DAT	Contains stresses/strains in beam element
RBARSS.DAT	Contains stresses/strains in reinforcing bar
TRUSS.DAT	Contains stresses/strains in truss element
DISPLT.DAT	Contains nodal displacements

It should be noted that all of your output files are saved in the Working Directory that you specified at the beginning.

SMAP-2D Graphical Output

SMAP-2D Post-processing programs can generate the following graphical output:

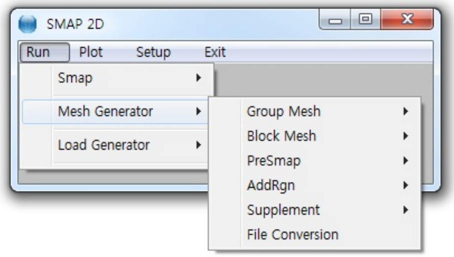
- Finite element mesh
- Deformed shape
- Principal stress distribution
- Section forces in beam element
- Extreme fiber stresses/strains in beam elements
- Axial force/stress/strain in truss element
- Contours of stresses, strains and factor of safety
- 3D iso surface of stresses and strains
- Time histories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

Graphical output can be followed by running RESULT from PLOT Menu.

3.2.2 MESH GENERATOR

MESH GENERATOR is mainly used to model two dimensional finite element meshes that are used as Mesh File.

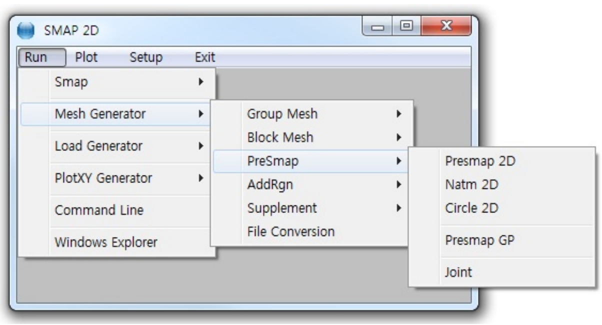
MESH GENERATOR Menu has the following Sub Menus; Group Mesh, Block Mesh, PreSmap, AddRgn, Supplement, and File Conversion.



GROUP MESH 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 5 in SMAP-2D Example Problems describes in detail about running Group Mesh.

BLOCK MESH 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 6 in SMAP-2D Example Problems describes in detail about running Block Mesh.

PRESMAP menu includes two dimensional pre-processing programs to generate finite element meshes: Section 7 in SMAP-2D Example Problems describes in detail about running PRESMAP Programs.



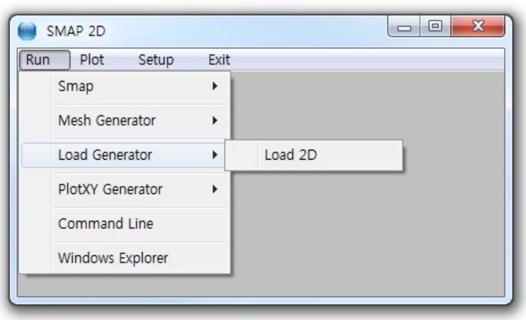
ADDRGN is the pre-processing program which has the following two basic functions: Combine two different meshes and modify existing meshes. Section 8 in SMAP-2D Example Problems describes in detail about running ADDRGN programs.

SUPPLEMENT contains supporting programs which are useful to prepare input data for pre- and main-processing programs. Section 9 in SMAP-2D Example Problems describes in detail about running SUPPLEMENT programs.

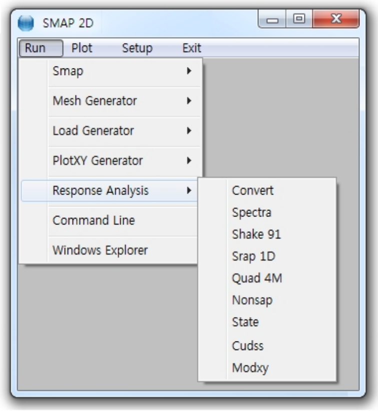
FILE CONVERSION is to convert Mesh File formats between different programs. IGES or FEMAP (Version 4.1- 4.5) can be converted to SMAP Mesh File format. Section 10 in SMAP-2D User's Manual describes in detail about running FILE CONVERSION program.

3.2.3 LOAD GENERATOR

LOAD GENERATOR includes the pre-processing program **LOAD-2D** which generates nodal values of external forces, specified velocities, initial velocities, accelerations and transmitting boundaries.
Section 10 in SMAP-2D Example Problems describes in detail about running **LOAD-2D** program.



3.2.4 PlotXY GENERATOR



PlotXY GENERATOR is the graphical user interface which is mainly used to generate or edit Simplified Time History and Simplified Snapshot of

Card Group 12 in SMAP Post File.

Section 12.7 in SMAP-2D User's Manual describes in detail about running PlotXY Generator program.

3.2.5 RESPONSE ANALYSIS

RESPONSE ANALYSIS runs following programs for seismic analysis:

Convert	Changing format of input earthquake acceleration data
Spectra	Constructing response spectra from acceleration history
Shake 91	Solving 1D seismic response by frequency domain analysis
Srap 1D	Solving 1D seismic response by finite element analysis
Quad 4M	Solving 2D seismic response by finite element analysis
Nonsap	Solving static and dynamic response of nonlinear systems
State	Plotting stress state on p-q space and octahedral plane
Cudss	Solving cyclic undrained direct simple shear for PM4Sand
Modxy	Modifying each XY data curve separately for PLOT-XY
	All Examples enclosed in the directory C:\Smap\Response

3.2.6 COMMAND LINE

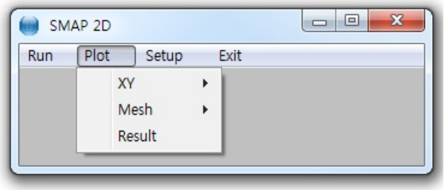
COMMAND LINE opens Windows Command Prompt at the current Working Directory. You can use a keyboard to navigate, access, and modify files and folders by entering commands. For example, COMMAND LINE is used when executing manually SMAP main solvers.

3.2.7 WINDOWS EXPLORER

WINDOWS EXPLORER opens Windows File Explorer at the current Working Directory. You can use a mouse to navigate and manage the drives, folders and files on your computer.

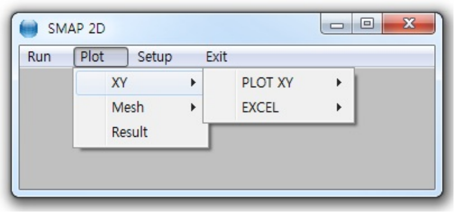
3.3 PLOT Menu

PLOT Menu is to show graphically XY graph, Mesh and Computed Result.



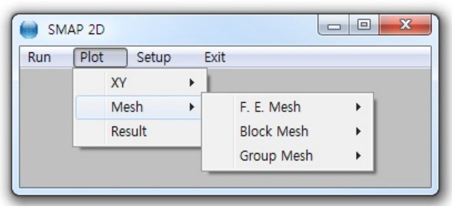
3.3.1 XY

XY graph can be displayed by PLOT-XY or EXCEL. Section 11 in SMAP-2D Example Problems describes in detail about running XY graph.



3.3.2 MESH

MESH has following Sub Menus; F. E. Mesh, Block Mesh and Group Mesh.



F. E. Mesh is used to open or create Finite Element Mesh File.

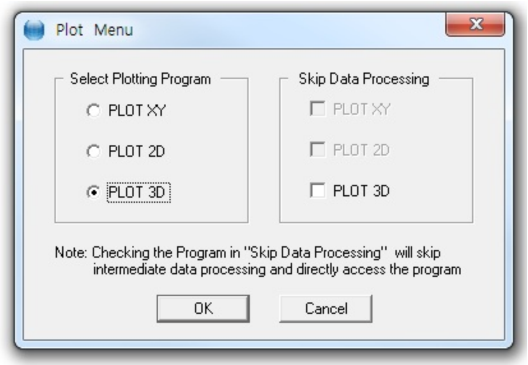
Block Mesh is used to open or build Block Mesh. Section 6 in SMAP-2D Example Problems describes in detail about running Block Mesh.

Group Mesh is used to open or build Group Mesh. Section 5 in SMAP-2D Example Problems describes in detail about running Group Mesh.

3.3.3 RESULT

Once you finished executing SMAP-2D main-processing program, you need to run post-processing programs to show graphically numerical results.

PLOT Menu contains PLOT-XY, PLOT-2D, and PLOT-3D.



PLOT-XY reads Card 12 in Post File and plots time histories of stress/strain/displacement and snapshots of stress/strain/displacement vs. distance. Refer to PLOT-XY User's Manual in Section 13.

PLOT-2D reads Card 11 in Post File and plots contours of continuum stress/strain, beam section forces, truss axial force/stress/strain, principal stress vectors, and deformed shapes. Refer to PLOT-2D User's Manual in Section 14.

PLOT-3D reads Mesh File and Smap Output Files and with no input for Post File, plots contours of stress/strain/displacement, iso surface, principal stress vectors, and deformed shapes. Refer to PLOT-3D User's Manual in Section 15.

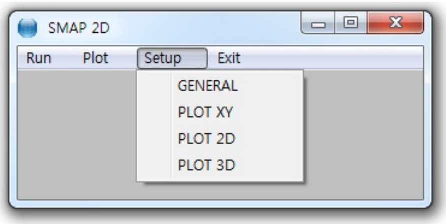
Note: When you first plot results, do not check the check box in Skip Data Processing. When you replot results, however, you can check the check box to skip intermediate data processing. This will save time and keep modified output data.

3.4 SETUP Menu

You need to run SETUP Menu

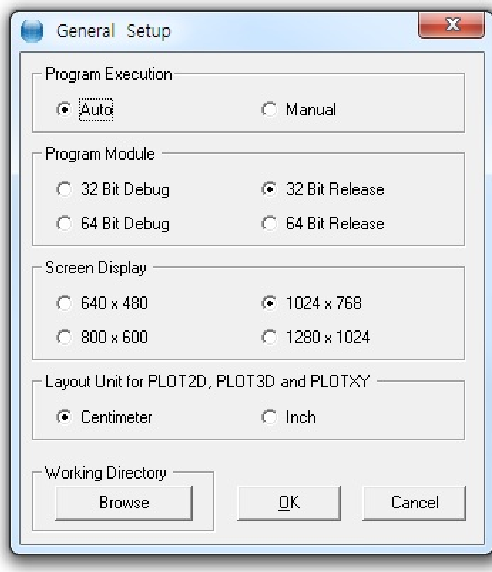
- To specify SMAP-2D main-processing program module.
- To adjust scales of graphical outputs from PLOT-XY, PLOT-2D, and PLOT-3D

SETUP Menu has four Sub Menus; General, PLOT-XY, PLOT-2D, and PLOT-3D



3.4.1 General Setup

General Setup has five different items; Program Execution, Program Module, Screen Display, Layout Unit, and Working Directory.



Program Execution has two options; Auto and Manual.
For Manual Execution, refer to Section 3.5 in User's Manual.

Program Module has four options. 32 Bit Debug, 32 Bit Release, 64 Bit Debug, and 64 Bit Release. Debug program modules run slower but gives more detailed information when run time errors occur. For most cases, 32 Bit Release is recommended. 64 Bit Modules are designed to run large problems.

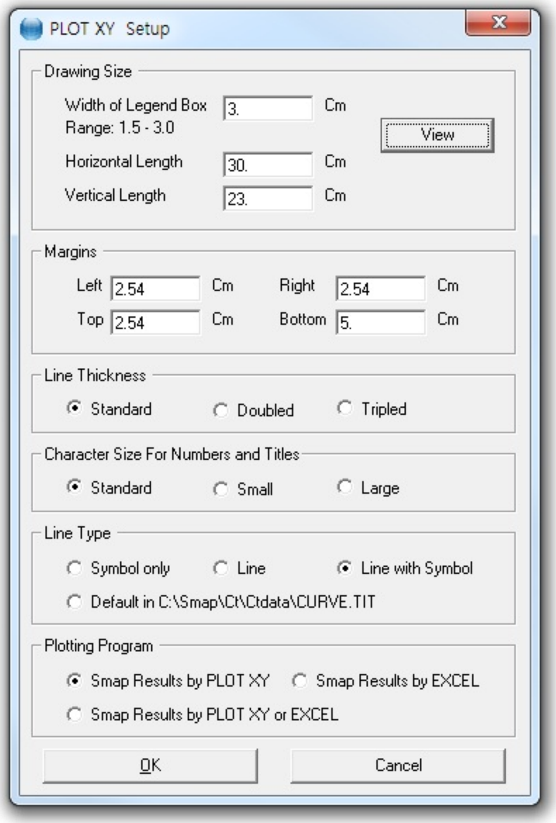
Screen Display has four options; 640x480, 800x600, 1024x768, and 1280x1024. This will affect the size of child window in PLOT-XY and PLOT-2D.

Layout Unit is used for PLOT-XY, PLOT-2D, and PLOT-3D. You can select either Centimeter or Inch in specifying plot scales and dimensions.

Working Directory is to change the current working directory. When you click the Browse button, Working Directory dialog will be shown so that you can select new directory.

3.4.2 PLOT-XY Setup

PLOT-XY Setup is mainly used to specify scales and dimensions of post processing program PLOT-XY. It has six different items; Drawing Size, Margins, Line Thickness, Character Size, Line Type, and Plotting Program.



Drawing Size controls the size of output. Once you specify Legend Box Width, Horizontal and Vertical Length, you can click **View** button to see the scaled layout.

Margins is used to shift the drawing area. Left margin is the distance from the left edge of printer page to the left frame line. In the similar way, you can specify Top, Right, and Bottom margins.

Line Thickness specifies the thickness of lines. This option is not used.

Character Size for Numbers and Titles specifies the size of characters for numbers and titles. It has three options; Standard, Small, and Large.

Line type is used to specify default line type and has four options; Symbol only, Line, Line with Symbol, and Default in C:\Smap \Ct \Ctdata \Curve.tit.

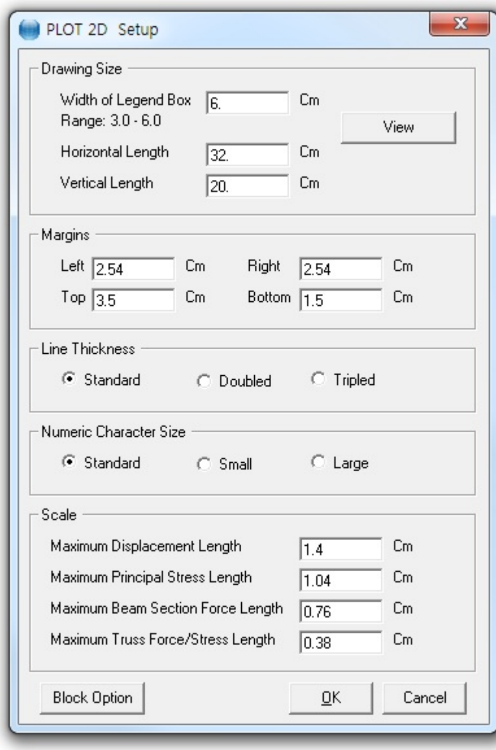
Plotting Program is used to specify default program to plot Smap results. It has three options; PLOT-XY, EXCEL, and PLOT-XY or EXCEL. Last option is to select either PLOT-XY or EXCEL at the time you plot results.

3.4.3 PLOT-2D Setup

PLOT-2D Setup is mainly used to specify scales and dimensions of post processing program PLOT-2D. It has six different items; Drawing Size, Margins, Line Thickness, Numeric Character Size, Scale and Block Option. The first four items are much similar to those described in PLOT-XY Setup.

Scale specifies Maximum Displacement Length, Maximum Principal Stress Length, Maximum Beam Section Force Length, and Maximum Truss Force/Stress Length, which will be shown on PLOT-2D.

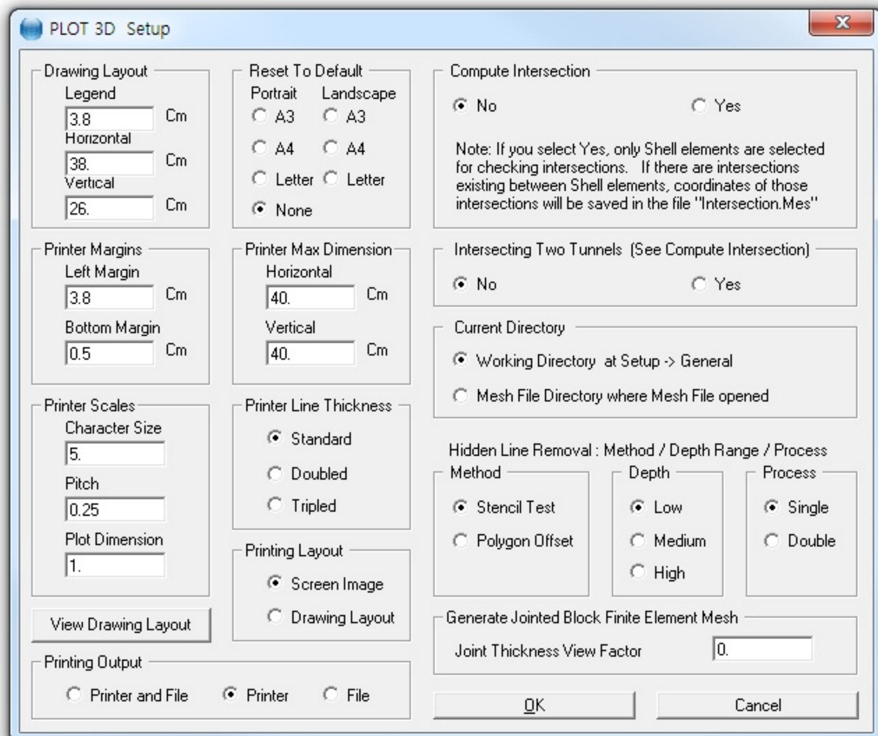
Block Option specifies options to generate either PRESMAP Output or Block Diagram.



3.4.4 PLOT-3D Setup

PLOT-3D Setup is mainly used to specify scales and dimensions of post processing program PLOT-3D. In addition, it can compute intersection of shell elements and intersecting two tunnels. And It can also generate jointed block finite element meshes. It has thirteen different items; Drawing Layout, Printer Margins, Printer Scales, Reset To Default, Printer Max Dimension, Printer Line Thickness, Printing Layout, Printing Output, Compute Intersection, Intersecting Two Tunnels, Current Directory, Hidden Line Removal and Generate Jointed Block Finite Element Mesh.

Refer to descriptions shown in the PLOT-3D Setup dialog.



3.5 Manual Procedure to Run SMAP-2D

Occasionally, you need to execute SMAP-2D main-processing program manually to see what is going on each step, specially when terminated due to some errors.

Method 1

1. Select Setup -> General -> Manual in Program Execution
2. Select Run -> Smap -> Execute
3. Select Smap project file when displaying file open dialog
4. Now Smap is running on Windows Command Line
5. Type Enter key to continue to next step or Control C to stop

Method 2

1. Select Run -> Command Line
2. Change to Temp sub directory

   Create Temp sub directory if not existing.

   Type MD Temp

   Then change to this sub directory.

   Type CD Temp

   Now, the files in the Working Directory can be accessed by prefixing "..\" to the file name.
3. Type C:\Smap\Ct\Ctbat\Smap2D
4. Type ..\VP1.Dat to access input file in Working Directory, for example
5. Type Enter key to continue to next step or Control C to stop

3.6 Debugging SMAP-2D Main-Processing Program

Debug information would be helpful in the following cases:

- Having run time errors
- Extracting convergence
- Checking elapsed time

In order to get debug information, you need to modify the file "Smap_2D.dat" in the directory C:\Smap\Ct\Ctdata\Debug

```
     1,     11,       1,       1,      1,    100,         90
IDEBUG, NCLDEB, IOUTDEB, ICONVER, NELDEB, NO_MAX, NO_RESTART

   This "DEBUG.DAT" file allows listing of status with
   elapsed time information while running main process of
   SMAP programs.   This is the very useful features to see
   where it spends most time and where it stops.

   IDEBUG   =  0 : Do not print debug information.
               1 : Print debug information. Refer to IOUTDEB.
               2 : Print debug information in each individual
                   files based on NO_MAX and NO_RESTART and
                   save  in C:\SMAP\SMAP2D\DEBUG for SMAP-2D
                   and   in C:\SMAP\SMAP3D\DEBUG for SMAP-3D

   NCLDEB        : Ending cycle number.
                   No printing debug information after NCLDEB.

   IOUTDEB =   0 : Debug information on screen.
               1 : Debug information on file,
                   Smap_2D.deb in Working Directory\Temp

   ICONVER =   0 : Do not print convergence information.
               1 : Print the ratio of displacement increment
                   to current displacement (DU/U)

   NELDEB   = -1 : Do not print element information in element
                   level operation.
            =  0 : Print current element number in element
                   level operation.
            >  0 : Print debug information for the element
                   number NELDEB in element level operation.

   NO_MAX        : Maximum number of individual files.
                   Used for IDEBUG = 2.

   NO_RESTART    : Restart number for individual file
                   once it reaches NO_MAX.
                   Used for IDEBUG = 2.
```

SMAP-2D
User's Manual

4.1 Introduction

To run SMAP-2D main-processing program, you need to prepare a Project File which contains Mesh File name, Main File name, and Post File name.

Mesh File contains nodal coordinates, boundary conditions, element indexes and material property numbers. This Mesh File is normally generated by Mesh Generator programs.

Main File contains all the other data required for the two-dimensional numerical analysis of static, consolidation, or dynamic problems.

Post File contains information which is used to show graphically the results from the main-processing program.

4.2 Project File

Project File is a collection of names of Mesh, Main, and Post Files with the following text format:

```
Mesh File Name
  Full path of Mesh File
Main File Name
  Full path of Main File
Post File Name
  Full path of Post File
```

As an example, a Project File **VP2.Dat** can be written as:

```
Mesh File Name
  D:\Example\VP2.Mes
Main File Name
  D:\Example\VP2.Man
Post File Name
  D:\Example\VP2.Pos
```

4.3 Mesh File

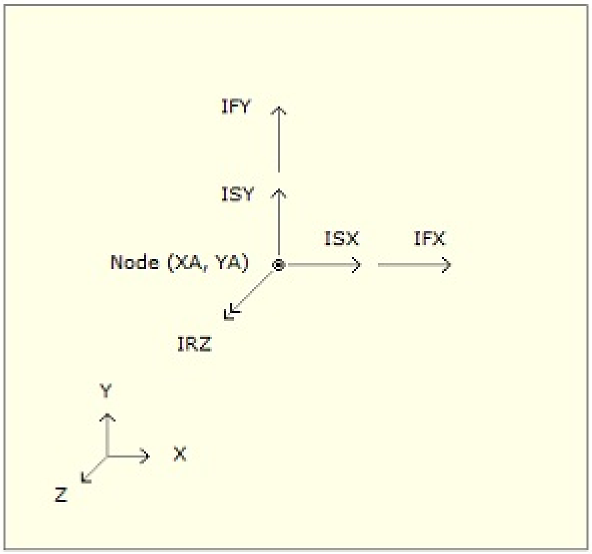
Mesh File contains nodal coordinates, boundary conditions, element indexes and material property numbers. This Mesh File is normally generated by Mesh Generator programs.

To plot Mesh File, select Mesh in Plot menu.

Mesh File

Card Group	Input Data and Definitions (Mesh File)
1 General Information	1.1 TITLE [Character string] TITLE Project title
	1.2 LABEL1 [Character string] LABEL1 Label for Card 1.3
	1.3 NUMNP, NCONT, NBEAM, NTRUSS NUMNP Total number of nodal points NCONT Total number of continuum elements NBEAM Total number of beam elements NTRUSS Total number of truss elements

Card Group	Input Data and Definitions (Mesh File)
2 Coordinate	2.1 LABEL2A [Character string] LABEL2B [Character string] LABEL2A Label for coordinate LABEL2B Label for Card 2.2
	2.2 NUMNP Cards: NODE, ISX, ISY, IFX, IFY, IRZ, IEX, IEY, XA, YA - - - - - - - - - - - - - - - - NODE Node Number ISX Skeleton X(radial) DOF (Degree of Freedom) ISY Skeleton Y(axial) DOF IFX X(radial) DOF for relative pore fluid motion IFY Y(axial) DOF for relative pore fluid motion IRZ Rotational DOF for beam IEX Slip X DOF IEY Slip Y DOF ISX, ISY, IFX, IFY, IRZ, IEX, IEY = 0 Free to move in specified direction = 1 Fixed in specified direction Note: For NCTYPE = 3 (1-D spherical analysis), set mesh height to 1.0, ISY=1, and IFY=1

Card Group	Input Data and Definitions (Mesh File)
2 Coordinate	2.2 

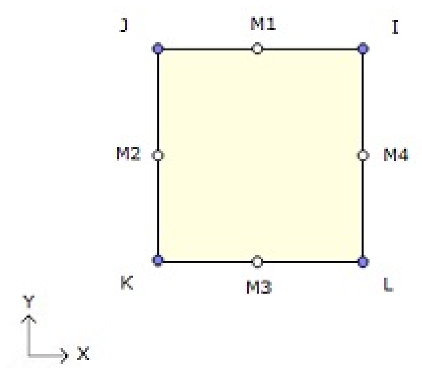
<table>
<tr><th>Card Group</th><th>Input Data and Definitions (Mesh File)</th></tr>
<tr><td rowspan="2">3

Continuum Element (If NCONT = 0, skip this card group)</td><td>3.1

LABEL3A [Character string]
LABEL3B [Character string]

LABEL3A — Label for continuum element
LABEL3B — Label for Card 3.2</td></tr>
<tr><td>3.2

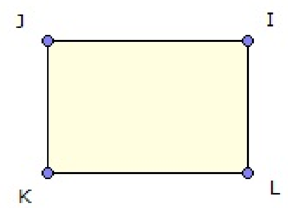
NCONT Cards ┌ NEL, I, J, K, L, M_1, M_2, M_3, M_4, MATC,
│ KS, KF, INTR, INTS, TBJWL
└ - - - - - - - -



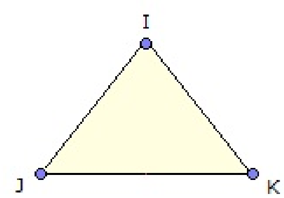
NEL — Element number
I, J, K, L — Element corner node numbers
M_1,M_2,M_3,M_4 — Element midside node numbers
If any of M_1, M_2, M_3 or M_4=0, that node is omitted</td></tr>
</table>

Card Group	Input Data and Definitions (Mesh File)		
3 Continuum Element	3.2		
	MATC		Material property number
	KS	=- 1	Element has high explosive solid phase
		= 0	Element has solid phase
		> 0	Element has joint and absolute value of KS represents face designation number. Refer to description in the following page
	KF	= 0	Element has fluid phase
		= 1	Element has no fluid phase
	INTR		Use INTR = 2
	INTS		Use INTS = 2
	TBJWL		Detonation time (required for KS = -1) Time from initial detonation to the detonation of this element

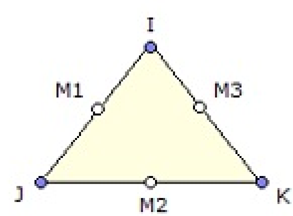
4-Node Quadrilateral Element ($M_1 = M_2 = M_3 = M_4 = 0$)



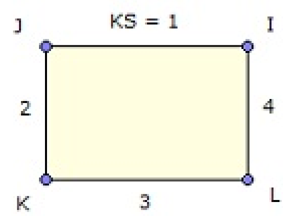
3-Node Triangular Element ($L = M_1 = M_2 = M_3 = M_4 = 0$)



6-Node Triangular Element ($L = M_4 = 0$)



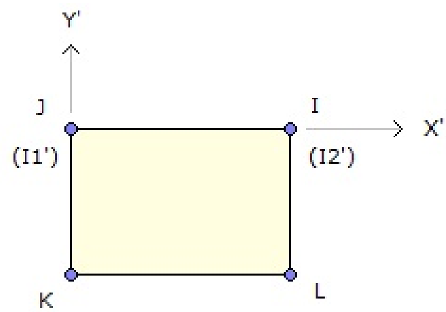
Joint Element Face Designation Number



Joint Element Local Coordinate Axes

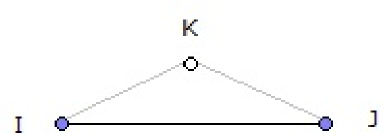
KS	I_1'	I_2'
1	J	I
2	K	J
3	L	K
4	I	L

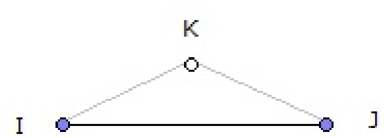
For KS = 1



It should be noted that the thickness of joint element is determined
Not by gap between two faces (JI and KL),
But by joint thickness (t) specified in Card 5.3.2.4.11 in Main File input.

The nodal coordinates of J and I represent the Location of Joint Face
but the nodal coordinates of K and L are used only For Plotting Purpose.

Card Group	Input Data and Definitions (Mesh File)
4 Beam Element (If NBEAM = 0, skip this card group)	4.1 LABEL4A [Character string] LABEL4B [Character string] LABEL4A Label for beam element LABEL4B Label for Card 4.2
	4.2 NBEAM Cards ┌ NEL, I, J, MSEC, K │ - - - - - └ - - - - - NEL Beam element number I, J Node numbers at beam end points MSEC Beam section number K Reference node number 

Card Group	Input Data and Definitions (Mesh File)
5 Truss Element (If NTRUSS = 0, skip this card group)	5.1 LABEL5A [Character string] LABEL5B [Character string] LABEL5A Label for truss element LABEL5B Label for Card 5.2
	5.2 NTRUSS Cards [NEL, I, J, MATT, K, NELPI, NELPJ - - - - - - - - - - - - - - NEL Truss element number I, J Node number at truss end points MATT Material property number K Reference node number NELPI Parent continuum element number for embedded truss node I NELPJ Parent continuum element number for embedded truss node J 

4.4 Main File

Mesh File in the previous section 4.3 contains the geometrical data of the structure to be analyzed.

Main File contains all the other data required for the two-dimensional numerical analysis of static, consolidation, or dynamic problems.

Main File consists of ten different card groups:

- System Control and Title
- Analysis Type
- Computational Parameters
- Coordinate
- Continuum Element
- Beam Element
- Truss Element
- Element Activity
- Loads
- Requested Output

Card Group	Input Data and Definitions (Main File)
Version No, System Control and Title	1.0 VERSION VERSION — Version No (Current Version = 7.05)
	1.1 IBATCH, IVMDK, IOPTDB, ISYMSOL IBATCH = 0 Interactive terminal job = 1 Batch job (not available) = 2 Generate Mesh File PlotMesh.Mes (This will not execute input) = -1 Terminal interactive job with beep sound when the calculation is finished. = -11 Same as IBATCH = -1 except long beep sound and character based screen display < -11 Same as IBATCH =- 11 except no display IVMDK = 0 Use hard disk to store internal variables = 1 Use addressable memory to store internal variables IOPTDB = 0 Use single precision to solve equation = 1 Use double precision to solve equation ISYMSOL = 0 Program determines solution scheme = 1 Impose symmetric solution scheme = 2 Impose unsymmetric solution scheme
	1.2 LTITLE LTITLE — Main title (80 characters maximum)
	1.3 LSUBTL LSUBTL — Subtitle (80 characters maximum)

Card Group	Input Data and Definitions (Main File)
2 Analysis Type	2.1 NTCSF, NLNR, NGEN, IQUAD, NTEMP, ITDIS, MODAL NTCSF = 1 Static analysis = 2 Consolidation analysis = 3 Dynamic analysis (Implicit method) = 4 Dynamic analysis (Explicit method) = 5 Mode superposition analysis For NTCSF = -5, computes only natural frequencies and mode shapes NLNR = 0 Linear elastic material = 1 Nonlinear material NGEN = 0 Small displacement = 1 Large displacement (Updated Lagrangian) IQUAD = 0 No automatic generation = 1 Automatic generation of quadratic elements If IQUAD = 1, all linear continuum elements are automatically transformed into quadratic elements NTEMP = 0 Thermal expansion is not considered = 1 Thermal properties and element temperatures are read from input the file ELTEMP.DAT that should be located in working directory. See Table in the next page Output motions when base acceleration applied ITDIS = 0 Relative displacement & Relative velocity = 1 Total displacement & Total velocity Modal analysis options for NTCSF = 5 or -5 MODAL = 0 Subspace iteration method = 1 Determinant search method = 2 Jacobi iteration method

Input File ELTEMP.DAT

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="4">Thermal Property</td><td colspan="2">1.1
TITLE [Character string]

TITLE   Project title</td></tr>
<tr><td colspan="2">2.1
LABEL 1 [Character string]
-
LABEL 6 [Character string]

LABEL 1-6   Labels for Card 2.2</td></tr>
<tr><td rowspan="2">2.2
For Each Material</td><td>2.2.1
MATNO, MODEL

MATNO   Material property number.
If MATNO = -1, end of Card 2.2

MODEL = 1   Constant thermal expansion
= 2   Step thermal expansion
= 3   Porosity rate dependent expansion

Note:
MODEL = 2 and 3 are not available</td></tr>
<tr><td>2.2.2
T_o, E_da

T_o   Freezing temperature (Degree C)
E_da   Anisotropic expansion parameter (ξ)</td></tr>
</table>

Input File ELTEMP.DAT

Card Group		Input Data and Definitions (Main File)
Thermal Property	2.2 For Each Material	2.2.3 E_u, V_u, E_f, V_f E_u — Unfrozen Young's modulus V_u — Unfrozen Poisson's ratio E_f — Frozen Young's modulus V_f — Frozen Poisson's ratio
		2.2.4 Required only for MODEL = 1 Alpha_c Alpha_c — Coefficient of thermal expansion (L/L/Temperature)
		2.2.5 Required only for MODEL = 2 Strain_m, dT_o Strain_m — Maximum expansive strain dT_o — Strain_m distributed over dT_o (Deg C)
		2.2.6 Required only for MODEL = 3 RateN_m, T_m, g_T, Z_eta RateN_m — Maximum porosity rate T_m — Temperature (Deg C) at RateN_m g_T — Temperature gradient (Deg C/m) at RateN_m Z_eta — Stress parameter, ζ , in stress unit (Mpa) used for reducing porosity rate

Input File ELTEMP.DAT

Card Group	Input Data and Definitions (Main File)
3 Temperature Profile, Can be repeated for each $TIME_i$	3.1 LABEL 2 [Character string] LABEL 1 — Label for Card 3.2
	3.2 $TIME_i$ $TIME_i$ — Time. $TIME_i$ should be 0.0 for initial state If $TIME_i$ = -1.0, end of data
	3.3 LABEL 3 [Character string] LABEL 3 — Label for Card 3.4
	3.4 NELNO, MATNO T_{top} T_{bot} $T_{,x}$ $T_{,y}$ $T_{,z}$ - - - - - - - - - - - - - - NELNO — Element number If $NELNO_i$ = -1, end of Card 3.4 MATNO — Material property number. T_{top} — Temperature on top surface T_{bot} — Temperature on bottom surface $T_{,x}$ — Temperature gradient in x direction $T_{,y}$ — Temperature gradient in y direction $T_{,z}$ — Temperature gradient in z direction

Card Group	Input Data and Definitions (Main File)
3 Computational Parameters	3.1 Cycles and Time Step NCYCL, DT, NDTGR, NITER, MNEWRP, TOLER, IRANGE, KRANGE NCYCL — Number of total solution cycles DT — Global time step: Duration of each cycle NDTGR — Number of time step group (Max=100) If NDTGR = 0, constant time steps are used. For NDTGR > 0, NLNR = 1 should be specified in Card Group 2.1 NITER — Number of maximum iteration (Iteration is available for NTCSF = 1) MNEWRP = 0 — Modified Newton-Raphson method = 1 — Newton-Raphson method =-1 — Newton-Raphson method with first iteration as trial guess. For specified velocity, use MNEWRP = 0 TOLER — Tolerance for convergence, defined as the ratio of displacement increment to current displacement. (Default TOLER = 0.001) IRANGE = 0 — NITER is applied throughout NCYCL = 1 — NITER is applied based on Cycle No = 2 — NITER is applied based on Time KRANGE = 0 — Stiffness update option is not used = 1 — Stiffness update option based on Cycle No = 2 — Stiffness update option based on Time

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="2">3

Computational Parameters</td><td colspan="2">3.1.1

If NDTGR = 0, go to Card Group 3.1.3
ICYCLTIME

ICYCLTIME = 0 Time step is based on Cycle No
= 1 Time step is based on Time</td></tr>
<tr><td>3.1.2

For Each Time Step Group</td><td>3.1.2.1

STIME, ITYPE

STIME Starting Cycle No for ICYCLTIME = 0
Starting Time (t_o) for ICYCLTIME = 1
For the first time group, use STIME = 0

ITYPE = 0 Constant time step
= 1 Constant log time step
= 2 Arbitrary specified time step

<u>If ITYPE = 0</u>
DT

DT Time step
<u>If ITYPE = 1</u>
DT_1, CLDT

DT_1 Starting time step
CLDT Constant log time step
$CLDT = \log_{10}(t_{i+1}-t_o) - \log_{10}(t_i - t_o)$
<u>If ITYPE = 2</u>
NUMDT,
DT_1, ..., DT_{NUMDT}

NUMDT Number of time step
DT_1, ..., DT_{NUMDT} Listing of specified time steps</td></tr>
</table>

Card Group			Input Data and Definitions (Main File)
3 Computational Parameters	3.1.3		If IRANGE = 0, go to Card Group 3.1.5 NRANGE NRANGE — Number of specified ranges where NITER is applied (Max=100)
	3.1.4 For Each Range	3.1.4.1	SFTIME, SLTIME SFTIME Starting Cycle No for IRANGE = 1 Starting Time for IRANGE = 2 SLTIME Ending Cycle No for IRANGE = 1 Ending Time for IRANGE = 2

Card Group		Input Data and Definitions (Main File)
3 Computational Parameters	3.1.5	If KRANGE = 0, go to Card Group 3.2 NRANGE NRANGE — Number of specified ranges where stiffness update option is applied (Max=100)
	3.1.6 For Each Range	3.1.6.1 SFTIME, SLTIME, NST SFTIME Starting Cycle No for KRANGE = 1 Starting Time for KRANGE = 2 SLTIME Ending Cycle No for KRANGE = 1 Ending Time for KRANGE = 2 NST — Number of time steps for which the global stiffness matrix is assumed to be constant. Ex. For NST = 2, stiffness matrix is updated every other step during the specified ranges from SFTIME to SLTIME

Card Group	Input Data and Definitions (Main File)
3 Computational Parameters	3.2 Numerical Time-Integration and Artificial Viscosity If NTCSF < 3, go to Card Group 3.3 TETA, BETA, GAMA, CQ, CL, F1, F3, RD, NTMODE TETA — θ See Table 1 BETA — β See Table 1 GAMA — γ See Table 1 CQ — Quadratic artificial viscosity coefficient CL — Linear artificial viscosity coefficient F1 — First natural frequency F3 — Third natural frequency or Predominant frequency of input motion RD — For NTCSF = 5, RD as critical damping ratio. For NTCSF = 3 and RD > 0, program first computes F1 and F3 and then automatically reruns. Actual RD is defined in element material input sections. Note: Both F1 and F3 are used to compute Rayleigh mass and stiffness proportional damping coefficients. NTMODE — Number of mode shapes to be considered Note: If NTCSF = 4, only CQ and CL are used

Card Group		Input Data and Definitions (Main File)
3 Computational Parameters	3.3 Calculational Mode Change	3.3.1 If NTCSF = 4, go to Card Group 4 NCLMCH NCLMCH = 0 Do not change calculation mode > 0 Change calculation mode at cycle NCLMCH.
		3.3.2 If NCLMCH = 0, go to Card Group 4 NTCNEW, DTNEW TETANEW, BETANEW, GAMANEW, CQNEW, CLNEW, F1NEW, F3NEW, RDNEW, NTMODENEW NTCNEW New value of NTCSF after NCLMCH DTNEW New value of DT after NCLMCH TETANEW, BETANEW, GAMANEW, CQNEW, CLNEW, F1NEW, F3NEW, RDNEW, NTMODENEW are new values of Card 3.2 after NCLMCH respectively

Table 1 Values of β and θ for γ =1/2*

Integration Method	β	θ
Explicit second central difference	0	1.0
Fox-Goodwin	1/12	1.0
Linear acceleration	1/6	1.0
Newmark's constant acceleration	1/4	1.0
Wilson	1/6	2.0
Stiff linear acceleration	1/6	1.5

*γ = 1/2 indicates no damping

γ > 1/2 introduces numerical damping and $\beta = (\gamma + 1/2)^2/4$

For more information, refer to
Ghaboussi and Wilson, "Variational Formulation of Dynamic of Fluid Saturated Porous Elastic Solids," ASCE Engineering Mechanics Journal, August 1972

Card Group	Input Data and Definitions (Main File)
4 Coordinate	4.1 NUMNP NUMNP — Total number of nodal points
	4.2 CMFAC, SCFP CMFAC — Coordinate multiplication factor (Use CMFAC = 1.0) SCFP — Stress conversion factor for converting pressure units to Pascals Note SCFP is used for nonlinear pore fluid and JWL model Stress Unit — SCFP kg/cm^2 — 98066.5 t/m^2 — 9806.65 kg/m^2 — 9.807 $Newton/cm^2$ — 10000 bar — 100000 psi — 6895 ksi — 6.895×10^6 psf — 47.88 MPa — 1000000

Card Group			Input Data and Definitions (Main File)
4 Coordinates	4.4 Boundary Code Change	4.4.1	NBNODE, NCLBCH, IFLCOD NBNODE — Number of nodes where boundary codes are changed NCLBCH — Cycle No where boundary codes are changed IFLCOD = 0 — Read Card 4.4.2 here = 1 — Read Card 4.4.2 from file NewBcode.dat starting with NBNODE as first card If NBNODE = 0, go to next Card Group 4.5
		4.4.2	NBNODE Cards: $NODE_1$, ISX_1, ISY_1, IFX_1, IFY_1, IRZ_1 - - - - - - - - - - - - - - - - - - Refer to Card Group 2.2 in Mesh File in page 4-5 for description.

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="2">4

Coordinates</td><td rowspan="2">4.5

Repeating Nodes</td><td>4.5.1

NREPEAT

NREPEAT Number of repeating nodes

If NREPEAT = 0, go to next Card Group 5.1</td></tr>
<tr><td>4.5.2

NREPEAT Cards: NODER, NODEP
- -
- -
- -

NODER Repeating node
NODEP Parent node

Note :
Repeating node NODER shares the same degrees of freedom as those of the corresponding parent node NODEP</td></tr>
</table>

Card Group	Input Data and Definitions (Main File)
5 Continuum Element	5.1 NCONT NCONT — Total number of continuum element If NCONT = 0, go to next Card Group 6
	5.2 NCTYPE, NSPTC, IEDOF NCTYPE = 0 — Axisymmetric element Y-axis is axis of symmetry = 1 — Plane strain element = 2 — Plane stress element = 3 — Spherically symmetric element NSPTC = 0 — Compute stresses and strains at integration points = 1 — Compute stresses and strains at center of element IEDOF = 0 — Do not include incompatible extra DOF = 1 — Include incompatible extra DOF

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data		5.3.1 NTNC NTNC Number of material property set for continuum element
		For Each Material Property Set	5.3.2.0 MATNO, MATNP MATNO Material number MATNP Parent material number MATNO will duplicate MATNP If MATNP > 0, go to next property set.
			5.3.2.1 TITLE TITLE Material name (Max 80 characters)
			5.3.2.2 POR, GW, G, PFMIN, DAMP, ICST POR Initial porosity (n_o) GW Unit weight of water at ( )° c G Gravity constant (g) PFMIN Minimum fluid pressure (Not used) DAMP Initial critical damping ratio ICST = 0 : Lumped mass = 1 : Consistent mass (Default)
			5.3.2.3 NF NF = 0 Linear fluid and solid grain = 1 Nonlinear fluid and solid grain

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.3 Fluid and Solid Grain Property (NF = 0)	5.3.2.3.1 For NF = 0 (Linear Fluid and Solid Grain) RK_1, BKG, SGG, BKF, SGF, NK, RK_1FAC, NPHNO RK_1 — Darcy's coefficient of permeability BKG — Bulk modulus of grain SGG — Specific gravity of solid grain BKF — Bulk modulus of pore fluid SGF — Specific gravity of pore fluid NK = 0 — Isotropic permeability = 1 — Anisotropic permeability RK_1FAC — Multiplication factor for RK_1, applied during NGSTEP NPHNO — Permeability intensity history number in Card Group 9.2.3 For NK = 1 α_{xx}, α_{yy}, α_{zz}, α_{xy}, α_{xz}, α_{yz} α_{ij} — Permeability component ($k_{ij} = \alpha_{ij} \cdot RK_1$)

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.3 Fluid and Solid Grain Property (NF = 1)	5.3.2.3.2 For NF = 1 (Nonlinear Fluid and Solid Grain) Permeability Property NP, RK1, RK2, RK3, NK, RK_1FAC, NPHNO NP = 0 Constant permeability Nonlinear permeability = 1 Function of porosity = 2 Function of flow velocity = 3 Function of porosity and flow velocity RK1, RK2, RK3 Permeability constants dependent on NP See Table 2 NK = 0 Isotropic permeability = 1 Anisotropic permeability RK_1FAC Multiplication factor for RK_1, applied during NGSTEP NPHNO Permeability intensity history number in Card Group 9.2.3 For NK = 1 α_{xx}, α_{yy}, α_{zz}, α_{xy}, α_{xz}, α_{yz} α_{ij} Permeability component ($k_{ij} = \alpha_{ij} \cdot k$)

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.3 Fluid and Solid Grain Property (NF = 1)	5.3.2.3.2 Solid Grain Property NG, BKG, SGG, CO, VO, S, PB NG = 0 Constant grain modulus = 1 Nonlinear grain modulus BKG Initial bulk modulus of grain SGG Initial specific gravity CO Initial wave velocity at relatively low pressure* VO Initial Poisson's ratio* S Experimentally determined constant relating loading wave velocity to peak particle velocity. Generally equal to about 1.5 for most rocks and minerals* PB Threshold pressure beyond which material tends to behave like a fluid (*) Not used for NG = 0

Table 2 Permeability Constants

NP	Equivalent Permeability k (length/time)	Input Variables
0	$k = RK_1$	RK_1 = Darcy's coefficient of permeability (length/time) RK_2, RK_3 not used
1	$k = 10^{RK_1 (n - RK_2)}$	RK_1 = Slope of n vs. log k line in units log (length/time). RK_2 = Porosity corresponding to k=1.0 RK_3 = Not used
2	$k = \dfrac{RK_1}{1 + \dfrac{RK_3}{\gamma_f}\sqrt{RK_1}\,\lvert\dot{w}_i\rvert}$	RK_1 = Darcy's coefficient of permeability (length/time) $= \dfrac{\gamma_f}{a}$ RK_2 = Not used. RK_3 = Ward's coeff. for turbulent flow $\beta_f = b\,k^{1/2}$
3	$k = \dfrac{K_l}{1 + \dfrac{RK_3}{\gamma_f}\sqrt{K_l}\,\lvert\dot{w}_i\rvert}$ $K_l = 10^{RK_1 (n-RK_2)}$	RK_1 See NP = 1 RK_2 See NP = 1 RK_3 See NP = 2

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.3 Fluid and Solid Grain Property (NF = 1)	5.3.2.3.2

Pore Fluid Property

NW, BKF, SGF, SO, GAMMA, PAO, T

NW = 0	Constant fluid modulus
= 1	Nonlinear modulus (Fresh water)
= 2	Nonlinear modulus (Sea water)
BKF	Initial bulk modulus of pore fluid
SGF	Initial specific gravity of pore fluid
SO	Initial degree of saturation* SO ≠1.0 invokes partial saturation model
GAMMA	Ratio of heat capacity *, $\gamma = C_p / C_v$
PAO	Initial pore air pressure (Absolute)*
T	Not used

(*) Not used for NW = 0

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property	5.3.2.4	MODELNO, DSRNMAX, MAXCYCL, Ko, NEHNO, NRHNO

MODELNO = 1 Elastic Model
= 2 Von Mises Model
= 3 Mohr-Coulomb Model
= 4 In Situ Rock Model
= 5 Generalized Hoek and Brown Model

= 6 Advanced Elasto-Plastic Model (N.A.)
= 7 Single Hardening Plastic Model (N.A.)
= 8 JWL High Explosive Model
= 9 Modified Cam Clay Model
= 10 Engineering Model

= 11 Joint Model
= 12 Duncan and Chang Hyperbolic Model

= 14 User Defined Model
= 15 User Defined Model
= 16 User Defined Model
= 17 User Defined Model
= 18 User Defined Model

= 21 PM4Sand Model (N.A.)

DSRNMAX = 0.0 Do not apply strain sub cycling
> 0.0 Maximum strain sub increment

MAXCYCL Maximum number of strain sub cycling

Ko Coefficient of earth pressure at rest
NEHNO Young□s modulus multiplication factor history number in Card Group 9.2.3
NRHNO Element volume multiplication factor history number in Card Group 9.2.3

Note: Ko, NEHNO, NRHNO are applicable only for MODELNO =1, 2, 3, 4, 5, 10, 12

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 1 (Elastic Model)	5.3.2.4.1 For MODELNO = 1 [Elastic Model] E, v E Young's modulus v Poisson's ratio

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 2 (Von Mises Model)	5.3.2.4.2 For MODELNO = 2 [Von Mises Model] E, ν σ E Young's modulus ν Poisson's ratio σ Shear strength in triaxial compression

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 3 (Mohr-Coulomb Model)	5.3.2.4.3 For MODELNO = 3 [Mohr-Coulomb Model] E, v φ, c, K, T, ST_n, ST_s E Young's modulus v Poisson's ratio φ Internal frictional angle (°) $C = \frac{(1 - \sin\varphi)}{2\cos\varphi}\sigma_c$ C Cohesion K The ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same pressure T Tensile strength ST_n Factor used to divide stiffness normal to tensile crack ST_s Factor used to divide shear modulus for the cracked zone Note To ignore stiffness reduction associated with tensile crack, use ST_n=ST_s=1.0

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 4 (In Situ Rock Model)	5.3.2.4.4 For MODELNO = 4 (In Situ Rock Model) E, v m, s, σ_c, K, T, ST_n, ST_s E — Young's modulus v — Poisson's ratio φ — Internal frictional angle (°) C — Cohesion $C = \frac{(1 - \sin\varphi)}{2\cos\varphi}\sigma_c$ K — The ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same pressure T — Tensile strength ST_n — Factor used to divide stiffness normal to tensile ST_s — Factor used to divide shear modulus for the cracked zone m,s — Hoek and Brown material parameters See Table 3 σ_c — Unconfined compressive strength Note : To ignore stiffness reduction associated with tensile crack, use ST_n = ST_s =1.0

Table 3 Hoek and Brown Material Parameters (m, s)

Rock Type / Rock Quality	Dolomite, Limestone & Marble	Mudstone, Siltstone, Shale and Slate (normal to cleavage)	Sandstone and Quartzite	Andesite, Dolerite & Rhyolite	Amphibolite, Gabbro, Gneiss, Norite and Quartz-Diorite
Intact CSIR rating = 100 NGI rating = 150	m = 7 s = 1	10.0 1.0	15.0 1.0	17.0 1.0	25.0 1.0
Very Good Quality CSIR rating = 85 NGI rating = 100	3.5 0.1	5.0 0.1	7.5 0.1	8.5 0.1	12.5 0.1
Good Quality CSIR rating = 65 NGI rating = 10	0.7 0.004	1.0 0.004	1.5 0.004	1.7 0.004	2.5 0.004
Fair Quality CSIR rating = 44 NGI rating = 1	0.14 0.001	0.20 0.0001	0.3 0.0001	0.34 0.0001	0.5 0.0001
Poor Quality CSIR rating = 23 NGI rating = 0.1	0.04 0.00001	0.05 0.00001	0.08 0.00001	0.09 0.00001	0.13 0.00001
Very Poor Quality CSIR rating = 3 NGI rating = 0.01	0.007 0.0	0.01 0.0	0.015 0.1	0.017 0.0	0.025 0.0

Table 3 Continued

Description of Rock Quality

Intact Rock Samples	Laboratory size specimens free from joints
Very Good Quality Rock Mass	Tightly interlocking undisturbed rock with unweathered joints at 1 to 3m
Good Quality Rock Mass	Fresh to slightly weathered rock, slightly disturbed with joints at 1 to 3m
Fair Quality Rock Mass	Several sets of moderately weathered joints spaced at 0.3 to 1m
Poor Quality Rock Mass	Numerous weathered joints at 30 to 500mm with sane gouge. Clean compacted waste rock
Very Poor Quality Rock Mass	Numerous heavily weathered joints spaced < 50m with gouge. Waste rock with fines

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property	Skeleton Property for MODELNO = 5 (Generalized Hoek and Brown Model)	5.3.2.4.5 For MODELNO = 5 [Generalized Hoek & Brown Model] Elastic Parameters E, v E Young's Modulus v Poisson's Ratio Tensile Strength Parameters NTCUT NTCUT = 0 No tension cut-off = 1 Tension cut-off For NTCUT = 1, otherwise go to next Card T, St_n, St_s T Tensile strength ST_n Factor used to divide stiffness normal to tensile crack ST_s Factor used to divide shear modulus for cracked zone Note: To ignore stiffness reduction associated with tensile crack, use ST_n = ST_s=1.0 Strength Parameters A_1, A_2, A_3, A_4, A_5, A_6, A_7, A_8 1. Von Mises (A_1 = 0.0) F = q - A_4 R(θ) A_2 = A_3 = 0.0 A_4 = A_6 = q_{VM} = σ Refer to Card 5.3.2.4.2

Card Group	Input Data and Definitions (Main File)
5 Continuum Element 5.3 Material Property Skeleton Property for MODELNO = 5 (Generalized Hoek and Brown Model)	5.3.2.4.5

2. Hoek and Brown (A_1 = 0.5)
   $F = q - ((A_2 + A_3 p)^{?} + A_4) R(\theta)$
   $A_2 = (m^2 / 36 + s) \sigma_c^2$
   $A_3 = m \sigma_c$
   $A_4 = m \sigma_c / 6$
   Refer to Card 5.3.2.4.4

3. Mohr-Coulomb (A_1 = 1.0)
   $F = q - ((A_2 + A_4) + A_3 p) R(\theta)$
   $A_2 + A_4 = 3 \sigma_c (1 - \sin\varphi) / (3 - \sin\varphi)$
   $A_3 = 6 \sin\varphi / (3 - \sin\varphi)$
   Refer to Card 5.3.2.4.3

4. Quadratic (A_1 = 2.0)
   $F = q - (A_2 + A_3 p + A_4 p^2) R(\theta)$

5. Elliptic (A_1 = 3.0)
   $F = q - (A_3 + (A_6 - A_3) (1 - ((p - A_2)/A_4)^2)^{1/2}) R(\theta)$

   $A_5 = K$ (See notes in previous page)
   $A_6 = q_{VM}$ (Von Mises limit stress)

The mean pressure (p_o) at which it reaches Von Mises limit stress (q_{VM}) is given by:

For A1 = 0.0, $p_o = \infty$
For A1 = 0.5, $p_o = ((A_6 - A_4)^2 - A_2)/A_3$
For A1 = 1.0, $p_o = (A_6 - (A_2 + A_4))/A_3$
For A1 = 2.0, $p_o = (-A_3 + (A_3^2 - 4A_4 (A_2 - A_6))^{1/2})/(2A_4)$
For A1 = 3.0, $p_o = A_2$

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property	Skeleton Property for MODELNO = 5 (Generalized Hoek and Brown Model)	5.3.2.4.5 $A_7 = p_{BD}$ Brittle-Ductile transition pressure $A_8 = r_i$ Initial dilatancy parameter Dilatancy parameter r is calculated as For $p_{BD} > 0.0$ and $p < p_{BD}$ $r = r_i \ (1 - p / p_{BD})$ For $p_{BD} > 0.0$ and $p \geq p_{BD}$ $r = 0.0$ For $p_{BD} = 0.0$ $r = r_i$ Note: Derivatives of potential function (Q) are related to the yield function (F) as $\partial Q / \partial p = (\partial F / \partial p)\ r$ $\partial Q / \partial q = (\partial F / \partial q)$ $\partial Q / \partial \theta = (\partial F / \partial \theta)$ For associated flow rule use $A_7 = 0.0$, $A = 1.0$ and set ISYMSOL=1 in Card 1.1 For no plastic volume change use $A_7 = 0.0$, $A_8 = 0.0$ and set ISYMSOL=2 in Card 1.1 For all non associated cases set ISYMSOL=2 in Card 1.1

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property	Skeleton Property for MODELNO = 5 (Generalized Hoek and Brown Model)	5.3.2.4.5

5.3.2.4.5

Pressure - Dependent Moduli

IBULK, ISHEAR

IBULK = 0 Constant bulk modulus
= 1 Nonlinear bulk modulus

ISHEAR = 0 Constant shear modulus
= 1 Constant Poisson's ratio

Loading Bulk Modulus Definition

NLPC

NLPC Number of volumetric pressure/modulus pairs describing the virgin loading bulk modulus

NLPC Cards
P_1, B_{L1}
P_2, B_{L2}
- -
P_n, B_{Ln}

P_i, B_{Li} Pressure and bulk modulus pairs

Unloading Bulk Modulus Definition

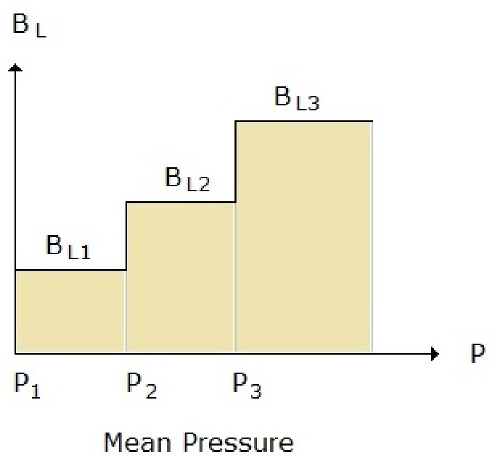
NUPC

NUPC Number of volumetric pressure/modulus pairs describing unloading bulk modulus

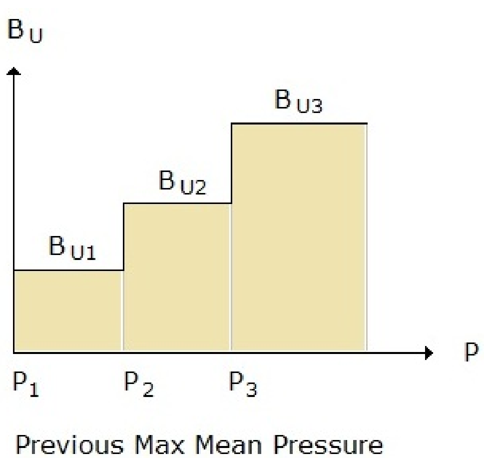
NUPC Cards
P_1, B_{U1}
P_2, B_{U2}
- -
P_n, B_{Un}

P_i, B_{Ui} Pressure and bulk modulus pairs

Loading Bulk Modulus as a Function of Mean Pressure



Unloading Bulk Modulus as a Function of Previous Max Pressure



Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property	Skeleton Property for MODELNO = 6 (Advanced Plastic Model)	5.3.2.4.6 For MODELNO = 6 [Advanced Elasto-plastic Model] Not Available

Card Group	Input Data and Definitions (Main File)
5 Continuum Element / 5.3 Material Property / Skeleton Property for MODELNO = 7 (Single Hardening Elasto-Plastic Model)	5.3.2.4.7

5.3.2.4.7

For MODELNO = 7 [Single Hardening Elasto-Plastic Model]

Not Available

Precision Parameters

NDVMIN, NDVMAX, DEEMAX, NUNLOAD, NDRIFT

NDVMIN		Min number of plastic strain sub increment
NDVMAX		Max number of plastic strain sub increment
DEEMAX		Max plastic strain sub increment
NUNLOAD	= 0	Smooth initial unloading
	= 1	No smooth unloading
NDRIFT	= 0	Drift correction
	= 1	No drift correction

Tensile Strength

APEX, ATMO

APEX	Tensile strength T
ATMO	Atmospheric pressure P_a

Elastic Constant

AKUR, AN, APOI

AKUR	Elastic Young's modulus constant Kur
AN	Elastic Young's modulus exponent n
APOI	Elastic Poisson's ratio v

Isotropic Hardening

NACRV

AACC(I), AAPC(I), ABRK(I) I = 1, NACRV

NACRV	Number of segments for isotropic hardening function
AACC	Isotropic hardening constant C

Card Group	Input Data and Definitions (Main File)
5 Continuum Element / 5.3 Material Property / Skeleton Property for MODELNO = 7 (Single Hardening Elasto-Plastic Model)	5.3.2.4.7

AAPC Isotropic hardening constant P
ABRK Break point in terms of W_p/P

Failure Constant
AK, AMY, AETA1

AK The ratio of triaxial extensive to compressive strength at a given pressure
AMY Failure exponent m
AETA1 Failure constant η_1

Yield Constant
AY1, AH, ALPHA

AY1 Yield constant ψ_1
AH Yield constant h
ALPHA Yield constant α

Potential Constant
AY2, AMU

AY2 Potential constant ψ_2
AMU Potential constant μ

Unload/Reload Constant
AHLAM, AHGAM, AHBET, APCO

AHLAM (λ), AHGAM (γ), AHBET (β)
These unload/reload constants are not used

APCO Effective mean pressure at which yielding begins

Card Group	Input Data and Definitions (Main File)
5 Continuum Element / 5.3 Material Property / Skeleton Property for MODELNO = 8 (JWL High Explosive Model)	5.3.2.4.8

5.3.2.4.8

For MODELNO = 8 [JWL High Explosive Model]

Elastic Constant

E, v

Note: When using JWL model, specify NLNR = 1 and NGEN = 1 in Card 2

JWL Model Parameters

A, B, R_1, R_2, ω, E_v

- A — JWL material constant (Megabar)
- B — JWL material constant (Megabar)
- R_1 — JWL material constant (Dimensionless)
- R_2 — JWL material constant (Dimensionless)
- ω — JWL material constant (Dimensionless)
- E_v — Chemical energy density of explosive (Megabar cc/cc)

Burn Fraction Parameters

C_d, B_s, XL

- C_d — Detonation velocity
- B_s — Constant used to spread the detonation front [Usually set B_s = 2.5]
- XL — Characteristic length of element. If XL = 0.0, program computes XL

Note:
If C_d = 0 and B_s = 0, XL represents LHNO (Pressure Load History Number) specified in Cards 9.2.3.1 through 9.2.3.5 and above JWL parameters are ignored

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 9 (Modified Cam Clay Model)	5.3.2.4.9

For MODELNO = 9 [Modified Cam Clay Model]

Cam-Clay Material Parameters

P_c, e_o, v, C_c, C_r, M, G_o

- P_c Preconsolidation pressure
- e_o Initial void ratio
- v Poisson's ratio
- C_c Virgin compression index
- C_r Swelling/recompression index
- M Strength parameter
- G_o Initial elastic shear modulus at P_c
  When G_o = 0, shear modulus is computed based on v

Tensile Strength Parameters

NTCUT

NTCUT = 0 No tension cut-off
= 1 Tension cut-off

For NTCUT = 1, otherwise go to next Card

T, ST_n, ST_s

- T Tensile strength
- ST_n Factor used to divide stiffness normal to tensile crack
- ST_s Factor used to divide shear modulus for cracked zone

Note:
To ignore stiffness reduction associated with tensile crack, use ST_n = ST_s = 1.0

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO =9 (Modified Cam Clay Model)	5.3.2.4.9 Creep Option NCREEP NCREEP = 0 No creep = 1 Only volumetric creep = 2 Only deviatoric creep = 3 Both volumetric and deviatoric creep Volumetric Creep Parameters (For NCREEP = 1 or 3) t_{vi}, C_{α} t_{vi} Initial volumetric age C_{α} Secondary compression coefficient Deviatoric Creep Parameters (For NCREEP = 2 or 3) t_{di}, A, α, m t_{di}, Initial deviatoric age A Sing-Mitchell creep parameter α Sing-Mitchell creep parameter m Sing-Mitchell creep parameter Note: Deviatoric creep is not available

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 10 (Engineering Model)	5.3.2.4.10 For MODELNO = 10 [Engineering Model] Strength Parameters NSTYPE ST1, Y1, S1, VM1 NSTYPE = 1 Single failure surface = 2 Double falling failure Surface Peak ST1 Tensile failure limit Y1 Yield stress intercept S1 Slope VM1 Von Mises limit For NSTYPE = 2 FSRATE ST2, Y2, S2, VM2 FSRATE Rate of deviatoric plastic strain at which failure surface drops to residual level Residual ST2 Tensile failure limit Y2 Yield stress intercept S2 Slope VM2 Von Mises limit Loading Modulus NLS EBL(i), BKL(i), POL(i) i = 1, NLS NLS Number of loading slopes EBL(i) Volume strain breakpoint between loading slopes i and i+1 BKL(i) Bulk modulus for loading slope i POL(i) Poisson's ratio for loading slope i

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 10 (Engineering Model)	5.3.2.4.10

Unloading Modulus

NUS
PBU(i), BKU(i), POU(i) i = 1, NUS

NUS — Number of unloading slopes

PBU(i) — Pressure breakpoint between unloading slopes i and i+1

BKU(i) — Bulk modulus for unloading slope i

POU(i) — Poisson's ratio for unloading slope i

Note: Special case for NLS = 1

1. Loading and unloading modulus are assumed to be the same. Input data for unloading Modulus is not considered

2. Tension cutoff is based on individual principal stress. The limit of tensile stress is equal to ST1 / 3

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 11 (Joint Model)	5.3.2.4.11

For MODELNO = 11 [Joint Model]

Elastic Modulus and Thickness

NM
E, G, t, v

NM	= 0	Linear elastic joint
	= 1	Nonlinear joint
	= 2	Lumped nonlinear joint
	= 3	Contact nonlinear joint
	= 4	Thin Layer Element

E	Elastic Young˙s modulus
G	Elastic shear modulus
t	Joint thickness
v	Poisson˙s ratio (Used for NM = 4)

Strength Parameters (Only for NM > 0)

C, φ, r

C	Cohesion
φ	Friction angle (˚)

r	= -1	Decoupled volume and shear
	= 0	No plastic volume change (N.A.)
	= 1	Associated flow rule (N.A.)
	= -2	Decoupled shear (N.A.)

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 11 (Joint Model)	5.3.2.4.11

5.3.2.4.11

Normal Stress-Strain Relation (Only for NM = 1,2,3)

ε_1, σ_1
ε_2, σ_2
ε_3, σ_3
ε_4, σ_4

ε_i, σ_i — Pair of strain and stress to define normal stress-strain relation (Tension is positive)

Tensile Strength (Only for NM = 4)

TENSTR

TENSTR — Tensile strength

Note:

1. For t > 0.0, coordinateso of joint element is adjusted based on t
2. For t < 0.0, no adjustment of coordinates. Users input mesh should represent joint thickness t
3. For t = 0.0 and NM = 4, joint thickness by user's input coordinate
4. Lumped nonlinear joint (NM=2) has better performance than nonlinear joint (NM=1). Contact nonlinear joint (NM=3) has no resistance in shear.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 12 (Duncan and Chang Hyperbolic Model)	5.3.2.4.12 For MODELNO = 12 ([Duncan and Chang Hyperbolic Model] A_1, A_2, A_3, A_4, A_5, R_f A_1 = 1.0 A_2 = 1000. A_3 = 6 sinφ / (3 - sinφ) A_4 = 6 cosφ C / (3 - sinφ) - 1000 A_5 = 1.0 R_f = 0.7 ~ 0.9 <u>Loading Bulk Modulus Definition</u> NLPC NLPC Number of volumetric strain/modulus/ Poisson's ratio pairs describing the virgin loading NLPC Cards ┌ EBL_1, BKL_1, POL_1 │ EBL_2, BKL_2, POL_2 │ - - - └ EBL_n, BKL_n, POL_n EBL, BKL, POL Refer to Card 5.3.2.4.10 <u>Unloading Bulk Modulus Definition</u> NUPC NUPC Number of volumetric pressure/modulus /Poisson's ratio pairs describing the unloading NUPC Cards ┌ PBU_1, BKU_1, POU_1 │ PBU_2, BKU_2, POU_2 │ - - - └ PBU_n, BKU_n, POU_n PBU, BKU, POU Refer to Card 5.3.2.4.10

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 14 (User Defined Model)	5.3.2.4.14 For MODELNO = 14 [User Defined Model] 60 Cards ┌ PROP (41) │ PROP (42) │ - │ - └ PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL14.FOR in the directory C:\SMAP\SMAP2D\PROGRAM\USER\MODEL-14. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL14.FOR. 2. MODEL14.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE14.BAT. 3. Text file LABEL14.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL14.DLL can be obtained once compiled. MODEL14.DLL should be saved in the directory C:\SMAP\SMAP2D\PROGRAM.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 15 (User Defined Model)	5.3.2.4.15 For MODELNO = 15 [User Defined Model] 60 Cards: PROP (41), PROP (42), -, -, PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL15.FOR in the directory C:\SMAP\SMAP2D\PROGRAM\USER\MODEL-15. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL15.FOR. 2. MODEL15.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE15.BAT. 3. Text file LABEL15.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL15.DLL can be obtained once compiled. MODEL15.DLL should be saved in the directory C:\SMAP\SMAP2D\PROGRAM.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 16 (User Defined Model)	5.3.2.4.16 For MODELNO = 16 [User Defined Model] 60 Cards ┌ PROP (41) │ PROP (42) │ - │ - └ PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL16.FOR in the directory C:\SMAP\SMAP2D\PROGRAM\USER\MODEL-16. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL16.FOR. 2. MODEL16.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE16.BAT. 3. Text file LABEL16.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL16.DLL can be obtained once compiled. MODEL16.DLL should be saved in the directory C:\SMAP\SMAP2D\PROGRAM.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 17 (User Defined Model)	5.3.2.4.17

For MODELNO = 17 [User Defined Model]

60 Cards
┌ PROP (41)
│ PROP (42)
│ -
│ -
└ PROP (100)

PROP (41) - PROP (100):
Material constants related to the User's Model.

Note:

1. Users can use their own material model by modifying file MODEL17.FOR in the directory C:\SMAP\SMAP2D\PROGRAM\USER\MODEL-17. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL17.FOR.

2. MODEL17.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE17.BAT.

3. Text file LABEL17.DAT can be modified appropriately.

4. Dynamic Link Library file MODEL17.DLL can be obtained once compiled.
   MODEL17.DLL should be saved in the directory C:\SMAP\SMAP2D\PROGRAM.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 18 (User Defined Model)	5.3.2.4.18 For MODELNO = 18 [User Defined Model] 60 Cards ┌ PROP (41) │ PROP (42) │ - │ - └ PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL18.FOR in the directory C:\SMAP\SMAP2D\PROGRAM\USER\MODEL-18. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL18.FOR. 2. MODEL18.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE18.BAT. 3. Text file LABEL18.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL18.DLL can be obtained once compiled. MODEL18.DLL should be saved in the directory C:\SMAP\SMAP2D\PROGRAM.

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.3 Skeleton Property for MODELNO = 21 (PM4Sand Model)	5.3.2.4.21 For MODELNO = 21 [PM4Sand Model] D_R G_o h_{po} p_a N_s S_{cheme} T_{antyp} Secondary Parameters (Skip these cards for N_s = 1) h_o e_{max} e_{min} n^b n^d A_{do} z_{max} c_z C_e ϕ_{cv} v_o C_{GD} C_{DR} c_{kaf} Q R m $F_{sed.min}$ p_{sed} D_R Apparent relative density (Fraction) G_o Shear modulus coefficient h_{po} Contraction rate parameter p_a Atmospheric pressure (10.33 for stress unit t/m^2) N_s Secondary parameter specification: 0 = Yes, 1 = No S_{cheme} Integration scheme (Use S_{cheme} = 0) T_{antyp} Drift correction method (Use T_{antyp} = 0) h_o Control parameter for ratio of plastic to elastic modulus e_{max} Maximum void ratio (Default 0.8) e_{min} Minimum void ratio (Default 0.5) n^b Control parameter for dilatancy & peak friction angle n^d Control parameter for transition from contr. to dilation A_{do} Bolton's dilatancy parameter z_{max} Maximum allowable fabric dilatancy tensor z c_z Control parameter when fabric effects get important C_e Control parameter for adjusting strain accumulation rate ϕ_{cv} Critical state effective friction angle (Default 33°) v_o Poisson's ratio (Default 0.3) C_{GD} Factor for shear modulus degradation (Default 2.0) C_{DR} Control parameter for rotated dilatancy surface c_{kaf} Control parameter for effects of sustained shear stress Q, R Parameters for Bolton's empirical critical state line m Parameter defining size of yield stress (Default 0.01) $F_{sed.min}$ Parameter for post-shaking elastic modulus reduction p_{sed} Mean effective stress for post-shaking reconsolidation Set -1 for default values of secondary model parameters. For description, refer to Boulanger, R. W. And ziotopoulou, k. PM4Sand (Version 3.1): A Sand Plasticity Model for Earthquake Engineering Applications, Report No UCD/CGM-17/01, Dept. of Civil & Env. Eng., U. of Cal., Davis, CA, 109 pp.

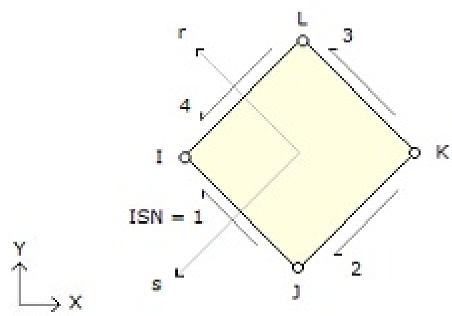
<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="2">5
Continuum Element</td><td rowspan="2">5.5
Skew Boundary</td><td colspan="2">5.5.1
NSKEW

NSKEW Number of element sides on skew boundary</td></tr>
<tr><td>For Each Skew Boundary (If NSKEW = 0, skip this Card)</td><td>5.5.2
NEL, NDT, NDH, ISN, MSF

NEL Element number

NDT Node number at tail of arrow
NDH Node number at head of arrow
ISN Side number on skew boundary

MSF = 1 Skew in fluid phase only
= 2 Skew in solid phase only
= 3 Skew in solid and fluid phases



Note:
1. Positive boundaries run parallel to element boundaries, from node K toward node L
2. Side numbers are counterclockwise from node I
3. Input forces or velocities and output motions on skew boundaries are parallel to the element boundaries
4. Only one skew boundary is allowed for each element</td></tr>
</table>

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.6 Initial Stress	5.6.1 IEFST IEFST = 0 Zero initial effective stress = 1 Specified initial effective stress
		5.6.2 If IEFST =1, list initial effective stresses for each element SXX, SYY, SZZ, SXY (NCONT Cards) SXX σ_x' (Normal stress in x direction) SYY σ_y' (Normal stress in y direction) SZZ σ_z' (Normal stress in z direction) SXY τ_{xy} (Shear stress in xy plane) Note: For joint element (KS > 0), SYY represents joint nomal stress, SXY represents joint shear stress and SXX = SZZ = 0.0
		5.6.3 IPOFP IPOFP = 0 Zero initial pore fluid pressure = 1 Specified initial pore fluid pressure
		5.6.4 If IPOFP =1, list initial pore fluid pressure for each element PRF (NCONT Cards) PRF List initial pore fluid pressures for each element, specified sequentially from 1 to NCONT

Card Group	Input Data and Definitions (Main File)

5 Continuum Element

5.7 Element Surface

5.7.1

NUMEST, MATEST

NUMEST Nunber of material & element surface traction
MATEST Nunber of material surface traction

If NUMEST = 0, go to Card Group 6

For Each Material / Element Surface

5.7.2.1

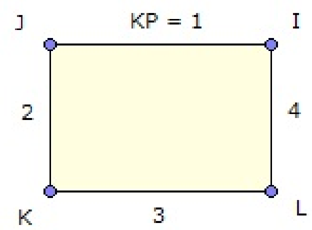
(MATEST) Cards
MAT, KP, KH, KD, a_0, a_1, a_2

(NUMEST - MATEST) Cards
NEL, KP, KH, KD, a_0, a_1, a_2

MAT	Material number
NEL	Element number
KP	Element surface designation number
KH	Load history number specified in Cards 9.2.3.1 through 9.2.3.5 If KH = 0, constant static pressure/ traction vector is acting all the time
KD = 0	Uniformly distributed traction vector defined in local coordinate system $P'_n = a_0$ $P_x = a_1$ $P_y = a_2$
= 1	Uniformly distributed traction vector defined in global coordinate system $P'_n = a_0$ $P_X = a_1$ $P_Y = a_2$ P'_n is static normal pressure Compression is positive
= 2	Linearly distributed static normal pressure $P_{n1} = a_1$ at I_1' $P_{n2} = a_2$ at I_2'

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.7 Element Surface	For Each Material / Element Surface	5.7.2.1 Linearly distributed surface tractions defined in global coordinate system = 3 q_X q_{X1} = a_1 at I_1' q_{X2} = a_2 at I_2' = 4 q_Y q_{Y1} = a_1 at I_1' q_{Y2} = a_2 at I_2' = 5 Static normal pressure given as functions of global X and Y coordinates $P'_n = a_0 + a_1 X + a_2 Y$ Global surface traction given as functions of global X, and Y coordinates = 6 q_X $q_X = a_0 + a_1 X + a_2 Y$ = 7 q_Y $q_Y = a_0 + a_1 X + a_2 Y$ Note1: Element traction is not available for KS = -1 (High Explosive Solid Element) Note2: (NEL1, -NEL2) generates the same surface traction from NEL1+1 to NEL2. This also applies to material based traction. Refer to description in next page

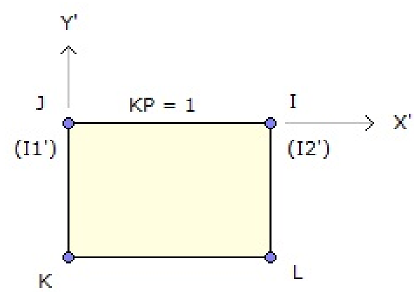
Element Surface Designation Number



Element Local Coordinate Axes

KP	Quadrilateral Element		Triangular Element	
	I_1'	I_2'	I_1'	I_2'
1	J	I	J	I
2	K	J	K	J
3	L	K	I	K
4	I	L	0	0

For KP = 1



Card Group	Input Data and Definitions (Main File)
6 Beam Element	6.1 NBEAM NBEAM Total number of beam element If NBEAM = 0, go to Card group 7
	6.2 NBTYPE, NSPTB, NBLT NBTYPE =0 Axially symmetric shell =2 Plane stress in direction transverse to beam axis and plane strain in z direction =3 Plane stress in both transverse and z directions NSPTB <u>3 Gauss points for integration</u> = 0 Stresses at integration points = 1 Stresses at center of each layer = 2 Stresses at integration points and member ends. <u>Equally spaced int. points with member ends</u> = 3 Stresses at 3 integration points = 5 Stresses at 5 integration points <u>Equally spaced int. points without member ends</u> =-3 Stresses at 3 integration points =-5 Stresses at 5 integration points NBLT = 0 Built-in layered beam = 1 User-defined layered beam = 2 Conventional elastic beam = 3 Reinforced axisymmetric shell for NBTYPE = 0. When used for NBTYPE = 2 or 3, A_{S1} and A_{S2} represent total area per unit depth and A_{S3} and A_{S4} are not considered.

Card Group			Input Data and Definitions (Main File)
6 Beam Element	6.3 For NBLT= 0 (Built-in Layered Beam)	Material Property Data	6.3.1.1 Concrete property E_c, υ_c, φ, C, K, T, ST_n, ST_s E_C Young's modulus υ_c Poisson's ratio φ Internal frictional angle (°) C Cohesion K The ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same pressure T Tensile strength ST_n Factor used to divide stiffness normal to tensile crack ST_s Factor used to divide shear modulus for the cracked zone Note: For ST_n= 0 and ST_s= 0, beam axial and shear deformations are assumed to be decoupled
			6.3.1.2 Steel plate property E_s, υ_s, σ_s E_s Young's modulus υ_s Poisson's ratio σ_s Shear strength in the triaxial compression
			6.3.1.3 Reinforcing bar property E_r, υ_r, σ_r E_r Young's modulus υ_r Poisson's ratio σ_r Shear strength in the triaxial compression

Card Group	Input Data and Definitions (Main File)		
6 Beam Element	6.3 For NBLT= 0 (Built-in Layered Beam)	Section Property Data	6.3.2.1 NTNS NTNS Number of different beam sections

Card Group	Input Data and Definitions (Main File)
6 Beam Element / 6.3 For NBLT = 0 (Built-in Layered Beam) / Section Property Data / For Each Section	6.3.2.2.1

6.3.2.2.1

NSEC, NFSHR, MR, NSTYPE, NLAYR, NEHNO,
WL, RHOL, CTS, DAMP

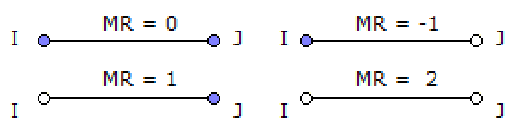
NSEC — Beam section number

NFSHR = 0 Neglect shear deformation
= 1 Include shear deformation

MR — Moment Release
= 0 No hinge
= 1 Hinge at node I
= -1 Hinge at node J
= 2 Hinge at node I and J

Spring Element at Node I
= 11 Axial spring ($Kx = E A / L$)
= 12 Shear spring ($Ky = 12 E I / L^3$)
= 13 Rotational spring ($Kr = 4 E I / L$)

Use Negative for Spring at Node J



NSTYPE — Type of built-in section

NLAYR — Total number of layers (Max=20)

NEHNO — Young's modulus multiplication factor history number in Card Group 9.2.3

WL — Weight per unit length of beam

RHOL — Mass per unit length of beam (used for dynamic analysis)

CTS — Timoshenko shear coefficient

DAMP — Critical damping ratio

Card Group	Input Data and Definitions (Main File)

6 Beam Element

6.3 For NBLT = 0 (Built-in Layered Beam)

Section Property Data

For Each Section

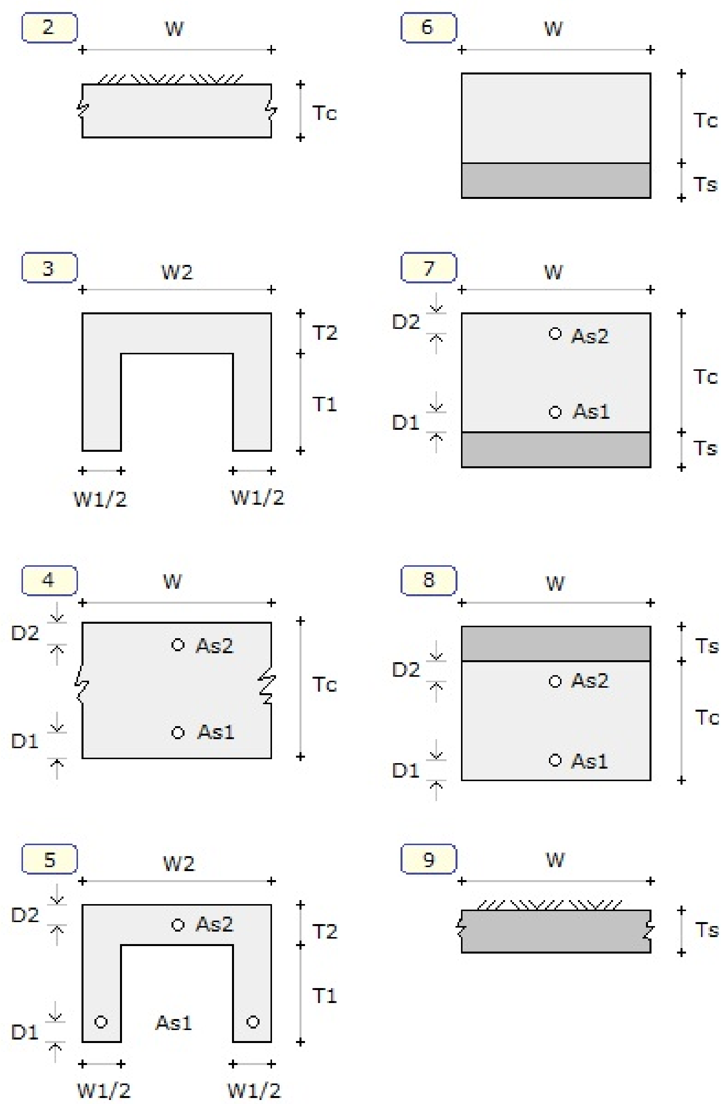
6.3.2.2.2

NSTYPE

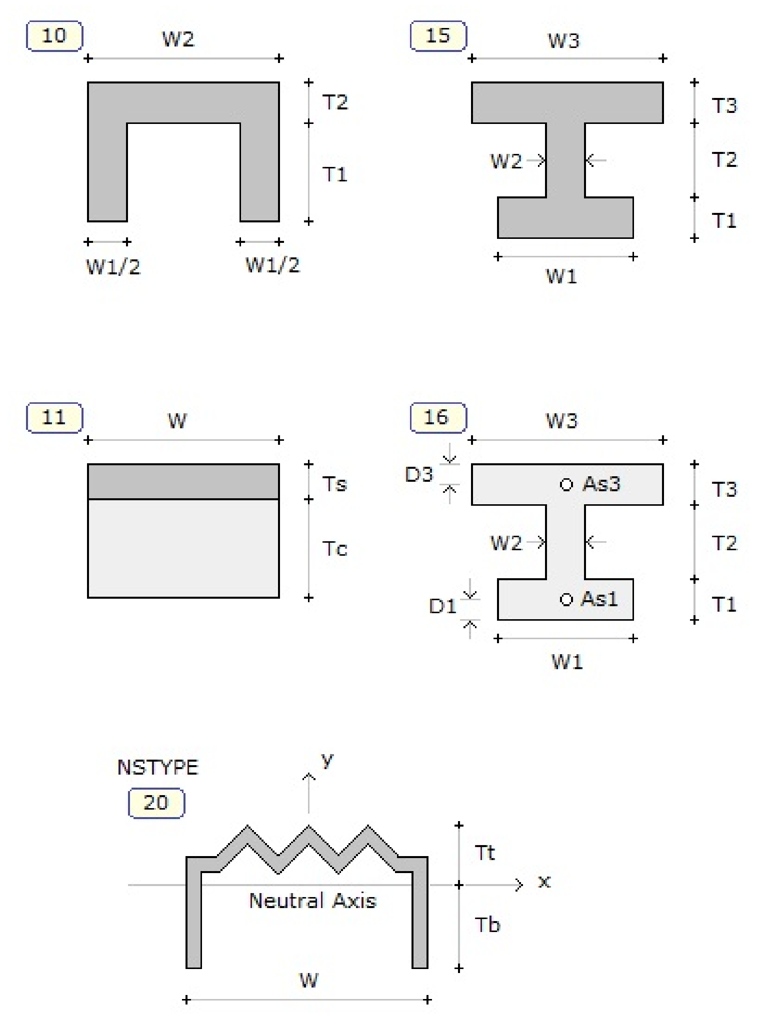
NSTYPE		
= 1	>	No beam, skip this Card
= 2	>	T_c, W
= 3	>	T_1, T_2, W_1, W_2
= 4	>	T_c, D_1, A_{s1}, D_2, A_{s2}, W
= 5	>	T_1, T_2, W_1, W_2, D_1, A_{s1}, D_2, A_{s2}
= 6	>	T_c, T_s, W
= 7	>	T_c, D_1, A_{s1}, D_2, A_{s2}, T_s, W
= 8	>	T_c, D_1, A_{s1}, D_2, A_{s2}, T_s, W
= 9	>	T_s, W
=10	>	T_1, T_2, W_1, W_2
=11	>	T_c, T_s, W
=15	>	T_1, T_2, T_3, W_1, W_2, W_3
=16	>	T_1, T_2, T_3, W_1, W_2, W_3 D_1, A_{s1}, D_3, A_{s3}
=20	>	T_b, T_t, W, A, I A: Cross section area I : Moment of inertia (Elastic material only)

Note: NSTYPE = 2 and 4 can be used for NBTYPE = 0 to represent axial reinforcement A_{S1} and A_{S2} per 1 radian

Liner types are shown in the next page



Type of Built-in Cross Section



Type of Built-in Cross Section

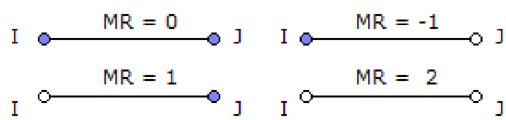
<table>
<tr><th>Card Group</th><th colspan="4">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="2">6
Beam Element</td><td rowspan="2">6.4
For NBLT = 1 (User-defined Layered Beam)</td><td rowspan="2">Material Property Data</td><td colspan="2">6.4.1.1
NTNB

NTNB   Number of material property (Max=50)</td></tr>
<tr><td>For Each Material</td><td>6.4.1.2.1
MATNO, MODELNO, NEHNO

MATNO   Material number
MODELNO   Material model number
NEHNO   Young's modulus multiplication factor history number in Card Group 9.2.3</td></tr>
</table>

Card Group				Input Data and Definitions (Main File)
6 Beam Element	6.4 For NBLT = 1 (User-defined Layered Beam)	Material Property Data	For Each Mateтial	6.4.1.2.2 MODELNO = 1 Elastic Model E, ν = 2 Von Mises Model E, ν σ = 3 Mohr-Coulomb Model E, ν φ, C, K, T, ST_n, ST_s E Young's modulus ν Poisson's ratio σ Shear strength in the triaxial compression φ Internal frictional angle (°) $C = \frac{(1 - \sin\varphi)}{2\cos\varphi}\sigma_c$ C Cohesion K The ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same pressure T Tensile strength $T = \frac{2\cos\varphi}{(1 + \sin\varphi)} C$ ST_n Factor used to divide stiffness normal to tensile crack ST_s Factor used to divide shear modulus for the cracked zone Note: For ST_n=0 and ST_s=0, beam axial and shear deformations are assumed to be decoupled. To ignore stiffness reduction associated with tensile crack, use ST_n=ST_s=1.0

Card Group			Input Data and Definitions (Main File)
6 Beam Element	6.4 For NBLT = 1 (User-defined Layered Beam)	Section Property Data	6.4.2.1 NTNS NTNS Number of beam sections (Max=50)

Card Group				Input Data and Definitions (Main File)
6 Beam Element	6.4 For NBLT = 1 (User-defined Layered Beam)	Section Property Data	For Each Section	6.4.2.2.1 NSEC, NFSHR, MR, WL, YNA, RHOL, CTS, DAMP NSEC — Beam section number NFSHR = 0 Neglect shear deformation = 1 Include shear deformation MR — Moment Release = 0 No hinge = 1 Hinge at node I = -1 Hinge at node J = 2 Hinge at node I and J Spring Element at Node I = 11 Axial spring (Kx = E A / L) = 12 Shear spring (Ky = 12 E I / L^3) = 13 Rotational spring (Kr = 4 E I / L) Use Negative for Spring at Node J  WL — Weight per unit length of beam YNA — Distance from neutral axis to top extreme fiber RHOL — Mass per unit length of beam (used for dynamic analysis) CTS — Timoshenko shear coefficient DAMP — Critical damping ratio

Card Group	Input Data and Definitions (Main File)

6 Beam Element

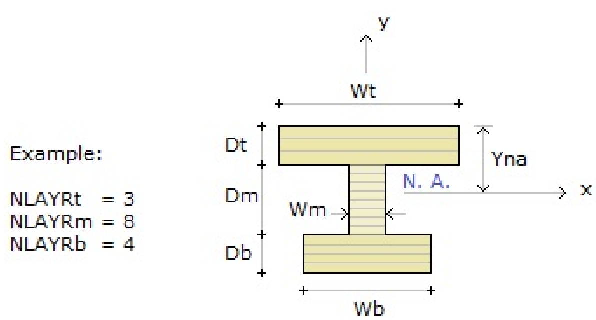
6.4 For NBLT = 1 (User-defined Layered Beam)

For Each Section

6.4.2.2.2

MATb, NLAYRb, Db, Wb
MATm, NLAYRm, Dm, Wm
MATt, NLAYRt, Dt, Wt

MATb	Material number for bottom component
NLAYRb	Number of layers for bottom component
MATm	Material number for middle component
NLAYRm	Number of layers for middle component
MATt	Material number for top component
NLAYRt	Number of layers for top component



Note: NLAYRb + NLAYRm + NLAYRt ≤ 20

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="2">6
Beam Element</td><td rowspan="2">For NBLT = 1 (User-defined Layered Beam)</td><td rowspan="2">For Each Section</td><td>6.4.2.2.3
NFRBR

NFRBR Number of reinforcing bars</td></tr>
<tr><td>6.4.2.2.4
If NFRBR = 0, skip this Card

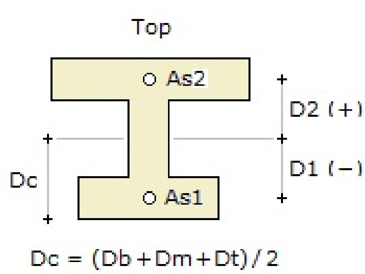
MATBR

NFRBR Cards: D_1, A_{S1} / D_2, A_{S2} / - - / - -

MATBR Material property number for reinforcing bar

D Distance from the mid height to the reinforcing bar. Upward is positive

A_s Cross section area of reinforcing bar at distance D

</td></tr>
</table>

Card Group			Input Data and Definitions (Main File)
6 Beam Element	6.5 For NBLT = 2 (Conventional Elastic Beam)		6.5.1.1 NTNS NTNS Number of beam sections (Max=50)
		For Each Section	6.5.1.2 NSEC, NFSHR, MR, NEHNO, WL, RHOL, CTS, DAMP Refer to Card 6.3.2.2.1 MR = 3 Joint spring element Available only for NBLT = 2
			6.5.1.3 For MR ≠ 3 T_b, T_t, W, A, I, E, v Refer to NSTYPE = 20 in Page 4-72 For MR = 3 K_X, K_y, K_R K_X Axial spring stiffness K_Y Shear spring stiffness K_R Rotational spring stiffness

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="4">6

Beam Element</td><td rowspan="4">6.6

For NBLT = 3 (Reinforced Axisymmetric Shell)</td><td colspan="2">6.6.1.1

NTNS

NTNS   Number of beam sections (Max=50)</td></tr>
<tr><td colspan="2">6.6.1.2

[Concrete and Reinforcing Bar Properties]

E_c, υ_c, ϕ, C, K, T, ST_n, ST_s (See 6.3.1.1)
E_r, υ_r, σ_r (See 6.3.1.3)</td></tr>
<tr><td rowspan="2">For Each Section</td><td>6.6.1.3

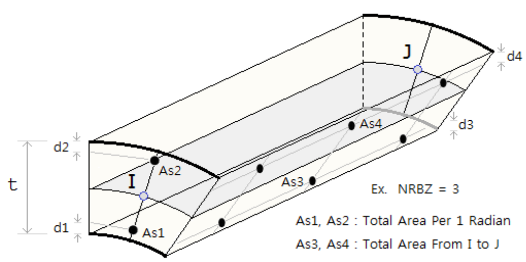
NSEC, NFSHR, MR, NEHNO, GAMA, RHO, CTS, DAMP

GAMA   Unit weight
RHO   Mass density

Moment Release (MR = 0, 1, -1, 2) is available
Refer to Card 6.3.2.2.1 for other parameters</td></tr>
<tr><td>6.6.1.4

t, D_1, A_{s1}, D_2, A_{s2}
NRBZ, D_3, A_{s3}, D_4, A_{s4}

To exclude particular rebar, set As = 0.


</td></tr>
</table>

Card Group	Input Data and Definitions (Main File)
7 Truss Element	7.1 NTRUSS NTRUSS Total number of truss elements If NTRUSS = 0, go to Card Group 8
	7.2 NTRST NTRST Use NTRST = 1
	7.3 NTNT, $MATP_1$, $MATP_2$, $MATP_3$ NTNT Number of material property set for truss element MATP Material number of parent continuum element which is not allowed to embed truss element

Card Group		Input Data and Definitions (Main File)
7 Truss Element	7.4 For Each Material	7.4.1 MATNO, ME, MS MATNO — Material number ME = 0 — No embedment = 1 — Embedded with auto subdivision = 2 — Embedded with no subdivision = 3 — Embedded using input NELPI and NELPJ See Card 5.2 in Mesh File description =-N — Embedded with N equal subdivision MS = 0 — No slip = 1 — Monotonic loading path = 2 — Arbitrary loading path = n — (n > 2) Plastic stiffness = Kslip x 10^{-n} Note: For ME = 1, 2, and -N, input files of Mesh and Main are automatically updated
		7.4.2 A, WL, RHO, E, STRSI, DAMP A — Cross section area WL — Weight per unit length of truss RHO — Mass density (Used for dynamic analysis) To lump all mass at node J, use -RHO E — Young's modulus STRSI — Initial stress. Tension is positive For constant initial stress, use E = 0 DAMP — Critical damping ratio Negative for viscous damping constant

Card Group		Input Data and Definitions (Main File)
7 Truss Element	7.4 For Each Material	7.4.3 If NLNR = 0 and NGEN = 0, skip this Card σ_{yc}, σ_{yt}, ε_f, I, y_{max} σ_{yc} Yield stress in compression σ_{yt} Yield stress in tension ε_f Strain at rupture For $\varepsilon_f \leq \sigma_y/E$, ε_f represents Yield strain at tension I Moment of inertia (Minimum) y_{max} Distance from neutral axis to extreme fiber (Maximum) $\sigma_{yc} = \sigma_{yt} = 0$: Linear elastic material $\sigma_{yc} = 0$: No compression (Cable) $\sigma_{yt} = 0$: No tension (Strut) $I = 0$: No buckling $y_{max} = 0$: No yield on buckling
		7.4.4 If MS = 0, skip this Card Kslip, Cmax, Cres, Umax, Ures, Dslip Kslip Stiffness for shear stress - slip displacement Cmax Maximum cohesion Cres Residual cohesion (N.A.) Umax Slip at the end of Cmax (N.A.) Ures Slip at the beginning of Cres (N.A.) Dslip Diameter of slip surface

Card Group	Input Data and Definitions (Main File)
8	8.1 NFAD, MCFAD, MBFAD, MTFAD NFAD Number of materials / elements with activity MCFAD Number of continuum materials with activity MBFAD Number of beam materials with activity MTFAD Number of truss materials with activity If NFAD = 0, go to Card Group 9
Element Activity	8.2 (MCFAD) Cards [MATC, NAC, NDAC (MBFAD) Cards [MATB, NAC, NDAC- (MTFAD) Cards [MATT, NAC, NDAC (NFAD - MCFAD - MBFAD - MTFAD) Cards [NEL, NAC, NDAC MATC Continuum material number MATB Beam material number MATT Truss material number NEL Element number NAC Load step at which an element is activated NDAC Load step at which an element is deactivated Note: If initially active and deactivated at step 5: NAC = 0, NDAC = 5 If active permanently from step 20: NAC = 20, NDAC > NCYCL (NEL1, -NEL2) generates the same activity from NEL1+1 to NEL2. This also applies to material based activity.

Card Group		Input Data and Definitions (Main File)
9 Loads	9.1 Gravity Load	9.1.1 NGSTEP NGSTEP = 0 No gravity load is applied > 0 Number of load steps (Cycles) through which the gravity load is applied incrementally Note: During gravity load step, inertia forces are not active To print time history output after NGSTEP, use negative value of NGSTEP. Output times would be relative to the time at NGSTEP
		9.1.2 If NGSTEP = 0, go to Card Group 9.1.3 IRELD, FRX, FRY, NHFRX, NHFRY IRELD = 0 Displacements/strains include gravity load = 1 Displacements/strains after NGSTEP are relative to gravity load FRX X component of unit gravity load FRY Y component of unit gravity load NHFRX Intensity history number in X direction NHFRY Intensity history number in Y direction Note: Intensity is specified through Card 9.2.3 Intensity Times Distribution Factor will be additive to FRX or FRY

Card Group		Input Data and Definitions (Main File)
9 Loads	9.1 Gravity Load	9.1.2.1 If NHFRX = 0, skip this card A_0, A_1, A_2, A_3, Y_1, Y_2 A_i Distribution factor Y_i Global Y coordinate For $Y < Y_1$ $A_i = A_0$ For $Y > Y_2$ $A_i = A_3$ For others $A_i = A_1 + (Y - Y_1) * (A_2 - A_1) / (Y_2 - Y_1)$
		9.1.2.2 If NHFRY = 0, skip this card A_0, A_1, A_2, A_3, Y_1, Y_2 A_i Distribution factor Y_i Global Y coordinate

Card Group		Input Data and Definitions (Main File)
9 Loads	9.1 Specified Displacement	9.1.3 NUMDIS NUMDIS — Total number of degrees of freedom at which input displacement time histories are specified If NUMDIS = 0, go to Card Group 9.2.1
		9.1.4 For each of NUMDIS where displacement is specified NODE, IDOF, LHNO, DINT NODE — Node number Skeleton displacement IDOF = 1 — x-direction = 2 — y-direction Apparent relative fluid displacement = 3 — x-direction = 4 — y-direction LHNO — Displacement history number corresponding to sequence of displacement specifications given in Card Group 9.1.5.3 DINT — Displacement intensity factor

Card Group			Input Data and Definitions (Main File)
9 Loads	9.1 Specified Displacement		9.1.5.1 NUMDH, NUMDTP, TDSTART, TDFAC NUMDH — Number of different input displacement time histories NUMDTP — Number of displacement-time pairs TDSTART — Starting time TDFAC — Time scale factor for TD
		For Each Load History	9.1.5.2 TD_1, TD_2, ..., TD_{NUMDTP} TD_i — Specified times
			9.1.5.3 $SDIS_1$, $SDIS_2$, ..., $SDIS_{NUMDTP}$ $SDIS_i$ — Displacement magnitude at corresponding time TD_i

Card Group		Input Data and Definitions (Main File)
9 Loads	9.2 Concentrated Nodal Force	9.2.1 NUMCON NUMCON — Total number of degrees of freedom at which input concentrated force time histories are specified If NUMCON = 0, skip the rest of this Card Group
		9.2.2 For each of the NUMCON where load is applied NODE, IDOF, LHNO, CINT NODE — Node number Total force acting on a given node IDOF = 1 — x-direction = 2 — y-direction Fluid force acting on a given node = 3 — x-direction = 4 — y-direction LHNO — Load history number corresponding to sequence of load specifications given in Card Group 9.2.3.4 or 9.2.3.5 CINT — Load intensity factor

Card Group					Input Data and Definitions (Main File)
9 Loads	9.2 Concentrated Nodal Force				9.2.3.1 NTFNC, NUMCH NTFNC = 0 User-specified arbitrary force = 1 Force is specified by math functions NUMCH Number of different force time histories
		NTFNC = 0 (User-Specified Arbitrary Force)			9.2.3.2 NUMCTP, NCTYPE, DTXC, TCSTART, TCFAC NUMCTP Number of force-time pairs NCTYPE = 0 Constant time increment = 1 Specified times for all time histories = 2 Specified times for each time history DTXC Constant time interval for NCTYPE = 0 TCSTART Starting time TCFAC Time scale factor for TC
			For Each Load History		9.2.3.3 If NCTYPE = 0, go to next Card TC_1, TC_2, ..., TC_{NUMCTP} TC_i Specified times For NCTYPE = 1, specify only once for the first load history
					9.2.3.4 $SCON_1$, $SCON_2$, ..., $SCON_{NUMCTP}$ $SCON_i$ Force magnitude at time TC_i

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Main File)</th></tr>
<tr><td>9
Loads</td><td>9.2
Concentrated Nodal Force</td><td>NTFNC = 1 (Math Function)</td><td>9.2.3.5

For each of NUMCH loading time histories
NFNC, a_1, a_2, a_3, a_4, a_5

NFNC = 1 Polynomial decaying load
= 2 Exponential decaying load
= 3 Trigonometric load

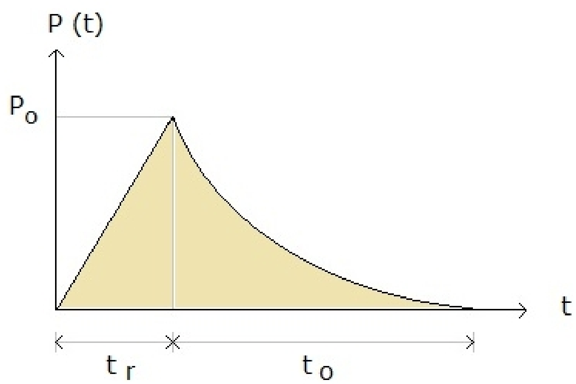
a_1,a_2,a_3,a_4 Force function coefficients defined in the next page

a_5 Starting time</td></tr>
</table>

Polynomial Decaying (NFNC = 1)

$a_1 = P_o \quad a_2 = t_r \quad a_3 = t_o \quad a_4 = n$

For $t_r \le t \le (t_r + t_o)$ $\qquad P(t) = P_o \left[1 - \frac{(t - t_r)}{t_o}\right]^n$



Exponential Decaying (NFNC = 2)

$$P(t) = a_1 + a_2\, e^{a_3 t}$$

Trigonometric (NFNC = 3)

$t \le a_4 \qquad P(t) = a_1 \sin(a_2\, t) + a_3 \cos(a_2\, t)$

$t > a_4 \qquad P(t) = 0$

Card Group		Input Data and Definitions (Main File)
9 Loads	9.3 Specified Velocity	9.3.1 NUMVEL NUMVEL — Total number of degrees of freedom at which velocity histories are specified If NUMVEL= 0, skip the rest of this Card Group
		9.3.2 For each of the NUMVEL where velocity is specified NODE, IDOF, LHNO, VINT NODE — Node number IDOF — Skeleton velocity = 1 x - direction = 2 y - direction Apparent relative fluid velocity = 3 x-direction = 4 y-direction LHNO — Velocity history number corresponding to sequence of velocity specifications given in Card Group 9.3.3.4 or 9.3.3.5 VINT — Velocity intensity factor
		9.3.3.1 NTFNV, NUMVH NTFNV = 0 User-specified arbitrary velocity = 1 Velocity specified by math function NUMVH — Number of different input velocity time histories

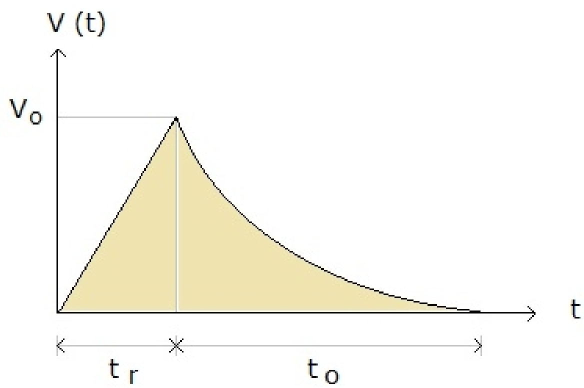
Card Group				Input Data and Definitions (Main File)
9 Loads	9.3 Specified Velocity	NTFNV = 0 (User-Specified Arbitrary Velocity)		9.3.3.2 NUMVTP, NVTYPE, DTXV, TVSTART, TVFAC NUMVTP — Number of velocity-time pairs NVTYPE = 0 — Constant time increment = 1 — Specified times for all time histories = 2 — Specified times for each time history DTXV — Constant interval for NVTYPE = 0 TVSTART — Starting time TVFAC — Time scale factor for TV
			For Each Load History	9.3.3.3 If NVTYPE = 0, go to next Card TV_1, TV_2, ..., TV_{NUMVTP} TV_i — Specified times For NVTYPE = 1, specify only once for the first load history
				9.3.3.4 $SVEL_1$, $SVEL_2$, ..., $SVEL_{NUMVTP}$ $SVEL_i$ — Velocity magnitude at time TV_i

Card Group			Input Data and Definitions (Main File)
9 Loads	9.3 Specified Velocity	NTFNV = 1 (Math Function)	9.3.3.5 For each of NUMVH velocity time histories NFNV, a_1, a_2, a_3, a_4, a_5 NFNV = 1 Polynomial decaying velocity = 2 Exponential decaying velocity = 3 Trigonometric velocity a_1,a_2,a_3,a_4 Velocity function coefficients defined the next page a_5 Starting time

Polynomial Decaying (NFNV = 1)

$a_1 = V_o \quad a_2 = t_r \quad a_3 = t_o \quad a_4 = n$

For $t_r \le t \le (t_r + t_o)$ $\qquad V(t) = V_o \left[1 - \frac{(t - t_r)}{t_o} \right]^n$



Exponential Decaying (NFNV = 2)

$$V(t) = a_1 + a_2 \, e^{a_3 t}$$

Trigonometric (NFNV = 3)

$t \le a_4 \qquad V(t) = a_1 \sin(a_2 t) + a_3 \cos(a_2 t)$

$t > a_4 \qquad V(t) = 0$

Card Group		Input Data and Definitions (Main File)
9 Loads	9.4 Initial Velocity	9.4.1 NINVEL NINVEL — Number of degrees of freedom where initial velocity is applied. If NINVEL= 0, skip the rest of this Card Group
		9.4.2 For each of the NINVEL where velocity is applied NODE, IDOF, VEL NODE — Node number Skeleton velocity IDOF = 1 — x-direction = 2 — y-direction Apparent relative fluid velocity = 3 — x-direction = 4 — y-direction VEL — Initial velocity

Card Group		Input Data and Definitions (Main File)
9 Loads	9.5 Specified Acceleration	9.5.1 NUMACC, MOTION, EYB, EDEN, ECP, ECS NUMACC Total number of directions at which input acceleration time histories are specified MOTION = 0 No EHS (Elastic Half Space) = 1 EHS with base acceleration applied = 2 EHS with base shear force applied = 3 EHS with conventional base accel. applied EYB, EDEN, ECP, ECS : Half space top y coordinate, Unit weight, Compression and Shear wave speeds used for elastic half space if MOTION is not zero If NUMACC = 0, Skip the rest of this Card Group
		9.5.2 For each of NUMACC where acceleration is specified NODE, IDOF, LHNO, AINT NODE Node number IDOF = 1 x-direction skeleton acceleration = 2 y-direction skeleton acceleration LHNO Acceleration history number corresponding to sequence of acceleration specifications given in Card Group 9.5.3.4 or 9.5.3.5 AINT Acceleration intensity factor Note: For uniformly distributed acceleration, set all node numbers to zero. Output motions are relative to base motion for MOTION = 0 or 1
		9.5.3.1 NTFNA, NUMAH NTFNA = 0 User-specified arbitrary acceleration = 1 Acceleration specified by math function NUMAH Number of different input time histories

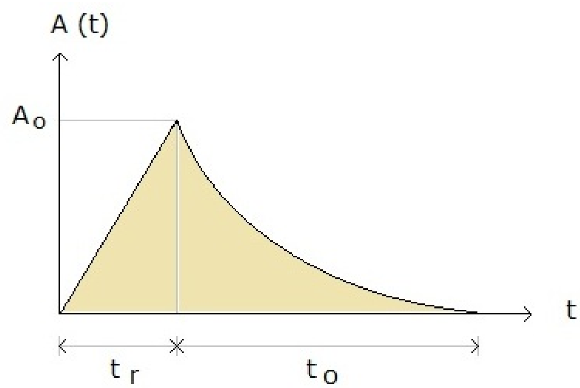
Card Group	Input Data and Definitions (Main File)
9 Loads — Specified Acceleration — NTFNA = 0 (User-Specified Arbitrary Acceleration)	9.5.3.2 NUMATP, NATYPE, DTXA, TASTART, TAFAC, IACCM NUMATP Number of acceleration-time pairs NATYPE = 0 Constant time increment = 1 Specified times for all time histories = 2 Specified times for each time history DTXA Constant time interval for NATYPE = 0 TASTART Starting time TAFAC Time scale factor for TA IACCM = 0 Input histories from Main File = 1 Input histories from External Files First 3 lines represent headers
For Each Load History	9.5.3.3 If NATYPE = 0, go to next Card TA_1, TA_2, ..., TA_{NUMATP} TA_i Specified times For NATYPE = 1, specify only once for the first load history For IACCM = 1, specified times read from Acc_Time_1.dat, Acc_Time_2.dat ...
For Each Load History	9.5.3.4 $SACC_1$, $SACC_2$, ..., $SACC_{NUMATP}$ $SACC_i$ Acceleration magnitude at time TA_i For IACCM = 1, specified histories read from Acc_History_1.dat, Acc_History_2.dat ...

Card Group	Input Data and Definitions (Main File)		
9 Loads	Specified Acceleration	NTFNA = 1 (Math Function)	9.5.3.5 For each of NUMAH acceleration time histories NFNA, a_1, a_2, a_3, a_4, a_5 NFNA = 1 Polynomial decaying acceleration = 2 Exponential decaying acceleration = 3 Trigonometric acceleration a_1,a_2,a_3,a_4 Acceleration function coefficients defined in the next page a_5 Starting time

Polynomial Decaying (NFNA = 1)

$a_1 = A_o \quad a_2 = t_r \quad a_3 = t_o \quad a_4 = n$

For $t_r \le t \le (t_r + t_o)$ $\qquad A(t) = A_o \left[1 - \frac{(t - t_r)}{t_o}\right]^n$



Exponential Decaying (NFNA = 2)

$$A(t) = a_1 + a_2 e^{a_3 t}$$

Trigonometric (NFNA = 3)

$t \le a_4 \qquad A(t) = a_1 \sin(a_2 t) + a_3 \cos(a_2 t)$

$t > a_4 \qquad A(t) = 0$

Card Group		Input Data and Definitions (Main File)
9 Loads	9.6 Transmitting Boundary	9.6.1 NODVIS, NELVIS, NOSVIS NODVIS — Number of transmitting degrees of freedom NELVIS — Number of continuum element surfaces NOSVIS — Number of outer boundary surfaces (Max = 4)
		9.6.2 If NODVIS = 0, go to Card Group 9.6.3 NODE, IDOF, VISC — For each of NODVIS NODE — Node number IDOF = 1 — Damping in x-direction = 2 — Damping in y-direction VISC — Constant which is proportional to the force on a given node (ρCA_c), equal to impedence times contributing area on the node $C = C_p$ for IDOF normal to the boundary $C = C_s$ for IDOF parallel to the boundary $C_{p,}\ C_s$: Compression & shear wave speed
		9.6.3 If NELVIS = 0, go to next Card Group NEL, KT — For each of the NELVIS NEL — Element number KT — Element surface designation number. Same as KP in Card Group 5.7.2.1
		9.6.4 If NOSVIS = 0, go to next Card Group NOS, VC — For each of the NOSVIS NOS — Outer surface number VC — NOS = 1: $VC = Y_{TOP}$ NOS = 2: $VC = X_{LEFT}$ NOS = 3: $VC = Y_{BOTOM}$ NOS = 4: $VC = X_{RIGHT}$

Card Group	Input Data and Definitions (Main File)
10 Requested Output	10.1 NTPRNT NTPRNT — Number of cycles between output data print
	10.2.1 NHPEL NHPEL — Number of elements at which stress/strain time histories are requested
	10.2.2 If NHPEL = 0, skip the following Card NEL_1, NEL_2, ..., NEL — Element numbers to be printed
	10.3.1 NHPMT NHPMT — Number of nodes at which motion time histories are requested
	10.3.2 If NHPMT = 0, skip the following Card $NODE_1$, $NODE_2$, ..., $NODE_{NHPMT}$ NODE — Node numbers to be printed
	10.4.1 NTIME NTIME — Number of times at which stress/strain/motion profiles are requested
	10.4.2 If NTIME = 0, skip the following Card $TIME_1$, $TIME_2$, ..., $TIME_{NTIME}$ TIME — Time to be printed

4.5 Post File

Post File contains information which are used to show graphically the results from the main-processing program.

Post File consists of three different card groups:

- Card Group 11 (PLOT-2D)
- Card Group 12 (PLOT-XY)
- Card Group 13 (FEMAP)

Card Group 11 contains the input data which are used to plot the following snapshots in two dimension:

- Finite element mesh/element/node number
- Principal stress distribution
- Deformed shape
- Beam section force/extreme fiber stress/strain
- Truss axial force/stress/strain
- Contours of continuum element data

Card Group 12 contains the input data for the following plots:

Time history

- Stress/strain/time
- Displacement/velocity/acceleration/time

Snapshot

- Stress/strain vs. distance
- Displacement/velocity/acceleration vs. distance

Card Groups 13 is no longer supported.
These plots can be performed automatically by using PLOT-3D.

PLOT-2D

Post-Processor

Card Group	Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.1 NPTYPE NPTYPE = 0 End of plotting output = 1 Finite element mesh / element number = 2 Principal stress distribution = 3 Deformed shape = 4 Beam section force / fiber stress / strain = 5 Truss axial force / stress / strain = 6 Contours of continuum element data = 7 Stress state in p-q space and octahedral plane. When NPTYPE = 7 is specified, all other cases of NPTYPE are not considered. If NPTYPE = 0, Skip rest of Card Group 11

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.2 For NPTYPE = 1 (Finite Element Mesh / Element Number)	11.2.1 TITLE TITLE Any title (Max = 70 characters)
		11.2.2 IUNIT IUNIT = 1 Inch = 2 Cm = 3 User-specified unit
		11.2.3 For IUNIT = 3 NCHR LABEL NCHR Number of characters for mesh unit LABEL Name of mesh unit

Card Group	Input Data and Definitions (Post File)

11

PLOT-2D Plot Information

For NPTYPE = 1 (Finite Element Mesh / Element Number)

11.2.4

IMODE

IMODE	=	1	Plot finite element mesh
	=	-1	Plot element and node numbers
	=	2	Plot element numbers
	=	-2	Plot node numbers
	=	3	Plot skeleton boundary codes
	=	-3	Plot fluid boundary codes
	=	4	Plot rotational boundary codes

11.2.5

NGROUP

NGROUP	=	0	Plot all elements
	>	0	Plot specified groups (Max=1000)

11.2.6

<u>If NGROUP = 0, Skip this Card</u>

NGROUP Cards	NSS,	NEE,	NIC,	NNN
	-	-	-	-
	-	-	-	-

NSS	Starting element number in a row
NEE	Number of elements in a row
NIC	Element number increment for next row
NNN	Total number of rows

10	11	12	13
20	21	22	23
30	31	32	33

Example

NSS = 10
NEE = 4
NIC = 10
NNN = 3

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.3 For NPTYPE = 2 (Principal Stress Distribution)	11.3.1 TITLE TITLE Any title (Max = 70 characters)
		11.3.2 IUNIT IUNIT = 1 In, Psi = 2 Cm, Kg/cm^2 = 3 User-specified unit
		11.3.3 <u>For IUNIT = 3</u> NCHR LABEL NCHRC LABELC NCHR Number of characters for mesh unit LABEL Name of mesh unit NCHRC Number of characters for stress unit LABELC Name of stress unit

Card Group	Input Data and Definitions (Post File)	
11 PLOT-2D Plot Information	For NPTYPE = 2 (Principal Stress Distribution)	11.3.4 NLTIME, $TIME_{REF}$ $TIME_1$, $TIME_2$, ..., $TIME_{NLTIME}$ NLTIME — Number of specified times (Max=1000) $TIME_{REF}$ — Reference time TIME — Specified time If $TIME_{REF}$ is not equal to 0.0, Stress at $TIME_i$ are relative to $TIME_{REF}$
		11.3.5 NGROUP, IAVG, ISCRIN, IMESH, IPSTRS NGROUP = 0 Plot stresses at all elements > 0 Plot stresses at specified groups (Max=1000) IAVG = 0 Do not plot averages = 1 Plot average stresses ISCRIN = 0 Do not screen the data = 1 Screen the data IMESH = 0 Do not plot meshes = 1 Plot meshes IPSTRS = 0 Do not store principal stresses = 1 Store principal stresses on file PSTRS.DAT

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	For NPTYPE = 2 (Principal Stress Distribution)	11.3.6 <u>If NGROUP = 0, Skip this Card</u> NGROUP Cards ┌ NSS, NEE, NIC, NNN │ - - - - └ - - - - Refer to Card Group 11.2.6
		11.3.7 NRL NRL Number of nodes to be connected by a solid line (Max=5000)
		11.3.8 <u>If NRL = 0, Skip this Card</u> $NODE_1$, $NODE_2$, …, $NODE_{NRL}$ NODE Reference node numbers. If $NODE_i$ has negative sign, a New Line is drawn

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.4 For NPTYPE = 3 (Deformed Shape)	11.4.1 TITLE TITLE Any title of up to 70 characters
		11.4.2 IUNIT IUNIT = 1 In = 2 Cm = 3 User-specified unit
		11.4.3 For IUNIT = 3 NCHR LABEL NCHR Number of characters for mesh and displacement unit. LABEL Name of mesh and displacement unit
		11.4.4 NLTIME, $TIME_{REF}$ $TIME_1$, $TIME_2$, ..., $TIME_{NLTIME}$ NLTIME Number of specified times (Max=1000) $TIME_{REF}$ Reference time TIME Specified time If $TIME_{REF}$ is not equal to 0.0, Displacement at $TIME_i$ are relative to $TIME_{REF}$

Card Group			Input Data and Definitions (Post File)
11	11.4	11.4.5	
PLOT-2D Plot Information	For NPTYPE = 3 (Deformed Shape)		

Row and Line Plots (Repeat in any order)

For Row Plot --> 1, IDISP
NSR, JCR, NJR, ICR, NIR

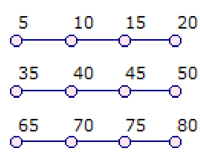
For Line Plot --> 2, IDISP
NPT
$NODE_1$, $NODE_2$, .., $NODE_{NPT}$

For End Plot --> 0, 0

IDISP = 0 Undeformed shape
= 1 Deformed shape
= 2 Displacement vector

For Row Plot (Max = 1000)

NSR	Starting node number of row plot
JCR	Node number increment in a row
NJR	Number of nodes in a row
ICR	Node number increment for next row
NIR	Total number of rows



Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

For Line Plot (Max = 160)

NPT	Number of nodes (Max=1000)
NODE	Node number

Card Group			Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.5 For NPTYPE = 4 (Beam Section Force / Extreme Fiber Stress / Strain)		11.5.1 TITLE TITLE Any title (Max = 70 characters)
			11.5.2 IUNIT IUNIT = 1 In, Psi = 2 Cm, Kg/cm^2 = 3 User-specified unit
			11.5.3 For IUNIT = 3 NCHR LABEL NCHRB LABELB NCHR Number of characters for mesh unit LABEL Name of mesh unit NCHRB Number of characters for section force / extreme fiber stress LABELB Name of section force / fiber stress

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.5 For NPTYPE = 4 (Beam Section Force / Extreme Fiber Stress / Strain)	11.5.4 NLTIME, $TIME_{REF}$ $TIME_1$, $TIME_2$, ..., $TIME_{NLTIME}$ NLTIME Number of specified times (Max=1000) $TIME_{REF}$ Reference time TIME Specified time If $TIME_{REF}$ is not equal to 0.0, Section force / Stress / Strain plots at $TIME_i$ are relative to $TIME_{REF}$
		11.5.5 NBTS NBTS = 1 Thrust = 2 Shear in member y direction = 3 Shear in member z direction = 4 Torque = 5 Bending moment about y axis = 6 Bending moment about z axis See Figure PL-4 for Sign Convention
		11.5.6 NBGROUP NBGROUP Number of beam groups (Max=280)

Card Group			Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.5 For NPTYPE = 4 (Beam Section Force / Extreme Fiber Stress / Strain)	For Each Beam Group	11.5.7.1 NBLIST NBLIST = 0 Elements from NFBEAM to NLBEAM = 1 Listing of individual elements
			11.5.7.2 <u>For NBLIST = 0</u> NFBEAM, NLBEAM NFBEAM Starting beam element number NLBEAM Ending beam element number
			11.5.7.3 <u>For NBLIST = 1</u> MBEAM N_1, N_2, …, N_{MBEAM} MBEAM Number of beam element (Max=280) N_1, N_2, N_{MBEAM} List of element number
		11.5.8 NRL NRL Number of nodes to be connected by a Solid Line (Max=280)	
		11.5.9 <u>If NRL = 0, Skip this Card</u> $NODE_1$, $NODE_2$, …, $NODE_{NRL}$ NODE Reference node numbers If $NODE_i$ has negative sign, a New Line is drawn	

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Post File)</th></tr>
<tr><td rowspan="3">11

PLOT-2D Plot Information</td><td rowspan="3">11.6

For NPTYPE = 5 (Truss Axial Force / Stress / Strain)</td><td>11.6.1

TITLE

TITLE Any title (Max = 70 characters)</td></tr>
<tr><td>11.6.2

IUNIT

IUNIT = 1 In, Pound
= 2 Cm, Kg
= 3 User-specified unit</td></tr>
<tr><td>11.6.3

For IUNIT = 3

NCHR
LABEL
NCHRT
LABELT

NCHR Number of characters for mesh unit
LABEL Name of mesh unit
NCHRT Number of characters for axial data
LABELT Name of axial force / stress / strain</td></tr>
</table>

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.6 For NPTYPE = 5 (Truss Axial Force / Stress / Strain)	11.6.4 NLTIME, $TIME_{REF}$ $TIME_1$, $TIME_2$, ..., $TIME_{NLTIME}$ NLTIME — Number of specified times (Max=1000) $TIME_{REF}$ — Reference time TIME — Specified times If $TIME_{REF}$ is not equal to 0.0, Force / Stress / Strain at $TIME_i$ are relative to $TIME_{REF}$
		11.6.5 NTTS NTTS = 1 — Axial force = 2 — Axial stress = 3 — Axial strain
		11.6.6 NTGROUP NTGROUP — Number of truss groups (Max=100)

Card Group	Input Data and Definitions (Post File)		
11 PLOT-2D Plot Information	For NPTYPE = 5 (Truss Axial Force / Stress / Strain)	For Each Truss	11.6.7.1 NTLIST NTLIST = 0 Elements from NFTRUS to NLTRUS = 1 Listing of individual elements
			11.6.7.2 For NTLIST = 0 NFTRUS, NLTRUS NFTRUS Starting truss element number NLTRUS Ending truss element number
			11.6.7.3 For NTLIST = 1 MTRUS N_1, N_2, ..., N_{MTRUS} MTRUS Number of element (Max=280) N_1, N_2, N_{MTRUS} List of element number
		11.6.8 NRL NRL Number of nodes to be connected by a Solid Line (Max=280)	
		11.6.9 If NRL = 0, Skip this Card $NODE_1$, $NODE_2$, ..., $NODE_{NRL}$ NODE Reference node numbers If $NODE_i$ has negative sign, a New Line is drawn	

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	For NPTYPE = 6 (Contours of Continuum Element Data)	11.7.1 TITLE TITLE Any title (Max = 70 characters)
		11.7.2 IUNIT IUNIT = 1 In, Pound = 2 Cm, Kg = 3 User-specified unit
		11.7.3 For IUNIT = 3 NCHR LABEL NCHRC LABELC NCHR Number of characters for mesh unit LABEL Name of mesh unit NCHRC Number of characters for contouring data LABELC Name of contouring data
		11.7.4 NLTIME, $TIME_{REF}$ $TIME_1$, $TIME_2$, ..., $TIME_{NLTIME}$ NLTIME Number of specified times (Max=1000) $TIME_{REF}$ Reference time TIME Specified time If $TIME_{REF}$ is not equal to 0.0, Contour plots at $TIME_i$ are relative to $TIME_{REF}$

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	For NPTYPE = 6 (Contours of Continuum Element Data)	11.7.5 NCTS NCTS Variable to be plotted. Select from Table PL-1
		11.7.6 DELTA, IRES, IRGP, IENL, R_x, R_y DELTA = -DELTA Line contour, absolute value of DELTA is desired contour interval = 0 Color-filled contour = 2 Smoothed color-filled contour IRES = 0 Draft copy = 1 Fine copy IRGP = 0 Values at ref. grid points are not added = 1 Values at ref. grid points are added IENL = 0 Standard view = 2 Laplacian & spline interpolation scheme = 3 Davis distance to a power interpolation For IENL= 2 R_x Weight factor applied to spline function If R_x =0.0, only Laplacian interpolation is used R_y is not used For IENL= 3 R_y Power applied to 1/(distance **power) interpolation scheme. Recommended starting value is 4.0. R_x is not used Reference [Davis, J.c., 1986, Statistics and Data Analysis in Geology, page 356]

Card Group			Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.7 For NPTYPE = 6 (Contours of Continuum Element Data)		11.7.7 NGROUP NGROUP = 0 Plot at all elements > 0 Plot at specified groups (Max=1000)
			11.7.8 If NGROUP = 0, Skip this Card NGROUP Cards: NSS, NEE, NIC, NNN - - - - - - - - Refer to Card Group 11.2.6
			11.7.9 NRL NRL Number of nodes to be connected by a Solid Line (Max=5000)
			11.7.10 If NRL = 0, Skip this Card $NODE_1$, $NODE_2$, ..., $NODE_{NRL}$ NODE Reference node numbers If $NODE_i$ has negative sign, a New Line is drawn

Card Group		Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.8 For NPTYPE = 7 (Stress State in p-q Space and Octahedral Plane)	11.8.1 TITLE TITLE Any title of up to 70 characters
		11.8.2 LABELC LABELC Label for stress unit
		11.8.3 NLTIME $TIME_1$, $TIME_2$, ..., $TIME_{NLTIME}$ NLTIME Number of specified times (Max=10) TIME Specified time
		11.8.4 NUMNEL NEL_1, NEL_2, ..., NEL_{NUMNEL} NUMNEL Number of specified elements (Max=10) NEL Element number

Table PL-1 Continuum Contour Plot

NCTS	Legend	Description	
		<u>Continuum Element (See Fig. PL-1)</u>	
2	STRESS-XX	Normal XX stress	(σ_x')
3	STRESS-YY	Normal YY stress	(σ_y')
4	STRESS-ZZ	Normal ZZ stress	(σ_z')
5	STRESS-XY	Shear XY stress	(τ_{xy})
6	STRESS-YZ	Shear YZ stress	(τ_{yz})
7	STRESS-XZ	Shear XZ stress	(τ_{xz})
8	PRESSURE	Mean pressure	(P')
9	FLUID-PRES	Fluid pressure	(π)
10	TSTRESS-XX	Normal XX total stress	($\sigma_x = \sigma_x' + \pi$)
11	TSTRESS-YY	Normal YY total stress	($\sigma_y = \sigma_y' + \pi$)
12	TSTRESS-ZZ	Normal ZZ total stress	($\sigma_z = \sigma_z' + \pi$)
13	TPRESSURE	Total mean pressure	($P = P' + \pi$)
14	D.STRES	Deviatoric stress	($Q = (3/\sqrt{2})\ \tau_{oct}$)
15	STRAIN-XX	Normal XX strain	(ε_x)
16	STRAIN-YY	Normal YY strain	(ε_y)
17	STRAIN-ZZ	Normal ZZ strain	(ε_z)
18	STRAIN-XY	Shear XY strain	(γ_{xy})
19	STRAIN-YZ	Shear YZ strain	(γ_{yz})
20	STRAIN-XZ	Shear XZ strain	(γ_{xz})
21	VOL-STRAIN	Volumetric strain	(ε_v)
22	GAMMA-OCT	Octahedral shear strain	(γ_{oct})
23	TAU-OCT	Octahedral shear stress	(τ_{oct})
24	FS	Safety factor	(Fig. PL-2)
25	YIELD-FLAG	Yield flag	(Fig. PL-3)
26	STRESS - 1	Major principal stress	(σ_1')
27	STRESS - 2	Inter. principal stress	(σ_2')
28	STRESS - 3	Minor principal stress	(σ_3')

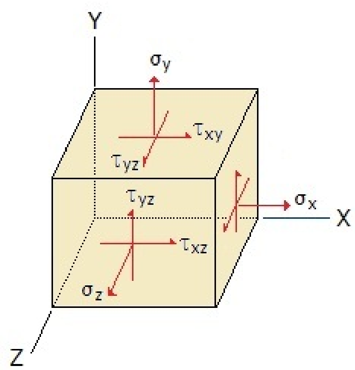


Figure PL-1 Sign Conventions for Continuum Stress

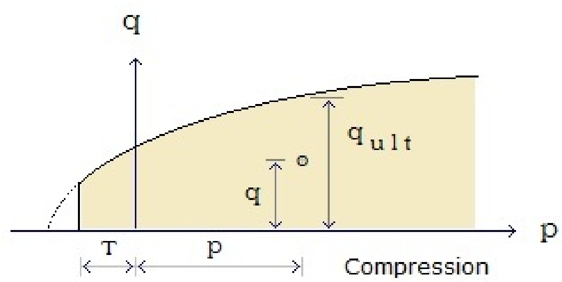


Figure PL-2 Definition of Safety Factor

Factor of Safety (FS) is defined as:

For elastic material FS = 10

For elasto-plastic material FS = q_{ult} / q

FS is limited to $1 \leq$ F.S. ≤ 10

For $p \leq -T$ FS = 1

$P = (\sigma_x + \sigma_y + \sigma_z) / 3$

$q = (3 / \sqrt{2})\ \tau_{oct}$

Yield Flag for Beam and Continuum Elements

Yield Flag	Stress Status
0	Stress point is in elastic
1	Stress point is in plastic
2	Stress Point develops crack

Stress Status for Truss Element

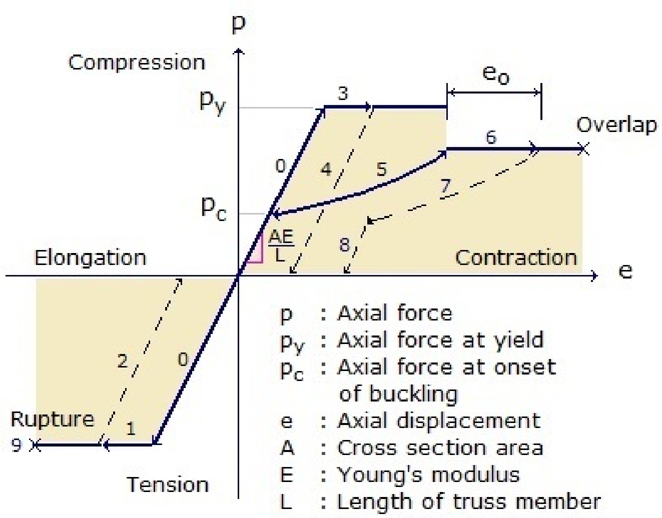


Figure PL-3 Description of Stress Status

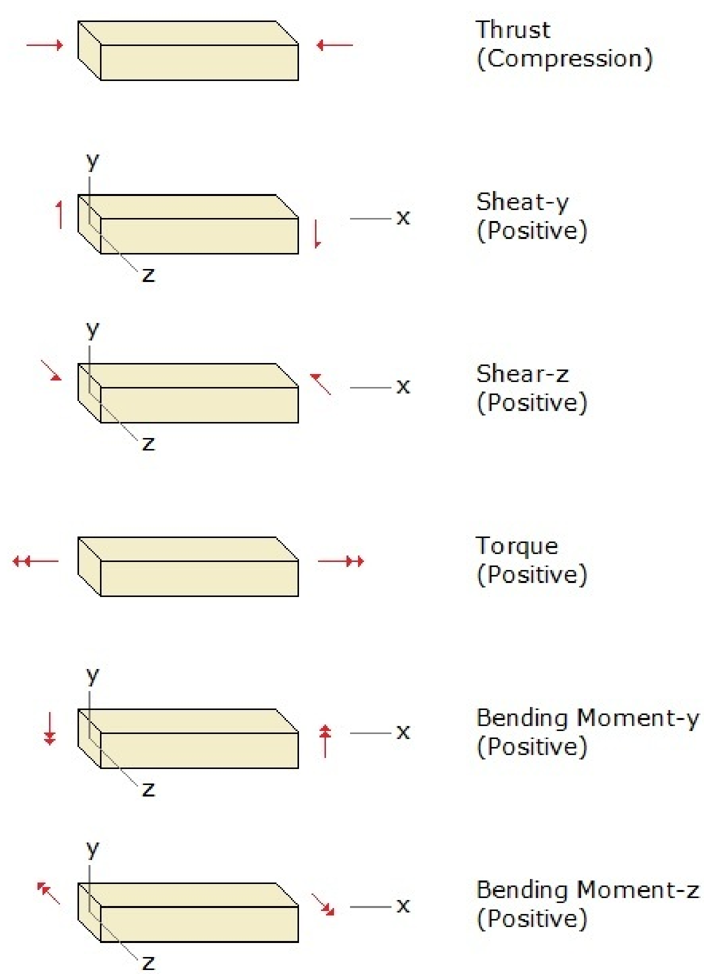


Figure PL-4 Sign Conventions for Beam

PLOT-XY

Post-Processor

Card Group	Input Data and Definitions (Post File)
12 PLOT-XY Information	12.1 IPTYPE IPTYPE 0 End of plotting output **Standard Time history** 1 Stress/Strain/Time 2 Displacement/Velocity/Accel./Time **Standard Snapshot** 3 Stress/Strain vs. Distance 4 Displacement/Velocity/Accel. vs. Distance **Simplified Time history** 5 Stresses/Strains for a Given Element 6 Stress/Strain Pair for Different Elements 7 Displacements/Velocities/Accel. for a Given Node 8 Displacement/Velocity/Accel. Pair for Different Nodes **Simplified Snapshot** 9 Stresses/Strains for a Given Time 10 Stress/Strain for Different Times 11 Displacements/Velocities/Accel. for a Given Time 12 Displacement/Velocity/Accel. for Different Times Note: Simplified plots (IPTYPE 5 to 12) should be specified after standard plots. You can edit simplified plots using PlotXY Generator in SMAP Run Menu.

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.2 For IPTYPE = 1 (Stress / Strain / Time History)	12.2.1 IPLOT IPLOT = 0 For each specified element, Number of different pair of variables = 1 For each specified pair of variables, Number of different element data
		12.2.2 NOEL NOEL Number of elements (Max 10)
		12.2.3 LIST (I) I = 1, NOEL LIST (I) List element numbers
		12.2.4 NDPQ NDPQ Number of different pair of variables

Card Group		Input Data and Definitions
12	12.2	12.2.5 NDPQ Cards ┌ K_{x1}, K_{y1} │ K_{x2}, K_{y2} │ - - └ - - K_x,K_y Select from Table PL-1
		12.2.6 TMFAC, STFAC, SNFAC Multiplication factor TMFAC Time STFAC Stress SNFAC Strain
PLOT-XY Information	For IPTYPE = 1 (Stress / Strain / Time History)	12.2.7 IPLOT = 0: For each element IPLOT = 1: For each pair of variables TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.3 For IPTYPE = 2 (Displacement / Velocity / Acceleration / Time History)	12.3.1 IPLOT IPLOT = 0 — For each specified node, Number of different pair of variables = 1 — For each specified pair of variables, Number of different node data
		12.3.2 NODE NODE — Number of nodes (Max 10)
		12.3.3 LIST (I), I = 1, NODE LIST (I) — List node numbers

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.3 For IPTYPE = 2 (Displacement / Velocity / Acceleration / Time History)	12.3.4 NDPQ NDPQ — Number of different pair of variables
		12.3.5 NDPQ Cards: K_{x1}, K_{y1} K_{x2}, K_{y2} - - - - K_x, K_y — Select from Table PL-2
		12.3.6 TMFAC, SND, SNV, SNA, NC, ANGLE Multiplication factor TMFAC — Time SND — Displacement SNV — Velocity SNA — Acceleration NC = 0 — No transfer = 1 — Transfer from X-Y to polar coordinate = 2 — Transfer from polar to X-Y coordinate ANGLE — Rotation angle (Degree)
		12.3.7 IPLOT = 0: For each node IPLOT = 1: For each pair of variables TITLE — (50 characters) X-LABEL — (50 characters) Y-LABEL — (50 characters)

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Post File)</th></tr>
<tr><td rowspan="5">12

PLOT-XY Information</td><td rowspan="5">12.4

For IPTYPE = 3 (Stress / Strain vs. Distance Snapshot)</td><td>12.4.1

IPLOT

IPLOT = 0 For each specified time, Number of different variables

= 1 For each specified variable, Number of different time data</td></tr>
<tr><td>12.4.2

NOTM

NOTM Number of times (Max 10)</td></tr>
<tr><td>12.4.3

TLIST (I), I = 1, NOTM

TLIST (I) List times in sequential order</td></tr>
<tr><td>12.4.4

NDPQ

NDPQ Number of different variables</td></tr>
<tr><td>12.4.5

NDPQ Cards: K_{y1}, K_{y2}, -

K_y Select from Table PL-1</td></tr>
</table>

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.4 For IPTYPE = 3 (Stress / Strain vs. Distance Snapshot)	12.4.6 ISCALD, ILTNUM, XSTART ISCALD = 0 Unscaled distance = 1 Scaled distance ILTNUM = 0 Do not list element numbers = 1 List Element No vs Value in PlotXy.Lin XSTART Reference starting X-coordinate Note: If ISCALD = 1 and ILTNUM = 1, X-LABEL is used for distance unit
		12.4.7 Element Number Specification (Max 800 Elements) For arbitrary order > 1 NRL N_1, N_2, N_{NRL} For sequential order > 2 NSTAR, NINCR, NPONT For end of generation > 0 NRL Number of elements $N_1,N_2,..,N_{NRL}$ Element numbers NSTAR Starting element numbers NINCR Element number increment NPONT Number of element

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.4 For IPTYPE = 3 (Stress / Strain vs. Distance Snapshot)	12.4.8 STFAC, SNFAC, SDFAC Multiplication factor STFAC Stress SNFAC Strain SDFAC Distance
		12.4.9 IPLOT = 0: For each specified time IPLOT = 1: For each variable TITLE (50 characters) X-LABEL (50 characters) Y-LABEL (50 characters)

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.5 For IPTYPE = 4 (Displacement / Velocity / Acceleration vs. Distance Snapshot)	12.5.1 IPLOT IPLOT = 0 For each specified time, Number of different variables = 1 For each specified variable, Number of different time data
		12.5.2 NOTM NOTM Number of times (Max 10)
		12.5.3 TLIST (I), I = 1, NOTM TLIST (I) List times in sequential order
		12.5.4 NDPQ NDPQ Number of different variables
		12.5.5 NDPQ Cards: K_{y1}, K_{y2}, -, - K_y Select from Table PL-2

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.5 For IPTYPE = 4 (Displacement / Velocity / Acceleration vs. Distance Snapshot)	12.5.6 ISCALD, ILTNUM, XSTART ISCALD = 0 Unscaled distance = 1 Scaled distance ILTNUM = 0 Do not list node numbers = 1 List Node No vs Value in PlotXy.Lin XSTART Reference starting X-coordinate Note: If ISCALD = 1 and ILTNUM = 1, X-LABEL is used for distance unit

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.5 For IPTYPE = 4 (Displacement / Velocity / Acceleration vs. Distance Snapshot)	12.5.7 Node Number Specification (Max 800 nodes) For Arbitrary Order > 1 NRL N_1, N_2, .., N_{NRL} For Sequential Order > 2 NSTAR, NINCR, NPONT For End Generation > 0 NRL: Number of nodes N_1,N_2,..,N_{NRL}: Node numbers NSTAR: Starting node numbers NINCR: Node number increment NPONT: Number of nodes
		12.5.8 SND, SNV, SNA, NC, ANGLE, SDFAC Multiplication factor SND: Displacement SNV: Velocity SNA: Acceleration NC = 0: No transfer = 1: Transfer from X-Y to polar coordinate = 2: Transfer from polar to X-Y coordinate ANGLE: Rotation angle (Degree) SDFAC: Multiplication factor for distance
		12.5.9 IPLOT = 0: For each specified time IPLOT = 1: For each variable TITLE (50 characters) X-LABEL (50 characters) Y-LABEL (50 characters)

Card Group		Input Data and Definitions
12 PLOT-XY Information	For IPTYPE = 5 (Time History of Stresses/Strains for a Given Element)	12.6.1 NEL NEL Element number
		12.6.2 NDQ NDQ Number of different quantities
		12.6.3 NDQ Cards: K_{y1}, K_{y2}, -, - K_y Select from Table PL-1
		12.6.4 TMFAC, STFAC, SNFAC Multiplication factor TMFAC Time STFAC Stress SNFAC Strain
		12.6.5 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Card Group		Input Data and Definitions
12 PLOT-XY Information	For IPTYPE = 6 (Time History of Stress/Strain Pair for Different Elements)	12.7.1 NOEL NOEL Number of elements (Max 10)
		12.7.2 LIST (I) I = 1, NOEL LIST (I) List element numbers
		12.7.3 K_x, K_y K_x, K_y Select from Table PL-1
		12.7.4 TMFAC, STFAC, SNFAC Multiplication factor TMFAC Time STFAC Stress SNFAC Strain
		12.7.5 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Card Group		Input Data and Definitions
12 PLOT-XY Information	For IPTYPE = 7 (Time History of Displacements/Vel./Accel. for a Given Node)	12.8.1 NOD NOD Node number
		12.8.2 NDQ NDQ Number of different quantities
		12.8.3 NDQ Cards: K_{y1}, K_{y2}, -, - K_y Select from Table PL-2
		12.8.4 TMFAC, SND, SNV, SNA Multiplication factor TMFAC Time SND Displacement SNV Velocity SNA Acceleration
		12.8.5 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Card Group		Input Data and Definitions
12	For IPTYPE = 8 (Time History of Displ./Vel./Accel. Pair for Different Nodes)	12.9.1 NODE NODE — Number of nodes (Max 10)
PLOT-XY Information		12.9.2 LIST (I) I = 1, NODE LIST (I) — List node numbers
		12.9.3 K_x, K_y K_x, K_y — Select from Table PL-2
		12.9.4 TMFAC, SND, SNV, SNA Multiplication factor TMFAC — Time SND — Displacement SNV — Velocity SNA — Acceleration
		12.9.5 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Card Group		Input Data and Definitions
12 PLOT-XY Information	For IPTYPE = 9 (Snap Shot of Stresses/Strains for a Given Time)	12.10.1 TIME TIME Specified time
		12.10.2 NDQ NDQ Number of different quantities
		12.10.3 NDQ Cards: K_{y1}, K_{y2}, - K_y Select from Table PL-1
		12.10.4 XSTART XSTART Reference starting X-coordinate
		12.10.5 Element Number Specification (Max 800 Elements) NRL N_1, N_2, N_{NRL} NRL Number of elements N_1, N_2, .., N_{NRL} Element numbers N_i, $-N_{i+1}$, N_{i+2} From N_i to N_{i+1} with increment N_{i+2}
		12.10.6 STFAC, SNFAC, SDFAC Multiplication factor STFAC Stress SNFAC Strain SDFAC Distance
		12.10.7 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Card Group		Input Data and Definitions
12 PLOT-XY Information	For IPTYPE = 10 (Snap Shot of a Stress/Strain for Different Times)	12.11.1 NOTM NOTM — Number of times (Max 10)
		12.11.2 TLIST (I), I = 1, NOTM TLIST (I) — List times in sequential order
		12.11.3 K_y K_y — Select from Table PL-1
		12.11.4 XSTART XSTART — Reference starting X-coordinate
		12.11.5 <u>Element Number Specification (Max 800 Elements)</u> NRL N_1, N_2, N_{NRL} NRL — Number of elements N_1, N_2, .., N_{NRL} — Element numbers N_i, $-N_{i+1}$, N_{i+2} — From N_i to N_{i+1} with increment N_{i+2}
		12.11.6 STFAC, SNFAC, SDFAC Multiplication factor STFAC — Stress SNFAC — Strain SDFAC — Distance
		12.11.7 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Card Group		Input Data and Definitions
12 PLOT-XY Information	For IPTYPE = 11 (Snap Shot of Displacements/Vel./Accel for a Given Time)	12.12.1 TIME TIME Specified time
		12.12.2 NDQ NDQ Number of different quantities
		12.12.3 NDQ Cards: K_{y1}, K_{y2}, - K_y Select from Table PL-2
		12.12.4 XSTART XSTART Reference starting X-coordinate
		12.12.5 Node Number Specification (Max 800 Nodes) NRL N_1, N_2, N_{NRL} NRL Number of nodes N_1, N_2, .., N_{NRL} Node numbers N_i, $-N_{i+1}$, N_{i+2} From N_i to N_{i+1} with increment N_{i+2}
		12.12.6 SND, SNV, SNA, SDFAC Multiplication factor SND Displacement SNV Velocity SNA Acceleration SDFAC Distance
		12.12.7 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Card Group		Input Data and Definitions
12 PLOT-XY Information	For IPTYPE = 12 (Snap Shot of a Displ./Vel./Accel. for Different Times)	12.13.1 NOTM NOTM Number of times (Max 10)
		12.13.2 TLIST (I), I = 1, NOTM TLIST (I) List times in sequential order
		12.13.3 K_y K_y Select from Table PL-2
		12.13.4 XSTART XSTART Reference starting X-coordinate
		12.13.5 Node Number Specification (Max 800 Nodes) NRL N_1, N_2, N_{NRL} NRL Number of nodes N_1, N_2, .., N_{NRL} Node numbers N_i, $-N_{i+1}$, N_{i+2} From N_i to N_{i+1} with increment N_{i+2}
		12.13.6 SND, SNV, SNA, SDFAC Multiplication factor SND Displacement SNV Velocity SNA Acceleration SDFAC Distance
		12.13.7 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

Table PL-1 (IPTYPE = 1, 3, 5, 6, 9, 10)

K_x, K_y	Legend	Description	
1	TIME	Time	(t)
		Continuum Element (See Fig. PL-1)	
2	STRESS-XX	Normal XX stress	(σ_x')
3	STRESS-YY	Normal YY stress	(σ_y')
4	STRESS-ZZ	Normal ZZ stress	(σ_z')
5	STRESS-XY	Shear XY stress	(τ_{xy})
6	STRESS-YZ	Shear YZ stress	(τ_{yz})
7	STRESS-XZ	Shear XZ stress	(τ_{xz})
8	PRESSURE	Mean pressure	(P')
9	FLUID-PRES	Fluid pressure	(π)
10	TSTRESS-XX	Normal XX total stress	($\sigma_x = \sigma_x' + \pi$)
11	TSTRESS-YY	Normal YY total stress	($\sigma_y = \sigma_y' + \pi$)
12	TSTRESS-ZZ	Normal ZZ total stress	($\sigma_z = \sigma_z' + \pi$)
13	TPRESSURE	Total mean pressure	($P = P' + \pi$)
14	D.STRES	Deviatoric stress	($Q = (3/\sqrt{2})\ \tau_{oct}$)
15	STRAIN-XX	Normal XX strain	(ε_x)
16	STRAIN-YY	Normal YY strain	(ε_y)
17	STRAIN-ZZ	Normal ZZ strain	(ε_z)
18	STRAIN-XY	Shear XY strain	(γ_{xy})
19	STRAIN-YZ	Shear YZ strain	(γ_{yz})
20	STRAIN-XZ	Shear XZ strain	(γ_{xz})
21	VOL-STRAIN	Volumetric strain	(ε_v)
22	GAMMA-OCT	Octahedral shear strain	(γ_{oct})
23	TAU-OCT	Octahedral shear stress	(τ_{oct})
24	FS	Safety factor	(Fig. PL-2)
25	YIELD-FLAG	Yield flag	(Fig. PL-3)
26	STRESS - 1	Major principal stress	(σ_1')
27	STRESS - 2	Inter. principal stress	(σ_2')
28	STRESS - 3	Minor principal stress	(σ_3')

Table PL-1 continued

K_x, K_y	Legend	Description	
		<u>Beam Element (See Fig. PL-4)</u>	
33	THRUST-H	Thrust in hoop direction	(F_h)
34	MOMENT-H	Moment in hoop direction	(M_h)
35	THRUST	Thrust	(F_x)
36	SHEAR-Y	Shear in y direction	(F_y)
40	MOMENT-Z	Moment about z axis	(M_z)
41	STRAIN-FT	Top fiber strain	(ε_{ft})
42	STRESS-FT	Top fiber stress	(σ_{ft})
43	STRAIN-RT	Top reinf. bar strain	(ε_{rt})
44	STRESS-RT	Top reinf. bar stress	(σ_{rt})
45	STRAIN-RB	Bot. reinf. bar strain	(ε_{rb})
46	STRESS-RB	Bot. reinf. bar stress	(σ_{rb})
47	STRAIN-FB	Bot. fiber strain	(ε_{fb})
48	STRESS-FB	Bot. fiber stress	(σ_{fb})
57	HSTRESS-FT	Top fiber hoop stress	(σ_{hft})
58	HSTRESS-FB	Bot. fiber hoop stress	(σ_{hfb})
59	HSTRESS-RT	Top rebar hoop stress	(σ_{hrt})
60	HSTRESS-RB	Bot. rebar hoop stress	(σ_{hrb})
		<u>Truss Element</u>	
61	FORCE-XX	Axial force	(F_x)
62	STRESS-XX	Axial stress	(σ_x)
63	STRAIN-XX	Axial strain	(ε_x)

Table PL-1 continued

K_x, K_y	Legend	Description	
		Shell element section forces and stresses	
71	MOMENT-XX	Bending moment	(M_{xx})
72	MOMENT-YY	Bending moment	(M_{yy})
73	MOMENT-XY	Twisting moment	(M_{xy})
74	M-MAX	Max bending moment	(M_{max})
75	M-MIN	Min bending moment	(M_{min})
76	MXY-MAX	Max twisting moment	($M_{xy\ max}$)
		Mid-surface stress	
77	SMID-XX	Normal xx stress	($\sigma_{xx\ mid}$)
78	SMID-YY	Normal yy stress	($\sigma_{yy\ mid}$)
79	SMID-XY	Shear xy stress	($\sigma_{xy\ mid}$)
80	SM-MAX	Max normal xx stress	($\sigma_{max\ mid}$)
81	SM-MIN	Min normal yy stress	($\sigma_{min\ mid}$)
82	SMXY-MAX	Max shear xy stress	($\sigma_{xy\ max\ mid}$)
		Top-surface stress	
83	STOP-XX	Normal xx stress	($\sigma_{xx\ top}$)
84	STOP-YY	Normal yy stress	($\sigma_{yy\ top}$)
85	STOP-XY	Shear xy stress	($\sigma_{xy\ top}$)
86	ST-MAX	Max normal xx stress	($\sigma_{max\ top}$)
87	ST-MIN	Min normal yy stress	($\sigma_{min\ top}$)
88	STXY-MAX	Max shear xy stress	($\sigma_{xy\ max\ top}$)
		Bottom-surface stress	
89	SBOT-XX	Normal xx stress	($\sigma_{xx\ bot}$)
90	SBOT-YY	Normal yy stress	($\sigma_{yy\ bot}$)
91	SBOT-XY	Shear xy stress	($\sigma_{xy\ bot}$)
92	SB-MAX	Max normal xx stress	($\sigma_{max\ bot}$)
93	SB-MIN	Min normal yy stress	($\sigma_{min\ bot}$)
94	SBXY-MAX	Max shear xy stress	($\sigma_{xy\ max\ bot}$)
		Note: Bending and Twisting moments are Moments per unit width. (See Fig. PL-5)	

Table PL-2 (IPTYPE = 2, 4, 7, 8, 11, 12)

K_x, K_y	Legend	Description	
1	TIME	Time	(t)
		<u>Skeleton displacement</u>	
2	X-DIS.	X-displacement	(u_x)
3	Y-DIS.	Y-displacement	(u_y)
4	Z-DIS.	Z-displacement	(u_z)
5	X-VEL.	X-velocity	(u_x)
6	Y-VEL.	Y-velocity	(u_y)
7	Z-VEL.	Z-velocity	(u_z)
8	X-ACC.	X-acceleration	(u_x)
9	Y-ACC.	Y-acceleration	(u_y)
10	Z-ACC.	Z-acceleration	(u_z)
		<u>Relative fluid displacement</u>	
11	R.FL.X-DIS	X-displacement	($w_x = n\ (U_x - u_x)$)
12	R.FL.Y-DIS	Y-displacement	(w_y)
13	R.FL.Z-DIS	Z-displacement	(w_z)
14	R.FL.X-VEL	X-velocity	(w_x)
15	R.FL.Y-VEL	Y-velocity	(w_y)
16	R.FL.Z-VEL	Z-velocity	(w_z)
17	R.FL.X-ACC	X-acceleration	(w_x)
18	R.FL.Y-ACC	Y-acceleration	(w_y)
19	R.FL Z-ACC	Z-acceleration	(w_z)

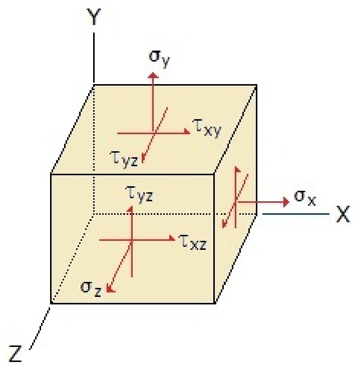


Figure PL-1 Sign Conventions for Continuum Stress

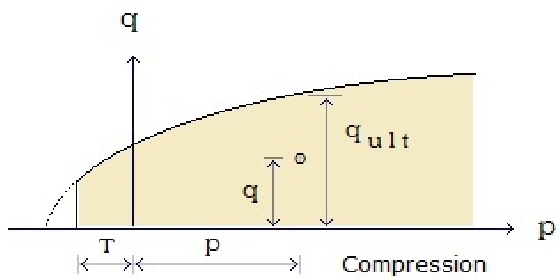


Figure PL-2 Definition of Safety Factor

Factor of Safety (FS) is defined as:

For elastic material FS = 10

For elasto-plastic material FS = q_{ult} / q
FS is limited to $1 \leq$ F.S. ≤ 10
For $p \leq -T$ FS = 1

$P = (\sigma_x + \sigma_y + \sigma_z) / 3$
$q = (3 / \sqrt{2})\, \tau_{oct}$

Yield Flag	Stress Status for Beam & Continuum Element
0	Stress point is in elastic
1	Stress point is in plastic
2	Stress Point develops crack

Stress Status for Truss Element

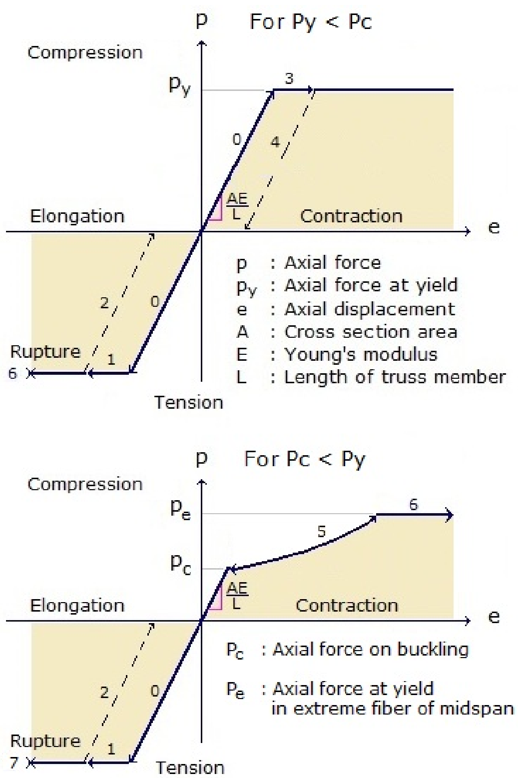


Figure PL-3 Description of Stress Status

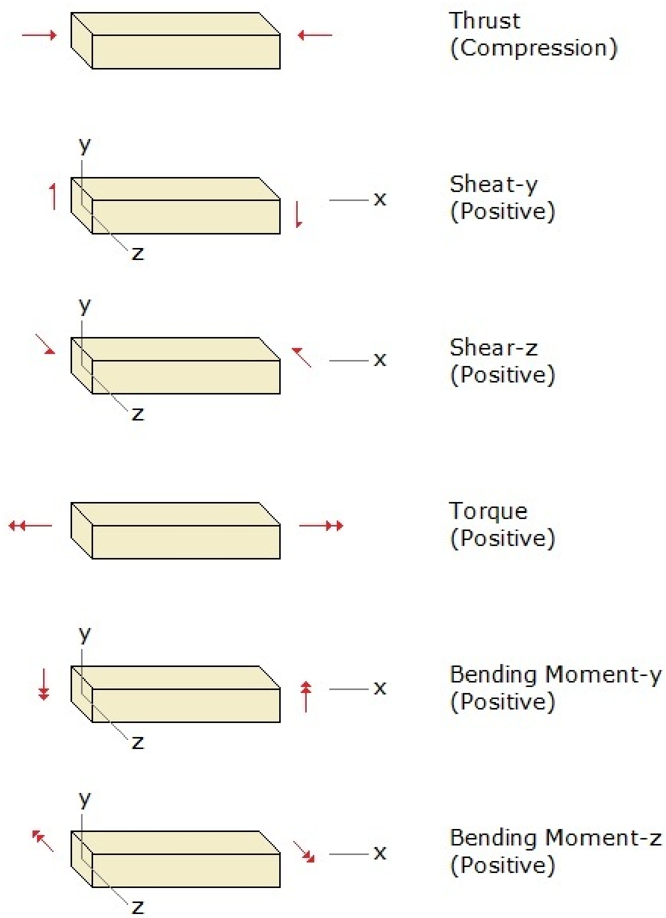
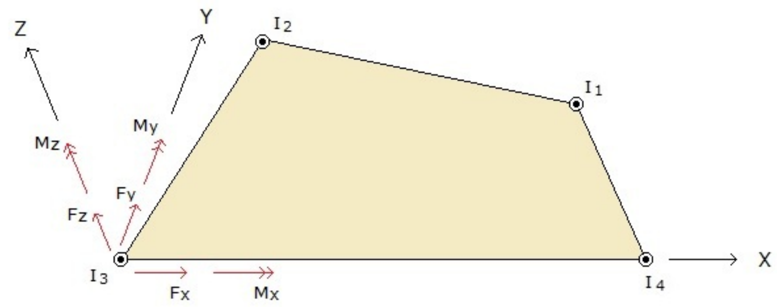
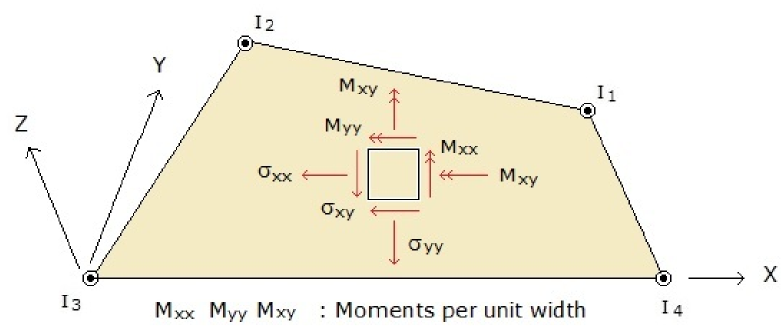


Figure PL-4 Sign Conventions for Beam



Shell Member End Forces at Element Nodes
[Output File SHELMEF.DAT]



Shell Stresses and Moments at Element Center
[Output File: SHELSM.DAT]

Figure PL-5 Sign Conventions for Shell

Group Mesh
User's Manual

5.1 Introduction

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.

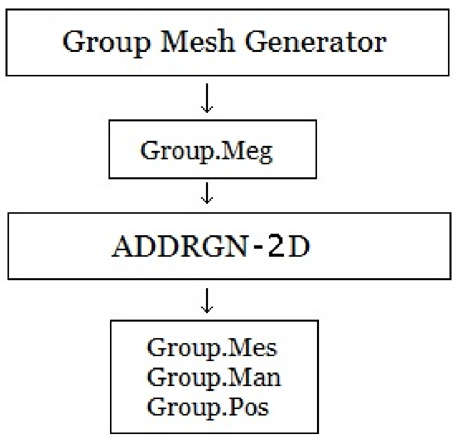


Figure 5.1 Flow diagram of group mesh generation

Group.Meg contains group mesh data that can be generated or modified by Group Mesh Generator. The file Group.Meg is used as input to the program ADDRGN-2D, thereby generating finite element mesh file Group.Mes along with the main file Group.Man for element activity and the post file Group.Pos for PLOT-2D plot.

Group Mesh Generator can be accessed through SMAP menu
Run or Plot as explained in Section 5.2.

ADDRGN-2D can be accessed from SMAP menu:
Run → Mesh Generator → AddRgn → Addrgn 2D.
This program can also be accessed indirectly by executing
F. E. Mesh Plot in Group dialog as explained in Section 5.3.8.

5.2 Group Mesh Generator

Group Mesh Generator can be accessed by selecting the following menu items in SMAP:

Run → Mesh Generator → Group Mesh or
Plot → Mesh → Group Mesh

When you build new group mesh, you can select either Built-in Base Mesh or Existing Finite Element Mesh. Built-in Base Mesh is explained in detail in Section 5.4.

Once you click OK button in Group Input dialog, PLOT-2D program is displayed along with group menu which is the main access to Group Mesh Generator.

When click Group menu in PLOT-2D, Group dialog is displayed.

5.3 Group

Group dialog in Figure 5.2 is the main dialog associated with group mesh generation or modification. Group dialog consists of following eight parts:

- Group Identity
- MTYPE and Material Parameter
- Line Options
- Coordinate Constraint
- Element Activity
- PLOT-2D Plot
- Translation
- Command Buttons

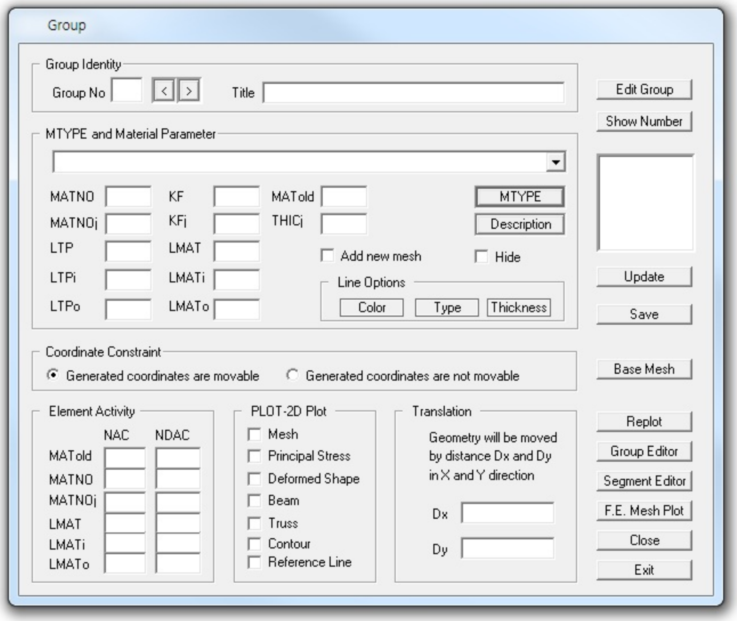


Figure 5.2 Group dialog.

5.3.1 Group Identity

Here, you type Group No and Title.

When you add a new group, first select an appropriate MTYPE and change all default parameters as you want. Then click Add Group button to build the geometry of new group.

When you type the existing Group No, all parameters of that group are shown on the screen. Click Edit Group button to modify the geometry of the group.

It should be noted that Add Group and Edit Group buttons share the same position in the Group dialog. And Add Group for new group and Edit Group for existing group will appear.

5.3.2 MTYPE and Material Parameter

MTYPE dialog with icons and MTYPE list box with brief explanations are shown in Figures 5.3 and 5.4, respectively.

You can select MTYPE from the list box or by clicking MTYPE button which opens MTYPE dialog with icons.
Selection of proper MTYPE is the most important to model the desired group. Once you select MTYPE, all input variables and options available for that MTYPE will be shown on the screen along with default values.

Figure 5.3
MTYPE dialog

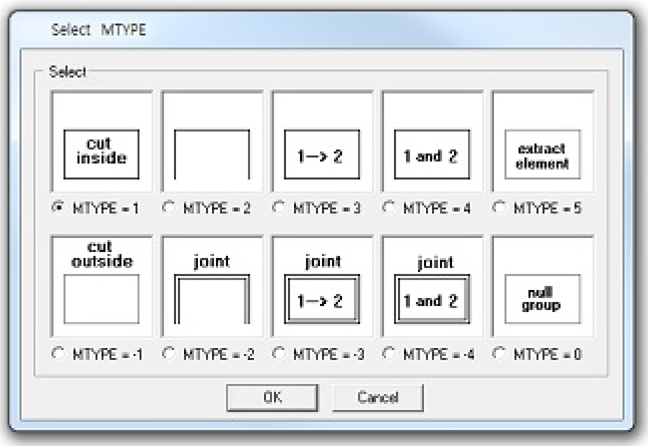
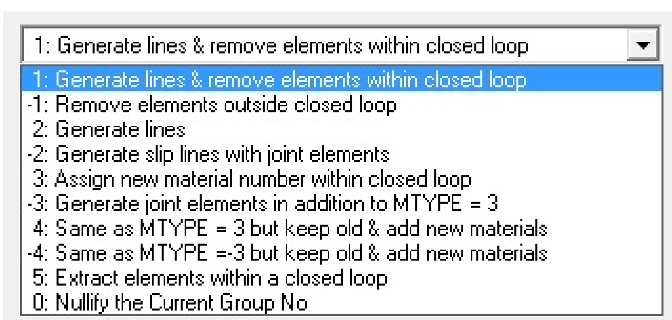


Figure 5.4
MTYPE list box



Click Description button to see description of material parameters and element activity as shown in Figure 5.5.

Add new mesh check box is available only for MTYPE = 3. When checked, new group is formed without interfering with the other groups.

Hide check box is to hide the current group geometry on the screen when checked.

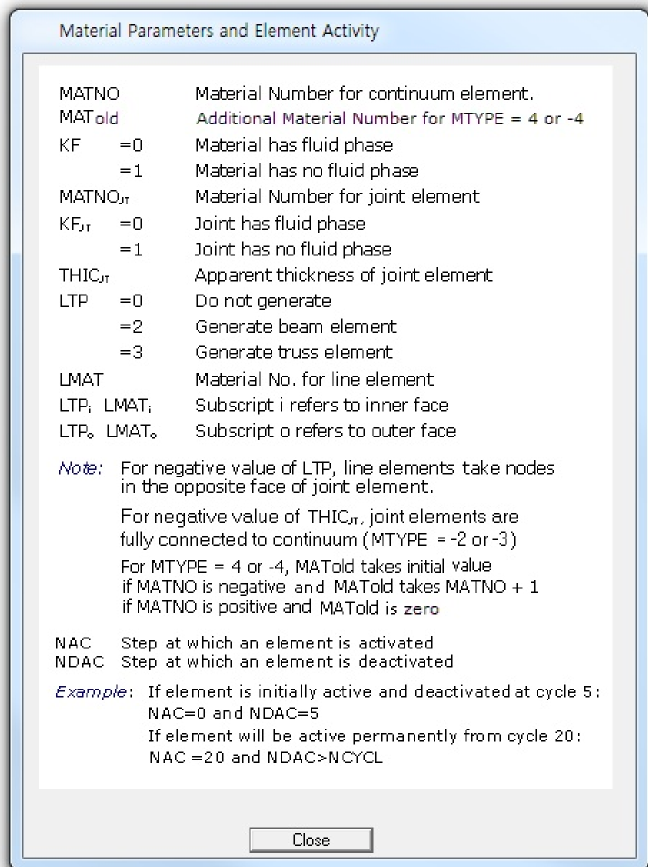


Figure 5.5 Material parameters & element activity (SMAP-2D)

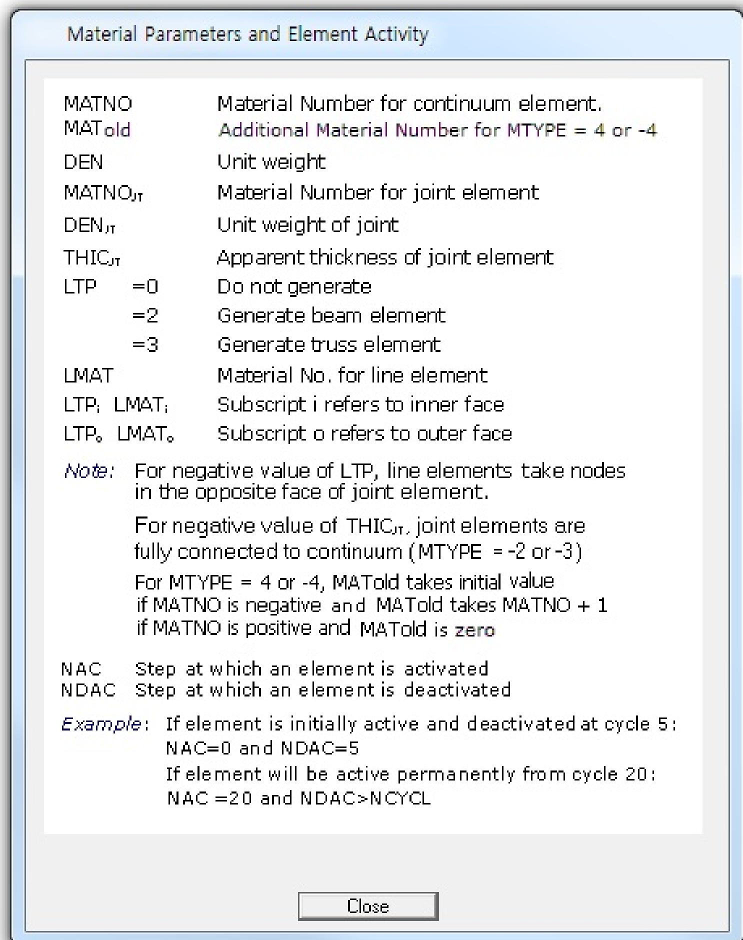


Figure 5.5 Material parameters & element activity (SMAP-S2)

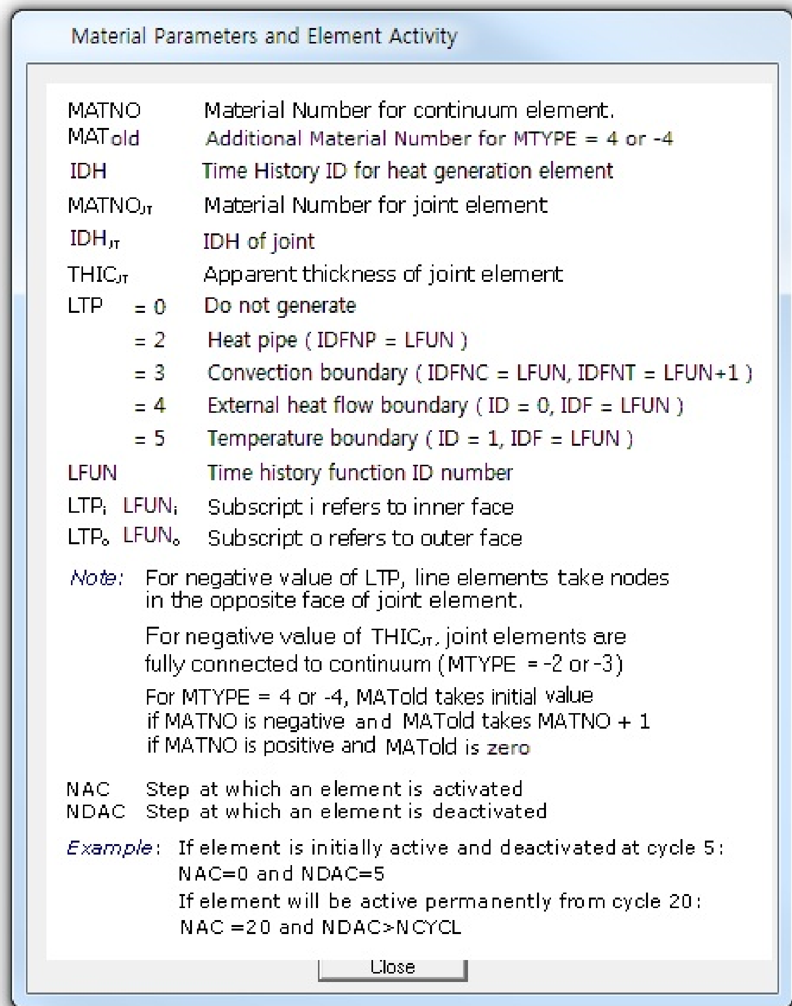


Figure 5.5 Material parameters & element activity (SMAP-T2)

5.3.3 Line Options

Line options are provided to distinguish the outline of the group from the other groups. Figure 5.6 shows available line color, line type and line thickness.

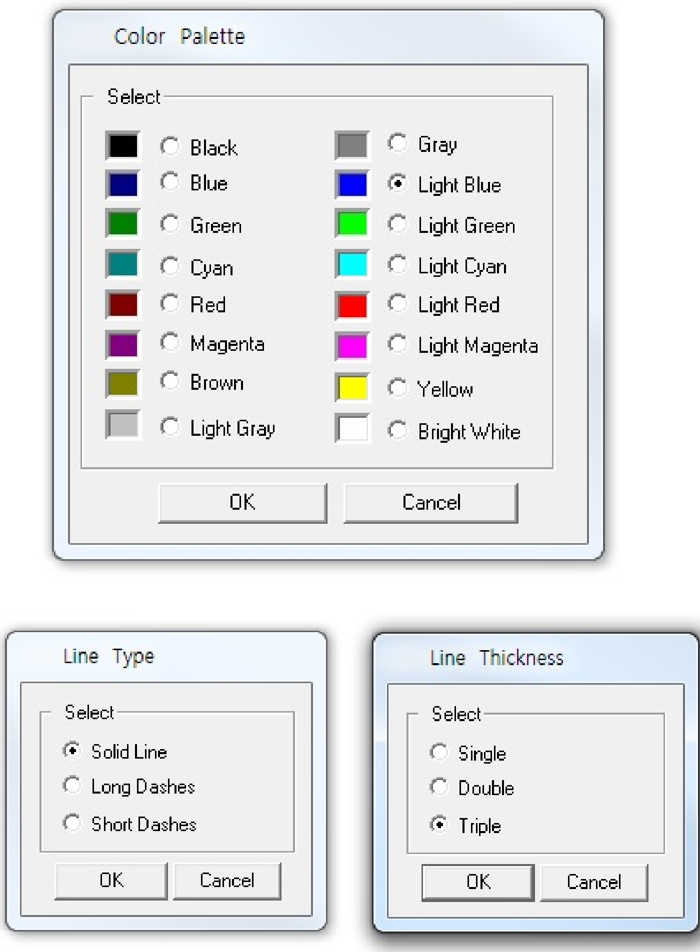


Figure 5.6 Line options.

5.3.4 Coordinate Constraint

Finite element meshes are generated when you click F. E. Mesh Plot button.

Normally, finite element nodal coordinates associated with the current group are adjusted to get the overall optimum meshes by selecting Generated coordinates are movable.

However, you can make such generated coordinates not movable by selecting Generated coordinates are not movable.

5.3.5 Element Activity

Element activity data is used in SMAP main program Card group 8. Elements in current group is to activate at step NAC and deactivate at step NDAC. Such activity data is generated and saved in Group.Man when executing group mesh, that is, by clicking F. E. Mesh Plot button.

Examples of element activity are shown at bottom of Fig. 5.5.

5.3.6 PLOT-2D Plot

PLOT-2D Plot data is used in SMAP post processing program PLOT-2D to plot computed results available for the current group. Such plot information is generated and saved in Group.Pos when executing group mesh.

It should be noted that SMAP post processing program PLOT-3D can automatically produce all such plots.

5.3.7 Translation

Translation is mainly used to move the geometry of the current group in x and y directions. Here D_x and D_y represent relative distances from the current position of the group to the new position.

Once you type in D_x and D_y, you need to click Update and then Replot buttons to confirm the translation of the current group.

5.3.8 Command Buttons

Command buttons are shown on the right side of Group dialog.

Add Group
This is used to build the geometry of the new group.
Line Segment dialog in Figure 5.14 will be displayed.

Edit Group
This is used to modify the geometry of the existing group.
Edit Segment dialog in Figure 5.7 will be displayed.

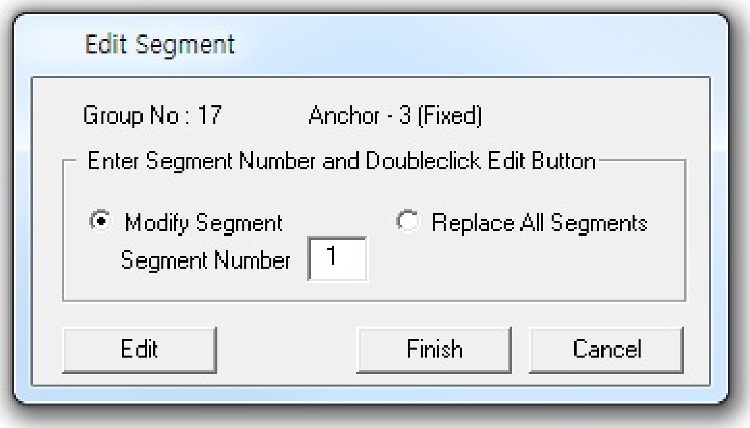


Figure 5.7 Edit segment dialog.

Show Number
This is used to show group and segment numbers.
Plot Group / Segment No dialog in Figure 5.8 will be displayed.

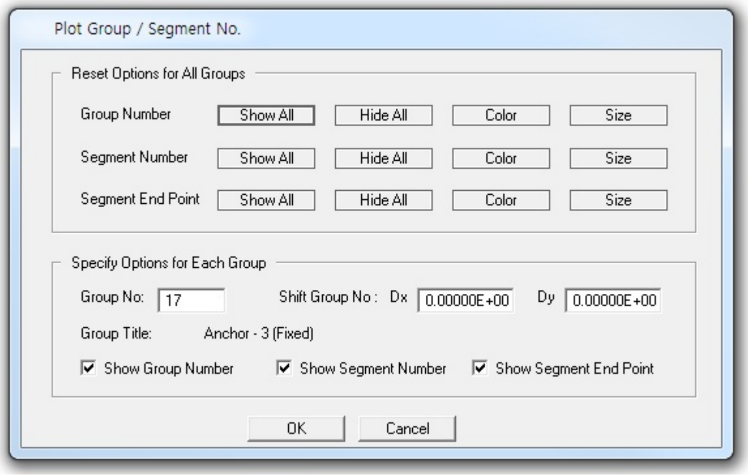


Figure 5.8 Plot Group / Segment No dialog.

Update
This is used to update the current group parameters shown on the screen. It should be noted that you need to click Update button before leaving the current group. Leaving the current group without clicking Update will not update all the changes you made on the current group.

Save
This is used to save all the works you have done . This includes updating the current group parameters shown in the Group dialog.

Base Mesh
This is used to edit Built-in Base Mesh which is explained in detail in Section 5.4. Base Mesh dialog in Figure 5.13 will be displayed.

Replot

This is used to show the geometry of groups you have updated so far.

Group Editor

This is used to delete, cut and paste, or copy and paste specified groups. Group Editor dialog in Figure 5.9 will be displayed.

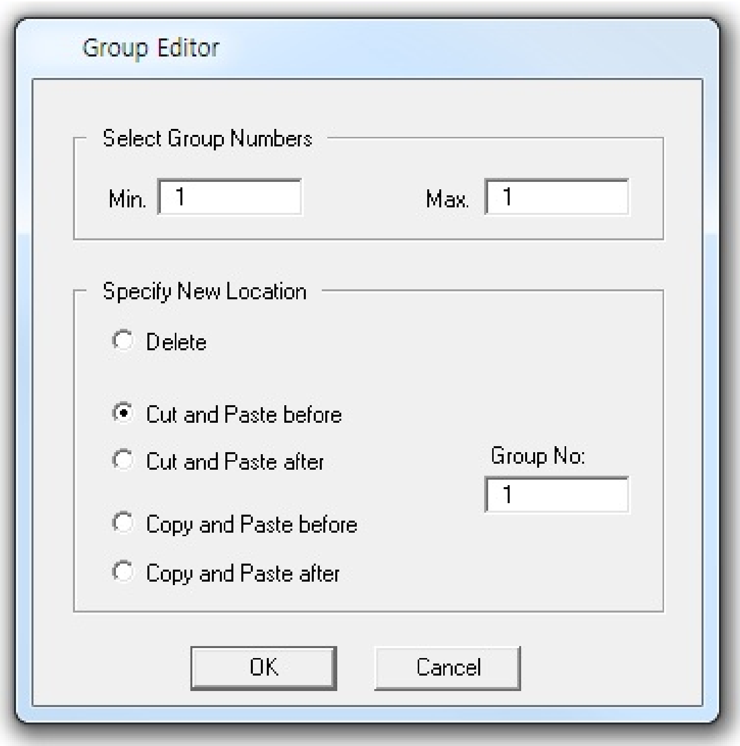


Figure 5.9 Group editor dialog.

Segment Editor

This is used to add or modify the segments of the existing group based on text input. Segment Editor dialog in Figure 5.10 will be displayed.

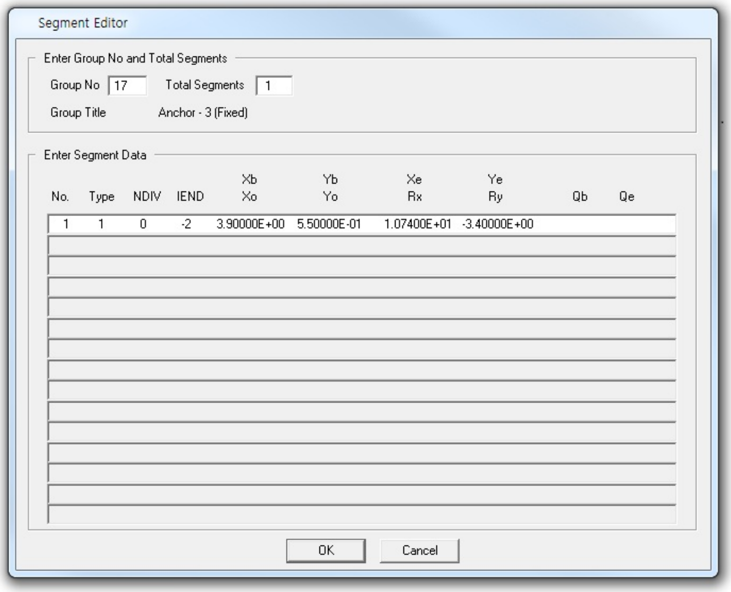


Figure 5.10 Segment editor dialog.

F. E. Mesh Plot

This is used to execute the group mesh and then plot the generated finite element mesh. It should be noted that you need to click Save button before executing F. E. Mesh Plot.

Once executed, new sub directory Plot_Mesh under working directory will be created along with following files:

Group.Mes	Mesh file with finite element.
Group.Man	Main file with element activity.
Group.Pos	Post file with PLOT-2D plot data.

Close

This is used to close the Group dialog.

Exit

This is used to exit from the Group Mesh Generator.
Exit dialog in Figure 5.11 will be displayed.

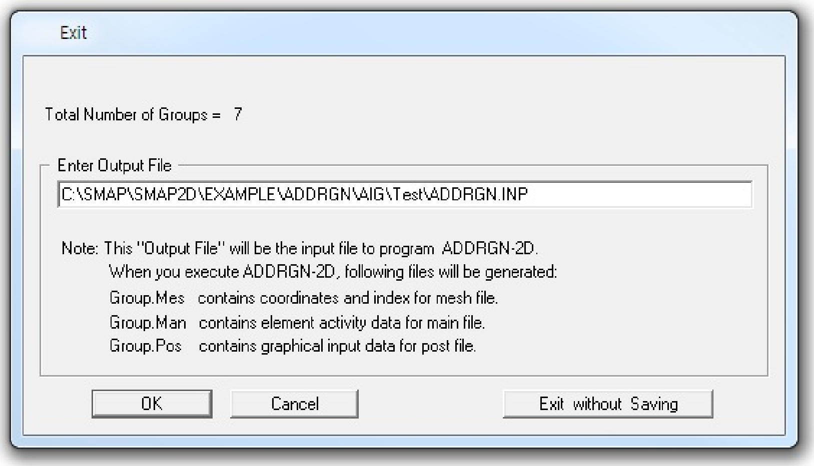


Figure 5.11 Exit dialog.

5.4 Base Mesh

Base Mesh is the finite element mesh where you build group meshes. You can select either Built-in Base Mesh or Existing Finite Element Mesh at the time when you first build new group mesh as discussed in Section 5.2.

5.4.1 Built-in Base Mesh

Figure 5.12 shows layout of Built-in Base Mesh which consists of rectangular blocks that will be filled with finite elements.

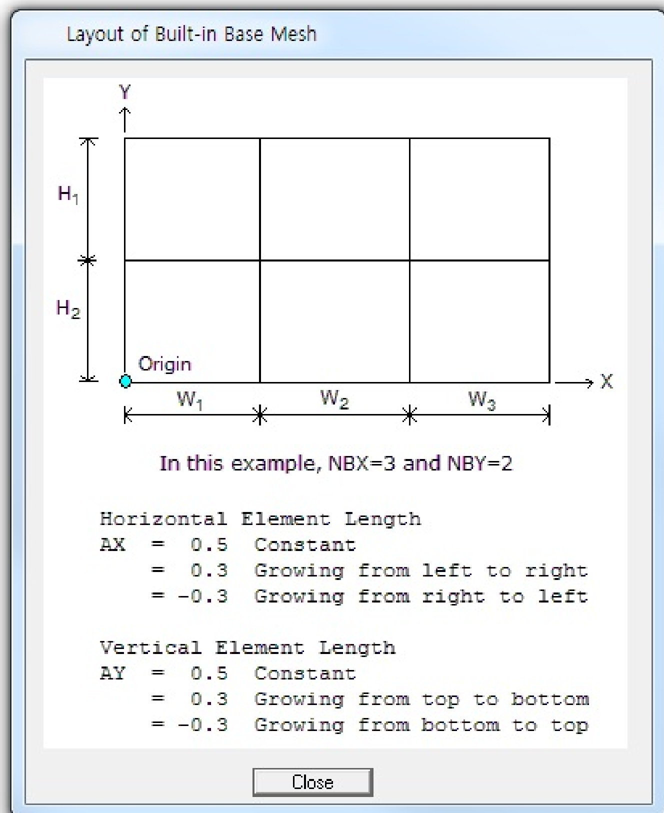


Figure 5.12 Layout of built-in base mesh.

Figure 5.13 shows Built-in Base Mesh dialog which is used to edit block dimensions, element sizes and boundary conditions.

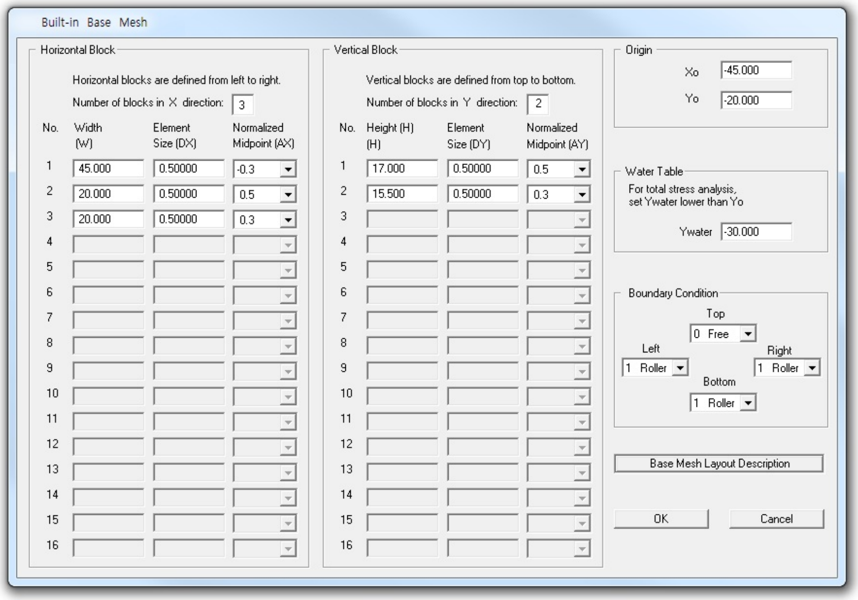


Figure 5.13 Built-in base mesh dialog.

5.4.2 Existing Finite Element Mesh

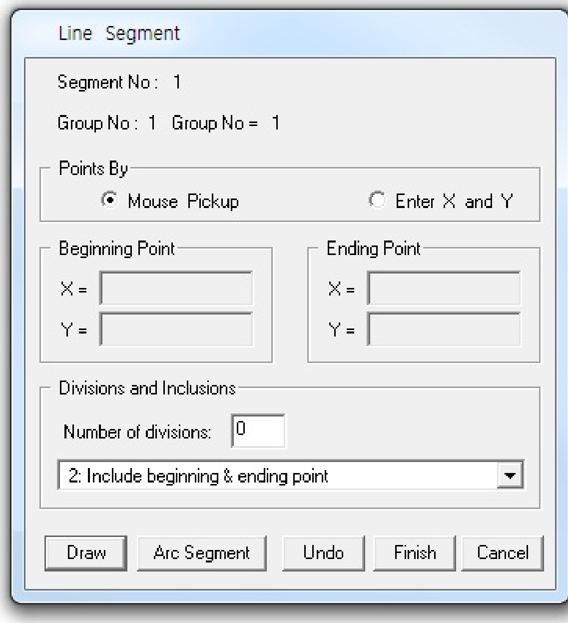
You can provide existing finite element mesh file to be used as base mesh. Group meshes will be built on this finite element mesh.

5.5 Segment

There are two types of segments, Line and Arc Segments which are used to build a group. Segment dialog will be displayed when you click Add Group or Edit Group button on the Group dialog screen.

5.5.1 Line Segment

Figure 5.14
Line segment dialog.



Line Segment dialog is shown in Figure 5.14.

Segment No
Current segment number will be displayed automatically.

Group No & Title
Current group number and title will be displayed automatically.

Point By
Select Mouse Pickup or Enter X and Y.

Beginning & Ending Point
Coordinates of beginning and ending points are required when Enter X and Y is selected.

Divisions and Inclusions
Use following default values.
Number of divisions 0
Combo box selection 2: Include beginning & ending point

Draw
Draw line segment.

For Mouse Pickup,
1. Click Draw button.
2. Move the mouse to the point and click left mouse button. Or hold down left mouse button, move the mouse and release the button at the point.

Note 1:
It is important to choose an appropriate mouse snap method before drawing by mouse. Mouse snap dialog in Figure 5.15 can be opened by clicking Mouse-Snap menu in PLOT-2D.
For example, when you choose Snap to Grid, mouse cursor will automatically move to the nearest grid point.

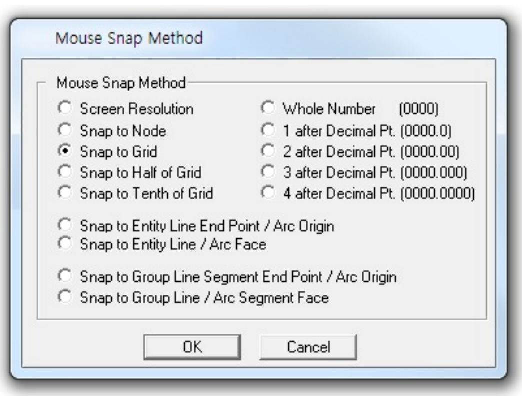


Figure 5.15
Mouse snap dialog

For Enter X and Y,

1. Type in the coordinates of beginning and ending points.
2. Click Draw button.

Note 2:
You can draw many segments continuously by repeating above Mouse Pickup or Enter X and Y procedure.

Arc Segment
Switch to arc segment.

Undo
Undo the changes you just made for line segment.

Finish
Finish and exit from drawing the current group.

Cancel
Cancel and exit from drawing the current group.

5.5.2 Arc Segment

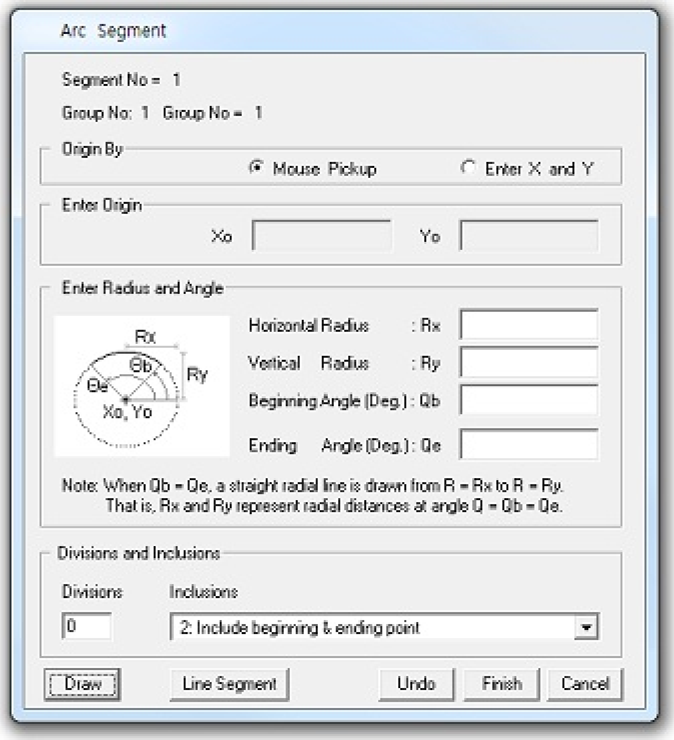


Figure 5.16 Arc segment dialog.

Arc Segment dialog is shown in Figure 5.16.

Segment No
Current segment number will be displayed automatically.

Group No & Title
Current group number and title will be displayed automatically.

Origin By
Select Mouse Pickup or Enter X and Y.

Enter Origin
Coordinates of origin are required for Enter X and Y.

Enter Radius and Angle
Enter Horizontal & vertical radii, and beginning & ending angles.

Divisions and Inclusions
Use following default values.
Number of divisions 0
Combo box selection 2: Include beginning & ending point

Draw
Draw arc segment.

For Mouse Pickup,
1. Type in R_x, R_y, Θ_b, Θ_e
2. Click Draw button
3. Move the mouse to the origin and click left mouse button. Or hold down left mouse button, move the mouse and release the button at the origin.

For Enter X and Y,
1. Type in X_o, Y_o, R_x, R_y, Θ_b, Θ_e
2. Click Draw button

Refer to Note 1 & 2 in Section 5.5.1.

Line Segment
Switch to line segment.

Undo
Undo the changes you just made for arc segment.

Finish
Finish and exit from drawing the current group.

Cancel
Cancel and exit from drawing the current group.

5.6 Modifying Finite Element Meshes

Group Mesh Generator can be used to directly modify finite element meshes.

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 5.17.

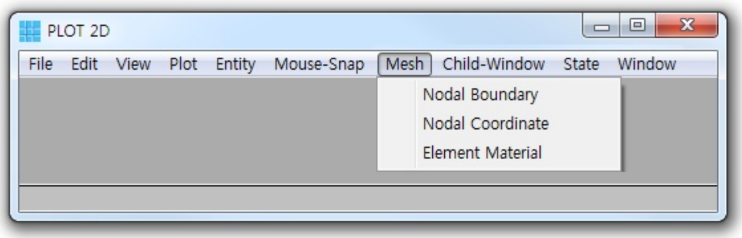


Figure 5.17 Menu for editing finite element mesh

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.

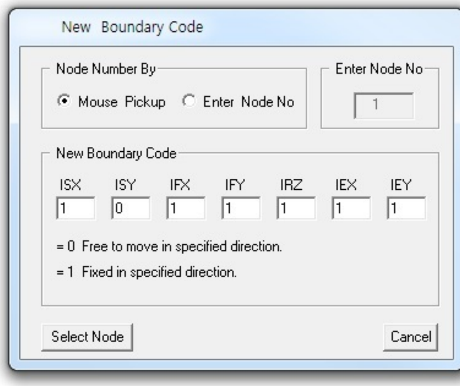
5.6.1 Edit Nodal Boundary

When you click Nodal Boundary from the Mesh menu,
Edit Boundary dialog will be displayed.

5.6.1.1 Mouse Pickup

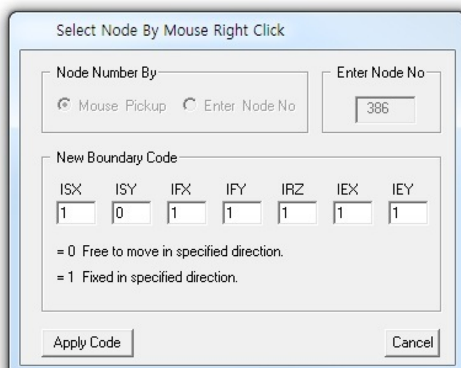
When you select Mouse Pickup mode as in Figure 5.18, you are supposed to select node number by mouse click. Click Select Node button.

Figure 5.18
Edit boundary
(Mouse Pickup)



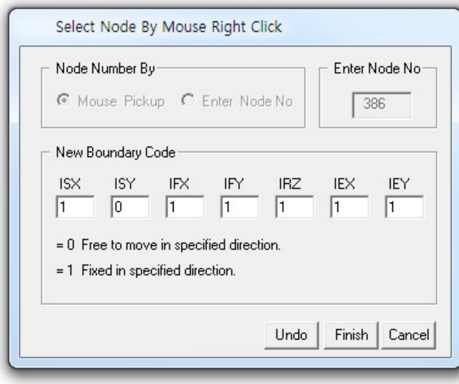
Click the node by Mouse Right Click, edit boundary codes and then click Apply Code button in Figure 5.19.

Figure 5.19
Edit boundary
(Apply Code)



You can repeat the same procedure many times for other nodes. Once finished, click Finish button in Figure 5.20.

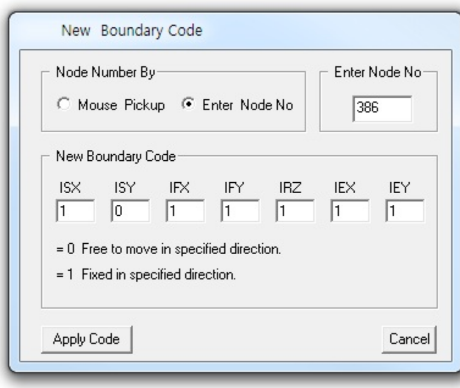
Figure 5.20
Edit boundary (Finish)



5.6.1.2 Enter Node No

When you select Enter Node No mode as in Figure 5.21, you are supposed to type in node number. Edit boundary codes and then click Apply Code button.

Figure 5.21
Edit boundary (Enter Node No)



You can repeat the same procedure many times for other nodes. Once finished, click Finish button.

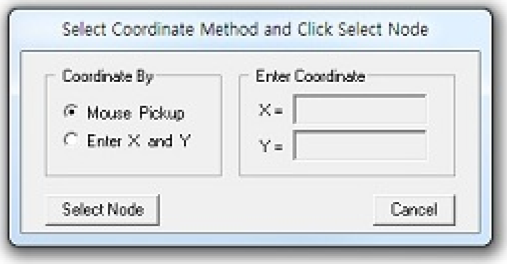
5.6.2 Edit Nodal Coordinate

When you click Nodal Coordinate from the Mesh menu, Edit Coordinate dialog will be displayed.

5.6.2.1 Mouse Pickup

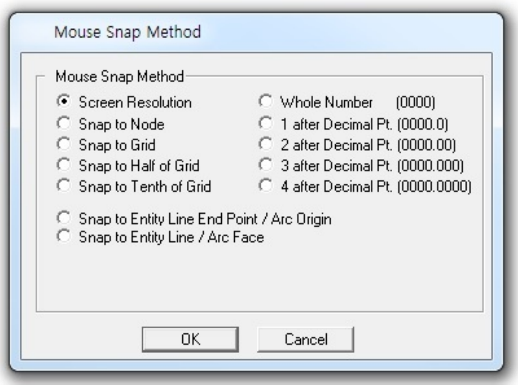
When you select Mouse Pickup mode as in Figure 5.22, you are supposed to select node number by mouse click. Click Select Node button.

Figure 5.22
Edit coordinate
(Mouse Pickup)



Select the node number by Mouse Right Click and then move the coordinate by Mouse Left Click. It is convenient to select an appropriate Mouse-Snap method in Figure 5.23 before moving the coordinates.

Figure 5.23
Mouse snap method



You can repeat the same procedure many times for other nodes. Once finished, click Finish button in Figure 5.24.

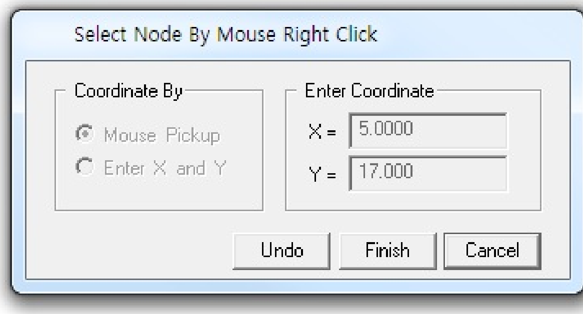


Figure 5.24 Edit coordinate (Finish)

5.6.2.2 Enter X and Y

When you select Enter X and Y mode as in Figure 5.25, you are supposed to type in nodal coordinates. Type in X and Y coordinates and then click Apply button.

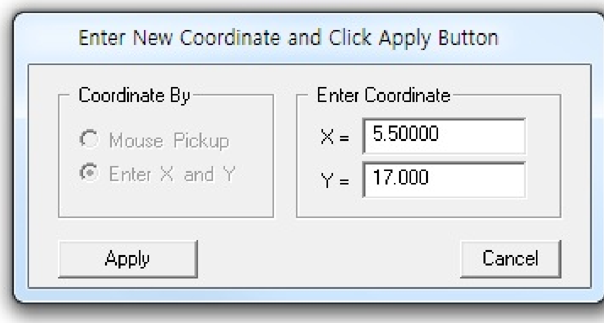


Figure 5.25 Edit coordinate (Enter X and Y)

You can repeat the same procedure many times for other nodes. Once finished, click Finish button.

5.6.3 Edit Element Material

When you click Element Material from the Mesh menu, Edit Element Material dialog will be displayed.

5.6.3.1 Mouse Pickup

When you select Mouse Pickup mode as in Figure 5.26, you are supposed to select element number by mouse click.
Click Select Element button.

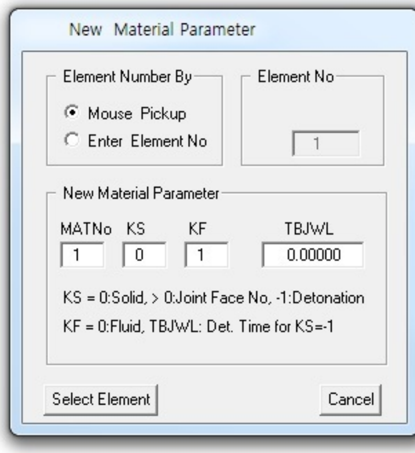


Figure 5.26
Edit element material
(Mouse Pickup)

Click the element by Mouse Right Click, edit material parameters and then click Apply button in Figure 5.27.

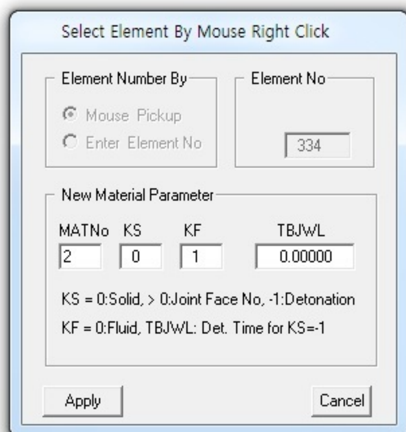
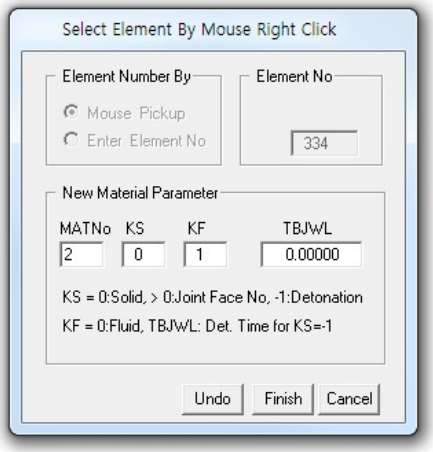


Figure 5.27
Edit element material
(Apply)

You can repeat the same procedure many times for other elements. Once finished, click Finish button in Figure 5.28.

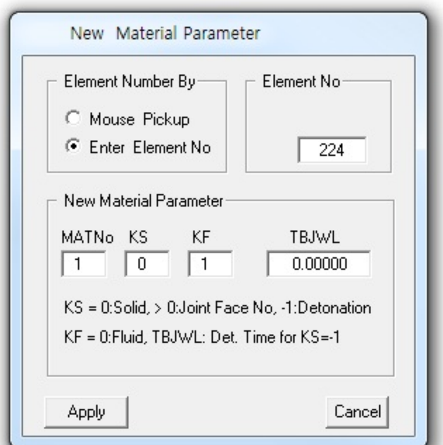
Figure 5.28
Edit element material
(Finish)



5.6.3.2 Enter Element No

When you select Enter Element No mode as in Figure 5.29, you are supposed to type in element number. Edit material parameters and then click Apply button.

Figure 5.29
Edit element material
(Enter Element No)



You can repeat the same procedure many times for other elements. Once finished, click Finish button.

5.7 Entities

Entities are graphical objects which are mainly used to assist editing the geometry of groups and elements.

There are four types of entities: Mark, Line, Arc, and Text.

Entities can be accessed from Entity menu in PLOT-2D as shown in Figure 5.30.

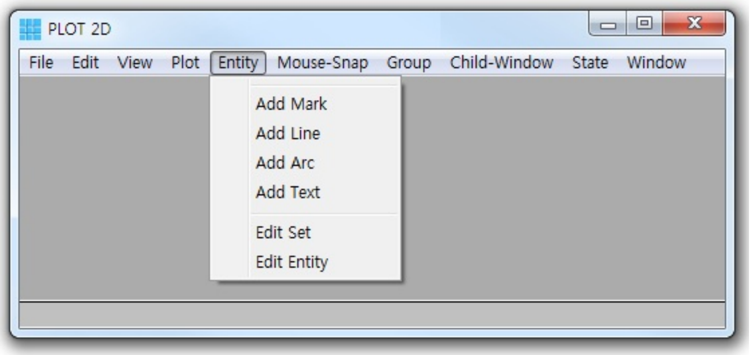


Figure 5.30 Entity menu

Entity menu has six parts:
Add Mark, Add Line, Add Arc, Add Text, Edit Set and Edit Entity.

First four Add Entities are to build new entities.

Edit Set is to assign entity set so that each plot number can include only selected entities.

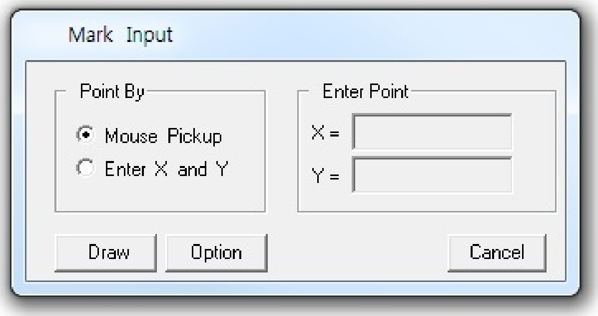
Edit Entity is to modify, delete or replace the selected entity.

5.7.1 Add Mark

Marks are graphical symbols which are mainly used to assist editing the geometry of groups and elements.

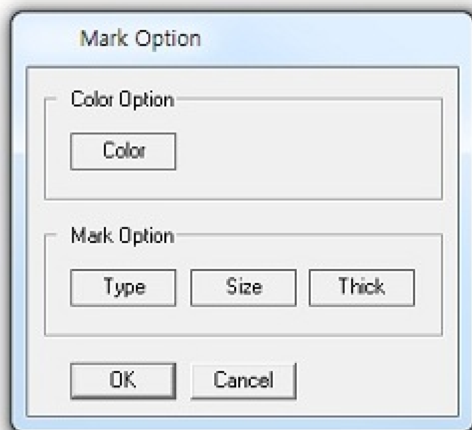
When you select Add Mark submenu, Mark Input dialog in Figure 5.31 is displayed.

Figure 5.31
Mark input
(Mouse Pickup)



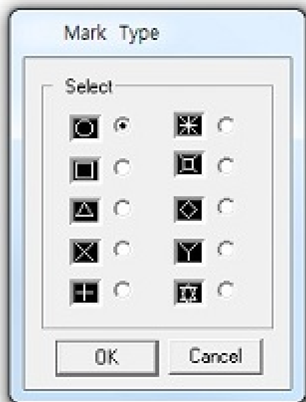
Option button is to show Mark Option in Figure 5.32.

Figure 5.32
Mark option dialog



Available Mark Types are shown in Figure 5.33.

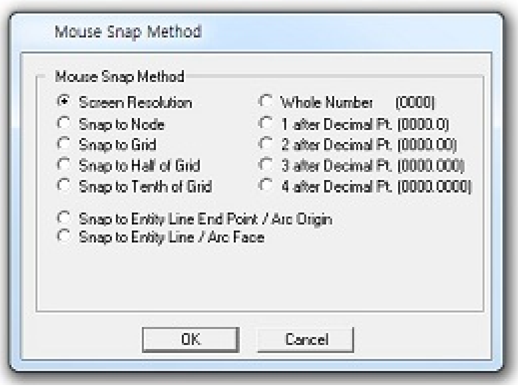
Figure 5.33 Mark type dialog



5.7.1.1 Mouse Pickup

When you select Mouse Pickup mode as in Figure 5.31, you are supposed to select the mark center position by mouse click. Click Draw button and then move the position by Mouse Left Click. It is convenient to select an appropriate Mouse-Snap method in Figure 5.34 before moving the position.

Figure 5.34
Mouse snap method



Once finished, click Finish button in Figure 5.35.

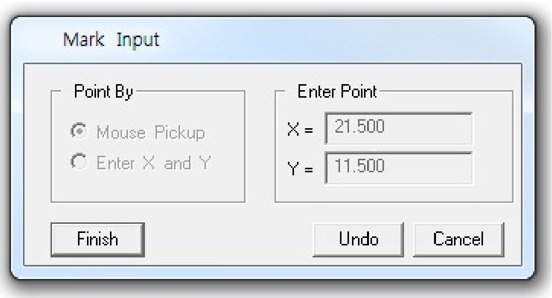


Figure 5.35
Mark input
(Finish)

5.7.1.2 Enter X and Y

When you select Enter X and Y mode as in Figure 5.36, you are supposed to type in the coordinates of the mark center position. Click Draw button.

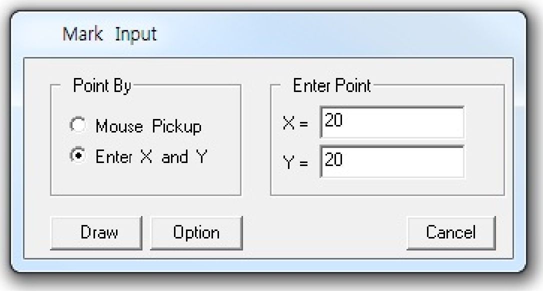


Figure 5.36
Mark input
(Enter X and Y)

Once finished, click Finish button in Figure 5.37.

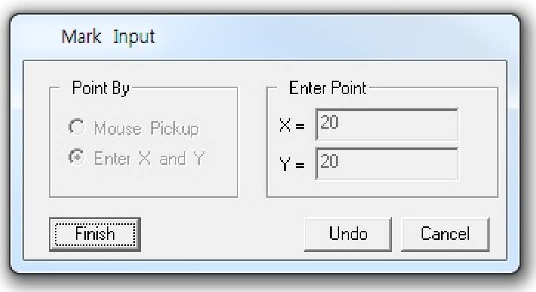


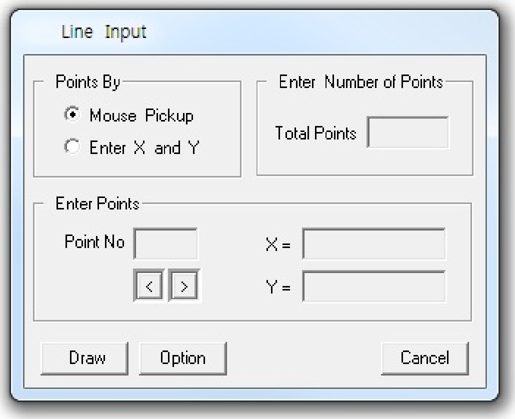
Figure 5.37
Mark input
(Finish)

5.7.2 Add Line

Lines are graphical objects which are mainly used to assist editing the geometry of groups and elements.

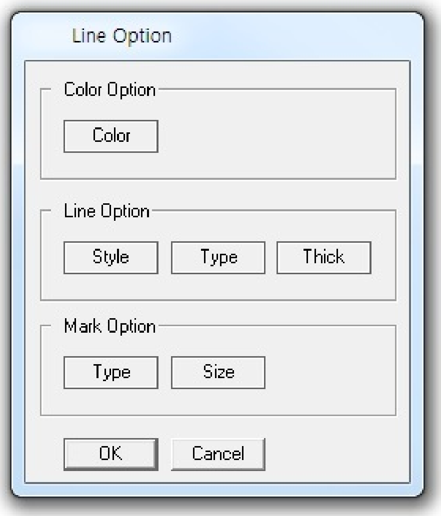
When you select Add Line submenu, Line Input dialog in Figure 5.38 is displayed.

Figure 5.38
Line input
(Mouse Pickup)



Option button is to show Line Option in Figure 5.39.

Figure 5.39
Line option dialog



Available Line Styles are shown in Figure 5.40.

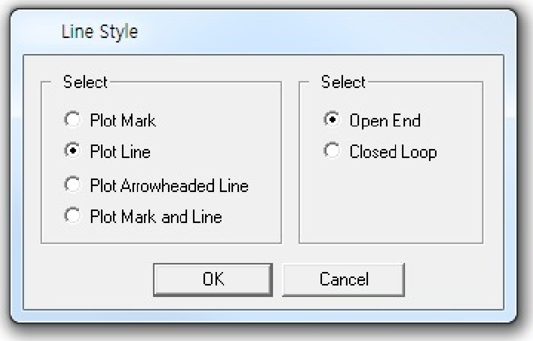


Figure 5.40 Line style dialog

Available Line Types are shown in Figure 5.41.

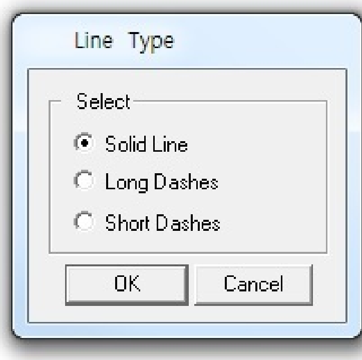


Figure 5.41 Line type dialog

5.7.2.1 Mouse Pickup

When you select Mouse Pickup mode as in Figure 5.38,
you are supposed to select the line end point by mouse click.
Click Draw button and then select the point by Mouse Left Click.

It is convenient to select an appropriate Mouse-Snap method in Figure 5.34 before moving the coordinate.

You can click many points to build continuous lines.
Once finished, click Finish button in Figure 5.42.

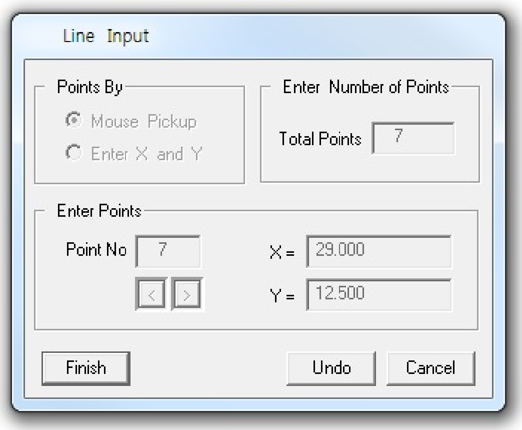
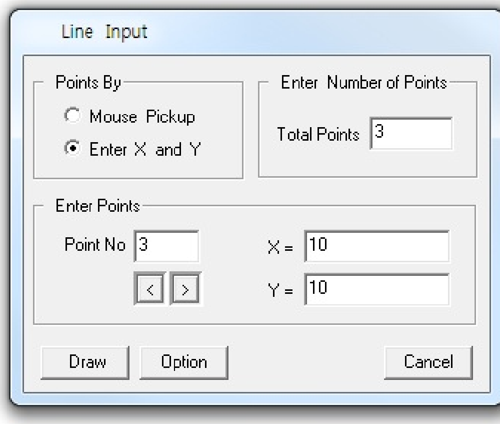


Figure 5.42 Line input (Finish)

5.7.2.2 Enter X and Y

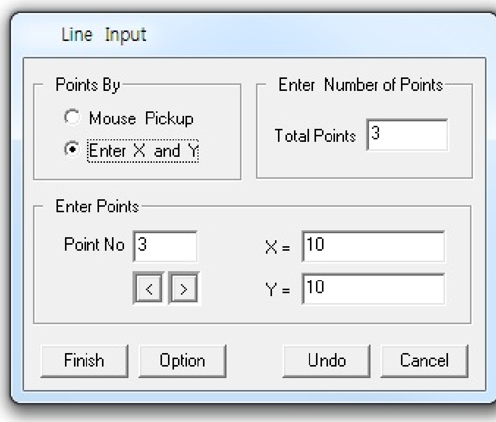
When you select Enter X and Y mode as in Figure 5.43, you are supposed to type the coordinates of the line. Click Draw button.

Figure 5.43
Line input
(Enter X and Y)



And then click Finish button in Figure 5.44.

Figure 5.44
Line input
(Finish)

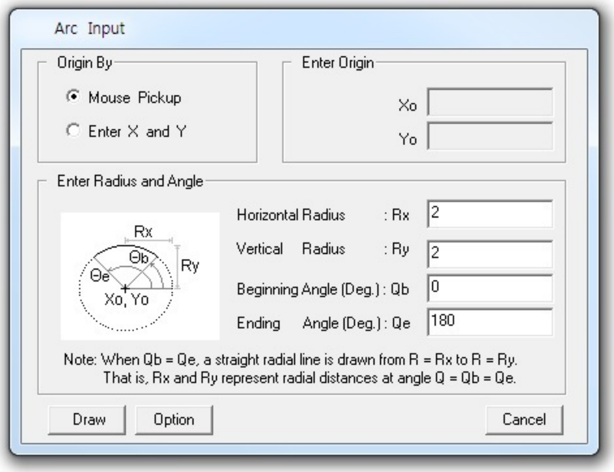


5.7.3 Add Arc

Arcs are graphical objects which are mainly used to assist editing the geometry of groups and elements.

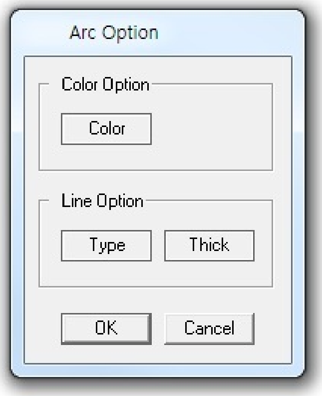
When you select Add Arc submenu, Arc Input dialog in Figure 5.45 is displayed.

Figure 5.45
Arc input
(Mouse Pickup)



Option button is to show Arc Option in Figure 5.46.

Figure 5.46 Arc option dialog



5.7.3.1 Mouse Pickup

When you select Mouse Pickup mode as in Figure 5.45,
you are supposed to select the arc origin by mouse click.

Type in Horizontal Radius, Vertical Radius, Beginning Angle and Ending Angle.

Click Draw button and then select the origin by Mouse Left Click.
It is convenient to select an appropriate Mouse-Snap method in Figure 5.34 before moving the coordinate.

Once finished, click Finish button in Figure 5.47.

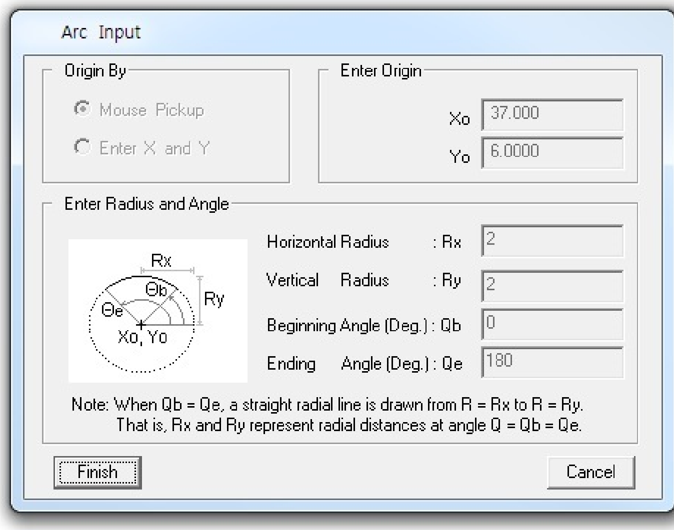


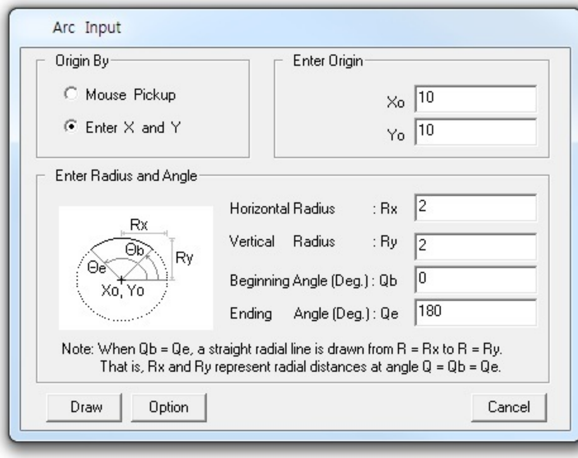
Figure 5.47 Arc input (Finish)

5.7.3.2 Enter X and Y

When you select Enter X and Y mode as in Figure 5.48, you are supposed to type in the coordinates of the arc origin.

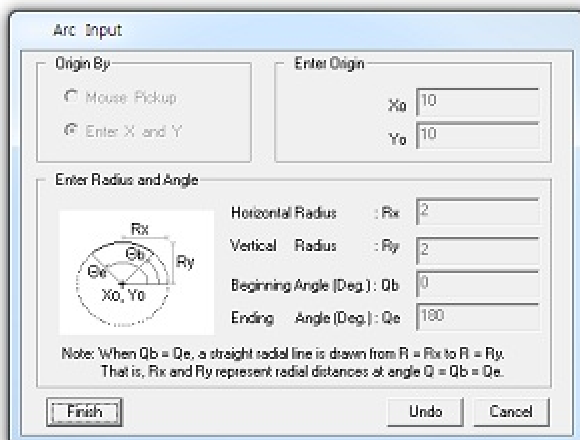
Type in Horizontal Radius, Vertical Radius, Beginning Angle and Ending Angle. And then click Draw button.

Figure 5.48
Arc input
(Enter X and Y)



Once finished, click Finish button in Figure 5.49.

Figure 5.49
Arc input
(Finish)

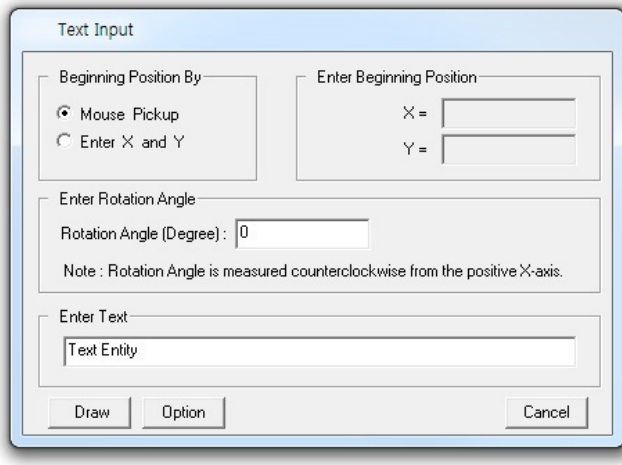


5.7.4 Add Text

Texts are characters which are mainly used to assist describing the geometry of groups and elements.

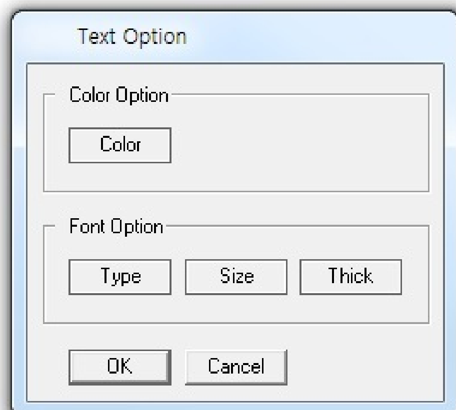
When you select Add Text submenu, Text Input dialog in Figure 5.50 is displayed.

Figure 5.50
Text input
(Mouse Pickup)



Option button is to show Text Option in Figure 5.51.

Figure 5.51
Text option dialog



Available Font Sizes are shown in Figure 5.52.

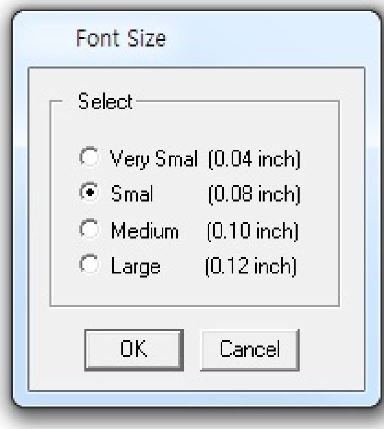


Figure 5.52 Font size dialog

5.7.4.1 Mouse Pickup

When you select Mouse Pickup mode as in Figure 5.50,
you are supposed to select the beginning position of text by mouse click.

Type in Rotation Angle and Text.

Click Draw button and then select the beginning position of the text by Mouse Left Click.   It is convenient to select an appropriate Mouse-Snap method in Figure 5.34 before moving the coordinate.

Once finished, click Finish button in Figure 5.53.

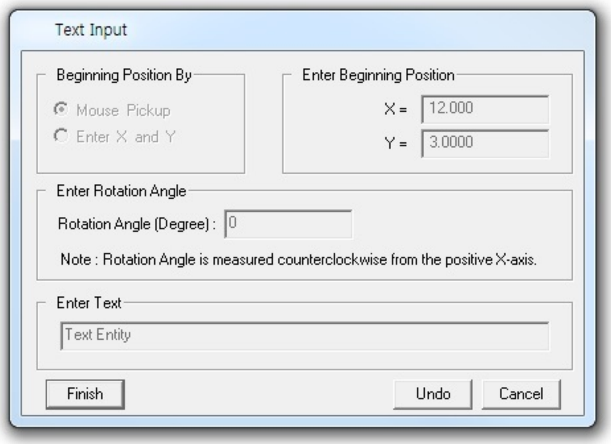


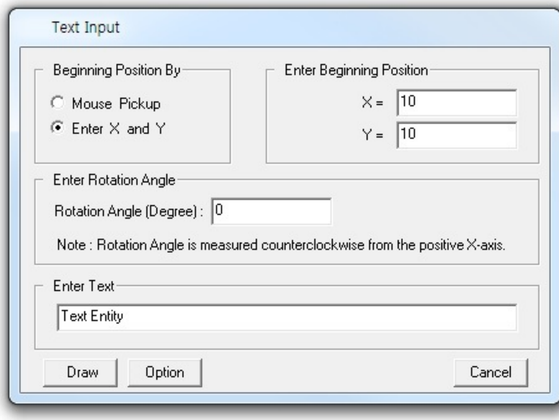
Figure 5.53   Text input (Finish)

5.7.4.2 Enter X and Y

When you select Enter X and Y mode as in Figure 5.54,
you are supposed to type in the coordinates of beginning position of text.

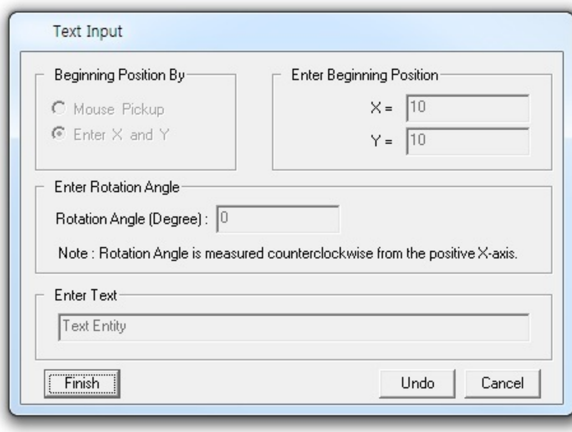
Type in Rotation Angle and Text. And then click Draw button.

Figure 5.54
Text input
(Enter X and Y)



Once finished, click Finish button in Figure 5.55.

Figure 5.55
Text input
(Finish)



5.7.5 Edit Set

Edit Set is to assign Entity Set as shown in Figure 5.56.

Edit Set consists of two parts:

1. Enter Entity Set Number and assign Entity Numbers.
2. Enter Plot Number and assign Entity Set Number.

Every time Enter Set Number or Plot Number is changed, click Update button. When finished, click Finish button.

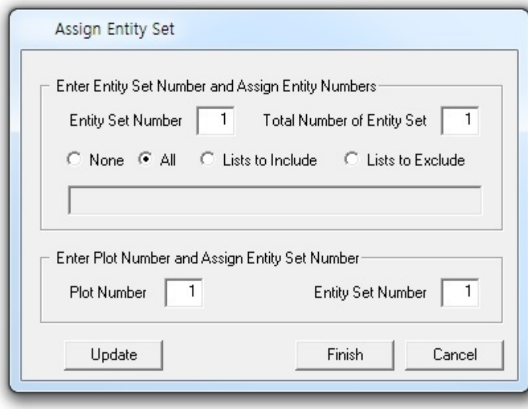


Figure 5.56
Assign entity set dialog

5.7.5.1 Enter Entity Set No & Assign Entity No

Here, you enter Entity Set Number, Total Number of Entity Set and then select Option for the current set.

When Lists to Include option is selected, type in entity numbers to be included in the current set.

When Lists to Exclude option is selected, type in entity numbers to be excluded in the current set.

5.7.5.2 Enter Plot No & Assign Entity Set No

Here, you enter Plot Number and assign Entity Set Number.

5.7.6 Edit Entity

Edit Entity is to modify, delete or replace the selected entity as shown in Figure 5.57.

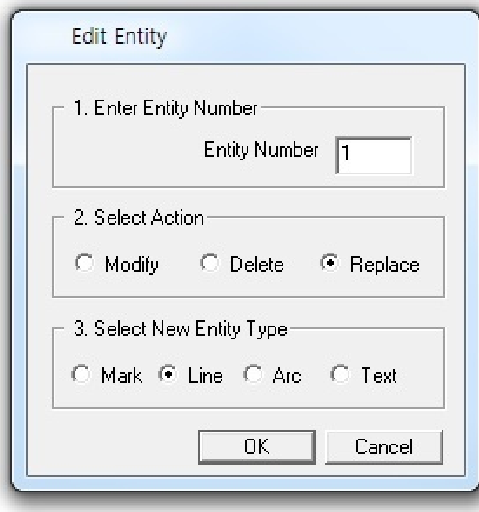


Figure 5.57
Edit entity dialog

5.7.6.1 Modify

Modify is to modify the current entity.

When OK button is clicked, Entity Input dialog corresponding to the current entity is displayed. Follow the same procedure as described in Add Entity.

5.7.6.2 Delete

Delete is to delete the current entity.

5.7.6.3 Replace

Replace is to replace the current entity by new entity type.

When OK button is clicked, Entity Input dialog corresponding to the new entity type is displayed. Follow the same procedure as described in Add Entity.

Block Mesh
User's Manual

6.1 Introduction

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.

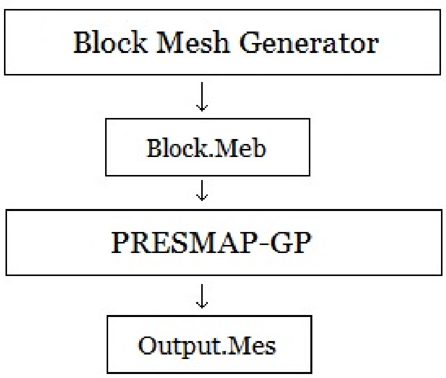


Figure 6.1 Flow diagram of block mesh generation

Block.Meb contains block mesh data that can be generated or modified by Block Mesh Generator. The file Block.Meb is used as input to the program PRESMAP-GP, thereby generating finite element mesh file Output.Mes.

Block Mesh Generator can be accessed through SMAP menu
Run or Plot as explained in Section 6.2.

PRESMAP-GP can be accessed from SMAP menu:
Run → Mesh Generator → PreSmap → Presmap GP.
This program can also be accessed indirectly by executing
Show F. E. Mesh in Block Editor dialog in Section 6.5.8.

6.2 Block Mesh Generator

Block Mesh Generator can be accessed by selecting the following menu items in SMAP:
Run → Mesh Generator → Block Mesh or
Plot → Mesh → Block Mesh

When you build new block mesh, PLOT-3D program in Figure 6.2 is displayed along with Work Plane Editor in Figure 6.3.
Click Block Editor toolbar in Figure 6.4. Building new block is discussed in detail in Section 6.5.8.

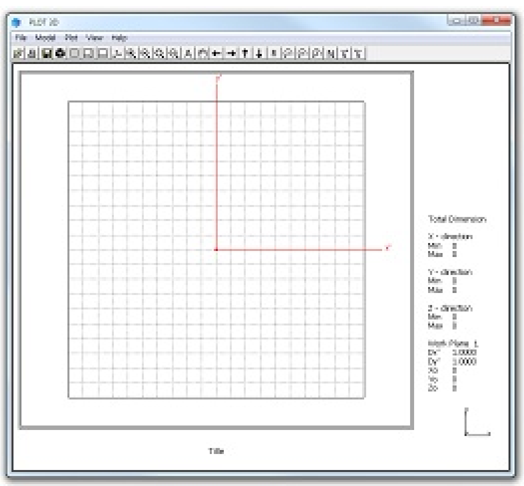


Figure 6.2 Prebuilt work plane on PLOT-3D

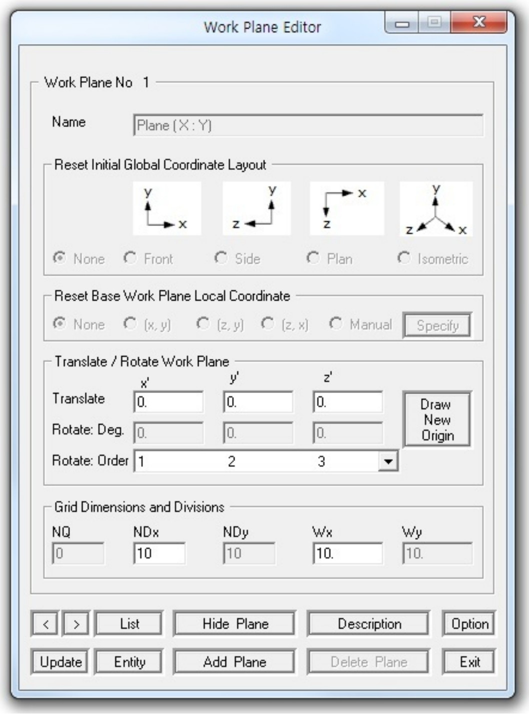


Figure 6.3 Prebuilt work plane editor

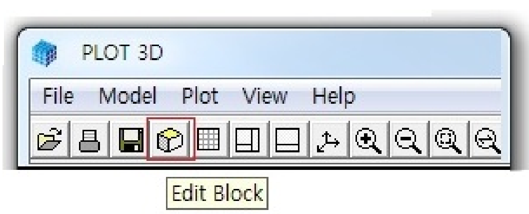


Figure 6.4 Block editor toolbar

When you open existing block mesh, Select Open in SMAP menu as shown in Figure 6.5 and then select the input file. Block mesh will be displayed on PLOT-3D as in Figure 6.6.

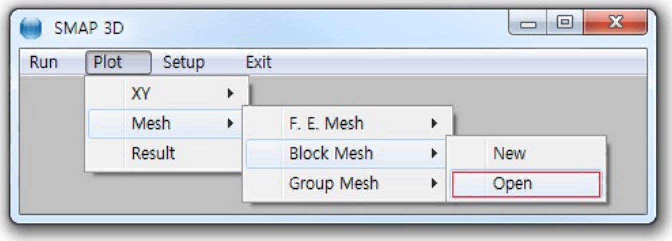


Figure 6.5 Open input file dialog

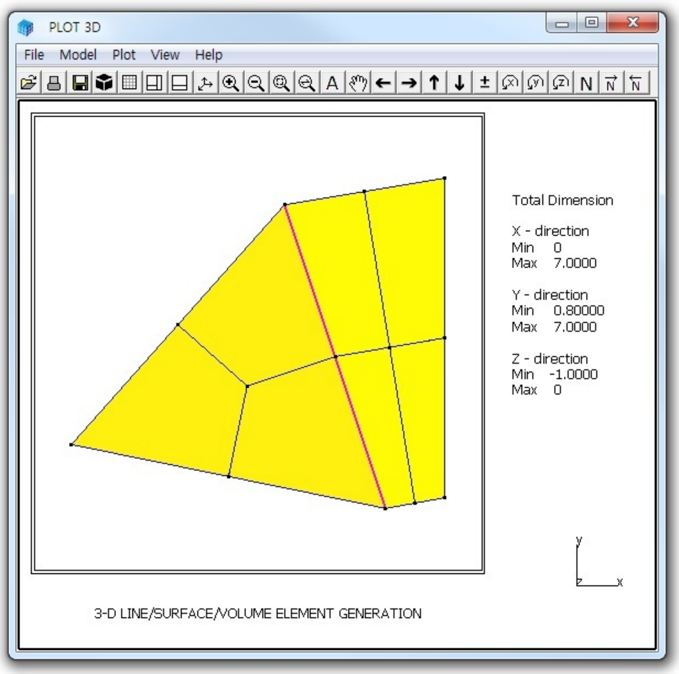


Figure 6.6 Block mesh on PLOT-3D

6.3 Work Plane

Work Planes are rectangular planes with grid lines and local coordinate axes, which are mainly used to assist editing the geometry of blocks and elements.

Work Plane Editor can be accessed by selecting the following menu items in PLOT-3D:
Model → Work Plane → Show Editor
or by clicking Work Plane toolbar as shown in Figure 6.7.

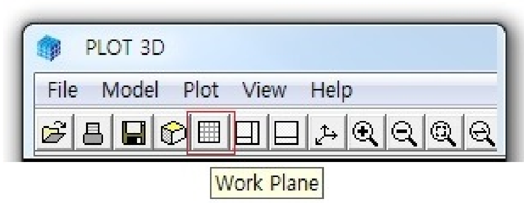


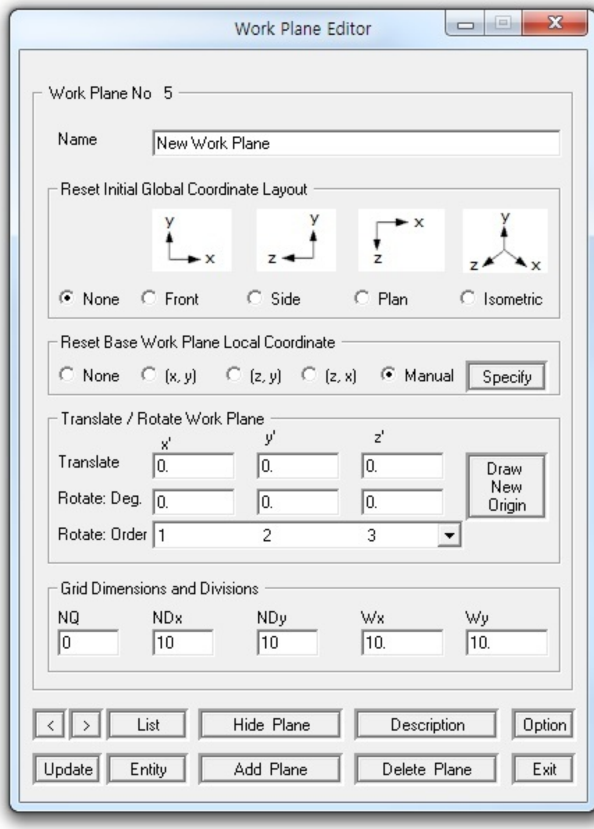
Figure 6.7 Work plane toolbar

Work Plane Editor dialog in Figure 6.8 consists of following six parts:

- Name
- Reset Initial Global Coordinate Layout
- Reset Base Work Plane Local Coordinate
- Translate / Rotate Work Plane
- Grid Dimensions and Divisions
- Command Buttons

First three work planes are prebuilt work planes:
(X : Y), (Z : Y) and (Z : X) planes. New work planes can be added by copying one of these prebuilt planes.

Figure 6.8
Work plane editor



6.3.1 Name

Name is work plane name you can specify for identification.

6.3.2 Reset Initial Global Coordinate Layout

This is used to reset initial global coordinate layout. You can select Front, Side, Plan or Isometric views. Once selected, click Update button to see the selected layout.

6.3.3 Reset Base Work Plane Local Coordinate

This is used to reset base work plane local coordinate.
You can select (x, y), (z, y), (z, x) or Manual.

For Manual, click Specify button to display Base Work Plane Local Coordinate dialog in Figure 6.9. Base work plane can be specified either by coordinates of three points or by three node numbers.

Once selected, click Update button to see the selected local coordinate.

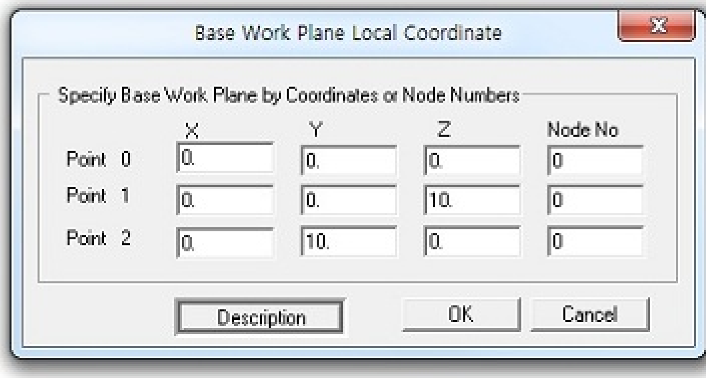


Figure 6.9 Base work plane local coordinate dialog

6.3.4 Translate / Rotate Work Plane

This is used to translate and rotate work plane.
When you rotate about more than one axis, select appropriate rotation order from the list box.

Click Draw New Origin button in Figure 6.8 to display Work Plane Origin dialog in Figure 6.10. This is a convenient way of moving the work plane origin.

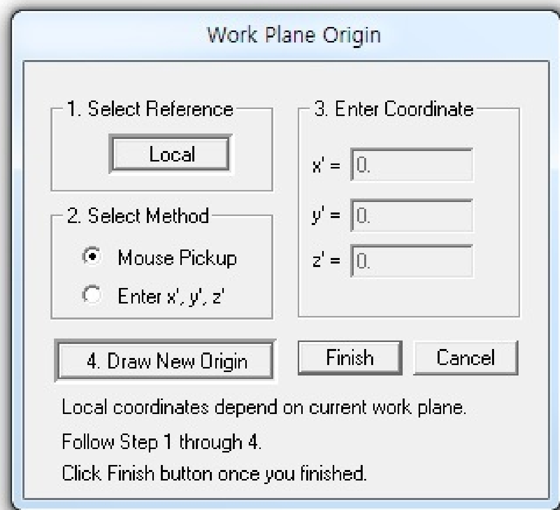


Figure 6.10 Work plane origin dialog

6.3.5 Grid Dimensions and Divisions

You can specify quadrant (NQ), grid divisions (NDx, NDy), and grid dimensions (Wx, Wy) as shown in work plane description in Figure 6.11.

Normally, you set the grid dimensions such that they include all blocks.

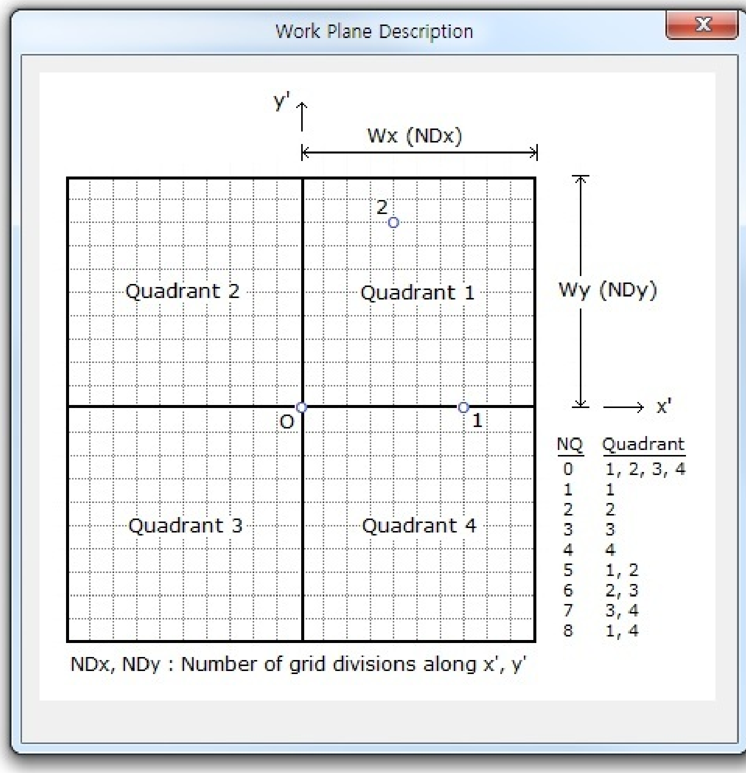


Figure 6.11 Work plane description

6.3.6 Command Buttons

Command buttons are shown on the bottom of Work Plane Editor dialog.

List

This is used to list all available work planes in Figure 6.12.
When you click OK button, selected work plane will be displayed as the current work plane.

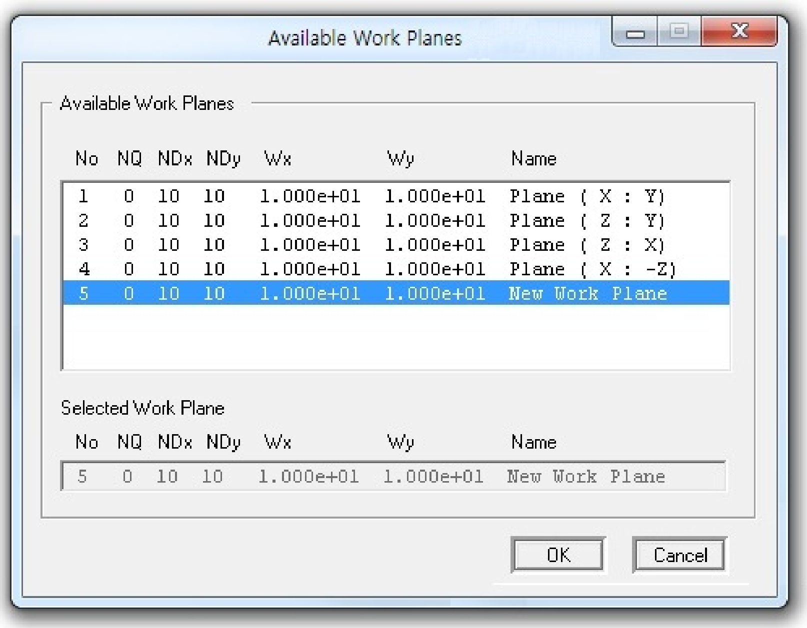


Figure 6.12 Work plane list

Hide Plane

This is used to hide the work plane and entities on the screen.

Description

This is used to show the description of work plane as shown in Figure 6.11.

Option
This is used to open work plane option dialog in Figure 6.13.
Click Update button on this dialog to see the changes made by selected options.

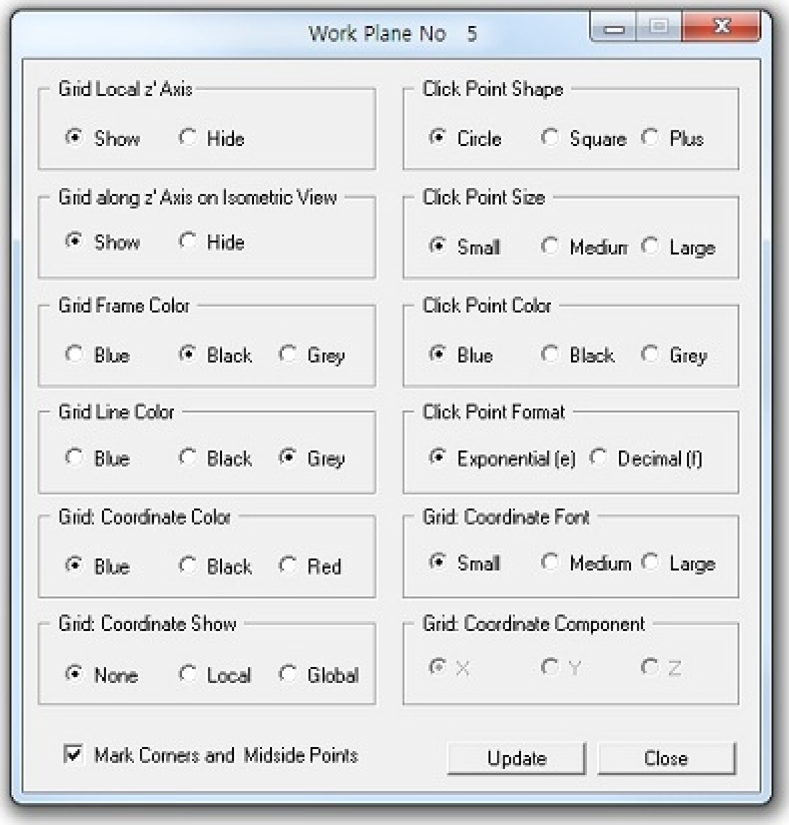


Figure 6.13 Work plane option dialog

Update
This is used to update the current work plane parameters shown on the Work Plane Editor dialog.

Entity

This is used to show Entity Editor dialog in Figure 6.17.

Entities are geometric objects under the current work plane, which are mainly used to assist editing the geometry of blocks and elements. Section 6.4 discusses entities in detail.

Add Plane

This is used to add new work plane.

New work plane is made by copying the work plane shown on the dialog. Once you edit work plane parameters, click Update button in the Work Plane Editor dialog to see the changes.

Delete Plane

This is used to delete the current work plane.

Exit

This is used to hide the work plane and exit from the dialog.

6.3.7 Prebuilt Work Planes

First three work planes are prebuilt work planes:
(x : y), (z : y) and (z : x) planes.

These Prebuilt Work Planes can be accessed by selecting the following menu items in PLOT-3D as shown in Figure 6.14:
Model → Work Plane

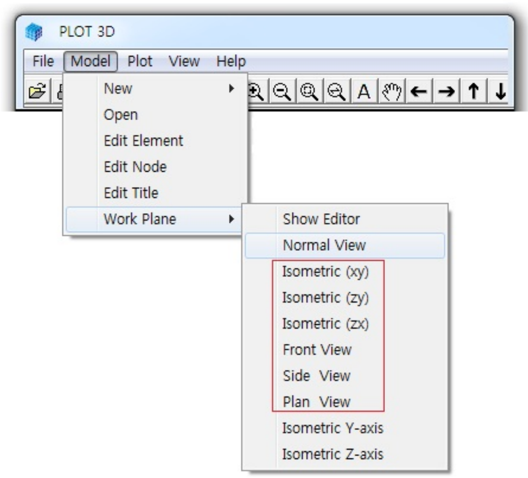


Figure 6.14 Prebuilt work plane menus

or by clicking Axis toolbar as shown in Figure 6.15.

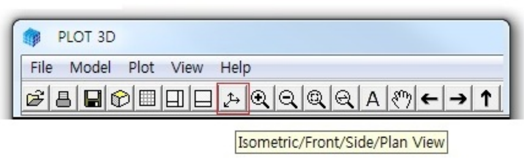


Figure 6.15 Axis toolbar

There are six different views associated with these prebuilt work planes as shown in Figure 6.16: Isometric (xy), Isometric (zy), Isometric (zx), Front, Side and Plan views.

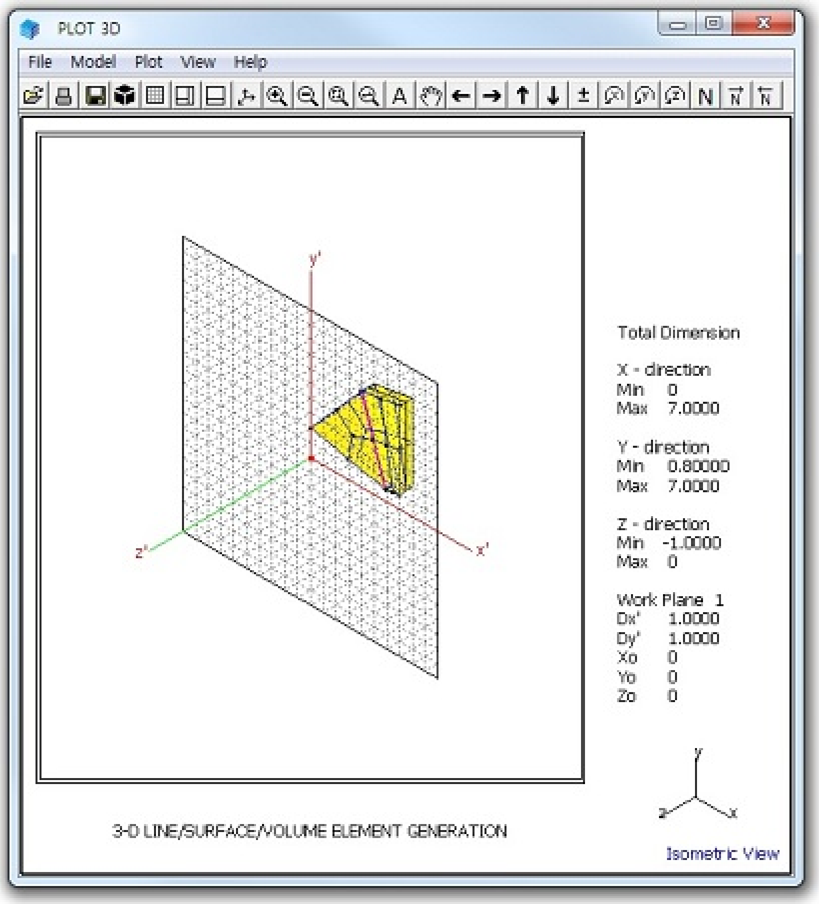


Figure 6.16 Prebuilt work plane: Isometric (xy) View

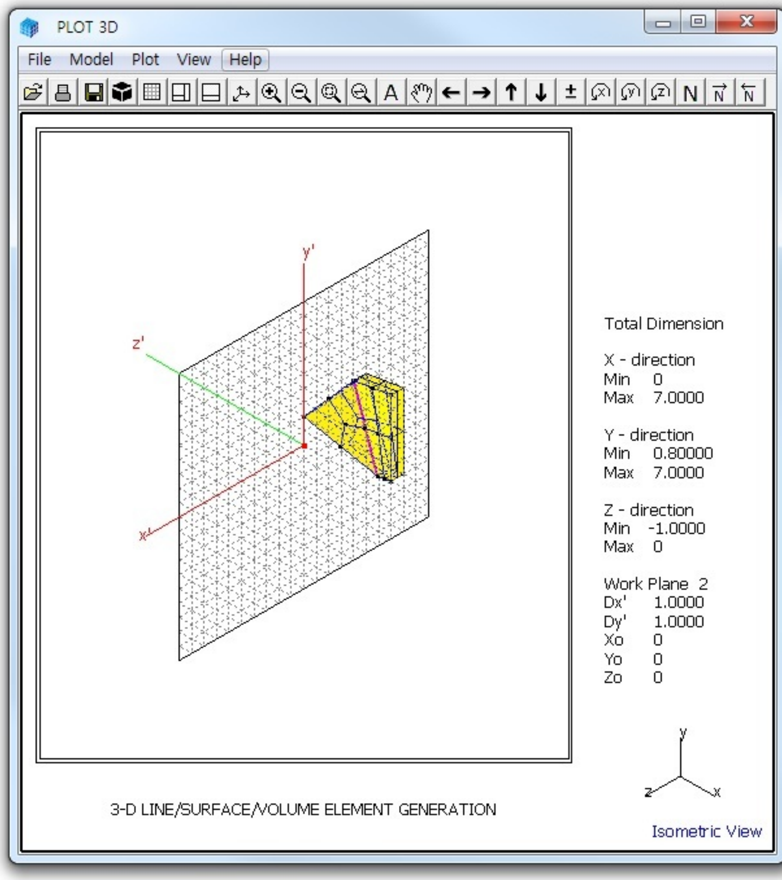


Figure 6.16 Prebuilt work plane: Isometric (zy) View

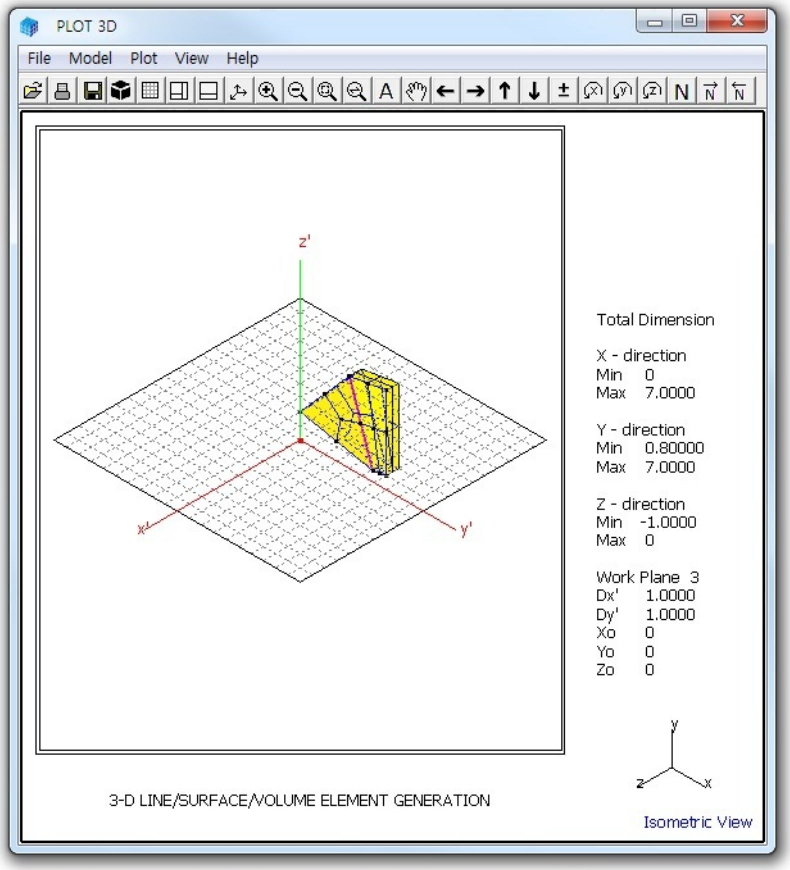


Figure 6.16 Prebuilt work plane: Isometric (zx) View

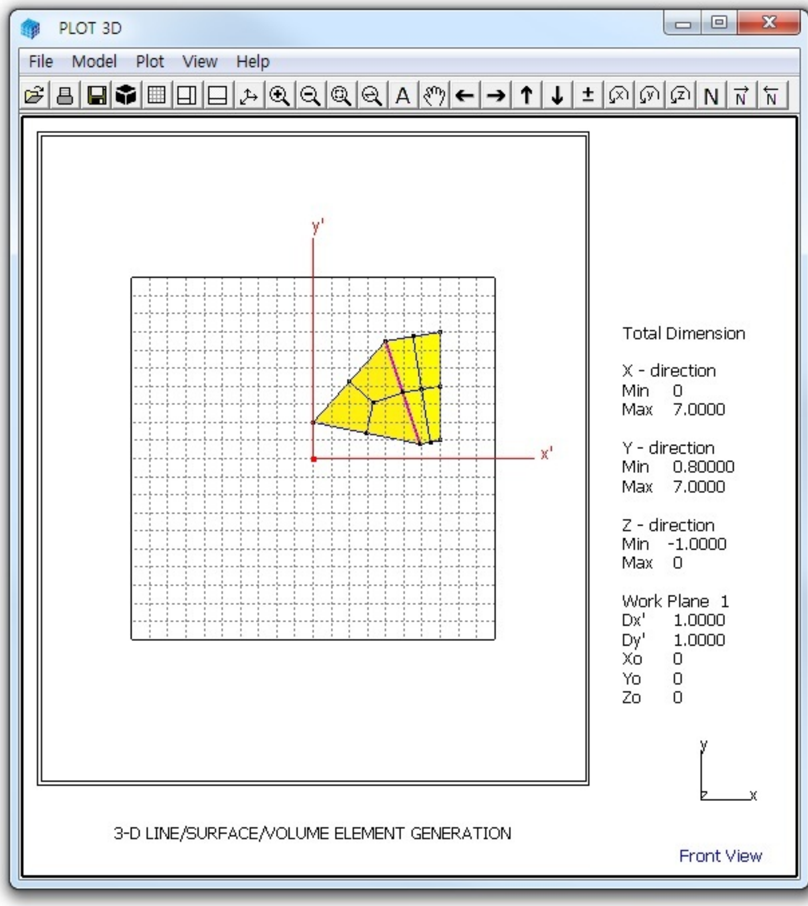


Figure 6.16 Prebuilt work plane: Front View

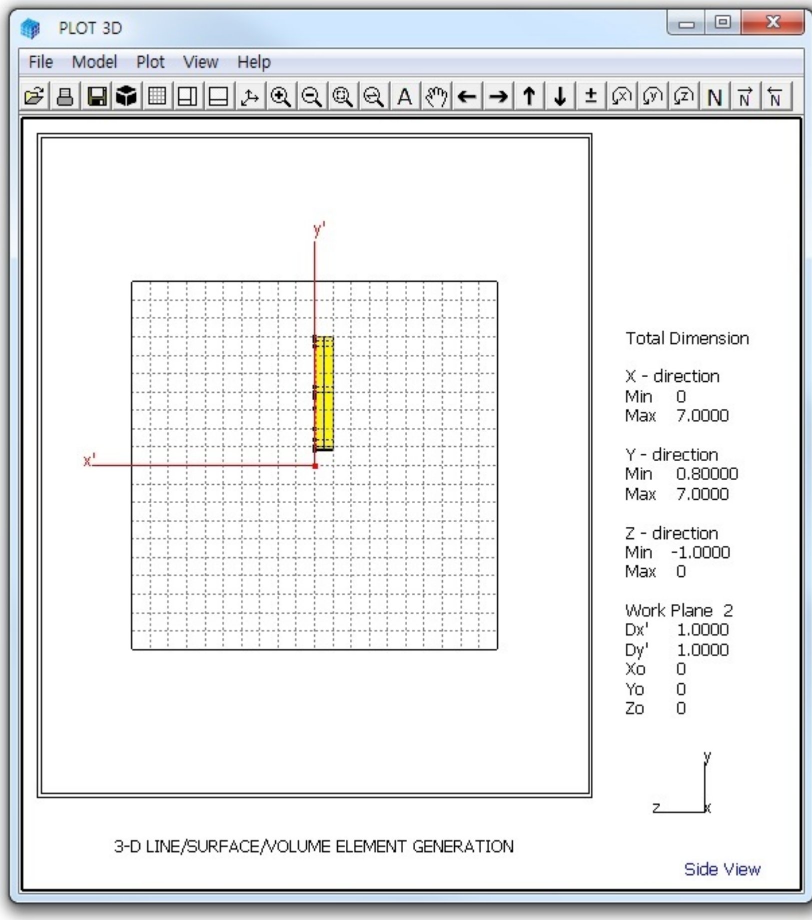


Figure 6.16 Prebuilt work plane: Side View

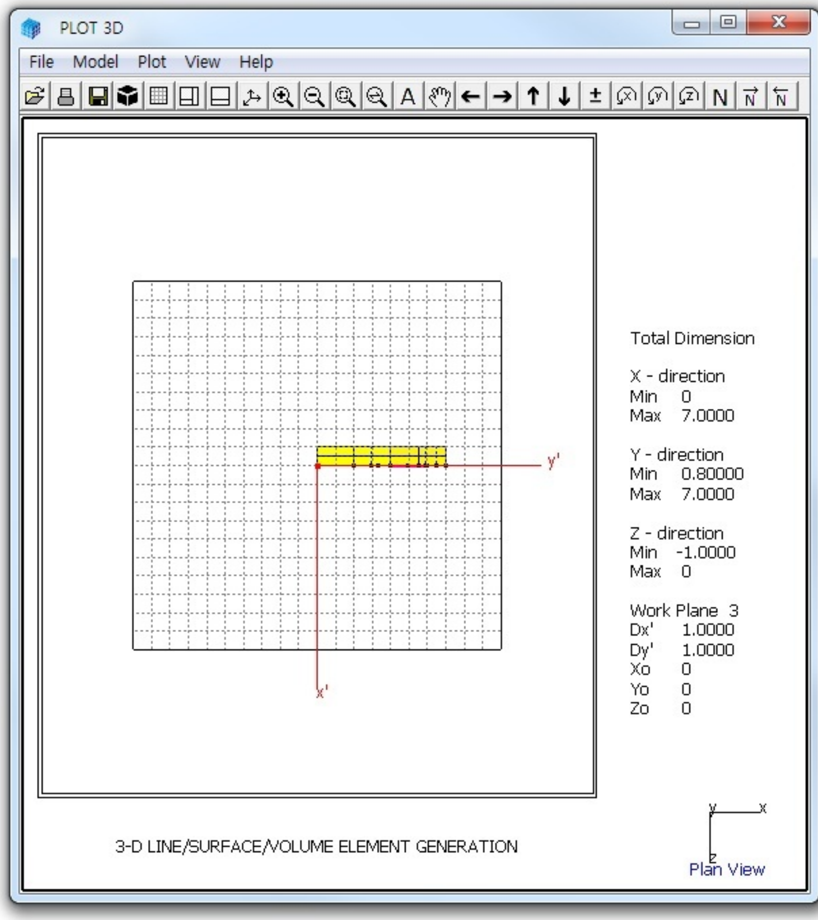


Figure 6.16 Prebuilt work plane: Plan View

6.4 Entities

Entities are geometric objects under the work plane, which are mainly used to assist editing geometry of blocks and elements.
There are five types of entities: Line, Arc, Cube, Ellipsoid, and Cylinder.

Entity Editor dialog in Figure 6.17 can be accessed by clicking Entity button on the Work Plane Editor dialog in Figure 6.8.

Entity Editor dialog consists of following seven parts:

- Entity Number
- Line Thickness
- Line Type
- Line Visibility
- Line Color
- Reference Coordinate
- Command Buttons

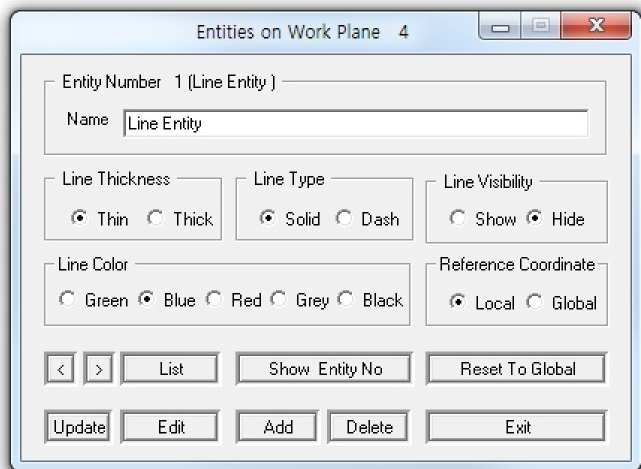


Figure 6.17 Entity editor dialog

6.4.1 Entity Number

Entity number and type are automatically displayed.
You can edit default entity name.

6.4.2 Line Thickness

Two options are available: Thin and Thick.

6.4.3 Line Type

Two options are available: Solid and Dash.

6.4.4 Line Visibility

Two options are available: Show and Hide.

6.4.5 Line Color

Five options are available: Green, Blue, Red, Grey, and Black.

6.4.6 Reference Coordinate

Two options are available: Local and Global.

6.4.7 Command Buttons

Command buttons are shown on Entity dialog in Figure 6.17.

List

This is used to list all available entities in current work plane.

When you click OK button, selected entity will be displayed as the current entity on the Entity Editor dialog.

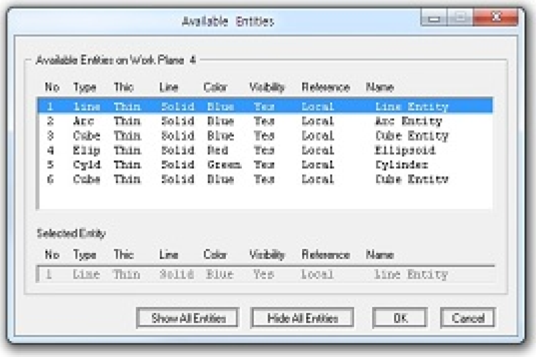


Figure 6.18
Entity list dialog

Show Entity No

This is used to show all entity numbers on the screen.

Reset To Global

This is used to reset the current entity global reference by the current local coordinate.

Update

This is used to update parameters of the current entity.

Edit

This is used to edit the geometry of the current entity.

Add

This is used to add new entity. Refer to Section 6.4.9

Delete

This is used to delete the current entity.

Exit

This is used to exit from the Entity Editor dialog.

6.4.8 Popup Menu for Entity

When Entity Editor dialog is opened, you can directly access an entity by Control + Right Click. Then the selected entity is displayed on the Entity Editor dialog along with Popup Menu as shown in Figure 6.19.

Popup Menu consists of eight submenus:
Edit, Copy, Add, Hide, Delete, List, Number and Exit.
These menus are essentially duplicates of command buttons on the Entity Editor dialog.

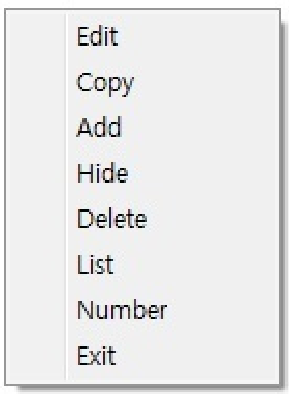


Figure 6.19 Popup menu for entity

6.4.9 Adding New Entity

To add a new entity, click Add button on Entity Editor dialog. Then Entity Type Selection dialog will be displayed as shown in Figure 6.20.

There are five types of entities:
Line, Arc, Cube, Ellipsoid and Cylinder. You can also select Copy Existing Entity and then type Entity No.

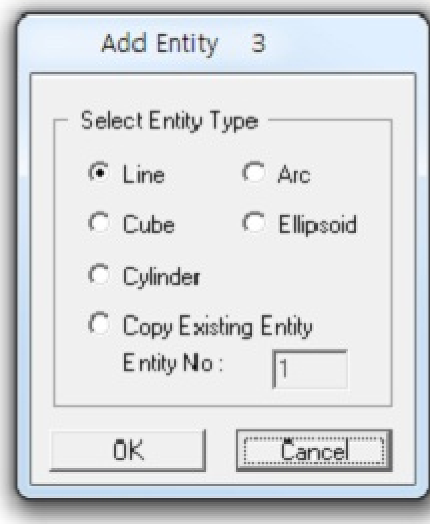


Figure 6.20 Entity type selection dialog

6.4.9.1 Line Entity

Line Entity dialog is shown in Figure 6.21.

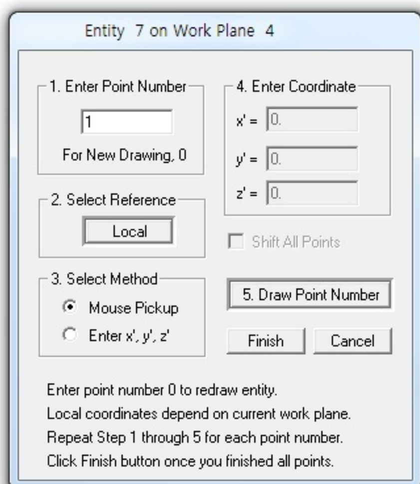
To draw Line Entity, follow five steps:

1. Enter Point Number
2. Select Reference
3. Select Method
4. Enter Coordinate
5. Draw Point Number

For Mouse Pickup method, when clicking Draw Point Number button at step 5, Coordinates on Work Plane dialog in Figure 6.22 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.23. Once finished, click Finish in Figure 6.22.

Finally, click Finish on Line Entity dialog in Figure 6.21.
Then you will be back to Entity Editor dialog where you can set the other parameters for the new entity.

Figure 6.21
Line entity dialog



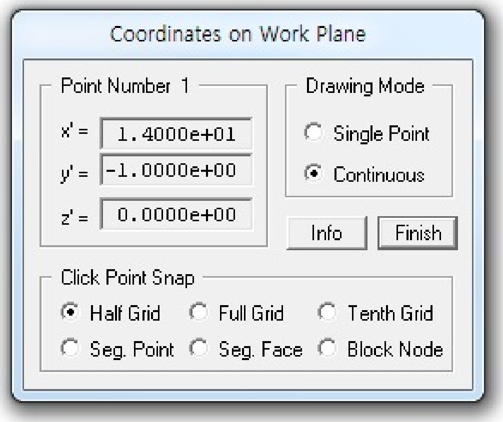


Figure 6.22 Coordinates on work plane

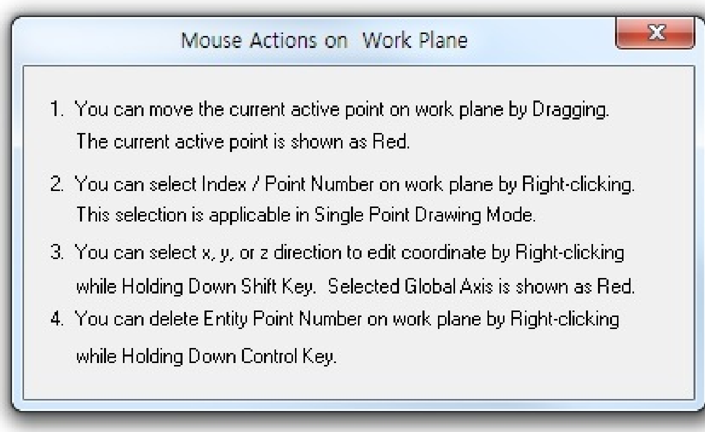


Figure 6.23 Mouse actions on work plane

6.4.9.2 Arc Entity

Arc Entity dialog is shown in Figure 6.24.

To draw Arc Entity, follow five steps:

1. Select Reference
2. Select Method
3. Enter Origin
4. Enter Dimensions
5. Draw Arc Entity

For Mouse Pickup method, when clicking Draw Arc Entity button at step 5, Coordinates on Work Plane dialog in Figure 6.22 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.23. Once finished, click Finish in Figure 6.22.

Finally, click Finish on Arc Entity dialog in Figure 6.24. Then you will be back to Entity Editor dialog where you can set the other parameters for the new entity.

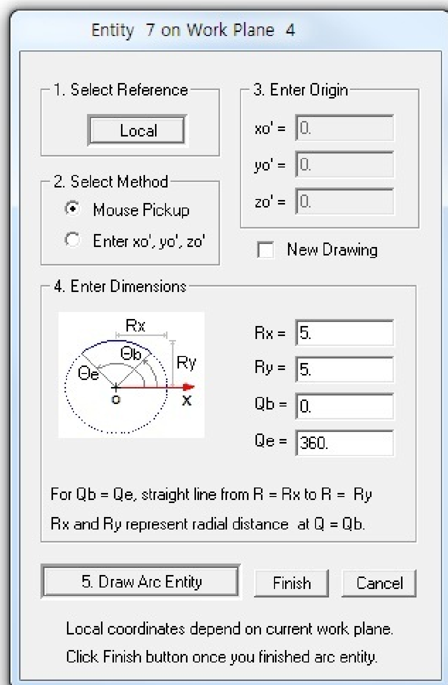


Figure 6.24
Arc entity dialog

6.4.9.3 Cube Entity

Cube Entity dialog is shown in Figure 6.25.

To draw Cube Entity, follow five steps:

1. Select Reference
2. Select Method
3. Enter Origin
4. Enter Dimensions
5. Draw Cube Entity

For Mouse Pickup method, when clicking Draw Cube Entity button at step 5, Coordinates on Work Plane dialog in Figure 6.22 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.23. Once finished, click Finish in Figure 6.22.

Finally, click Finish on Cube Entity dialog in Figure 6.25. Then you will be back to Entity Editor dialog where you can set the other parameters for the new entity.

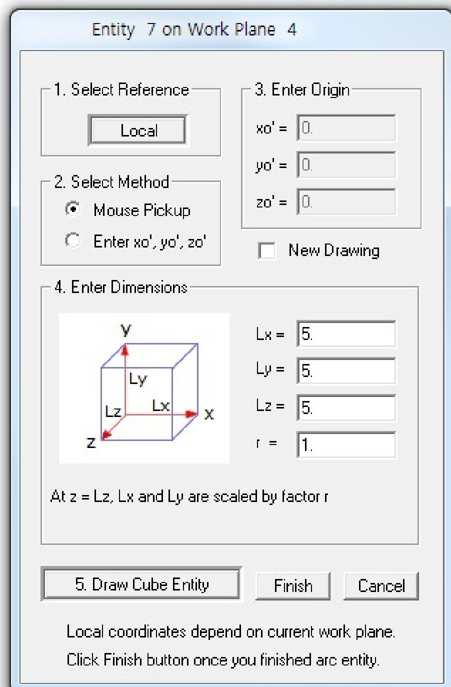


Figure 6.25
Cube entity dialog

6.4.9.4 Ellipsoid Entity

Ellipsoid Entity dialog is shown in Figure 6.26.

To draw Ellipsoid Entity, follow five steps:

1. Select Reference
2. Select Method
3. Enter Origin
4. Enter Dimensions
5. Draw Ellipsoid Entity

For Mouse Pickup method, when clicking Draw Ellipsoid Entity button at step 5, Coordinates on Work Plane dialog in Figure 6.22 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as in Figure 6.23. Once finished, click Finish in Figure 6.22.

Finally, click Finish on Ellipsoid Entity dialog in Figure 6.26. Then you will be back to Entity Editor dialog where you can set the other parameters for the new entity.

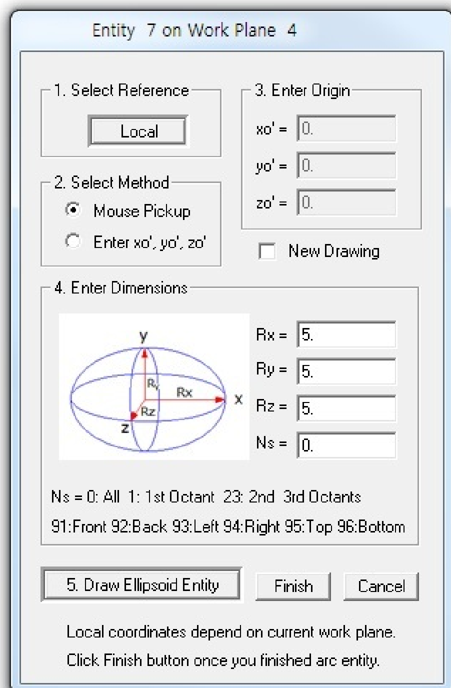


Figure 6.26
Ellipsoid entity dialog

6.4.9.5 Cylinder Entity

Cylinder Entity dialog is shown in Figure 6.27.

To draw Cylinder Entity, follow five steps:

1. Select Reference
2. Select Method
3. Enter Origin
4. Enter Dimensions
5. Draw Cylinder Entity

For Mouse Pickup method, when clicking Draw Cylinder Entity button at step 5, Coordinates on Work Plane dialog in Figure 6.22 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as in Figure 6.23. Once finished, click Finish in Figure 6.22.

Finally, click Finish on Cylinder Entity dialog in Figure 6.27. Then you will be back to Entity Editor dialog where you can set the other parameters for the new entity.

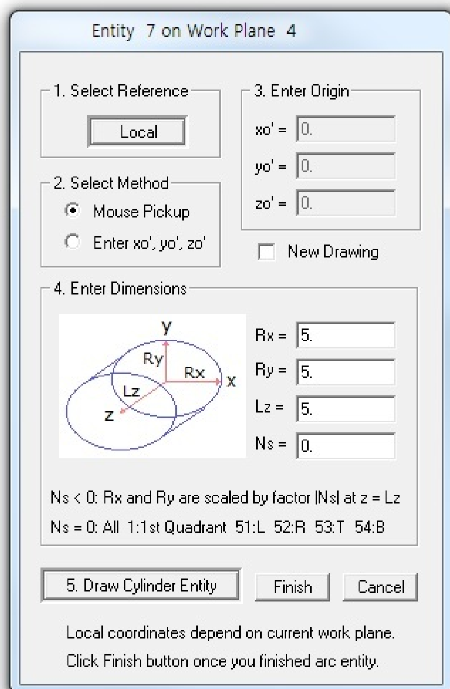


Figure 6.27
Cylinder entity dialog

6.5 Block

Blocks are groups of elements. Each block consist of the same type of finite elements.

Block Editor can be accessed by selecting the following menu items in PLOT-3D:
Model → Block Editor
or by clicking Block Editor toolbar as shown in Figure 6.28.

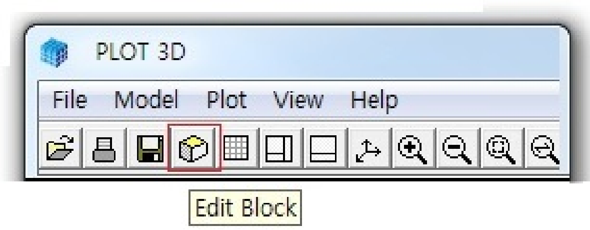


Figure 6.28 Block editor toolbar

Block Editor dialog in Figure 6.29 consists of following eight parts:

- Title
- Block Number
- Interpolation Coordinate System
- Coordinate Modification
- Interpolation Scheme / Element Type
- Reference Node Numbers
- Material and Element Generation Parameters
- Command Buttons

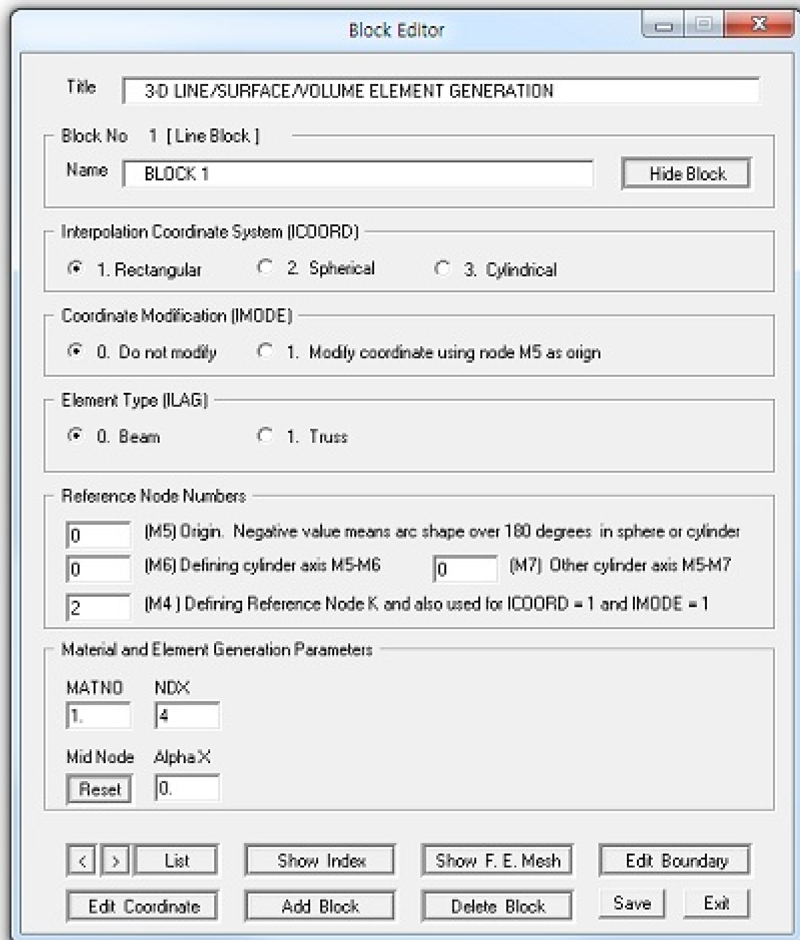


Figure 6.29 Block editor (Line Block)

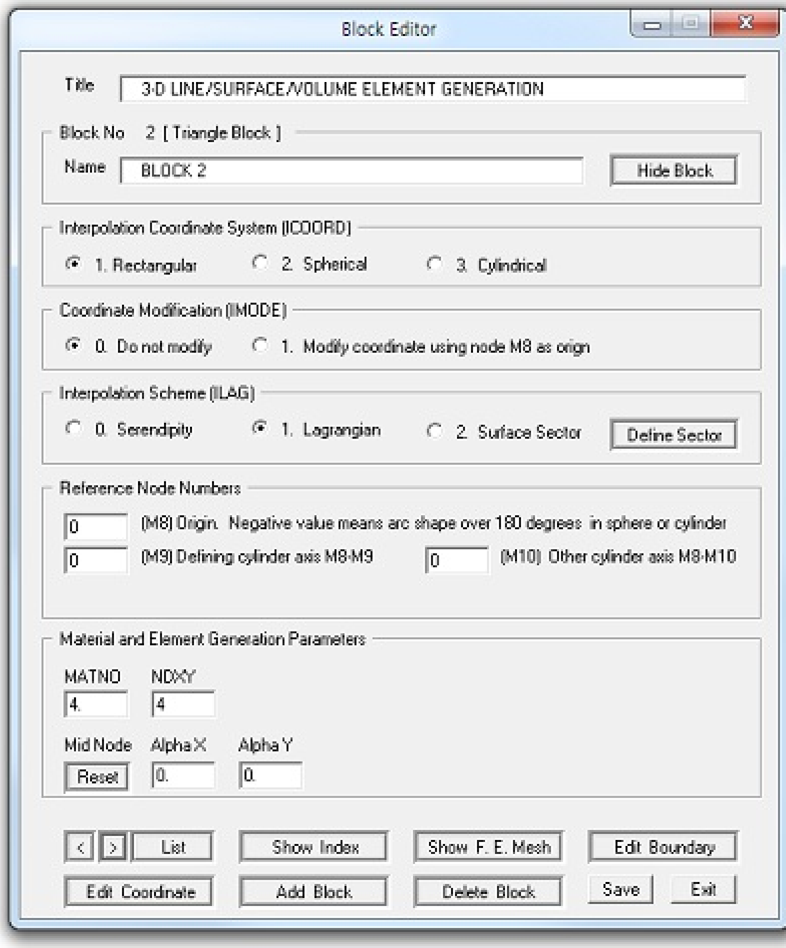


Figure 6.29 Block editor (Triangle Block)

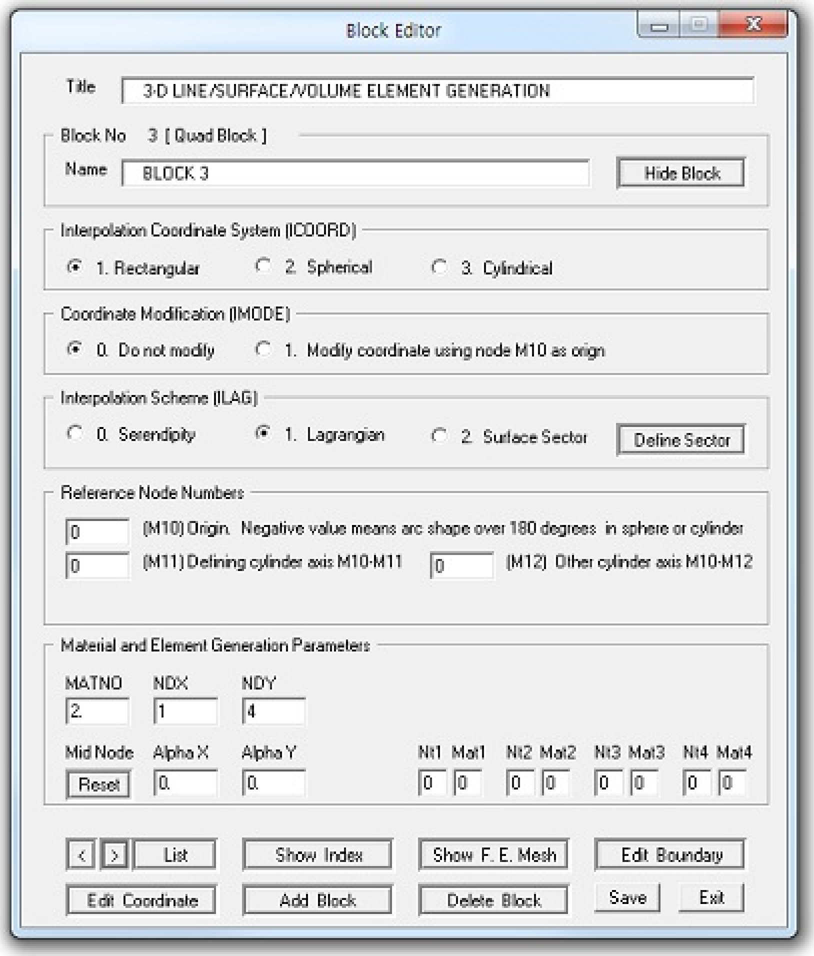


Figure 6.29 Block editor (Quad Block)

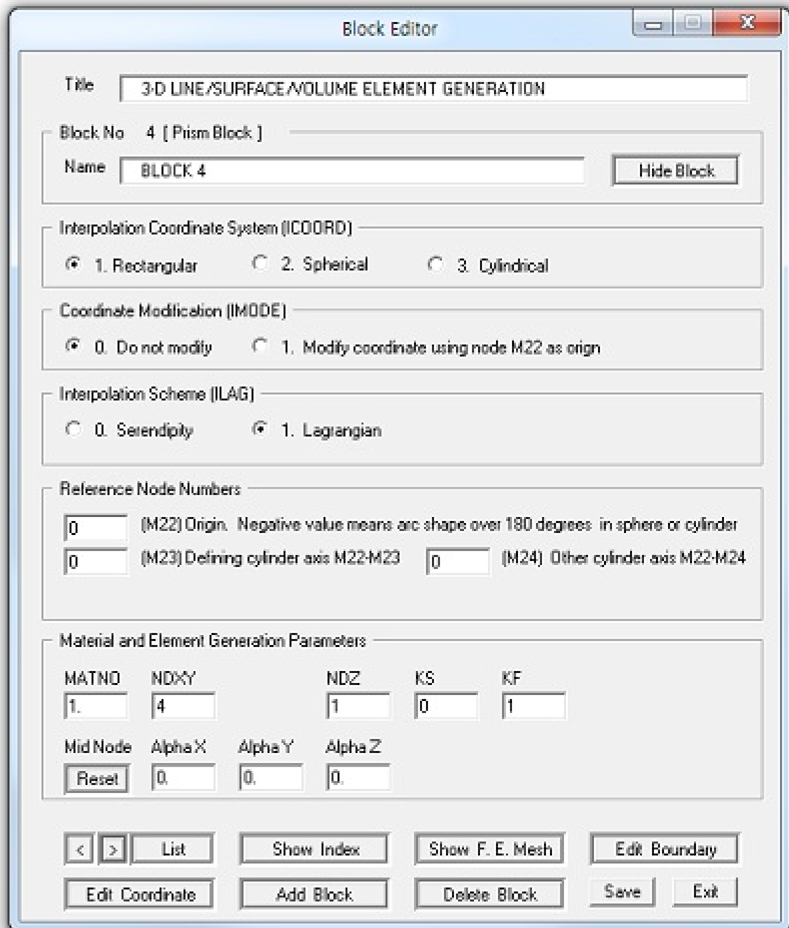


Figure 6.29 Block editor (Prism Block)

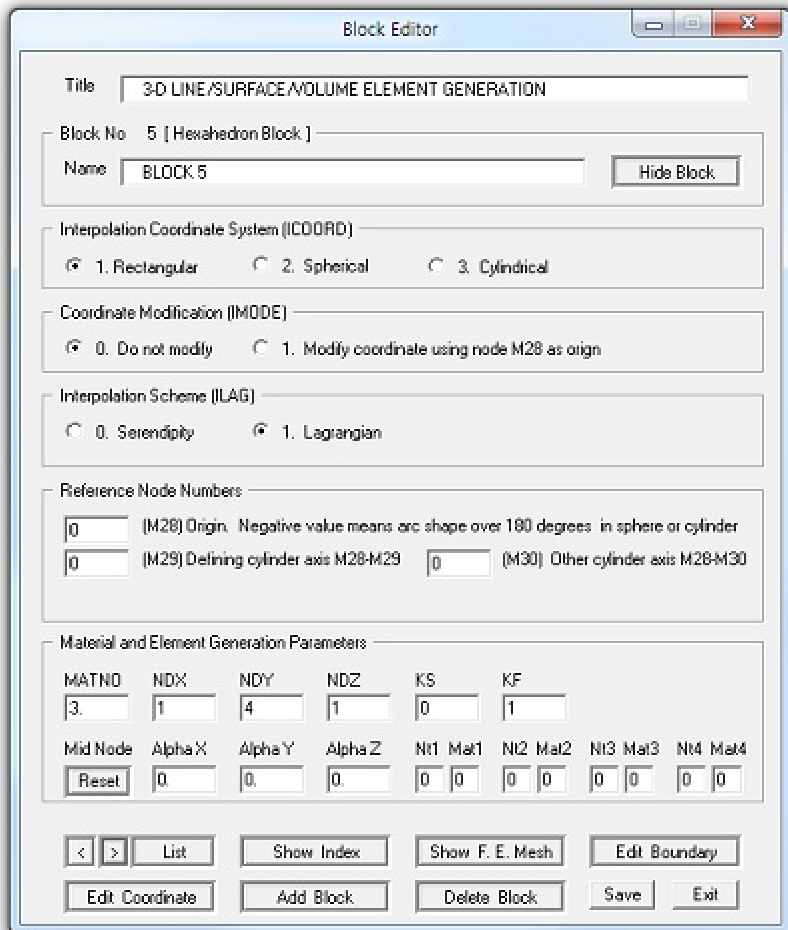


Figure 6.29 Block editor (Hexahedron Block)

6.5.1 Title

This is the title for the block mesh file.

6.5.2 Block Number

Block number and type are automatically displayed as the label of the frame. You can specify block name for identification.

Hide Block button is to hide the current block on the screen.

6.5.3 Interpolation Coordinate System

This is to select the coordinate system for interpolation. Three options are available: Rectangular, Spherical and Cylindrical.

6.5.4 Coordinate Modification

This is to modify generated coordinates based on the reference node as origin.

6.5.5 Interpolation Scheme / Element Type

For line blocks, two options are available for the type of line element: Beam and Truss.

For surface blocks, three options are available: Serendipity, Lagrangian and Surface Sector.

For volume blocks, two options are available: Serendipity and Lagrangian.

When you click Define Sector button, Surface Sector dialog is displayed to edit input parameters as shown in Figure 6.30.

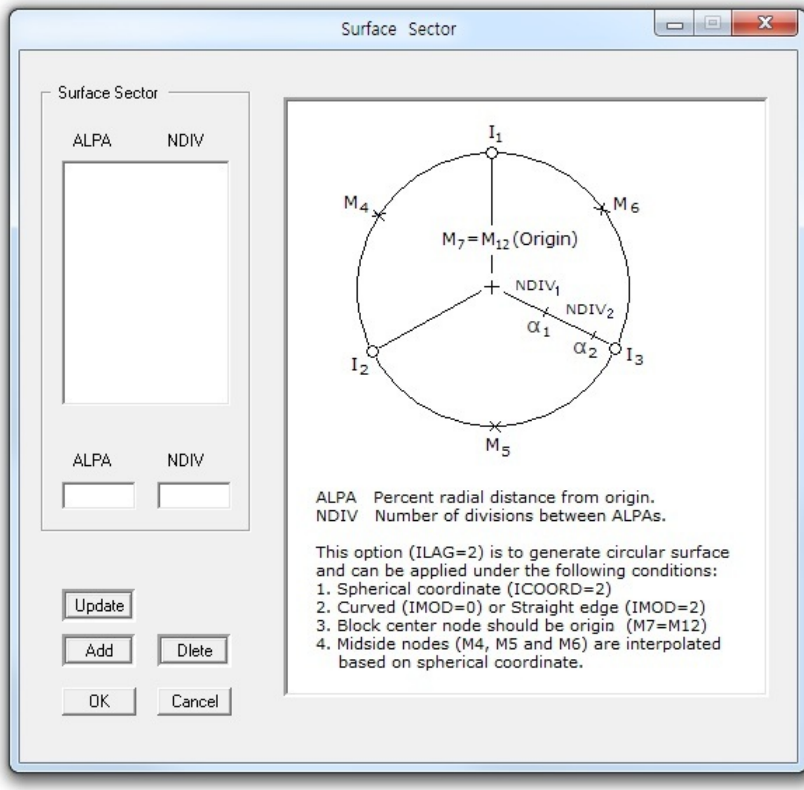


Figure 6.30 Surface sector (Triangle Block)

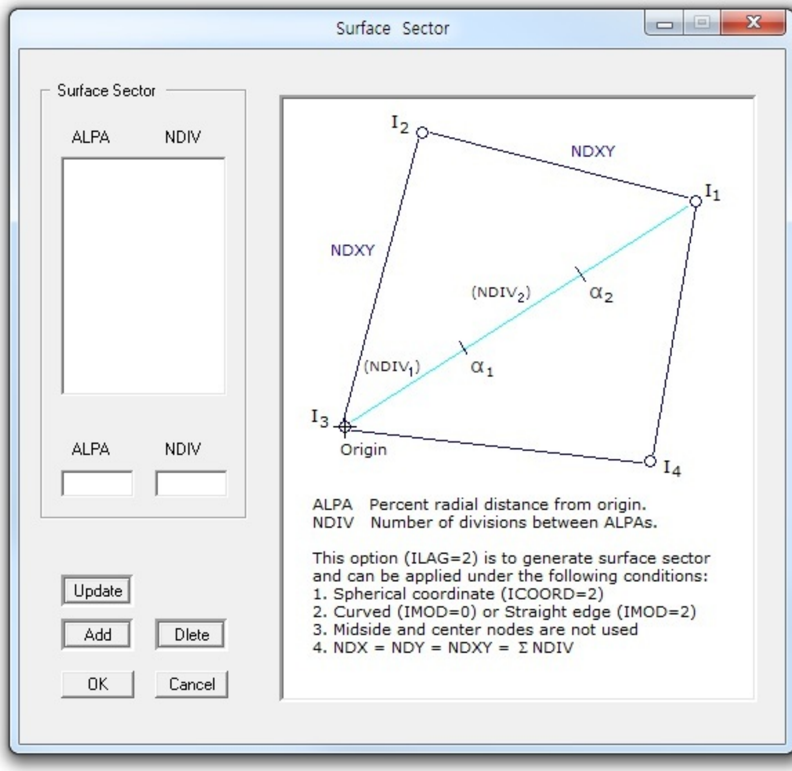


Figure 6.30 Surface sector (Quad Block)

6.5.6 Reference Node Numbers

This is to specify reference node numbers which are associated with block type.

6.5.7 Material & Element Generation Parameters

This is to specify material number and element generation parameters for the block.

6.5.8 Command Buttons

Command buttons are shown on the bottom of Block Editor dialog in Figure 6.29.

List
This is used to list all available blocks in the current block mesh as shown in Figure 6.31.

When you click OK button, selected block will be displayed as the current block on the Block Editor dialog.

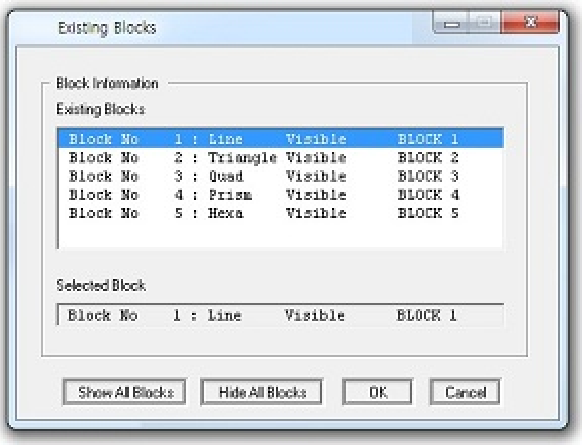


Figure 6.31 Block list

Show Index
This is used to show block index numbers.

Show F. E. Mesh
This is used to execute block mesh and then plot the generated finite element mesh.

Edit Boundary

This is used to edit boundary conditions shown in Figure 6.32.

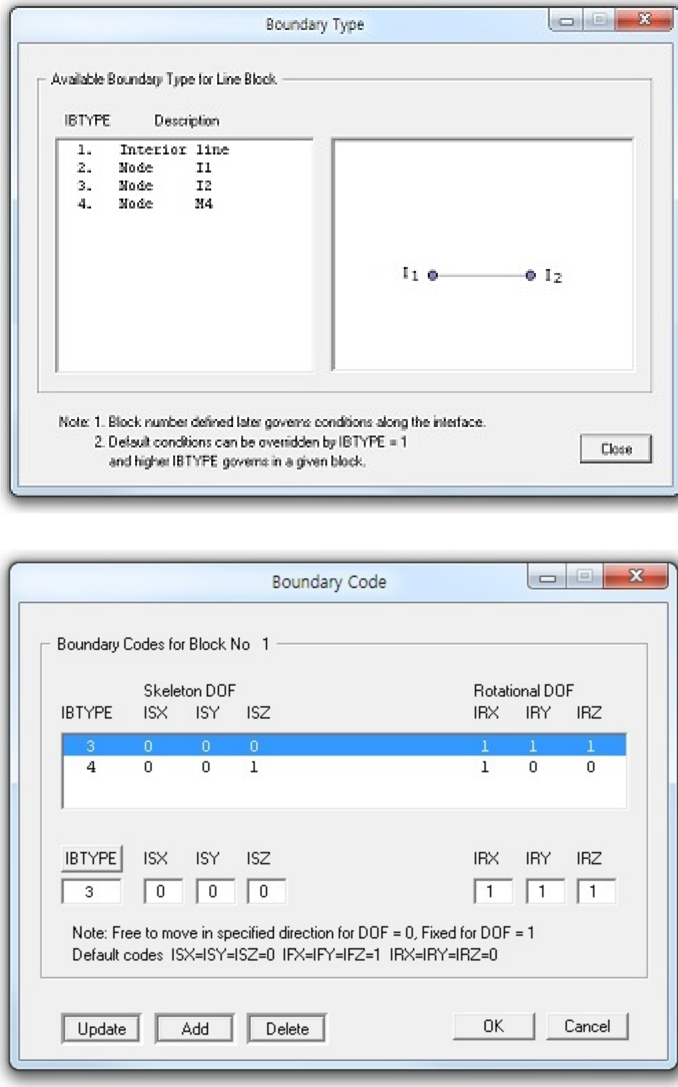
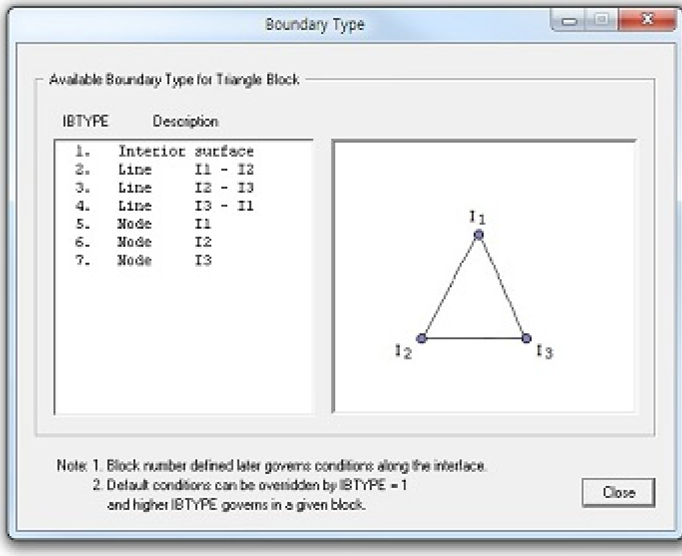


Figure 6.32 Boundary code (Line Block)



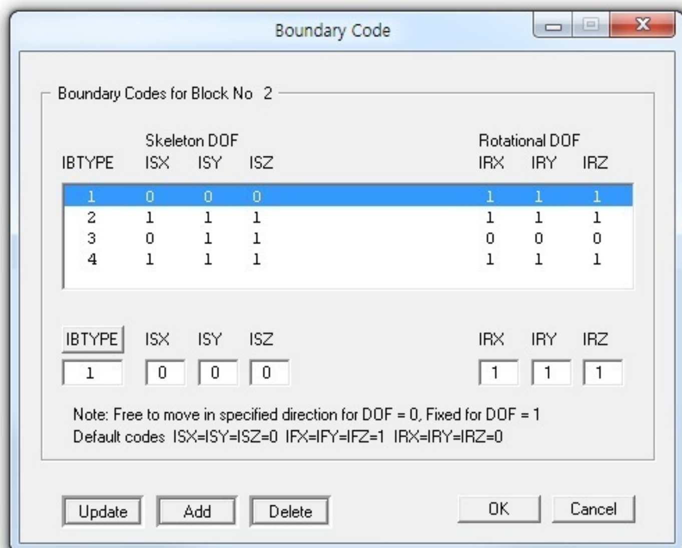


Figure 6.32 Boundary code (Triangle Block)

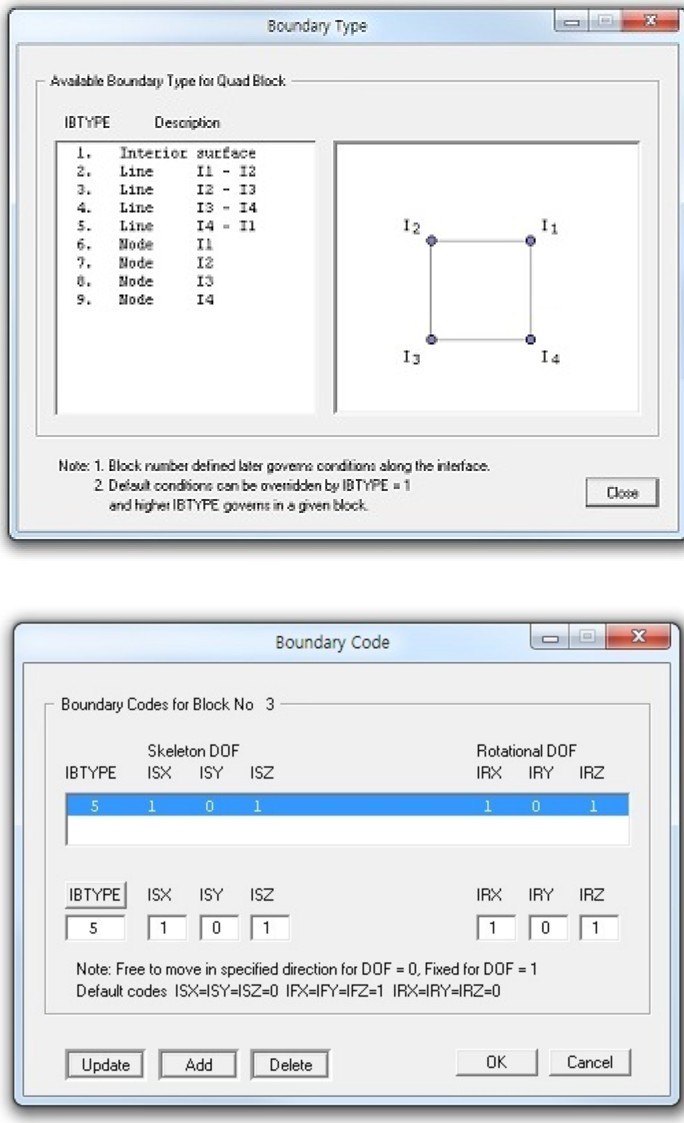


Figure 6.32 Boundary code (Quad Block)

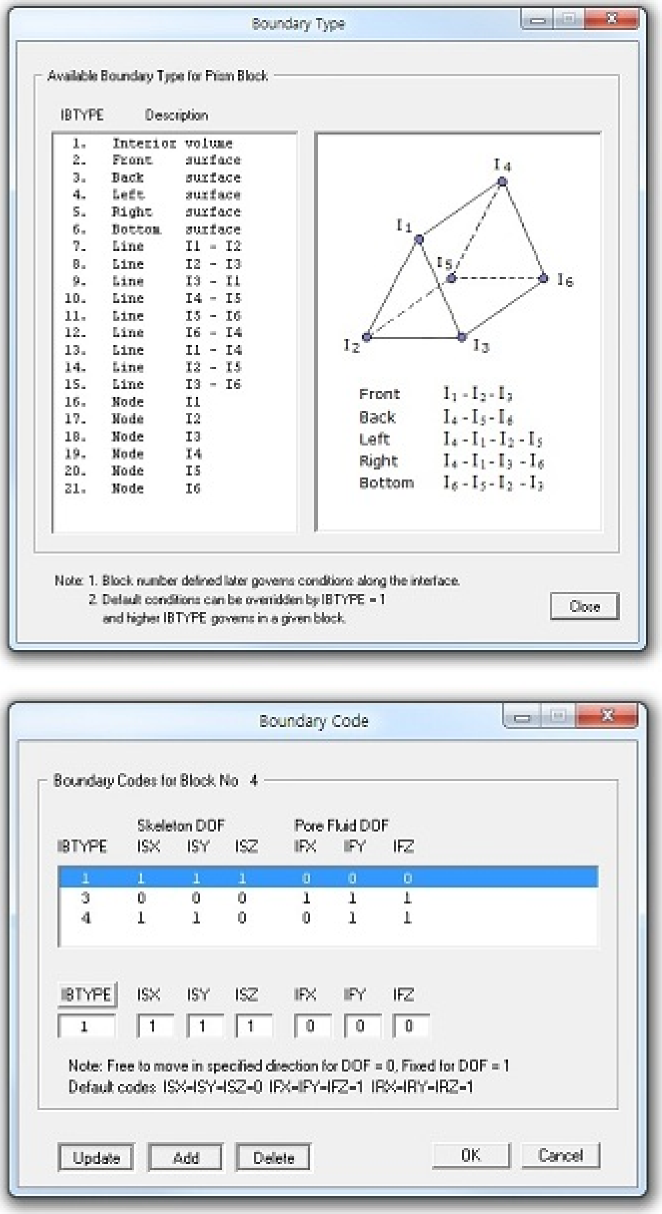


Figure 6.32 Boundary code (Prism Block)

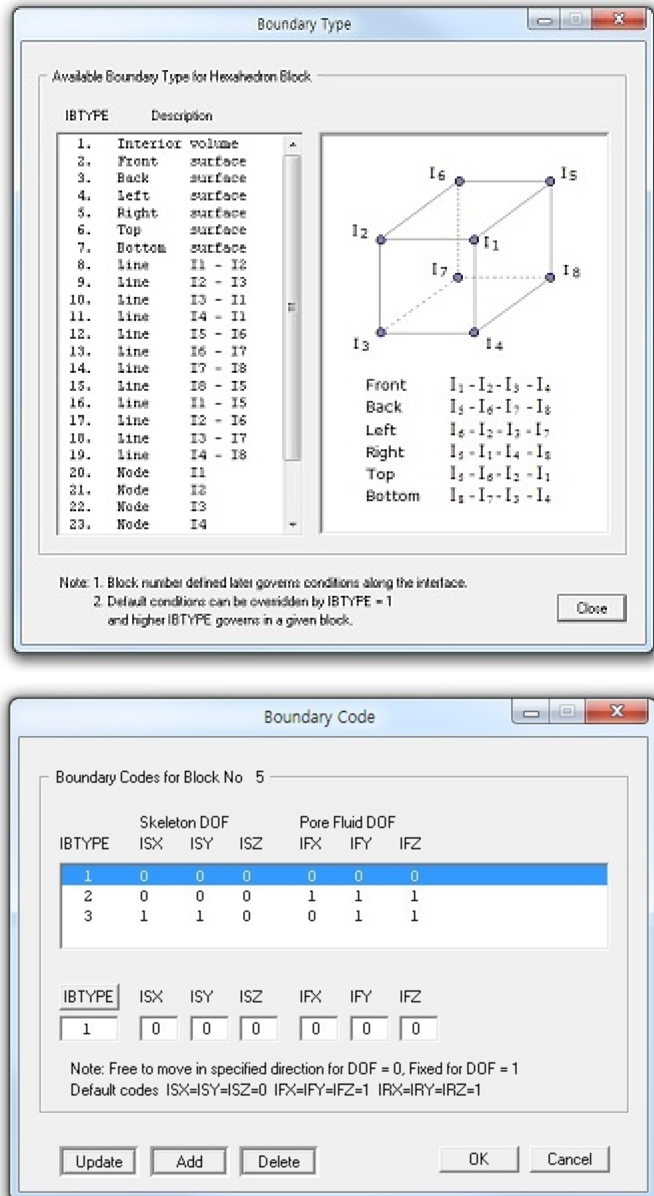


Figure 6.32 Boundary code (Hexahedron Block)

Edit Coordinate

This is used to edit the geometry of the block.
Before editing, work plane should be displayed on the screen.

Type Block No on Edit Current Block dialog in Figure 6.33
and then click OK button.

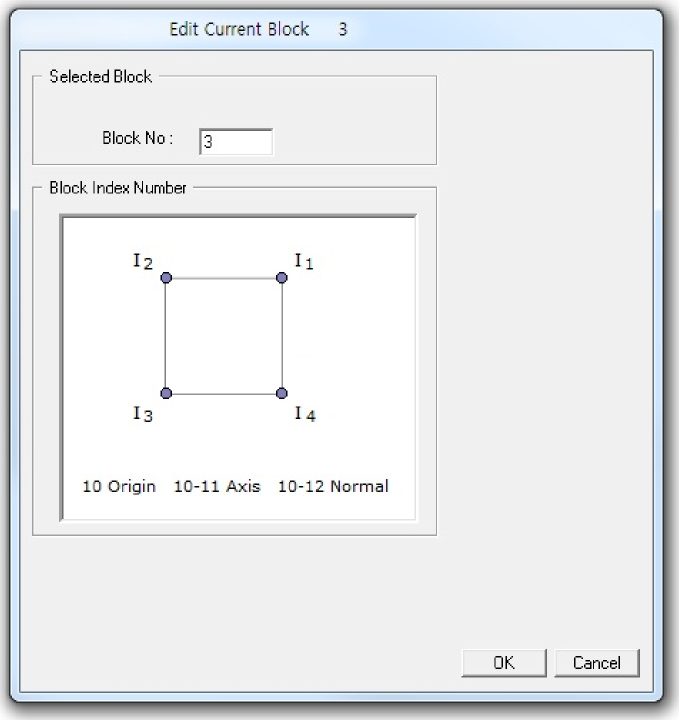


Figure 6.33 Edit current block (Selection Mode)

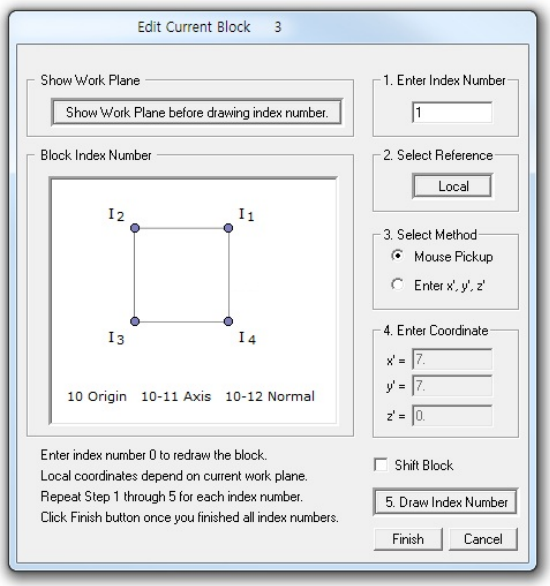
Edit Current Block dialog now shows input parameters required
to edit the geometry of the block as shown in Figure 6.34.

To edit block, follow five steps:
1. Enter Index Number
2. Select Reference
3. Select Method
4. Enter Coordinate
5. Draw Index Number

For Mouse Pickup method, when clicking Draw Index Number button at step 5, Coordinates on Work Plane dialog in Figure 6.35 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.36.   Once finished, click Finish in Figure 6.35.

Finally, click Finish on Edit Current Block dialog in Figure 6.34. Then you will be back to Block Editor dialog where you can set the other parameters for the current block.

Figure 6.34
Edit current block
(Edit Mode)



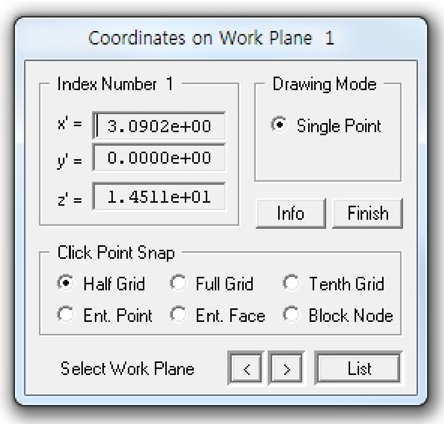


Figure 6.35 Coordinates on work plane

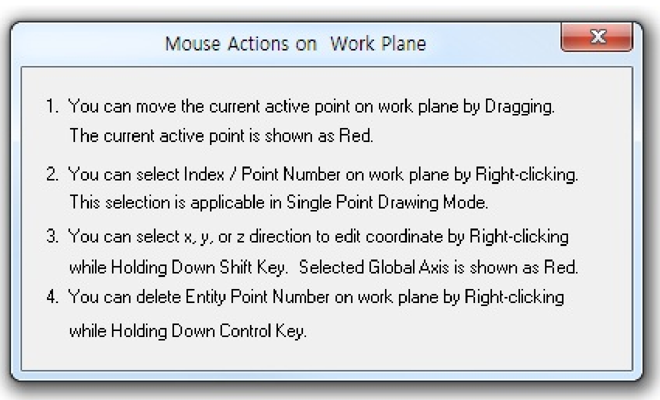


Figure 6.36 Mouse actions on work plane

Add Block

This is used to add the geometry of the new block.
Before building, work plane should be displayed on the screen.

Build New Block dialog in Figure 6.37 will be displayed.
Select Block Type, Interpolation Coordinate System
and then click OK button.

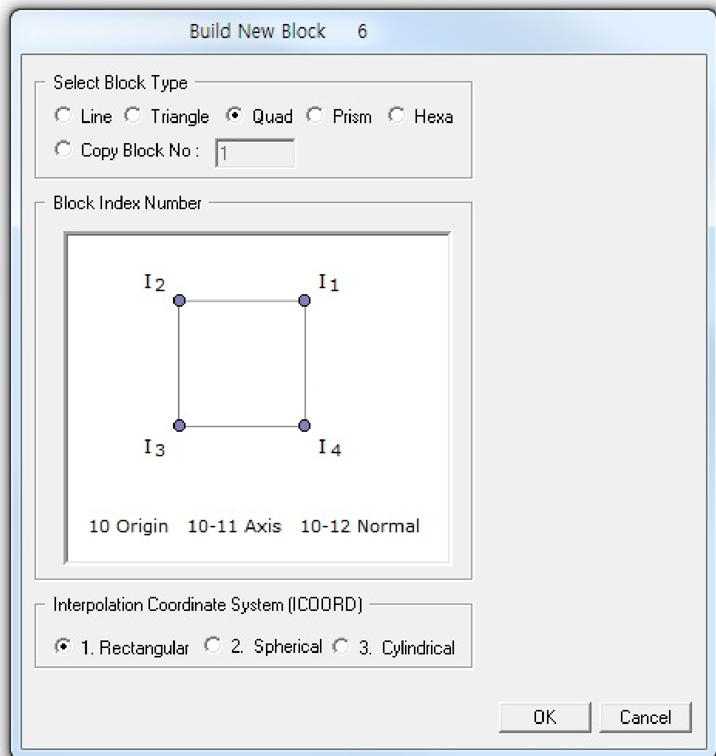


Figure 6.37 Build new block (Selection Mode)

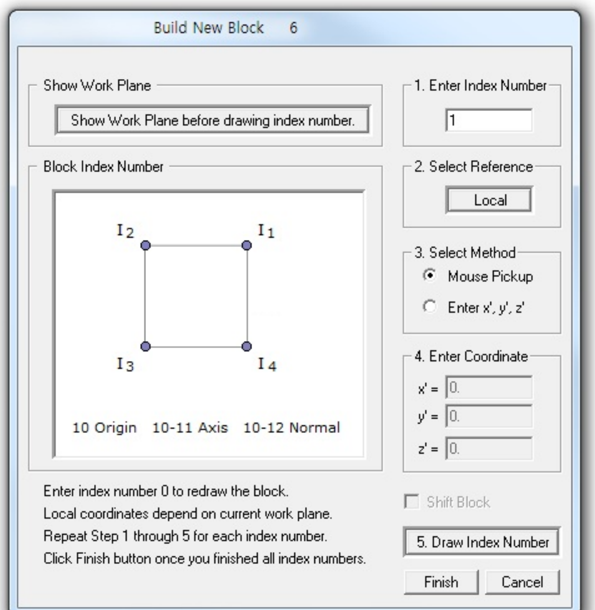
Build New Block dialog now shows input parameters required to build the geometry of new block as shown in Figure 6.38.

To build new block, follow five steps:
1. Enter Index Number
2. Select Reference
3. Select Method
4. Enter Coordinate
5. Draw Index Number

For Mouse Pickup method, when clicking Draw Index Number button at step 5, Coordinates on Work Plane dialog in Figure 6.39 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.36. Once finished, click Finish in Figure 6.39.

Finally, click Finish on Build New Block dialog in Figure 6.38. Then you will be back to Block Editor dialog where you can set the other parameters for the new block.

Figure 6.38
Build new block
(Build Mode)



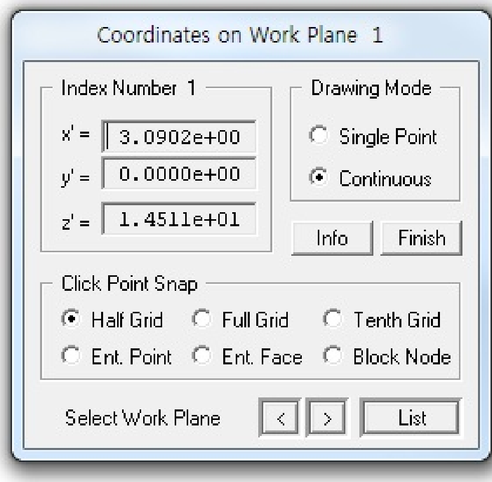


Figure 6.39 Coordinates on work plane

Delete Block

This is used to delete the current block.

Save

This is used to save all the works you have done.

Exit

This is used to exit from the block editor.

6.5.9 Popup Menu for Block

When Block Editor dialog is opened, you can directly access a block by Shift + Right Click. Then the selected block is displayed on the Block Editor dialog along with Popup Menu as shown in Figure 6.40.

Popup Menu consists of eleven submenus:
Edit, Copy, Add, Hide, Delete, List, Index, Boundary, F.E. Mesh, Save and Exit. These menus are essentially duplicates of command buttons on the Block Editor dialog.

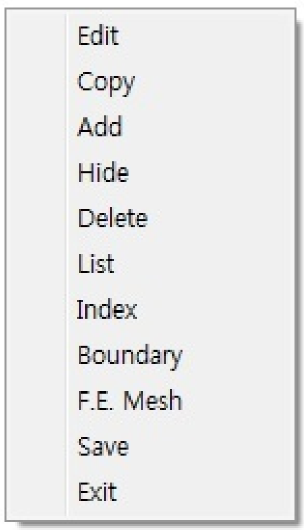


Figure 6.40 Popup menu for block

6.6 Modifying Finite Element Meshes

Block Mesh Generator can be used to directly modify finite element mesh.

When you open input file, Mesh Generator reads the format of the input file and automatically identifies whether it is block mesh file or finite element mesh file.

Editing finite element mesh has three parts: Edit Element, Edit Node and Edit Title. These editing modes can be accessed from Model menu in PLOT-3D as shown in Figure 6.41.

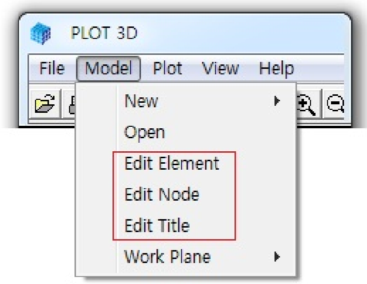


Figure 6.41 Menu for editing finite element mesh

You can check the current editing mode by moving the mouse on Editing Mode toolbar as shown in Figure 6.42.

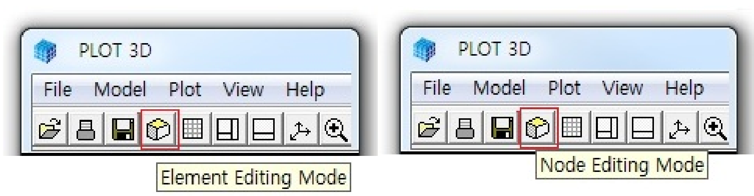
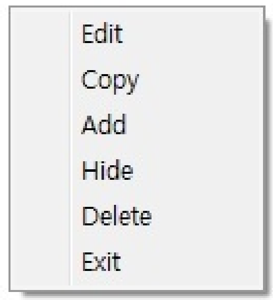


Figure 6.42 Toolbar for editing finite element mesh

6.6.1 Edit Element

When you are in Element Editing Mode, you can access popup menu for element in Figure 6.43 by Shift + Right Click.

Figure 6.43 Popup menu for element



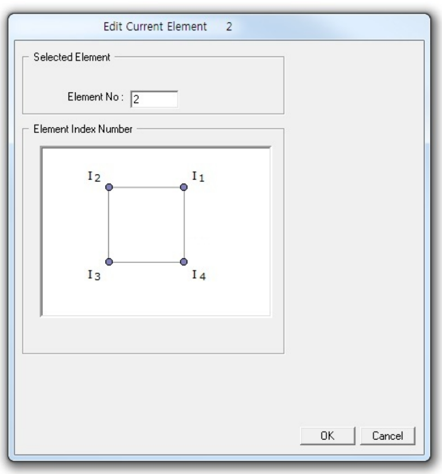
Element popup menu consists of six submenus:
Edit, Copy, Add, Hide, Delete and Exit.

Edit

This is used to edit the geometry of element.
Before editing, work plane should be displayed on the screen.

Edit Current Element dialog is displayed in Figure 6.44.
Type Element No and click OK button.

Figure 6.44
Edit current element
(Selection Mode)



Edit Current Element dialog now shows input parameters required to edit the geometry of element as shown in Figure 6.45.

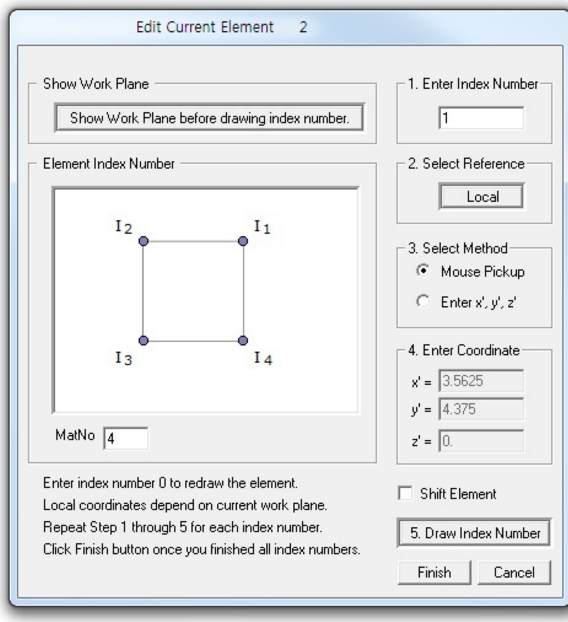
To edit element, follow five steps:

1. Enter Index Number
2. Select Reference
3. Select Method
4. Enter Coordinate
5. Draw Index Number

For Mouse Pickup method, when clicking Draw Index Number button at step 5, Coordinates on Work Plane dialog in Figure 6.46 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.47. Once finished, click Finish in Figure 6.46.

Finally, click Finish on Edit Current Element dialog in Figure 6.45.

Figure 6.45
Edit current element
(Edit Mode)



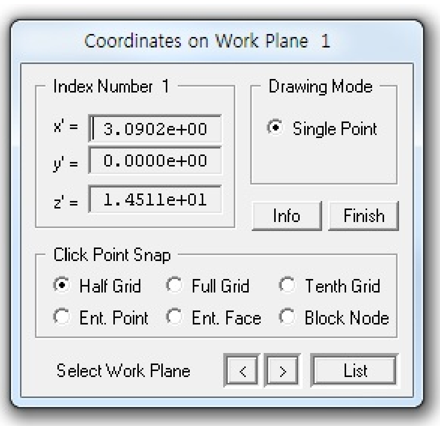


Figure 6.46 Coordinates on work plane (Edit Mode)

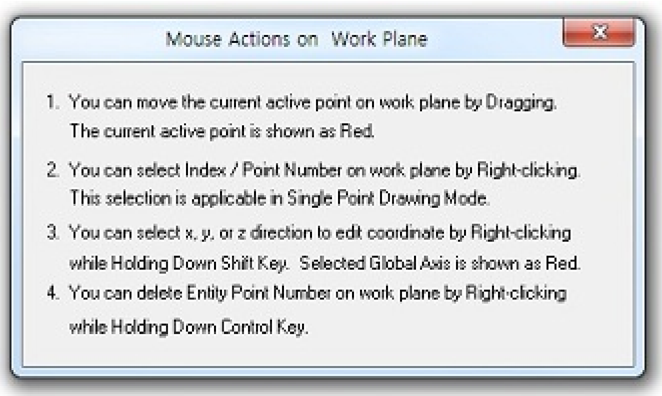


Figure 6.47 Mouse actions on work plane

Copy

This is used to copy the selected element and paste it as new element.

Edit Current Element dialog with new element number is displayed as shown in Figure 6.48. Shift Element check box should be checked to move this new element.

Follow the same procedure as in Edit.

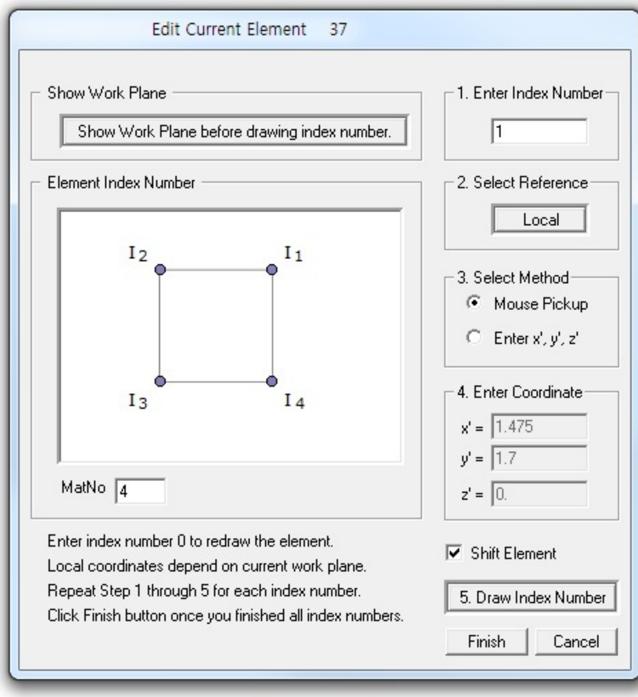


Figure 6.48 Edit current element (Copy Mode)

Add

This is used to add the geometry of the new element.
Before building, work plane should be displayed on the screen.

Build New Element dialog in Figure 6.49 will be displayed.
Select Element Type and then click OK button.

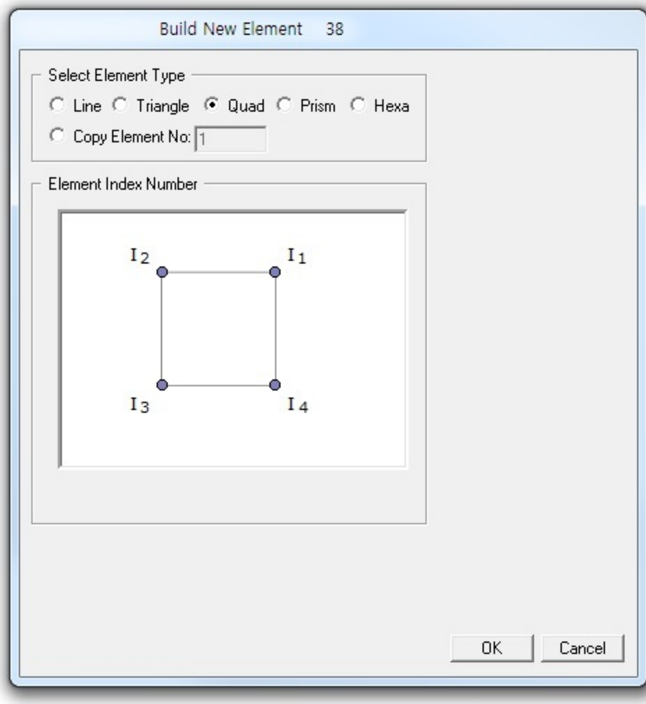


Figure 6.49 Build new element (Selection Mode)

Build New Element dialog now shows input parameters required to build the geometry of new element as shown in Figure 6.50.

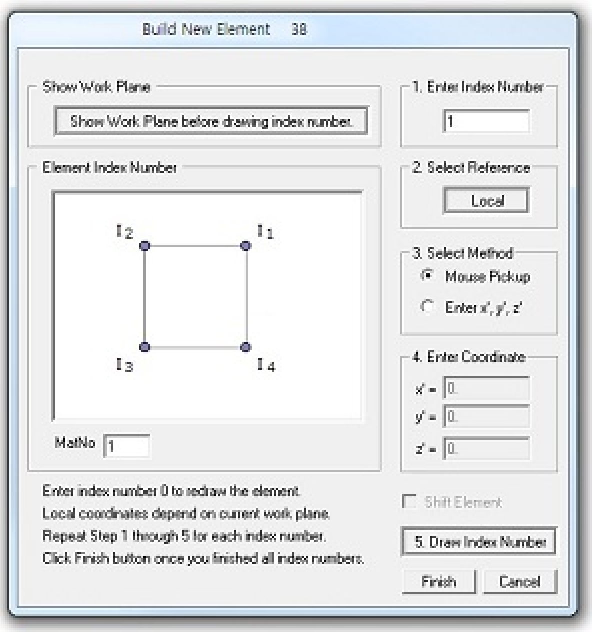
To build new element, follow five steps:

1. Enter Index Number
2. Select Reference
3. Select Method
4. Enter Coordinate
5. Draw Index Number

For Mouse Pickup method, when clicking Draw Index Number button at step 5, Coordinates on Work Plane dialog in Figure 6.51 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.47. Once finished, click Finish in Figure 6.51.

Finally, click Finish on Build New Element dialog in Figure 6.50.

Figure 6.50
Build new element
(Edit Mode)



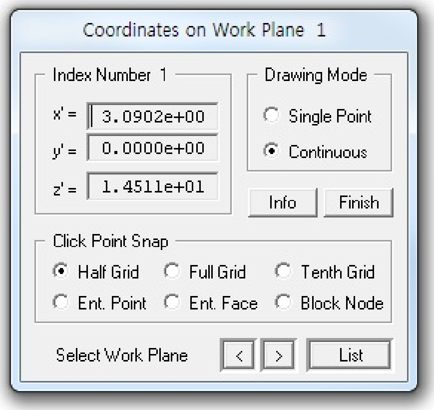


Figure 6.51 Coordinates on work plane (Add Mode)

Hide

This is used to hide the selected element from the screen.
To show the hidden element, follow instructions in Figure 6.52.

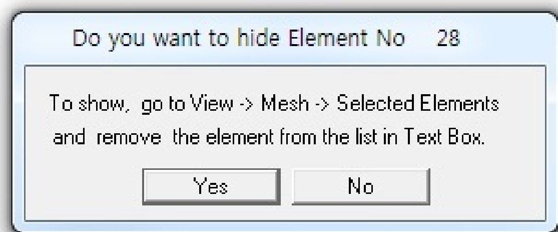


Figure 6.52 Instructions to show the hidden element

Delete

This is used to delete the selected element.

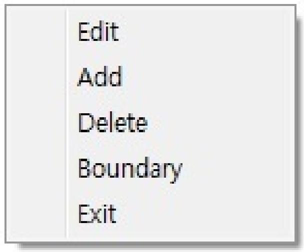
Exit

This is used to exit from the element editing mode.

6.6.2 Edit Node

When you are in Node Editing Mode, you can access popup menu for node in Figure 6.53 by Shift + Right Click.

Figure 6.53 Popup menu for node



Node popup menu consists of five submenus:
Edit, Add, Delete, Boundary and Exit.

Edit

This is used to edit the coordinates of node.
Before editing, work plane should be displayed on the screen.

Edit Current Node dialog is displayed in Figure 6.54.

To edit current node, follow five steps:

1. Enter Node Number
2. Select Reference
3. Select Method
4. Enter Coordinate
5. Draw Node Number

For Mouse Pickup method, when clicking Draw Node Number button at step 5, Coordinates on Work Plane dialog in Figure 6.55 will be opened. Click Info button to see the notes on Mouse Actions on Work Plane as shown in Figure 6.47. Once finished, click Finish in Figure 6.55.

Finally, click Finish on Edit Current Node dialog in Figure 6.54.

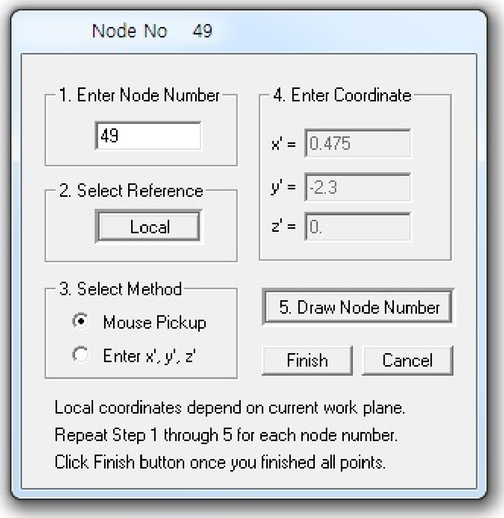


Figure 6.54 Edit current node dialog

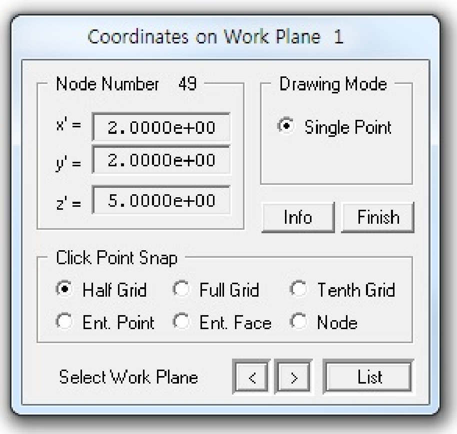


Figure 6.55 Coordinates on work plane (Edit Node)

Add

This is used to add new node.

Before adding, work plane should be displayed on the screen.

Build New Node dialog similar to Figure 6.54 is displayed.

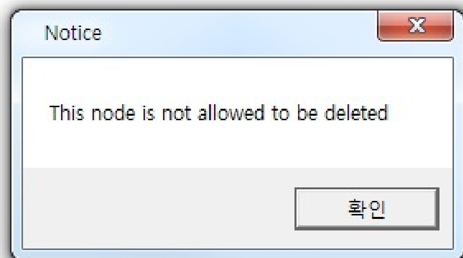
Follow the same procedure as in Edit.

Delete

This is used to delete the selected node.

You can delete only standalone nodes which are not connected to elements. Refer to the notice in Figure 6.56.

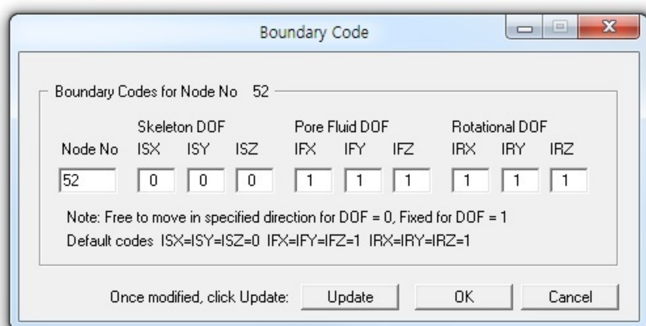
Figure 6.56
Notice on deleting connected nodes



Boundary

This is used to edit boundary codes associated with the current selected node as shown in Figure 6.57.

Figure 6.57
Boundary editor dialog



Exit

This is used to exit from the node editing mode.

6.6.3 Edit Title

This is used to edit the title of the finite element mesh file as shown in Figure 6.58.

Figure 6.58 Mesh title editor dialog

PRESMAP
User's Manual

7.1 Introduction

PRESMAP programs are mainly used to model the geometry of the structures to be analyzed. Mesh File described in Section 4.3 can be created using PRESMAP programs.

Seven PRESMAP programs are provided in this manual; PRESMAP-2D, NATM-2D, CIRCLE-2D, PRESMAP-3D, CROSS-3D, GEN-3D, PILE-3D, PRESMAP-GP, JOINT-3D. and INTERSECTION.

PRESMAP-2D includes Model 1, 2, 3, and 4. Model 1 is basic pre-processor which can be applied to model various types of problem geometry. Model 2 is the special pre-processor developed to model near-fields around underground openings such as tunnels, culverts, etc. Model 3 is the special pre-processor developed to model triangular and rectangular shape geometry. Model 4 is the useful pre-processor to generate layered embankments having slope.

NATM-2D is the special pre-processing program developed to generate automatically two-dimensional finite element meshes and boundary conditions for NATM (New Austrian Tunneling Method) tunnels.

CIRCLE-2D is the special pre-processing program developed to generate automatically two-dimensional finite element meshes for circular cross section with joint interface.

PRESMAP-3D is the basic pre-processor which can be applied to model various types of three dimensional geometries.

CROSS-3D is the special pre-processing program developed to generate automatically three dimensional finite element meshes and boundary conditions for crossing tunnels. The intermediate output file with file extension *.TMP* from CROSS-3D contains finite element block coordinates, indexes, and boundary conditions which are essentially input data to PRESMAP-3D.

GEN-3D generates coordinates, element indexes, boundary codes, external loads, and transmitting boundaries in three dimensional coordinate system by extending typical two dimensional output files from PRESMAP-2D, NATM-2D or CIRCLE-2D.

PILE-3D is the special pre-processor which can be used to generate all input files required for pile foundation analysis. It can generate Concrete Pile with Anchor Bolts or Steel Pipe with Concrete Cap.

PRESMAP-GP is the general purpose pre-processing program which can be used to generate coordinates, element indexes and boundary codes for truss, beam, shell or continuum elements. Users can select rectangular, spherical or cylindrical coordinate for interpolation.

JOINT-3D is the special pre-processor which can be used to generate jointed continuum finite element meshes given the conventional continuum SMAP-3D Mesh File input. For the jointed continuum analysis, each continuum finite element is surrounded by joint elements which allow slippage along the joint when reaching shear strength and debonding normal to the joint face when exceeding tensile strength.

INTERSECTION programs are mainly used to compute the locations of the 3D surfaces crossing each other. These surfaces consist of Shell Elements with different materials. The computed coordinates of intersections can be used for the construction of complicated three-dimensional meshes. Two methods are available: Shell Element and Two Tunnels.

PRESMAP-2D

Model 1

User's Manual

Card Group	Input Data and Definitions (Model 1)
1	1.1 TITLE TITLE Any title of (Max = 60 characters)
	1.2 IP IP = 0 Plane strain or plane stress = 1 Axisymmetry
General Information	1.3 NBLOCK, NBNODE, NSNEL, CMFAC (SMAP-S2/2D) NBLOCK, NBNODE, NSNEL, CMFAC, TEMPI (SMAP-T2) See Figure 7.1 NBLOCK Number of blocks NBNODE Number of block nodes NSNEL Starting element number CMFAC Coordinate magnification factor TEMPI Initial temperature

Card Group	Input Data and Definitions (Model 1)
1 General Information	1.4 NBX, NBY, MIDX, MIDY, NF, NSNODE See Figure 7.2 NBX — Number of blocks in x-direction NBY — Number of blocks in y-direction MIDX = 0 — Element has no side nodes in x-direction = 1 — Element has side nodes in x-direction MIDY = 0 — Element has no side nodes in y-direction = 1 — Element has side nodes in y-direction NF = 0 — Element and node numbering sequence from top to bottom and left to right. = 1 — Element and node numbering sequence from left to right and top to bottom. NSNODE — Starting node number

Card Group	Input Data and Definitions (Model 1)
2 Block Coordinate	2.1 NBNODE Cards: $NODE_1$, X_1, Y_1 / $NODE_2$, X_2, Y_2 / - - - / - - - NODE — Node number X — X-coordinate Y — Y-coordinate

<table>
<tr><th>Card Group</th><th>Input Data and Definitions (Model 1)</th></tr>
<tr><td rowspan="4">3

Data for Each Block</td><td>3.1
BLNAME
BLNAME Block name (up to 60 characters)</td></tr>
<tr><td>3.2
IBLNO
IBLNO Block number</td></tr>
<tr><td>3.3
I_1, I_2, I_3, I_4, M_5, M_6, M_7, M_8, M_9

See Figure 7.1

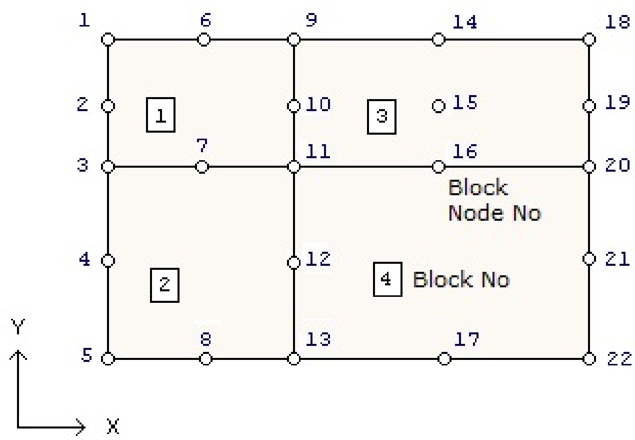
I_1, I_2, I_3, I_4 Corner node number
M_5, M_6, M_7, M_8 Side node number
M_9 Center node number</td></tr>
<tr><td>3.4
IBASE, IB_1, IB_2, IB_3, IB_4, IB_5, IB_6, IB_7, IB_8 (SMAP-2D)
IB_1, IB_2, IB_3, IB_4, IB_5, IB_6, IB_7, IB_8 (SMAP-S2)

See Figure 7.3

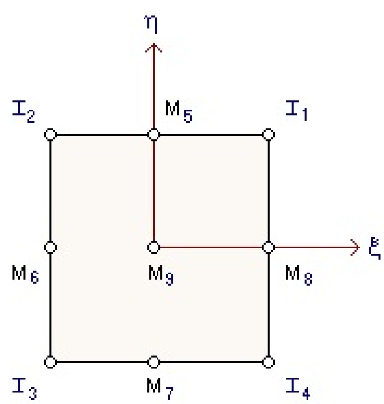
IBASE Base boundary code
IB_1, IB_2, IB_3, IB_4 Corner boundary code
IB_5, IB_6, IB_7, IB_8 Edge boundary code</td></tr>
</table>

Card Group	Input Data and Definitions (Model 1)
3 Data for Each Block	3.5 MATNO, NDX, NDY, KS, KF (SMAP-2D) MATNO, NDX, NDY, THICK, DENSITY (SMAP-S2) MATNO, NDX, NDY, IDH (SMAP-T2) MATNO — Material property number If MATNO = 0, the block is void. NDX — Number of elements in x-direction NDY — Number of elements in y-direction KS = 0 — Has solid phase = 1 — No solid phase KF = 0 — Has fluid phase = 1 — No fluid phase THICK — Thickness of element. For plane strain, use THICK=1.0 DENSITY — Unit weight of element IDH — Heat generation history ID number

Card Group	Input Data and Definitions (Model 1)
3	3.6 NFSIDE NFSIDE — Number of block sides where boundary forces are specified
Data for Each Block Force Data for Each Specified Side (see Figure 7.4)	3.7.1 IEDGE, LHNO, IBF IEDGE — Edge designation number LHNO — Load history number IBF = 0 — No applied force = 1 — Static fluid pressure = 2 — Horizontal force = 3 — Vertical force = 4 — Horizontal and vertical force
	3.7.2 IBF = 1 > $IDIR_n$, q_{n1}, q_{n2} = 2 > $IDIR_h$, q_{h1}, q_{h2} = 3 > $IDIR_v$, q_{v1}, q_{v2} = 4 > $IDIR_h$, q_{h1}, q_{h2} $IDIR_v$, q_{v1}, q_{v2} IDIR = 1 — Pressure/force increases linearly with x = 2 — Pressure/force increases linearly with y q_{n1}, q_{n2} — Static pressure coefficient at edge ends q_{h1}, q_{h2} — Horizontal components of load coefficients at edge ends q_{v1}, q_{v2} — Vertical components of load coefficients at edge ends



NBLOCK = 4, NBNODE = 22
Block number should be in order from top to bottom and left to right



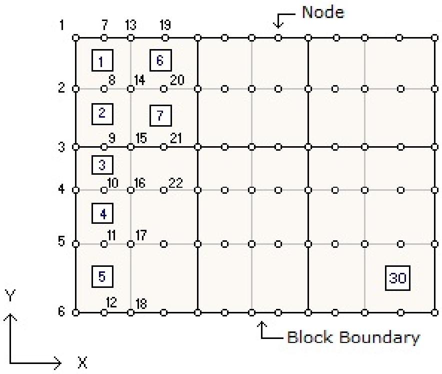
For Block Number 3

I_1 = 18
I_2 = 9
I_3 = 11
I_4 = 20
M_9 = 15

M_5 = 14
M_6 = 10
M_7 = 16
M_8 = 19

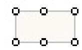
PRESMAP uses Serendipity interpolation if $M_9 = 0$
and Lagrangion interpolation if $M_9 \neq 0$

Figure 7.1 Block Specification and Block Index



NBX = 3
NBY = 2

MIDX = 1
MIDY = 0

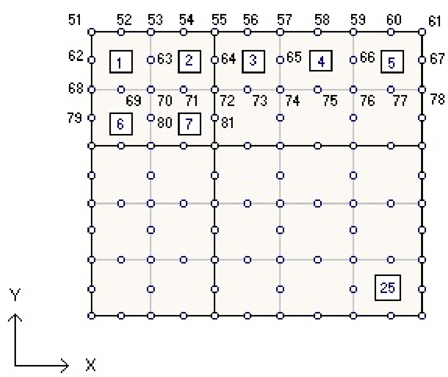


6 Node Element

NF = 0 (Element and node numbers generated from top to bottom and left to right)

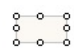
NSNODE = 1

□ Element Number



NBX = 2
NBY = 2

MIDX = 1
MIDY = 1



8 Node Element

NF = 1 (Element and node numbers generated from left to right and top to bottom)

NSNODE = 51

□ Element Number

Figure 7.2 Element and Node Numbering Sequence for Model 1 of PRESMAP-2D

Boundary Codes				
IBASE or IB	ISX	ISY	IFX	IFY
0	0	0	0	0
1	1	0	0	0
2	0	1	0	0
3	1	1	0	0
4	0	0	1	0
5	1	0	1	0
6	0	1	1	0
7	1	1	1	0
8	0	0	0	1
9	1	0	0	1
10	0	1	0	1
11	1	1	0	1
12	0	0	1	1
13	1	0	1	1
14	0	1	1	1
15	1	1	1	1

ISX Specifies skeleton X(radial) degree of freedom
ISY Specifies skeleton Y(axial) degree of freedom

IFX Specifies X(radial) degree of freedom for relative pore fluid motion.
IFY Specifies Y(axial) degree of freedom for relative pore fluid motion.

ISX, ISY, IFX, IFY = 0 Free to move in specified direction
= 1 Fixed in specified direction

Figure 7.3a Boundary Codes for SMAP-2D

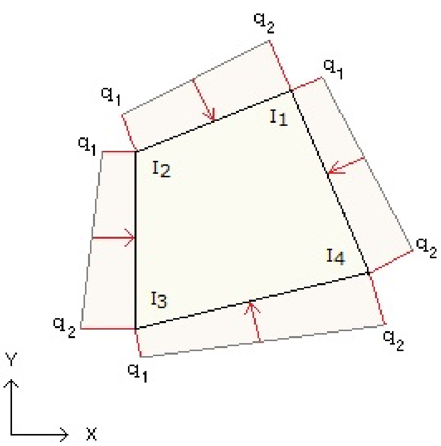
Boundary Type	Boundary Codes		
IB	IDX	IDY	IDT
0	0	0	1
1	1	0	1
2	0	1	1
3	1	1	1
4	0	0	0
5	1	0	0
6	0	1	0
7	1	1	0

IDX = 0 Displacement in x-direction is free
= 1 Displacement in x-direction is fixed

IDY = 0 Displacement in y-direction is free
= 1 Displacement in y-direction is fixed

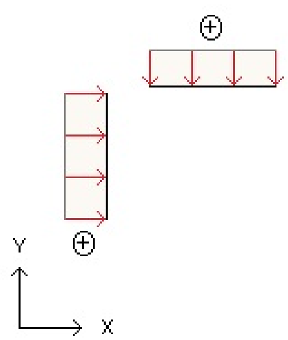
IDT = 0 Rotational degree of freedom is free
= 1 Rotational degree of freedom is fixed

Figure 7.3b Boundary Codes for SMAP-S2



IBF = 1 Static fluid pressure
Compression is positive

I_2 I_1	IEDGE = 1
I_2 I_3	IEDGE = 2
I_3 I_4	IEDGE = 3
I_1 I_4	IEDGE = 4



Horizontal / Vertical Force
Sign Convention

IBF	= 1	Static fluid pressure
	= 2	Horizontal force
	= 3	Vertical force
	= 4	Horizontal / Vertical force

Figure 7.4 Boundary Force/Pressure Sign Conventions

PRESMAP-2D
Model 2
User's Manual

Card Group	Input Data and Definitions (Model 2)
1	1.1 TITLE TITLE — Any title (Max = 60 characters)
	1.2 IP IP = 0 — Plane strain or plane stress = 1 — Axisymmetry
General Information	1.3 NSNEL, NSNODE, NF, CMFAC (SMAP-S2/2D) NSNEL, NSNODE, NF, CMFAC, TEMPI (SMAP-T2) NSNEL — Starting element number NSNODE — Starting node number NF = 0 — Element and node numbering sequence from top to bottom and left to right = 1 — Element and node numbering sequence from left to right and top to bottom CMFAC — Coordinate magnification factor TEMPI — Initial temperature
	1.4 NSUBR, NDRF, NDRS, NDRT, DRF, DRS See Figure 7.5 NSUBR — Number of subregions NDRF — Number of divisions in the first row block NDRS — Number of divisions in the second row block NDRT — Number of divisions in the third row block DRF — Length of the first row block DRS — Length of the second row block

Card Group	Input Data and Definitions (Model 2)
2 Data for Each Subregion	2.1 SUBNAME SUBNAME Subregion name (up to 60 characters)
	2.2 ISUBNO ISUBNO Subregion number
	2.3 ISBTYPE, LSFTYPE, NSEG See Figure 7.6 and 7.7 ISBTYPE = 0 Column grids are normal to subregion surface = 1 Column grids are straight line LSFTYPE = 0 Straight line subregion surface = 1 Circular subregion surface NSEG Number of segments along subregion surface

Card Group	Input Data and Definitions (Model 2)		
2 Data for Each Subregion	2.4 Subregion Surface (Figure 7.6 and 7.7)	For LSFTYPE= 0	2.4.1 X_A, Y_A, X_B, Y_B X_A, Y_A X and Y coordinate of point A X_B, Y_B X and Y coordinate of point B
		For LSFTYPE1=1	2.4.2 R, X_O, Y_O, θ_a, θ_B R Radius of arc AB X_O, Y_O X and Y coordinate of circle origin θ_A, θ_B Polar angle (degree) of point A and B

Card Group	Input Data and Definitions (Model 2)			
2 Data for Each Subregion	2.5 Subregion Outer Edge	For ISBTYPE=0	Point C	2.5.1.1 LCTYPE LCTYPE = 0 X_C and Y_C are specified = 1 X_C is specified = 2 Y_C is specified = 3 DRT_C is specified
				2.5.1.2 If LCTYPE = 0 - - > X_C, Y_C = 1 - - > X_C = 2 - - > Y_C = 3 - - > DRT_C X_C, Y_C X and Y coordinate of point C DRT_C Length of third row block along the edge AC
			Point D	2.5.2.1 LDTYPE LDTYPE = 0 X_D and Y_D are specified = 1 X_D is specified = 2 Y_D is specified = 3 DRT_D is specified
				2.5.2.2 If LDTYPE = 0 - - > X_D, Y_D = 1 - - > X_D = 2 - - > Y_D = 3 - - > DRT_D X_D,Y_D X and Y coordinate of point D DRT_D Length of third row block along the edge BD.

Card Group			Input Data and Definitions (Model 2)
2 Data for Each Subregion	2.5 Subregion Outer Edge	For ISBTYPE =1	2.5.3 X_C, Y_C, X_D, Y_D X_C, Y_C X and Y coordinate of point C X_D, Y_D X and Y coordinate of point D

<table>
<tr><th>Card Group</th><th>Input Data and Definitions (Model 2)</th></tr>
<tr><td rowspan="2">2

Data for Each Subregion</td><td>2.6

$IBASE_1$, $IBASE_2$, $IBASE_3$ (SMAP-2D)
IB_B, IB_A, IB_C, IB_D, IB_{AB}, IB_{AC}, IB_{CD}, IB_{BD} (SMAP-2D/S2)

See Figure 7.3 in Model 1

$IBASE_1$, $IBASE_2$, $IBASE_3$ — First, second, and third block base boundary code
IB_B , IB_A , IB_C , IB_D — Corner boundary code
IB_{AB}, IB_{AC}, IB_{CD}, IB_{BD} — Edge boundary code</td></tr>
<tr><td>2.7

1st Block: $MATNO_1$, KS_1, KF_1 (SMAP-2D)
$MATNO_1$, $DENSITY_1$ (SMAP-S2)
$MATNO_1$, IDH_1 (SMAP-T2)
2nd Block: - -
3rd Block: - -

$MATNO_1$ — Material property number of first block
KS_1, KF_1 — Solid and fluid phase flag of first block
$DENSITY_1$ — Unit weight of first block
IDH_1 — Heat generation history ID of first block

Note: For KS and KF, refer to Card Group 3.5 in PRESMAP-2D Model 1 User's Manual</td></tr>
</table>

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Model 2)</th></tr>
<tr><td rowspan="3">2
Data for Each Subregion</td><td colspan="2">2.8
NFSIDE
NFSIDE — Number of edge where boundary forces are specified</td></tr>
<tr><td rowspan="2">2.9
Force Data for Each Specified Edge (see Figure 7.8)</td><td>2.9.1
IEDGE, LHNO, IBF

IEDGE — Edge designation number
LHNO — Load history number

IBF = 0 — No applied force
= 1 — Static fluid pressure
= 2 — Horizontal force
= 3 — Vertical force
= 4 — Horizontal and vertical force</td></tr>
<tr><td>2.9.2
IBF = 1 --> $IDIR_n$, q_{n1}, q_{n2}
= 2 --> $IDIR_h$, q_{h1}, q_{h2}
= 3 --> $IDIR_v$, q_{v1}, q_{v2}
= 4 --> $IDIR_h$, q_{h1}, q_{h2}
$IDIR_v$, q_{v1}, q_{v2}

IDIR = 1 — Pressure/force increases linearly with x
= 2 — Pressure/force increases linearly with y

q_{n1}, q_{n2} — Static pressure coefficients
q_{h1}, q_{h2} — Horizontal load coefficients
q_{v1}, q_{v2} — Vertical load coefficients</td></tr>
</table>

Block numbers are in order from surface to outer edge and counterclockwise

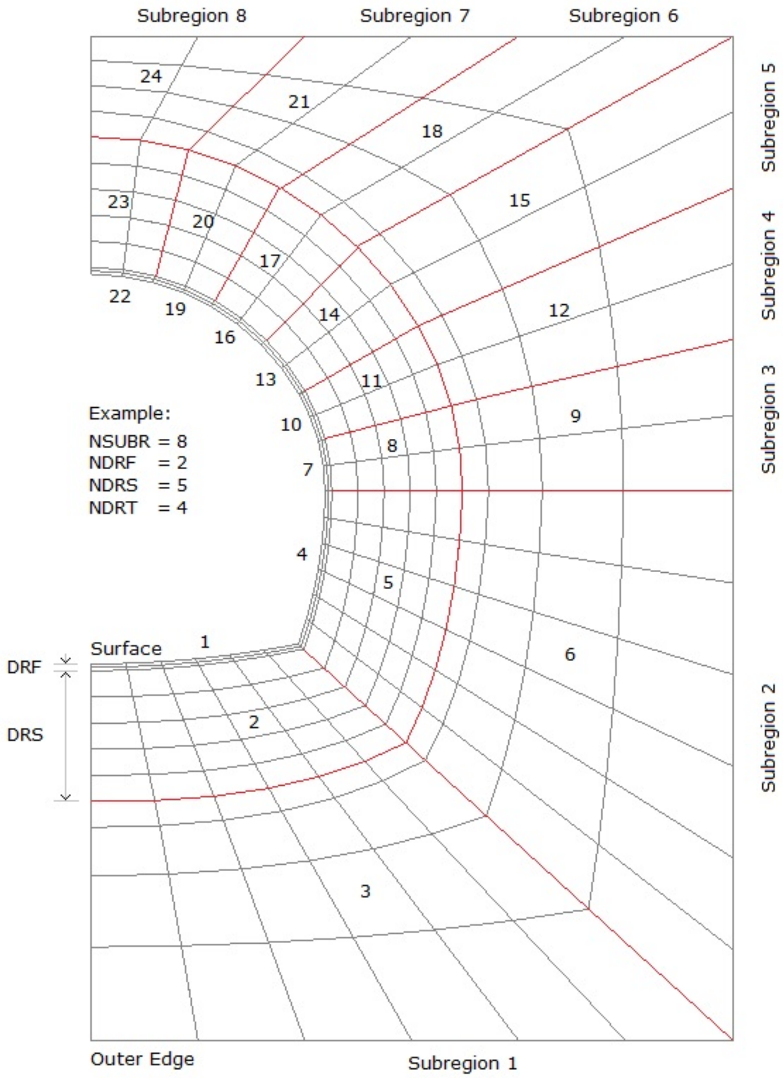


Figure 7.5 Subregion and block numbering sequence

Table 7.1 Subregion parameters in Example Figure 7.5

Subregion	ISBTYPE	LSFTYPE	NSEG
1	1	1	6
2	1	1	6
3	0	1	2
4	0	1	2
5	0	1	2
6	0	1	2
7	0	1	2
8	0	1	2

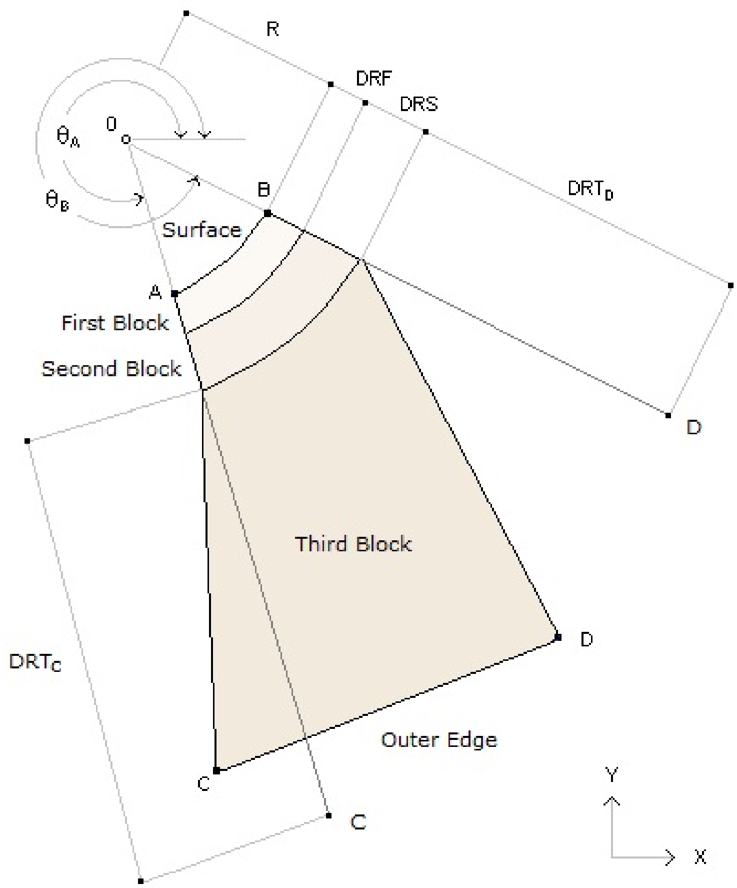


Figure 7.6 Typical Subregion for ISBTYPE = 0

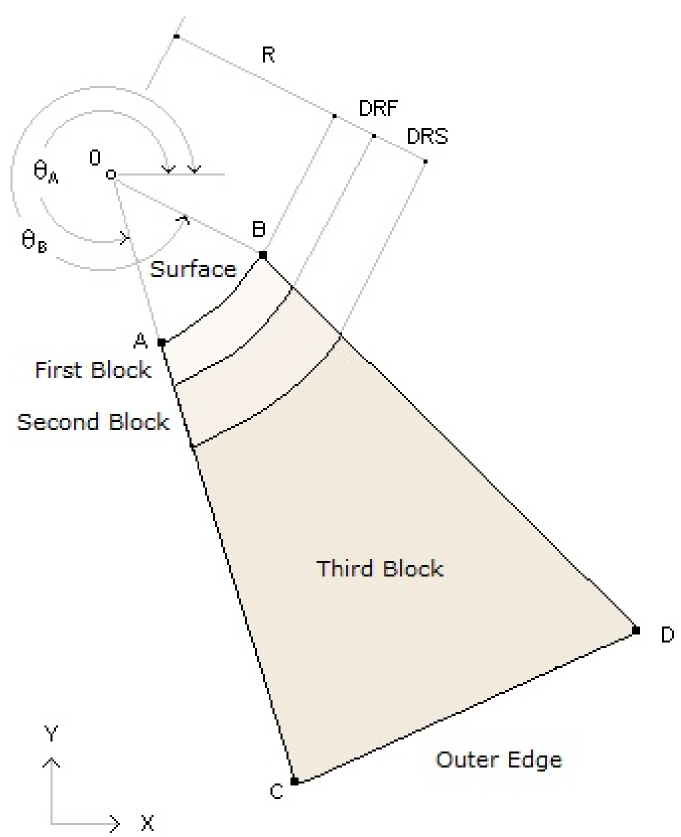
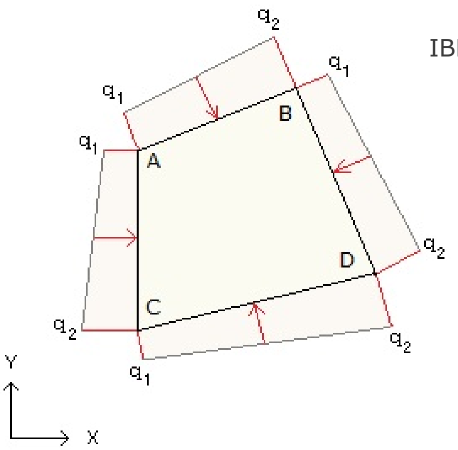
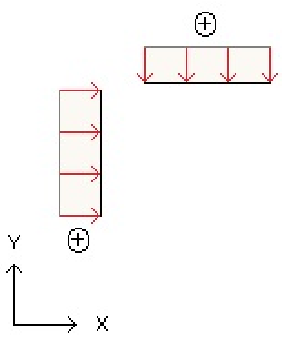


Figure 7.7 Typical Subregion for ISBTYPE = 1



IBF = 1 Static fluid pressure
Compression is positive

AB:	IEDGE = 1
AC:	IEDGE = 2
CD:	IEDGE = 3
BD:	IEDGE = 4



Horizontal / Vertical Force
Sign Convention

IBF	= 1	Static fluid pressure
	= 2	Horizontal force
	= 3	Vertical force
	= 4	Horizontal / Vertical force

Figure 7.8 Boundary Force/Pressure Sign Conventions

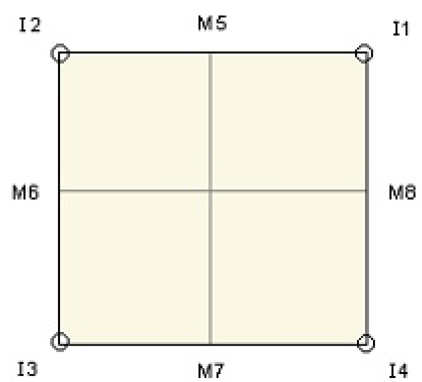
PRESMAP-2D

Model 3

User's Manual

Card Group	Input Data and Definitions (Model 3)
1 General Information	1.1 TITLE TITLE — Any title (Max = 60 characters)
	1.2 IP IP = 0 — Plane geometry = 1 — Axisymmetry geometry
	1.3 NBLOCK, NBNODE, NSNEL, NSNODE, CMFAC See Figure 7.9 NBLOCK — Number of blocks NBNODE — Number of block nodes NSNEL — Starting element number NSNODE — Starting node number CMFAC — Coordinate magnification factor
2 Block Coordinates	2.1 NBNODE Cards: $NODE_1$, X_1, Y_1 $NODE_2$, X_2, Y_2 - - - - - - NODE — Node number X — X-coordinate Y — Y-coordinate

Card Group	Input Data and Definitions (Model 3)
3 Data for Each Block (see Figure 7.9)	3.1 IBLNO, IBLTYPE, MATNO, KS, KF (SMAP-2D) IBLNO, IBLTYPE, MATNO, DENSITY (SMAP-S2) IBLNO, IBLTYPE, MATNO, IDH (SMAP-T2) IBLNO Block number IBLTYPE Block type MATNO Material number KS = 0 Has solid phase = 1 No solid phase KF = 0 Has fluid phase = 1 No fluid phase DENSITY Unit weight IDH Heat generation history ID number

Card Group	Input Data and Definitions (Model 3)
3 Data for Each Block	3.2 For IBLTYPE = 1 I_1, I_2, I_3, I_4, M_5, M_6, M_7, M_8 I_1, I_2, I_3, I_4 Corner node number M_5, M_6, M_7, M_8 Side node number  Note: IBLTYPE = 1 generates 4 elements

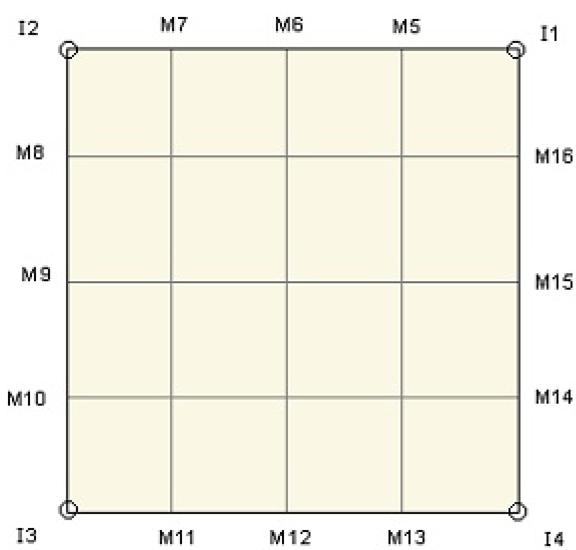
Card Group	Input Data and Definitions (Model 3)
3	For IBLTYPE = 2

Data for Each Block

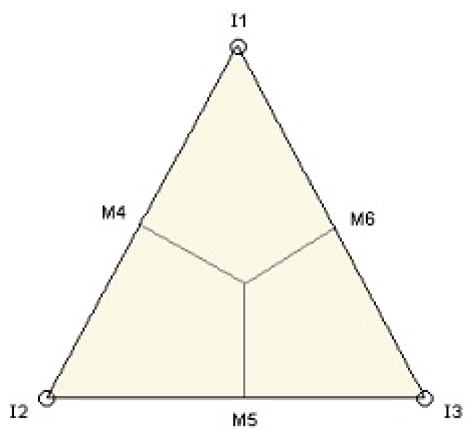
For IBLTYPE = 2

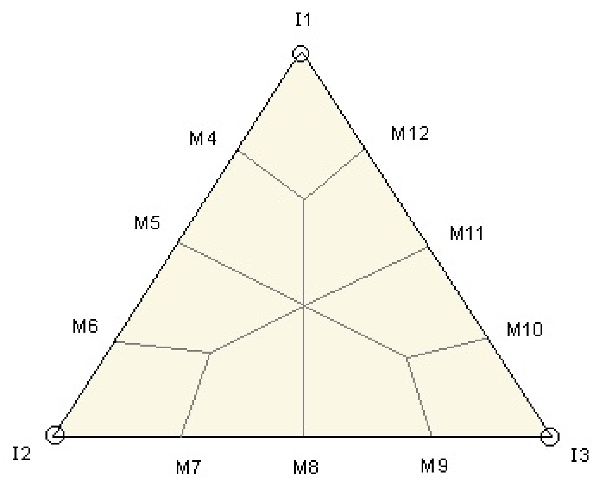
I_1, I_2, I_3, I_4,
M_5, M_6, M_7,
M_8, M_9, M_{10},
M_{11}, M_{12}, M_{13},
M_{14}, $M_{15,}$ M_{16}

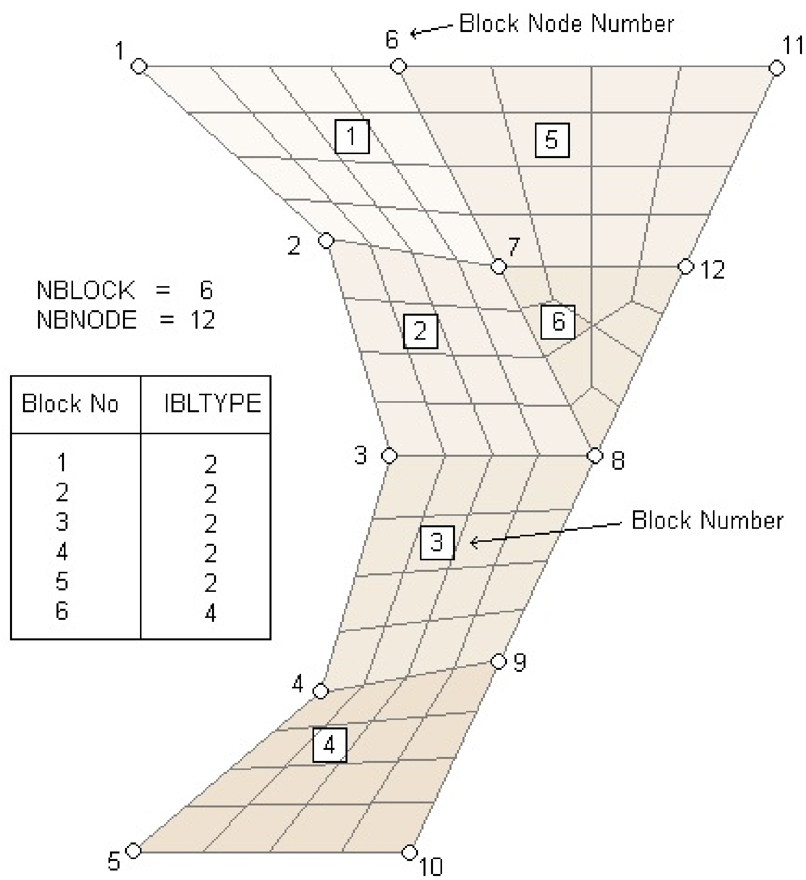
I_1, I_2, I_3, I_4	Corner node number
M_5, M_6,, M_{16}	Side node number



Note: IBLTYPE = 2 generates 16 elements

Card Group	Input Data and Definitions (Model 3)
3 Data for Each Block	For IBLTYPE = 3 I_1, I_2, I_3, M_4, M_5, M_6 I_1, I_2, I_3 Corner node number M_4, M_5, M_6 Side node number  **Note:** IBLTYPE = 3 generates 3 elements

Card Group	Input Data and Definitions (Model 3)
3 Data for Each Block	For IBLTYPE = 4 I_1, I_2, I_3, M_4 M_5, M_6, M_7, M_8, M_9, M_{10} M_{11}, M_{12} I_1, I_2, I_3 Corner node number M_4 - M_{12} Side node number  Note: IBLTYPE = 4 generates 9 elements



Block No	IBLTYPE
1	2
2	2
3	2
4	2
5	2
6	4

Figure 7.9 Block Node Number for Model 3 of PRESMAP-2D

PRESMAP-2D

Model 4

User's Manual

Card Group	Input Data and Definitions (Model 4)
1 General Information	1.1 TITLE TITLE Any title (Max = 60 characters)
	1.2 NLAYER, NDIV, ITRANGL See Figure 7.10 NLAYER Number of layer NDIV Number of elements in first layer ITRANGL = 0 Last element in each layer is rectangle = 1 Last element in each layer is triangle
	1.3 NSNEL, NSNODE, CMFAC NSNEL Starting element number NSNODE Starting node number CMFAC Coordinate magnification factor
2 Block Coordinates	2.1 XB1, YB1, YB2, XB3 See Figure 7.10 XB1, YB1 X, Y coordinate of block node 1 YB2 Y coordinate of block node 2 XB3 X coordinate of block node 3

Card Group	Input Data and Definitions (Model 4)
3 Material Parameters	3.1 MATNO, KS, KF (SMAP-2D) MATNO, DENSITY (SMAP-S2) MATNO, IDH (SMAP-T2) MATNO Material number KS = 0 Has solid phase = 1 No solid phase KF = 0 Has fluid phase = 1 No fluid phase DENSITY Unit weight IDH Heat generation history ID number

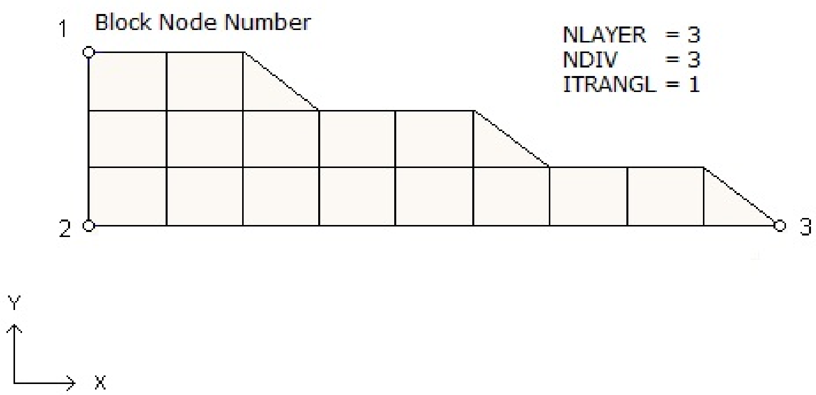


Figure 7.10 Block node number for Model 4 of PRESMAP-2D

NATM-2D

User's Manual

Card Group	Input Data and Definitions
1 General Information	1.1 TITLE TITLE Any title (Max = 60 characters)
	1.2 IUNIT IUNIT Length Force Pressure Unit Weight 1 in lb lb/in^2 lb/in^3 2 m ton ton/m^2 ton/m^3
	1.3 MODEL, IGEN, IEXMESH, ILNCOUPL, IAUTO MODEL = 1 Single tunnel (Half section) = 2 Single tunnel (Full section) = 3 Two tunnels (Symmetric) = 4 Two tunnels (Unsymmetric) IGEN = 0 Generate whole mesh = 1 Generate core = 2 Generate surrounding IEXMESH = 0 No user supplied mesh = 1 Add generated mesh to user supplied mesh For Lining analysis ILNCOUPL= 0 Surrounding rock by continuum element = 1 Surrounding rock by spring element IAUTO = 0 Generate Mesh file = 1 Generate Mesh, Main and Post files Available only for SMAP-S2 See Figure 7.11

Card Group	Input Data and Definitions
2 Tunnel Analysis Boundary	2.1 MODEL = 1: HT, HL, W, DX, DY, NY = 2: HT, HL, W, DX, DY, NY = 3: HT, HL, W, WP, DX, DY, NY = 4: HT, HL, W, WP, HP, DX, DY, NY HT Tunnel depth HL Depth from springline to bottom boundary W Horizontal distance from left to right boundary WP Horizontal distance from left tunnel center line to right tunnel center line HP Vertical distance from right tunnel springline to left tunnel springline. When HP is positive, left tunnel springline is above the right tunnel springline. DX Far-field horizontal element length DY Far-field vertical element length NY Maximum number of elements in the vertical direction See Figure 7.11

Card Group	Input Data and Definitions
3 Soil / Rock Layer Information	3.1 NLAYER NLAYER Total number of layers. Max = 10
	3.2 NLAYER Cards: $LAYERNO_1$, H_1, DD_1 $LAYERNO_2$, H_2, DD_2 - - - - - - LAYERNO Soil/rock layer number H Thickness of soil/rock layer DD = GAMA SMAP-S2 = IDH SMAP-T2 = KF SMAP-2D GAMA Unit weight IDH Heat generation history ID number KF = 0 Has fluid phase = 1 No fluid phase See Figure 7.11

<table>
<tr><th>Card Group</th><th>Input Data and Definitions</th></tr>
<tr><td rowspan="3">4

Tunnel Dimension (Repeat this card group for the left tunnel when MODEL = 4)</td><td>4.1

R_1, A_1, R_2, A_2, R_3, A_3, R_4, GR, GA

R_1, R_2, R_3, R_4 Radius as shown in Figure 7.12
A_1, A_2, A_3 Angle (˚) as shown in Figure 7.12

GR Growing rate for near-field element. Use GR = 1
GA Normalized mid length. Use GA= 0.5</td></tr>
<tr><td>4.2

INVSHOT, T_S, T_l

INVSHOT = 0 No shotcrete at invert
= 1 Shotcrete at invert

T_S Thickness of shotcrete
T_l Thickness of lining

Note: For A_1+A_2 > 90, invert shotcrtete is always included</td></tr>
<tr><td>4.3

NUMRB, L_{RB}, $L_{SPACING}$, $T_{SPACING}$, NSRB

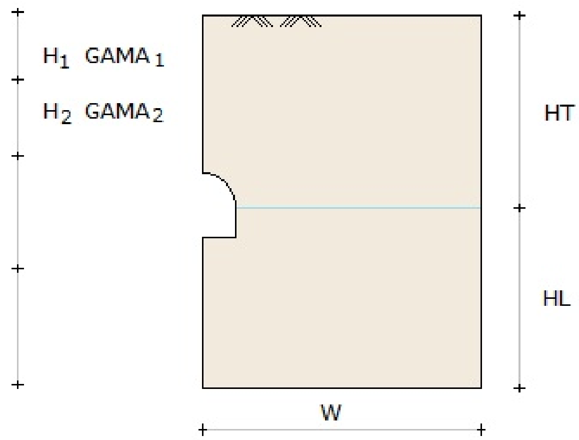
NUMRB Number of rock bolts
Example: NUMRB = 11 in Figure 7.12

L_{RB} Length of rock bolt
$L_{SPACING}$ Rock bolt spacing in longitudinal direction
$T_{SPACING}$ Rock bolt spacing in tangential direction

NSRB Number of elements between rock bolts
Use NSRB = 2 or 3</td></tr>
</table>

Card Group	Input Data and Definitions
5 Water Pressure and Loosening Load	5.1 LDTYPE, DGW, GAMAW, HPRES, VPRES, SUBGK, ITSPR, NUMSJ LDTYPE = 0 No external load = 1 Water pressure only = 2 Loosening load only = 3 Water pressure and loosening load DGW Depth of ground water table from ground surface GAMAW Unit weight of water HPRES Horizontal pressure due to loosening load VPRES Vertical Pressure due to loosening load SUBGK Coefficient of subgrade reaction (ILCOUPL = 1) ITSPR = 0 No tangential spring = 1 Add tangential spring NUMSJ Number of segment joints Available for circular shape of MODEL 2
	5.2 Joint Locations If NUMSJ = 0, skip this card AJ_1, AJ_1, ... , AJ_{NUMSJ} AJ_i Angle (degrees) from crown top ($AJ_i \leq 180$)

MODEL = 1 Single Tunnel (Half Section)



MODEL = 2 Single Tunnel (Full Section)

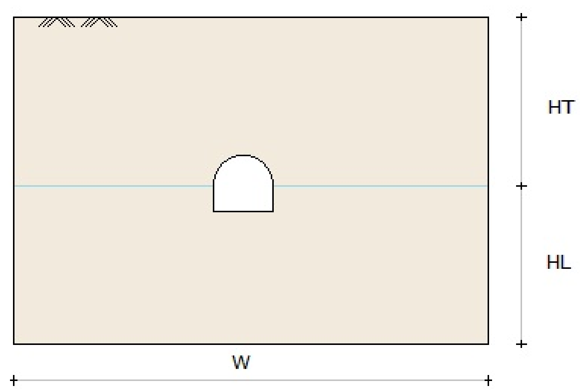
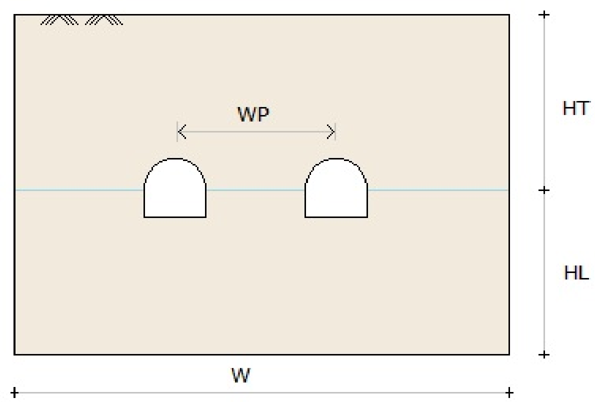


Figure 7.11 Schematic tunnel section view for MODEL = 1 and 2

MODEL = 3 Two Tunnel (Symmetric Section)



MODEL = 4 Two Tunnel (Unsymmetric Section)

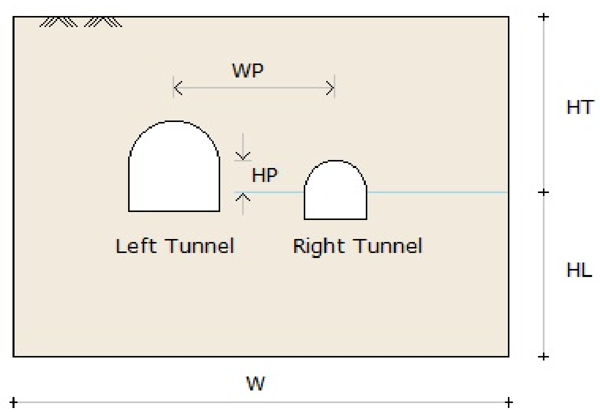
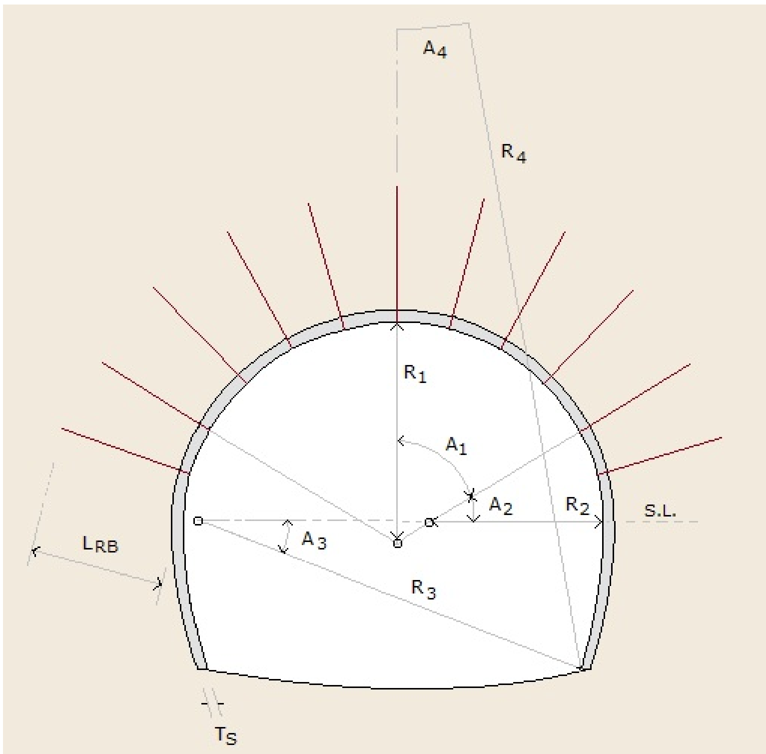


Figure 7.11 Schematic tunnel section view for MODEL = 3 and 4



R_4 = 0 : Invert is flat
R_4 < 0 : Invert depth is given as absolute value of R_4

Refer to Example problem MODEL 4-1 and 4-3

Figure 7.12 Tunnel dimension (A_1+A_2 = 90)

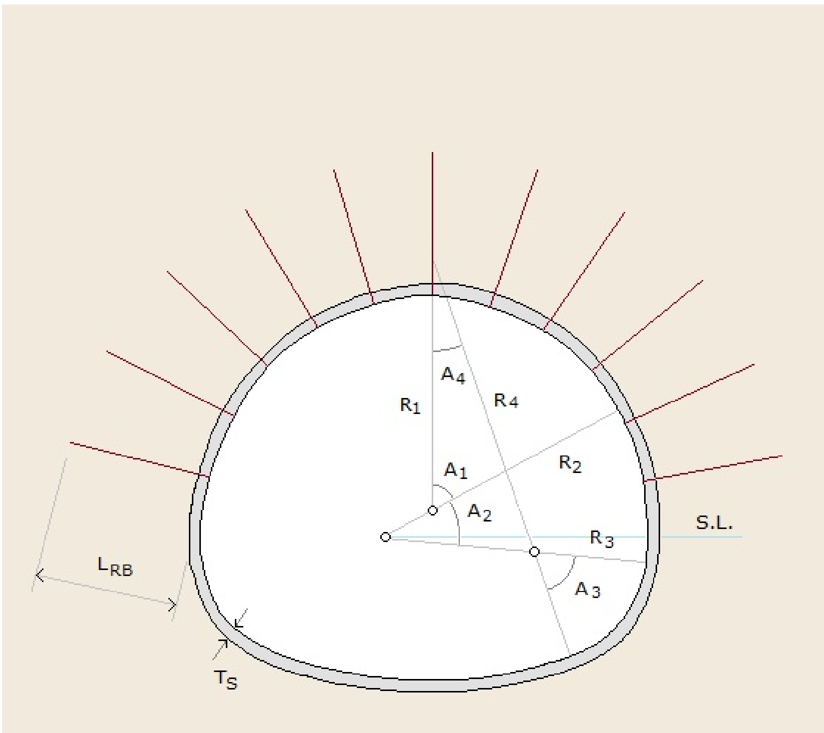


Figure 7.12 Tunnel dimension (A_1+A_2 > 90)

CIRCLE-2D

User's Manual

Card Group	Input Data and Definitions
1 General Information	1.1 TITLE TITLE — Any title (Max = 80 characters)
	1.3 MODEL, NSNEL, NSNODE MODEL = 1 — Quarter Section = 2 — Half Section = 3 — Full Section NSNEL — Starting element number NSNODE — Starting node number See Figure 7.13
2 Geometry	2.1 R, FINEMESH, NEARMESH, NDIV, BH, BV R — Radius of Circular Core FINEMESH = 0 — Coarse Mesh = 1 — Fine Mesh NEARMESH = 0 — All Quad Mesh = 1 — Quad and Triangle Mesh NDIV — Number of divisions for outer zone BH, BV — Horizontal and Vertical dimensions

Card Group	Input Data and Definitions
3 Material Number	3.1 $COREMAT_1$, $COREMAT_2$, $COREMAT_{2j}$, JOINTMAT, NEARMAT $COREMAT_1$ Material No for Core 1 $COREMAT_2$ Material No for Core 2 $COREMAT_{2j}$ Material No for Core 2 facing Joint JOINTMAT Material No for Joint NEARMAT Material No for Near Note $COREMAT_1$ and $COREMAT_2$ have the common interface with NEARMAT and JOINTMAT, respectively. When material number for $COREMAT_1$ or JOINTMAT is zero, meshes corresponding to that material will not be generated.

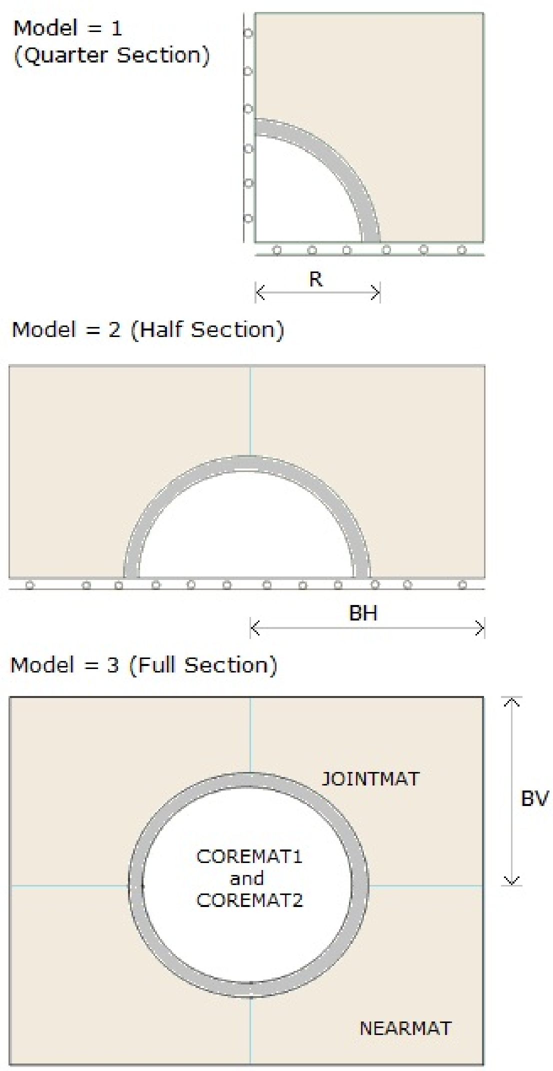


Figure 7.13 Model type for CIRCLE-2D

PRESMAP-GP

User's Manual

<table>
<tr><th>Card Group</th><th>Input Data and Definitions</th></tr>
<tr><td rowspan="2">1

General Information</td><td>1.1

TITLE

TITLE Any title (Max = 80 characters)

Note: Following two cards are required at the beginning
StartPresmap
VersionNo = 7.000</td></tr>
<tr><td>1.2

NBLOCK, NBNODE, NSNODE, NSNEL, IGBND, ISMAP, CMFAC, ICOMP

NBLOCK Number of blocks
NBNODE Number of block nodes
NSNODE Starting node number
NSNEL Starting element number

IGBND = 0 Do not generate
= 1 Generate global boundary conditions based on Card 1.3

ISMAP = 1 Mesh generation for SMAP-S2
= 2 Mesh generation for SMAP-2D
= -2 Mesh generation for SMAP-T2
= 3 Mesh generation for SMAP-3D & S3
= -3 Mesh generation for SMAP-T3

CMFAC Coordinate magnification factor

ICOMP = 0 Do not impose
= 1 Impose compatibility between blocks

Note: If NBLOCK is negative value, the output file contains plotting information for block diagram</td></tr>
</table>

Card Group	Input Data and Definitions
1 General Information	1.3 Six cards starting from right, left, top, bottom, front, back For SMAP-S2/S3/2D/3D ISG, ISX, ISY,ISZ, IFG, IFX, IFY,IFZ, IRG, IRX, IRY,IRZ For SMAP-T2/T3 ITG, IDF, T, CF ISG, IFG, IRG = 0 None = 1 Free boundary = 2 Fixed boundary = 3 Roller boundary = 4 Specified in X, Y, Z directions ITG = 0 None = 1 Heat Flow = 2 Temperature IDF Time function identification number T Initial temperature CF Time function coefficient
	1.4 ELMIN, MAXNEL ELMIN Minimum element length MAXNEL Maximum number of elements Note: ELMIN and MAXNEL are used in PLOT-3D as control parameters to generate automatically finite elements

Card Group	Input Data and Definitions
2 Block Coordinate	2.1 NBNODE Cards: $NODE_1$, X_1, Y_1, Z_1 $NODE_2$, X_2, Y_2, Z_2 - - - - - - - - NODE — Node number X — X-coordinate Y — Y-coordinate Z — Z-coordinate

Card Group	Input Data and Definitions
3 Data for Each Block	3.0 IBETYPE IBETYPE = 1 Line block (Beam or Truss Element) = 2 Quad surface block = -2 Triangle surface block Surface block generates plane strain/stress, or axisymmetric element for ISMAP = 1 or 2 and shell/ membrane element for ISMAP = 3 = 3 Hexahedron volume block = -3 Prism volume block. Volume block generates 3-D Continuum element or 3-D Joint element. Note: Card Group 3 requires following cards: At the beginning of each block StartBlock At the end of each block EndBlock At the end of last block EndOfLastBlock

Card Group	Input Data and Definitions
3	3.1 BLNAME BLNAME Block name (Max = 60 characters)
Data for Each Line Block [IBETYPE =1]	3.2 ICOORD, IMODE, ILAG Interpolation based on ICOORD = 1 Rectangular coordinate = 2 Spherical coordinate = 3 Cylindrical coordinate Modify generated coordinate IMODE = 0 Do not modify = 1 Modify using reference node (M_5) as origin for ICOORD = 1. Modify coordinate based on rectangular grid for ICOORD = 2 or 3. ILAG = 0 Generate Beam element = 1 Generate Truss element

Card Group	Input Data and Definitions
3 Data for Each Line Block [IBETYPE =1]	3.3 I_1, I_2 M_3 M_4 M_5, M_6, M_7 See Figure 7.22 I_1 - I_2 Corner node number of a block M_3 Side node number of a block M_4 Reference node number For ICOORD = 2 M_5 Node number defining origin of spherical coordinate For ICOORD = 3 M_5 Node number defining reference origin of cylindrical coordinate M_6 Node number defining cylinder axis M_5 - M_6 M_7 Node number defining other local axis M_5 - M_7 which is normal to cylinder axis.

Card Group	Input Data and Definitions	
3	3.4	3.4.1 NBOUND NBOUND — Number of boundaries to be specified If NBOUND = 0, go to Card group 3.5
Data for Each Line Block [IBETYPE =1]		3.4.2 NBOUND cards For SMAP-S2/S3/2D/3D IBTYPE, ISX, ISY,ISZ, IFX, IFY,IFZ, IRX, IRY,IRZ For SMAP-T2/T3 IBTYPE, ID, IDF, T, CF IBTYPE = 1 Interior line = 2 Node I_1 = 3 Node I_2 = 4 Node M_4 Skeleton X, Y, Z DOF : ISX, ISY, ISZ Pore fluid X, Y, Z DOF relative to skeleton : IFX, IFY, IFZ Rotational DOF about X, Y, Z axis : IRX, IRY, IRZ ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ = 0 Free to move in specified direction = 1 Fixed in specified direction Default boundary conditions ISX=ISY=ISZ=0, IFX=IFY=IFZ=1, IRX=IRY=IRZ=0 For SMAP-T2/T3 ID = 0 Heat flow is specified = 1 Temperature is specified IDF — Time function identification number T — Initial temperature CF — Time function coefficient

Card Group	Input Data and Definitions
3 Data for Each Line Block [IBETYPE =1]	3.5 MATNO, NDX MATNO Material property number NDX Number of elements in x-direction

Card Group	Input Data and Definitions
3 Data for Each Quad Surface Block [IBETYPE =2]	3.1 BLNAME BLNAME Block name (Max = 60 characters)
	3.2 ICOORD, IMODE, ILAG Interpolation based on ICOORD = 1 Rectangular coordinate = 2 Spherical coordinate = 3 Cylindrical coordinate Modify generated coordinate IMODE = 0 Do not modify = 1 Modify using reference node (M_{10}) as origin for ICOORD = 1. Modify coordinate based on rectangular grid for ICOORD = 2 or 3. ILAG = 0 Serendipity interpolation = 1 Lagrangian interpolation = 2 Surface sector generation

Card Group	Input Data and Definitions
3 Data for Each Quad Surface Block [IBETYPE =2]	3.3 I_1, I_2, I_3, I_4 M_5, M_6, M_7, M_8 M_9 M_{10}, M_{11}, M_{12} See Figure 7.22 I_1 - I_4 Corner node number of a block M_5-M_8 Side node number of a block M_9 Center node number of a block, used for ILAG = 1 For ICOORD = 2 M_{10} Node number defining origin of spherical coordinate For ICOORD = 3 M_{10} Node number defining reference origin of cylindrical coordinate M_{11} Node number defining cylinder axis M_{10} - M_{11} M_{12} Node number defining other local axis M_{10} - M_{12} which is normal to cylinder axis

Card Group		Input Data and Definitions
3 Data for Each Quad Surface Block [IBETYPE =2]	3.4	3.4.1 NBOUND NBOUND — Number of boundaries to be specified If NBOUND = 0, go to Card group 3.5
		3.4.2 NBOUND cards For SMAP-S2/S3/2D/3D IBTYPE, ISX, ISY,ISZ, IFX, IFY,IFZ, IRX, IRY,IRZ For SMAP-T2/T3 IBTYPE, ID, IDF, T, CF IBTYPE = 1 Interior surface = 2 Line I_1 - I_2 = 3 Line I_2 - I_3 = 4 Line I_3 - I_4 = 5 Line I_4 - I_1 = 6 Node I_1 = 7 Node I_2 = 8 Node I_3 = 9 Node I_4 Skeleton X, Y, Z DOF : ISX, ISY, ISZ Pore fluid X, Y, Z DOF relative to skeleton : IFX, IFY, IFZ Rotational DOF about X, Y, Z axis : IRX, IRY, IRZ ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ = 0 Free to move in specified direction = 1 Fixed in specified direction Default boundary conditions ISX=ISY=ISZ=0, IFX=IFY=IFZ=1, IRX=IRY=IRZ=0 For SMAP-T2/T3 ID = 0 Heat flow is specified = 1 Temperature is specified IDF Time function identification number T Initial temperature CF Time function coefficient

Card Group	Input Data and Definitions
3	3.5
Data for Each Quad Surface Block [IBETYPE =2]	MATNO, NDX, NDY NT_1, NT_2, NT_3, NT_4 MAT_1, MAT_2, MAT_3, MAT_4 THICK, DENSITY (For ISMAP = 1) KS, KF (For ISMAP = 2) IDH (For ISMAP =-2 or -3) MATNO — Material property number NDX — Number of elements in I_2 to I_1 direction NDY — Number of elements in I_2 to I_3 direction NT — For NT i is greater than zero, a triangle at block node i with NT i divisions along the triangle base. NT i ≤ min (NDX, NDY) and NT i + NT j ≤ min (NDX, NDY) where i =1, 2, 3, 4 j =2, 3, 4, 1 MAT_i — Material property number for the triangle at block node i. Zero value of MAT will remove the triangle. THICK — Thickness of element. For plane strain, use THICK = 1.0 DENSITY — Unit weight of element KS = -1 — Element has high explosive solid phase = 0 — Element has solid phase > 0 — Element has joint and absolute value of KS represents face designation number. KF = 0 — Element has fluid phase = 1 — Element has no fluid phase IDH — Heat generation history ID number

Card Group	Input Data and Definitions
3	3.6

Data for Each Quad Surface Block [IBETYPE =2]

Only for ICOORD = 2 and ILAG = 2

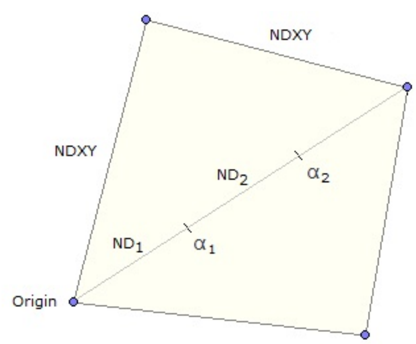
NSEG

NSEG Cards
- $ALPA_1$, $NDIV_1$
- $ALPA_2$, $NDIV_2$
- \- \-

NSEG	Number of segments
ALPA	Percent radial distance from origin
NDIV	Number of divisions between $ALPA_{i-1}$ and $ALPA_i$

Note: This option (ILAG=2) is to generate surface sector and has the following restrictions:

1. ICOORD = 2 (Spherical Coordinate)
2. IMOD = 0 Curved edge
   = 2 Straight edge
3. Midside and center nodes are not used.
4. NDX = NDY = NDXY = Σ $NDIV_i$



Card Group	Input Data and Definitions
3	3.1 BLNAME BLNAME Block name (Max = 60 characters)
Data for Each Triangle Surface Block [IBETYPE =-2]	3.2 ICOORD, IMODE, ILAG Interpolation based on ICOORD = 1 Rectangular coordinate = 2 Spherical coordinate = 3 Cylindrical coordinate Modify generated coordinate IMODE = 0 Do not modify = 1 Modify using reference node (M_8) as origin for ICOORD = 1. Modify coordinate based on rectangular grid for ICOORD = 2 or 3. ILAG = 0 Serendipity interpolation = 1 Lagrangian interpolation = 2 Circular surface generation

Card Group	Input Data and Definitions
3 Data for Each Triangle Surface Block [IBETYPE =-2]	3.3 I_1, I_2, I_3 M_4, M_5, M_6 M_7 M_8, M_9, M_{10} See Figure 7.22 I_1 - I_3 Corner node number of a block M_4- M_6 Side node number of a block M_7 Center node number of a block for ILAG = 1 For ICOORD = 2 M_8 Node number defining origin of spherical coordinate For ICOORD = 3 M_8 Node number defining reference origin of cylindrical coordinate. M_9 Node number defining cylinder axis M_8-M_9 M_{10} Node number defining other local axis M_8-M_{10} which is normal to cylinder axis.

Card Group	Input Data and Definitions	
3 Data for Each Triangle Surface Block [IBETYPE=-2]	3.4	3.4.1 NBOUND NBOUND Number of boundaries to be specified If NBOUND = 0, go to Card group 3.5
		3.4.2 NBOUND cards For SMAP-S2/S3/2D/3D IBTYPE, ISX, ISY,ISZ, IFX, IFY,IFZ, IRX, IRY,IRZ For SMAP-T2/T3 IBTYPE, ID, IDF, T, CF IBTYPE = 1 Interior surface = 2 Line I_1 - I_2 = 3 Line I_2 - I_3 = 4 Line I_3 - I_1 = 5 Node I_1 = 6 Node I_2 = 7 Node I_3 Skeleton X, Y, Z DOF : ISX, ISY, ISZ Pore fluid X, Y, Z DOF relative to skeleton : IFX, IFY, IFZ Rotational DOF about X, Y, Z axis : IRX, IRY, IRZ ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ = 0 Free to move in specified direction = 1 Fixed in specified direction Default boundary conditions ISX=ISY=ISZ=0, IFX=IFY=IFZ=1, IRX=IRY=IRZ=0 For SMAP-T2/T3 ID = 0 Heat flow is specified = 1 Temperature is specified IDF Time function identification number T Initial temperature CF Time function coefficient

Card Group	Input Data and Definitions
3	3.5

Data for Each Triangle Surface Block [IBETYPE =-2]

3.5

MATNO, NDXY
THICK, DENSITY (For ISMAP = 1)
KS, KF (For ISMAP = 2)
IDH (For ISMAP =-2 or -3)

MATNO	Material property number
NDXY	Number of elements along triangle edge For wedge surface block, use negative NDXY Refer to Example problem 11
THICK	Thickness of element. For plane strain, use THICK = 1.0
DENSITY	Unit weight of element
KS = -1	Element has high explosive solid phase
= 0	Element has solid phase
> 0	Element has joint and absolute value of KS represents face designation number.
KF = 0	Element has fluid phase
= 1	Element has no fluid phase
IDH	Heat generation history ID number

Card Group	Input Data and Definitions
3	3.6

Data for Each Triangle Surface Block [IBETYPE =-2]

Only for ICOORD = 2 and ILAG = 2

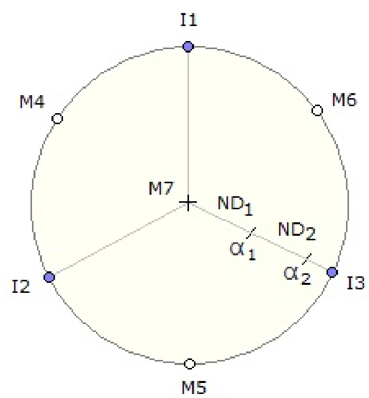
NSEG

NSEG Cards ┌ $ALPA_1$, $NDIV_1$
│ $ALPA_2$, $NDIV_2$
└ - -

NSEG	Number of segments
ALPA	Percent radial distance from origin
NDIV	Number of divisions between $ALPA_{i-1}$ and $ALPA_i$

Note: This option (ILAG = 2) is to generate circular surface and has the following restrictions:

1. ICOORD =2 (Spherical Coordinate)
2. IMOD = 0 Curved edge
   = 2 Straight edge
3. Block center node should be origin (M_7=M_8)
4. Midside nodes (M_4, M_5 and M_6) are interpolated based on spherical coordinate



Card Group	Input Data and Definitions
3	3.1 BLNAME BLNAME Block name (Max = 60 characters)
Data for Each Hexahedron Volume Block [IBETYPE = 3]	3.2 ICOORD, IMODE, ILAG Interpolation based on ICOORD = 1 Rectangular coordinate = 2 Spherical coordinate = 3 Cylindrical coordinate Modify generated coordinate IMODE = 0 Do not modify = 1 Modify using reference node (M_{28}) as origin for ICOORD = 1. Modify coordinate based on rectangular grid for ICOORD = 2 or 3. ILAG = 0 Serendipity interpolation = 1 Lagrangian interpolation

Card Group	Input Data and Definitions
3 Data for Each Hexahedron Volume Block [IBETYPE =3]	3.3 I_1, I_2, I_3, I_4, I_5, I_6, I_7, I_8 M_9, M_{10}, M_{11}, M_{12}, M_{13}, M_{14}, M_{15}, M_{16}, M_{17}, M_{18}, M_{19}, M_{20} M_{21}, M_{22}, M_{23}, M_{24}, M_{25}, M_{26}, M_{27} M_{28} M_{28}, M_{29}, M_{30} See Figure 7.22 I_1 - I_8 Corner node number of a block M_9 - M_{20} Side node number of a block M_{21} -M_{27} Side node number of a block required for Lagrangian interpolation For ICOORD = 2 or IMODE = 1 M_{28} Node number defining origin of spherical coordinate for ICOORD = 2, or node number defining reference origin to the whole volume for IMODE = 1 For ICOORD = 3 M_{28} Node number defining reference origin of cylindrical coordinate M_{29} Node number defining cylinder axis M_{28}-M_{29} M_{30} Node number defining other local axis M_{28}-M_{30} which is normal to cylinder axis

Card Group		Input Data and Definitions
3	3.4	3.4.1 NBOUND NBOUND Number of boundaries to be specified If NBOUND = 0, go to Card group 3.5
Data for Each Hexahedron Volume Block [IBETYPE =3]		3.4.2 NBOUND cards For SMAP-S2/S3/2D/3D IBTYPE, ISX, ISY,ISZ, IFX, IFY,IFZ, IRX, IRY,IRZ For SMAP-T2/T3 IBTYPE, ID, IDF, T, CF IBTYPE = 1 Interior Volume = 2 Front surface = 3 Back surface = 4 Left surface = 5 Right surface = 6 Top surface = 7 Bottom surface = 8 Line I_1 - I_2 = 9 Line I_2 - I_3 = 10 Line I_3 - I_4 = 11 Line I_4 - I_1 = 12 Line I_5 - I_6 = 13 Line I_6 - I_7 = 14 Line I_7 - I_8 = 15 Line I_8 - I_5 = 16 Line I_1 - I_5 = 17 Line I_2 - I_6 = 18 Line I_3 - I_7 = 19 Line I_4 - I_8 = 20 Node I_1 = 21 Node I_2 = 22 Node I_3 = 23 Node I_4 = 24 Node I_5

Card Group	Input Data and Definitions
3 Data for Each Hexahedron Volume Block [IBETYPE =3]	3.4.2 IBTYPE = 25 Node I_6 = 26 Node I_7 = 27 Node I_8 See Figure 7.23 Skeleton X, Y, Z DOF : ISX, ISY, ISZ Pore fluid X, Y, Z DOF relative to skeleton : IFX, IFY, IFZ Rotational DOF about X, Y, Z axis : IRX, IRY, IRZ ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ = 0 Free to move in specified direction = 1 Fixed in specified direction Default boundary conditions ISX=ISY=ISZ=0, IFX=IFY=IFZ=1, IRX=IRY=IRZ=1 For SMAP-T2/T3 ID = 0 Heat flow is specified = 1 Temperature is specified IDF Time function identification number T Initial temperature CF Time function coefficient

Card Group	Input Data and Definitions
3 Data for Each Hexahedron Volume Block [IBETYPE =3]	3.5 MATNO, NDX, NDY, NDZ, KS, KF (For ISMAP = 3) MATNO, NDX, NDY, NDZ, IDH (For ISMAP =-3) NT_1, NT_2, NT_3, NT_4 MAT_1, MAT_2, MAT_3, MAT_4 MATNO Material property number NDX Number of elements in I_2 - I_1 direction NDY Number of elements in I_2 - I_3 direction NDZ Number of elements in I_2 - I_6 direction KS = -1 Element has high explosive solid phase = 0 Element has solid phase > 0 Element has joint and absolute value of KS represents face designation number. KF = 0 Element has fluid phase = 1 Element has no fluid phase IDH Heat generation history ID number NT & MAT See descriptions on page 7-92

Card Group	Input Data and Definitions
3 Data for Each Prism Volume Block [IBETYPE = -3]	3.1 BLNAME BLNAME Block name (Max = 60 characters)
	3.2 ICOORD, IMODE, ILAG Interpolation based on ICOORD = 1 Rectangular coordinate = 2 Spherical coordinate = 3 Cylindrical coordinate Modify generated coordinate IMODE = 0 Do not modify = 1 Modify using reference node (M_{22}) as origin for ICOORD = 1 Modify coordinate based on rectangular grid for ICOORD = 2 or 3 ILAG = 0 Serendipity interpolation = 1 Lagrangian interpolation

Card Group	Input Data and Definitions
3 Data for Each Prism Volume Block [IBETYPE =-3]	3.3 I_1, I_2, I_3, I_4, I_5, I_6 M_7, M_8, M_9, M_{10}, M_{11}, M_{12}, M_{13}, M_{14}, M_{15}, M_{16}, M_{17} M_{18}, M_{19}, M_{20}, M_{21} M_{22}, M_{23}, M_{24} See Figure 7.22 I_1 - I_6 Corner node number of a block M_7- M_{20} Side node number of a block M_{21} Center node number of a block For ICOORD = 2 or IMODE = 1 M_{22} Node number defining origin of spherical coordinate for ICOORD = 2, or node number defining reference origin to the whole volume for IMODE = 1 For ICOORD = 3 M_{22} Node number defining reference origin of cylindrical coordinate. M_{23} Node number defining cylinder axis M_{22}-M_{23} M_{24} Node number defining other local axis M_{22}-M_{24} which is normal to cylinder axis.

Card Group	Input Data and Definitions	
3 Data for Each Prism Volume Block [IBETYPE =-3]	3.4	3.4.1 NBOUND NBOUND — Number of boundaries to be specified If NBOUND = 0, go to Card group 3.5
		3.4.2 NBOUND cards For SMAP-S2/S3/2D/3D IBTYPE, ISX, ISY,ISZ, IFX, IFY,IFZ, IRX, IRY,IRZ For SMAP-T2/T3 IBTYPE, ID, IDF, T, CF IBTYPE = 1 Interior volume = 2 Front surface = 3 Back surface = 4 Left surface = 5 Right surface = 6 Bottom surface = 7 Line I_1 - I_2 = 8 Line I_2 - I_3 = 9 Line I_3 - I_1 = 10 Line I_4 - I_5 = 11 Line I_5 - I_6 = 12 Line I_6 - I_4 = 13 Line I_1 - I_4 = 14 Line I_2 - I_5 = 15 Line I_3 - I_6 = 16 Node I_1 = 17 Node I_2 = 18 Node I_3 = 19 Node I_4 = 20 Node I_5 = 21 Node I_6 See Figure 7.24

Card Group	Input Data and Definitions
3 Data for Each Prism Volume Block [IBETYPE =-3]	3.4.2 Skeleton X, Y, Z DOF : ISX, ISY, ISZ Pore fluid X, Y, Z DOF relative to skeleton : IFX, IFY, IFZ Rotational DOF about X, Y, Z axis : IRX, IRY, IRZ ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ = 0 Free to move in specified direction = 1 Fixed in specified direction Default boundary conditions ISX=ISY=ISZ=0, IFX=IFY=IFZ=1, IRX=IRY=IRZ=1 For SMAP-T2/T3 ID = 0 Heat flow is specified = 1 Temperature is specified IDF Time function identification number T Initial temperature CF Time function coefficient
	3.5 MATNO, NDXY, NDZ, KS, KF (For ISMAP = 3) MATNO, NDXY, NDZ, IDH (For ISMAP =-3) MATNO Material property number NDXY Number of elements along triangular edge For wedge volume block, use negative NDXY Refer to Example problem 11 NDZ Number of elements in z-direction KS = -1 Element has high explosive solid phase = 0 Element has solid phase > 0 Element has joint and absolute value of KS represents face designation number. KF = 0 Element has fluid phase = 1 Element has no fluid phase IDH Heat generation history ID number

Note: Mesh Control Data on File DV-GP.DAT

To control mesh generation, users can change the values in file DV-GP.DAT in the directory C:\SMAP\CT\CTDATA.

1. Variables Controlling Coinsident Nodes
   RLIMIT
   When the distance between two adjacent nodes is less than RLIMIT, those two nodes are assumed to be coinsident.

2. Variables Contolling Spherical Coordinate
   SDCLOSE, SDTOL, SDZERO
   When the angle of block corner node reaches SDCLOSE (degree), program will set 360 degrees. The tolerance angle is SDTOL (degree). When the angle of block corner node is greater than (360-SDZERO), program will set zero degree.

3. Variables Contolling Cylindrical Coordinate
   CDCLOSE, CDTOL, CDZERO
   When the angle of block corner node reaches CDCLOSE (degree), program will set 360 degrees. The tolerance angle is CDTOL (degree). When the angle of block corner node is greater than (360-CDZERO), program will set zero degree.

4. For spherical block having the angle of longitude greater than π and for the cylindrical block occupying more than two quadrants, the block node numbers referring to the origin should be prefixed by negative sign.

5. Current Default Values
   RLIMIT = 0.001
   SDCLOSE = 359.1 SDTOL = 0.001 SDZERO = 0.001
   CDCLOSE = 359.1 CDTOL = 0.001 CDZERO = 0.001

Note: Boundary Conditions

Boundary conditions at nodes are generated based on following rules:

1. Default conditions are applied first based on block type
2. Default conditions can be overrided by specifying IBTYPE = 1
3. Higher IBTYPE overrides lower IBTYPE in a given block
4. Each block number defined later governs conditions along the block interface

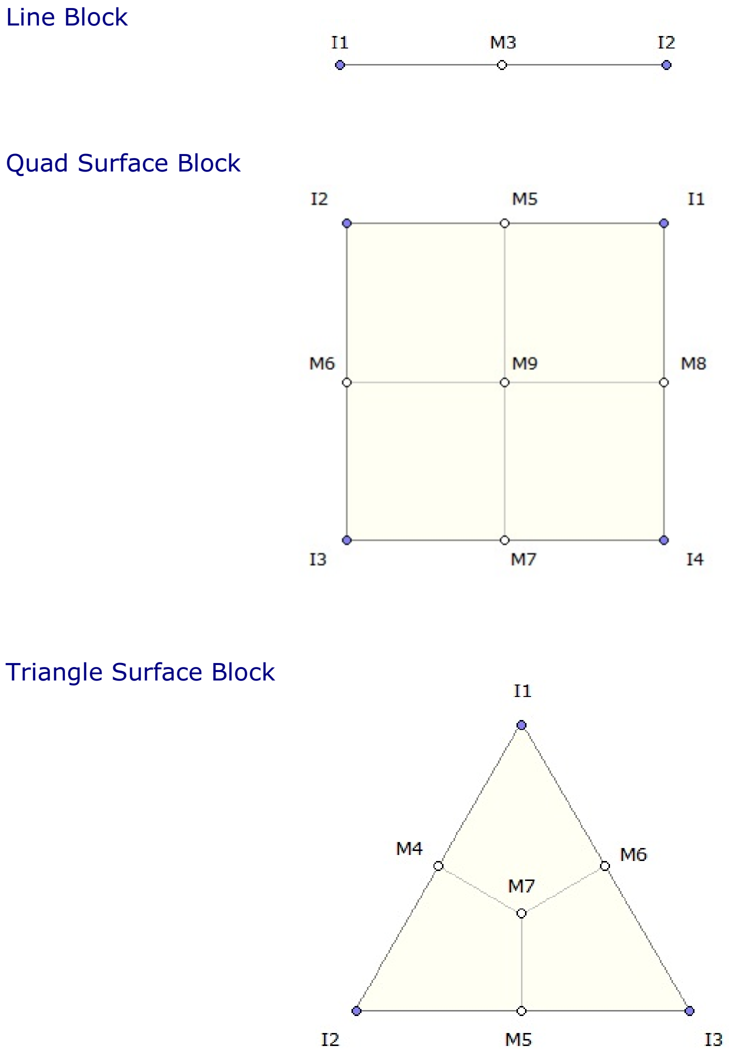


Figure 7.22 Block index for PRESMAP-GP

Hexahedron Volume Block

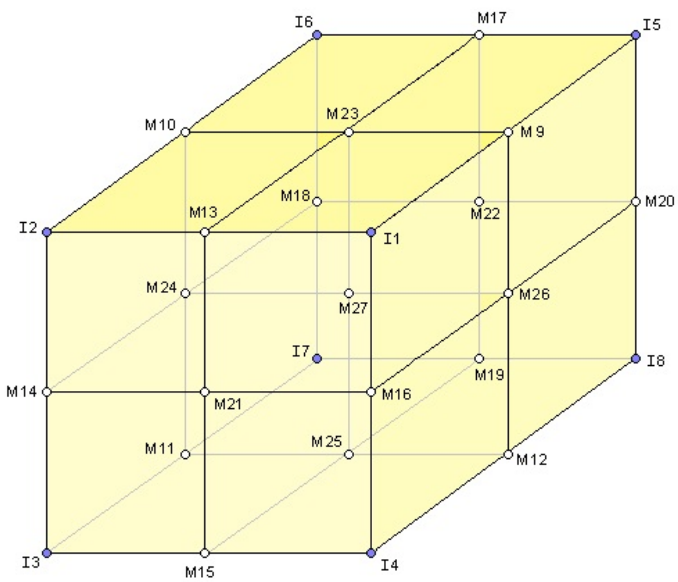


Figure 7.22 Block index for PRESMAP-GP (Continued)

Prism Volume Block

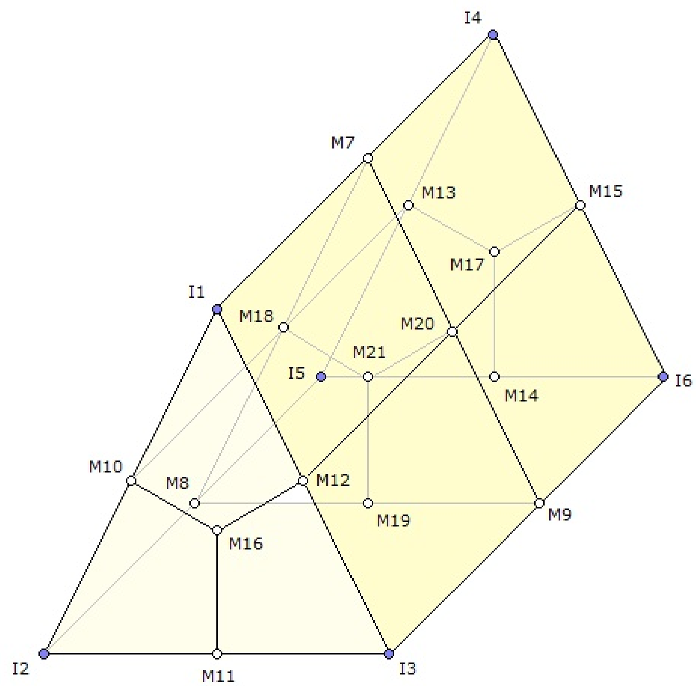


Figure 7.22 Block index for PRESMAP-GP (Continued)

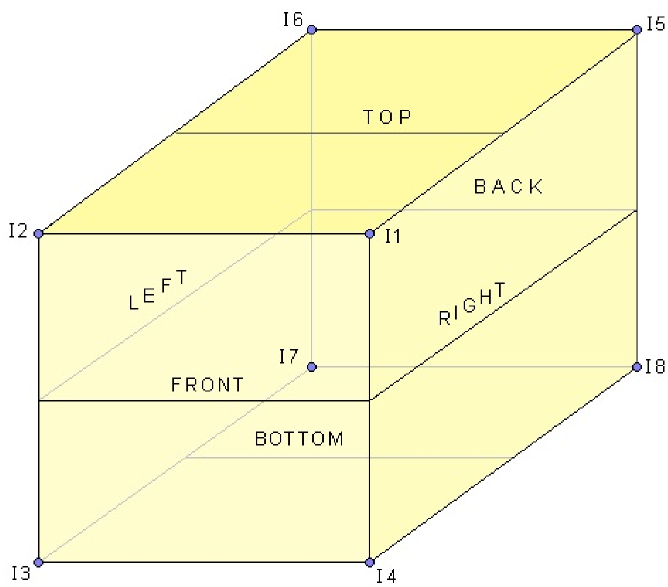


Figure 7.23 Boundary surface designation for Hexahedron Volume Block

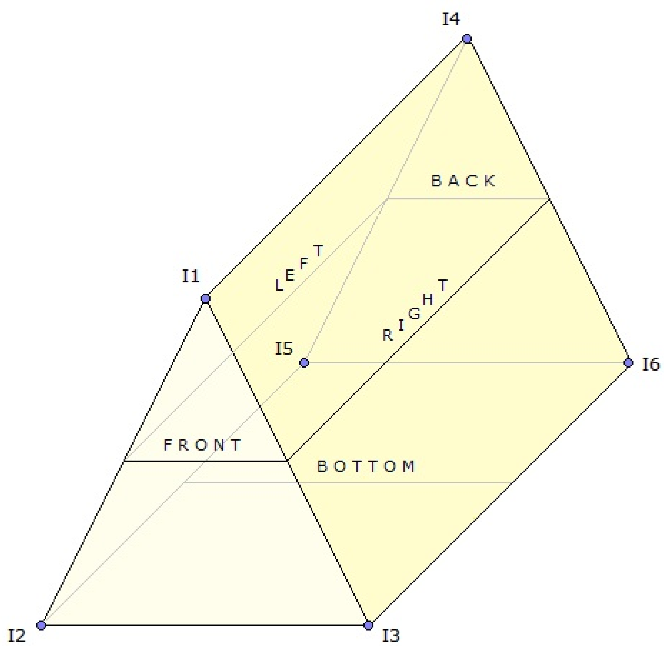


Figure 7.24 Boundary surface designation for Prism Volume Block

JOINT-2D

User's Manual

Card Group	Input Data and Definitions
1	1.1 TITLE TITLE Any title of up to 80 characters
General Information	1.2 AllJoint, ThicAJ AllJoint = 0 Generates Joint Elements along the All Interfaces between Continuum Elements. Cards 2, 3, and 4 are not used. = 1 Generates Joint Elements for the Material Numbers of Continuum Elements as specified in Cards 2 and 3. Card 4 is not used. = 2 Generates Joint Elements for the Element Surface Numbers of Continuum Elements as specified in Card 4. Cards 2 and 3 are ignored. ThicAJ Thickness used for AllJoint = 0. To Run JOINT-2D **Method 1** SMAP-2D > Run > Mesh Generator > PreSmap > Joint Specify input and output file names shown on the screen. **Method 2** 1. Select SMAP-2D > Setup > PLOT 3D Specify Joint Thickness View Factor which is greater than 0.0 Example: Joint Thickness View Factor = 1.0 2. Select SMAP-2D > Mesh > F.E. Mesh > Open This wil open Mesh File of Continuum Elements. Input file Joint.inp should exist in the Working Directory. Output File JointedMesh.Mes is shown in Working Directory.

Card Group		Input Data and Definitions
AllJoint = 1: Internal / Boundary Joint Generation	Internal Joint Generation	2.1 NumIJ, ThicIJ NumIJ — Number of continuum materials for Internal Joint. If NumIJ = 0, go to Card 3 ThicIJ — Thickness of Internal Joints
		2.2 NumIJ Cards: $MatIJ_1$ $InnerBeam_1$ $OuterBeam_1$ $MatIJ_2$ $InnerBeam_2$ $OuterBeam_2$ - - - MatIJ — Material property number of continuum element for Internal Joints (See Fig. 1) InnerBeam = 0 Do not include = 1 Include Inner Beam element OuterBeam = 0 Do not include = 1 Include Outer Beam element
	Boundary Joint Generation	3.1 NumBJ, ThicBJ, InterfaceJoint NumBJ — Number of continuum materials for Boundary Joint. If NumBJ = 0, go to Card 4 ThicBJ — Thickness of Boundary Joints. If negative, inside continuum elem. contacts joint face InterfaceJoint = 0 Do not include = 1 Include Interface Joint Element
		3.2 NumBJ Cards: $MatBJ_1$ $InnerBeam_1$ $OuterBeam_1$ $MatBJ_2$ $InnerBeam_2$ $OuterBeam_2$ - - - MatBJ — Material property number of continuum element for Boundary Joints (See Fig. 1) InnerBeam = 0 Do not include = 1 Include Inner Beam element OuterBeam = 0 Do not include = 1 Include Outer Beam element

Card Group	Input Data and Definitions
4 AllJoint = 2 : Surface Joint Generation	4.1 NumSJG NumSJG — Number of Groups for Surface Joints If NumSJG = 0, end of data
	4.2 NumSJG Cards: $NumSJ_1$ $ThicSJ_1$ / $NumSJ_2$ $ThicSJ_2$ / - - / - - $NumSJ_i$ — Number of element surfaces in Group i $ThicSJ_i$ — Thickness of Surface Joint in Group i
	For Each Surface Joint Group — 4.3 $NumSJ_i$ Cards: $ElementNo_1$ $SurfaceNo_1$ / $ElementNo_2$ $SurfaceNo_2$ / - - / - - ElementNo — Continuum Element No SurfaceNo — Continuum Element Surface No where Surface Joint is generated Note: To take new node number for corner contact element, set SurfaceNo = 0 Refer to page 4-67 of SMAP-2D User's Manual for Element Surface designation

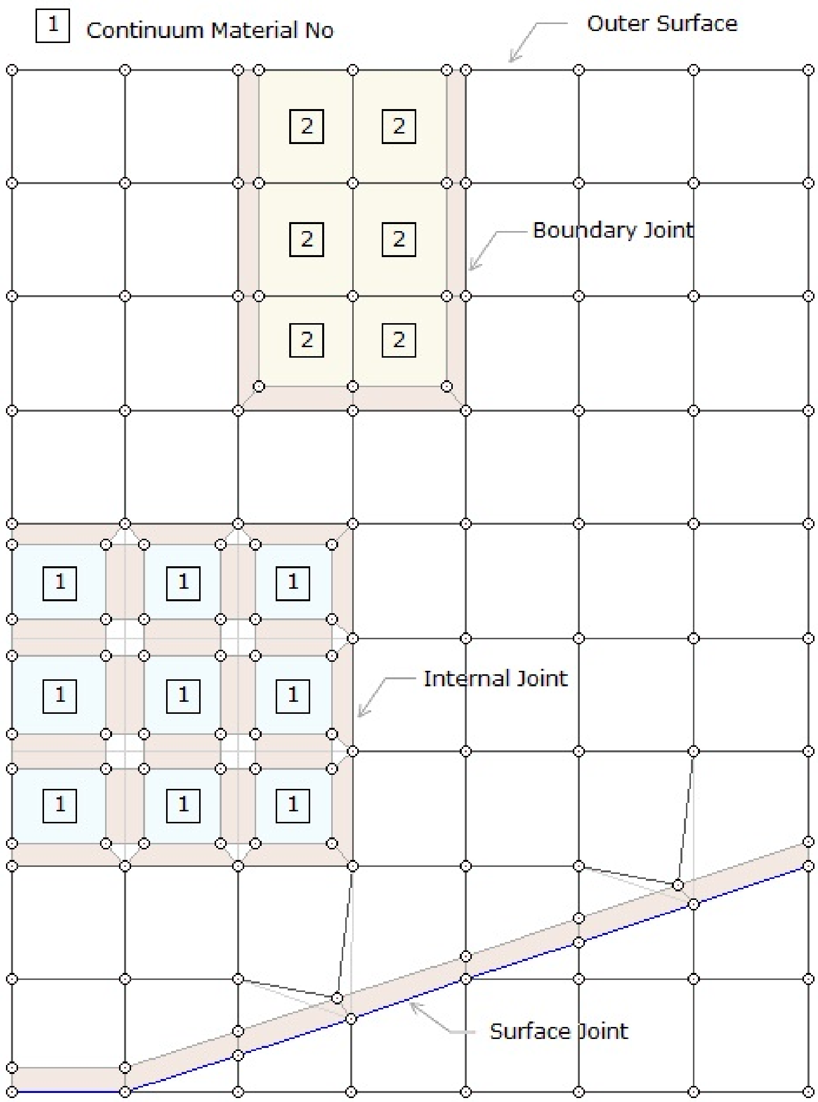


Figure 1 Joint element generation

ADDRGN
User's Manual

8.1 Introduction

ADDRGN is the pre-processing program which has the following two basic functions:

- Combine two different meshes
- Modify existing meshes

A problem geometry can be composed of a number of regions. Parts of the problem geometry can be generated using the PRESMAP programs described in Section 7. Then ADDRGN is used to combine two different regions (Region A and Region B). When Region B is added to Region A to make Combined Region, following restrictions are applied:

- Element numbers for Region A and Region B should be continuous
- Only those node numbers for Region B are modified to be consistent with the Region A, but element numbers for both regions do not change.

Though the program ADDRGN combines only two regions at a time, users can apply ADDRGN many times to assemble all the different regions.

ADDRGN can also be used to modify the existing meshes:

- Change coordinates
- Change boundary codes
- Cut elements
- Change material numbers

ADDRGN-2D deals with two dimensional meshes and ADDRGN-3D deals with three dimensional meshes.

ADDRGN-2D has an additional powerful feature which is very useful to generate meshes for complicated underground structures. This special feature modifies the existing meshes such that new structures can be easily added by simply specifying the geometries and material properties of structures. It can even generate a base mesh and then add new structures (IMOD=2).

ADDRGN-2D

User's Manual

Card Group	Input Data and Definitions
1 Mode Type	1.1 IMOD, JK IMOD = 0 Add Region B to Region A = 1 Modify existing mesh = 2 Generate base mesh and then modify. Generated base mesh is saved as BMESH.Dat =-1 Same as IMOD = 0 except it uses DOF of Region B mesh along the interface JK 1 (T2), 2 (S2), 3 (2D), 9 (W2)
2 Adding Region B to Region A (IMOD = 0)	2.1 FILEA FILEB FILEC FILEA Input file name containing Region A mesh FILEB Input file name containing Region B mesh FILEC Output file name to store Combined Region mesh When combining Region B mesh to Region A mesh, only Region B node numbers are changed. Element numbers for Region A and Region B should be continuous, otherwise element numbers are automatically reordered by program.
	2.2 INTERFACE INTERFACE = 0 Interface is found automatically = 1 Interface is specified by user
	2.3 Required only for INTERFACE = 1 NODE $NODA_1$, $NODA_2$, ..., $NODA_{NODE}$ $NODB_1$, $NODB_2$, ..., $NODB_{NODE}$ NODE Number of interface nodes. $NODA_i$ Interface node numbers in Region A $NODB_i$ Interface node numbers in Region B Note: $NODB_i$ should be the same location as $NODA_i$

Card Group	Input Data and Definitions
3 Modifying Existing Mesh (IMOD =1)	3.1 FILEA FILEM FILEA Input file name containing existing mesh FILEM Output file name to store modified mesh
	3.2 NSNEL, NSNODE, NBNEL, NTNEL NSNEL New starting continuum element number NSNODE New starting node number NBNEL New starting beam element number NTNEL New starting truss element number Note: NBNEL & NTNEL are used for IEDIT = 0, 1, 6
	3.3 IEDIT, MC_1, MC_2, MC_3, MB, MT IEDIT = 0 Change coordinates = 1 Change boundary codes = 2 Cut elements = 3 Change material numbers = 4 Build user-defined curves and material zones = 6 Change element index order MC Continuum material number to be kept MB Beam material number to be kept MT Truss material number to be kept Note: MC, MB, and MT are applicable only for IEDIT = 2 and 3

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Changing Coordinates (IEDIT = 0)	3.3.1.1 X_o, Y_o, X_{oNew}, Y_{oNew} X_o, Y_o Reference origin X_{oNew}, Y_{oNew} New origin
		3.3.1.2 X_{scale}, Y_{scale} X_{scale}, Y_{scale} Scale factors for X, Y coordinates Note: New coordinates $X_{(new)}$ and $Y_{(new)}$ are computed as follows: $X_{(new)} = X_{oNew} + (X-X_0)\ X_{scale}$ $Y_{(new)} = Y_{oNew} + (Y-Y_0)\ Y_{scale}$

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Changing Boundary Codes (IEDIT = 1)	3.3.2.1 IRANGE IRANGE = 0 Range specified by coordinates = 1 Range specified by node numbers = 2 Range specified by line strip = 3 Range specified by material numbers
		3.3.2.2.1 Required only for IRANGE = 0 X_{start}, Y_{start}, X_{end}, Y_{end} X_{start}, Y_{start} Coordinates for lower left boundary X_{end}, Y_{end} Coordiantes for upper right boundary
		3.3.2.2.2 Required only for IRANGE = 1, 2, 3 NODE NOD_1, NOD_2, ..., NOD_{NODE} NODE Number of nodes/materials to be specified NOD_i Node/Material number (Note 1 in page 8-7) Line strip is defined counterclockwise. For IRANGE = 3, Nodes refer to Material numbers.
		3.3.2.3 INSIDE (Not applicable for IRANGE= 3) INSIDE = 0 Apply inside of range = 1 Apply outside of range
		3.3.2.4 ISX, ISY, IFX, IFY, IRZ (SMAP-2D) IDX, IDY, IDT (SMAP-S2) ID, IDF (SMAP-T2) ISX, ISY X and Y DOF for skeleton motion IFX, IFY X and Y DOF for relative motion IRZ Z DOF for beam rotation IDX, IDY X and Y DOF for skeleton motion IDT Z DOF for beam rotation ID Heat flow (0), Temperature (1) specified IDF Time history identification number

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Cutting Elements (IEDIT = 2)	3.3.3.1 IRANGE IRANGE = 0 Range specified by coordinates = 1 Range specified by element numbers
		3.3.3.2.1 Required only for IRANGE = 0 X_{start}, Y_{start}, X_{end}, Y_{end} X_{start}, Y_{start} Coordinates for lower left boundary X_{end}, Y_{end} Coordinates for upper right boundary
		3.3.3.2.2 Required only for IRANGE = 1 NOEL NEL_1, NEL_2, ..., NEL_{NOEL} NOEL Number of elements to be specified NEL_i Element number (See Note 2)
		3.3.3.3 INSIDE INSIDE = 0 Apply inside of range = 1 Apply outside of range Note 1: NOD_1, $-NOD_2$ generates from NOD_1 to NOD_2 Note 2: NEL_1, $-NEL_2$ generates from NEL_1 to NEL_2

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Change Material No (IEDIT = 3)	3.3.4 IRANGE IRANGE = 0 Range specified by coordinates = 1 Range specified by element numbers
		3.3.4.1 Required only for IRANGE = 0 X_{start}, Y_{start}, X_{end}, Y_{end} X_{start}, Y_{start} Coordinates for lower left boundary X_{end}, Y_{end} Coordiantes for upper right boundary
		3.3.4.2 Required only for IRANGE = 1 NOEL NEL_1, NEL_2, ..., NEL_{NODE} NOEL Number of elements to be specified NEL_i Element number (See Note 2 in page 8-7)
		3.3.4.3 INSIDE INSIDE = 0 Apply inside of range = 1 Apply outside of range
		3.3.4.4 MATC, MATB, MATT MATC New continuum material number MATB New beam material number MATT New truss material number Note: When new material number is zero, keep the old material number

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	3.3.5 Build User-Defined Curves and Material Zones (IEDIT = 4)	3.3.5.1 NODE NOD_1, NOD_2, ..., NOD_{NODE} NODE — Number of nodes which are not movable NOD_i — Node number
		3.3.5.2 NOEL NEL_1, NEL_2, ..., NEL_{NOEL} NOEL — Number of elements whose nodal coordinates are not movable NEL_i — Element number
		3.3.5.3 IBOUND IBOUND = 0 Do not apply = 1 Nodal coordinates outside of rectangle are not movable Required only for IBOUND = 1 X_{LEFTt}, X_{RIGHT}, Y_{BOTTOM}, Y_{TOP} X_{LEFTt}, X_{RIGHT}, Y_{BOTTOM}, Y_{TOP} — Coordinates of rectangle
		3.3.5.4 NGROUP, IGTITL X_{REF}, Y_{REF} NGROUP — Number of curve groups. X_{REF}, Y_{REF} — Coordinates of reference point IGTITL = 0 Do not specify = 1 Specify group title

Card Group			Input Data and Definitions
Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.1 GTITL (For IGTITL= 1) MTYPE, IGPOST, OVERLAY, GCOLOR, GLTYPE, GLTHIC, GHIDE GTITL Group title MTYPE = 1 Generate lines & remove within closed loop = -1 Remove elements outside closed loop = 2 Generate lines = -2 Generate slip lines with joint elements = 3 Assign new material number within the closed loop = -3 Assign new material number within the closed loop and generate slip lines with joint elements along the loop. MTYPE = 4 and -4 are the same as MTYPE=3 and -3, respectively, except that old material zone is not removed for MTYPE = 4 and -4. To make the group null, use MTYPE = 0. IGPOST Generate Post file for element activity (1) OVERLAY Overlaid over existing group mesh (1) GCOLOR Group color index number GLTYPE Group line type index number GLTHIC Group line thickness index number GHIDE Group hide (1)

Card Group			Input Data and Definitions
Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.1 For MTYPE = 1 or MTYPE = 2 LTP, LMAT For MTYPE =-2 $MATNO_{JT}$, DD_{JT}, $THIC_{JT}$, LTP_I, $LMAT_I$, LTP_O, $LMAT_O$ For MTYPE = 3 MATNO, DD, LTP, LMAT For MTYPE =-3 MATNO, DD, $MATNO_{JT}$, DD_{JT}, $THIC_{JT}$, LTP_I, $LMAT_I$, LTP_O, $LMAT_O$ For MTYPE = 4 MATNO, DD, LTP, LMAT, MATold For MTYPE =-4 MATNO, DD, $MATNO_{JT}$, DD_{JT}, $THIC_{JT}$, LTP_I, $LMAT_I$, LTP_O, $LMAT_O$, MATold DD = KF (SMAP-2D) = DEN (SMAP-S2) = IDH (SMAP-T2) DD_{JT} = KF_{JT} (SMAP-2D) = DEN_{JT} (SMAP-S2) = IDH_{JT} (SMAP-T2) For MTYPE = 4 or -4 MATold takes initial value if MATNO < 0 MATold takes MATNO + 1 if MATold = 0

Card Group			Input Data and Definitions
Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.1 MATNO — Material No for continuum element MATold — Additional MATNO for MTYPE = 4 or -4 KF = 0 Material has fluid phase = 1 Material has no fluid phase DEN — Unit weight IDH — Heat generation ID $MATNO_{JT}$ — Material No for joint element KF_{JT} = 0 Joint has fluid phase = 1 Joint has no fluid phase DEN_{JT} — Unit weight for joint element IDH_{JT} — Heat generation ID for joint element $THIC_{JT}$ — Apparent thickness of joint element LTP = 0 Do not generate = 2 Generate beam element Heat pipe (IDFNP=LFUN), T2 = 3 Generate truss element Convection (IDFNC=LFUN, IDFNT=LFUN+1), T2 = 4 External heat flow (ID=0, IDF=LFUN), T2 = 5 Temperature boun. (ID=1, IDF=LFUN), T2 LMAT — Material No for line element LTP_I, $LMAT_I$ — Subscript i refers to inner face LTP_o, $LMAT_o$ — Subscript o refers to outer face Note: For negative value of LTP, line elements take nodes in opposite face of joint element For negative value of $THIC_{JT}$, joint elements are fully connected to the surrounding continuum elements (MTYPE = -2 or -3)

Card Group			Input Data and Definitions
Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.1 Required only for IGPOST= 1 NAC, NDAC (MATold) NAC, NDAC (MATNO) NAC, NDAC ($MATNO_{JT}$) NAC, NDAC (LMAT) NAC, NDAC ($LMAT_I$) NAC, NDAC ($LMAT_o$) NAC Active step number NDAC Deactive step number Required only for IGPOST= 1 CHKBOX (Mesh) CHKBOX (Principal Stress) CHKBOX (Deformed Shape) CHKBOX (Beam) CHKBOX (Truss) CHKBOX (Contour) CHKBOX (Reference Line) CHKBOX = 0 Do not plot = 1 Plot the checked item Note: IGPOST= 1 will generate main file Group.man for element activity and post file Group.pos for PLOT-2D

Card Group			Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.2 NPOINT, MOVE, IREF, X_{LO}, Y_{LO} NPOINT — Number of points defining X and Y coordinates of segments. Point numbering is counter-clockwise MOVE = 0 — Generated coordinates are movable = 1 — Generated coordinates are not movable IREF = 0 — Do not apply = 1 — Local Origin (X_{LO}, Y_{LO}) is relative to Reference Point in Card 3.3.5.4 X_{LO}, Y_{LO} — Coordinates of Local Origin NPOINT Cards: NP_1, X_1, Y_1 NP_2, X_2, Y_2 - - - - - - NP — Point number X — X-coordinate Y — Y-coordinate

Card Group	Input Data and Definitions
3	

Modifying Existing Mesh (IMOD = 1) / Build User-Defined Curves and Material Zones (IEDIT = 4) / For Each Curve Group

3.3.5.4.3

NSEGMENT, GX, GY

- NSEGMENT — Number of segments
  If NSEGMENT is equal to NPOINT, the generated curve is closed loop.
  If NSEGMENT is less than NPOINT, the generated curve is open.
- GX, GY — Group No coordinates used in AIG

For Each Segment

3.3.5.4.3.1

SEGNO, LTYPE, NDIV, IEND

SEGNO — Segment No in sequential order

LTYPE = 1 Straight line
= 2 Elliptical line

NDIV — Number of divisions.
Use NIDV=0 for default divisions.
Use negative value to consider intermediate points as line path only.

IEND = 0 Include beginning and ending points but do not register contact information
=-1 Include beginning point
= 1 Include ending point
= 2 Same as IEND=0 but register and split
=-2 Same as IEND=2 but do not split
= 3 This segment is only for reference line

For LTYPE = 2

X_O, Y_O, R_X, R_Y, θ_b, θ_e

- X_O, Y_O — Arc Origin relative to (X_{LO},Y_{LO})
- R_X, R_Y — Radius in X and Y axis, respectively
- θ_b, θ_e — Beginning and ending angle (°)

See Figure 8.2

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	3.6 Change Element Index Order (IEDIT = 6)	3.6.1 NumMATC MAT, I_1, I_2, I_3, I_4, MATC, KS, KF (SMAP-2D) MAT, I_1, I_2, I_3, I_4, MATC, THIC, DEN (SMAP-S2) MAT, I_1, I_2, I_3, I_4, MATC, IDH (SMAP-T2) NumMATC — Number of continuum materials MAT — Material number I_1, I_2, I_3, I_4 — Element corner index numbers MATC — New material property number KS, KF, THIC, DEN. IDH — Refer to Mesh File user manual
		3.6.2 NumSECB SEC, I, J, MSEC, K NumSECB — Number of beam sections SEC — Section number I, J — Element corner index numbers MSEC — New material section number K — New reference node number
		3.6.3 NumMATT MAT, I, J, MATT, K NumMATT — Number of truss materials MAT — Material number I, J — Element corner index numbers MATT — New material property number K — New reference node number Note: Index numbers are required as input. To keep the existing value, set it to -10.

Card Group	Input Data and Definitions
4 Generate Base Mesh and then Modify (IMOD = 2) See Figure 8.1	4.1 NBX, NBY, IB_LEFT, IB_RIGHT, IB_TOP, IB_BOTTOM NBX — Number of blocks in X direction NBY — Number of blocks in Y direction IB = 0 Free boundary = 1 Roller boundary
	4.2 X_O, Y_O, Y_{WT} X_O, Y_O — Origin of X and Y coordinates Y_{WT} — Y coordinate of water table (SMAP-2D) Initial temperature (SMAP-T2)
	4.3 NBX Cards [W_1, ΔX_1, α_{X1} / W_2, ΔX_2, α_{X2} / - - - / - - -] W_i — Horizontal length of block Δx_i — Minimum horizontal element length α_{Xi} = 0.5 Element length is constant = 0.3 Element length is growing from left to right =-0.3 Element length is growing from right to left
	4.4 NBY Cards [H_1, ΔY_1, α_{Y1} / H_2, ΔY_2, α_{Y2} / - - - / - - -] H_i — Vertical length of block ΔY_i — Minimum vertical element length α_Y = 0.5 Element length is constant = 0.3 Element length is growing from top to bottom =-0.3 Element length is growing from bottom to top
	4.5 IGMOD IGMOD = 0 Do not modify = 1 Modify generated base mesh If IGMOD = 1, go to Card 3.1

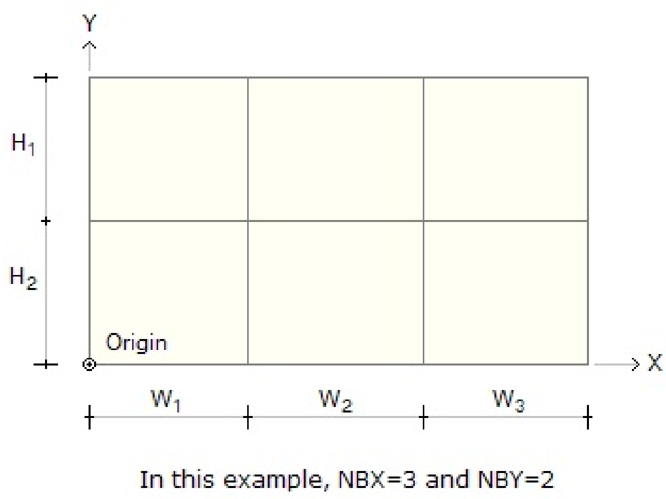


Figure 8.1 Layout of Base Mesh

Case	θ_b	θ_e
1	30 °	310 °
2	310 °	30 °
3	-50 °	30 °
4	30 °	-50 °

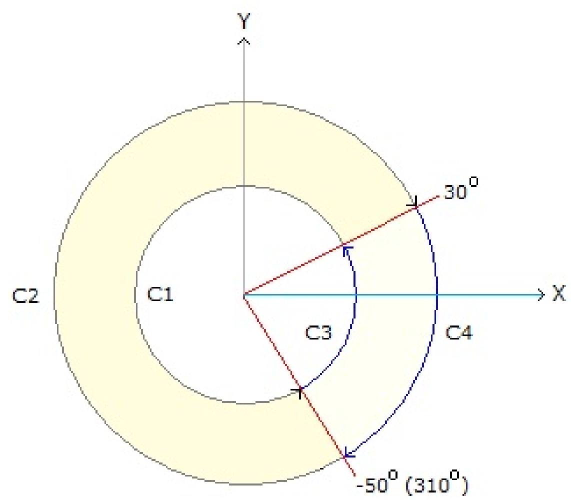


Figure 8.2 Examples of arc specification

Supplement Program

9.1 Introduction

Supplement programs contain supporting programs which are useful to prepare input data for pre-and main-processing programs and can be accessed through Run → Mesh Generater → Supplement menu.
Currently, there are five programs available:
EDIT, XY, CARDS, SHRINK FILE and CUDSS.

EDIT is used to run text editor.

XY computes coordinates of mid points, cross points, or normal points.

CARDS generates Element Activity data in Card Group 8 in Section 4.4 Main File.

SHRINK FILE removes extra blank spaces before carriage return.
This will reduce the size of the file.

CUDSS simulates cyclic undrained direct simple shear test.

9.2 EDIT

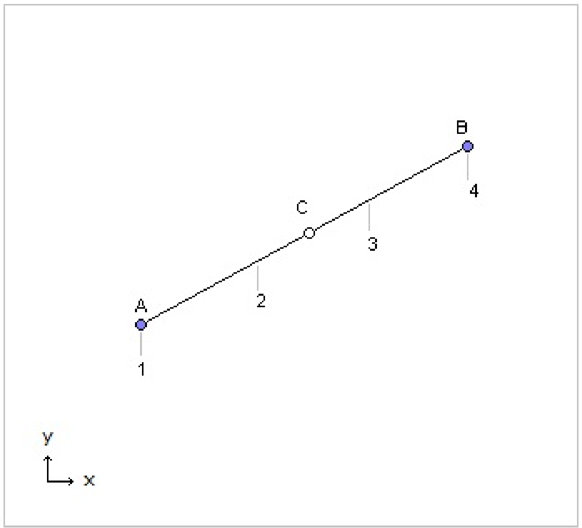
EDIT uses Windows text editor Wordpad to creat, modify, or list file.

9.3 XY

Program XY can be used to compute midpoints, intersection points and normal points of straight line and circular arc. The program is useful to construct the block diagrams of the problem geometry.

To run program XY, simply select XY from SUPPLEMENT Menu and follow instructions shown on the screen.

NF = 1 Compute Midpoint on Straight Line

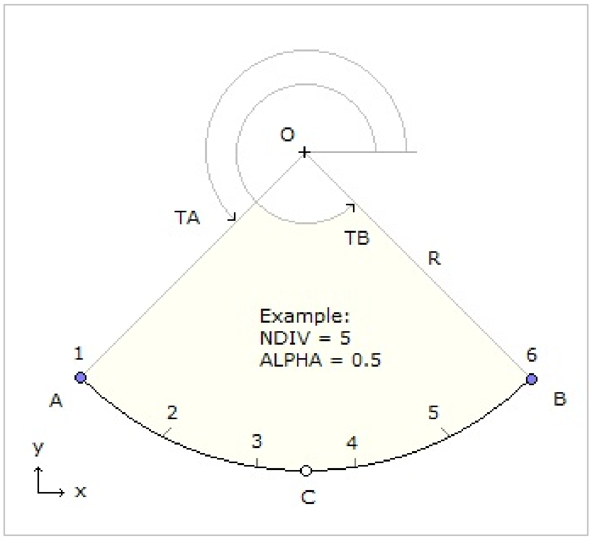


Example: NDIV = 3 and ALPHA = 0.5

INPUT:
XA, YA, XB, YB
NDIV, ALPHA

XA, YA	=	X and Y coordinates of A
XB, YB	=	X and Y coordinates of B
NDIV	=	Number of division
ALPHA	=	Geometric ratio

NF = 2 Compute Midpoint on Circular Arc



INPUT:

R, X_o, Y_o, TA, TB

NDIV, ALPHA

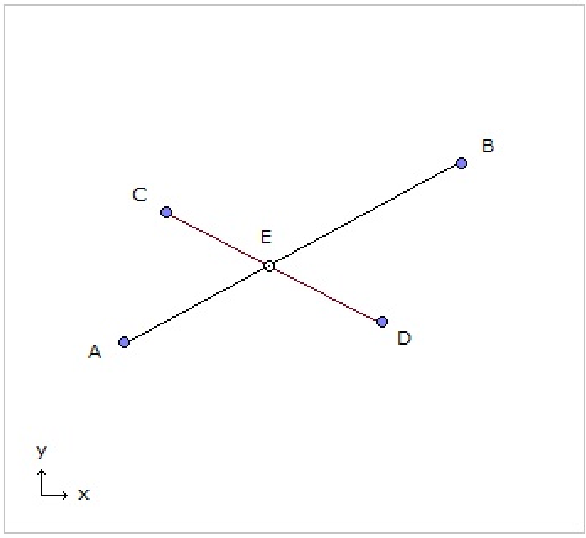
R	=	Radius
X_o, Y_o	=	X and Y coordinates of origin O
TA,TB	=	Angles (degrees) of A and B
NDIV	=	Number of division
ALPHA	=	Geometric ratio

If ALPHA = 0.5, midpoint C is located in half way between A and B

If ALPHA < 0.5, midpoint is close to A

If ALPHA > 0.5, midpoint is close to B

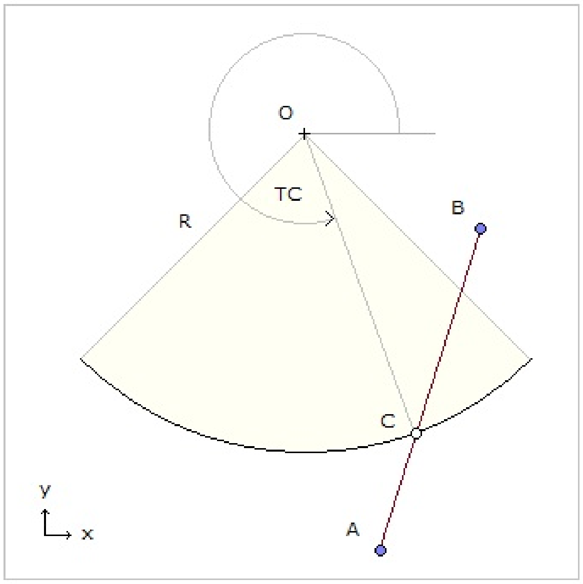
NF = 3 Compute Intersection Point of Two Straight Lines



INPUT:
XA, YA, XB, YB
XC, YC, XD, YD

XA, YA = X and Y coordinates of A
XB, YB = X and Y coordinates of B
XC, YC = X and Y coordinates of C
XD, YD = X and Y coordinates of D

NF = 4   Compute Intersection point of Arc & Straight Line



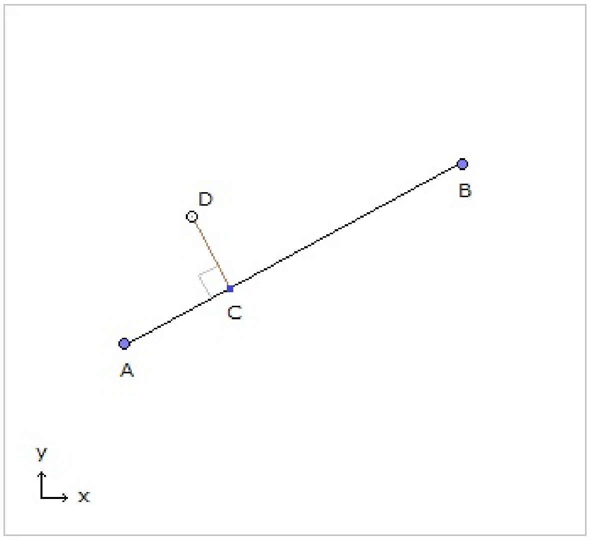
INPUT:

R, X_o, Y_o

XA, YA, XB, YB

R	=	Radius
X_o, Y_o	=	X and Y coordinates of origin O
XA, YA	=	X and Y coordinates of point A
XB, YB	=	X and Y coordinates of point B

NF = 5 Compute Points Normal to Straight Line



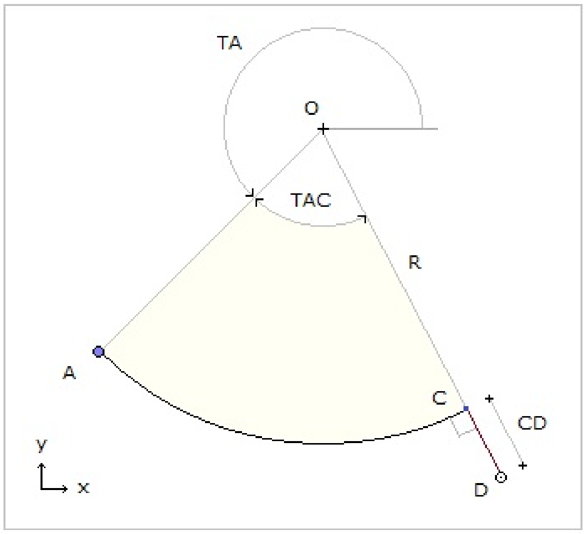
INPUT:

XA, YA, XB, YB

AC, CD

XA, YA	=	X and Y coordinates of A
XB, YB	=	X and Y coordinates of B
AC	=	Distance between A and C
CD	=	Distance between C and D

NF = 6 Compute Points Normal to Circular Arc



INPUT:

R, X_o, Y_o, TA

TAC, CD

R	=	Radius
X_o, Y_o	=	X and Y coordinates of origin O
TA	=	Angle (degree) of A
TAC	=	Angle (degree) between A and C
CD	=	Distance between C and D

9.4 CARDS

Program CARDS is included to aid for users to prepare SMAP-3D input cards. Currently, there is only one routine which generates Element Activity in Card Group 8 in Section 4.4 Main File.

You are asked to type in following input data to generate element activity and deactivity;

NEL (start)   NEL(end)   NAC   NDAC

where

NEL (start)	Starting element number
NEL (end)	Ending element number
NAC	Load step at which elements from NEL(start) to NEL(end) are activated.
NDAC	Load step at which elements from NEL(start) to NEL(end) are deactivated.

Generated element activity data will be written in the output file you specified.

9.5 SHRINK FILE

SHRINK FILE is included to remove extra blank spaces before carriage return. This will reduce the size of the file where blank spaces are existing before the carriage return.

9.6 CUDSS

CUDSS is to simulate cyclic undrained direct simple shear test. PM4Sand material model is used to represent skeleton behavior, developed by Boulanger, R. W. And ziotopoulou, k. (Version 3.1).

Input File CUDSS.inp for PM4Sand Material Model

<table>
<tr><th>Card Group</th><th>Cyclic Undrained Direct Simple Shear Simulation</th></tr>
<tr><td rowspan="4">PM4Sand Material Model</td><td>1.0
Title

Title — Title</td></tr>
<tr><td>2.0
σ_{vo}' K_o α_s

σ_{vo}' — Initial effective vertical stress
K_o — Coefficient of earth pressure at rest
α_s — Initial static shear stress ratio : $\alpha_s = \tau_s / \sigma_{vo}'$
where τ_s is initial static shear stress</td></tr>
<tr><td>3.0
CSR γ_{max}

CSR — Cyclic stress ratio : $CSR = \tau_p / \sigma_{vo}'$
where τ_p is cyclic peak shear stress
γ_{max} — Maximum cutoff shear strain</td></tr>
<tr><td>4.0
NCYCLE $\Delta\gamma$

NCYCLE — Maximum number of cycles
$\Delta\gamma$ — Shear strain increment (Default 1.0e-05)</td></tr>
</table>

<table>
<tr><th>Card Group</th><th>Cyclic Undrained Direct Simple Shear Simulation</th></tr>
<tr><td>5

PM4Sand Material Model</td><td>5.3.2.4.21

For MODELNO = 21 [PM4Sand Model]
D_R G_o h_{po} p_a N_s

Secondary Parameters (Skip these cards for N_s = 1)
h_o e_{max} e_{min} n^b n^d A_{do}
z_{max} c_z C_e φ_{cv} v_o C_{GD}
C_{DR} c_{kaf} Q R m $F_{sed.min}$ p_{sed}

D_R Apparent relative density (Fraction)
G_o Shear modulus coefficient
h_{po} Contraction rate parameter
p_a Atmospheric pressure (10.33 for stress unit t/m^2)
N_s Secondary parameter specification: 0 = Yes, 1 = No

h_o Control parameter for ratio of plastic to elastic modulus
e_{max} Maximum void ratio (Default 0.8)
e_{min} Minimum void ratio (Default 0.5)
n^b Control parameter for dilatancy & peak friction angle
n^d Control parameter for transition from contr. to dilation
A_{do} Bolton’s dilatancy parameter

z_{max} Maximum allowable fabric dilatancy tensor z
c_z Control parameter when fabric effects get important
C_e Control parameter for adjusting strain accumulation rate
φ_{cv} Critical state effective friction angle (Default 33˚)
v_o Poisson’s ratio (Default 0.3)
C_{GD} Factor for shear modulus degradation (Default 2.0)

C_{DR} Control parameter for rotated dilatancy surface
c_{kaf} Control parameter for effects of sustained shear stress
Q, R Parameters for Bolton’s empirical critical state line
m Parameter defining size of yield stress (Default 0.01)
$F_{sed.min}$ Parameter for post-shaking elastic modulus reduction
p_{sed} Mean effective stress for post-shaking reconsolidation

Set -1 for default values of secondary model parameters.

For description, refer to Boulanger, R. W. And ziotopoulou, k. PM4Sand (Version 3.1): A Sand Plasticity Model for Earthquake Engineering Applications, Report No UCD/CGM-17/01, Dept. of Civil & Env. Eng., U. of Cal., Davis, CA, 109 pp.</td></tr>
</table>

File Conversion

10.1 Introduction

PRESMAP programs described in Section 7 generate Mesh Files which contain the geometric information of structures to be analyzed. The format of SMAP-2D Mesh File is presented in detail in Section 4.3.

In this section, we will briefly discuss Mesh File conversion under Mesh Generater → File Conversion menu:

10.2 Conversion to SMAP-2D Mesh File

Following Mesh Files can be converted to SMAP-2D Mesh File format:

- Mesh Files generated for two-dimensional SMAP programs (SMAP-S2 and SMAP-T2)
- FEMAP (Version 4.1 - 4.5, neutral format)

Figure 10.1 shows File Conversion dialog box with Input Mesh File options.

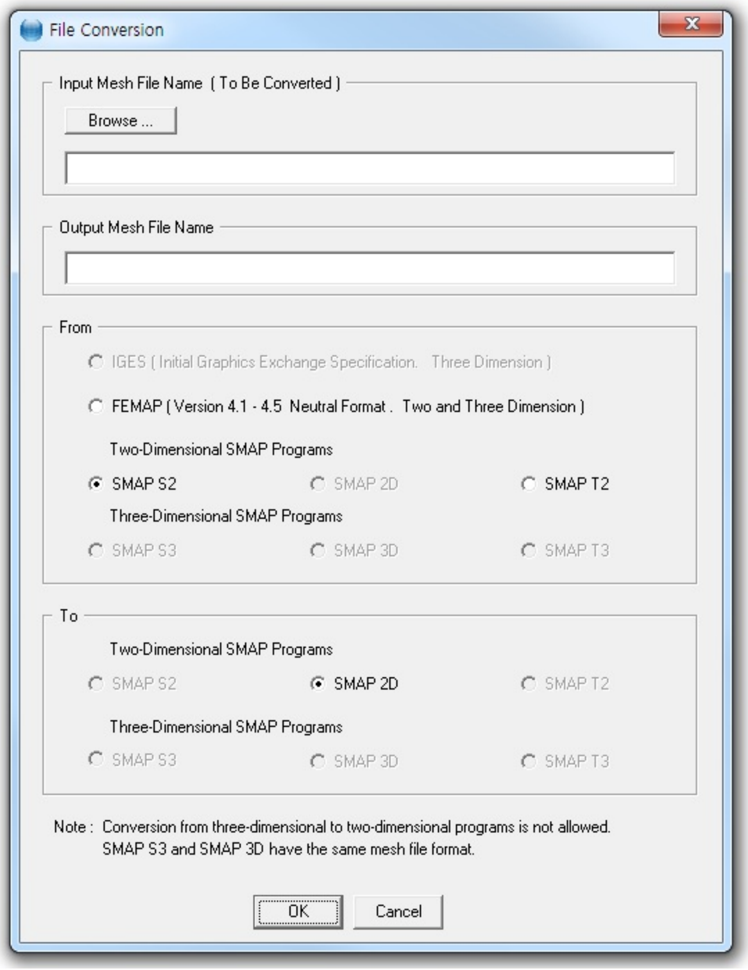


Figure 10.1 File Conversion dialog box

LOAD
User's Manual

11.1 Introduction

LOAD is the pre-processing program which generates nodal values of external forces, specified velocities, initial velocities, accelerations and transmitting boundaries.

Before you prepare LOAD input data in this section, you should have a Mesh File generated from PRESMAP/ADDRGN programs. That is, LOAD input is referred to the geometric surfaces given in the Mesh File.

Generated LOAD output file contains load data which is compatible to the format of Card Group 9 described in Section 4.4 Main File.

LOAD-2D deals with two dimensional meshes and LOAD-3D deals with three dimensional meshes.

LOAD-2D

LDTYPE = 1 [Pressure: SMAP-2D/S2]

Card Group	Input Data and Definitions (Pressure)
1 Title & Element	1.1 TITLE TITLE Any title (Max = 60 characters)
	1.2 NCTYPE NCTYPE = 0 Axisymmetric element Y-axis is axis of symmetry = 1 Plane strain element (Thickness=1.0) = 2 Plane stress element (Thickness=1.0) = 3 Spherically symmetric element (SMAP-2D)
2 Loading Surface	2.1 NUMLS NUMLS Number of loading surfaces where external tractions are specified (Max = 20)
2.2 For Each Loading Surface	2.2.1 LSNO, LSTYPE LSNO Loading surface number LSTYPE = 0 All specified nodes = 1 Line strip = 2 Points = 3 Node group = 4 Element group
LSTYPE = 0, 1, 2	2.2.2 NUMNODE NUMNODE Number of nodes on this loading surface (Max = 9990)
	2.2.3 NOD_1, NOD_2, ..., $NOD_{NUMNODE}$ NOD_i Specified node Line strip (LSTYPE=1) is defined counterclockwise. For LSTYPE=1 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.

Card Group	Input Data and Definitions (Pressure)

2 Loading Surface

2.2 For Each Loading Surface

LSTYPE = 3 (Node Group)

2.2.4

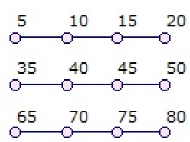
NUMNODG

NUMNODG	Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR For Each Group

NSR	Starting node number of the first row
JCR	Node number increment in a row
NJR	Number of nodes in a row
ICR	Node number increment for next row
NIR	Total number of rows



Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

LSTYPE = 4 (Element Group)

2.2.6

NUMNELG

NUMNELG	Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS For Each Group

NSR	Starting element number of the first row
JCR	Element number increment in a row
NJR	Number of elements in a row
ICR	Element number increment for next row
NIR	Total number of rows
NS	Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

Card Group		Input Data and Definitions (Pressure)
3 Pressure Function	3.1	NUMLP NUMLP Number of pressure functions (Max = 20)
	3.2 For Each Pressure Function	3.2.1 LPNO, LPTYPE LPNO Pressure function number LPTYPE = 0 Use effective surface = 1 Use actual surface Note: Effective surface is normal to force direction (Ex. Wind load)
		3.2.2 a_{xo}, a_{xx}, a_{xy} a_{xi} Coefficients defining surface traction in the x-direction. $P_x = a_{xo} + a_{xx}x + a_{xy}y$
		3.2.3 a_{yo}, a_{yx}, a_{yy} a_{yi} Coefficients defining surface traction in the y-direction. $P_y = a_{yo} + a_{yx}x + a_{yy}y$
		3.2.4 a_{no}, a_{nx}, a_{ny} a_{ni} Coefficients defining surface traction normal to surface. Acting on actual surface $P_n = a_{no} + a_{nx}x + a_{ny}y$

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Pressure)</th></tr>
<tr><td rowspan="5">4
Pressure History</td><td colspan="2">4.1
NUMLH
NUMLH Number of pressure histories (Max = 20)</td></tr>
<tr><td rowspan="4">4.2
For Each Pressure History</td><td>4.2.1
LHNO
LHNO Pressure history number</td></tr>
<tr><td>4.2.2
NUMTP
NUMTP Number of time points (Max = 1000)</td></tr>
<tr><td>4.2.3
T_1, T_2, …, T_{NUMTP}
T_i Specified time</td></tr>
<tr><td>4.2.4
C_1, C_2, …, C_{NUMTP}
C_i Pressure intensity at time T_i</td></tr>
</table>

Card Group	Input Data and Definitions (Pressure)
5 Pressure Specification	5.1 LSNO, LPNO, LHNO LSNO Loading surface number LPNO Pressure function number LHNO Pressure history number Repeat Card 5.1 until the last card (LSNO=0) is specified

LOAD-2D

LDTYPE = 2 [Velocity: SMAP-2D]

Card Group	Input Data and Definitions (Velocity)
1 Title & Element	1.1 TITLE TITLE Any title (Max = 60 characters)
	1.2 NCTYPE NCTYPE = 0 Axisymmetric element Y-axis is axis of symmetry = 1 Plane strain element (Thickness=1.0) = 2 Plane stress element (Thickness=1.0) = 3 Spherically symmetric element (SMAP-2D)
2 Loading Surface	2.1 NUMLS NUMLS Number of loading surfaces where velocities are specified (Max = 20)
2.2 For Each Loading Surface	2.2.1 LSNO, LSTYPE LSNO Loading surface number LSTYPE = 0 All specified nodes = 1 Line strip = 2 Points = 3 Node group = 4 Element group
LSTYPE = 0, 1, 2	2.2.2 NUMNODE NUMNODE Number of nodes on this loading surface (Max = 9990)
	2.2.3 NOD_1, NOD_2, ..., $NOD_{NUMNODE}$ NOD_i Specified node Line strip (LSTYPE=1) is defined counterclockwise. For LSTYPE=1 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.

Card Group	Input Data and Definitions (Velocity)

2 — Loading Surface

2.2 — For Each Loading Surface

LSTYPE = 3 (Node Group)

2.2.4

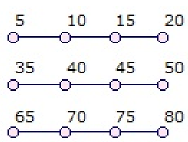
NUMNODG

NUMNODG — Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR — For Each Group

NSR	Starting node number of the first row
JCR	Node number increment in a row
NJR	Number of nodes in a row
ICR	Node number increment for next row
NIR	Total number of rows



Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

LSTYPE = 4 (Element Group)

2.2.6

NUMNELG

NUMNELG — Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS — For Each Group

NSR	Starting element number of the first row
JCR	Element number increment in a row
NJR	Number of elements in a row
ICR	Element number increment for next row
NIR	Total number of rows
NS	Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

Card Group	Input Data and Definitions (Velocity)	
3 Velocity Function	3.1 NUMLV NUMLV Number of velocity functions (Max = 20)	
	3.2 For Each Velocity Function	3.2.1 LVNO LVNO Velocity function number
		3.2.2 a_{xo}, a_{xx}, a_{xy} a_{xi} Coefficients defining velocity in x-direction $V_x = a_{xo} + a_{xx}\ x + a_{xy}\ y$
		3.2.3 a_{yo}, a_{yx}, a_{yy} a_{yi} Coefficients defining velocity in y-direction $V_y = a_{yo} + a_{yx}\ x + a_{yy}\ y$
		3.2.4 a_{no}, a_{nx}, a_{ny} a_{ni} Coefficients defining velocity normal to surface $V_n = a_{no} + a_{nx}\ x + a_{ny}\ y$

Card Group		Input Data and Definitions (Velocity)
4 Velocity History	4.1	NUMLH NUMLH Number of velocity histories (Max = 20)
	4.2 For Each Velocity History	4.2.1 LHNO LHNO Velocity history number
		4.2.2 NUMTP NUMTP Number of time points (Max = 1000)
		4.2.3 T_1, T_2, ..., T_{NUMTP} T_i Specified time
		4.2.4 C_1, C_2, ..., C_{NUMTP} C_i Velocity intensity at time T_i

Card Group	Input Data and Definitions (Velocity)
5 Velocity Specification	5.1 LSNO, LVNO, LHNO LSNO Loading surface number LVNO Velocity function number LHNO Velocity history number Repeat Card 5.1 until the last card (LSNO=0) is specified

LOAD-2D

LDTYPE = 3 [Initial Velocity: SMAP-2D]

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Initial Velocity)</th></tr>
<tr><td rowspan="2">1
Title & Element</td><td colspan="3">1.1
TITLE
TITLE — Any title (Max = 60 characters)</td></tr>
<tr><td colspan="3">1.2
NCTYPE
NCTYPE = 0 Axisymmetric element
Y-axis is axis of symmetry
= 1 Plane strain element (Thickness=1.0)
= 2 Plane stress element (Thickness=1.0)
= 3 Spherically symmetric element (SMAP-2D)</td></tr>
<tr><td rowspan="4">2
Loading Surface</td><td colspan="3">2.1
NUMLS
NUMLS — Number of loading surfaces where initial velocities are specified (Max = 20)</td></tr>
<tr><td rowspan="3">2.2
For Each Loading Surface</td><td colspan="2">2.2.1
LSNO, LSTYPE
LSNO — Loading surface number
LSTYPE = 0 All specified nodes
= 1 Line strip = 2 Points
= 3 Node group = 4 Element group</td></tr>
<tr><td rowspan="2">LSTYPE = 0, 1, 2</td><td>2.2.2
NUMNODE
NUMNODE — Number of nodes on this loading surface (Max = 9990)</td></tr>
<tr><td>2.2.3
NOD_1, NOD_2, ..., $NOD_{NUMNODE}$
NOD_i — Specified node

Line strip (LSTYPE=1) is defined counterclockwise. For LSTYPE=1 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.</td></tr>
</table>

Card Group	Input Data and Definitions (Initial Velocity)

2 — Loading Surface

2.2 — For Each Loading Surface

LSTYPE = 3 (Node Group)

2.2.4

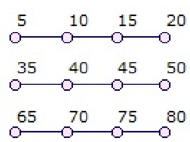
NUMNODG

NUMNODG	Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR — For Each Group

NSR	Starting node number of the first row
JCR	Node number increment in a row
NJR	Number of nodes in a row
ICR	Node number increment for next row
NIR	Total number of rows



Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

LSTYPE = 4 (Element Group)

2.2.6

NUMNELG

NUMNELG	Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS — For Each Group

NSR	Starting element number of the first row
JCR	Element number increment in a row
NJR	Number of elements in a row
ICR	Element number increment for next row
NIR	Total number of rows
NS	Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Initial Velocity)</th></tr>
<tr><td rowspan="5">3
Initial Velocity Function</td><td colspan="2">3.1
NUMLIV

NUMLIV Number of initial velocity functions (Max = 20)</td></tr>
<tr><td rowspan="4">3.2
For Each Initial Velocity Function</td><td>3.2.1
LIVNO

LIVNO Initial velocity function number</td></tr>
<tr><td>3.2.2
a_{xo}, a_{xx}, a_{xy}

a_{xi} Coefficients defining initial velocity in the x-direction
$V_{ix} = a_{xo} + a_{xx}x + a_{xy}y$</td></tr>
<tr><td>3.2.3
a_{yo}, a_{yx}, a_{yy}

a_{yi} Coefficients defining initial velocity in the y-direction
$V_{iy} = a_{yo} + a_{yx}x + a_{yy}y$</td></tr>
<tr><td>3.2.4
a_{no}, a_{nx}, a_{ny}

a_{ni} Coefficients defining initial velocity normal to the surface
$V_{in} = a_{no} + a_{nx}x + a_{ny}y$</td></tr>
</table>

Card Group	Input Data and Definitions (Initial Velocity)
4 Initial Velocity Specification	4.1 LSNO, LIVNO LSNO Loading surface number LIVNO Initial velocity function Repeat Card 4.1 until the last card (LSNO=0) is specified

LOAD-2D

LDTYPE = 4 [Acceleration: SMAP-2D]

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Acceleration)</th></tr>
<tr><td rowspan="2">1
Title & Element</td><td colspan="3">1.1
TITLE
TITLE Any title (Max = 60 characters)</td></tr>
<tr><td colspan="3">1.2
NCTYPE
NCTYPE = 0 Axisymmetric element
Y-axis is axis of symmetry
= 1 Plane strain element (Thickness=1.0)
= 2 Plane stress element (Thickness=1.0)
= 3 Spherically symmetric element (SMAP-2D)</td></tr>
<tr><td rowspan="4">2
Loading Surface</td><td colspan="3">2.1
NUMLS
NUMLS Number of loading surfaces where accelerations are specified (Max = 20)</td></tr>
<tr><td rowspan="3">2.2
For Each Loading Surface</td><td colspan="2">2.2.1
LSNO, LSTYPE
LSNO Loading surface number
LSTYPE = 0 All specified nodes
= 1 Line strip = 2 Points
= 3 Node group = 4 Element group</td></tr>
<tr><td rowspan="2">LSTYPE = 0, 1, 2</td><td>2.2.2
NUMNODE
NUMNODE Number of nodes on this loading surface (Max = 9990)</td></tr>
<tr><td>2.2.3
NOD_1, NOD_2, ..., $NOD_{NUMNODE}$
NOD_i Specified node

Line strip (LSTYPE=1) is defined counterclockwise. For LSTYPE=1 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.</td></tr>
</table>

Card Group	Input Data and Definitions (Acceleration)

2 — Loading Surface

2.2 — For Each Loading Surface

LSTYPE = 3 (Node Group)

2.2.4

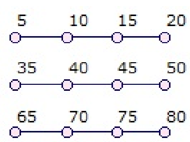
NUMNODG

NUMNODG	Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR For Each Group

NSR	Starting node number of the first row
JCR	Node number increment in a row
NJR	Number of nodes in a row
ICR	Node number increment for next row
NIR	Total number of rows



Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

LSTYPE = 4 (Element Group)

2.2.6

NUMNELG

NUMNELG	Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS For Each Group

NSR	Starting element number of the first row
JCR	Element number increment in a row
NJR	Number of elements in a row
ICR	Element number increment for next row
NIR	Total number of rows
NS	Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

Card Group		Input Data and Definitions (Acceleration)
3 Acceleration Function	3.1	NUMLA NUMLA — Number of acceleration functions (Max = 20)
	3.2 For Each Acceleration	3.2.1 LANO LANO — Acceleration function number
		3.2.2 a_{xo}, a_{xx}, a_{xy} a_{xi} — Coefficients defining acceleration in the x-direction $A_x = a_{xo} + a_{xx}x + a_{xy}y$
		3.2.3 a_{yo}, a_{yx}, a_{yy} a_{yi} — Coefficients defining acceleration in the y-direction $A_y = a_{yo} + a_{yx}x + a_{yy}y$
		3.2.4 a_{no}, a_{nx}, a_{ny} a_{ni} — Coefficients defining acceleration normal to the surface $A_n = a_{no} + a_{nx}x + a_{ny}y$

Card Group		Input Data and Definitions (Acceleration)
4 Acceleration History		4.1 NUMLH NUMLH Number of acceleration histories (Max = 20)
	4.1 For Each Acceleration History	4.2.1 LHNO LHNO Acceleration history number
		4.2.2 NUMTP NUMTP Number of time points (Max = 1000)
		4.2.3 T_1, T_2, ..., T_{NUMTP} T_i Specified time
		4.2.4 C_1, C_2, ..., C_{NUMTP} C_i Acceleration intensity at time T_i

Card Group	Input Data and Definitions (Acceleration)
5 Acceleration Specification	5.1 LSNO, LANO, LHNO LSNO Loading surface number LANO Acceleration function number LHNO Acceleration history number Repeat Card 5.1 until the last card (LSNO=0) is specified

LOAD-2D

LDTYPE = 5 [Transmitting Boundary: SMAP-2D]

Card Group	Input Data and Definitions (Transmitting Boundary)
1 Title & Element	1.1 TITLE TITLE Any title (Max = 60 characters)
	1.2 NCTYPE NCTYPE = 0 Axisymmetric element Y-axis is axis of symmetry = 1 Plane strain element (Thickness=1.0) = 2 Plane stress element (Thickness=1.0) = 3 Spherically symmetric element (SMAP-2D)
2 Loading Surface	2.1 NUMLS NUMLS Number of loading surfaces where transmitting boundaries are specified (Max = 20)
	2.2 For Each Loading Surface 2.2.1 LSNO, LSTYPE LSNO Loading surface number LSTYPE = 0 All specified nodes = 1 Line strip = 2 Points = 3 Node group = 4 Element group
	LSTYPE = 0, 1, 2 2.2.2 NUMNODE NUMNODE Number of nodes on this loading surface (Max = 9990)
	2.2.3 NOD_1, NOD_2, ..., $NOD_{NUMNODE}$ NOD_i Specified node Line strip (LSTYPE=1) is defined counterclockwise. For LSTYPE=1 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.

Card Group	Input Data and Definitions (Transmitting Boundary)

2 — Loading Surface

2.2 — For Each Loading Surface

LSTYPE = 3 (Node Group)

2.2.4

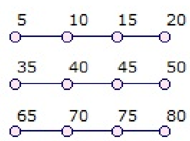
NUMNODG

NUMNODG — Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR — For Each Group

NSR	Starting node number of the first row
JCR	Node number increment in a row
NJR	Number of nodes in a row
ICR	Node number increment for next row
NIR	Total number of rows



Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

2.2.6

NUMNELG

NUMNELG — Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS — For Each Group

NSR	Starting element number of the first row
JCR	Element number increment in a row
NJR	Number of elements in a row
ICR	Element number increment for next row
NIR	Total number of rows
NS	Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

Example
NSR = 5
JCR = 5
NJR = 4
ICR = 30
NIR = 3

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Transmitting Boundary)</th></tr>
<tr><td rowspan="3">3
Material Property</td><td colspan="2">3.1
NUMMP

NUMMP   Number of different material property (Max=5)</td></tr>
<tr><td rowspan="2">3.2
For Each Material Property</td><td>3.2.1
MATNO

MATNO   Material property number</td></tr>
<tr><td>3.2.2
RO, E, V

RO   Mass density
E   Young's modulus
V   Poisson's ratio</td></tr>
</table>

Card Group	Input Data and Definitions (Transmitting Boundary)
4 Transmitting Boundary Specification	4.1 LSNO, MATNO LSNO Loading surface number MATNO Material property number For MATNO = 0, loading surface is related to continuum element surface Refer to Card 9.6.3 in SMAP-2D User's Manual Repeat Card 4.1 until the last card (LSNO=0) is specified

XY Graph
User's Manual

12.1 Introduction

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. Figure 12.1 shows schematic flow diagram of plotting simple form of Draft XY data in Table 12.1.

This Draft XY is changed into Standard XY by Converter DS. Then Standard XY can be plotted by directly PLOT XY or by EXCEL with the aid of Converter SE.

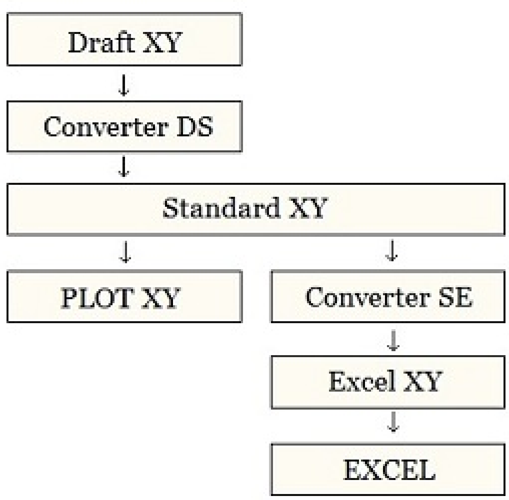


Figure 12.1 Flow diagram of plotting XY graph

Table 12.1 Draft XY Data Format

Card Group		Input Data and Definitions		
First Plot	Title	Title		(Max 50 Characters)
		Sub Title		(Max 50 Characters)
		X-Label		(Max 50 Characters)
		Y-Label		(Max 50 Characters)
	First Curve	X_1	Y_1	
		X_2	Y_2	
		-	-	
		X_n	Y_n	
		0.0	123456	(End of Curve)
		Legend 1		(Max 20 Characters)
		Legend 2		(Max 20 Characters))
	Second Curve	X_1	Y_1	
		X_2	Y_2	
		-	-	
		X_n	Y_n	
		0.0	123456	(End of Curve)
		Legend 1		(Max 20 Characters)
		Legend 2		(Max 20 Characters)
	Last Curve	X_1	Y_1	
		X_2	Y_2	
		-	-	
		X_n	Y_n	
		0.0	123456	(End of Curve)
		Legend 1		(Max 20 Characters)
		Legend 2		(Max 20 Characters)
		0.0	987654	(End of Plot)
Next Plot		Next Plot can be added using the same format as the First Plot		

12.2 New Graph

XY Graph can be created by performing the following steps:

Step 1:

Select the following menu items in SMAP:
Plot → XY → PLOT XY → New

Step 2:

Once selected, initial default file XY.dat will be opened
by Notepad as listed in Table 12.2.

Edit this default file according to the format of Draft XY Data
in Table 12.1. And then save and exit.

Step 3:

Draft XY.dat is automatically changed into Standard Form
by Converter DS as listed in Table 12.3.
Modified graph will be displayed on PLOT XY drawing board.

Step 4:

XY Graph can be further modified by Edit Dialog explained in detail in the next Section 12.3.

Table 12.2 Draft XY Data (Initial Default File XY.dat)

```
Plot No. 1
Sub Title 1
XLabel-1
YLabel-1
 0       10
 100     20
 .000000E+00  .123456E+06
Curve 1
Legend
10,     20
90,     30
 .000000E+00  .123456E+06
Curve 2
Legend
 .000000E+00  .987654E+06
Plot No. 2
Sub Title 2
XLabel-2
YLabel-2
0       100
1000    200
 .000000E+00  .123456E+06
Curve 1
Legend
100     200
900     300
.000000E+00  .123456E+06
Curve 2
Legend
.000000E+00  .987654E+06
Plot No. 3
Sub Title 3
XLabel-3
YLabel-3
0       100
1000    200
.000000E+00  .123456E+06
Curve 1
Legend
200,    200
900,    300
.000000E+00  .123456E+06
Curve 2
Legend
.000000E+00  .987654E+06
```

Table 12.3 Standard XY Data (Initial Default File XY.dat)

```
*************************************************************
*                            PLOT NO:    1                  *
*************************************************************
C Following data can be modified for plotting configuration
 TITLE     (50 CHAR) = Plot No. 1
 SUB-TITLE (50 CHAR) = Sub Title 1
 XLABLE    (50 CHAR) = XLabel-1
 YLABLE    (50 CHAR) = YLabel-1
C
 MAN.-SCALE :     IXY = 1
 LEGEND-OPT. :    ILG = 1
 TOTAL CURVE :    NLG = 2
 LEGEND-LEN : DXLEGN = 0.0
C
C IELEM= 0: no list data, list X-label & X-tick number
C        1:    list data, list X-label & X-tick number
C       -2:    node data, list node numbers only
C        2: element data, list element numbers only
C       -3:    node data, list node no, X-tick no. & X-label
C        3: element data, list elem no, X-tick no. & X-label
 EL-LIST-OPT : IELEM = 0
C
 FRAMING :        IFM = 1
 CENTERING :    ICENL = 1
 GRIDDING :     IGRID = 1
C X-coordinate data
                 XMAX = 5.0
                 NODX = 6
                   XS =  .000000E+00
                   XE =  .120000E+03
                NXDEC =-1
               XSCALE = 1.0
C
                IGENX = 0
               XDELTA = 0.0
C
                 LOGX = 0
                  NXD = 0
C Y-coordinate data
                 YMAX = 5.0
                 NODY = 6
                   YS =  .800000E+01
                   YE =  .320000E+02
                NYDEC = 2
               YSCALE = 1.0
C
                 LOGY = 0
                  NYD = 0
C Individual Curve
C                  NO :  1   2   3   4   5   6   7   8   9  10
                 HIDE =  0   0   0   0   0   0   0   0   0   0
                 LINE =  1   1   1   1   1   1   1   1   1   1
                 DASH =  1   2   3   4   5   6   7   8   9  10
                 MARK =  1   2   3   4   5   6   7   8   9  10
                 COLR =  1   2   3   4   5   6   7   8   9  10
C *************************************************************
  .000000E+00   .100000E+02
  .100000E+03   .200000E+02
  .000000E+00   .123456E+06
Curve 1
Legend
  .100000E+02   .200000E+02
  .900000E+02   .300000E+02
  .000000E+00   .123456E+06
Curve 2
Legend
  .000000E+00   .987654E+06
```

12.3 Edit Dialog

Edit Dialog in Figure 12.2 can be accessed by selecting the Edit menu in PLOT-XY.

Edit Dialog consists of following six parts:

- Titles and Labels
- General Options
- Dimensions and Scales
- Manual Scales
- Curve Data
- Command Buttons & Check Box

Refer to description in Sample Graph in Figure 12.3.

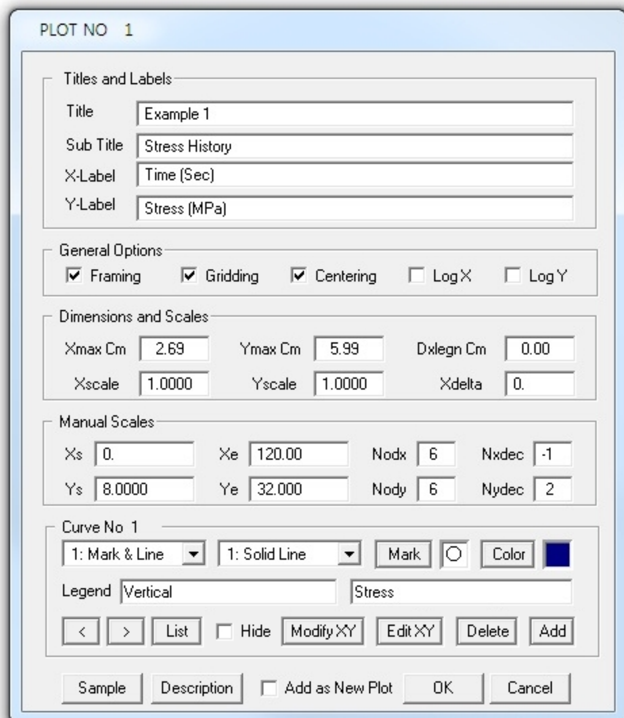


Figure 12.2
Edit dialog

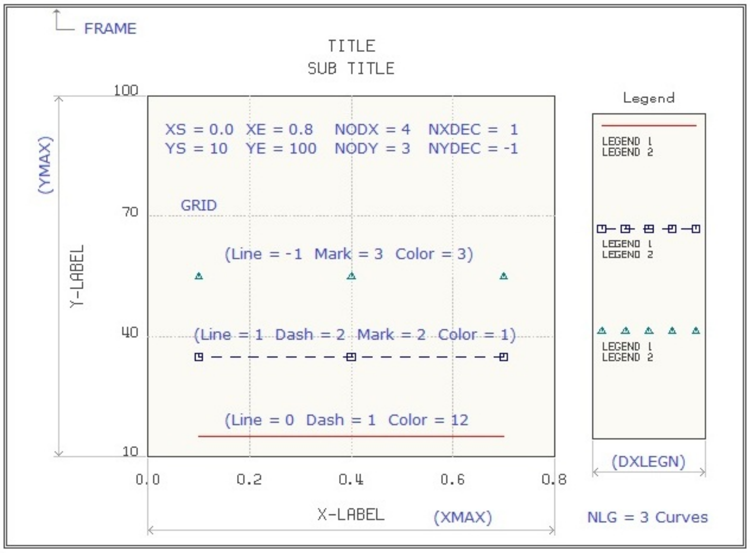


Figure 12.3 Sample graph

12.3.1 Titles and Labels

Here, you type:
Title, Sub Title, X-Label, and Y-Label.

12.3.2 General Options

Check the box for the option item to be active:

Framing	Draw Frame
Gridding	Draw Grid lines
Centering	Center Titles and X & Y Labels
Log X	Log scale in X axis
Log Y	Log scale in Y axis

12.3.3 Dimensions and Scales

Refer to description in Sample Graph in Figure 12.3.

12.3.4 Manual Scales

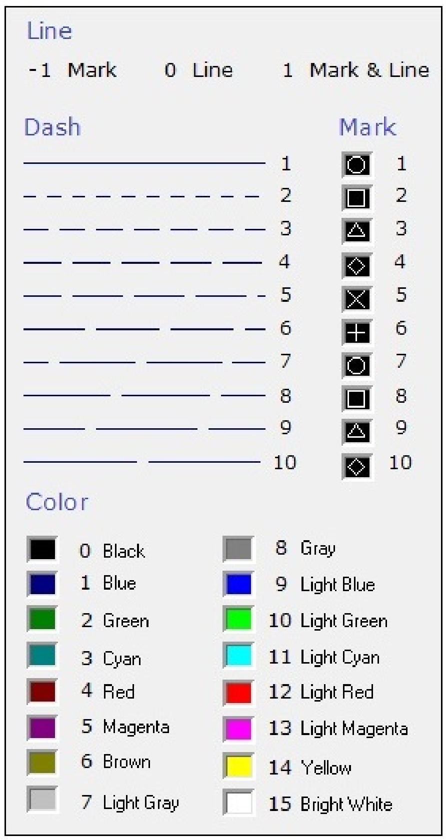
Refer to description in Sample Graph in Figure 12.3.

12.3.5 Curve Data

For each curve, you can select Line type, Dash type, Mark type, Color as in Figure 12.4, and type in Legends.

Check Hide Curve to hide the current curve.

Figure 12.4
Curve options



Curve Data has the following seven command buttons:

Back	Open previous curve
Next	Open next curve
List	List all curves as in Figure 12.5a
Modify XY	Modify current curve XY data as in Figure 12.5b
Edit XY	Edit current curve XY data
Delete	Delete current curve
Add	Add new curve to current plot

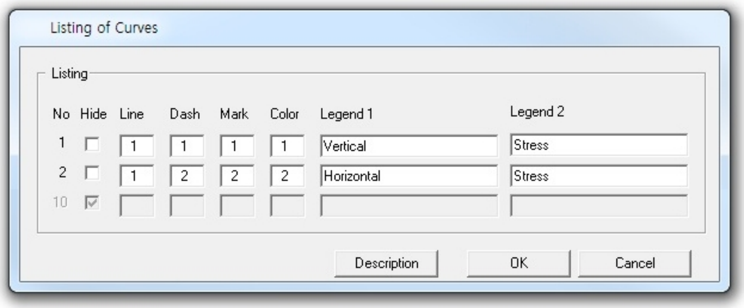


Figure 12.5a Listing of curves

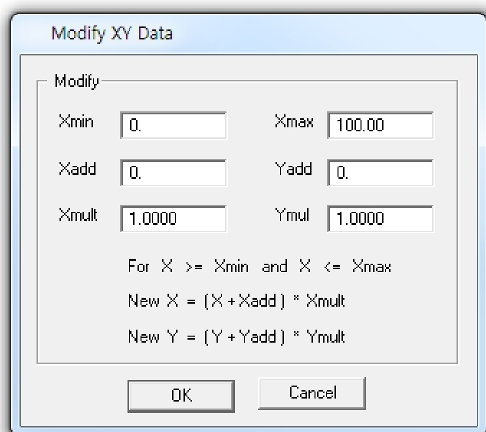


Figure 12.5b Modify current curve XY data

12.3.6 Command Buttons & Check Box

Sample	Show Sample graph in Figure 12.3
Description	Show Curve options in Figure 12.4
Add as New Plot	Copy Current plot and Add as New plot
OK	Save and exit Edit dialog
Cancel	Cancel and exit Edit dialog

12.4 Existing Graph

XY Graph can be opened by performing the following steps:

Step 1:

Select the following menu items in SMAP:
Plot → XY → PLOT XY → Open

Step 2:

If input file is Draft Form, then it will be automatically changed into Standard Form by Converter DS as listed in Table 12.3.

XY Graph will be displayed on PLOT XY drawing board.

Step 3:

XY Graph can be modified by Edit Dialog as explained in detail in the previous Section 12.3.

Refer to samples in the following directory:

C:\Smap \Smap3D \Example \XY_Graph \PLOT XY Graph Sample.docx

12.5 Excel XY Graph

Excel XY Graph can be made by performing the following steps:

Step 1:

Select the following menu items in SMAP:
Plot → XY → EXCEL → Open

Step 2:

If input file is Draft Form, then it will be automatically changed into Standard Form by Converter DS as listed in Table 12.3.

Then this Standard XY Graph will be changed into Excel Form by Converter SE and displayed on EXCEL Spreadsheet as shown in Figure 12.6.

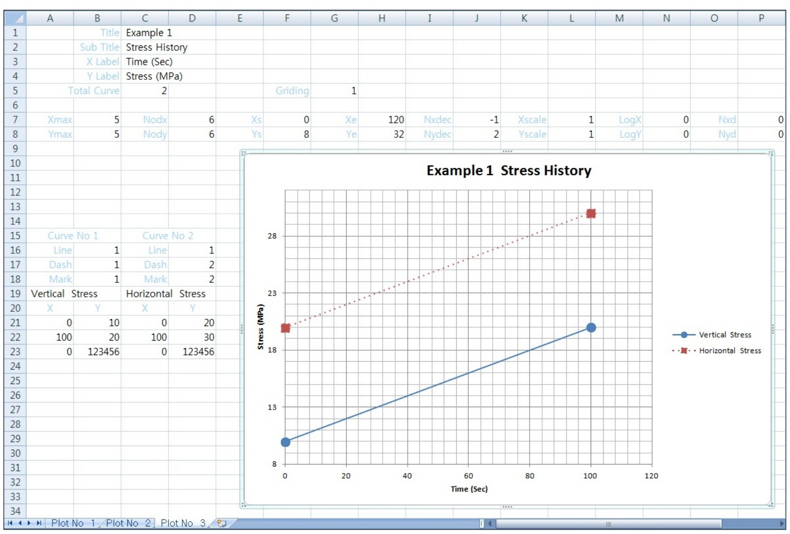


Figure 12.6 XY graph on Excel spread sheet

Notes on Excel XY Graph

Excel XY Graph can be influenced by the following input parameters in Standard Form:

Note 1: Input Parameters Not Considered

Following parameters are not considered:
Plot dimensions: XMAX , YMAX
Number of digits after decimal point: NXDEC, NYDEC

Note 2: Automatic Scaling (Xscale = 0, Yscale = 0)

For XSCALE = 0
X axis is automatically scaled and XS, XE and NODX are not used.

For YSCALE = 0
Y axis is automatically scaled and YS, YE and NODY are not used.

Note 3: Logarithmic Scaling (Logx = 1, Logy = 1)

For LOGX = 1
NODX and NXD are not used.
If XSCALE $\neq$ 0 and XS < 1 and XE > 1, XS is automatically scaled.

For LOGY = 1
NODY and NYD are not used.
If YSCALE $\neq$ 0 and YS < 1 and YE > 1, YS is automatically scaled.

Refer to samples in the following directory:
C:\Smap \Smap2D \Example \XY_Graph \Excel XY Graph Sample.pdf

12.6 SMAP Results

Figure 12.7 shows schematic flow diagram of processing SMAP Results corresponding to Card Group 12 in SMAP Post File.

This Standard Form of PlotXy.dat can be opened by either PLOT XY or EXCEL spreadsheet.

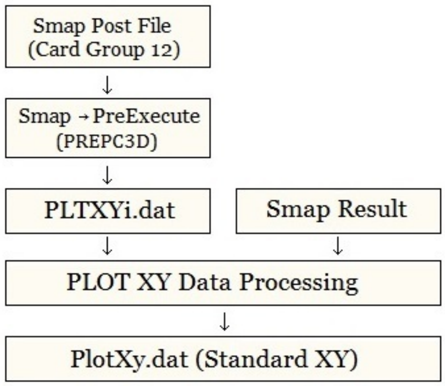


Figure 12.7 Processing SMAP results

SMAP Results can be plotted by performing following steps:

Step 1:

Select the following menu items in SMAP:
Plot → Result

Step 2:

Select PLOT XY
in Plot Menu dialog
in Figure 12.8.

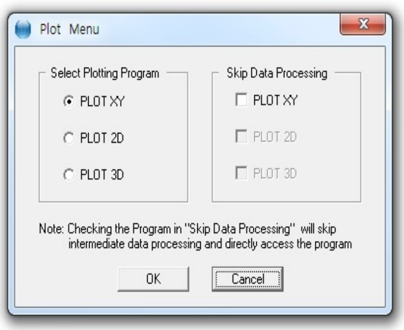


Figure 12.8
Plot menu dialog

12.6.1 PLOT XY Setup

PLOT XY Setup in Figure 12.9 can be accessed by selecting the following item in SMAP main menu.
Setup → PLOT XY

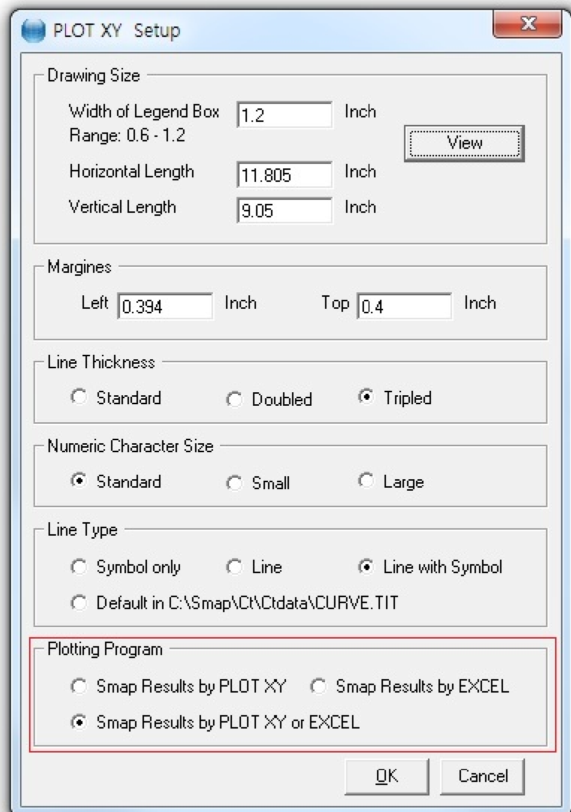


Figure 12.9 PLOT XY setup dialog

Refer to description in Sample Graph in Figure 12.4.

12.7 PlotXY Generator

PlotXY Generator is the graphical user interface which is mainly used to generate or edit Simplified Time History and Simplified Snapshot of Card Group 12 in SMAP Post File.

All different cases will be discussed in the following sections.

12.7.1 Accessing PlotXY Generator

PlotXY Generator can be accessed by selecting the following item in SMAP main menu as in Figure 12.10.
Run → PlotXY Generator → New / Open

New is used to generate new Post File.

You can edit sample input with all different cases.

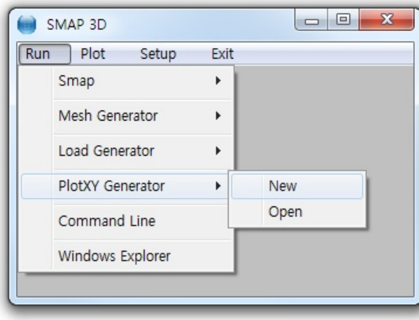


Figure 12.10
Menu for PlotXY Generator

Open is used to edit existing Post File. You can specify different output Post File name as shown in Figure 12.11.

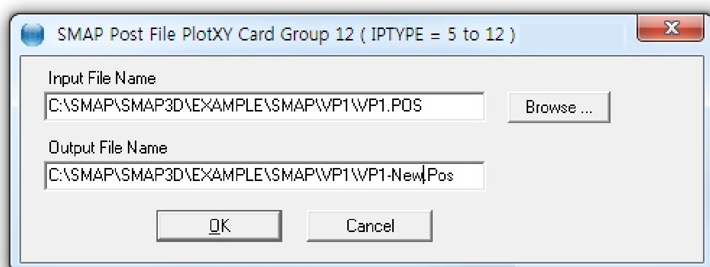


Figure 12.11 PlotXY input and output file dialog

12.7.2 Time History for a Given Element

Main Dialog for Time History of Stresses / Strains for a Given Element (IPTYPE = 5) is shown in Figure 12.12.

Element should be listed in Card 10.2.2 in SMAP Main File.
Table shows available data as in Figure 12.13.

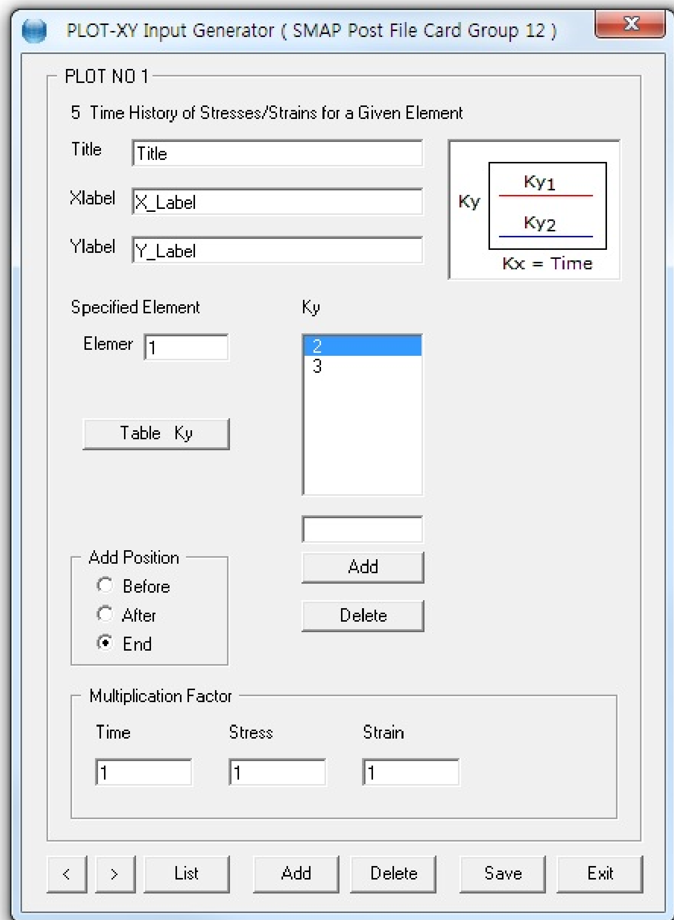


Figure 12.12 Time history for a given element

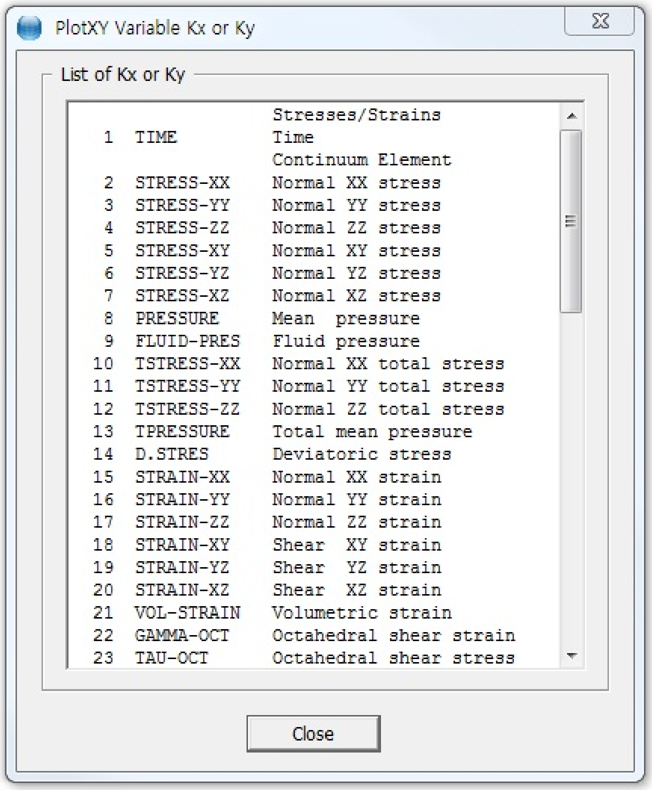


Figure 12.13 Available data for stresses / strains

Buttons at Main Dialog Bottom

Back	Show previous plot
Next	Show next plot
List	Show listing of all plots
Add	Add new plot at the end
Delete	Delete the current plot
Save	Save all updates
Exit	Save and exit

List shows summary of all plots as shown in Figure 12.14.

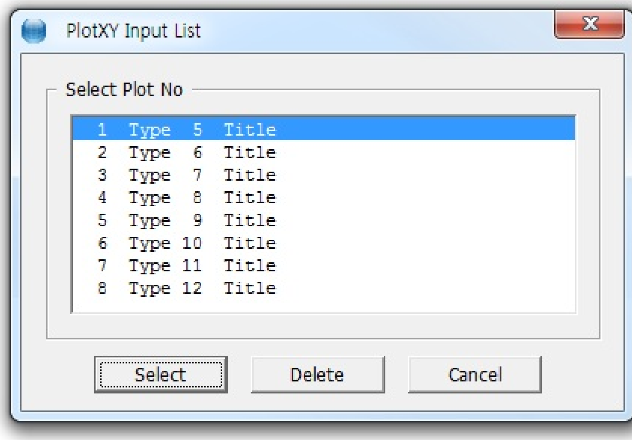


Figure 12.14 Listing of plots

Add shows new plot type to be added as in Figure 12.15.

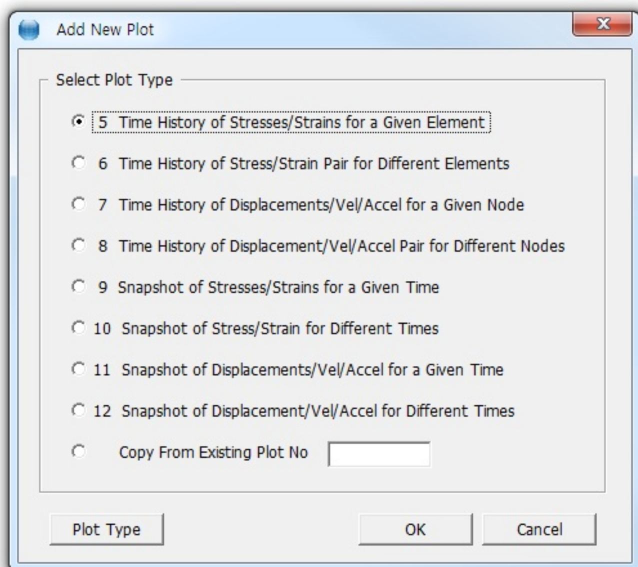


Figure 12.15 Add options for new plot

Plot Type in Add dialog illustrates graphically available plot types as shown schematically in Figure 12.16.

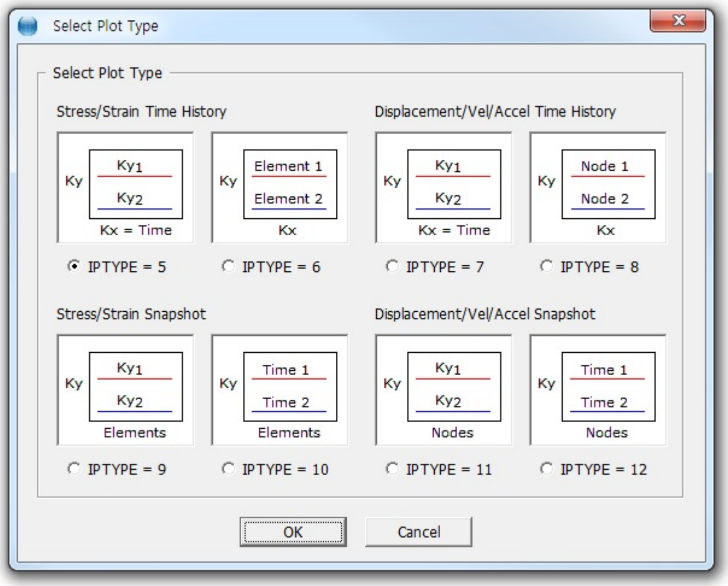


Figure 12.16 Available plot types

12.7.3 Time History for Different Elements

Main Dialog for Time History of Stresses / Strains for Different Elements (IPTYPE = 6) is shown in Figure 12.17.

Elements should be listed in Card 10.2.2 in SMAP Main File.
Table shows available data as in Figure 12.13.

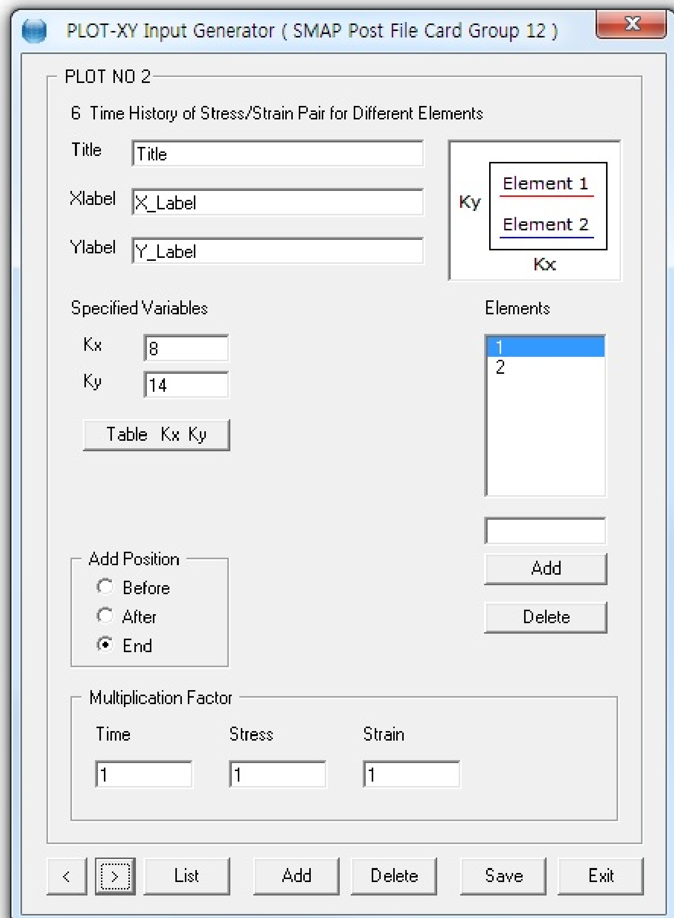


Figure 12.17 Time history for different elements

12.7.4 Time History for a Given Node

Main Dialog for Time History of Displacement / Vel / Accel for a Given Node (IPTYPE = 7) is shown in Figure 12.18.

Node should be listed in Card 10.3.2 in SMAP Main File.
Table shows available data as shown in Figure 12.19.

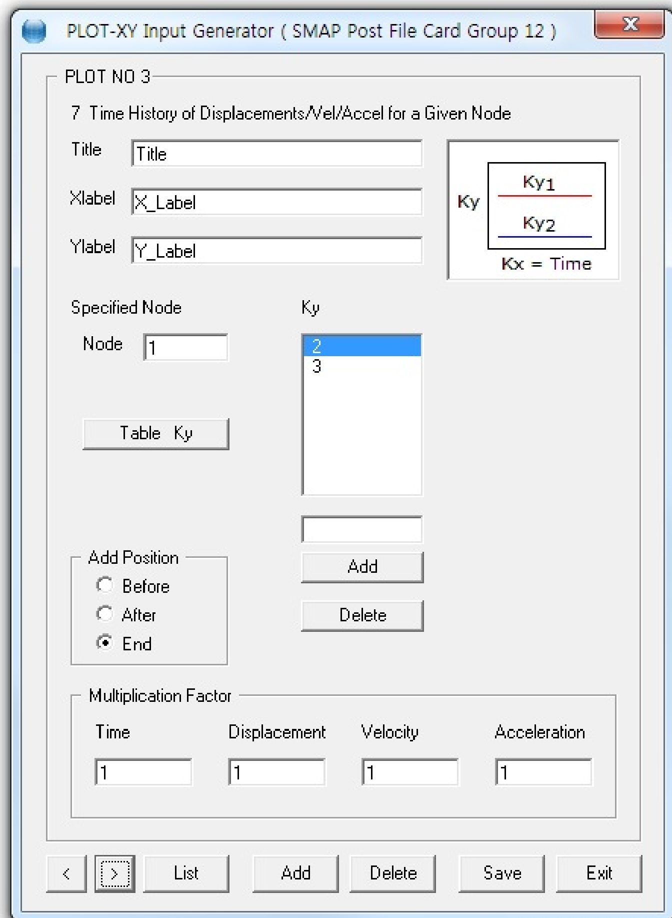


Figure 12.18 Time history for a given node

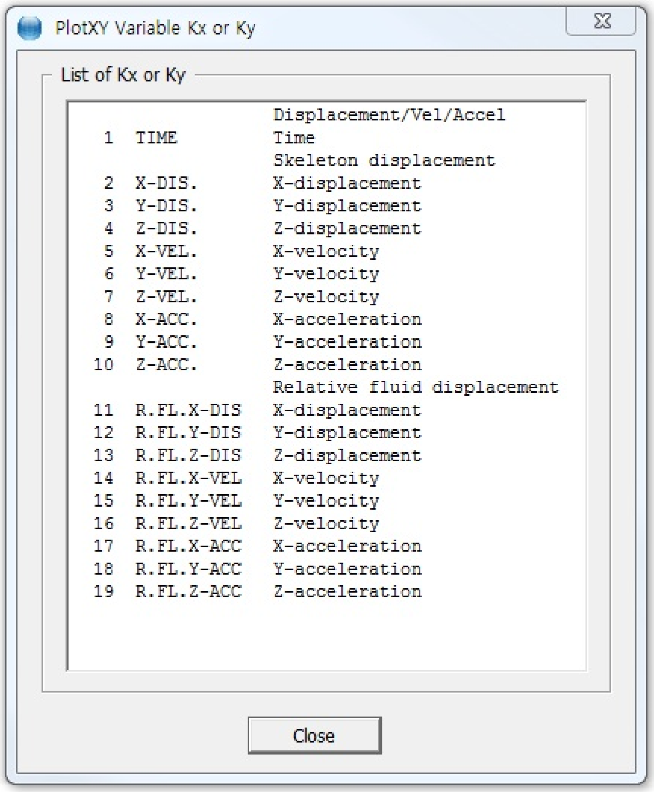


Figure 12.19 Available data for displacement/vel/accel

12.7.5 Time History for Different Nodes

Main Dialog for Time History of Displacement / Vel / Accel for Different Nodes (IPTYPE = 8) is shown in Figure 12.20.

Nodes should be listed in Card 10.3.2 in SMAP Main File.
Table shows available data as in Figure 12.19.

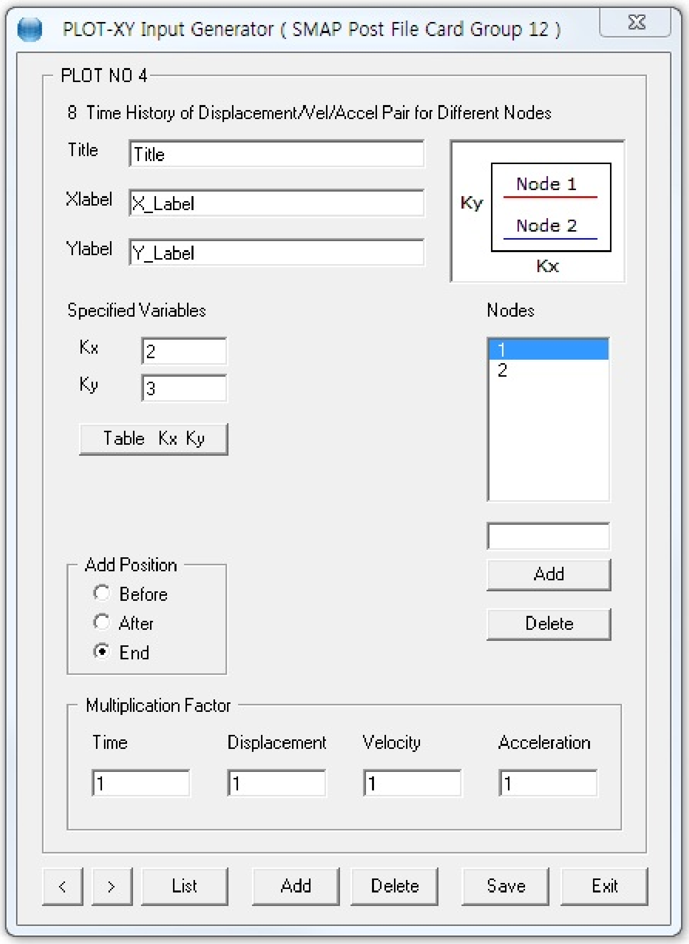


Figure 12.20 Time history for different nodes

12.7.6 Stress/Strain Snapshot for a Given Time

Main Dialog for Snapshot of Stresses / Strains for a Given Time (IPTYPE = 9) is shown in Figure 12.21.

Time should be listed in Card 10.4.2 in SMAP Main File.
Table shows available data as in Figure 12.13.
Elements represent a series of data points in SMAP Mesh.

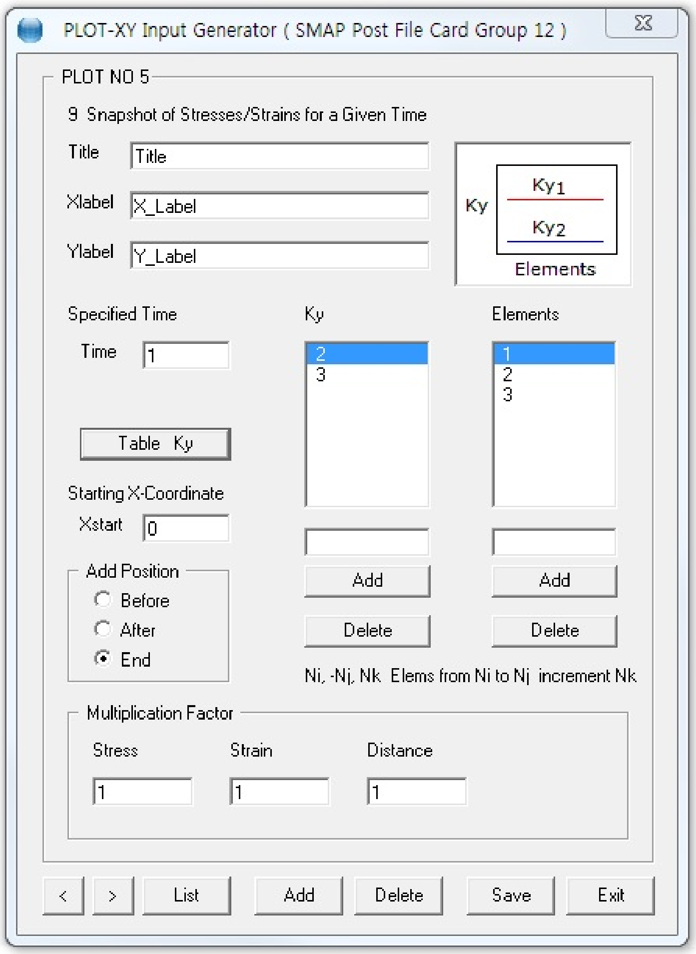


Figure 12.21 Stress/strain snapshot for a given time

12.7.7 Stress/Strain Snapshot for Different Times

Main Dialog for Snapshot of Stresses / Strains for Different Times (IPTYPE = 10) is shown in Figure 12.22.

Times should be listed in Card 10.4.2 in SMAP Main File.
Table shows available data as in Figure 12.13.
Elements represent a series of data points in SMAP Mesh.
This example will select a series of Elements (1,2,3,4,5,6,7,8,9,10).

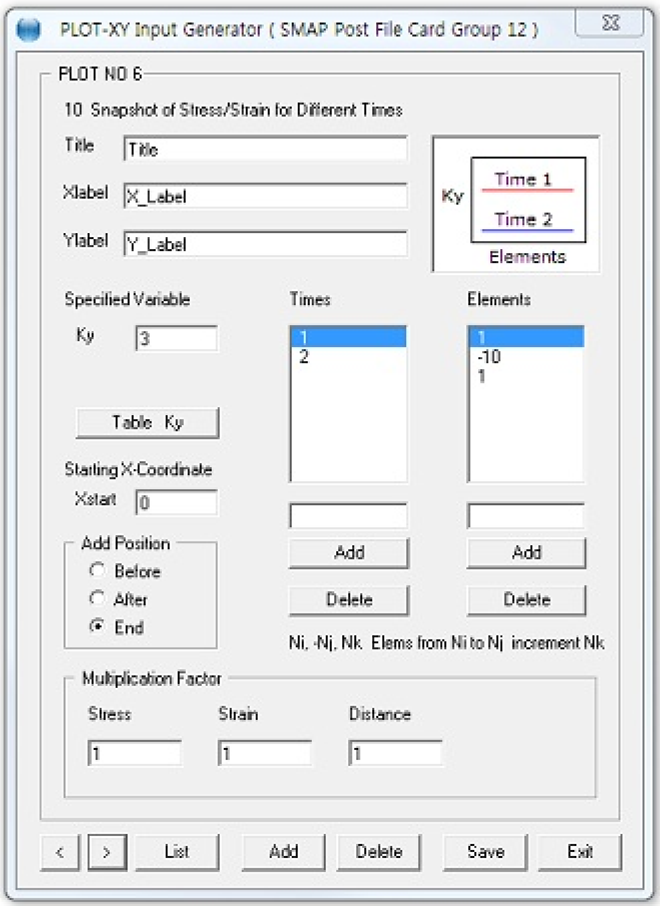


Figure 12.22 Stress/strain snapshot for different times

12.7.8 Displ/Vel/Acc Snapshot for a Given Time

Main Dialog for Snapshot of Displacement / Vel / Accel for a Given Time (IPTYPE = 11) is shown in Figure 12.23.

Time should be listed in Card 10.4.2 in SMAP Main File.
Table shows available data as in Figure 12.19.
Nodes represent a series of data points in SMAP Mesh.

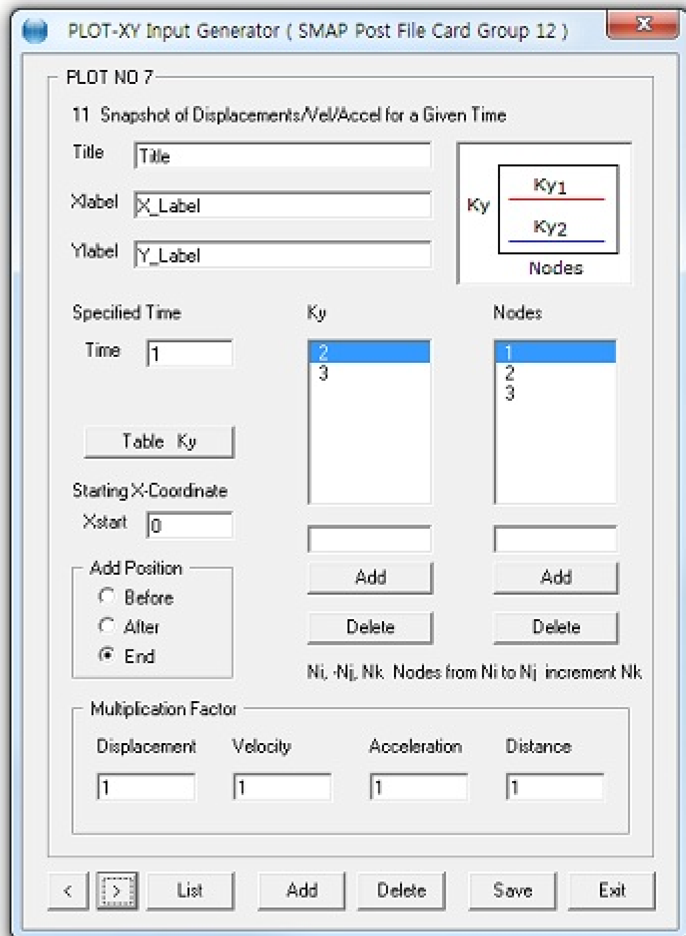


Figure 12.23 Displ/vel/accel snapshot for a given time

12.7.9 Displ/Vel/Acc Snapshot for Different Times

Main Dialog for Snapshot of Displacement / Vel / Accel for Different Times (IPTYPE = 12) is shown in Figure 12.24.

Times should be listed in Card 10.4.2 in SMAP Main File.
Table shows available data as in Figure 12.19.
Nodes represent a series of data points in SMAP Mesh.
This example will select a series of Nodes (1,2,3,11,13,15,17,19,21).

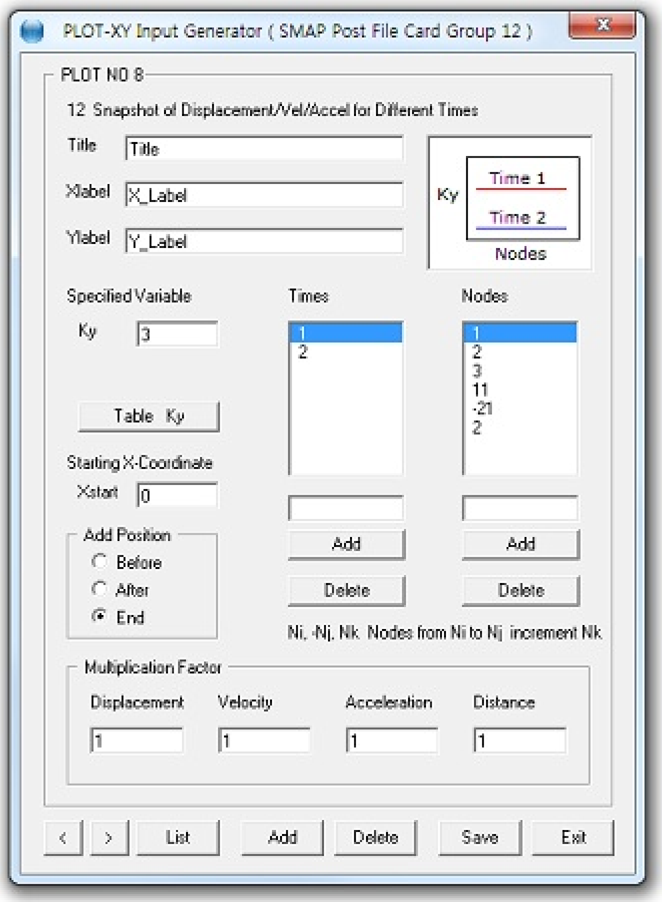


Figure 12.24 Displ/vel/accel snapshot for different times

PLOT-XY
User's Manual

13.1 Introduction

PLOT-XY is a two-dimensional graphical program specially designed to perform scatter plotting and post processing for SMAP programs. The key features of PLOT-XY are:

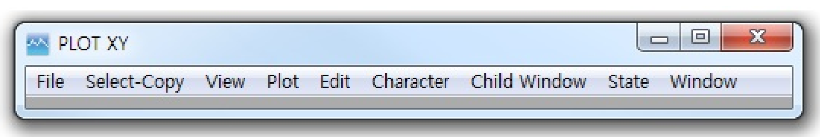
- Plot scatterplot data
  It reads the scatterplot data in text file and plots lines connecting each pair of data points.

- Plot results of analyses
  It reads Card 12 of Post File and SMAP Output and plots time histories of stress/strain/displacement/temperature and snap shots of stress/strain/displacement/temperature vs. distance.

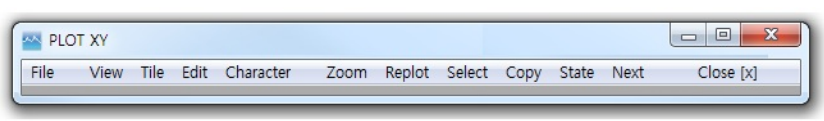
- Edit XY graph
  It reads XY data, edits titles and scales, adds user-defined additional curves.

PLOT-XY has two menu styles, General and Express.

General Style includes 9 menus consisting of all menu items available.
For General Style, specify 1 in C:\Smap\Ct\Ctdata\MenuStyle_XY.dat

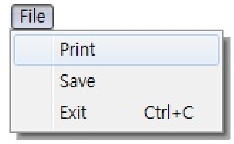


Express Style includes 12 menus which are rearranged so as to quickly access most frequently used menu items in practice.
For Express Style, specify 0 in C:\Smap\Ct\Ctdata\MenuStyle_XY.dat



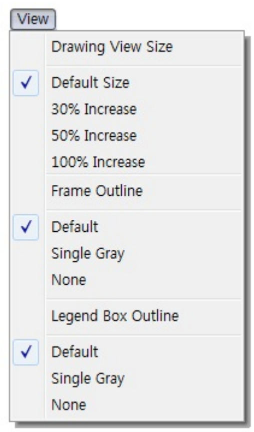
13.2 Menus

File has three sub menus.
Print is to get the hard copy of the current view.
Save is to save the current view.
Exit is to exit PLOT-XY.



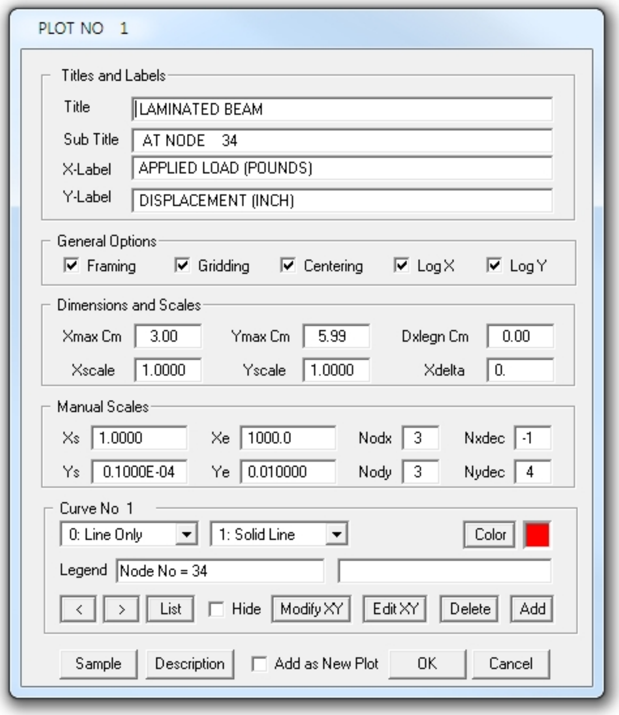
Select-Copy is mainly used to select and then copy the current view.

View is mainly used to select
Drawing View Size:
Default Size, 30%, 50%, or 100%
Increase Frame Outline:
Default, Single Gray, or None
Legend Box Outline:
Default, Single Gray, or None

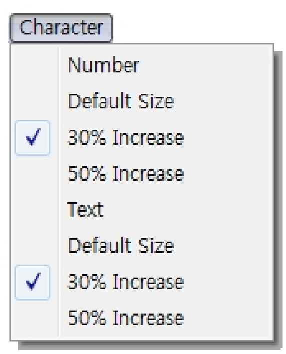


Plot has the following five sub menus.
Replot is to replot the currently focused child window. Zoom is to zoom the currently focused child window. Once this sub menu is selected, you can specify the rectangular zoom area by left mouse button down at the left top corner and then left mouse button up at the right bottom corner.
Hardcopy is to print the currently focused window.
Next is to plot the next graph.
Stop is to stop plotting.

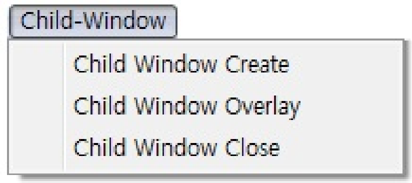
Edit opens following dialog to edit XY graph data.
It is described in detail in Section 12.3 in XY graph User's Manual.



Character is used to change sizes of number and text fonts. Default sizes are specified in PLOT-XY setup menu.



Child-Window is used to create, overlay, or close child window. A maximum of 40 child windows can be opened.



PLOT-2D
User's Manual

14.1 Introduction

PLOT-2D is a two-dimensional graphical program specially designed to perform pre and post processing for SMAP programs.
The key features of PLOT-2D are:

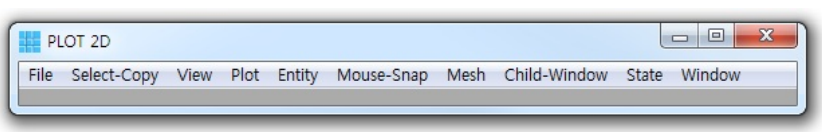
- **Plot finite element meshes**
  It reads the Mesh File and plots meshes along with node, element, boundary code, and material numbers.

- **Plot results of analyses**
  It reads Mesh File, Card 11 of Post File, SMAP Output Files and plots contours of continuum stress/strain/temperature, beam section forces, truss axial force/stress/strain, principal stress vectors, and deformed shapes.

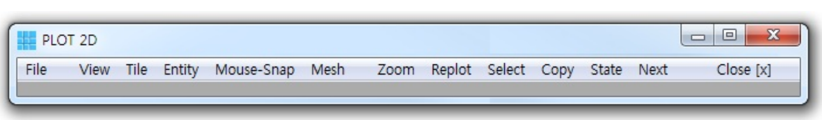
- **Edit finite element or group meshes**
  It reads finite element or group mesh files and edit these meshes.

PLOT-2D has two menu styles, General and Express.

General Style includes 11 menus consisting of all menu items available.
For General Style, specify 1 in C:\Smap\Ct\Ctdata\MenuStyle_2D.dat

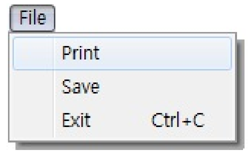


Express Style includes 13 menus which are rearranged so as to quickly access most frequently used menu items in practice.
For Express Style, specify 0 in C:\Smap\Ct\Ctdata\MenuStyle_2D.dat



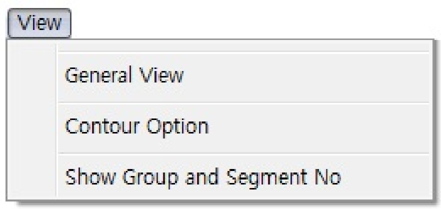
14.2 Menus

File has three sub menus.
Print is to get the hard copy of the current view.
Save is to save the current mesh file. Exit is to exit PLOT-2D.

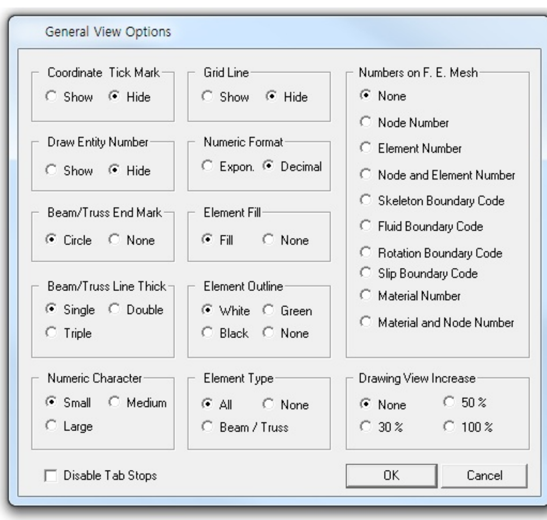


Select-Copy is mainly used to select and then copy the current view.

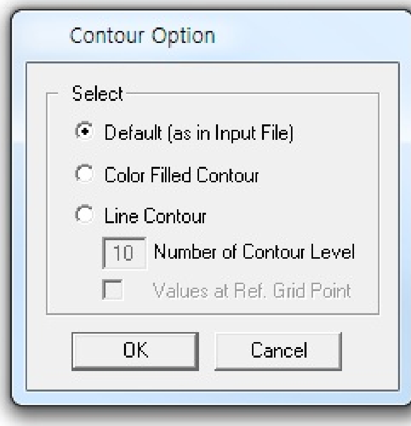
View has three sub menus;
General View, Contour Option,
and Show Group and Segment No.



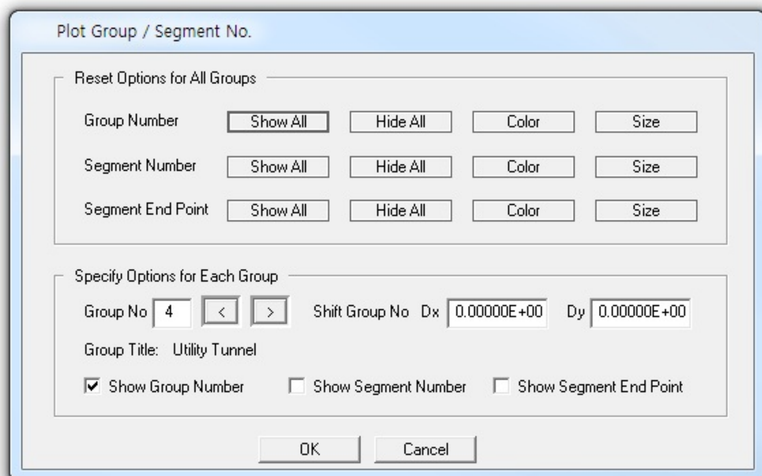
General View options affect all types of plots.



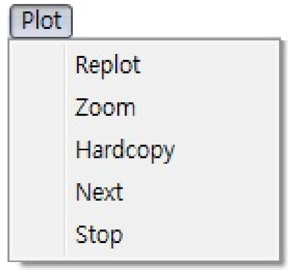
Contour Options affect contour plots of continuum element data for analysis results.



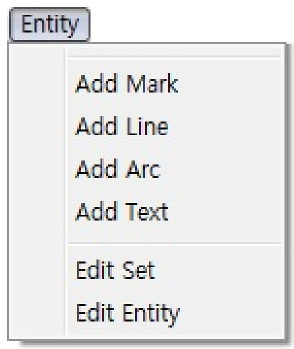
Show Group and Segment No is to show group and segment numbers when editing group meshes. It is described in detail in Section 5.3 in Group Mesh User's Manual.



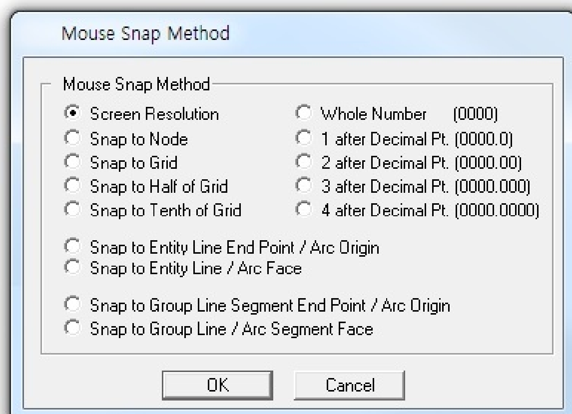
Plot has the following five sub menus.
Replot is to replot the currently focused child window. Zoom is to zoom the currently focused child window.
It zooms only mesh. Once this sub menu is selected, you can specify the rectangular zoom area by left mouse button down at the left top corner and then left mouse button up at the right bottom corner.
Hardcopy is to print the currently focused window.
Next is to plot the next graph.
Stop is to stop plotting.



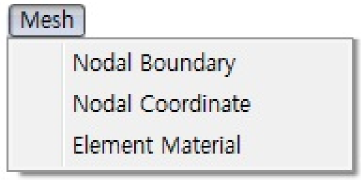
Entity is the graphical object which is mainly used to assist editing geometry of groups and elements. It has following six sub menus; Add Mark, Add Line, Add Arc, Add Text, Edit Set, and Edit Entity. It is described in detail in Section 5.7 in Group Mesh User's Manual.



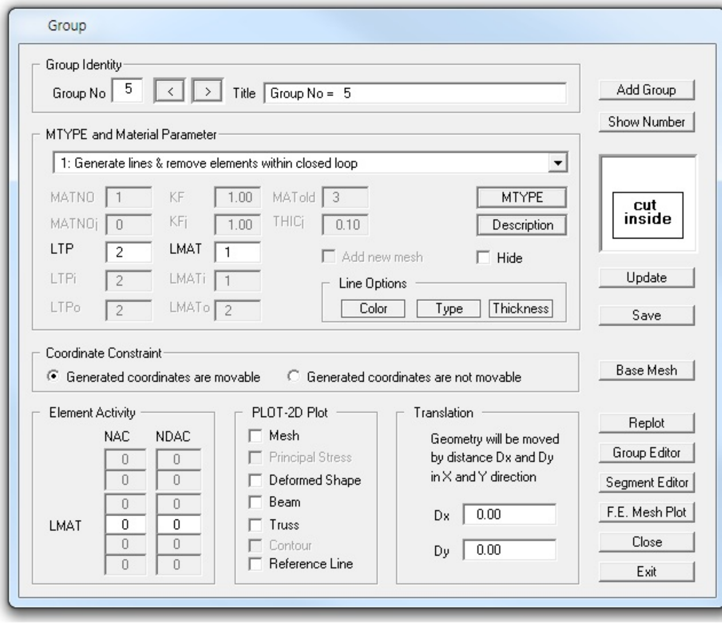
Mouse-Snap is to control the position of mouse cursor when you work for finite element mesh, group mesh, or entities. Mouse Snap Method helps you place the mouse cursor more accurately.



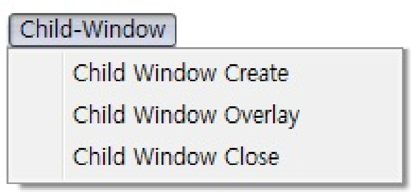
Mesh is used to directly modify finite element meshes. It has three sub menus; Nodal Boundary, Nodal Coordinate, and Element Material.
It is described in detail in Section 5.6 in Group Mesh User's Manual.



Group is used to build or edit group mesh.
It is described in detail in Section 5.3 in Group Mesh User's Manual.



Child-Window is used to create, overlay, or close child window.
A maximum of 40 child windows can be opened.



PLOT-3D
User's Manual

15.1 Introduction

PLOT-3D is a three-dimensional graphical program specially designed to perform pre and post processing for SMAP programs. The key features of PLOT-3D are:

- **Plot finite element meshes**
  It reads the Mesh File and plots meshes along with node, element, boundary code, and material numbers.

- **Plot results of analyses automatically**
  It reads Mesh File and SMAP Output Files and with no input for Post File, plots contours of stress/strain/displacement, iso surface, principal stress vectors, load vectors and deformed shapes.

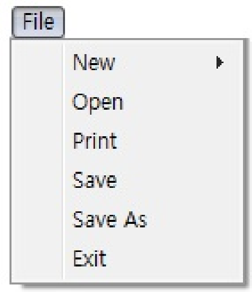
- **Compute intersections of surfaces**
  It reads the Mesh File containing shell elements for 3D surfaces and shows the locations of the computed intersections.
  The computed coordinates of intersections are saved in a file "Intersection.dat" which can be used for the construction of complicated 3D meshes.

PLOT-3D has 5 menus; File, Model, Plot, View and Help along with 25 toolbars.

15.2 Menus

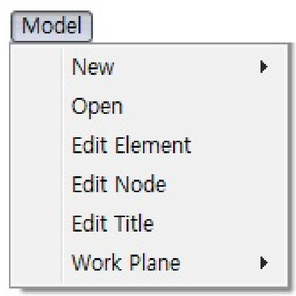
File has six sub menus.
New is used to build Finite Element Mesh or Block Mesh.
Open is used to open existing mesh file.
Print is to get the hard copy of the current view.
Save is to save the current mesh file or current view.

Save As is to save the current mesh file as another name.
Exit is to exit PLOT-3D.

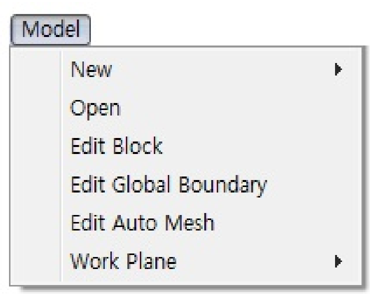


Model is mainly used to edit Finite Element or Block Mesh file.
For detailed description, refer to Block Mesh User's Manual in Section 6.

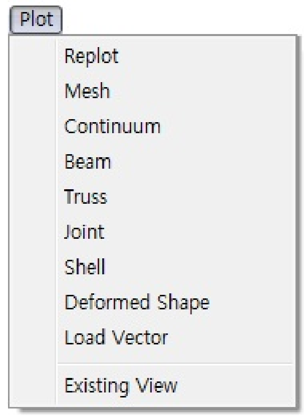
For editing Finite Element Mesh, 6 menus are shown.
New is to build new mesh file.
Open is to open existing mesh file.
Edit Element is to edit parameters related to element.
Edit Node is to edit parameters related to node.
Edit Title is to edit title.
Work Plane is to show prebuilt work planes.



For editing Block Mesh, 6 menus are shown.
New is to build new mesh file.
Open is to open existing mesh file.
Edit Block is to edit parameters related to block.
Edit Global Boundary is to edit parameters related to boundary.
Edit Auto Mesh is to edit parameters related to auto mesh.
Work Plane is to show prebuilt work planes.

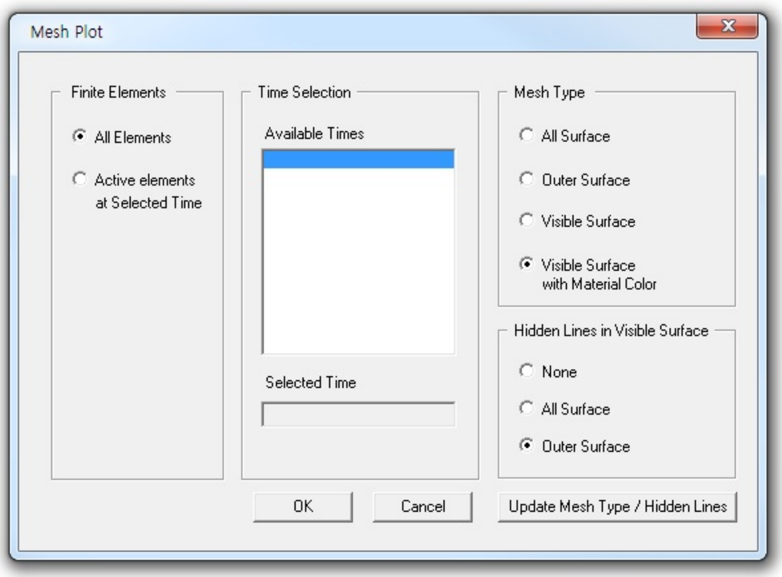


Plot is mainly used to plot Finite Element mesh and analysis results.
It has 10 sub menus; Replot, Mesh, Continuum, Beam, Truss, Joint, Shell, Deformed Shape, Load Vector, Existing View.
Joint plot is not available.

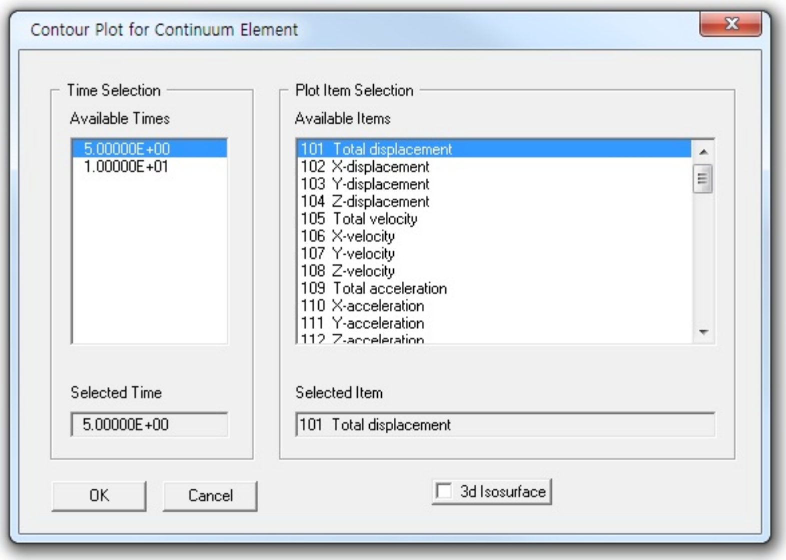


Replot is mainly used to refresh the current view.

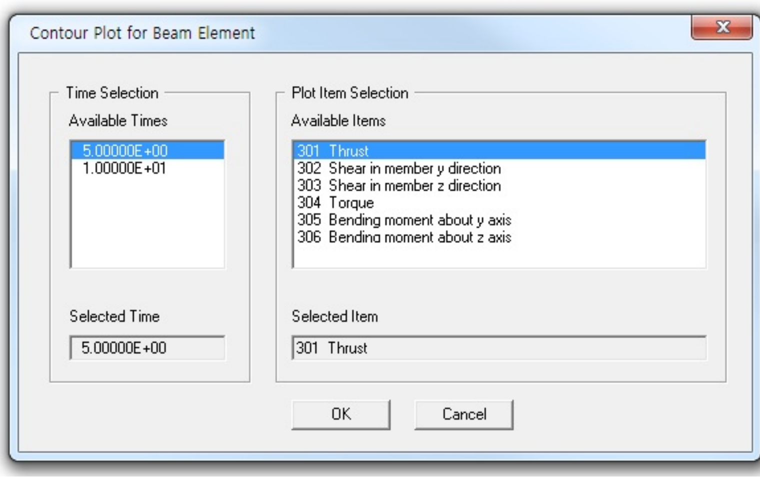
Mesh is to plot Finite Element meshes (Default plot type).
Mesh plot requires only Mesh File.



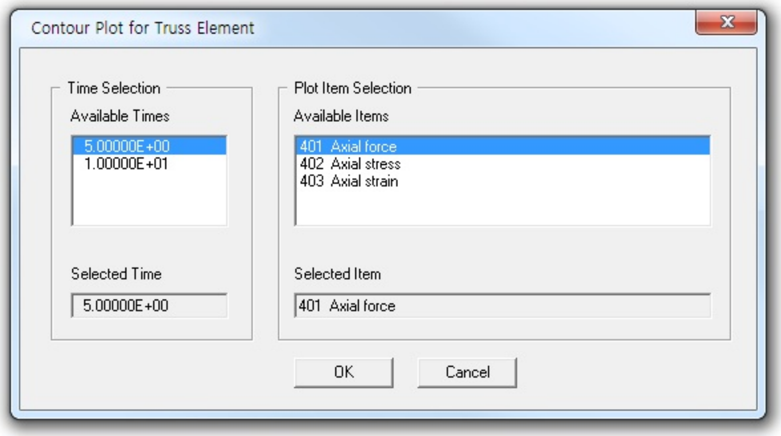
Continuum is to plot contours or principal stress vectors for continuum elements. By checking "3d Isosurface", iso surface will be shown.



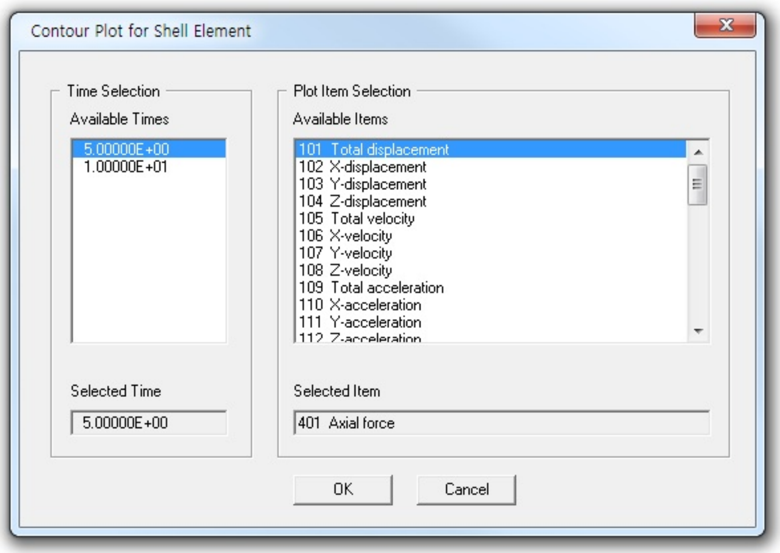
Beam is to plot section forces of beam elements.



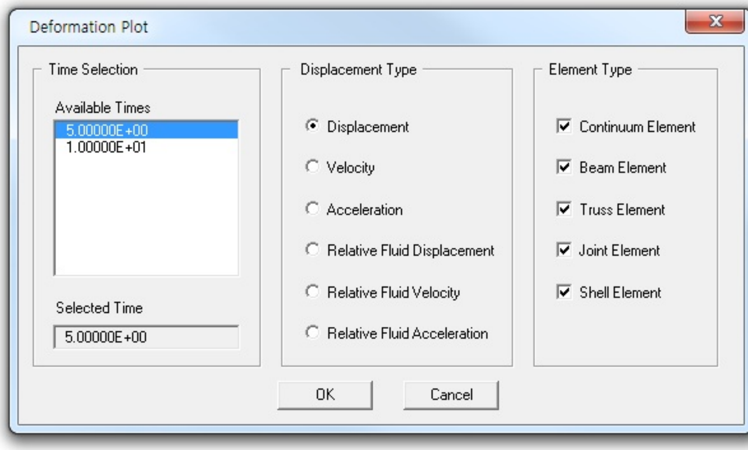
Truss is to plot axial force/stress/strain of truss elements.



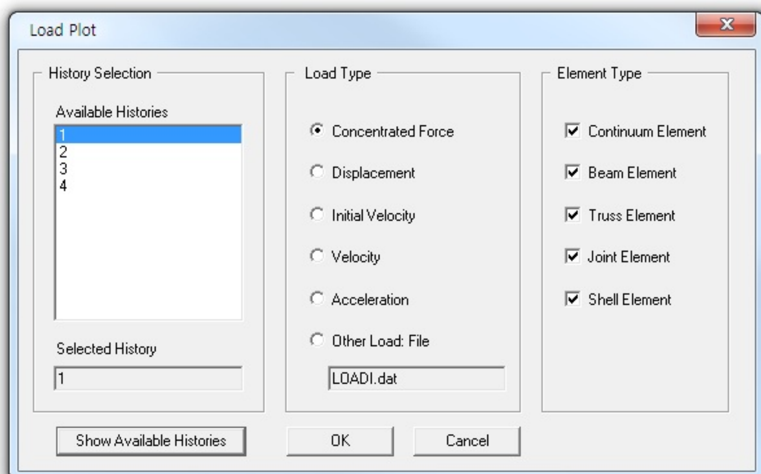
Shell is to plot contours or principal stress vectors for shell elements.



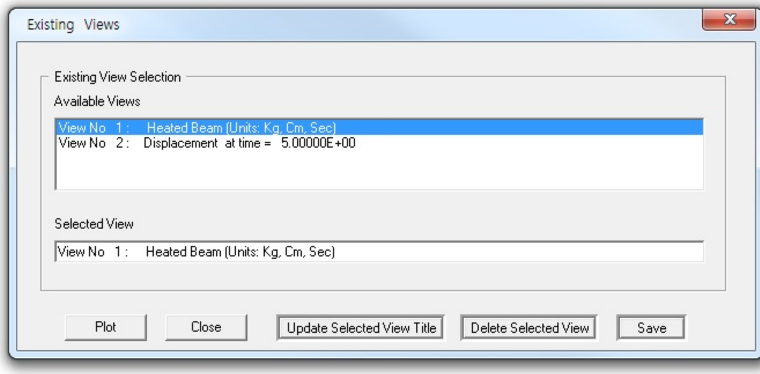
Deformed Shape is to plot the snap shot of all kinds of displacement/velocity/accelerations.
Note that deformed meshes can be combined with other plot types as discussed in "Displacement" option in view menu.



Load Vector is to plot the external loads of concentrated forces/ displacements/velocities/accelerations along with load intensity.
Note that load vectors can be plotted on deformed meshes as discussed in "Load Vector" option in view menu.



Existing View is to replot the saved views.

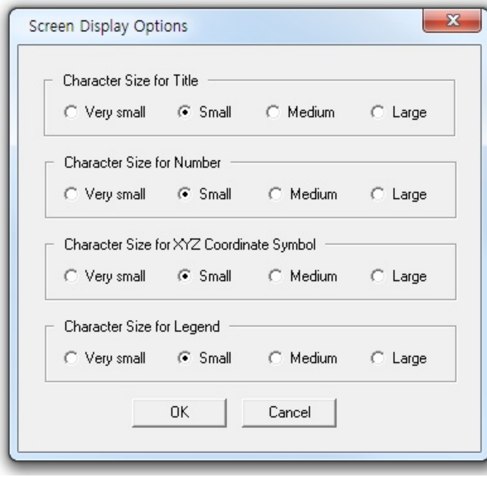


View is used to change the appearance of a selected plot.

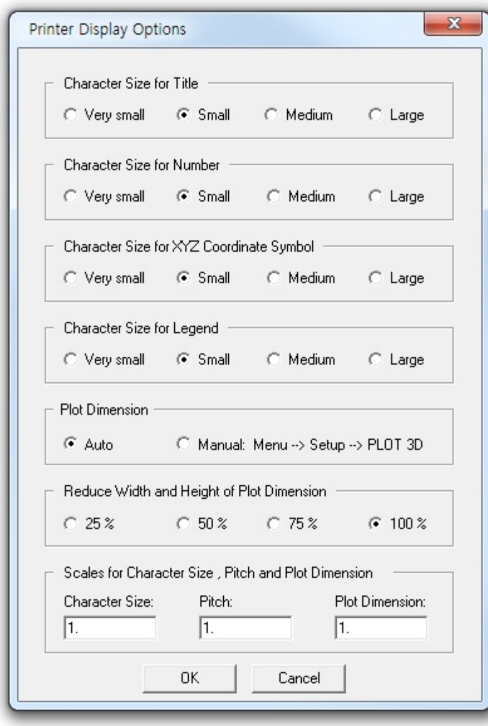
It has eleven sub menus; General, Screen, Printer, Contour, Clip Plane, Mesh, Beam, Truss, Principal Stress, Displacement, and Load Vector.

General view options affect most plot types.

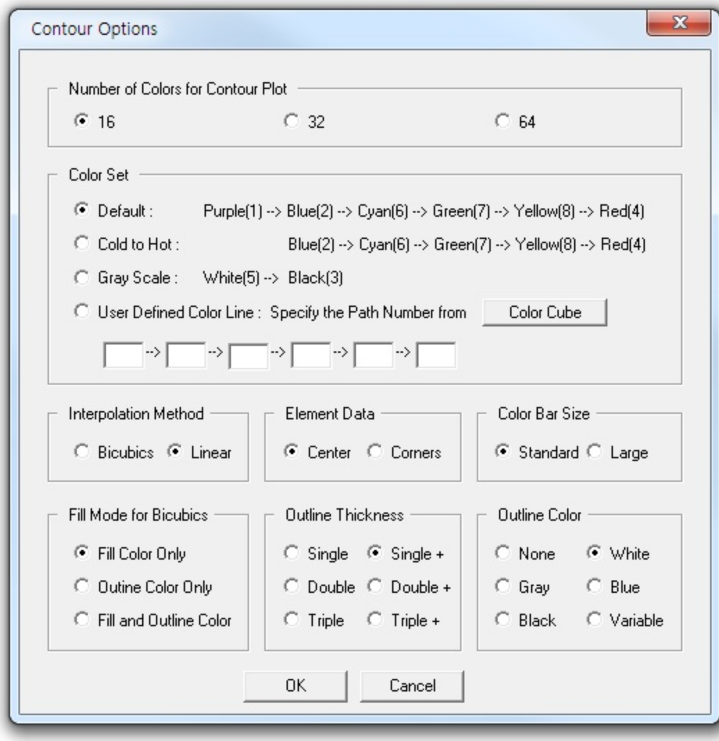
Screen display options affect character sizes shown on the monitor.



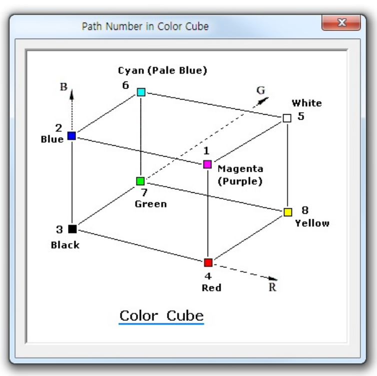
Printer display options affect character sizes and plot dimensions shown on the hard copy.



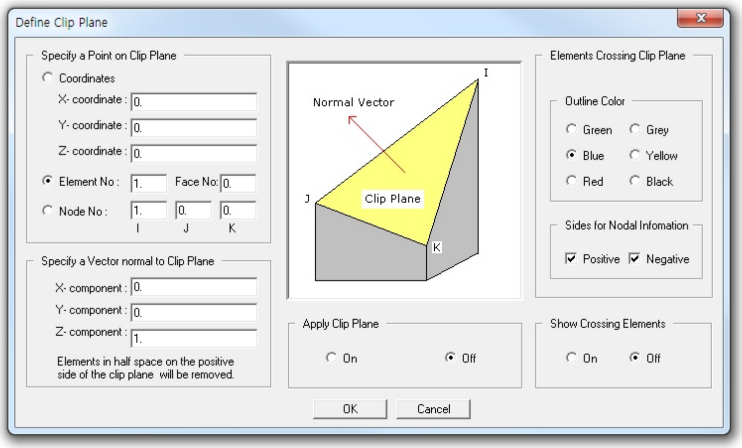
Contour options affect all types of plots involving contours.



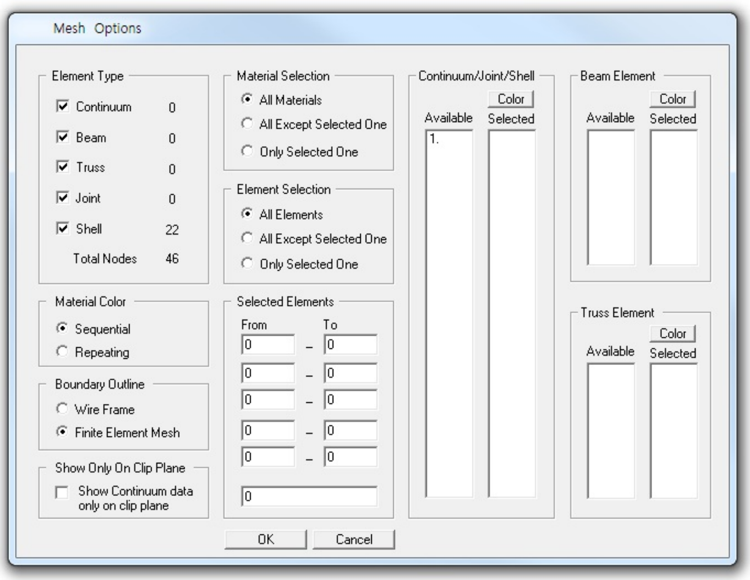
Color cube is to use for user defined color line.



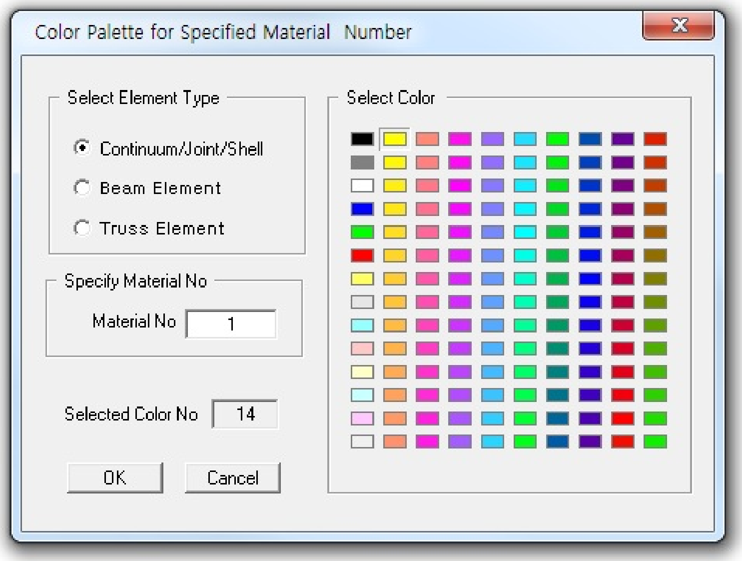
Clip plane defines parameters associated with the clip plane which cuts through the internal part of the 3D domain. When "Apply Clip Plane" is on, contours or deformed shapes are shown on such user defined plane.



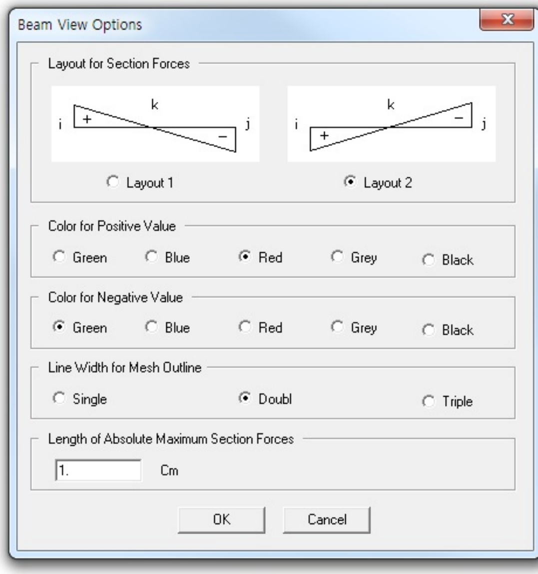
Mesh options affect all plot types. As one of useful features, it can select particular types of elements and materials.



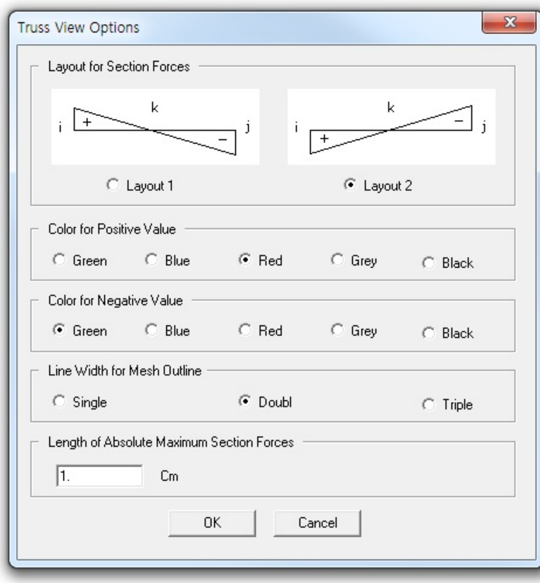
Color is to use for user defined mesh color.



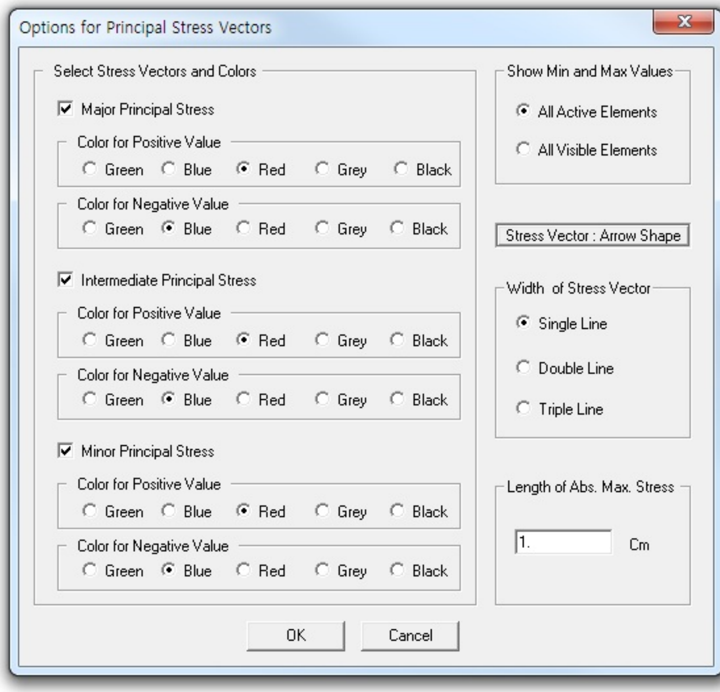
Beam view options affect only beam plot.



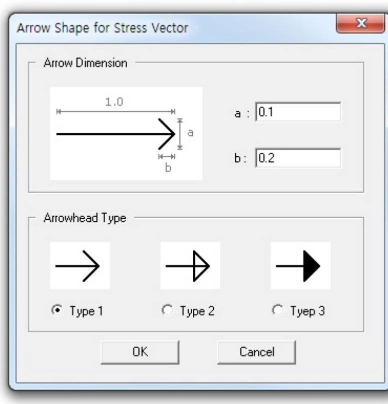
Truss view options affect only truss plot.



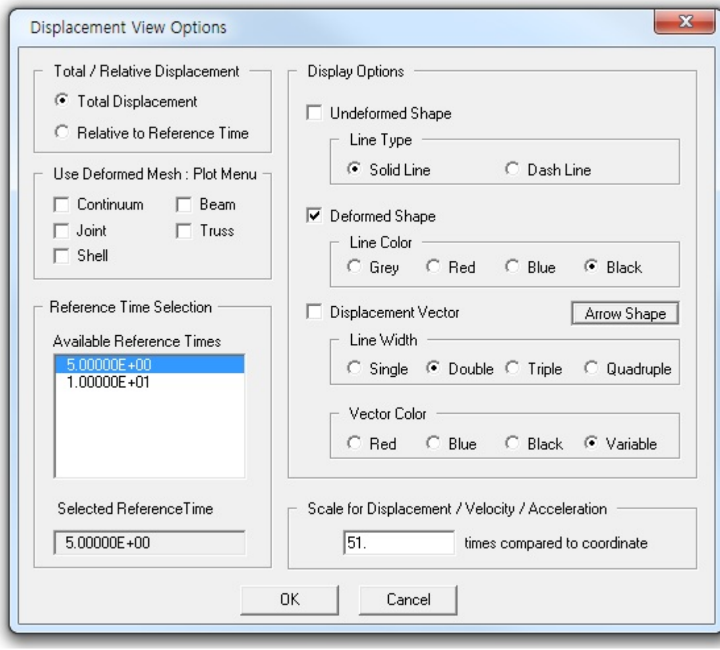
Principal Stress options affect only plots of principal stress vectors in continuum or shell elements.



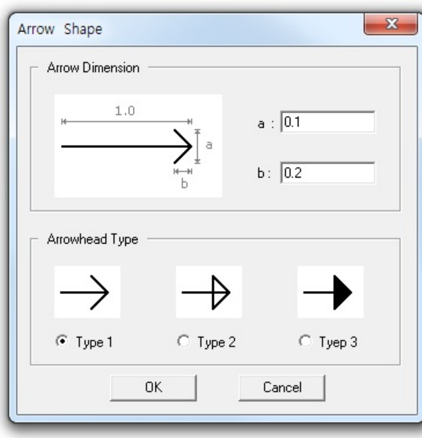
Users can specify the arrow shape for stress vector.



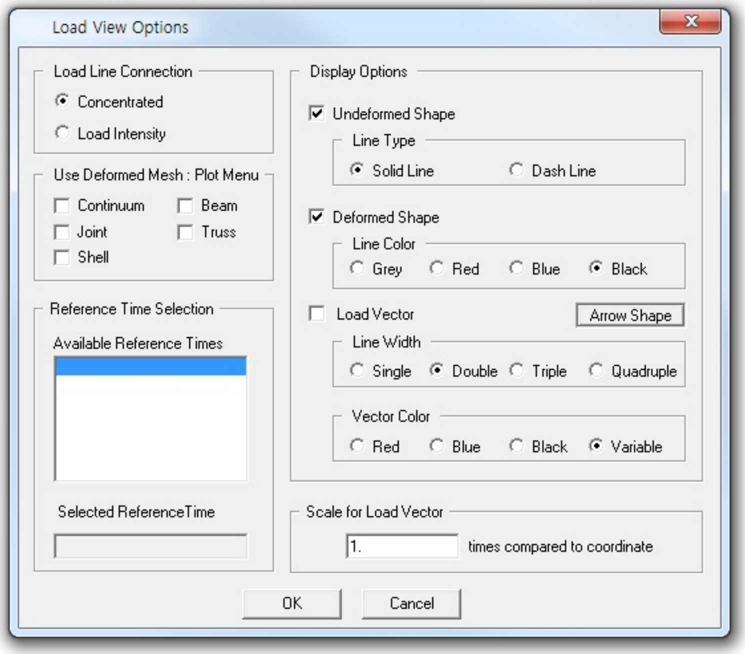
Displacement view options affect only deformed shape plot.
Continuum, Beam, Truss, and Shell plots can be displayed
over deformed mesh by checking types in "Use Deformed Mesh".



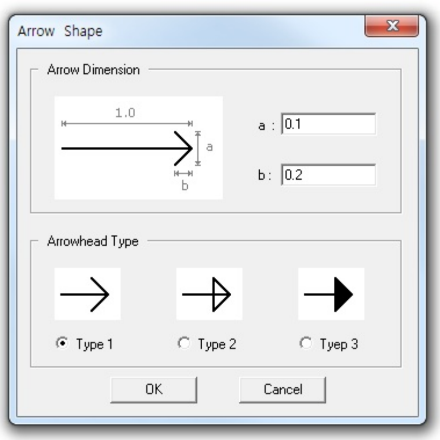
Users can specify the arrow shape
for displacement vector.



Load Vector view options affect only load vector plot.
Load vectors can be displayed over deformed mesh by checking "Deformed Shape" in Display Options



Users can specify the arrow shape for load vector.



15.3 Toolbars

Open Toolbar

This button activates the file open dialog box to open mesh file.

Print Toolbar

This button is used to get the hard copy of current view.

Save Toolbar

This button is used to save current view or working file.

Model Toolbar

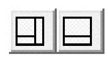
This button is used to edit finite element or block mesh.

Work Plane Toolbar

This button is to set work plane used for Model.

Layout Toolbar

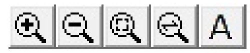
These buttons are used to show different layouts.
The first button divides the plot area into three parts; mesh, title, and legend. The second button divides the plot area into two parts; mesh and title.



XYZ Toolbar

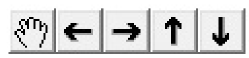
This button is used to locate position of XYZ coordinate symbol in the two part layout mode. Each time you click this button, the XYZ symbol moves counterclockwise along the corners of rectangle. XYZ button is also used to control the amount of movement, rotation, and zoom.

Zoom Toolbar



The first button is used to magnify the mesh.
And the second button is used to reduce the mesh.
The third button is used to activate the selection of zoom area.
Once this button is on, you can specify the rectangular zoom area by left mouse button down at the left top corner and left mouse button up at the right bottom corner. To deactivate, click the button again.
The fourth button is used to switch from the currently zoomed view to the previously zoomed view or vice versa. The last button with "A" is to go back to the initial default configuration.

Translation Toolbar



The first button is to activate drag mode. Once this button is on, you can move the mesh by dragging the mouse. To deactivate, click the button again.
The other buttons move the mesh to the left, right, up, and down, respectively.

Rotation Toolbar

The first button changes direction of rotation.
The other three buttons rotate the mesh about X, Y, and Z axes, respectively.

Number Toolbar

The first button is to activate number mode.
Once this button is on, the selected data will be shown.
Clicking the button again will hide the selected data.
The other two buttons are used to select next and previous number, respectively. The description of selected number is listed at the bottom of PLOT-3D window.

SMAP ® - 2D

Structure Medium Analysis Program

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

Example Problems

Printed in the United States of America.

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-2D main example problems as summarized in Table 1.1. First 9 problems are presented to demonstrate the accuracy and validity of SMAP-2D 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 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 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-2D 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	0.30	Planar compression wave propagation Check: • Dynamic two-phase response
	VP3-1.dat		Using transmitting boundary
4	VP4.dat	0.22	Circular tunnel in Drucker-Prager medium Check: • 2-D elasto-plastic matrix of Generalized Hoek and Brown Model
	VP4-1.dat		Using element surface load
	VP4-2.dat		Using Triangular element
5	VP5.dat	0.01	Laminated beam with slip interface Check: • Joint element • Joint model
	VP5-1.dat		Thin layer joint element, NM=4 Joint thickness by CARD 5.3.2.4.11
	VP5-2.dat		Thin layer joint element, NM=4, t=0 Joint thickness by user coordinate

Table 1.1 List of SMAP-2D example problem, continued

Problem Number	Project File Name	Run Time Pent. III 850	Description
6	VP6.dat	0.01 min.	Gibson's construction pore pressure • Consolidation • Variable time step • Moving boundary
	VP6-1.dat		Using Triangular element
7	VP7.dat	0.01	Drained triaxial compression test • Modified Cam Clay Model • Drained triaxial compression path
8	VP8.dat	0.01	Undrained plane strain comp. test. • Modified Cam Clay Model • Undrained plane compression path
9	VP9.dat	0.01	Volumetric creep in isotropic undrained test. • Modified Cam Clay Model • Volumetric creep
10	VP10.dat	0.33	Spherical wave propagation
11	VP11.dat	0.01	Elastic truss analysis
12	VP12.dat	0.01	Fixed end beam analysis
13	VP13.dat	0.07	Beam dynamic analysis
14	VP14.dat	0.25	Burn's and Siess' beam analysis
15	VP15.dat	0.20	William's toggled beam analysis
16	VP16.dat	0.02	Plane strain tunnel analysis
17	VP17.dat	8.75	Embankment construction
18	VP18.dat	0.01	Heated beam modeled by beam
	VP18-1.dat		Heated beam modeled by continuum
19	VP19.dat	9.03	Preload consolidation & excavation

Table 1.1 List of SMAP-2D example problem, continued

Problem Number	Project File Name	Run Time Pent. III 850	Description
20	VP20.dat	3.95	Seismic tunnel analysis
21	VP21.dat	0.01	Frames with hinge connection
	VP21-1.dat		NBLT = 0 NSPTB = 2 NSTYPE = 2
	VP21-2.dat		NBLT = 0 & NSPTB = 5
	VP21-3.dat		NBLT = 0 & NSPTB = -5
	VP21-4.dat		NBLT = 1 & NSPTB = 2
	VP21-5.dat		NBLT = 0 NSPTB = 2 NSTYPE = 20
	VP21-6.dat		Same as VP21-4 except variable E
	VP21-7.dat		NBLT = 2 NSPTB = 2
	VP21-8.dat		NBLT = 0 NSPTB = 2 NSTYPE = 9
22	VP22.dat		Embedded rebars with slip
23	VP23.dat		Pseudo dynamic embankment fill
24	VP24.dat		Excavation on nearby box frame
25	VP25.dat		Plane strain tunnel in jointed continuum
26	VP26.dat		Spring analysis
27	VP27.dat		Nonlinear truss analysis
28	VP28.dat		SDOF System To Ground Acceleration
29	VP29.dat		Frames with Rotational Spring Connection
30	VP30.dat		Reinforced Concrete Cylinder
31	VP31.dat		Beam 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-2D 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-2D, you may generate such Mesh Files using the following method:

Method

First, generate 2D Mesh File using Group Mesh Generator, Block Mesh Generator, or 2D PRESMAP. Then combine or modify these Mesh Files using ADDRGN-2D if you need to do it.

1. Generate 2D Mesh File

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

2. Combine or modify Mesh File

ADDRGN-2D

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-2D 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-2D 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
│  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 two and 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 (3D)

SMAP-2D
Example Problem

SMAP-2D is the main-processing program which computes static, consolidation and dynamic response of two-dimensional problems. Input parameters of SMAP-2D are described in detail in Section 4 of SMAP-2D User's Manual.

Running SMAP-2D 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-2D is outlined in Section 3.5 of User's Manual. Once you finished execution of SMAP-2D, 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-2D User's Manual.

Table 1.1 in Section 1 shows the summary of SMAP-2D 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-2D.

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

C:\Smap\Smap2D\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-2D 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-2D is identical to the exact solution.

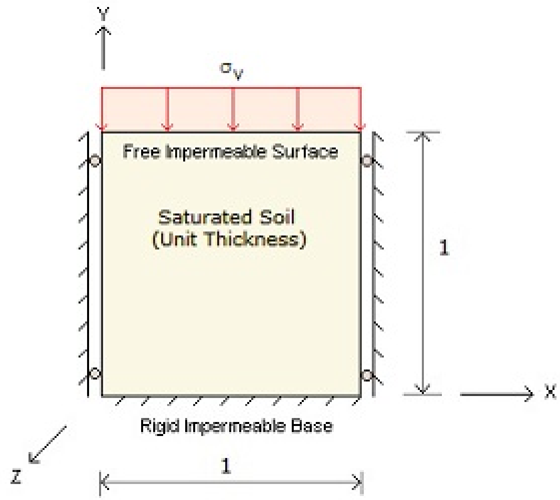


Figure 4.1 A cubic element subjected to undrained uniaxial strain loading

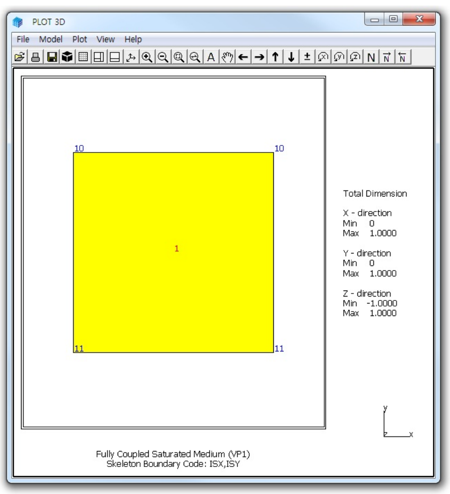


Figure 4.2 Finite element mesh

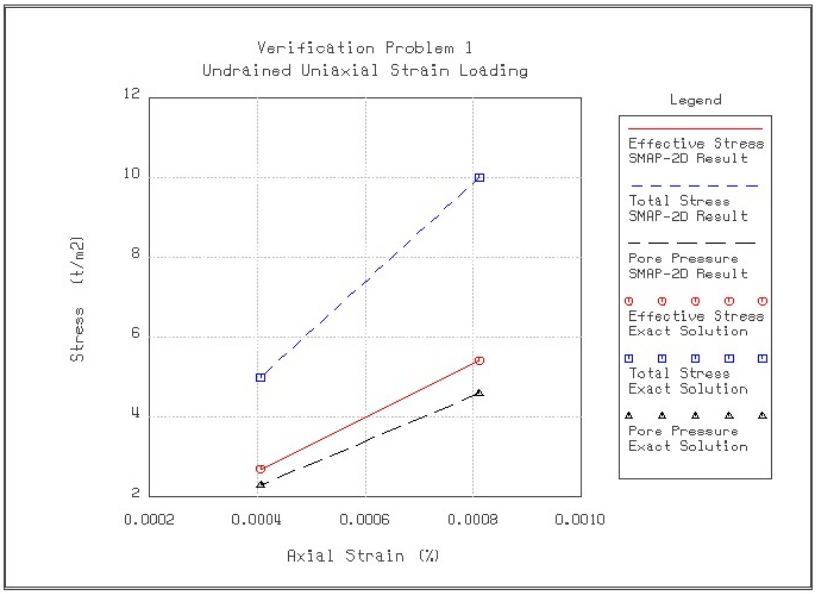


Figure 4.3 Computed undrained stress response compared with exact solution

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

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

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-2D calculations are very close to the exact solution.

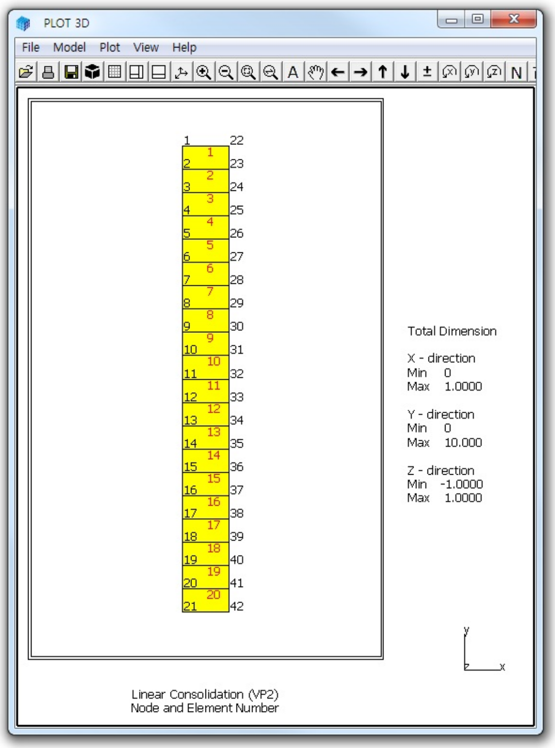


Figure 4.4 Finite element mesh

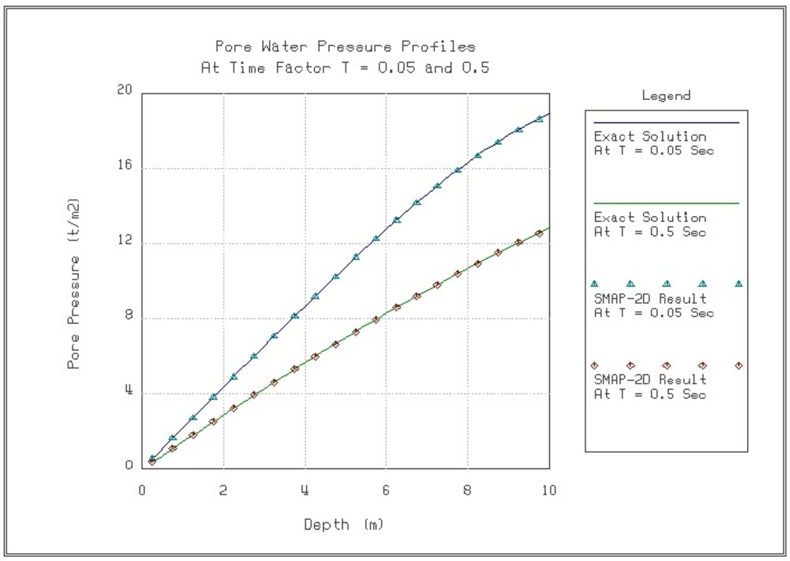


Figure 4.5 Pore water pressure profiles at T = 0.05 and 0.5

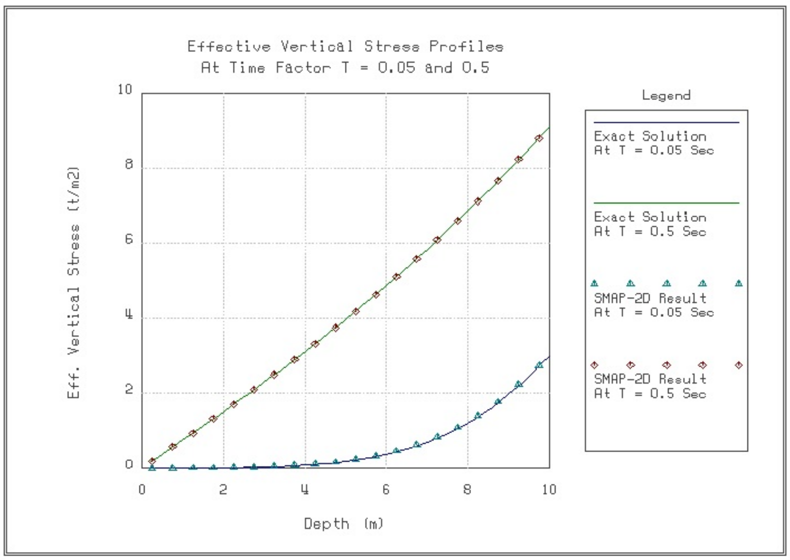


Figure 4.6 Effective vertical stress profiles at T = 0.05 and 0.5

4.3 Planar Compression Wave Propagation

The problem is to check overall two-phase dynamic equations implemented in the program SMAP-2D. 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-2D results.

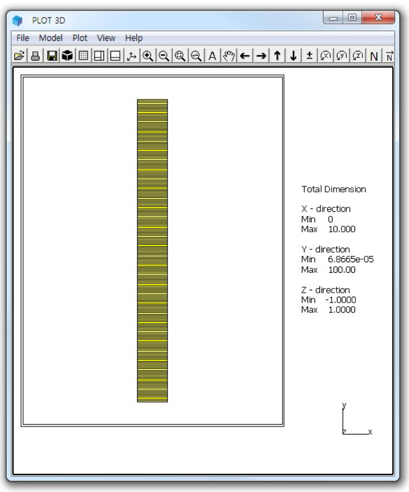
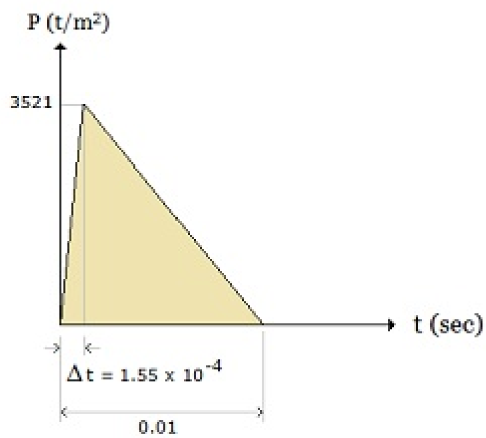


Figure 4.7 Finite element mesh



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

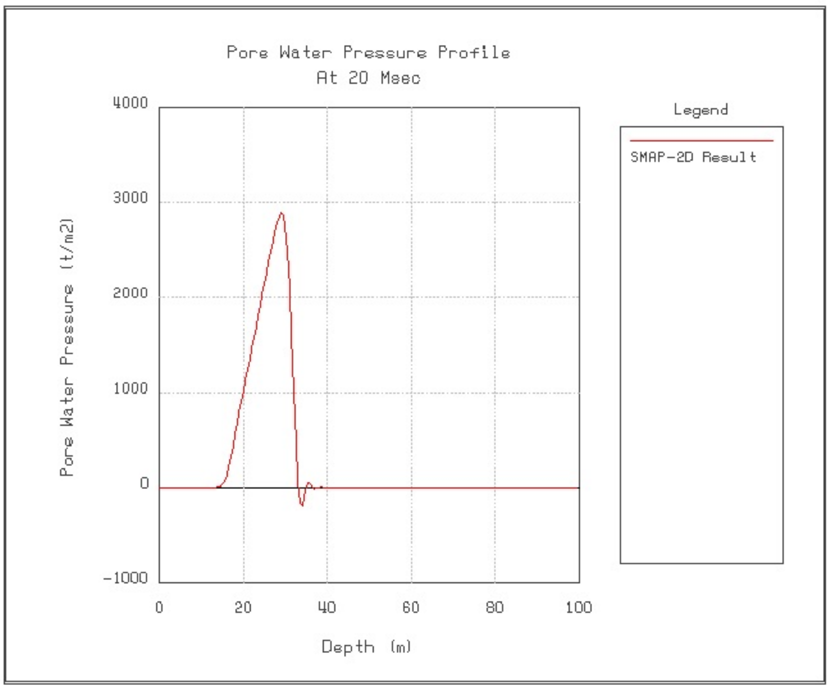


Figure 4.9 Profiles of pore water pressure at 20 msec

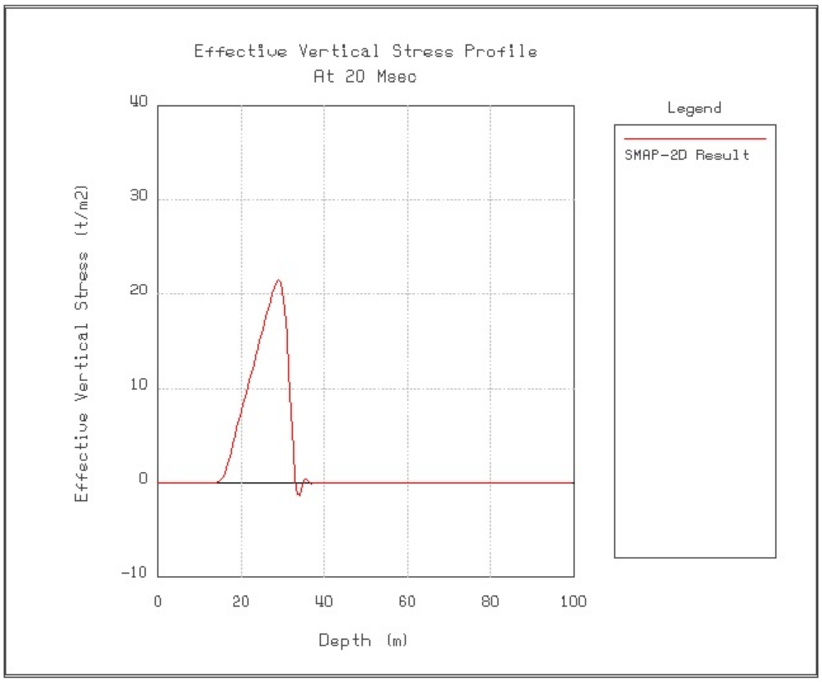


Figure 4.10 Profiles of effective vertical stress at 20 msec

4.4 Circular Tunnel in Drucker-Prager Medium

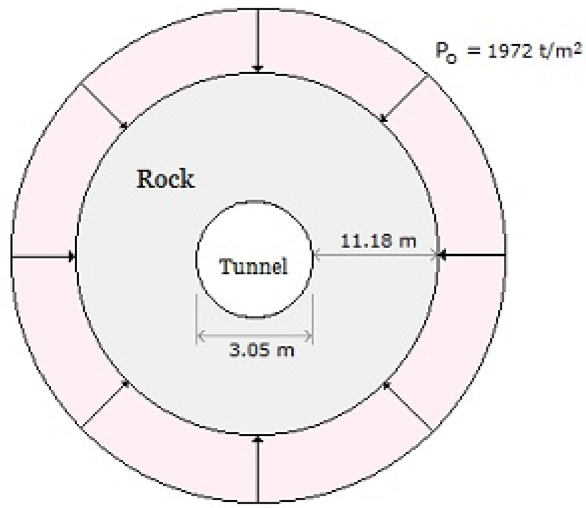
The problem is to check the implementation of the 2-dimensional formulation of elasto-plastic matrix derived for the Generalized Hoek and Brown Model. In this problem, the plane strain response of a tunnel subjected to axisymmetric loading as calculated using SMAP-2D is compared to a semi-analytical 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. This is to check the uniform response of the integrated two-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. And Figure 4.15 shows stresses along the 85.5° from the X-axis. 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:

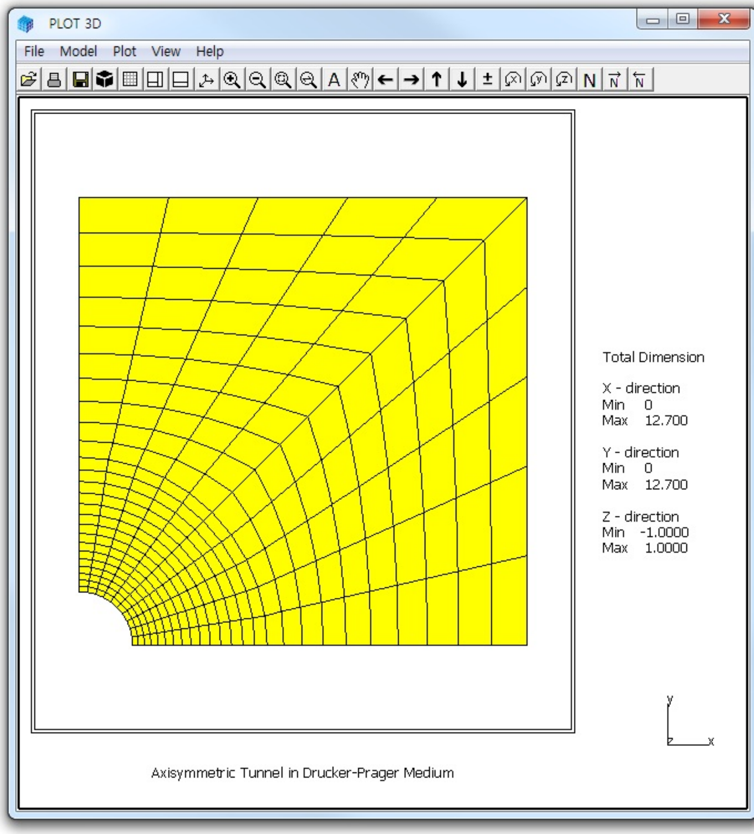
E = 810,000 t/m^2 Young's Modulus

ν = 0.33 Poisson's Ratio

σ_c = 1,268 t/m^2 (1800 psi) Unconfined Strength

ϕ = 18° Friction Angle

Figure 4.11 Circular tunnel subjected to axisymmetric loading



4.12 Finite element mesh

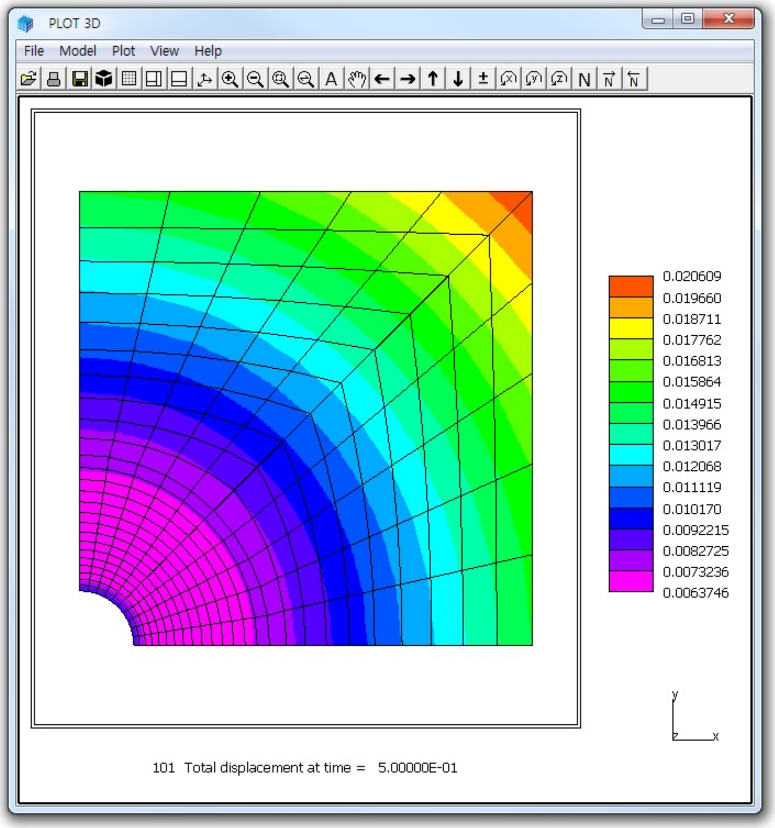


Figure 4.13 Tunnel displacement contour

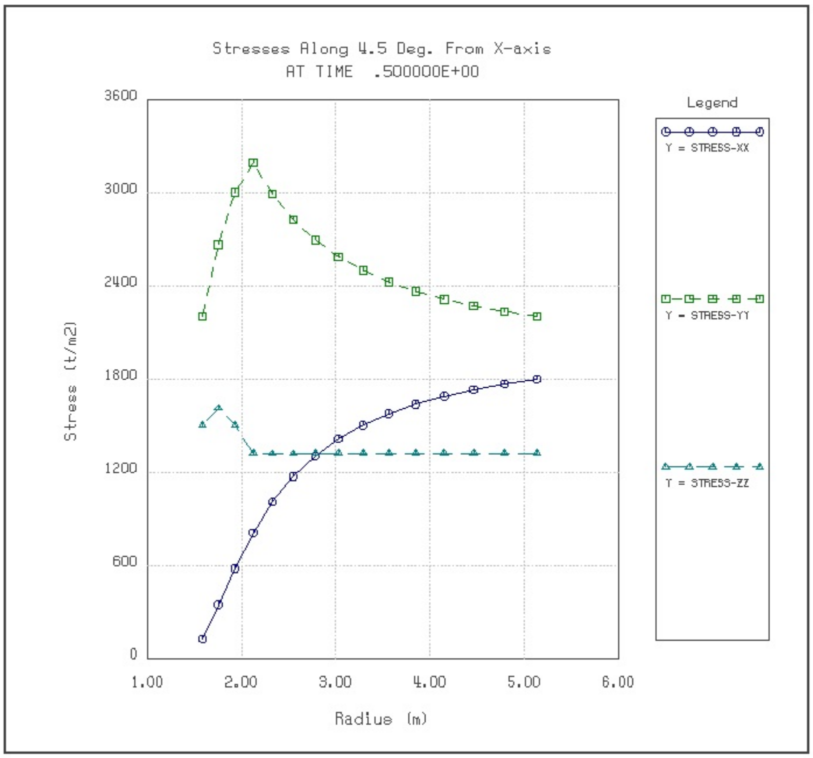


Figure 4.14 Stresses along 4 degree from X-axis

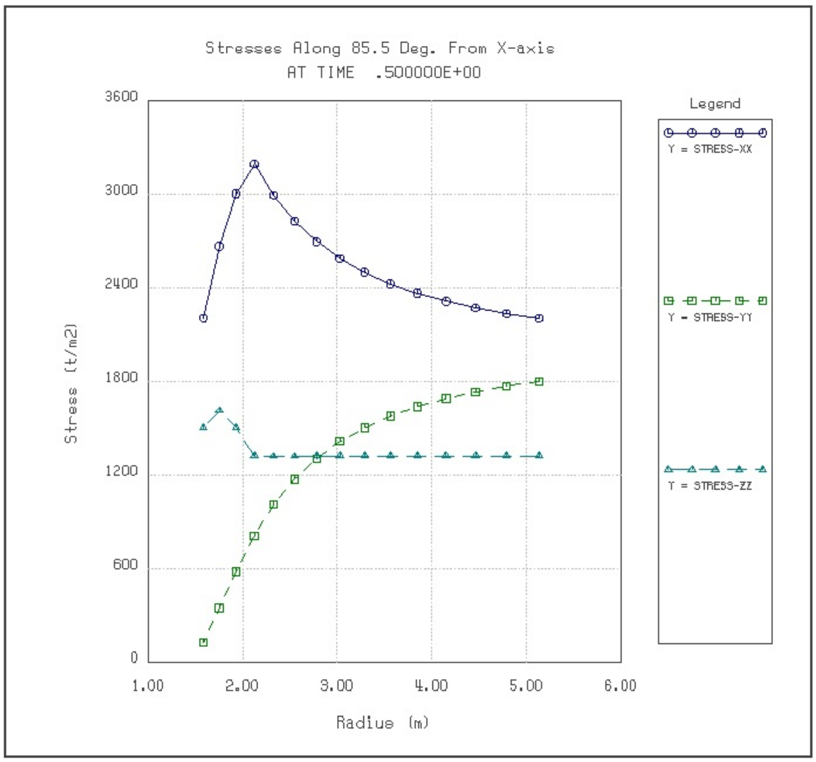


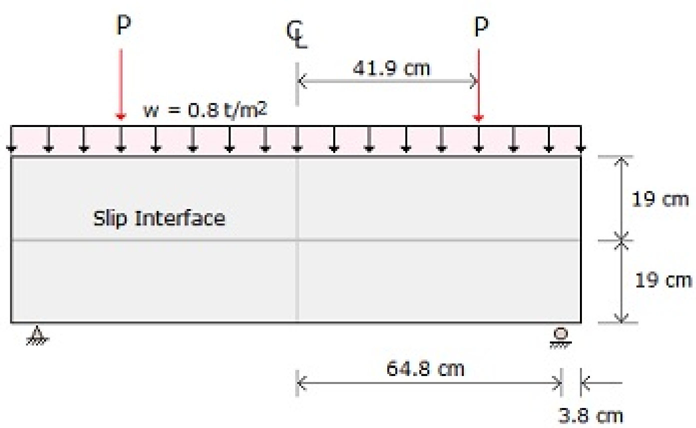
Figure 4.15 Stresses along 85.5 degree from X-axis

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 74. Thus, nodal coordinates along the other side of joint face are used mainly for visual presentation of joint elements. That is, program SMAP-2D resets internally the nodal coordinates of nodes from 79 to 88 equal to the nodal coordinates of the joint face (nodes from 4 to 74). 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-2D are compared to the closed-form solution derived from beam theory (Agbabian Associates, 1981). Overall, SMAP-2D 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-2D 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

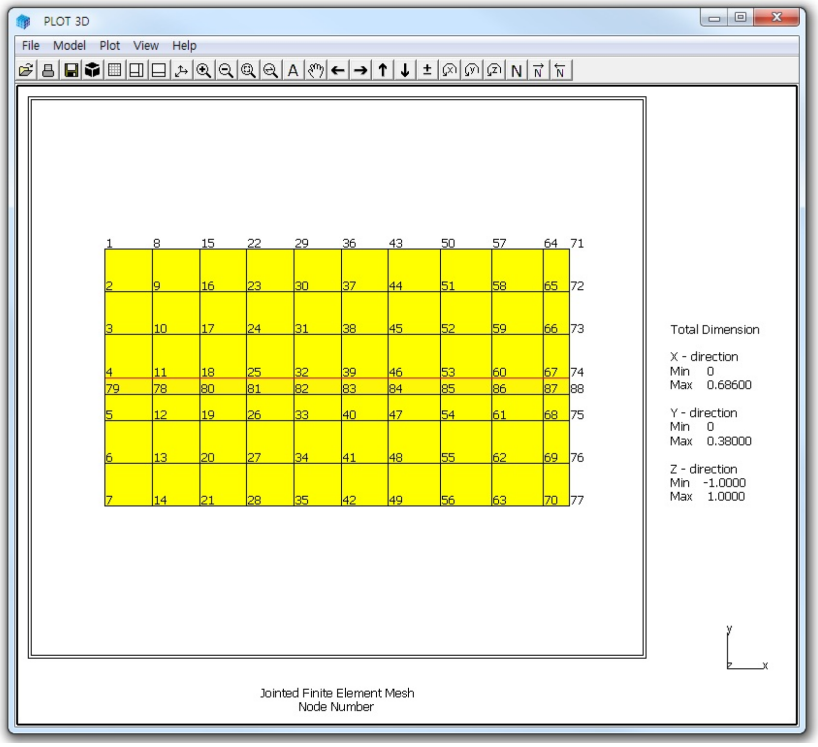


Figure 4.17 Node numbers of laminated beam

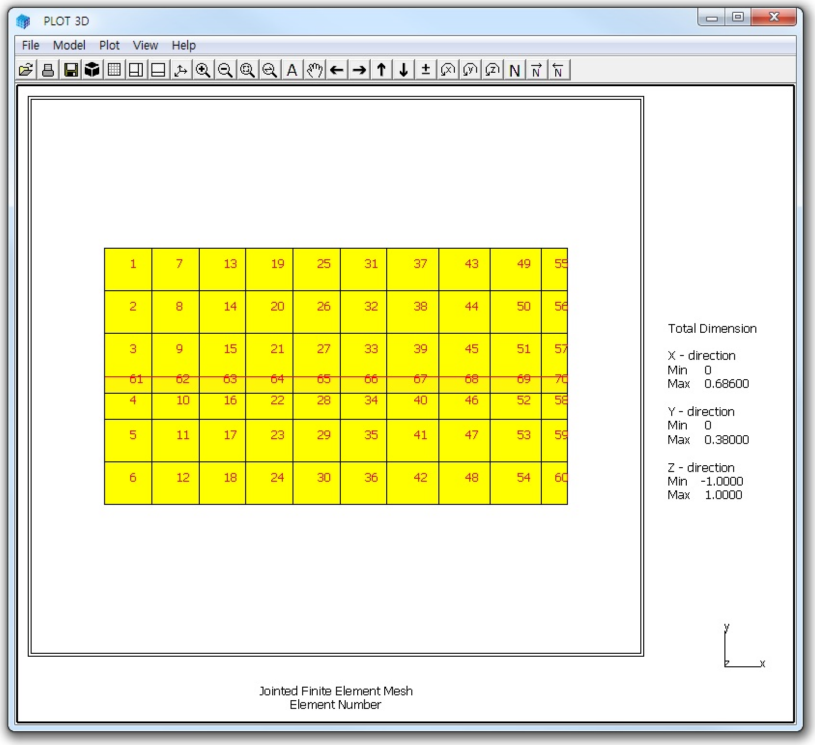


Figure 4.18 Element numbers of laminated beam

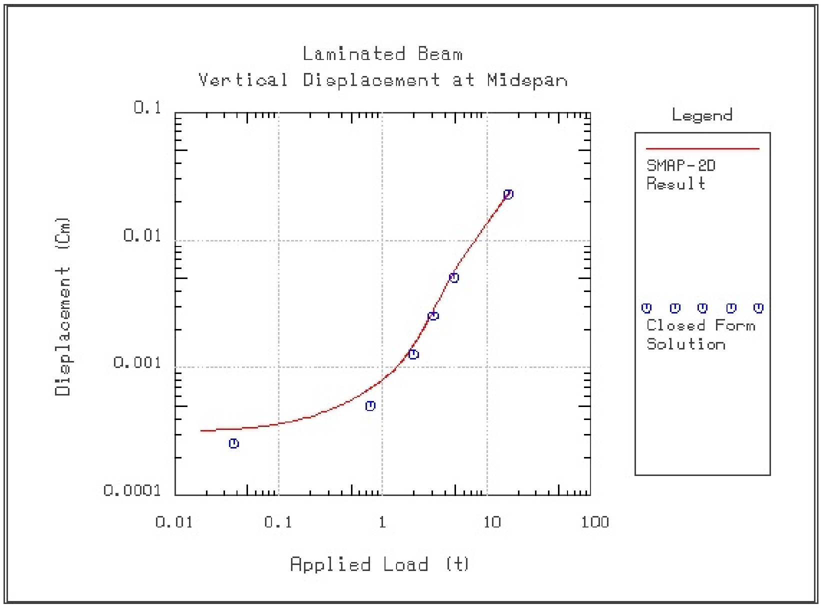


Figure 4.19 Vertical displacement at midspan

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 2D 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

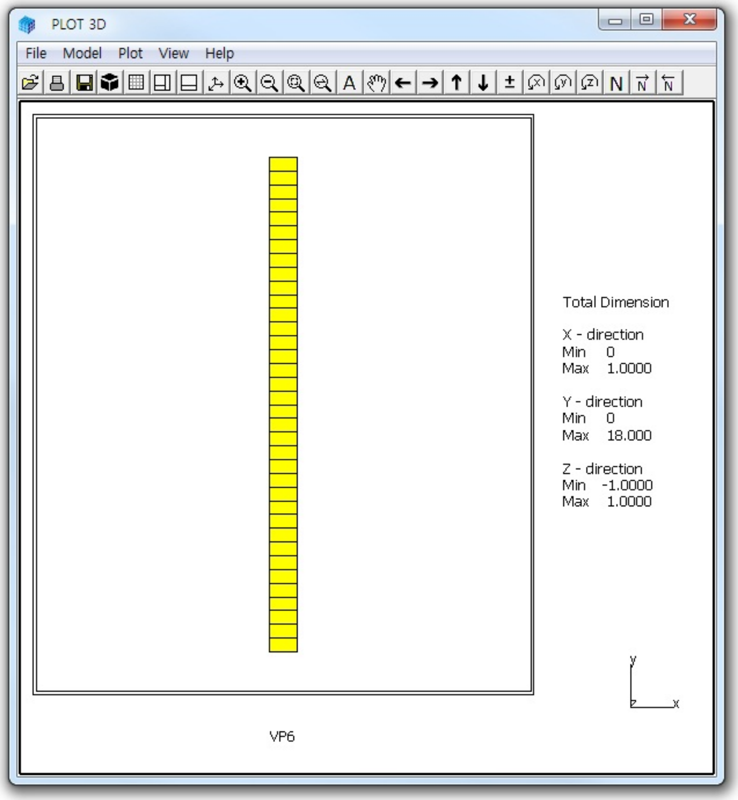
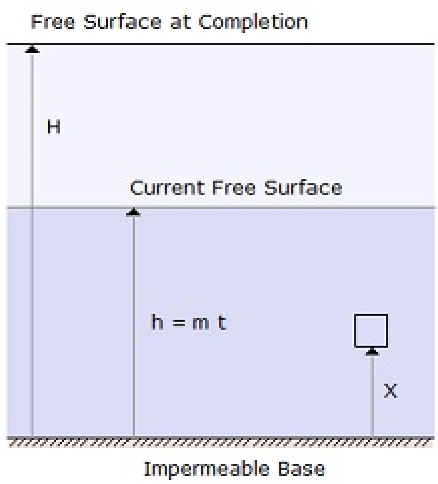


Figure 4.20 Finite element mesh



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 / γ_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-2D are very close to Gibson's exact solution.

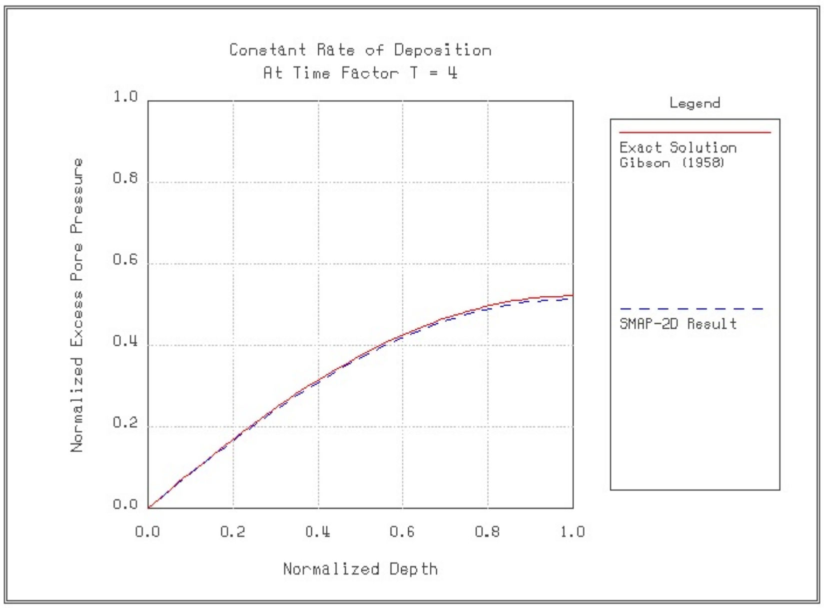


Figure 4.22 Normalized excess pressure profiles at T = 4

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 an axisymmetric 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-2D results reflect well the overall behavior of test results for the normally consolidated clay.

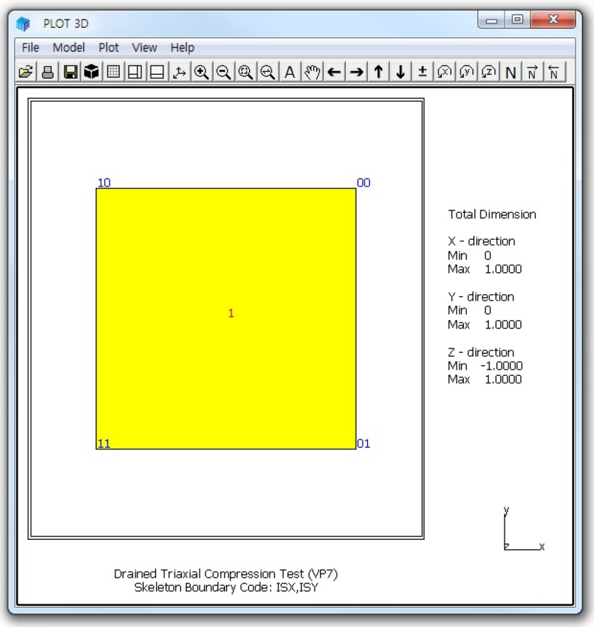
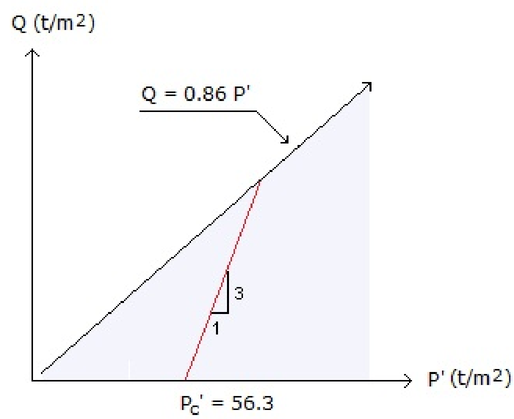


Figure 4.23 Finite element mesh



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

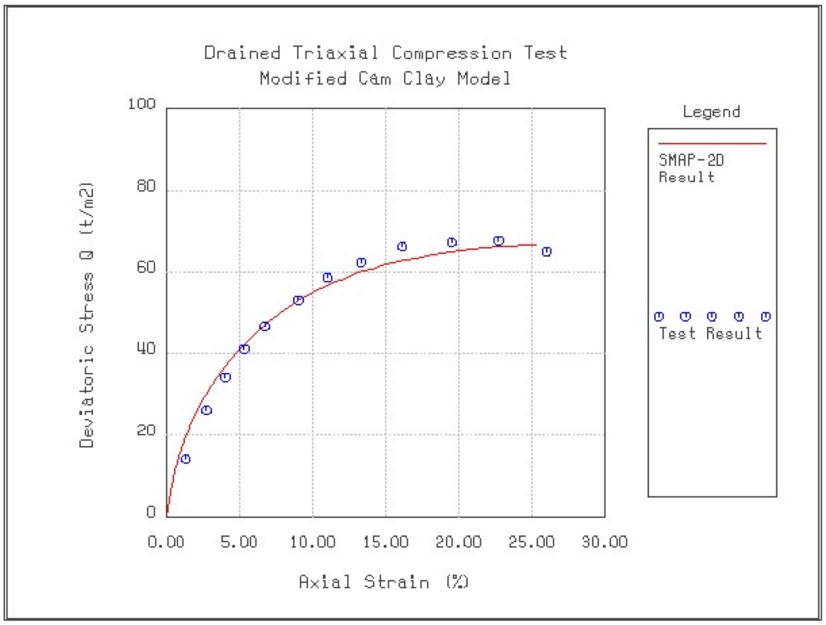


Figure 4.25 Deviatoric stress vs. axial strain for drained triaxial compression test

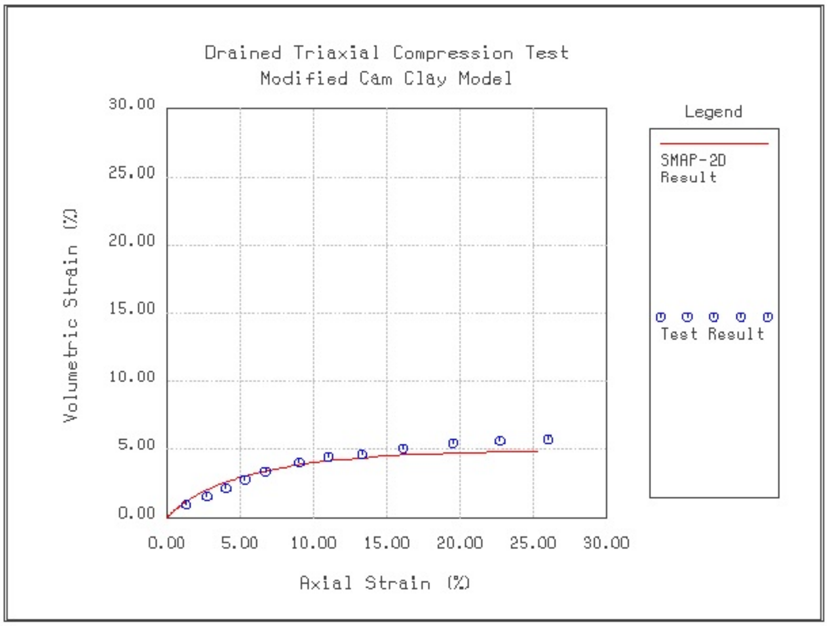


Figure 4.26 Volumetric strain vs. axial strain for drained triaxial compression test

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-2D 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.

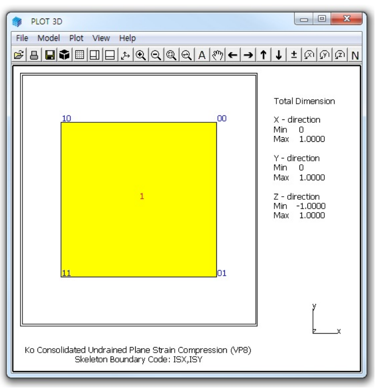


Figure 4.27 Finite element mesh

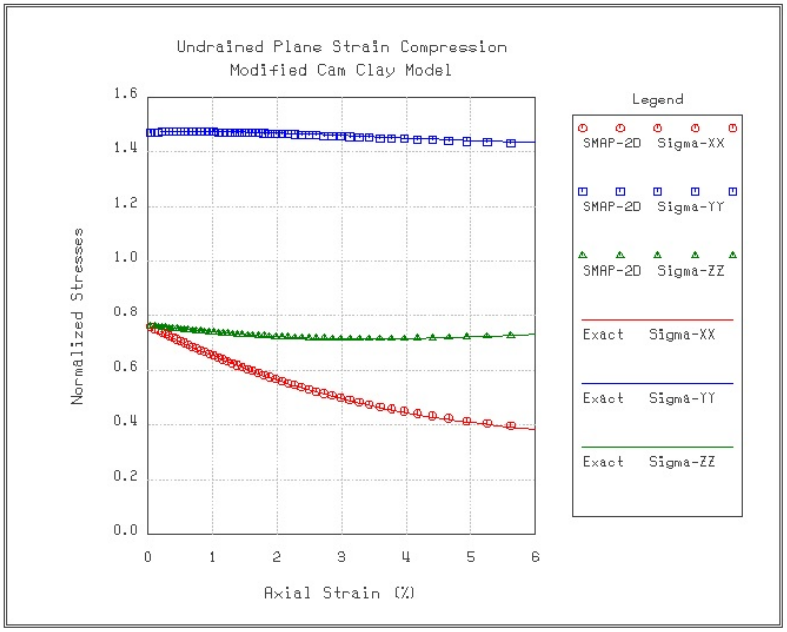


Figure 4.28 Effective stresses as a function of axial strain in K_o consolidated undrained plane strain compression test

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 axisymmetric element with unit length as shown in Figure 4.29.

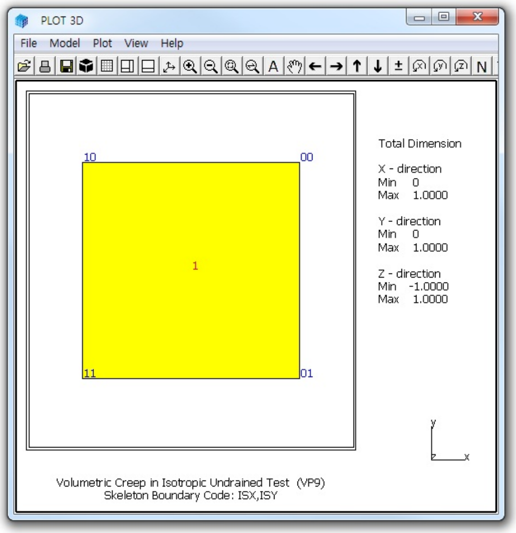


Figure 4.29 Finite element mesh

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-2D results are almost identical to the closed-form solution.

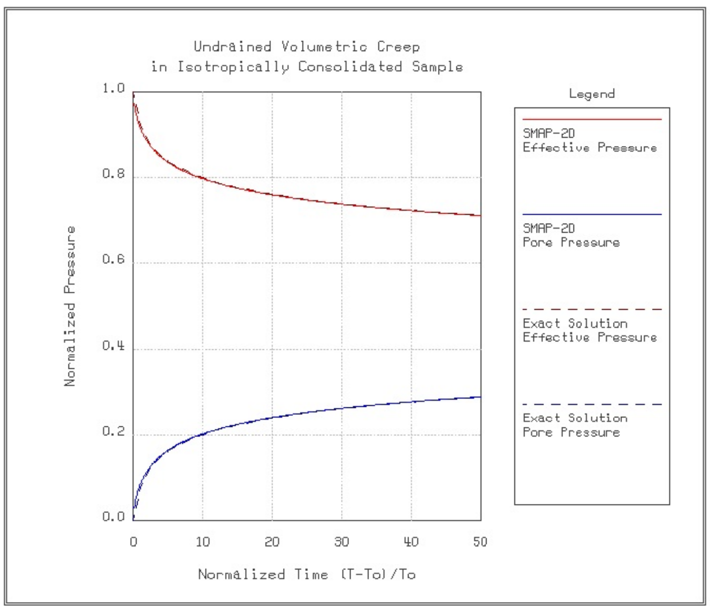
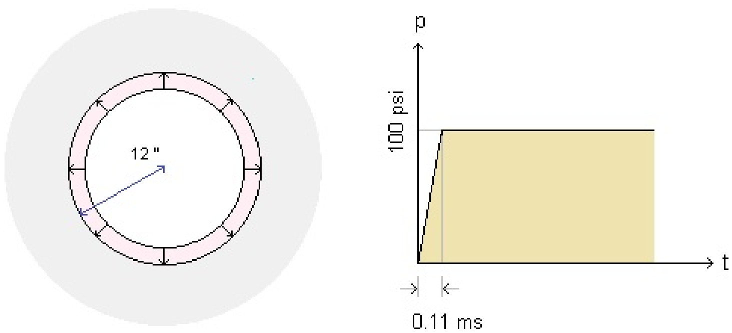


Figure 4.30 Volumetric creep in isotropically undrained test

4.10 Spherical Wave Propagation

Figure 4.31 shows a 12 inch hollow spherical hole in an infinite elastic medium subjected to a 100 psi internal step load. Material properties and time steps used for the calculations are included in Figure 4.31.

Graphical output from PLOT-XY is shown in Figure 4.32 along with the exact solution. SMAP-2D calculation gives good agreement with the exact solution.



Time Step	Δt	=	0.022 Msec
Young's Modulus	E	=	12,457 psi
Poisson's Ratio	v	=	0.25
Mass Density	ρ	=	1.88×10^{-4} lb-sec^2 /in^4

Figure 4.31 Loading time history and material property

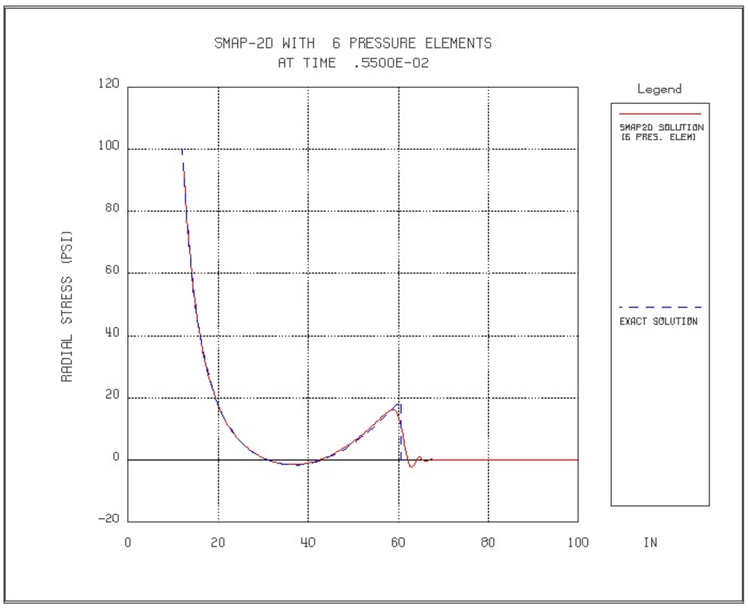
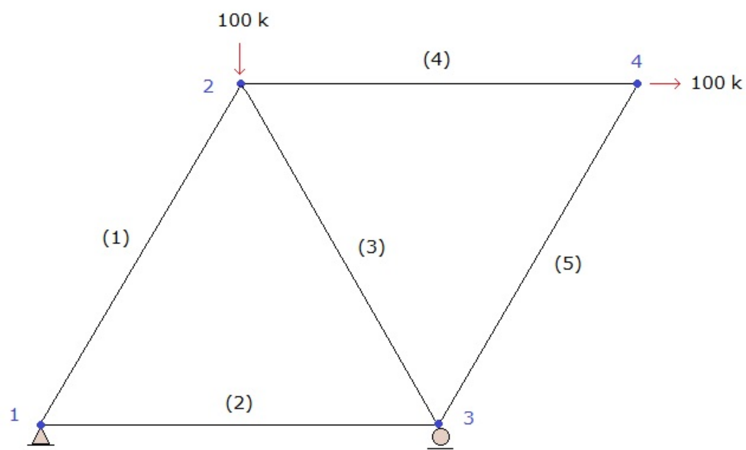


Figure 4.32 Radial stress profile at 5.5 msec

4.11 Elastic Truss Analysis

This example problem is to solve the static response of truss structure subjected to vertical and horizontal loads as shown in Figure 4.33. Member properties are listed in Figure 4.33.

Graphical outputs from PLOT-2D are shown in Figures 4.34 and 4.35 for deformed shape and member forces, respectively. Computed displacements and member forces are exact.



All members	L	=	10 ft.
	E	=	30×10^3 ksi
Member 1	A	=	2.5 in^2
Member 2	A	=	4.5 in^2
Member 3	A	=	12.0 in^2
Member 4	A	=	6.0 in^2
Member 5	A	=	6.0 in^2

Figure 4.33 Truss section view and member properties

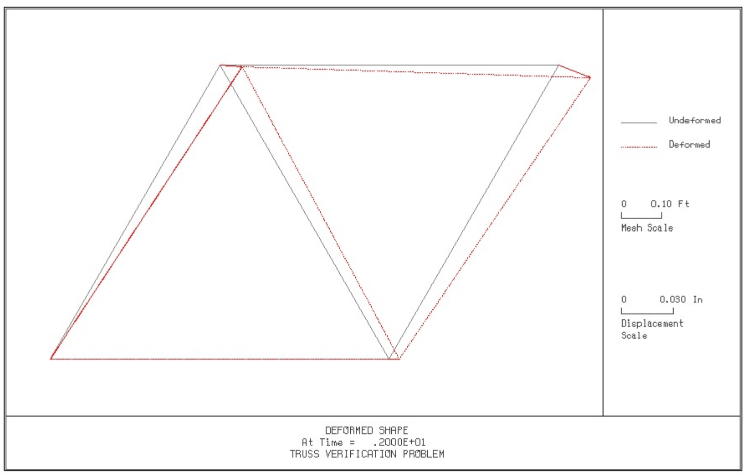


Figure 4.34 Truss deformed shape

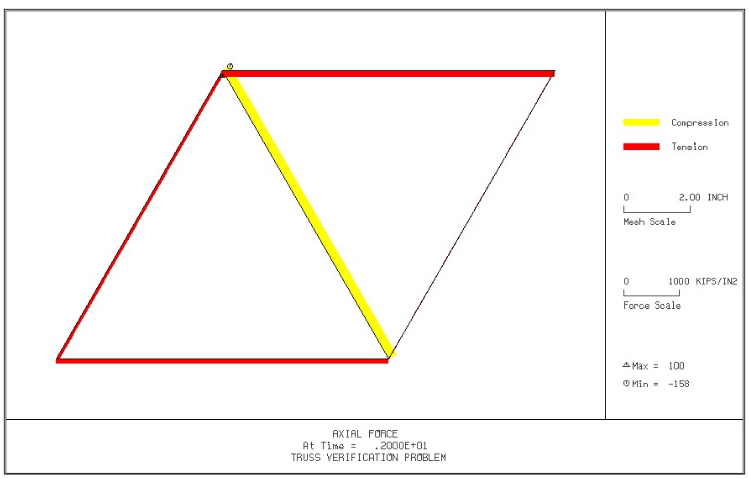


Figure 4.35 Truss member forces

4.12 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 4.36.

The exact solution for this beam is given below

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

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

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

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 4.37. Graphical outputs are plotted in Figures 4.38 and 4.39 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.

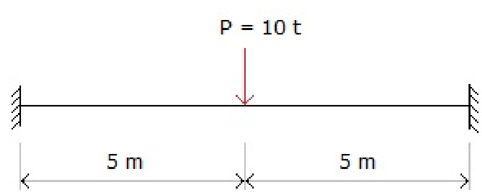


Figure 4.36 Fixed end beam subjected to concentrated load

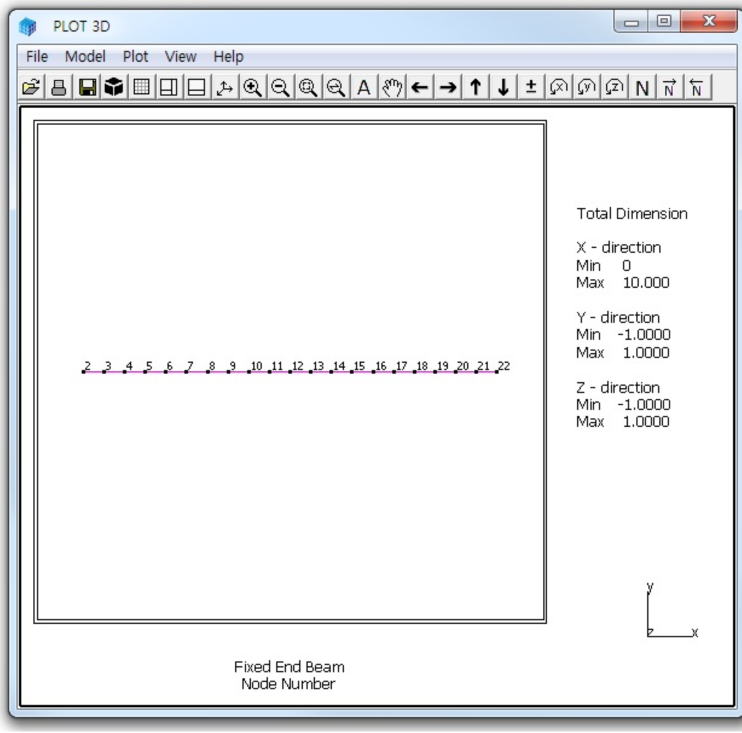


Figure 4.37 Beam node numbers

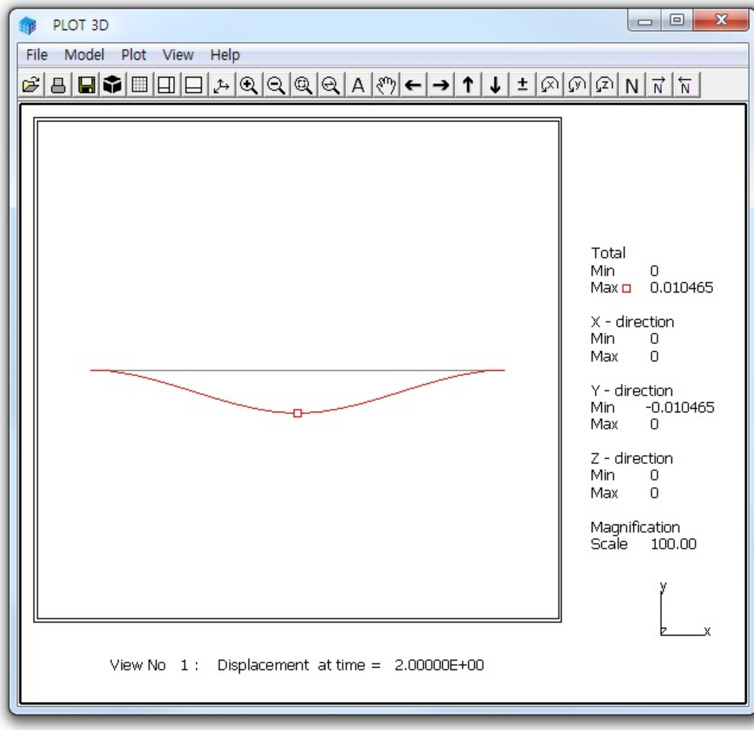


Figure 4.38 Beam deformed shape

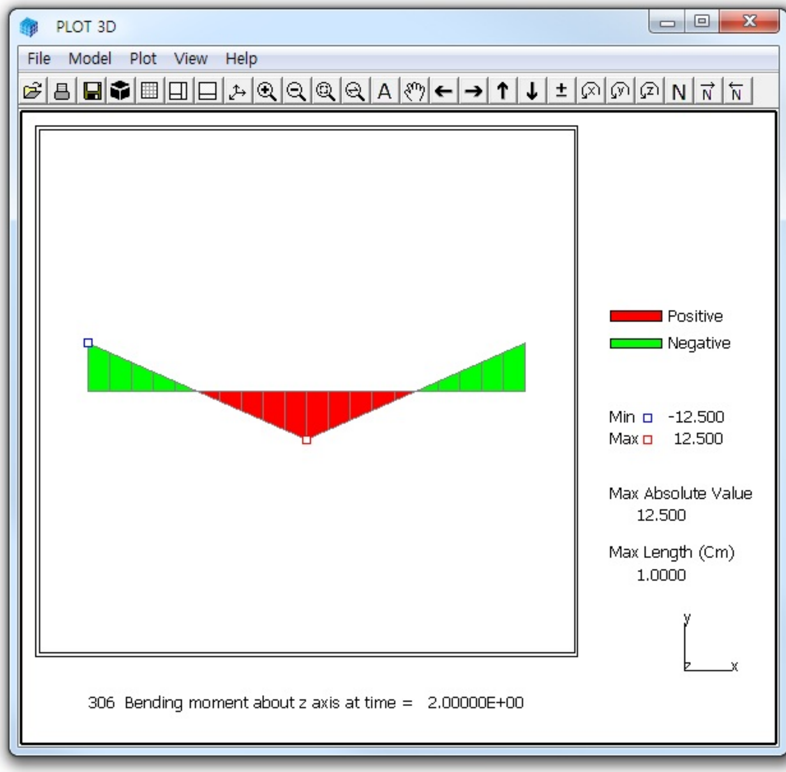


Figure 4.39 Bending moment diagram

4.13 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:

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

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

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 4.40 has been performed using the following parameters:

I_o = 0.1 t-sec

ρ	=	0.786 $t\text{-}s^2/m^4$	L	=	10 m
A	=	0.008412 m^2	I	=	2.37×10^{-4} m^4
E	=	21×10^6 t/m^2	ν	=	0.3

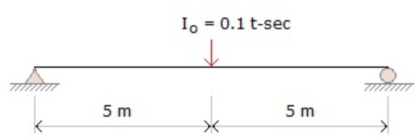


Figure 4.40 Simply supported beam subjected to impact load

The problem is modeled by 20 beam elements as shown in Fig. 4.41.
And impact load is simulated by the initial velocity applied at mid span.

Figure 4.42 shows the deformed shape at time t = 0.1 second.
Figure 4.43 shows time history plot of deflection at mid span.
SMAP-2D results agree well with the exact solution.

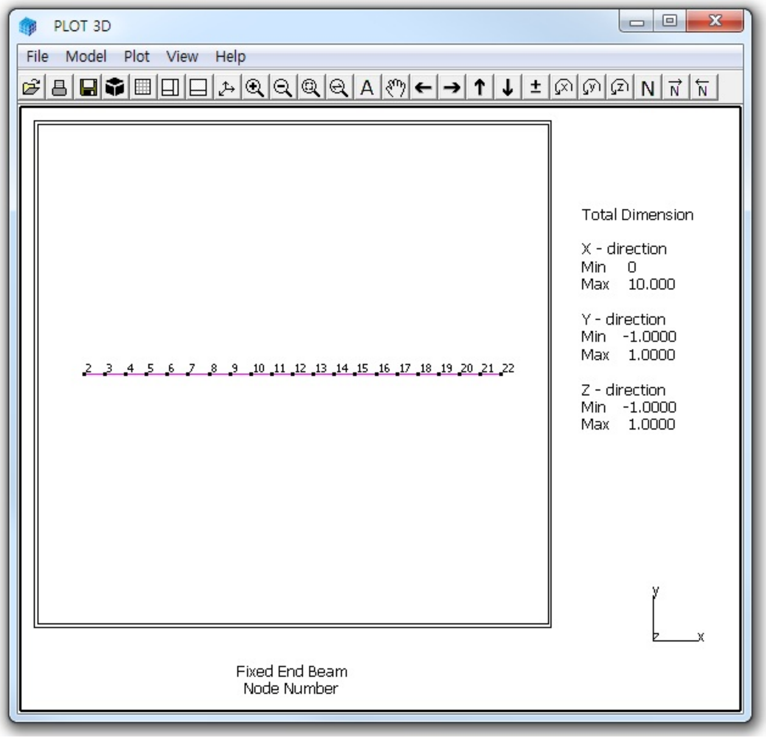


Figure 4.41 Beam node numbers

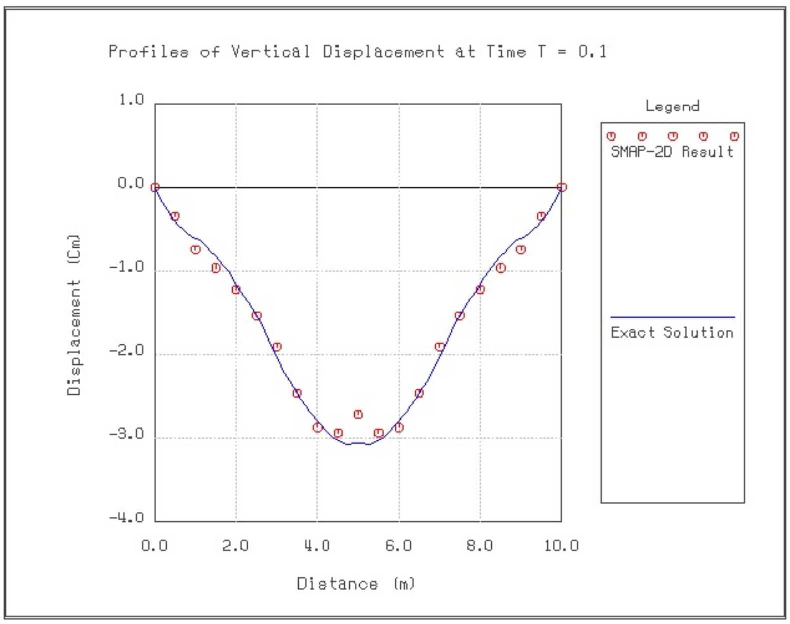


Figure 4.42 Beam deformed shape at 100 msec

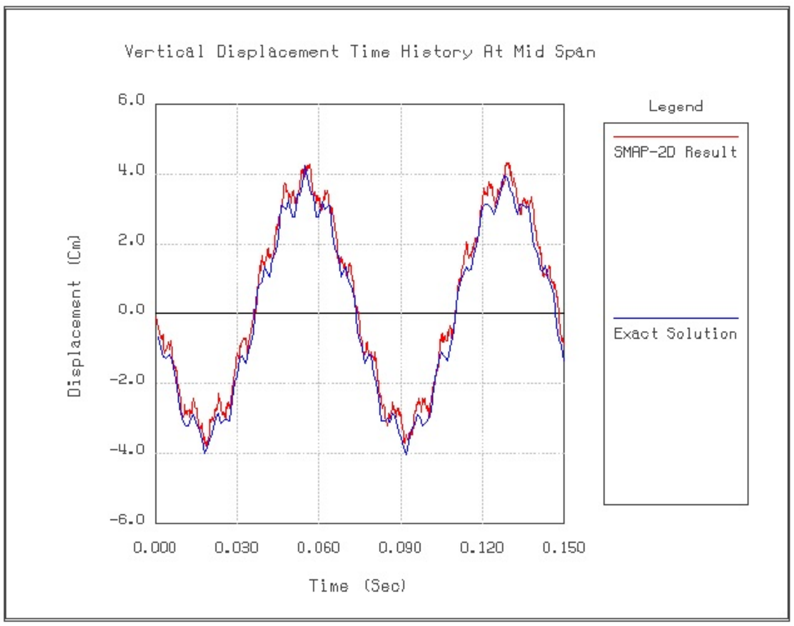


Figure 4.43 Time history of deflection at mid span

4.14 Burn's and Siess' Beam Analysis

Burns and Siess' beam, shown in Figure 4.44 is the reinforced concrete beam which was tested at the University of Illinois. The objective of this problem is to verify concrete cracking and steel bar yielding when the concentrated load at midspan is increased monotonically to failure. The cracking moment of concrete is given by

$$M_c = \frac{f_t \, I_t}{C}$$

where

- f_t Tensile strength (546 psi)
- I_t Moment of inertia of transformed section (5988 in^4)
- C Distance from the neutral axis of the transformed section to the bottom of the beam (9.489 in)

And the cracking load, P_c, is given by

$$P_c = \frac{4M_c}{L} = \frac{4(28.71)}{12} = 9.57 \text{ kips}$$

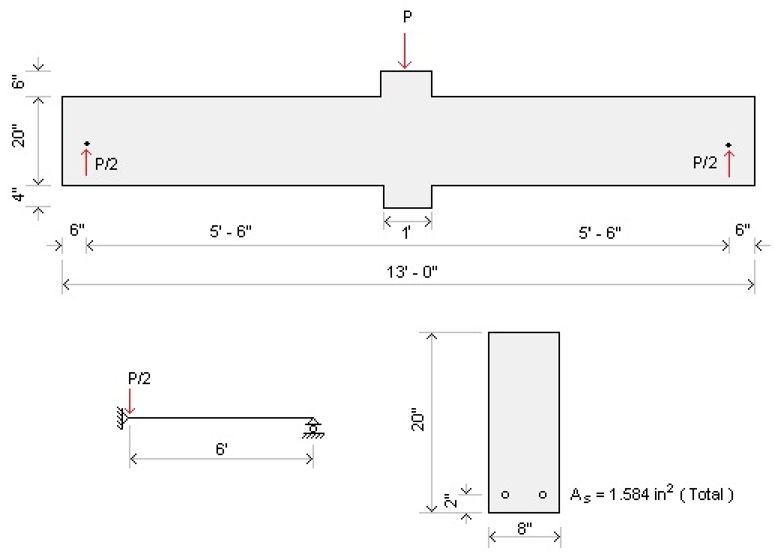
The maximum moment capacity, M_{max}, can be estimated by simplified ultimate strength theory as

$$M_{max} = A_s f_y \left(d - \frac{0.5 \, A_s \, F_y}{0.85 \, f_c' \, b}\right)$$

and the corresponding maximum load, P_{max}, is given by

$$P_{max} = \frac{4M_{max}}{L} = \frac{4(100.24)}{12} = 33.41 \text{ kips}$$

At cracking load, it is expected that the change of load-deflection curve from the initial elastic response would occur. And at the maximum load, it is expected that deflections would begin to increase rapidly.



Concrete

E = 3.8 x 10^6 psi

υ = 0.15

f'_c = 4820 psi

f_t = 546 psi

φ = 40°

Steel Bar

E = 30 x 10^6 psi

υ = 0.3

f_y = 44,900 psi

Figure 4.44 Burns and Siess' beam

SMAP-2D model is shown in Figure 4.45. A total of 22 beam elements is used to model the right half of the structure. A constant concentrated load increment of 40 lbs is applied.

Figure 4.46 shows the load-deflection response at midspan. Figure 4.47 shows the deformed shapes at the applied loads; 20 and 32 kips. As seen, there is very good agreement between the experimental and SMAP-2D results.

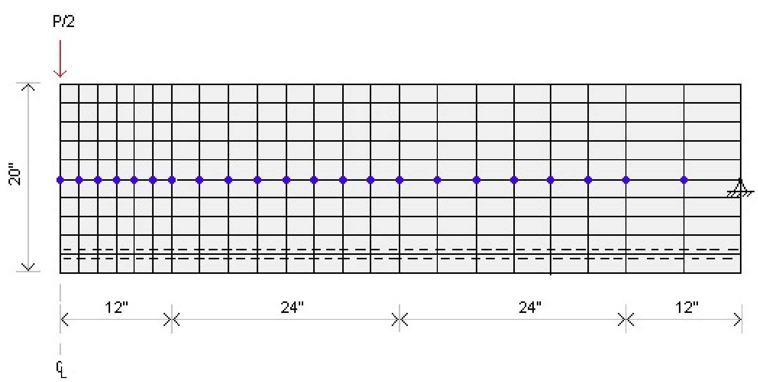


Figure 4.45 SMAP-2D model of Burns and Siess' beam

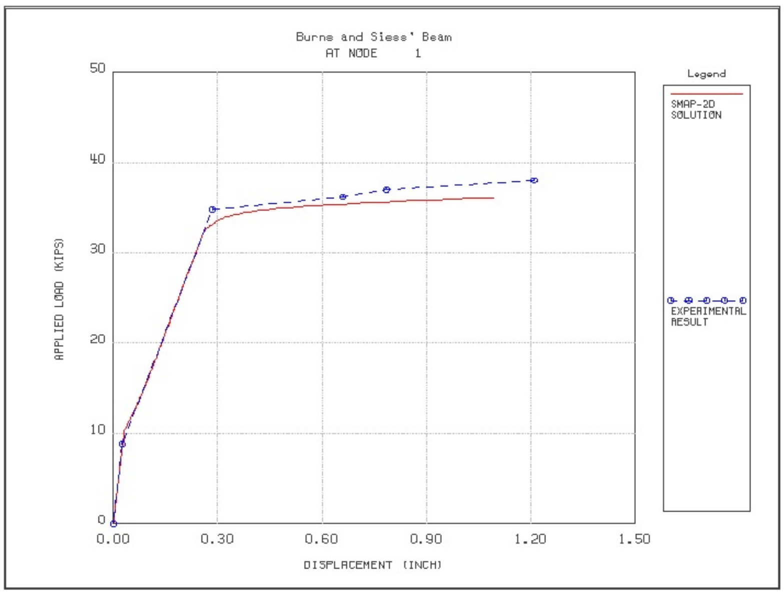


Figure 4.46 Load-deflection curve

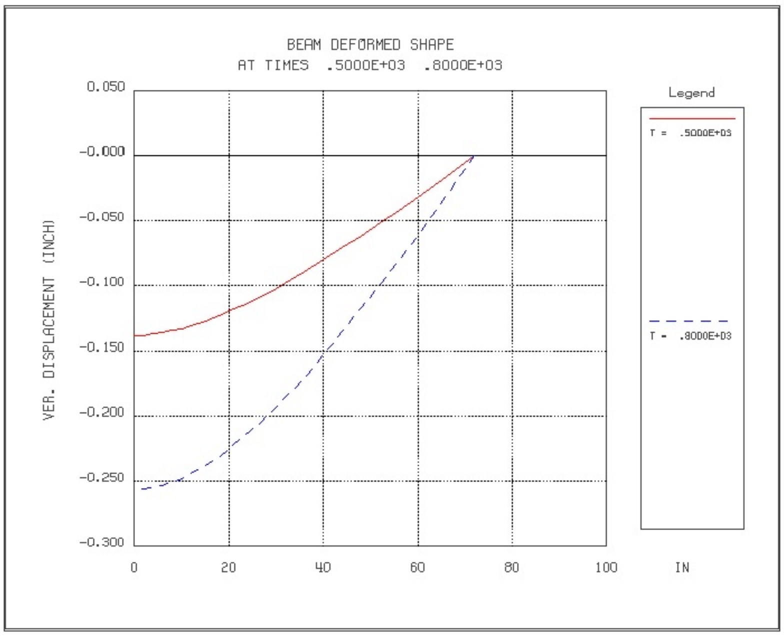


Figure 4.47 Deformed shapes at 20 and 32 kips

4.15 William's Toggled Beam Analysis

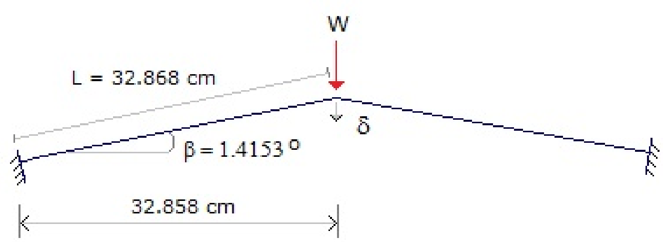
This classic problem of a rigidly jointed toggle is selected to verify the geometric nonlinear behavior of the continuum element.

For the toggle shown in Figure 4.48 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 and 4 elements across the depth.

Figures 4.49 and 4.50 show the load-deflection response at mid span and deformed shape at applied load of 16 kg, respectively.
SMAP-2D results are close to the Williams' closed form solution.

The same problem is modeled by 20 beam elements. Results are very close to closed form solution as shown in Figure 4.51.



$E = 7.238 \times 10^6 \ t/m^2$
$I = 3.746 \times 10^{-2} \ cm^4$ $A = 1.18 \ cm^2$

Figure 4.48 William's toggled beam (Not Scaled)

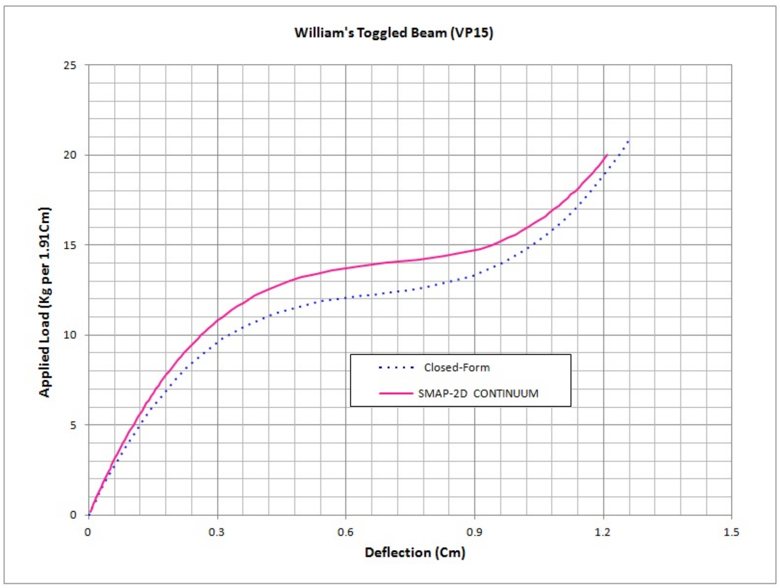


Figure 4.49 Load-deflection curve using continuum element

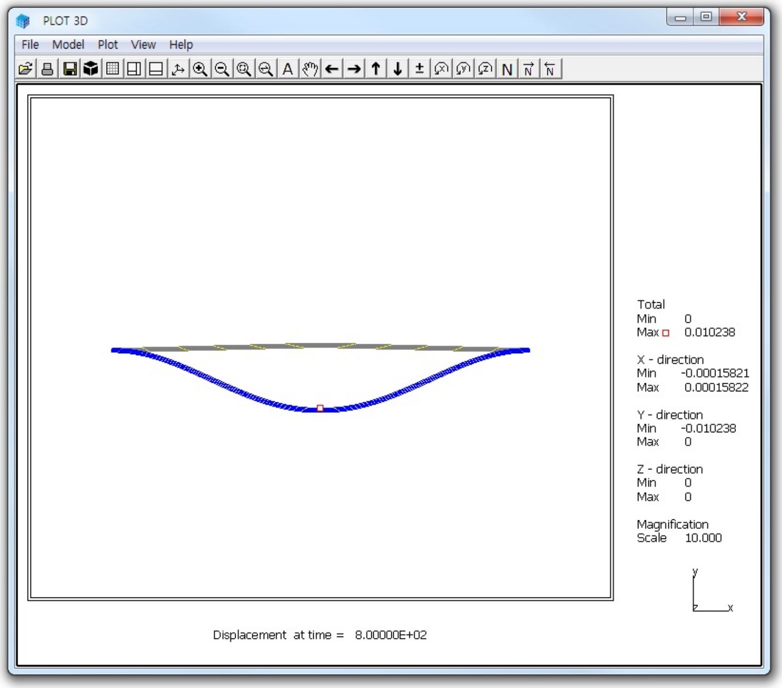


Figure 4.50 Deformed shape at applied load of 16 kg

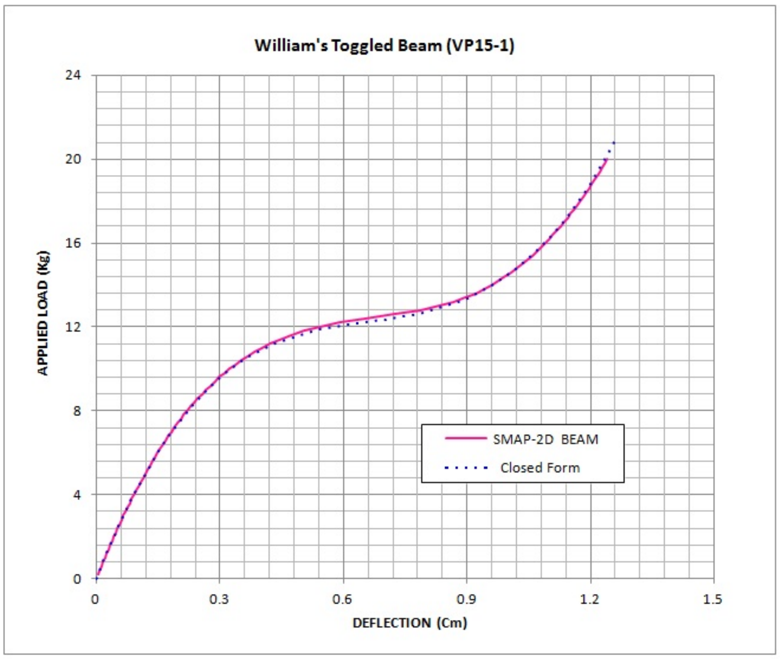


Figure 4.51 Load-deflection curve using beam element

4.16 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 4.52 shows schematic tunnel section view and material properties of soil and steel liner.

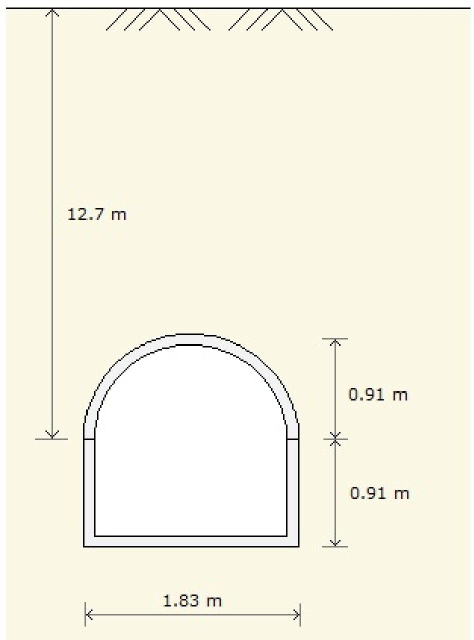
Figure 4.53 shows Finite element mesh. By symmetry, only the right half of the tunnel is modeled. Tunnel liner is modeled by beam elements as shown in Figure 4.54. Block mesh example 4 illustrates how to generate this mesh.

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 4.55 Tunnel deformed shape
- Figure 4.56 Tunnel liner bending moment
- Figure 4.57 Tunnel liner axial stress
- Figure 4.58 Principal stress vector
- Figure 4.59 Major principal stress distribution
- Figure 4.60 Minor principal stress distribution

SMAP-2D results are close to SMAP-S2 results.



Soil

E = 7,030 t/m^2

ν = 0.23

K_o = 0.3

Steel Liner

E = 20.4 $x10^6$ t/m^2

ν = 0.3

t = 2.54 cm (Thickness)

Figure 4.52 Schematic tunnel section view (Not Scaled)

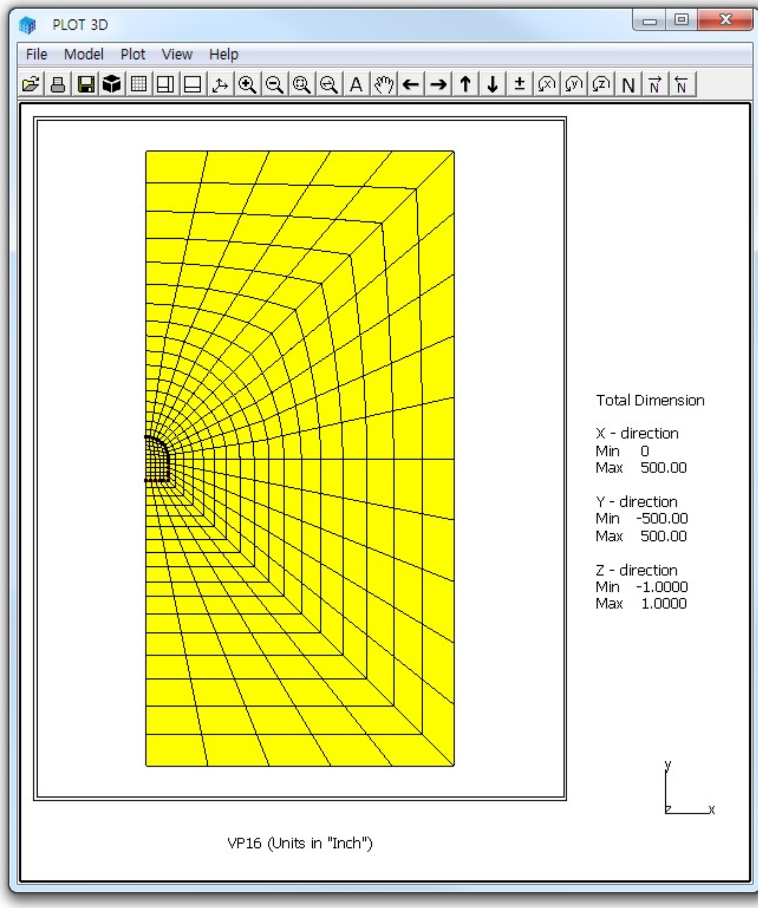


Figure 4.53 Finite element mesh

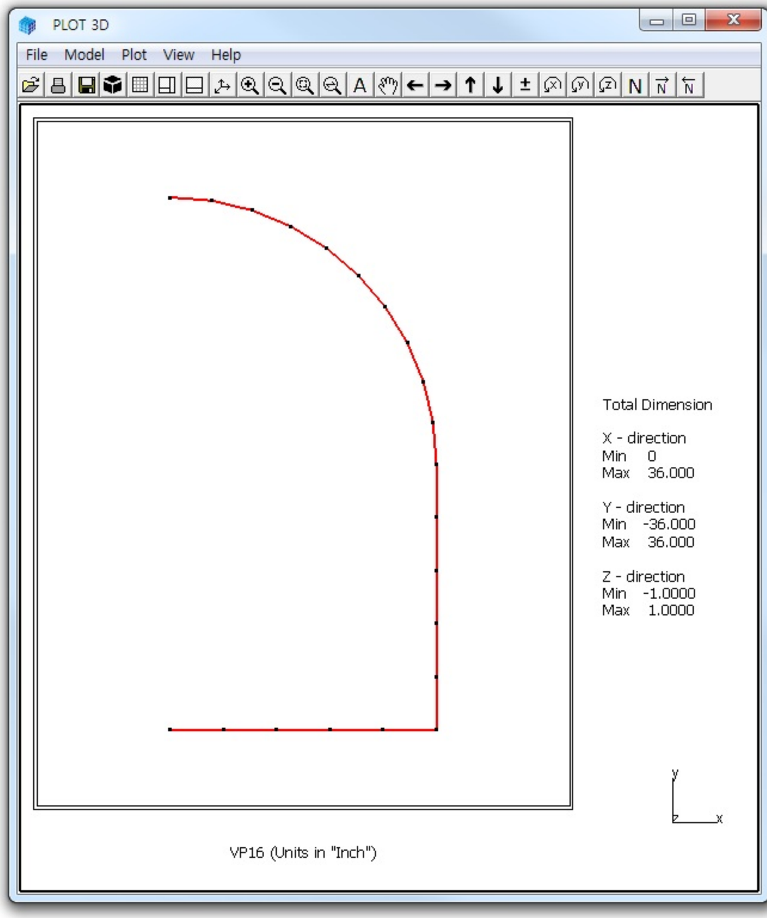


Figure 4.54 Finite element mesh for liner

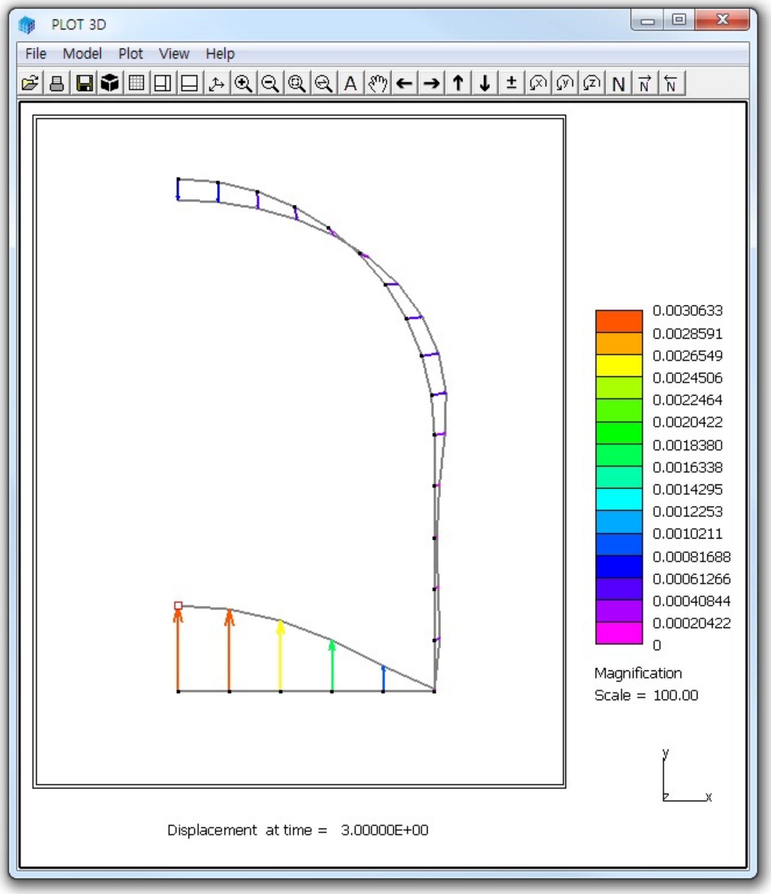


Figure 4.55 Tunnel deformed shape

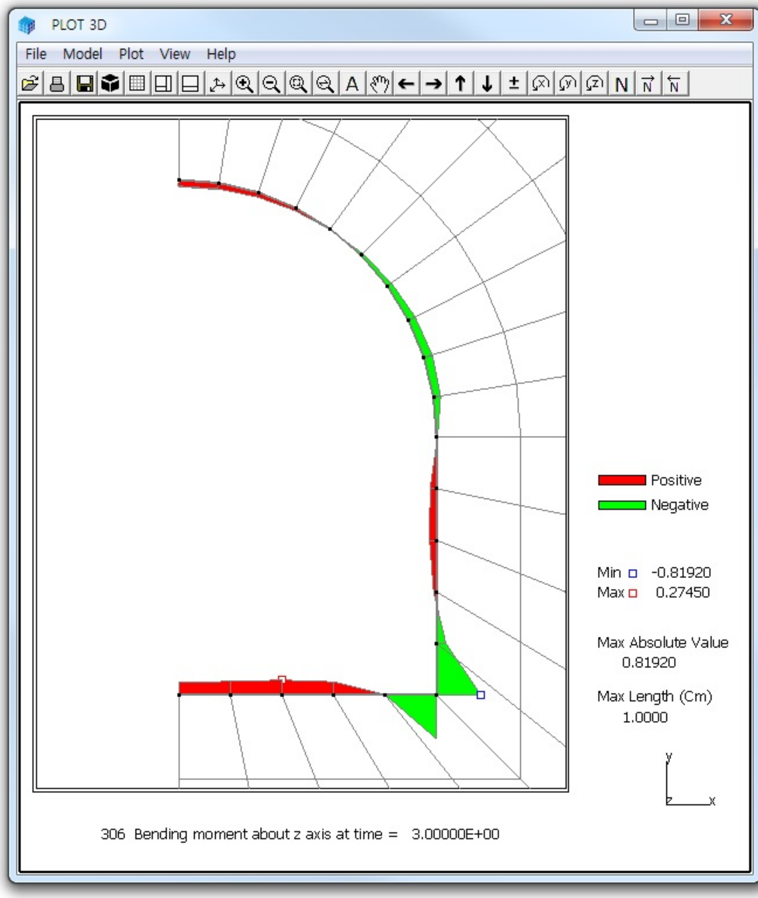


Figure 4.56 Tunnel liner bending moment

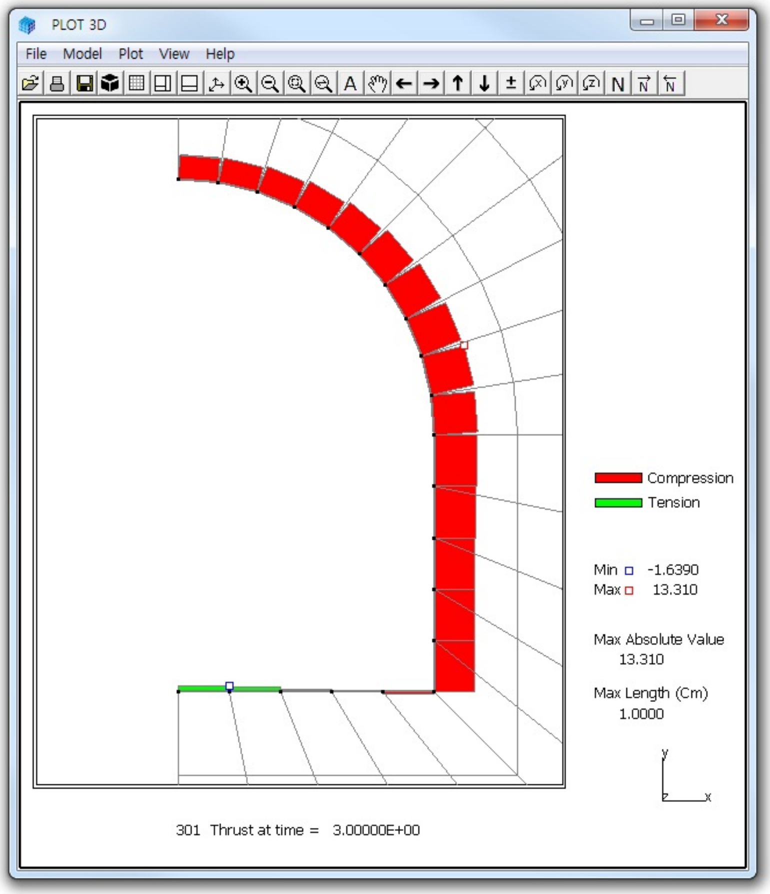


Figure 4.57 Tunnel liner axial force

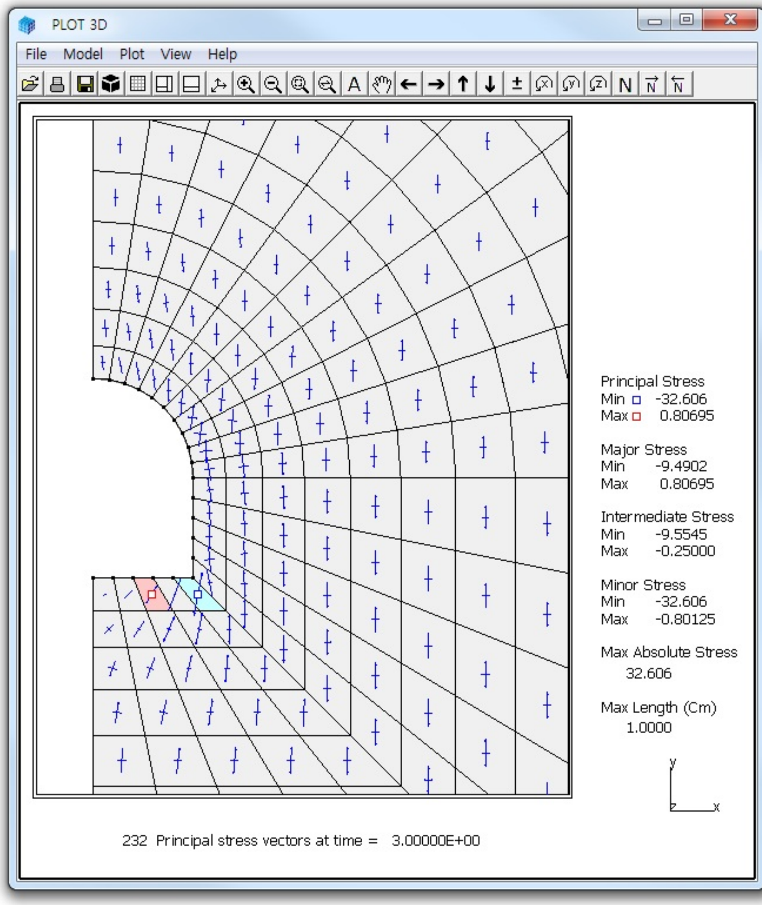


Figure 4.58 Principal stress vector

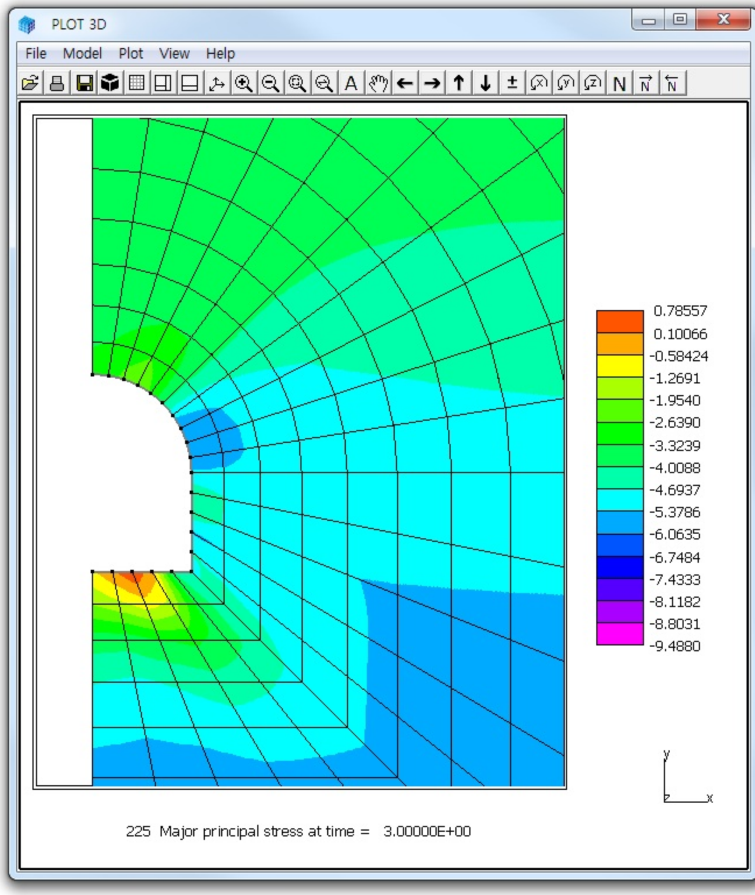


Figure 4.59 Major principal stress distribution

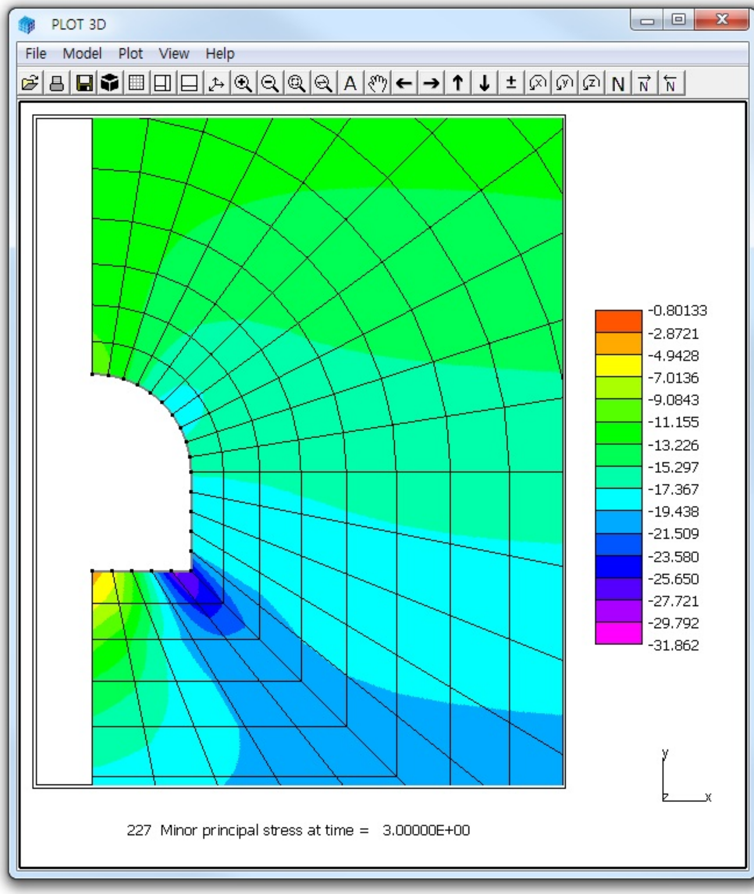


Figure 4.60 Minor principal stress distribution

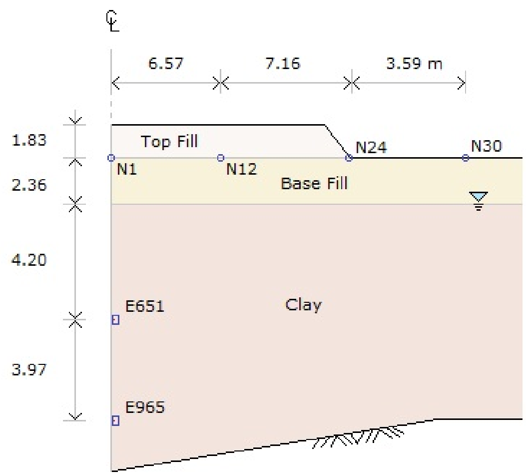
4.17 Embankment Construction

Figure 4.61 shows schematic embankment section view and material properties. Embankment construction consists of pervious base and top fills. These fills were placed on saturated clay deposit at constant construction rate of 2.62 Cm/day.

First 100 steps were used to simulate in situ stress state of normally consolidated clay. Consolidation analysis was performed with constant time step of 1 day.

Graphical outputs from PLOT-XY are presented in Figures 4.62 to 4.64. Figures 4.62 and 4.63 show time histories of effective vertical stress and pore water pressure at selected elements 651 and 965, respectively. Figure 4.64 shows vertical displacement time history at selected nodes 1, 12, 24 and 30.

Graphical outputs from PLOT-3D are presented in Figures 4.65 to 4.71. Figure 4.65 shows undeformed shape of finite element mesh. Figures 4.66 and 4.67 show deformed shapes of finite element mesh at the completion of embankment and at two years after completion, respectively. Pore water pressure profiles are shown in Figures 4.68 and 4.69 at the completion of embankment and two years after completion, respectively. Effective vertical stress profiles are shown in Figures 4.70 and 4.71 at the completion of embankment and two years after completion.



Top Fill

n = 0.4 G_s = 2.7 E = 1000 t/m^2 ν = 0.2

Base Fill

n = 0.4 G_s = 2.7 E = 1500 t/m^2 ν = 0.2

Clay

n = 0.45 G_s = 2.7 e_o = 1.74 ν = 0.3

C_c = 0.338 C_r = 0.138 M = 1.05

Construction Rate m = 2.62 cm/day

Figure 4.61 Schematic section view and material properties

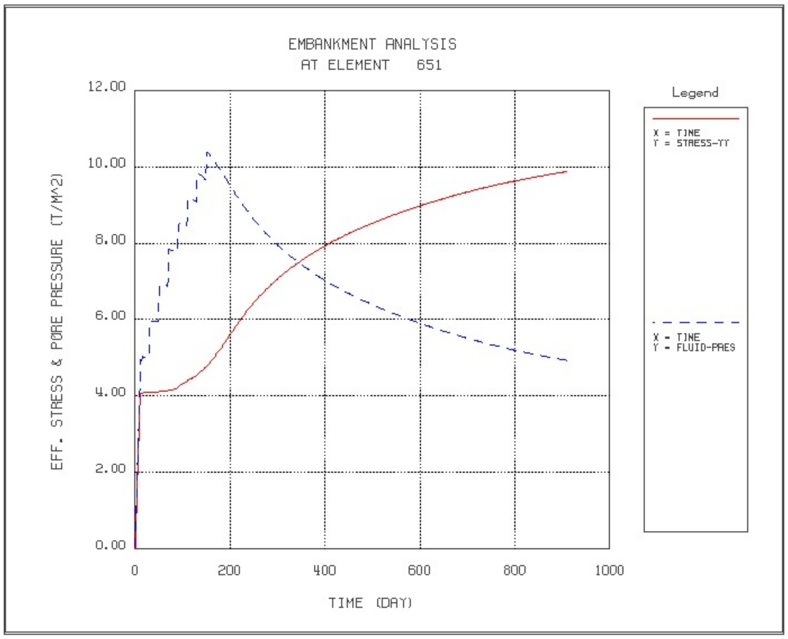


Figure 4.62 Stress time history at element 651

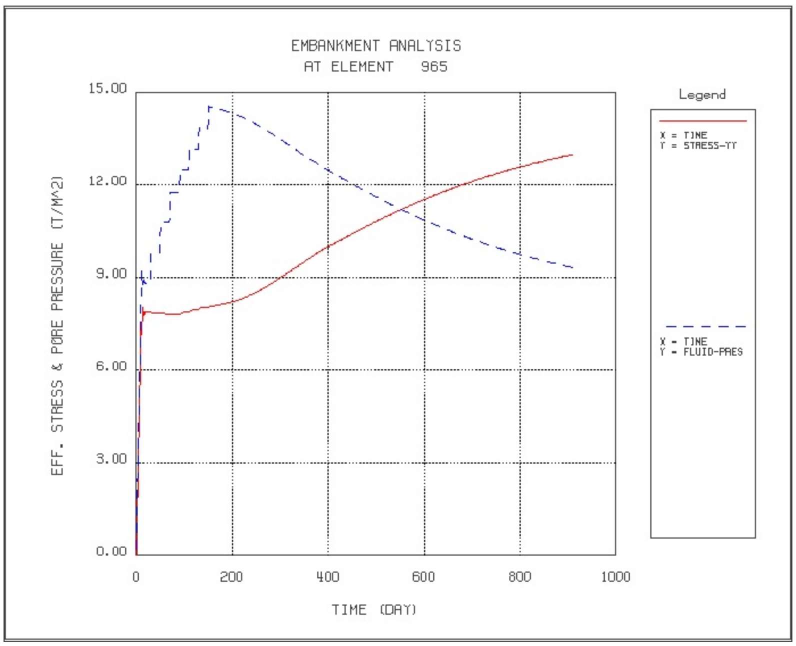


Figure 4.63 Stress time history at element 965

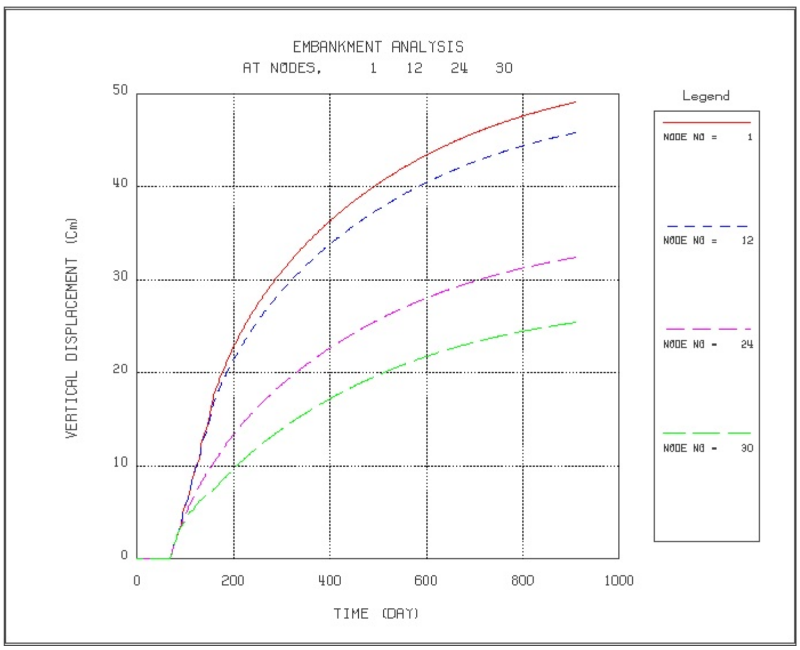


Figure 4.64 Vertical displacement history at selected nodes

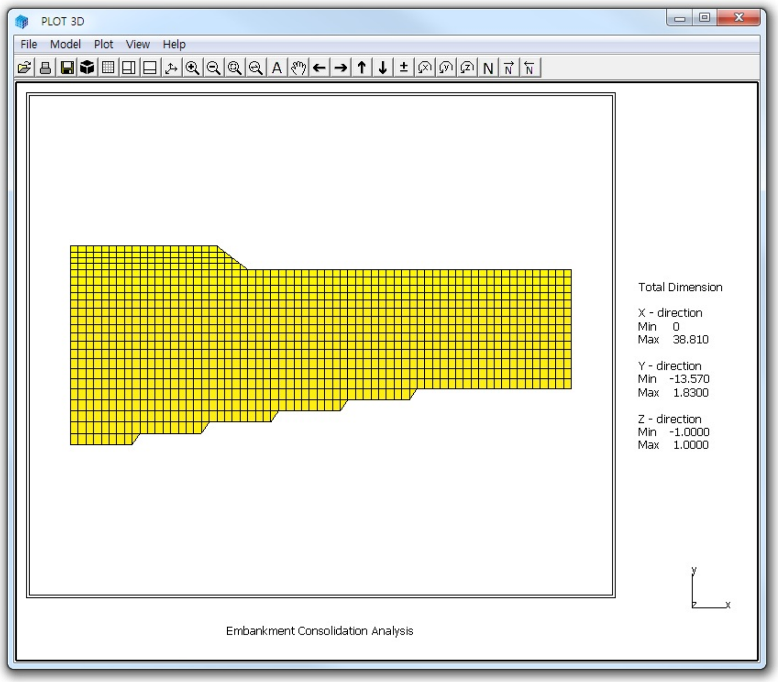


Figure 4.65 Undeformed shape of finite element mesh

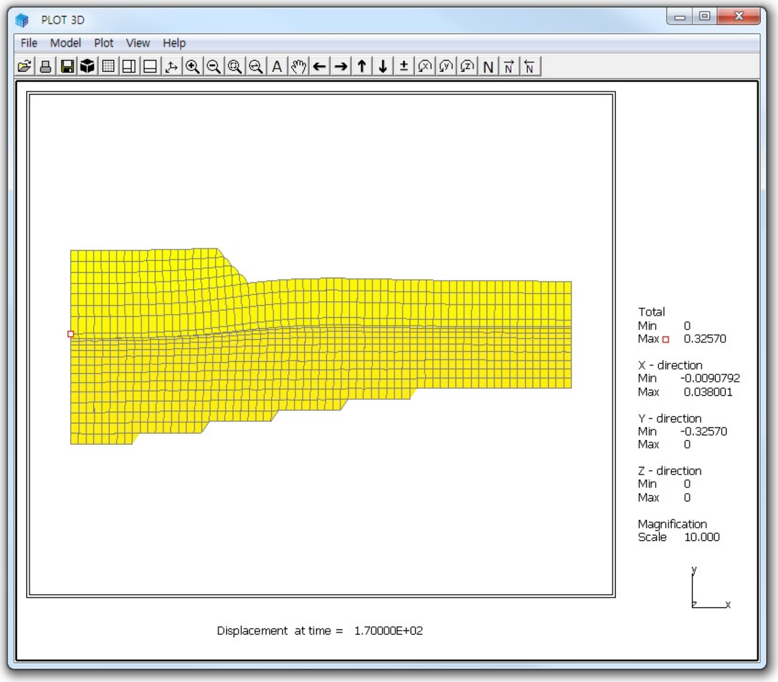


Figure 4.66 Deformed shape at completion of embankment

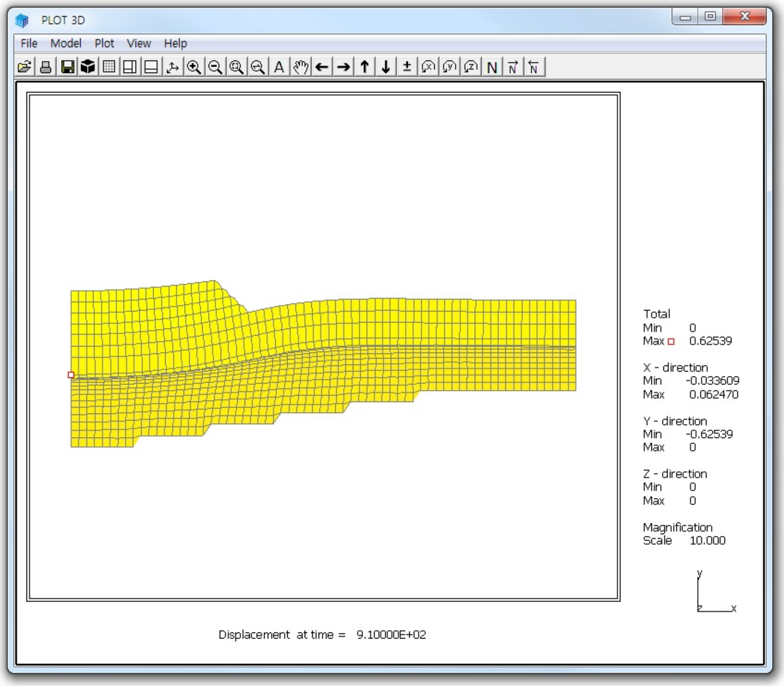


Figure 4.67 Deformed shape at two years after completion

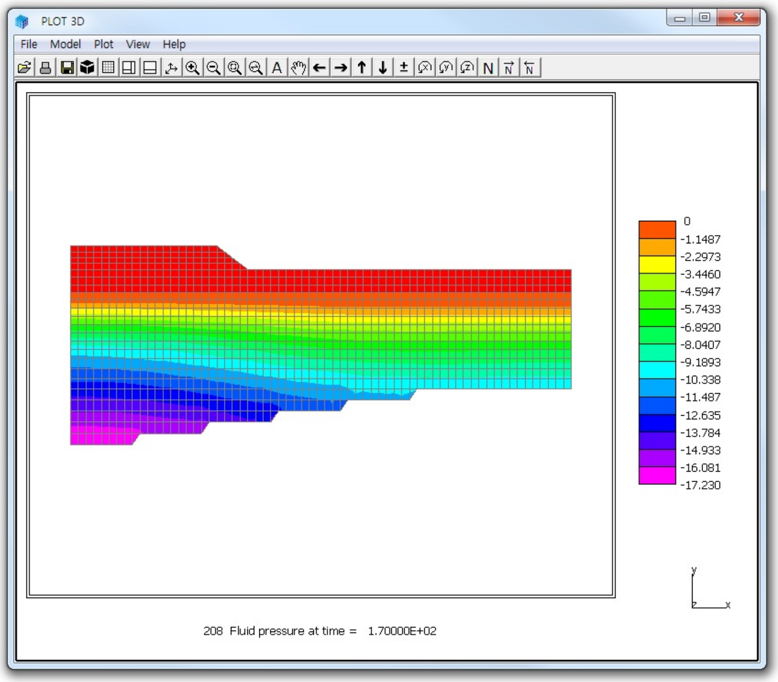


Figure 4.68 Pore pressure profile at completion of embankment

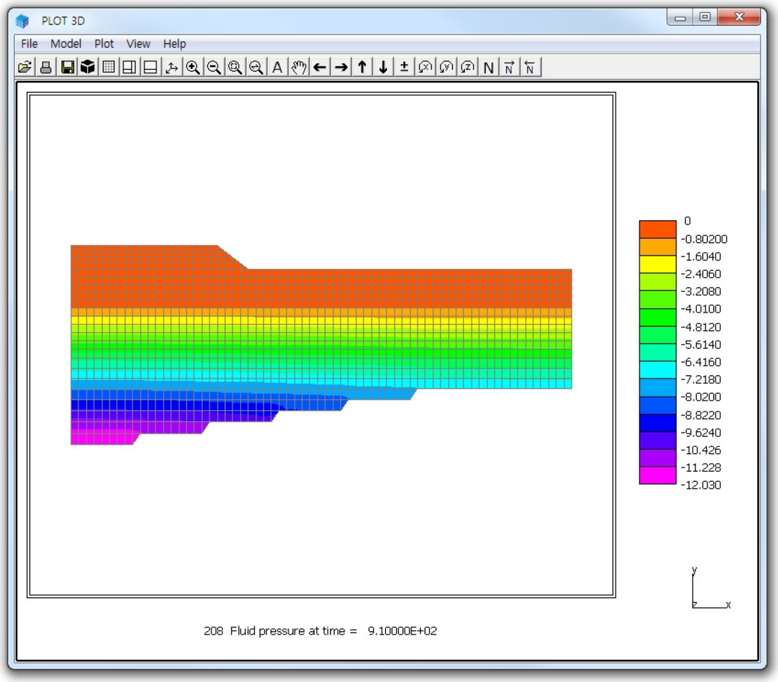


Figure 4.69 Pore pressure profile at two years after completion

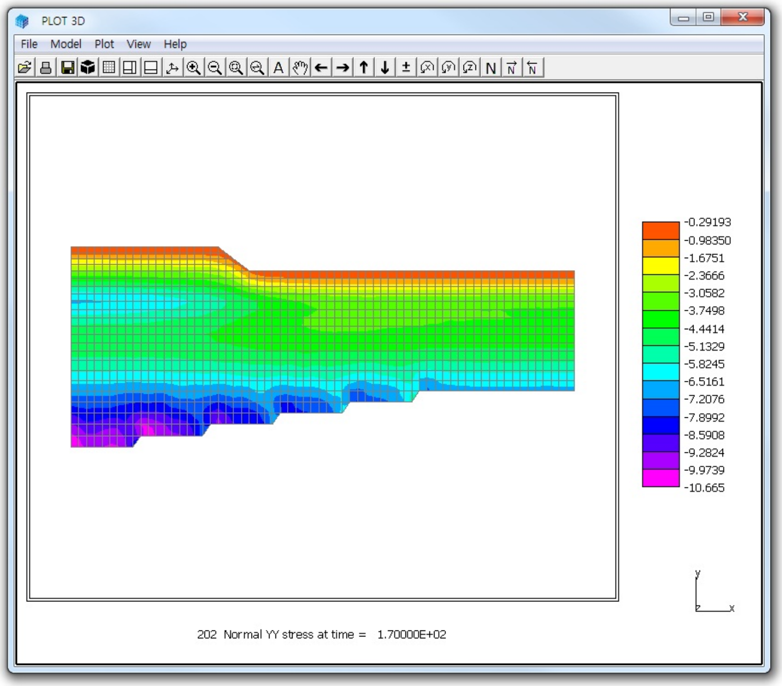


Figure 4.70 Effective stress profile at completion of embankment

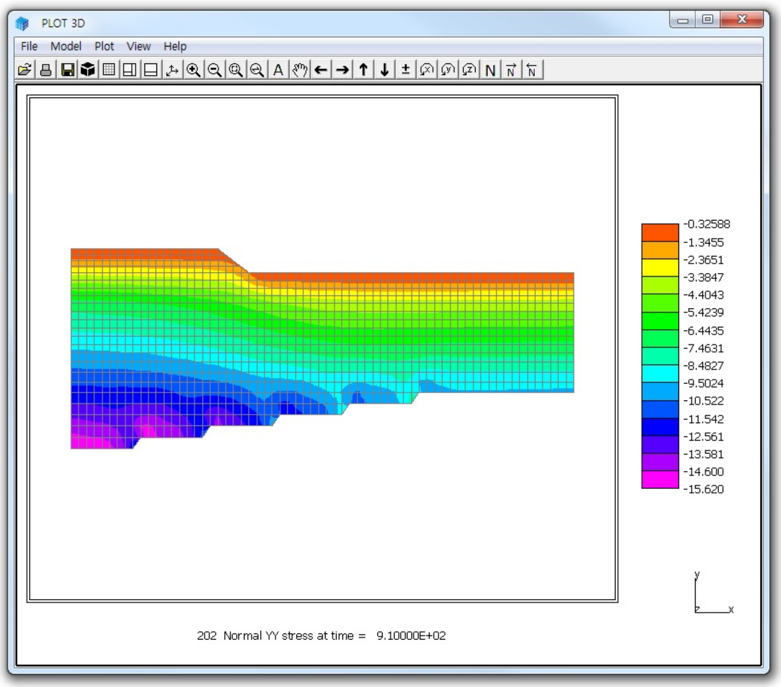


Figure 4.71 Effective stress profile at two years after completion

4.18 Heated Beam Analysis

A Simply supported plain concrete beam, shown schematically in Figure 4.72, is subjected to linear temperature increase through depth.

The temperature of top surface of 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.

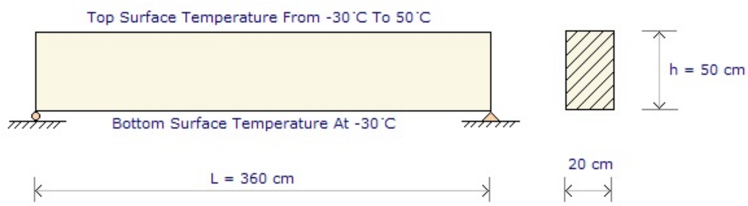


Figure 4.72 Heated beam subjected to temperature difference

By symmetry, only right half of the beam is modeled using a total of 22 beam elements as shown in Figure 4.73.

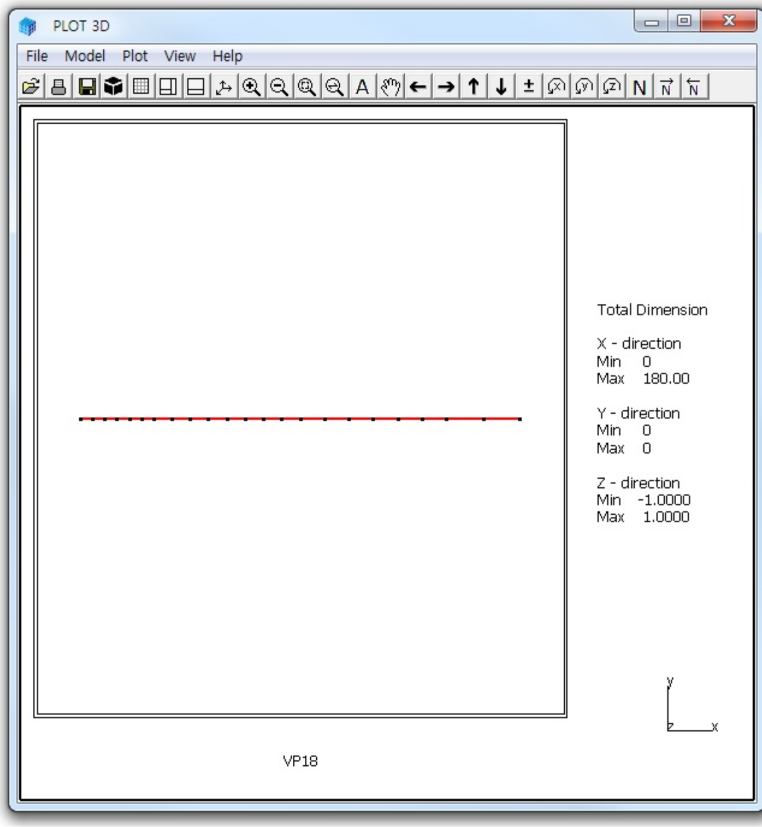


Figure 4.73 Finite element mesh

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 4.74 shows beam deflections. SMAP-2D result gives excellent results for vertical displacement at center of the beam.

Theoretical solution = 0.8294 cm

SMAP-2D solution = 0.82946 cm

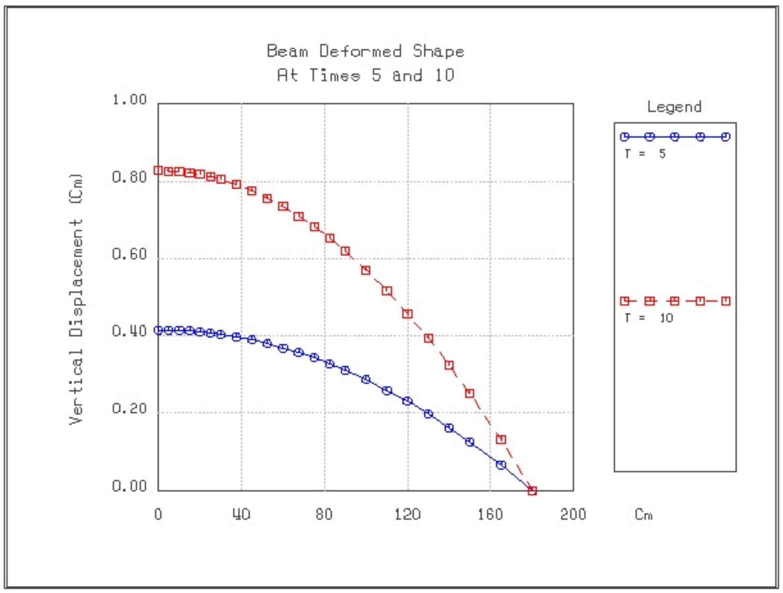


Figure 4.74 Beam deformed shapes

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 4.75 shows 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 4.76 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 4.77 along with computational steps used for SMAP-2D analysis.

Finite element meshes used for the analysis are shown:
- Figure 4.78 Finite element mesh
- Figure 4.79 Finite element mesh around preload
- Figure 4.80 Finite element mesh at completion
- Figure 4.81 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 4.82 Deformed shape around slope
- Figure 4.83 Horizontal displacement distribution
- Figure 4.84 Pore pressure distribution
- Figure 4.85 Effective mean pressure distribution
- Figure 4.86 Deviatoric stress distribution

The horizontal contour lines of the hydrostatic water pressure in Figure 4.84 indicates that there will be no further consolidation settlement at 152 days after completion of excavation. Figure 4.86 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 4.87 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 4.88 Stress time history at element 120
Figure 4.89 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

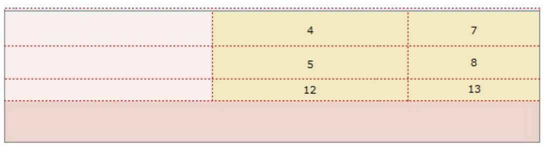
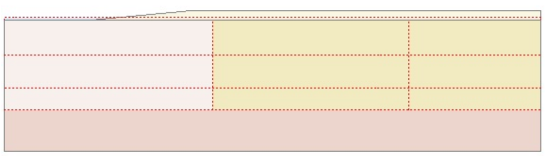
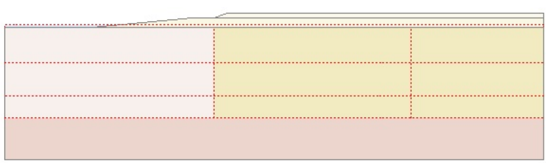
Construction State	Description
Step 101 (2000 days) 	In Situ State Sand Drain: Material 4, 5, 12 PDB: Material 7, 8, 13
Step 104 (+ 15 days) 	Completion of Stage 1 Embankment
Step 165 (+ 321 days) 	Completion of Stage 2 Embankment

Figure 4.75 Construction sequence

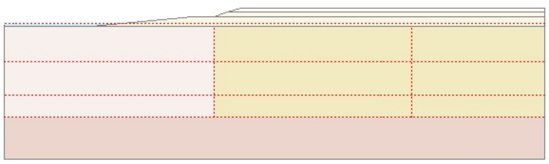
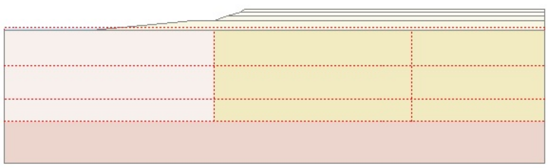
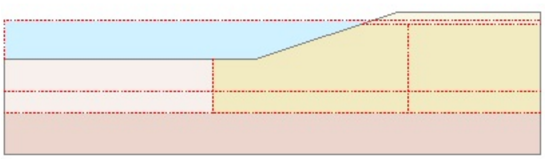
Construction State	Description
Step 216 (+ 627 days) 	Completion of Stage 3 Embankment
Step 267 (+ 933 days) 	Completion of Stage 4 Embankment
Step 333 (+ 1265 days) 	Completion of Final Excavation (Dredging)

Figure 4.75 Construction sequence (Continued)

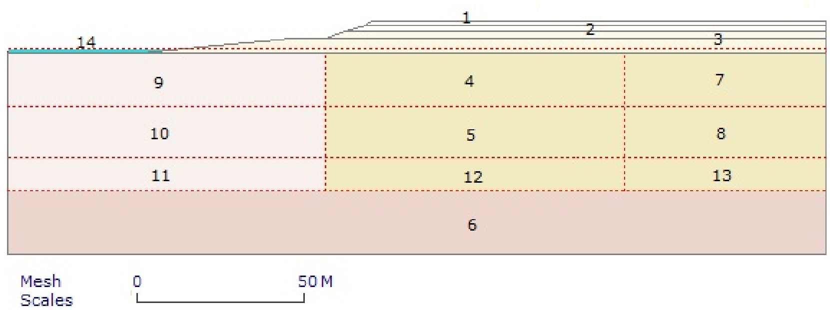


Figure 4.76 Material number

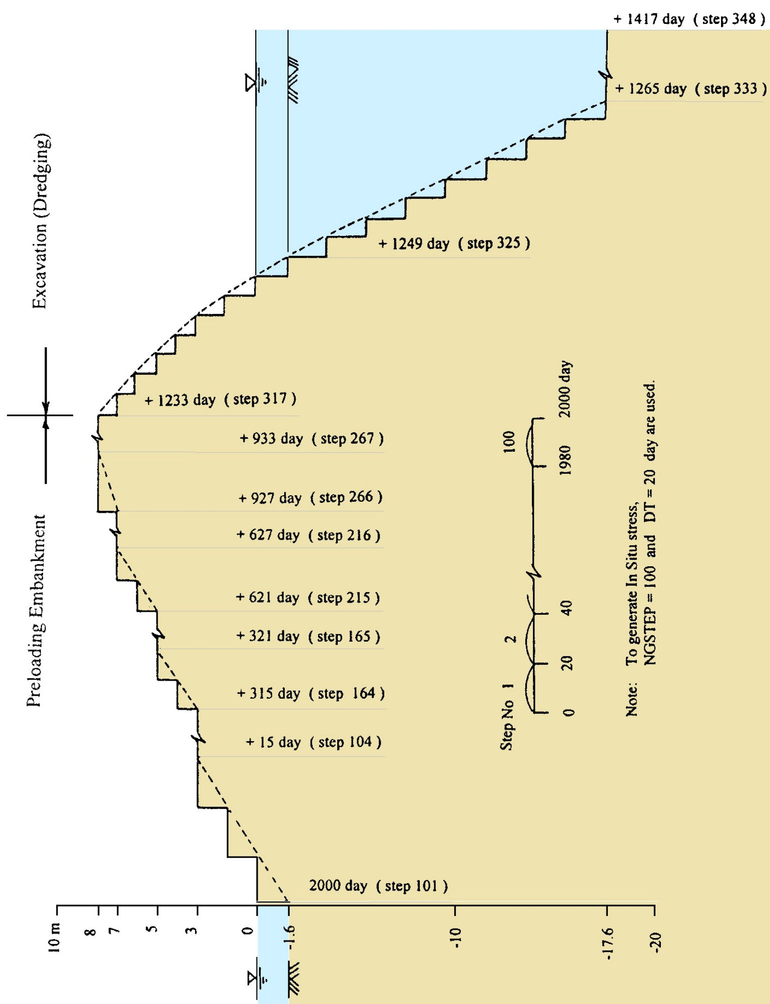


Figure 4.77 Construction rate

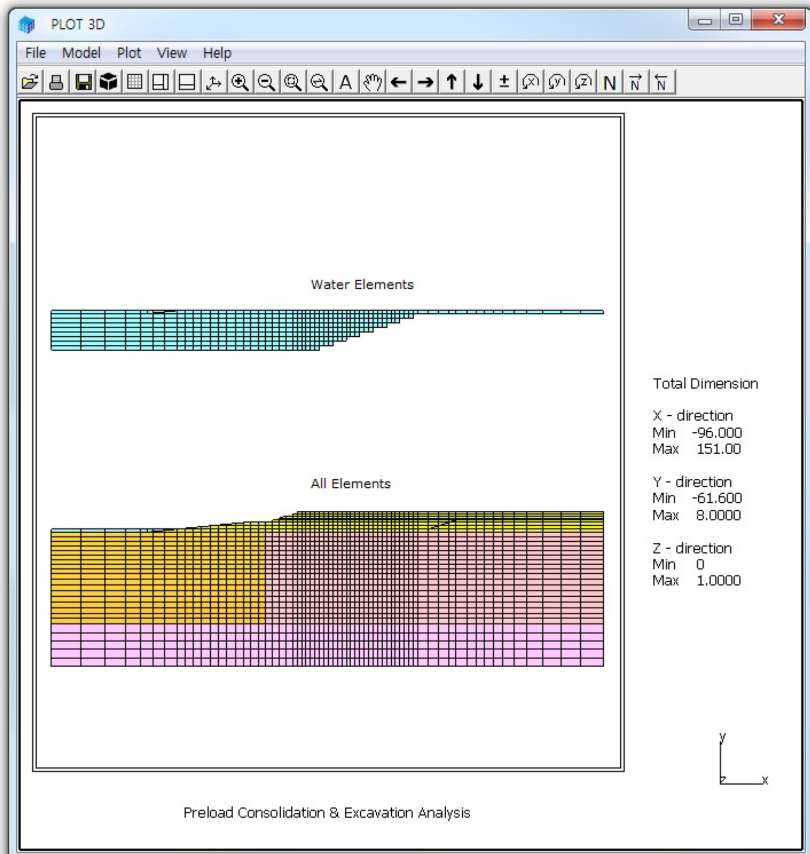


Figure 4.78 Finite element mesh

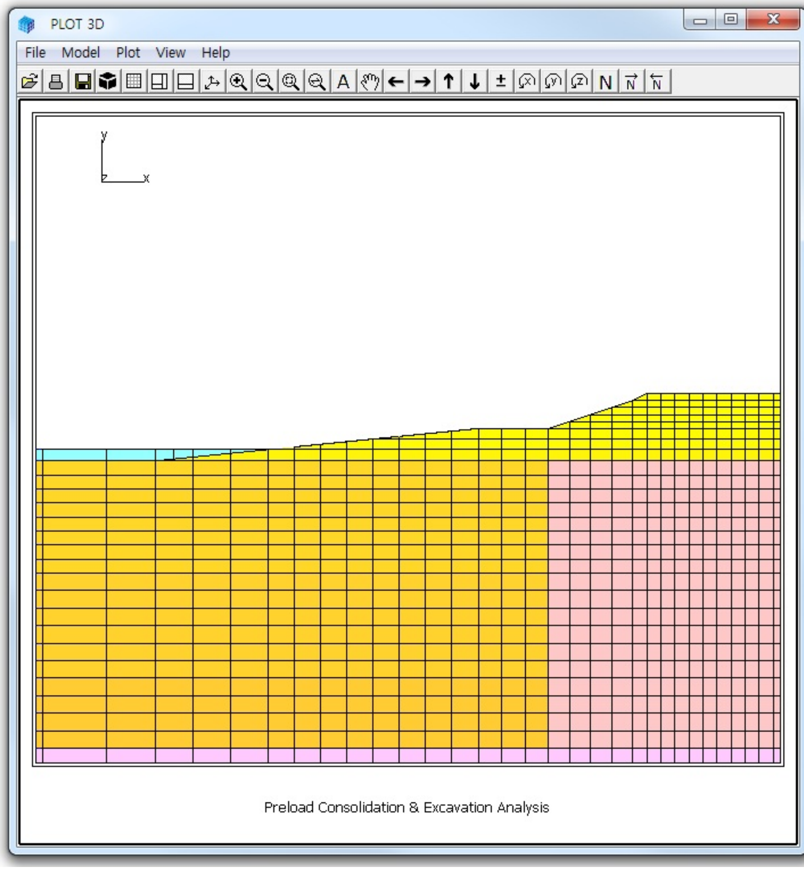


Figure 4.79 Finite element mesh around preload

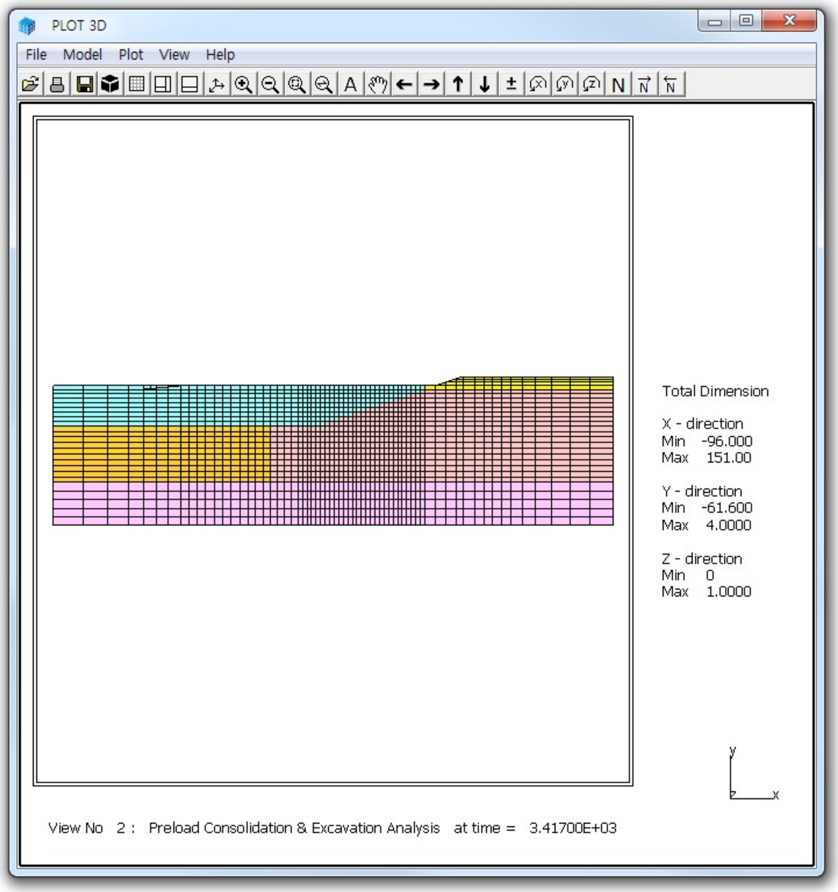


Figure 4.80 Finite element mesh at completion

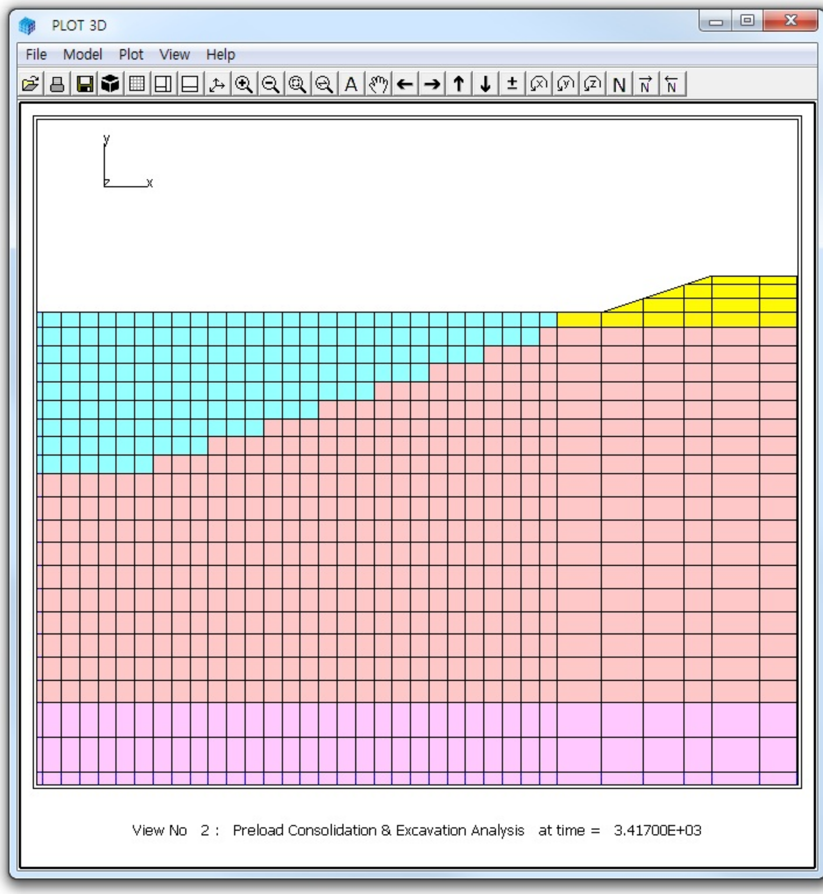


Figure 4.81 Finite element mesh around slope

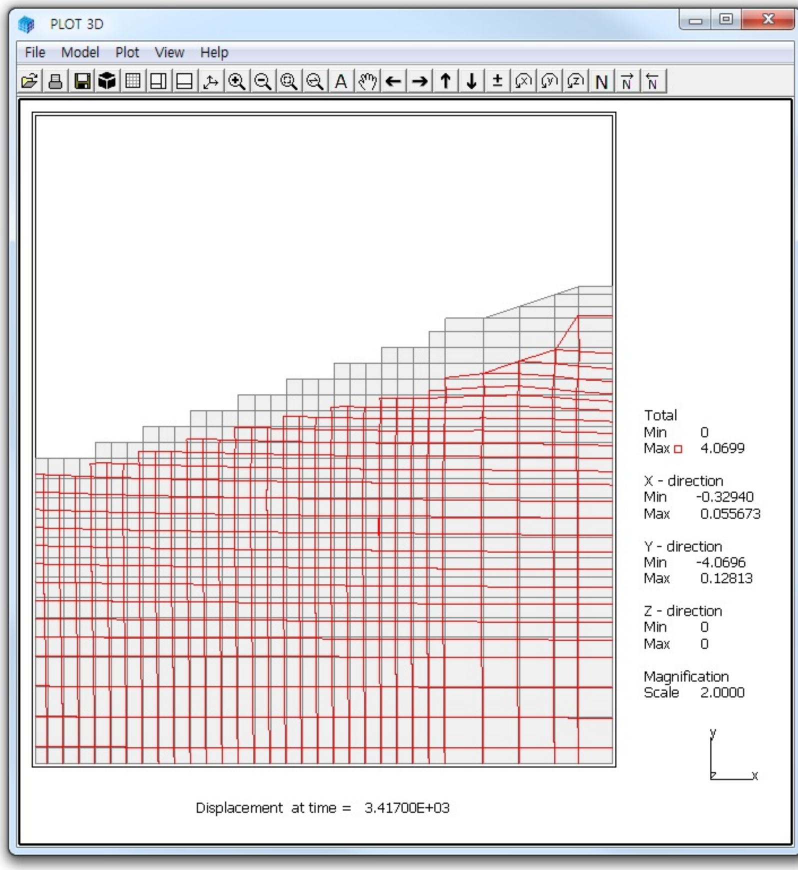


Figure 4.82 Deformed shape around slope

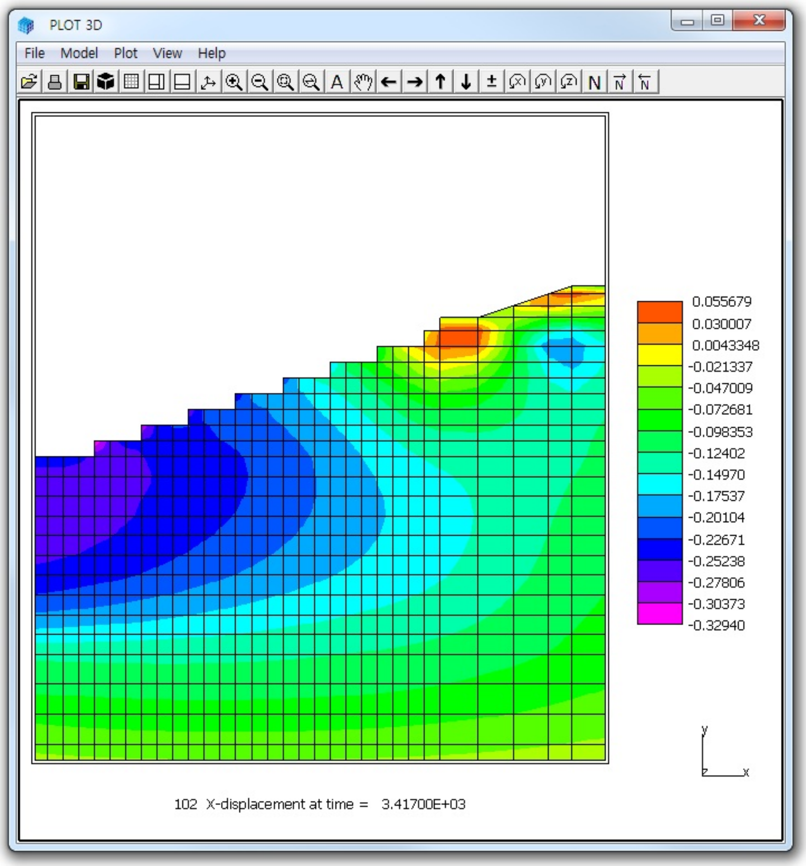


Figure 4.83 Horizontal displacement distribution

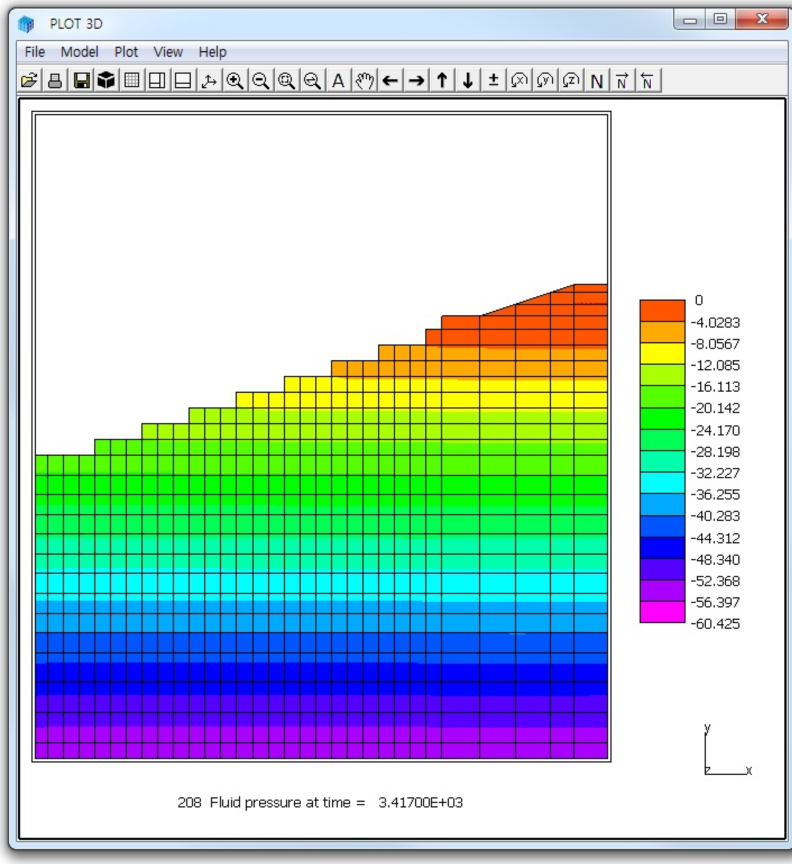


Figure 4.84 Pore pressure distribution

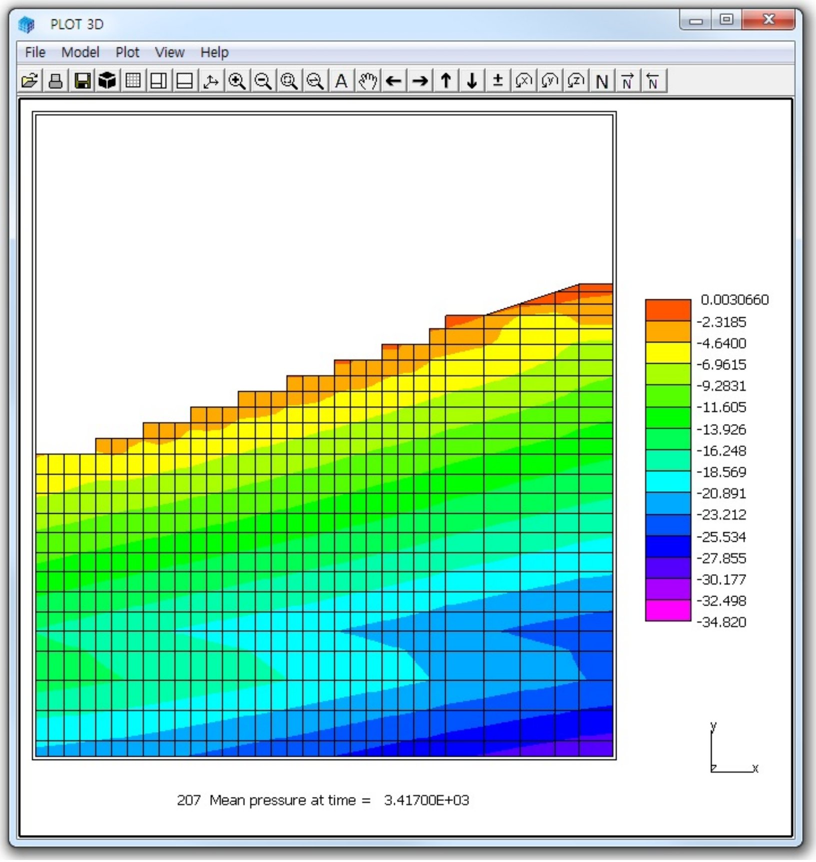


Figure 4.85 Effective mean pressure distribution

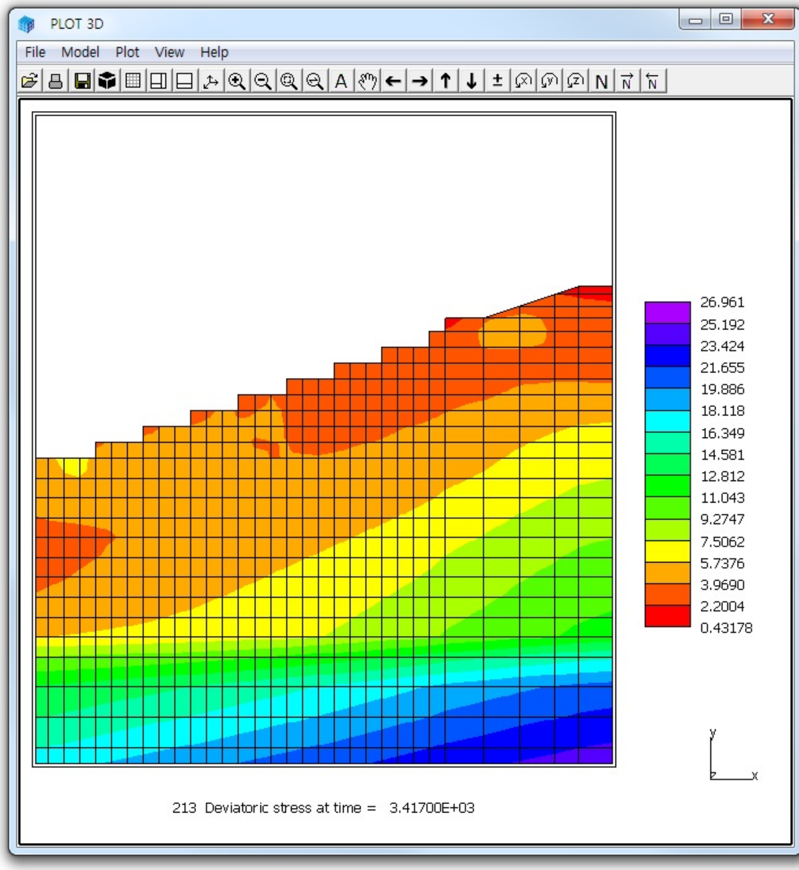


Figure 4.86 Deviatoric stress distribution

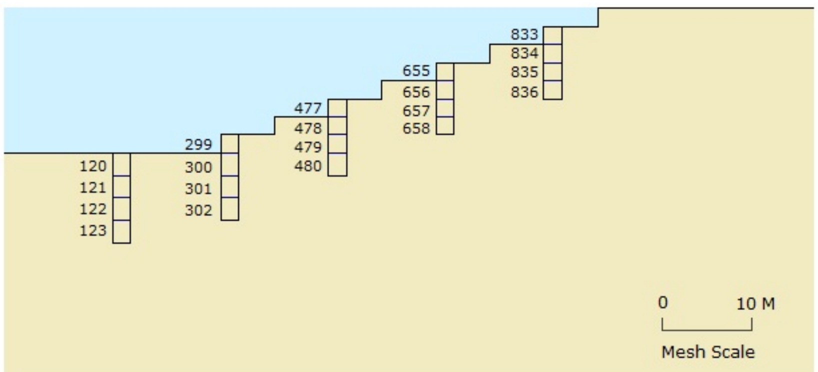


Figure 4.87 Element locations for time history plot

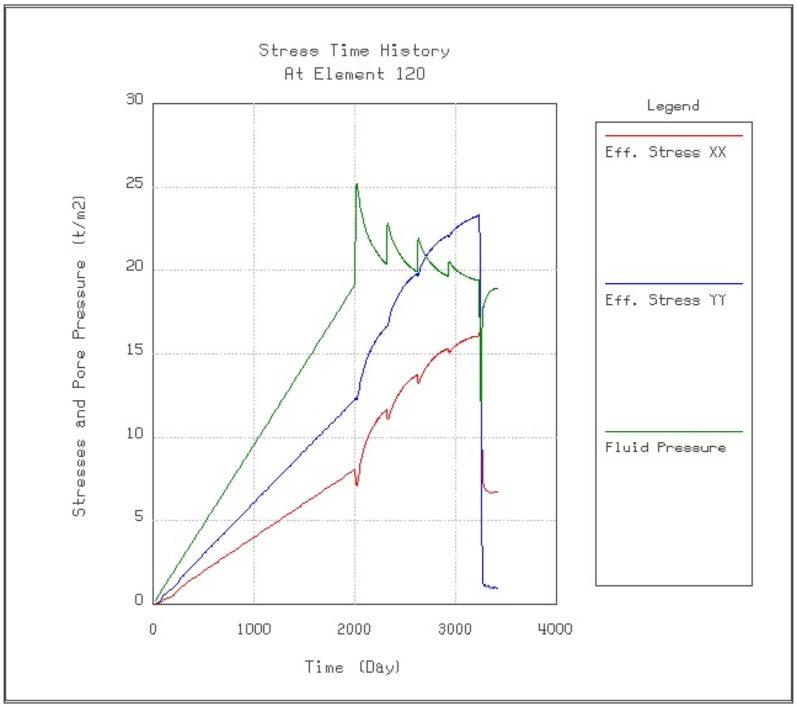


Figure 4.88 Stress time history at element 120

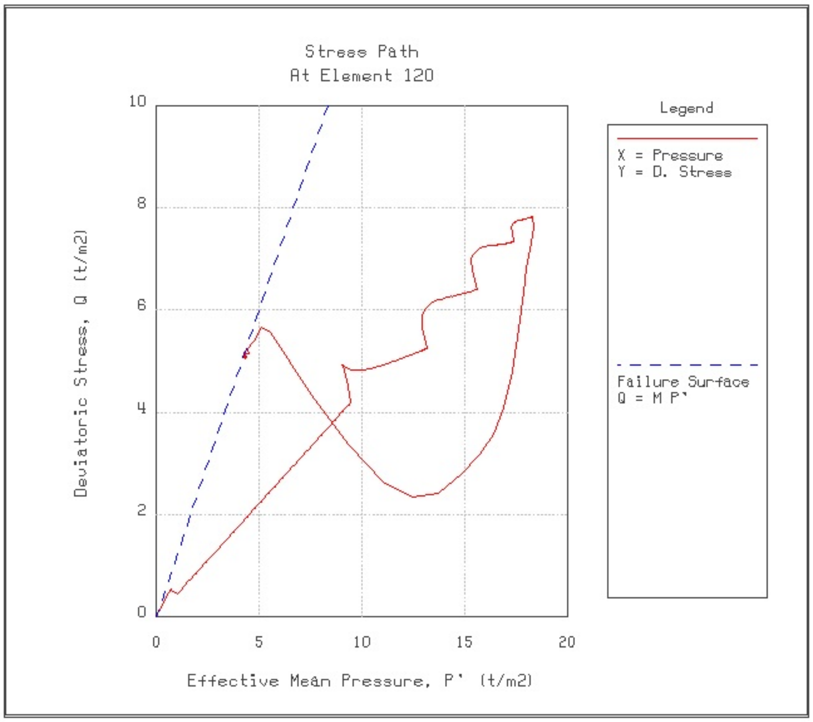


Figure 4.89 Stress path at element 120

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 4.90. Figure 4.91 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 4.92 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 Beam element with plain concrete.

The dynamic part starting from Step 10 as in Figure 4.93 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 4.94 shows key location selected for displacement time history plot. Numbers shown in the figure represent node numbers. Figure 4.95 thru 4.97 show finite element meshes used for the analysis.

Figure 4.98 shows tunnel deformed shape at 5 seconds after the onset of earthquake loading. Figures 4.99 and 4.100 show bending moment and axial force at 5 seconds after onset of earthquake loading.

Figure 4.101 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 4.102 and 4.103 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 4.94 and 4.97.

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	$2.00x10^3$	0.33	30	3	20
Weathered Rock	1.90	0.43	$5.000x10^3$	0.30	35	30	30
Soft Rock	2.40	0.33	$2.00x10^4$	0.25	40	70	40
Hard Rock	2.55	0.25	$2.00x10^5$	0.20	45	100	50
Shotcrete (Soft)	2.40		$0.50x10^6$	0.20	30	500	100
Shotcrete (Hard)	2.40		$1.50x10^6$	0.20	30	500	100
Rock Bolt			$2.10x10^7$				
Reinforced Concrete Lining	2.50		$2.10x10^6$	0.20	30	500	300
Reinforcing Bar			$2.10x10^7$	0.20			
Interface Joint			$2.00x10^5$		5	0.001	0.02

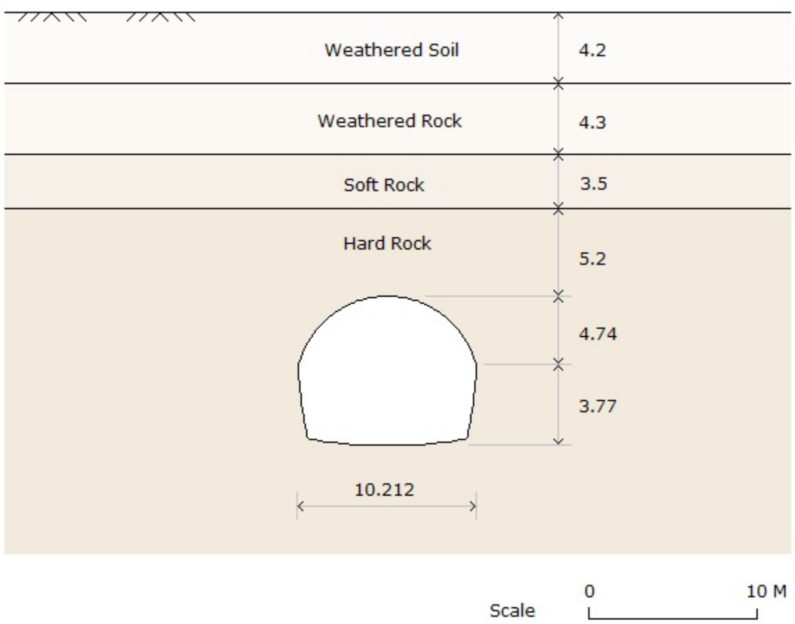
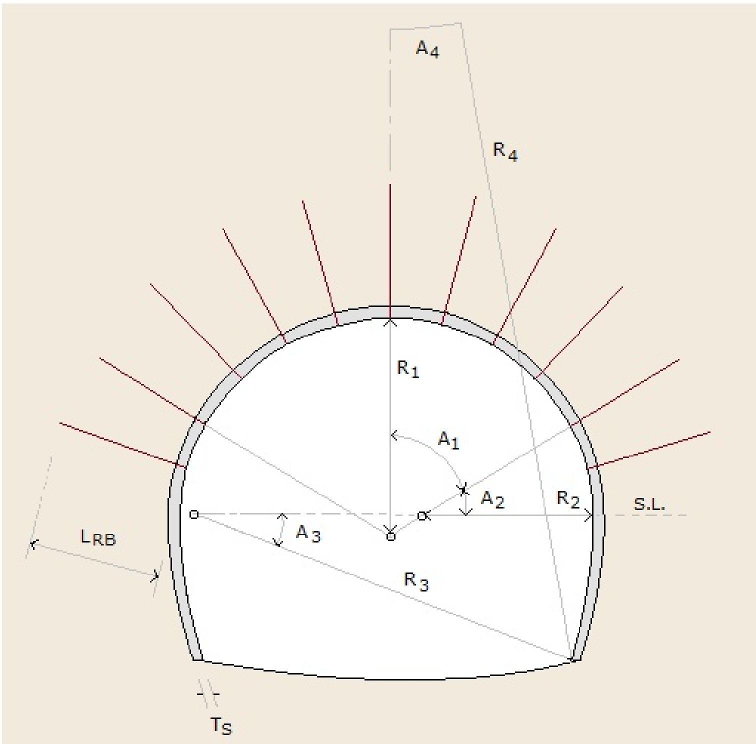


Figure 4.90 Geological profile



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 4.91 Tunnel cross section

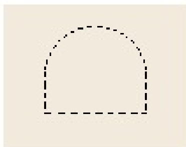
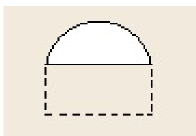
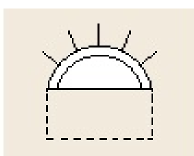
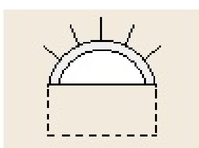
Step	Construction State	Descriptions	
1,2		In Situ K_o State	
3		50 % Stress Relief	Upper Core Excavation
4		75 % Stress Relief Soft Shotcrete Rock Bolt	
5		100 % Stress Relief Hard Shotcrete Rock Bolt	

Figure 4.92 Construction sequence, static part

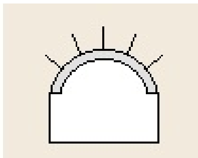
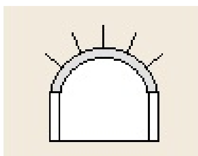
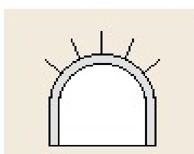
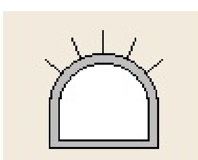
<table>
<tr><th>Step</th><th>Construction State</th><th colspan="2">Descriptions</th></tr>
<tr><td>6</td><td></td><td>50% Stress Relief</td><td rowspan="3">Lower Core Excavation</td></tr>
<tr><td>7</td><td></td><td>75% Stress Relief
Soft Shotcrete</td></tr>
<tr><td>8</td><td></td><td>100% Stress Relief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

Figure 4.92 Construction sequence, static part (Continued)

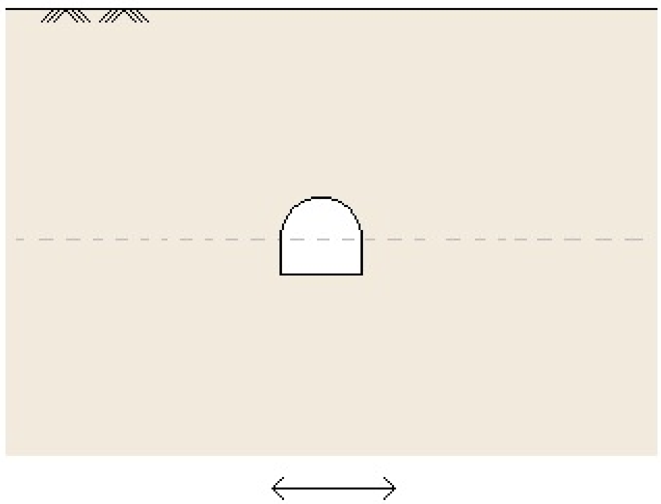


Figure 4.93 Seismic load, dynamic part

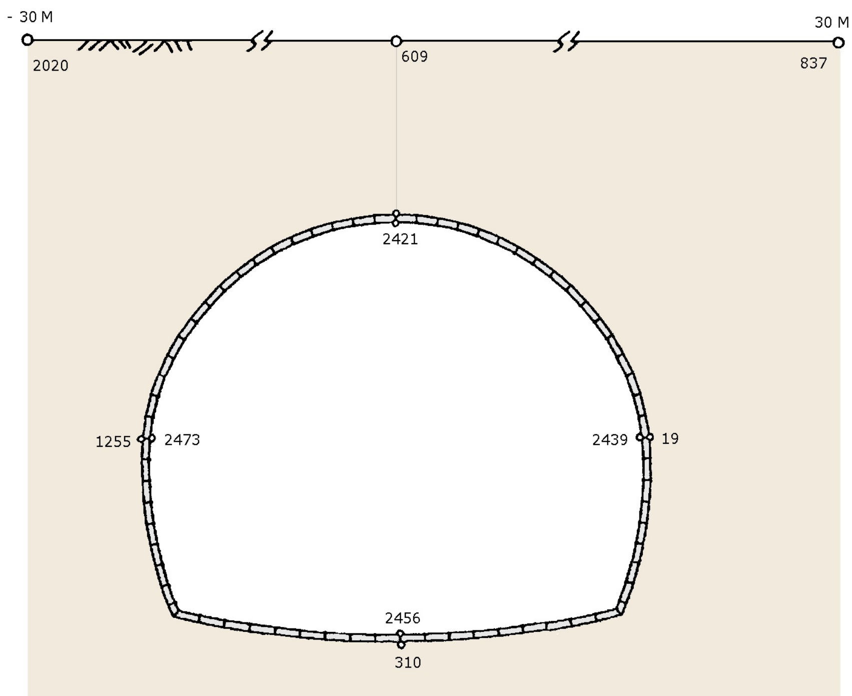


Figure 4.94 Key locations for displacement time history plot

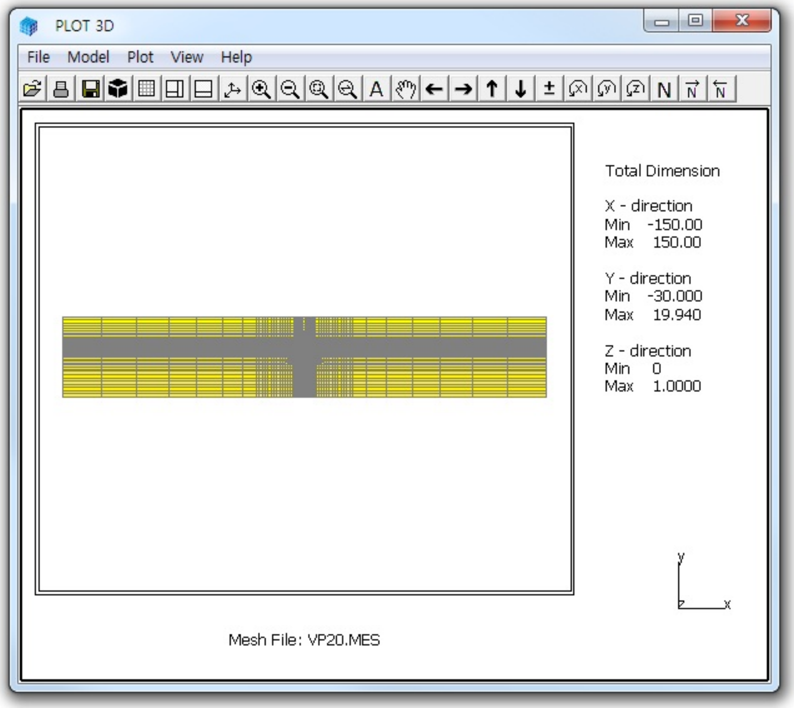


Figure 4.95 Finite element mesh

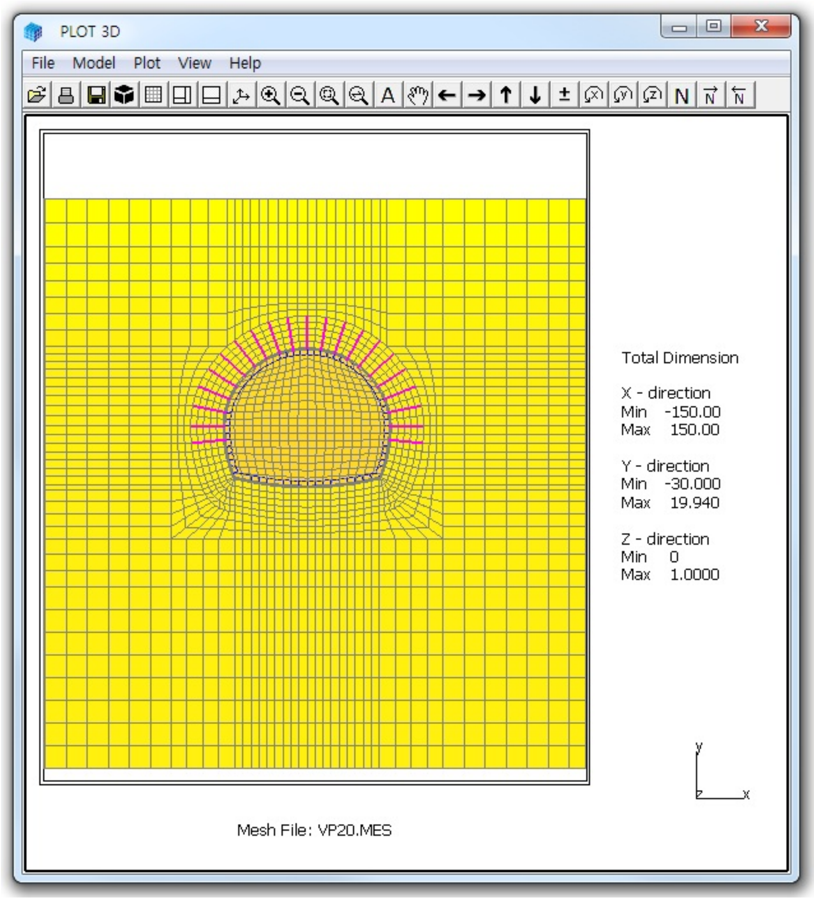


Figure 4.96 Finite element mesh around tunnel

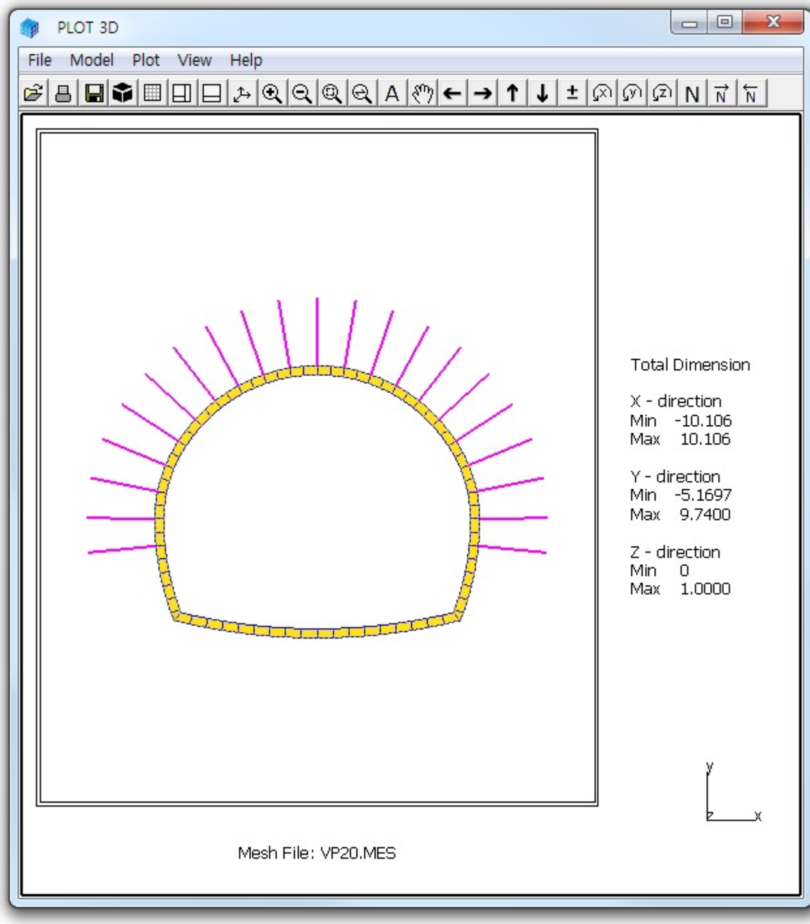


Figure 4.97 Interface between shotcrete and lining

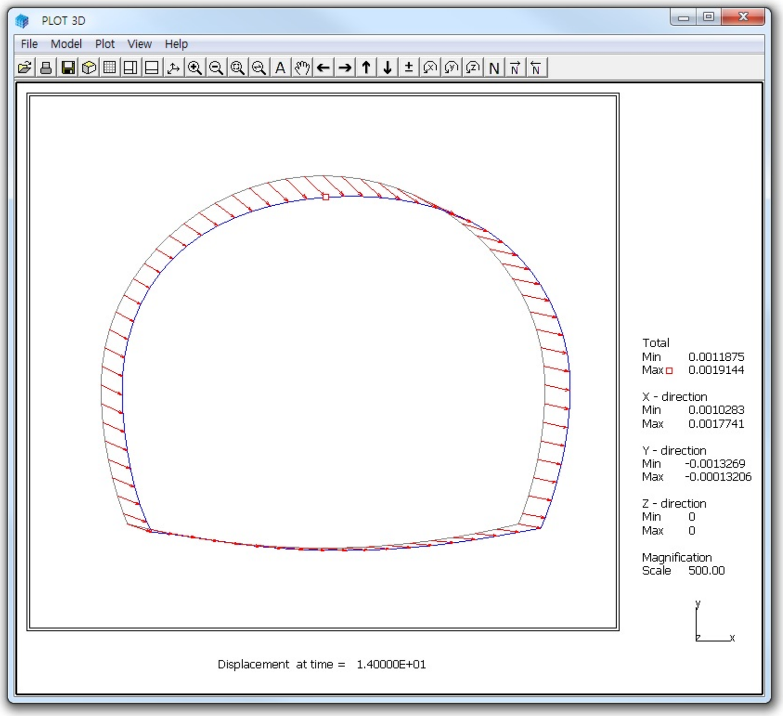


Figure 4.98 Tunnel deformed shape at t=14 sec

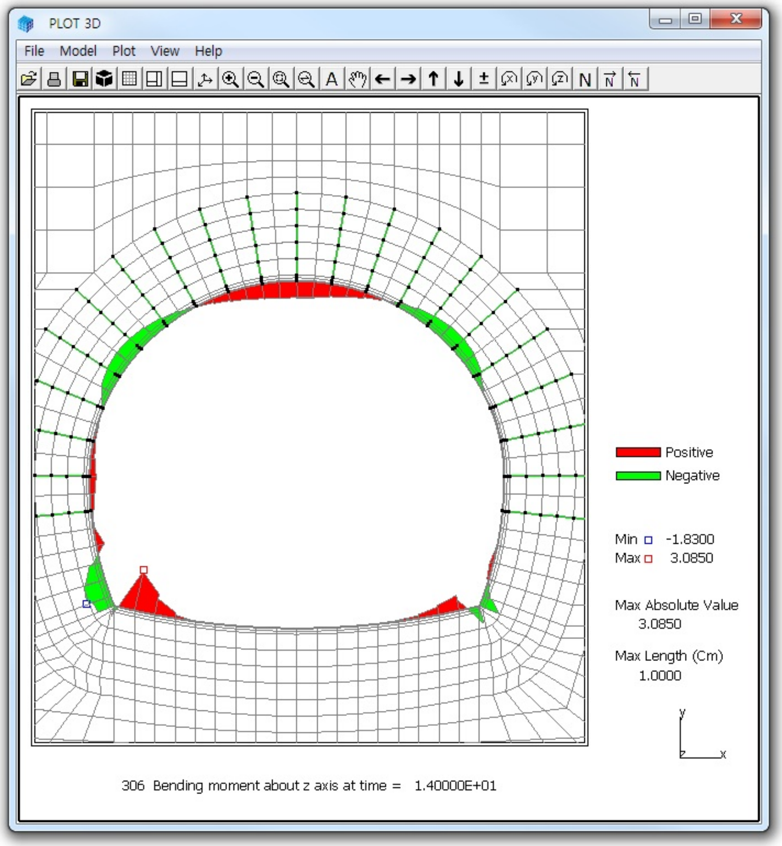


Figure 4.99 Bending moment at t=14 sec

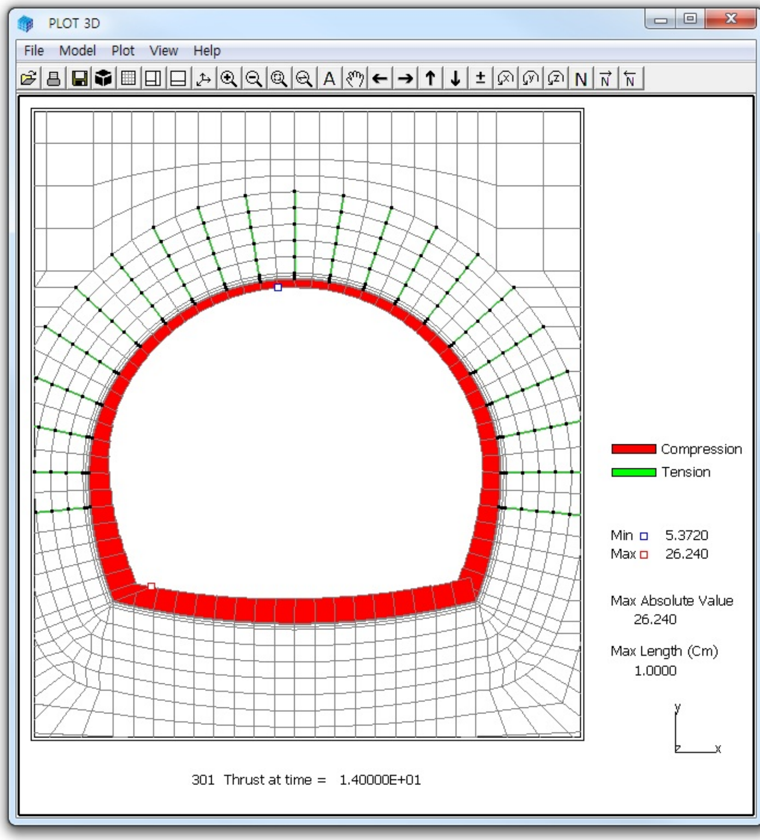


Figure 4.100 Axial force at t=14 sec

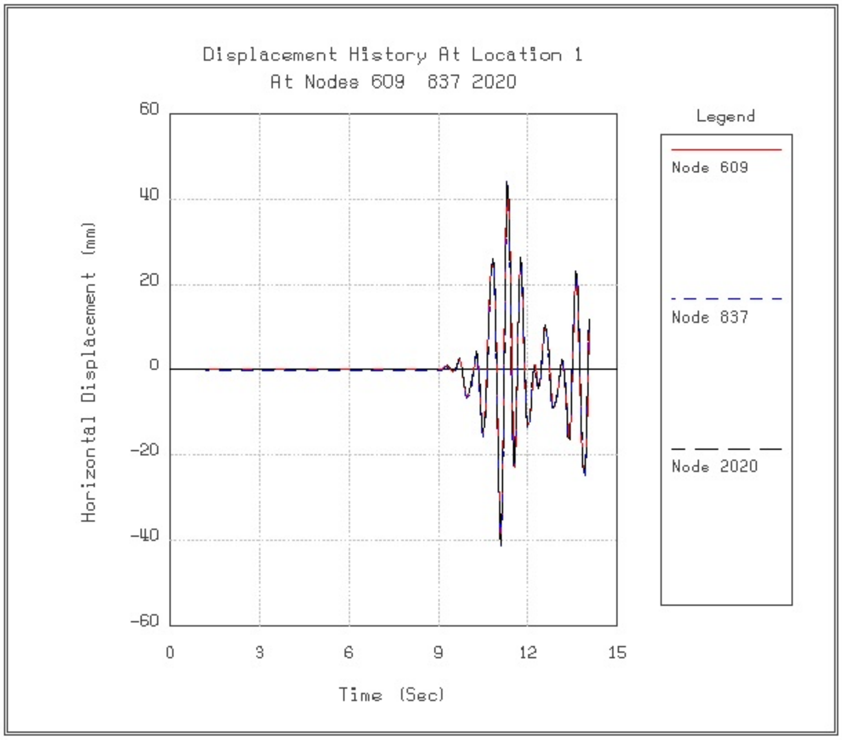


Figure 4.101 Horizontal displacement at nodes 609, 837, and 2020

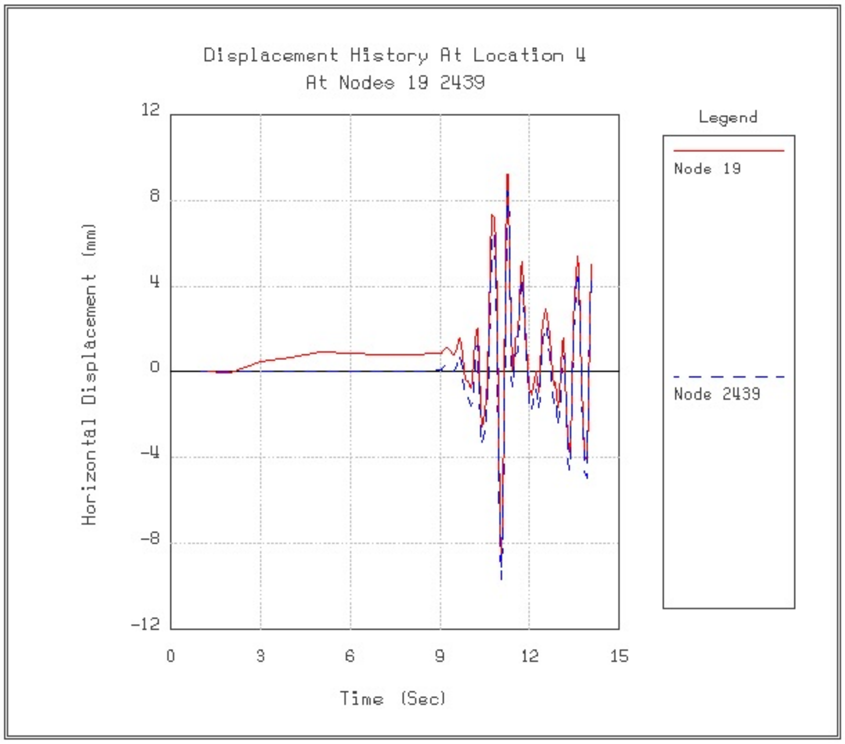


Figure 4.102 Horizontal displacement at nodes 19 and 2439

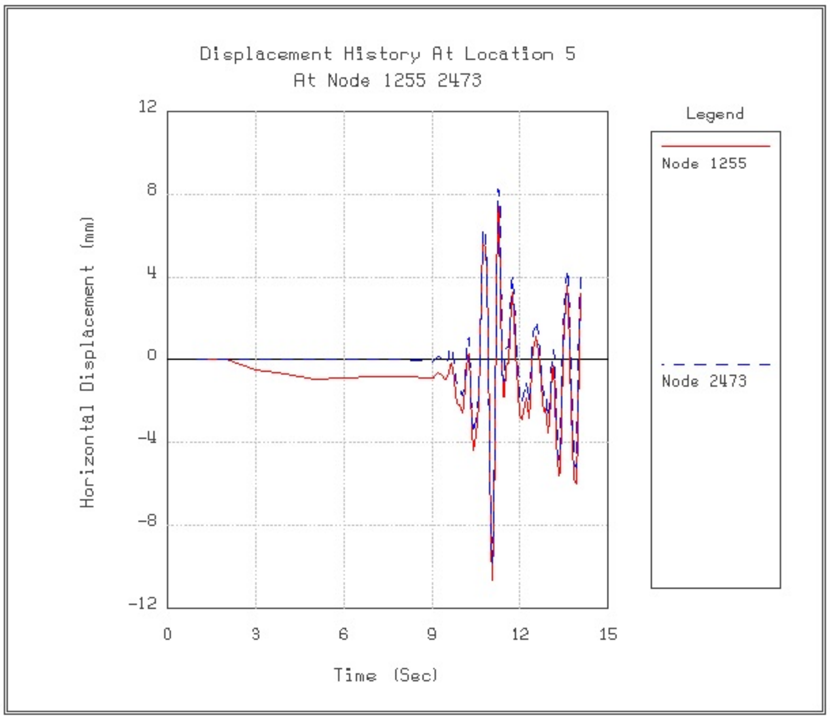


Figure 4.103 Horizontal displacement at nodes 1255 and 2473

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 4.104.

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

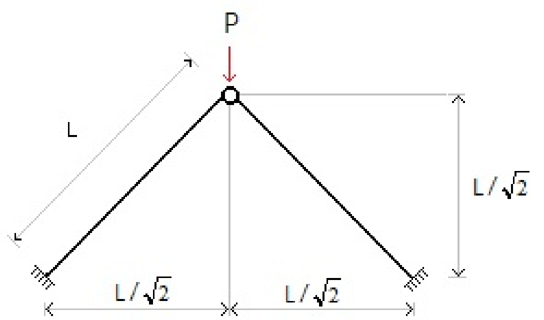
SMAP-2D calculations are performed using the geometrical and material parameters listed in Figure 4.104. The frame is modeled by 10 beam elements as shown in Figure 4.105.

Figures 4.106 and 4.107 show beam deformed shape and bending moment diagram, respectively.

SMAP-2D results show good agreement with the exact solutions.

Maximum deflection at the center (δ)
- Exact solution = 0.01768 cm
- SMAP-2D (Beam) = 0.01767 cm

Maximum moment at fixed end (M_{max})
- Exact solution = 0.1000 t-m
- SMAP-2D (Beam) = 0.1000 t-m



P = 100 t

E = 20×10^6 t/m^2

A = 0.2 m^2

L = 7.071 m

v = 0.0

I = 0.000667 m^4

Figure 4.104 Frames with hinge connection

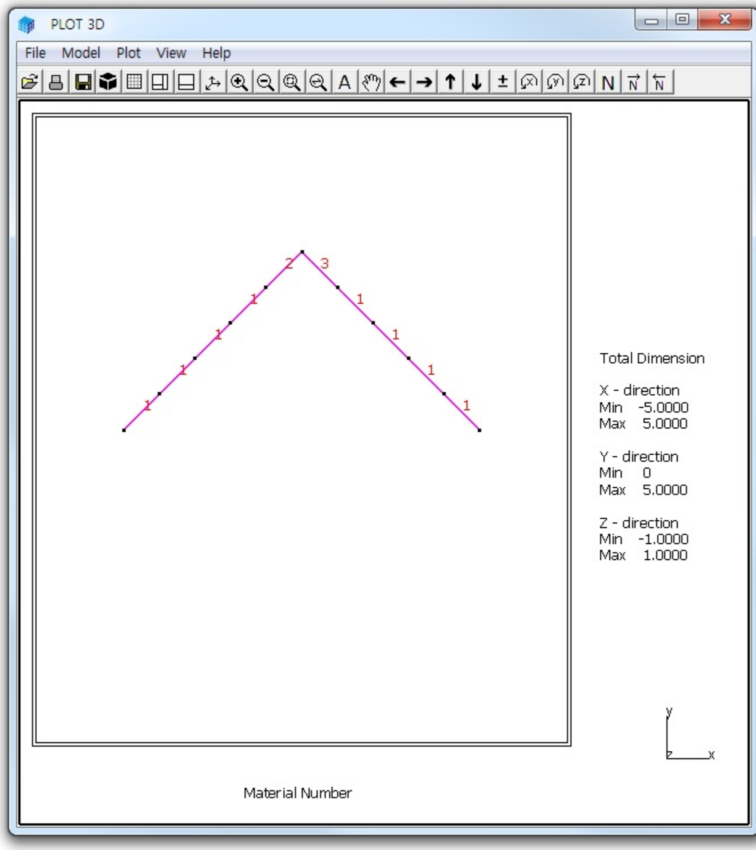


Figure 4.105 Beam element with material number

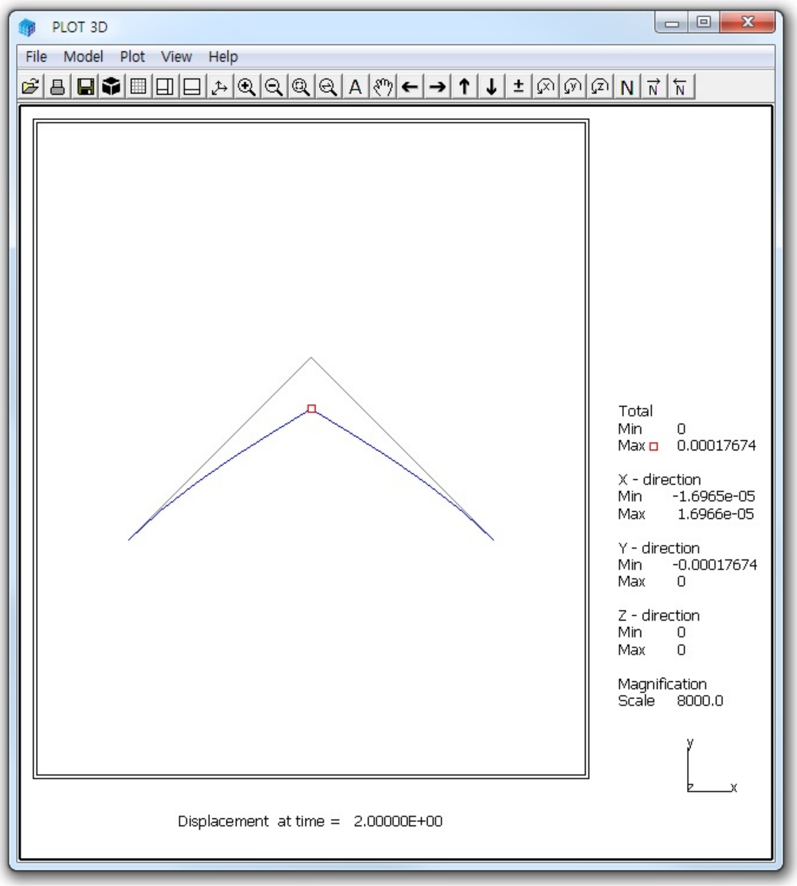


Figure 4.106 Beam deformed shape

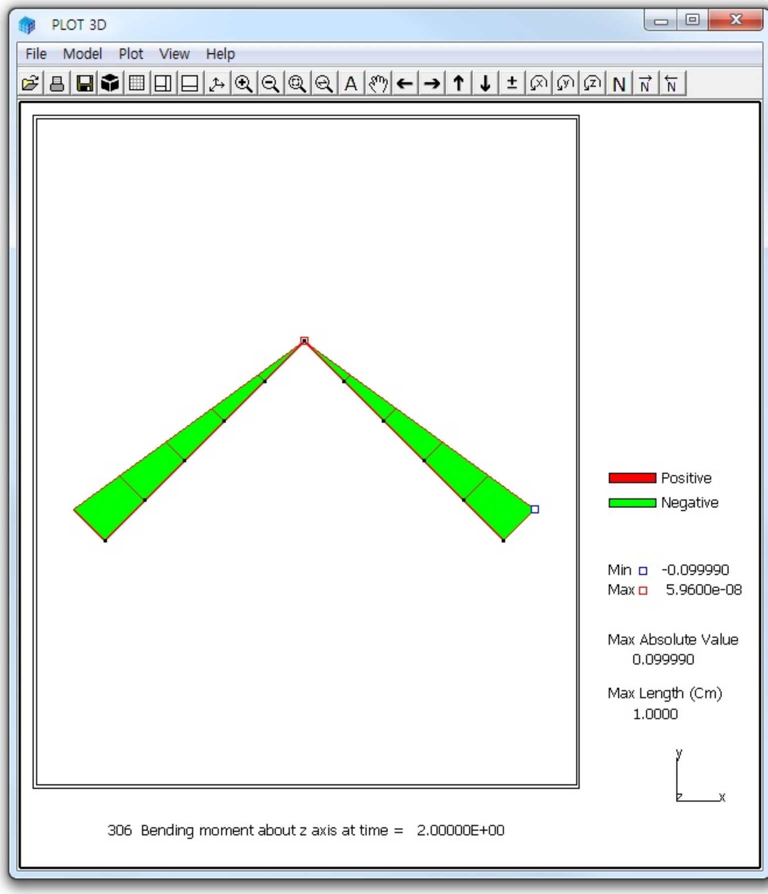


Figure 4.107 Beam bending moment diagram

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 4.108 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 4.109. It should be noted that the end points of embedded truss elements do not belong to the corner nodes of continuum elements.

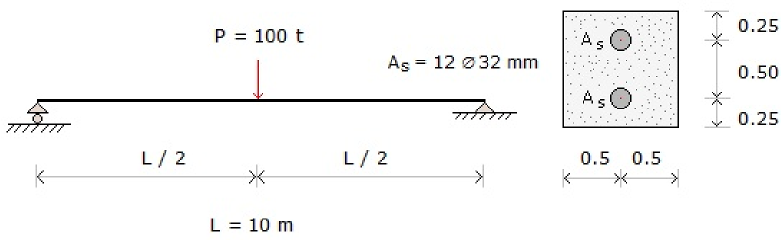
The computed center deflections are compared with the exact beam solution as shown in Table 4.3. SMAP-2D results approach to the upper bound beam solution at lower cohesion and the lower bound beam solution at higher cohesion. At the intermediate cohesion, however, the computed deflection is in between upper and lower bound beam solutions, indicating some resistance from slip strength.

Figures 4.110 and 4.111 show the deformed shape and the axial stress distribution, respectively, from SMAP-2D result at the intermediate cohesion of 5 t/m^2.

Table 4.3 Computed center deflections

Cmax (t/m^2)	SMAP-2D 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 = $2\,(E_s / E_c)\,A_s\,(0.25)^2$ = 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 4.108 Embedded rebars with slip

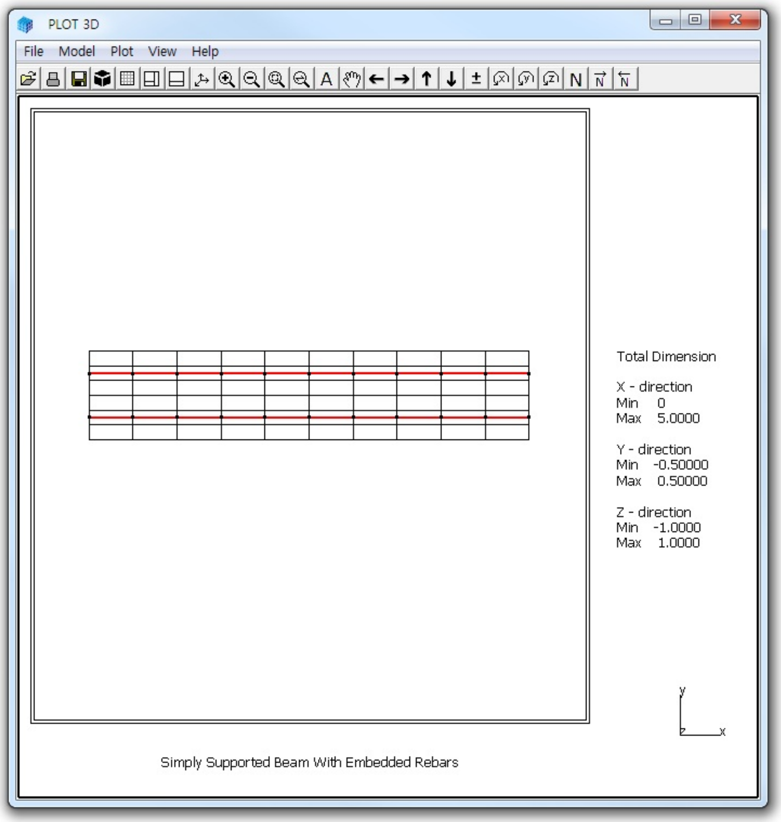


Figure 4.109 Finite element mesh

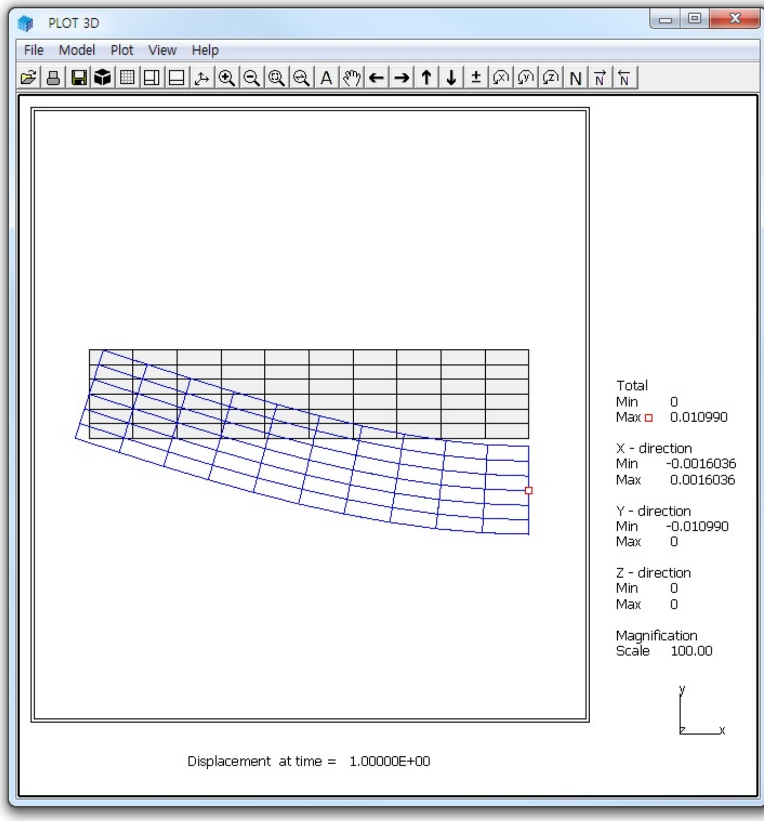


Figure 4.110 Deformed shape

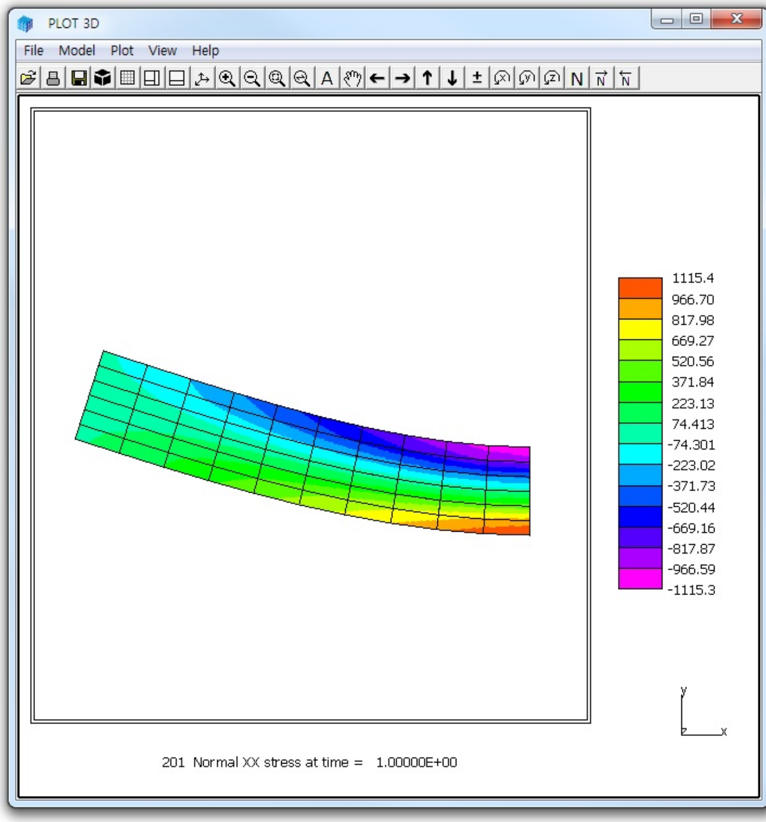


Figure 4.111 Axial stress distribution

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 4.112.

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 4.113 shows the finite element mesh used for the analysis. Figures 4.114 and 115 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-3D analyses show the same results.

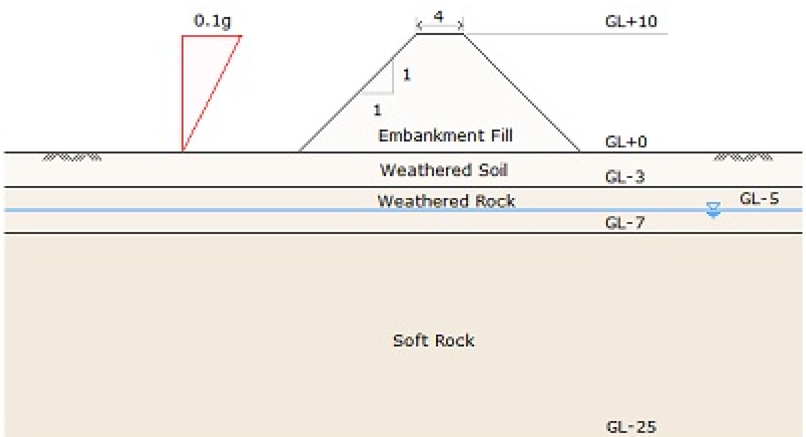


Figure 4.112 Schematic section view of embankment

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)	ν	φ deg.	C (t/m^2)	T (t/m^2)
Weathered Soil	1.90	0.50	2.0 x10^3	0.33	30	3	20
Weathered Rock	1.90	0.43	5.0 x10^3	0.30	35	30	30
Soft Rock	2.40	0.33	2.0 x10^4	0.25	40	70	40
Embankment Fill	2.00	0.50	3.0 x10^3	0.33	30	3	20

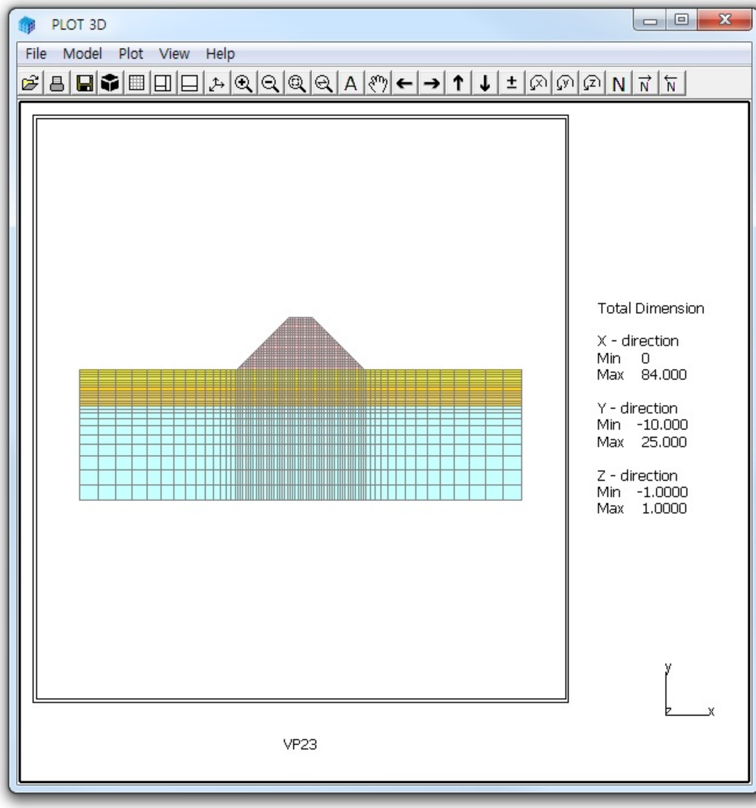


Figure 4.113 Finite element mesh

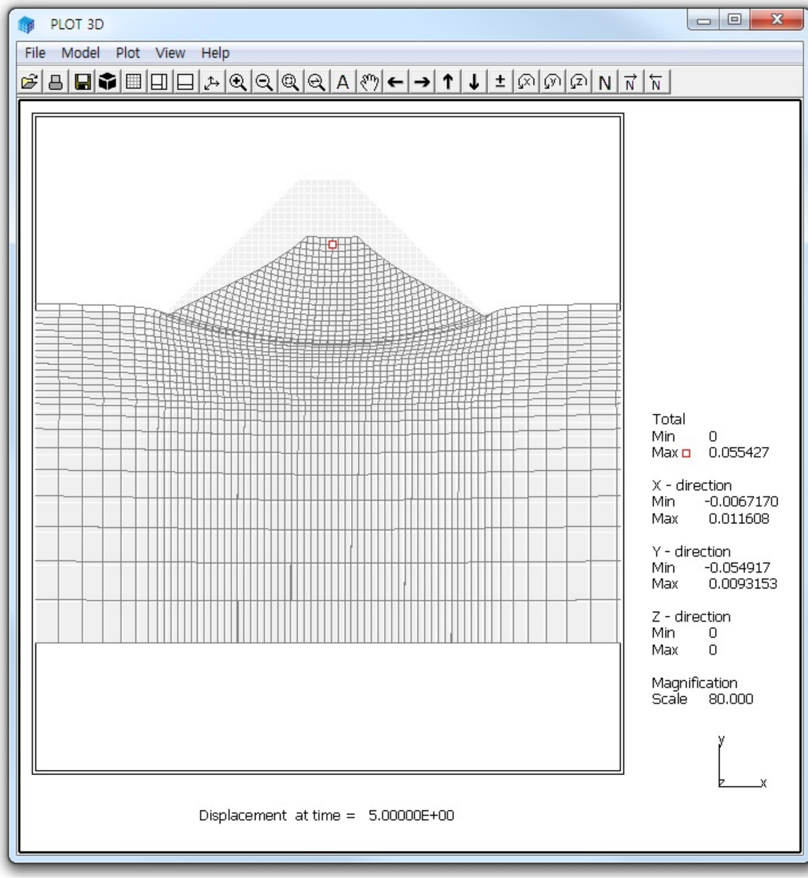


Figure 4.114 Deformed shape

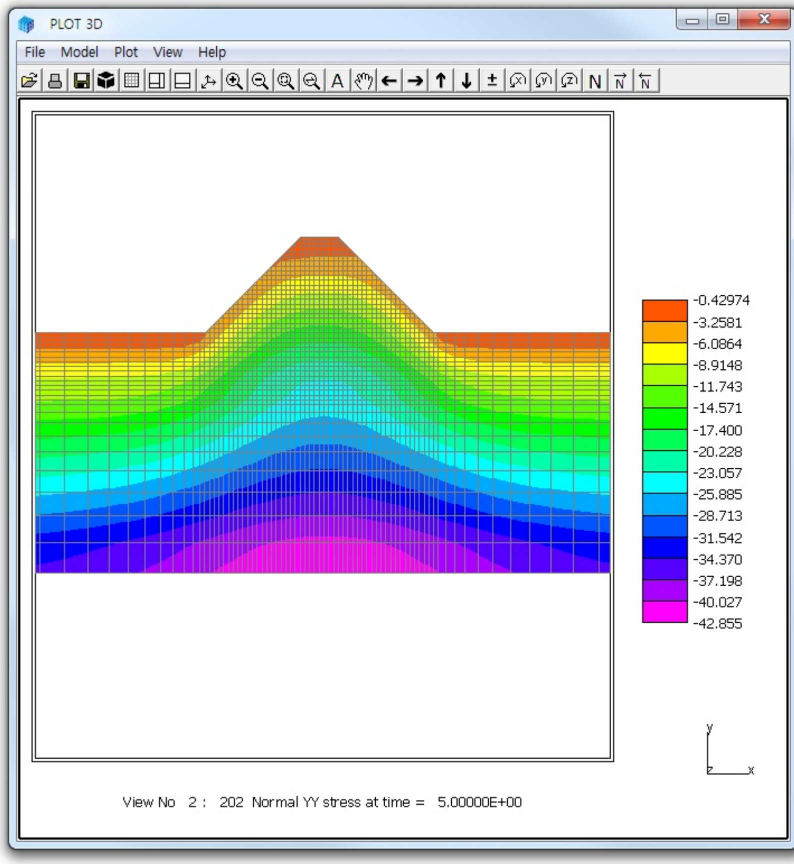


Figure 4.115 Vertical stress distribution

4.24 Excavation on Nearby Box Frame

This example problem is to investigate the influence of excavation on the nearby reinforced concrete box structure as shown in Figures 4.116 and 4.117. Table 4.6 lists the sequence of construction which consists of 10 steps. And Table 4.7 lists material properties.

The finite element meshes for this example problem were prepared by Group Mesh Generator as illustrated in Group Mesh Example 3. Figure 4.118 shows all groups used for mesh generation.

Following graphical results are obtained from PLOT-3D:

Figure 4.119	Vertical stresses on deformed mesh at Step 10
Figure 4.120	Bending moments on deformed mesh at Step 10
Figure 4.121	Bending moments in RC box frame at Step 3
Figure 4.122	Bending moments in RC box frame at Step 10
Figure 4.123	Inner extreme fiber stress in concrete at Step 10
Figure 4.124	Outer extreme fiber stress in concrete at Step 10
Figure 4.125	Inner reinforcing bar stress at Step 10
Figure 4.126	Outer reinforcing bar stress at Step 10

It shows that the maximum bending moment is reduced slightly due to excavation. The maximum compressive concrete stress of 311 t/m2 is much lower than the allowable compressive stress and the maximum tensile reinforcing bar stress of 1286 t/m2 is much lower than the allowable tensile stress. Thus the influence of excavation on the nearby box structure is insignificant and the structure is safe.

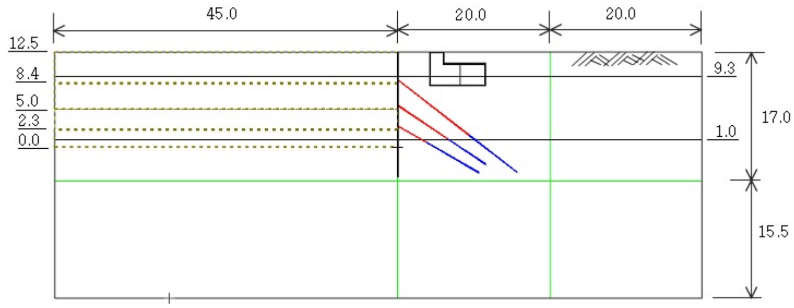


Figure 4.116 Schematic section view

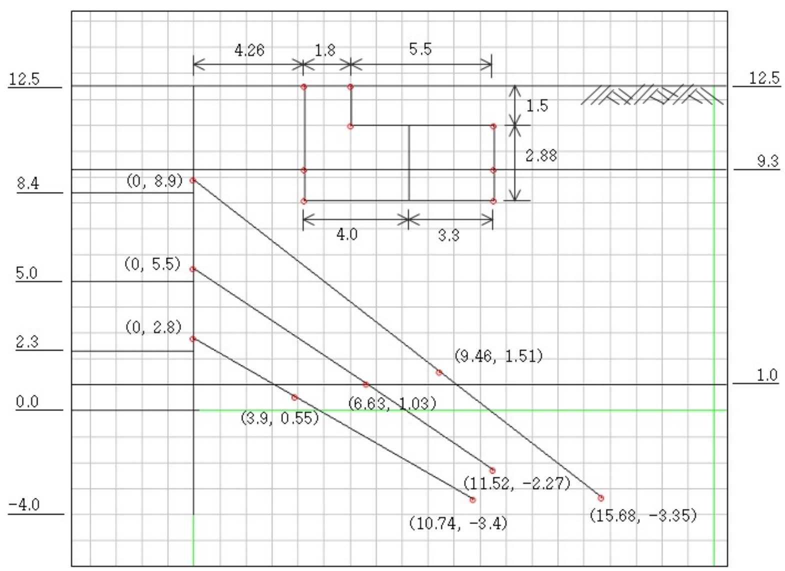


Figure 4.117 Cross section near box structure

Table 4.6 Construction sequence

Step	Description
1, 2	In Situ Ko State
3	Construction of R.C. Box Structure
4	Installation of SCE Wall Followed by First Excavation to Y = 8.4 m
5	Anchor - 1 Installation
6	Second Excavation to Y = 5.0 m
7	Anchor - 2 Installation
8	Third Excavation to Y = 2.3 m
9	Anchor - 3 Installation
10	Final Excavation to Y = 0.0 m

Table 4.7 Material property

Material Type	γ (t/m^3)	K_o	E (t/m^2)	ν	φ deg.	C (t/m^2)	T (t/m^2)
Fill	1.8	0.54	1000.	0.35	25	0.5	1
Silty Sand	1.8	0.54	1000.	0.35	25	0.5	1
Sand Gravel	1.8	0.47	3000.	0.32	32	0.5	1
SCE Wall			2.1×10^7	0.2			
R.C. Box			2.1×10^6	0.2	45	250	300
Anchor			2.1×10^7				

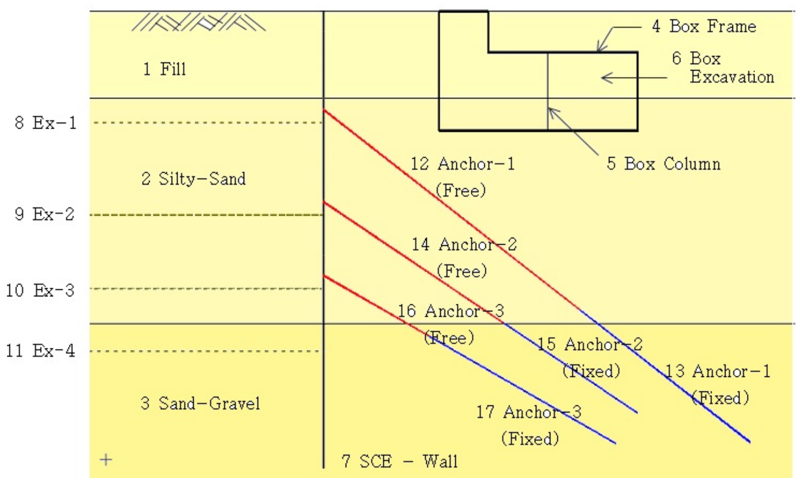


Figure 4.118 All groups used for mesh generation

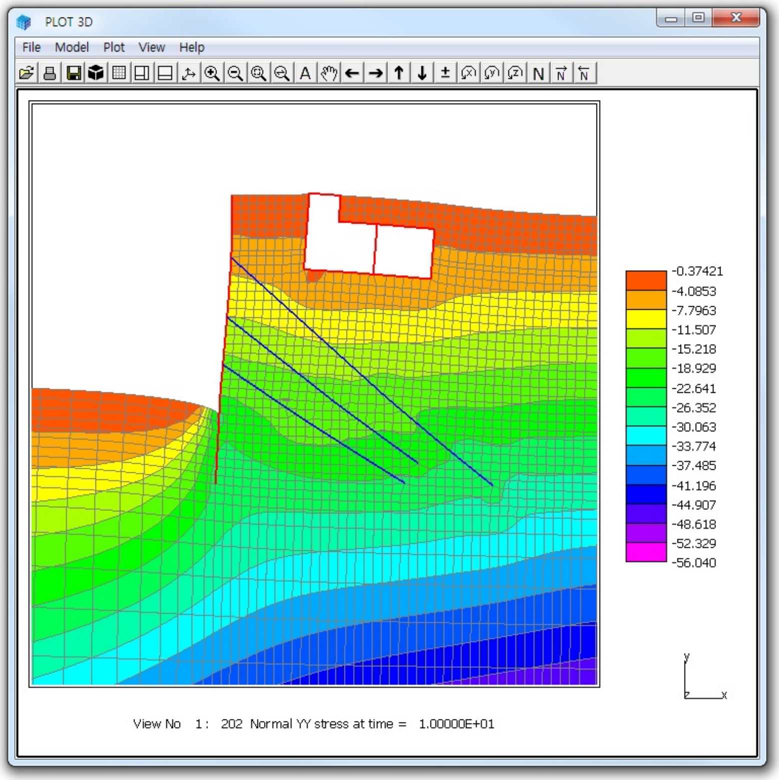


Figure 4.119 Vertical stresses on deformed mesh at Step 10

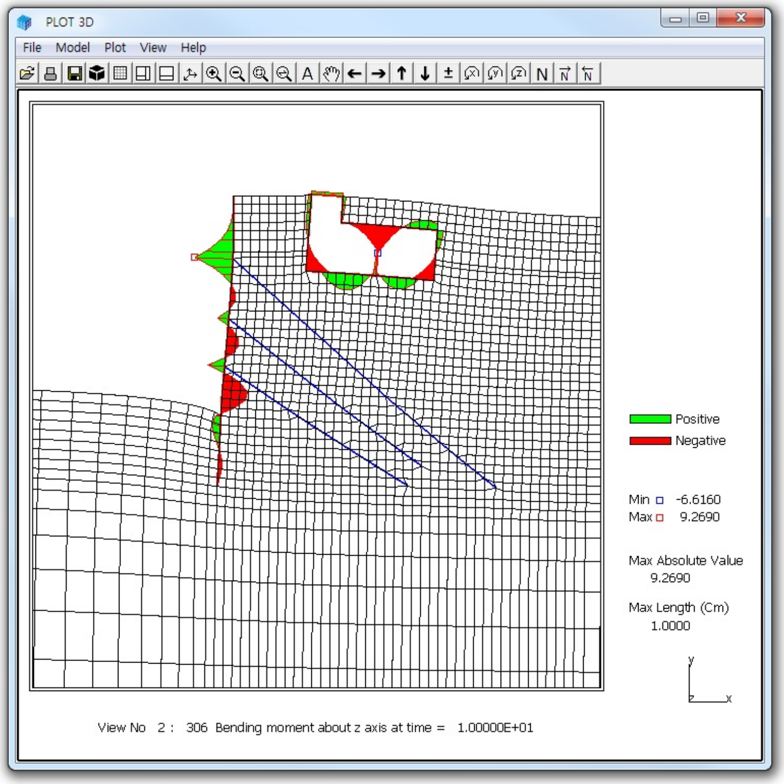


Figure 4.120 Bending moments on deformed mesh at Step 10

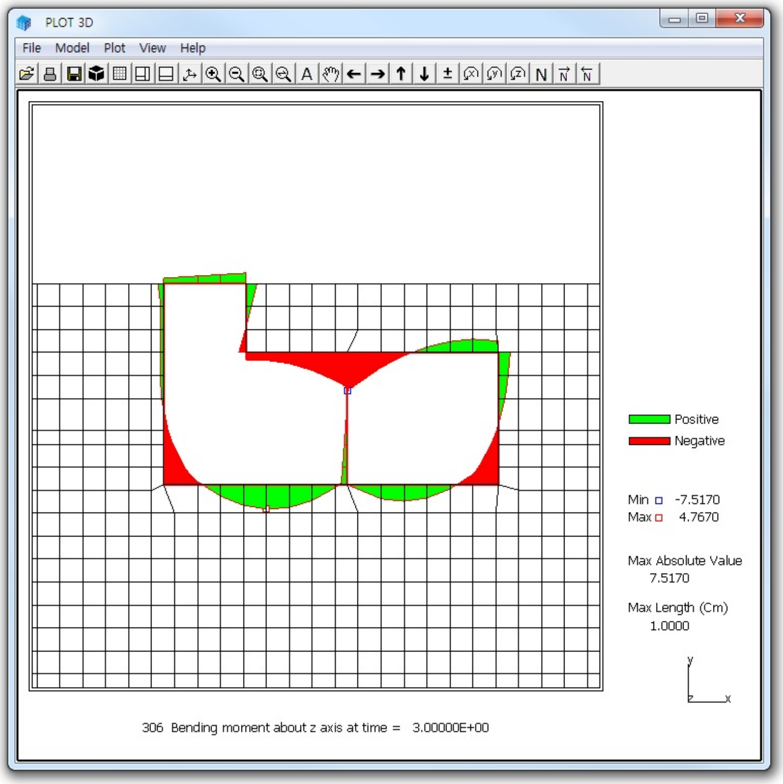


Figure 4.121 Bending moments in RC box frame at Step 3

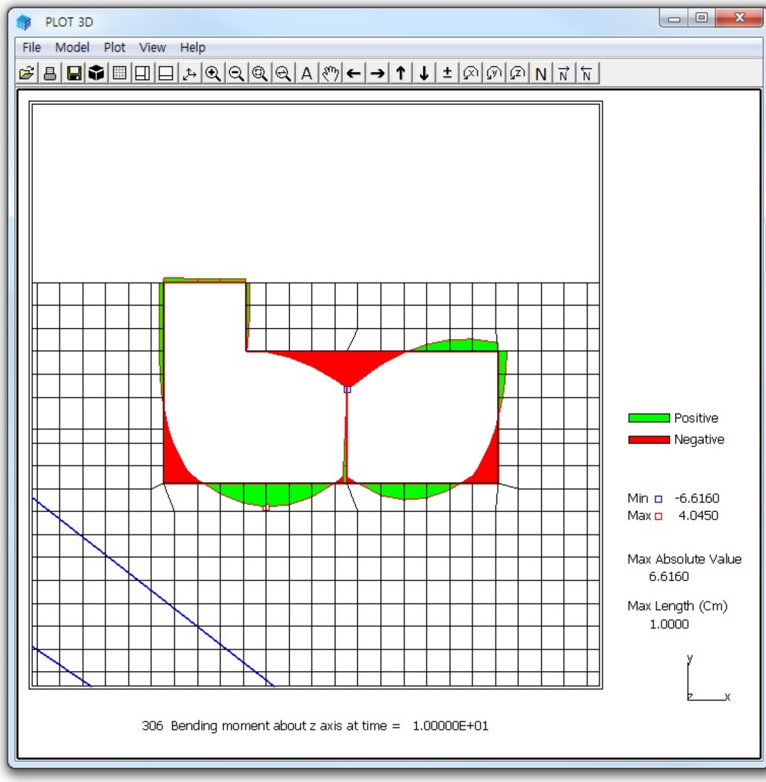


Figure 4.122 Bending moments in RC box frame at Step 10

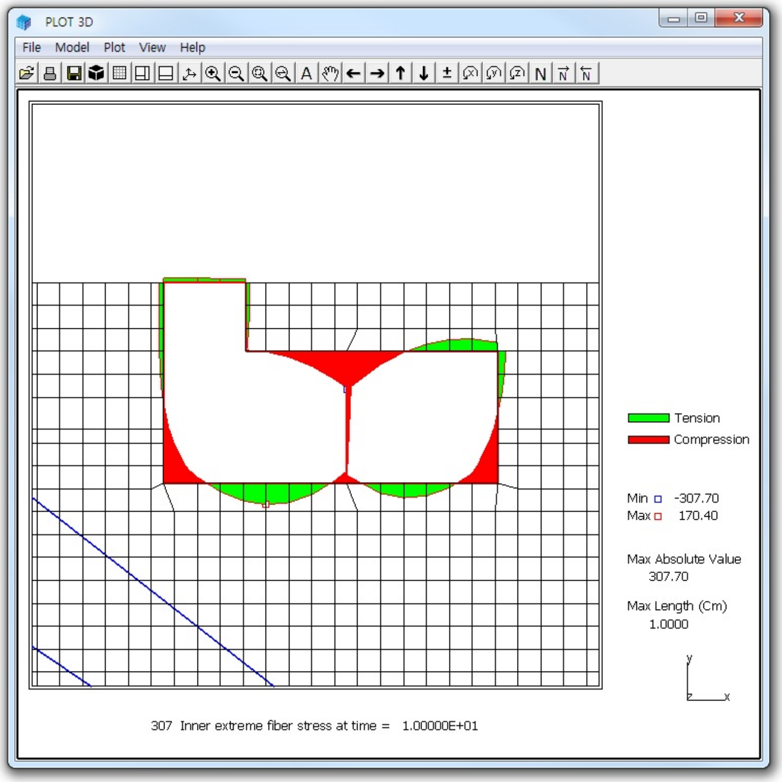


Figure 4.123 Inner extreme fiber stress in concrete at Step 10

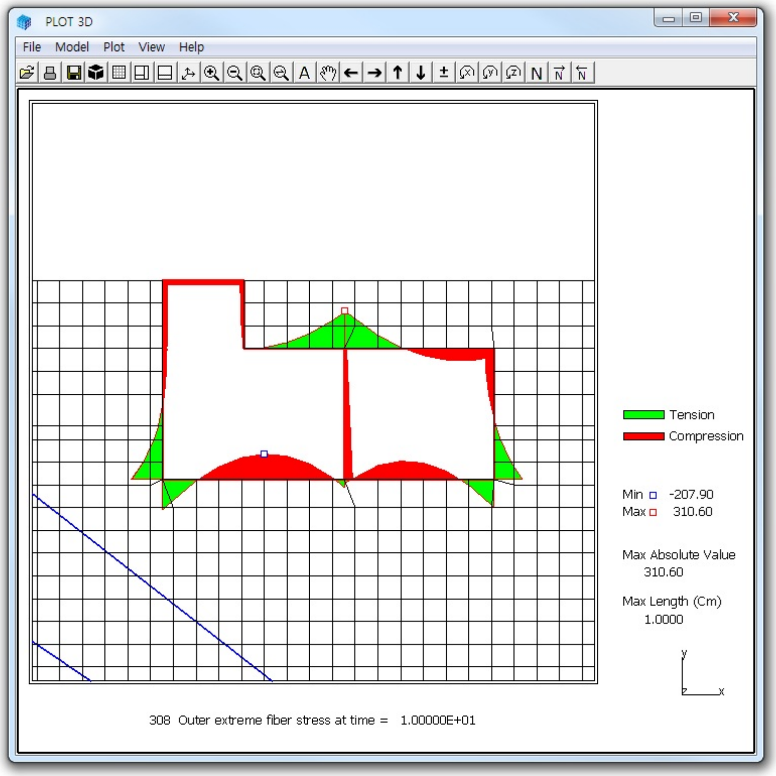


Figure 4.124 Outer extreme fiber stress in concrete at Step 10

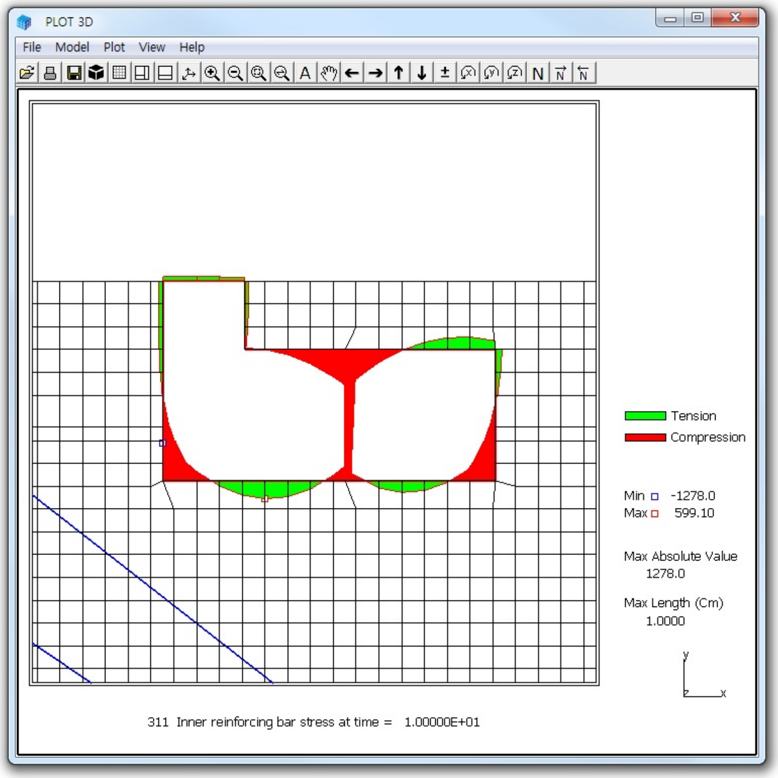


Figure 4.125 Inner reinforcing bar stress at Step 10

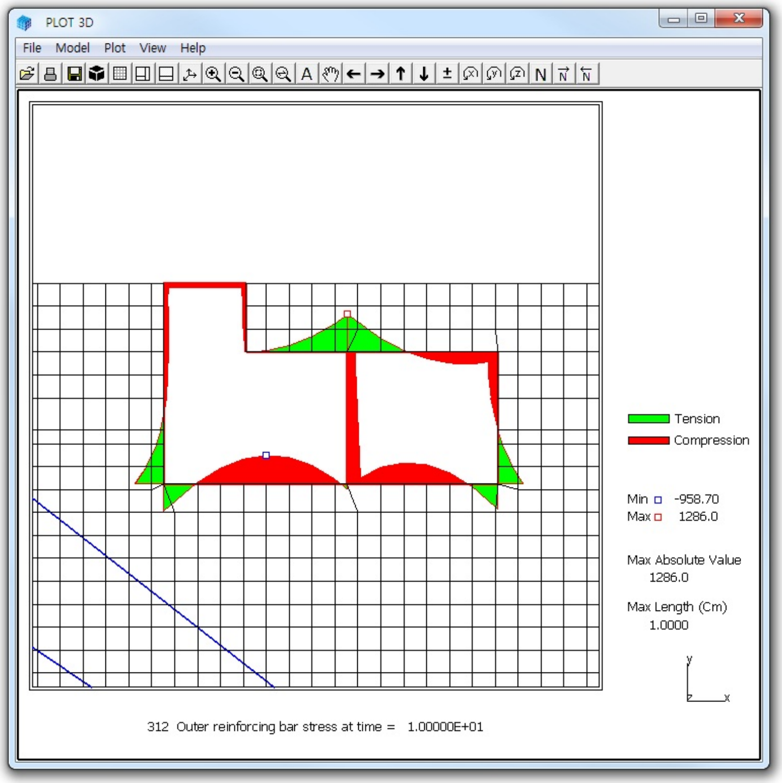


Figure 4.126 Outer reinforcing bar stress at Step 10

4.25 Plane Strain Tunnel in Jointed Continuum

This example problem is to verify the jointed continuum mesh generated by JOINT-2D 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 16 except that the tunnel is located in the jointed continuum. The jointed continuum mesh is generated by JOINT-2D program with the input file Joint.inp. Refer to JOINT-2D User's Manual.

Figure 4.127 shows the finite element mesh consisting of the jointed continuum around tunnel.

To compare with continuum model (Example Problem 16), 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 4.128 Deformed shape for Elastic Joint
- Figure 4.129 Principal stress vector for Elastic Joint
- Figure 4.130 Bending moment for Elastic Joint
- Figure 4.131 Deformed shape for Plastic Joint
- Figure 4.132 Principal stress vector for Plastic Joint
- Figure 4.133 Bending moment for Plastic Joint

In general, rersults of the Elastic Joint Model are close to those of conventional continuum analysis in Example Problem 16.

On the other hand, Plastic Joint Model shows considerable amount of slippage below bottom corner of tunnel as in Figures 4.131 and 4.132. Stress distributions are quite different from Elastic Joint Model.

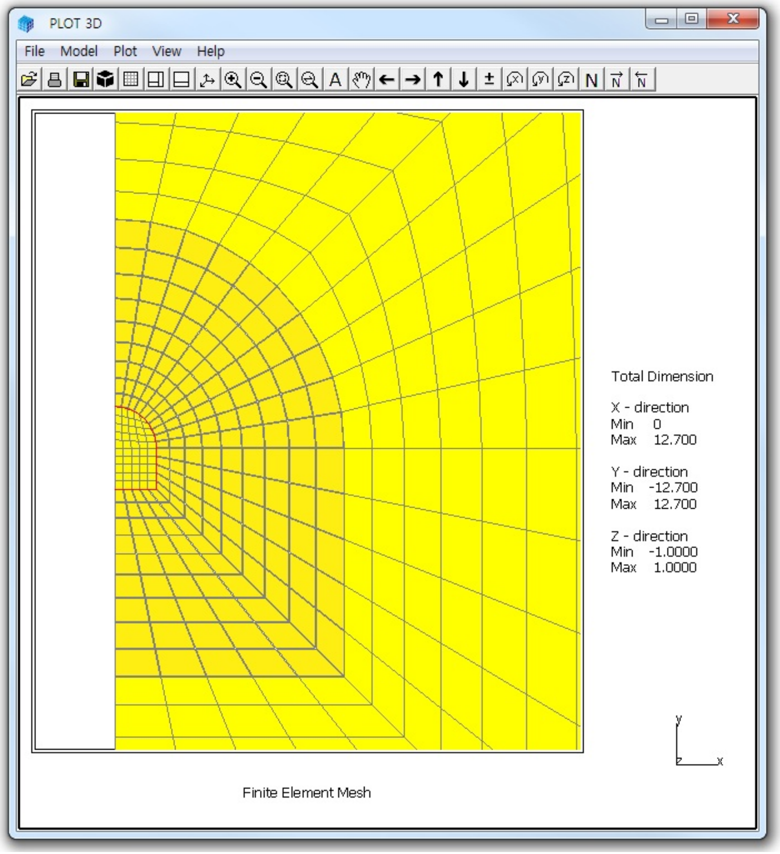


Figure 4.127 Finite element mesh around tunnel

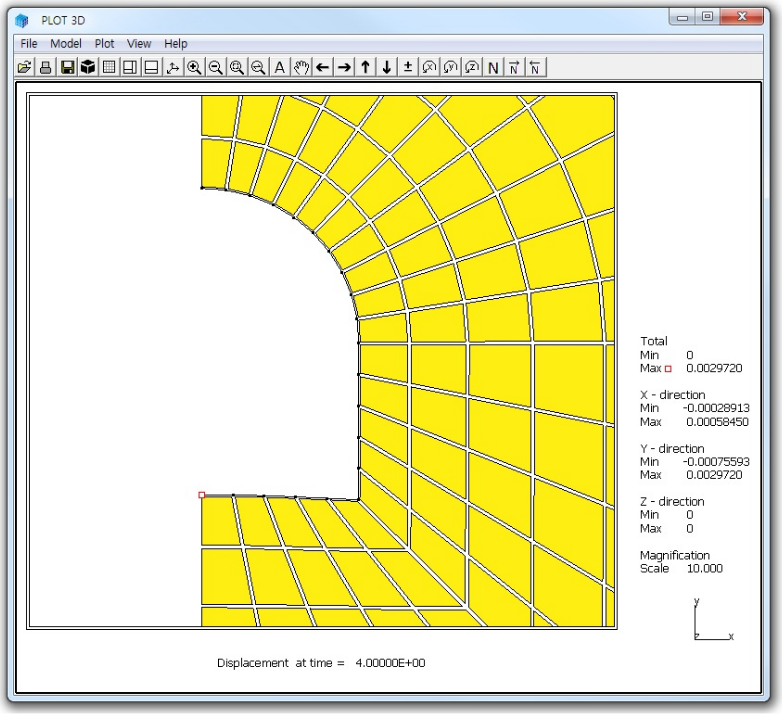


Figure 4.128 Deformed shape for Elastic Joint

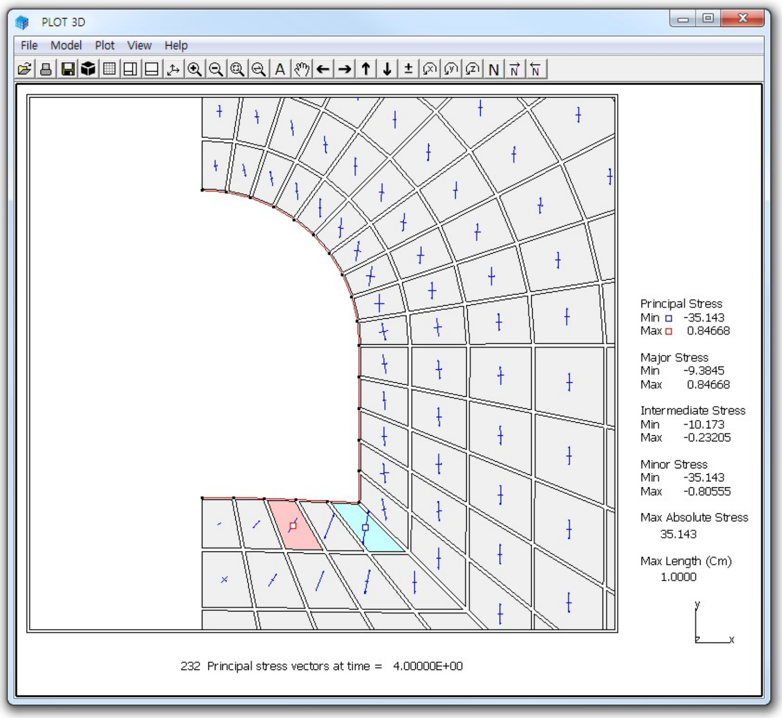


Figure 4.129 Principal stress vector for Elastic Joint

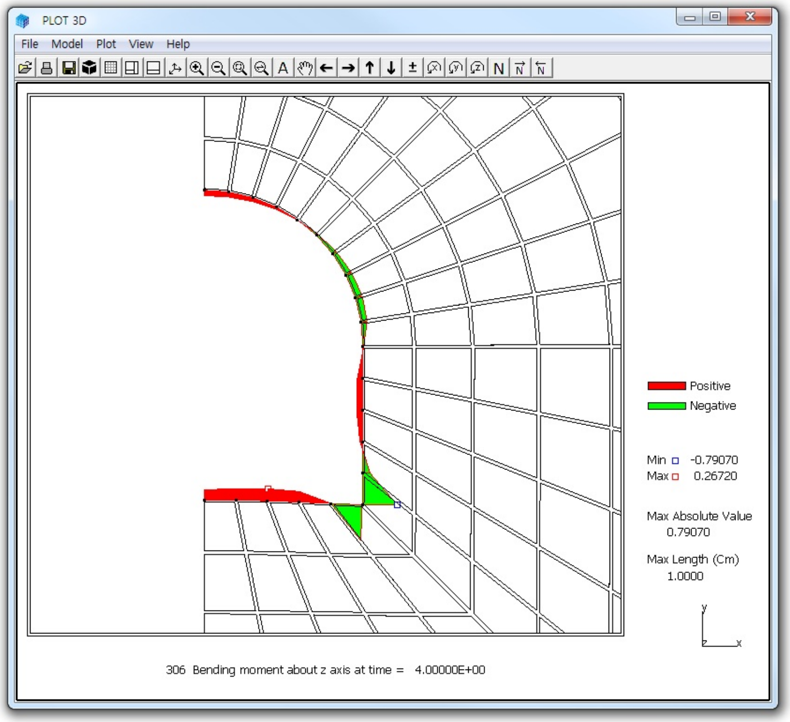


Figure 4.130 Bending moment for Elastic Joint

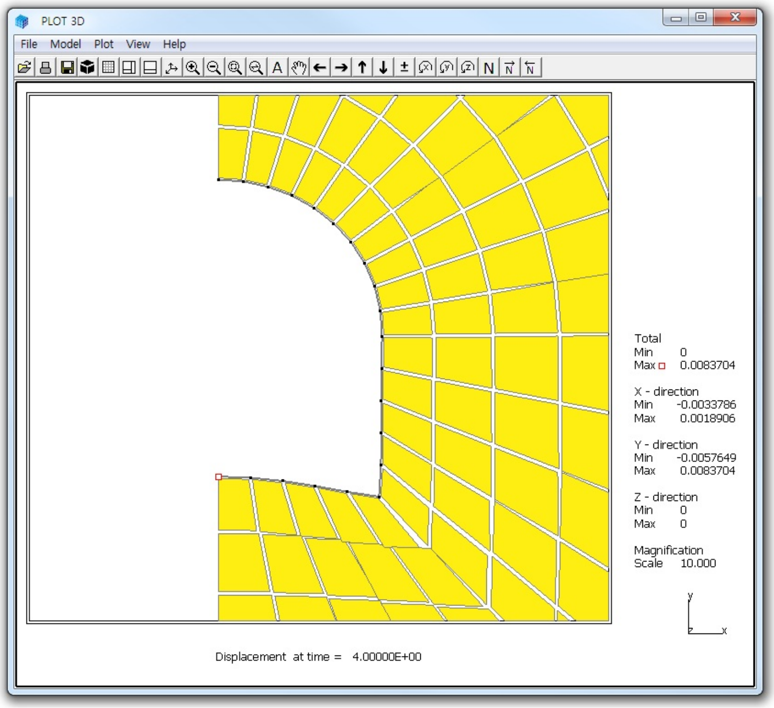


Figure 4.131 Deformed shape for Plastic Joint

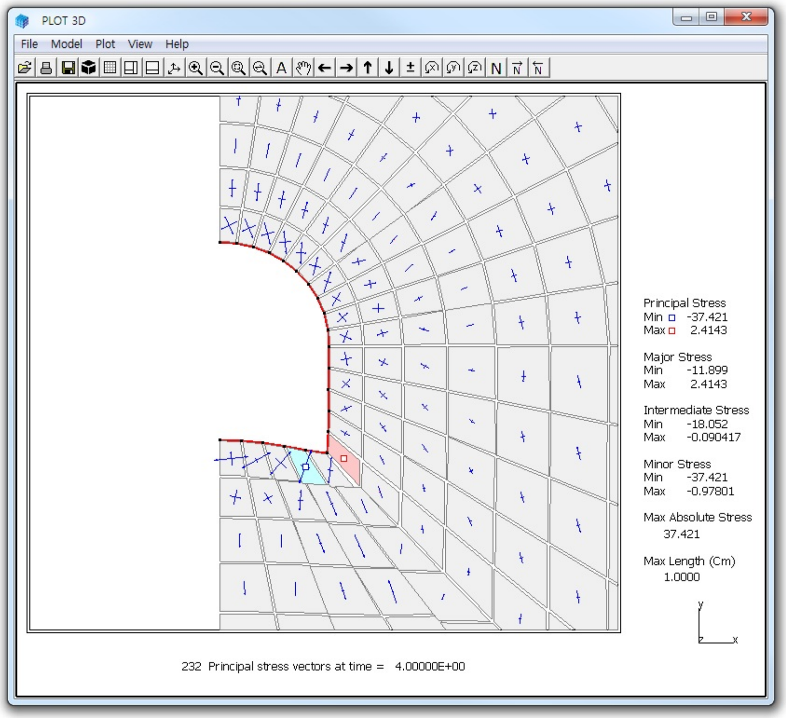


Figure 4.132 Principal stress vector for Plastic Joint

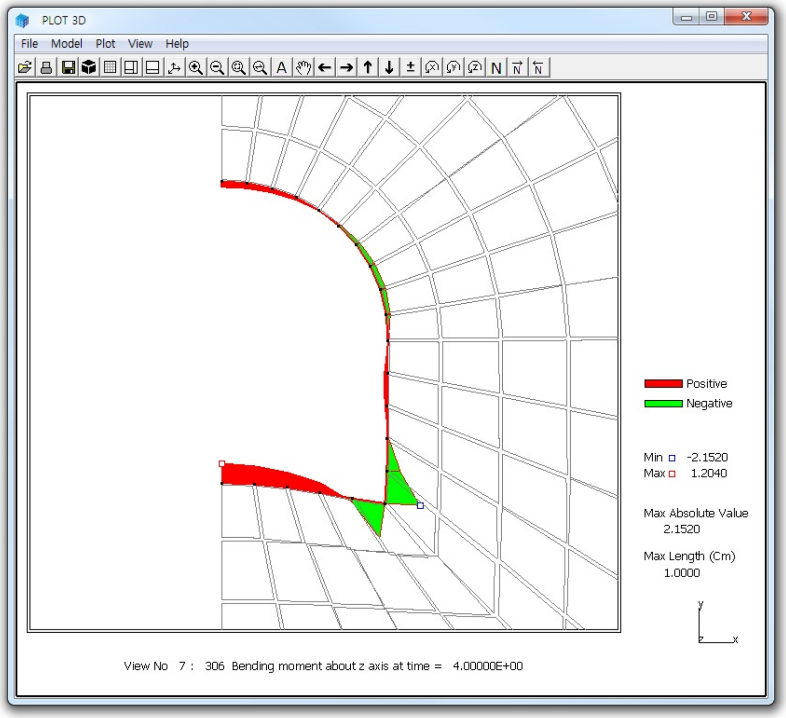


Figure 4.133 Bending moment for Plastic Joint

4.26 Spring Analysis

This example problem is to show how to model springs using special features in beam element in Card 6.5 of SMAP-2D User's Manual.

The example is composed of two truss members connected by horizontal and vertical springs as shown in Figure 4.134. The structure is subjected to external horizontal and vertical nodal forces.

Figure 4.135 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 4.134, 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-2D results show exact as shown in Figures 4.136, and 4.137 for displacements and truss axial forces, respectively.

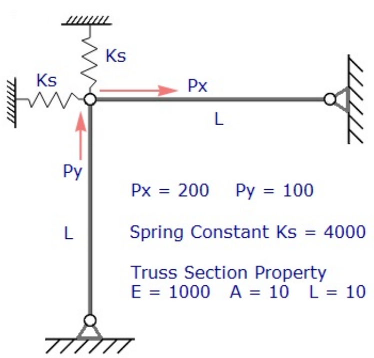


Figure 4.134 Truss members connected by springs

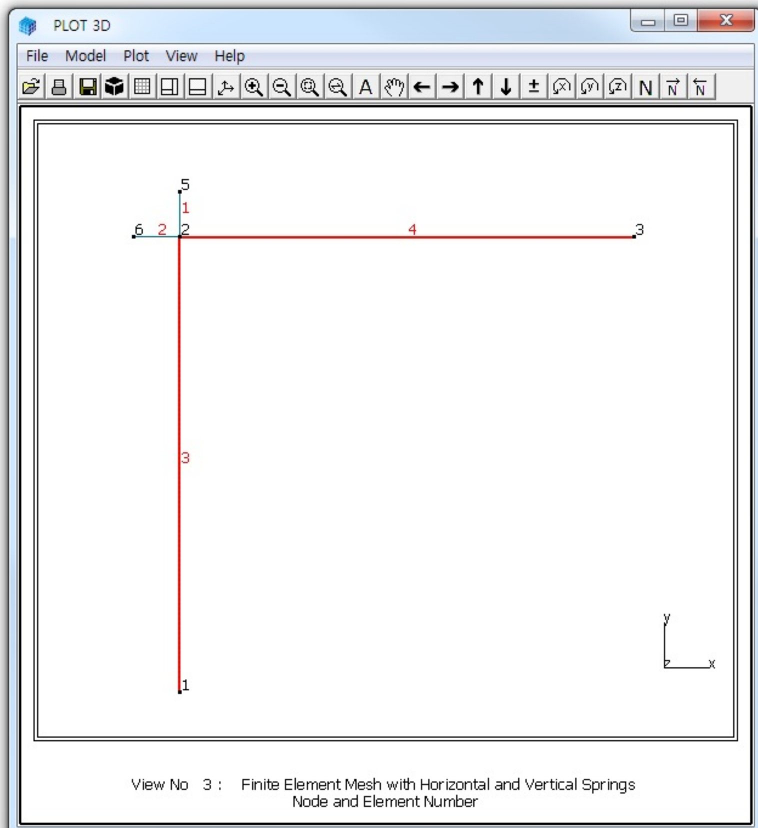


Figure 4.135 Finite element mesh for Example Problem 26

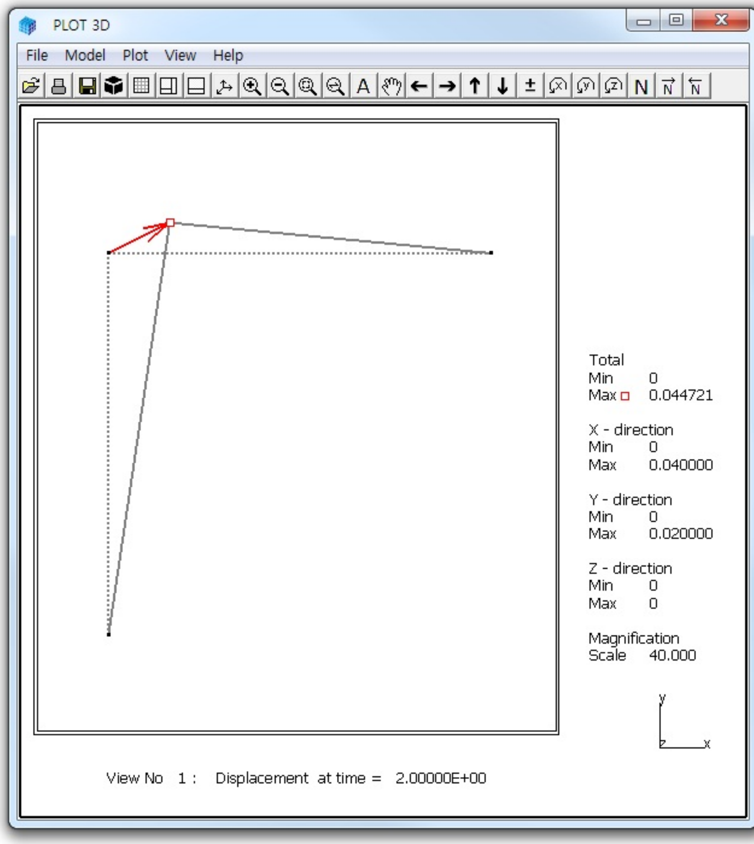


Figure 4.136 Deformed shape for Example Problem 26

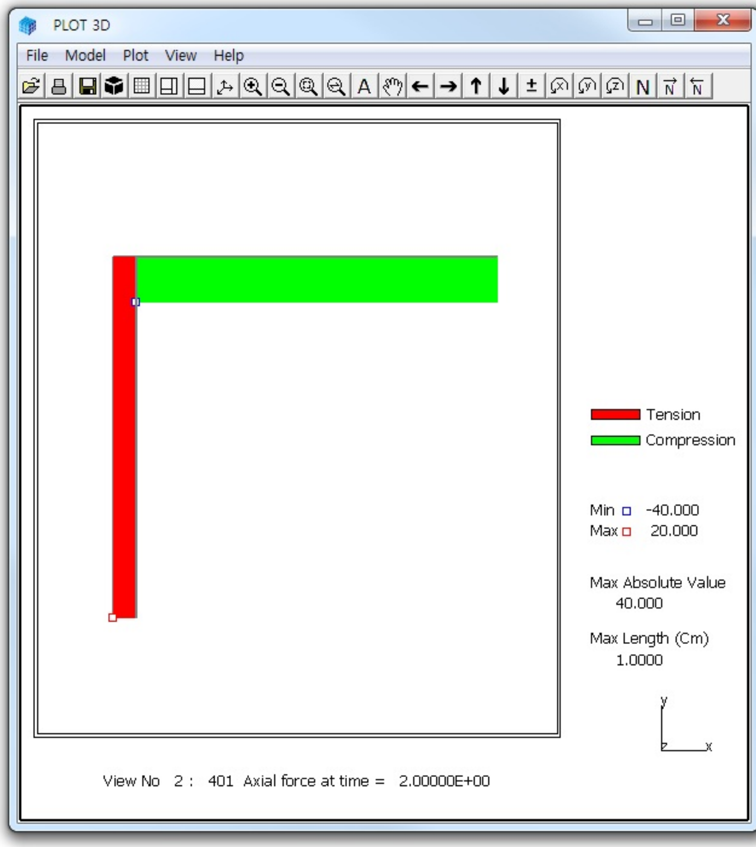


Figure 4.137 Truss axial force for Example Problem 26

4.27 Nonlinear Truss Analysis

Truss elements in SMAP can consider nonlinear behavior such as yielding and post buckling as schematically illustrated in Figure 4.139. Following examples are to show how to use such material parameters in truss element in Card 7.4.3 of SMAP-2D User's Manual.

Figure 4.138 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 4.140)
2. Compression and Tension Yielding (Figure 4.141)
3. Tension Yielding for No Compression Member (Figure 4.142)
4. Compression Yielding for No Tension Member (Figure 4.143)
5. Buckling for No Tension Member (Figure 4.144)
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.

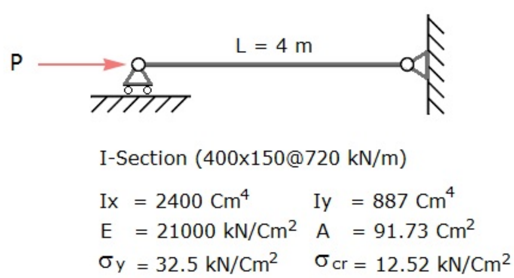


Figure 4.138 Truss member subjected to axial force

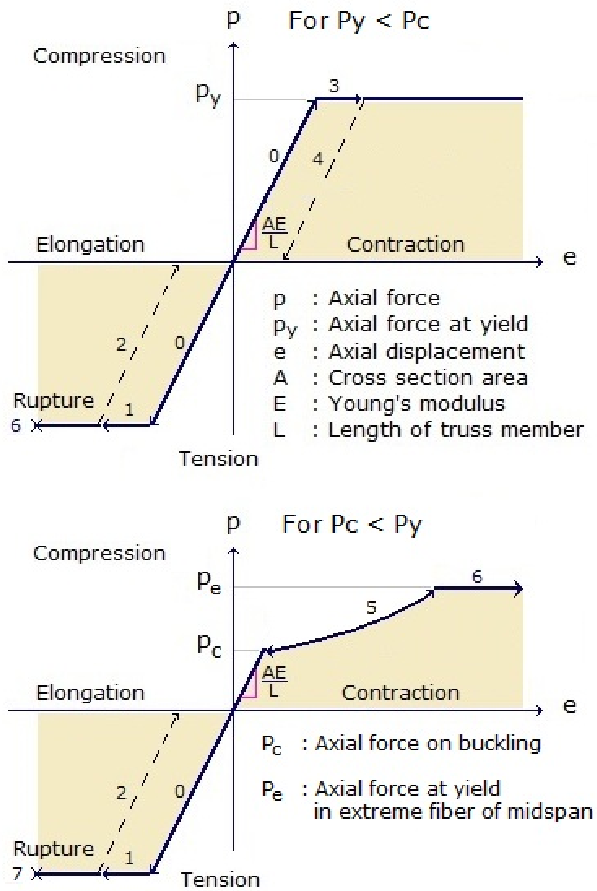


Figure 4.139 Nonlinear force displacement curve

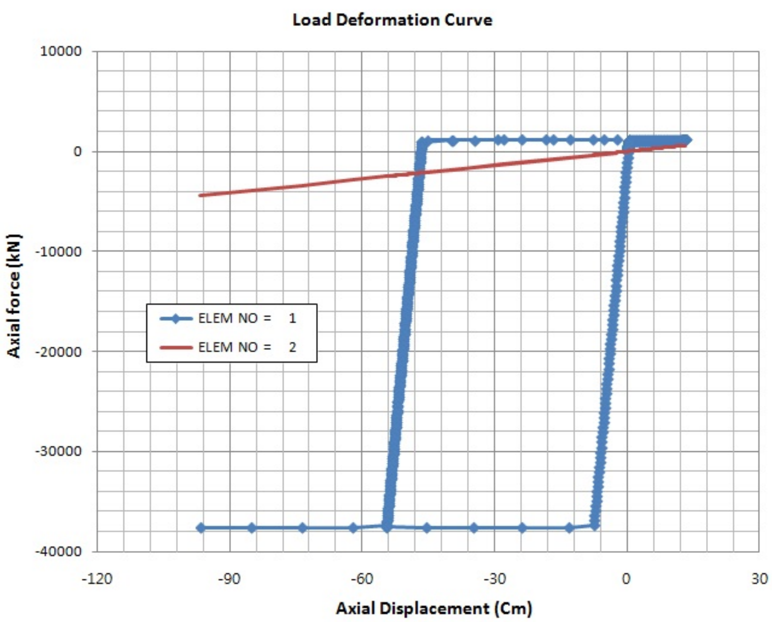


Figure 4.140 Buckling and Tension Yielding (VP27-1)

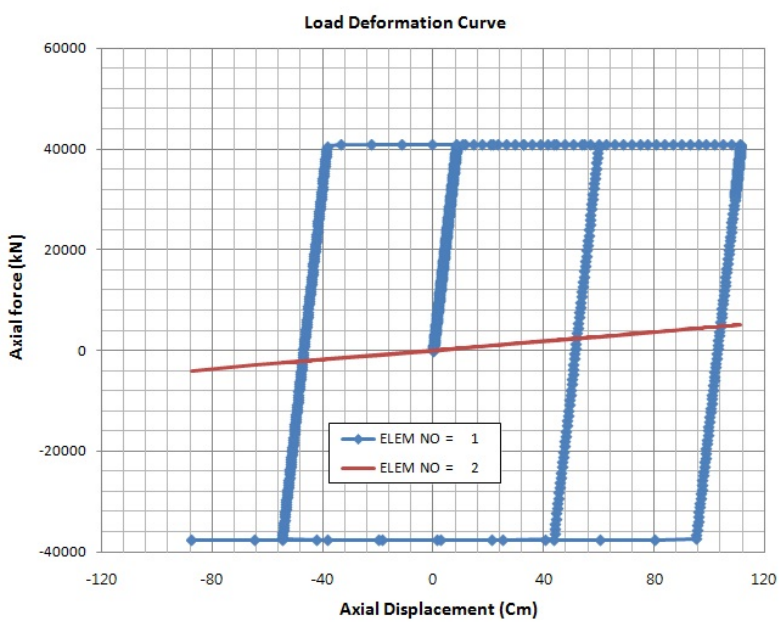


Figure 4.141 Compression and Tension Yielding (VP27-2)

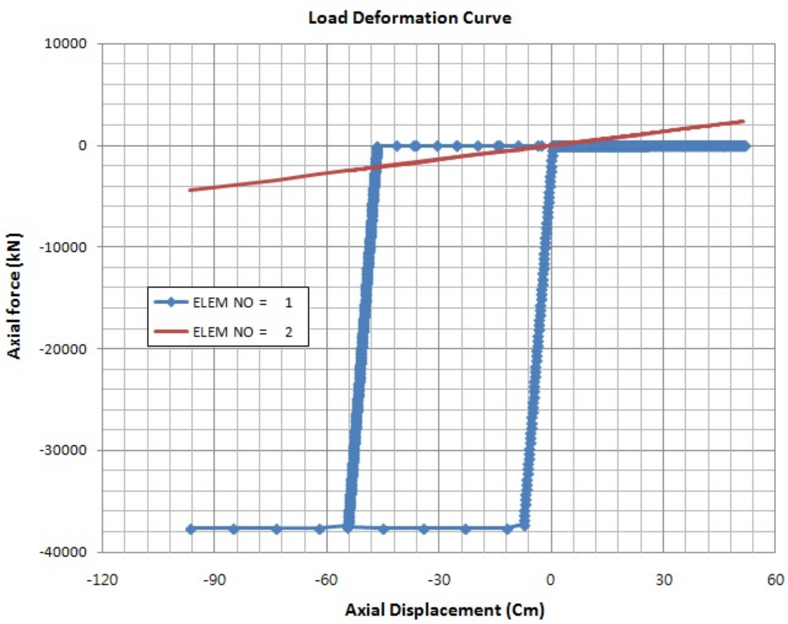


Fig 4.142 Tension Yielding for No Compression Member (VP27-3)

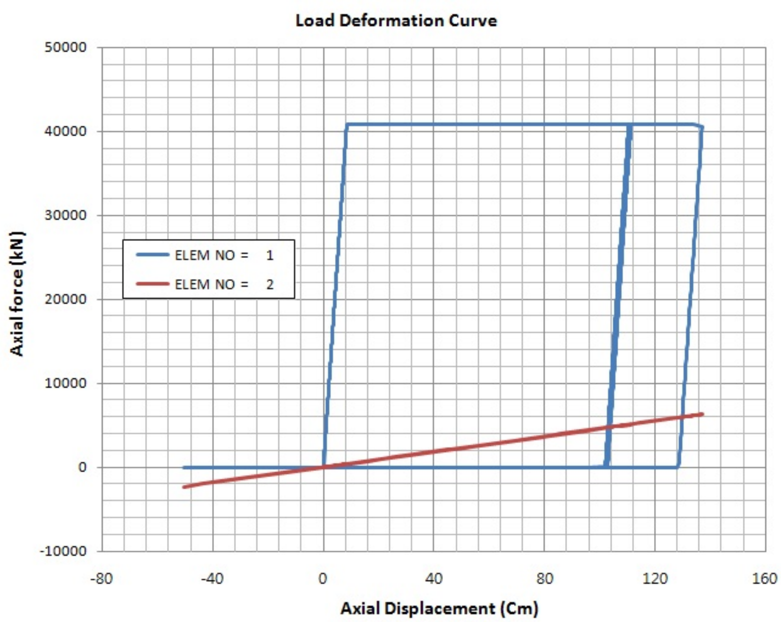


Fig 4.143 Compression Yielding for No Tension Member (VP27-4)

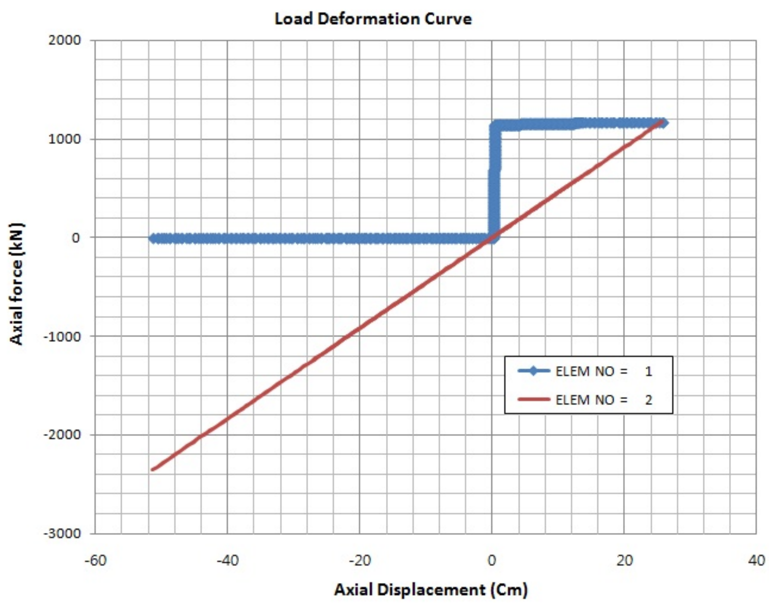


Figure 4.144 Buckling for No Tension Member (VP27-5)

Case 6 Initial Stress

For this example, following parameters are used:

$L = 400$ Cm $\quad E_1 = 21000$ kN/Cm^2 $\quad E_2 = 1000$ 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,

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

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

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

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

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

$$\varepsilon_2 = E_1 (1 - L / Lo) / (E_2 + E_1 \cdot L / Lo) \qquad (5)$$
$$= 0.00045475$$

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.

4.28 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 4.145. Mass is lumped at the node in the right side of truss member.

Following parameters are assumed:

L = 120 inch A = 1 in^2 $E = 30x10^6$ psi
ρ = (1/1.2) lb-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-s^2/in

Equivalent spring constant:
$k = E A / L = (30x10^6)(1)/(120) = 250{,}000$ 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

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

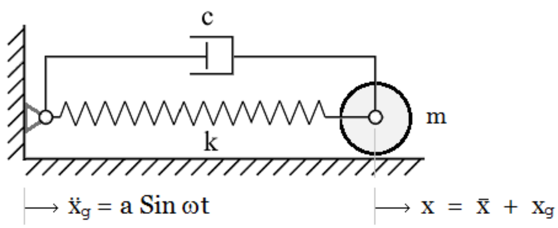


Figure 4.145 SDOF system subjected to ground acceleration

Figure 4.146 shows time history of computed relative displacements. SMAP results are almost identical to the exact solution.

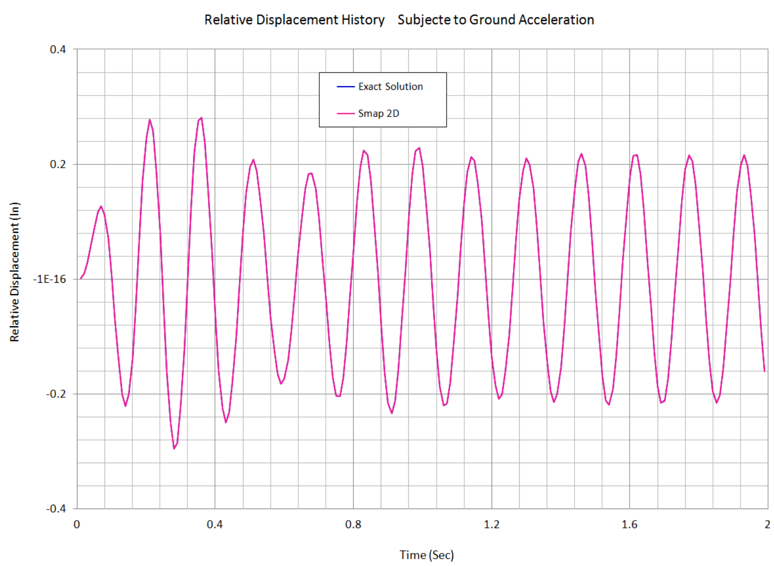


Fig 4.146 Time history of relative displacements

4.29 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 4.147.

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 = 1×10^{6} t-m/rad
4. Rotational spring connection, very flexible Kr = 1×10^{-3} t-m/rad
5. Rotational spring connection, somewhat rigid Kr = 1×10^{4} t-m/rad

Computed results are summarized in detail in Joint_Spring_2D.pdf. It approaches to rigid connection when the rotational spring is rigid and hinge connection when the spring constant is very flexible.

Figures 4.148 to 4.152 show finite element mesh, deformed shape, thrust, shear and bending moment distributions, respectively, for the rotational spring connection with Kr = 1×10^{4} t-m/rad.

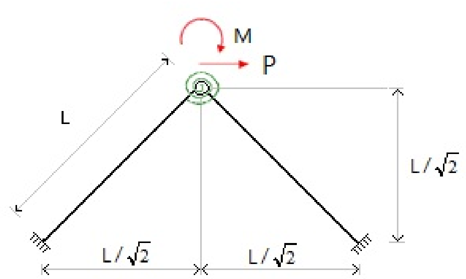


Figure 4.147 Frames with rotational spring connection

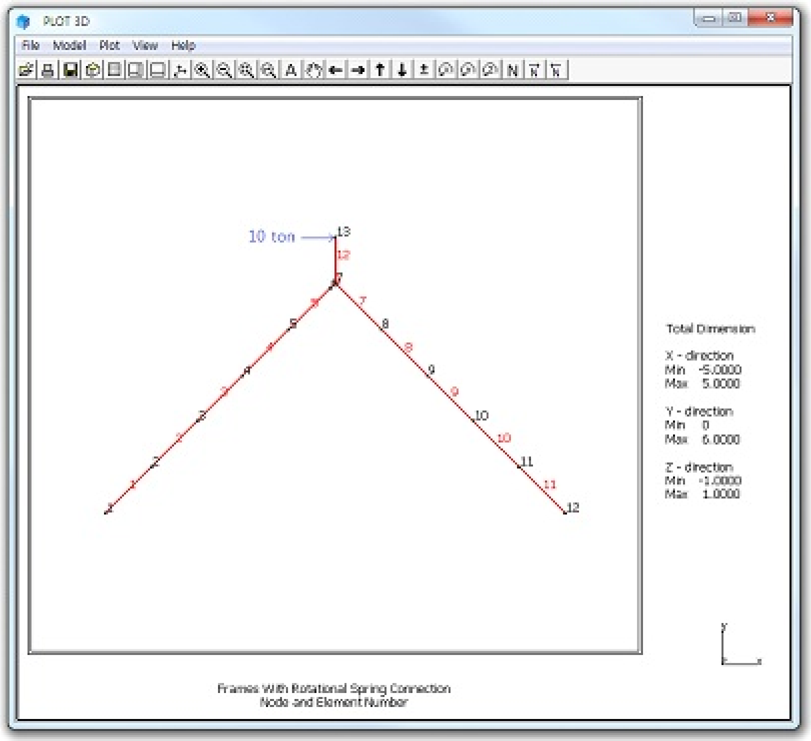


Figure 4.148 Finite element mesh

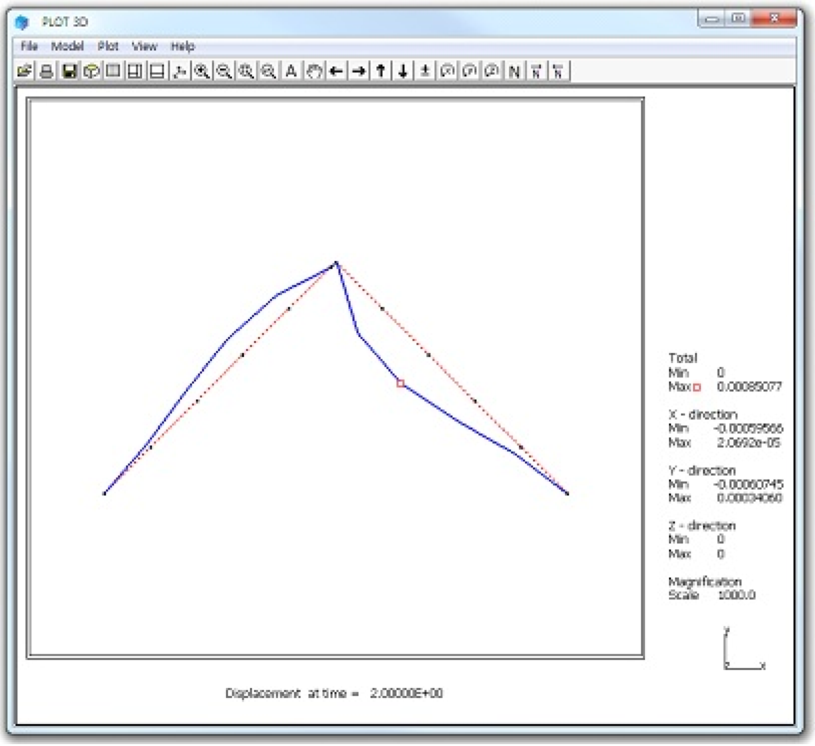


Figure 4.149 Deformed shape

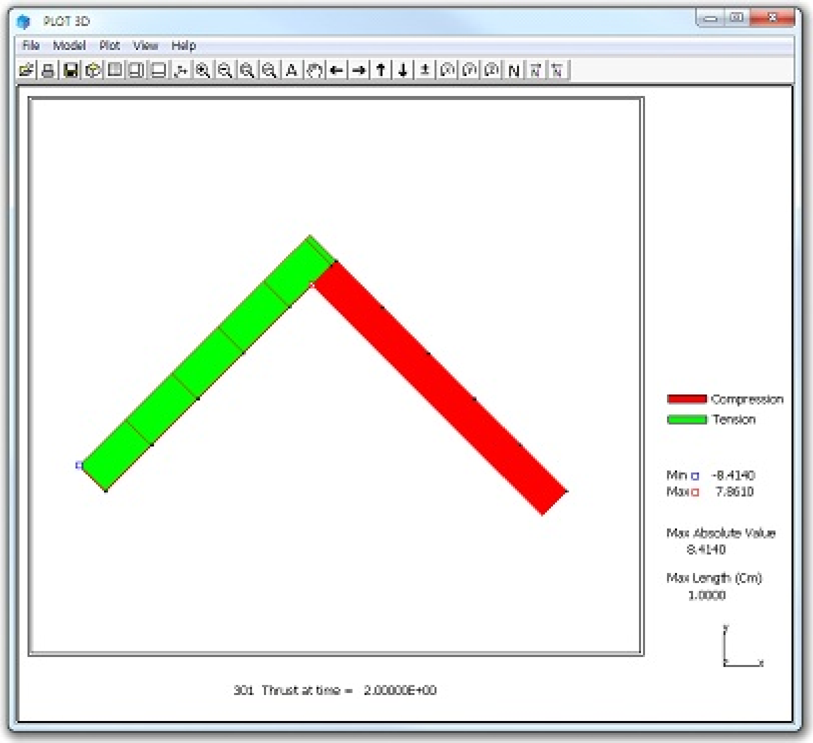


Figure 4.150 Thrust distribution

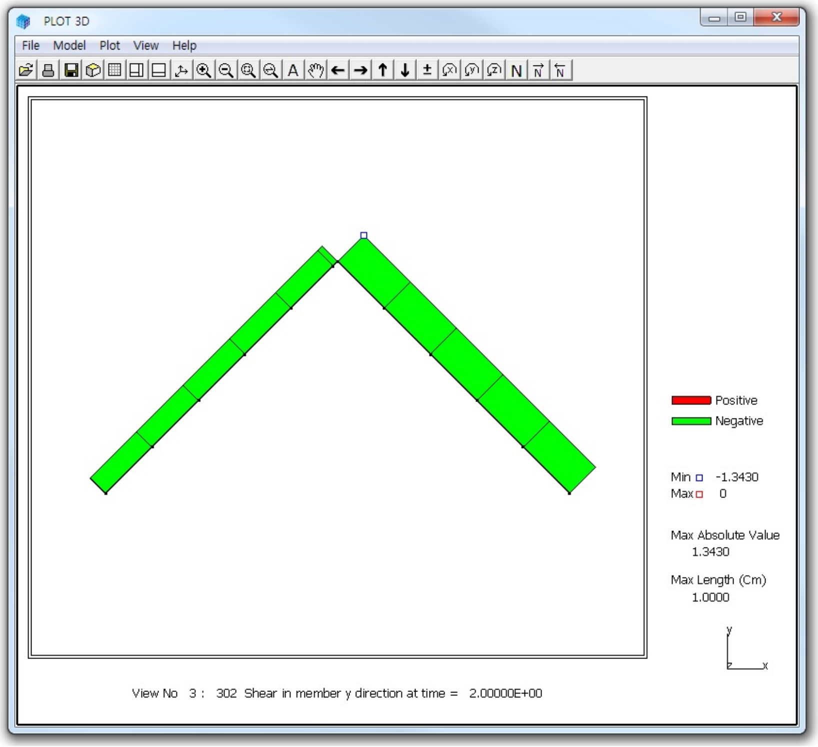


Fig 4.151 Shear distribution

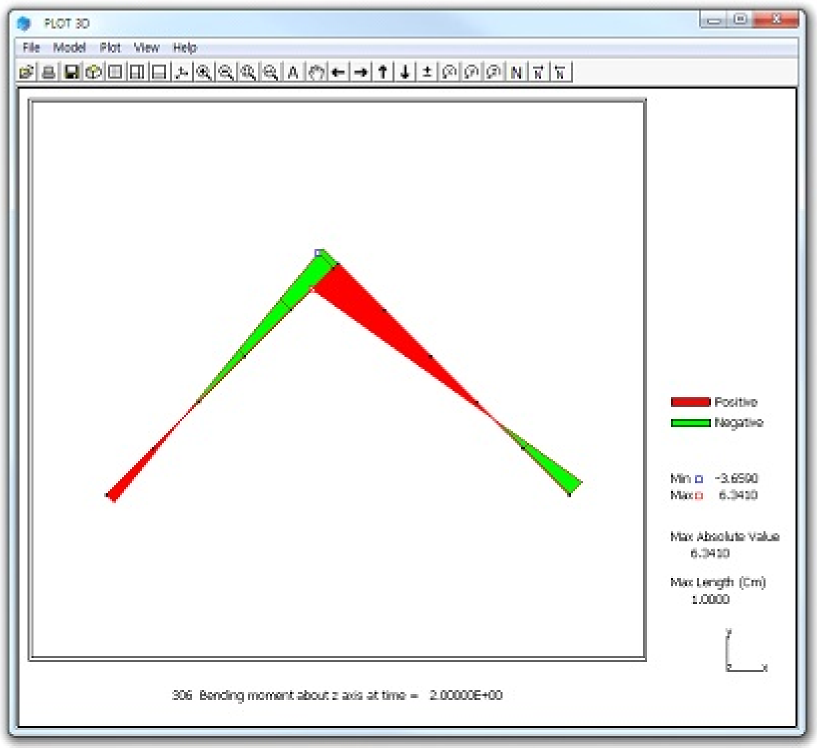


Fig 4.152 Bending moment distribution

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 4.153.
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_2D.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 shown in Figure 4.154, the structure is modeled by beam elements which have capability of modeling axially symmetric reinforced shell.

Overall, SMAP-2D results are very close to the exact solutions.
Refer to the following two files for detailed graphical outputs:
Reinforced_Cylinder_2D.pdf and Smap-2D_Vp30.pdf.

SMAP-2D results for case 3 are compared with closed-form solutions:

Figure 4.155 Radial displacement profile
Figure 4.156 Meridian bending moment profile

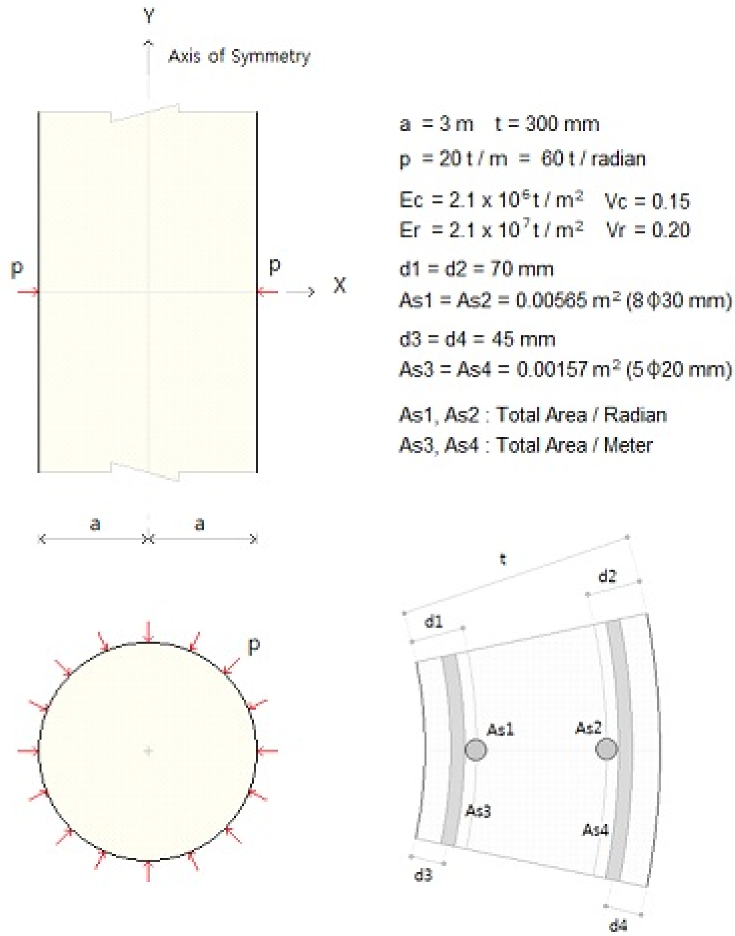


Figure 4.153 Reinforced cylinder section view

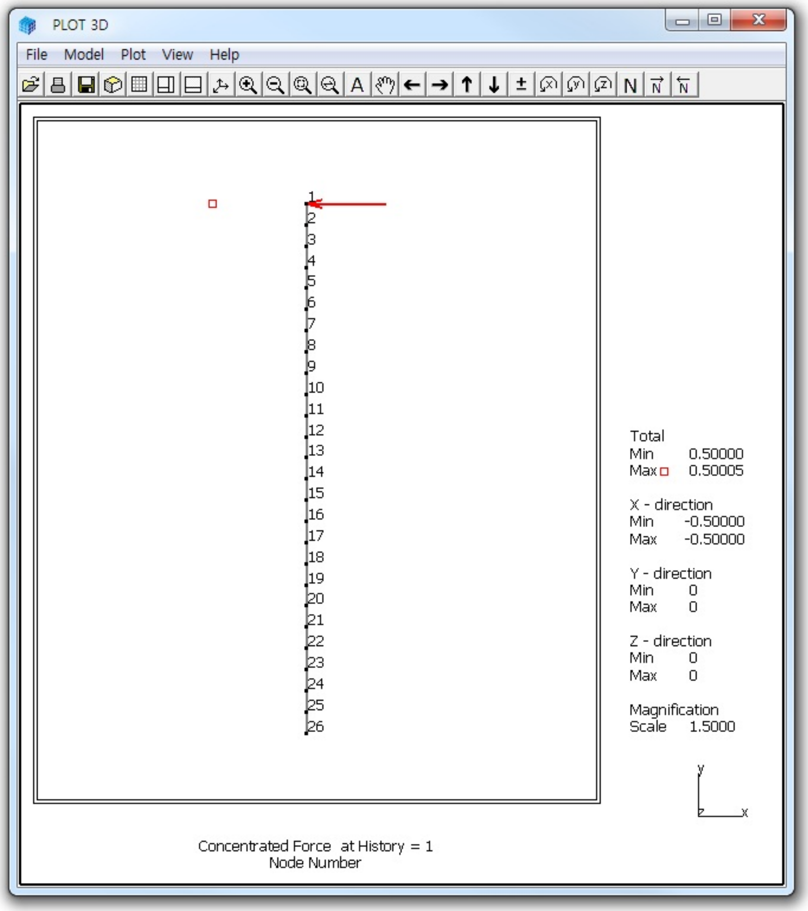


Figure 4.154 Finite element mesh with applied load

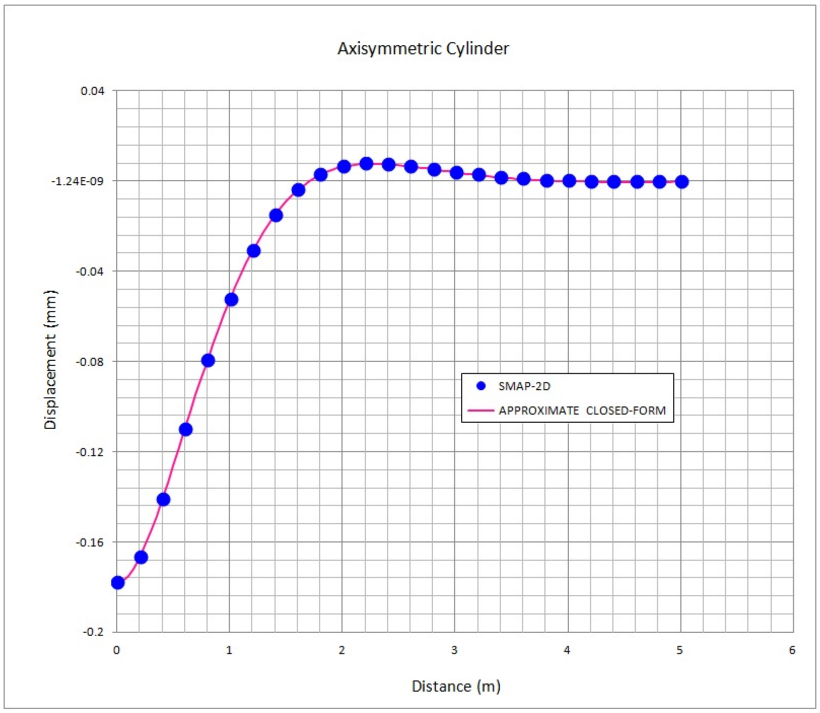


Figure 4.155 Radial displacement profile

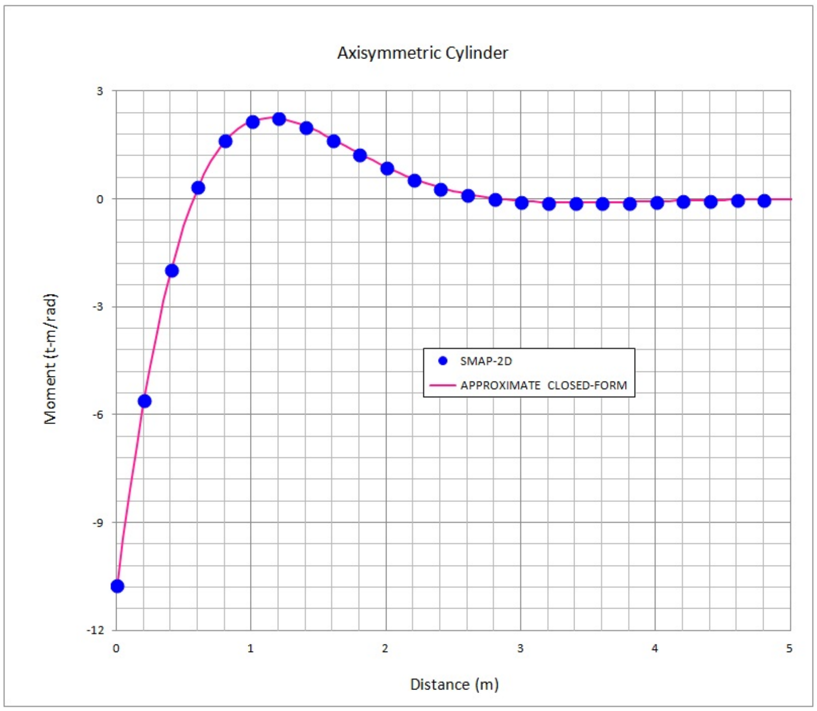


Figure 4.156 Meridian bending moment profile

4.31 Beam Modal Analysis

This example problem is to solve dynamic response of a simply supported beam subjected to a concentrated step load at mid span by Modal Superposition method. See Figure 4.157.

The exact deflection for undamped beam is given by:

$$\delta = \frac{2\,P\,L^3}{\pi^4\,E\,I} \sum_{n=1}^{\infty} \frac{\alpha_n}{n^4} \left(1 - \mathrm{Cos}\ \omega_n\, t\right)\ \mathrm{Sin}\ \frac{n\,\pi\,x}{L}$$

$$\omega_n = n^2\,\omega_1 \qquad \omega_1 = \frac{\pi^2}{L^2}\sqrt{\frac{E\,I}{m}} \qquad m = \rho\,A \qquad \alpha_n = (-1)^{i+1}$$

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

Following parameters are used for modal superposition analyses:

	=	2.753	$kN\text{-}s^2 / m^4$	L	=	10 m
A	=	0.09	m^2	I	=	$6.75 \times 10^{-4}\ m^4$
E	=	7×10^4	kN / m^2		=	0

Table 4.8 summarizes the computed natural frequencies along with closed form solution. All 2D and 3D modal analyses predict pretty well natural frequencies of the simply supported beam.

Figure 4.158 shows the contours of the first three modes solved by two dimensional continuum modal analysis. Figure 4.159 (a) shows deflection time history at beam center as predicted by modal superposition method using only first 4 mode shapes. SMAP-2D/3D modal superposition solutions predict very well the dynamic response of simply supported beam.

Figure 4.159 (b) shows deflection time history at beam center by modal superposition for 5% damping using first 4 mode shapes. All SMAP-2D/3D modal solutions agree very well with each other.

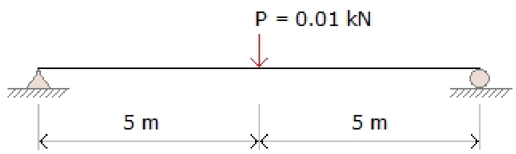


Figure 4.157 Simply supported beam subjected to step load

Table 4.8 Computed natural frequencies (rad/s)

Analysis Type	Mode Number		
	1	2	3
Closed Form	1.362	5.448	12.258
2D Continuum	1.358	5.369	11.774
2D Beam	1.357	5.364	11.830
3D Continuum	1.358	5.369	11.774
3D Shell	1.357	5.363	11.829
3D Beam	1.357	5.364	11.830

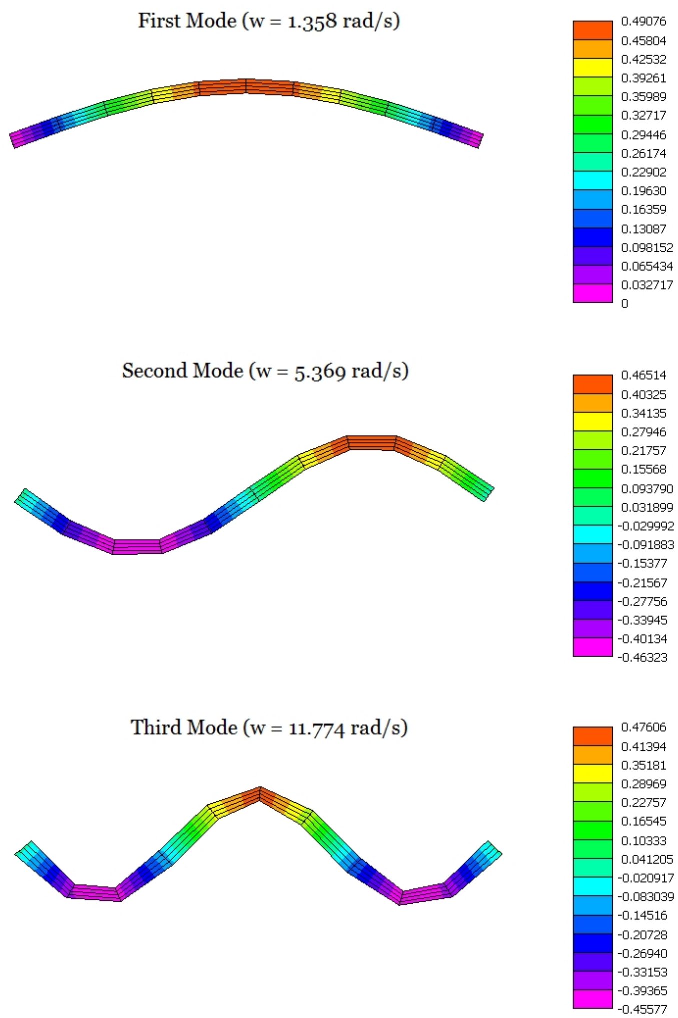


Figure 4.158 First three modes solved by Continuum(10x4)

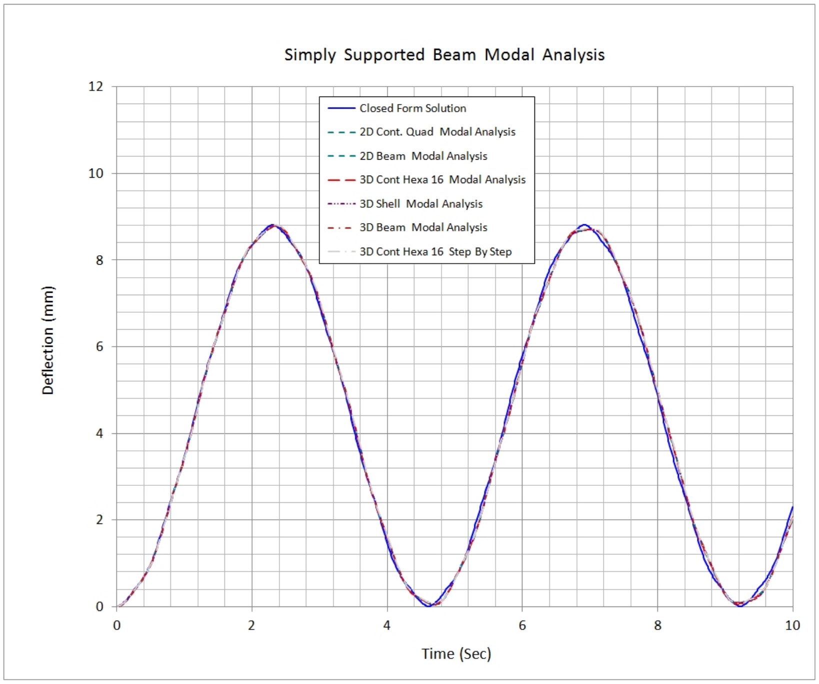


Figure 4.159 (a) Deflection time history at beam center

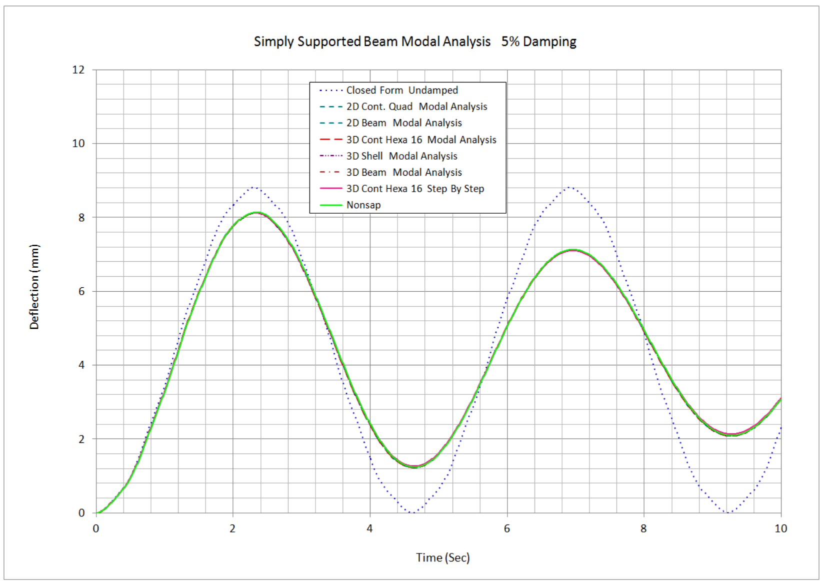


Figure 4.159 (b) Deflection history at beam center, 5% damping

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 4.160 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 4.161 and 4.162, 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:

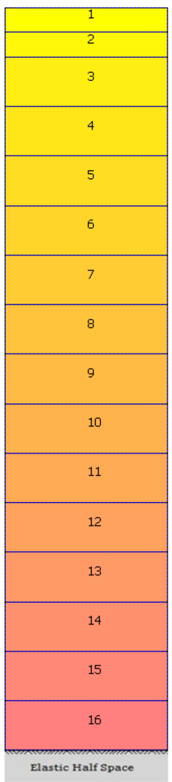
$$a = 2\,\beta\,\omega_1\,\omega_i \,/\, (\,\omega_1 + \omega_i\,) \qquad b = 2\,\beta \,/\, (\,\omega_1 + \omega_i\,)$$

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 4.163 shows computed acceleration time histories on the ground surface and Figure 4.164 shows the same accelerations between 10 and 12 seconds where strong motions occur. SMAP-2D solutions predict very closely the closed-form frequency domain SHAKE91 solution.

Figure 4.165 shows spectral accelerations with 5% structural damping on the ground surface and Figure 4.166 shows the same accelerations between 0.1 and 1 seconds. SMAP-2D 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 4.160 Finite element meshes and material properties

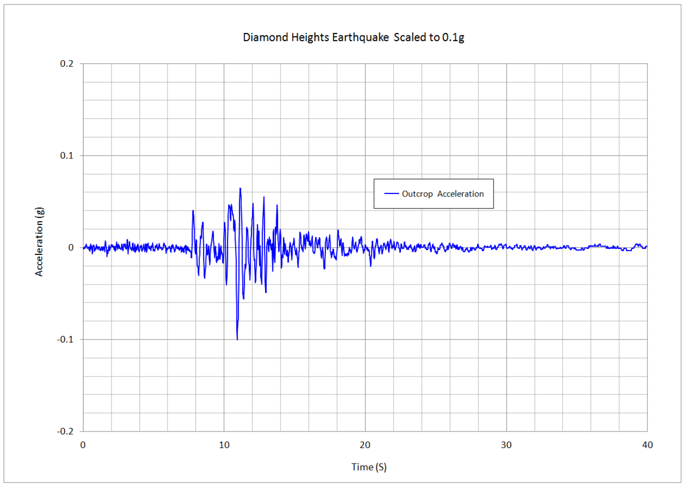


Figure 4.161 Diamond Heights acceleration time history

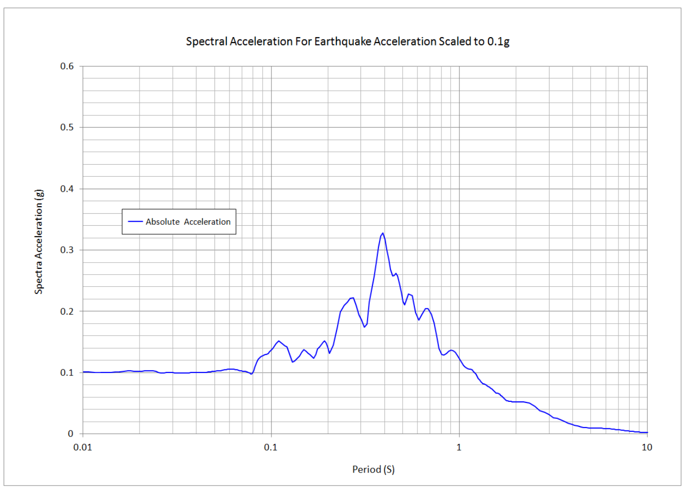


Figure 4.162 Spectral acceleration for input earthquake

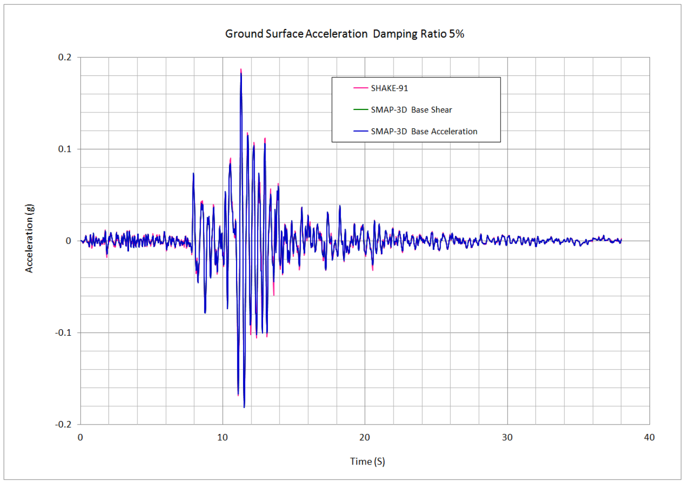


Figure 4.163 Ground surface accelerations

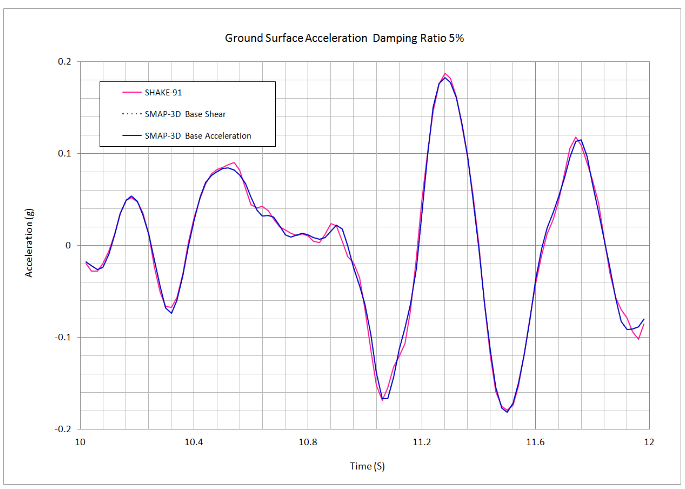


Figure 4.164 Ground surface accelerations between 10 and 12 sec.

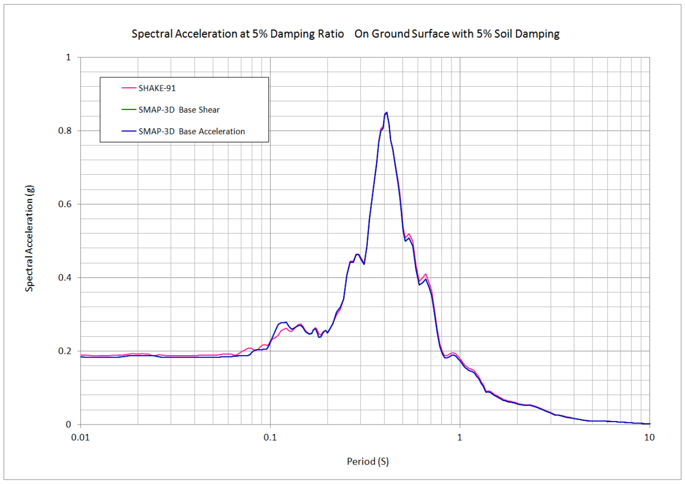


Figure 4.165 Spectral accelerations on ground surface

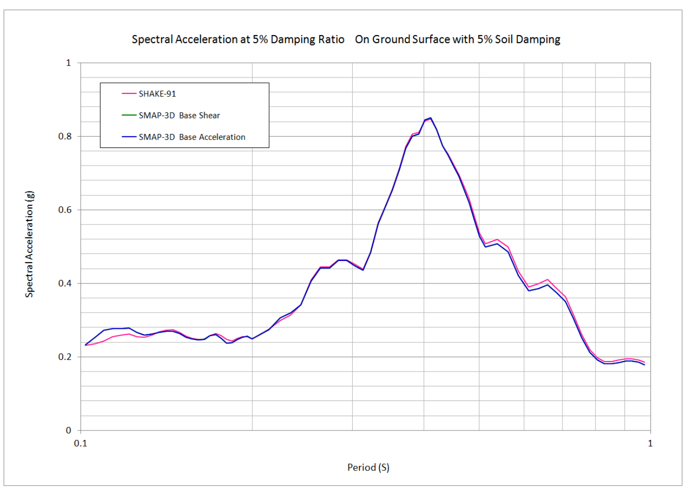


Figure 4.166 Spectral accelerations between 0.1 and 1 sec.

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 4.167 and 4.168 show finite element meshes and close-up view around silo, respectively. The program used only the right half of the whole mesh because the problem is axially symmetric about Y axis.

Table 4.9 lists material properties and Figure 4.169 shows schematic view of detailed silo lining structure. Table 4.10 lists lining thickness and reinforcement. Figure 4.170 shows silo lining material numbers. Table 4.11 shows schematically the sequence of silo construction including residual water pressure applied at step 5. Figure 4.171 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 4.172 Deformed shape of silo lining
- Figure 4.173 Dome deflection along A-B
- Figure 4.174 Storage wall radial displacement along C-D
- Figure 4.175 Dome lining inner hoop stress along A-B
- Figure 4.176 Dome outer rebar meridian stress along A-B
- Figure 4.177 Storage wall lining inner hoop stress along C-D
- Figure 4.178 Storage wall outer rebar meridian stress along C-D

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

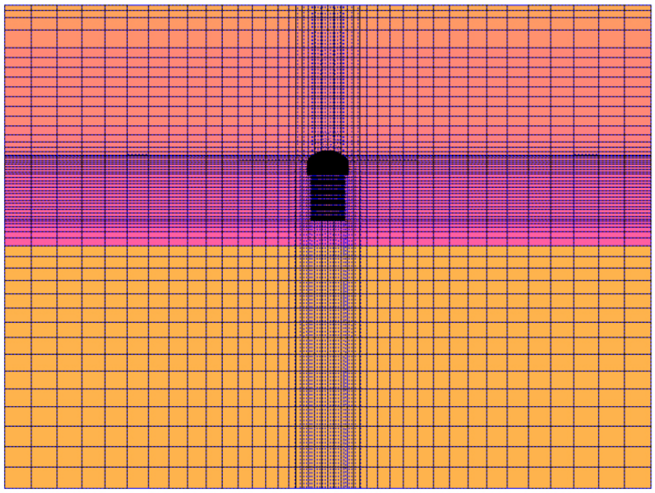


Figure 4.167 Finite element meshes

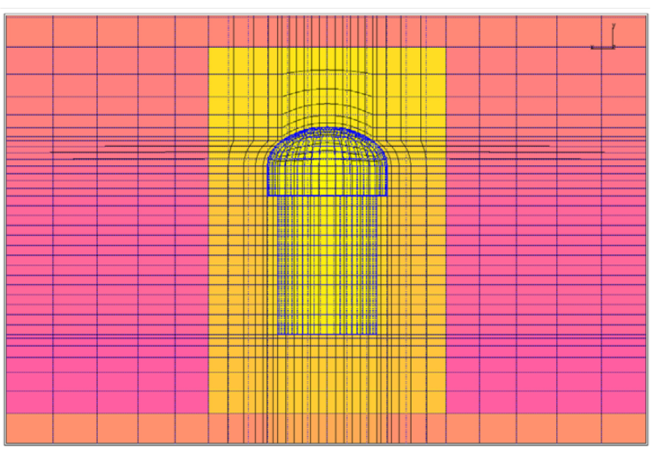


Figure 4.168 Finite element meshes around silo

Table 4.9 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	-

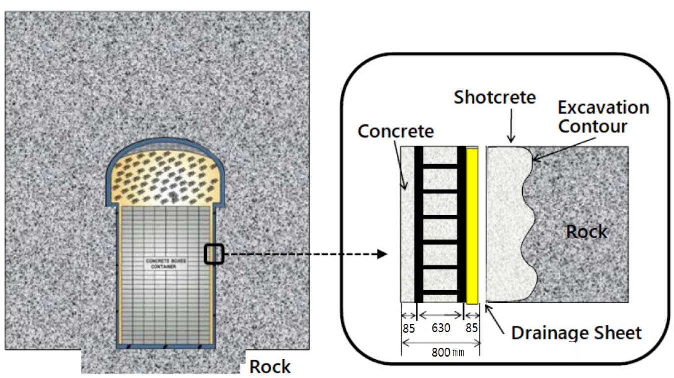


Figure 4.169 Schematic view of detailed silo lining structure

Table 4.10 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

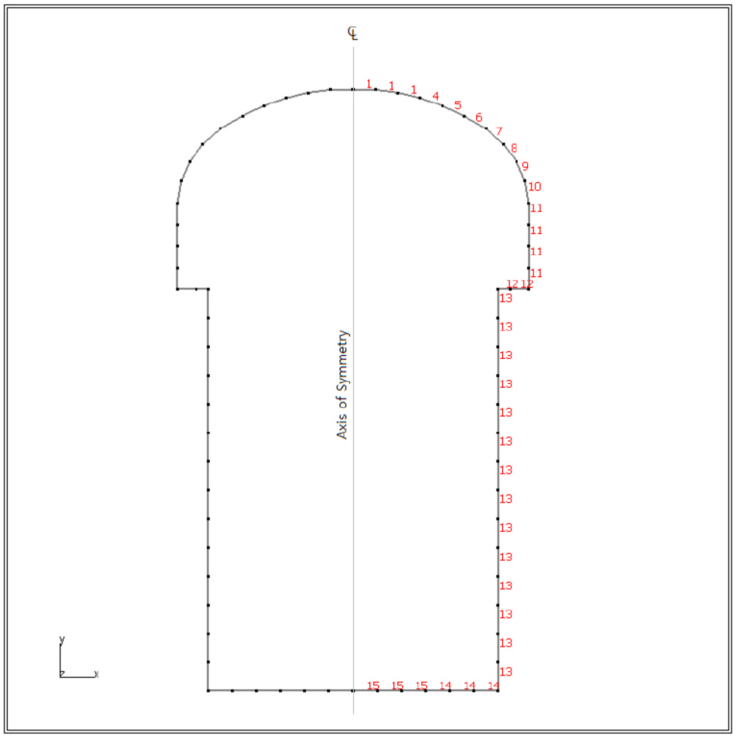
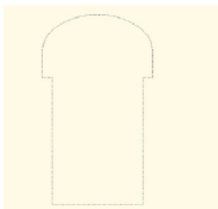
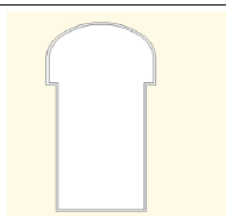
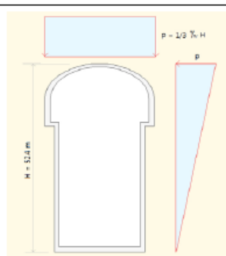


Figure 4.170 Silo lining material numbers

Table 4.11 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

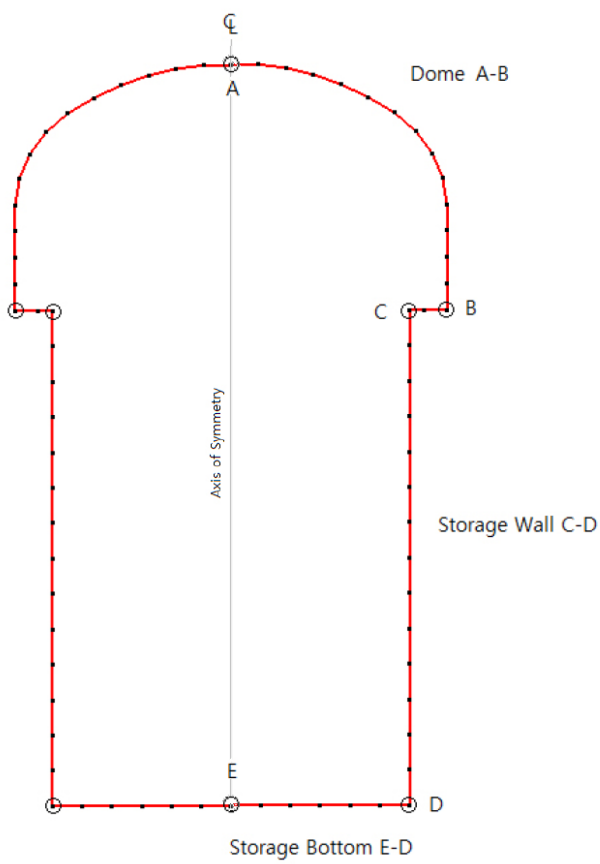


Figure 4.171 Key locations along silo lining

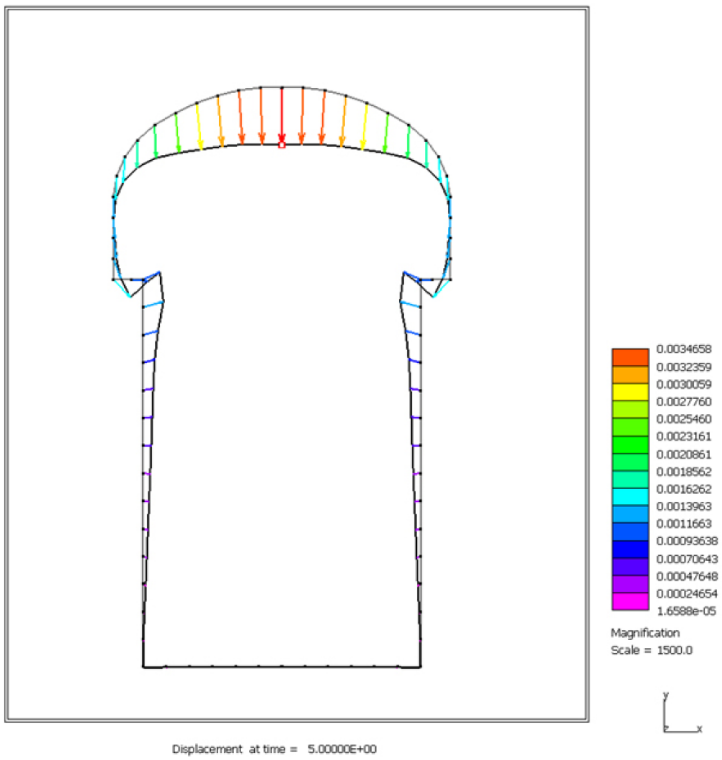


Figure 4.172 Deformed shape of silo lining

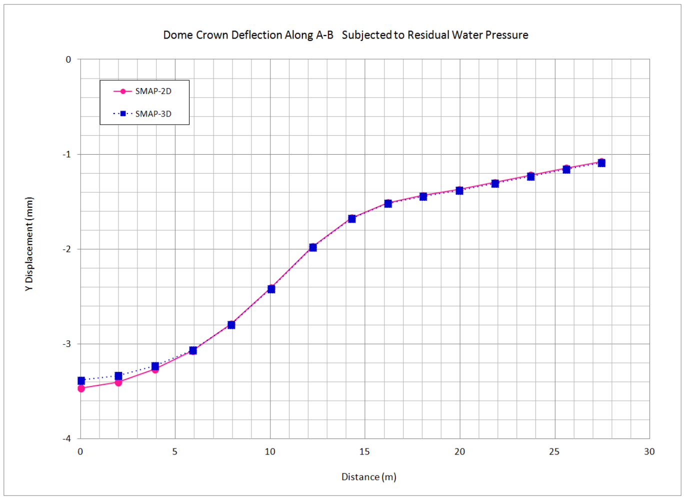


Figure 4.173 Dome deflection along A-B

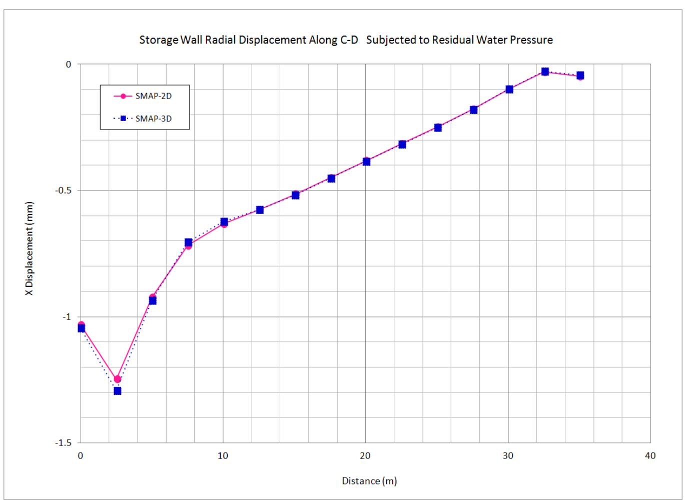


Figure 4.174 Storage wall radial displacement along C-D

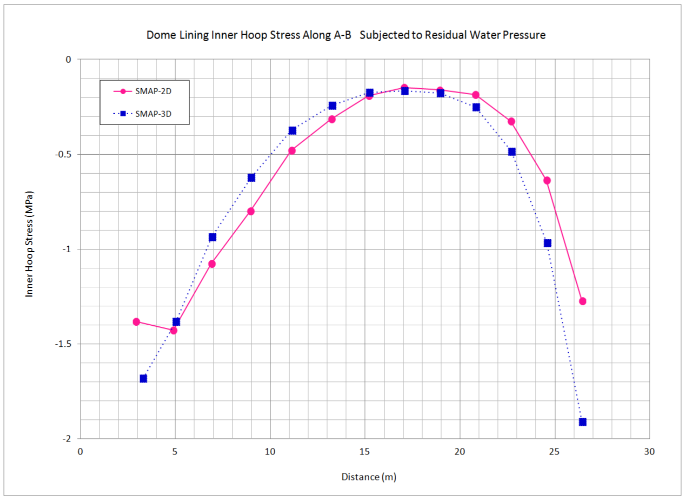


Figure 4.175 Dome lining inner hoop stress along A-B

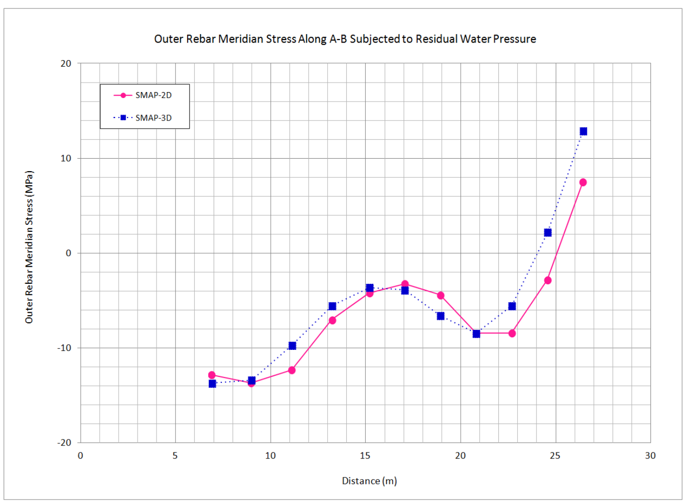


Figure 4.176 Dome outer rebar meridian stress along A-B

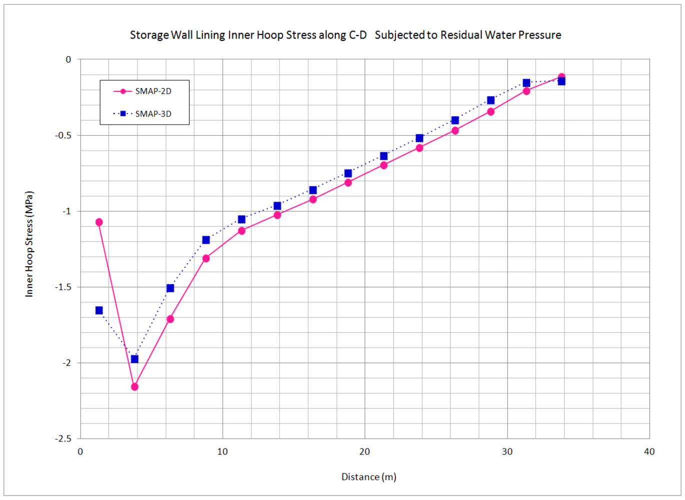


Figure 4.177 Storage wall lining inner hoop stress along C-D

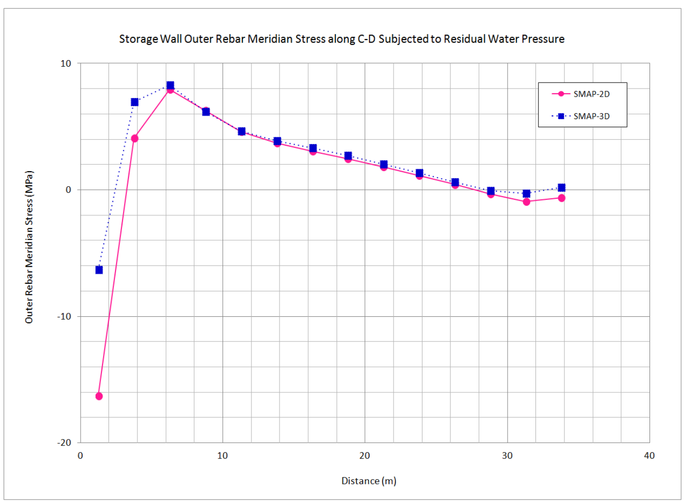


Figure 4.178 Storage wall outer rebar meridian stress along C-D

4.34 Liquefaction Analysis with PM4Sand

The main objective of this example is to verify PM4Sand model implemented in SMAP-2D 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 4.179 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 4.180 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 4.181 and 4.182, respectively.

Figures 4.183 and 4.184 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 4.185 Maximum acceleration profile (PGA)
Figure 4.186 Maximum displacement profile
Figure 4.187 Maximum shear strain profile
Figure 4.188 Maximum r_u profile
r_u = Excess Pore Pressure / Initial Effective Ver. Stress

Overall, PM4Sand in SMAP-2D is performing very well in predicting the stress-strain responses compared to the calibrated FLAC results.

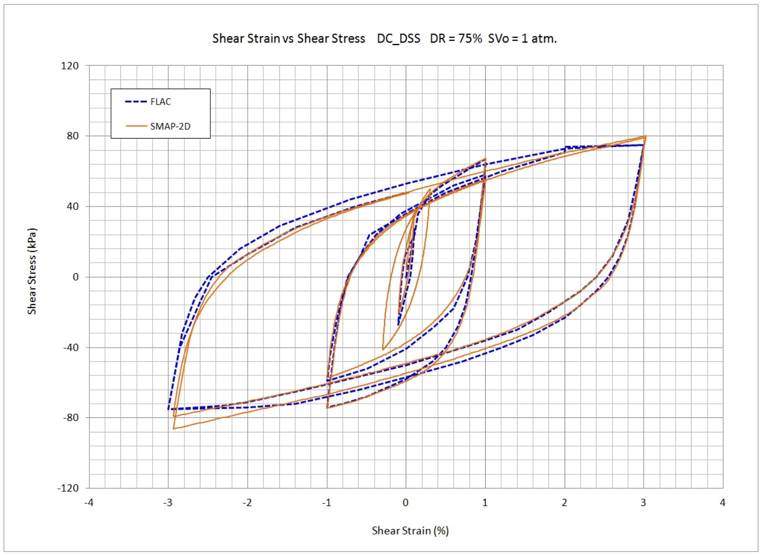
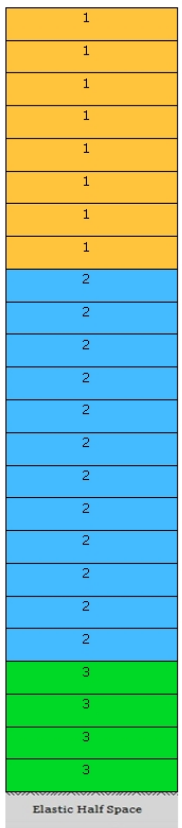


Figure 4.179 Isotropic consolidated drained cyclic direct simple shear test



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	

Figure 4.180 Finite element meshes and material properties

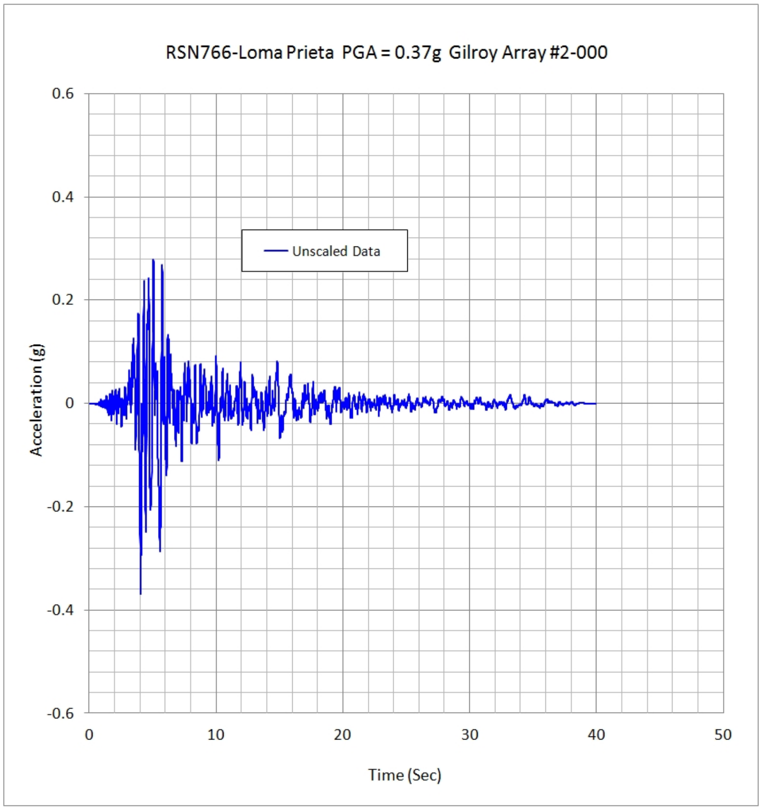


Figure 4.181 Loma Prieta (RSN766) acceleration time history

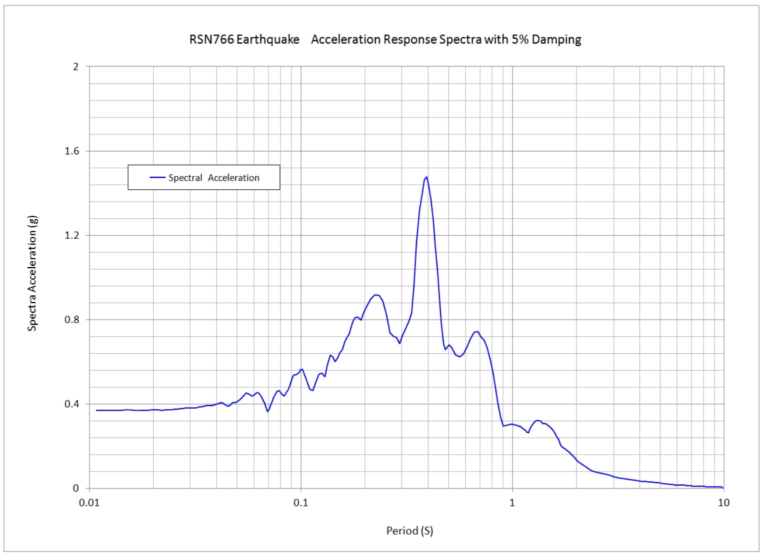


Figure 4.182 Spectral acceleration for input earthquake (RSN766)

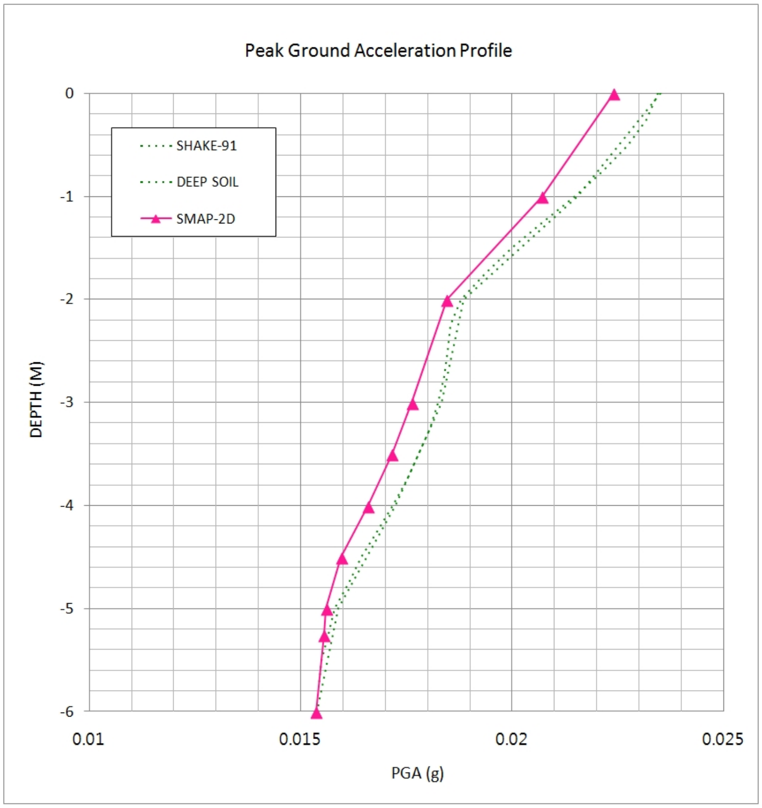


Figure 4.183 Peak ground acceleration profile, Elastic analysis

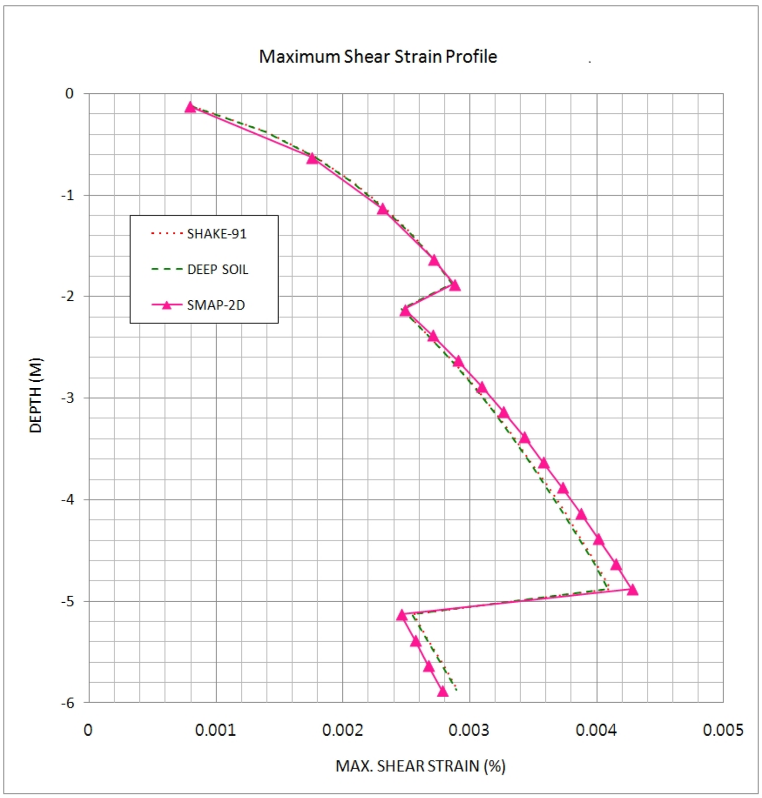


Figure 4.184 Maximum shear strain profile, Elastic analysis

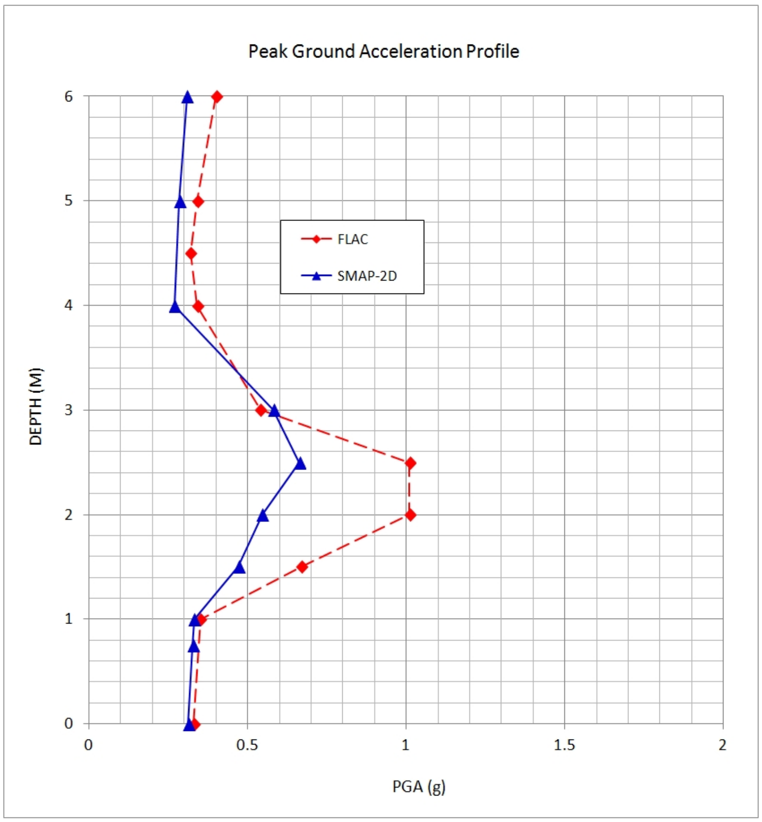


Figure 4.185 Maximum acceleration profile, Liquefaction analysis

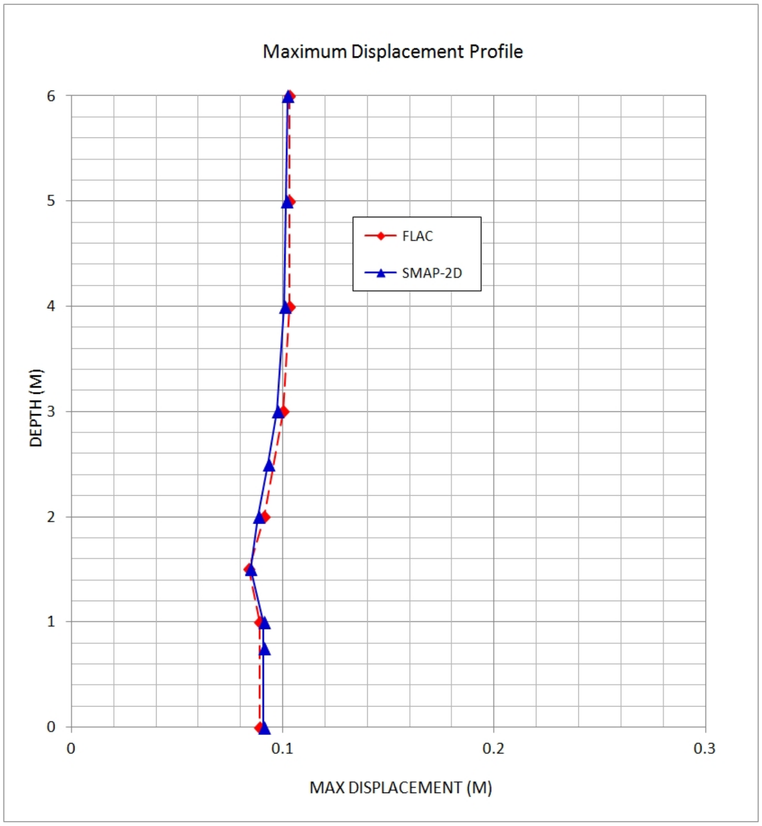


Figure 4.186 Maximum displacement profile, Liquefaction analysis

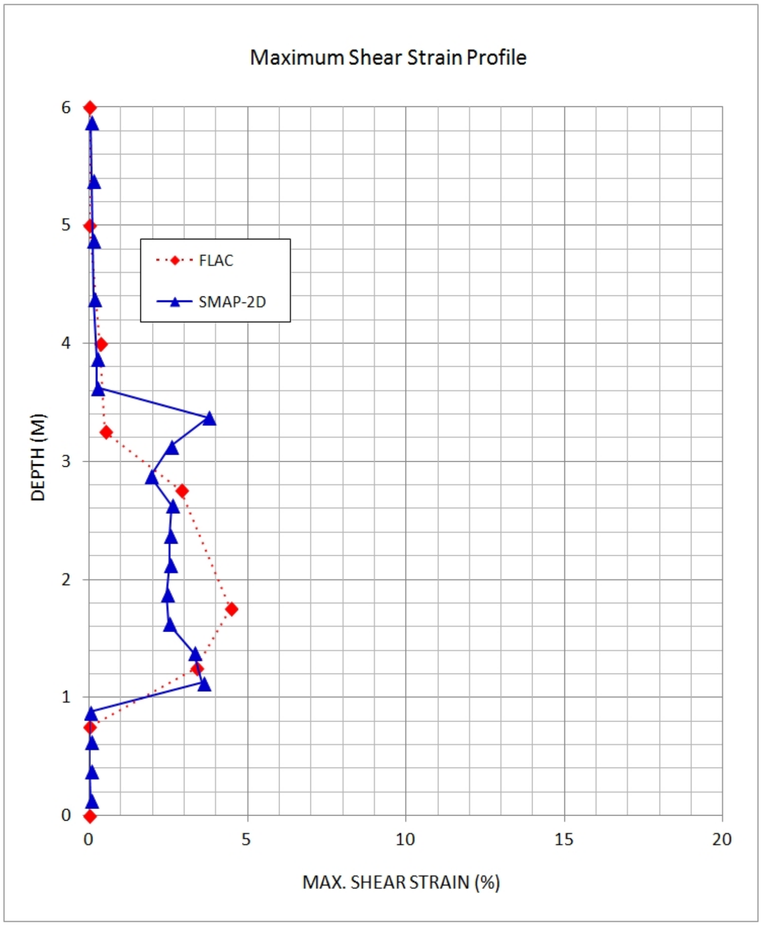


Figure 4.187 Maximum shear strain profile. Liquefaction analysis

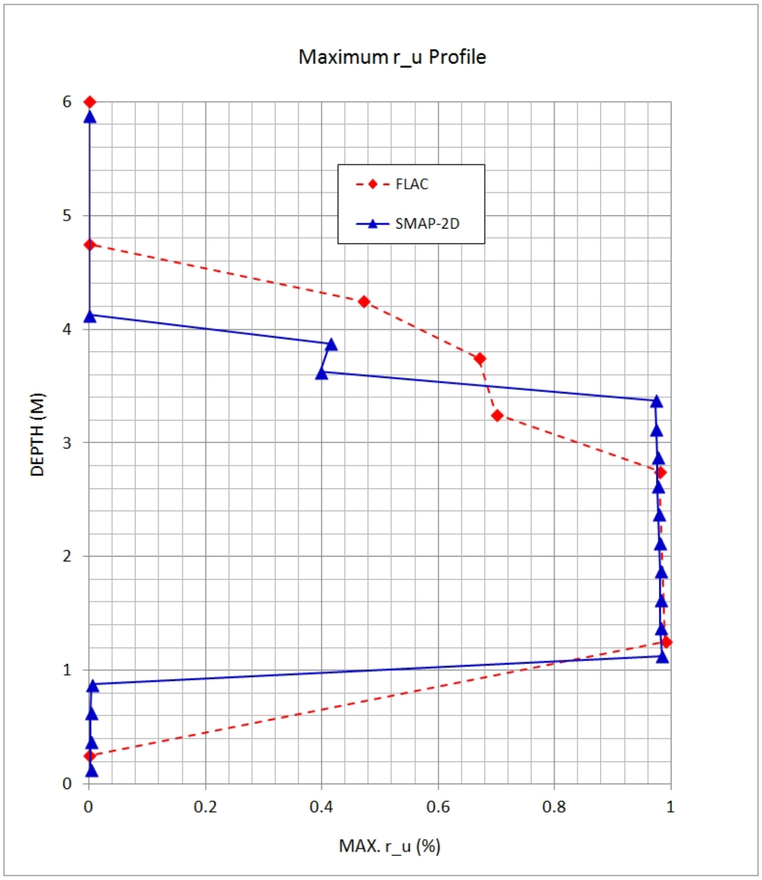


Figure 4.188 Maximum r_u profile, Liquefaction analysis

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.

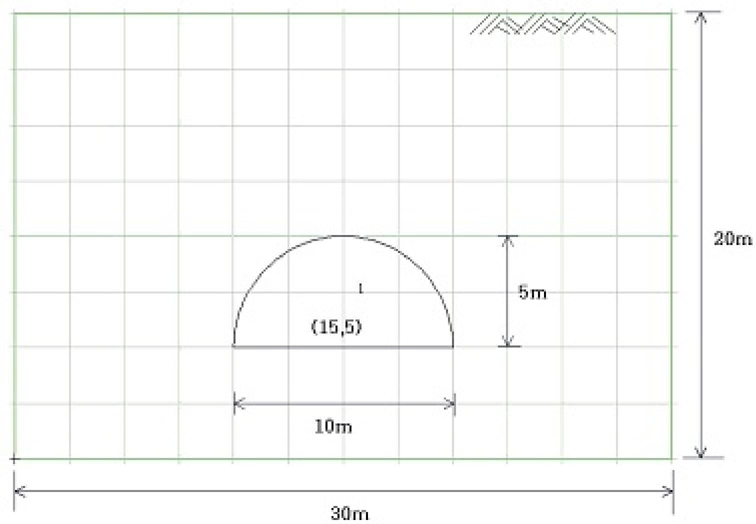


Figure 5.1 Arch tunnel (Part 1)

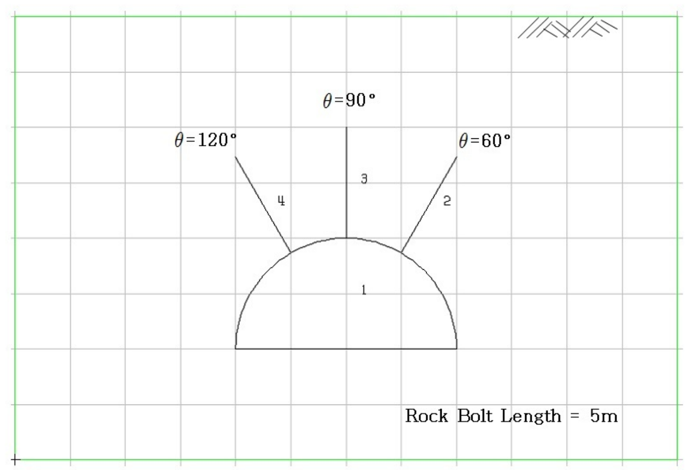


Figure 5.2 Arch tunnel with rock bolts (Part 2)

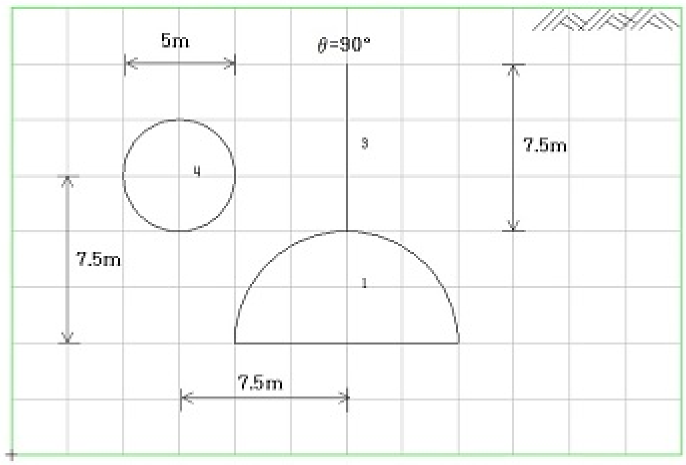


Figure 5.3 Arch tunnel with utility tunnel (Part 3)

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

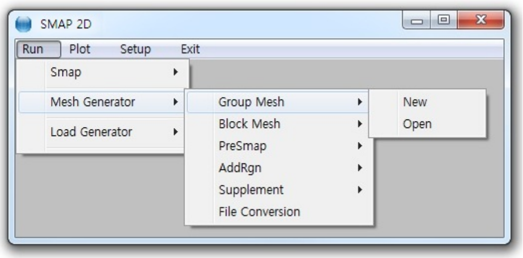


Figure 5.4 Accessing group mesh generator (New)

Step 2: Group Input (New)

Select Built-in Base Mesh in Figure 5.5.

Click OK.

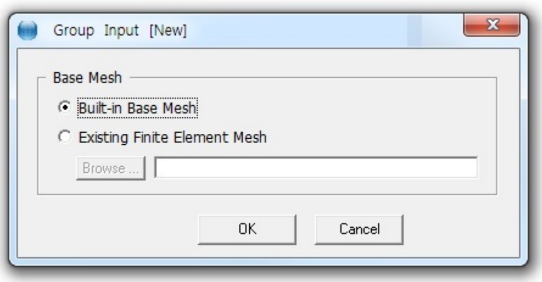


Figure 5.5 Group input (New)

Step 3: Group Menu and Dialog

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

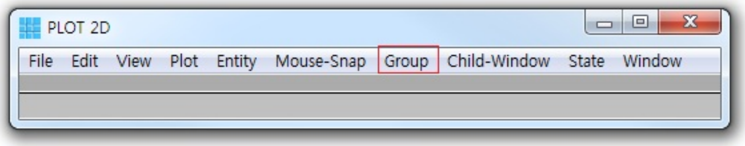


Figure 5.6 Group menu

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

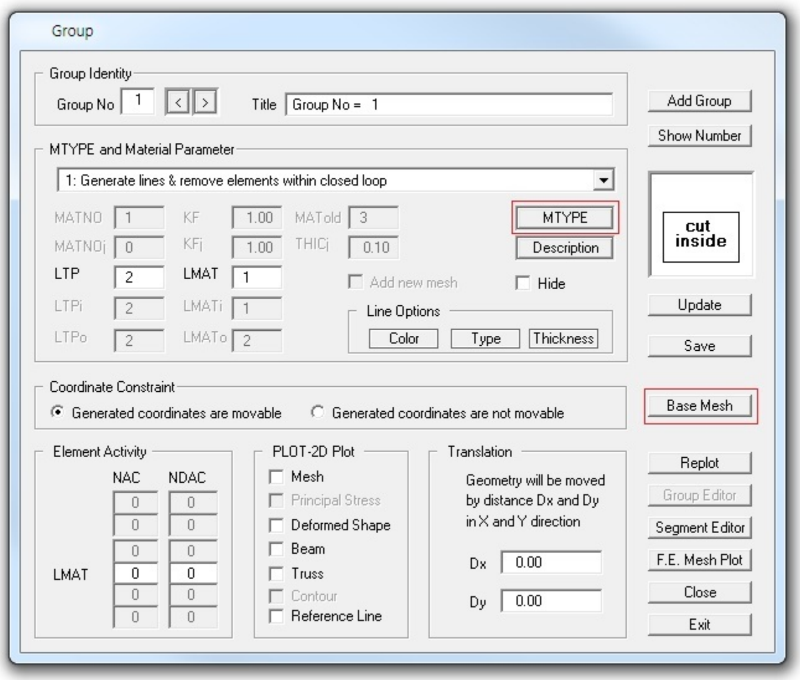


Figure 5.7 Group dialog with initial default values

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.

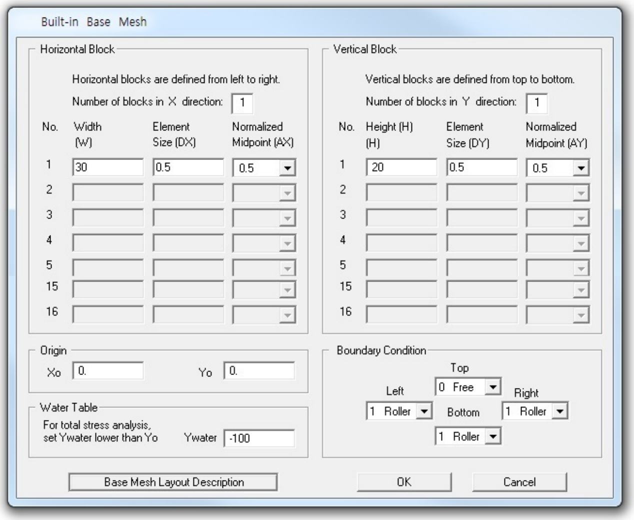


Figure 5.8 Built-in base mesh dialog

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

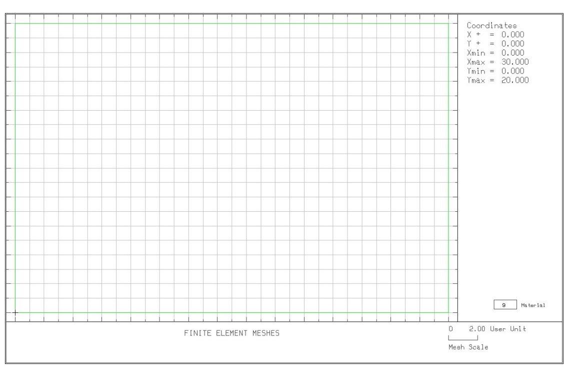


Figure 5.9 Base mesh on drawing board

Step 5: MTYPE

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

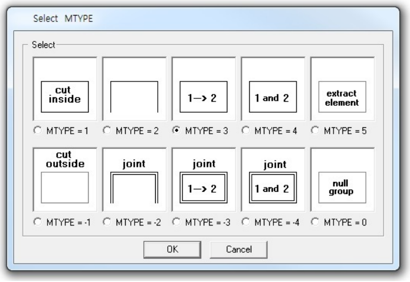


Figure 5.10 MTYPE dialog

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

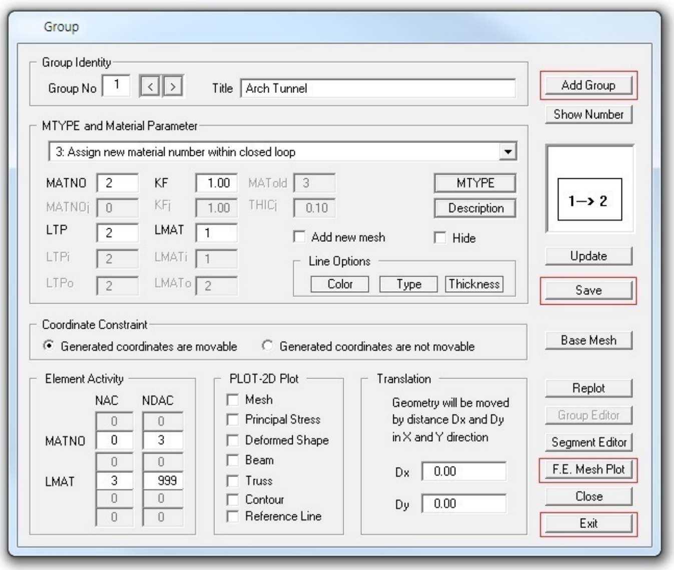
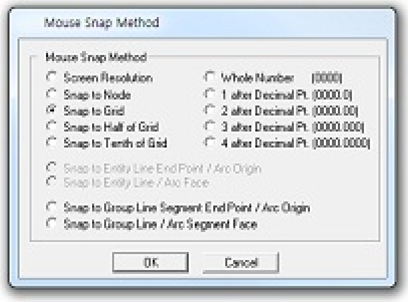


Figure 5.11 Group dialog with MTYPE = 3

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.

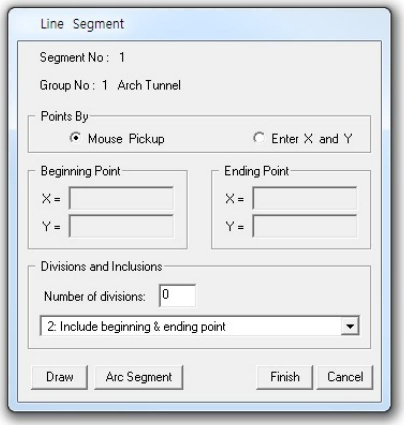


Figure 5.13 Line segment dialog

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

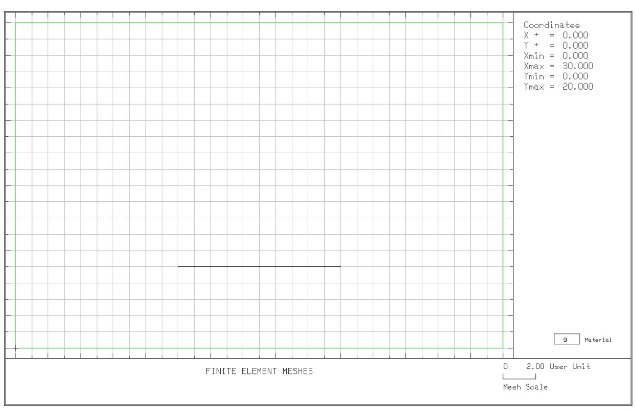


Figure 5.14 Line segment on drawing board

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.

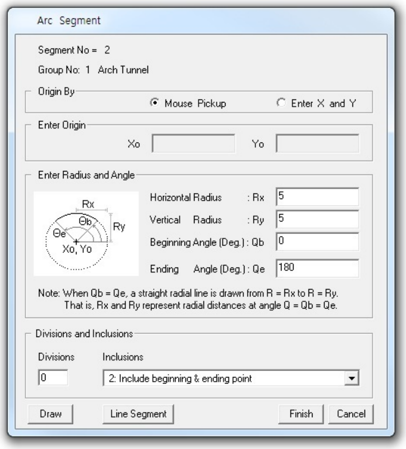


Figure 5.15 Arc segment dialog

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.

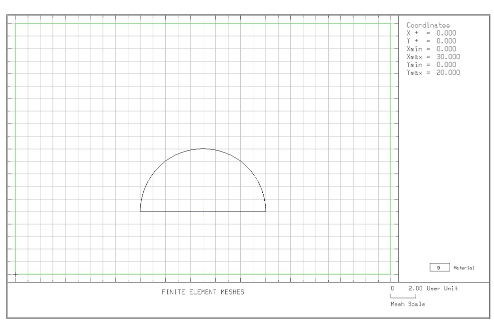


Figure 5.16 Arc segment on drawing board

Step 10: Save

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

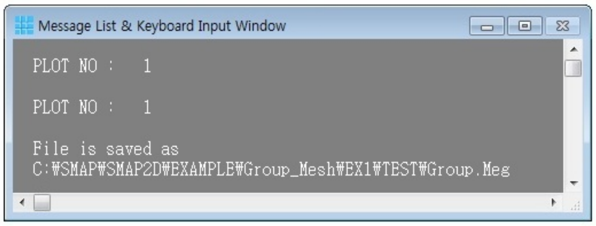


Figure 5.17 Message for file save

Step 11: Finite Element Mesh

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

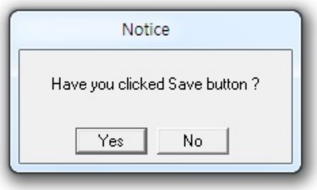


Figure 5.18 Notice for finite element mesh plot

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

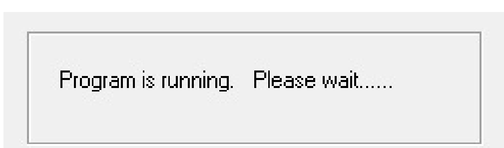


Figure 5.19 Notice while generating finite element mesh plot

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.

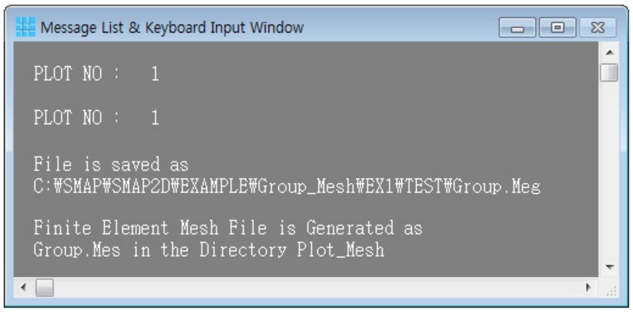


Figure 5.20 Message for finite element mesh file

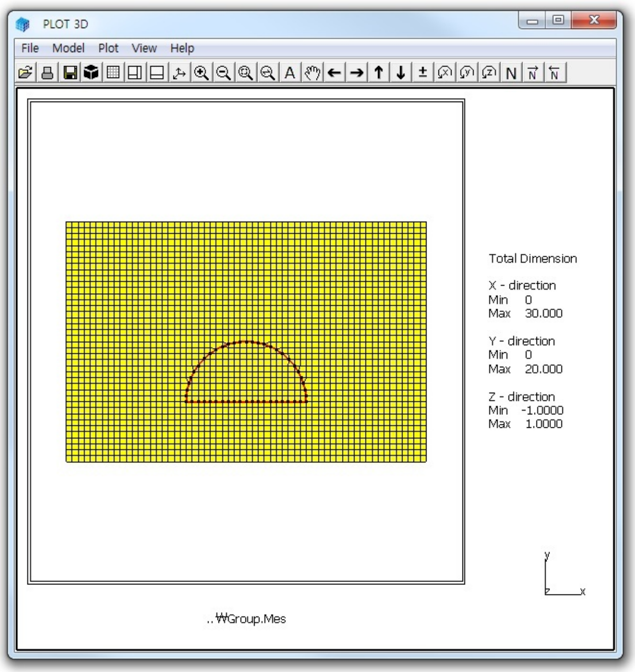


Figure 5.21 Finite element mesh plot

Step 12: Exit

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

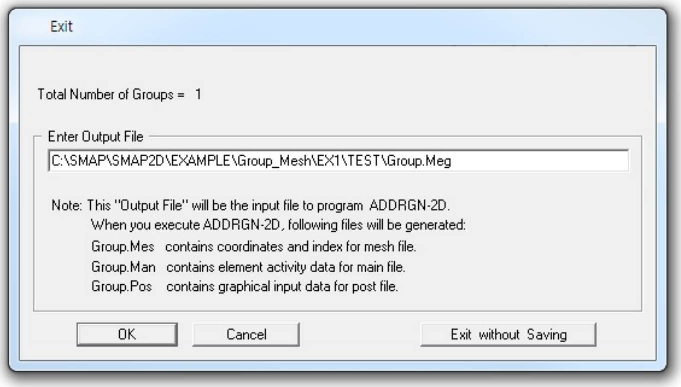


Figure 5.22 Exit dialog

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.

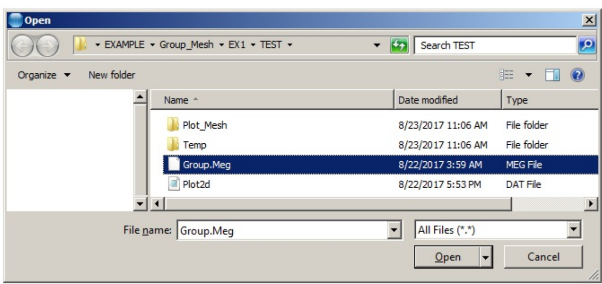


Figure 5.23 File open dialog

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.

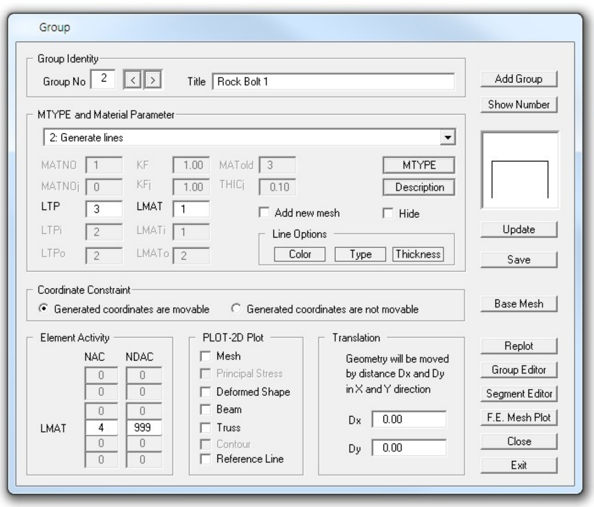


Figure 5.24 Group dialog for Rock Bolt 1

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.

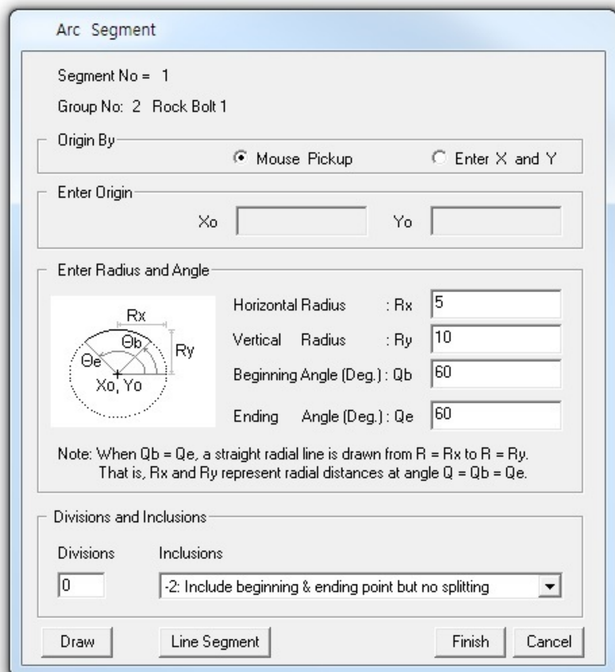


Figure 5.25 Arc segment dialog for Rock Bolt 1

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.

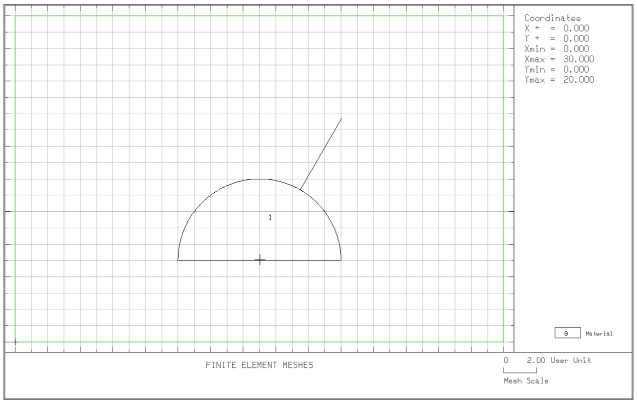


Figure 5.26 Rock Bolt 1 on drawing board

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.

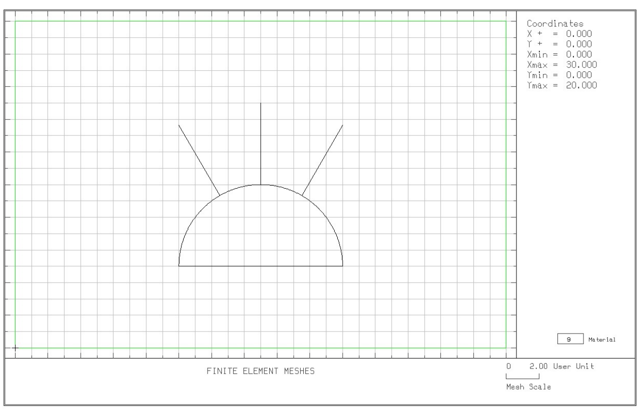


Figure 5.27 All three rock bolts on drawing board

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

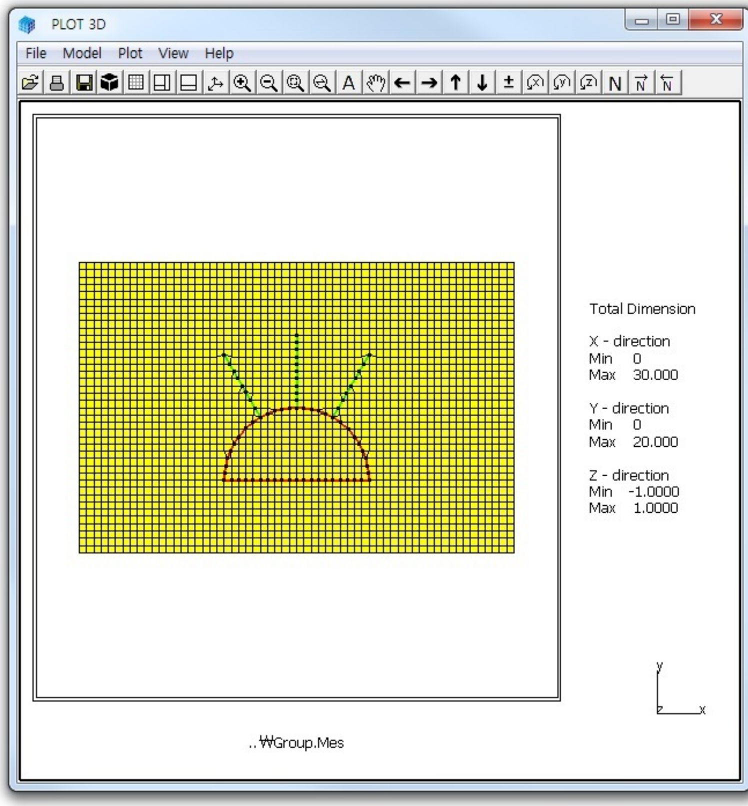


Figure 5.28 Finite element mesh plot

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

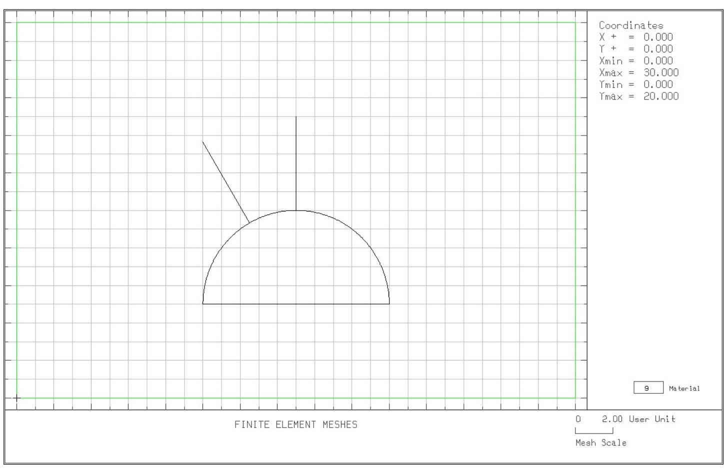


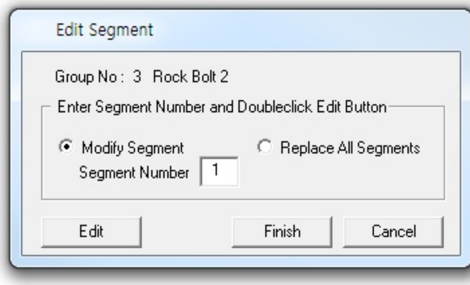
Figure 5.29 Rock Bolt 1 removed on drawing board

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

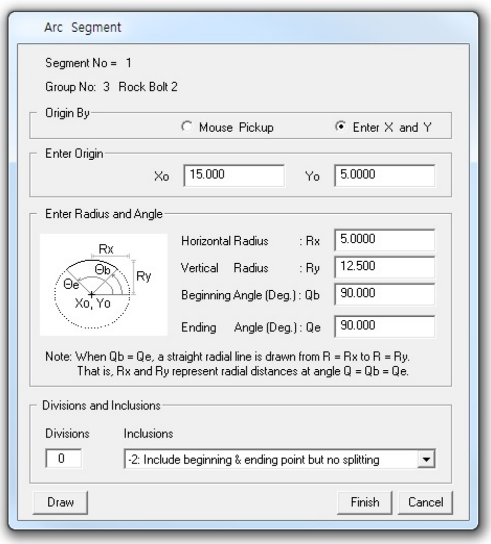


Figure 5.32 shows a new plot with longer Rock Bolt 2.

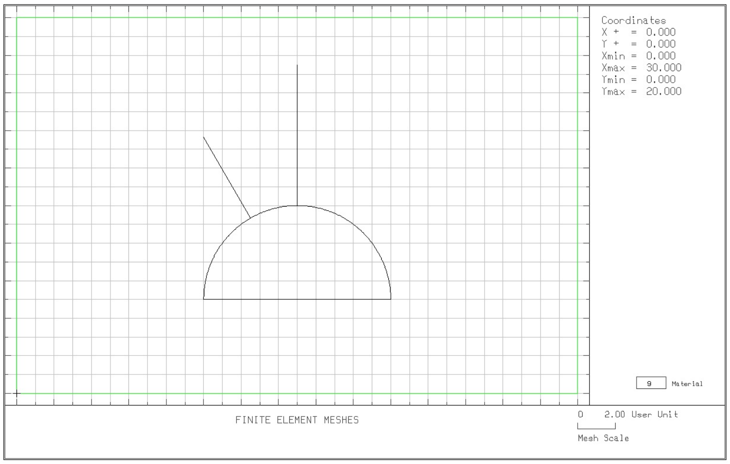


Figure 5.32 Longer Rock Bolt 2 on drawing board

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.

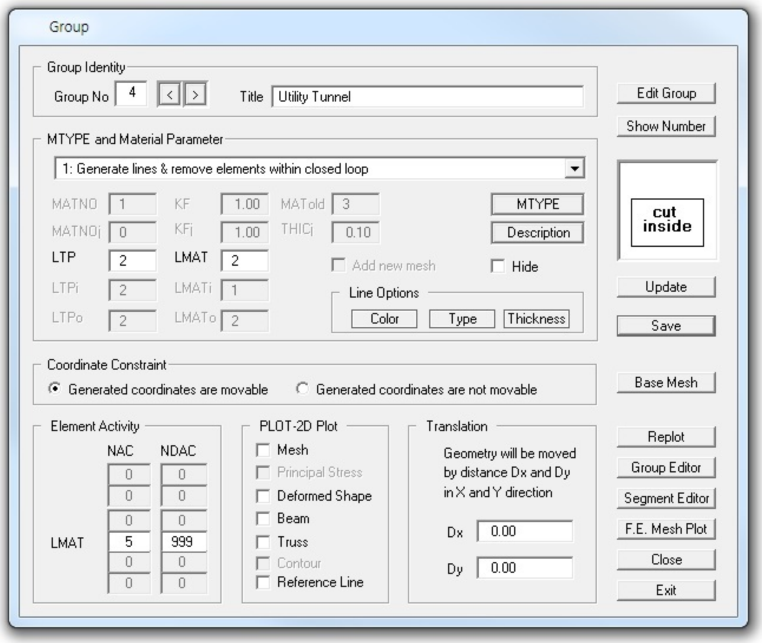


Figure 5.33 Group dialog for Utility Tunnel

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

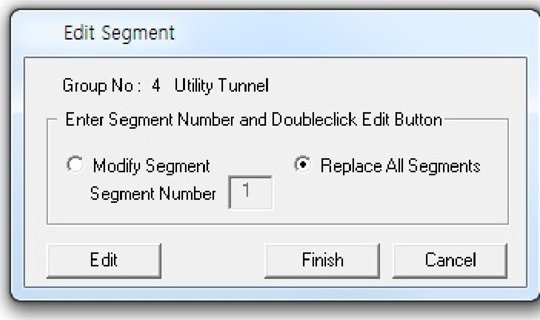


Figure 5.34 Edit segment dialog for Group No 4

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

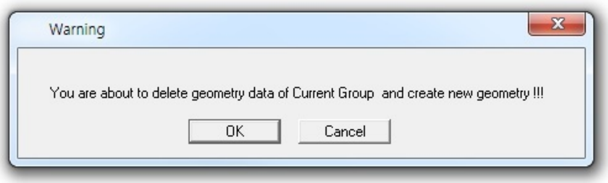


Figure 5.35 Warning message

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.

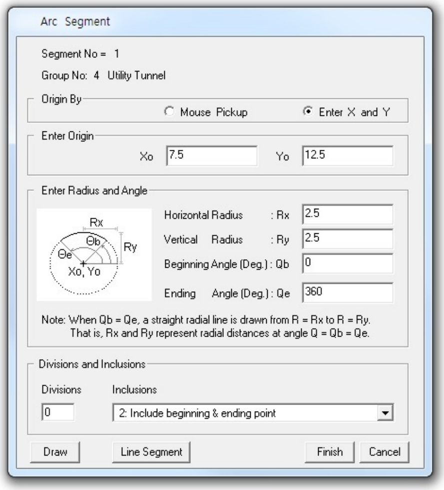


Figure 5.36 Arc segment dialog for Utility Tunnel

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

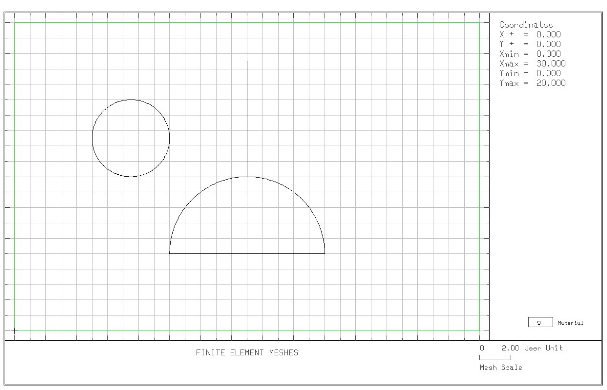


Figure 5.37 Arch and Utility Tunnels on drawing board

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

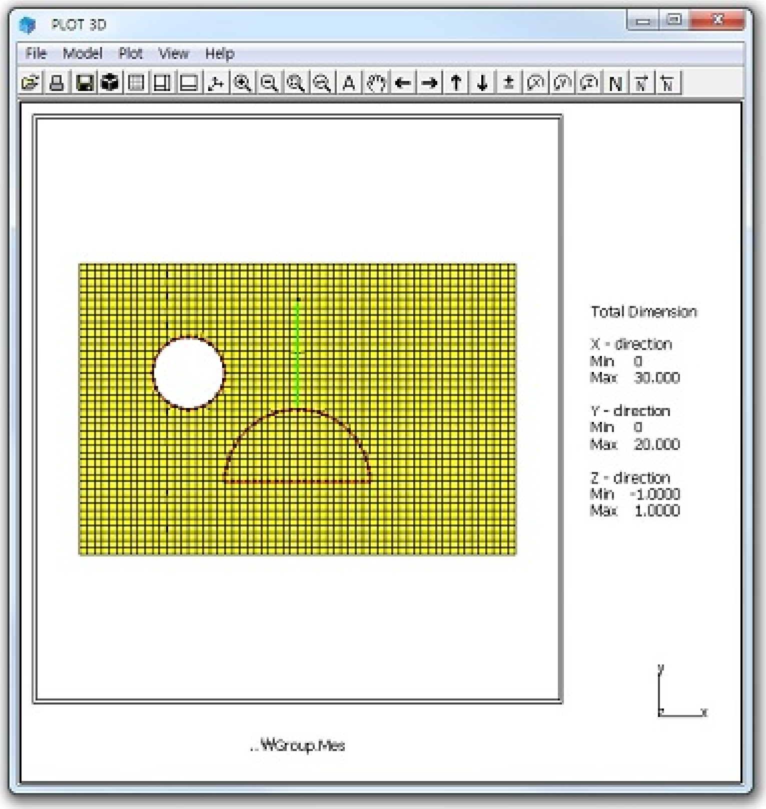


Figure 5.38 Finite element mesh plot

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.

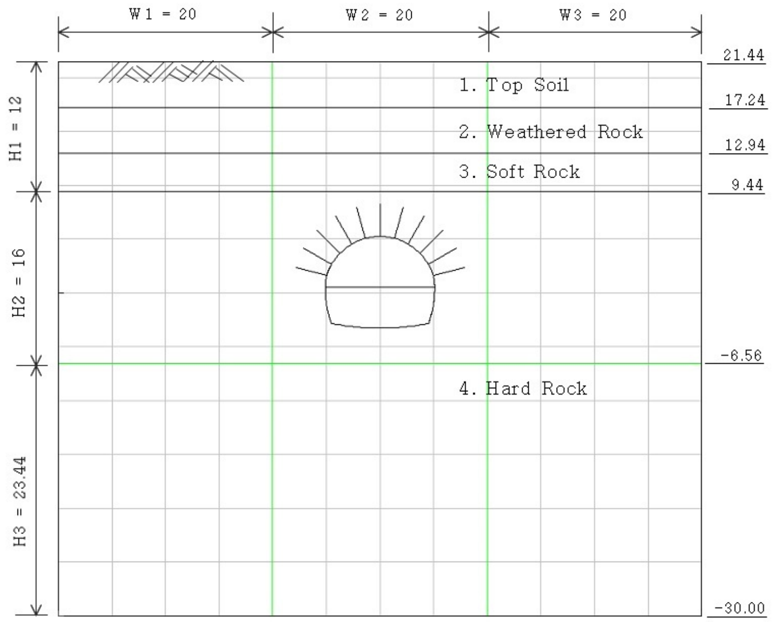


Figure 5.39 Tunnel cross section

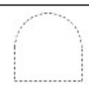
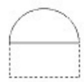
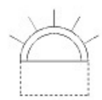
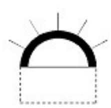
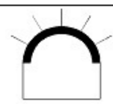
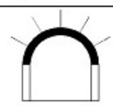
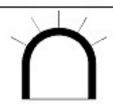
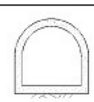
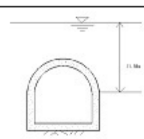
Step	Construction State	Description	
1, 2		In Situ Ko State	
3		50 % Stress Relief	Upper Core Excavation
4		75 % Stress Relief Soft Shotcrete, Rock Bolt	
5		100 % Stress Relief Hard Shotcrete, Rock Bolt	
6		50 % Stress Relief	Lower Core Excavation
7		75 % Stress Relief, Soft Shotcrete	
8		100 % Stress 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.

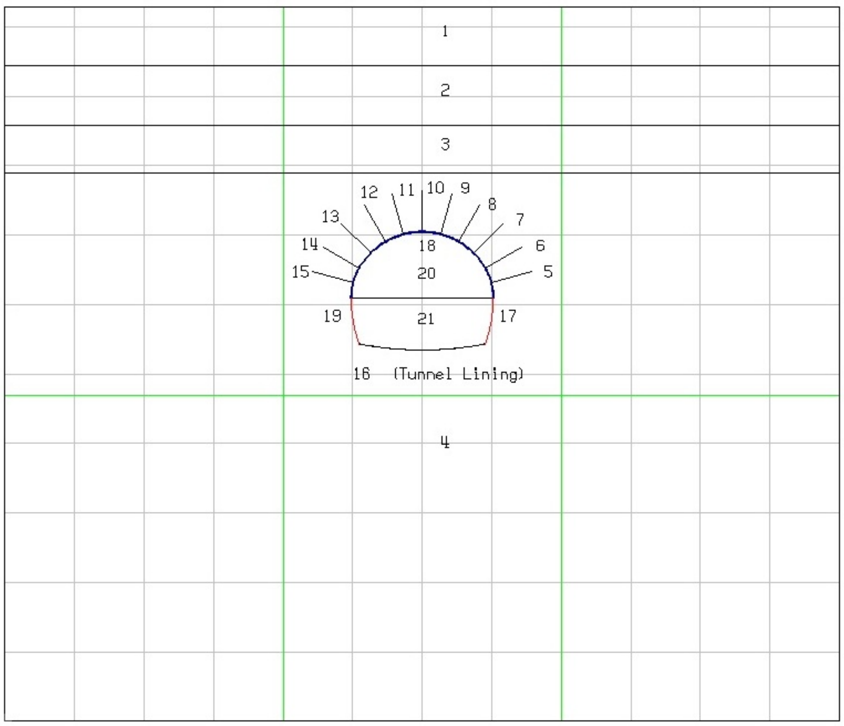


Figure 5.40 Group section view

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.

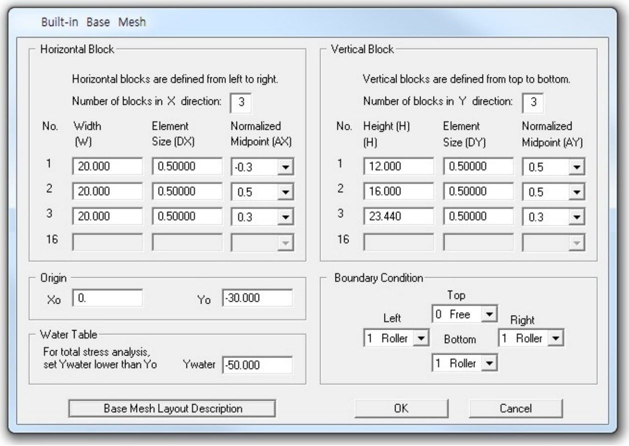


Figure 5.41 Built-in base mesh dialog

Figure 5.42 shows base mesh plot on drawing board.

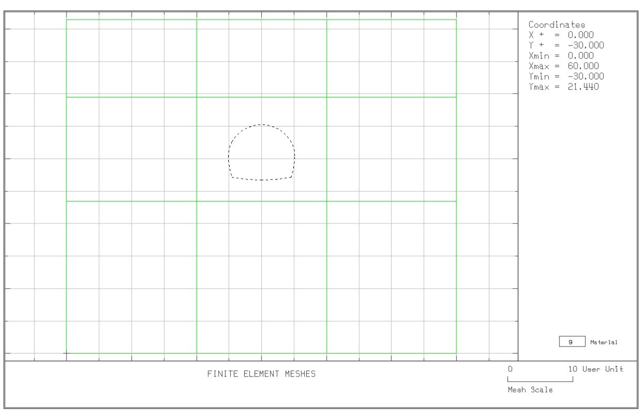


Figure 5.42 Base mesh plot on drawing board

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.

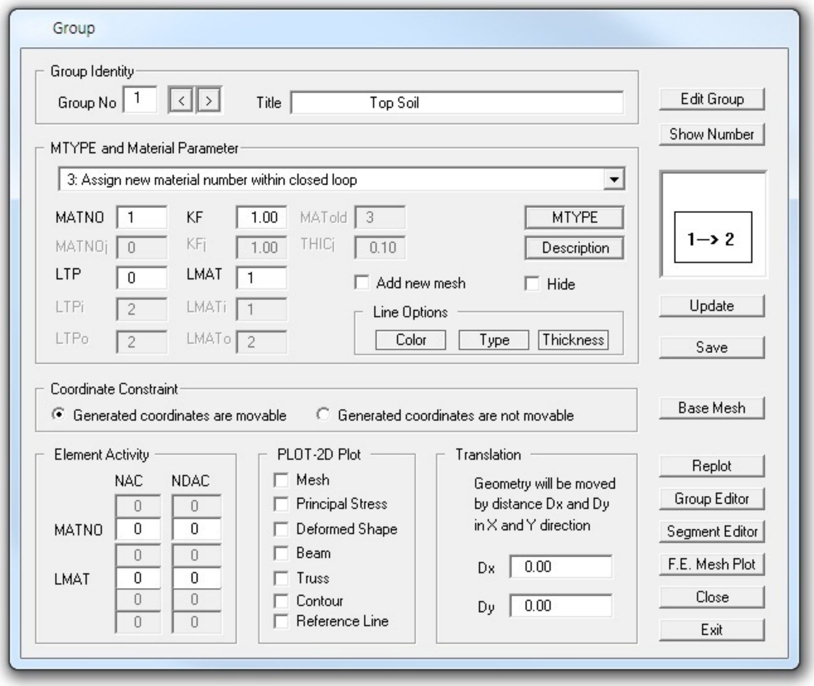


Figure 5.43 Group dialog for top soil layer

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.

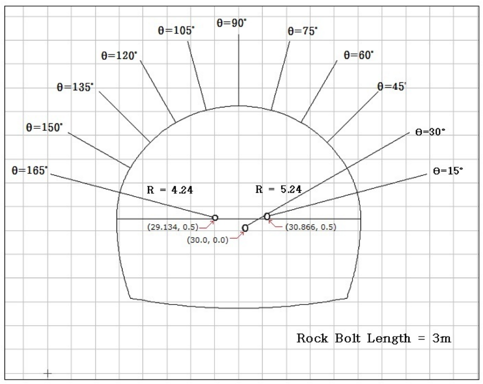


Figure 5.44 Rock bolt layout

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.

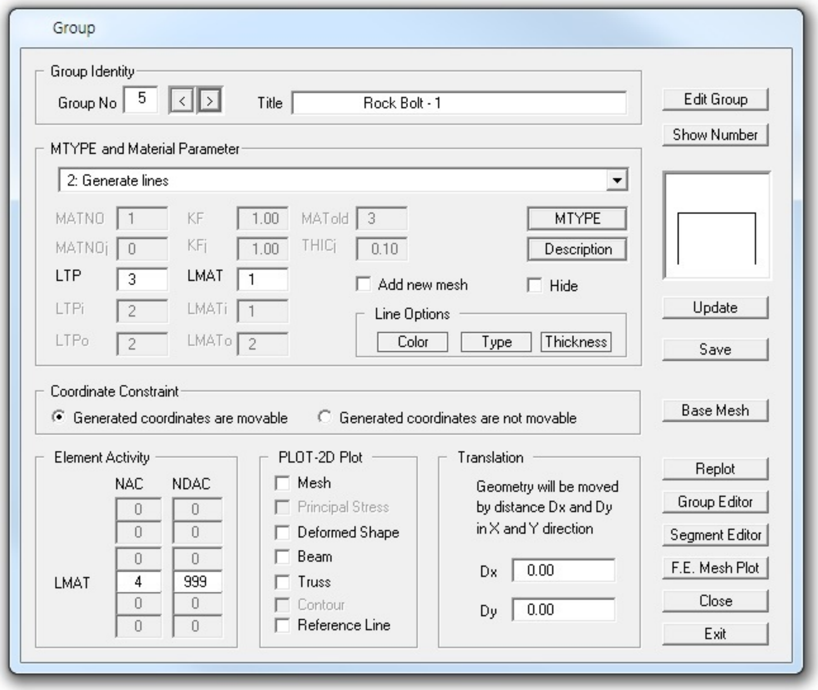


Figure 5.45 Group dialog for rock bolt at 15 degrees

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.

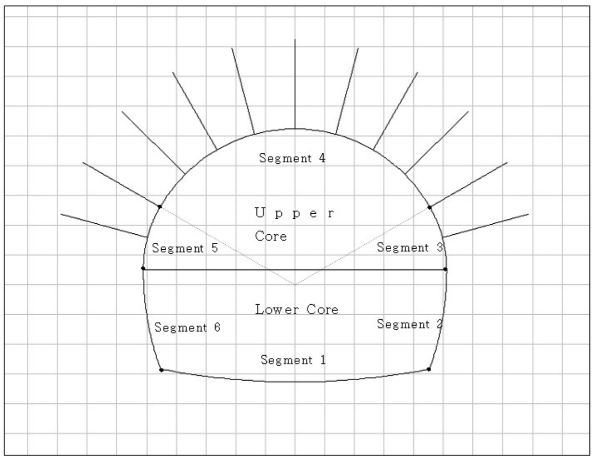


Figure 5.46 Lining segments

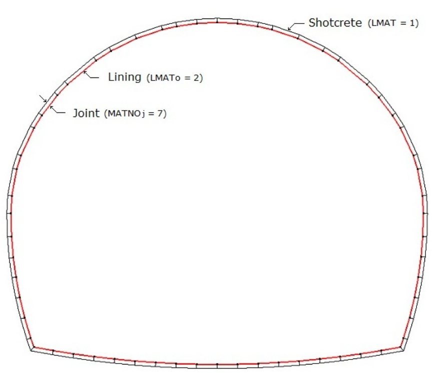


Figure 5.47 Interface joint element

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.

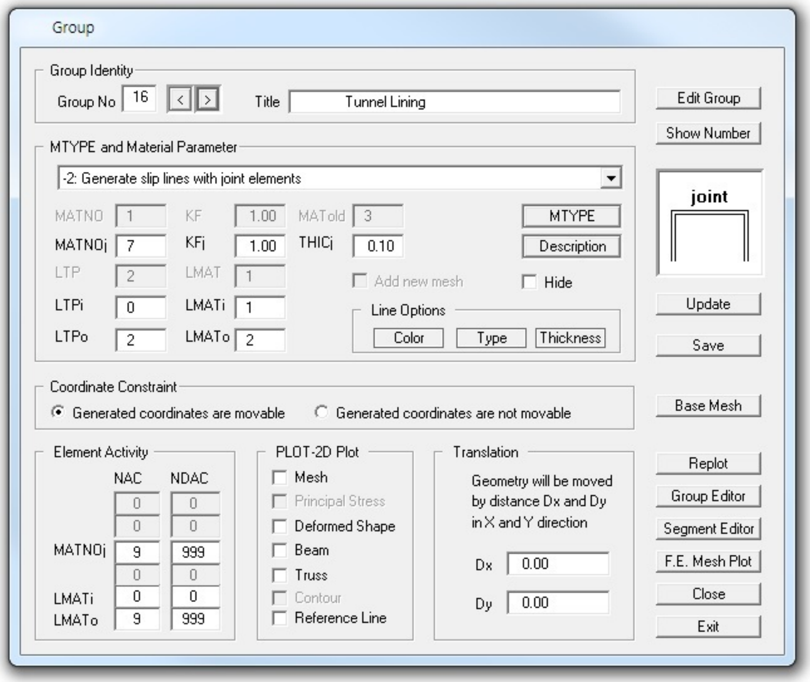
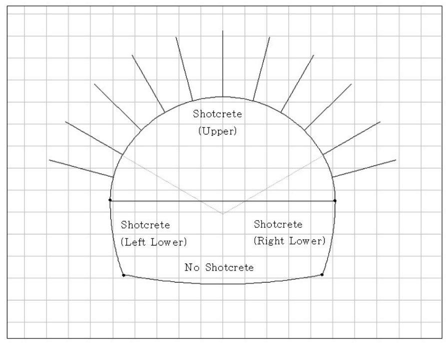


Figure 5.48 Group dialog for tunnel lining

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.

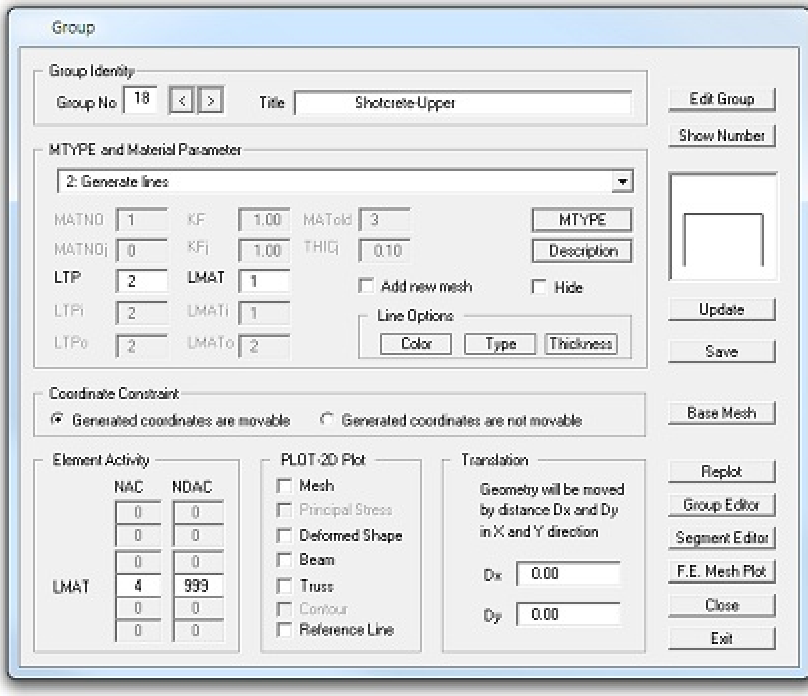


Figure 5.50 Group dialog for upper shotcrete

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.

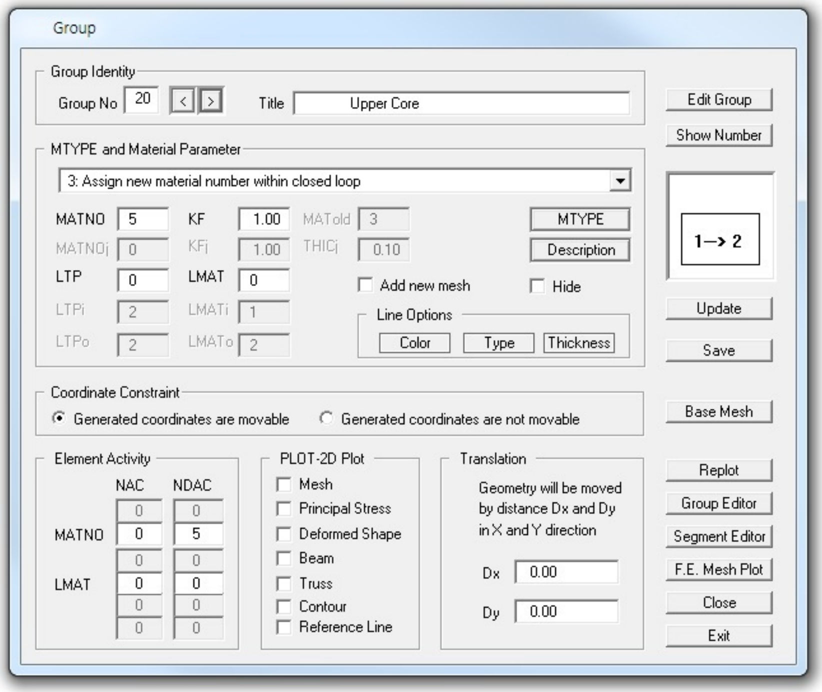


Figure 5.51 Group dialog for upper core

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.

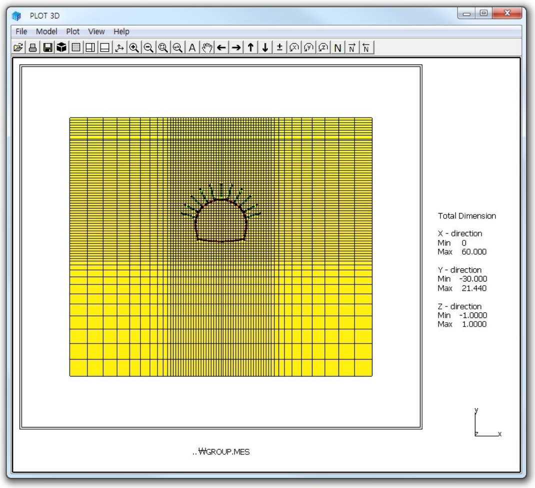


Figure 5.52 Finite element meshes for NATM tunnel

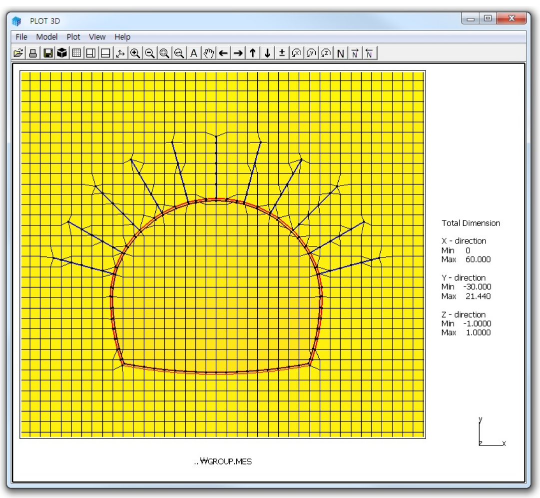


Figure 5.53 Finite element meshes around tunnel

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.

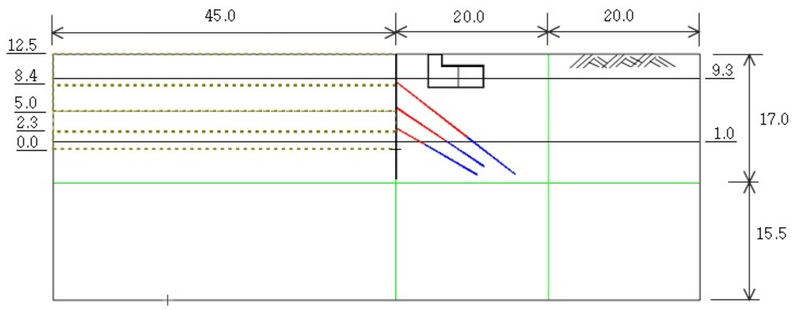


Figure 5.54 Schematic section of excavation problem

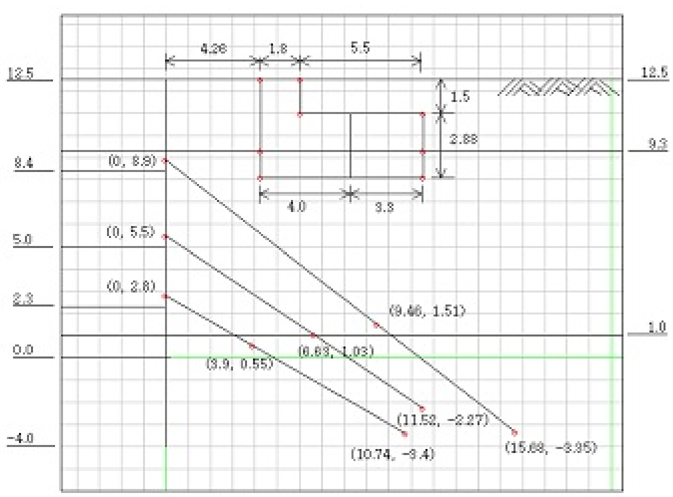


Figure 5.55 Cross section near box structure

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.

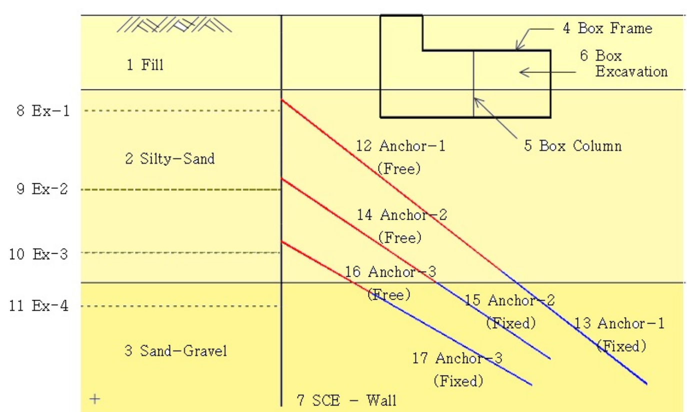


Figure 5.56 Group section view

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.

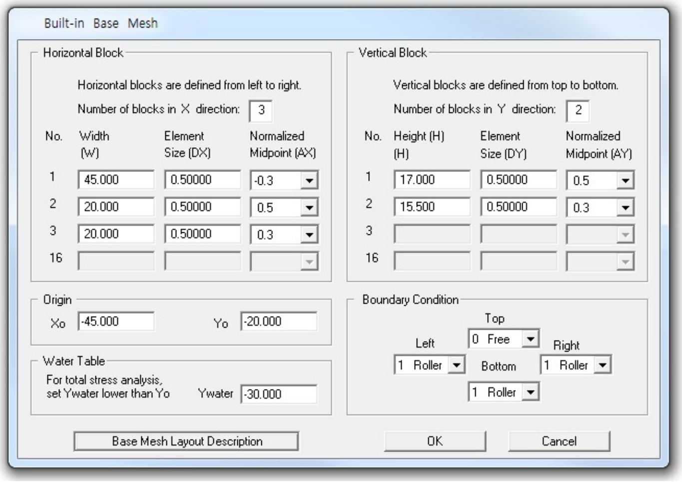


Figure 5.57 Built-in base mesh dialog

Figure 5.58 shows base mesh plot on drawing board.

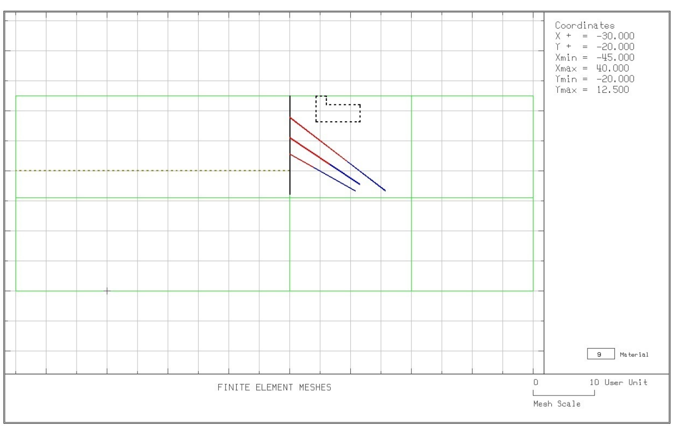


Figure 5.58 Base mesh plot on drawing board

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.

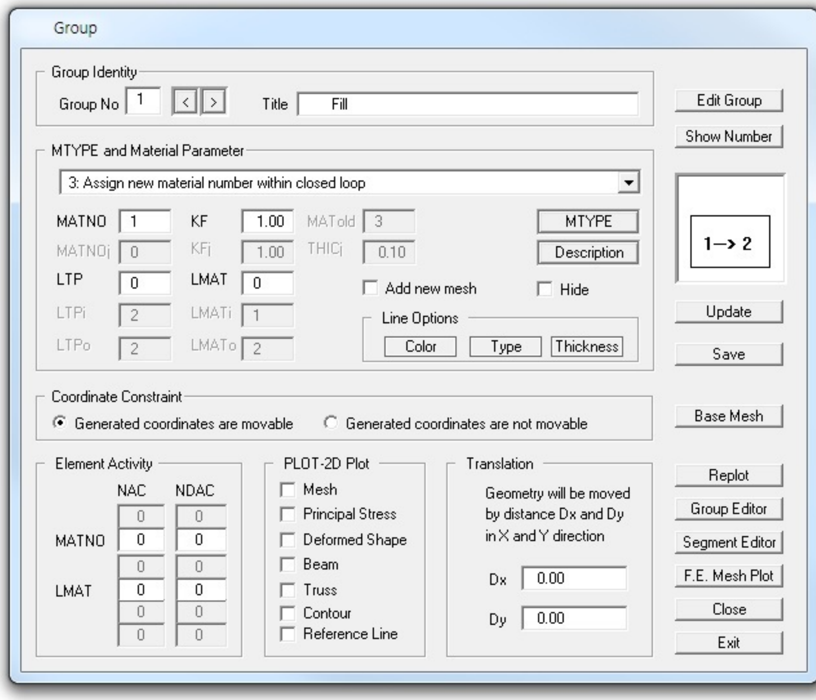


Figure 5.59 Group dialog for top fill

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.

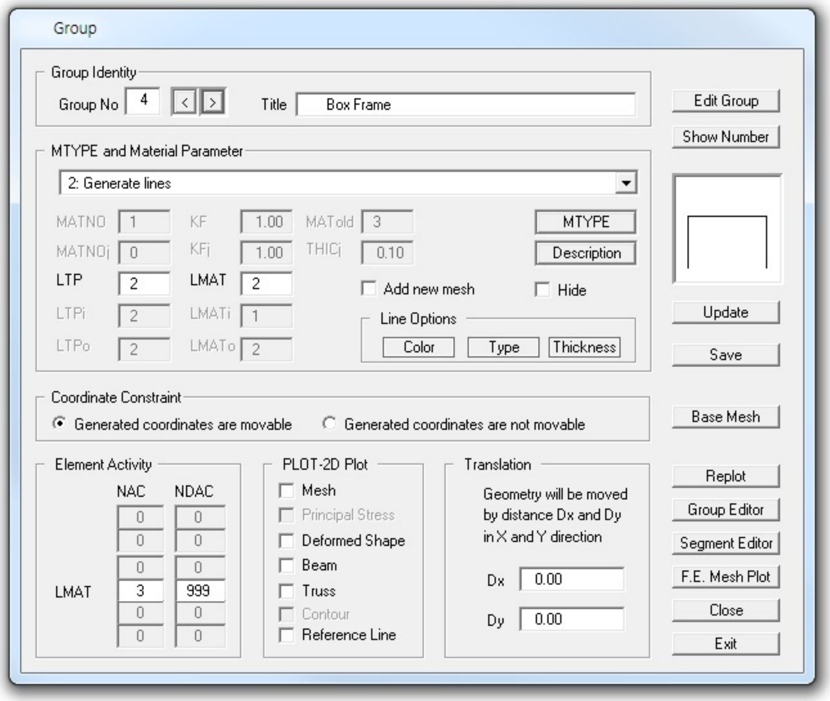


Figure 5.60 Group dialog for box frame

Figure 5.61 shows Group dialog for the box excavation.

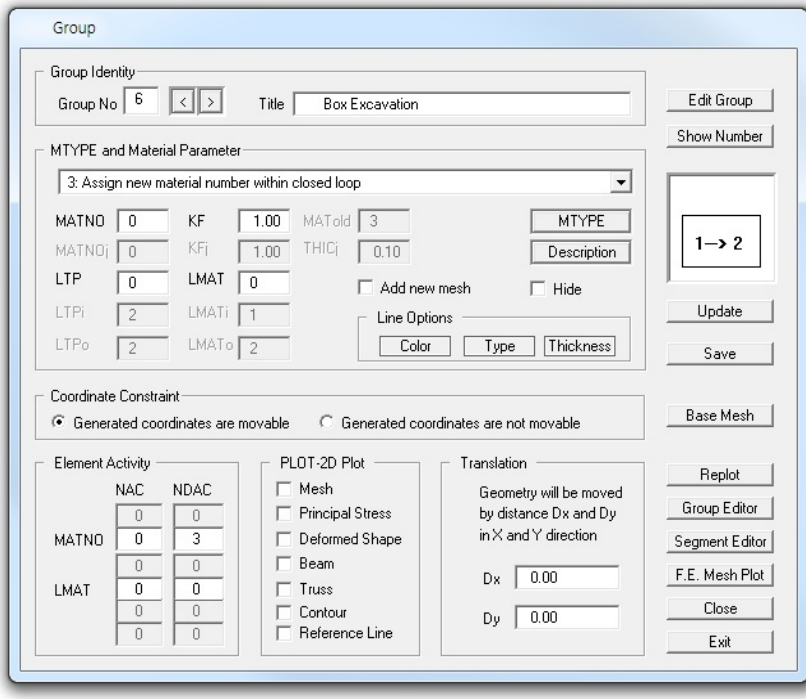


Figure 5.61 Group dialog for box excavation

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.

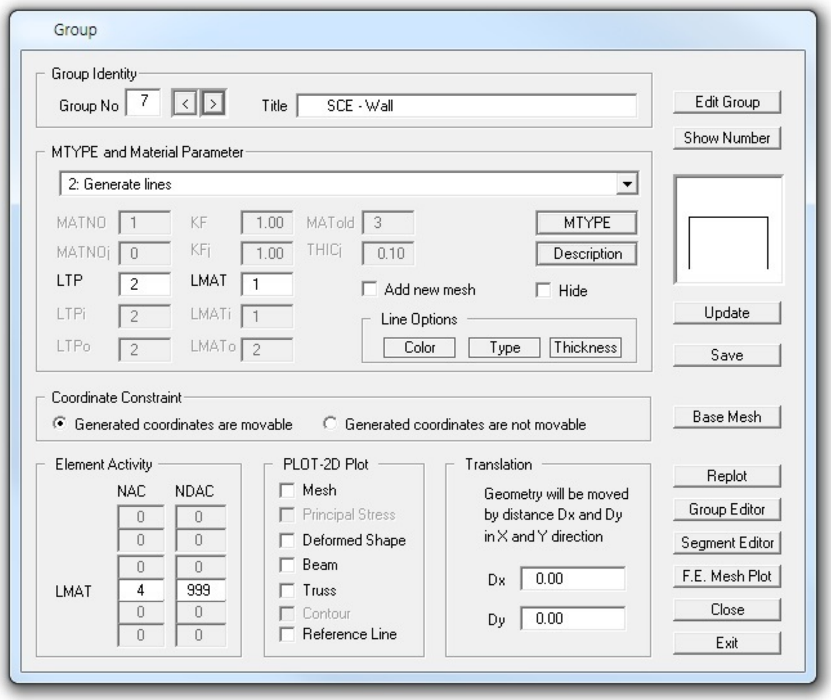


Figure 5.62 Group dialog for SCE-wall

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.

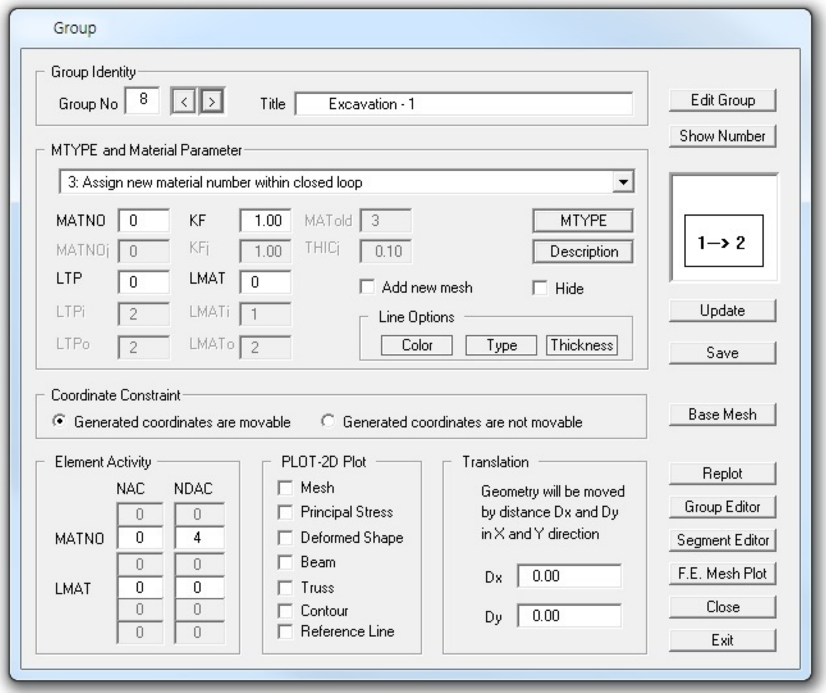


Figure 5.63 Group dialog for the first excavation

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.

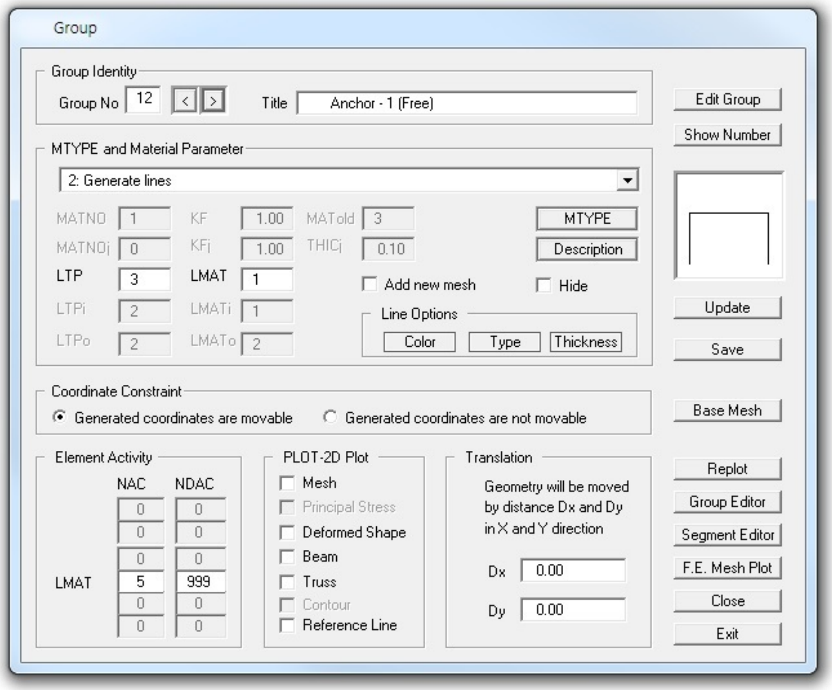


Figure 5.64 Group dialog for the first anchor (free part)

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

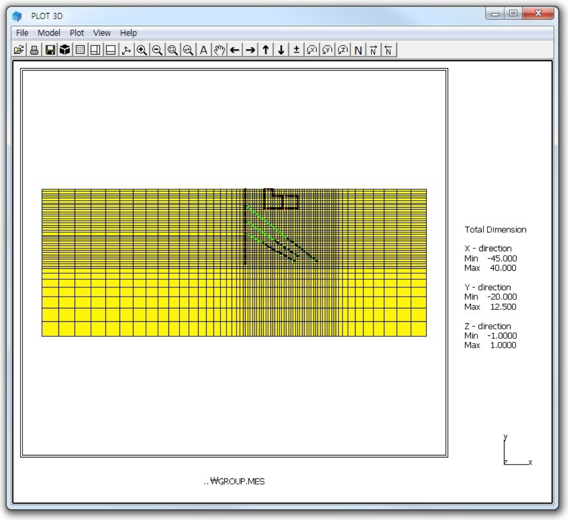
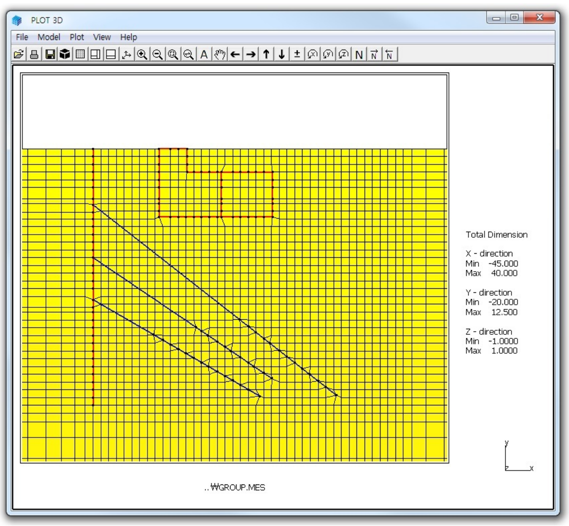


Figure 5.66
Finite element meshes
near box structure



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.

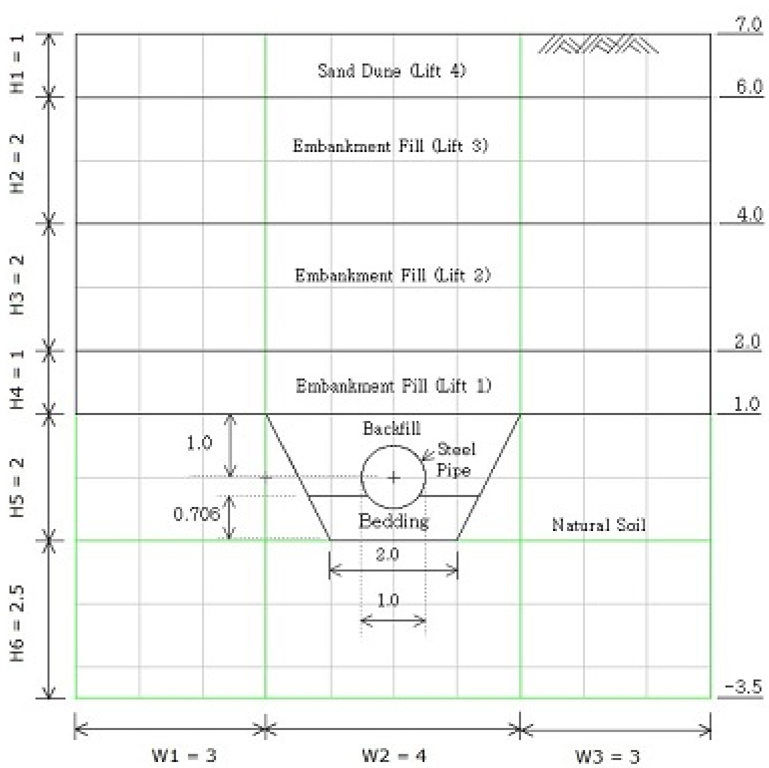


Figure 5.67 Schematic section of buried pipe

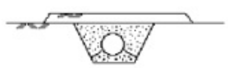
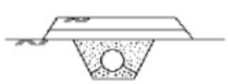
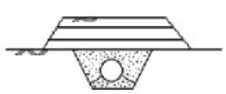
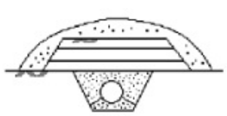
Step	Construction Sequence	Description	Element Activity
1,2		In situ K_o state	Active elements: Natural soil within trench
3		Excavate trench	Deactive elements: Natural soil within trench
4		Place bedding	Active elements: Compacted sand for bedding
5		Place steel pipe Fill the backfill	Active elements: 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.

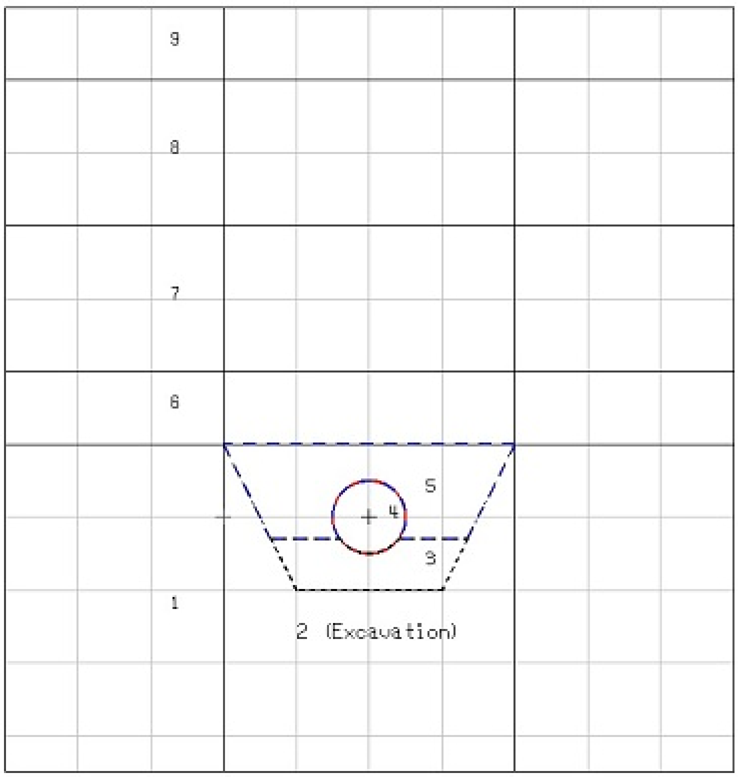


Figure 5.68 Group section view

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.

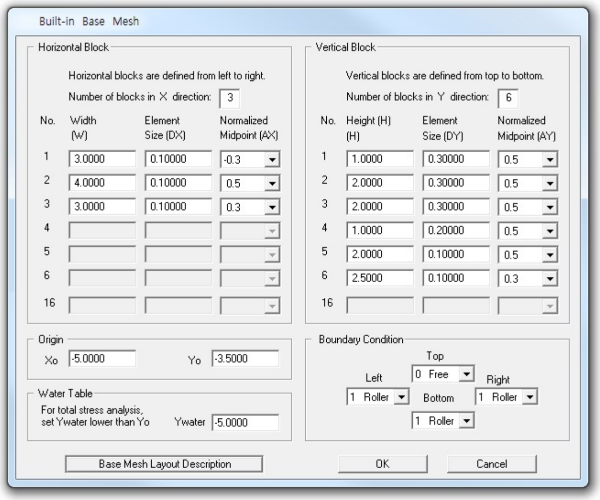


Figure 5.69 Built-in base mesh dialog

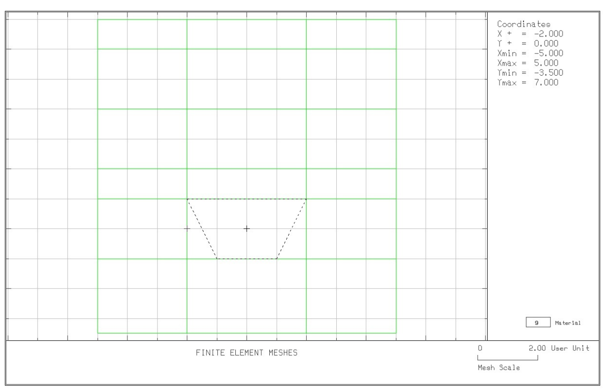


Figure 5.70 Base mesh plot on drawing board

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.

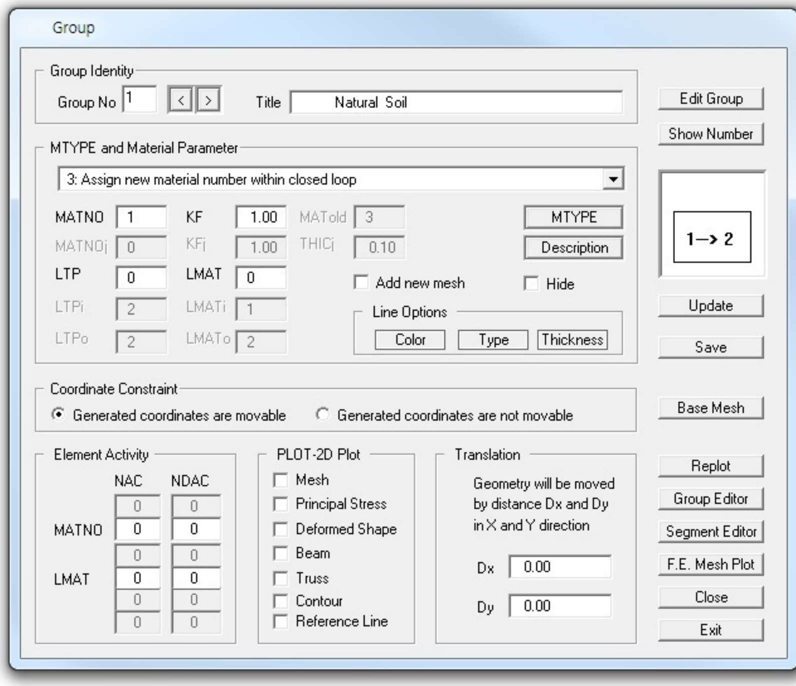


Figure 5.71 Group dialog for natural soil

Figure 5.72 shows Group dialog for excavation.

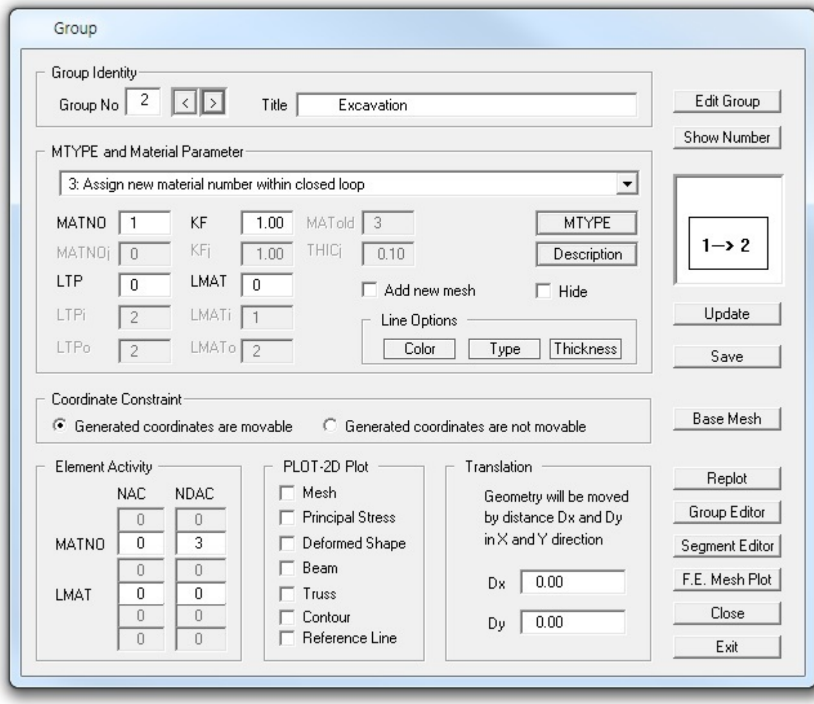


Figure 5.72 Group dialog for excavation

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.

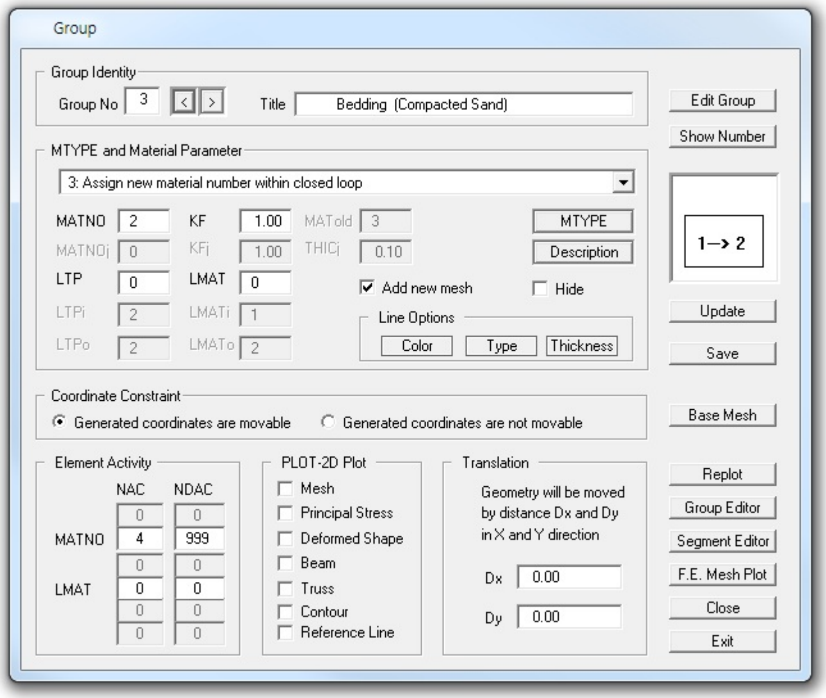


Figure 5.73 Group dialog for bedding

Figure 5.74 shows Group dialog for steel pipe.

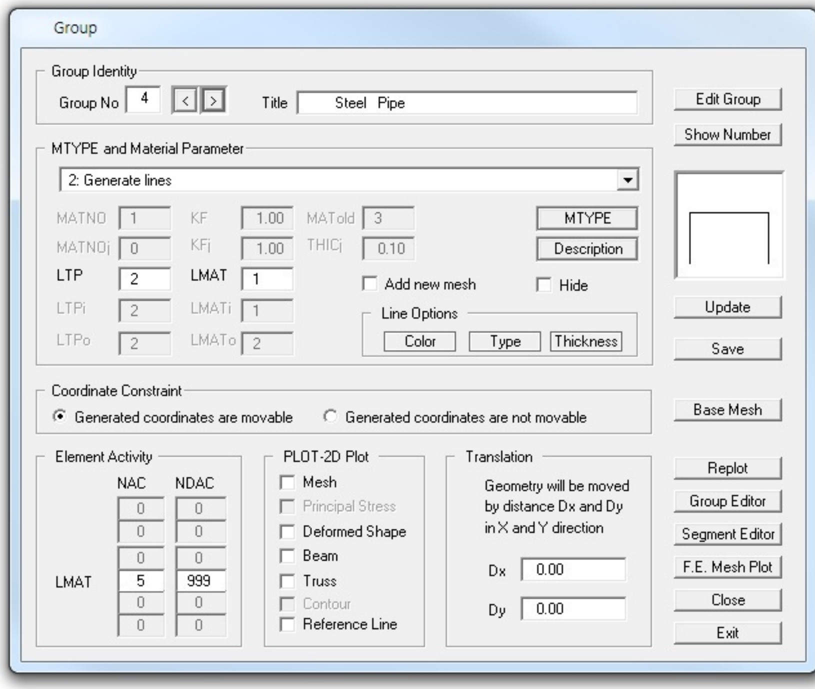


Figure 5.74 Group dialog for steel pipe

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.

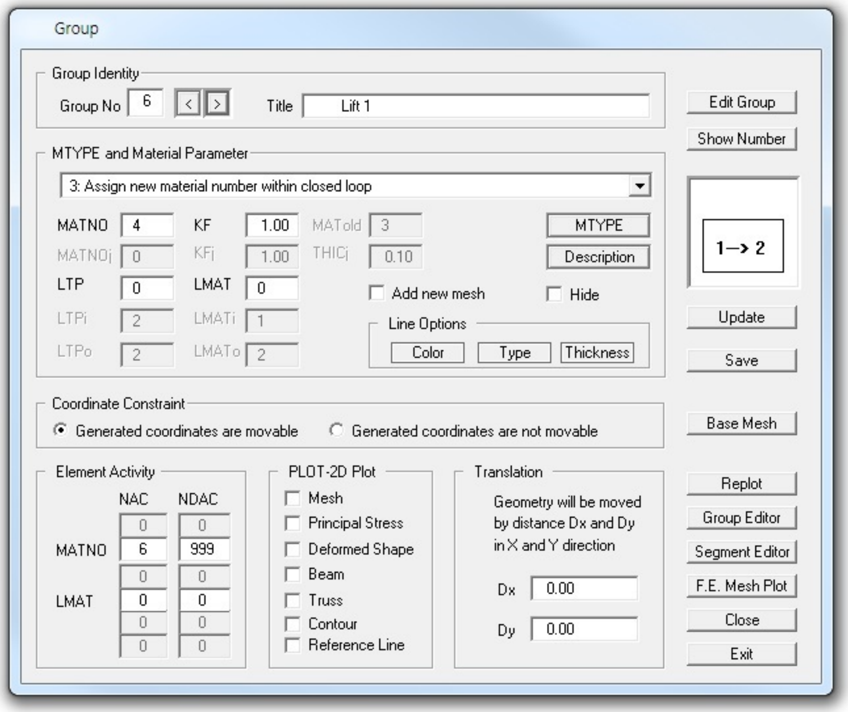


Figure 5.75 Group dialog for first lift

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

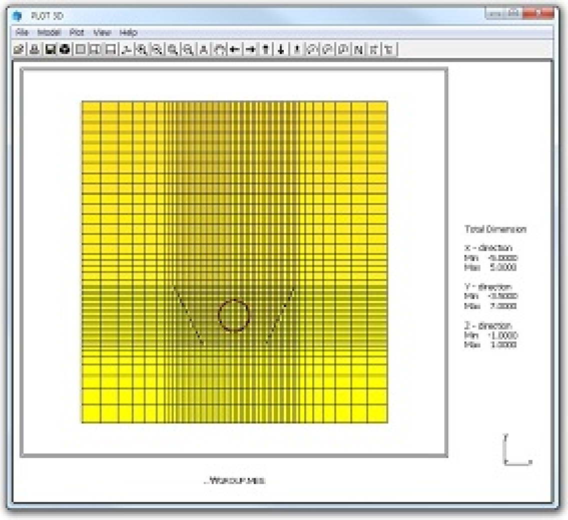
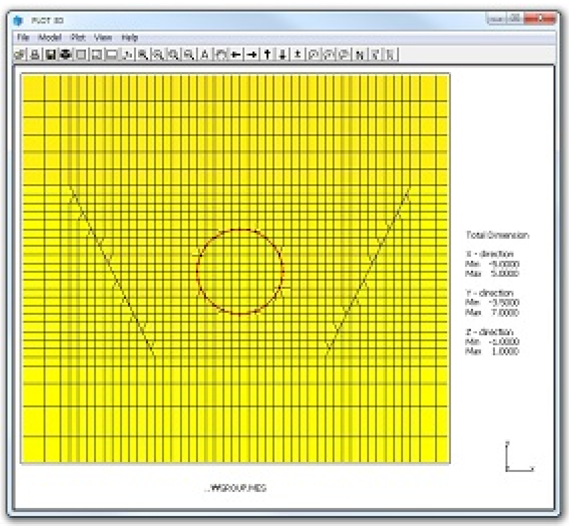


Figure 5.77
Finite element meshes near buried pipe



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.

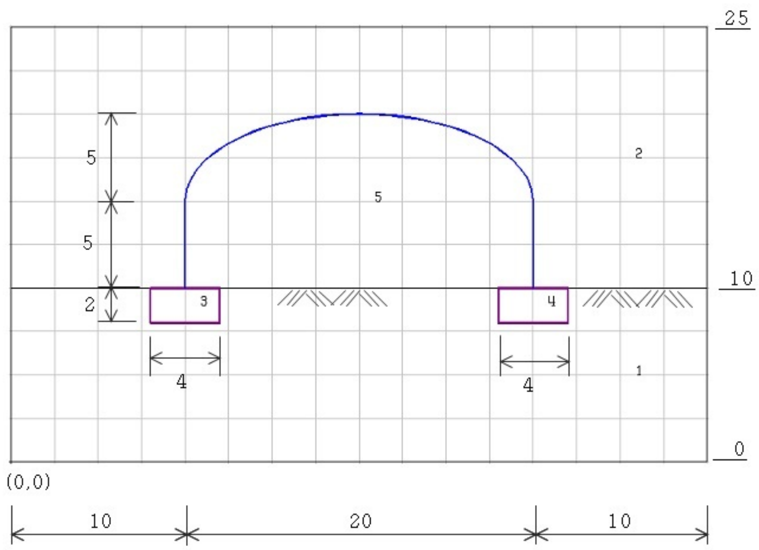


Figure 5.78 Schematic section of arch warehouse

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.

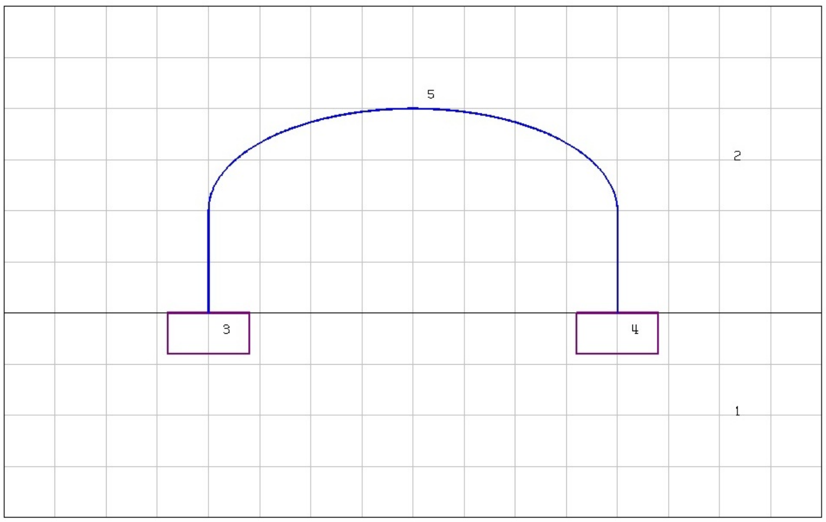


Figure 5.79 Group section view

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.

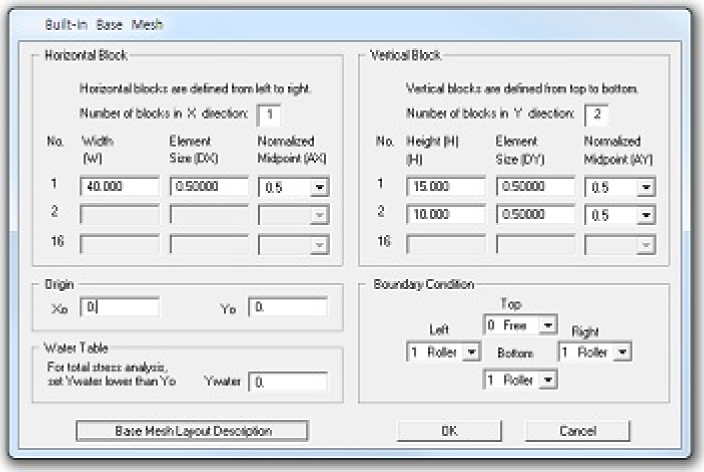


Figure 5.80 Built-in base mesh dialog

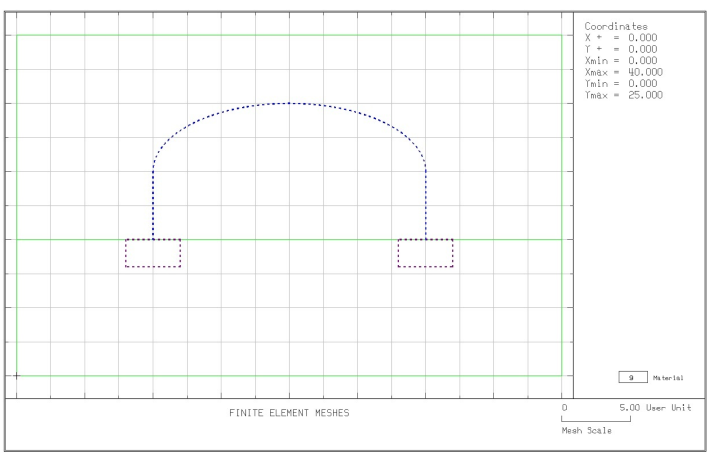


Figure 5.81 Base mesh plot on drawing board

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.

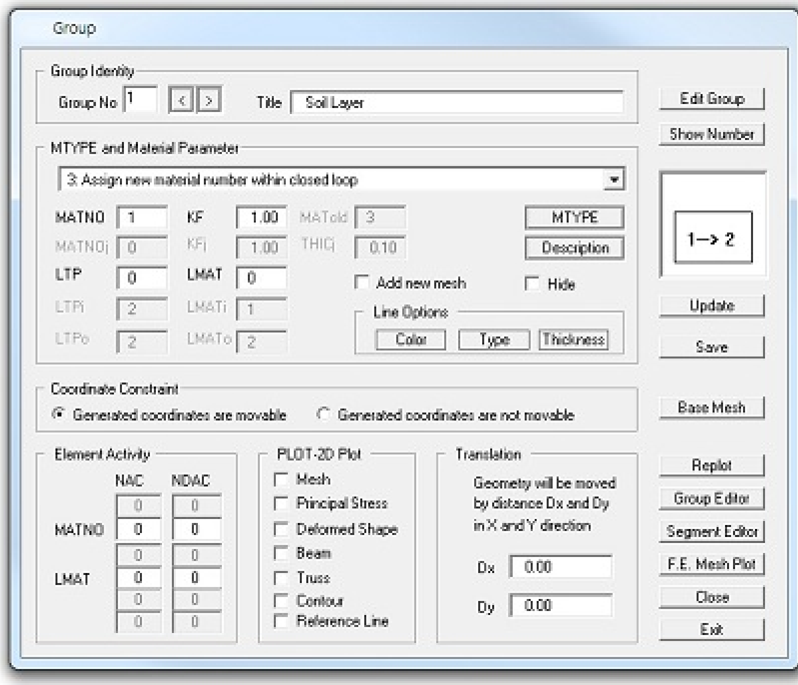


Figure 5.82 Group dialog for soil layer

Figure 5.83 shows Group dialog for above ground.

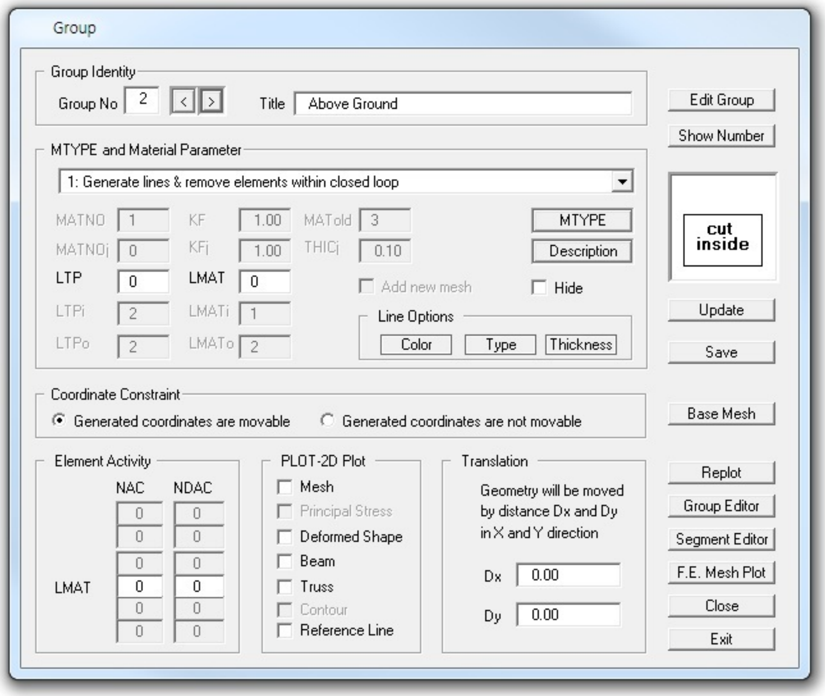


Figure 5.83 Group dialog for above ground

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.

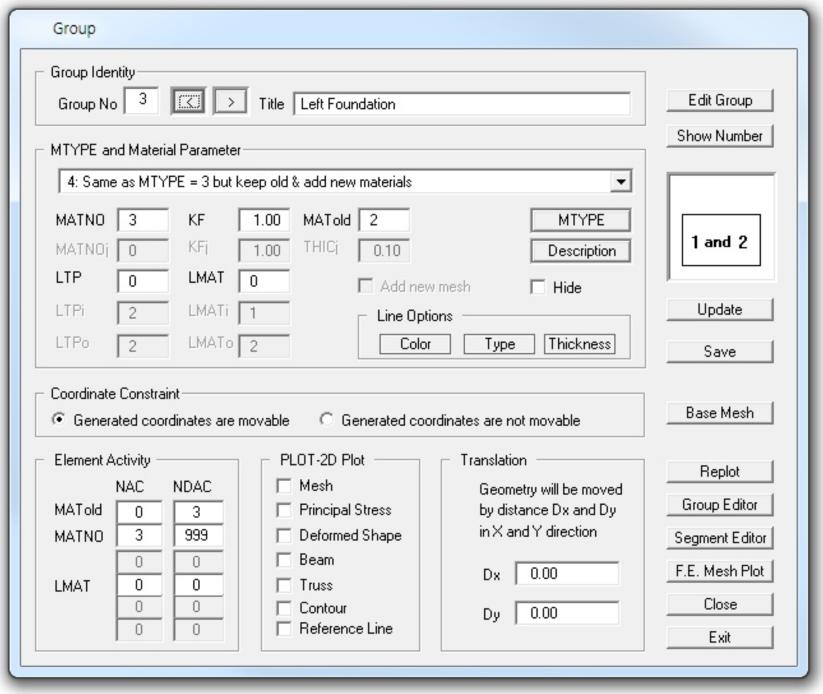


Figure 5.84 Group dialog for left foundation

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.

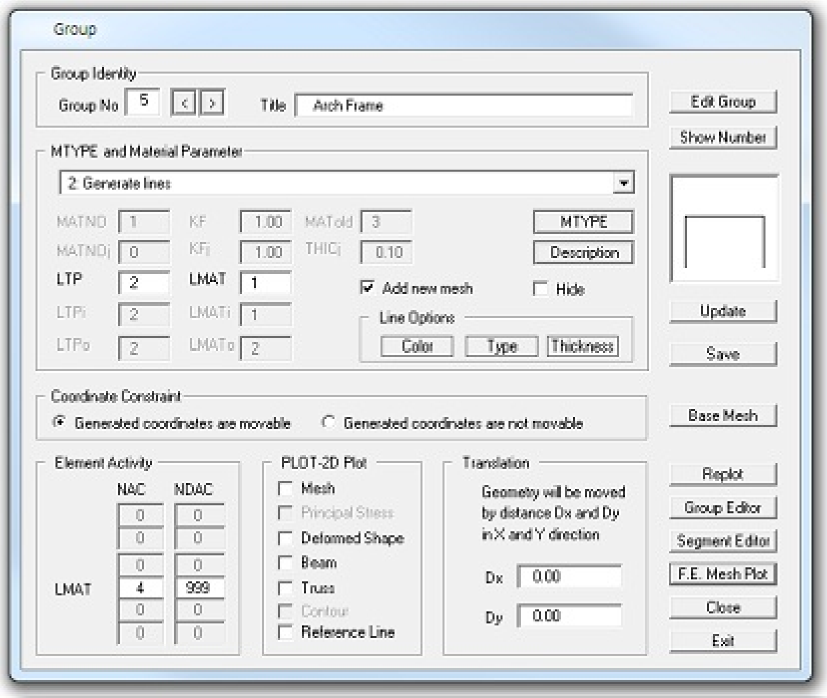


Figure 5.85 Group dialog for arch frame

5.5.4 Finite Element Mesh Plot

Figure 5.86 shows finite element meshes generated from group meshes.

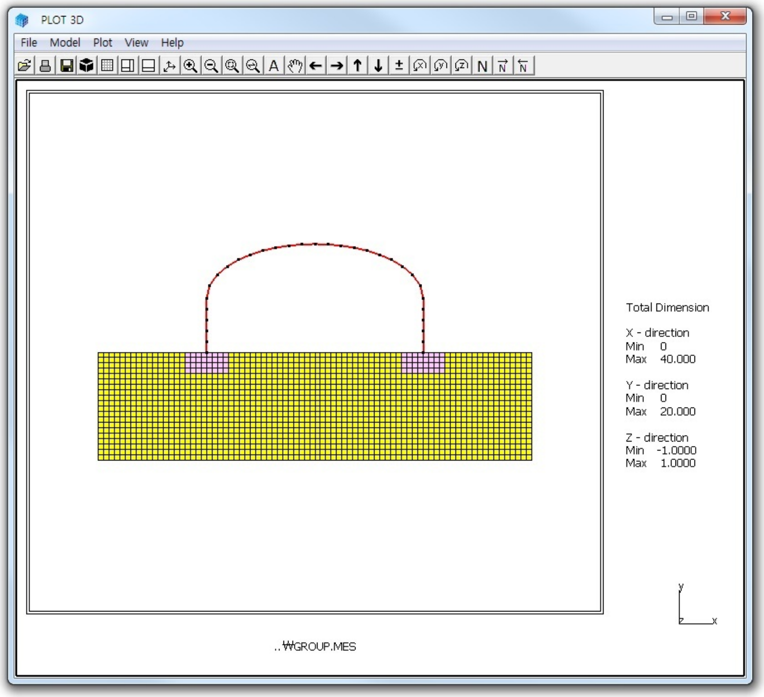


Figure 5.86 Finite element meshes

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.

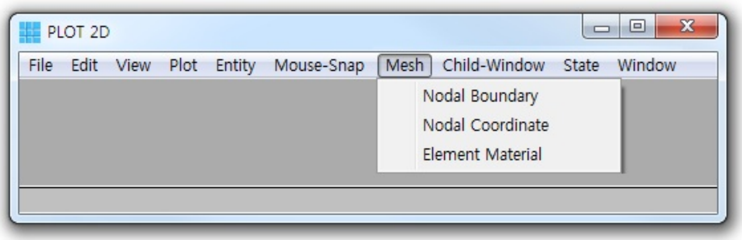


Figure 5.87 Menu for editing finite element mesh

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

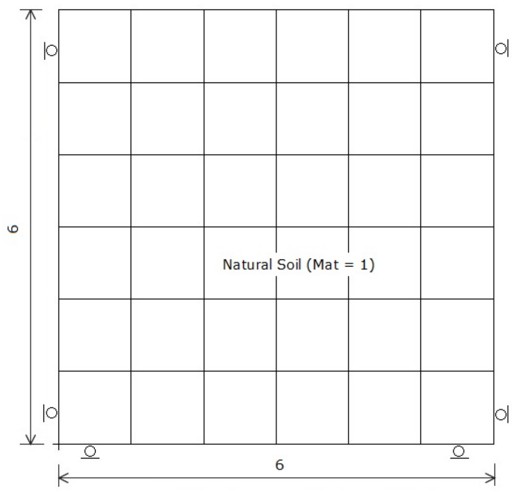


Figure 5.88 Existing finite element mesh

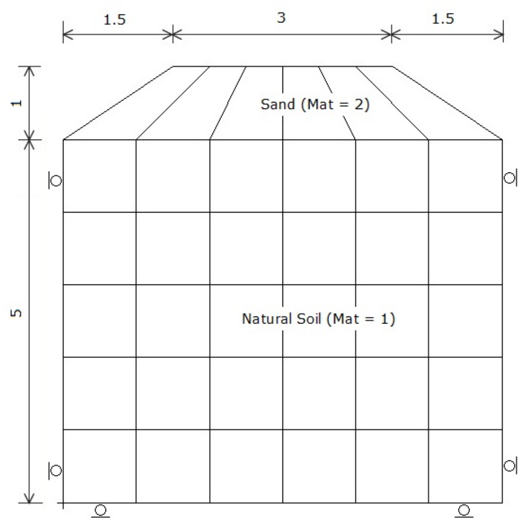


Figure 5.89 Modified finite element mesh

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.

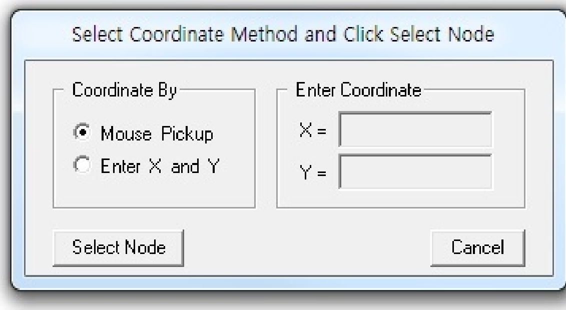


Figure 5.90 Edit coordinate dialog

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

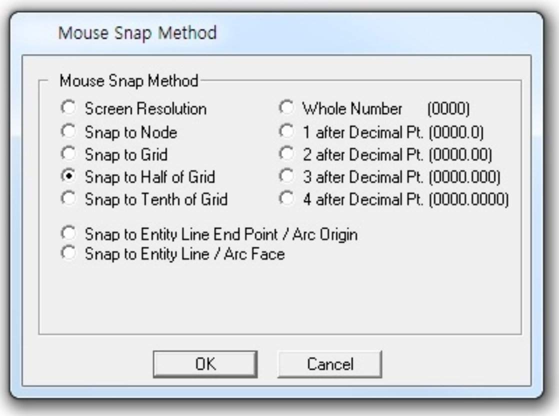


Figure 5.91 Mouse snap method

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.

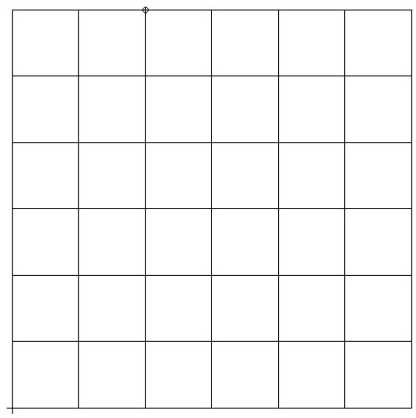


Figure 5.92 First selected node on drawing board

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

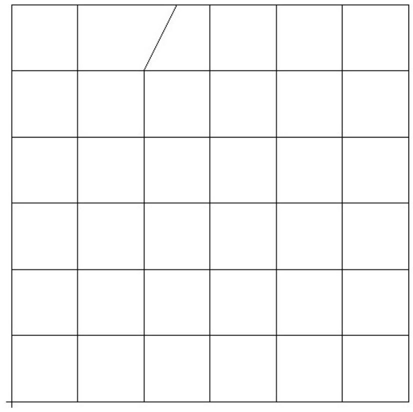


Figure 5.93 New position of first selected node

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

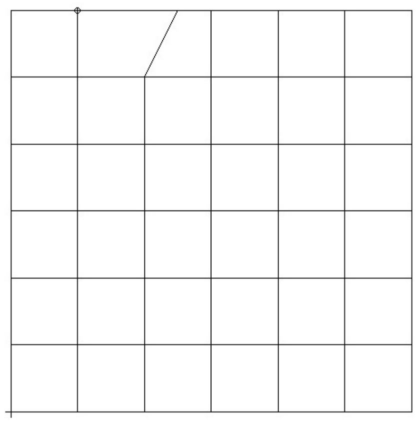


Figure 5.94 Second selected node on drawing board

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

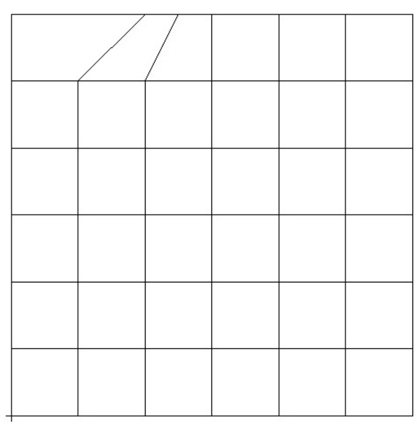


Figure 5.95 New position of second selected node

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

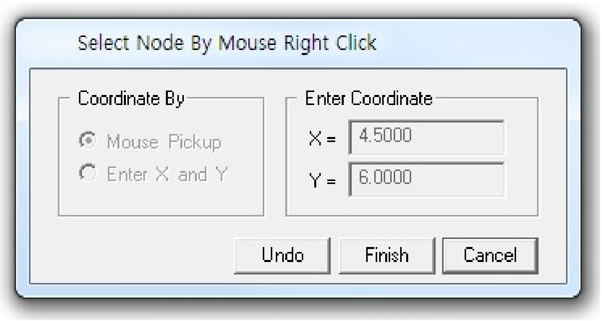


Figure 5.96 Edit coordinate dialog

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

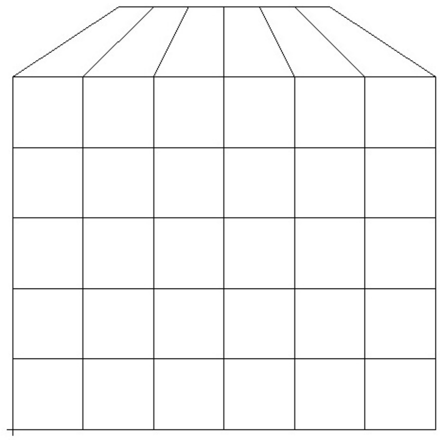


Figure 5.97 Final finite element mesh

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.

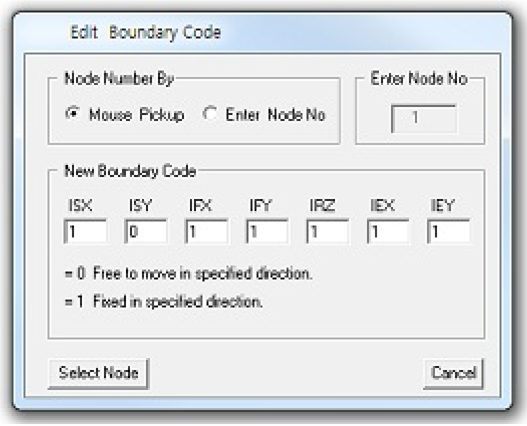


Figure 5.98
Edit boundary dialog

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.

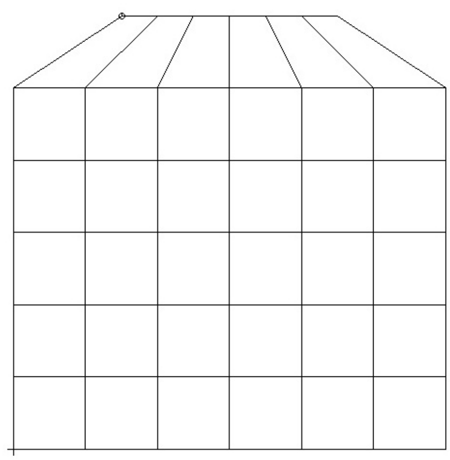
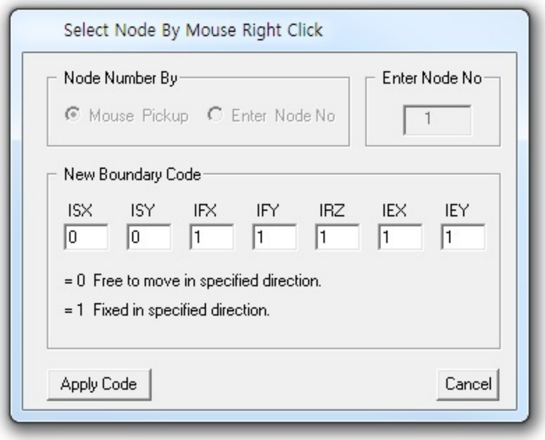


Figure 5.99
Selected node on drawing board

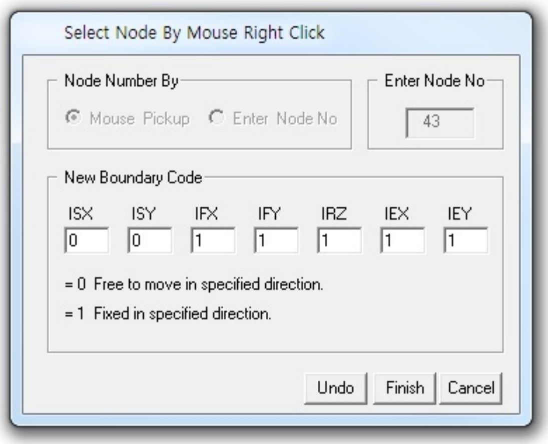
Change the boundary codes as in Figure 5.100 so that the top left node can be free to move in both horizontal and vertical directions and then click Apply Code button.

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.

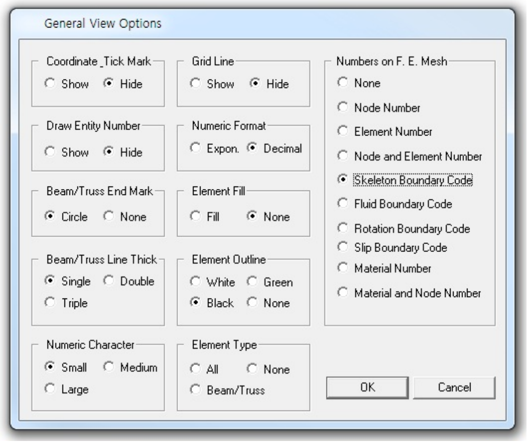


Figure 5.102
General view
for skeleton boundary code

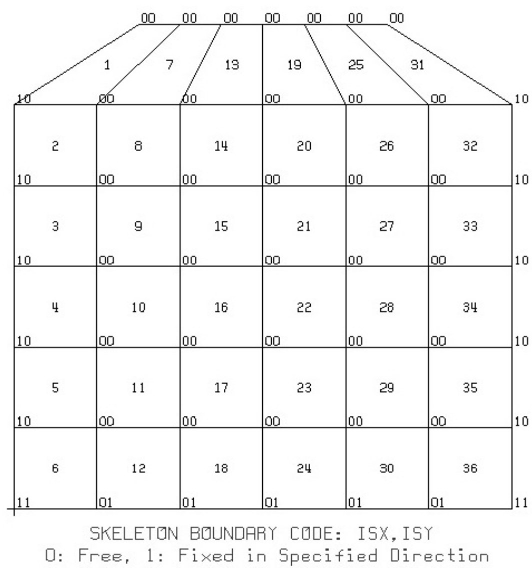


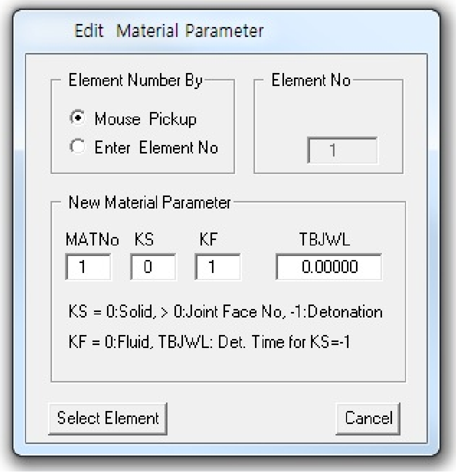
Figure 5.103
Modified skeleton
boundary code plot

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.

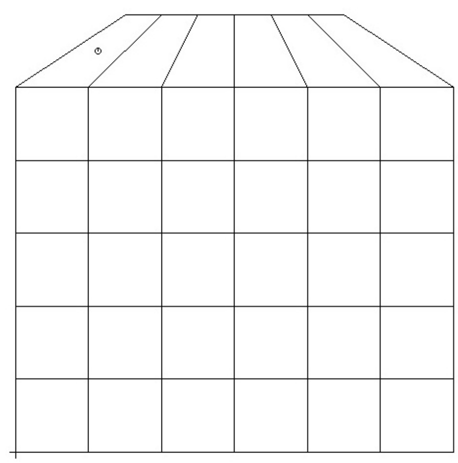
Figure 5.104

Edit element material dialog



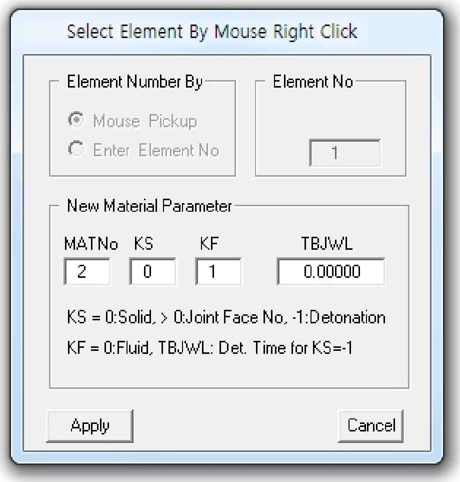
Click Select Element button.
Click the element on the top layer by Mouse Right Click.
Selected element is marked as an open circle as shown in Figure 5.105.

Figure 5.105
Selected element on drawing board



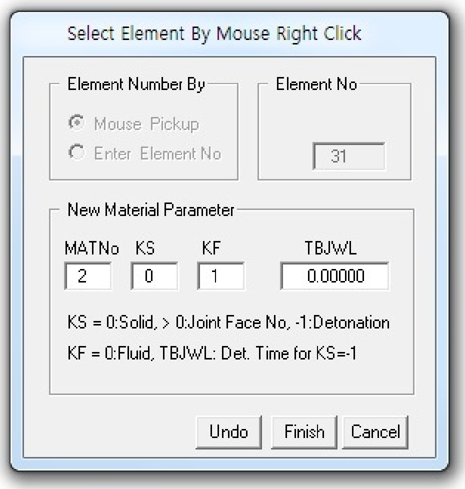
Change the material number as shown in Figure 5.106 and then click Apply button.

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

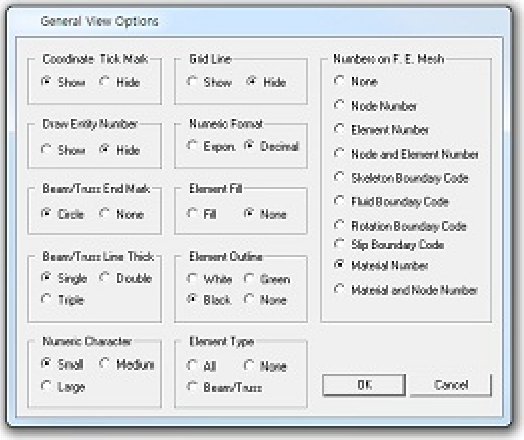
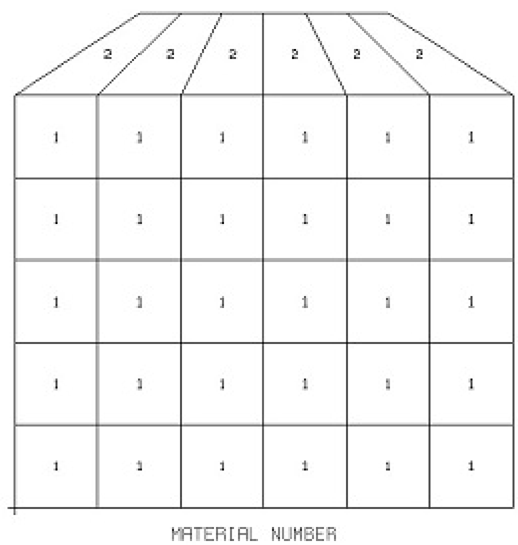


Figure 5.109
Modified material number
plot



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.

Two example problems are presented:

1. Single Element
   Shows step by step procedure to create block mesh.
2. Square Foundation
   Builds block mesh for square foundation.

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 square 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 quad block
5. Edit block boundary code
6. View skeleton boundary code
7. Plot finite element mesh

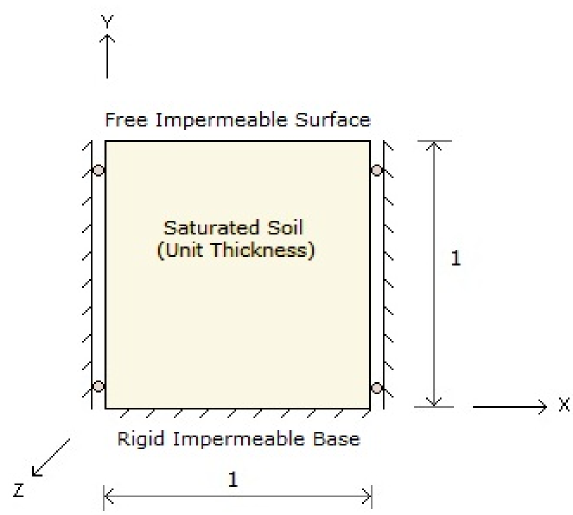


Figure 6.1 Single element in uniaxial strain condition

Step 1: Access Block Mesh Generator (New)

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

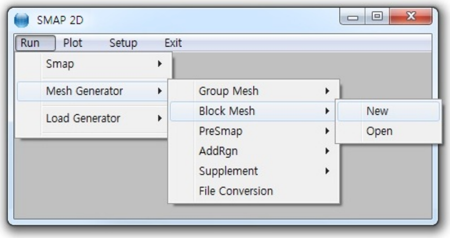


Figure 6.2 Accessing block mesh generator

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.

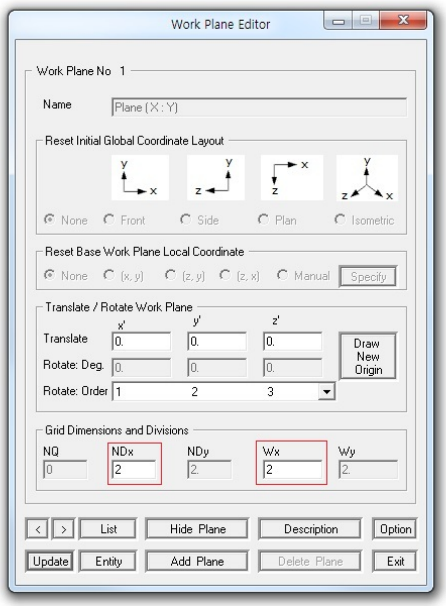


Figure 6.3 Work plane editor

Step 3: Build Cube Entity

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

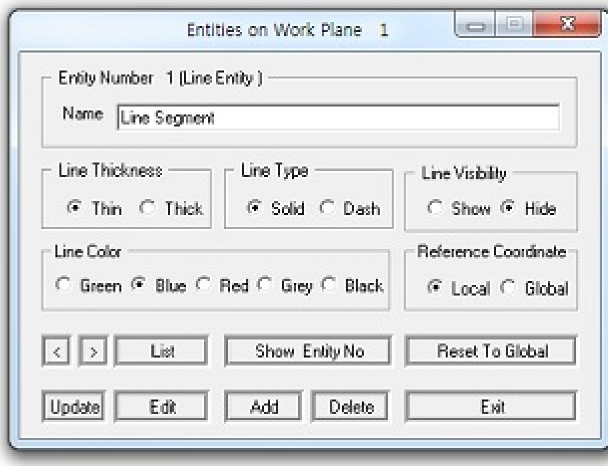


Figure 6.4 Entity editor

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

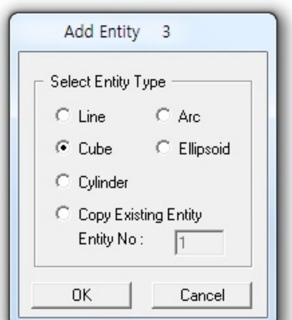


Figure 6.5 Entity type selection

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

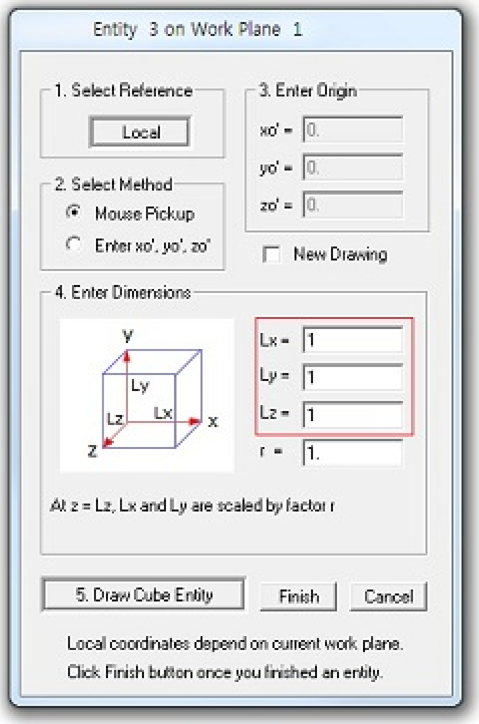


Figure 6.6 Cube entity

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.

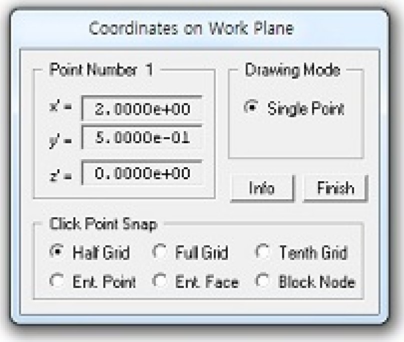


Figure 6.7 Coordinates on work plane

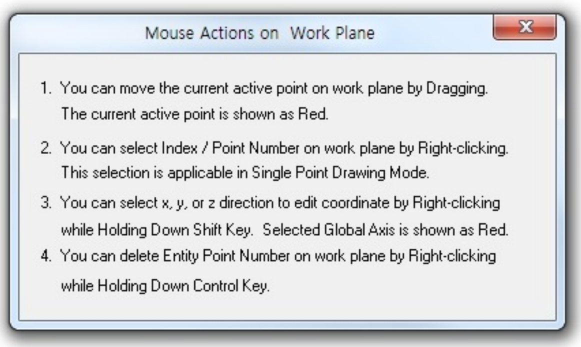


Figure 6.8 Mouse actions on work plane

9. Click Axis toolbar as shown in Figure 6.9.

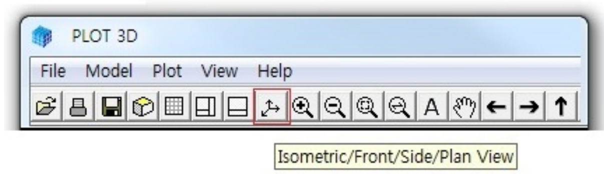


Figure 6.9 Axis toolbar

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

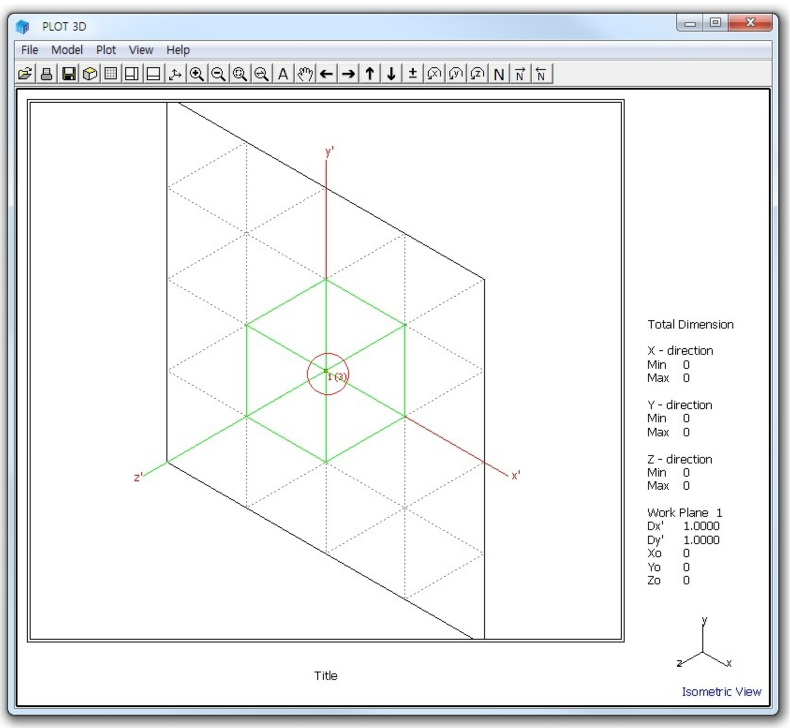


Figure 6.10 Cube entity on isometric work plane

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.

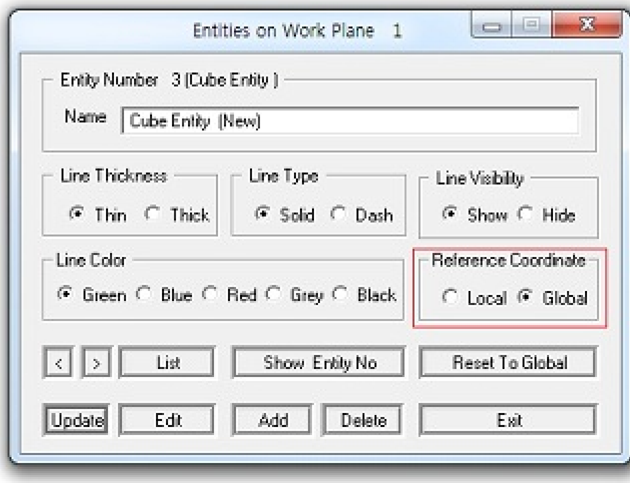


Figure 6.11 Entity editor

Step 4: Build Quad Block

1. Click Block Editor toolbar in Figure 6.12.

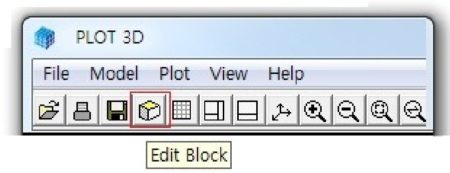


Figure 6.12 Block editor toolbar

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

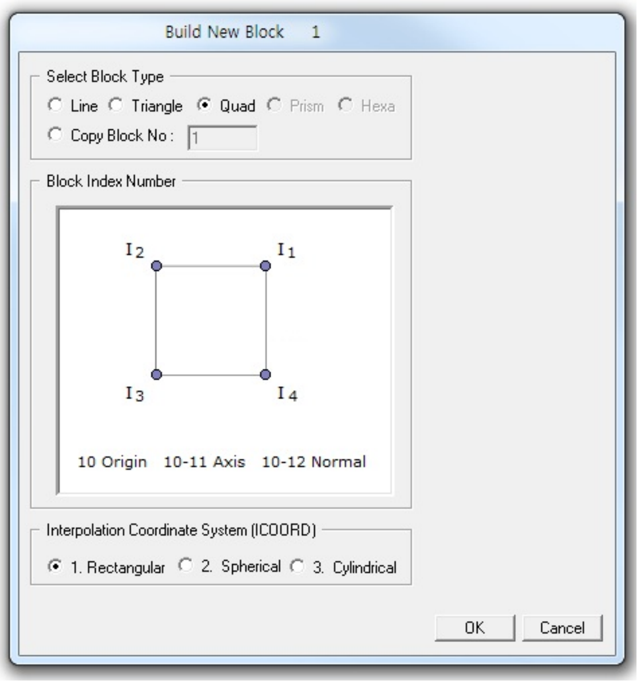


Figure 6.13 Block type selection

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

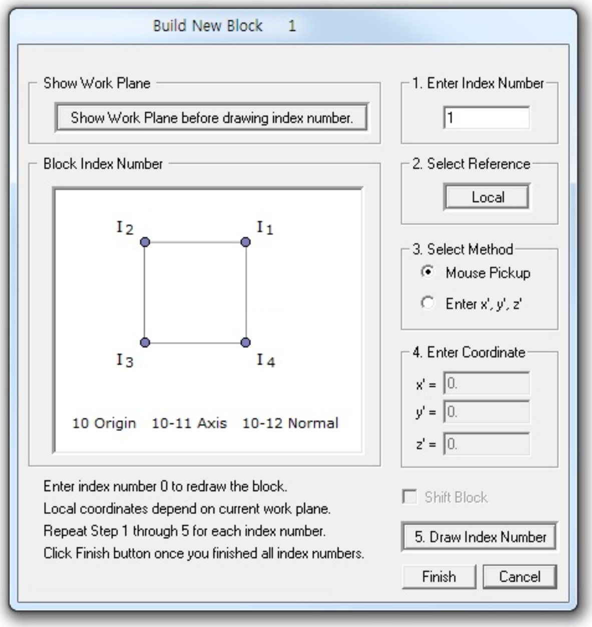


Figure 6.14 Quad block

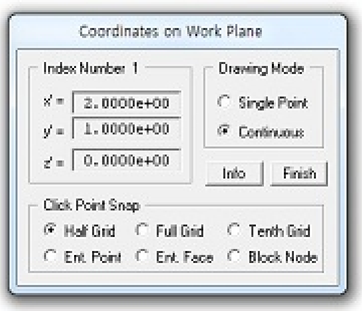


Figure 6.15 Coordinates on work plane

5. Click the points for index numbers for quad block as in Fig. 6.16.

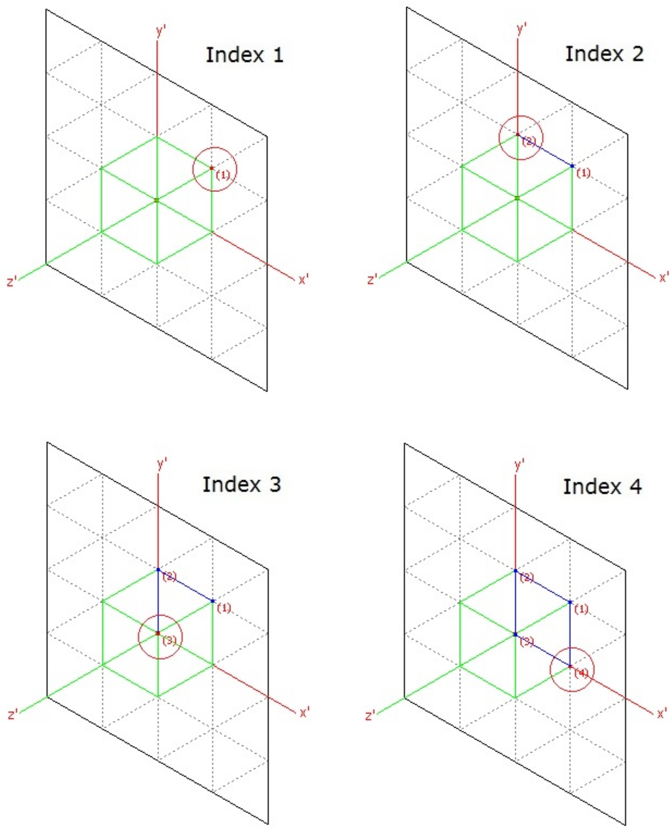


Figure 6.16 Index numbers for quad block

Now, the geometry of quad block is completed.
6. Click Finish in Figure 6.17 and then click Finish in Figure 6.14.

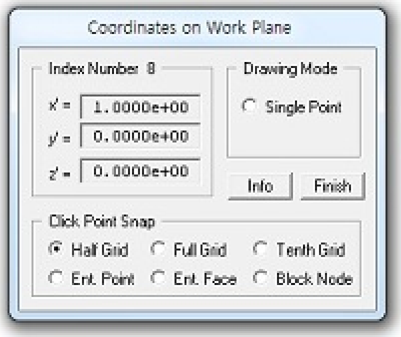


Figure 6.17 Coordinates on work plane

7. Get back to Work Plane Editor dialog and click Entity.
8. Select Entity Number 3, Hide for line visibility, click Update, and click Exit in Figure 6.18.

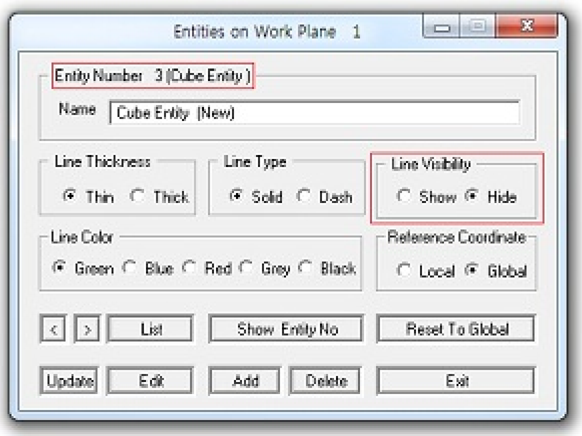


Figure 6.18 Entity editor

9. Modify Title and Material & Element Generation Parameters in Block Editor as shown in Figure 6.19.
10. Click Save and type in file name as EX1.

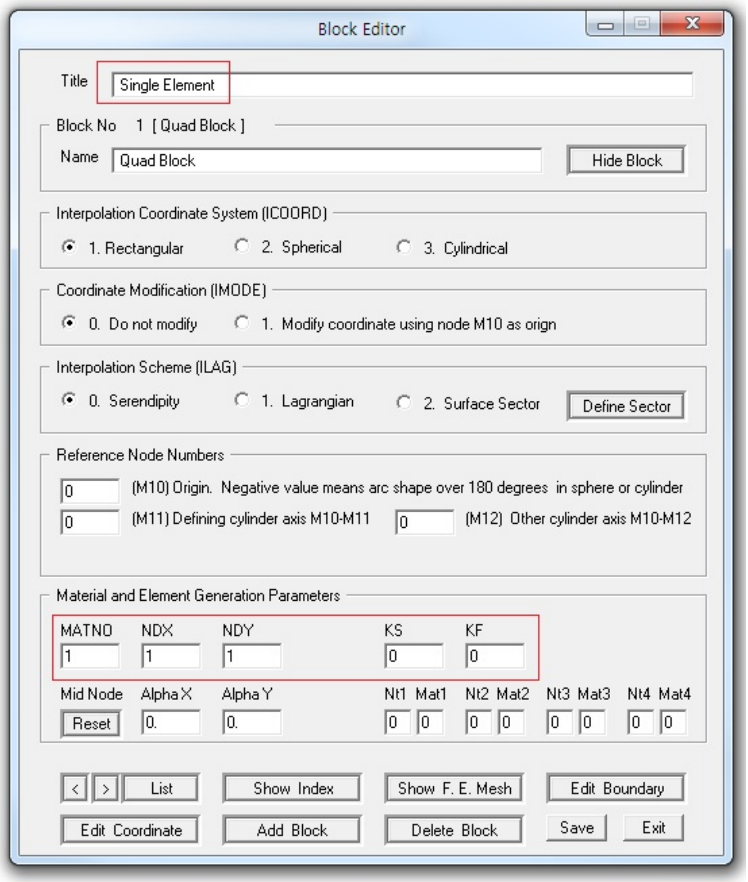


Figure 6.19 Block editor

Step 5: Edit Block Boundary Code

1. Click Edit Boundary in Figure 6.19.
2. Set the boundary codes as shown in Figure 6.20.
3. Click IBTYPE button to see description of boundary type in Fig. 6.21.
4. Click Update and then OK buttons.

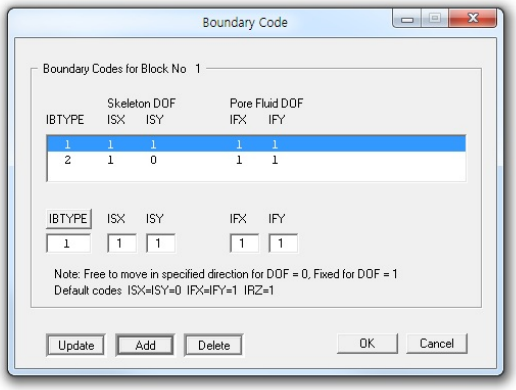


Figure 6.20 Boundary code editor

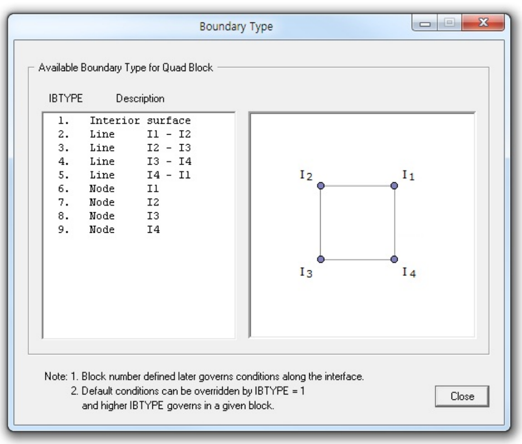


Figure 6.21 Boundary type for quad block

Step 6: View Skeleton Boundary Code

1. Select View → General in PLOT-3D menu.
2. Select Skeleton Boundary Code and click OK in Figure 6.22.
3. Click Save in Figure 6.19.

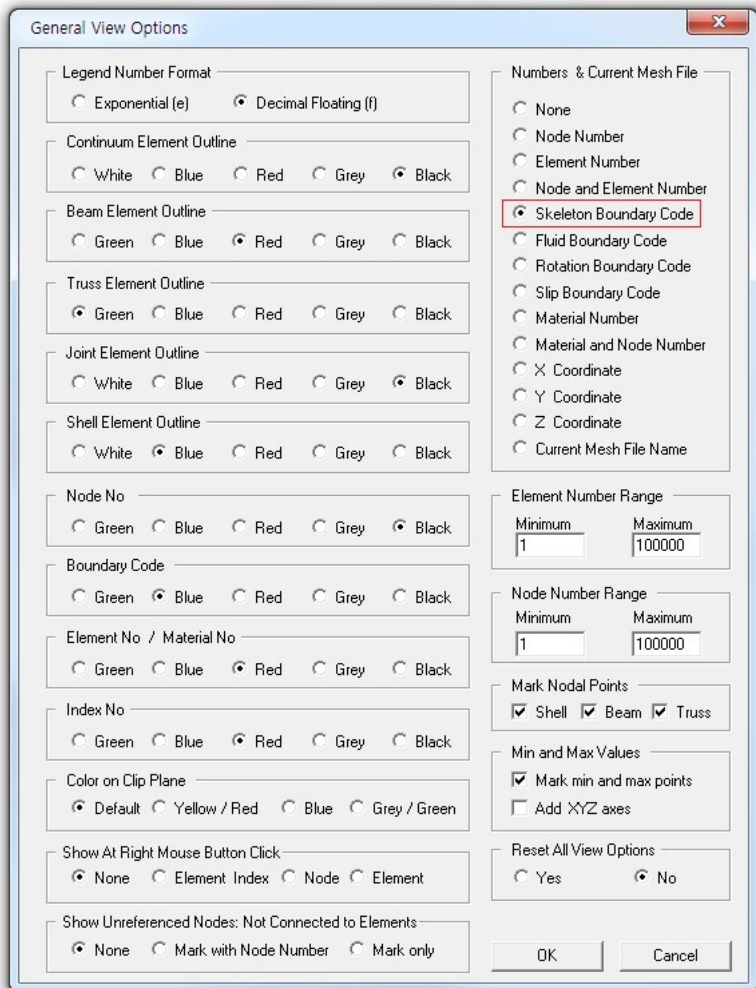


Figure 6.22 General view options

4. Click Show Numbers toolbar as shown in Figure 6.23.

Figure 6.23 Show numbers toolbar

5. Skeleton boundary codes are shown in Figure 6.24.

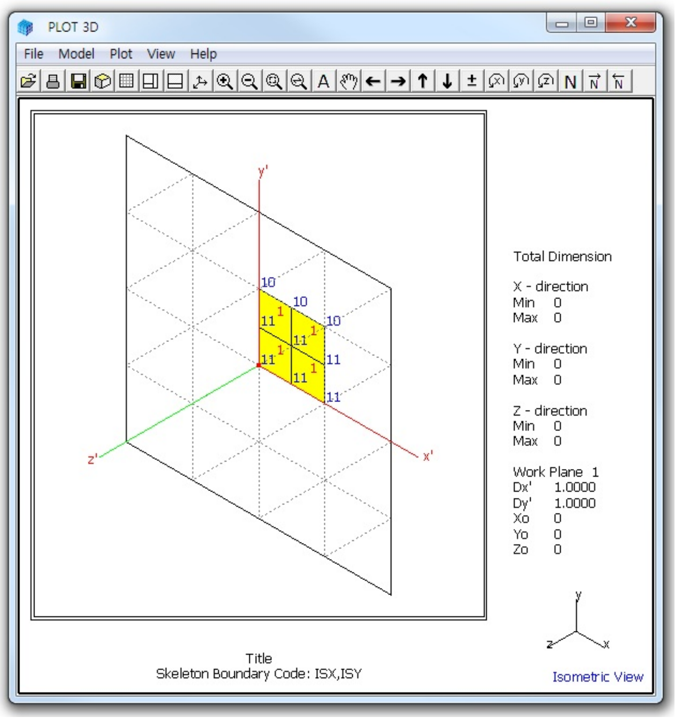


Figure 6.24 Skeleton boundary codes on drawing board

Step 7: Plot Finite Element Mesh

1. Click Show F. E. Mesh in Figure 6.19.
2. Figure 6.25 shows the finite element mesh.

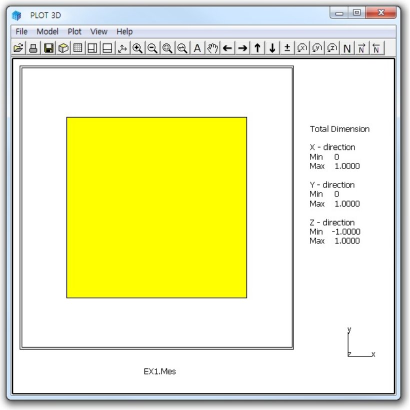


Figure 6.25 Finite element mesh

3. Follow same procedure to plot skeleton boundary code in Step 6.
4. Figure 6.26 shows skeleton boundary code for finite element mesh.

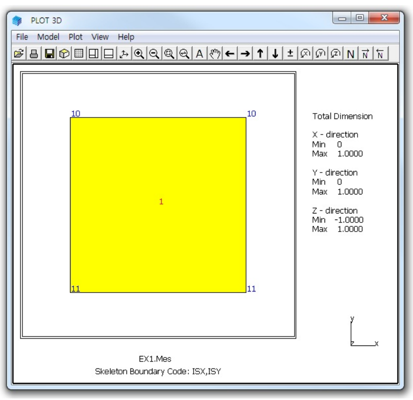


Figure 6.26 Skeleton boundary codes

6.2 Square Foundation

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

This example has the following two parts:

Part 1: Creating Square Foundation (Figure 6.27)

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

Part 2: Modifying Square Foundation (Figure 6.28)

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

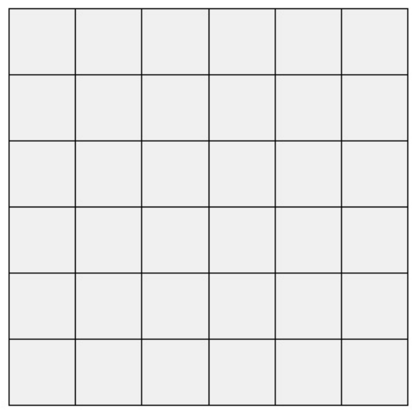


Figure 6.27 Square foundation with constant element size

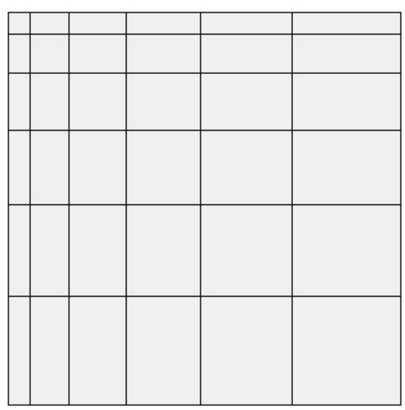


Figure 6.28 Square foundation with variable element size

6.2.1 Part 1: Creating Square Foundation

Part 1 consists of the following seven main steps:

1. Access block mesh generator (New)
2. Set work plane
3. Build quad 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 6.29 and click Update button.

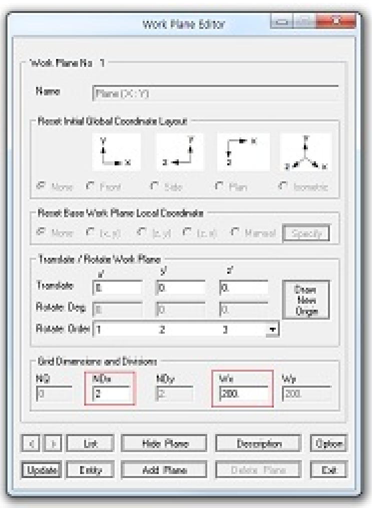


Figure 6.29 Work plane editor

Step 3: Build Quad 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 Quad 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 Quad Block

6. Click the points for index numbers for quad block as in Fig. 6.30.

Now, the geometry of quad block is completed.

7. Click Finish in Figure 6.17.
8. Click Finish in Figure 6.14.

9. Modify Title and Material & Element Generation Parameters in Block Editor dialog as shown in Figure 6.31.

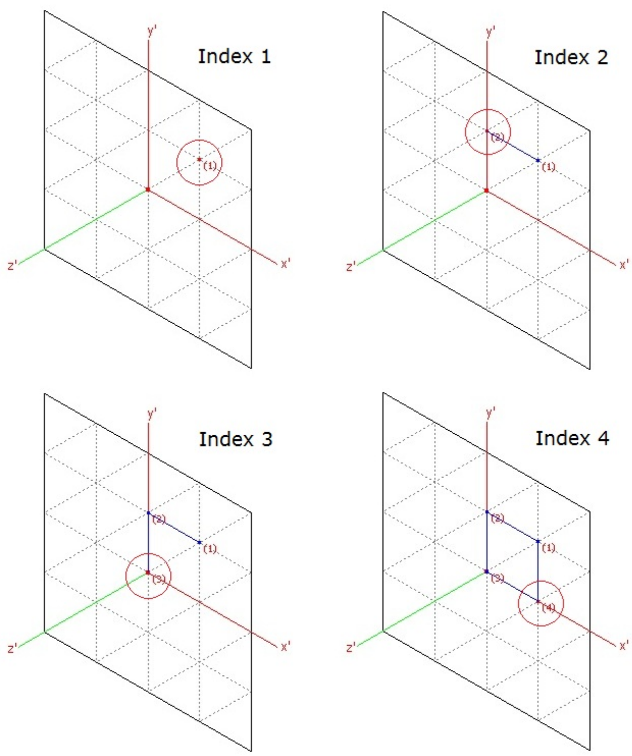


Figure 6.30 Index numbers for quad block

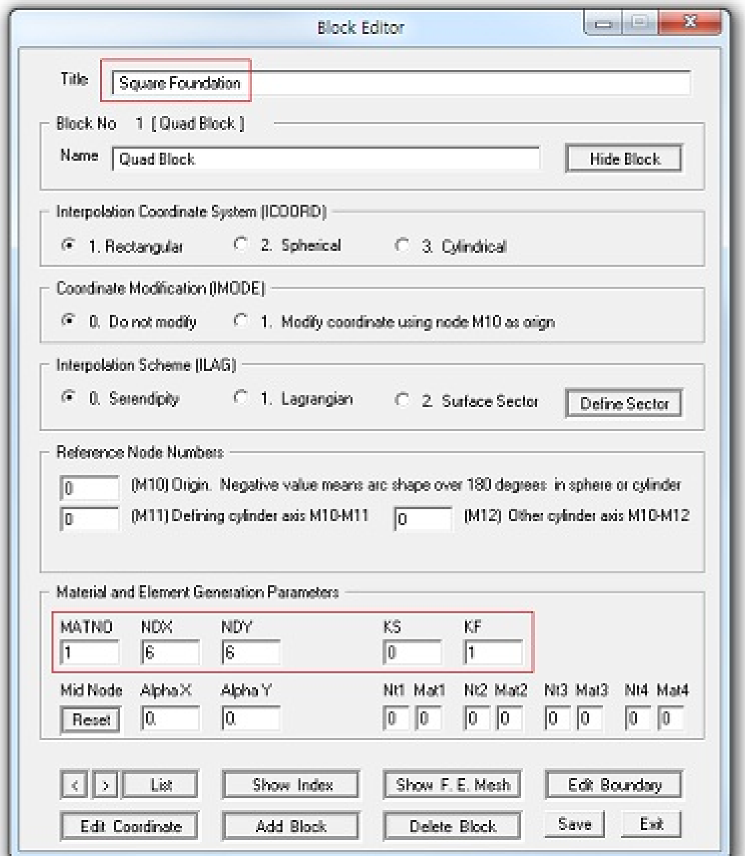


Figure 6.31 Block editor

Step 4: Edit Block Boundary Code

1. Click Edit Boundary in Figure 6.31.
2. Set the boundary codes as shown in Figure 6.32.
3. Click IBTYPE button to see description of boundary type in Fig. 6.33.
4. Click Update and then OK buttons in Figure 6.32.

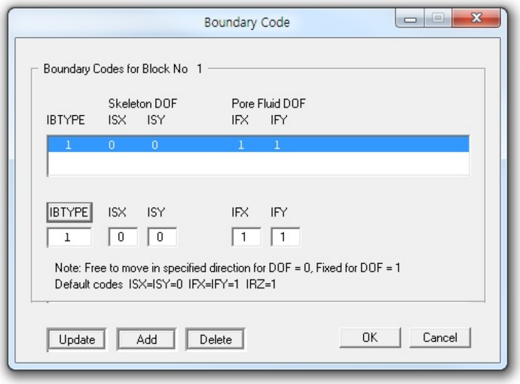


Figure 6.32 Boundary code editor

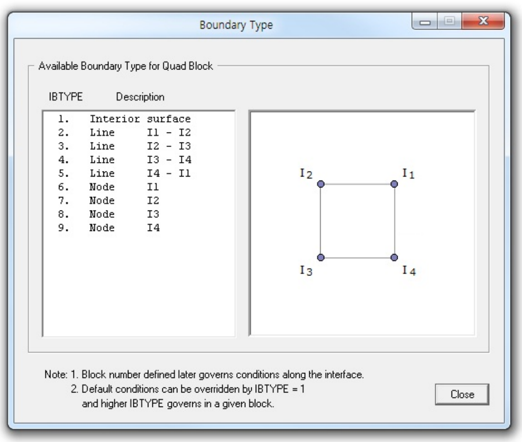


Figure 6.33 Boundary type for quad block

Step 5: Set Global Boundary Code

1. Select Model → Edit Global Boundary in Figure 6.34.

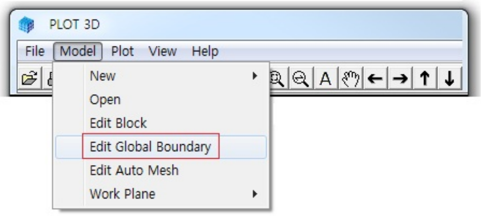


Figure 6.34 Edit global boundary menu

2. Set the boundary codes as shown in Figure 6.35.
3. Select Yes override block boundary.
4. Click Save and type in file name as EX2.

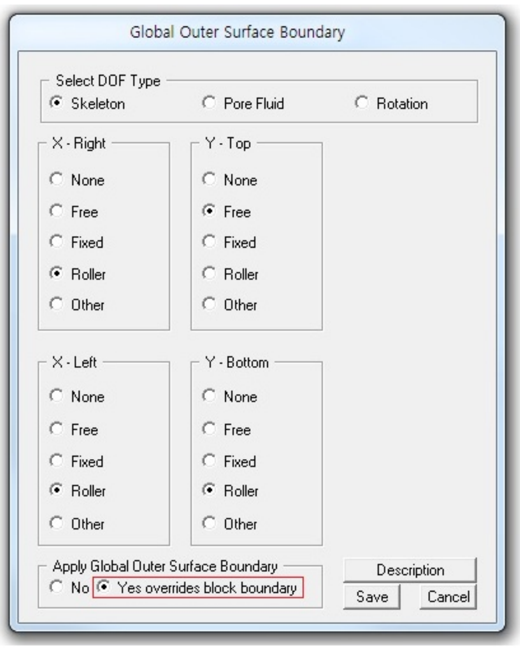


Figure 6.35 Global outer surface boundary

Step 6: View Skeleton Boundary Code

1. Select View → General in PLOT-3D menu.
2. Select Skeleton Boundary Code and click OK in Figure 6.22.
3. Click Show Numbers toolbar as shown in Figure 6.23.
4. Skeleton boundary codes are shown in Figure 6.36.

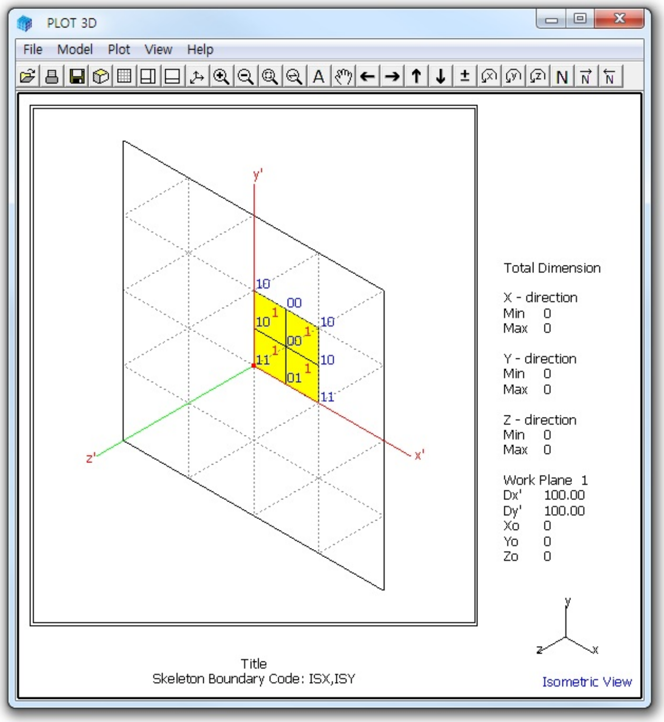


Figure 6.36 Skeleton boundary codes on drawing board

Step 7: Plot Finite Element Mesh

1. Click Show F. E. Mesh in Figure 6.31.
2. Figure 6.37 shows the finite element mesh.

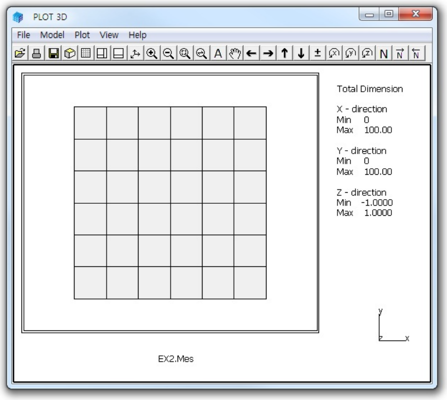


Figure 6.37 Finite element mesh

3. Follow same procedure to plot skeleton boundary code in Step 6.
4. Figure 6.38 shows skeleton boundary code for finite element mesh.

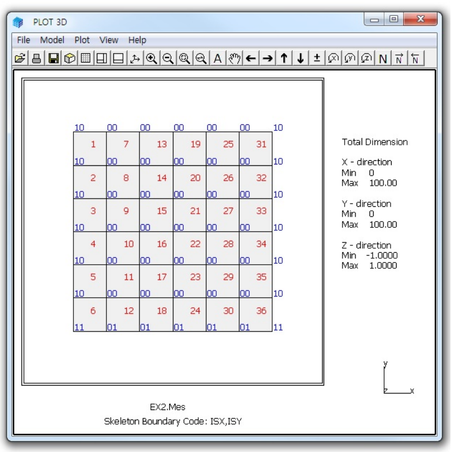


Figure 6.38 Skeleton boundary codes

6.2.2 Part 2: Modifying Square 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 6.39.
3. Select the input file EX2.Meb generated in Part 1.

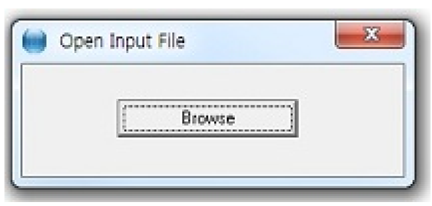


Figure 6.39 Open input file

Step 9: Modify Element Generation Parameters

1. Click Block Editor toolbar in Figure 6.12.
2. Modify Alpha X, Alpha Y as in Figure 6.40.
3. Click Reset.
4. Click Save.

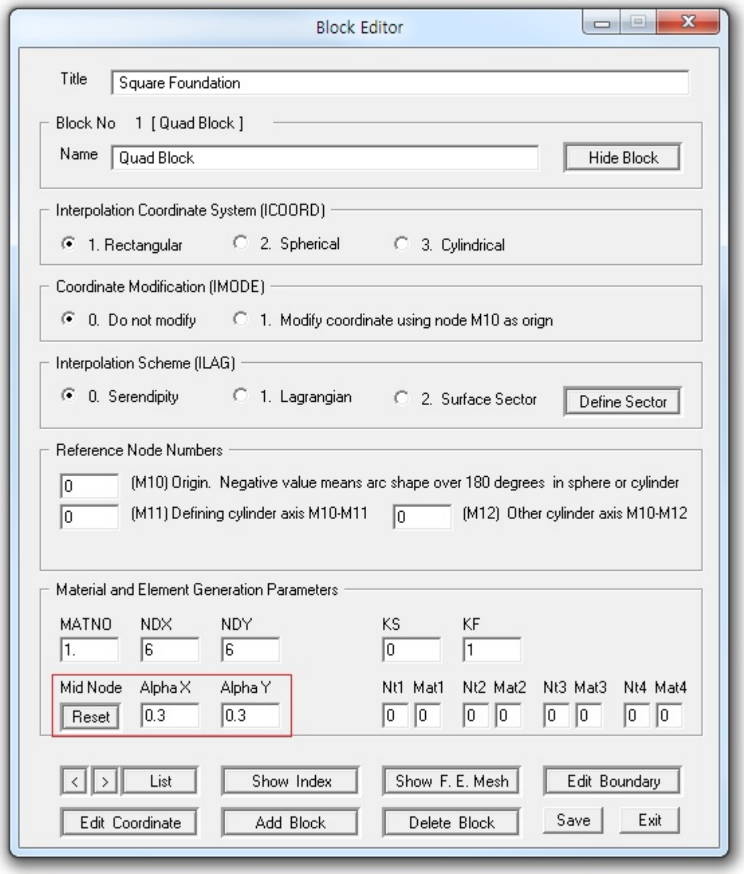


Figure 6.40 Block editor

5. Block mesh is shown in Figure 6.41.

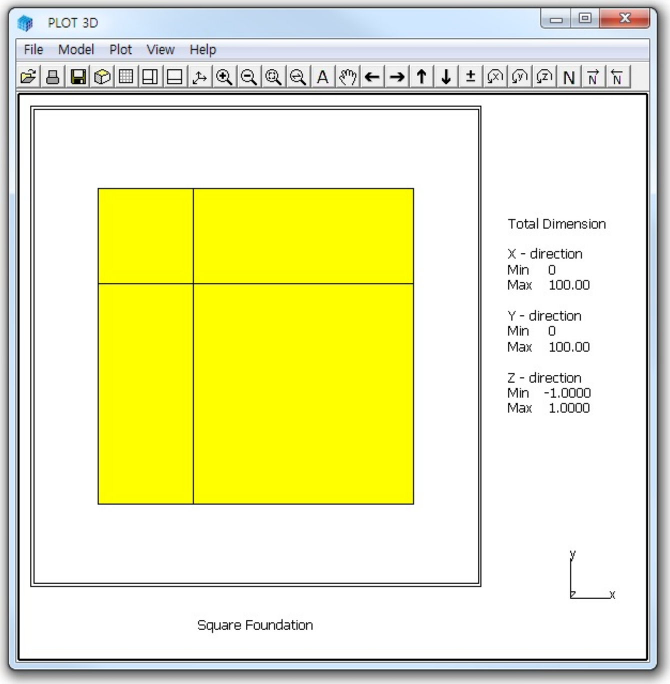


Figure 6.41 Block mesh on drawing board

Step 7: Plot Finite Element Mesh

1. Click Show F. E. Mesh in Figure 6.40.
2. Figure 6.42 shows the finite element mesh.

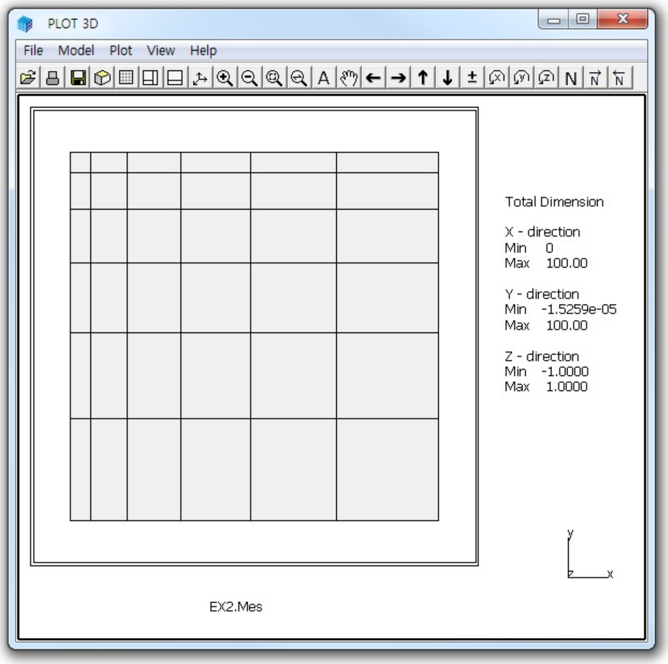


Figure 6.42 Finite element mesh

PRESMAP
Example Problem

PRESMAP menu includes four Pre-Processing programs: PRESMAP-2D, NATM-2D, CIRCLE-2D, and PRESMAP-GP. These Pre-Processing programs are mainly used to generate Mesh File described in Section 4.3 of User's Manual. Refer to SMAP-2D User's Manual:

- Section 5 for input parameters for PRESMAP programs.
- Section 3.2.2 for running PRESMAP programs.

7.1 PRESMAP-2D

PRESMAP-2D includes Model 1, 2, 3, and 4. Model 1 is basic pre-processor which can be applied to model various types of problem geometry.

Model 2 is the special pre-processor developed to model near-field around underground openings such as tunnels, culverts, etc. Model 3 is the special pre-processor developed to model triangular and rectangular shape geometry. Model 4 is the useful pre-processor to generate layered embankments having slope.

7.1.1 Model 1

A typical underground tunnel is chosen here to illustrate mesh generations using PRESMAP-2D Model 1 and 2. Figure 7.1 shows geological condition around tunnel consisting of four layers: weathered soil, weathered rock, soft rock, and hard rock. Figure 7.2 shows in detail tunnel cross section including shotcrete and rock bolt dimensions.

For convenience, the tunnel problem geometry is divided into three regions as shown in Figure 7.3; Core, Near-field, and Far-field regions. By symmetry, only right half of the tunnel geometry is considered. Model 1 is used to generate Core and Far-field region meshes. And Model 2 is used to generate Near-field region mesh. Near-field region mesh generation will be explained in the next section. And assembly of Core, Near-field, and Far-field regions will be explained in ADDRGN-2D Example Problems in Section 8.1.

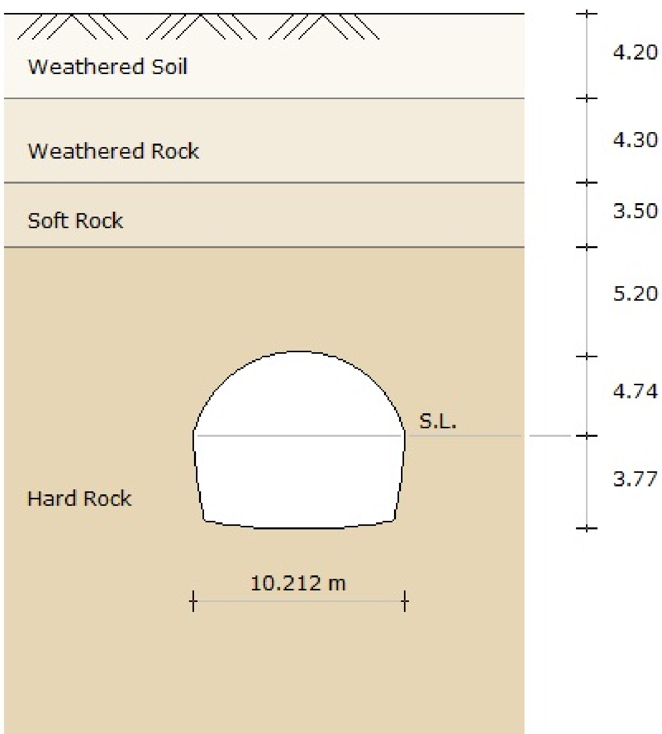
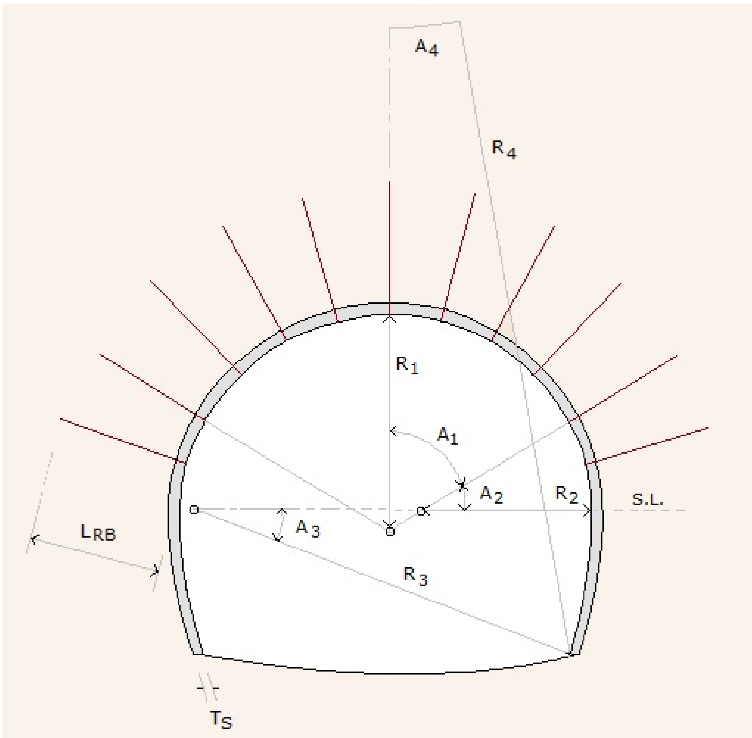


Figure 7.1 Geological condition



R_1 = 5.24 m A_1 = 60°
R_2 = 4.24 m A_2 = 30°
R_3 = 9.86 m A_3 = 19.781°
R_4 = 23.86 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)	=	15 Cm
Thickness of Liner (TL)	=	30 Cm
Reinforcing Bar Area (ASI)	=	22 Cm^2
Reinforcing Bar Area (ASO)	=	22 Cm^2

Figure 7.2 PD-2 tunnel section detail

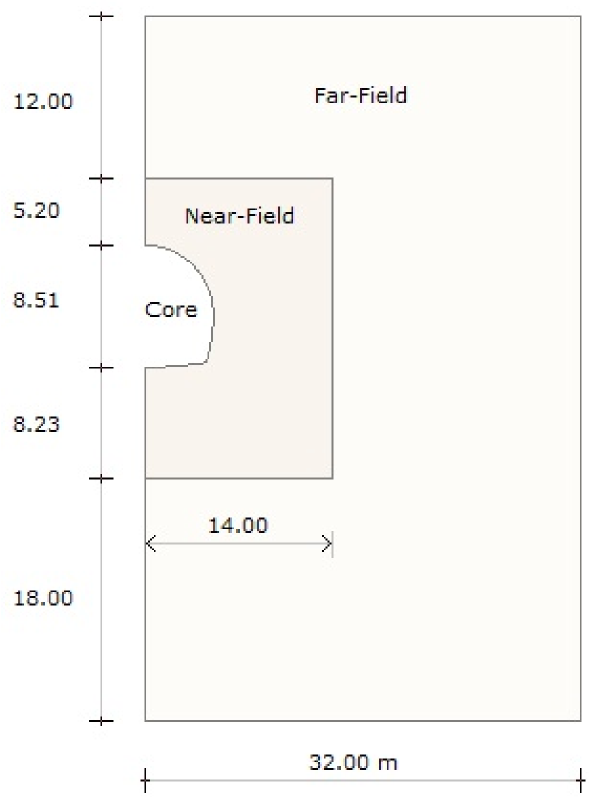


Figure 7.3 Core, Near-field, and Far-field regions

7.1.1.1 Core Region Mesh Generation

Figure 7.4 shows the block diagram for the Core region. Three blocks are used in the horizontal direction (NBX=3) and four blocks in the vertical direction (NBY=4). Block numbers should be in order from top to bottom and left to right. Top 9 blocks (Block numbers 1,2,3,5,6,7,9,10, and 11) represent upper half of tunnel core to be excavated first and bottom 3 blocks (Block numbers 4,8, and 12) represent lower half of tunnel core to be excavated later.

Each block can be consisted of 4 to 9 block nodes depending on whether you can include side and center block nodes. For those blocks facing the tunnel wall of the Core region, side block nodes are included to form the curve. Note that when the side block node is not specified, the straight line will be formed along that side.

Block index should be specified in counterclockwise. For example, the index of Block 4 can be written as I_1=11, I_2 =4, I_3=5, I_4=12, M_5=0, M_6=0, M_7 =7, M_8=0, M_9=0. Next, each block is further divided into elements. For example, Block 4 has 2 elements in the horizontal direction (NDX=2) and 6 elements in the vertical direction (NDY=6). It should be noted that to be compatible, the same number of divisions be specified along the two adjacent blocks. For example, Blocks 4, 8, and 12 have 6 elements in the vertical direction so that the generated elements can share the same nodal points along the boundaries of these blocks.

Since the tunnel is symmetry about y axis, the boundary condition along the y axis is specified as the roller which allows the displacement in the y direction and the boundary condition at all other nodes is specified to be free. And material number.4 representing hard rock is specified for all blocks since the Core region belongs to the hard rock layer as shown in Figure 7.1.

Table 7.1 shows the listing of input file, CORE.Rgn, which has been prepared according to the PRESMAP-2D Model 1 in Section 7.2.1 of User's Manual. Note that the format of the PRESMAP-2D output file is the same as that of Mesh File in SMAP-2D User's Manual. Graphical outputs are shown in Figure 7.5.

Table 7.1 Listing of input file CORE.Rgn

```
* INPUT DATA FOR PRESMAP-2D MODEL 1
* CARD 1.1
  PD-2 CORE REGION GENERATION
* CARD 1.2
* IP
  0
* CARD 1.3
* NBLOCK  NBNODE  NSNEL  CMFAC
    12      30      1     1.0
* CARD 1.4
* NBX  NBY  MIDX  MIDY  NF  NSNODE
   3    4     0     0    1    1
* CARD 2.1
* NODE  X         Y
   1   0.0       4.74
   2   0.0       3.16
   3   0.0       1.58
   4   0.0       0.0
   5   0.0      -3.77
   6   0.684     4.695
   7   0.76     -3.7579
   8   1.356     4.562
   9   1.488     2.819
   10  1.594     1.425
   11  1.702     0.0
   12  1.517    -3.722
   13  2.005     4.341
   14  2.273    -3.662
   15  2.62      4.038
   16  2.9204    2.4907
   17  3.157     1.273
   18  3.404     0.0
   19  3.025    -3.577
   20  3.19      3.66
   21  3.776    -3.47
   22  3.705     3.205
   23  4.157     2.69
   24  4.538     2.12
   25  4.783     1.623
   26  4.962     1.097
   27  5.07      0.5534
   28  5.106     0.0
   29  4.96     -1.693
   30  4.524    -3.337
```

```
* ====================================================
* CARD 3.1
* BLNAME
  BLOCK 1
* CARD 3.2
* IBLNO
    1
* CARD 3.3
* I1  I2  I3  I4  M5  M6  M7  M8  M9
   8   1   2   9   6   0   0   0   0
* CARD 3.4
* IBASE IB1 IB2 IB3 IB4 IB5 IB6 IB7 IB8
    12    12  13  13  12  12  13  12  12
* CARD 3.5
* MATNO NDX NDY  KS  KF
    4     2   2   0   1
* CARD 3.6
* NFSIDE
    0
* ====================================================
  BLOCK 2
  2
  9  2  3  10 0  0  0  0  0
  12 12 13 13 12 12 13 12 12
  4  2  2  0  1
  0
* ====================================================
  BLOCK 3
  3
  10 3  4  11 0  0  0  0  0
  12 12 13 13 12 12 13 12 12
  4  2  2  0  1
  0
* ====================================================
  BLOCK 4
  4 3.337
  11 4  5  12 0  0  7  0  0
  12 12 13 13 12 12 13 12 12
  4  2  6  0  1
  0
* ====================================================
  BLOCK 5
  5
  15 8  9  16 13 0  0  0  0
  12 12 12 12 12 12 12 12 12
  4  2  2  0  1
  0
```

```
* =====================================================
  BLOCK 6
  6
  16 9  10 17 0  0  0  0  0
  12 12 12 12 12 12 12 12 12
  4  2  2  0  1
  0
* =====================================================
  BLOCK 7
  7
  17 10 11 18 0  0  0  0  0
  12 12 12 12 12 12 12 12 12
  4  2  2  0  1
  0
* =====================================================
  BLOCK 6
  6
  16 9  10 17 0  0  0  0  0
  12 12 12 12 12 12 12 12 12
  4  2  2  0  1
  0
* =====================================================
  BLOCK 7
  7
  17 10 11 18 0  0  0  0  0
  12 12 12 12 12 12 12 12 12
  4  2  2  0  1
  0
* =====================================================
  BLOCK 8
  8
  18 11 12 19 0  0  14 0  0
  12 12 12 12 12 12 12 12 12
  4  2  6  0  1
  0
* =====================================================
  BLOCK 9
  9
  22 15 16 24 20 0  0  23 0
  12 12 12 12 12 12 12 12 12
  4  2  2  0  1
  0
```

```
* =====================================================
  BLOCK 10
  10
  24 16 17 26 0  0  0  25 0
  12 12 12 12 12 12 12 12 12
  4  2  2  0  1
  0
* =====================================================
  BLOCK 11
  11
  26 17 18 28 0  0  0  27 0
  12 12 12 12 12 12 12 12 12
  4  2  2  0  1
  0
* =====================================================
  BLOCK 12
  12
  28 18 19 30 0  0  21 29 0
  12 12 12 12 12 12 12 12 12
  4  2  6  0  1
  0
* =====================================================
```

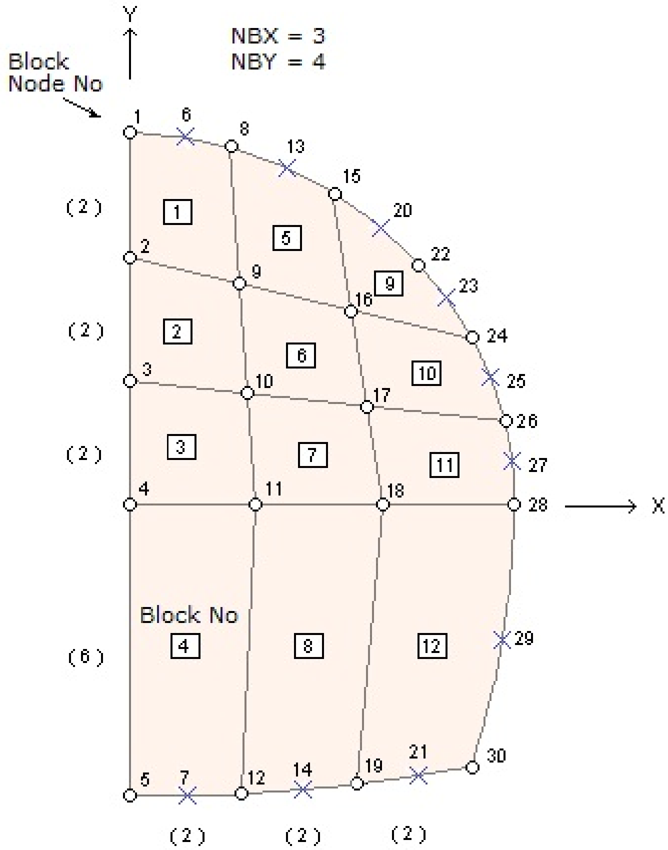


Figure 7.4 Core region block diagram

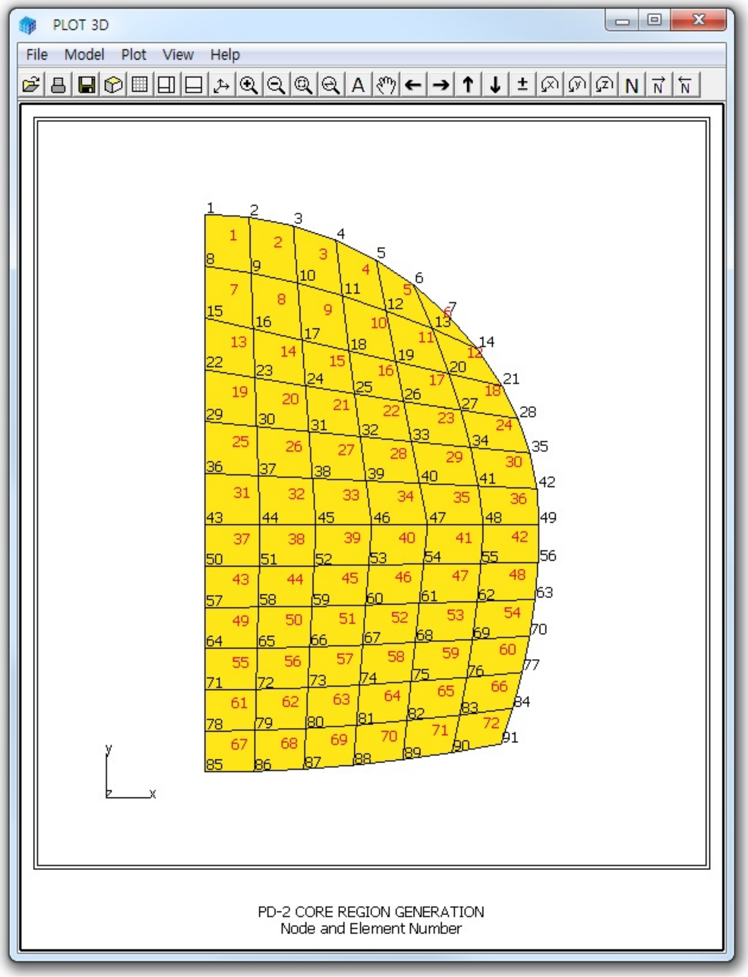


Figure 7.5 Generated element and node numbers for Core region

7.1.1.2 Far-Field Region Mesh Generation

Figure 7.6 shows the block diagram for the Far-field region. Two blocks are used in the horizontal direction (NBX=2) and 6 blocks in the vertical direction (NBY=6). Block numbers 1 and 7 represent weathered soil (MATNO=1). Block numbers 2 and 8 represent weathered rock (MATNO= 2). Block numbers 3 and 9 represent soft rock (MATNO=3). And the rest of blocks represent hard rock (MATNO=4) except Block numbers 4 and 5 (MATNO=0). Note that Block numbers 4 and 5 are void blocks. Elements in this void blocks are not generated in Far-field region, but will be generated in Core and Near-field regions.

You can specify the index of each block as for Core region. Side block nodes are used here to make element sizes bigger as the elements are away from the tunnel core. To simulate plane strain condition at the remote boundary, boundary conditions for the left, right, and bottom are specified as the roller.

Table 7.2 shows the listing of input file, FAR.Rgn, which has been prepared according to the PRESMAP-2D Model 1 in Section 7.2.1 of User's Manual. Generated element and node numbers are shown in Figure 7.7. Note that the Far-field element number starts from 337, considering that there are 336 elements in Core and Near-field regions.

Table 7.2 Listing of input file FAR.Rgn

```
* INPUT DATA FOR PRESMAP-2D MODEL 1
* CARD 1.1
  PD-2 FAR-FIELD REGION GENERATION
* CARD 1.2
* IP
  0
* CARD 1.3
* NBLOCK  NBNODE  NSNEL  CMFAC
   12       31      337    1.0
* CARD 1.4
* NBX  NBY  MIDX  MIDY  NF  NSNODE
   2    6    0     0    1     1
* CARD 2.1
* NODE   X       Y
   1    0.0    21.94
   2    0.0    17.74
   3    0.0    13.44
   4    0.0    9.94
   5    0.0    0.0
   6    0.0   -12.0
   7    0.0   -19.2
   8    0.0   -30.0
   9    14.0   21.94
   10   14.0   17.74
   11   14.0   13.44
   12   14.0   9.94
   13   14.0   0.0
   14   14.0  -12.0
   15   14.0  -19.2
   16   14.0  -30.0
   17   21.2   21.94
   18   21.2   17.74
   19   21.2   13.44
   20   21.2   9.94
   21   21.2   0.0
   22   21.2  -12.0
   23   21.2  -30.0
   24   32.0   21.94
   25   32.0   17.74
   26   32.0   13.44
   27   32.0   9.94
   28   32.0   0.0
   29   32.0  -12.0
   30   32.0  -19.2
   31   32.0  -30.0
```

```
* ====================================================
* CARD 3.1
* BLNAME
  BLOCK 1
* CARD 3.2
* IBLNO
    1
* CARD 3.3
* I1 I2 I3 I4 M5 M6 M7 M8 M9
   9  1  2 10  0  0  0  0  0
* CARD 3.4
* IBASE IB1 IB2 IB3 IB4 IB5 IB6 IB7 IB8
    12   12  13  13  12  12  13  12  12
* CARD 3.5
* MATNO  NDX  NDY  KS   KF
    1     6    1    0    1
* CARD 3.6
* NFSIDE
    0
* ====================================================
  BLOCK 2
  2
  10 2  3  11 0  0  0  0  0
  12 12 13 13 12 12 13 12 12
  2  6  1  0  1
  0
* ====================================================
  BLOCK 3
  3
  11 3  4  12 0  0  0  0  0
  12 12 13 13 12 12 13 12 12
  3  6  2  0  1
  0
* ====================================================
  BLOCK 4
  4
  12 4  5  13 0  0  0  0  0
  12 12 13 13 12 12 13 12 12
  0  6  6  0  1
  0
* ====================================================
  BLOCK 5
  5
  13 5  6  14 0  0  0  0  0
  12 12 13 13 12 12 13 12 12
  0  6  6  0  1
  0
```

```
* ======================================================
  BLOCK 6
  6
  14 6  8  16 0  7  0  15 0
  12 12 13 15 14 12 13 14 12
  4  6  4  0  1
  0
* ======================================================
  BLOCK 7
  7
  24 9  10 25 17 0  18 0  0
  12 13 12 12 13 12 12 12 13
  1  4  1  0  1
  0
* ======================================================
  BLOCK 8
  8
  25 10 11 26 18 0  19 0  0
  12 13 12 12 13 12 12 12 13
  2  4  1  0  1
  0
* ======================================================
  BLOCK 9
  9
  26 11 12 27 19 0  20 0  0
  12 13 12 12 13 12 12 12 13
  3  4  2  0  1
  0
* ======================================================
  BLOCK 10
  10
  27 12 13 28 20 0  21 0  0
  12 13 12 12 13 12 12 12 13
  4  4  6  0  1
  0
* ======================================================
  BLOCK 11
  11
  28 13 14 29 21 0  22 0  0
  12 13 12 12 13 12 12 12 13
  4  4  6  0  1
  0
```

```
* =====================================================
  BLOCK 12
  12
  29 14 16 31 22 15 23 30 0
  12 13 12 14 15 12 12 14 13
  4  4  4  0  1
  0
* =====================================================
* END OF DATA
```

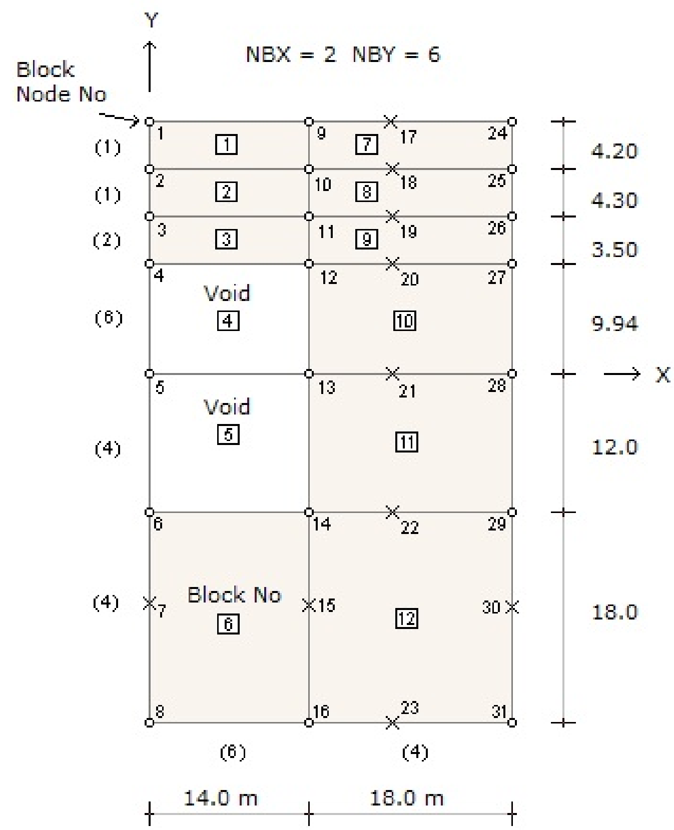


Figure 7.6 Far-field region block diagram

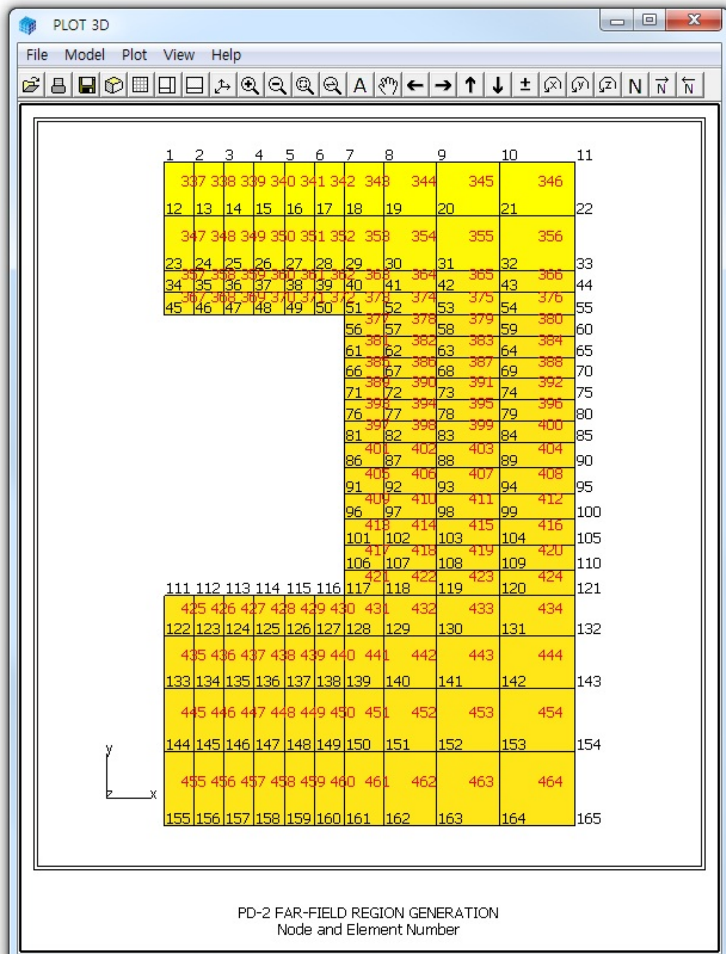


Figure 7.7 Generated element and node numbers for Far-field region

7.1.2 Model 2

Model 2 is the special pre-processor developed to model Near-field region around the underground openings. The Near-field region shown in Figure 7.3 is taken here as an example problem.

As shown in Figure 7.8, eight subregions are used to construct the Near-field region. And each subregion consists of three blocks. Then each block is further divided in radial and tangential directions. For example, Block number 5 in Subregion 2 has 5 elements in radial direction and 6 elements in the tangential direction. Note that element sizes in the third block increase gradually in the radial direction. Parameters specific to each subregion are tabulated in Table 7.3.

Table 7.4 shows the listing of input file, NEAR.Rgn, which has been prepared according to the PRESMAP-2D Model 2 in Section 7.2.2 of User's Manual. Generated element mesh is shown in Figure 7.9.

Table 7.3 Parameters specific in Near-field region

NSUBR = 8 NDRF = 2 NDRS = 5 NDRT = 4
DRF = 0.15 m DRS = 2.85 m

Subregion	ISBTYPE	LSFTYPE	NSEG
1	1	1	6
2	1	1	6
3	0	1	2
4	0	1	2
5	0	1	2
6	0	1	2
7	0	1	2
8	0	1	2

Global block numbers are in order from surface

to outer edge and counterclockwise.

Local block numbers in each subregion are in order
from surface to outer edge.

Example : In Subregion 2,
First block = 4 , Second block = 5, Third block = 6

Table 7.4 Listing of input file NEAR.Rgn

```
* INPUT DATA FOR PRESMAP-2D MODEL 2
* CARD 1.1
  PD-2 NEAR-FIELD MESH GENERATION
* CARD 1.2
* IP
  0
* CARD 1.3
* NSNEL  NSNODE  NF  CMFAC
   73      67    1   1.0
* CARD 1.4
* NSURB  NDRF  NDRS  NDRT  DRF   DRS
    8     2     5     4    0.15  2.85
* ====================================================
* CARD 2.1
* SUBNAME
  SUBREGION 1
* CARD 2.2
* ISUBNO
     1
* CARD 2.3
* ISBTYPE  LSFTYPE  NSEC
     1        1       6
* CARD 2.4.2 (LSFTYPE = 1)
* R       Xo   Yo     TA    TB
  23.86  0.0  20.09  270.  280.93
* (ISBTYPE = 1)
* CARD 2.5.3
* Xc    Yc    Xd    Yd
  0.0   -12.  14.0  -12.
* CARD 2.6
* IBASE1  IBASE2  IBASE3
    12      12      12
* IBb  IBa  IBc  IBd  IBab  IBac  IBcd  Ibbd
   12   13   13   12   12    13    12    12
* CARD 2.7
* MATNO1  KS1   KF1
      4    0     1
* MATNO2  KS2   KF2
      4    0     1
* MATNO3  KS3   KF3
      4    0     1
* CARD
* NFSIDE
      0
```

```
* =====================================================
      SUBREGION 2
      2
      1  1  6
      9.86  -4.754  0.0  340.22  360.
      14.0  -12.  14.0   0.0
      12 12 12
      12 12 12 12 12 12 12 12
      4  0  1
      4  0  1
      4  0  1
      0
* =====================================================
      SUBREGION 3
      3
      0  1  2
      4.24  0.866  0.0  0.0  15.0
      1
      14.0
      0
      14.0  3.31
      12 12 12
      12 12 12 12 12 12 12 12
      4  0  1
      4  0  1
      4  0  1
      0
* =====================================================
      SUBREGION 4
      4
      0  1  2
      4.24  0.866  0.0  15.0  30.0
      0
      14.0  3.31
      0
      14.0  6.63
      12 12 12
      12 12 12 12 12 12 12 12
      4  0  1
      4  0  1
      4  0  1
      0
```

```
* =====================================================
      SUBREGION 5
      5
      0  1  2
      5.24  0.0  -0.5  30.0  45.0
      0
      14.0  6.63
      0
      14.0  9.94
      12 12 12
      12 12 12 12 12 12 12 12
      4  0  1
      4  0  1
      4  0  1
      0
* =====================================================
      SUBREGION 6
      6
      0  1  2
      5.24  0.0  -0.5  45.0  60.0
      0
      14.0  9.94
      0
      9.33  9.94
      12 12 12
      12 12 12 12 12 12 12 12
      4  0  1
      4  0  1
      4  0  1
      0
* =====================================================
      SUBREGION 7
      7
      0  1  2
      5.24  0.0  -0.5  60.  75.0
      0
      9.33  9.94
      0
      4.67  9.94
      12 12 12
      12 12 12 12 12 12 12 12
      4  0  1
      4  0  1
      4  0  1
      0
```

```
* =====================================================
      SUBREGION 8
      8
      0  1  2
      5.24  0.0  -0.5  75.0  90.0
      0
      4.67  9.94
      0
      0.0   9.94
      12 12 12
      13 12 12 13 12 12 12 13
      4  0  1
      4  0  1
      4  0  1
      0
* =====================================================
```

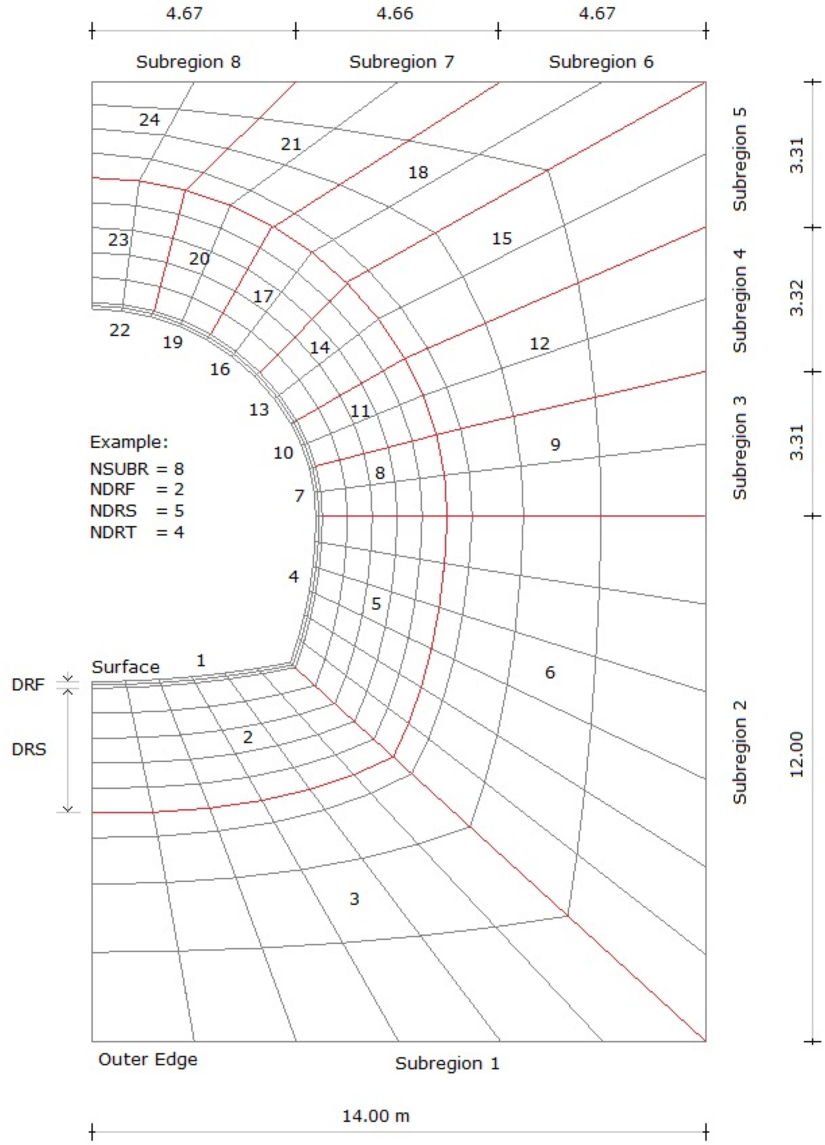


Figure 7.8 Subregions in Near-field region

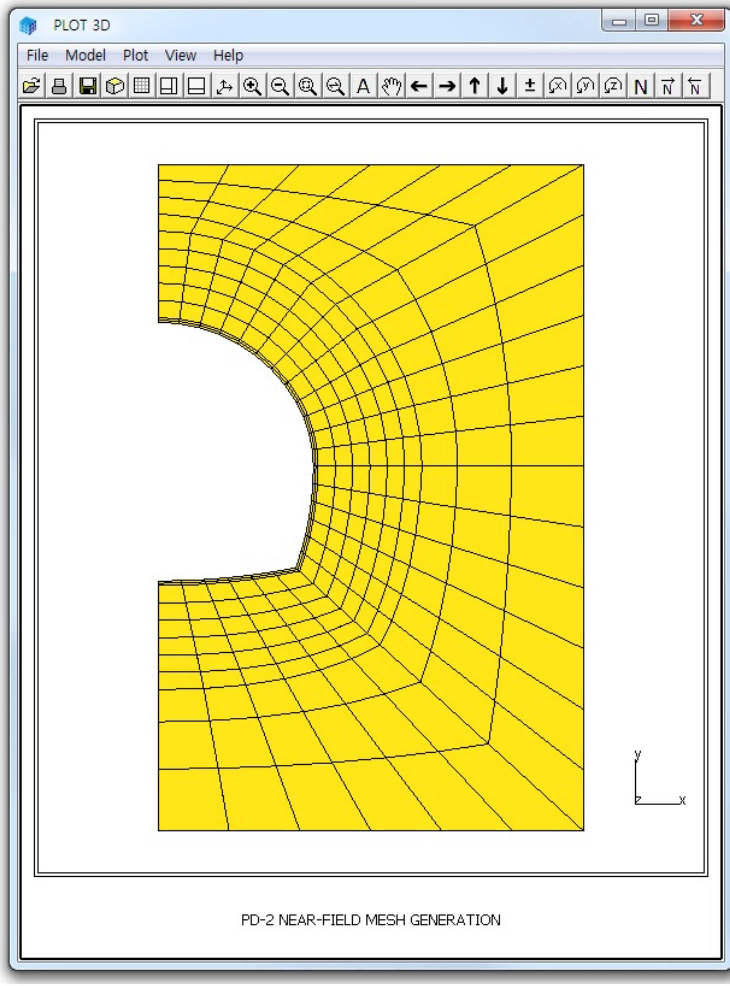


Figure 7.9 Generated mesh for Near-field region

7.1.3 Model 3

Model 3 is a useful pre-processor to generate triangular or rectangular meshes. It is much easier to use compared to Models 1 and 2.
But you have to specify the boundary codes manually.

Figure 7.10 shows block diagram for Model 3 example problem.
Block numbers 1 to 5 are 4 x 4 rectangular shape and Block number 6 is the 9-element triangular shape.

Table 7.5 shows the listing of input file, GM3.Rgn, which has been prepared according to the PRESMAP-2D Model 3 in Section 7.2.3 of User's Manual. Generated element and node numbers are shown in Figure 7.11.

Table 7.5 Listing of input file GM3.Rgn

```
* INPUT DATA FOR PRESMAP-2D MODEL 3
* CARD 1.1
  MESH GENERATION SURROUNDING PIPE ( GM3 )
* CARD 1.2
* IP
  0
* CARD 1.3
* NBLOCK  NBNODE  NSNEL  NSNODE  CMFAC
     6       12     171     1       1.0
* CARD 2.1
* NODE      X            Y
   1  .324920E+02  .100000E+03
   2  .809020E+02  .587790E+02
   3  .100000E+03  .000000E+00
   4  .809020E+02 -.587790E+02
   5  .324920E+02 -.100000E+03
   6   100.          100.
   7   125.          50.
   8   150.          0.
   9   125.         -50.
   10  100.         -100.
   11  200.          100.
   12  175.          50.
* =======================================================
* CARD 3.1
* =======================================================
* IBLNO  IBLTYPE  MATNO  KS  KF
    1       2       2     0   1
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
   6  1  2  7  0  0  0  0  0   0   0   0   0   0   0   0
* =======================================================
* IBLNO  IBLTYPE  MATNO  KS  KF
    2       2       2     0   1
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
   7  2  3  8  0  0  0  0  0   0   0   0   0   0   0   0
* =======================================================
* IBLNO  IBLTYPE  MATNO  KS  KF
    3       2       2     0   1
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
   8  3  4  9  0  0  0  0  0   0   0   0   0   0   0   0
```

```
* =======================================================
* IBLNO  IBLTYPE  MATNO  KS  KF
    4       2       2     0   1
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
   9  4  5 10  0  0  0  0  0   0   0   0   0   0   0   0
* =======================================================
* IBLNO  IBLTYPE  MATNO  KS  KF
    5       2       2     0   1
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
  11  6  7 12  0  0  0  0  0   0   0   0   0   0   0   0
* =======================================================
* IBLNO  IBLTYPE  MATNO  KS  KF
    6       4       2     0   1
* FOR IBLTYPE = 2
* I1 I2 I3 M4 M5 M6 M7 M8 M9 M10 M11 M12
   7  8 12  0  0  0  0  0  0   0   0   0
* =======================================================
```

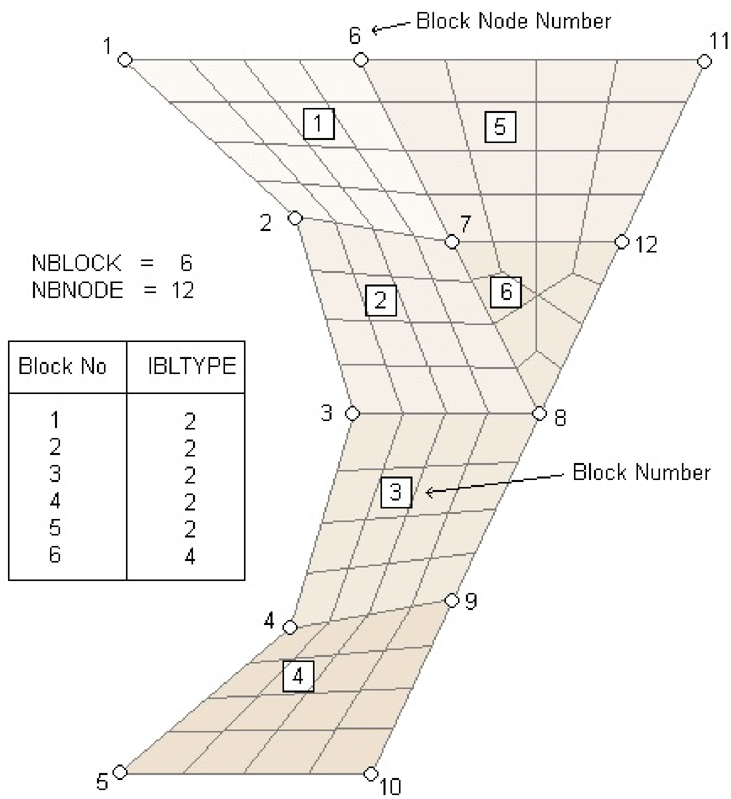


Figure 7.10 Block diagram for Model 3 example problem

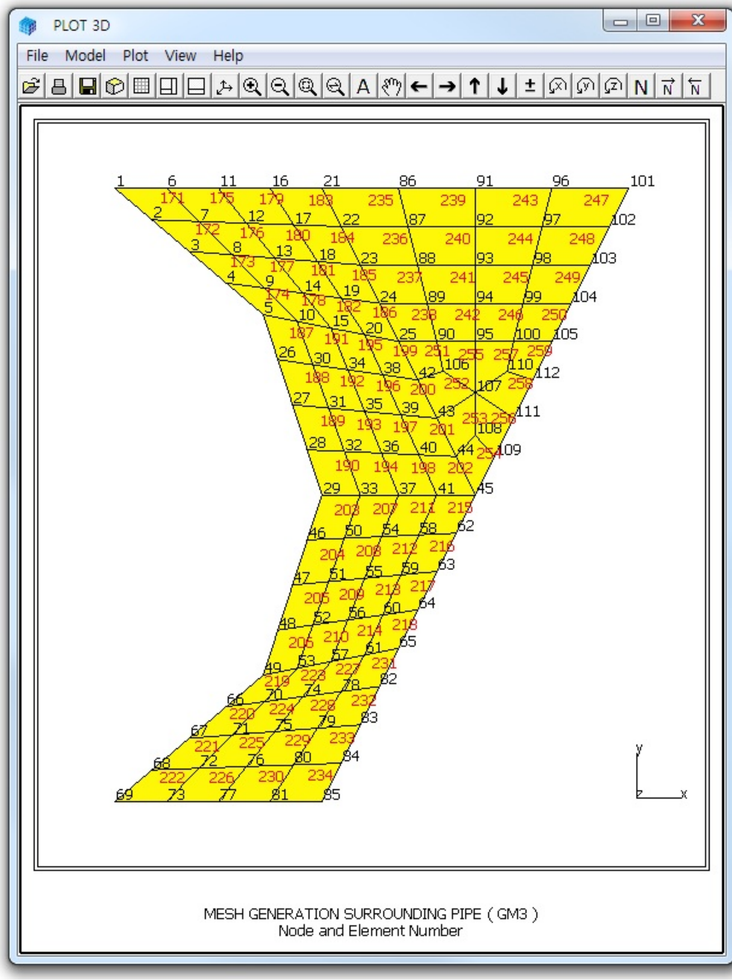


Figure 7.11 Generated element and node numbers for Model 3 example problem

7.1.4 Model 4

Model 4 is a useful pre-processor to generate horizontally layered dams or embankments. It is easy to use but the boundary codes should be specified manually.

As Model 4 example problem, an embankment with 3 layers is considered. Table 7.6 shows the listing of input file, GM4.Rgn, which has been prepared according to the PRESMAP-2D Model 4 in Section 7.2.4 of User's Manual. Generated element and node numbers are shown in Figure 7.12.

Table 7.6 Listing of input file GM4.Rgn

```
* CARD 1.1
* TITLE
  EXAMPLE PROBLEM FOR PRESMAP-2D MODEL 4
* CARD 1.2
* NLAYER  NDIV  ITRANGL
   3       3     1
* CARD 1.3
* NSNEL  NSNODE  CMFAC
   1      1       1.0
* CARD 2.1
* XB1  YB1  YB2  XB2
  0.0  3.0  0.0  12.
* CARD 3.1
* MATNO  KS  KF
   3     0   1
* END OF DATA
```

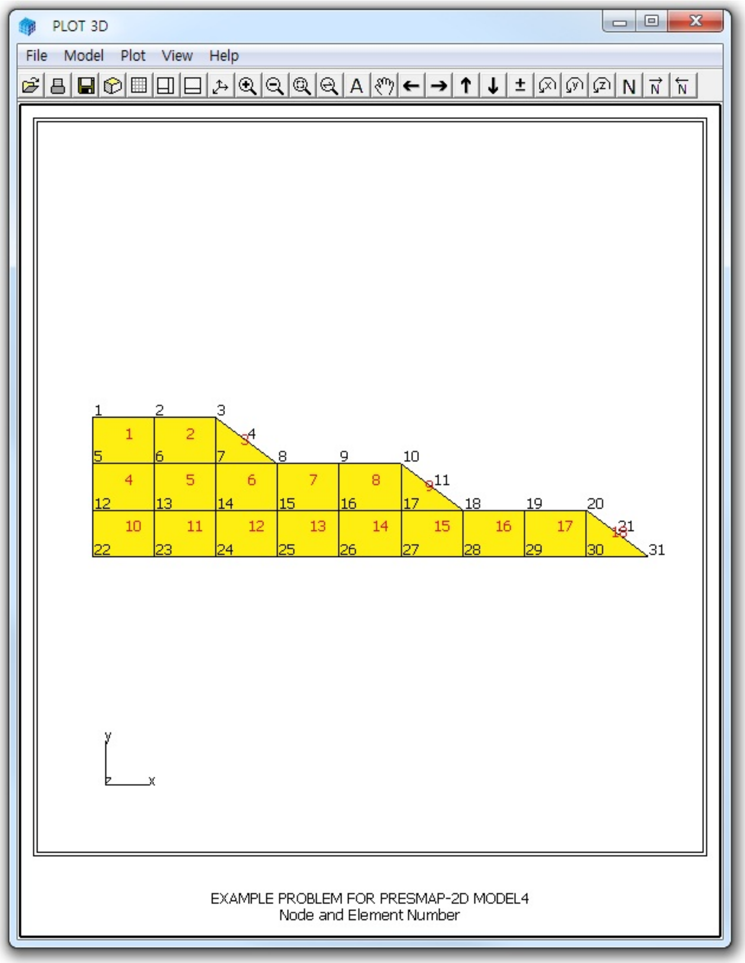


Figure 7.12 Generated element and node numbers for Model 4 example problem

7.2 NATM-2D

NATM-2D is the special pre-processing program to generate automatically two-dimensional finite element meshes and boundary conditions for NATM tunnels. NATM-2D has four different models:

Model 1	Single Tunnel (Half Section)
Model 2	Single Tunnel (Full Section)
Model 3	Two Tunnel (Symmetric Section)
Model 4	Two Tunnel (Unsymmetric Section)

Once you have executed NATM-2D, you will obtain following files:

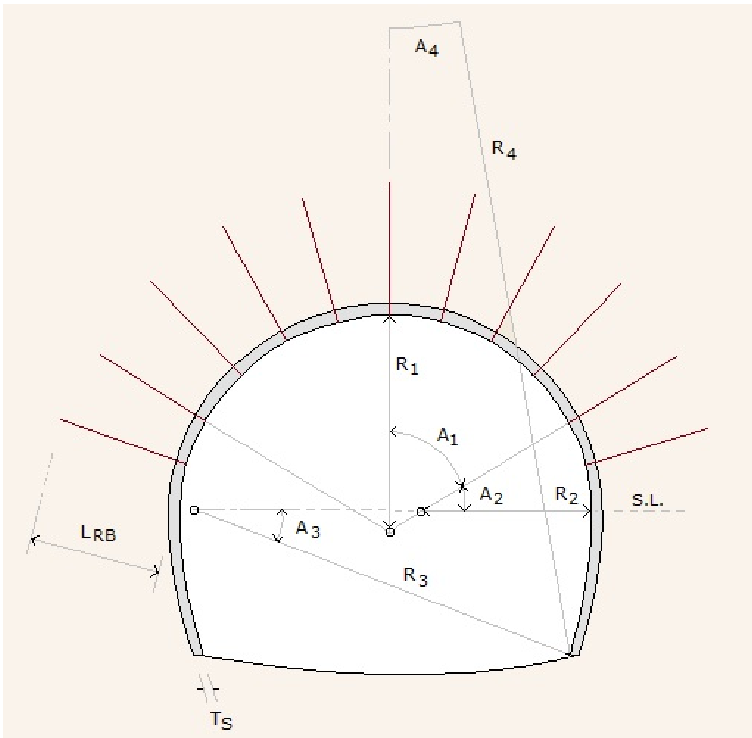
Output File	Mesh File including all elements (Continuum, Beam, and Truss). Output File is the user specified name.
BEAM.Dat	Mesh File including only beam elements.
TRUSS.Dat	Mesh File including only truss elements.
AD.Dat	Card Group 8 in Main File representing default element activities for upper and lower parts of Core, Shotcrete, and Rock Bolt including Joint and Lining elements.
LINING.Dat	Mesh File for Beam-Spring Lining Analysis. This file will be generated only for ILNCOUPL=1.

A typical PD2 tunnel shape is chosen here to illustrate mesh generation using NATM-2D as shown in Figure 7.13. For each model, we will present:

- Listing of input file
- Schematic tunnel section view
- Graphical output of finite element mesh

Table 7.7 Listing of input file PD2-1.Dat

```
* CARD 1.1
* TITLE
  NATM-2D MODEL 1 EXAMPLE PROBLEM
* CARD 1.2
* IUNIT
   2
* CARD 1.3
* MODEL  IGEN  IEXMESH   ILNCOUPL
   1      0     0                0
* CARD 2.1
* HT    HL   W    DELTAX  DELTAX  NDYMAX
  21.94 30.  20.  2.0     2.0     40
* CARD 3.1
* NLAYER
   4
* CARD 3.2
* LAYERNO   H     KF
   1        4.2   1
   2        4.3   1
   3        3.5   1
   4        39.94 1
* CARD 4.1
* R1    A1    R2    A2  R3    A3     R4    GR    GA
  5.24  60.   4.24  30. 9.86  19.781 23.86 1.0   0.5
* CARD 4.2
* INVSHOT  TS
  0        0.3
* CARD 4.3
* NUMRB  LRB    LSPACING  TSPACING  NSRB
  11     3.0    0.8       1.2        2
* CARD 5.1
* LDTYPE  DGW  GAMAW
  1       2.0  1.0
* END OF DATA
```

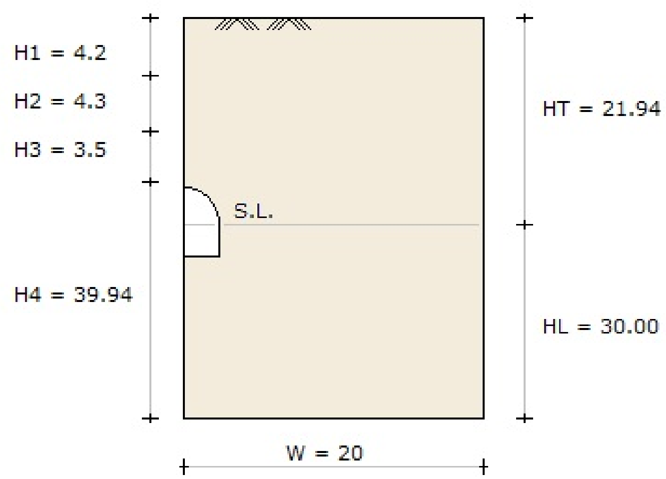


R_1	=	5.24 M	A_1 =	60°	
R_2	=	4.24 M	A_2 =	30°	
R_3	=	9.86 M	A_3 =	19.781°	
R_4	=	23.86 M			

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)	=	15 Cm
Thickness of Liner (TL)	=	30 Cm
Reinforcing Bar Area (ASI)	=	22 Cm^2
Reinforcing Bar Area (ASO)	=	22 Cm^2

Figure 7.13 Tunnel dimensions used for example problem

MODEL=1   Single Tunnel (Half Section)



DELTAX = 2.0   DELTAY = 2.0   NDYMAX = 40

Figure 7.14 Schematic tunnel section view for Model 1 example

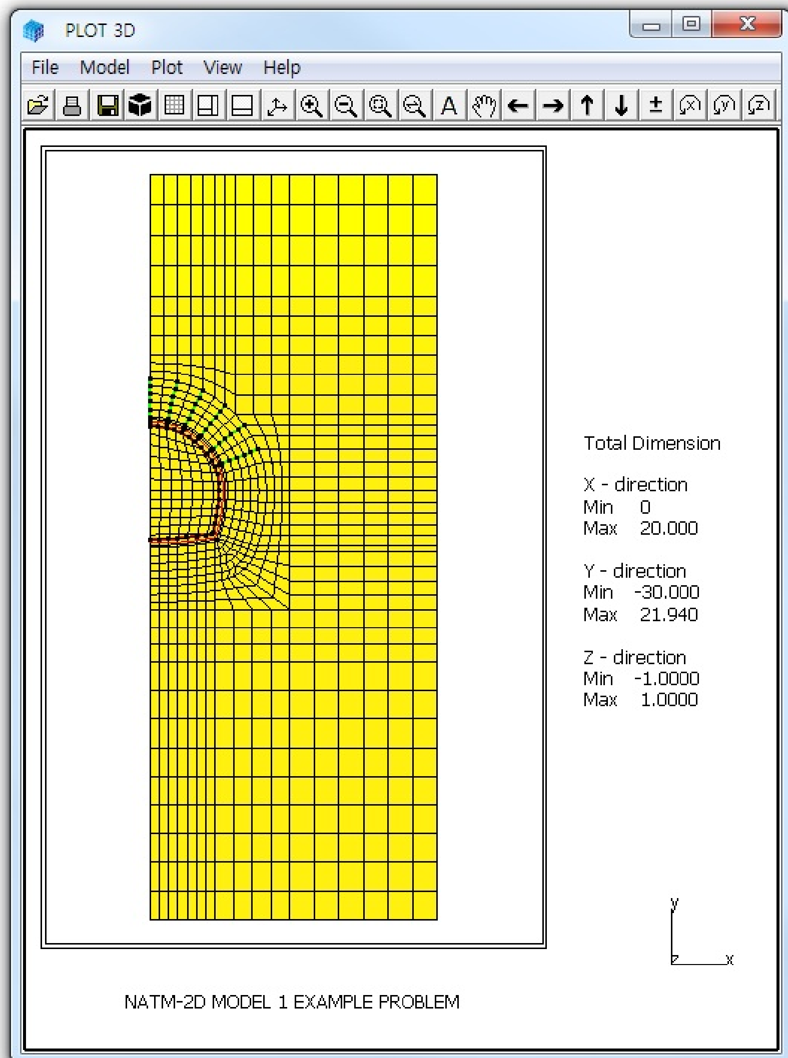


Figure 7.15 Generated finite element mesh for Model 1 example

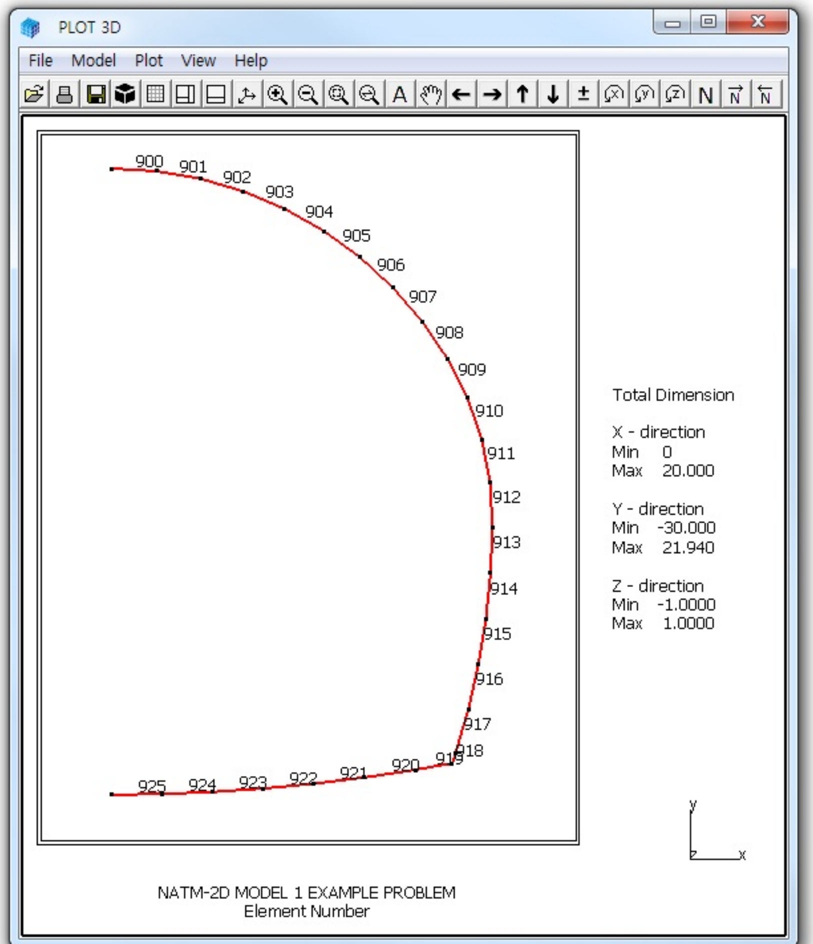


Figure 7.16 Generated beam element number for Model 1 example

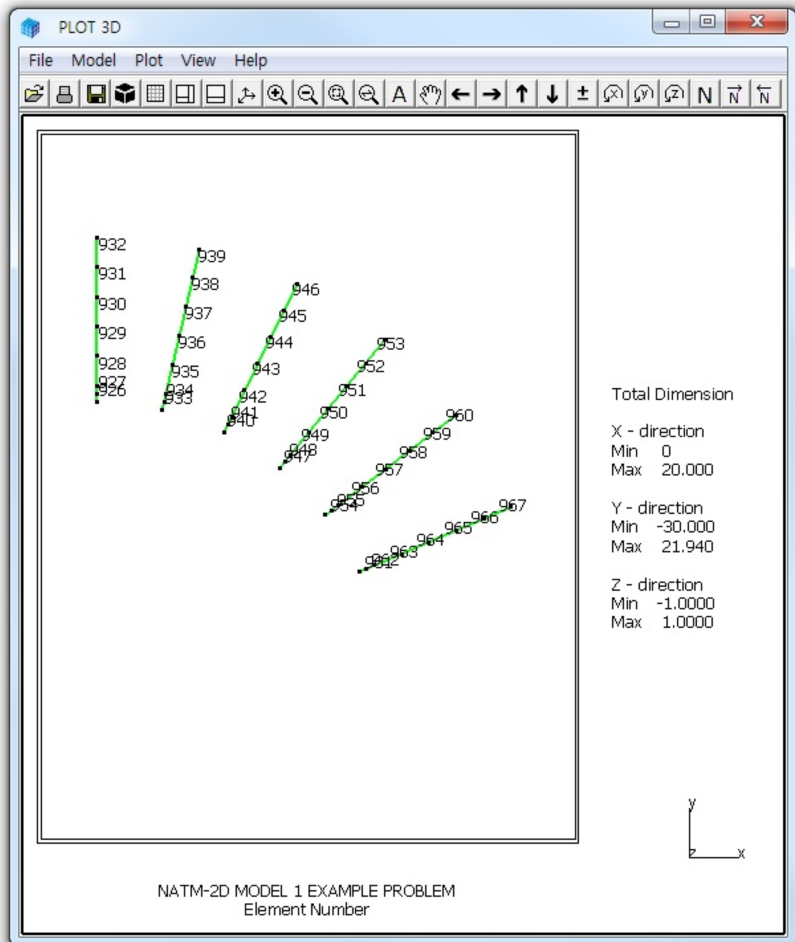
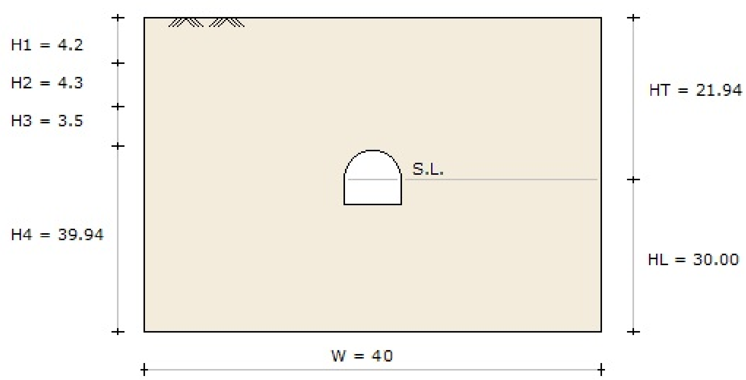


Figure 7.17 Generated truss element number for Model 1 example

Table 7.8 Listing of input file PD2-2.Dat

```
* CARD 1.1
* TITLE
  NATM-2D MODEL 2 EXAMPLE PROBLEM
* CARD 1.2
* IUNIT
   2
* CARD 1.3
* MODEL  IGEN  IEXMESH      ILNCOUPL
   2     0     0            0
* CARD 2.1
* HT    HL    W    DELTAX  DELTAX  NDYMAX
  21.94 30.   40.  2.0     2.0     40
* CARD 3.1
* NLAYER
   4
* CARD 3.2
* LAYERNO   H     KF
   1        4.2   1
   2        4.3   1
   3        3.5   1
   4        39.94 1
* CARD 4.1
* R1    A1    R2    A2  R3    A3     R4    GR    GA
  5.24  60.   4.24  30. 9.86  19.781 23.86 1.0   0.5
* CARD 4.2
* INVSHOT  TS
  0        0.3
* CARD 4.3
* NUMRB  LRB    LSPACING  TSPACING  NSRB
  11     3.0    0.8       1.2        2
* CARD 5.1
* LDTYPE  DGW  GAMAW
  1       2.0  1.0
* END OF DATA
```

MODEL=2 Single Tunnel (Full Section)



DELTAX = 2.0 DELTAY = 2.0 NDYMAX = 40

Figure 7.18 Schematic tunnel section view for Model 2 example

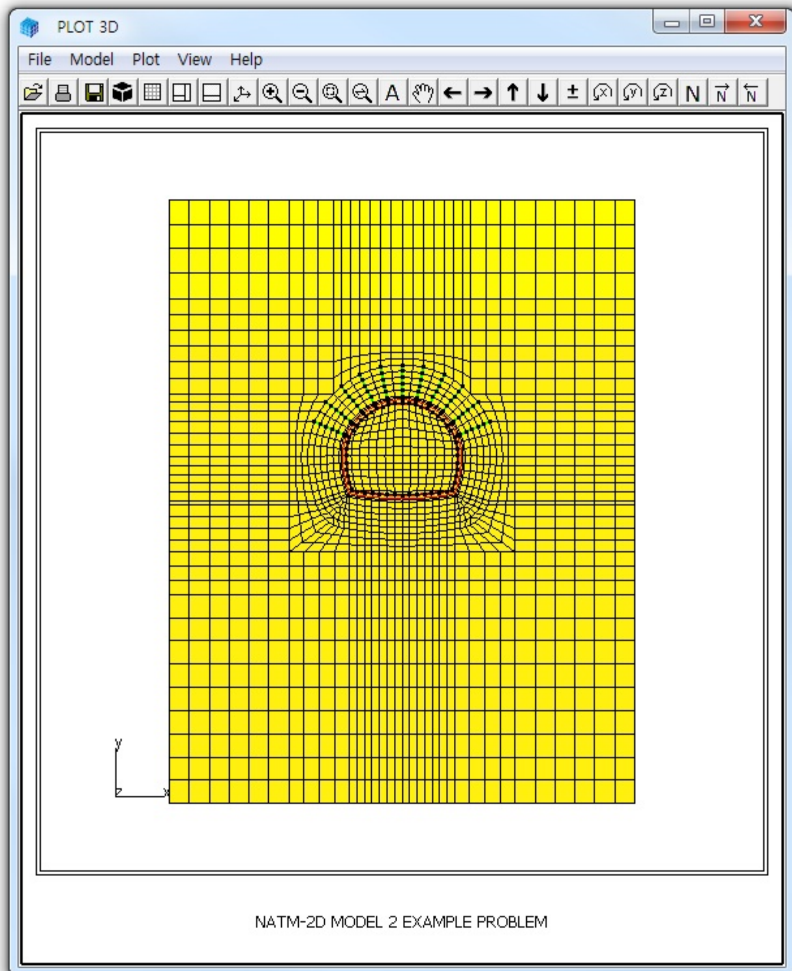
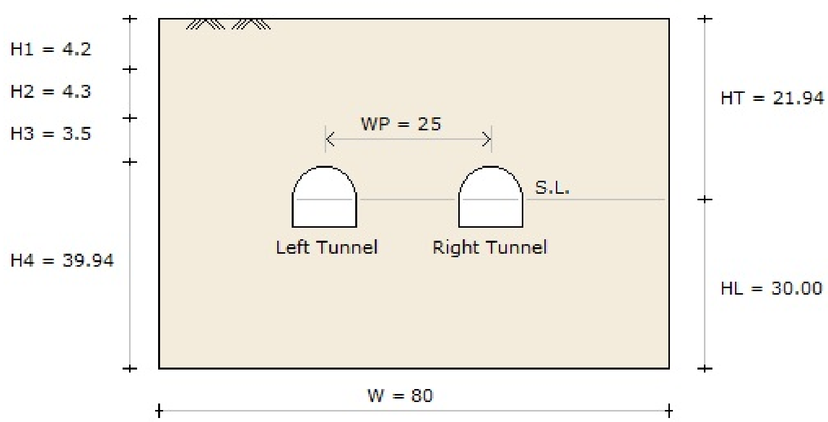


Figure 7.19 Generated finite element mesh for Model 2 example

Table 7.9 Listing of input file PD2-3.Dat

```
* CARD 1.1
* TITLE
  NATM-2D MODEL 3 EXAMPLE PROBLEM
* CARD 1.2
* IUNIT
   2
* CARD 1.3
* MODEL  IGEN  IEXMESH   ILNCOUPL
   3     0     0        0
* CARD 2.1
* HT    HL    W    WP     DELTAX  DELTAY  NDYMAX
  21.94 30.  80.  25.    2.0     2.0     40
* CARD 3.1
* NLAYER
   4
* CARD 3.2
* LAYERNO   H     KF
   1        4.2   1
   2        4.3   1
   3        3.5   1
   4        39.94 1
* CARD 4.1
* R1    A1    R2    A2  R3    A3     R4    GR    GA
  5.24  60.   4.24  30. 9.86  19.781 23.86 1.0   0.5
* CARD 4.2
* INVSHOT  TS
  0        0.3
* CARD 4.3
* NUMRB  LRB    LSPACING  TSPACING  NSRB
  11     3.0    0.8       1.2        2
* CARD 5.1
* LDTYPE  DGW  GAMAW
  1       2.0  1.0
* END OF DATA
```

MODEL=3 Two Tunnel (Symmetric Section)



DELTAX = 2.0 DELTAY = 2.0 NDYMAX = 40

Figure 7.20 Schematic tunnel section view for Model 3 example

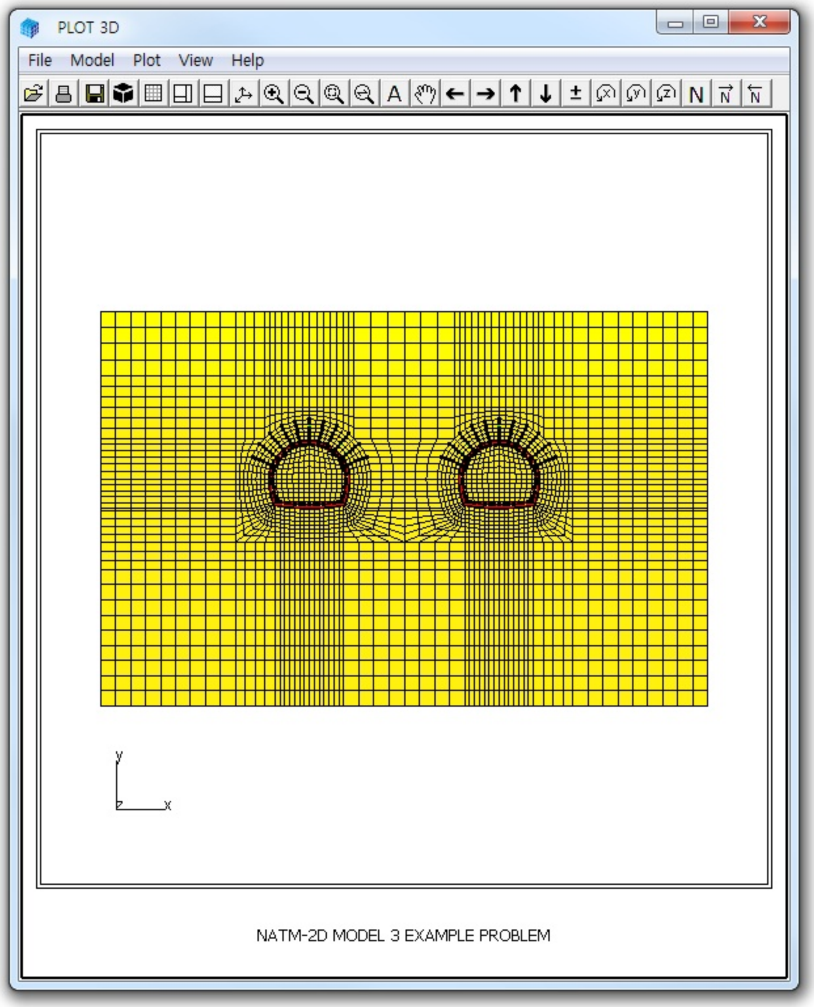
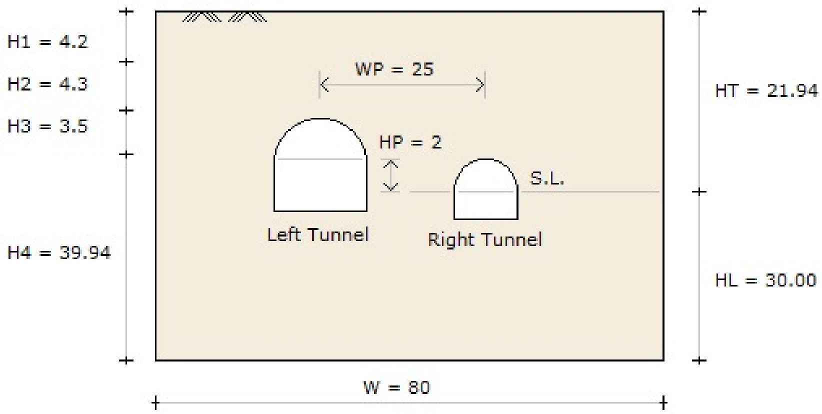


Figure 7.21 Generated finite element mesh for Model 3 example

Table 7.10 Listing of input file PD2-4.Dat

```
* CARD 1.1
* TITLE
  NATM-2D MODEL 4 EXAMPLE PROBLEM
* CARD 1.2
* IUNIT
   2
* CARD 1.3
* MODEL  IGEN  IEXMESH   ILNCOUPL
   4     0     0        0
* CARD 2.1
* HT    HL   W    WP    HP      DELTAX  DELTAY  NDYMAX
  21.94 30.  80.  25.   2.0     2.0     2.0     40
* CARD 3.1
* NLAYER
   4
* CARD 3.2
* LAYERNO   H     KF
   1        4.2   1
   2        4.3   1
   3        3.5   1
   4        39.94 1
* RIGHT TUNNEL
* CARD 4.1
* R1    A1    R2    A2  R3    A3     R4    GR    GA
  5.24  60.   4.24  30. 9.86  19.781 23.86 1.0   0.5
* CARD 4.2
* INVSHOT  TS
  0        0.3
* CARD 4.3
* NUMRB  LRB    LSPACING  TSPACING  NSRB
  11     3.0    0.8       1.2        2
* LEFT TUNNEL
* CARD 4.1
* R1    A1    R2    A2  R3    A3     R4    GR    GA
  7.24  60.   6.24  30. 11.86 21.781 25.86 1.0   0.5
* CARD 4.2
* INVSHOT  TS
  0        0.35
* CARD 4.3
* NUMRB  LRB    LSPACING  TSPACING  NSRB
  15     3.0    0.8       1.2        2
* CARD 5.1
* LDTYPE  DGW  GAMAW
  1       2.0  1.0
* END OF DATA
```

MODEL=4 Two Tunnel (Unsymmetric Section)



DELTAX = 2.0 DELTAY = 2.0
NDYMAX = 40

Right Tunnel Tunnel dimensions are shown in Figure 7.16

Left Tunnel R_1 = 7.24 M A_1 = 60°
R_2 = 6.24 M A_2 = 30°
R_3 = 11.86 M A_3 = 21.781°
R_4 = 25.86 M

Number of Rock Bolts (NUMRB)	=	15
Length of Rock Bolts (LRB)	=	3.0 M
Spacing of Rock Bolts (TSPACING)	=	1.2 M
Thickness of Shotcrete (TS)	=	35 Cm

Figure 7.22 Schematic tunnel section view for Model 4 example

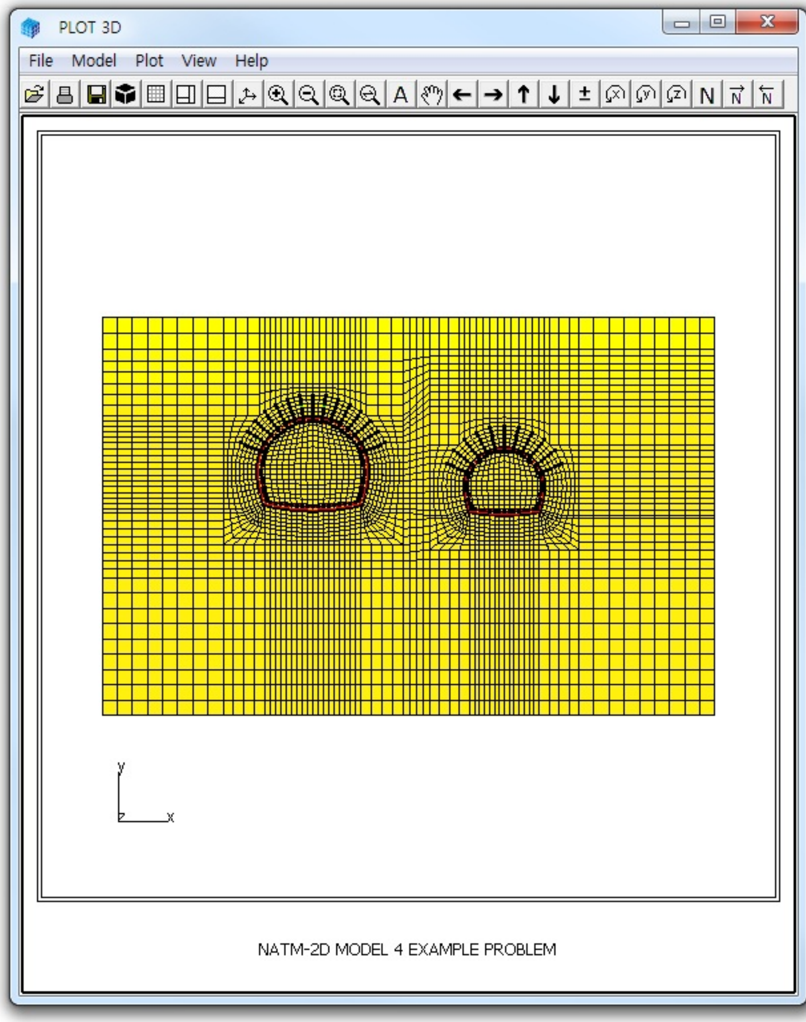


Figure 7.23 Generated finite element mesh for Model 4 example

Table 7.11 Listing of input file Shield.Dat

```
* CARD 1.1
* TITLE
  NATM-2D MODEL 2 FOR SEGMENT LINING
* CARD 1.2
* IUNIT
   2
* CARD 1.3
* MODEL  IGEN  IEXMESH    ILNCOUPL
   2     0     0          1
* CARD 2.1
* HT    HL   W    DELTAX  DELTAX  NDYMAX
  21.94 30.  40.  2.0     2.0     40
* CARD 3.1
* NLAYER
   4
* CARD 3.2
* LAYERNO   H     KF
   1        4.2   1
   2        4.3   1
   3        3.5   1
   4        39.94 1
* CARD 4.1
* R1    A1    R2    A2  R3    A3     R4    GR    GA
  5.3   60.   5.3   60. 5.3   30.    5.3   1.0   0.5
* CARD 4.2
* INVSHOT  TS   TL
  0        0.3  0.3
* NOTE: TUNNEL LINING RADIUS = R1 - TL = 5.3 - 0.3 = 5.0 M
* CARD 4.3
* NUMRB  LRB    LSPACING  TSPACING  NSRB
  11     3.0    0.8       1.2       2
* FOR FINE MESH, USE NSRB = 3
* CARD 5.1
* LDTYPE  DGW  GAMAW  HPRES  VPRES  SUBGK    ITSPR  NUMSJ
  1       2.0  1.0    20.    30.    1.0E+05  1      4
* CARD 5.2
* JOINT LOCATIONS (ANGLES FROM CROWN TOP)
* AJ1    AJ2    AJ3    AJ4
  0      60     120    180
* END OF DATA
```

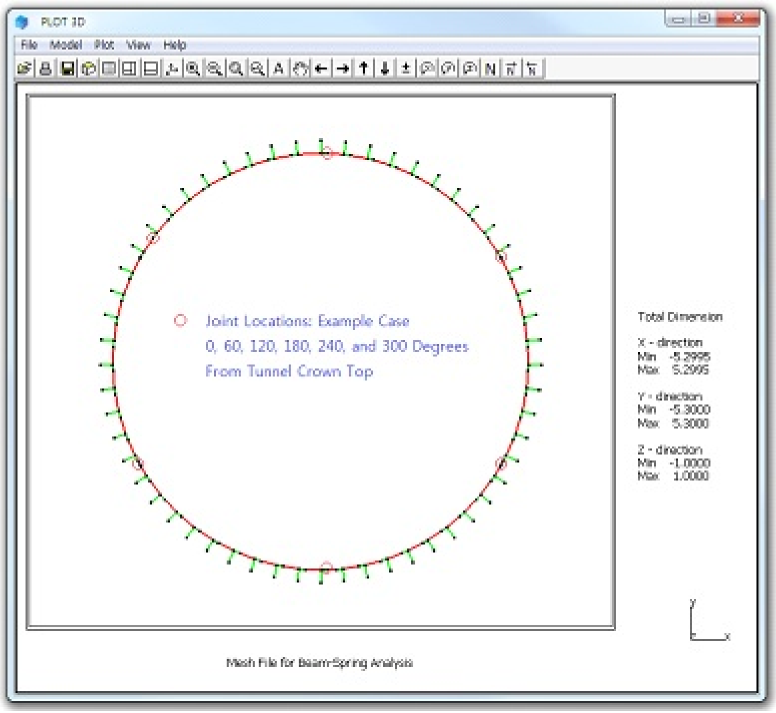


Figure 7.24 Finite element mesh for Model 2-1

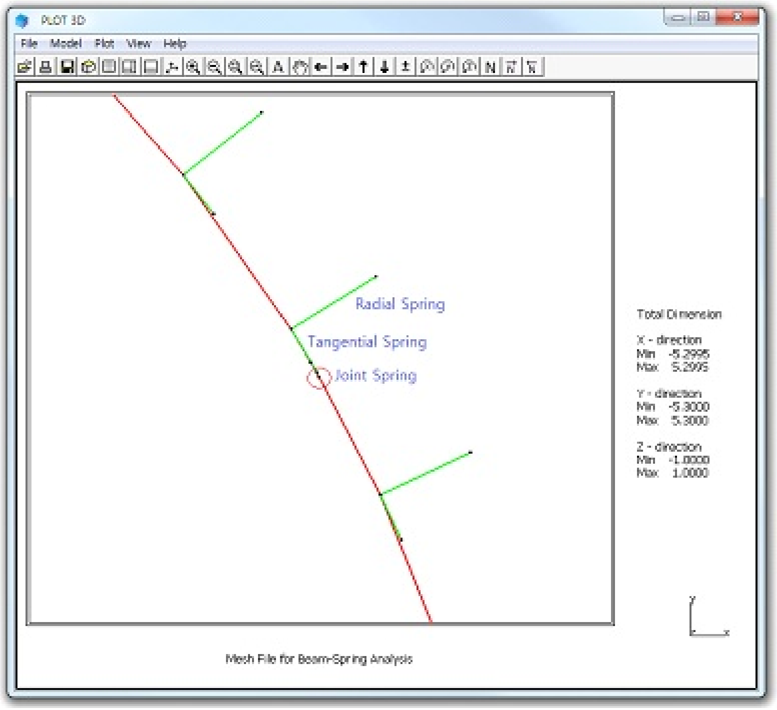


Figure 7.25 Detailed mesh around joint spring element

7.3 CIRCLE-2D

CIRCLE-2D is the special pre-processing program to generate automatically two-dimensional finite element meshes and boundary conditions for circular sections. CIRCLE-2D has three different models:

Model 1	Quarter	Section
Model 2	Half	Section
Model 3	Full	Section

CIRCLE-2D is described in Section 7.4 of User's Manual and can be selected in the following order:

Run → Mesh Generator → PreSmap → Circle 2D

When you finish the execution of CIRCLE-2D, select PLOT-3D to plot the generated finite element mesh.

Three example problems are presented here to show all three types of available models. Figure 7.26 shows schematic section views which are used for example problems.

For each model, we will present:

- Listing of input file
- Graphical output of finite element mesh

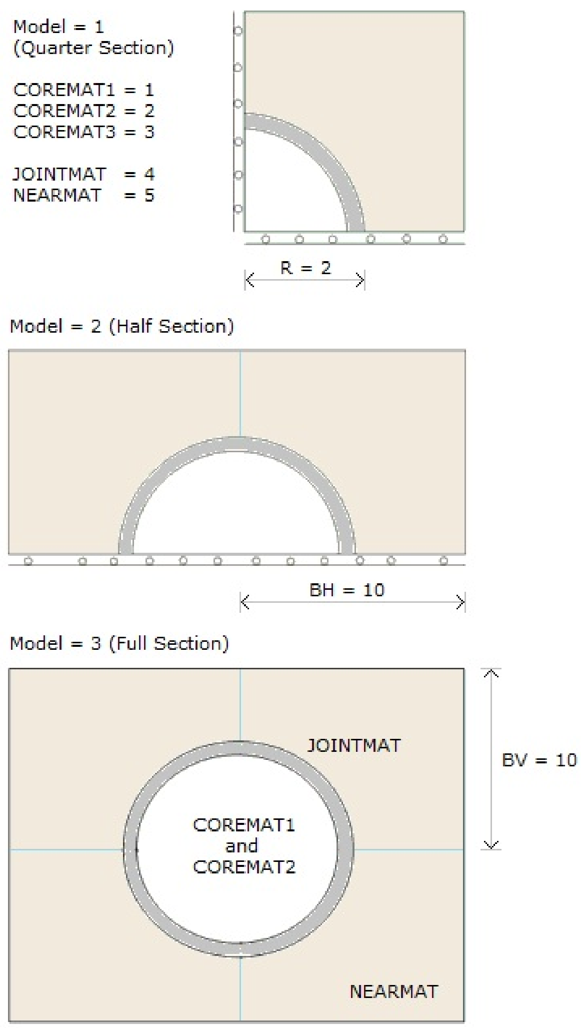


Figure 7.26 Schematic section views for CIRCLE-2D examples

Table 7.12 Listing of input file CIR1C_Q.Dat (MODEL = 1)

```
* CARD 1.1
* TITLE
  MODEL 1 (COARSE, ALL QUAD)
* CARD 1.2
* MODEL     NSNEL      NSNODE
  1         1          1
* CARD 2.1
* R         FINEMESH   NEARMESH   NDIV      BH       BV
  2.0       0          0          5         10.0     10.0
* CARD 3.1
* COREMAT1  COREMAT2   COREMAT2J  JOINTMAT  NEARMAT
  1         2          3          4         5
* END OF DATA
```

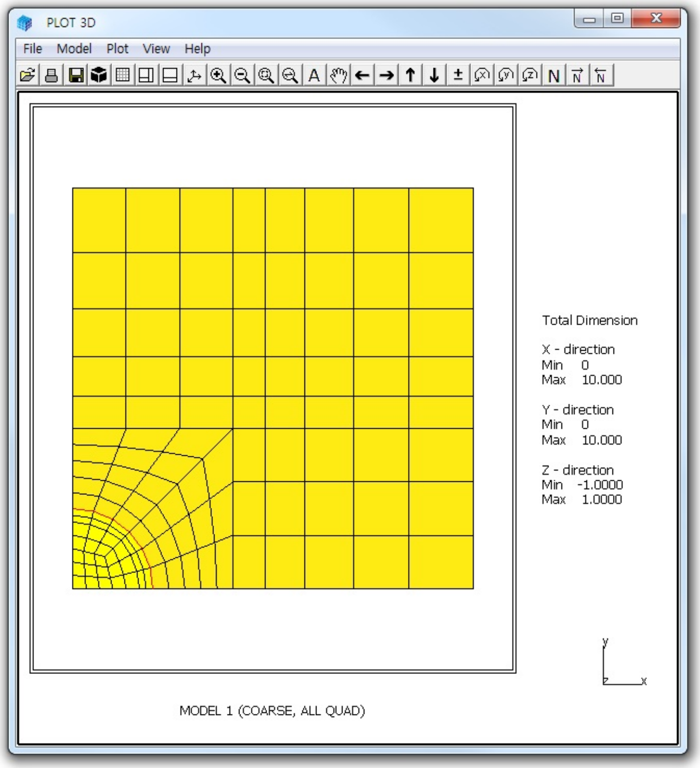


Figure 7.27 Generated finite element mesh for MODEL = 1

Table 7.13 Listing of input file CIR2C_Q.Dat (MODEL = 2)

```
* CARD 1.1
* TITLE
  MODEL 2 (COARSE, ALL QUAD)
* CARD 1.2
* MODEL     NSNEL      NSNODE
  2         1          1
* CARD 2.1
* R         FINEMESH   NEARMESH    NDIV       BH       BV
  2.0       0          0           5          10.0     10.0
* CARD 3.1
* COREMAT1  COREMAT2   COREMAT2J   JOINTMAT   NEARMAT
  1         2          3           4          5
* END OF DATA
```

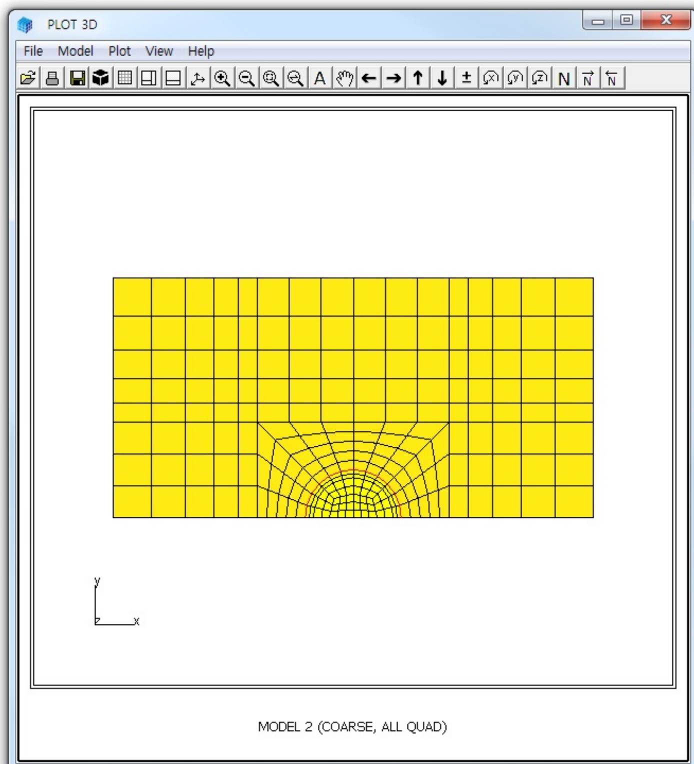


Figure 7.28 Generated finite element mesh for MODEL = 2

Table 7.14 Listing of input file CIR3C_Q.Dat (MODEL = 3)

```
* CARD 1.1
* TITLE
  MODEL 3 (COARSE, ALL QUAD)
* CARD 1.2
* MODEL     NSNEL       NSNODE
  3         1           1
* CARD 2.1
* R         FINEMESH    NEARMESH     NDIV        BH        BV
  2.0       0           0            5           10.0      10.0
* CARD 3.1
* COREMAT1  COREMAT2    COREMAT2J    JOINTMAT    NEARMAT
  1         2           3            4           5
* END OF DATA
```

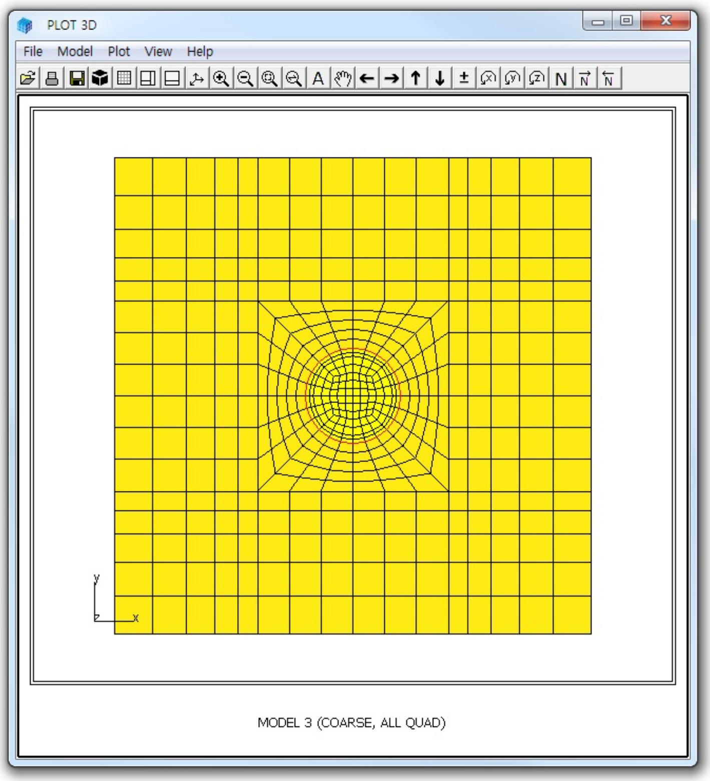


Figure 7.29 Generated finite element mesh for MODEL = 3

7.4 PRESMAP-GP

PRESMAP-GP is the general purpose pre-processor which can be used to generate coordinates, element indexes, and boundary codes of various geometries modeled by truss, beam, shell or continuum elements. Input parameters of PRESMAP-GP have been described in detail in Section 7.9 of User's Manual.

Input file for PRESMAP-GP is also called block mesh file which can be generated or modified by Block Mesh Generator described in Section 6 of User's Manual.

PRESMAP-GP can be selected in the following order.

Run → Mesh Generator → PreSmap → Presmap GP

When you finish the execution of PRESMAP-GP, select PLOT-3D to plot the generated mesh.

7.4.1 Example 1: 2-D Line/Surface Blocks

Example 1 shows you how Beam and Continuum elements are generated using various types of blocks. There are a total of 3 blocks consisting of a line block, a triangle surface block, and a quad surface block. Detailed block information is listed in Table 7.24.

Input block meshes and generated finite element meshes are presented in the following order:

Input Block Meshes
Figure 7.56 Node and block numbers
Figure 7.57 Material numbers
Figure 7.58 Skeleton boundary codes
Figure 7.59 Pore fluid boundary codes
Figure 7.60 Rotation boundary codes

Generated Finite Element Meshes

Table 7.24 Listing of input file EX1.Meb

```
StartPresmap
VersionNo = 7.000
* CARD 1.1
* TITLE
  LINE/SURFACE/ ELEMENT GENERATION
* CARD 1.2
* NBLOCK    NBNODE    NSNODE    NSNEL     IGBND     ISMAP     CMFAC    ICOMP
  3         6         1         1         0         2         1.000    1
*=======================================================================
* CARD 1.3
* Global Outer Surface Boundary
* X - Right Boundary
* ISG  ISX  ISY  ISZ  IFG  IFX  IFY  IFZ  IRG  IRX  IRY  IRZ
  3    0    0    0    0    0    0    0    0    0    0    0
* X - Left Boundary
* ISG  ISX  ISY  ISZ  IFG  IFX  IFY  IFZ  IRG  IRX  IRY  IRZ
  3    0    0    0    0    0    0    0    0    0    0    0
* Y - Top Boundary
* ISG  ISX  ISY  ISZ  IFG  IFX  IFY  IFZ  IRG  IRX  IRY  IRZ
  4    1    1    0    4    1    1    1    0    0    0    0
* Y - Bottom Boundary
* ISG  ISX  ISY  ISZ  IFG  IFX  IFY  IFZ  IRG  IRX  IRY  IRZ
  3    0    0    0    4    1    0    1    0    0    0    0
* Z - Front Boundary
* ISG  ISX  ISY  ISZ  IFG  IFX  IFY  IFZ  IRG  IRX  IRY  IRZ
  3    0    0    0    0    0    0    0    4    0    1    0
* Z - Back Boundary
* ISG  ISX  ISY  ISZ  IFG  IFX  IFY  IFZ  IRG  IRX  IRY  IRZ
  3    0    0    0    0    0    0    0    4    1    0    1
*=======================================================================
```

```
* CARD 1.4
* Automatic Finite Element Generation Control Parameters
* Min Length    Max Element
  1.000         10000
*=======================================================================
* CARD 2.1
* NODE  X        Y        Z
  1     4.0      6.5      0.0
  2     0.0      2.0      0.0
  3     5.9      0.8      0.0
  4     7.0      7.0      0.0
  5     7.0      1.0      0.0
  6     5.72     3.87     0.0
*=======================================================================
StartBlock
* CARD 3.0
* IBETYPE
  1
* CARD 3.1
* BLNAME
  BLOCK 1
* CARD 3.2
* ICOORD IMODE  ILAG
  1      0      0
* CARD 3.3
* I1     I2
  1      3
* M3
  0
* M4
  0
* M5     M6     M7
  0      0      0
* CARD 3.4.1
* NBOUND
  2
* CARD 3.4.2
* IBTYPE ISX    ISY    ISZ    IFX    IFY    IFZ    IRX    IRY    IRZ
  3      0      0      0      1      1      1      1      1      1
  4      0      0      1      1      1      1      1      0      0
* CARD 3.5
* MATNO  NDX
  1      4
EndBlock
```

```
*======================================================================
StartBlock
* CARD 3.0
* IBETYPE
  -2
* CARD 3.1
* BLNAME
  BLOCK 2
* CARD 3.2
* ICOORD IMODE  ILAG
  1      0      1
* CARD 3.3
* I1     I2     I3
  1      2      3
* M4     M5     M6
  0      0      0
* M7
  0
* M8     M9     M10
  0      0      0
* CARD 3.4.1
* NBOUND
  4
* CARD 3.4.2
* IBTYPE ISX    ISY    ISZ    IFX    IFY    IFZ    IRX    IRY    IRZ
  1      0      0      0      0      0      0      1      1      1
  2      1      1      1      0      0      0      1      1      1
  3      0      1      1      1      1      1      0      0      0
  4      1      1      1      1      1      1      1      1      1
* CARD 3.5
* MATNO  NDXY
  4      4
* KS     KF
  0      1
EndBlock
*======================================================================
StartBlock
* CARD 3.0
* IBETYPE
  2
* CARD 3.1
* BLNAME
  BLOCK 3
* CARD 3.2
* ICOORD IMODE  ILAG
  1      0      1
```

```
* CARD 3.3
* I1      I2      I3      I4
  4       1       3       5
* M5      M6      M7      M8
  0       0       0       0
* M9
  0
* M10     M11     M12
  0       0       0
* CARD 3.4.1
* NBOUND
  1
* CARD 3.4.2
* IBTYPE ISX     ISY     ISZ     IFX     IFY     IFZ     IRX     IRY     IRZ
  5       1       0       1       0       1       0       1       0       1
* CARD 3.5
* MATNO  NDX     NDY
  2       1       4
* NT1     NT2     NT3     NT4
  0       0       0       0
* MAT1    MAT2    MAT3    MAT4
  0       0       0       0
* KS      KF
  0       0
EndBlock
*========================================================================
EndOfLastBlock
```

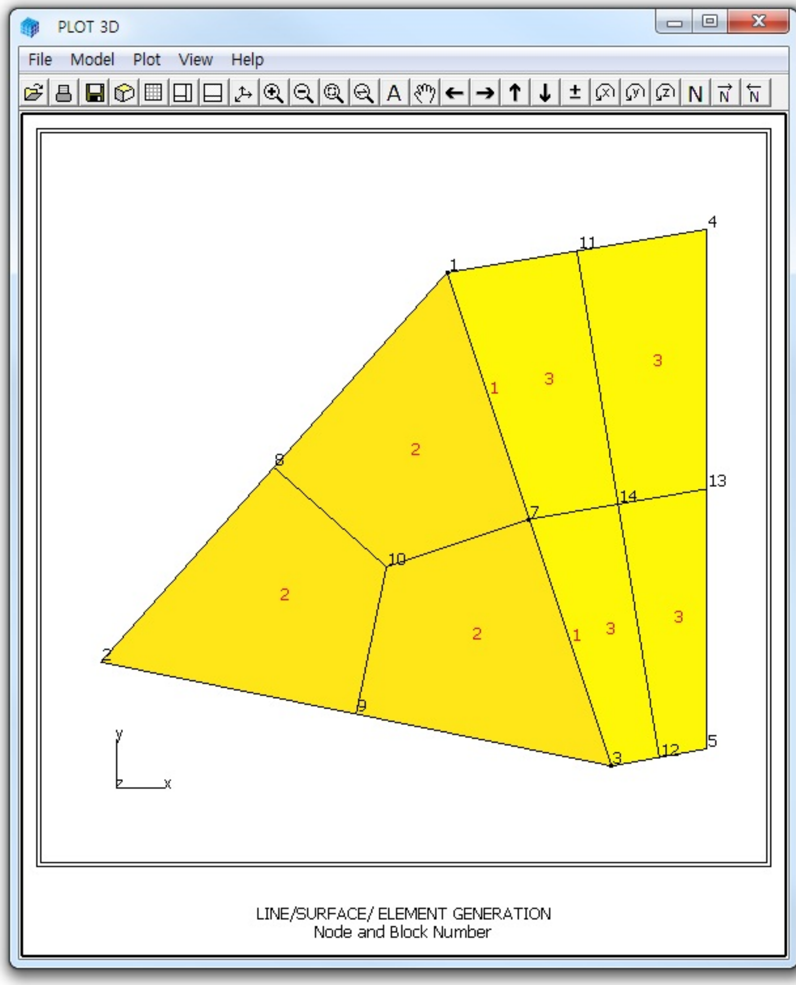


Figure 7.56 Node and block numbers for Example 1

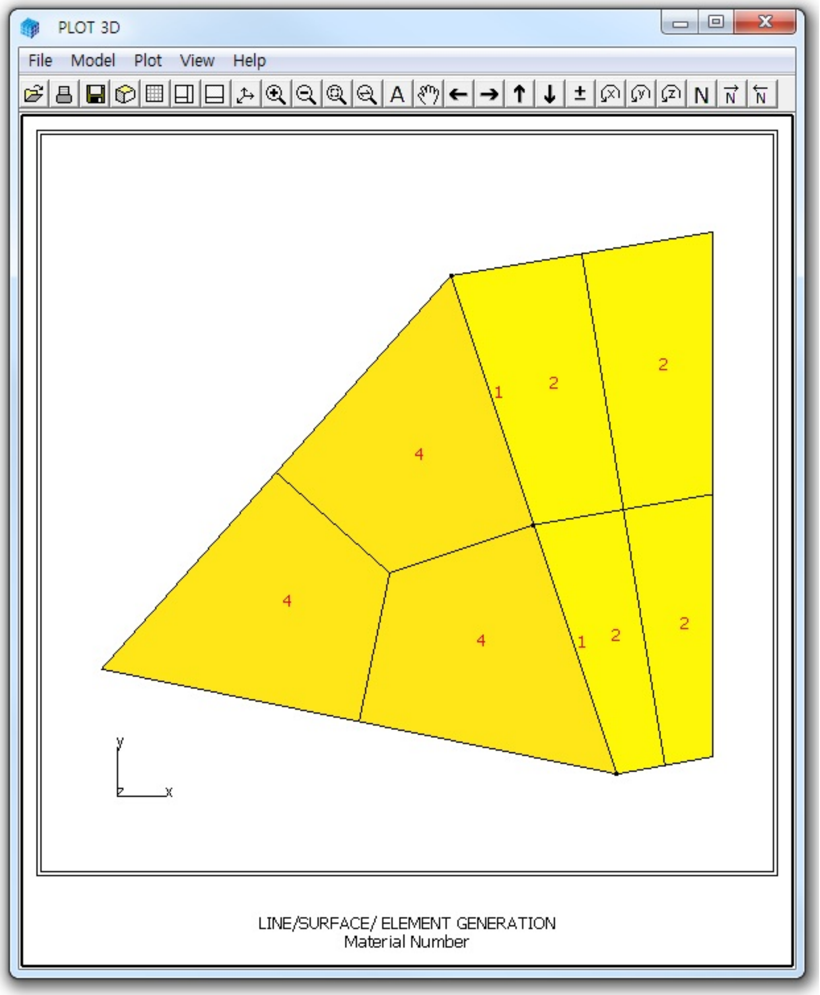


Figure 7.57 Material numbers

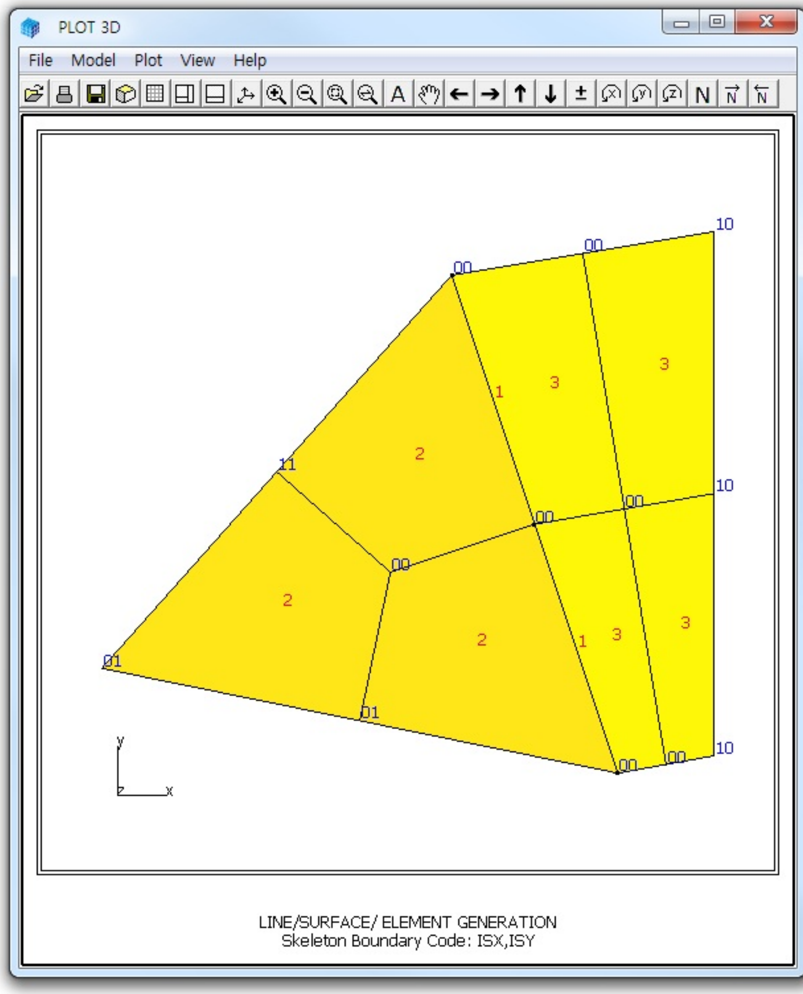


Figure 7.58 Skeleton boundary codes

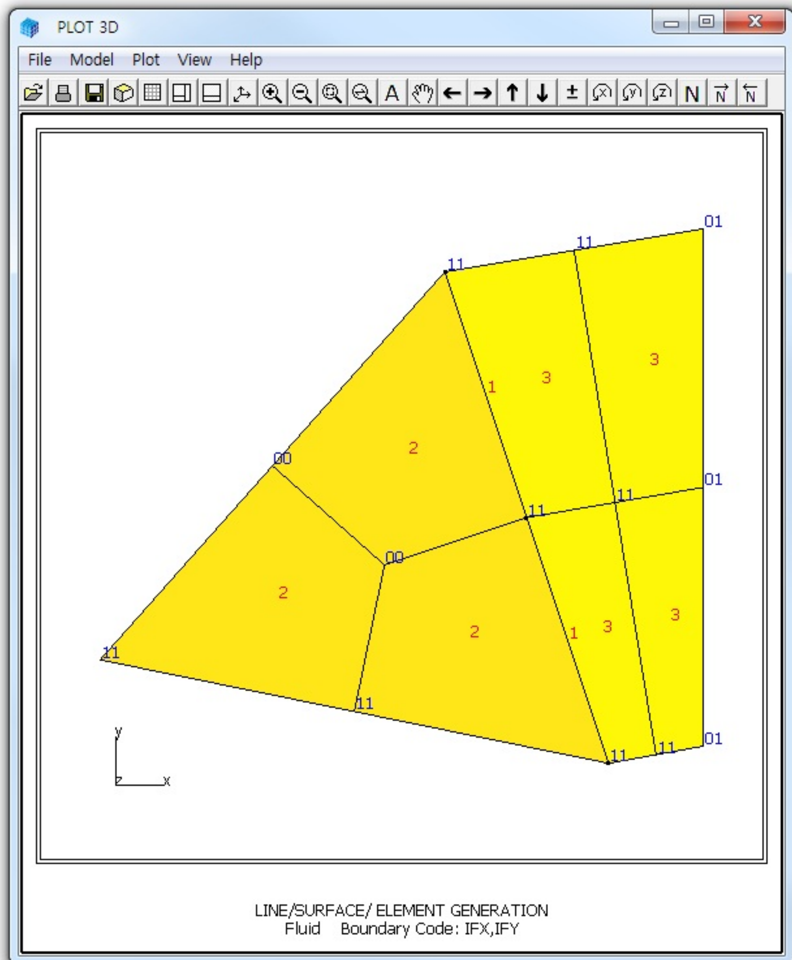


Figure 7.59 Pore fluid boundary codes

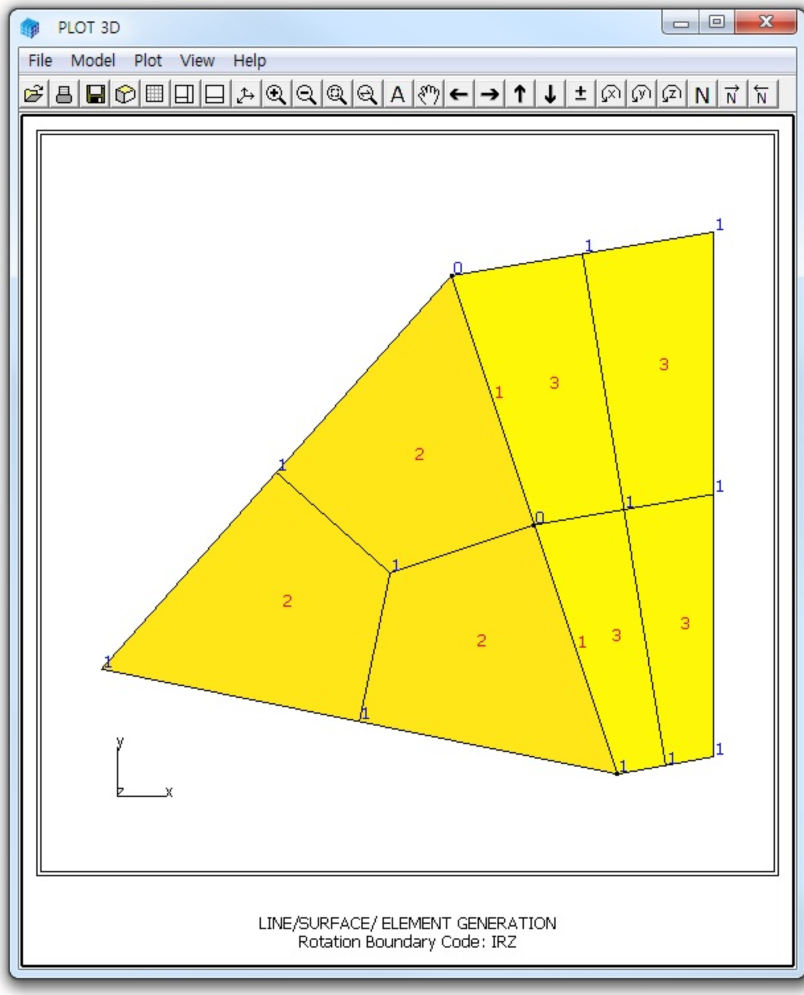


Figure 7.60 Rotation boundary codes

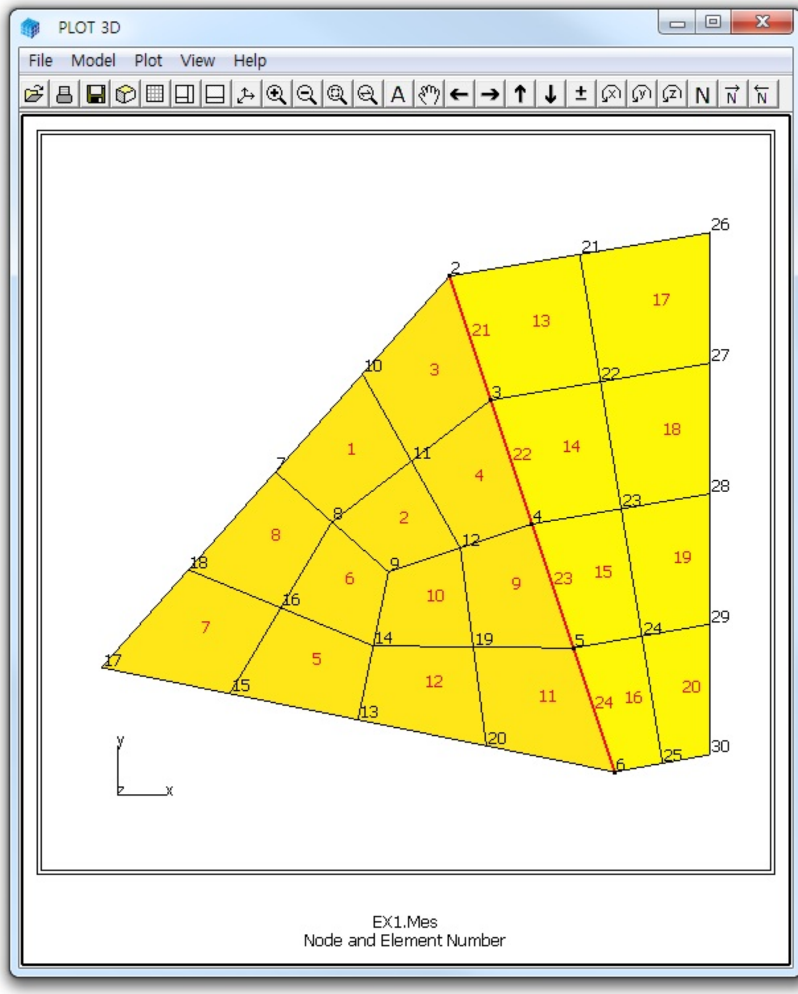


Figure 7.61 Node and element numbers

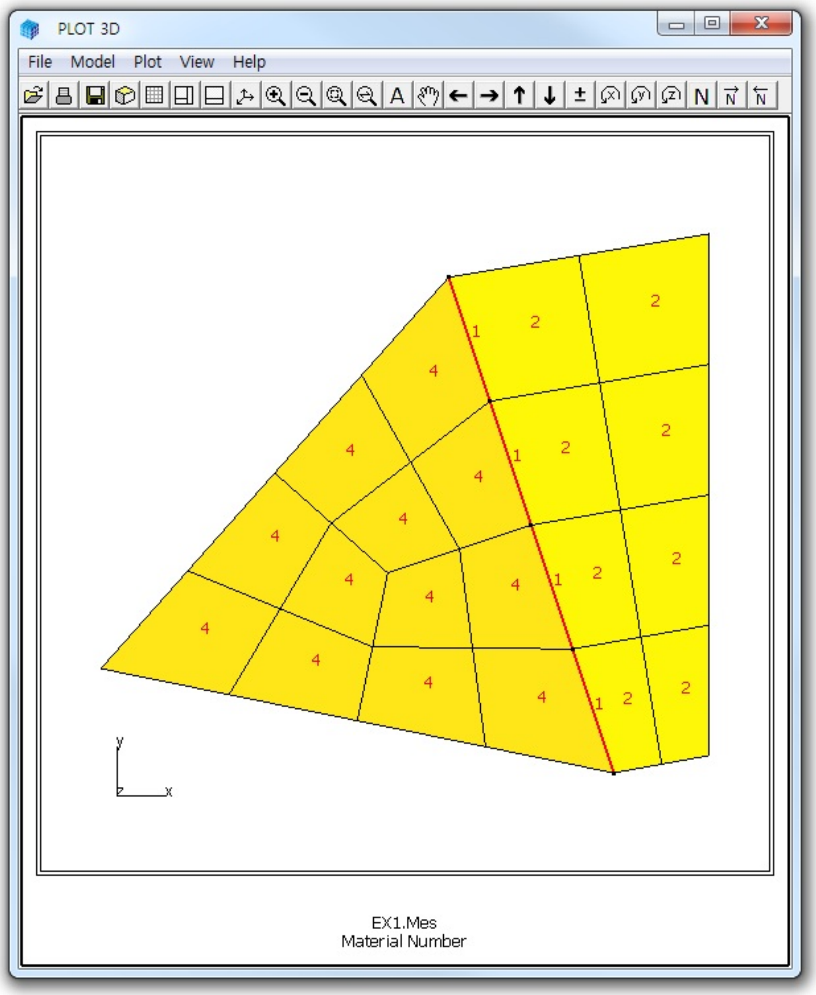


Figure 7.62 Material numbers

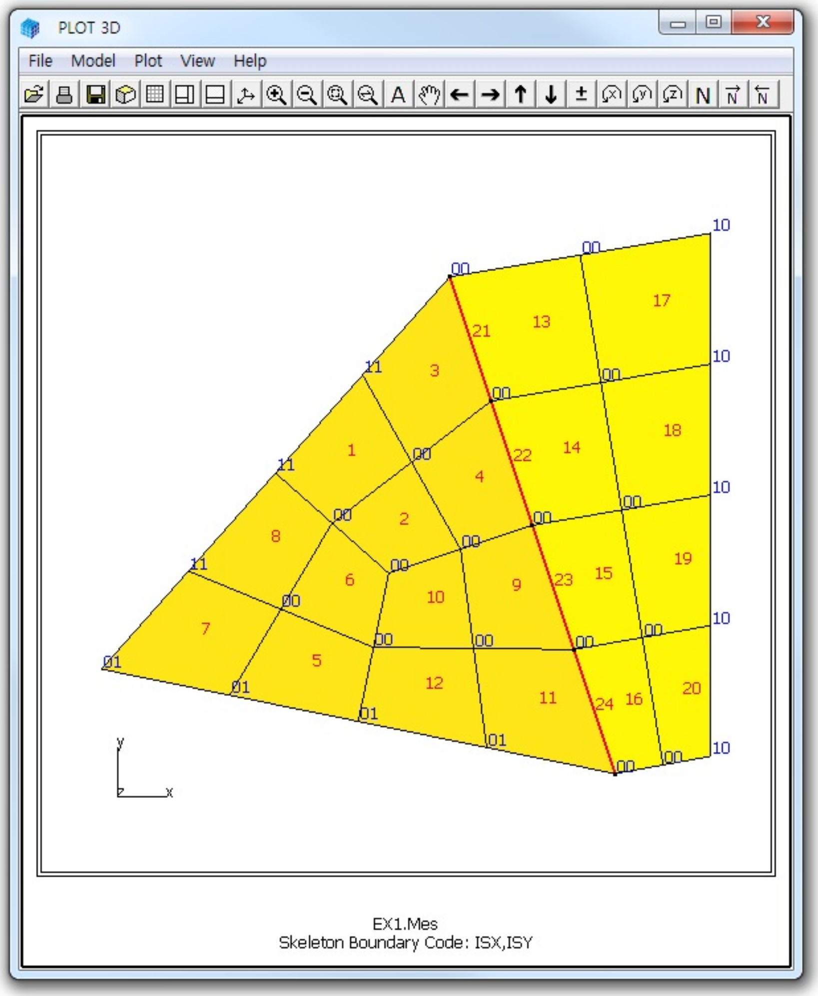


Figure 7.63 Skeleton boundary codes

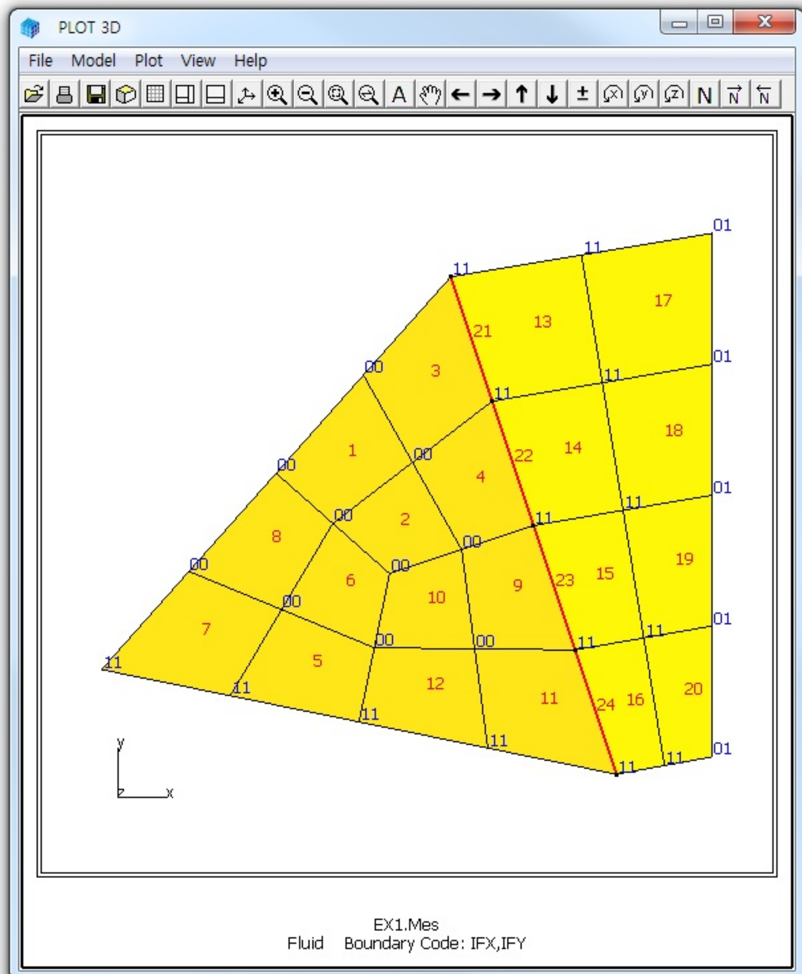


Figure 7.64 Pore fluid boundary codes

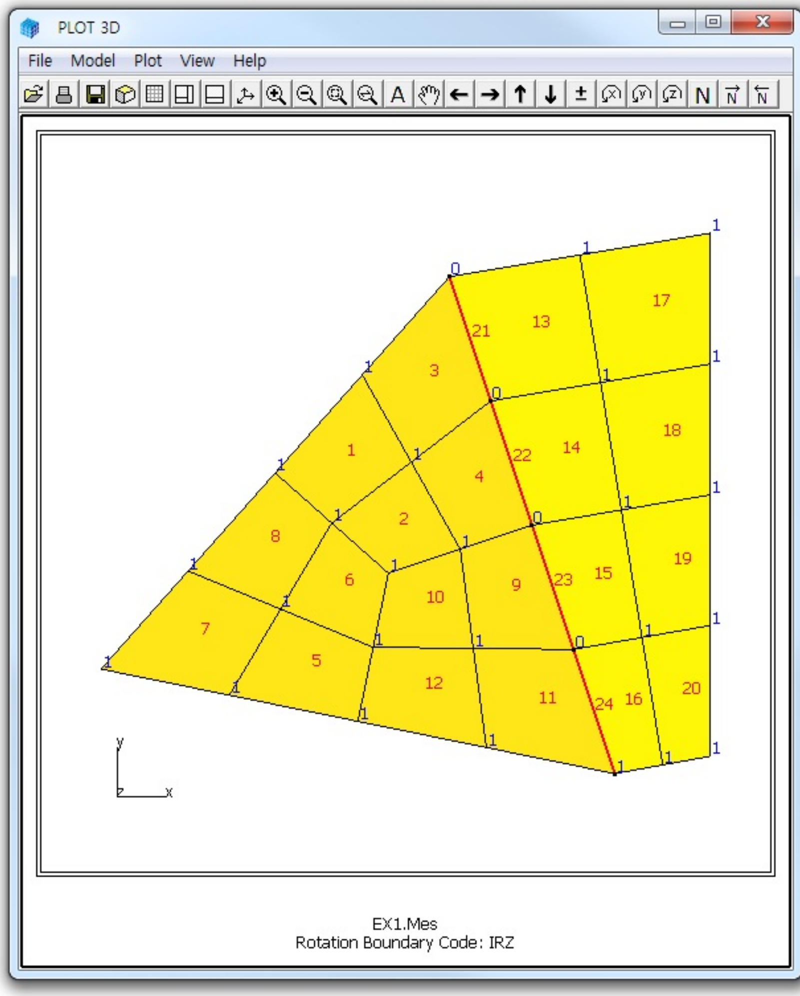


Figure 7.65 Rotation boundary codes

7.4.2 Example 2: Surface with Corner Triangles

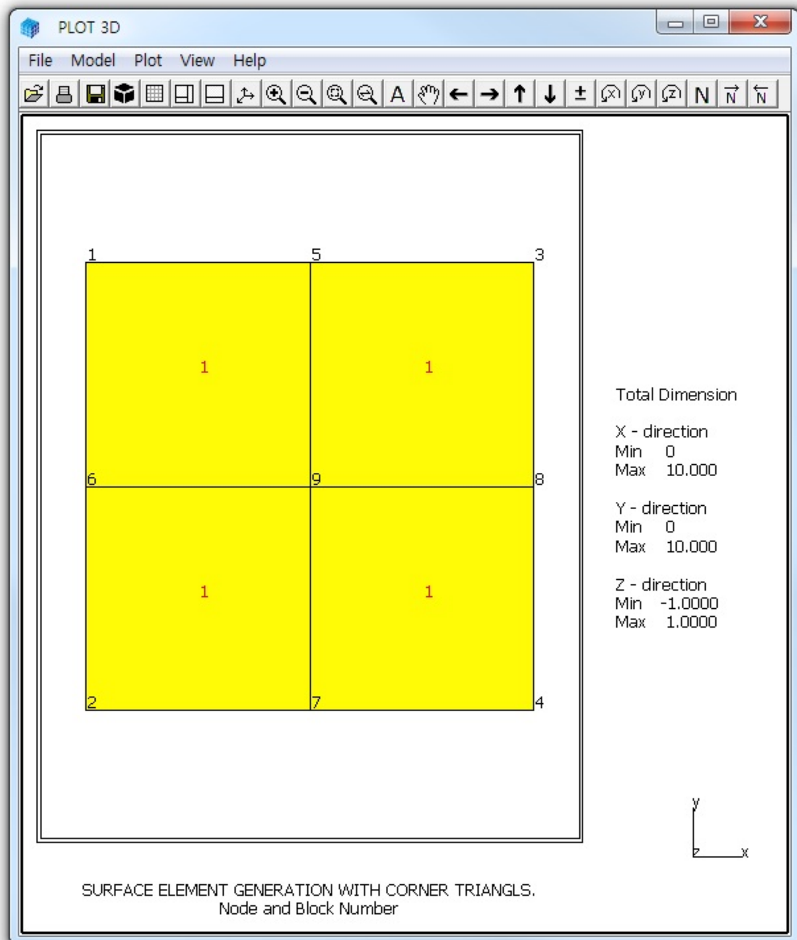


Figure 7.66 Block mesh for Example 2

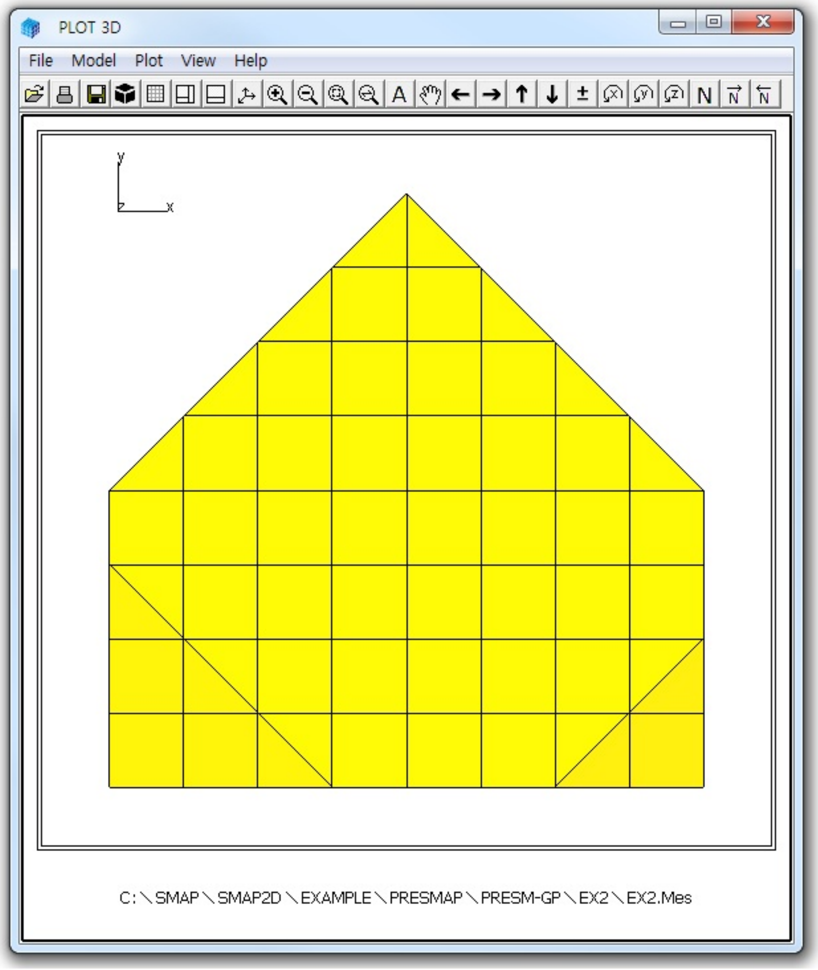


Figure 7.67 Finite element mesh for Example 2

7.4.3 Example 3: Circular Sector

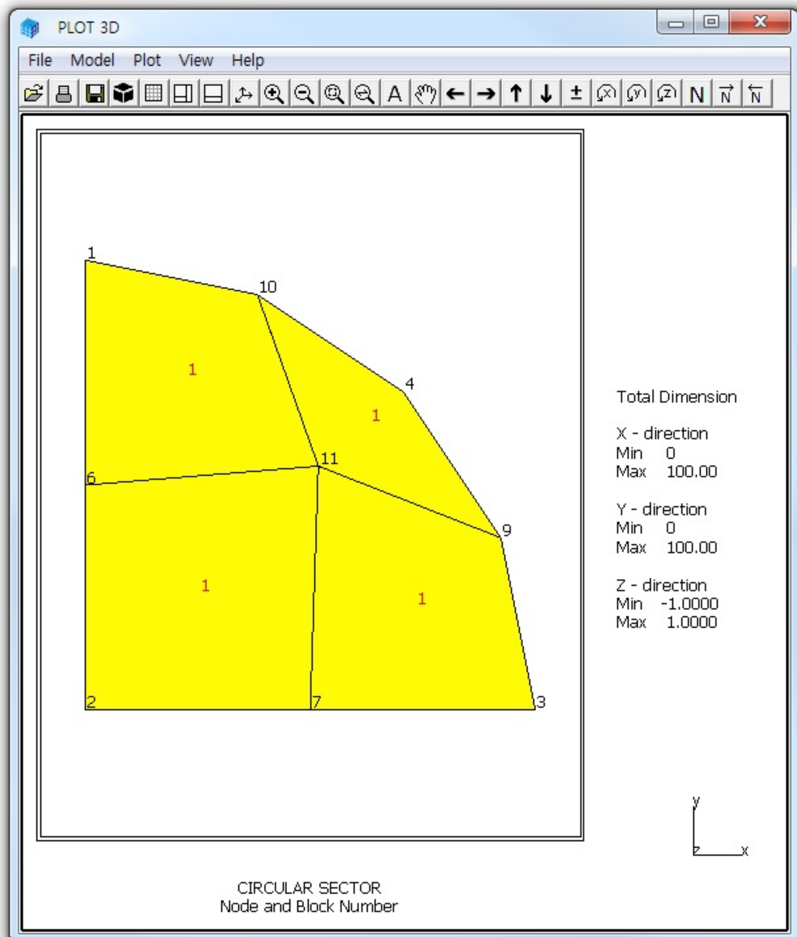


Figure 7.68 Block mesh for Example 3

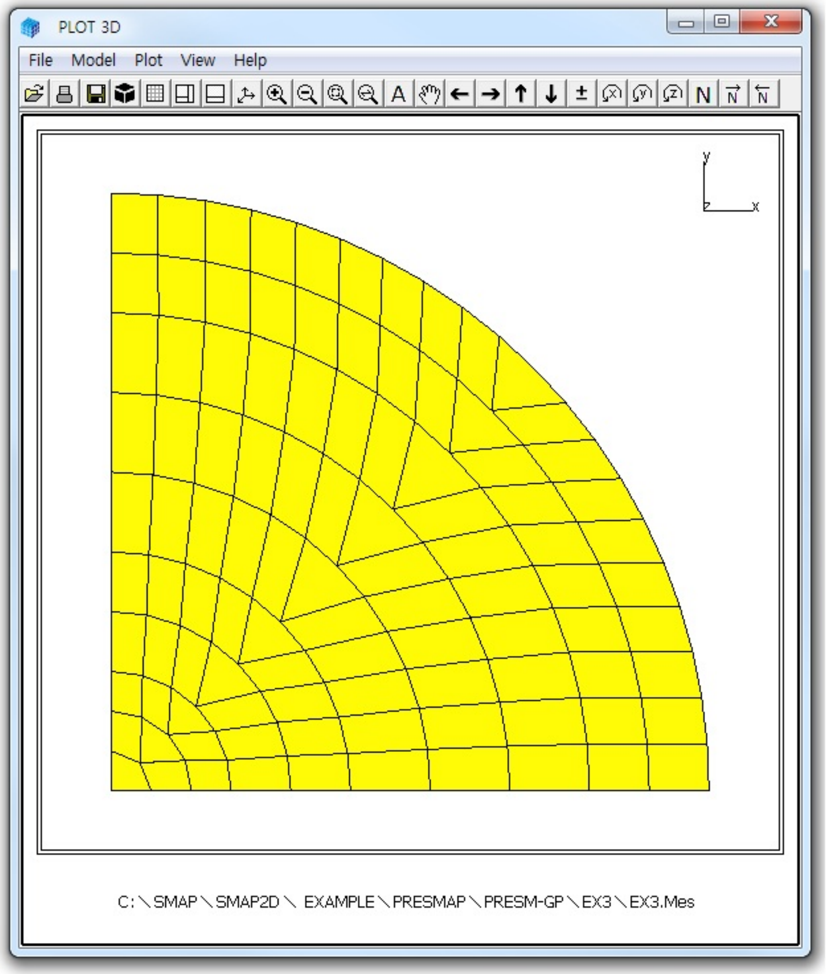


Figure 7.69 Finite element mesh for Example 3

7.4.4 Example 4: Straight Line Sector

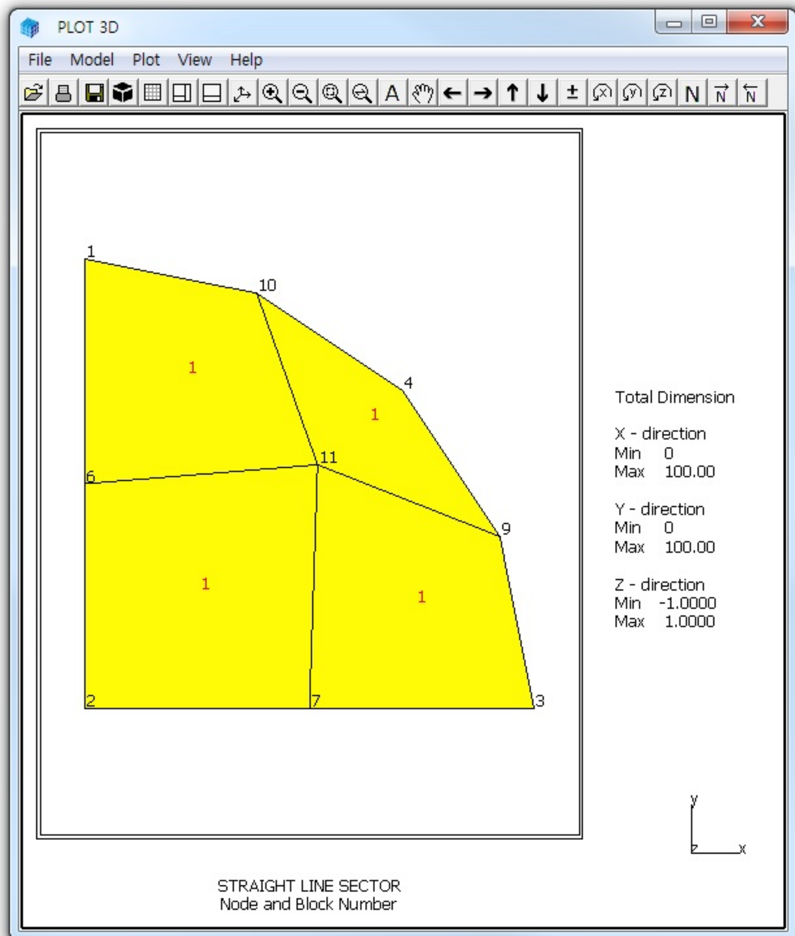


Figure 7.70 Block mesh for Example 4

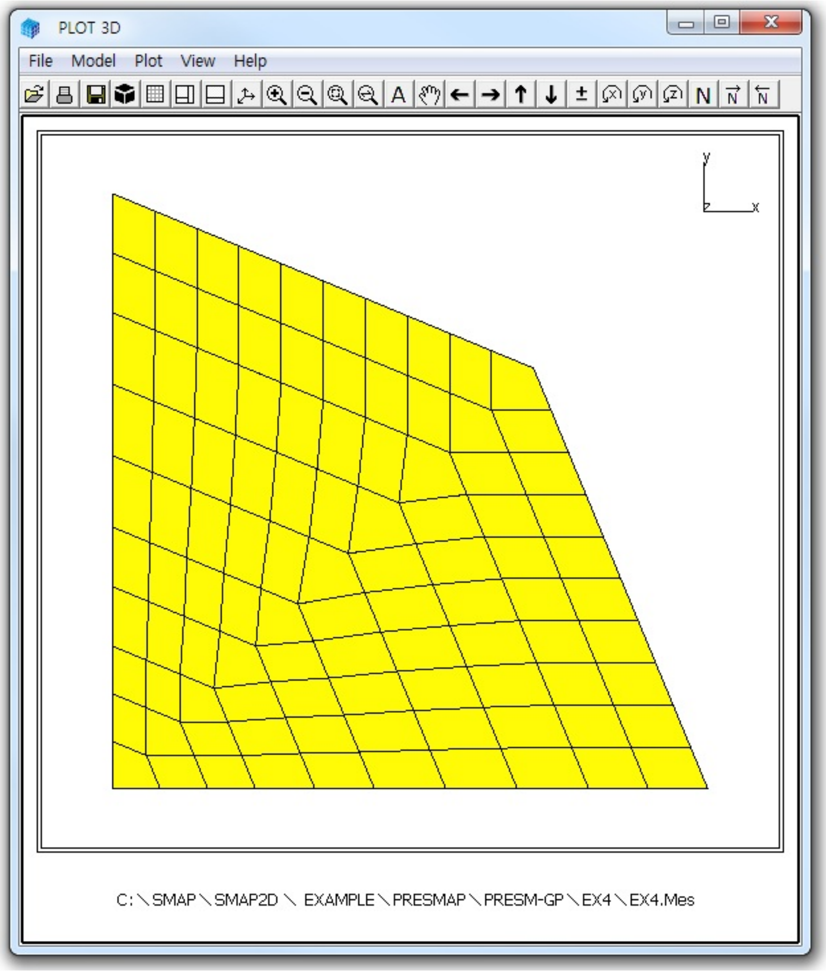


Figure 7.71 Finite element mesh for Example 4

7.4.5 Example 5: Surface and Line Element (1)

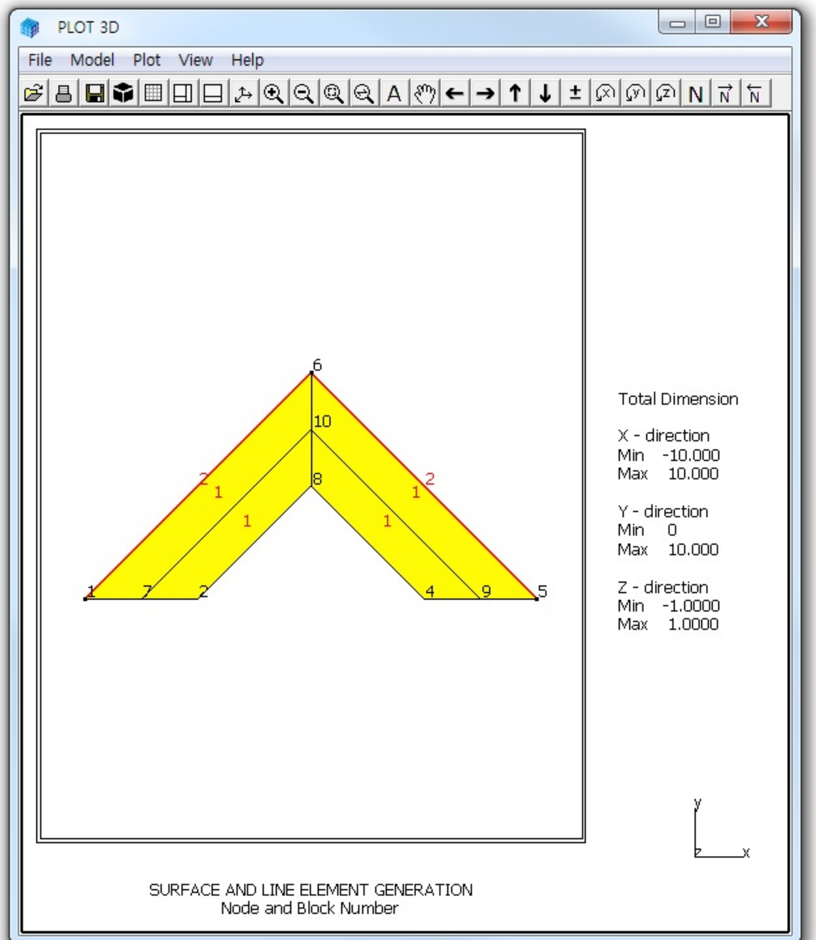


Figure 7.72 Block mesh for Example 5

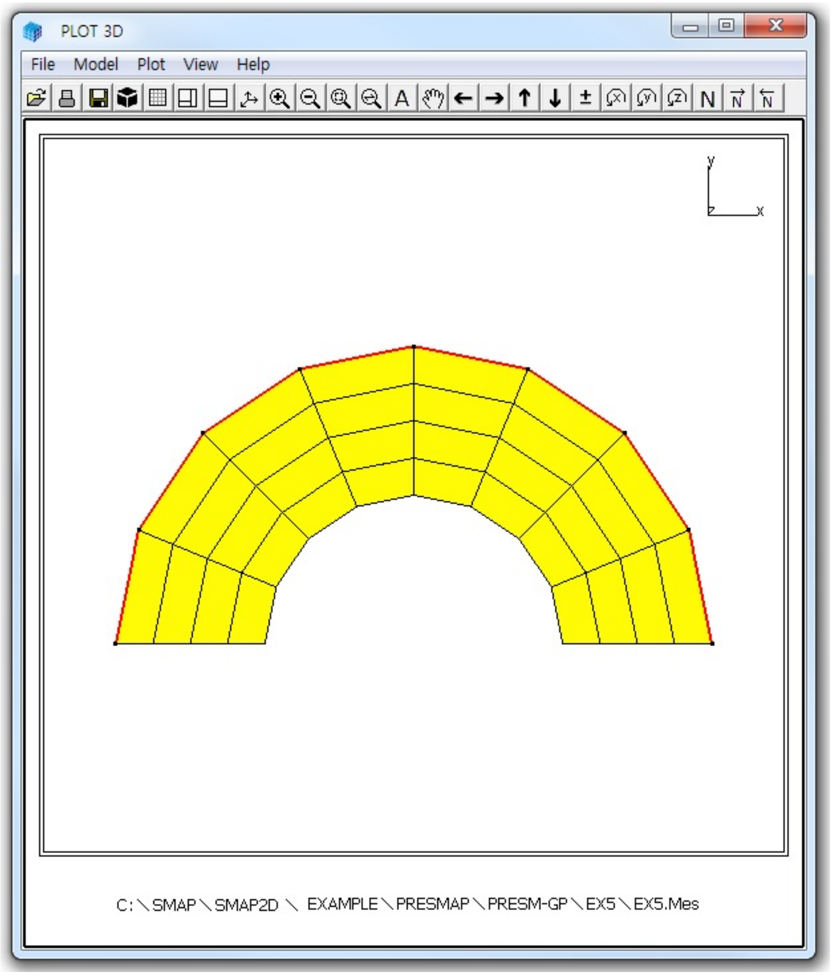


Figure 7.73 Finite element mesh for Example 5

7.4.6 Example 6: Surface and Line Element (2)

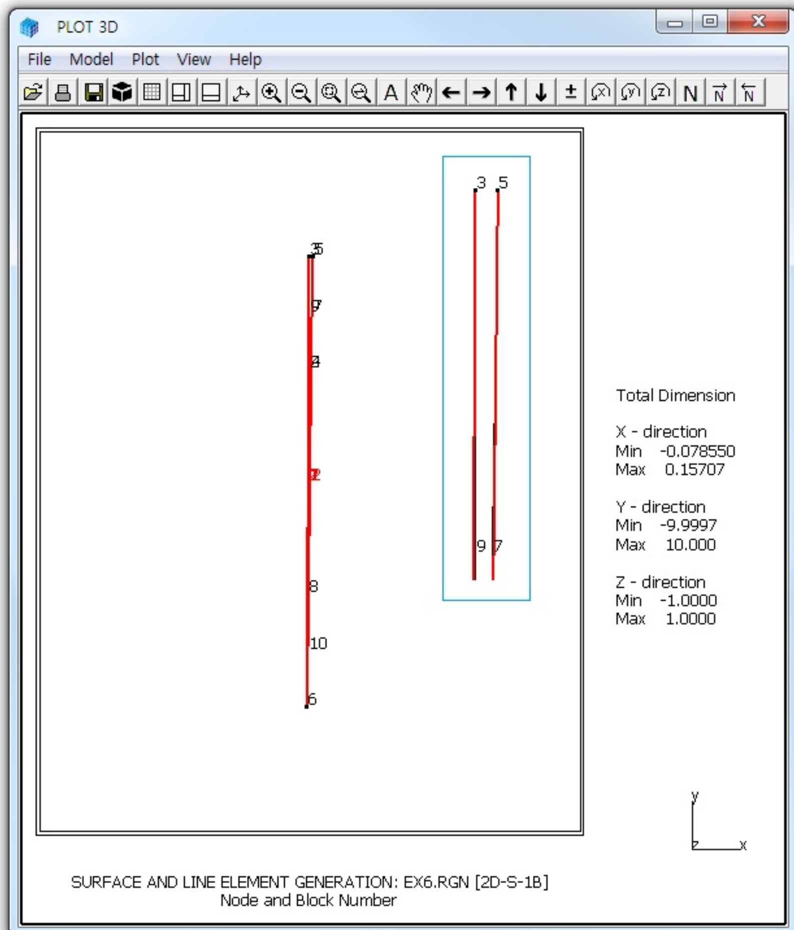


Figure 7.74 Block mesh for Example 6

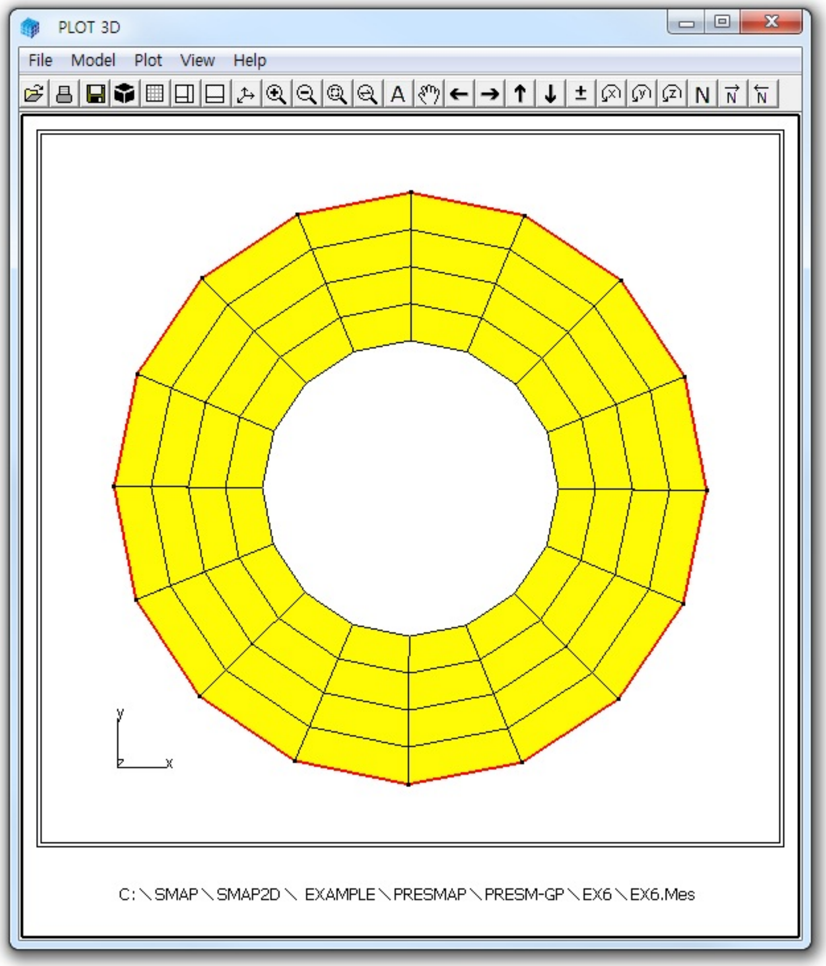


Figure 7.75 Finite element mesh for Example 6

7.4.7 Example 7: Surface and Line Element (3)

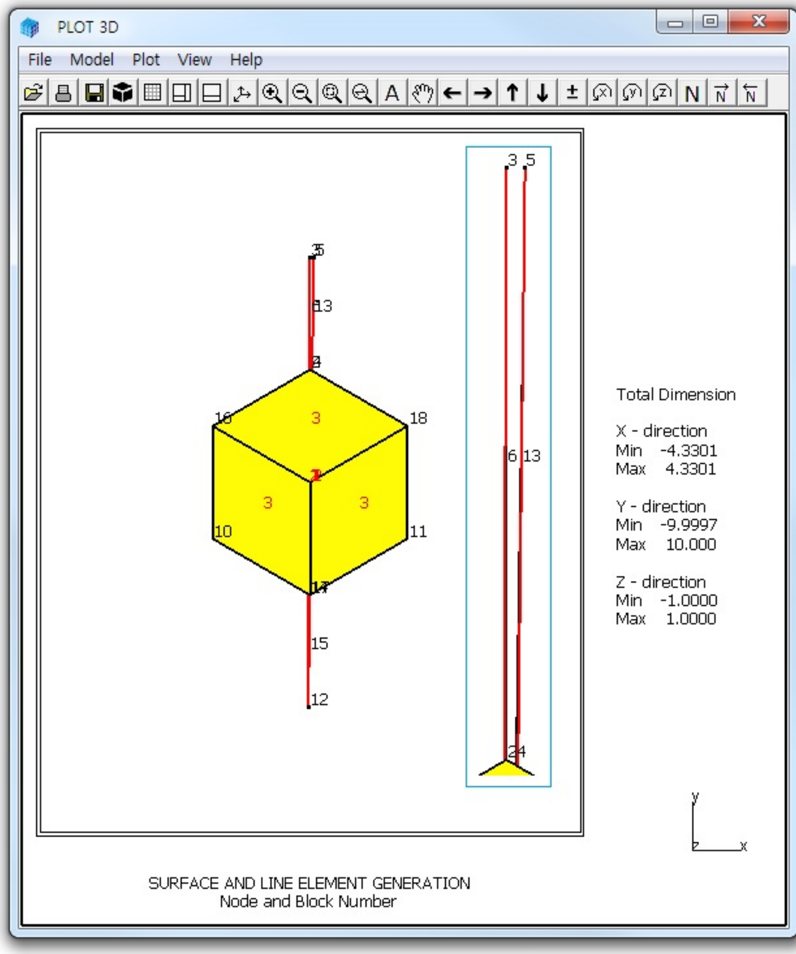


Figure 7.76 Block mesh for Example 7

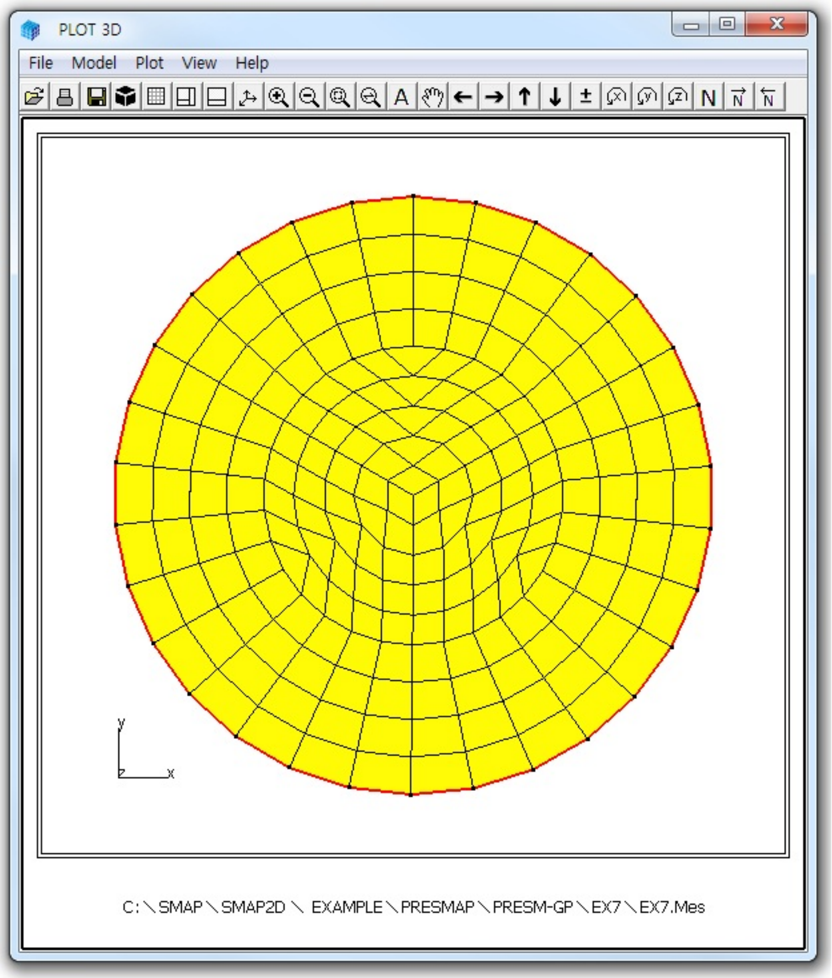


Figure 7.77 Finite element mesh for Example 7

7.4.8 Example 8: Cement-Soil Road

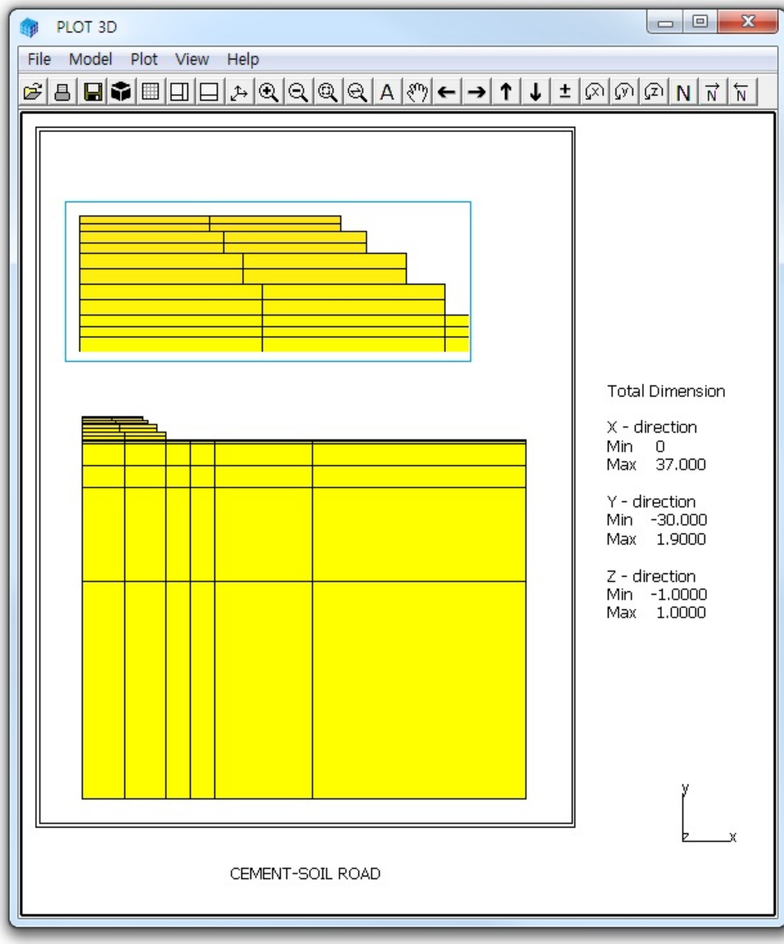


Figure 7.78 Block mesh for Example 8

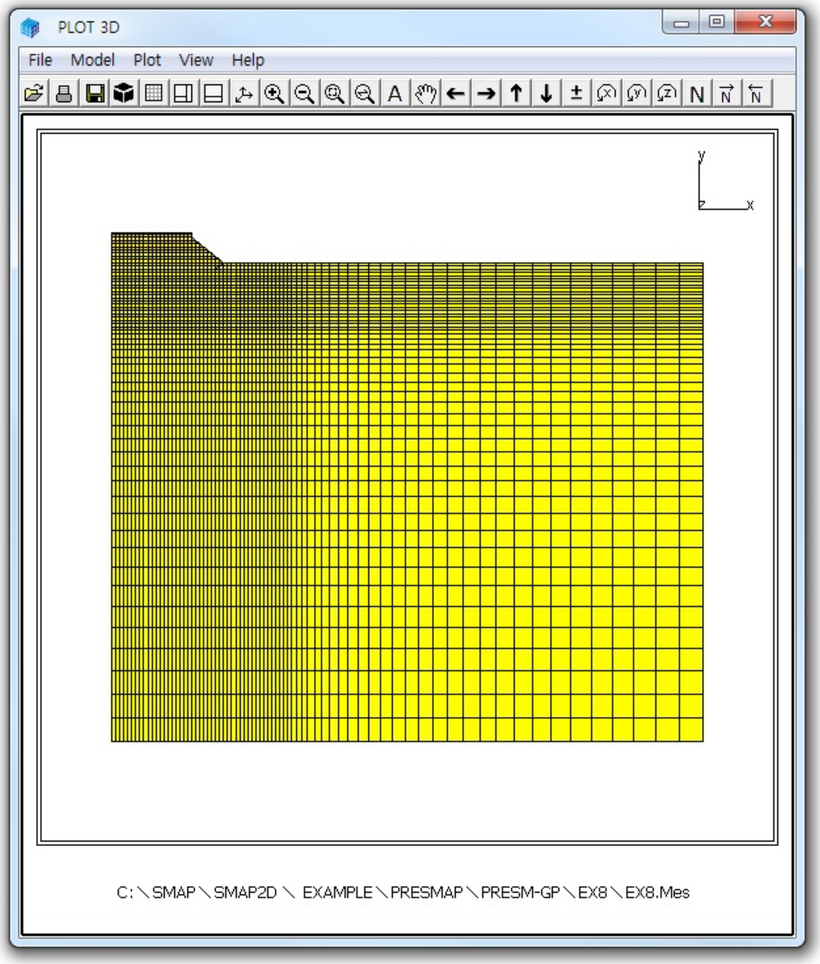


Figure 7.79 Finite element mesh for Example 8

7.4.9 Example 9: Tunnel in Spherical Geometry

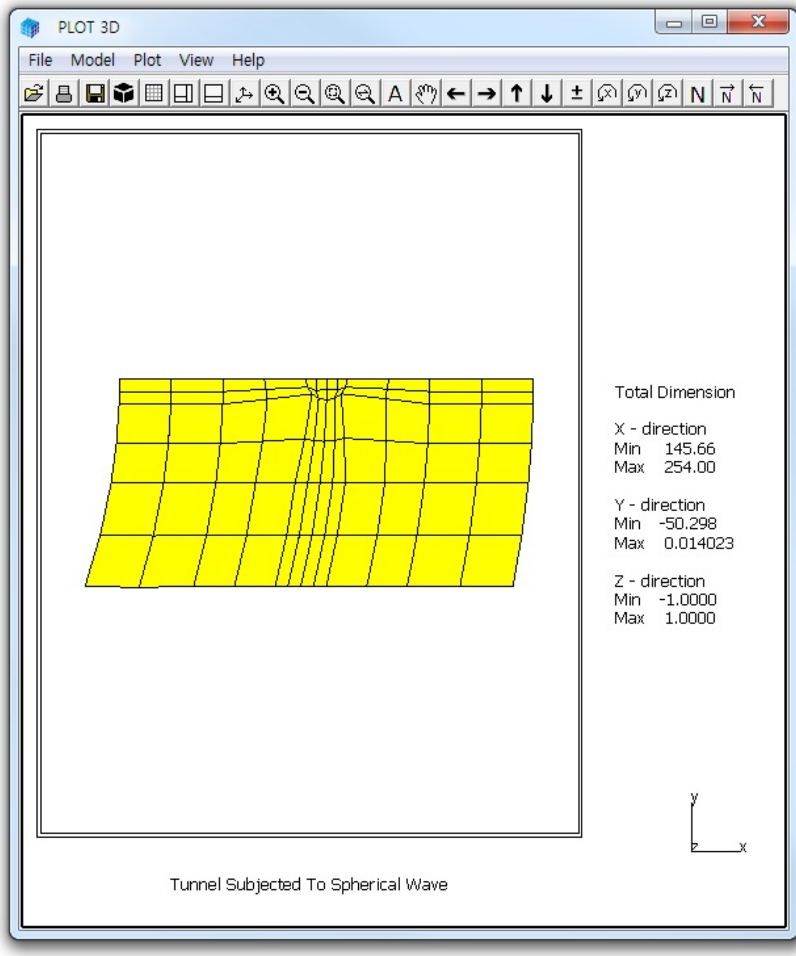


Figure 7.80 Block mesh for Example 9

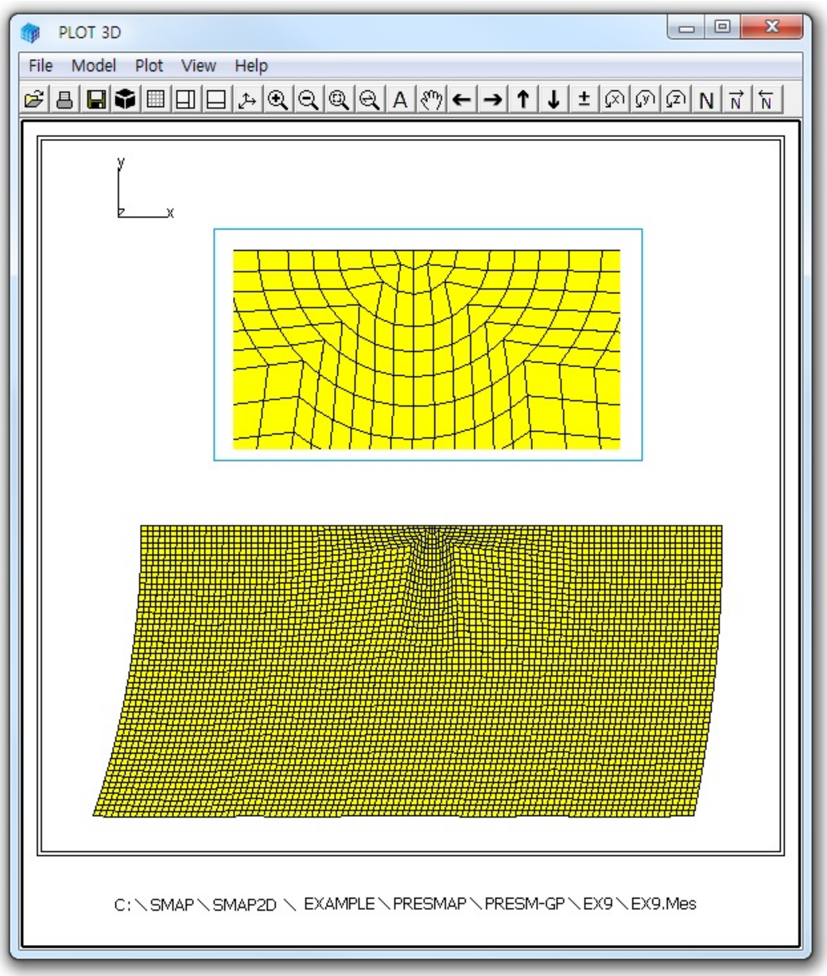


Figure 7.81 Finite element mesh for Example 9

7.4.10 Example 10: Horseshoe Tunnel

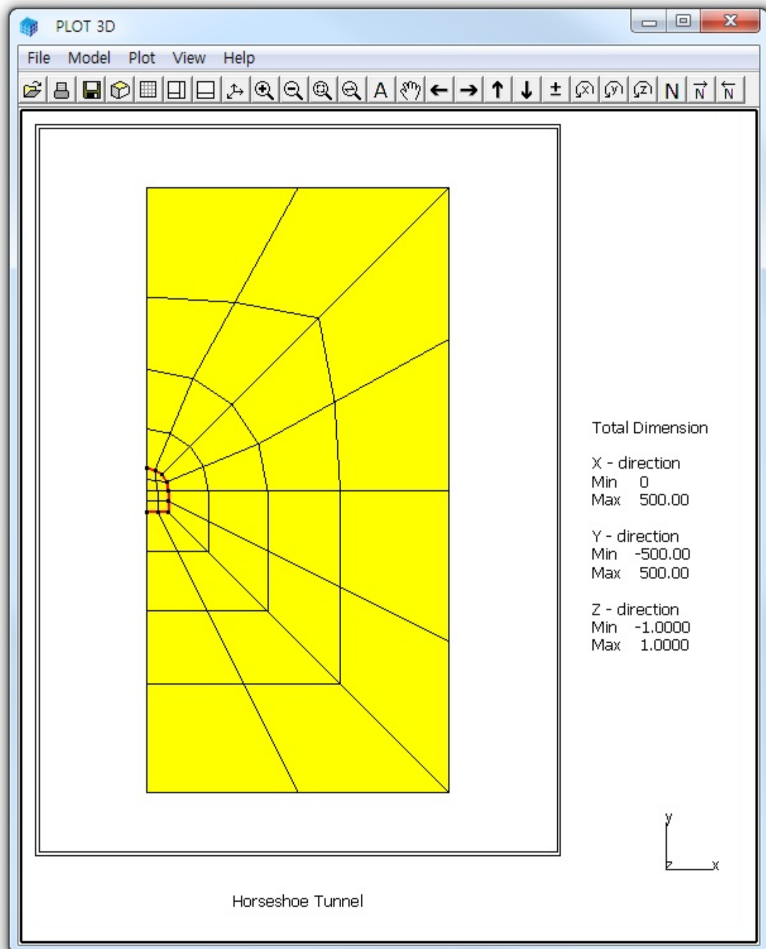


Figure 7.82 Block mesh for Example 10

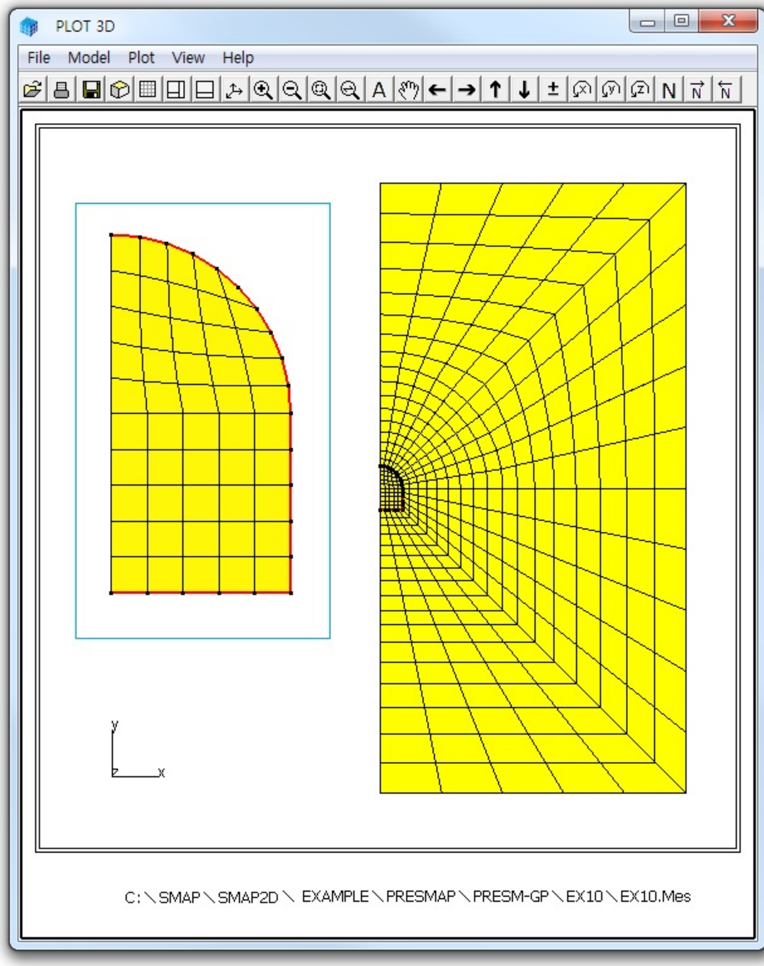


Figure 7.83 Finite element mesh for Example 10

7.4.11 Example 11: Wedge Surface Block

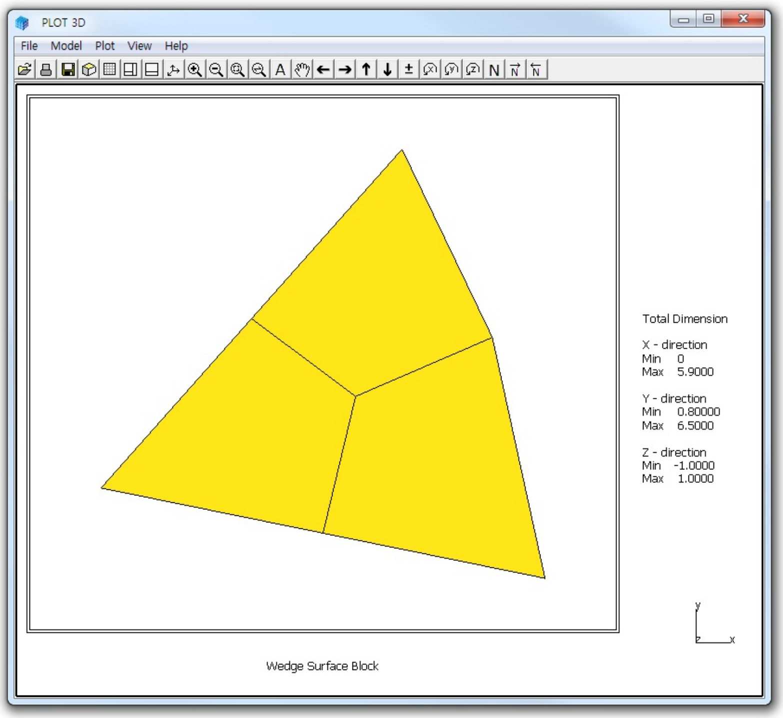


Figure 7.84 Block mesh for Example 11

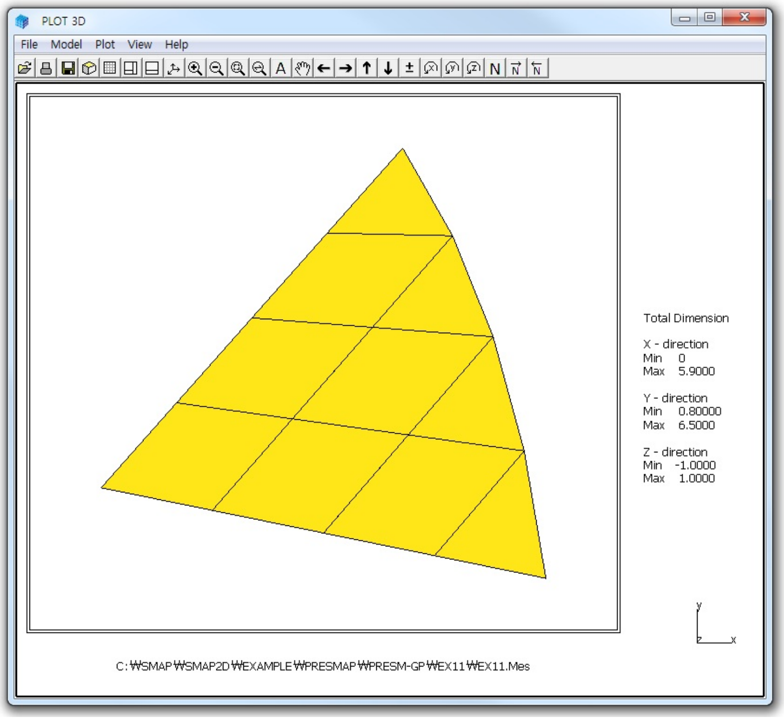


Figure 7.85 Finite element mesh for Example 11

7.5 JOINT-2D

JOINT-2D is the special pre-processor which can be used to generate jointed continuum finite element meshes given the conventional continuum SMAP-2D Mesh File input. For the jointed continuum analysis, each continuum finite element is surrounded by joint elements which allow slippage along the joint when reaching shear strength and debonding normal to the joint face when exceeding tensile strength.

JOINT-2D includes following features:

- Internal Joints within the specified group of materials
- Boundary Joints along the specified group of materials
- Surface Joints along the specified group of element surfaces

First, you need to prepare SMAP-2D Mesh File consisting of continuum elements. Copy C:\Smap\Ct\Ctdata\Joint-2D.dat into Working Directory and then modify input parameters as described in Section 7.10 of User's Manual.

JOINT-2D can be selected in the following order.

Run → Mesh Generator → PreSmap → Joint

Dialog for Input and Output File Names will be displayed as in Figure 7.90.

7.5.1 Example 1: Horseshoe Tunnel

Example 1 shows you how jointed continuum elements are generated around the horseshoe tunnel. Figure 7.91 shows material numbers of continuum elements: 1 representing for Far Field, 2 for Tunnel Core and 3 for Near Field in the input mesh. Joint data is prepared to generate internal joints within the Near Field as listed in Table 7.25. Note that it also specify Outer Beam between Tunnel Core and and Near Field to generate tunnel liner.

Figure 7.92 shows generated jointed finite element meshes around tunnel core along with beam elements representing for tunnel liner.

Table 7.25 Listing of input file Joint.inp for Example 1

```
****************************************************************
* Jointed Continuum Generation                                 *
* ==============================================================*
*                                                              *
* Card 1.1                                                     *
* Title                                                        *
  Example 1: Horseshoe Tunnel
* Card 1.2                                                     *
* AllJoint                                                     *
*  = 0 Generate Joint Elements along all interfaces            *
*      between continuum elements.                             *
*      Cards 2, 3 and 4 are not used.                          *
*                                                              *
*  = 1 Generate Joint Elements for material numbers of         *
*      continuum elements as specified in Cards 2 and 3.       *
*      Card 4 is not used.                                     *
*                                                              *
*  = 2 Generate Joint Elements for element surface numbers of  *
*      continuum elements as specified in Card 4.              *
*      Cards 2 and 3 are ignored.                              *
*                                                              *
* ThicAJ  Joint Thickness Used For AllJoint = 0                *
*                                                              *
*--------------------------------------------------------------*
*                                                              *
*  To Run JOINT-2D                                             *
*                                                              *
*  Method 1                                                    *
*  ========                                                    *
*                                                              *
*  SMAP-2D > Run > Mesh Generator > PreSmap > Joint            *
*  Specify input and output file names shown on the screen.    *
*                                                              *
*  Method 2                                                    *
*  ========                                                    *
*                                                              *
*  1. Select SMAP-2D > Setup > PLOT 3D                         *
*  Specify Joint Thickness View Factor greater than 0.0        *
*  Example: Joint Thickness View Factor = 1.0                  *
*                                                              *
*  2. Select SMAP-2D > Mesh > F.E. Mesh > Open
*
*  This wil open Mesh File of Continuum Elements.              *
*                                                              *
*  Input file Joint.inp should exist in Working Directory.     *
*  Output file JointedMesh.Mes is shown in Working Directory.  *
*                                                              *
```

```
* AllJoint  ThicAJ                                                  *
*-------------------------------------------------------------------*
  1         0.03
*                                                                   *
*********************************************************************
* Card 2                                                            *
* Internal Joint Generation By AllJoint = 1                         *
*********************************************************************
*                                                                   *
* Card 2.1                                                          *
* NumIJ  (Number of Continuum Materials for Internal Joints)        *
* ThicIJ (Joint Thickness)                                          *
*                                                                   *
* NumIJ     ThicIJ                                                  *
*-------------------------------------------------------------------*
  1         1.0
*                                                                   *
* Card 2.2                                                          *
* MatIJ (Material No of Continuum Element for Internal Joints)      *
*           InnerBeam = 0: No  1: Includes Inner Beam               *
*           OuterBeam = 0: No  1: Includes Outer Beam               *
* MatIJ     InnerBeam   OuterBeam                                   *
*-------------------------------------------------------------------*
  3         0           1
*                                                                   *
*********************************************************************
* Card 3                                                            *
* Boundary Joint Generation By AllJoint = 1                         *
*********************************************************************
*                                                                   *
* Card 3.1                                                          *
* NumBJ  (Number of Continuum Materials for Boundary Joints)        *
* ThicBJ (Joint Thickness)                                          *
*         InterfaceJoint = 0: No  1: Includes Joint Element         *
* NumBJ     ThicBJ      InterfaceJoint                              *
*-------------------------------------------------------------------*
* 3         0.03        1
  0         0.03        1
*                                                                   *
* Card 3.2                                                          *
* MatBJ (Material No of Continuum Element for Boundary Joints)      *
*        InnerBeam = 0: No  1: Includes Inner Beam                  *
*        OuterBeam = 0: No  1: Includes Outer Beam                  *
*                                                                   *
* MatBJ     InnerBeam   OuterBeam                                   *
*-------------------------------------------------------------------*
* 1         1           1
* 2         1           1
* 3         1           1
*                                                                   *
*********************************************************************
```

```
* Card 4                                                        *
* Surface Joint Generation By AllJoint = 2                      *
*****************************************************************
*                                                               *
* Card 4.1                                                      *
* NumSJG (Number of Groups for Surface Joints)                  *
*                                                               *
* NumSJG                                                        *
*---------------------------------------------------------------*
  0
* 2
*                                                               *
* Card 4.2                                                      *
* NumSJG [i]   : Number of Element Surfaces in Group i          *
* ThicSJG[i]   : Thickness of Surface Joint in Group i          *
*                                                               *
* NumSJG [i]   ThicSJG[i]                                       *
*---------------------------------------------------------------*
*  2           0.5
*  2           0.4
*****************************************************************
* Group (1)                                                     *
* ElementNo    SurfaceNo                                        *
*---------------------------------------------------------------*
*  1           1
*  2           2
*****************************************************************
* Group (2)                                                     *
* ElementNo    SurfaceNo                                        *
*---------------------------------------------------------------*
*  3           4
*  4           4
* End of Data                                                   *
*****************************************************************
```

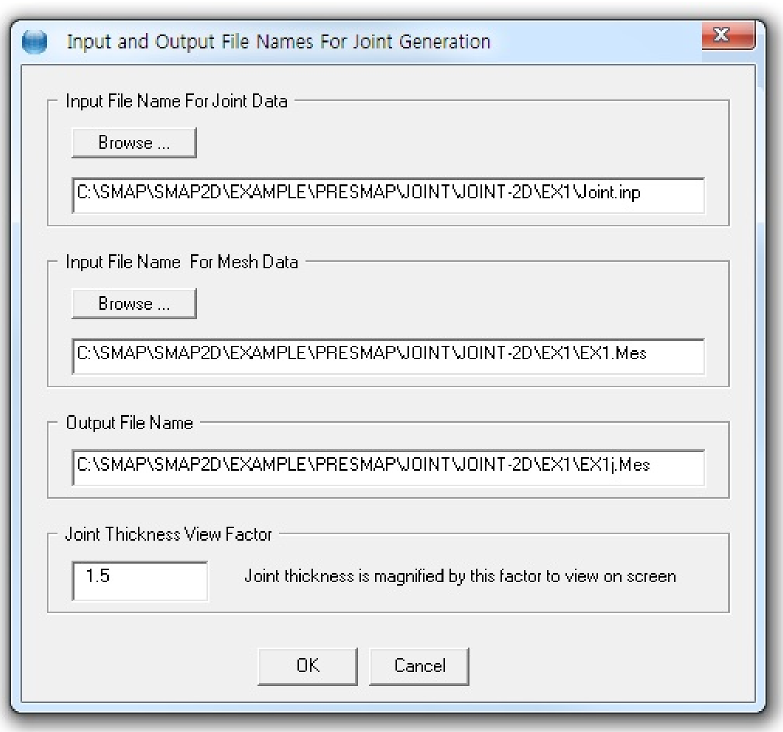


Figure 7.90 Dialog for input and output file names for joint generation

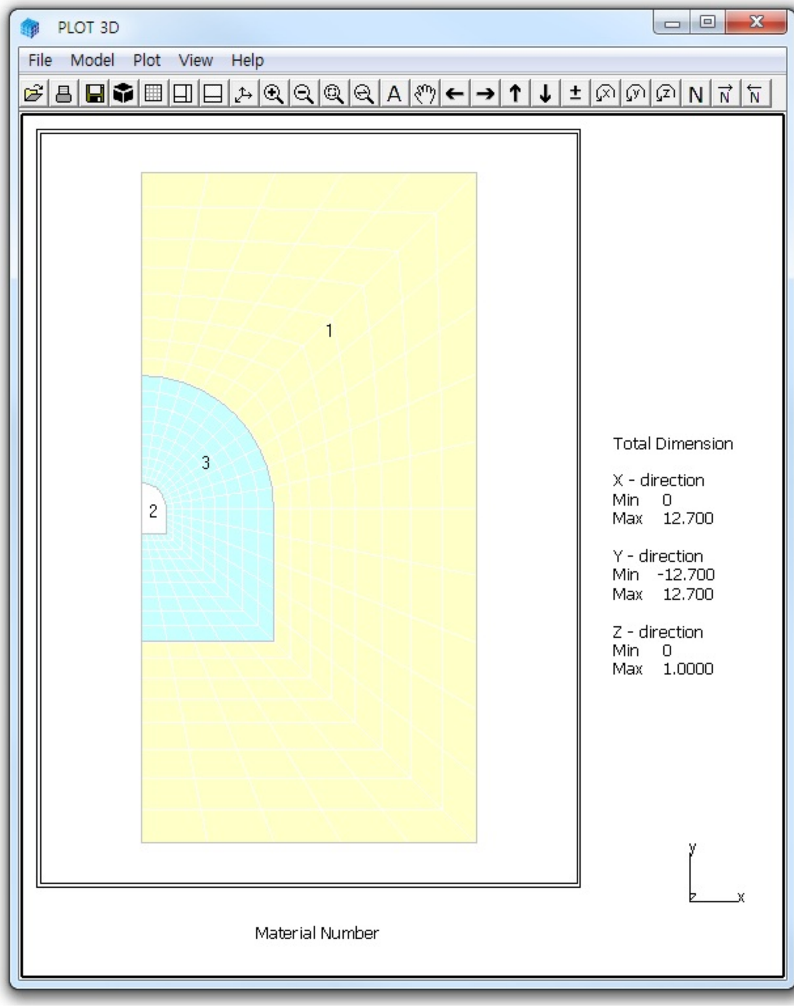


Figure 7.91 Material numbers in input mesh for Example 1

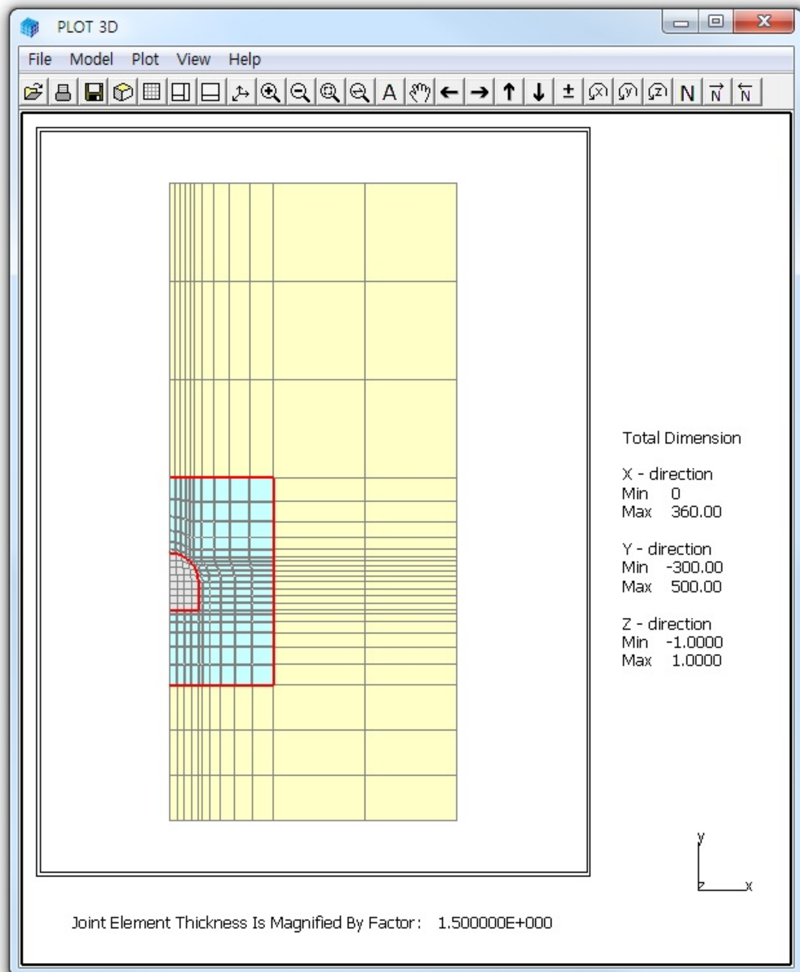


Figure 7.92 Generated jointed finite element mesh for Example 1

7.5.2 Example 2: Arch Tunel with Internal Joints

Example 2 shows jointed continuum elements which are generated within the arch tunnel. Figure 7.93 shows material numbers of continuum elements: 1 to 3 representing for arch tunnel in the input mesh. Joint data is prepared to generate internal joints within the arch tunnel as listed in Table 7.26. Note that it also specify Inner and Outer Beams between arch tunnel and surrounding soils.

Figure 7.94 shows generated jointed finite element meshes within arch tunnel and beam elements along the boundary.

Table 7.26 Listing of input file Joint.inp for Example 2

```
****************************************************************
* Jointed Continuum Generation                                 *
* =============================================================*
*                                                              *
* Card 1.1                                                     *
* Title                                                        *
  Example 2: Arch Tunnel with Internal Joints
* Card 1.2                                                     *
* AllJoint                                                     *
*  = 0 Generate Joint Elements along all interfaces            *
*      between continuum elements.                             *
*      Cards 2, 3 and 4 are not used.                          *
*                                                              *
*  = 1 Generate Joint Elements for material numbers of         *
*      continuum elements as specified in Cards 2 and 3.       *
*      Card 4 is not used.                                     *
*                                                              *
*  = 2 Generate Joint Elements for element surface numbers of  *
*      continuum elements as specified in Card 4.              *
*      Cards 2 and 3 are ignored.                              *
*                                                              *
* ThicAJ  Joint Thickness Used For AllJoint = 0                *
*                                                              *
*--------------------------------------------------------------*
*                                                              *
* AllJoint  ThicAJ                                             *
*--------------------------------------------------------------*
  1         0.03
*                                                              *
****************************************************************
```

```
* Card 2                                                        *
* Internal Joint Generation By AllJoint = 1                     *
*****************************************************************
*                                                               *
* Card 2.1                                                      *
* NumIJ  (Number of Continuum Materials for Internal Joints)    *
* ThicIJ (Joint Thickness)                                      *
*                                                               *
* NumIJ     ThicIJ                                              *
*---------------------------------------------------------------*
  2         0.03
* Card 2.2                                                      *
* MatIJ (Material No of Continuum Element for Internal Joints) *
*           InnerBeam = 0: No  1: Includes Inner Beam           *
*           OuterBeam = 0: No  1: Includes Outer Beam           *
* MatIJ     InnerBeam   OuterBeam                               *
*---------------------------------------------------------------*
  1         1           1
  3         1           1
*                                                               *
*****************************************************************
* Card 3                                                        *
* Boundary Joint Generation By AllJoint = 1                     *
*****************************************************************
*                                                               *
* Card 3.1                                                      *
* NumBJ  (Number of Continuum Materials for Boundary Joints)    *
* ThicBJ (Joint Thickness)                                      *
*         InterfaceJoint = 0: No  1: Includes Joint Element     *
* NumBJ     ThicBJ      InterfaceJoint                          *
*---------------------------------------------------------------*
  0         0.03        1
*
*****************************************************************
* Card 4                                                        *
* Surface Joint Generation By AllJoint = 2                      *
*****************************************************************
*                                                               *
* Card 4.1                                                      *
* NumSJG (Number of Groups for Surface Joints)                  *
*                                                               *
* NumSJG                                                        *
*---------------------------------------------------------------*
  0
* End of Data                                                   *
*****************************************************************
```

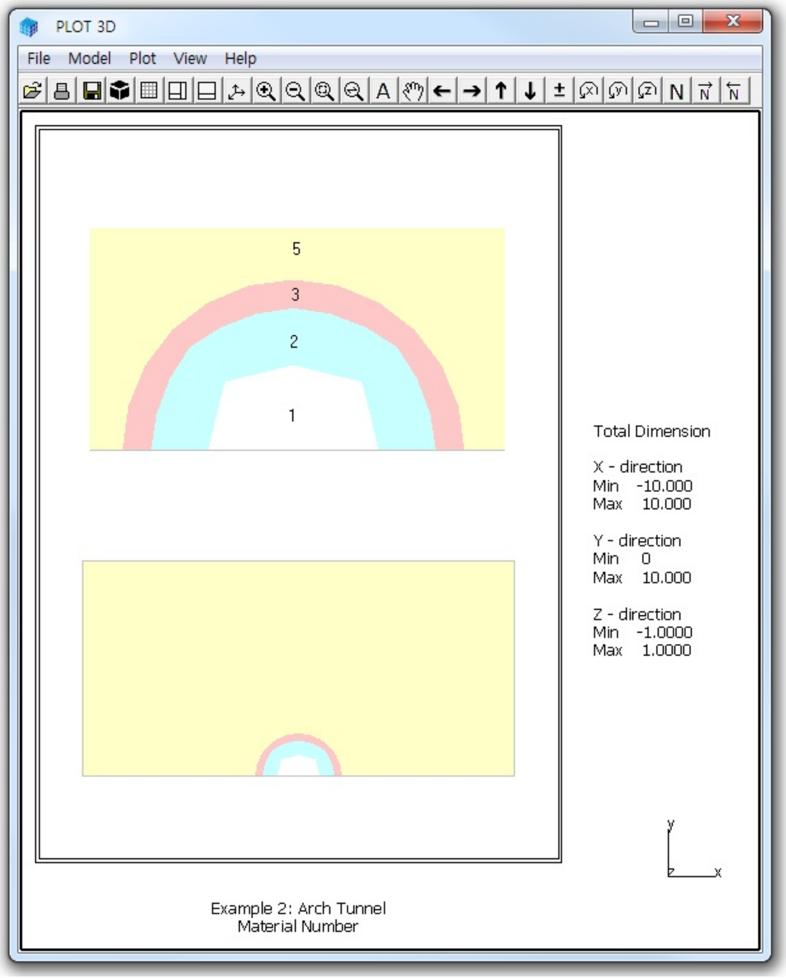


Figure 7.93 Material numbers in input mesh for Example 2

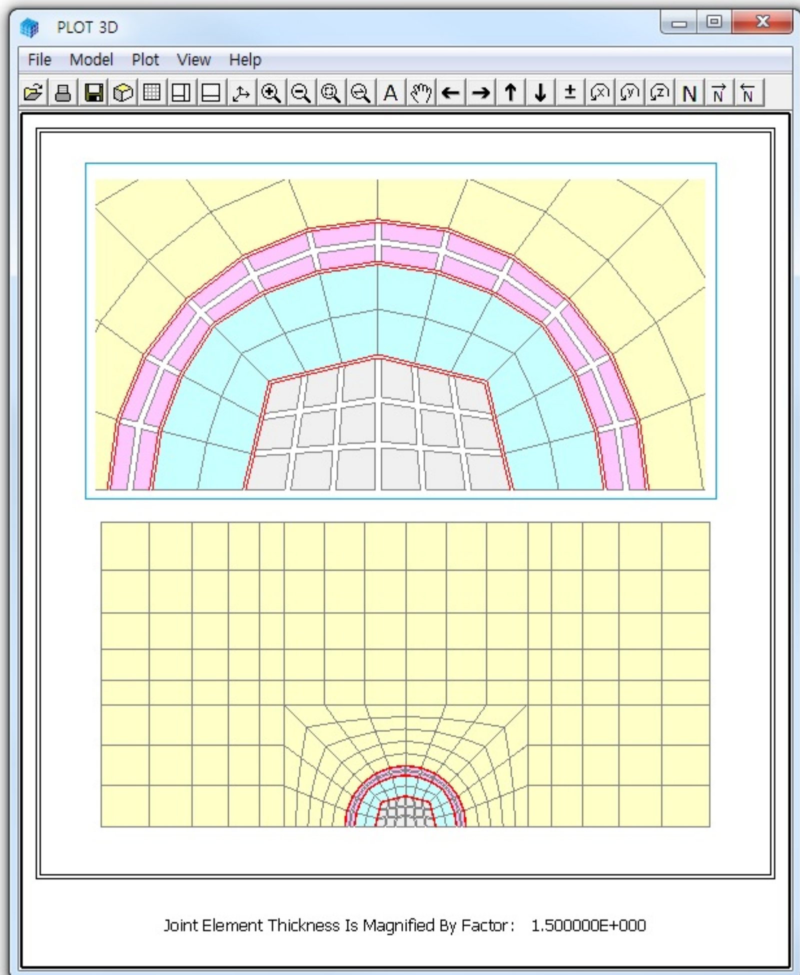


Figure 7.94 Generated jointed finite element mesh for Example 2

7.5.3 Example 3: Arch Tunnel with Boundary Joints

Example 3 is the same as Example 2 except that it generates boundary joints along the interface between the arch tunnel and surrounding soils. Joint data is prepared to generate boundary joints along the interface as listed in Table 7.27. Note that it also specify Inner and Outer Beams between arch tunnel and surrounding soils.

Figure 7.95 shows generated boundary joint elements and beam elements along the interface between arch tunnel and surrounding soils.

Table 7.27 Listing of input file Joint.inp for Example 3

```
****************************************************************
* Jointed Continuum Generation                                 *
* ==============================================================*
*                                                              *
* Card 1.1                                                     *
* Title                                                        *
  Example 3: Arch Tunnel with Boundary Joints
* Card 1.2                                                     *
* AllJoint                                                     *
*  = 0 Generate Joint Elements along all interfaces            *
*      between continuum elements.                             *
*      Cards 2, 3 and 4 are not used.                          *
*                                                              *
*  = 1 Generate Joint Elements for material numbers of         *
*      continuum elements as specified in Cards 2 and 3.       *
*      Card 4 is not used.                                     *
*                                                              *
*  = 2 Generate Joint Elements for element surface numbers of  *
*      continuum elements as specified in Card 4.              *
*      Cards 2 and 3 are ignored.                              *
*                                                              *
* ThicAJ  Joint Thickness Used For AllJoint = 0                *
*                                                              *
*--------------------------------------------------------------*
*                                                              *
* AllJoint  ThicAJ                                             *
*--------------------------------------------------------------*
  1         0.03
*                                                              *
****************************************************************
```

```
* Card 2                                                        *
* Internal Joint Generation By AllJoint = 1                     *
*****************************************************************
*                                                               *
* Card 2.1                                                      *
* NumIJ  (Number of Continuum Materials for Internal Joints)    *
* ThicIJ (Joint Thickness)                                      *
*                                                               *
* NumIJ     ThicIJ                                              *
*---------------------------------------------------------------*
  0         0.03
*                                                               *
*****************************************************************
* Card 3                                                        *
* Boundary Joint Generation By AllJoint = 1                     *
*****************************************************************
*                                                               *
* Card 3.1                                                      *
* NumBJ  (Number of Continuum Materials for Boundary Joints)    *
* ThicBJ (Joint Thickness)                                      *
*         InterfaceJoint = 0: No  1: Includes Joint Element     *
* NumBJ     ThicBJ      InterfaceJoint                          *
*---------------------------------------------------------------*
  2         0.03        1
*                                                               *
* Card 3.2                                                      *
* MatBJ (Material No of Continuum Element for Boundary Joints) *
*        InnerBeam = 0: No  1: Includes Inner Beam              *
*        OuterBeam = 0: No  1: Includes Outer Beam              *
*                                                               *
* MatBJ     InnerBeam   OuterBeam                               *
*---------------------------------------------------------------*
  1         1           1
  3         1           1
*                                                               *
*****************************************************************
* Card 4                                                        *
* Surface Joint Generation By AllJoint = 2                      *
*****************************************************************
*                                                               *
* Card 4.1                                                      *
* NumSJG (Number of Groups for Surface Joints)                  *
*                                                               *
* NumSJG                                                        *
*---------------------------------------------------------------*
  0
* End of Data                                                   *
*****************************************************************
```

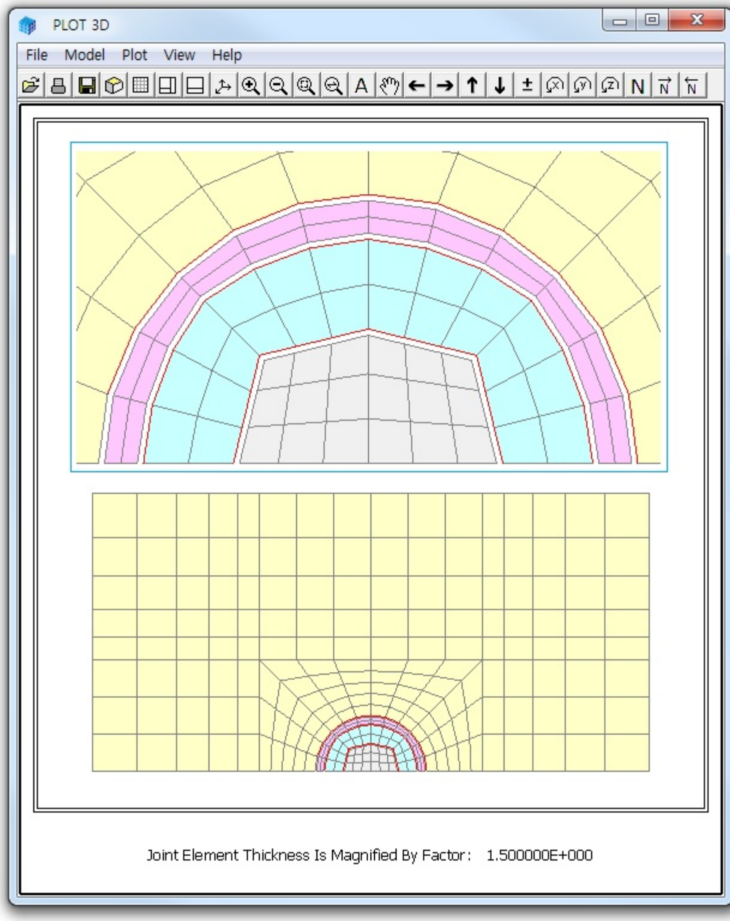


Figure 7.95 Generated jointed finite element mesh for Example 3

ADDRGN
Example Problem

ADDRGN is the pre-processing program which has the following functions:

- Combine two different meshes
- Modify existing meshes
  - Change coordinates
  - Change boundary codes
  - Cut elements
  - Change material numbers
- Generate finite element meshes (ADDRGN-2D)

Refer to SMAP-2D User's Manual:

- Section 5 for group mesh generation (ADDRGN-2D)
- Section 8 for input parameters

8.1 ADDRGN-2D

ADDRGN-2D is the two dimensional pre-processor which is used to combine, modify, or generate finite element meshes.

ADDRGN-2D can be selected in the following order:

Run → Mesh Generator → AddRgn → Addrgn 2D

When you finish the execution of ADDRGN-2D, select PLOT-3D to plot modified or generated mesh.

8.1.1 Combining Meshes

In the PRESMAP-2D Example Problem in Sections 7.1.1 and 7.1.2, three different regions (Core, Near-field, and Far-field) are generated using Models 1 and 2. Now, we want to combine all these different regions into one using ADDRGN-2D. Note that CORE.Mes, NEAR.Mes and FAR.Mes are the output files corresponding to the input file CORE.Rgn, NEAR.Rgn and FAR.Rgn respectively.

Element numbers 1 to 72 are assigned for CORE.Mes, 73 to 336 for NEAR.Mes and 337 to 464 for FAR.Mes. When we combine two regions, element numbers should be continuous through the regions. So, let's first add NEAR.Mes (called REGION B) to CORE.Mes (called REGION A) to make CONE.Mes (called COMBINED REGION). Next, let's add FAR.Mes (called REGION B) to CONE.Mes (called REGION A) to make the final mesh CNF.Mes (called COMBINED REGION). ADDRGN input files are listed in Tables 8.1 and 8.2.

Figure 8.1 shows the element meshes of combined region representing all three regions.

Table 8.1 Listing of input file ADD2D-1.Dat

```
* ADD2D-1.Dat
* CARD 1.1
* IMOD = 0 : ADD REGION B TO REGION A
  0
* CARD 2.1
* FILEA : Input file name containing REGION A
  CORE.Mes
* FILEB : Input file name containing REGION B
  NEAR.Mes
* FILEC : Output file name to store COMBINED REGION
  CONE.Mes
* CARD 2.2
* INTERFACE
  0
* END OF DATA
```

Table 8.2 Listing of input file ADD2D-2.Dat

```
* ADD2D-2.Dat
* CARD 1.1
* IMOD = 0 : ADD REGION B TO REGION A
  0
* CARD 2.1
* FILEA : Input file name containing REGION A
  CONE.Mes
* FILEB : Input file name containing REGION B
  FAR.Mes
* FILEC : Output file name to store COMBINED REGION
  CNF.Mes
* CARD 2.2
* INTERFACE
  0
* END OF DATA
```

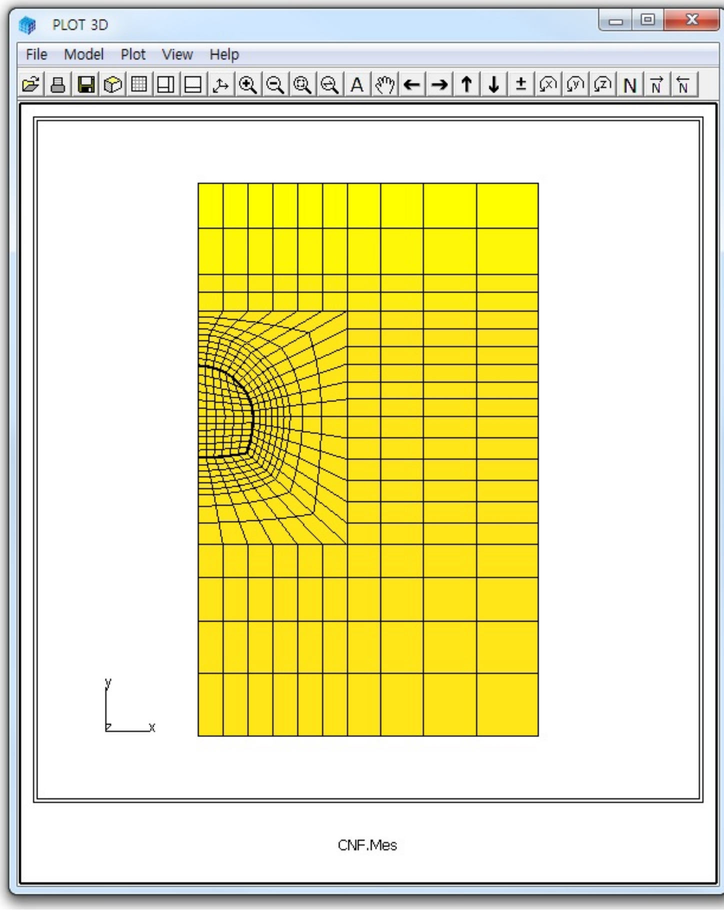


Figure 8.1 Final element meshes representing Core, Near-field, and Far-field regions, CNF.Mes

8.1.2 Modifying Mesh

In this example, we want to generate symmetric meshes using ADDRGN-2D. As the existing mesh, we take the CORE.Mes which has been generated using PRESMAP-2D Model 1 (refer to Section 7.1.1.2). Note that CORE.Mes represents the right side of the tunnel core. ADDRGN input file to generate Left Core is listed in Table 8.3. The output file LCORE.Mes contains Left Core whose graphical output is shown in Figure 8.2.

By combining both left and right core regions as instructed in Table 8.4, we can generate a whole core region, WCORE.Mes. Graphical output of WCORE.Mes is shown in Figure 8.3.

Table 8.3 Listing of input file ADD2D-3.Dat

```
* ADD2D-3.Dat
* CARD 1.1
* IMOD = 1 : MODIFY EXISTING MESH
  1
* CARD 3.1
* FILEA : Input file name to be modified
  CORE.Mes
* FILEM : Output file name to store modified mesh
  LCORE.Mes
* CARD 3.2
* NSNEL  NSNODE
  73     1
* CARD 3.3
* IEDIT = 0 : CHANGE COORDINATES
  0
* CARD 3.3.1.1
* Xo    Yo    Xonew   Yonew
  0.0   0.0   0.0     0.0
* CARD 3.3.1.2
* Xscale  Yscale
 -1.0      1.0
* END OF DATA
```

Table 8.4 Listing of input file ADD2D-4.Dat

```
* ADD2D-4.Dat
* CARD 1.1
* IMOD = 0 : ADD REGION B TO REGION A
  0
* CARD 2.1
* FILEA : Input file name containing REGION A
  CORE.Mes
* FILEB : Input file name containing REGION B
  LCORE.Mes
* FILEC : Output file name to store COMBINED REGION
  WCORE.Mes
* CARD 2.2
* INTERFACE
  0
* END OF DATA
```

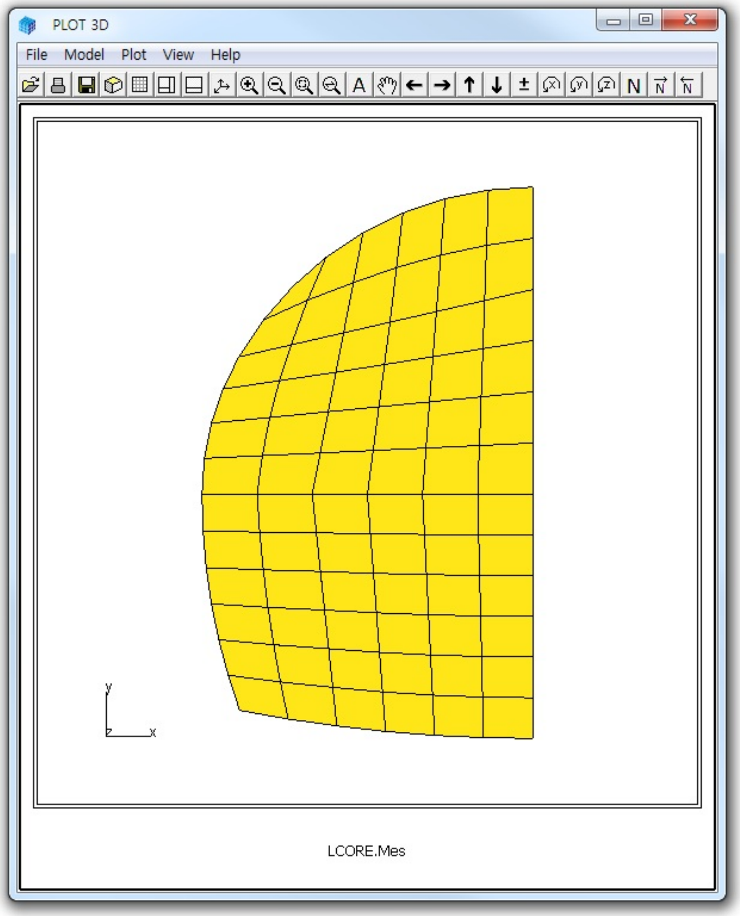


Figure 8.2 Left core mesh, LCORE.Mes

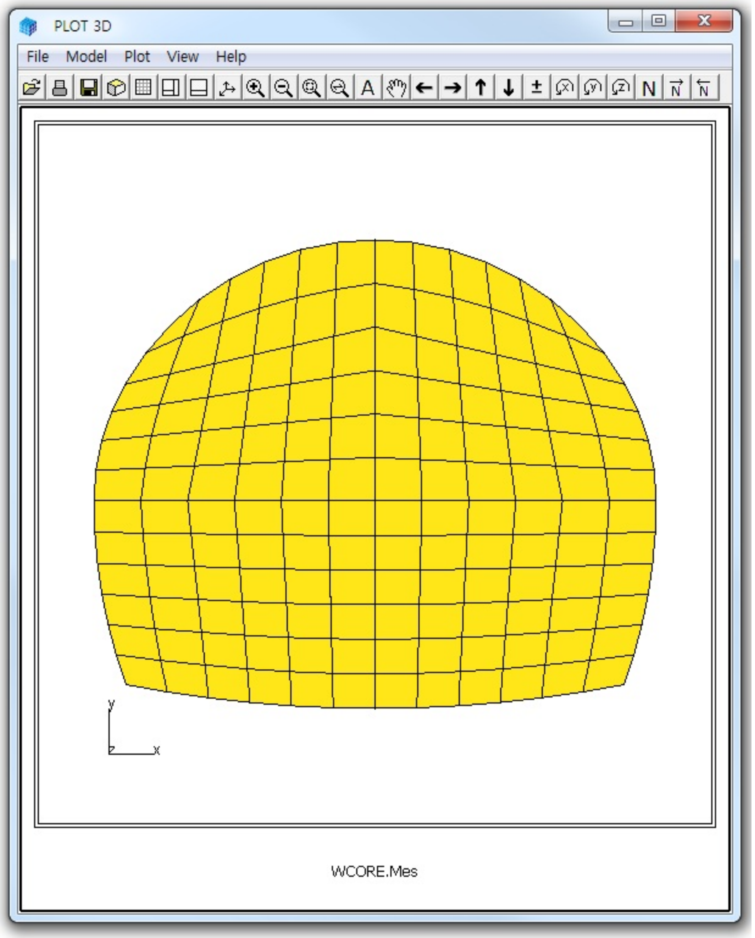


Figure 8.3 Combined whole core mesh, WCORE.Mes

8.1.3 Generating Mesh

This example is to show a powerful mesh generation feature using ADDRGN-2D. All you need to do is to specify the locations, dimensions and material numbers of structures along with few instructions for mesh generation. ADDRGN-2D will do the rest of the work to build the Mesh File.

As the first example, we take a simple problem as schematically shown in Figure 8.4. A utility tunnel with a diameter of 4 meters is located 6 meters below the ground surface. Table 8.5 shows the full listing of input file ADD2D-5.Dat. The base mesh consists of 3 blocks in the horizontal direction and 1 block in the vertical direction.

The first group represents soft rock underlying soil. And the second group represents the utility tunnel. Tunnel liner is modeled by beam element and the interface between the liner and the surrounding soil is modeled by joint element which will allow the slippage and separation. Finite element meshes generated by ADDRGN-2D are shown in Figures 8.5 and 8.6. It should be noted that the joint thickness in Figure 8.6 is exaggerated to show clearly both inner and outer joint faces. The real joint thickness is specified in material property card in Main File.

Table 8.5 Listing of input file ADD2D-5.Dat

```
* ADD2D-5.Dat
* CARD 1.1
* IMOD = 2 : GENERATE BASE MESH AND THEN MODIFY
* IMOD JK
   2    3
* CARD 4.1
* NBX  NBY
   3    1
* CARD 4.2
* XO    YO
  0.0   0.0
* CARD 4.3
* W     DX     ALPAX
  14.0  0.3   -0.3
  21.0  0.3    0.5
  11.0  0.3    0.3
* CARD 4.4
* H     DY     ALPAY
  20.0  0.3    0.5
* CARD 4.5
* IGMOD
  1
* -------------------------------------------------------------
* CARD 3.1
* FILEA
  BMESH.DAT
* FILEM
  ADD2D-5.Mes
* CARD 3.2
* NSNEL  NSNODE
      1       1
* CARD 3.3
* IEDIT = 4 : BUILD USER-SPECIFIED CURVES.
     4
* CARD 3.3.5.1
* NODE
     0
* CARD 3.3.5.2
* NOEL
     0
* CARD 3.3.5.3
* IBOUND
     0
```

```
* CARD 3.3.5.4
* NGROUP
    2
* XREF  YREF
  14.0  20.0
* -------------------- GROUP 1  ------------------------------
*
*                      SOFT ROCK
*
* CARD 3.3.5.4.1.1
* MTYPE
     3
* CARD 3.3.5.4.1.2
* MATNO  KF   LTPI  LMAT
     7   0     0     0
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO   YLO
     6     1     0    0.0   0.0
* CARD 3.3.5.4.2.2
* NP   X     Y
   1   0.0   0.0
   2  46.0   0.0
   3  46.0  13.0
   4  31.0  12.0
   5  19.0   8.0
   6   0.0   4.0
* CARD 3.3.5.4.3
* NSEGMENT
     6
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     1      1    0     3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     2      1    0     3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     3      1    0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     4      1    0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     5      1    0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     6      1    0     2
```

```
* --------------------- GROUP 2 -------------------------------
*                  UTILITY TUNNEL
* CARD 3.3.5.4.1.1
* MTYPE
    -3
* CARD 3.3.5.4.1.2
* MATNO  KF  MATNOJT  KFJT  THICJT  LTPI, LMATI, LTPO, LMATO
     3   0     4       0     0.1     2     5      2     6
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO   YLO
     1      0    1    8.0  -6.0
* CARD 3.3.5.4.2.2
* NP   X     Y
   1   2.0   0.0
* CARD 3.3.5.4.3
* NSEGMENT
     1
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV IEND
     1      2     0   2
* CARD 3.3.5.4.3.2
* X0    Y0    RX   RY   THETA_B   THETA_E
  0.0   0.0   2.0  2.0  0.0       360.
* ------------------------------------------------------------
* END OF DATA
```

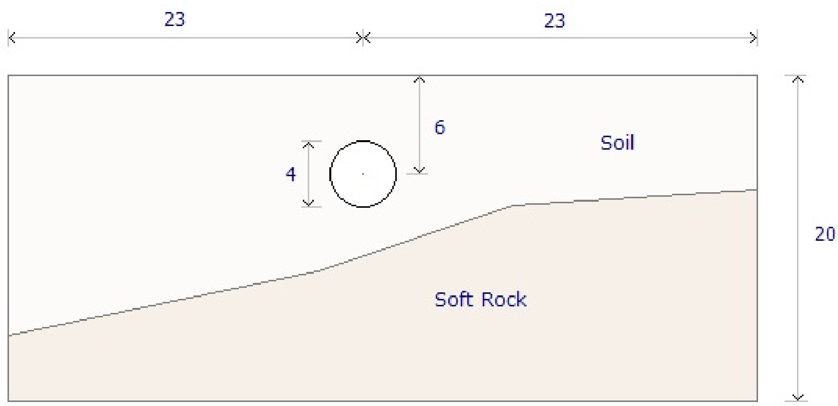


Figure 8.4 Schematic section view.

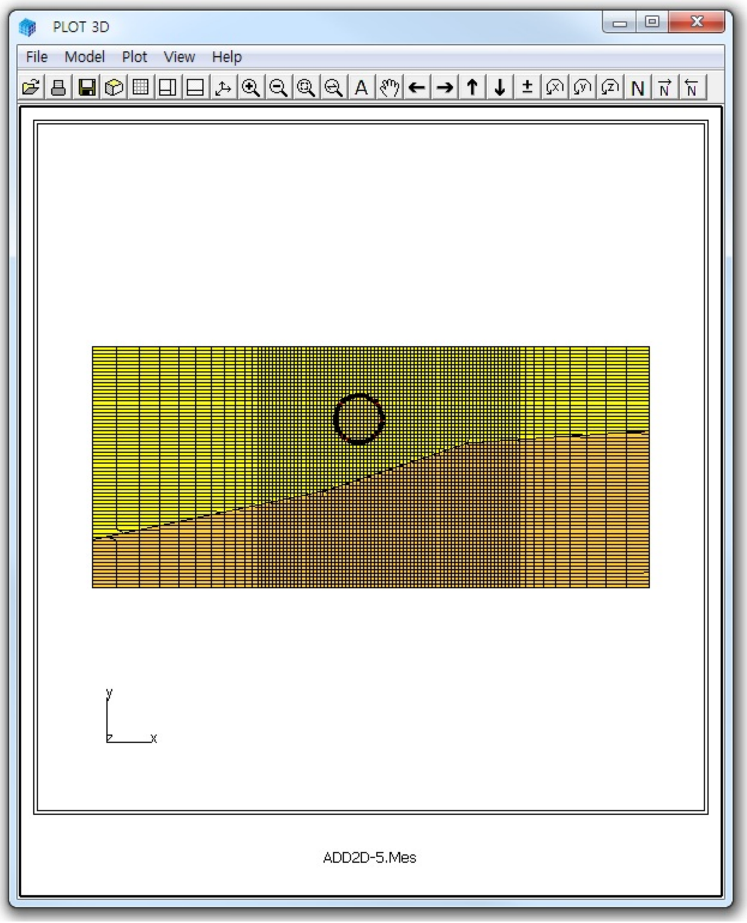


Figure 8.5 Overall finite element mesh

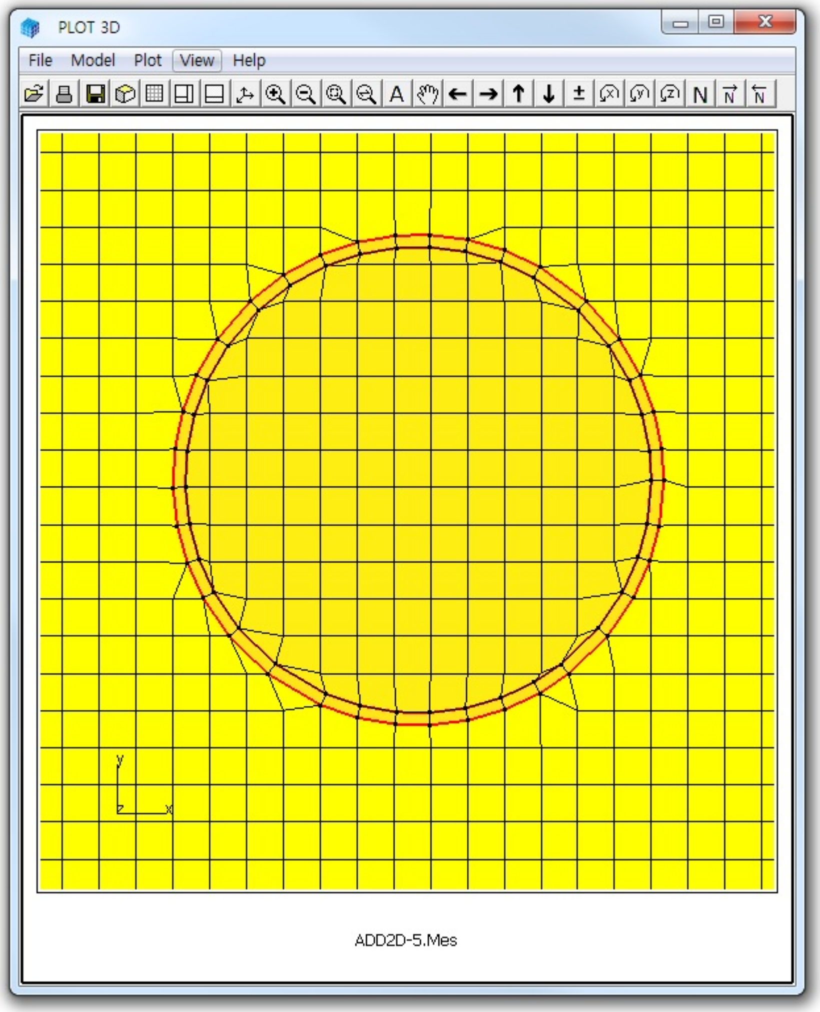


Figure 8.6 Finite element mesh around utility tunnel.

As the second example, we take a complex problem as schematically shown in Figure 8.7. The problem geometry includes different types of underground structures; strut, anchor bar, pile, utility tunnel, subway tunnel, rock bolt, foundation and fault zone. Table 8.6 shows the partial listing of input file ADD2D-6.Dat. The base mesh consists of 3 blocks in the horizontal direction and 2 blocks in the vertical direction.

For detailed description of input parameters, refer to Section 8.2 in SMAP-2D User's Manual. Joint elements are used to model the fault zone and the interfaces between surrounding medium and the structures such as pile and tunnels. Figure 8.8 shows overall finite element mesh generated by ADDRGN-2D. Detailed finite element meshes are shown in Figure 8.9 for the excavation zone and in Figure 8.10 for the tunnels and foundation. As in the previous example, the joint thickness in Figures 8.9 and 8.10 is exaggerated to show clearly both inner and outer joint faces. The real joint thickness is specified in material property card in Main File.

Table 8.6 Listing of input file ADD2D-6.Dat (Partial Listing)

```
* ADD2D-6.Dat
* CARD 1.1
* IMOD = 2 : GENERATE BASE MESH AND THEN MODIFY
* IMOD JK
   2    3
* CARD 4.1
* NBX  NBY
   3    2
* CARD 4.2
* XO    YO
  0.0   0.0
* CARD 4.3
* W     DX     ALPAX
  14.0  0.3   -0.3
  21.0  0.3    0.5
  11.0  0.3    0.3
* CARD 4.4
* H     DY     ALPAY
  23.0  0.3    0.5
  16.0  0.3    0.3
```

```
* CARD 4.5
* IGMOD
  1
* -------------------------------------------------------------
* CARD 3.1
* FILEA
  BMESH.DAT
* FILEM
  ADD2D-6.Mes
* CARD 3.2
* NSNEL  NSNODE
     1       1
* CARD 3.3
* IEDIT = 4 : BUILD USER-SPECIFIED CURVES.
     4
* CARD 3.3.5.1
* NODE
     0
* CARD 3.3.5.2
* NOEL
     0
* CARD 3.3.5.3
* IBOUND
     0
* CARD 3.3.5.4
* NGROUP
   22
* XREF  YREF
  14.0  39.0
* --------------------- GROUP 1  ------------------------------
*
*               MAKING GROUND SURFACE
*
* CARD 3.3.5.4.1.1
* MTYPE
    -1
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO   YLO
     8     1     0    0.0   0.0
* CARD 3.3.5.4.2.2
* NP    X     Y
   1  46.0  34.0
   2  39.0  34.0
   3  33.0  39.0
   4  18.0  39.0
   5  12.0  34.0
   6   0.0  34.0
   7   0.0   0.0
   8  46.0   0.0
```

```
* CARD 3.3.5.4.3
* NSEGMENT
     8
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     1      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
     2      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     3      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
     4      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     5      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
     6      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     7      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
     8      1     0     2
* --------------------- GROUP 2  ------------------------------
*
*                      SOFT ROCK
*
* CARD 3.3.5.4.1.1
* MTYPE
     3
* CARD 3.3.5.4.1.2
* MATNO  KF   LTPI  LMAT
     7   0     0     0
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO   YLO
     6     1     0    0.0   0.0
* CARD 3.3.5.4.2.2
* NP   X     Y
   1  46.0   0.0
   2  46.0  33.0
   3  31.0  32.0
   4  19.0  28.0
   5   0.0  24.0
   6   0.0   0.0
```

```
* CARD 3.3.5.4.3
* NSEGMENT
     6
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     1      1    0     3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     2      1    0     3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     3      1    0     0
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     4      1    0     0
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     5      1    0     0
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     6      1    0     3
* --------------------- GROUP 3  ------------------------------
*
*                        FAULT
*
* MTYPE
    -2
* CARD 3.3.5.4.1.2
* MATNOJT  KFJT  THICJT  LTPI, LMATI, LTPO, LMATO
    5        0   -0.1     0     0      0     0
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO   YLO
     4     1     0    0.0   0.0
* CARD 3.3.5.4.2.2
* NP    X      Y
   1  46.0  29.0
   2  29.0  19.0
   3  16.0  14.0
   4   0.0  10.0
* CARD 3.3.5.4.3
* NSEGMENT
     3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     1      1    0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
     2      1    0     2
```

```
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     3      1     0     2
* --------------------- GROUP 4  ------------------------------
*
*                      FOUNDATION
*
* CARD 3.3.5.4.1.1
* MTYPE
     4
* CARD 3.3.5.4.1.2
* MATNO   KF  LTPI  LMAT
     2    0    0     0
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO   YLO
     8     1     0    0.0   0.0
* CARD 3.3.5.4.2.2
* NP   X     Y
   1  34.5  29.0
   2  34.5  30.0
   3  32.5  30.5
   4  32.5  39.0
   5  31.5  39.0
   6  31.5  30.5
   7  29.5  30.0
   8  29.5  29.0
* CARD 3.3.5.4.3
* NSEGMENT
     8
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     1      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     2      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     3      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     4      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     5      1     0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     6      1     0     2
* CARD 3.3.5.4.3.1
```

```
* SEGNO  LTYPE  NDIV  IEND
     7      1    0     2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     8      1    0     2
* --------------------- GROUP 5 -------------------------------
*
*                  LEFT UTILITY TUNNEL
*
* CARD 3.3.5.4.1.1
* MTYPE
    -3
* CARD 3.3.5.4.1.2
* MATNO  KF  MATNOJT  KFJT  THICJT  LTPI, LMATI, LTPO, LMATO
     3   0     4       0    -0.1     2     5      2     6
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO   YLO
     1      0    1    8.0  -6.0
* CARD 3.3.5.4.2.2
* NP   X     Y
   1   2.0   0.0
* CARD 3.3.5.4.3
* NSEGMENT
     1
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV IEND
     1      2     0   2
* CARD 3.3.5.4.3.2
* X0    Y0    RX   RY   THETA_B   THETA_E
  0.0   0.0   2.0  2.0  0.0       360.
* --------------------- GROUP 6 -------------------------------
*
*                  RIGHT UTILITY TUNNEL
*
* CARD 3.3.5.4.1.1
* MTYPE
    -3
* CARD 3.3.5.4.1.2
* MATNO  KF  MATNOJT  KFJT  THICJT  LTPI, LMATI, LTPO, LMATO
     3   0     4       0    -0.1     2     5      2     6
* CARD 3.3.5.4.2.1

---
---
```

```
* --------------------- GROUP 22 ------------------------------
*
*                      SUBWAY TUNNEL
*
* CARD 3.3.5.4.1.1
* MTYPE IGPOST OVERLAY GCOLOR GLTYPE GLTHIC GHIDE
  -3     0      0       0      0      0      0
* Card 3.3.5.4.1-1
* MAT  KF  MATj KFj  THICj  LTi LMi LTo LMo
    3   0    4   0  -0.100   2   5   2   6
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO   YLO
    4      1     1    0.0   0.0
* CARD 3.3.5.4.2.2
* NP   X     Y
   1   26.  24.
   2   20.  24.
   3   20.  20.
   4   26.  20
* CARD 3.3.5.4.3
* NSEGMENT
     4
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
     1      2    0     2
* Card 3.3.5.4.3.1-1
* Xo     Yo     Rx    Ry    Qb    Qe
  23.    24.    3.0   3.0   0.0   180.
* Card 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV   IEND
    2     1       0     2
* Card 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV   IEND
    3     1       0     2
* Card 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV   IEND
    4     1       0     2
* ------------------------------------------------------------
* END OF DATA
```

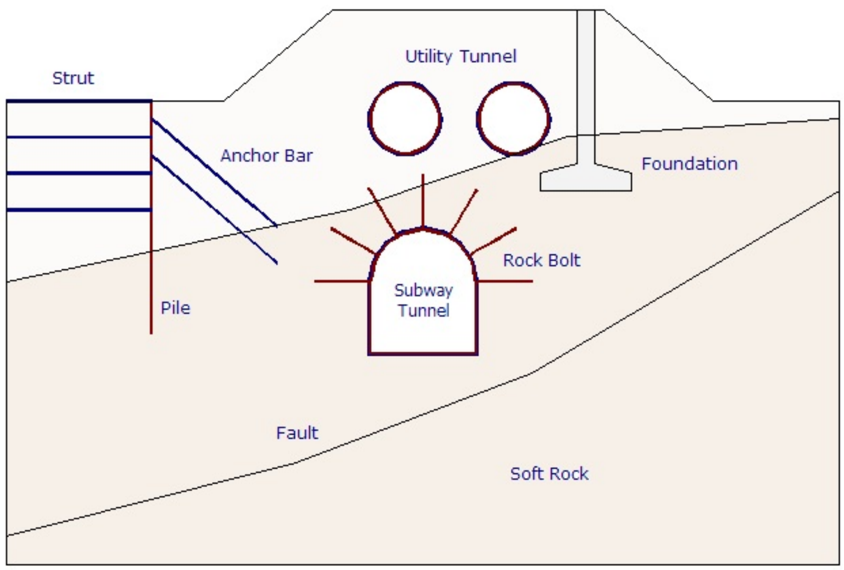


Figure 8.7 Schematic section view

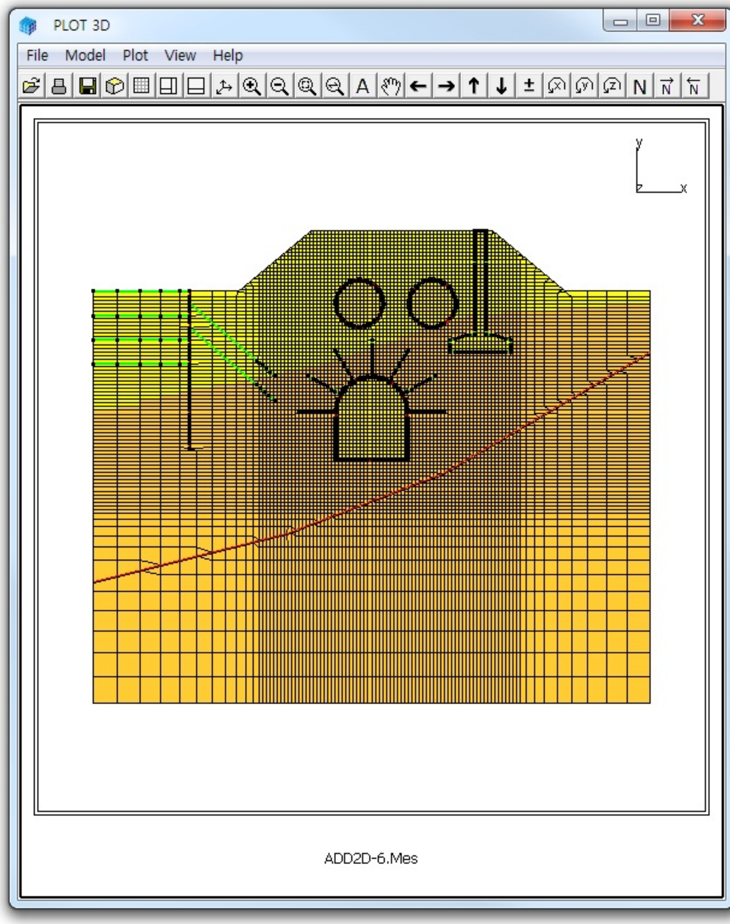


Figure 8.8 Overall finite element mesh

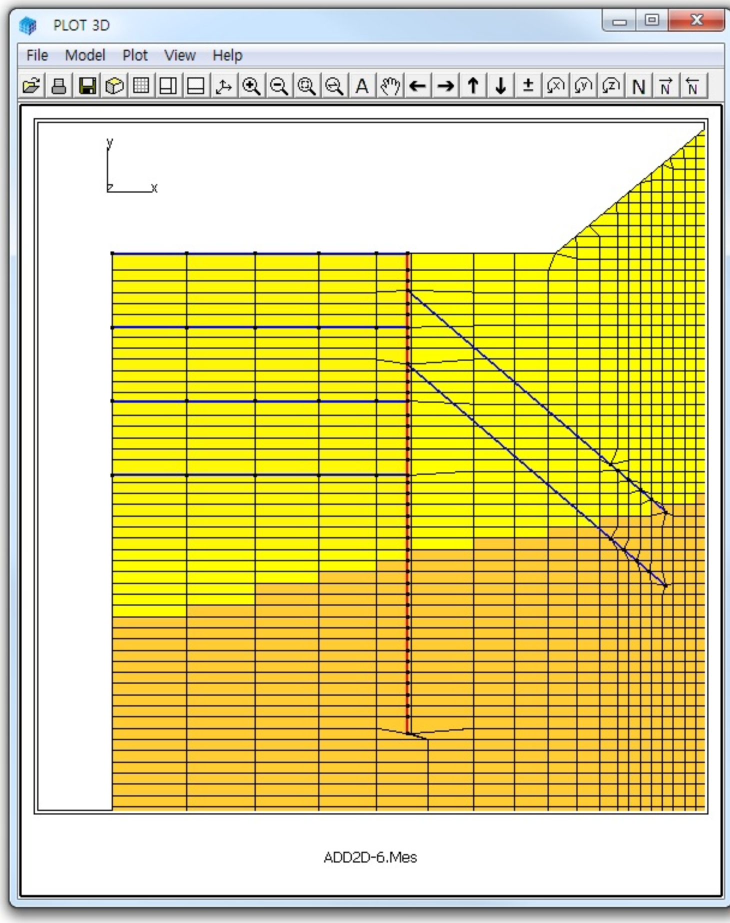


Figure 8.9 Finite element mesh around excavation zone

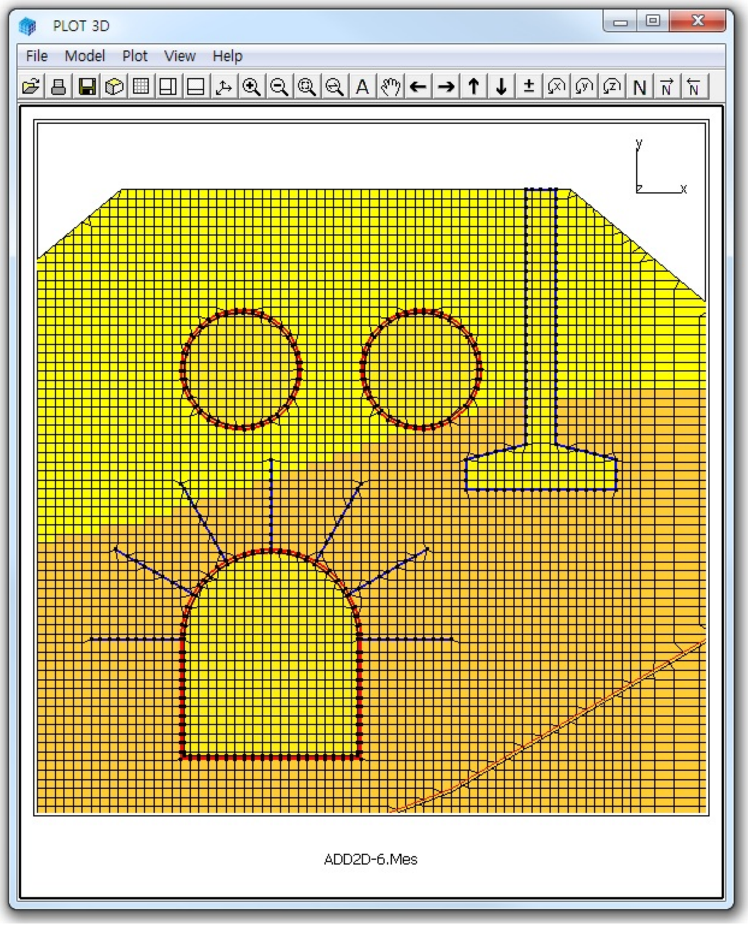


Figure 8.10 Finite element mesh around tunnels and foundation

SUPPLEMENT
Example Problem

SUPPLEMENT Menu contains supporting programs which are useful to prepare input data for pre- and main-processing programs of SMAP-2D.

Running SUPPLEMENT is described in Section 3.2.6 of User's Manual and can be selected in the following order:

Run → Mesh Generater → Supplement → Edit, XY. Cards or Shrink File

EDIT is used to run text editor.

XY computes coordinates of mid points, cross points, or normal points.

CARDS generates element activity data in Card 8 in Section 4.4 Main File.

SHRINK FILE removes extra blank spaces before carriage return.
This will reduce the size of the file.

9.1 XY Example Problem

XY is the supporting program which computes coordinates of mid points, cross points, or normal points. Full description of XY is presented in Section 9.3 of User's Manual.

As an example, we select NF=6 which computes coordinates of point normal to the circular arc as shown in Figure 9.1.

Table 9.1 illustrates options available to the program XY and the user inputs specific to NF=6. Computed coordinates of the normal point are stored in the output file XY.Out and are listed in bottom part of Table 9.l.

Table 9.1 XY Example Problem

Type file name to store output: XY.Out

NF =	0	END OF COMPUTATION.
	1	COMPUTE MIDPOINT ON STRAIGHT LINE.
	2	COMPUTE MIDPOINT ON CIRCULAR ARC.
	3	COMPUTE INTERSECTION POINT OF TWO STRAIGHT LINES.
	4	COMPUTE INTERSECTION POINT OF CIRCULAR ARC AND STRAIGHT LINE.
	5	COMPUTE POINTS NORMAL TO STRAIGHT LINE.
	6	COMPUTE POINTS NORMAL TO CIRCULAR ARC.

NF= **6**

R, Xo, Yo, TA

5.0 0.0 0.0 0.0

TAC, CD

45.0 3.0

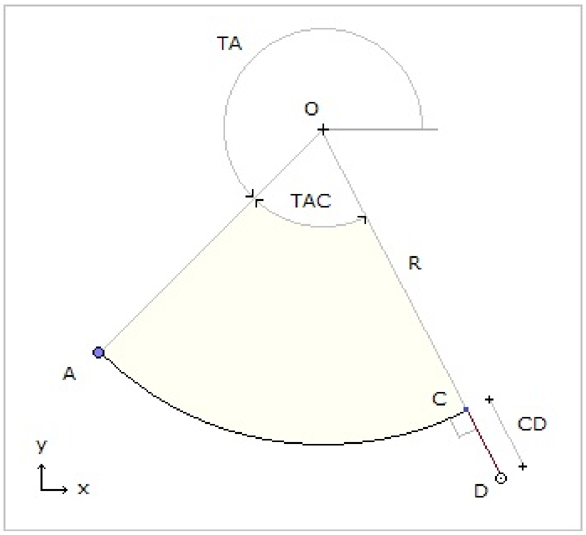
User inputs are **bold**.

Output file contains following information:

COMPUTED POINTS NORMAL TO CIRCULAR ARC

R	=	5.000000			
Xo	=	0.000000E+00	Yo	=	0.000000E+00
TA	=	0.000000E+00			
TAC	=	45.000000	CD	=	3.000000
XC	=	3.535527	YC	=	3.535540
XD	=	5.656844	YD	=	5.656865

NF = 6 Compute Points Normal to Circular Arc



INPUT:
R, X_o, Y_o, TA
TAC, CD

R	=	5.0
X_o, Y_o	=	0.0, 0.0
TA	=	0.0
TAC	=	45.0
CD	=	3.0

Figure 9.1 XY example problem

9.2 CARDS Example Problem

CARDS is the supporting program which is written to aid the preparation of SMAP-2D input cards. Currently, there is only one routine available to generate element activity data in Card Group 8.2 of Users Manual.

Table 9.2 shows user inputs for the example problem. Generated element activity data is stored in the output file, CARDS.Out, which is listed in Table 9.3.

Table 9.2 User inputs for CARDS example problem

CARD NO =	0	EXIT
	8.2	ELEMENT ACTIVITY

CARD NO = **8.2**
Type file name to store output: **CARDS.OUT**

NF =	0	END OF GENERATION
	1	GENERATE ELEMENT ACTIVITY/ DEACTIVITY

NF = **1**
NEL (start), NEL (end), NAC, NDAC
101 120 0 6

NF =	0	END OF GENERATION
	1	GENERATE ELEMENT ACTIVITY/ DEACTIVITY

NF = **1**
NEL (start), NEL (end), NAC, NDAC
121 130 3 50

NF =	0	END OF GENERATION
	1	GENERATE ELEMENT ACTIVITY/ DEACTIVITY

NF = **0**

User inputs are **bold**.

Table 9.3 Listing of output file CARDS.Out

```
* NEL   NAC  NDAC
*
  101     0     6
  102     0     6
  103     0     6
  104     0     6
  105     0     6
  106     0     6
  107     0     6
  108     0     6
  109     0     6
  110     0     6
  111     0     6
  112     0     6
  113     0     6
  114     0     6
  115     0     6
  116     0     6
  117     0     6
  118     0     6
  119     0     6
  120     0     6
*
  121     3    50
  122     3    50
  123     3    50
  124     3    50
  125     3    50
  126     3    50
  127     3    50
  128     3    50
  129     3    50
  130     3    50
* NFAD =    30
```

LOAD
Example Problem

10.1 LOAD-2D

LOAD-2D is the pre-processing program which can be used to generate external force (pressure), specified velocity, initial velocity, acceleration, and transmitting boundary. For the detailed description of input parameters, refer to section 11 of User's Manual.

LOAD-2D can be selected in the following order:

Run → Load Generator → Load 2D

When you select LOAD-2D, Load Generation Dialog will be displayed as in Figure 10.1. You need to specify input file names for Load and Mesh Data.

10.1.1 Example 1

Example 1 is to show the pressure load generation along the surfaces of elements 1, 2, 3 and 4 as schematically shown in Figure 10.2. Triangular pressure loads are acting on the surfaces of elements 1, 2 and 3. Right surfaces of elements 3 and 4 are subjected to the uniformly distributed pressure of 1.0. Two different load time histories, as shown in Figure 10.3, are considered.

Mesh Data contains information for nodal coordinates and element indexes. Mesh2D.Mes represents Mesh Data graphically shown in Figure 10.4 along with listing in Table 10.1. Load Data contains information for loads to be generated. Load2D.Dat in Table 10.2, has been prepared according to LOAD-2D User's Manual.

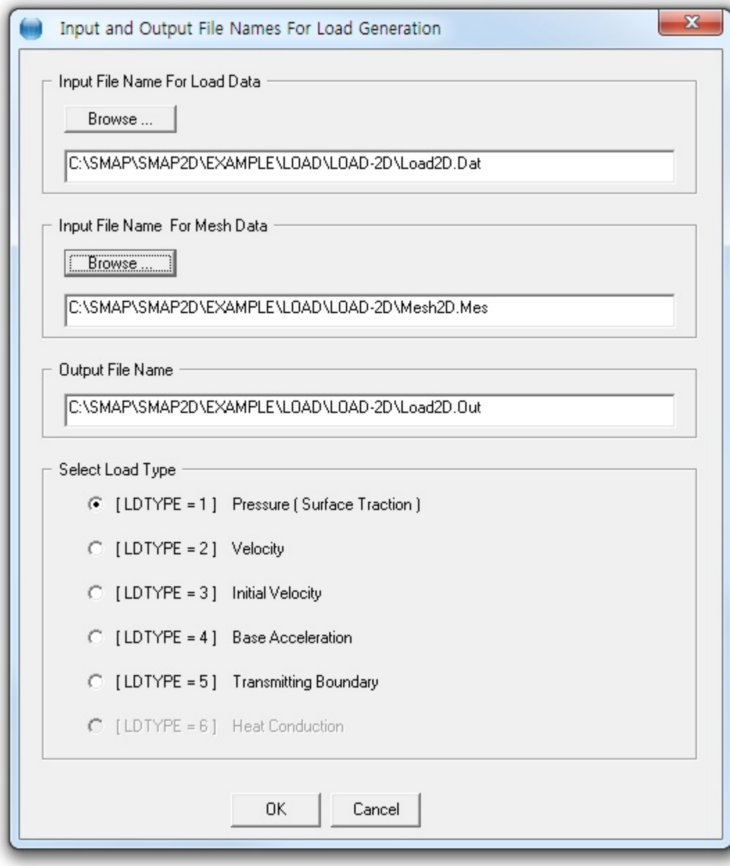
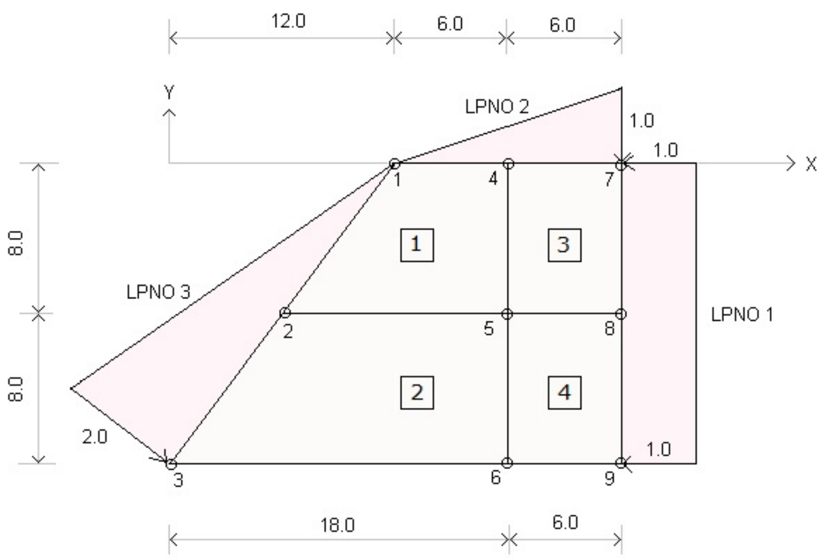


Figure 10.1 Load generation dialog



Loading Surface 1 (LSNO = 1) consists of nodes 9, 7, 8

Loading Surface 2 (LSNO = 2) consists of nodes 7, 4, 1

Loading Surface 3 (LSNO = 3) consists of nodes 1, 2, 3

Pressure Function 1 (LPNO = 1) $P_x = -1.0$

Pressure Function 2 (LPNO = 2) $P_Y = 1.0 - (1/12) X$

Pressure Function 3 (LPNO = 3) $P_n = -0.125 Y$

Note: Mesh is axially symmetric about Y axis

Figure 10.2 Schematic view of pressure loads for Example 1

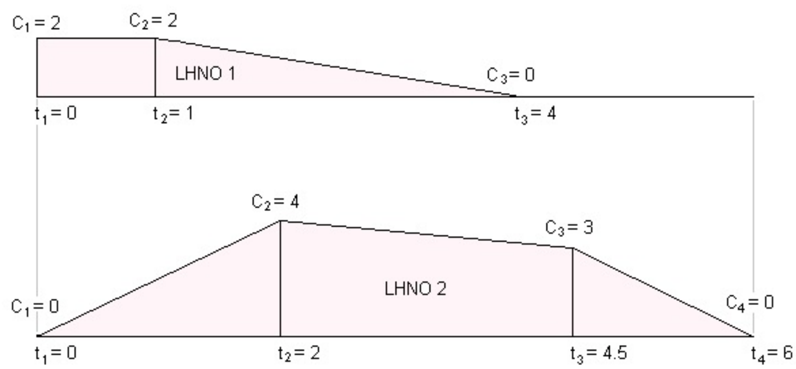


Figure 10.3 Load time histories for Example 1

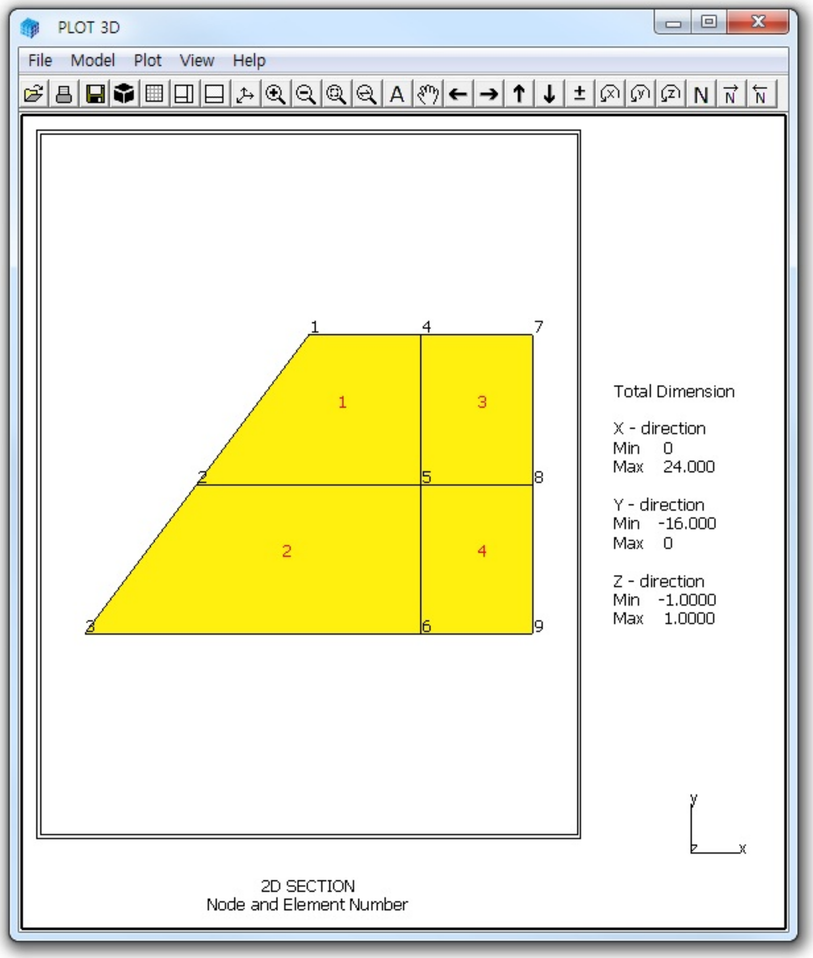


Figure 10.4 Finite element mesh for Example 1

Table 10.1 Listing of mesh data input file Mesh2D.Mes for Example 1

```
2D SECTION
NUMNP   NCONT   NBEAM  NTRUS
   9      4       0      0
NODAL COORDINATES
NODE  ISX   ISY  IFX  IFY  IRZ      XC           YC
  1    1     0    1    1    1       12.           0.
  2    0     0    1    1    0        6.          -8.
  3    0     0    1    1    0        0.         -16.
  4    0     0    1    1    0       18.           0.
  5    0     0    1    1    0       18.          -8.
  6    0     0    1    1    0       18.         -16.
  7    0     0    1    1    0       24.           0.
  8    0     0    1    1    0       24.          -8.
  9    0     0    1    1    0       24.         -16.
ELEMENT INDEX
NEL   I1    I2   I3   I4   M5   M6   M7   M8 MATC KS KF IR IS TBJWL
  1    4     1    2    5    0    0    0    0    4  0  1 2  2  .000E+00
  2    5     2    3    6    0    0    0    0    4  0  1 2  2  .000E+00
  3    7     4    5    8    0    0    0    0    4  0  1 2  2  .000E+00
  4    8     5    6    9    0    0    0    0    4  0  1 2  2  .000E+00
```

Table 10.2 Listing of load data inut file Load2D.Dat for Example 1

```
*
* LOAD-2D  INPUT
*
* CARD 1.1
* TITLE
  EXAMPLE 1  LOAD-2D Pressure [LDTYPE = 1]
* -----------------------------------------
* CARD 1.2
* NCTYPE
   0
* =========================================
* CARD 2.1
* NUMLS
   3
* -----------------------------------------
* CARD 2.2.1
* LSNO
   1
* CARD 2.2.2
* NUMNODE
   3
* CARD 2.2.3
* LISTING OF NODES
   9, 7, 8
* -----------------------------------------
* CARD 2.2.1
* LSNO
   2
* CARD 2.2.2
* NUMNODE
   3
* CARD 2.2.3
* LISTING OF NODES
   7, 4, 1
* -----------------------------------------
* CARD 2.2.1
* LSNO
   3
* CARD 2.2.2
* NUMNODE
   3
```

```
* CARD 2.2.3
* LISTING OF NODES
  1, 2, 3
* =========================================
* CARD 3.1
* NUMLF
   3
* -----------------------------------------
* CARD 3.2.1
* LFNO   LPTYPE
   1      0
* CARD 3.2.2
* A-X0   A-XX   A-XY
  -1.,   0.0,   0.0
* CARD 3.2.3
* A-Y0   A-YX   A-YY
  0.0,   0.0,   0.0
* CARD 3.2.4
* A-N0   A-NX   A-NY
  0.0,   0.0,   0.0
* -----------------------------------------
* CARD 3.2.1
* LFNO   LPTYPE
   2      0
* CARD 3.2.2
* A-X0   A-XX   A-XY
  0.0,   0.0,   0.0
* CARD 3.2.3
* A-Y0   A-YX   A-YY
  1.0,-0.083333,0.0
* CARD 3.2.4
* A-N0   A-NX   A-NY
  0.0,   0.0,   0.0
* -----------------------------------------
* CARD 3.2.1
* LFNO   LPTYPE
   3      1
* CARD 3.2.2
* A-X0   A-XX   A-XY
  0.0,   0.0,   0.0
* CARD 3.2.3
* A-Y0   A-YX   A-YY
  0.0,   0.0,   0.0
```

```
* CARD 3.2.4
* A-N0    A-NX    A-NY
  0.0,    0.0,   -0.125
* =========================================
* CARD 4.1
* NUMLH
   2
* -----------------------------------------
* CARD 4.2.1
* LHNO
   1
* CARD 4.2.2
* NUMTP
   3
* CARD 4.2.3
* T1    T2    T3
  0.0   1.0   4.0
* CARD 4.2.4
* C1    C2    C3
  2.0   2.0   0.0
* -----------------------------------------
* CARD 4.2.1
* LHNO
   2
* CARD 4.2.2
* NUMTP
   4
* CARD 4.2.3
* T1    T2    T3    T4
  0.0   2.0   4.5   6.0
* CARD 4.2.4
* C1    C2    C3    C4
  0.0   4.0   3.0   0.0
* =========================================
* CARD 5.1
* LSNO  LFNO  LHNO
   1,    1,    1
   2,    2,    1
   3,    3,    2
   0,    0,    0
* END OF INPUT DATA
```

The output file, Load2D.Out listed in Table 10.3, contains generated concentrated nodal forces and load time histories. Figure 10.5 shows time history curves for each load history number. The format of the generated load output is compatible to the format of Card Group 9 in SMAP-2D main input.

Table 10.3 Listing of load output file Load2D.Out for Example 1

```
* CARD 9.2.1
* NUMLP
   12
* LOAD HISTORY NO:  1
* CARD 9.2.2
* NODE   IDOF   LHNO   CINT
    1      2      1   -.74998E+01
    4      2      1   -.56999E+02
    7      1      1   -.96000E+02
    7      2      1   -.55500E+02
    8      1      1   -.19200E+03
    9      1      1   -.96000E+02
* LOAD HISTORY NO:  2
* CARD 9.2.2
* NODE   IDOF   LHNO   CINT
    1      1      2    .12000E+02
    1      2      2   -.90000E+01
    2      1      2    .40000E+02
    2      2      2   -.30000E+02
    3      1      2    .12000E+02
    3      2      2   -.90000E+01
* END OF LOAD HISTORY
* CARD 9.2.3.1
* NTFUN  NUMLH
    0      2
* CARD 9.2.3.2
* NUMTP  NTYPE   DTXX
    6      1    .00000E+00
* CARD 9.2.3.3
* LISTING OF TIME POINTS
  .0000E+00  .10000E+01  .20000E+01  .40000E+01  .4500E+01  .6000E+01
* CARD 9.2.3.4
* LISTING OF LOAD FOR HISTORY NO:  1
  .2000E+01  .20000E+01  .13333E+01 -.59605E-07  .0000E+00  .0000E+00
* CARD 9.2.3.4
* LISTING OF LOAD FOR HISTORY NO:  2
  .0000E+00  .20000E+01  .40000E+01  .32000E+01  .3000E+01  .0000E+00
* END OF LOAD DATA
```

Generated load vectors for concentrated forces can be plotted graphically. Refer to the step by step procedure in the file Running LOAD-2D.pdf.
The effect of LPTYPE (Effective vs Actual Surface) is described in the file LOAD-2D Example.pdf.

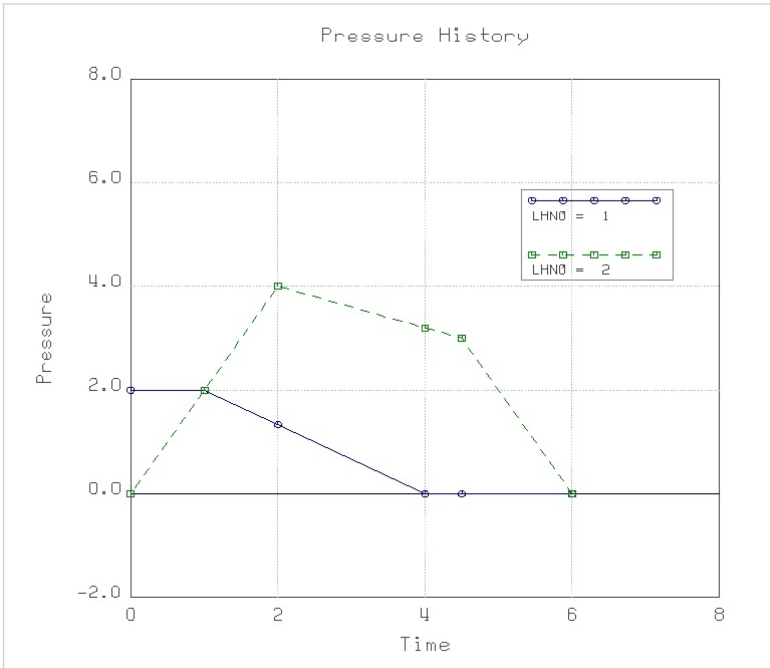


Figure 10.5 Generated load time histories for Example 1

XY Graph
Example Problem

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. XY Graph User's Manual describes all the basic functions associated with XY graph creation and modifications.

Two example problems are presented:

1. New Graph
   Shows step by step procedure to create and modify XY graph.
   Main actions:

- Access XY graph
- Edit initial Draft XY
- Modify XY graph by Edit dialog
- Open XY graph on Excel Spreadsheet

2. SMAP Result
   Plots SMAP results specified in Card Group 12 in SMAP Post File.
   Main actions:

- Execute SMAP-2D example
- Access SMAP result
- Access PLOT XY in Plot menu
- Modify XY graph by Edit dialog
- Open XY graph on Excel Spreadsheet

11.1 New Graph

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

This example consists of the following main actions:

- Access XY graph
- Edit initial Draft XY
- Modify XY graph by Edit dialog
- Open XY graph on Excel Spreadsheet

Step 1: Access XY Graph (New)

Access XY Graph by selecting following items in SMAP (Figure 11.1):
Plot → XY → PLOT XY → New

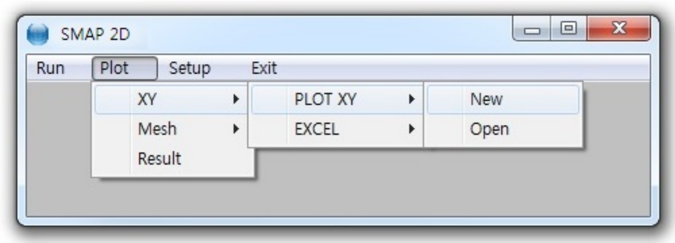


Figure 11.1 Accessing XY graph (New)

Step 2: Edit Initial Draft XY

Once selected, initial default file XY.dat will be opened by Notepad as listed in Table 11.1.

Edit the first plot in this default file as listed in Table 11.2.
And then save and exit.

Modified graph will be displayed on PLOT XY drawing board as shown in Figure 11.2.

Table 11.1 Draft XY Data (Initial Default File XY.dat)

```
Plot No. 1
Sub Title 1
XLabel-1
YLabel-1
 0          10
 100        20
 .000000E+00  .123456E+06
Curve 1
Legend
10,         20
90,         30
 .000000E+00  .123456E+06
Curve 2
Legend
 .000000E+00  .987654E+06
Plot No. 2
Sub Title 2
XLabel-2
YLabel-2
0           100
1000        200
 .000000E+00  .123456E+06
Curve 1
Legend
100         200
900         300
.000000E+00  .123456E+06
Curve 2
Legend
.000000E+00  .987654E+06
Plot No. 3
Sub Title 3
XLabel-3
YLabel-3
0            100
1000         200
.000000E+00  .123456E+06
Curve 1
Legend
200,        200
900,        300
.000000E+00  .123456E+06
Curve 2
Legend
.000000E+00  .987654E
```

Table 11.2 Modified Draft XY Data (File XY.dat)

```
Example 1
Stress History
Time (Sec)
Stress (MPa)
 0          10
 100        20
 .000000E+00   .123456E+06
Vertical
Stress
0           20
100         30
 .000000E+00   .123456E+06
Horizontal
Stress
 .000000E+00   .987654E+06
Plot No. 2
Sub Title 2
XLabel-2
YLabel-2
0           100
1000        200
 .000000E+00   .123456E+06
Curve 1
Legend
100         200
900         300
.000000E+00   .123456E+06
Curve 2
Legend
.000000E+00   .987654E+06
Plot No. 3
Sub Title 3
XLabel-3
YLabel-3
0           100
1000        200
.000000E+00   .123456E+06
Curve 1
Legend
200,        200
900,        300
.000000E+00   .123456E+06
Curve 2
Legend
.000000E+00   .987654E
```

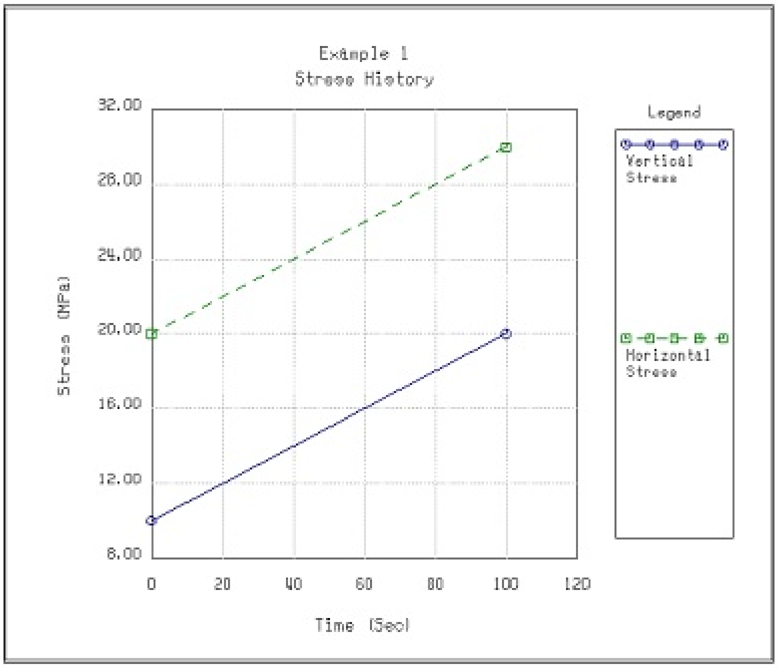


Figure 11.2 Modified graph on PLOT XY

Step 3: Modify XY Graph by Edit Dialog

Access Edit dialog by clicking the Edit menu in PLOT XY (Figure 11.3):

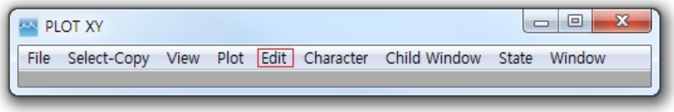


Figure 11.3 Edit menu in PLOT XY

Edit dialog will be displayed as shown in Figure 11.4.

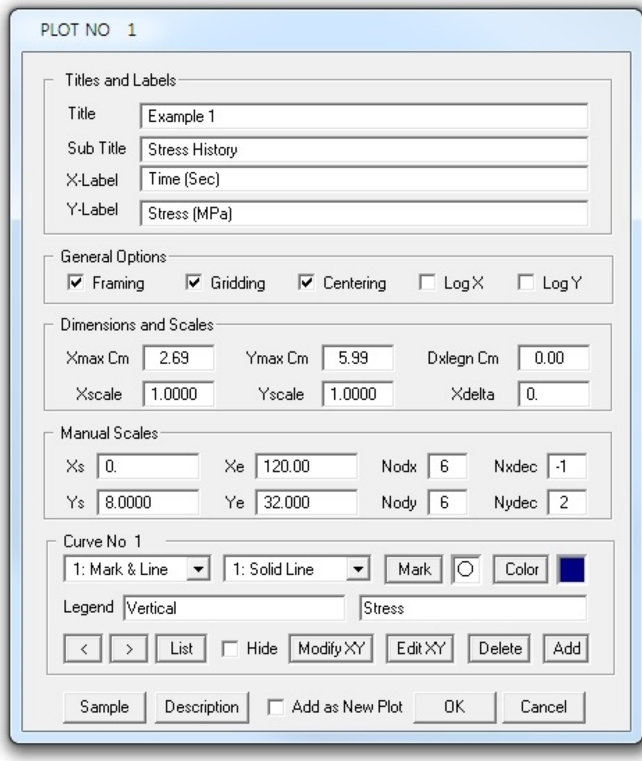


Figure 11.4 Edit dialog

There are many different options available for changing view of XY graphas described in detail in Section 12.3 in XY Graph User's Manual.

Here, change the color of the first curve into Red.
Click Color button and select Red from Color Palette dialog.
Then Red color will be updated for first curve as shown in Figure 11.5.

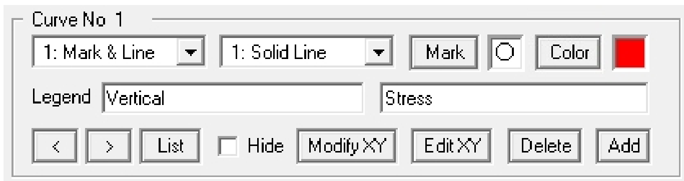


Figure 11.5 Updated red color in Edit dialog

Click OK button in Edit dialog.
Then updated plot will be displayed on PLOT XY as in Figure 11.6.

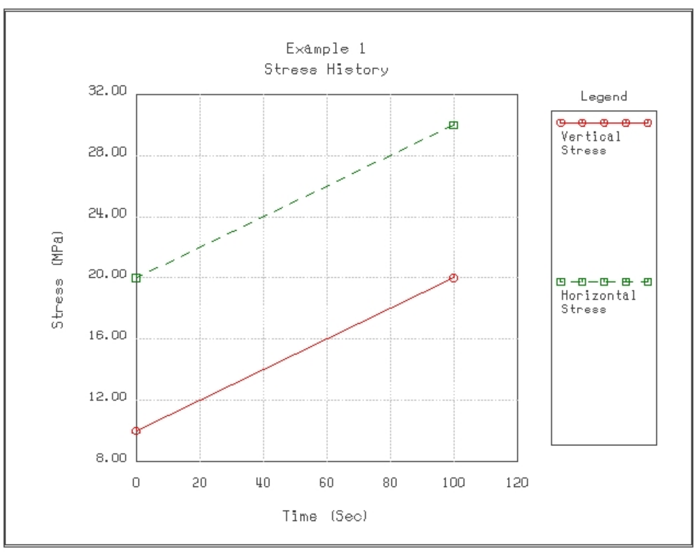


Figure 11.6 Updated first curve on PLOT XY

Step 4: Open XY Graph on Excel Spreadsheet

Access XY Graph by selecting following items in SMAP (Figure 11.7):
Plot → XY → EXCEL → Open

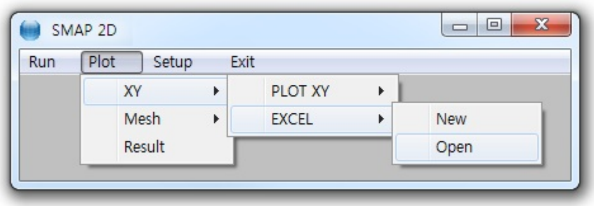


Figure 11.7 Accessing XY graph on Excel (Open)

Open XY.dat in the current working directory.
XY graph will be displayed on Excel Spreadsheet as in Figure 11.8.

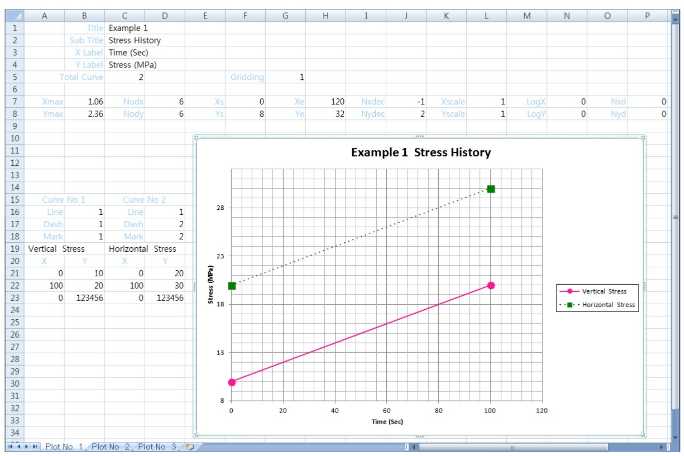


Figure 11.8 XY graph on Excel spreadsheet

Refer to more samples in the following directory:
C:\Smap \Smap2D \Example \XY_Graph \Excel XY Graph Sample.pdf

11.2 SMAP Result

The main objective of this second example is to show the step by step procedure to plot SMAP results specified in Card Group 12 in SMAP Post File. This example involves SMAP-2D Example Problem 5 (Laminated Beam with Slip Interface).

This example consists of the following main actions:

- Execute SMAP-2D example
- Access SMAP result
- Access PLOT XY in Plot menu
- Modify XY graph by Edit dialog
- Open XY graph on Excel Spreadsheet

Step 1: Execute SMAP-2D Example

Execute SMAP-2D by selecting the following menu items in SMAP (Figure 11.9): Run → Smap → Execute

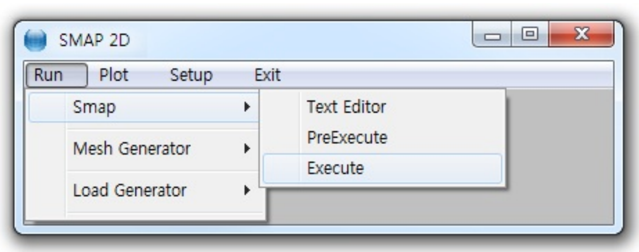


Figure 11.9 Execute SMAP-2D example problem

Note that SMAP-2D Example Problem 5 includes XY graph specified in Card Group 12 in SMAP Post File Vp5.Pos as listed in Table 11.3

Step 2: Access SMAP Result

Access SMAP Result by selecting the following menu items in SMAP : Plot → Result

Table 11.3 SMAP-2D post file (File Vp5.Pos)

```
* Card 11.1
* NPTYPE
  0
* P L O T - X Y
* Card 12.1
* IPTYPE
  2
* Card 12.3.1
* IPLOT
  1
* Card 12.3.2
* NODE
  1
* LIST1,  LIST2,  ...
  4
* Card 12.3.4
* NDPQ
  1
* Card 12.3.5
* KX      KY
  1,      3
* Card 12.3.6
* TMFAC   SND    SNV   SNA   NC  ANGLE
  0.018  -100    1     1     0   0
* Card 12.3.7
* TITLE / X-LABEL / Y-LABEL
  Laminated Beam
  Applied Load (t)
  Displacement (Cm)
* Card 12.1
* IPTYPE
  0
* End of Data
```

Step 3: Access PLOT XY in Plot Menu

Select PLOT XY in Plot Menu dialog in Figure 11.10.

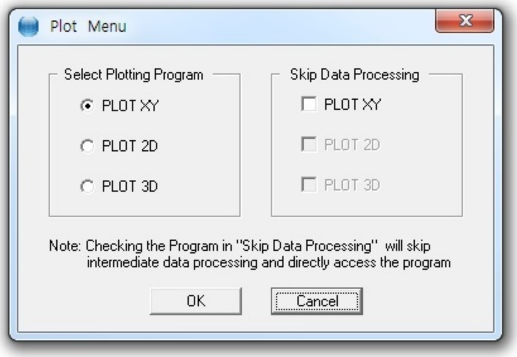


Figure 11.10 Plot menu dialog

Select PLOT XY in Select Plotting Program dialog in Figure 11.11.
Click OK button.

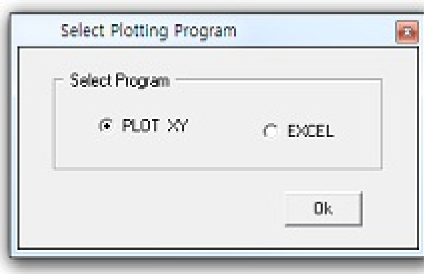


Figure 11.11 Select plotting program dialog

Step 4: Modify XY Graph by Edit Dialog

Once XY graph is displayed on PLOT XY, access Edit dialog by clicking the Edit menu in PLOT XY as shown in Figure 11.12

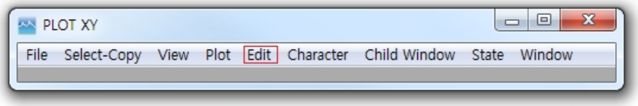


Figure 11.12 Edit menu in PLOT XY

Modify Edit dialog as shown in Figure 11.13.
The main modification is to plot the XY graph in log scales.
Click OK button in Edit dialog.

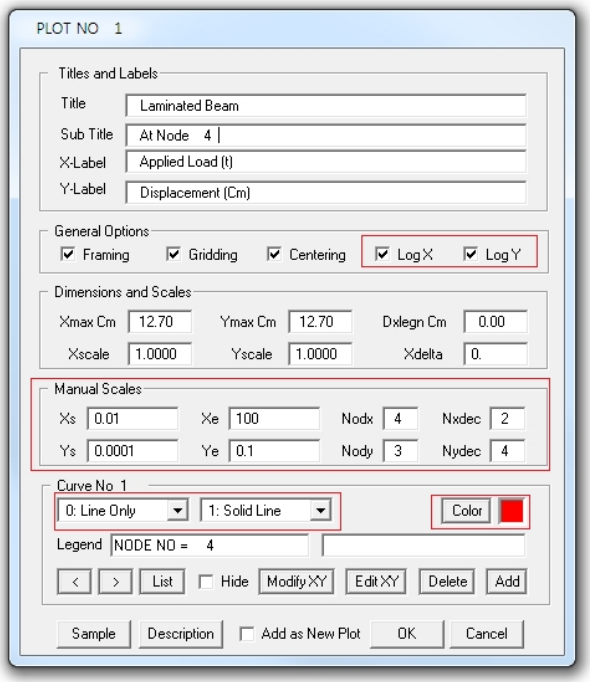


Figure 11.13 Edit dialog

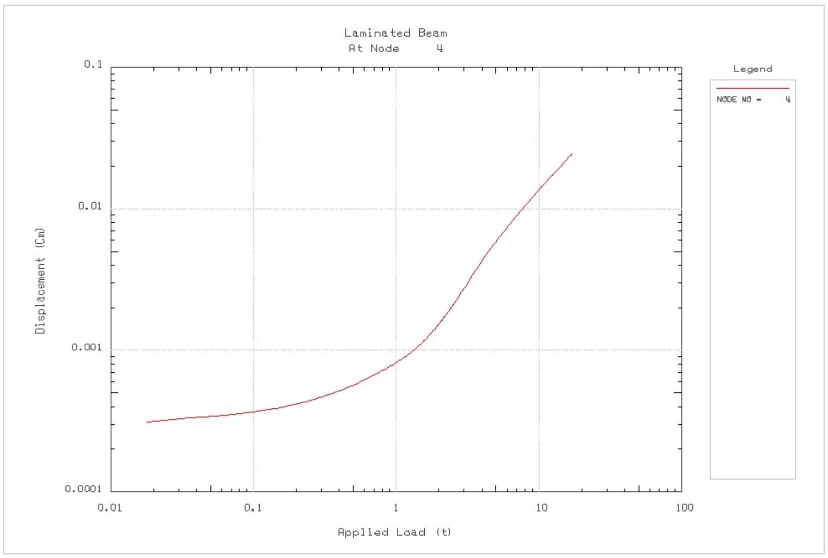


Figure 11.14 XY graph in log scales on PLOT XY

Refer to more samples in the following directory:
C:\Smap \Smap2D \Example \XY_Graph \PLOT XY Graph Sample.pdf

Step 5: Open XY Graph on Excel Spreadsheet

Access XY Graph by selecting following items in SMAP (Figure 11.15):
Plot → XY → EXCEL → Open

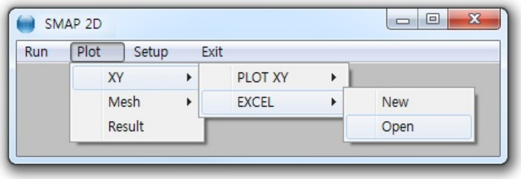


Figure 11.15 Accessing XY graph on Excel (Open)

Open PlotXy.dat in the current working directory.
XY graph will be displayed on Excel Spreadsheet as in Figure 11.16.

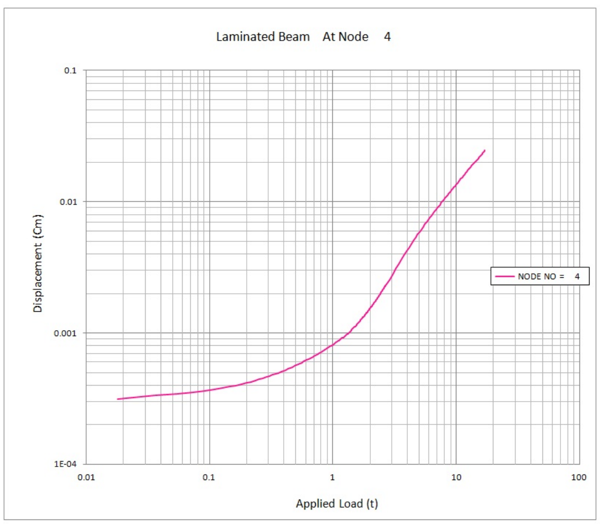


Figure 11.16 XY graph on Excel spreadsheet

Refer to more samples in the following directory:
C:\Smap \Smap2D \Example \XY_Graph \Excel XY Graph Sample.pdf

Go to Edit > Preferences > Page Display > Uncheck Enhance Thin Lines

SMAP ® - 2D

Structure Medium Analysis Program

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

Theory

Printed in the United States of America.

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

1.1 Introduction

SMAP-2D, which is an upgraded follow-on to the original MPDAP (Multi-Phase Dynamic Analysis Program), is a two-dimensional finite element computer program which has been continuously improved based on theoretical and experimental works since 1982. The program has been used to study fundamental mechanics of saturated porous medium. The program can be a powerful tool for the geomechanical analysis since it can solve static, consolidation and dynamic problems in dry, partially saturated or fully saturated soils and porous rock mass. The program considers material, geometric and boundary condition nonlinearities. Next two sections describe theoretical backgrounds of program SMAP-2D.

Section 2 describes theoretical formulations of nonlinear two-phase medium. Nonlinear compressibility equations are derived in detail for grains, saturated pore water, and partially saturated pore water. Field equations representing fundamental mechanics of two-phase medium are presented.

These field equations include effective stress law, constitutive equation for skeleton deformation, continuity equation of pore fluid, equation of motion for the bulk mixture, and equation of motion for pore fluid. Then, these field equations are discretized in space and expressed in incremental forms. Finally, global equilibrium equations are derived by principle of virtual work and then linearized to be solved by linear equation solver.

Section 3 describes constitutive relations of various nonlinear material models available in the program SMAP-2D. These nonlinear models include; Generalized Hoek and Brown Model, Single Hardening Elasto-Plastic Model, JWL High Explosive Model, Modified Cam Clay Model with Creep, Engineering Model, Joint Model, and Generalized Decoupled Hyperbolic Model.

To see the validation of the computational algorithms of the computer program SMAP-2D, refer to SMAP-2D Example Problems.

Finite Element Formulation of Nonlinear Two-Phase Medium

2.1 Introduction

Biot introduced fundamental analytical work describing the behavior of saturated porous media in a series of papers extending over many years (e.g. 1956, 1962a and 1962b). Other investigators have applied Biot's analytic results using techniques which approximate his equations with varying degrees of accuracy and sophistication (e.g. Ghaboussi and Wilson 1972, Mengi and McNiven, 1977). Theoretical formulations incorporated in the code SMAP-2D are the extension of Biot's two-phase theory to nonlinear region. These nonlinear two-phase theories have been developed over a decade under the sponsorship of Air Force Office of Scientific Research (e.g. Kim and Blouin 1984, Kim et al. 1986, 1987 and 1988).

In this section, the fundamental equations implemented in the code SMAP-2D are described. First the individual material components compressibility models are described in Subsection 2.2 for the solid grain, Subsection 2.3 for the pore water and Subsection 2.4 for the partially saturated water. Nonlinear material models of the skeleton are presented in Section 3. Field equations described in Subsection 2.5 include effective stress law, constitutive equation for skeleton deformation, continuity equation of pore fluid flow, equation of motion for the bulk mixture and equation of motion for pore fluid.

These field equations are described in terms of nodal values and expressed in incremental form in Subsection 2.6. Finally, global equilibrium equations for the two-phase medium are formulated in Subsection 2.7 and linearized to be solved by linear equation solver in Subsection 2.8.

2.2 Grain Model

To model the nonlinear response of the solid grains to both the applied pore pressure and effective stress, analytic expressions for the deformation of solids at high pressure are employed. High pressure data for many rocks and minerals show a linear relationship between loading wave velocity and particle velocity (e.g. Allen, 1967).
The loading wave velocity can be expressed as:

$$c_L = c_o + S\, v_p \tag{2.1}$$

where:

- c_L = Loading wave velocity
- c_o = The initial wave velocity at relatively low pressure
- v_p = Peak particle velocity
- S = Experimentally determined constant relating c_L to v_p (generally equal to about 1.5 for most dense rocks and minerals)

Conservation of mass and momentum on either side of the wave front yields the familiar relationships:

$$\sigma_p = \rho_o\, c_L\, v_p \tag{2.2}$$

$$M = \rho_o\, c_L^2 \tag{2.3}$$

where:

σ_p	=	Peak axial stress
ρ_o	=	Initial material density
M	=	Constrained secant modulus = σ_p / ε_p
ε_p	=	Peak axial strain corresponding to the peak stress σ_p

Substitution of Equation 2.1 into 2.2 gives:

$$\sigma_p = \rho_o\, c_o\, v_p + \rho_o\, S\, v_p^2 \tag{2.4}$$

and solving for peak particle velocity as a function of peak stress yields

$$v_p = \frac{f(\sigma_p)}{2\,\rho_o\, S} \tag{2.5}$$

where

$$f(\sigma_p) = (\rho_o^2\, c_o^2 + 4\,\rho_o\, S\, \sigma_p)^{1/2} - \rho_o\, c_o \tag{2.6}$$

Substitution of Equation 2.1, 2.5, and 2.6 into Equation 2.3 gives:

$$M = F(\sigma_p) = \rho_o\, c_o^2 + c_o\, f(\sigma) + \frac{f^2(\sigma)}{4\rho_o} \tag{2.7}$$

The tangent constrained modulus, M_t, used in the numerical model is defined as the slope of the stress strain curve by:

$$M_t = \frac{d\sigma}{d\varepsilon} \tag{2.8}$$

From Equation 2.7 and the definition of constrained modulus, M:

$$\varepsilon_p = \frac{\sigma_p}{F(\sigma_p)} \tag{2.9}$$

Differentiating Equation 2.9 with respect to σ_p and inverting gives the tangent constrained modulus as

$$M_t = \frac{F^2(\sigma_p)}{F(\sigma_p) - \sigma_p F'(\sigma_p)} \tag{2.10}$$

Differentiating Equations 2.6 and 2.7 with respect to σ_p yields:

$$F'(\sigma_p) = c_o f'(\sigma_p) + \frac{f(\sigma_p) f'(\sigma_p)}{2 \rho_o} \tag{2.11}$$

and

$$f'(\sigma_p) = \frac{2 \rho_o S}{(\rho_o^2 c_o^2 + 4 \rho_o S \sigma_p)^{1/2}} \tag{2.12}$$

Hence, Equations 2.5 through 2.12 can be used to define high pressure constrained stress strain and modulus relationships for the solid grains.

For two phase, coupled calculations, the volumetric relationships for the solid grains should be specified in terms of the bulk modulus, K_g, rather than in terms of the constrained modulus. At high pressures, the shear strength of the grain materials becomes insignificant compared to the applied stress and the materials tend to behave like fluids. At these pressures, the tangent bulk modulus equals the tangent constrained modulus with Poisson's ratio equal to 0.5. Beneath some threshold pressure, p_b, Poisson's ratio begins to decrease from 0.5 at p_b to an initial value of Poisson's ratio, v_o, at a low value of mean stress. We have used a simple relationship to approximate the influence of mean stress on Poisson's ratio for the solid grains:

$$K_g = g(p) M_t \tag{2.13}$$

The ratio of the bulk modulus to the tangent constrained modulus, g(p) at pressures less than p_b is given by:

$$g(p) = \frac{2}{3}\frac{(1-2v_o)}{(1-v_o)}\frac{p}{p_b} + \frac{(1+v_o)}{3(1-v_o)} \tag{2.14}$$

For pressures greater than p_b;

$$g(p) = 1 \tag{2.15}$$

Poisson's ratio can be computed as a function of the modulus ratio at a given pressure as:

$$v = \frac{3\,g(p) - 1}{1 + 3\,g(p)} \tag{2.16}$$

2.3 Pore Water Model

The model for the nonlinear, elastic compressibility of the pore water is derived from an equation of state reported by Ahrens (1988) and attributed to Bakanova, et. al. (1976). This equation relates the shock velocity in water to the peak particle velocity. In the lower pressure regime, a quadratic relation is used while a linear relation is used in the higher pressure regime. The transition point between the two regimes is defined in terms of a peak particle velocity at the transition, V_{pt}. Bakanova's equations can be expressed as:

$$v_s \le v_{pt}: \quad c = c_1 + S_1\,v_p + S_2\,v_p^2 \tag{2.17}$$

$$v_s > v_{pt}: \quad c = c_2 + S_3\,v_p \tag{2.18}$$

where:

c	=	Shock propagation velocity in the fluid
v_p	=	Peak fluid particle velocity
c_1, S_1, S_2	=	Constants used to fit data below the transition
c_2, S_3	=	Constants used to fit data above the transition

Equation 2.18 can also be expressed in terms of the shock velocity at the transition point, c_t. Substituting v_{pt} into Equation 2.18 yields:

$$c_2 = c_t - S_3 v_{pt} \tag{2.19}$$

Substituting 2.19 into 2.18 produces this expression for the shock velocity above the transition:

$$v_s > v_{pt} : \qquad c = c_t + S_3 (v_p - v_{pt}) \tag{2.20}$$

where:

c_t	=	Shock velocity at the transition
v_{pt}	=	Peak particle velocity at the transition (Model constant)

At the transition point, the shock velocity from Equations 2.17 and 2.20 should be equal to preserve continuity. Setting Equations 2.17 and 2.20 equal at $v_p = v_{pt}$ gives:

$$c_t = c_1 + S_1 v_{pt} + S_2 v_{pt}^2 \tag{2.21}$$

thereby defining c_t in terms of the model constants. Equations 2.17, 2.20, and 2.21 (with the constants c_1, S_1, S_2, and S_3) define the shock velocity as a function of peak particle velocity.

To derive a bulk modulus for water as a function of pressure, we first need an expression for peak particle velocity as a function of pressure.

Conservation of mass and momentum on either side of the wave front yields the familiar relationship from shock physics:

$$\pi_p = \rho_o \, c \, v_p \tag{2.22}$$

where:

π_p = Pore fluid pressure

ρ_o = Mass density of fluid

Substitution of Equation 2.17 into 2.22 yields an expression for the transition fluid pressure (π_{pt}):

$$\pi_{pt} = \rho_o \, v_{pt} \, (c_1 + S_1 \, v_{pt} + S_2 \, v_{pt}^2) \tag{2.23}$$

For water, the transition pressure is greater than 30,000 MPa. Below the transition pressure, substitution of Equation 2.17 into 2.22 will give:

$$v_p^3 + \frac{S_1}{S_2} v_p^2 + \frac{c_1}{S_2} v_p - \frac{\pi_p}{\rho_o \, S_2} = 0 \tag{2.24}$$

This cubic equation can be solved to yield an expression for v_p as a function of fluid pressure below the transition pressure π_{pt} : where:

$$v_p = m \cos\left[\frac{1}{3}\cos^{-1}\left(\frac{3\beta}{\alpha \, m}\right) + \frac{4\pi}{3}\right] - \frac{S_1}{3 S_2} \tag{2.25}$$

where

$$\alpha = \frac{c_1}{S_2} - \frac{1}{3}\left(\frac{S_1}{S_2}\right)^2 \tag{2.26}$$

$$\beta = \frac{-\pi_p}{\rho_o S_2} - \frac{1}{3}\left(\frac{S_1}{S_2}\right)\left(\frac{c_1}{S_2}\right) + \frac{2}{27}\left(\frac{S_1}{S_2}\right)^3 \quad (2.27)$$

$$m = 2\sqrt{\frac{-\alpha}{3}} \quad (2.28)$$

Above the transition pressure, substitution of Equation 2.20 into 2.22 yields a quadratic equation:

$$v_p^2 + \left(\frac{c_t - S_3\, v_{pt}}{S_3}\right) v_p - \frac{\pi_p}{\rho_o\, S_3} = 0 \quad (2.29)$$

Solving this equation for v_p as a function of fluid pressure gives v_p for pressures above the transition pressure π_{pt} :

$$v_p = -\left(\frac{c_t - S_3\, v_{pt}}{2S_3}\right) + \left[\left(\frac{c_t - S_3\, v_{pt}}{2S_3}\right)^2 + \frac{\pi_p}{\rho_o\, S_3}\right]^{\frac{1}{2}} \quad (2.30)$$

The elastic bulk modulus of water (K_w) is defined as:

$$K_w = \frac{d\pi_p}{d\varepsilon_v} = \frac{d\pi_p \;/\; dv_p}{d\varepsilon_v \;/\; dv_p} \quad (2.31)$$

where ε_v is the volume strain corresponding to the pressure π_p.
Taking the derivative of Equation 2.22:

$$\frac{d\pi_p}{dv_p} = \rho_o\,(c'\, v_p + c) \quad (2.32)$$

The volume strain is given by:

$$\varepsilon_v = \frac{v_p}{c} \tag{2.33}$$

and taking the derivative yields:

$$\frac{d\varepsilon_v}{dv_p} = \frac{c - v_p\, c'}{c^2} \tag{2.34}$$

Substitution of Equations 2.32 and 2.34 into 2.31 gives an expression for the bulk modulus in terms of the shock and peak particle velocities:

$$K_w = \frac{\rho_o\, c^2\, (c + v_p\, c')}{c - v_p\, c'} \tag{2.35}$$

The derivatives of the shock velocity with respect to the peak particle velocity are given by:

$$\pi_p \le \pi_{pt} : \quad c' = S_1 + 2\, S_2\, v_p \tag{2.36}$$

$$\pi_p > \pi_{pt} : \quad c' = S_3 \tag{2.37}$$

The material constant values for this model are given in Table 2.1 for fresh water and sea water. The fresh water values are from Bakanova, et. al. (1976) as reported by Ahrens (1988). Parameters for sea water were fit to compressibility data described by Kim, et. al. (1986) and attributed to Britt (1985).

2.4 Partially Saturated Pore Water Model

When rock or soil is unsaturated, compression of the pore water and solid grains is nearly insignificant when compared with the compression of pore air. Under these conditions, material behavior is governed mostly by the skeleton model. With sufficient compression, the pore air gets squeezed out and the material becomes saturated. Rischbieter, et. al. (1977) demonstrated that even a minute amount of entrapped air drastically alters the pore pressure response in multiphase porous materials. To simulate this behavior, the pore fluid model is modified to account for the compressibility of pore air and converges to a saturated condition. Note that this model is invoked only when the initial saturation is less than 100%.

The compressibility of the air-water mixture, C_{aw}, is defined as:

$$C_{aw} = \frac{d\varepsilon_{v,aw}}{d\pi_p} \tag{2.38}$$

where π_p is the fluid pressure. The volumetric strain in the air-water mixture, $\varepsilon_{v,aw}$, is the sum of volume strain in the air and water. Using the definition of the initial saturation, it can be shown that:

$$\varepsilon_{v,aw} = (1 - S_o)\, \varepsilon_{v,a} + S_o\, \varepsilon_{v,w} \tag{2.39}$$

where:

$\varepsilon_{v,aw}$ = Volume strain of air-water mixture
$\varepsilon_{v,a}$ = Volume strain of air bubbles
$\varepsilon_{v,w}$ = Volume strain of water (from Equation 2.33)
S_o = Initial saturation

From Equations 2.38 and 2.39 we can get an expression for the compressibility of the air-water mixture:

$$C_{aw} = (1 - S_o)\ C_a + S_o\ C_w \tag{2.40}$$

Since the compressibility is the inverse of the bulk modulus, Equation 2.40 can be expressed as:

$$\frac{1}{K_{aw}} = \frac{1 - s_o}{K_a} + \frac{s_o}{K_w} \tag{2.41}$$

where:

K_{aw} = Bulk modulus of air-water mixture
K_a = Equivalent bulk modulus of air bubbles in the fluid
K_w = Bulk modulus of water (from Equation 2.35)

The volume strain and the equivalent bulk modulus of the air bubbles in the pore fluid are derived here using the adiabatic ideal gas law (γ-law). The model has been shown to be applicable when the degree of pore water saturation is above approximately 85% where the pore air is thought to exist as small bubbles within the fluid (occluded state).

The model is derived from the adiabatic ideal gas law:

$$\pi_a \cdot V_a^{\gamma} = \pi_{ao} \cdot V_{ao}^{\gamma} \tag{2.42}$$

where

π_{ao} Initial air pressure (absolute pressure)
π_a Current air pressure (absolute pressure)
V_{ao} Initial air volume
V_a Current air volume
γ Ratio of heat capacity (c_p/c_v)

The volume strain of air can be defined in terms of engineering strain:

$$\varepsilon_{v,a} = 1 - \left(\frac{V_a}{V_{ao}}\right) \tag{2.43}$$

Substituting Equation 2.42 into Equation 2.43, we can express the volume strain of air bubble in terms of air pressure:

$$\varepsilon_{v,a} = 1 - \left[\frac{\pi_{ao}}{\pi_a}\right]^{\frac{1}{\gamma}} \tag{2.44}$$

Neglecting the influence of surface tension,

$$\pi_a = \pi + p_a \tag{2.45}$$

where

π	Current pore water pressure (gage pressure)
P_a	Reference atmospheric pressure

Substitution of Equation 2.45 into Equation 2.44 yields

$$\varepsilon_{v,\,a} = 1 - \left(\frac{\pi_a}{\pi + P_a}\right)^{\frac{1}{\gamma}} \tag{2.46}$$

Tangent bulk modulus of air bubbles can be defined as

$$K_a = \frac{d\pi_a}{d\varepsilon_{v,a}} \tag{2.47}$$

Differentiating Equation 2.46 with respect to π,

$$\frac{d\varepsilon_{v,a}}{d\pi_a} = \frac{1}{\gamma \cdot \pi_{ao}}\left(\frac{\pi_{ao}}{\pi + P_a}\right)^{\left(1+\frac{1}{\gamma}\right)} \tag{2.48}$$

Substitution of Equation 2.48 into Equation 2.47 yields

$$K_a = \gamma \cdot \Pi_{ao} \left[\frac{\Pi + P_a}{\Pi_{ao}} \right]^{\left(1 + \frac{1}{\gamma}\right)} \tag{2.49}$$

Equations 2.35 and 2.49, when substituted into Equation 2.41, define the compressibility of the pore air-water mixture. The model does not employ an explicit expression for the saturation point, where the air bubbles no longer exist. However, Equation 2.49 implies that the stiffness of the pore air increases with the pressure. As the pressure increases, the contribution of the air to the net compressibility of the mixture becomes insignificant when compared to the compressibility of the water. This, in essence, results in fully saturated behavior but with a smooth model transition during collapse of the air bubbles. An example pressure-volume curve for water with an initial air content of 5% is shown in Figure 2.1. Notice that the mixture becomes pressure saturated at a volume strain of about 5%.

Table 2.1 Fluid compressibility model constants
(See Section 2.3 for definitions of constants)

Parameter	Unit	Fresh Water	Sea Water
ρ_0	kg /m^3	1002.8	1026
c_1	m/s	1500	1522
S_1	-	2.00	1.97
S_2	s/m	-1.07×10^{-4}	-0.898×10^{-4}
S_3	-	1.144	1.123
v_{pt}	m/s	4000	4573
c_t	m/s	7788	8653
π_{pt}	MP_a	31,240	40,600

2.5 Field Equations

Effective Stress Law

Terzaghi's effective stress equation is fundamental to the development of the fully coupled model. It relates the total applied stress, σ, to the pore pressure, π, and the effective stress, σ', according to

$$\sigma_{ij} = \sigma'_{ij} + \delta_{ij}\,\pi \qquad (2.50)$$

where

σ_{ij} = Total stress
σ'_{ij} = Effective stress
δ_{ij} = Kronecker's delta
δ_{ij} = 0 if $i \neq j$
δ_{ij} = 1 if $i = j$

Constitutive Equation for Skeleton Deformation

The deformation of the porous skeleton is related to the applied effective stress and the pore pressure acting on the solid grains. The stress-strain relationship is given by

$$\{d\sigma'\} = [D^{ep}]\left(\{d\varepsilon\} - \frac{1}{3K_g}\{1\}\,d\pi\right) \qquad (2.51)$$

The last term in Equation 2.51 is the strain in the skeleton resulting from compression of the solid grains by the pore pressure.

Continuity Equation of Pore Fluid Flow

The continuity equation for pore fluid flow is derived from mass conservation relationships. The volumetric strain of the pore fluid, ε_f, is given by

$$d\varepsilon_f = -\frac{d\rho_f}{\rho_f} = C_f\,d\pi \qquad (2.52)$$

where

C_f = Pore fluid compressibility

π = Pore fluid pressure

The volume strain of the solid grains, ε_g , is give by

$$d\varepsilon_g = -\frac{d\rho_g}{\rho_g} = C_g\, d\pi + \frac{C_g}{1-n}\, dp' \tag{2.53}$$

where

C_g = Bulk compressibility of solid grains

p' = Effective mean pressure

The dry density, ρ_d , is given by

$$\rho_d = \frac{m_g}{V_t} = (1-n)\,\rho_g \tag{2.54}$$

where m_g is the mass of the solid grains in skeleton volume V_t . The change in dry density is given by

$$d\rho_d = -\rho_d\, d\varepsilon_v \tag{2.55}$$

where ε_v is the volumetric strain of the skeleton. Differentiating Equation 2.54 with respect to n and ρ_g gives

$$d\rho_d = (1-n)\, d\rho_g - \rho_g\, dn \tag{2.56}$$

Equating 2.55 and 2.56 yields

$$d\varepsilon_v = \frac{dn}{1-n} - \frac{d\rho_g}{\rho_g} \tag{2.57}$$

Conservation of mass for the pore fluid within a specified initial volume of saturated porous material is given by

$$n \, \rho_f \, V_t = \bar{n} \, \bar{\rho}_f \, \bar{V}_t \tag{2.58}$$

where as illustrated in Figure 2.2, the terms to the left of the equal sign represent the fluid mass under the initial conditions and the terms to the right represent the same fluid mass under deformed conditions.

Equation 2.58 may be expressed in infinitesimal incremental form as

$$n \, \rho_f \, V_t = (n + dn) \, (\rho_f + d\rho_f) \, (1 + d\epsilon_F) \, V_t \tag{2.59}$$

where

ϵ_F = Volumetric diffusion of pore fluid as depicted in Figure 2.2

Solving Equation 2.59 for $d\epsilon_F$ and discarding second order terms yields

$$d\epsilon_F = - \frac{dn}{n} - \frac{d\rho_f}{\rho_f} \tag{2.60}$$

Equation 2.60 is combined with Equation 2.57 by elimination of dn to yield

$$(1 - n) \, d\epsilon_v + n \, d\epsilon_F + (1 - n) \frac{d\rho_g}{\rho_g} + n \frac{d\rho_f}{\rho_f} = 0 \tag{2.61}$$

Combining Equations 2.52 and 2.53 with 2.61 gives

$$n \, (d\epsilon_F - d\epsilon_v) + d\epsilon_v - \frac{1}{K_m} d\pi - c_g \, dp' = 0 \tag{2.62}$$

where K_m is the bulk modulus of the solid/fluid mixture which is expressed by

$$K_m = \frac{1}{n \, C_f + (1 - n) \, C_g} \tag{2.63}$$

The change in effective mean pressure is given by

$$dp' = K_s \, (d\varepsilon_v - C_g \, d\pi) \tag{2.64}$$

Substituting Equation 2.64 into 2.62 gives

$$n \, (d\varepsilon_F - d\varepsilon_v) + (1 - C_g \, K_s) \, d\varepsilon_v + \left(C_g^2 \, K_s - \frac{1}{K_m} \right) d\pi = 0 \tag{2.65}$$

or

$$n \, (d\varepsilon_F - d\varepsilon_v) = \left(\alpha - \frac{C_g^2}{9} \{1\}^T [D^{ep}] \{1\} \right) d\pi - \left(\{1\}^T - \frac{C_g}{3} \{1\}^T [D^{ep}] \right) \{d\varepsilon\} \tag{2.66}$$

Equation 2.66 can be expressed in the following convenient form:

$$d\pi = \bar{m}_2 \cdot d\varepsilon_v + \bar{m} \cdot n \, (d\varepsilon_F - d\varepsilon_v) \tag{2.67}$$

where

$$\bar{m} = \frac{1}{\left[\dfrac{1}{K_m} - \dfrac{K_s^{ep}}{K_g^2} \right]} \tag{2.68}$$

$$\bar{m}_2 = \left[1 - \frac{K_s^{ep}}{K_g} \right] \cdot \bar{m} \tag{2.69}$$

Equation of Motion for the Bulk Mixture

The differential equation of motion governing the bulk mixture is expressed by equating the stress gradient to the inertial resistance as

$$\sigma_{ij,\,j} = (1-n)\,\rho_s\,\ddot{u}_i + n\,\rho_f\,\ddot{U}_i \tag{2.70}$$

$\sigma_{ij,j}$ is the total stress gradient applied to an infinitesimal element of saturated material at some given time. $\sigma_{ij,j}$ is expressed in tensor and represents the stress gradient in each of three mutually perpendicular coordinates (e.g. see Mendleson, 1968). For instance, in the x direction,

$$\sigma_{xj,\,j} = \frac{\partial\sigma_{xx}}{\partial x} + \frac{\partial\sigma_{xy}}{\partial y} + \frac{\partial\sigma_{xz}}{\partial z} = (1-n)\,\rho_s\,\ddot{u}_x + n\,\rho_f\,\ddot{U}_x \tag{2.71}$$

The term (1-n) ρ_s is the mass of the soil skeleton per unit volume of saturated material, where n is the porosity and ρ_s is the mass density of the solid grains. u_i is the displacement of the skeleton in the i direction and $\ddot{u}$ is the acceleration of the skeleton in the i direction. The term n ρ_f is the mass of pore fluid per unit volume of saturated material where ρ_f is the mass density of the pore fluid. U_i is the absolute displacement of the pore fluid in the i direction.

The bulk mass density of the saturated material, ρ, is given by

$$\rho = (1-n)\,\rho_s + n\,\rho_f \tag{2.72}$$

Substitution of the value for (1-n) ρ_s from Equation 2.72 into Equation 2.70 gives

$$\sigma_{ij,\,j} = (\rho - n\,\rho_f)\,\ddot{u}_i + n\,\rho_f\,\ddot{U}_i \tag{2.73}$$

A term w_i is introduced which is the apparent fluid displacement in the i direction relative to the soil skeleton and is given by

$$w_i = n\,(U_i - u_i) \tag{2.74}$$

In seepage problems, w_i , is referred to as the discharge displacement. It describes the discharge of fluid through a soil mass of unit area. The discharge velocity, or apparent relative velocity, $\dot{w}_I$, between the soil particles and pore water is the velocity of water in a discharge duct of unit area needed to maintain the actual relative velocity in the porous soil of the same unit area. The actual relative velocity between the skeleton and the pore water is given by $\dot{w}_i / n$. Finally, $\ddot{w}_i$ is the apparent relative acceleration between the soil skeleton and pore water given by

$$\ddot{w}_i = n (\ddot{U}_i - \ddot{u}_i) \qquad (2.75)$$

Equation 2.73 can be expressed in terms of the apparent relative fluid acceleration as simply

$$\sigma_{ij,\,j} = \rho\, \ddot{u}_i + \rho_f\, \ddot{w}_i \qquad (2.76)$$

Equation of Motion for Pore Fluid

The finite element code SMAP-2D is capable of calculating the flow of pore fluid between elements. The flow of fluid with respect to the skeleton is controlled by Forchheimer's permeability model as described in a series of reports to the Air Force Office of Scientific Research (Kim, et. al., 1986, 1987, 1988; Blouin et. al., 1990, 1991). The Forchheimer model, as described by Kim, et. al. (1988) can be expressed as:

$$\pi_{,i} = \frac{\rho_f\, g}{k}\, \dot{w}_i + \frac{\beta_f}{k^{1/2}}\, \dot{w}_i^2 + \rho_f\, \ddot{U}_i \qquad (2.77)$$

where

$\pi_{,i}$	=	Pore pressure gradient
g	=	Acceleration of gravity
ρ_f	=	Mass density of pore fluid
k	=	Darcy's coefficient of permeability (function of skeleton and fluid properties)
β_f	=	Ward's turbulent flow coefficient (function of skeleton and fluid properties)
$\dot{w}$	=	Apparent flow velocity relative to the skeleton
$\ddot{U}$	=	Absolute acceleration of pore fluid

The first term in Equation 2.77 is simply Darcy's law while the velocity squared term was apparently first proposed by Forchheimer (1901). The first two terms represent the frictional component of the pressure gradient while the last term accounts for the inertial effect of fluid flow.

Equation 2.77 can also be written in the form:

$$\pi_{,i} = \frac{\rho_f\, g}{k'}\,\dot{w}_i + \rho_f\,\ddot{U}_i \tag{2.78}$$

where k' represents an equivalent permeability coefficient given by:

$$k' = \frac{k}{1 + \dfrac{\beta_f}{\rho_f\, g}\sqrt{k}\,|\dot{w}_i|} \tag{2.79}$$

Hence, the flow of pore fluid in the soil skeleton is governed by Equations 2.78 and 2.79 and the flow coefficients k and β_f which can be determined from laboratory test data. Using the Equation 2.75, Equation 2.78 can be expressed in terms of skeleton and apparent relative fluid motions given by

$$\pi_{,i} = \frac{\rho_f}{n}\,\ddot{w}_i + \rho_f\,\ddot{u}_i + k'\,\dot{w}_i \tag{2.80}$$

More recently, Blouin and his coworkers (1991) have proposed a refined expression of the Forchheimer model that, while remaining equivalent to Equation 2.77, clarifies the distinction between fluid-related and skeleton-related permeability properties. This new expression is:

$$\pi_{,i} = \frac{\mu}{\alpha}\,\dot{w}_i + \frac{\rho_f}{\beta}\,\dot{w}_i^2 + \rho_f\,\ddot{U}_i \tag{2.81}$$

where

μ	=	Dynamic viscosity of the fluid
α, β	=	Flow coefficients that are properties of the porous skeleton only

The conversions between the different permeability parameters are obtained from equating the corresponding terms of Equations 2.77 and 2.81 to obtain:

$$k = \frac{\alpha \rho_f g}{\mu} \tag{2.82}$$

$$\beta_f = \frac{k^{\frac{1}{2}} \rho_f}{\beta} \tag{2.83}$$

While the parameters α and β and Equation 2.81 form the preferred expression for the permeability model, the current implementation of the model in our numerical codes follow the form of Equations 2.77 through 2.80.

2.6 Spatial Discretization and Incremental Relationships of Field Variables

Within each element, field variables can be discretized into element nodal values.

$$\begin{aligned} \{\Delta w\} &= [N]\,\{\Delta \underline{w}\}_e \\ \{\Delta \varepsilon\} &= [B]\,\{\Delta \underline{u}\}_e \\ \Delta w_{i,i} &= \{1\}^T\,[B]\,\{\Delta \underline{w}\}_e \\ \{\Delta u\} &= [N]\,\{\Delta \underline{u}\}_e \end{aligned} \tag{2.84}$$

Stress vector at time step n can be expressed as:

$$\{\sigma_n\} = \{\sigma_{n-1}\} + \{\Delta\sigma'\} + \{1\}\,\Delta\pi \tag{2.85}$$

Combining Equations 2.50, 2.51, 2.67 and 2.84 yields

$$\begin{aligned}\{\Delta\sigma\} = &\left([D^{ep}]\,[B] + \bar{m}_1\,\{1\}\,\{1\}^T\,[B]\right)\{\Delta\underline{u}\} \\ &+ \bar{m}_2\,\{1\}\,\{1\}^T\,[B]\,\{\Delta\underline{w}\}\end{aligned} \tag{2.86}$$

where

$$\bar{m}_1 = \left[1 - \frac{K_s^{ep}}{K_g}\right]^2 \cdot \bar{m} \tag{2.87}$$

Equation 2.67 can be rewritten in incremental form as:

$$\Delta\pi = \bar{m}_2 \cdot \Delta u_{i,i} + \bar{m} \cdot \Delta w_{i,i} \tag{2.88}$$

2.7 Global Equilibrium Equations

Two global equilibrium equations are derived, first in terms of field variables and then discretized using nodal variables.

The first equates the total internal stresses plus the inertia forces to the applied boundary traction. Letting the solid skeleton movement be the virtual displacement, δu, the following global equilibrium equation for the bulk mixture is established:

$$\begin{aligned}\int_v \{\delta\varepsilon\}^T\,\{\sigma\}\,dv = &\int_s \{\delta u\}^T\,\{T\}\,ds - \int_v \{\delta u\}^T\,\rho\,\{\ddot{u}\}\,dv \\ &- \int_v \{\delta u\}^T\,\rho_f\,\{\ddot{w}\}\,dv\end{aligned} \tag{2.89}$$

where

$\delta\epsilon$ is the virtual strain corresponding to virtual displacement δu.

The second equates the applied pore pressure on the boundary to the internal pore pressure plus the flow resistance force plus the inertia force on the pore fluid. Taking the apparent relative fluid movement as the virtual displacement, δw, the internal virtual work done by the pore pressure should be equal to the external virtual work. That is,

$$\int_v (\delta w_{i,i})^T \pi \cdot dv = \int_s \{\delta w\}^T \hat{n}\, ds - \int_v \{\delta w\}^T [r] \cdot \{\dot{w}\}\, dv - \int_v \{\delta w\}^T \rho_f \{\ddot{u}\}\, dv - \int_v \{\delta w\}^T \frac{1}{n} \rho_f \{\ddot{w}\}\, dv \qquad (2.90)$$

Replacing the field variables in Equation 2.89 and 2.90 by the discretized nodal variables using Equation 2.84 gives the following global equilibrium equation at time step n:

$$\begin{bmatrix} M_m & M_c \\ M_c^T & M_f \end{bmatrix} \begin{Bmatrix} \underline{\ddot{u}}_n \\ \underline{\ddot{w}}_n \end{Bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & H \end{bmatrix} \begin{Bmatrix} \underline{\dot{u}}_n \\ \underline{\dot{w}}_n \end{Bmatrix} + \begin{bmatrix} K_t\text{+EE} & C \\ C^T & E \end{bmatrix} \begin{Bmatrix} \Delta\underline{u}_n \\ \Delta\underline{w}_n \end{Bmatrix} = \begin{Bmatrix} F_n \\ G_n \end{Bmatrix} - \begin{Bmatrix} R^s_{n-1} + R^f_{n-1} \\ R^f_{n-1} \end{Bmatrix} \qquad (2.91)$$

where

$$M_m = \Sigma \int_v [N]^T \rho\, [N]\, dv$$

$$M_c = \Sigma \int_v [N]^T \rho_f\, [N]\, dv$$

$$M_f = \Sigma \int_v [N]^T \frac{1}{n} \rho_f\, [N]\, dv$$

$$H = \Sigma \int_v [r]\, [N]^T [N]\, dv$$

$$K_t = \Sigma \int_v [B]^T [D^{ep}]\, [B]\, dv$$

$$EE = \Sigma \int_v \bar{m}_1 [B]^T \{1\} \{1\}^T [B] \, dv$$

$$C = \Sigma \int_v \bar{m}_2 [B]^T \{1\} \{1\}^T [B] \, dv$$

$$F_n = \Sigma \int_s [N]^T \{T\} \, ds + \Sigma \int_v [N]^T \rho \{b\} \, dv$$

$$E = \Sigma \int_v \bar{m} [B]^T \{1\} \{1\}^T [B] \, dv$$

$$G_n = \Sigma \int_s [N]^T \hat{\pi}_n \, ds + \Sigma \int_v [N]^T \rho_f \{b\} \, dv$$

$$R^s_{n-1} = \Sigma \int_v [B]^T \{\sigma'_{n-1}\} \, dv$$

$$R^f_{n-1} = \Sigma \int_v [B]^T \{1\} \pi_{n-1} \, dv$$

$[r]$ = Inverse of permeability matrix

$\{b\}$ = Component of body force vector

Equation 2.91 can be rewritten in the simpler form:

$$[M] \{\ddot{\underline{d}}_n\} + [D] \{\dot{\underline{d}}_n\} + [K] \{\Delta \underline{d}_n\} = \{P_n\} - \{R_{n-1}\} \qquad (2.92)$$

2.8 Linearized Global Equilibrium Equations

Introducing a time integration method which incorporates both Newmark's β method and Wilson's θ method, the generalized acceleration vector is expressed as

$$\{ \ddot{\underline{d}}_n \} = C_1 \{ \Delta \underline{d}_n \} + C_2 \{ \dot{\underline{d}}_{n-1} \} + C_3 \{ \ddot{\underline{d}}_{n-1} \} \tag{2.93}$$

where

$$C_1 = \frac{1}{\beta\, \theta^3\, \Delta t^2}$$
$$C_2 = -\frac{1}{\beta\, \theta^2\, \Delta t} \tag{2.94}$$
$$C_3 = 1 - \frac{1}{2\, \beta\, \theta}$$

and the generalized velocity vector is expressed as

$$\{ \dot{\underline{d}}_n \} = B_1 \{ \Delta \underline{d}_n \} + B_2 \{ \dot{\underline{d}}_{n-1} \} + B_3 \{ \ddot{\underline{d}}_{n-1} \} \tag{2.95}$$

where

$$B_1 = \frac{\gamma}{\beta\, \theta^3\, \Delta t}$$
$$B_2 = 1 - \frac{\gamma}{\beta\, \theta^2} \tag{2.96}$$
$$B_3 = \Delta t - \frac{\gamma}{2\, \beta\, \theta}\, \Delta t$$

Substituting Equations 2.93 and 2.95 into Equation 2.92 and rearranging, we can obtain the following linearized global equilibrium equations which can be solved simultaneously at each step:

$$[\tilde{K}]\{\Delta \underline{d}_n\} = \{\tilde{P}_n\} \tag{2.97}$$

where the generalized stiffness matrix is given by

$$[\tilde{K}] = C_1 [M] + B_1 [D] + [K] \tag{2.98}$$

and the generalized force vector is given by

$$\begin{aligned} \{\tilde{P}_n\} = \{P_n\} - \{R_{n-1}\} - [M] (C_2 \{\dot{\underline{d}}_{n-1}\} + C_3 \{\ddot{\underline{d}}_{n-1}\}) \\ - [D] (B_2 \{\dot{\underline{d}}_{n-1}\} + B_3 \{\ddot{\underline{d}}_{n-1}\}) \end{aligned} \tag{2.99}$$

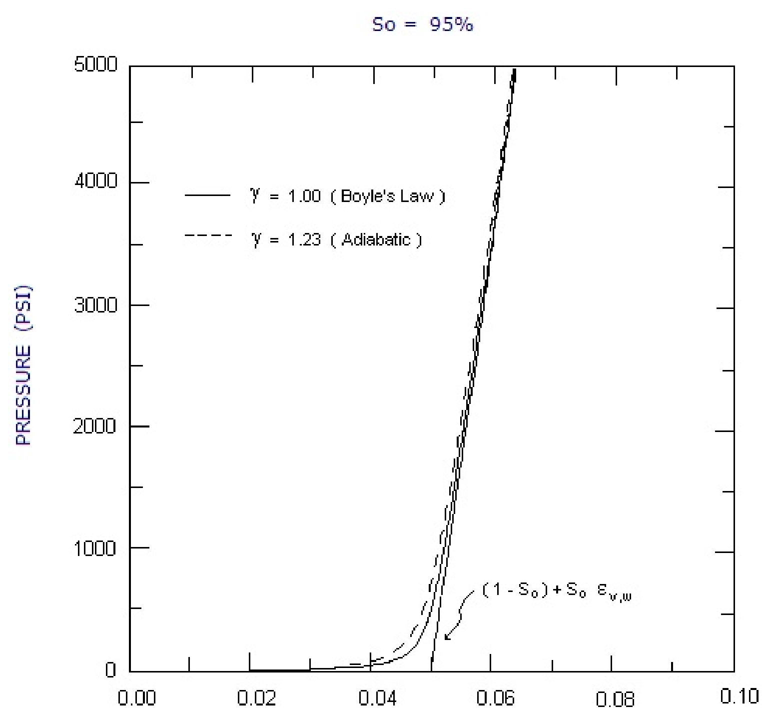


Figure 2.1 Prediction of air-water compressibility

Conservation of Fluid Mass

$$n \ \rho_f \ v_t = n' \ \rho_f' \ v_t'$$

v_t = Apparent fluid volume before compression
v_t' = $(1+ \varepsilon_F)\ v_t$: apparent fluid volume after compression
ε_v = Volumetric strain of porous skeleton
ε_F = Volumetric diffusion of pore fluid

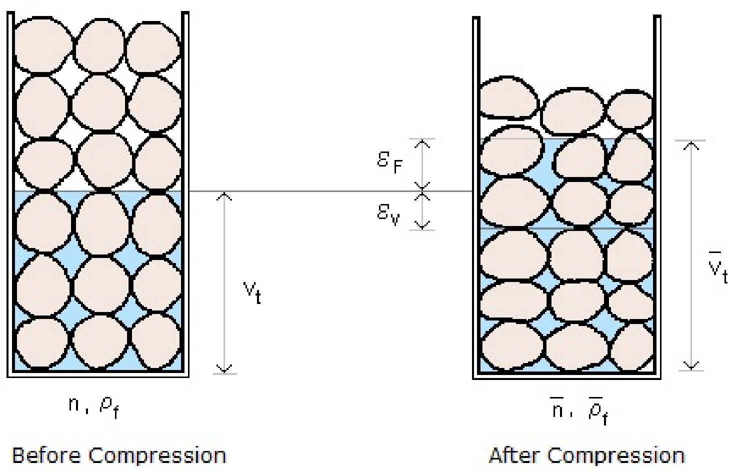


Figure 2.2 Schematic illustration of conservation of pore fluid mass in saturated porous materials

Nonlinear Material Models

3.1 Generalized Hoek and Brown Model

3.1.1 Introduction

Generalized Hoek and Brown Model represents the skeleton constitutive relations of soils or porous materials. In its generalized form, the model includes the empirically based Hoek and Brown failure equation as well as the classical Von Mises, Mohr-Coulomb, and Drucker-Prager failure equations. As one of the useful features, the model can use empirical data base for the strength of in situ rock mass when the in situ strength data are not available.

In this section, the 2-dimensional elasto-plastic matrix is derived for the Generalized Hoek and Brown Model. The model is elastic below the failure surface and perfectly plastic along the failure surface with the volumetric and deviatoric behaviors dependent upon one another once the failure surface is reached.

And the failure equation is expressed in terms of the alternate stress invariant (p, q, and θ) given by

$$p = \frac{1}{3}\sigma_{ii}$$

$$
\begin{aligned}
S_{ij} &= \sigma_{ij} - p \cdot \delta_{ij} \\
J_2 &= \frac{1}{2} S_{ij} S_{ij} \\
J_3 &= \frac{1}{3} S_{ij} S_{jk} S_{ki} \\
q &= \sqrt{3J_2} \\
\theta &= \frac{1}{3} \mathrm{Sin}^{-1}\left(- \frac{27}{2} \frac{J_3}{q^3} \right)
\end{aligned}
\qquad (3.1)
$$

where σ_{ij} is the total stress tensor and S_{ij} is the deviatoric stress tensor.

3.1.2 Elastic Stress-Strain Relationship

The incremental elastic constitutive law can be expressed in the following matrix form:

$$\{d\sigma\} = [D^e]\{d\varepsilon^e\} \qquad (3.2)$$

where

$\{d\sigma\}$	Stress increment
$[D^e]$	Elastic stress-strain matrix
$\{d\varepsilon^e\}$	Elastic strain increment

3.1.3 Failure Surface

The failure surface is described by the following equation:

$$F(p, q, \theta) = q - ((\alpha + \beta p)^n + \kappa) R(\theta) = 0 \qquad (3.3)$$

The expression for R(θ) in Equation 3.3 is given by

$$R(\theta) = \frac{x\,(\sqrt{3}\,\cos\theta + \sin\theta) + (2k-1)\,[(2+\cos2\theta+\sqrt{3}\,\sin2\theta)\,x + 5k^2-4k]^{1/2}}{[\,x\,(2+\cos2\theta+\sqrt{3}\,\sin2\theta) + (1-2k)^2\,]} \tag{3.4}$$

where

$$\left(-\frac{\pi}{6} \le \theta \le \frac{\pi}{6}\right)$$

x = $(1\text{-}k^2)$

k = the ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same mean pressure

The function R(θ) describes the shape of the yield surface, as projected in the π plane (octahedral plane). Figure 3.1 and 3.2 show the influence of the parameter k on the shape of the yield surface. k is the ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same mean pressure. k is a measure of the influence of the intermediate principal stress on the yield surface and can vary from 0.5 to 1.0. When k is equal to unity, R(θ) is circular, indicating a Drucker-Prager or Von Mises failure model. When k is less than unity, R(θ) is a smooth cornered approximation to the Mohr-Coulomb failure envelope.

The parameter n in Equation 3.3 determines the shape of the yield surface in the p-q plane. For n=0, the shear strength is constant with respect to the mean pressure and the strength envelope reduces to the Von Mises or Tresca yield surface. For n=1/2, the strength envelope represents Hoek and Brown (1982) failure surface. This nonlinear failure model is a multidimensional generalization of the original one-dimensional axisymmetric Hoek and Brown model which is based on extensive laboratory and field data (Kim, Piepenburg and Merkle, 1986).

For n = 1, shear strength is linearly proportional to the mean pressure and the strength envelope in the p-q plane is representative of the Drucker-Prager or Mohr-Coulomb failure surface.

The parameters α, β and κ of Equation 3.3 define the failure envelope in the p-q plane. They can be determined from laboratory tests. Recommended relationships for determining these parameters for Von Mises, Hoek and Brown and Mohr-Coulomb type materials are listed in Table 3.1. The empirical material parameters for n=1/2 are tabulated in Table 3.2 for several different rock types as a function of rock quality. Detailed description of rock quality is shown in Table 3.3.

3.1.4 Flow Rule

A variable dilatancy potential function, G, is defined such as

$$\frac{\partial G}{\partial p} = \left(\frac{\partial F}{\partial p}\right) r$$
$$\frac{\partial G}{\partial q} = \frac{\partial F}{\partial q} \qquad (3.5)$$
$$\frac{\partial G}{\partial \theta} = \frac{\partial F}{\partial \theta}$$

where r is a dilatancy parameter ($0 \le r \le 1$)

r = 0 No plastic volume change
= 1 Associated flow

Thus, in general,

$$\{d\varepsilon^p\} = d\lambda \, \{g\} \qquad (3.6)$$

where

$$\{g\} = \left\{\frac{\partial G}{\partial \sigma}\right\}$$

3.1.5 Consistency Equation

During yielding , the consistency equation forces the stress to move along the failure surface

$$dF = \{a\}^T \{d\sigma\} = 0 \tag{3.7}$$

where

$$\{a\} = \left\{\frac{\partial F}{\partial \sigma}\right\} \tag{3.8}$$

3.1.6 Incremental Elasto-Plastic Constitutive Law

Total strain is defined as the sum of elastic and plastic strains

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^p\} \tag{3.9}$$

Substituting Equation 3.9 into 3.2, we have

$$\{d\sigma\} = [D^e] (\{d\varepsilon\} - \{d\varepsilon^p\}) \tag{3.10}$$

From the flow rule defined in Equation 3.6, we can rewrite Equation 3.10 as

$$\{d\sigma\} = [D^e] \{d\varepsilon\} - d\lambda [D^e] \{g\} \tag{3.11}$$

Substituting Equation 3.11 into 3.7 and solving for dλ, we obtain

$$d\lambda = \frac{\{a\}^T [D^e] \{d\varepsilon\}}{\{a\}^T [D^e] \{g\}} \tag{3.12}$$

Back substituting Equation 3.12 into Equation 3.11, the stress increment is directly related to the total strain increment as follows:

$$\{d\sigma\} = [D^{ep}]\{d\varepsilon\} \tag{3.13}$$

where

$$[D^{ep}] = [D^e] - \frac{[D^e]\{g\}\{a\}^T[D^e]}{\{a\}^T[D^e]\{g\}} \tag{3.14}$$

3.1.7 Calculation of {a}

Differentiating the yield function with respect to p, q, and θ, we have

$$\begin{aligned} \frac{\partial F}{\partial p} &= -n(\alpha + \beta p)^{n-1} \cdot \beta \cdot R(\theta) \\ \frac{\partial F}{\partial q} &= 1 \\ \frac{\partial F}{\partial \theta} &= -\{(\alpha + \beta p)^n + \kappa\} \frac{\partial R(\theta)}{\partial \theta} \end{aligned} \tag{3.15}$$

where

$$\begin{aligned} \frac{\partial R}{\partial \theta} &= \frac{1}{R_D}\left[\frac{\partial R_N}{\partial \theta} - R(\theta)\frac{\partial R_D}{\partial \theta}\right] \\ R_N &= x(\sqrt{3}\cos\theta + \sin\theta) + (2k-1)[(2 + \cos 2\theta + \sqrt{3}\sin 2\theta)x + 5k^2 - 4k]^{1/2} \\ R_D &= x(2 + \cos 2\theta + \sqrt{3}\sin 2\theta) + (1-2k)^2 \\ \frac{\partial R_N}{\partial \theta} &= x(\cos\theta - \sqrt{3}\sin\theta) + \frac{x(2k-1)(\sqrt{3}\cos 2\theta - \sin 2\theta)}{[x(2 + \cos 2\theta + \sqrt{3}\sin 2\theta) + 5k^2 - 4k]^{1/2}} \\ \frac{\partial R_D}{\partial \theta} &= 2x(\sqrt{3}\cos 2\theta - \sin 2\theta) \end{aligned} \tag{3.16}$$

The derivative of the yield function with respect to stress can be written in general 3-dimensional condition as

$$\{a\} = \frac{\partial F}{\partial p}\left\{\frac{\partial p}{\partial \sigma}\right\} + \frac{\partial F}{\partial q}\left\{\frac{\partial q}{\partial \sigma}\right\} + \frac{\partial F}{\partial \theta}\left\{\frac{\partial \theta}{\partial \sigma}\right\} \tag{3.17}$$

where

$$\left\{\frac{\partial p}{\partial \sigma}\right\} = \frac{1}{3} < 1\ 1\ 1\ 0 >^T$$

$$\left\{\frac{\partial \theta}{\partial \sigma}\right\} = \frac{9}{2q^3 \cos 3\theta}\left(\frac{3J_3}{q}\left\{\frac{\partial q}{\partial \sigma}\right\} - \left\{\frac{\partial J_3}{\partial \sigma}\right\}\right)$$

$$\left\{\frac{\partial q}{\partial \sigma}\right\} = \frac{3}{2q} < S_x \quad S_y \quad S_z \quad 2\,\sigma_{xy} >^T$$

$$\left\{\frac{\partial J_3}{\partial \sigma}\right\} = \left\{\begin{matrix} S_y S_z + \frac{1}{9} q^2 \\ S_x S_z + \frac{1}{9} q^2 \\ S_x S_y - \sigma_{xy}^2 + \frac{1}{9} q^2 \\ -2\, S_z \sigma_{xy} \end{matrix}\right\}$$

$$\{\sigma\}^T = < \sigma_x \quad \sigma_y \quad \sigma_z \quad \sigma_{xy} >$$

$$\{\varepsilon\}^T = < \varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} >$$

$$\gamma_{xy} = 2\, \varepsilon_{xy}$$

Table 3.1 Material Constants in Grenerlized Hoek and Brown Model

	n = 0 Von Mises or Tresca	n = 1/2 Hoek and Brown	n = 1 Mohr-Coulomb or Drucker-Prager
α	N/A	$\left(\frac{m^2}{36} + s\right) \sigma_c^2$	1000
β	N/A	$m\ \sigma_c$	$\frac{6 \sin\phi}{(3 - \sin\phi)}$
κ	$q' - 1$	$\frac{1}{6}\ m\ \sigma_c$	$\frac{3(1 - \sin\phi)}{(3 - \sin\phi)} \sigma_c - 1000$

q' = $\sigma_1 - \sigma_3$
where σ_1 and σ_3 are major and minor pricipal stresses at failure.

σ_c = Unconfined compressive strength

ϕ = Internal friction angle

m,s = Hoek and Brown's material constants as tabulated in Table 3.2.

Table 3.2 Hoek and Brown Material Parameters (m, s)

Rock Type / Rock Quality	Dolomite, Limestone & Marble	Mudstone, Siltstone, Shale and Slate (normal to cleavage)	Sandstone and Quartzite	Andesite, Dolerite & Rhyolite	Amphibolite, Gabbro, Gneiss, Norite and Quartz-Diorite
Intact CSIR rating = 100 NGI rating = 150	m = 7 s = 1	10.0 1.0	15.0 1.0	17.0 1.0	25.0 1.0
Very Good Quality CSIR rating = 85 NGI rating = 100	3.5 0.1	5.0 0.1	7.5 0.1	8.5 0.1	12.5 0.1
Good Quality CSIR rating = 65 NGI rating = 10	0.7 0.004	1.0 0.004	1.5 0.004	1.7 0.004	2.5 0.004
Fair Quality CSIR rating = 44 NGI rating = 1	0.14 0.001	0.20 0.0001	0.3 0.0001	0.34 0.0001	0.5 0.0001
Poor Quality CSIR rating = 23 NGI rating = 0.1	0.04 0.00001	0.05 0.00001	0.08 0.00001	0.09 0.00001	0.13 0.00001
Very Poor Quality CSIR rating = 3 NGI rating = 0.01	0.007 0.0	0.01 0.0	0.015 0.1	0.017 0.0	0.025 0.0

Table 3.3 Description of Rock Quality in Table 3.2

Intact Rock Samples	Laboratory size specimens free from joints
Very Good Quality Rock Mass	Tightly interlocking undisturbed rock with unweathered joints at 1 to 3m
Good Quality Rock Mass	Fresh to slightly weathered rock, slightly disturbed with joints at 1 to 3m
Fair Quality Rock Mass	Several sets of moderately weathered joints spaced at 0.3 to 1m
Poor Quality Rock Mass	Numerous weathered joints at 30 to 500mm with sane gouge. Clean compacted waste rock
Very Poor Quality Rock Mass	Numerous heavily weathered joints spaced < 50m with gouge. Waste rock with fines

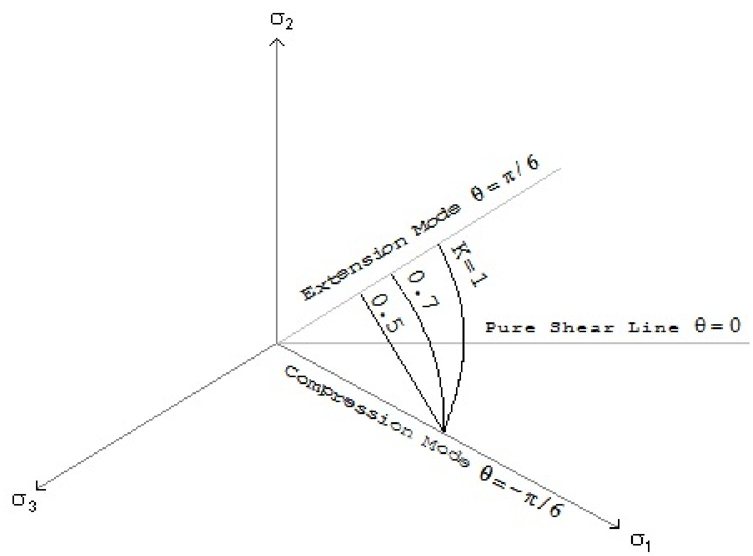


Figure 3.1 Shape of strength envelope, R(θ), on octahedral plane

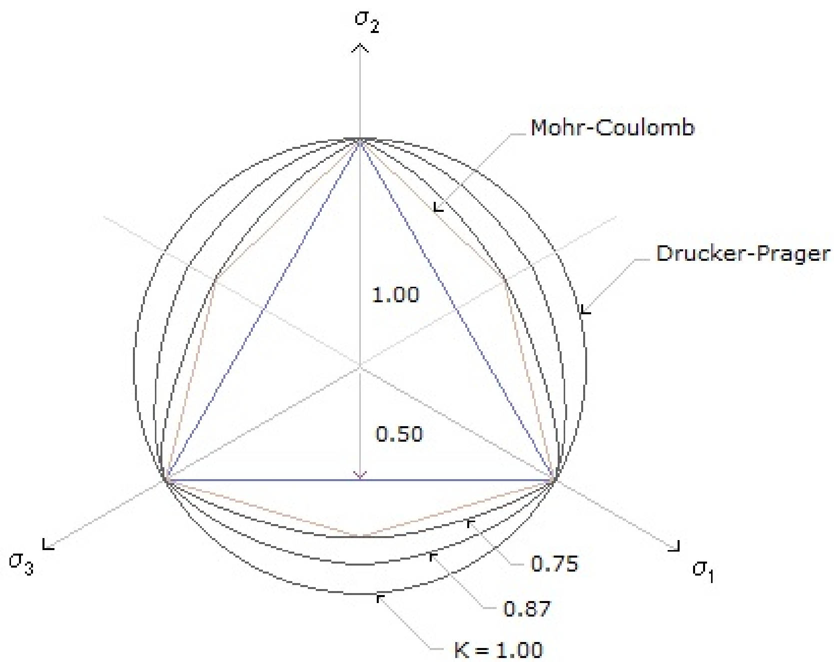


Figure 3.2 Shape of strength envelope, R(θ), on octahedral plane

3.2 Single Hardening Elasto-Plastic Model

3.2.1 Introduction

The Single Hardening Elasto-Plastic Model is a 3 invariant, single hardening surface, material model representing the drained response of the porous skeleton. The Single Hardening Model is the simplified version of the existing three invariant model which has been continuously upgraded since 1985 and reported by Merkle and Dass (1985), Dass and Merkle (1986), and Blouin, Chitty, Rauch, and Kim (1990). The major change from the existing three invariant model is the replacement of two hardening yield surfaces by the single hardening yield surface developed by Lade (1990). The advantages of this new model over the three invariant model include requirement of a lesser number of material constants, simple procedures of material parameter determination, and computational efficiency.

The Single Hardening Elasto-Plastic Model is a non-associated, isotropic, work hardening, elasto-plastic model with a single hardening yield surface bounded by a failure envelope, as shown in Figure 3.3. The yield surface has the shape of a teardrop with its pointed apex at the origin in principal stress space. The failure surface is a hyperboloid with its apex on the hydrostatic axis in the principal stress space. The shape of both yield and failure surfaces in the π-plane, perpendicular to the hydrostatic axis (see Figure 3.3), is a triple ellipse in polar coordinates.

3.2.2 Notations

Positive signs are used throughout this section to represent compression. Only those symbols which are not explicitly defined in the main text will be described below.

P_a	Atmospheric pressure
$\{\varepsilon\}$	Total strain vector
$\{\varepsilon_e\}$	Elastic strain vector
$\{\varepsilon_p\}$	Plastic strain vector associated with yield surface
$\{\varepsilon_u\}$	Plastic strain vector associated with failure surface

$\{\sigma\}$	Stress vector
σ_{oct}	Octahedral normal stress
τ_{oct}	Octahedral shear stress
ν	Poisson's ratio

3.2.3 Total Strain Formulation

In general, the total strain is comprised of the elastic component and two plastic components related to the yield and failure surfaces as given respectively by:

$$\{d\varepsilon\} = \{d\varepsilon_e\} + \{d\varepsilon_p\} + \{d\varepsilon_u\} \quad (3.18)$$

3.2.4 Elastic Response

At stress states inside the yield surfaces, the skeleton response is treated as nonlinear elastic and governed by the previous maximum peak stress. Two options are available for modeling the elastic response within the framework of the three invariant skeleton model: the modified elastic model and the Lade and Nelson elastic model. In both options, Poisson's ratio is assumed to remain constant.

Modified elastic model

During virgin unloading, the elastic bulk modulus is given by:

$$K = \frac{K_{ur}\,P_a}{3(1-2\nu)}\left[\frac{\sigma_{oct}}{P_a}\right]^n \geq K_i \quad (3.19)$$

where K_{ur} and n are material constants obtained in the parameter fitting. K_i represents the initial bulk modulus at low pressures and is necessary for modeling the behavior of rock-type materials that have a definite initial elastic behavior. In uncemented soils, K_i can be taken as a very small value. The initial bulk modulus is also used to determine the initial position of the yield surface by defining the initial elastic range.

During unloading or reloading, the skeleton modulus is described by one of two segments as depicted in Figure 3.4. Between the previous peak mean stress, $\sigma_{oct,max}$, and the transition into the nonlinear segment at $\sigma_{oct,\,b}$, the elastic bulk modulus is constant and is given by:

$$K = K_1 = \frac{K_{ur}\,P_a}{3(1-2\nu)}\left[\frac{\sigma_{oct,\,max}}{P_a}\right]^n \tag{3.20}$$

The transition into the nonlinear segment occurs at:

$$\sigma_{oct,b} = \lambda\,\sigma_{oct,\,max} \tag{3.21}$$

where λ is a model parameter. At mean stresses less than $\sigma_{oct,\,b}$ the nonlinear bulk modulus is given by:

$$K = K_1\left[1 - \gamma\left[\frac{\beta}{\gamma}\right]^{\frac{\sigma_{oct}}{\sigma_{oct,b}}}\right] \tag{3.22}$$

Referring to Figure 3.4, the model parameters γ and β are given by:

$$\gamma = 1 - \frac{K_o}{K_1} \tag{3.23}$$

where K_0 is the bulk modulus at zero pressure and

$$\beta = 1 - \frac{K_*}{K_1} \tag{3.24}$$

where K_* is the bulk modulus at one quarter of the transition pressure $\sigma_{oct,b}$. While this formulation allows for relatively accurate curve fitting of observed soil response, the model has three disadvantages:

1. For certain closed-loop stress/strain paths, the model may violate the energy conservation principle;
2. Unloading at low pressures could potentially generate expansive volumetric strains; and
3. At the transition pressure, $\sigma_{oct,b}$ the modulus is not continuous.

Lade and Nelson elastic model

The second elastic model option is based on a relationship derived by Lade and Nelson (1987). This formulation is continuous and was derived from the energy conservation principle. Lade and Nelson's model can be expressed as:

$$K = \frac{K_{ur} P_a}{3^{n+1}(1-2\nu)} \left[\left[\frac{3\sigma_{oct}}{P_a} \right]^2 + \frac{6(1+\nu)}{1-2\nu} \frac{J_2^1}{P_a^2} \right]^{\frac{n}{2}} \geq K_i \qquad (3.25)$$

where the parameters K_{ur}, n, and K_i are the same as used in Equation 3.19. Since this model is fit strictly using the slope of an initial unload curve, it can be difficult to closely match the observed characteristics of an unload cycle.

Fitting

Poisson's ratio (ν) for a given material can be determined in a number of ways using unload/reload data which represent the elastic response of the skeleton. Lade and Nelson (1987) recommended obtaining Poisson's ratio directly from strain measurements in triaxial compression unload/reload cycles, right after stress reversal at hydrostatic conditions where:

$$\nu = -\frac{\varepsilon_r}{\varepsilon_a} = \frac{1}{2}\left[1 - \frac{\varepsilon_v}{\varepsilon_a}\right] \qquad (3.26)$$

where

ε_a	Axial strain
ε_r	Radial strain
ε_v	Volume strain

In addition, since the elastic response is completely defined by any two independent elastic parameters, Poisson's ratio can be obtained from the bulk modulus, (K), measured in a hydrostatic compression unload, and any other elastic modulus. For example, a triaxial compression unload yields the shear modulus (G), an unconfined compression unload gives the

Young's modulus (E), and an uniaxial strain unload produces the constrained modulus (M). Any one of these parameters can be used with the bulk modulus to obtain Poisson's ratio:

$$\nu = \frac{3K - 2G}{2(3K + G)} \tag{3.27}$$

$$\nu = \frac{3K - E}{6K} \tag{3.28}$$

$$\nu = \frac{3K - M}{3K + M} \tag{3.29}$$

To obtain the elastic model parameters K_{ur} and n, Equation 3.19 is rewritten in the form:

$$\log\left[\frac{3K(1-2\nu)}{P_a}\right] = \log K_{ur} + n \log\left[\frac{\sigma_{oct}}{P_a}\right] \tag{3.30}$$

Values of K and σ_{oct} from the initial unloading response at various pressures in the hydrostatic compression test, are then plotted as log $(3K(I\text{-}2\nu)/P_a)$ versus log (σ_{oct}/P_a). A least squares linear regression is then applied in log-log space. The parameter n is the slope of this line, while K_{ur} is the intercept where (σ_{oct}/P_a) is 1.0. The parameters λ, γ, and β for the modified elastic unload model are determined from a single unload/reload cycle in the hydrostatic compression test as depicted in Figure 3.4. The parameters are computed using Equations 3.21, 3.23, and 3.24.

3.2.5 Failure Surface

The failure surface is a hyperboloid with its apex on the hydrostatic axis in principal stress space as shown in Figure 3.3. The shape of the failure surface in the π -plane, perpendicular to the hydrostatic axis is a triple ellipse in polar coordinates.

The failure criteria are given by:

$$f_u = \frac{\tau_{oct}}{R(\theta)}\left(\frac{m}{P_a} + \frac{1}{\bar{\sigma}_{oct}}T\right) - \eta_1 = 0 \tag{3.31}$$

where

$$R(\theta) = \frac{2K}{(1+K) + (1-K)\,\mathrm{Sin}3\theta} \tag{3.32}$$

$$\bar{\sigma}_{oct} = \sigma_{oct} + T \tag{3.33}$$

σ_{oct}	Octahedral normal stress
τ_{oct}	Octahedral shear stress
θ	Lode angle
T	Tensile strength
K	The ratio of extensive to compressive strength at given mean pressure

m and η_1 are the failure constants which can be determined from the following fitting procedure. In triaxial compression mode, $R(\theta) = 1$ and Equation 3.31 reduces to:

$$\frac{\bar{\sigma}_{oct}}{\tau_{oct}} = \frac{1}{\eta_1} + \frac{m}{\eta_1}\left(\frac{\bar{\sigma}_{oct}}{P_a}\right) \tag{3.34}$$

By plotting the failure stress points from each triaxial compression test in terms of $\bar{\sigma}_{oct}/\tau_{oct}$ versus $\bar{\sigma}_{oct}/P_a$, a straight line fit will yield an intercept of $1/\eta_1$ and a slope of m/η_a. Then the parameter η_1 is obtained simply by taking the inverse value of intercept and the parameter m is obtained by multiplying the slope by η_1.

3.2.6 Plastic Response Related to Yield Surface

Both yield and potential equations are based on Lade's single hardening model (Lade, 1990) which replaces previous two yield surface model (Lade, 1977).

To be consistent with the failure equation described in the previous subsection, however, Lade's equations were modified such that the shape of both yield and potential surfaces in the π-plane consists of triple ellipse given by Equation 3.32.

The yield equation is composed of the stress function (f_p') and the hardening function (f_p'').

$$f_p = f_p'(I_1, J_2, \theta) - f_p''(W_p) = 0 \tag{3.35}$$

The stress function is given by:

$$f_p' = \left(\psi_1 \frac{\bar{I}_1^{\,3}}{\bar{I}_3} - \frac{\bar{I}_1^{\,2}}{\bar{I}_2}\right)\left[\frac{\bar{I}_1}{p_a}\right]^h - e^q \tag{3.36}$$

where the stress quantities I_1, I_2, and I_3 are defined by:

$$\bar{I}_1 = I_1 + 3T \tag{3.37}$$

$$\bar{I}_2 = \left(\frac{J_2}{R(\theta)^2}\right) - \frac{\bar{I}_1^{\,3}}{3} \tag{3.38}$$

$$\bar{I}_3 = 2\left[\frac{J_2}{3R(\theta)^2}\right]^{\frac{3}{2}} - \frac{\bar{I}_1}{3}\left(\frac{J_2}{R(\theta)^2}\right) + \frac{\bar{I}_1^{\,3}}{27} \tag{3.39}$$

Note that I_1 is the first invariant of the total stress tensor, J_2 is the second invariant of deviatoric stress tensor and $R(\theta)$ is given by Equation 3.32. The variable q in Equation 3.36 is related to the shear stress level S as:

$$q = \frac{\alpha \cdot s}{1 - (1 - \alpha) \cdot s} \tag{3.40}$$

The shear stress level is defined as:

$$S = \frac{\frac{\tau_{oct}}{R_{(\theta)}}\left(\frac{m}{P_a} + \frac{1}{\bar{\sigma}_{oct}}\right)}{\eta_1} \tag{3.41}$$

The parameter q has the value of zero along the hydrostatic axis and unity along the failure surface. Thus, the material constants which are specific to the stress function are ψ_1, h and α. Determination of these material constants will be described at the end of this subsection.

The hardening function is given by:

$$f_p'' = \left[\frac{W_p}{D \cdot P_a}\right]^{\frac{h}{p}} \tag{3.42}$$

where the plastic work is expressed as:

$$W_p = \int \{\sigma\}^T \{d\epsilon_p\} \tag{3.43}$$

and the constant D is related to the isotropic hardening constants (C and P) as:

$$D = \frac{C}{(27\psi_1 + 3)^{P/h}} \tag{3.44}$$

The constant ψ_1 in Equation 3.36 and 3.44 is assumed to depend on the type of material.

Material Type	ψ_1
Sand	0.018
Clay	0.006
Mortar	0.004
Sandstone	0.0013
Concrete	0.0015
Reinforced Concrete	0.0007

It should be noted that the values of ψ_1 in the above table are based on Lade's data (Kim and Lade, 1988) but ψ_1 does not have any influence on the shape of yield surfaces on the π-plane.

Isotropic hardening constants (C and P) can be determined by fitting to the isotropic compression test. For the isotropic compression loading, Equation 3.35 reduces to:

$$\frac{W_p}{P_a} = C\left[\frac{I_1}{P_a}\right]^p \tag{3.46}$$

Taking the logarithm of both sides of Equation 3.46 yields

$$\log_{10}\left(\frac{W_p}{P_a}\right) = \log_{10}C + P\log_{10}\left(\frac{I_1}{P_a}\right) \tag{3.47}$$

so that the parameters C and P can be found from a log-log plot of (W_p/P_a) versus (I_1/P_a).

Yield constants (h and α) can be obtained by fitting to the triaxial compression test data. Along the isotropic and triaxial compression stress paths, Equation 3.35 has the same form as the Lade's single hardening yield equation so that the same procedure as described by Lade and Kim, 1988 can be used to determine yield constants (h and α).

The yield constant h can be obtained from:

$$h = \frac{\log_{10}\left(\frac{\left(\Psi_1 \frac{I_{1F}^3}{I_{3F}} - \frac{I_{1F}^2}{I_{2F}}\right)e}{27\psi_1 + 3}\right)}{\log_{10}\left(\frac{I_{1H}}{I_{1F}}\right)} \tag{3.48}$$

where I_{IF}, I_{2F}, and I_{3F} are the first, second and third invariant of the total stress tensor, respectively, at the failure point of triaxial compression test; I_{1H} is the first invariant of the total stress tensor in the hydrostatic compression test, measured at the same plastic work as for the failure point of triaxial compression test.

The yield constant α can be determined from

$$\alpha = \frac{1}{4} \frac{q_{80}}{1 - q_{80}} \tag{3.49}$$

Where q_{80} is value of q at the stress level S= 0.8 and is obtained from

$$q = l_n \frac{\left[\frac{W_p}{D P_a}\right]^{\frac{h}{p}}}{\left(\psi_1 \frac{I_1^3}{I_3} - \frac{I_1^2}{I_2}\right)\left[\frac{I_1}{P_a}\right]^h} \tag{3.50}$$

The potential equation is expressed in terms of stress invariants as

$$g_p = \left(\psi_1 \frac{\bar{I}_1^3}{\bar{I}_3} - \frac{\bar{I}_1^2}{\bar{I}_2} + \psi_2\right)\left[\frac{\bar{I}_1}{P_a}\right]^\mu \tag{3.51}$$

Material constants (ψ_2 and μ) which are specific to the potential surface can be determined in the same way as for Lade's single hardening model (Kim and Lade, 1988).

For the triaxial compression test, the potential constants (ψ_2 and μ) are related to the stresses and the plastic strains as

$$\xi_y = \frac{1}{\mu}\xi_x - \psi_2 \tag{3.52}$$

where

$$\xi_x = \psi_1 \frac{I_1^{\,3}}{I_3} - \frac{I_1^{\,2}}{I_2} \tag{3.53}$$

$$\xi_y = \frac{1}{1+\upsilon_p}\left(\frac{I_1^{\,3}}{I_2^2}(\sigma_a + \sigma_r + 2\upsilon_p\sigma_r) + \psi_1\frac{I_1^{\,4}}{I_3^{\,2}}(\sigma_a\sigma_r + \upsilon_p\sigma_r^2)\right)$$

$$- 3\Psi_1\frac{I_1^{\,3}}{I_3} + 2\frac{I_1^{\,2}}{I_2} \tag{3.54}$$

and

$$\upsilon_p = -\frac{\varepsilon_r^p}{\varepsilon_a^p} \tag{3.55}$$

Note that σ_a and σ_r are the axial and radial stress, respectively, and ε_a^p and ε_r^p are the axial and radial plastic strain, respectively. As described in Equation 3.52, the constants ψ_2 and μ now can be determined by the least square fit of a series of ξ_x and ξ_y data set.

The potential surface in Equation 3.51 is mainly used to compute the direction of the plastic strain increment during yielding.

$$\{d\varepsilon_p\} = d\lambda_p\left\{\frac{\partial g_p}{\partial\sigma_{ij}}\right\} \tag{3.56}$$

where $d\lambda_p$ is the scalar quantity.

3.2.7 Plastic Response Along the Failure Surface

When materials are hardening along the failure surface, the yield surface is crossing over the failure surface so that the state of stresses should satisfy both yield and failure equations. For simplicity, it has been assumed that there are no plastic volume changes and no strain softenings associated with the failure surface. However, there will be plastic volume changes associated with the yield surface along the failure surface. The exact method to derive the elasto-plastic stress-strain matrix [D_{ep}] is presented by Merkle and Dass (1985).

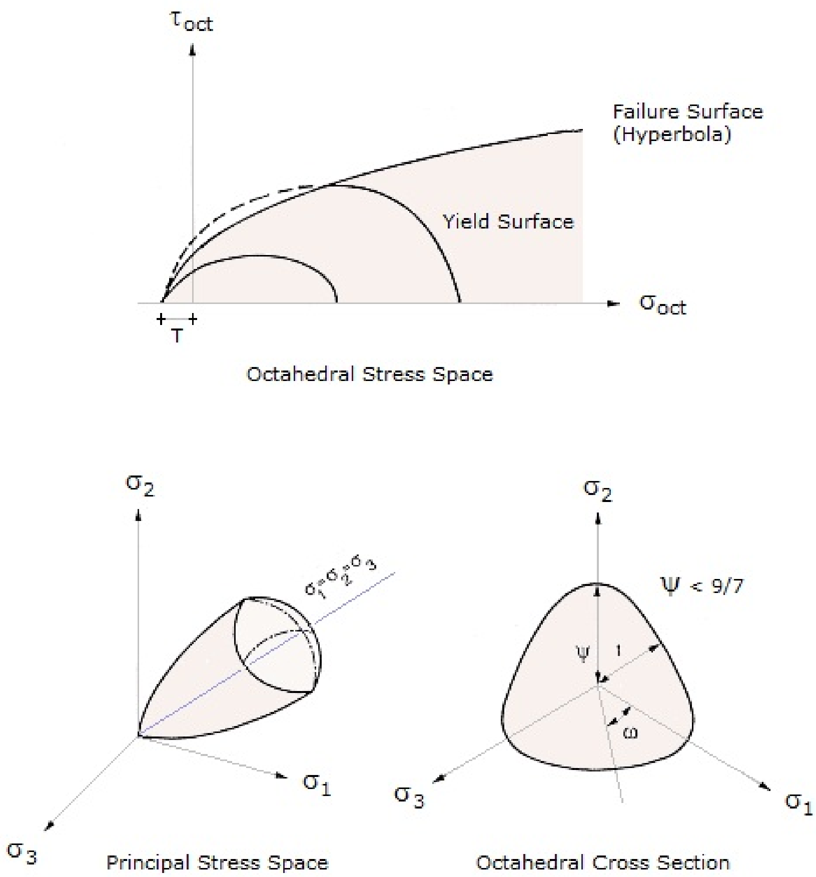


Figure 3.3 Yield and Failure Surfaces used in the Single Hardening Elasto-Plastic Model

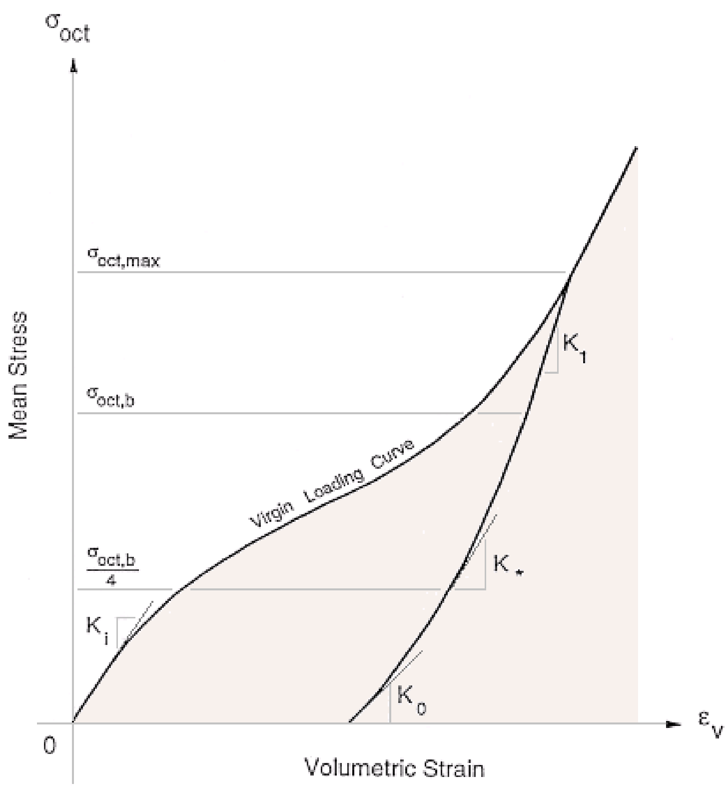


Figure 3.4 Fitting of modified elastic formulation to unload/reload hydrostatic compression response

3.3 JWL High Explosive Model

JWL High Explosive Model represents JWL equation of state (Jones, Wilkens, and Lee, 1968).

JWL is a relatively simple pressure-volume-energy equation of state developed to predict the adiabatic expansion of detonation products. JWL equation of state is given by:

$$\bar{P} = A\left(1-\frac{\omega}{R_1 V}\right)e^{-R_1 V} + B\left(1-\frac{\omega}{R_2 V}\right)e^{-R_1 V} + \frac{\omega E}{V} \tag{3.57}$$

where $\bar{p}$ is the pressure, V is the relative volume (ρ_o/ρ), and E is the internal energy density. And A, B, R_1, R_2, and ω are material constants.

To simulate progress of chemical reaction, Burn Fraction (BF) is used.

$$BF = \frac{(t - t_b)\, C_d}{B_s\, \ell} \tag{3.58}$$

where

t	Current time
t_b	Detonation time
C_d	Detonation velocity
B_s	Constant used to spread the detonation front (usually 2.5)
ℓ	Element characteristic length

The value of Burn Fraction is limited as follows:

For $t \le t_b$,

$$BF = 0 \tag{3.59}$$

For $t > t_b + B_s \ell / C_d$,

$$BF = 1 \tag{3.60}$$

Thus, the actual pressure (P) developing in the element is obtained by combining Equations 3.57 and 3.58. That is

$$P = BF \cdot \bar{P} \tag{3.61}$$

The following JWL model parameters represents the properties of typical ANFO:

A	20 GPa
B	0.2 GPa
R_1	3.7
R_2	0.9
ω	0.2
E	7.08 GPa (Initial chemical energy)
C_d	3048 m/s
ρ_o	830 Kg/m^3 (Initial density)

3.4 Modified Cam Clay Model with Creep

3.4.1 Introduction

Long term deformations of embankments on saturated clay soils are generally associated with consolidation and creep effects. Consolidation settlements are primarily due to the expulsion of pore water while creep strains are time-dependent deformations taking place under constant stress.

This section presents detailed derivation of constitutive relations when both plastic and creep strains develop. Modified Cam Clay Model is used to represent the elasto-plastic behavior of clay soils. In computing creep strains, averaging scheme is introduced such that volumetric scaling governs at low stress ratio and deviatoric scaling governs at high stress ratio.

In this section, the elasto-plastic matrix is derived for the Modified Cam Clay Model incorporating Singh-Mitchell creep equations.

3.4.2 Yield and Failure Equations

The failure surface (critical state line) of the Modified Cam Clay Model is given by

$$q = M P' \tag{3.62}$$

where M is the failure constant and P′ and q are the alternate stress invariants given by

$$\begin{aligned} P' &= \frac{1}{3} \sigma'_{ij} \delta_{ij} \\ q &= \sqrt{3J_2} \\ J_2 &= \frac{1}{2} S_{ij} S_{ij} \\ S_{ij} &= \sigma'_{ij} - P' \delta_{ij} \end{aligned} \tag{3.63}$$

The yield surface of the Modified Cam Clay Model is given by

$$F = \frac{q^2}{M^2} + P' (P' - P'_o) = 0 \tag{3.64}$$

where P_o' is the preconsolidation pressure which grows not only with plastic volumetric strain but also with time.

3.4.3 Elastic Stress-Strain Relationship

The elastic tangent bulk modulus is based on the recompression or swelling response in isotropic compression test.

$$B_k = \frac{2.3 (1 + e_o)}{C_r} P' \tag{3.65}$$

Where

e_o Initial void ratio

C_r Recompression or swelling index

Assuming the constant Poisson's ratio (υ), the elastic tangent shear modulus would be

$$G = 3.45 \frac{(1-2v)}{(1+v)} \frac{(1+e_o)}{C_r} p' \tag{3.66}$$

With these effective mean pressure dependent elastic tangent moduli, the incremental elastic constitutive law can be expressed in the following matrix form:

$$\{d\sigma'\} = [D^e] \{d\varepsilon^e\} \tag{3.67}$$

where

$\{d\sigma'\}$	Effective stress increment
$[D^e]$	Elastic stress-strain matrix
$\{d\varepsilon^e\}$	Elastic strain increment

3.4.4 Plastic Strain Increment

Plastic strain increment is assumed to be normal to the yield surface.

$$\{d\varepsilon^p\} = d\lambda_p \left\{ \frac{\partial F}{\partial \sigma'} \right\} \tag{3.68}$$

where

$\{d\varepsilon^p\}$	Plastic strain increment
$d\lambda_p$	Proportional constant for plastic strain
$\{\partial F/\partial \sigma'\}$	Derivative of yield surface with respect to stress

3.4.5 Creep Strain Increment

Creep strain increment is assumed to be normal to the equivalent yield surface.

$$\{d\varepsilon^c\} = d\lambda_c \left\{ \frac{\partial F_e}{\partial \sigma'} \right\} dt \quad (3.69)$$

where

$\{d\varepsilon^c\}$	Creep strain increment
$d\lambda_c$	Proportional constant for creep strain
$\{\partial F_e / \partial \sigma'\}$	Derivative of equivalent yield surface with respect to stress
dt	Time increment

Note that the equivalent yield surface is defined as

$$F_e = \frac{q^2}{M^2} + P'(P' - P'_e) = 0 \quad (3.70)$$

where

$$P'_e = \frac{q^2}{M^2 P'} + P' \quad (3.71)$$

The proportional constant for creep strain can be computed by averaging both volumetric and deviatoric scaling.

$$d\lambda_c = \frac{\eta}{M} d\lambda_d + \left(1 - \frac{\eta}{M}\right) d\lambda_v \quad (3.72)$$

where

η	Stress ratio (q/p')
$d\lambda_v$	Volumetric scaling factor
$d\lambda_d$	Deviatoric scaling factor

The volumetric scaling factor is based on the secondary consolidation curve.

$$d\lambda_v = \frac{C_\alpha}{2.3\,(1+e_o)\;t_v}\left(\frac{\partial F_e}{\partial P^{/}}\right)^{-1} \tag{3.73}$$

The volumetric age (t_v) in Equation 3.73 is given by

$$t_v = t_{vi}\left(\frac{P_o^{/}}{P_e^{/}}\right)^{\frac{C_c - C_r}{C_\alpha}} \tag{3.74}$$

where

t_{vi}	Reference volumetric time
C_c	Virgin compression index
C_α	Secondary compression coefficient

The deviatoric scaling factor is based on Singh-Mitchell creep equation (1968).

$$d\lambda_d = \sqrt{\frac{3}{2}}\;A e^{\alpha\eta}\left(\frac{t_{di}}{t}\right)^m\left(\frac{\partial F_e}{\partial\sigma_{ij}^{/}}\frac{\partial F_e}{\partial\sigma_{ij}^{/}} - \frac{1}{3}\frac{\partial F_e}{\partial P^{/}}\right)^{-1/2} \tag{3.75}$$

Where

t_{di}	Reference deviatoric time
t	Current time
A	Singh-Mitchell creep parameter
α	Singh-Mitchell creep parameter
m	Singh-Mitchell creep parameter

3.4.6 Total Strain Increment

Total strain increment consists of elastic, plastic and creep strains.

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^p\} + \{d\varepsilon^c\} \tag{3.76}$$

3.4.7 Consistency Equation

During the subsequent yielding, the equation forces the stress increment to move on the subsequent yield loci.

$$dF = \left\{\frac{\partial F}{\partial \sigma'}\right\}^T \{d\sigma'\} + \frac{\partial F}{\partial P_o'} dP_o' = 0 \tag{3.77}$$

The preconsolidation pressure increment (dP_o') is related to the plastic volumetric strain increment ($d\varepsilon_v^p$) and the time increment (dt).

$$dP_o' = \frac{2.3\,(1 + e_o)}{(c_c - c_r)} p_o' \, d\varepsilon_v^p + \frac{c_\alpha}{(c_c - c_r)} \frac{p_o'}{t_v} dt \tag{3.78}$$

From Equation 3.68 the plastic volumetric strain increment ($d\varepsilon_v^p$) can be expressed in terms of $d\lambda_p$

$$d\varepsilon_v^p = d\lambda_p \frac{\partial F}{\partial P'} \tag{3.79}$$

3.4.8 Evaluation of $d\lambda_p$

The elastic strain increment in Equation 3.67 can be expressed in terms of $d\lambda_p$ by combining Equations 3.76 and 3.67.

$$\{d\sigma'\} = [D^e]\left(\{d\varepsilon\} - d\lambda_p \left\{\frac{\partial F}{\partial \sigma'}\right\} - \{d\varepsilon^c\}\right) \tag{3.80}$$

Substituting Equations 3.78, 3.79 and 3.80 into the Consistency Equation 3.77 and solving for $d\lambda_p$.

$$d\lambda_p = \frac{\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e] (\{d\varepsilon\} - \{d\varepsilon^c\}) + p_n}{\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e] \left\{\frac{\partial F}{\partial \sigma'}\right\} - p_d} \tag{3.81}$$

Where

$$P_n = \frac{\partial F}{\partial P'_o} \frac{P'_o}{t_v} \frac{c_\alpha}{(c_c - c_r)} dt$$

$$P_d = \frac{\partial F}{\partial P'_o} \frac{\partial F}{\partial P'} \frac{(1 + e_o)}{2.3(c_c - c_r)} P'_o$$

3.4.9 Effective Stress Increment

The effective stress increment can be obtained by backsubstituting Equation 3.81 into 3.80.

$$\{d\sigma'\} = [D^{ep}] \{d\varepsilon\} - \{d\sigma'_c\} \tag{3.82}$$

where the incremental elasto-plastic matrix is expressed as

$$[D^{ep}] = [D^e] - \frac{[D^e] \left\{\frac{\partial F}{\partial \sigma'}\right\} \left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e]}{\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e] \left\{\frac{\partial F}{\partial \sigma'}\right\} - P_d} \tag{3.83}$$

and the stress increment associated with creep is given by

$$\{d\sigma_c'\} = [D^{ep}]\{d\varepsilon^c\} + \frac{[D^e]\left\{\frac{\partial F}{\partial \sigma'}\right\} P_n}{\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e]\left\{\frac{\partial F}{\partial \sigma'}\right\} - P_d} \qquad (3.84)$$

3.4.10 Evaluation of Derivatives

$$\frac{\partial F}{\partial P'} = 2P' - P_o'$$

$$\left\{\frac{\partial P'}{\partial \sigma'}\right\} = \frac{1}{3} <1\ 1\ 1\ 0>^T$$

$$\frac{\partial F}{\partial q} = \frac{2q}{M^2}$$

$$\left\{\frac{\partial q}{\partial \sigma'}\right\} = \frac{3}{2q}\{S_{ij}\}$$

$$\frac{\partial F}{\partial P_o'} = -P'$$

3.5 Engineering model

3.5.1 Introduction

The Engineering Model is hypoelastic-perfectly plastic in shear and hypoelastic in compression. A hypoelastic material is one for which the stress increments are homogeneous linear functions of the strain increments. In general, the coefficients in the linear functions depend on the stress. The principal advantages of the Engineering Model are ease of fitting to laboratory or in situ test data, simplicity of shear plasticity formulation, and the simple form of compressive hysteresis, which most soils exhibit. Its principal disadvantages are lack of hysteresis in pure shear at constant volume below the failure surface, and lack of dilatancy because the plastic strain increments are assumed to be normal to the hydrostatic axis. The Engineering Model is completely described by a pressure-volume strain curve for hydrostatic compression and a two-invariant failure surface.

3.5.2 Hydrostatic Response

The hydrostatic response is represented by the incremental elastic (hypoelastic) bulk modulus as a function of current compressive volumetric strain (ε_v), maximum past compressive volumetric strain (ε_{vm}) and compressive volumetric strain increment ($d\varepsilon_v$) as shown in Figure 3.5a.

$$K = K(\varepsilon_v, \varepsilon_{vm}, d\varepsilon_v) \tag{3.85}$$

Poisson's ratio is also defined for each hydrostat segment.

$$\upsilon = \upsilon(\varepsilon_v, \varepsilon_{vm}, d\varepsilon_v) \tag{3.86}$$

The corresponding hypoelastic constrained compression and shear moduli are then computed from the following expressions respectively:

$$M = \frac{3K(1 - \upsilon)}{(1 + \upsilon)} \tag{3.87}$$

and

$$G = \frac{3K(1 - 2\upsilon)}{2(1 + \upsilon)} \tag{3.88}$$

3.5.3 Plastic Shear Response

The failure surface is composed of three segments of conical surfaces as shown in Figure 3.5b, each having an equation of the form:

$$f(I_1, \sqrt{J_2}) = \sqrt{J_2} - (a + bI_1) = 0 \tag{3.89}$$

The material is assumed to behave incrementally elastically when the stress point lies below the failure surface. When the stress point moves along the failure surface, the material response is assumed to be hypoelastic-perfectly plastic in shear. The plastic strain increments are assumed to be normal to the hydrostatic axis so that there would be no plastic volume changes associated with the failure surface.

The derivation of elasto-plastic stress-strain matrix $[D_{ep}]$ is given by Merkle and Dass (1985).

3.5.4 Parameter Determination

The Engineering Model parameters can be obtained by fitting a series of straight lines to shear strength, hydrostatic compression, and constrained compression or K_0 test data.

The parameters of shear strength envelope shown in Figure 3.5b can be determined by fitting to shear strength data in drained triaxial compression tests which arc expressed as functions of I_1, and $\sqrt{J_2}$.

Note that the values of I_1 and $\sqrt{J_2}$ at the failure points of triaxial compression are computed in terms of σ_{af} and σ_r.

$$I_1 = \sigma_{af} + 2\sigma_r \tag{3.90}$$

and

$$\sqrt{J_2} = \frac{|\sigma_{af} - \sigma_r|}{\sqrt{3}} \qquad (3.91)$$

where σ_{af} is the axial stress at failure and σ_r is the confining stress.

The hypoelastic bulk modulus and Poisson's ratio in Equations 3.85 and 3.86 respectively can be determined from hydrostatic and constrained compression tests. Then the Poisson's ratio is computed from Equation 3.87.

$$\upsilon = \frac{3K - M}{3K + M} \qquad (3.92)$$

When K_o test data are available from constrained compression tests conducted in a triaxial cell, in which the confining stress is measured, Poisson's ratio can be directly computed from:

$$\upsilon = \frac{K_o}{1 + K_o} \qquad (3.93)$$

where

$$K_o = \frac{d\sigma_r}{d\sigma_a} \qquad (3.94)$$

When only uniaxial compression test data are available, it is customary to assume Poisson's ratio and to compute bulk modulus from

$$K = \frac{M(1 + \upsilon)}{3(1 - \upsilon)} \qquad (3.95)$$

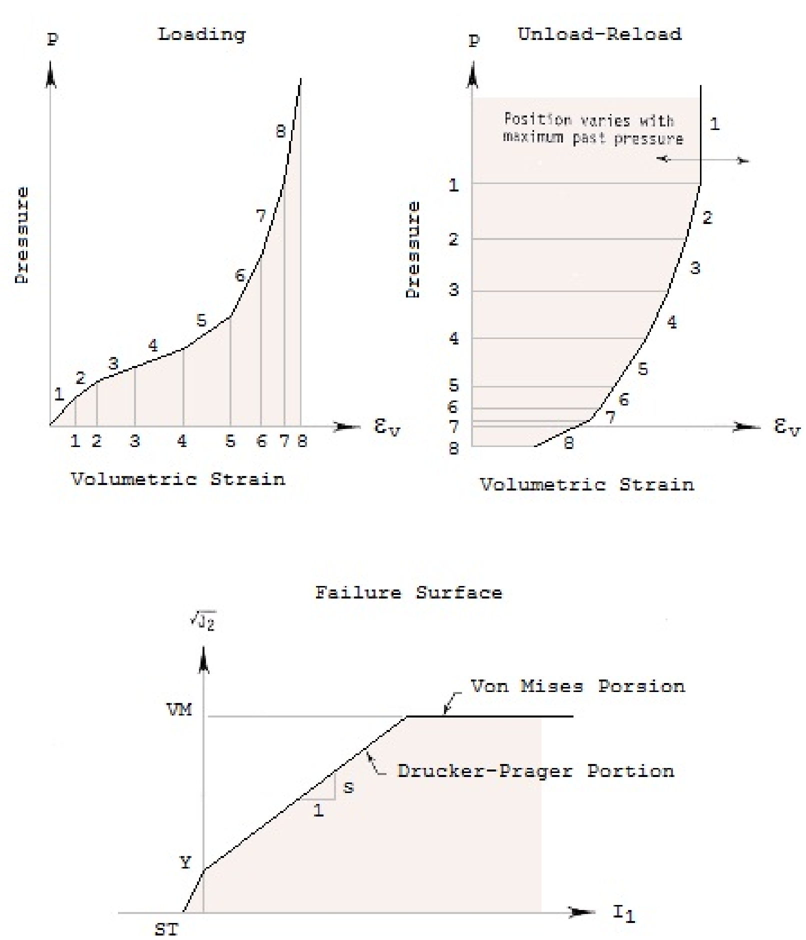


Figure 3.5 Engineering Model

3.6 Joint Model

3.6.1 Introduction

Joint Model is often used to represent rock joints, faults, and interfaces. Along the joint face, slipping takes place when the shear stress exceeds shear strength and debonding occurs when adjacent two blocks are not in contact.

Joint Model is to be used as a material model for the joint element (see Figure 3.6) as described in Card 5.4 of SMAP-2D User's Manual.

3.6.2 Strain-Displacement Relation

Strains in the joint local coordinate are

$$\{\Delta\varepsilon\} = \begin{Bmatrix} \Delta\gamma_{xy}' \\ \Delta\varepsilon_{yy}' \end{Bmatrix} \tag{3.96}$$

where

$\Delta\gamma_{xy}'$ Shear strain increment

$\Delta\varepsilon_{yy}'$ Normal strain increment

Local displacement increment, $\{\Delta u'\}$, is related to the global displacement increment, $\{\Delta u\}$, as follows:

$$\{\Delta u'\} = [\beta]\,[\Delta u] \tag{3.97}$$

where

$$\{\Delta u'\} = \begin{Bmatrix} \Delta u_x' \\ \Delta u_y' \end{Bmatrix} \qquad \{\Delta u\} = \begin{Bmatrix} \Delta u_x \\ \Delta u_y \end{Bmatrix}$$

$[\,\beta\,]$ Coordinate transformation matrix

Strain-displacement relation in the local coordinate is given by

$$\{\Delta\epsilon'\} = \frac{1}{\delta}\{\Delta u'\} \tag{3.98}$$

where δ is the thickness of joint. And global displacement increment can be expressed in terms of global nodal displacement increment, $\{\Delta\bar{u}\}$, using the shape function matrix, [h], as

$$\{\Delta u\} = [h]\{\Delta\bar{u}\} \tag{3.99}$$

Now, Substituting Equations 3.97 and 3.99 into the Equation 3.98, we obtain

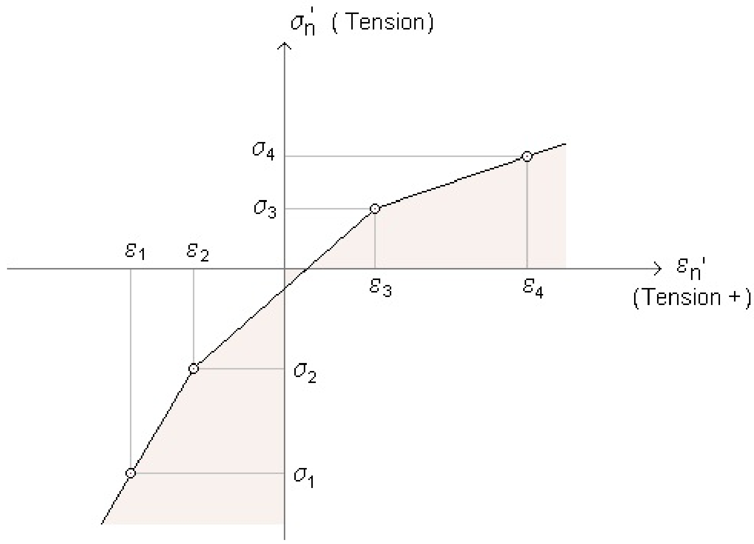
$$\{\Delta\epsilon'\} = [B]\{\Delta\bar{u}\} \tag{3.100}$$

where

$$[B] = \frac{1}{\delta}[\beta][h] \tag{3.101}$$

3.6.3 Normal Stress-Strain Relation

Normal Stress-strain relation is assumed to be nonlinear elastic as shown below



Thus, Young's modulus (E) is computed as follows:

For $\varepsilon_n' < \varepsilon_2$

$$E = \frac{\sigma_2 - \sigma_1}{\varepsilon_2 - \varepsilon_1}$$

For $\varepsilon_2 \leq \varepsilon_n' < \varepsilon_3$

$$E = \frac{\sigma_3 - \sigma_2}{\varepsilon_3 - \varepsilon_2} \tag{3.102}$$

For $\varepsilon_n' \geq \varepsilon_3$

$$E = \frac{\sigma_4 - \sigma_3}{\varepsilon_4 - \varepsilon_3}$$

3.6.4 Shear Stress-Strain Relation

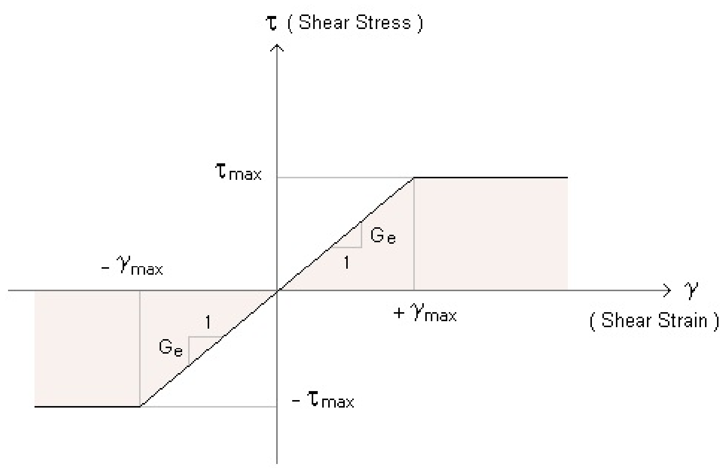
The shear strength of joint is assumed to follow Mohr-Coulomb failure criterion.

$$\tau_{max} = C - \sigma_n' \tan \varphi \qquad (3.103)$$

where

τ_{max}	Maximum shear stress
C	Cohesion
φ	Friction angle
σ_n'	Normal stress (Tension is positive)

Shear stress-strain relation is assumed to be elastic below the strength envelope and perfectly plastic along strength envelope as shown below:



Thus, shear modulus (G) is computed as follow:

$$\text{For } |\gamma| < \gamma_{max} \quad G = G_e \qquad \text{For } |\gamma| \geq \gamma_{max} \quad G = 0 \qquad (3.104)$$

Note that G_e is the elastic shear modulus.

3.6.5 Element Stiffness Matrix

Joint stress-strain relation can be given by

$$\{\Delta\sigma'\} = [C'] \{\Delta\varepsilon'\} \tag{3.105}$$

where

$$\{\Delta\sigma'\} = \begin{Bmatrix} \Delta\tau'_{xy} \\ \Delta\sigma'_{yy} \end{Bmatrix}$$

$$[C'] = \begin{bmatrix} G & 0 \\ 0 & E \end{bmatrix}$$

Note that both volumetric and shear responses are assumed to be decoupled.

Following element stiffness matrix, [K], can be derived using the principle of virtual work:

$$[K] = \int_v [B]^T [C'] [B] \, dv \tag{3.106}$$

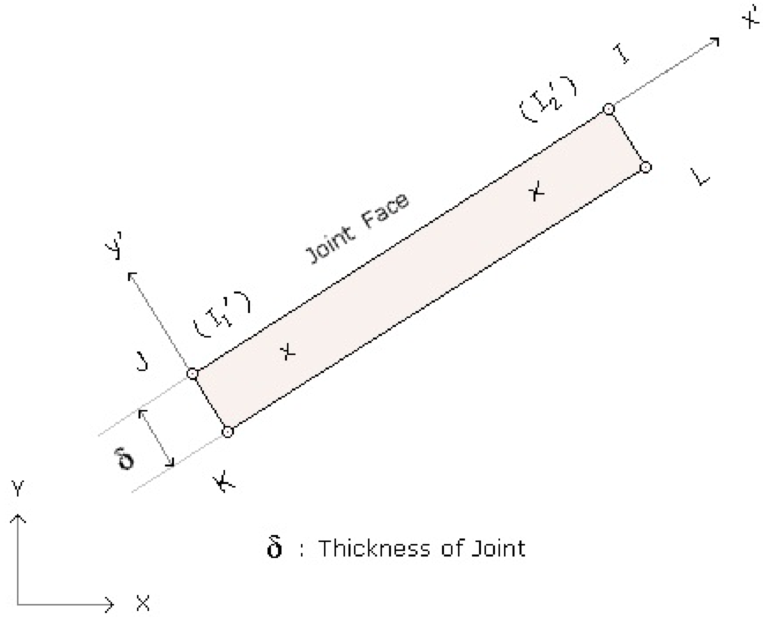


Figure 3.6 Local and Global Coordinates of Joint Element

3.7 Generalized Decoupled Hyperbolic Model

3.7.1 Introduction

GDHM (Generalized Decoupled Hyperbolic Model) is the decoupled material model which is the generalized form of the original hyperbolic model (Duncan and Chang, 1970). Main features of GDHM include:

- Hoek and Brown in situ rock strength model and
- Strength envelope expressed as a function of Lode angle on octahedral plane

3.7.2 Stress-Strain Relation

It is assumed that volumetric behavior is not coupled with deviatoric behavior. Thus, we have two independent equations:

$$dp = K \cdot d\varepsilon_v \qquad (3.107)$$

$$d\tau_{oct} = 2 \cdot G \cdot d\gamma_{oct}$$

where

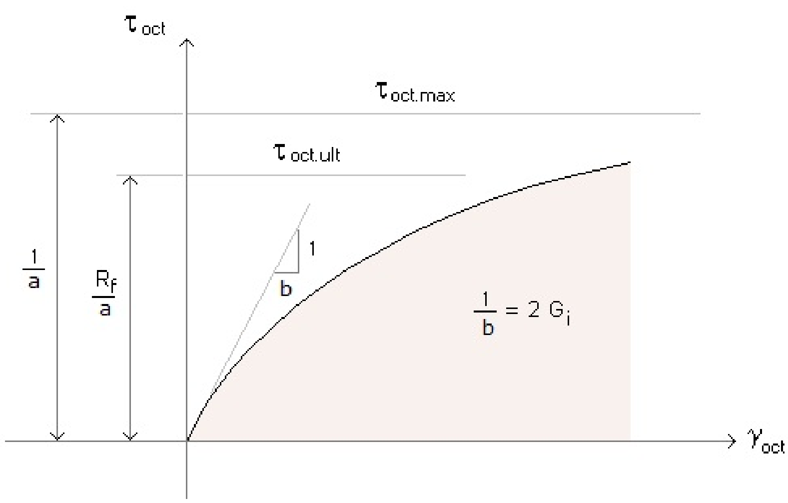
p	Mean pressure
ε_v	Volumetric strain
τ_{oct}	Octahedral shear stress
γ_{oct}	Octahedral shear strain
K	Bulk modulus
G	Shear modulus

At constant mean pressure and constant Lode angle, the shear stress-strain relation is assumed to be hyperbolic.

That is

$$\tau_{oct} = \frac{\gamma_{oct}}{b + a\,\gamma_{oct}} \qquad (3.108)$$

As shown in the following figure, τ_{oct} approaches to the maximum shear stress, $\tau_{oct.max}$, as γ_{oct} goes to infinity. And the slope of the Equation 3.109 at γ_{oct} = 0 is equal to 1/b. Most triaxial compression tests, however, show that ultimate shear strength, $\tau_{oct.ult}$, is reached at finite value of γ_{oct} and $\tau_{oct.ult}$ is approximately 70~90% of $\tau_{oct.max}$.



That is,

$$\frac{1}{a} = \tau_{oct.max} = \frac{1}{R_f}\tau_{oct.ult} \qquad (3.109)$$

$$\frac{1}{b} = 2\ G_i \qquad (3.110)$$

where

R_f	Material constant (0.7~0.9)
G_i	Initial shear modulus

Differentiating Equation 3.109 with respect to γ_{oct},

$$\frac{d\tau_{oct}}{d\gamma_{oct}} = \frac{b}{(b + a\gamma_{oct})^2} \tag{3.111}$$

Solving for γ_{oct} from Equation 3.108,

$$\gamma_{oct} = \frac{b\,\tau_{oct}}{(1 - a\,\tau_{oct})} \tag{3.112}$$

Now, substituting Equation 3.113 into 3.112, we obtain the following loading shear modulus:

$$G = G_i\left(1 - \frac{\tau_{oct}}{\left(\frac{1}{a}\right)}\right)^2 \tag{3.113}$$

3.7.3 Shear Strength Equation

Kim (1984) presented the following generalized form of ultimate shear strength equation:

$$\tau_{oct.ult} = \frac{\sqrt{2}}{3}\,[(\alpha + \beta\,p)^n + \kappa]\;R(\theta) \tag{3.114}$$

The strength parameters (n, α, β, κ) are tabulated in Table 3.1 and $R(\theta)$ is given by the Equation 3.4. Note that the strength equation represents Von Mises Model when n =1, Hoek and Brown In Situ Rock Model when n=1/2, and Mohr-Coulomb or Drucker-Prager Model when n=1.

Substituting Equation 3.110 into 3.115,

$$\frac{1}{a} = \frac{1}{R_f}\,\frac{\sqrt{2}}{3}\,[(\alpha + \beta p)^n + \kappa]\;R(\theta) \tag{3.115}$$

Now, combining Equations 3.114, 3.115 and 3.116, the generalized loading shear modulus, G_i is given by

$$G = G_i \left[1 - \frac{R_f \, \tau_{oct}}{\tau_{oct.ult}} \right]^2 \qquad (3.116)$$

The initial shear modulus, G_i, in Equation 3.117 may be obtained from the following empirical equations:

For cohesive soil (Hardin and Black, 1968)

$$G_i = 1230 \frac{(2.973 - e)^2}{(1 + e)} \bar{\sigma}_{oct}^{1/2} \; OCR^K \qquad (3.117)$$

where

e	Void ratio
OCR	Overconsolidation ratio
K	Parameter expressed as a function of plasticity index. Note that stress unit is psi

For cohesionless soil (Janbu, 1963)

$$G_i = \frac{1}{2(1 + \nu)} K_{ur} \, P_a \left(\frac{\sigma_3}{P_a} \right)^n \qquad (3.118)$$

where

ν	Poisson's ratio
P_a	Atmospheric pressure
K_{ur}, n	Material constants
σ_3	Confining pressure

It should be noted that Equation 3.117 can be degenerated to the original Duncan and Chang's hyperbolic model when Mohr-Coulomb strength envelope is specified along the triaxial compression mode.

That is,

$$\tau_{oct} = \frac{\sqrt{2}}{3}(\sigma_1 - \sigma_3) \tag{3.119}$$

$$\tau_{oct.ult} = \frac{\sqrt{2}}{3}(\sigma_1 - \sigma_3)_{ult}$$

$$= \frac{6\ \text{Sin}\varphi}{(3 - \text{Sin}\varphi)} P + \frac{6\ \text{Cos}\varphi}{(3 - \text{Sin}\varphi)} C \tag{3.120}$$

where

$$P = \frac{1}{3}(\sigma_1 + 2\ \sigma_3) \tag{3.121}$$

Substituting Equation 3.122 into 3.121 and solving for σ_1, we obtain

$$\sigma_1 = \frac{(1 + \text{Sin}\varphi)}{(1 - \text{Sin}\varphi)}\sigma_3 + \frac{2\ \text{Cos}\varphi}{(1 - \text{Sin}\varphi)} C \tag{3.122}$$

Backsubstituting σ_1 in Equation 3.123 into the right hand side of Equation 3.121,

$$\tau_{oct.ult} = \frac{\sqrt{2}}{3}\ \frac{(2\ \text{Sin}\varphi\ \sigma_3 + 2\ \text{Cos}\varphi\ C)}{(1 - \text{Sin}\varphi)} \tag{3.123}$$

Now, substituting Equations 3.120 and 3.124 into 3.117, we can obtain the following Duncan and Chang Hyperbolic Model (1970):

$$G = G_i \left[1 - \frac{R_f\ (1 - \text{Sin}\varphi)\ (\sigma_1 - \sigma_3)}{2\ \text{Sin}\varphi\ \sigma_3 + 2\ \text{Cos}\varphi\ C} \right]^2 \tag{3.124}$$

References

Ahrens, T.J., Equation of State of Earth Media, Report to Defense Nuclear Agency, DNA-TR-88-265, Washington, D.C. November 1988.

Allen, R.T., Equation of State of Rocks and Minerals, Defense Atomic Support Agency, DA49-146-XZ-462, Project No. 532, March 1967.

Bakanova, A.A., V.N. Zubarev, Y.N. Sutulov, and R.F. Trunin, Thermodynamic Properties of Water at High Pressures and Temperatures, Soviet Phys. JETP, 41, 544, 1976.

Biot, M.A., Theory of Propagation of Elastic Waves in Fluid Saturated Porous Solid. I, II, Journal of Acoustical Society of America, Vol. 28, pp 168-191, 1956.

Biot, M.A., Mechanics of Deformation and Acoustic Propagtion in Porous media, Journal of Applied Physics, Vol. 33, pp 1482-1498, 1962A.

Biot, M.A., Generalized Thoery of Acoustic Propagation in Porous Dissipative Media, Journal of Acoustical Society of America, Vol. 34, pp 1254-1264, 1962B.

Blouin, S.E., and K.J. Kim, Undrained Compressibility of Saturated Soil, DNA-TR-87-42, Defense Nuclear Agency, Washington, D.C., February 1984.

Blouin, S.E., D.E. Chitty, A.F. Rauch, and K.J. Kim, Dynamic Response of Multiphase Porous Media, Annual Technical Report 1, Report to U.S. Air Force Office of Scientific Research, Washington D.C., March 1990.

Blouin, S.E., D.E. Chitty, A.F. Rauch, and K.J. Kim, Dynamic Response of Multiphase Porous Media, Annual Technical Report 2, Report to U.S. Air Force Office of Scientific Research, Washington D.C., June 1991

Borja, Ronaldo I., Generalized Creep and Stress Relaxation Model for Clays, Journal of Geotechnical Engineering, Vol. 118, No. 11, November 1992.

Britt, J.R., Behavior of Water in Low Pressure Region, unpublished manuscript, 1985.

Dass, W.C. and D.H. Merkle, Computational Aspects of the ARA Three Invariant Constitutive Model, Report to U.S. Air Force Office of Scientific Research Washington, D.C., May 1986.

D avid E. Van Dillen, etc., Modernization of the BMINES Computer Code Vo. I: User's Guide, Agbabian Associates, Sep. 1981.

Duncan, J.M., and C. Y. Chang, Nonlinear Analysis of Stress and Strain in Soils, J. Soil Mech. Found. Div. ASCE, vol. 96, no. SM5, pp. 1629-1653, September, 1970.

E.L.Lee, H.C.Hornig, and J.K.Kury, Adiabatic Expansion of High Explosive Detonation Products, UCRL-50422, Lawrence Livermore National Laboratory, California, May 2, 1968.

Hardin, B.O. and W.L Black, Vibration Modulus of Normally Consolidated Clays, J. Soil Mech. Found. Div., ASCE, vol. 94, no. SM2, pp. 353-369, March 1968.

Forchheimer, P., Wasserbewegung durch Boden, Z. Ver. Deutsch Ing., 45 pp. 1782-1788, 1901.

Ghaboussi, J., and E. L. Wilson, Variational Formulation of Dynamics of Fluid-Saturated Porous Elastic Solids, Journal of the Engineering Mechanics Division, ASCE, Vol. 98, pp 947-963, 1972.

Gibson, R.E., The Progress of Consolidation in a Clay Layer Increasing in Thickness with Time, Geotechnique, Vol. 8, No. 4, pp. 171-182.

Hoek, E. and E.T. Brown, Underground Excavations in Rock, The Institution of Mining and Metallurgy, London, England, 1982.

Janbu, Nilmar, Soil Compressibility as Determined by Oedometer and Triaxial Tests, Proc. Eur. Conf. Soil Mech. Found. Eng., Wiesbaden, 1963, vol. 1, pp. 19-25.

Karshenas, M. and J. Ghaboussi, Modeling and Finite Element Analysis of Soil Behavior, Civil Engineering Studies, Geotechnical Research Series No. 17, University of Illinois, Urbana, 1979.

Kim, K. J., Finite Element Analysis of Nonlinear Consolidation, Ph.D. Thesis, University of Illinois at Urbana-Champaign, 1982.

Kim, K.J. and S.E. Blouin, Response of Saturated Parous Nonlinear Materials to Dynamic Loadings, Report to Air Force Office of Scientific Research, Washington, DC, F49620-81-C-0014 (May 1984).

Kim, K.J., S.E. Blouin, and D.A. Timian, Experimental and Theoretical Response of Multiphase Porous Media to Dynamic Loads, Annual Report No. 1 to Air Force Office of Scientific Research, Washington, D.C., 1986.

Kim, K.J., S.E. Blouin, and D.A. Timian, Experimental and Theoretical Response of Multiphase Porous Media to Dynamic Loads, Annual Report No. 2 to Air Force Office of Scientific Research, Washington, D.C., 1987.

Kim, K.J., S.E. Blouin, D.E. Chitty, and D.H. Merkle, Experimental and Theoretical Response of Multiphase Porous Media to Dynamic Loads, Final Report to Air Force Office of Scientific Research, Washington, D.C., 1988.

Kim, M.K. and P.V. Lade, Single Hardening Constitutive Model for Frictional Materials, 1. Plastic Potential Function, Computers and Geotechnics, 5(4), 307- 324, 1988.

Lade, P.V., Elasto-plastic Stress-Strain Theory for Cohesionless Soil with Curved Yield Surfaces, Int. J. Solids Struct., 13, 1019-1035, 1977.

Lade, P.V. and R.B. Nelson, Modelling the Elastic Behavior of Granular Materials, International Journal for Numerical and analytical Methods in Geomechanics, Vol. II, pp.521-542, 1987.

Lade, P.V., Single-Hardening Model with Application to NC Clay, J. Geotech. Engrg., ASCE, 116 (3), 394-414, 1990.

Lade, P.V. and M.K. Kim, Single Hardening Constitutive Model for Frictional Materials, III. Comparisons with Experimental Data, Computers and Geotechnics, 6(1), 30-47. 1988b.

Mendelson, A., Plasticity: Theory and Application, The MacMillan Company, New York (1968).

Mengi, Y., and H.D. McNiven, Fluid-filled Porous Media to a Transient Input, Journal of Acoustical Society of America, Vol. 61, pp 84-94. 1977.

Merkle, D.H. and W.C. Dass, Fundamental Properties of Soils for Complex Dynamic Loadings: Development of a Three Invariant Constitutive Model, Report to the U.S. Air Force Office of Scientific Research, Washington, D.C.,April 1985.

Piepenburg, D.D., K.J. Kim and M.D. Davister, Numerical Analysis of Nonlinear Liner-Medium Interaction. Tunnels Subjected to Biaxial Loading Vol. III, Technical Report to Defense Nuclear Agency, Washington, D.C., DNA-TR-86-138-V3, December 1986.

Rischbieter, F., et al., Studies of Soil Liquefaction by Shock Wave Loading, Fifth International Symposium on Military applications of Blast Simulation, Vol. III, Royal Swedish Fortifications Administration, Stockholm, Sweden, May 1977.

Singh, Awtar and James K. Mitchel, Generalized Stress-Strain-Time Functions for Soils, Journal of the Soil Mechanics and Foundation Division, ASCE, 21-46, 1968.