SMAP ® - S2

Structure Medium Analysis Program

2-D Static Analysis

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-S2 is an advanced two-dimensional, static, finite element computer program developed for the geometric and material nonlinear structure-medium interaction analysis. The program has specific applications for modeling geomechanical problems associated with multi-staged excavation or embankment. 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-S2 include:

- Two-dimensional plane strain, plane stress or axisymmetric isoparametric continuum element.
  - Models soils, rocks and concrete media
  - Allows yielding and tension cut-off
- 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
- Concentrated, distributed and gravity loads
- Simulation of a sequence of excavation and construction
- Nonlinear material model
  - Von Mises model
  - Mohr-Coulomb model
  - In Situ Rock model
- Large deformations
  - Use updated Lagrangian

1.3 Applications

Applications of SMAP-S2 include:

- 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.
  - NATM (New Austrian Tunneling Method) analysis.
- Shallow and deep foundation analysis
- Slope stability analysis
- Framed structural analysis

Overview of SMAP-S2 Program Structure

USER INPUT User prepares Mesh, Main, and Post Files according to SMAP-S2 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-S2 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 -S2

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

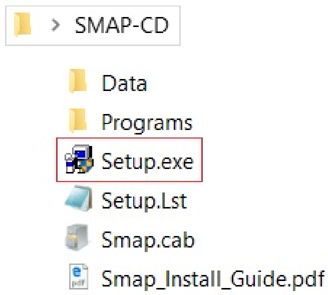
2.2 Installation Procedure

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

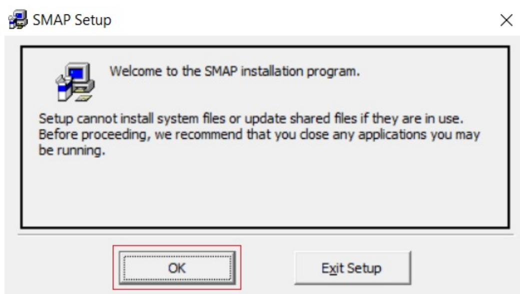
2. Download SMAP-CD.exe from the Download section of www.ComtecResearch.com

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

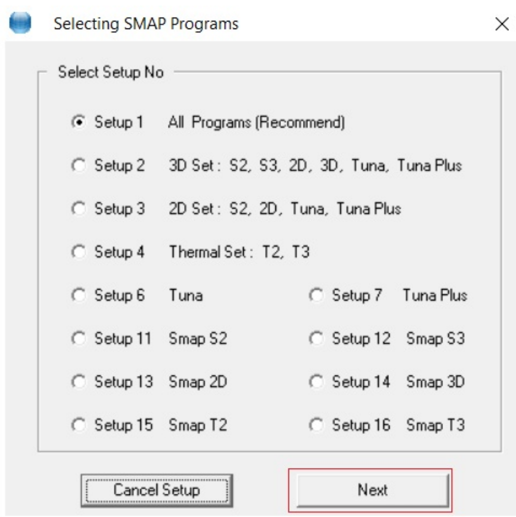
4. Double-click **Setup.exe**



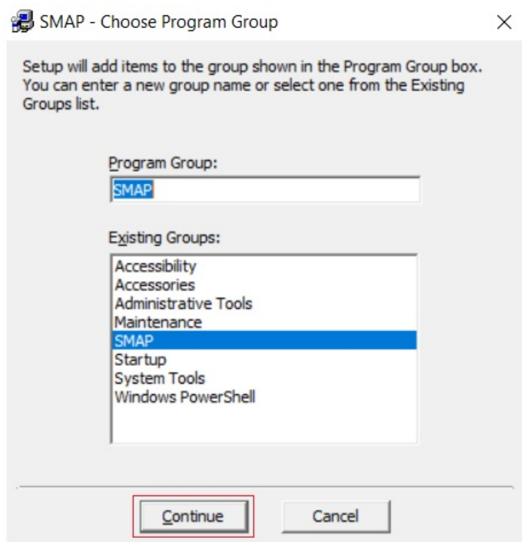
5. Click **OK**



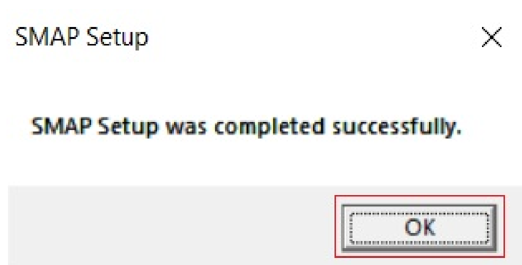
6. Click **Next**
   It will take few minutes.
   Wait until next step.



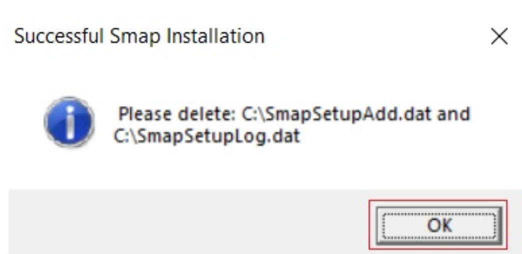
7. Click **Continue**



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

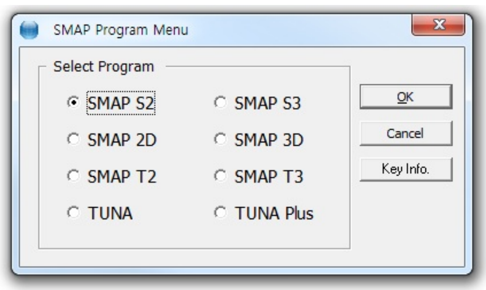
3.1 Introduction

Generally, SMAP-S2 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-S2 is the one which computes static response of two-dimensional problems. Post-processing programs are used to show graphically the results from the main-processing program.

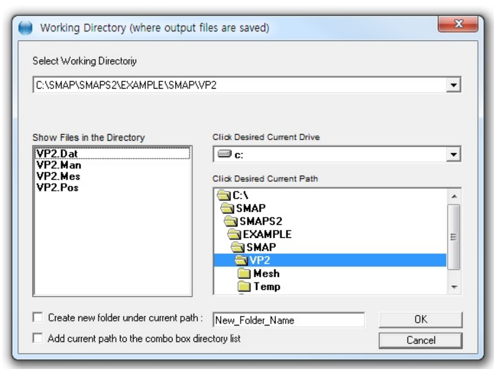
Accessing SMAP-S2 Programs

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-S2** radio button and then click **OK** button.

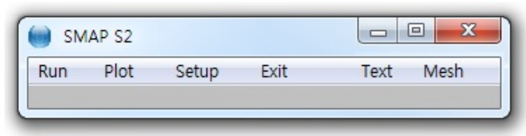


3. Next, you need to select **Working Directory**. Working Directory should be the existing directory where all the output files are saved. It is a good idea to have all your input files for the current project in this Working Directory. Click the disk drive, double-click the directory, and then **OK**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

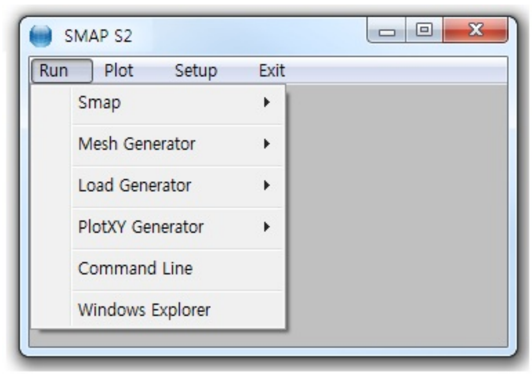


SMAP-S2 Menu

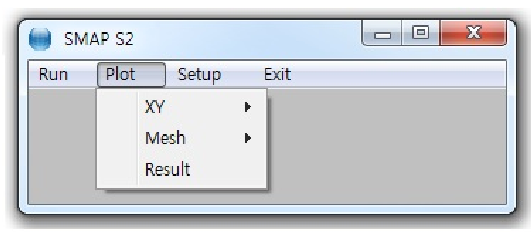
SMAP-S2 provides following Main Menus; Run, Plot, Setup, Exit, Text and Mesh.



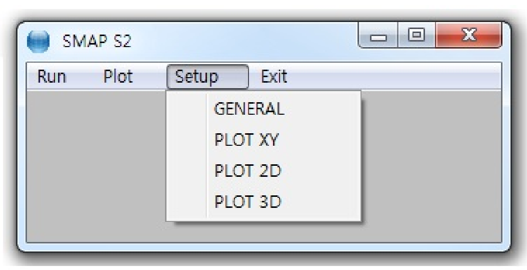
RUN executes main- and pre-processing programs and has following Sub Menus;
Smap,
Mesh Generator,
Load Generator,
PlotXY Generator,
Command Line and
Windows Explorer.



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-S2.

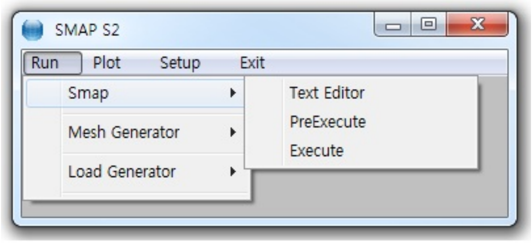
TEXT is used to edit Text files. **MESH** is used to plot F. E. Mesh files.

3.2 RUN Menu

3.2.1 SMAP

Once you have prepared the input files (Mesh, Main, and Post) according to the SMAP-S2 User's Manual in Section 4, you are ready to execute SMAP-S2 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-S2 main-processing program.

SMAP-S2 Output Files

Once you execute SMAP-S2, 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-S2 Graphical Output

SMAP-S2 Post-processing programs can generate the following graphical output:

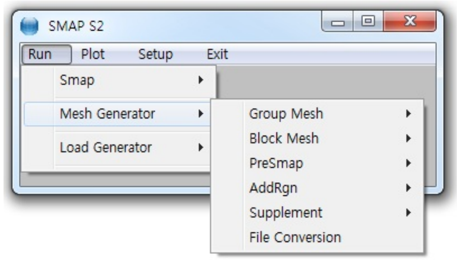
- Finite element mesh
- Deformed shape
- Principal stress distribution
- Section forces in beam element
- Extreme fiber stresses/strains in beam elements
- Axial force/stress/strain in truss element
- Contours of stresses, strains and factor of safety
- 3D iso surface of stresses and strains
- Time histories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

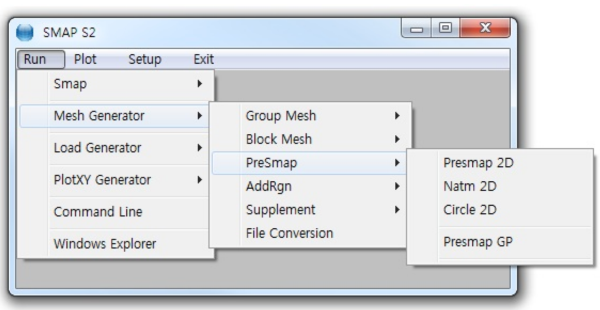
MESH GENERATOR Menu has the following Sub Menus; Group Mesh, Block Mesh, PreSmap, AddRgn, Supplement, and File Conversion.



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-S2 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-S2 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-S2 Example Problems describes in detail about running PRESMAP Programs.



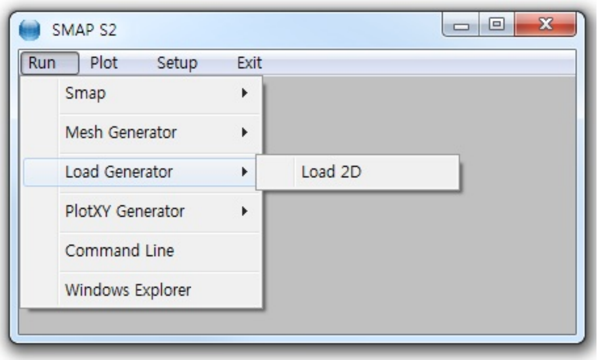
ADDRGN is the pre-processing program which has the following two basic functions: Combine two different meshes and modify existing meshes. Section 8 in SMAP-S2 Example Problems describes in detail about running ADDRGN programs.

SUPPLEMENT contains supporting programs which are useful to prepare input data for pre- and main-processing programs. Section 9 in SMAP-S2 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-S2 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. Section 10 in SMAP-S2 Example Problems describes in detail about running **LOAD-2D** program.



3.2.4 PlotXY GENERATOR

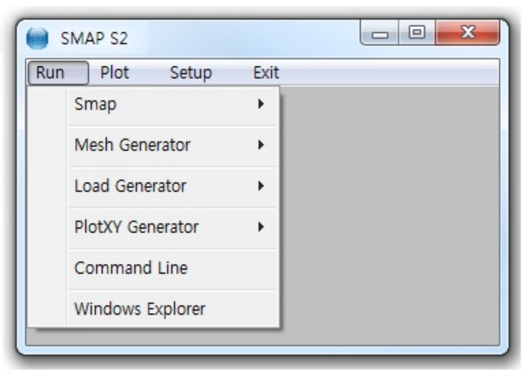
PlotXY GENERATOR is the graphical user interface which is mainly used to generate or edit Simplified Time History and Simplified Snapshot of Card Group 12 in SMAP Post File. Section 12.7 in SMAP-S2 User's Manual describes in detail about running PlotXY Generator program.

3.2.5 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.

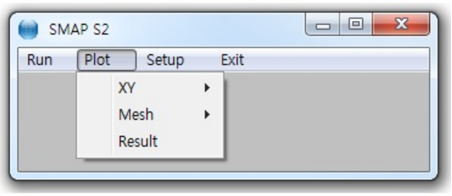
3.2.6 WINDOWS EXPLORER

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.



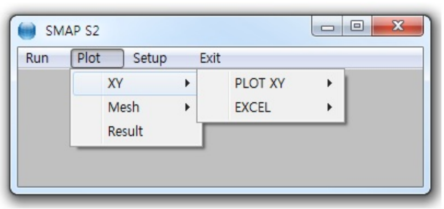
3.3 PLOT Menu

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



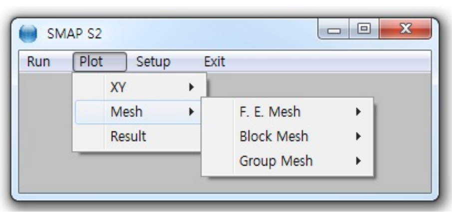
3.3.1 XY

XY graph can be displayed by PLOT-XY or EXCEL. Section 11 in SMAP-S2 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.

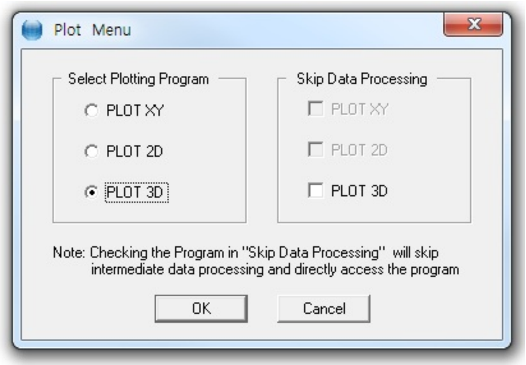
Block Mesh is used to open or build Block Mesh. Section 6 in SMAP-S2 Example Problems describes in detail about running Block Mesh.

Group Mesh is used to open or build Group Mesh. Section 5 in SMAP-S2 Example Problems describes in detail about running Group Mesh.

3.3.3 RESULT

Once you finished executing SMAP-S2 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.

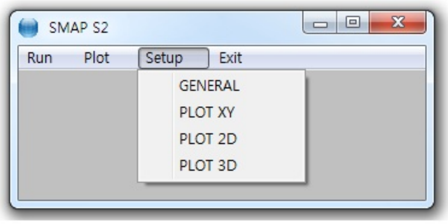
Note: When you first plot results, do not check the check box in Skip Data Processing. When you replot results, however, you can check the check box to skip intermediate data processing. This will save time and keep modified output data.

3.4 SETUP Menu

You need to run SETUP Menu

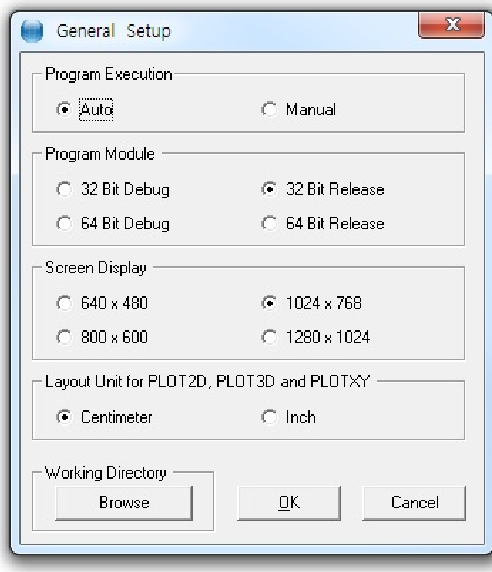
- To specify SMAP-S2 main-processing program module.
- To adjust scales of graphical outputs from PLOT-XY, PLOT-2D, and PLOT-3D

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.

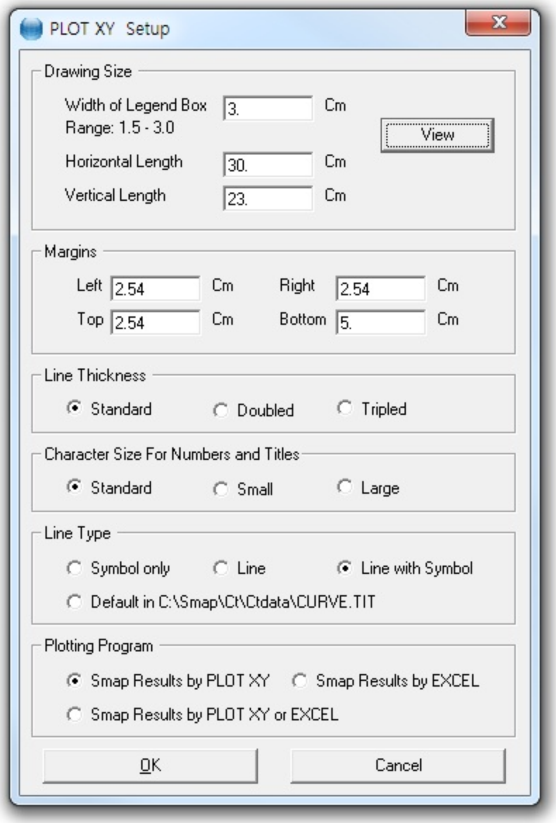
Screen Display has four options; 640x480, 800x600, 1024x768, and 1280x1024. This will affect the size of child window in PLOT-XY and PLOT-2D.

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.

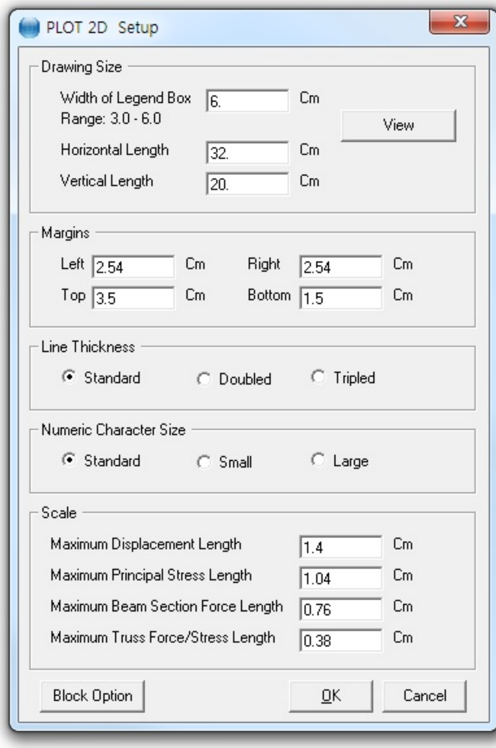
Plotting Program is used to specify default program to plot Smap results. It has three options; PLOT-XY, EXCEL, and PLOT-XY or EXCEL. Last option is to select either PLOT-XY or EXCEL at the time you plot results.

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.

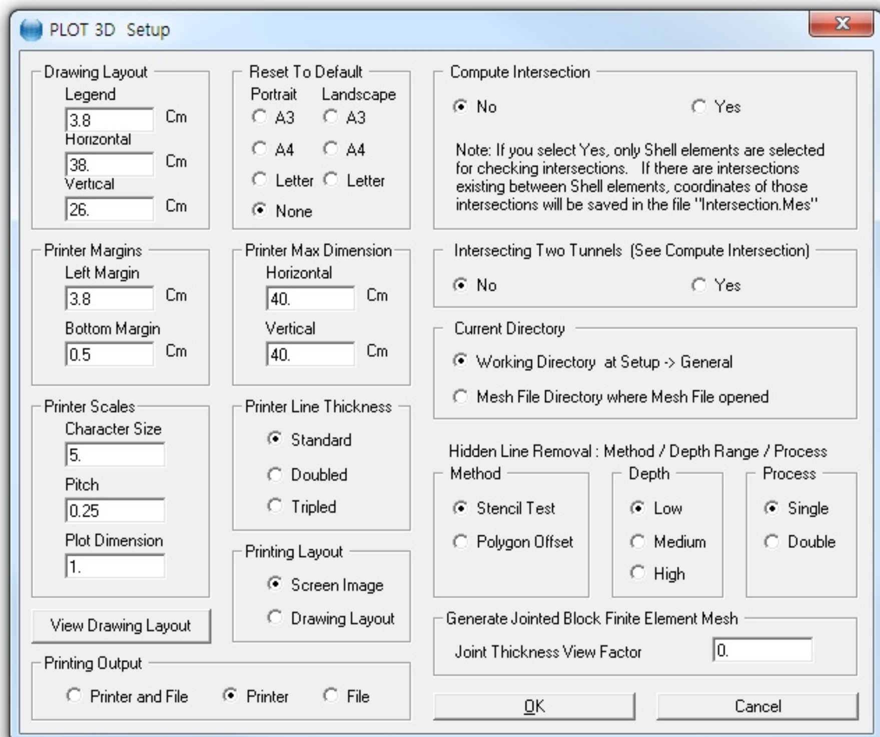
Block Option specifies options to generate either PRESMAP Output or Block Diagram.



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-S2

Occasionally, you need to execute SMAP-S2 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\SmapS2
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-S2 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_S2.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\SMAPS2\DEBUG (NOT AVAILABLE)

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

   IOUTDEB =   0 : Debug information on screen.
               1 : Debug information on file,
                   Smap_S2.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-S2
User's Manual

4.1 Introduction

To run SMAP-S2 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 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

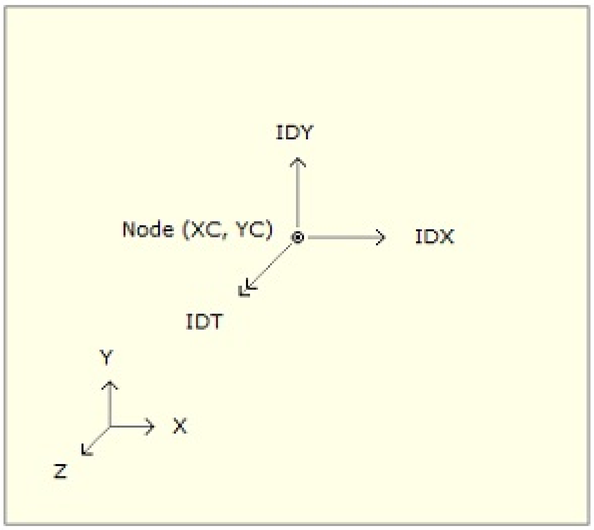
Mesh File contains nodal coordinates, boundary conditions, element indexes and material property numbers. This Mesh File is normally generated by Mesh Generator programs.

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, IDX, IDY, IDT, IEX, IEY, XC, YC - NODE Node number 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 IEX = 0 Slip displacement in x-direction is free = 1 Slip displacement in x-direction is fixed IEY = 0 Slip displacement in y-direction is free = 1 Slip displacement in y-direction is fixed XC X-coordinate YC Y-coordinate

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

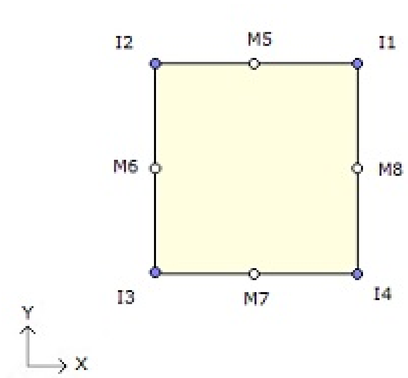
<table>
<tr><th>Card Group</th><th>Input Data and Definitions (Mesh File)</th></tr>
<tr><td rowspan="2">3

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

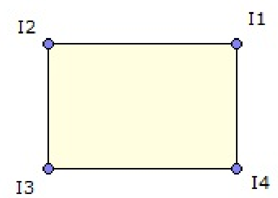
NCONT Card ┌ NEL, I_1, I_2, I_3, I_4, M_5, M_6, M_7, M_8, MATC,
│ THIC, DEN
└ - - - - - - - - - - -



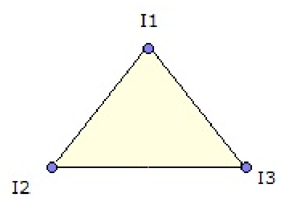
NEL Element number
I_1, I_2, I_3, I_4 Element corner node numbers
M_5,M_6,M_7 M_8 Element midside node numbers
If any of M_5, M_6, M_7 or M_8=0,
that node is omitted</td></tr>
</table>

Card Group	Input Data and Definitions (Mesh File)
3 Continuum Element	3.2 MATC — Material property number THIC — Thickness for plane stress element DEN — Unit weight of element. If DEN < 0.0, element has joint and absolute value of DEN represents face designation number. Refer to description in the following page

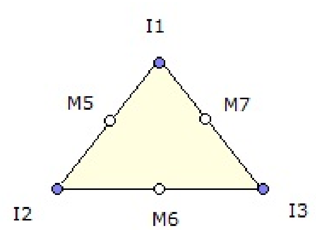
4-Node Quadrilateral Element_ ($M_5 = M_6 = M_7 = M_8 = 0$)



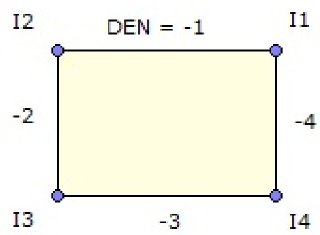
3-Node Triangular Element ($I_4 = M_5 = M_6 = M_7 = M_8 = 0$)



6-Node Triangular Element ($I_4 = M_8 = 0$)



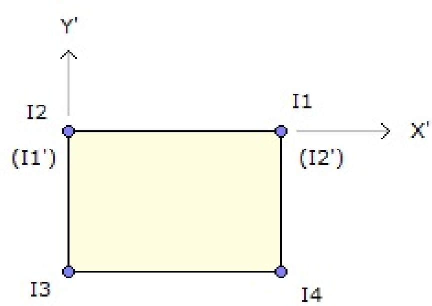
Joint Element Face DesignationNumber



Joint Element Local Coordinate Axes

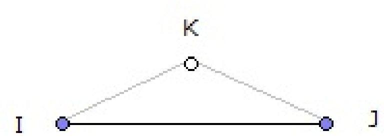
DEN	I_1'	I_2'
-1	I_2	I_1
-2	I_3	I_2
-3	I_4	I_3
-4	I_1	I_4

For DEN = -1



It should be noted that the thickness of joint element is determined
Not by the gap between two faces (I_2 I_1 and I_3 I_4),
But by the joint thickness (t) specified in Card 5.3.2.2 in Main File input

The nodal coordinates of I_2 and I_1 represent the Location of Joint Face
but the nodal coordinates of I_3 and I_4 are used only For Potting 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)

Card Group 5 — Truss Element (If NTRUSS = 0, skip this card group)

5.1

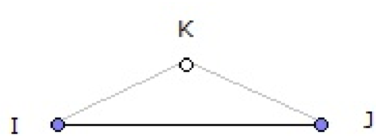
LABEL5A [Character string]
LABEL5B [Character string]

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 problems.

Main File consists of ten different card groups:

- Batch
- Title
- Analysis Type
- Coordinate
- Continuum Element
- Beam Element
- Truss Element
- Element Activity
- Gravity Load
- Concentrated Load
- Requested Output

Card Group	Input Data and Definitions (Main File)
1 Batch	1.1 IBATCH, VERSION IBATCH = 0 Interactive terminal job = 1 Batch job (not available) = 2 Generate Mesh File PlotMesh.Mes (This will not execute input file) = -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 VERSION Version Number (Current Version= 7.0)
2 Title	2.1 TITLE TITLE Any title of up to 70 characters

Card Group	Input Data and Definitions (Main File)
3 Analysis Type	3.1.1 NMAGE, NSTEP, NPRNT, IQUAD, DSRNMAX NMAGE = 0 Linear elastic analysis = 1 Material nonlinearity only = 2 Geometric nonlinearity only = 3 Material and geometric nonlinearity NSTEP Total number of load steps NPRNT Number of steps between printing output IQUAD = 0 No automatic generation = 1 Automatic generation of quadratic elements (Not available). If IQUAD = 1, all linear elements are automatically transformed into quadratic elements. DSRNMAX = 0.0 Do not apply strain subcycling > 0.0 Maximum strain subincrement

Card Group	Input Data and Definitions (Main File)
3 Analysis Type	3.1.2 MAXCYCL, NFDRIFT, NITER, MNEWRP, TOLER MAXCYCL — Maximum number of strain subcycling NFDRIFT — Drift correction option. (Use NFDRIFT=1) NITER — Number of maximum iteration MNEWRP = 0 — Modified Newton-Raphson method = 1 — Newton-Raphson method =-1 — Newton-Raphson method with first iteration as trial guess TOLER — Tolerance for convergence, defined as the ratio of displacement increment to current displacement. (Default TOLER=0.001)
	3.1.3 IRANGE, NTEMP IRANGE = 0 — NITER is applied throughout NSTEP = 1 — NITER is applied during the specified ranges of load step. NTEMP = 0 — Thermal expansion is not considered = 1 — Thermal properties and element temperatures are read from input file ELTEMP.DAT Note: File ELTEMP.DAT should be located in the working directory. See Table in next page

Card Group	Input Data and Definitions (Main File)	
3 Analysis Type	3.2.1 If IRANGE = 0, go to Card Group 4.1 NRANGE NRANGE Number of specified ranges where NITER is applied. (Max=100)	
	3.2.2 For Each Range	3.2.2.1 SFSTEP, SLSTEP SFSTEP Starting load step SLSTEP Ending load step

Input File ELTEMP.DAT

Card Group	Input Data and Definitions (Main File)
1	1.1 TITLE [Character string] TITLE Project title
2 Thermal Property	2.1 LABEL 1 [Character string] LABEL 1 Label for Card 2.2
	2.2 ┌ $MATNO_i$, $ALPHA_i$ │ - - └ - - $MATNO_i$ Material property number If $MATNO_i$ = -1, end of Card 2.2 $ALPHA_i$ Coefficient of thermal expansion (L/L/Temperature)

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 2 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 ┌ $NELNO_i$, $MATNO_i$, $TEMP_{topi}$, $TEMP_{boti}$ │ - - - - └ - - - - $NELNO_i$ Element number If $NELNO_i$ = -1, end of Card 3.4 $MATNO_i$ Material property number. $TEMP_{topi}$ Temperature on top surface $TEMP_{boti}$ Temperature on bottom surface

Card Group	Input Data and Definitions (Main File)
4 Coordinate	4.1 NUMNP NUMNP Total number of nodal points

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
	5.2.1 NCTYPE, NSPTC, IEDOF NCTYPE = 0 — Axisymmetric element Y-axis is axis of symmetry = 1 — Plane strain element = 2 — Plane stress 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
	5.2.2 NKOTYPE, YWATER NKOTYPE = 0 — Element has zero initial stresses = 1 — Element initial stresses are computed with $K_o = v/(1-v)$ = 2 — Element initial stresses are computed with input K_o values specified at Card 5.3.2.2 YWATER — Y coordinate of ground water table Note: If NKOTYPE = 2, Y-axis should be gravitational direction

<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">5.3
Material Property Data</td><td colspan="2">5.3.1
NTNC

NTNC Number of material property set for continuum element (Max=50)</td></tr>
<tr><td>For Each Material Property Set</td><td>5.3.2.1
MATNO, MODELNO, GAMMAW, MATNP

MATNO Material number
MODELNO Material model number
GAMMAW Unit weight of ground water
MATNP Parent material number
MATNO will duplicate MATNP

If MATNP > 0, go to next property set.</td></tr>
</table>

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	For Each Material Property Set	5.3.2.2 MODELNO = 1 Elastic Model E, v, K_o = 2 Von Mises Model E, v, K_o σ = 3 Mohr-Coulomb Model E, v, K_o φ, C, K, T, ST_n, ST_s = 4 In Situ Rock Model E, v, K_o m, s, σ_c, K, T, ST_n, ST_s E Young's modulus v Poisson's ratio K_o Coefficient of earth pressure at rest For NKOTYPE = 0 or 1, use K_o = 0.0 σ Shear strength in the triaxial compression φ Internal frictional angle (˚) C Cohesion $C = \frac{(1 - \sin \varphi)}{2 \cos \varphi} \sigma_c$ K The ratio of th 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	For Each Material Property Set	5.3.2.2

m, s — Hoek and Brown material parameters
See table in the next page

σ_c — Unconfined compressive strength

If MODELNO = 5 (Joint Model)

<u>Elastic Modulus and Joint Thickness</u>

NM, E, G, t

NM	= 0	Linear elastic joint
	= 1	Nonlinear joint
	= 2	Lumped nonlinear joint
	= 3	Contact nonlinear joint

E	Elastic Young's modulus
G	Elastic shear modulus
t	Joint thickness

<u>Strength Parameters (only for NM > 0)</u>

C, φ

C	Cohesion
φ	Friction angle (˚)

<u>Normal Stress-Strain Relation (only for NM > 0)</u>

ε_1, ε_2, ε_3, ε_4, σ_1, σ_2, σ_3, σ_4

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

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 represents joint thickness t
3. For t = 0.0 and NM = 4, oint 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 shear

Hoek and Brown Material Parameters (m, s)

Rock Type / Rock Quality	Dolomite, Limestone &	Mudstone, Siltstone, Shale and Slate (normal to	Sandstone and Quartzite	Andesite, Dolerite & Rhyolite	Amphibolite, Gabbro, Gneiss,
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

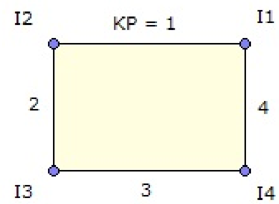
Description of Rock Quality

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 / 5.5 Element Surface	5.5.1 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	5.5.2.1 (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 Card 10.4 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	Element Surface	For Each Material / Element Surface	5.5.2.1 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 P'_n is given as a function of global X and Y coordinate $P'_n = a_0 + a_1 X + a_2 Y$ 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 $q_Y = a_0 + a_1 X + a_2 Y$ Note: (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

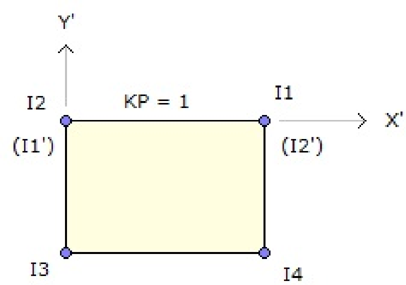
Element Surface Designation Number



Element Local Coordinate Axes

KP	Quadrilateral Element		Triangular Element	
	I_1'	I_2'	I_1'	I_2'
1	I_2	I_1	I_2	I_1
2	I_3	I_2	I_3	I_2
3	I_4	I_3	I_1	I_3
4	I_1	I_4	0	0

For KP =1



<table>
<tr><th>Card Group</th><th>Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="2">6

Beam Element</td><td>6.1

NBEAM

NBEAM Total number of beam element

If NBEAM = 0, go to Card group 7</td></tr>
<tr><td>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.</td></tr>
</table>

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

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

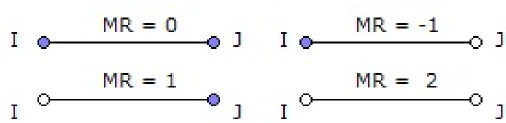
NSEC — Beam section number

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

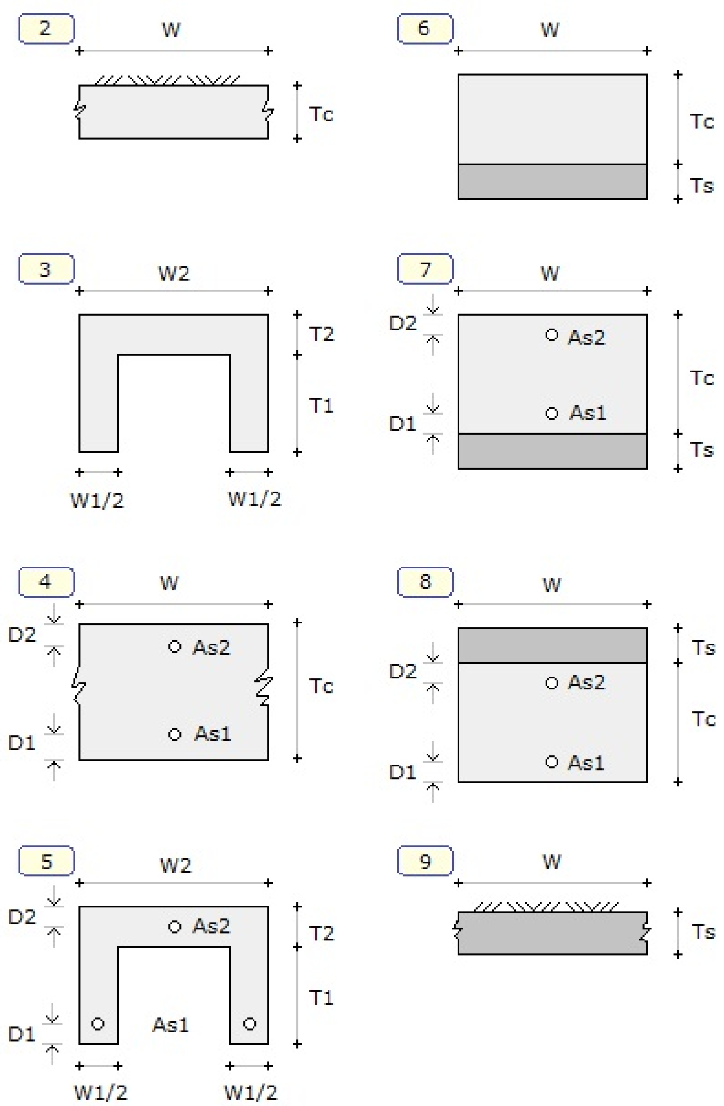
NLAYR — Total number of layers (Max=20)

NEHNO — Element stiffness/volume history No
Override MCH in Card Group 8.3

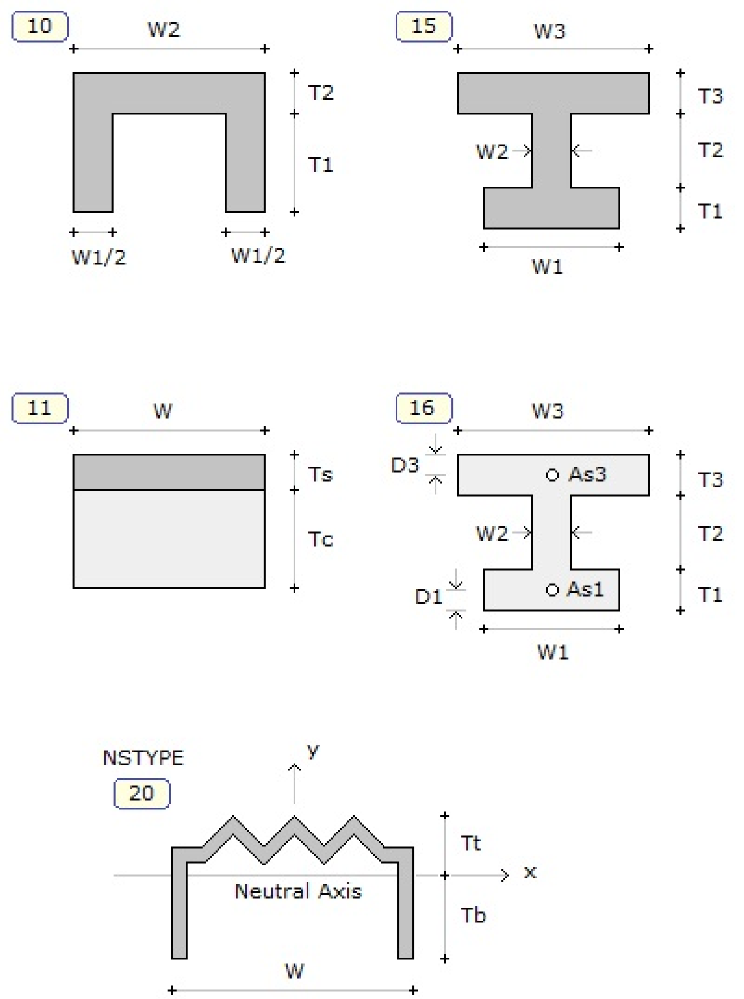
WL — Weight per unit length of beam

CTS — Timoshenko shear coefficient

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 = 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 history number in Card Group 10.4</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 Matetial	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 triaxial compression φ Internal frictional angle (°) C Cohesion K The ratio of shear strength in triaxial extension to the shear strength in triaxial compression at 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 $C = \frac{(1 - \sin \varphi)}{2 \cos \varphi} \sigma_c \qquad T = \frac{2 \cos \varphi}{(1 + \sin \varphi)} C$ 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)

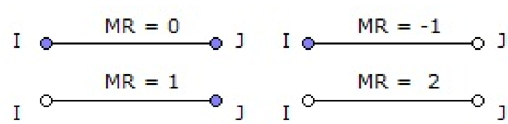
Section Property Data

For Each Section

6.4.2.2.1

NSEC, NFSHR, MR, WL, YNA, CTS

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
CTS	Timoshenko shear coefficient

Card Group	Input Data and Definitions (Main File)

6 Beam Element

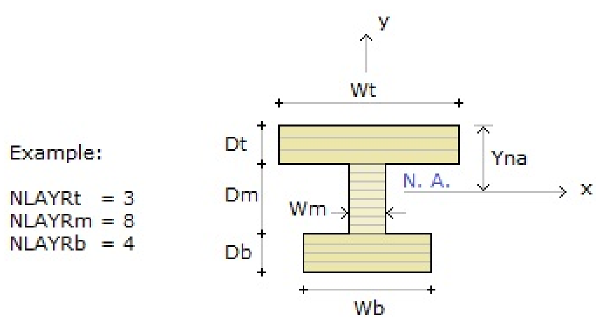
6.4 For NBLT = 1 (User-defined Layered Beam)

For Each Section

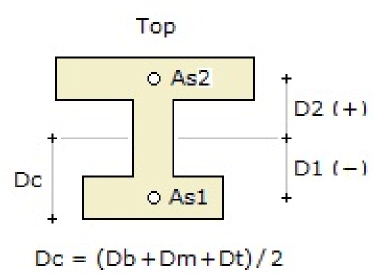
6.4.2.2.2

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 $\leq$ 20

Card Group	Input Data and Definitions (Main File)		
6 Beam Element	For NBLT = 1 (User-defined Layered Beam)	For Each Section	6.4.2.2.3 NFRBR NFRBR Number of reinforcing bars
			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 

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

Beam Element</td><td rowspan="3">6.5

For NBLT = 2 (Conventional Elastic Beam)</td><td colspan="2">6.5.1.1

NTNS

NTNS Number of beam sections (Max=50)</td></tr>
<tr><td rowspan="2">For Each Section</td><td>6.5.1.2

NSEC, NFSHR, MR, NEHNO, WL, CTS

Refer to Card 6.3.2.2.1

MR = 3 Joint spring element
Available only for NBLT = 2</td></tr>
<tr><td>6.5.1.3

For MR ≠ 3
T_b, T_t, W, A, I, E, v

Refer to NSTYPE = 20 in Page 4-34

For MR = 3
K_X, K_y, K_R

K_X Axial spring stiffness
K_Y Shear spring stiffness
K_R Rotational spring stiffness</td></tr>
</table>

<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

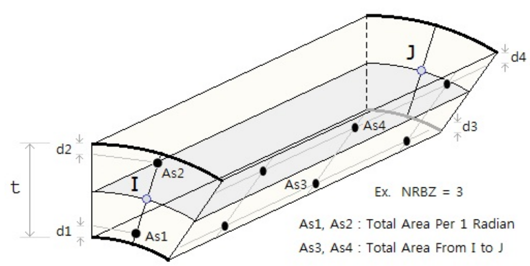
NSEC, NFSHR, MR, NEHNO, GAMA, CTS

GAMA Unit weight

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, E, STRSI A — Cross section area WL — Weight per unit length of truss E — Young's modulus STRSI — Initial stress. Tension is positive For constant initial stress, use E = 0

Card Group		Input Data and Definitions (Main File)
7 Truss Element	7.4 For Each Material	7.4.3 If NMAGE = 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

<table>
<tr><th>Card Group</th><th>Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="3">8

Element Activity</td><td>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</td></tr>
<tr><td>8.2

KSTEP

KSTEP Number of sub steps to unload deactive elements</td></tr>
<tr><td>8.3

(MCFAD) Cards
MATC, NAC, NDAC, MCH

(MBFAD) Cards
MATB, NAC, NDAC, MCH

(MTFAD) Cards
MATT, NAC, NDAC, MCH

(NFAD - MCFAD - MBFAD - MTFAD) Cards
NEL, NAC, NDAC, MCH

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
MCH Element stiffness/volume history number
Refer to Card Group 9.2 and 9.3

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

Card Group		Input Data and Definitions (Main File)
9 Loads	Gravity Load	9.1 NFDL, FRX, FRY, NHFRX, NHFRY NFDL = 0 Do not include gravity load = 1 Include 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 10.4 Intensity times Distribution Factor will be additive to FRX or FRY
		9.1.1 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 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

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="4">Variable Element Stiffness/Volume Data</td><td colspan="2">9.2

If NFAD = 0, go to Card Group 10

NUMMCH

NUMMCH — Number of histories for variable element stiffness and element volume. Max=1000

If NUMMCH = 0, go to Card Group 10</td></tr>
<tr><td rowspan="3">9.3

For Each Stiffness / Volume History</td><td>9.3.1

MCH

MCH — Element stiffness/volume history number</td></tr>
<tr><td>9.3.2

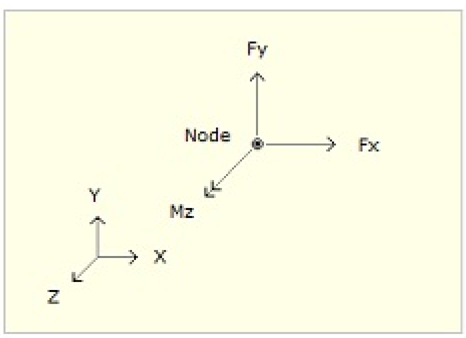
NSTMCH

NSTMCH — Number of specified load steps. Max=1000</td></tr>
<tr><td>9.3.3

NSTMCH Cards:
$STEP_1$, $PECE_1$, $PECV_1$
$STEP_2$, $PECE_2$, $PECV_2$
- - -
$STEP_{NSTMCH}$, $PECE_{NSTMCH}$, $PECV_{NSTMCH}$

$STEP_i$ — Specified load step
$PECE_i$ — Fraction of Young's Modulus at $STEP_i$
$PECV_i$ — Fraction of element volume at $STEP_i$

Note — $STEP_1$ should start from 0.0 and $STEP_{NSTMCH}$ > NSTEP</td></tr>
</table>

Card Group	Input Data and Definitions (Main File)
10 Concentrated Load	10.1 NLOAD NLOAD Total number of applied forces If NLOAD = 0, skip the rest of this Card Group
	10.2 NLOAD Cards: NODE, IDOF, LHNO, CINT - - - - - - - - - - - - NODE Node number at which force/moment is specified IDOF = 1 Horizontal force F_x = 2 Vertical force F_y = 3 Moment about z-axis M_z LHNO Load history number CINT Load intensity factor 

Card Group	Input Data and Definitions (Main File)
10	10.3 NUMLH, NUMSP NUMLH Number of different load histories. Max=100 NUMSP Number of load steps. Max=1000
Concentrated Load	10.4 $STEP_1$, $C_{1\,1}$, $C_{1\,2}$, - -, $C_{1\,NUMLH}$ $STEP_2$, $C_{2\,1}$, $C_{2\,2}$, - -, $C_{2\,NUMLH}$ - - - - - - - - - - - - $STEP_{NUMSP}$, $C_{NUMSP\,1}$, $C_{NUMSP\,2}$, - - $C_{NUMSP\,NUMLH}$ STEP Load step C Load coefficient Note Product of load intensity and load coefficient represents force/moment $STEP_{NUMSP} > NSTEP$

Card Group	Input Data and Definitions (Main File)
10 Requested Output	10.5.1 NHPEL NHPEL — Number of elements at which stress/strain time histories are requested
	10.5.2 If NHPEL = 0, skip the following Card NEL_1, NEL_2, …, NEL — Element number to be printed
	10.6.1 NHPMT NHPMT — Number of nodes at which displacement time histories are requested
	10.6.2 If NHPMT = 0, skip the following Card $NODE_1$, $NODE_2$, …, $NODE_{NHPMT}$ NODE — Node numbers to be printed
	10.7.1 NTIME NTIME — Number of steps at which stress/strain/ displacement profiles are requested
	10.7.2 If NTIME = 0, skip the following Card $STEP_1$, $STEP_2$, …, $STEP_{NTIME}$ STEP — STEP 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 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

<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">11.2

For NPTYPE = 1 (Finite Element Mesh / Element Number)</td><td>11.2.1

TITLE

TITLE Any title (Max = 70 characters)</td></tr>
<tr><td>11.2.2

IUNIT

IUNIT = 1 Inch
= 2 Cm
= 3 User-specified unit</td></tr>
<tr><td>11.2.3

For IUNIT = 3
NCHR
LABEL

NCHR Number of characters for mesh unit
LABEL Name of mesh unit</td></tr>
</table>

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 For IUNIT = 3 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 If NGROUP = 0, Skip this Card 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 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	11.4 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 PLOT-2D Plot Information	11.4 For NPTYPE = 3 (Deformed Shape)	11.4.5

11.4.5

Row and Line Plots (Repeat in any order)

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

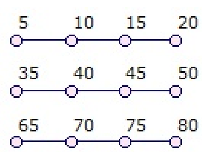
For Line Plot --> 2, IDISP
NPT
$NODE_1$, $NODE_2$, .., $NODE_{NPT}$

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 For NBLIST = 0 NFBEAM, NLBEAM NFBEAM Starting beam element number NLBEAM Ending beam element number
			11.5.7.3 For NBLIST = 1 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 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	11.6 For NPTYPE = 5 (Truss Axial Force / Stress / Strain)	11.6.1 TITLE TITLE Any title (Max = 70 characters)
		11.6.2 IUNIT IUNIT = 1 In, Pound = 2 Cm, Kg = 3 User-specified unit
		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

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 <u>For IENL= 2</u> R_x Weight factor applied to spline function If R_x =0.0, only Laplacian interpolation is used R_y is not used <u>For IENL= 3</u> 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	
		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')

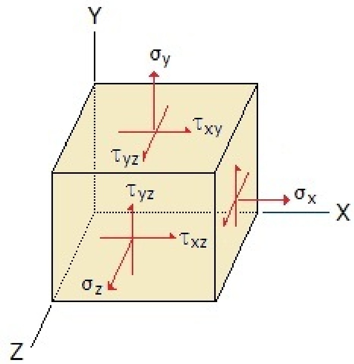


Figure PL-1 Sign Conventions for Continuum Stress

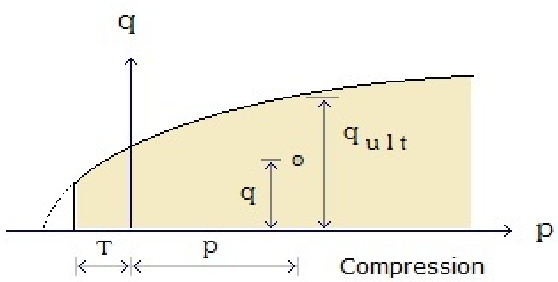


Figure PL-2 Definition of Safety Factor

<u>Factor of Safety (FS) is defined as:</u>

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

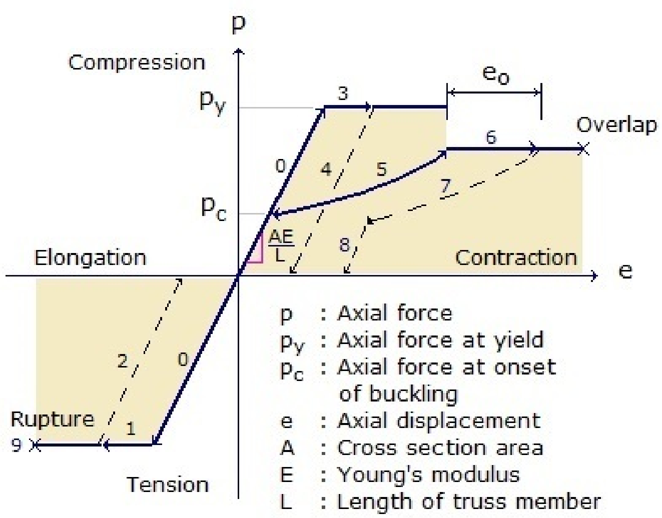


Figure PL-3 Description of Stress Status

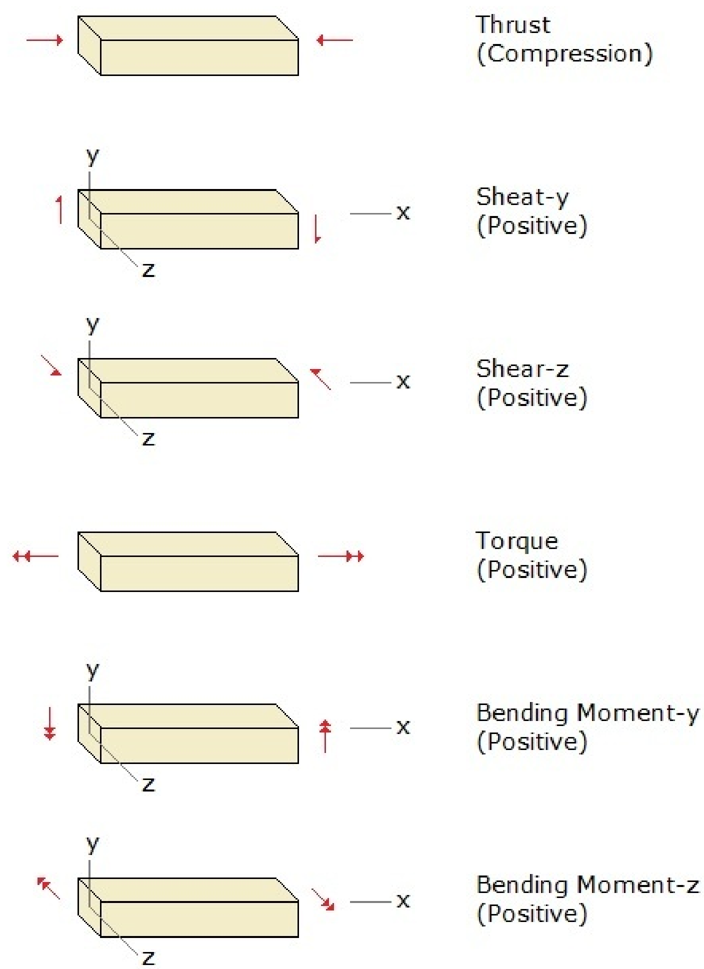


Figure PL-4 Sign Conventions for Beam

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

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions</th></tr>
<tr><td rowspan="3">12

PLOT-XY Information</td><td rowspan="3">12.2

For IPTYPE = 1 (Stress / Strain / Time History)</td><td>12.2.5

NDPQ Cards
K_{x1}, K_{y1}
K_{x2}, K_{y2}
- -
- -

K_x,K_y Select from Table PL-1</td></tr>
<tr><td>12.2.6

TMFAC, STFAC, SNFAC

Multiplication factor
TMFAC Time
STFAC Stress
SNFAC Strain</td></tr>
<tr><td>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)</td></tr>
</table>

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)

Card Group		Input Data and Definitions (Post File)
12 PLOT-XY Information	12.4 For IPTYPE = 3 (Stress / Strain vs. Distance Snapshot)	12.4.1 IPLOT IPLOT = 0 For each specified time, Number of different variables = 1 For each specified variable, Number of different time data
		12.4.2 NOTM NOTM Number of times (Max 10)
		12.4.3 TLIST (I), I = 1, NOTM TLIST (I) List times in sequential order
		12.4.4 NDPQ NDPQ Number of different variables
		12.4.5 NDPQ Cards: K_{y1}, K_{y2}, -, - K_y Select from Table PL-1

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)

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

PLOT-XY Information</td><td rowspan="5">For IPTYPE = 5 (Time History of Stresses/Strains for a Given Element)</td><td>12.6.1

NEL

NEL   Element number</td></tr>
<tr><td>12.6.2

NDQ

NDQ   Number of different quantities</td></tr>
<tr><td>12.6.3

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

K_y Select from Table PL-1</td></tr>
<tr><td>12.6.4

TMFAC, STFAC, SNFAC

Multiplication factor
TMFAC   Time
STFAC   Stress
SNFAC   Strain</td></tr>
<tr><td>12.6.5

TITLE   (50 characters)
X - LABEL   (50 characters)
Y - LABEL   (50 characters)</td></tr>
</table>

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)

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

PLOT-XY Information</td><td rowspan="5">For IPTYPE = 8 (Time History of Displ./Vel./Accel. Pair for Different Nodes)</td><td>12.9.1

NODE

NODE   Number of nodes (Max 10)</td></tr>
<tr><td>12.9.2

LIST (I)   I = 1,   NODE

LIST (I)   List node numbers</td></tr>
<tr><td>12.9.3

K_x, K_y

K_x, K_y Select from Table PL-2</td></tr>
<tr><td>12.9.4

TMFAC, SND, SNV, SNA

Multiplication factor
TMFAC   Time
SND   Displacement
SNV   Velocity
SNA   Acceleration</td></tr>
<tr><td>12.9.5

TITLE   (50 characters)
X - LABEL   (50 characters)
Y - LABEL   (50 characters)</td></tr>
</table>

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 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}
		12.11.6 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 <u>Node Number Specification (Max 800 Nodes)</u> 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)
		<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')

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)

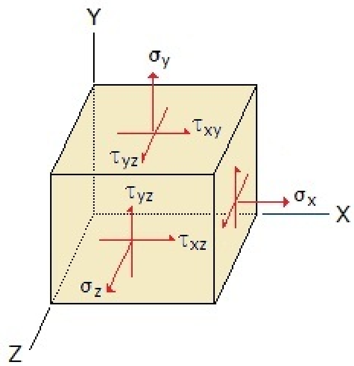


Figure PL-1 Sign Conventions for Continuum Stress

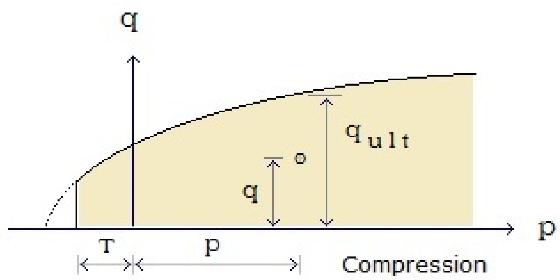


Figure PL-2 Definition of Safety Factor

<u>Factor of Safety (FS) is defined as:</u>

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

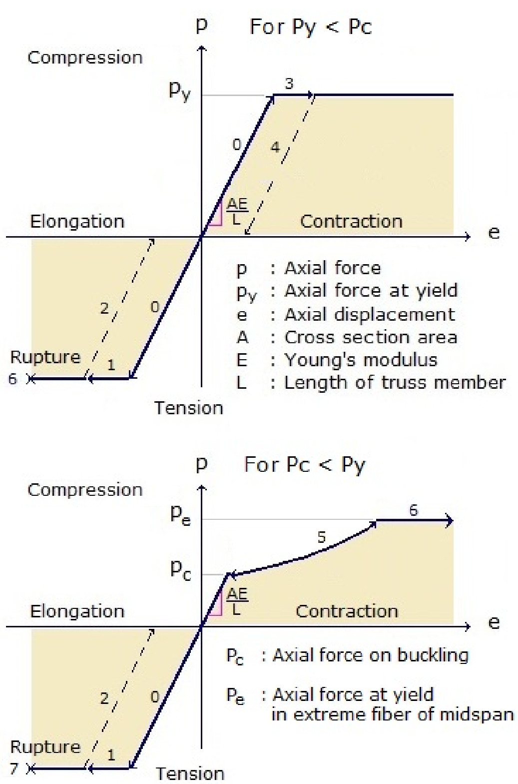


Figure PL-3 Description of Stress Status

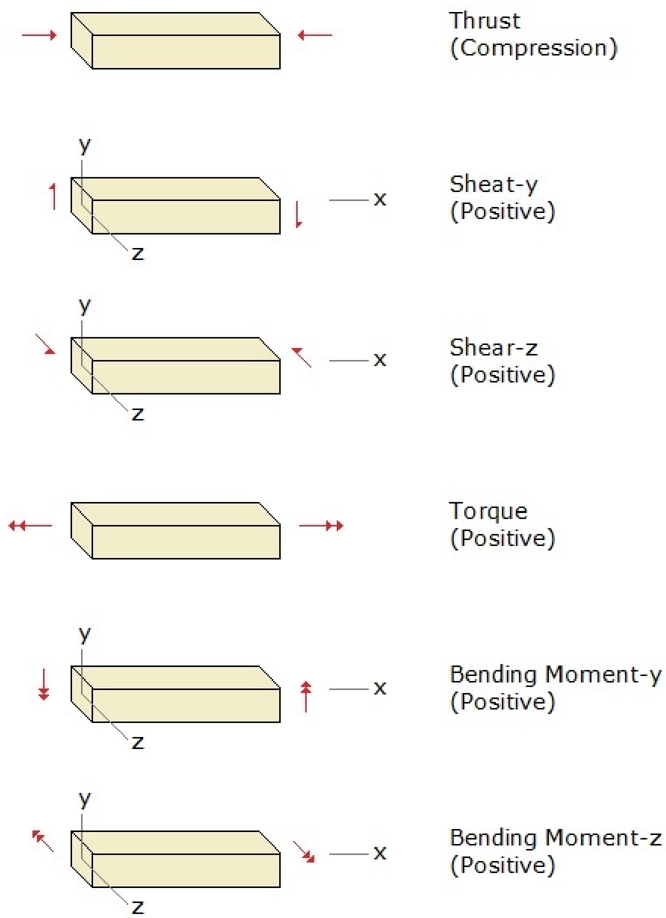
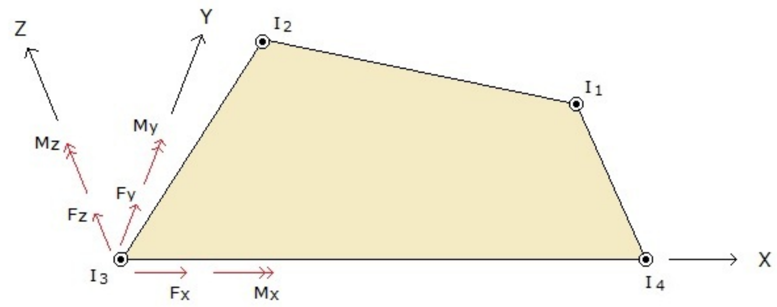
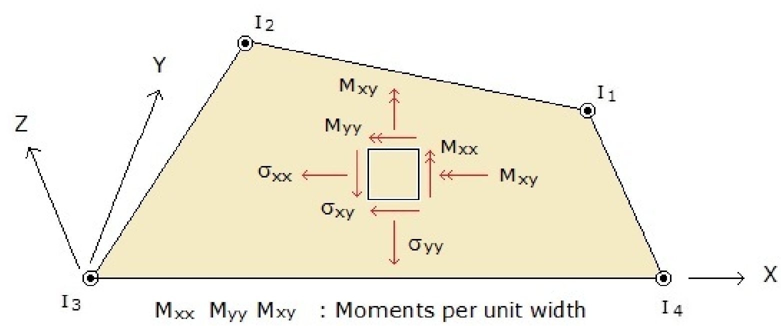


Figure PL-4 Sign Conventions for Beam



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.

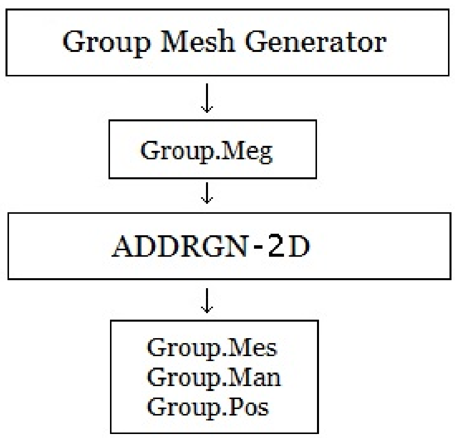


Figure 5.1 Flow diagram of group mesh generation

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

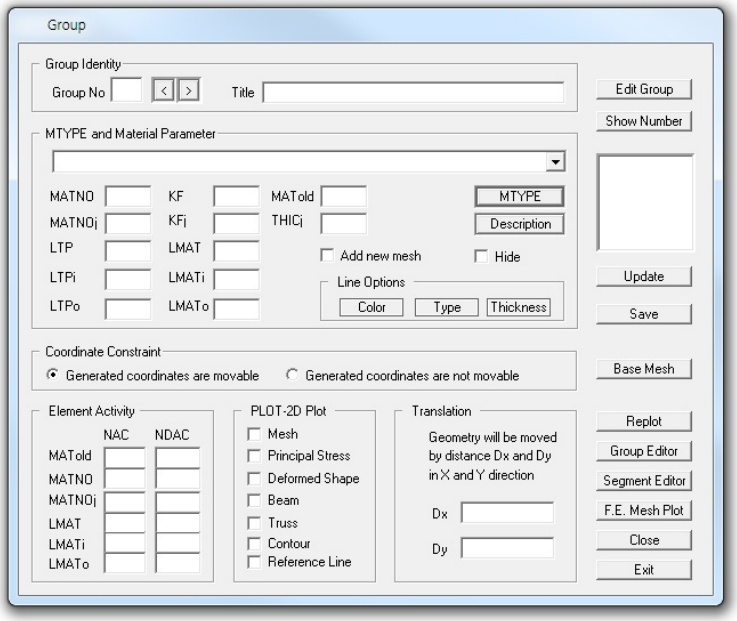


Figure 5.2 Group dialog.

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

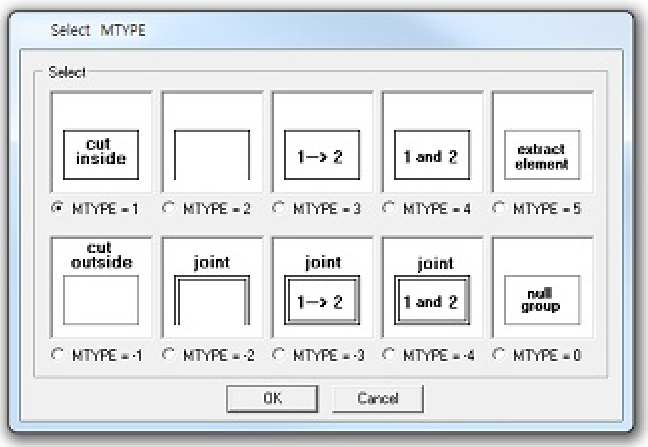
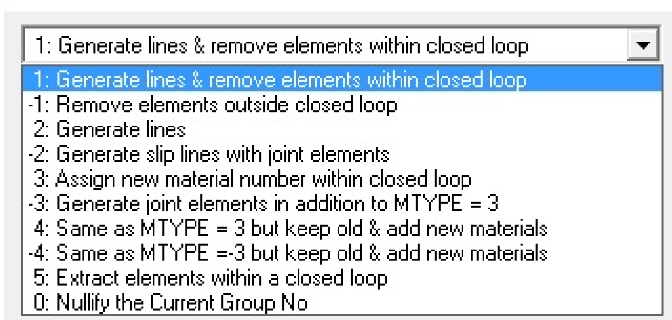


Figure 5.4
MTYPE list box



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.

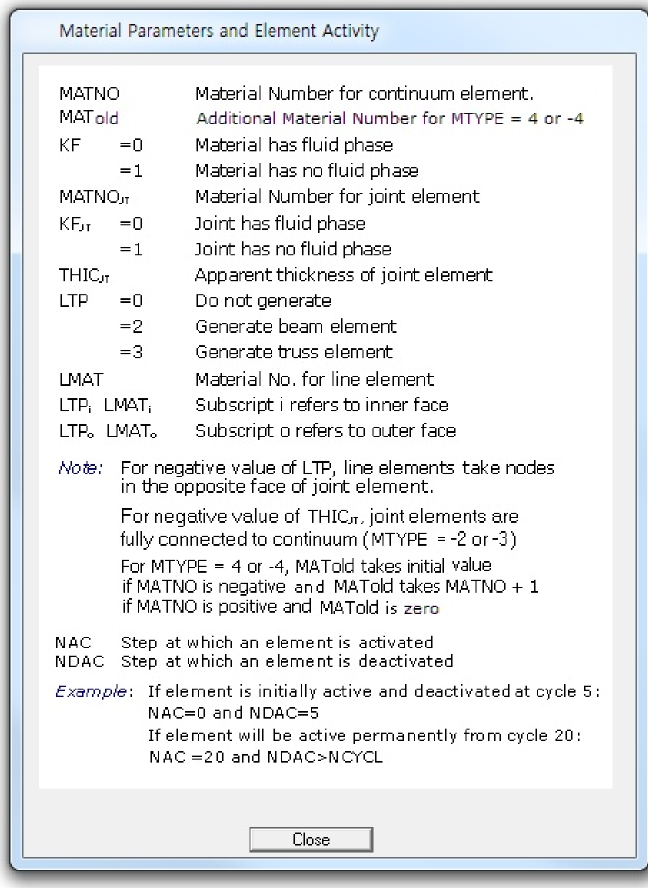


Figure 5.5 Material parameters & element activity (SMAP-2D)

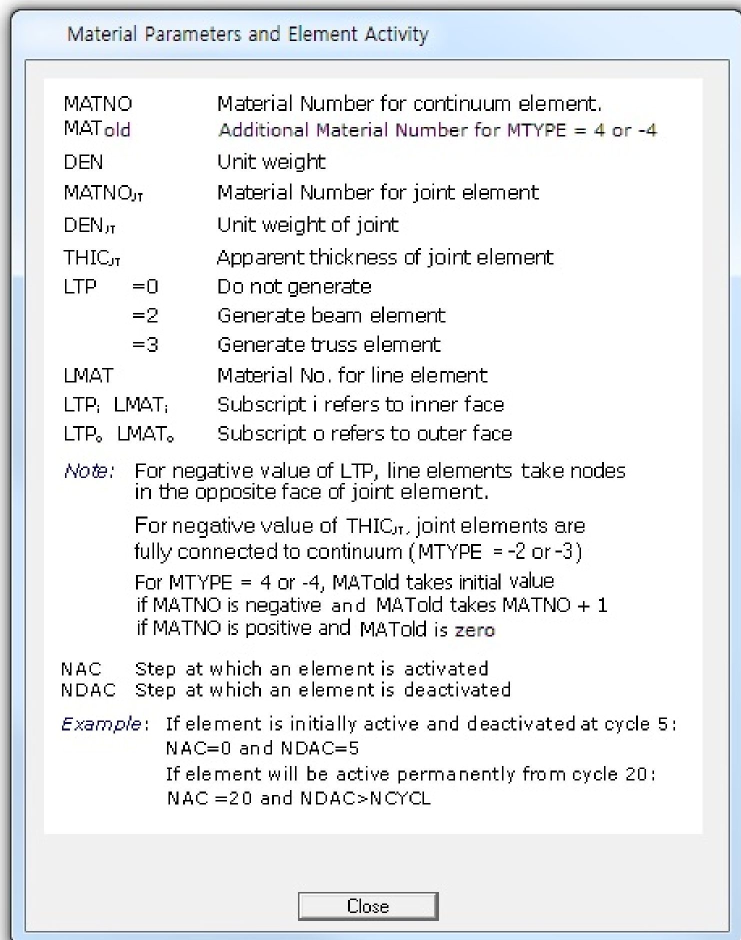


Figure 5.5 Material parameters & element activity (SMAP-S2)

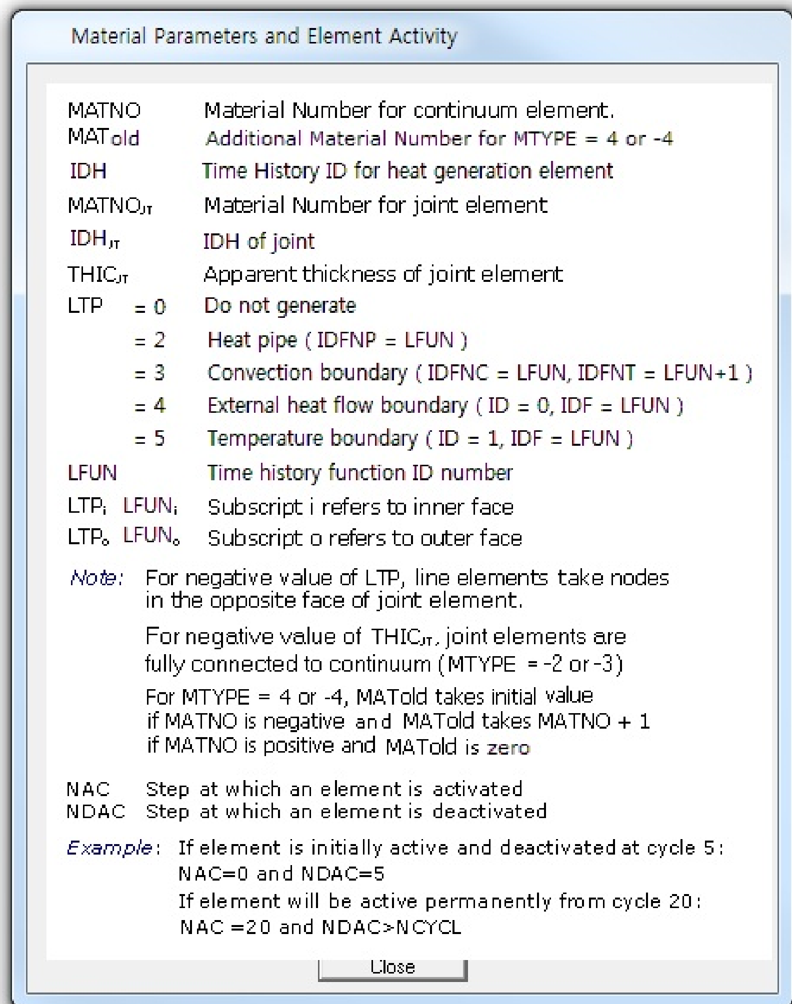


Figure 5.5 Material parameters & element activity (SMAP-T2)

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.

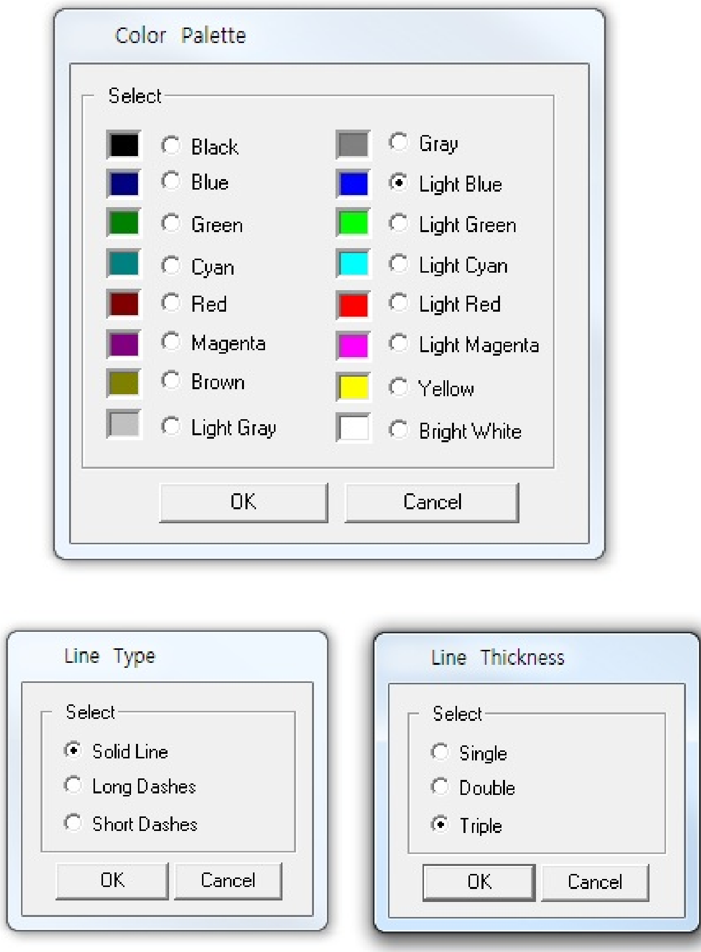


Figure 5.6 Line options.

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.

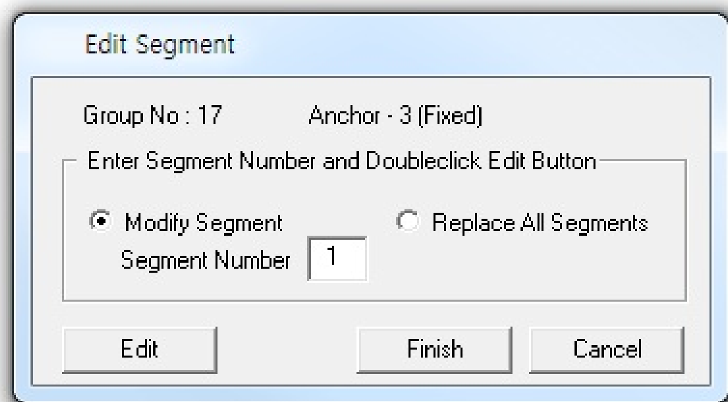


Figure 5.7 Edit segment dialog.

Show Number

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

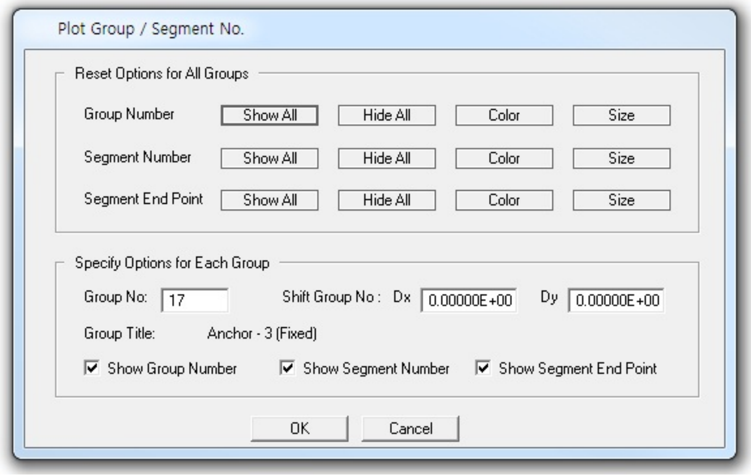


Figure 5.8 Plot Group / Segment No dialog.

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.

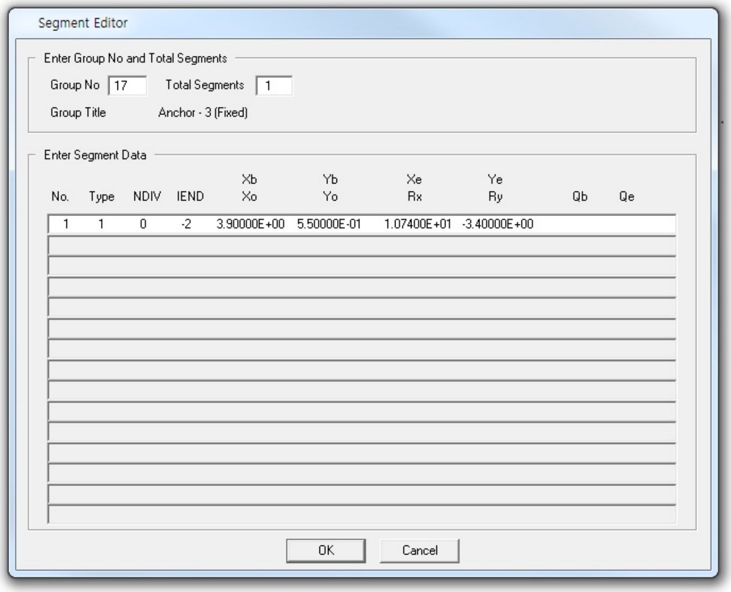


Figure 5.10 Segment editor dialog.

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.

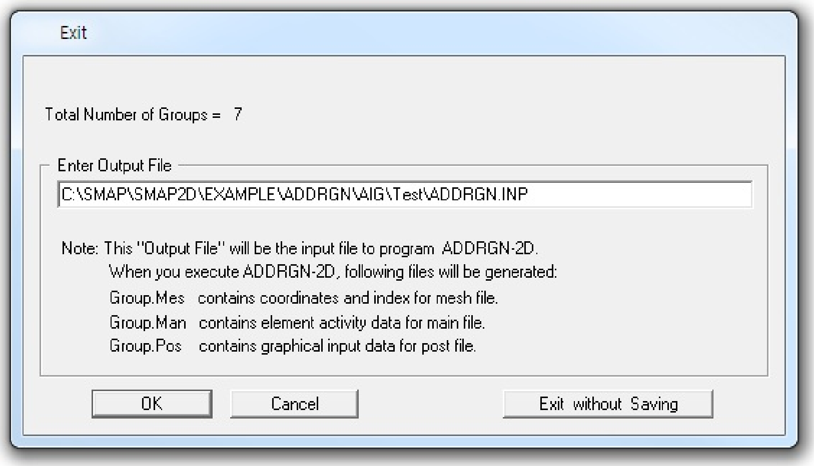


Figure 5.11 Exit dialog.

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.

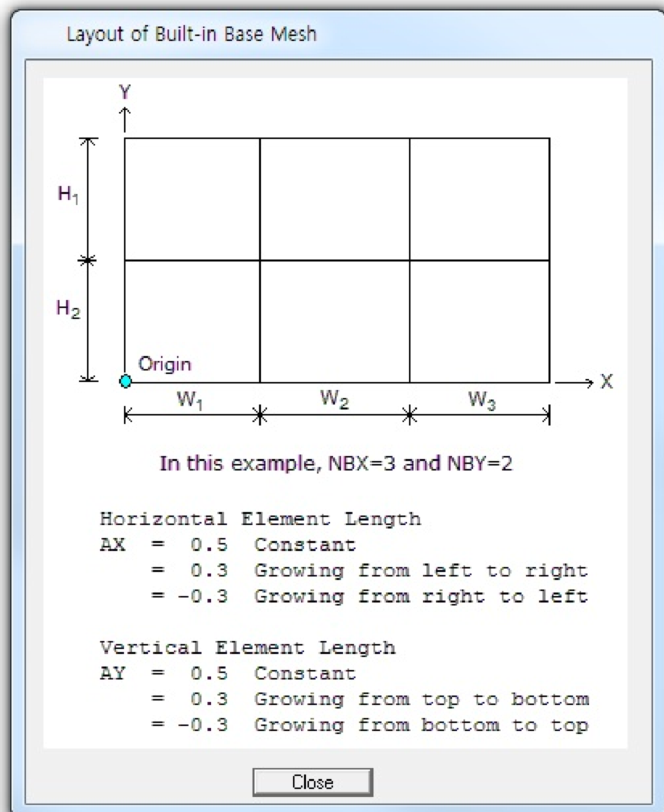


Figure 5.12 Layout of built-in base mesh.

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

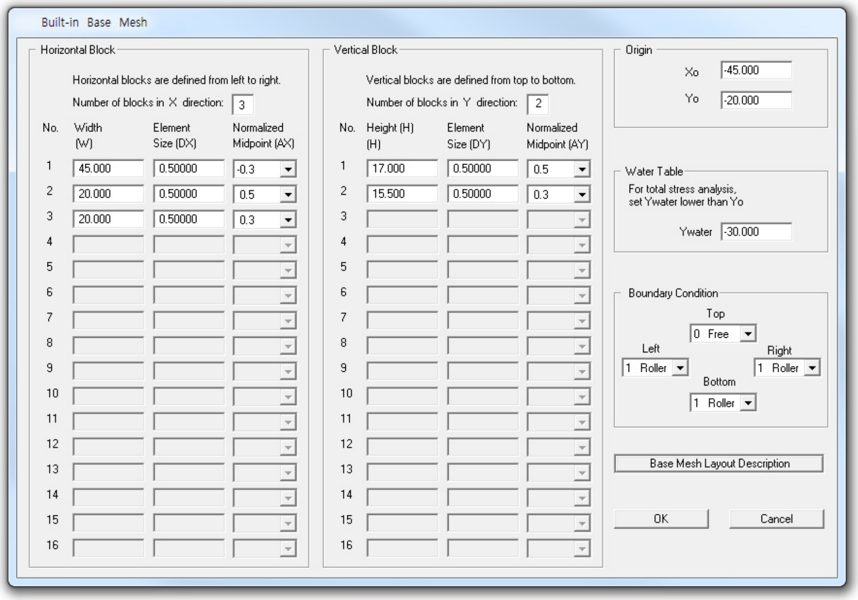


Figure 5.13 Built-in base mesh dialog.

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

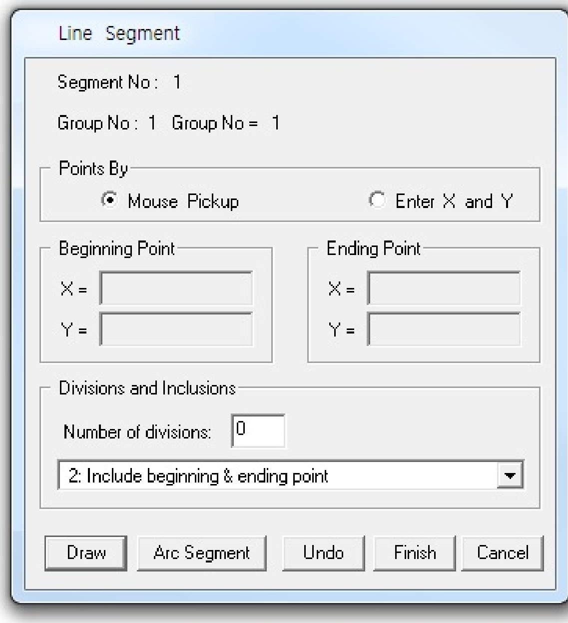


Figure 5.14
Line segment dialog.

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.

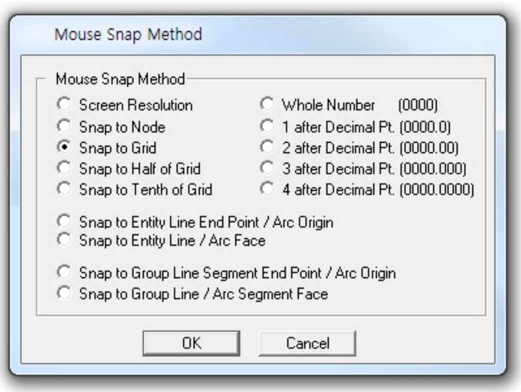
Divisions and Inclusions
Use following default values.
Number of divisions 0
Combo box selection 2: Include beginning & ending point

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

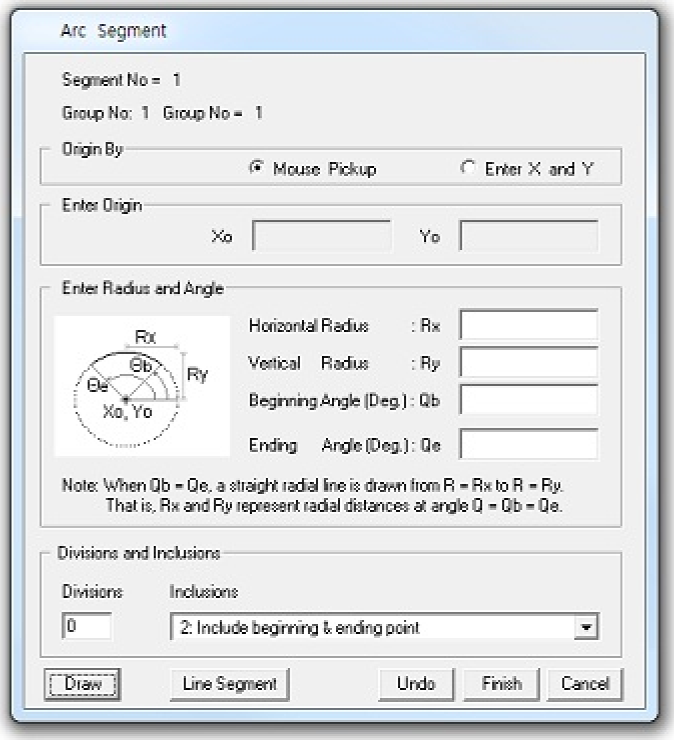


Figure 5.16 Arc segment dialog.

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.

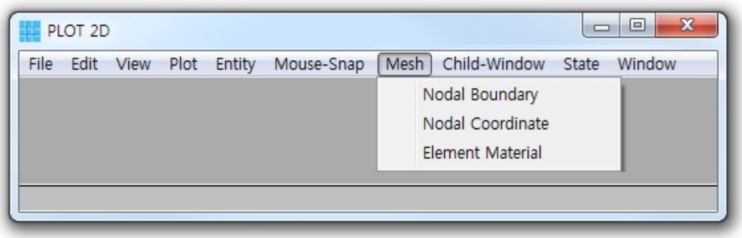


Figure 5.17 Menu for editing finite element mesh

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.

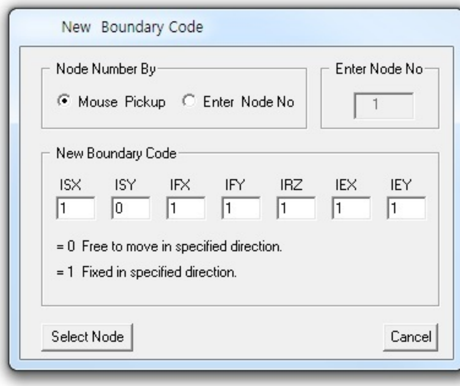
5.6.1 Edit Nodal Boundary

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

5.6.1.1 Mouse Pickup

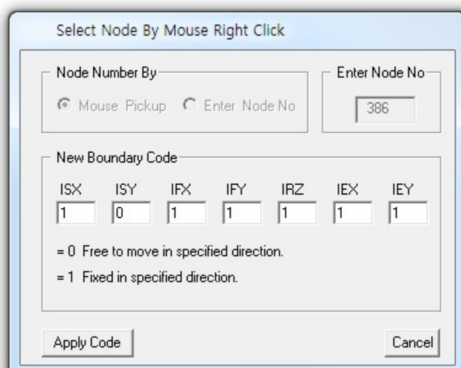
When you select Mouse Pickup mode as in Figure 5.18, you are supposed to select node number by mouse click. Click Select Node button.

Figure 5.18
Edit boundary
(Mouse Pickup)



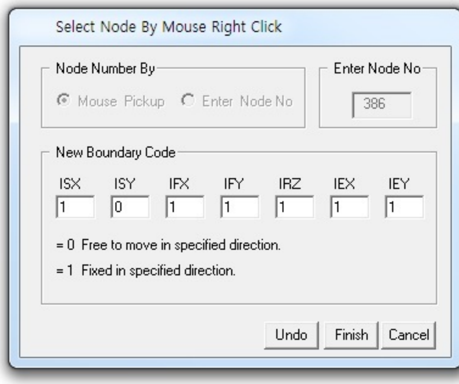
Click the node by Mouse Right Click, edit boundary codes and then click Apply Code button in Figure 5.19.

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.

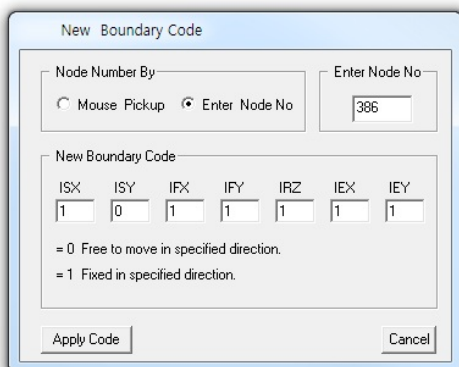
Figure 5.20
Edit boundary (Finish)



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.

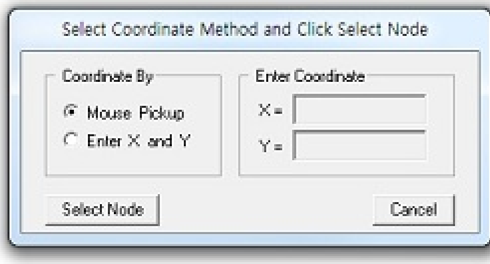
5.6.2 Edit Nodal Coordinate

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

5.6.2.1 Mouse Pickup

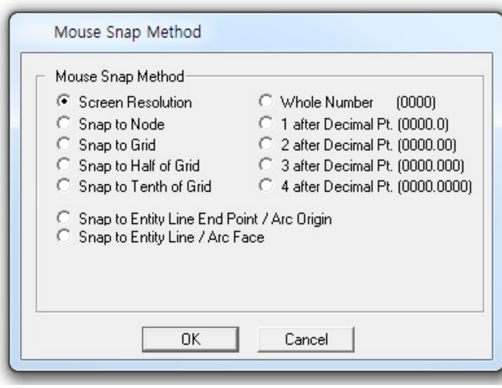
When you select Mouse Pickup mode as in Figure 5.22, you are supposed to select node number by mouse click. Click Select Node button.

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.

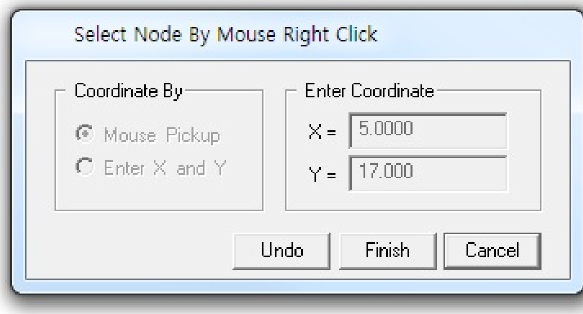


Figure 5.24 Edit coordinate (Finish)

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.

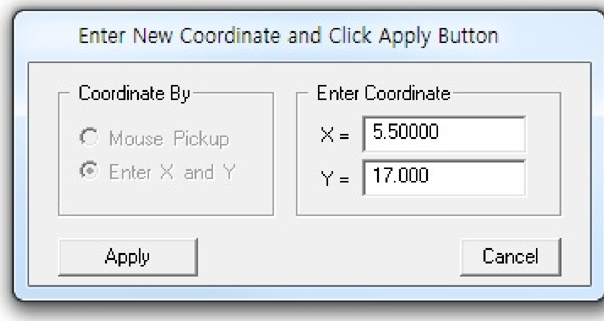


Figure 5.25 Edit coordinate (Enter X and Y)

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.

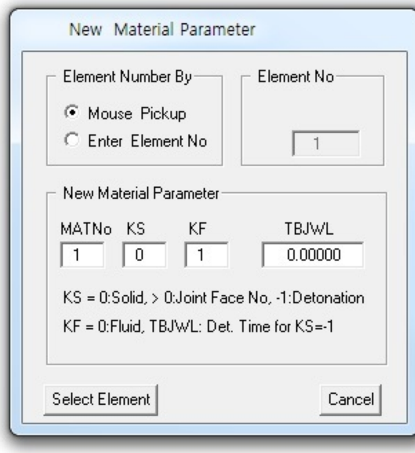


Figure 5.26
Edit element material
(Mouse Pickup)

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

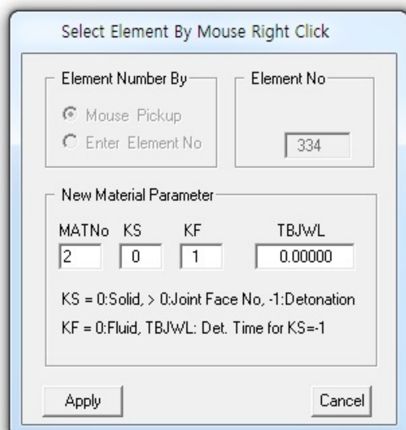


Figure 5.27
Edit element material
(Apply)

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

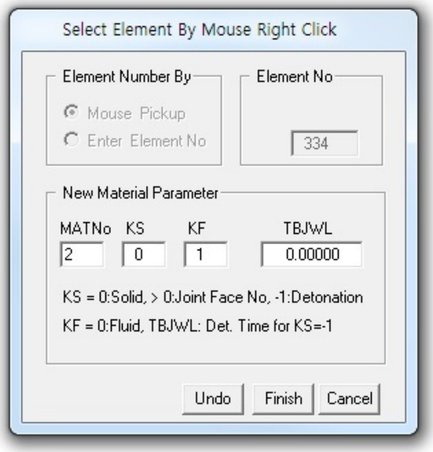


Figure 5.28
Edit element material
(Finish)

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.

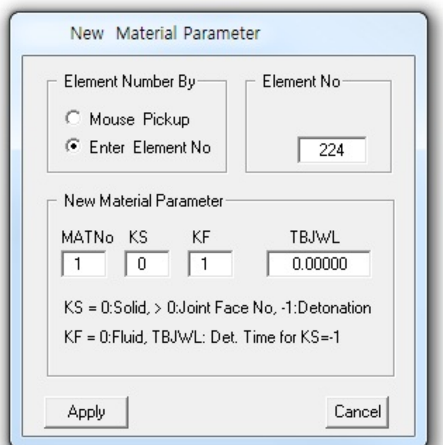


Figure 5.29
Edit element material
(Enter Element No)

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.

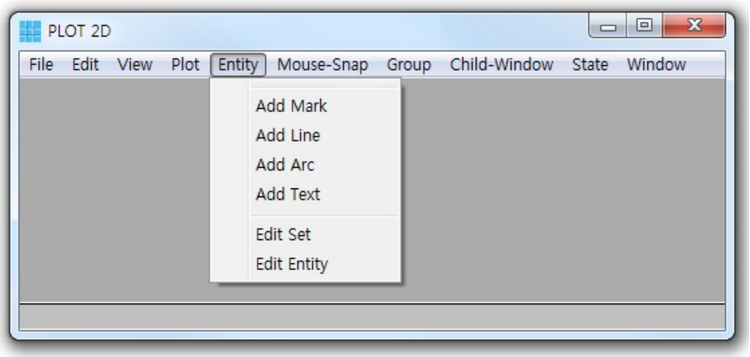


Figure 5.30 Entity menu

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.

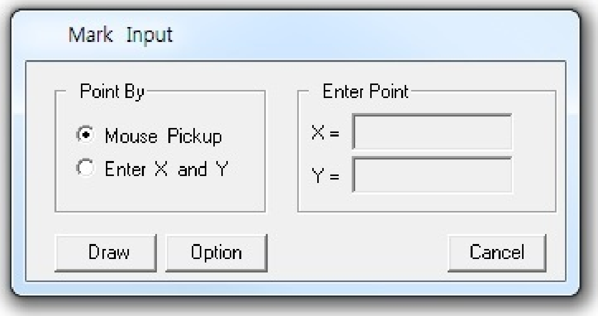
Edit Entity is to modify, delete or replace the selected entity.

5.7.1 Add Mark

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

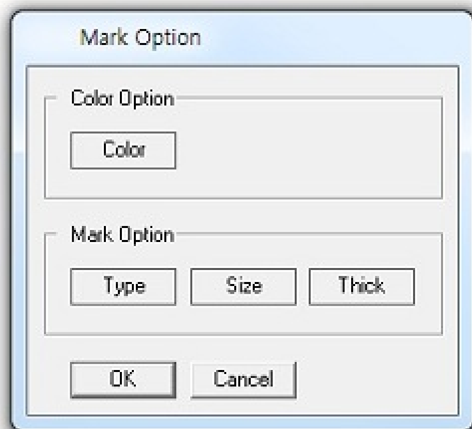
When you select Add Mark submenu, Mark Input dialog in Figure 5.31 is displayed.

Figure 5.31
Mark input
(Mouse Pickup)



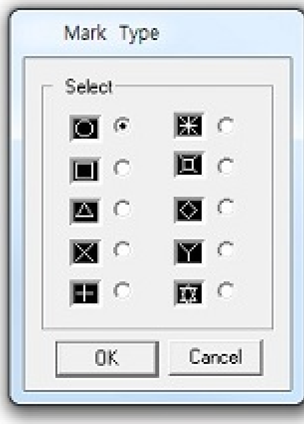
Option button is to show Mark Option in Figure 5.32.

Figure 5.32
Mark option dialog



Available Mark Types are shown in Figure 5.33.

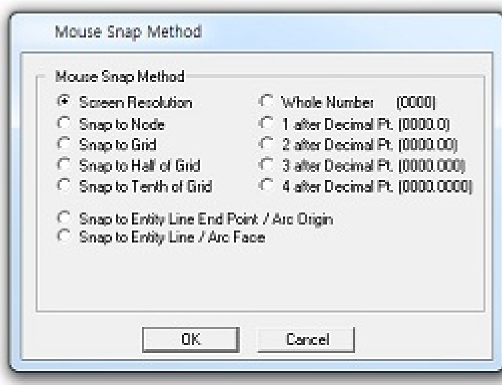
Figure 5.33 Mark type dialog



5.7.1.1 Mouse Pickup

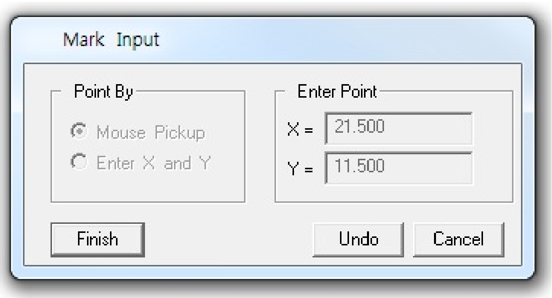
When you select Mouse Pickup mode as in Figure 5.31, you are supposed to select the mark center position by mouse click. Click Draw button and then move the position by Mouse Left Click. It is convenient to select an appropriate Mouse-Snap method in Figure 5.34 before moving the position.

Figure 5.34
Mouse snap method



Once finished, click Finish button in Figure 5.35.

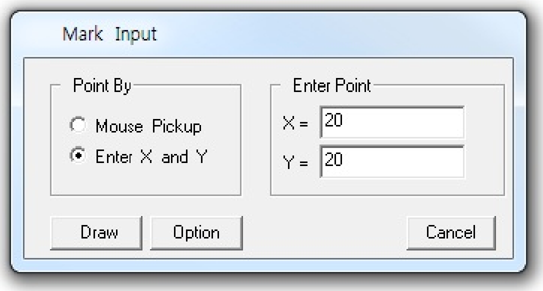
Figure 5.35
Mark input
(Finish)



5.7.1.2 Enter X and Y

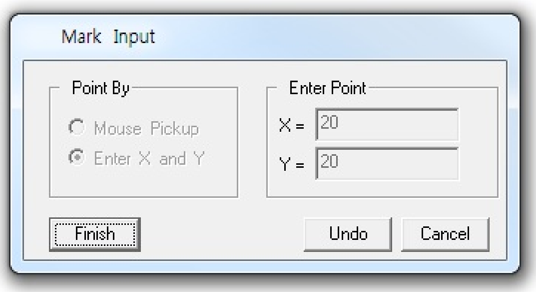
When you select Enter X and Y mode as in Figure 5.36, you are supposed to type in the coordinates of the mark center position. Click Draw button.

Figure 5.36
Mark input
(Enter X and Y)



Once finished, click Finish button in Figure 5.37.

Figure 5.37
Mark input
(Finish)

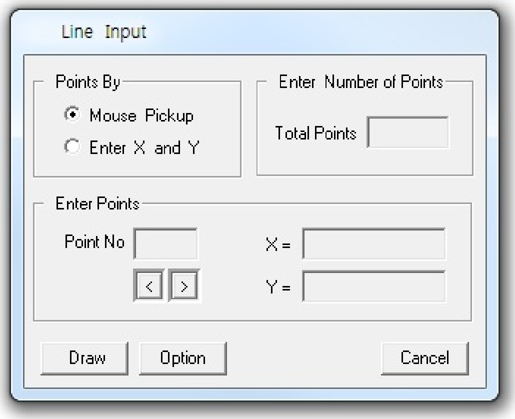


5.7.2 Add Line

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

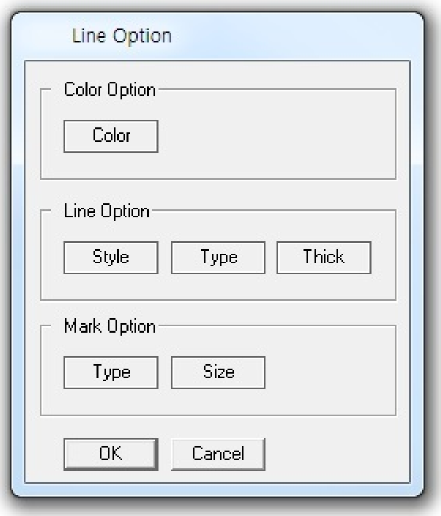
When you select Add Line submenu, Line Input dialog in Figure 5.38 is displayed.

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.

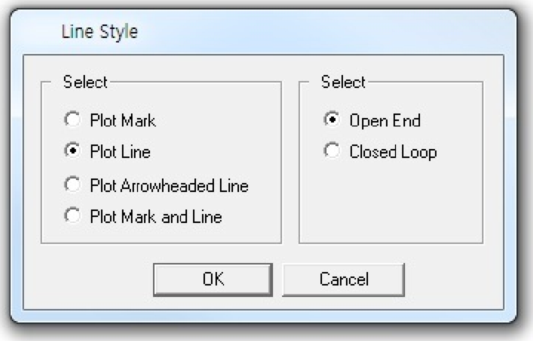


Figure 5.40 Line style dialog

Available Line Types are shown in Figure 5.41.

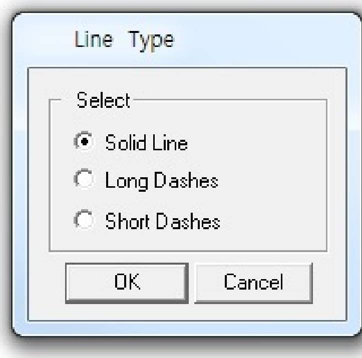


Figure 5.41 Line type dialog

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.

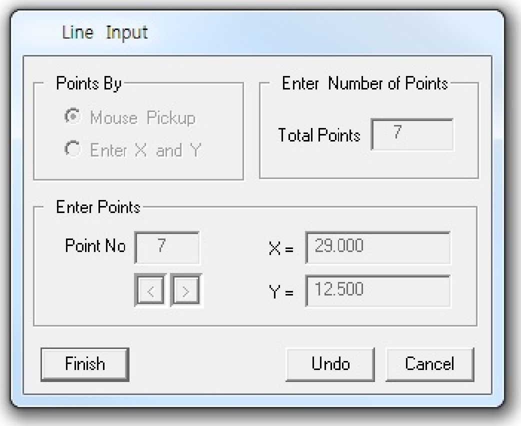
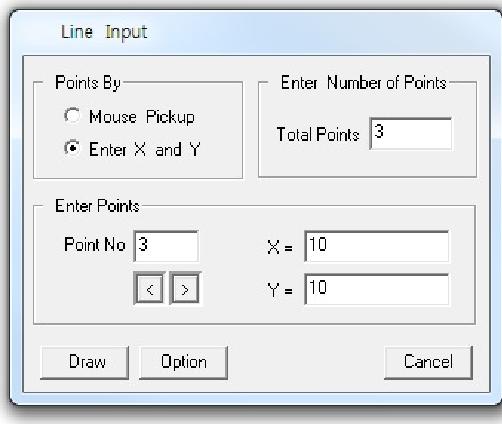


Figure 5.42 Line input (Finish)

5.7.2.2 Enter X and Y

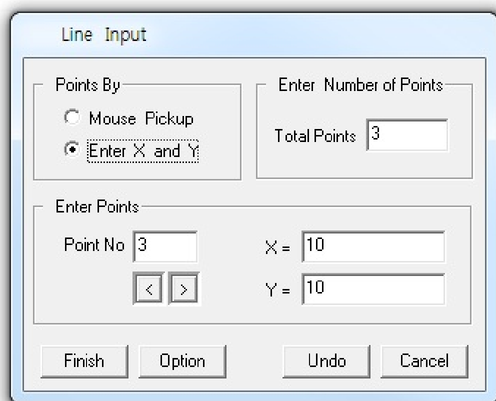
When you select Enter X and Y mode as in Figure 5.43, you are supposed to type the coordinates of the line. Click Draw button.

Figure 5.43
Line input
(Enter X and Y)



And then click Finish button in Figure 5.44.

Figure 5.44
Line input
(Finish)

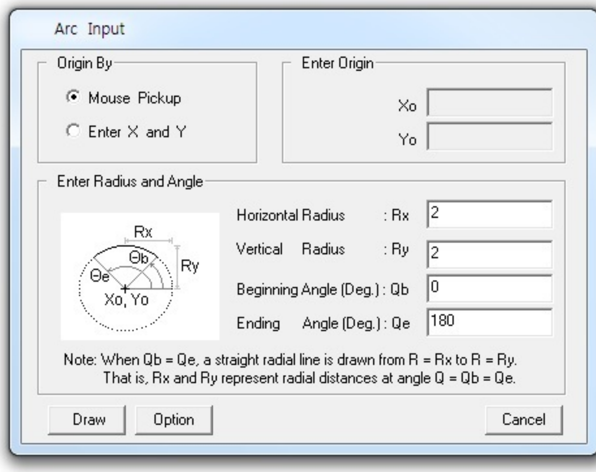


5.7.3 Add Arc

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

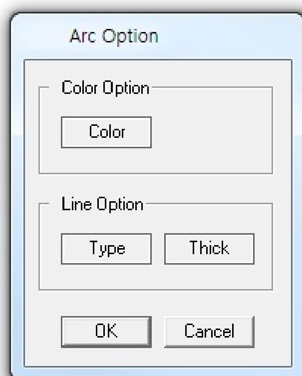
When you select Add Arc submenu, Arc Input dialog in Figure 5.45 is displayed.

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.

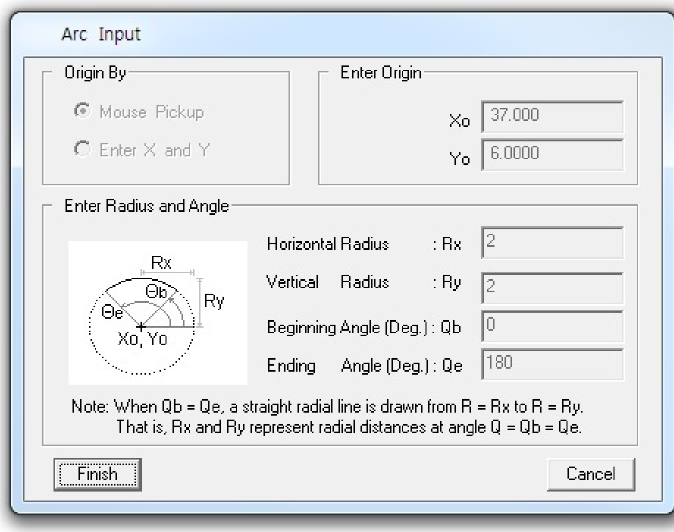


Figure 5.47 Arc input (Finish)

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.

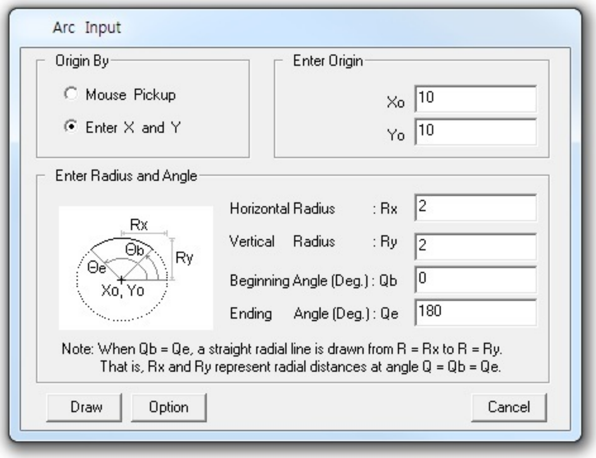


Figure 5.48
Arc input
(Enter X and Y)

Once finished, click Finish button in Figure 5.49.

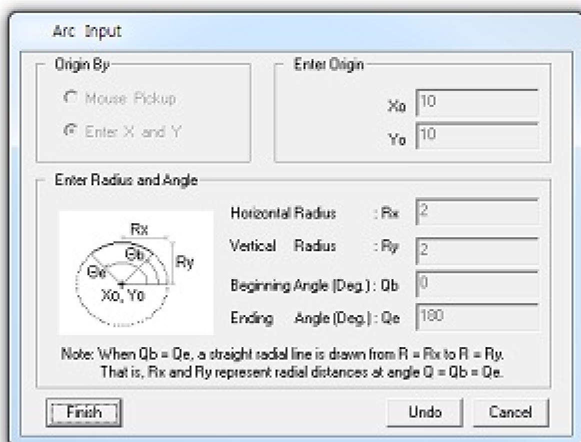


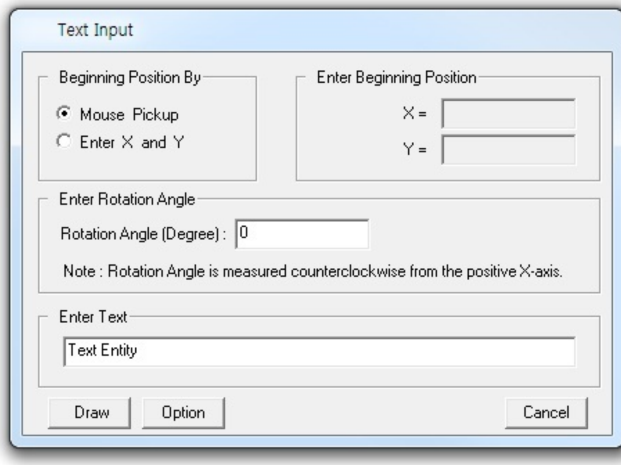
Figure 5.49
Arc input
(Finish)

5.7.4 Add Text

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

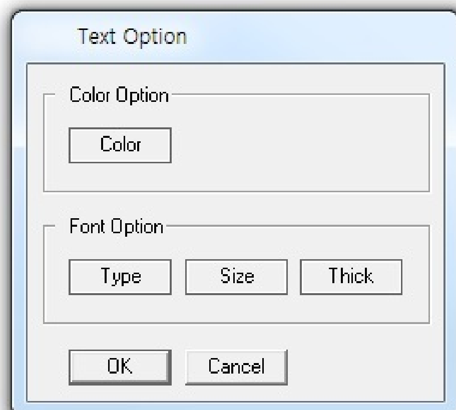
When you select Add Text submenu, Text Input dialog in Figure 5.50 is displayed.

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.

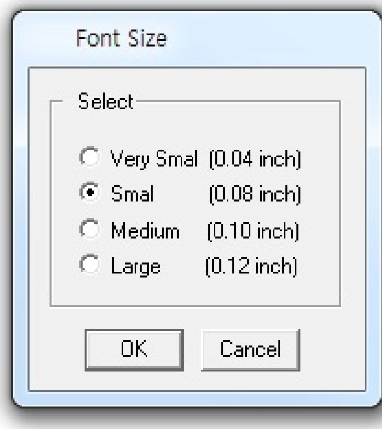


Figure 5.52 Font size dialog

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.

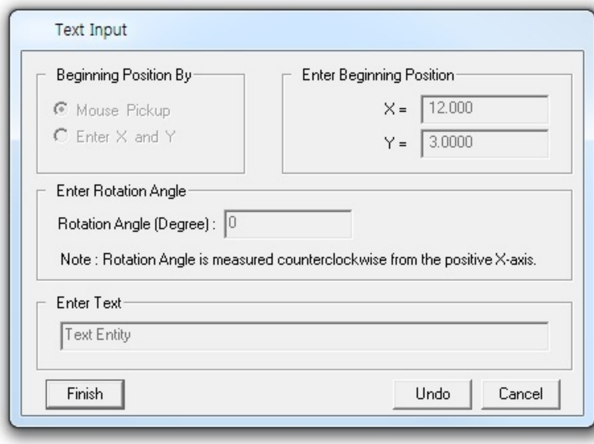


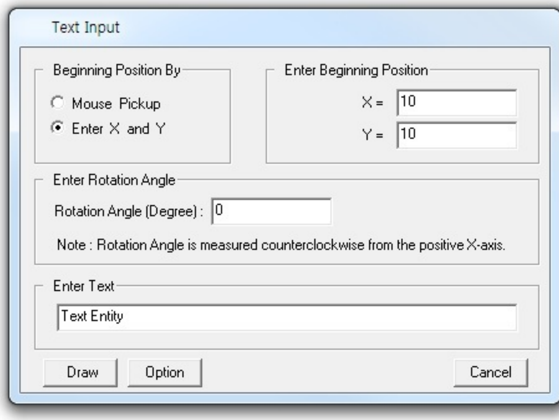
Figure 5.53 Text input (Finish)

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.

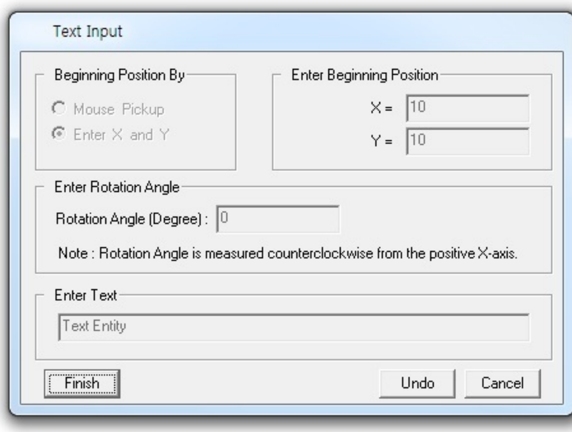
Type in Rotation Angle and Text. And then click Draw button.

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.

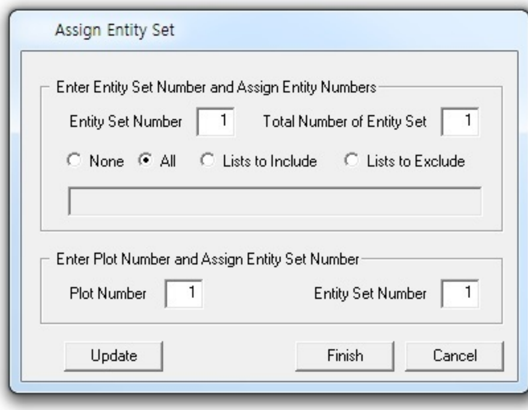


Figure 5.56
Assign entity set dialog

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.

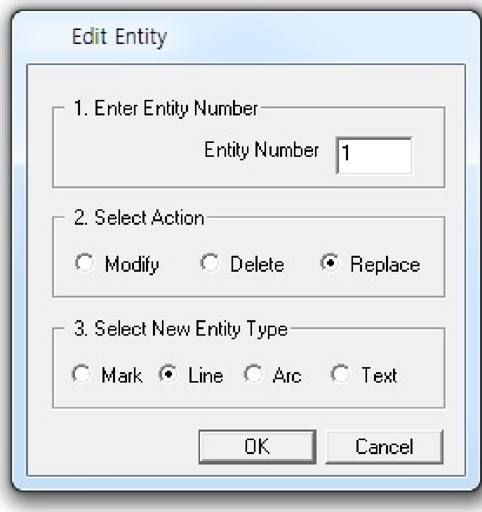


Figure 5.57
Edit entity dialog

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.

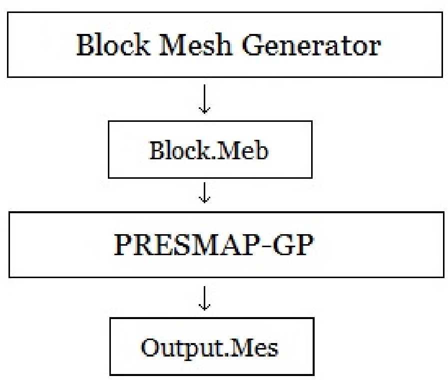


Figure 6.1 Flow diagram of block mesh generation

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.

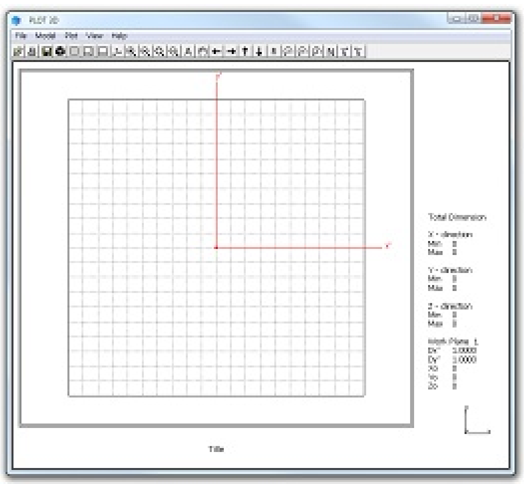


Figure 6.2 Prebuilt work plane on PLOT-3D

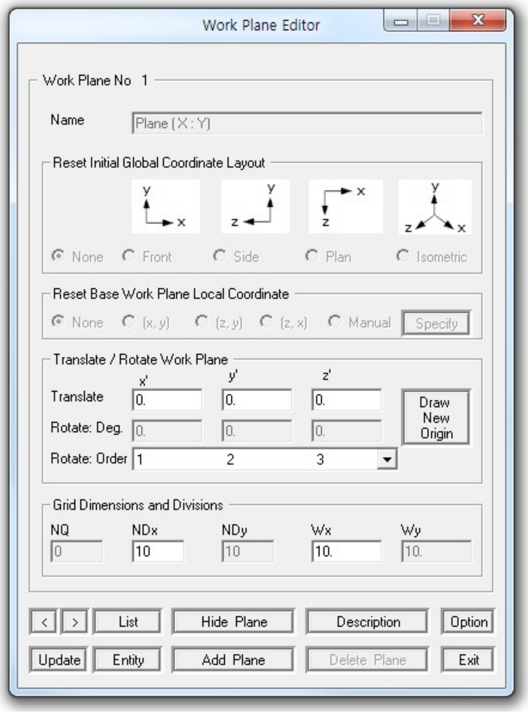


Figure 6.3 Prebuilt work plane editor

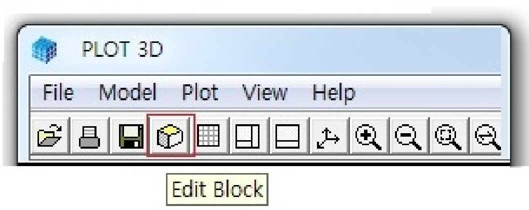


Figure 6.4 Block editor toolbar

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.

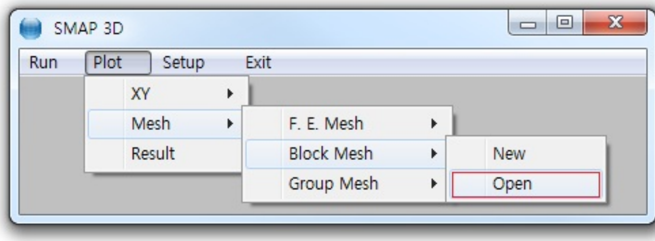


Figure 6.5 Open input file dialog

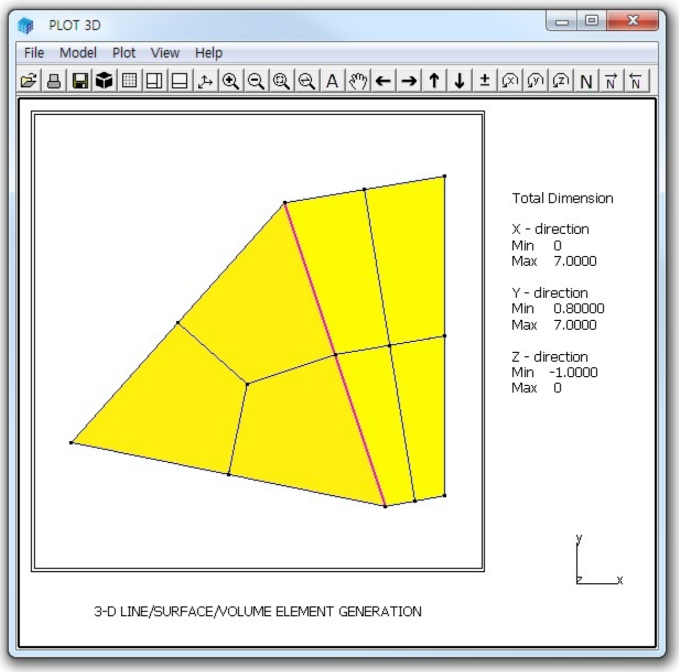


Figure 6.6 Block mesh on PLOT-3D

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.

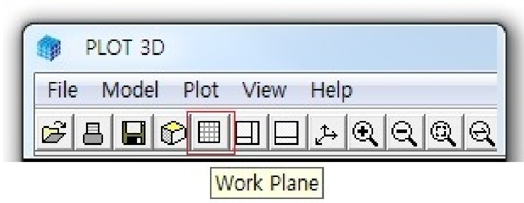


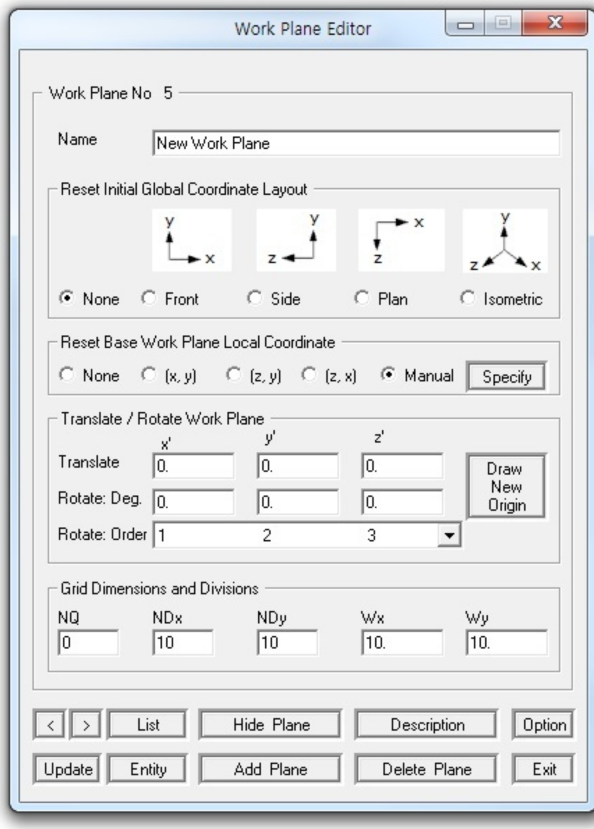
Figure 6.7 Work plane toolbar

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.

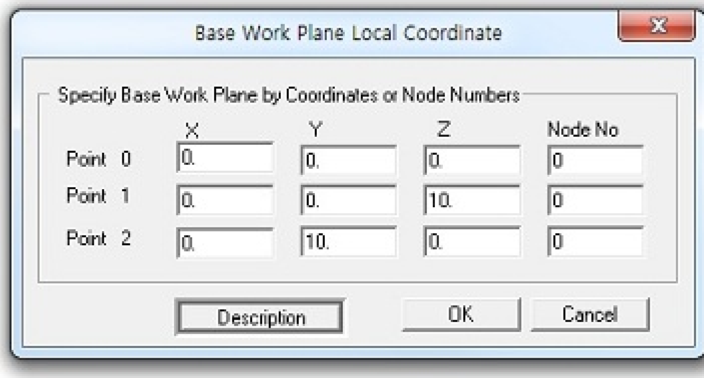


Figure 6.9 Base work plane local coordinate dialog

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.

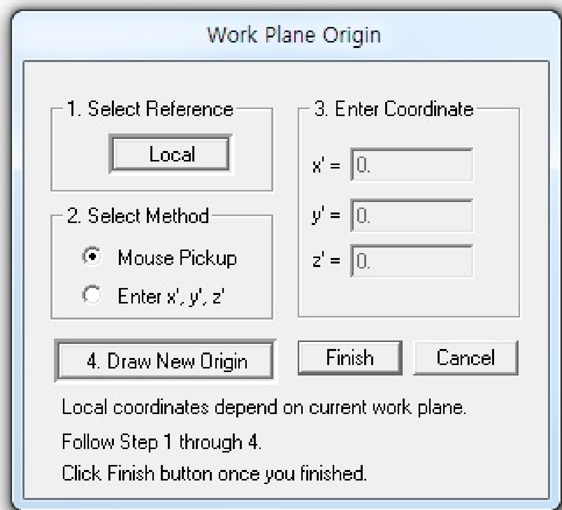


Figure 6.10 Work plane origin dialog

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.

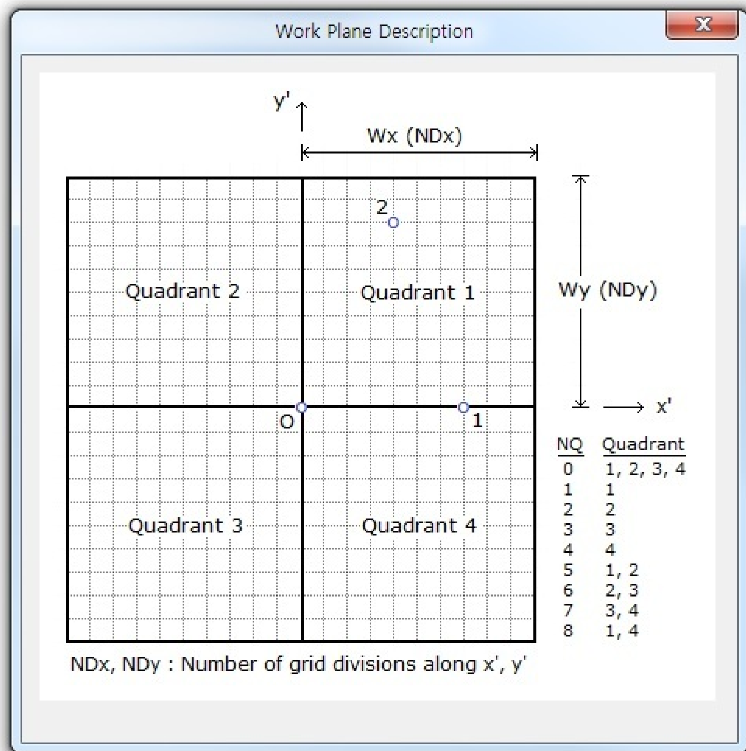


Figure 6.11 Work plane description

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.

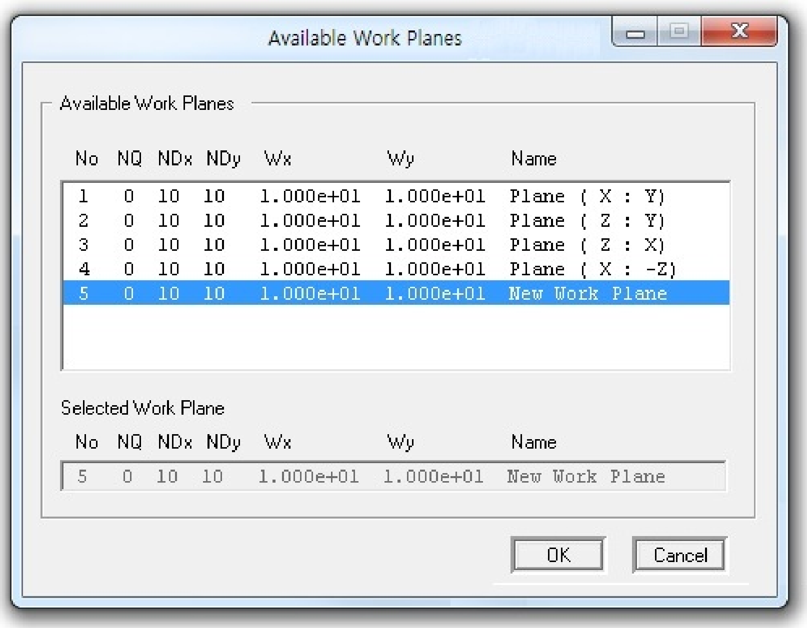


Figure 6.12 Work plane list

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.

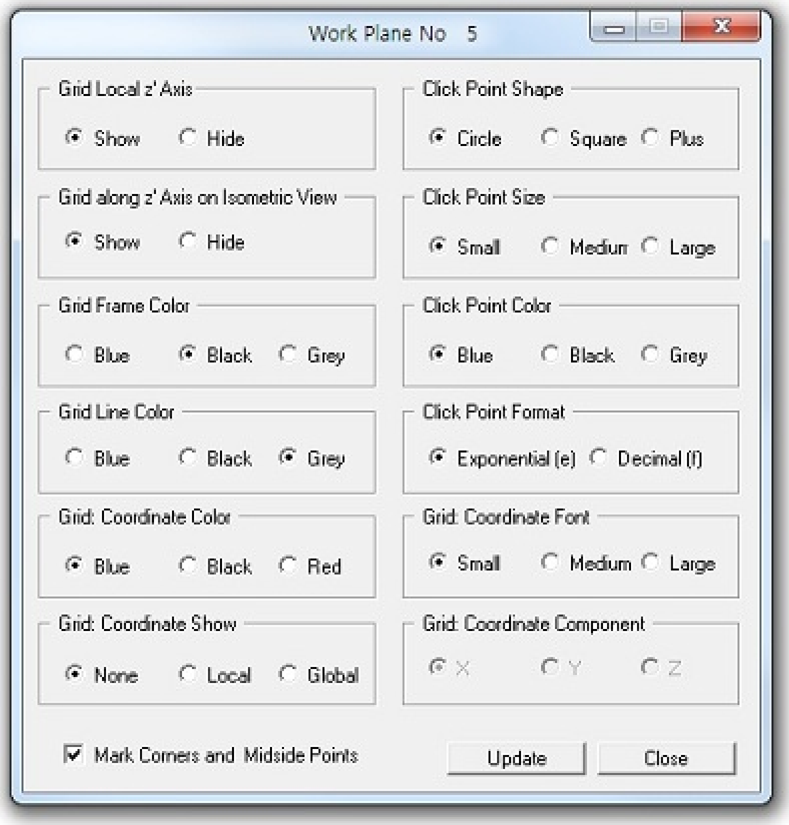


Figure 6.13 Work plane option dialog

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

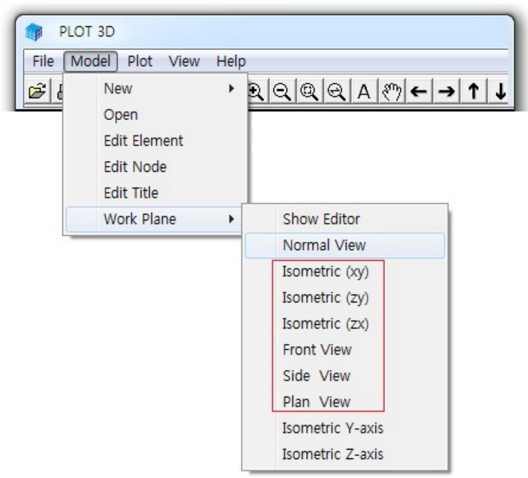


Figure 6.14 Prebuilt work plane menus

or by clicking Axis toolbar as shown in Figure 6.15.

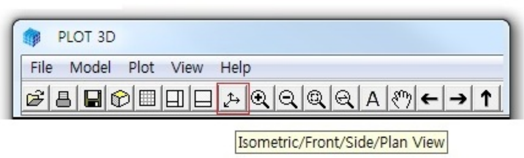


Figure 6.15 Axis toolbar

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.

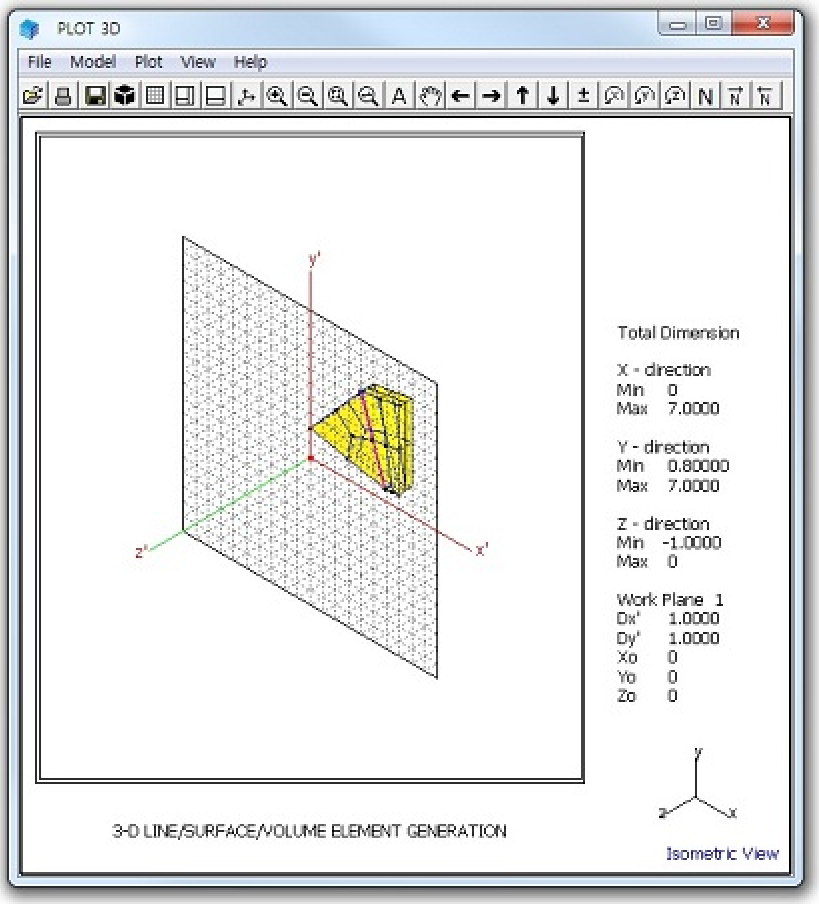


Figure 6.16 Prebuilt work plane: Isometric (xy) View

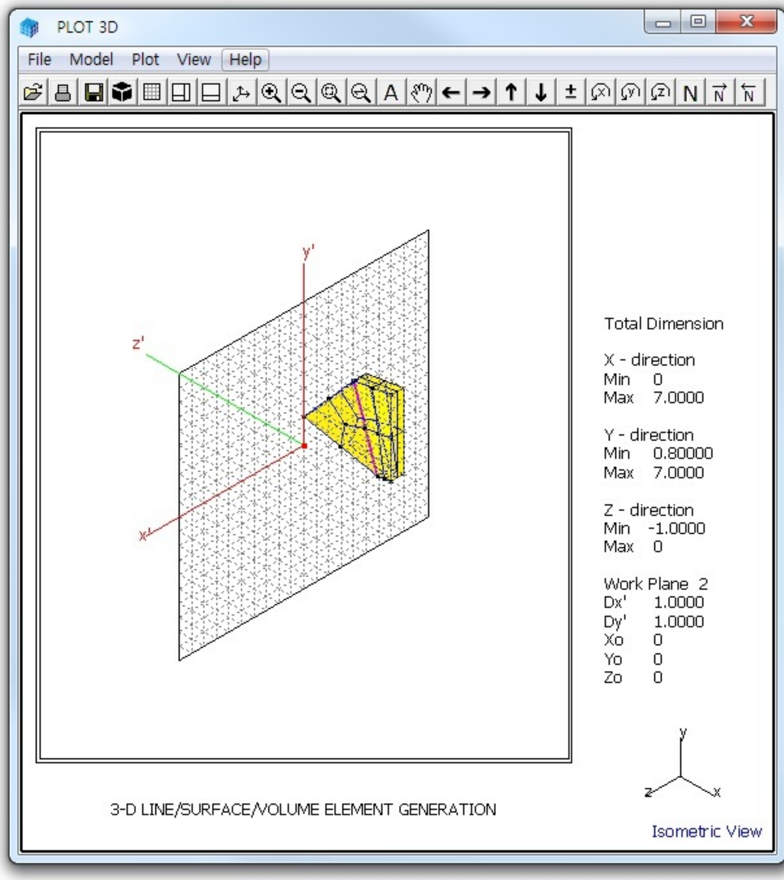


Figure 6.16 Prebuilt work plane: Isometric (zy) View

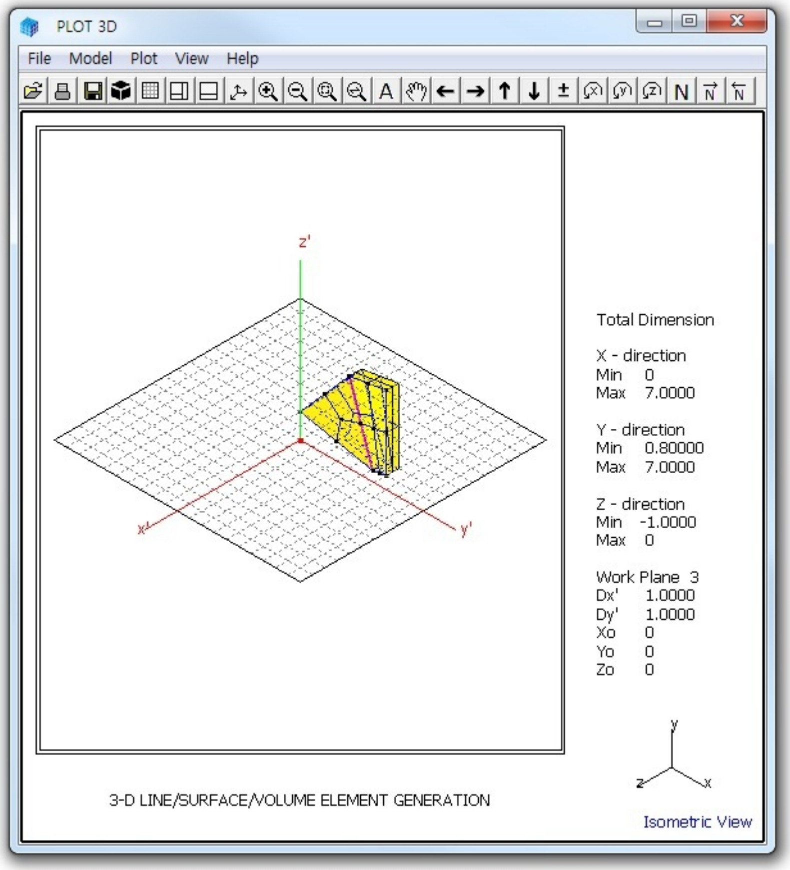


Figure 6.16 Prebuilt work plane: Isometric (zx) View

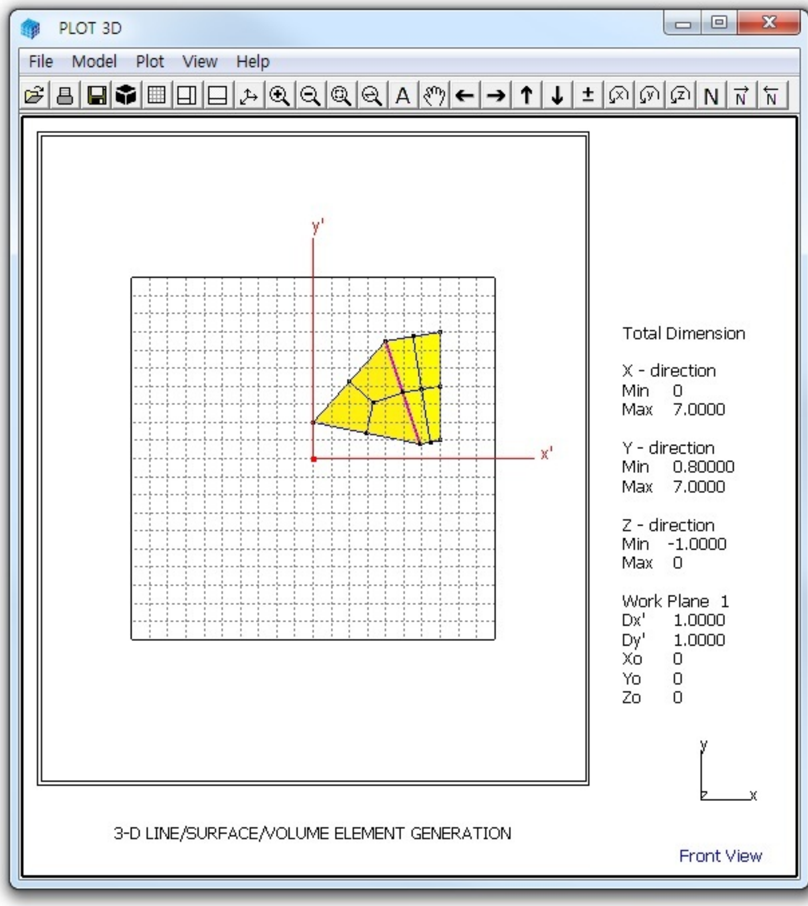


Figure 6.16 Prebuilt work plane: Front View

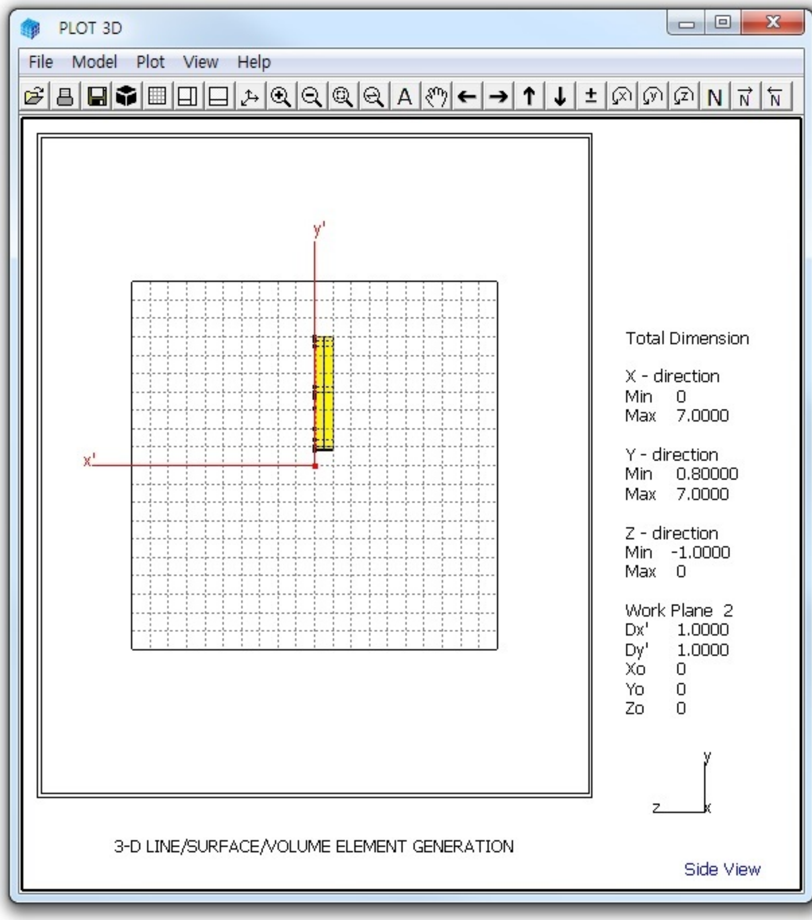


Figure 6.16 Prebuilt work plane: Side View

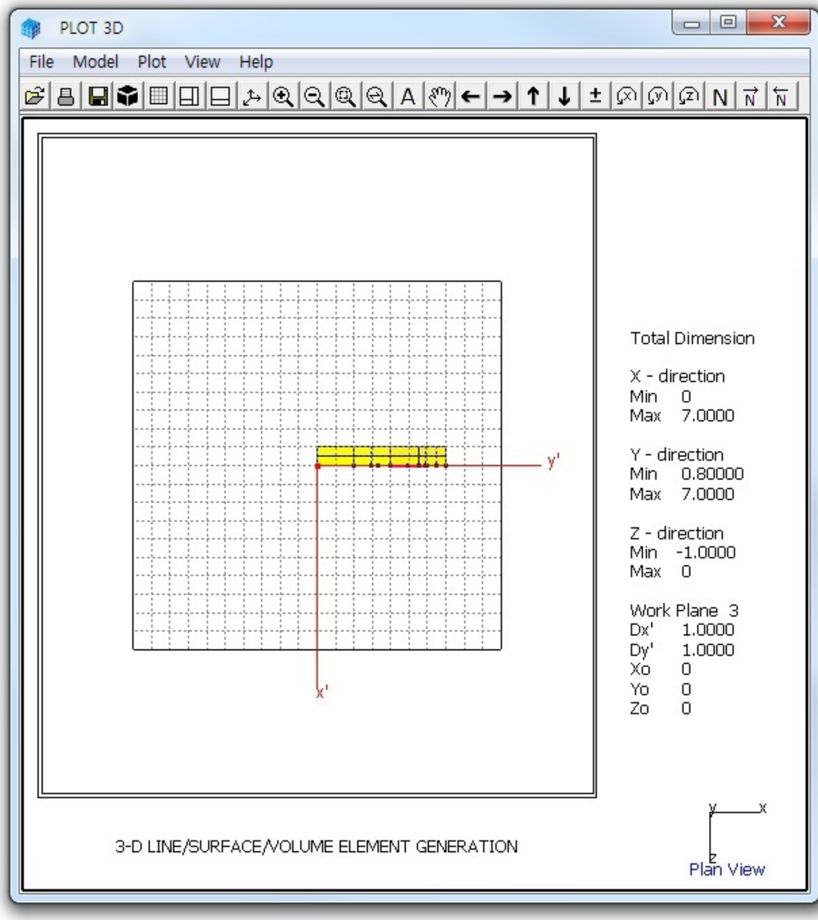


Figure 6.16 Prebuilt work plane: Plan View

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

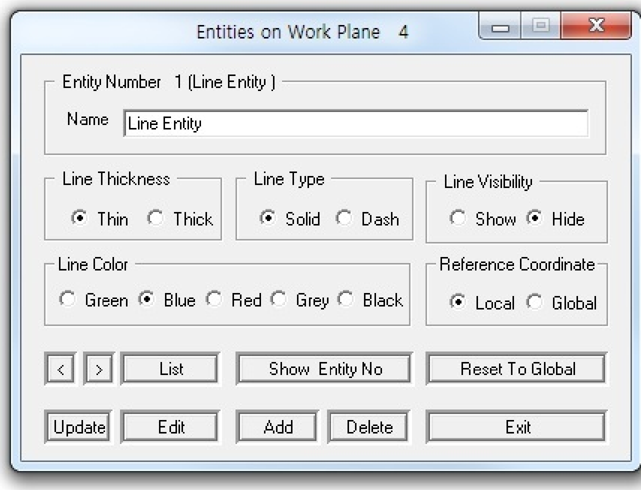


Figure 6.17 Entity editor dialog

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.

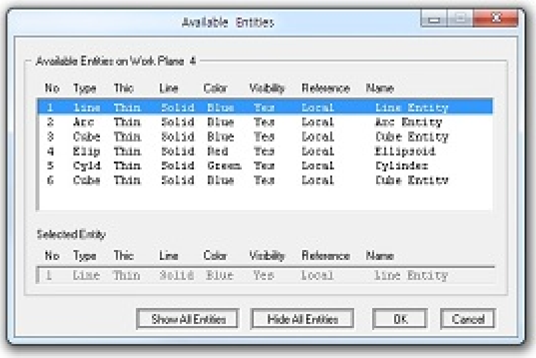


Figure 6.18
Entity list dialog

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.

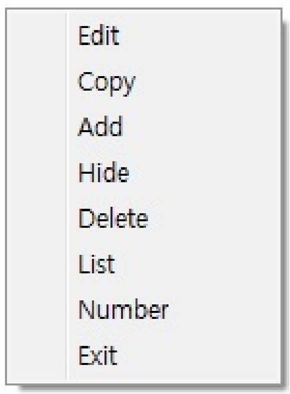


Figure 6.19 Popup menu for entity

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.

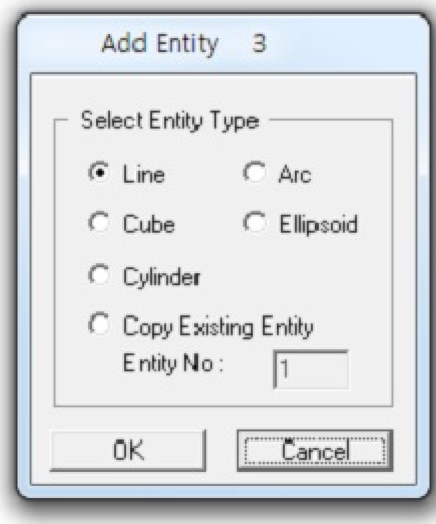


Figure 6.20 Entity type selection dialog

6.4.9.1 Line Entity

Line Entity dialog is shown in Figure 6.21.

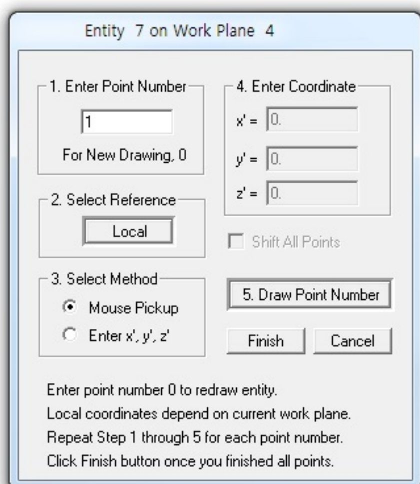
To draw Line Entity, follow five steps:

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



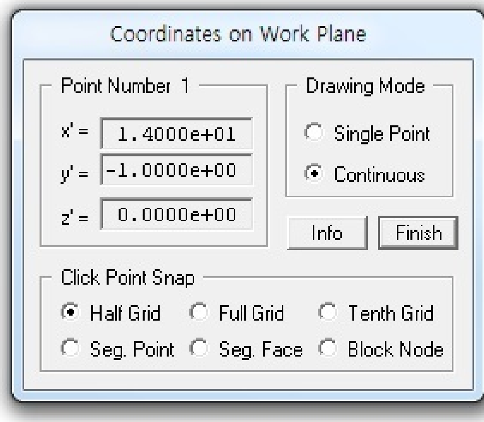


Figure 6.22 Coordinates on work plane

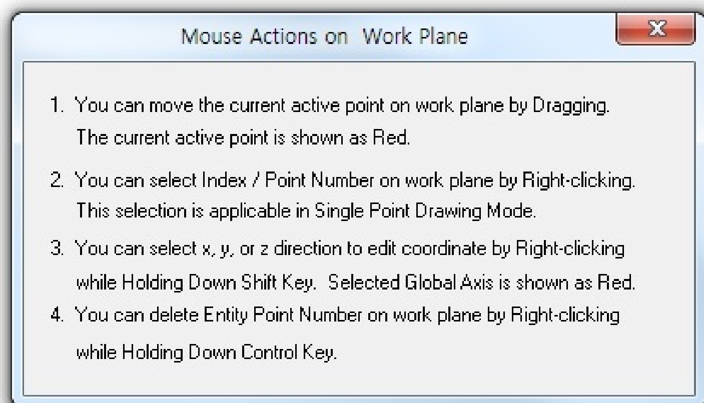


Figure 6.23 Mouse actions on work plane

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.

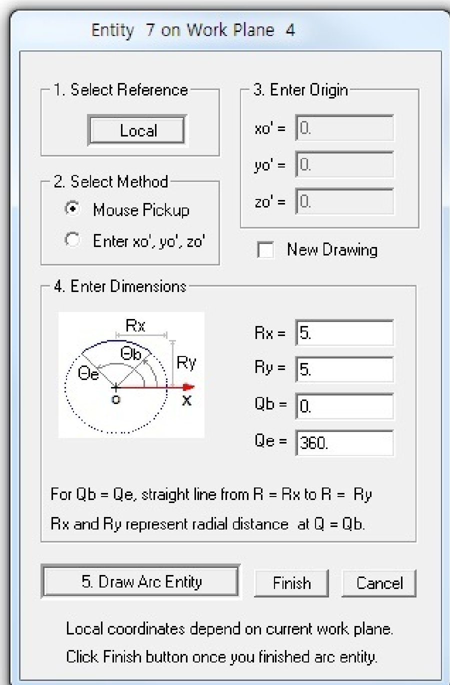


Figure 6.24
Arc entity dialog

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.

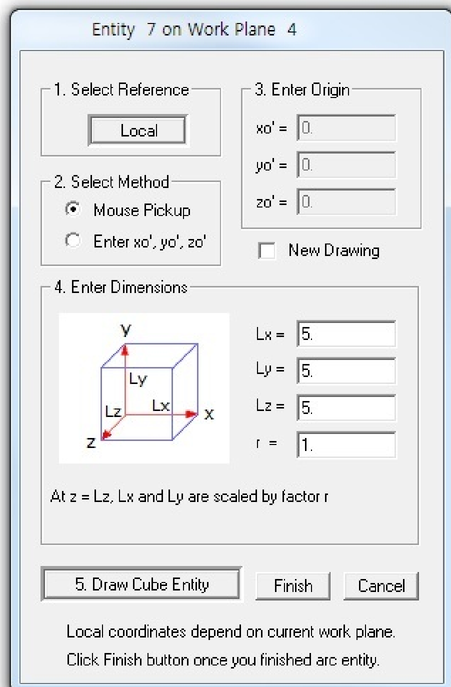


Figure 6.25
Cube entity dialog

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.

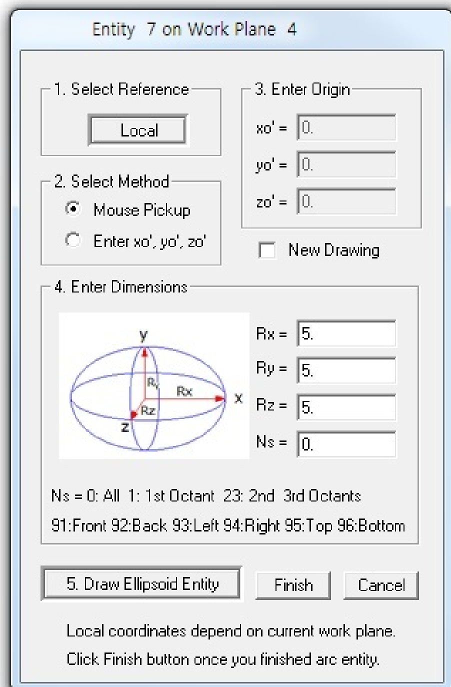


Figure 6.26
Ellipsoid entity dialog

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.

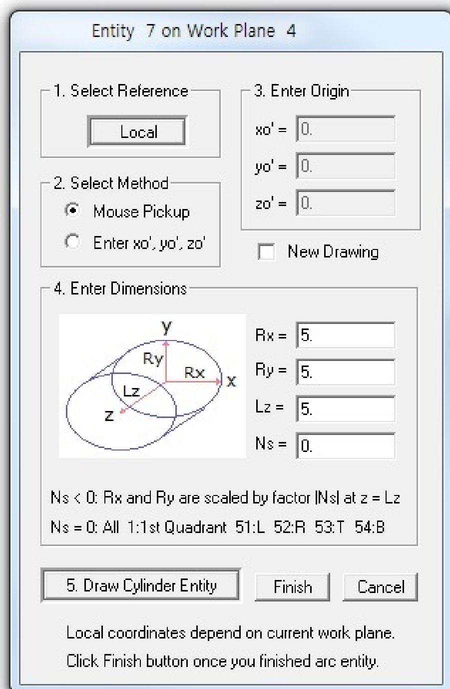


Figure 6.27
Cylinder entity dialog

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.

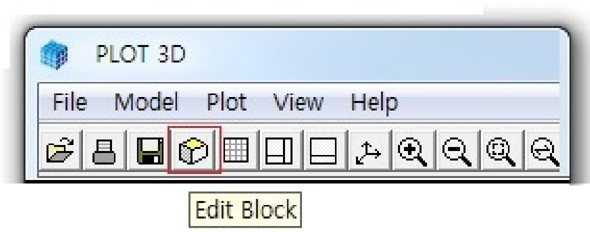


Figure 6.28 Block editor toolbar

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

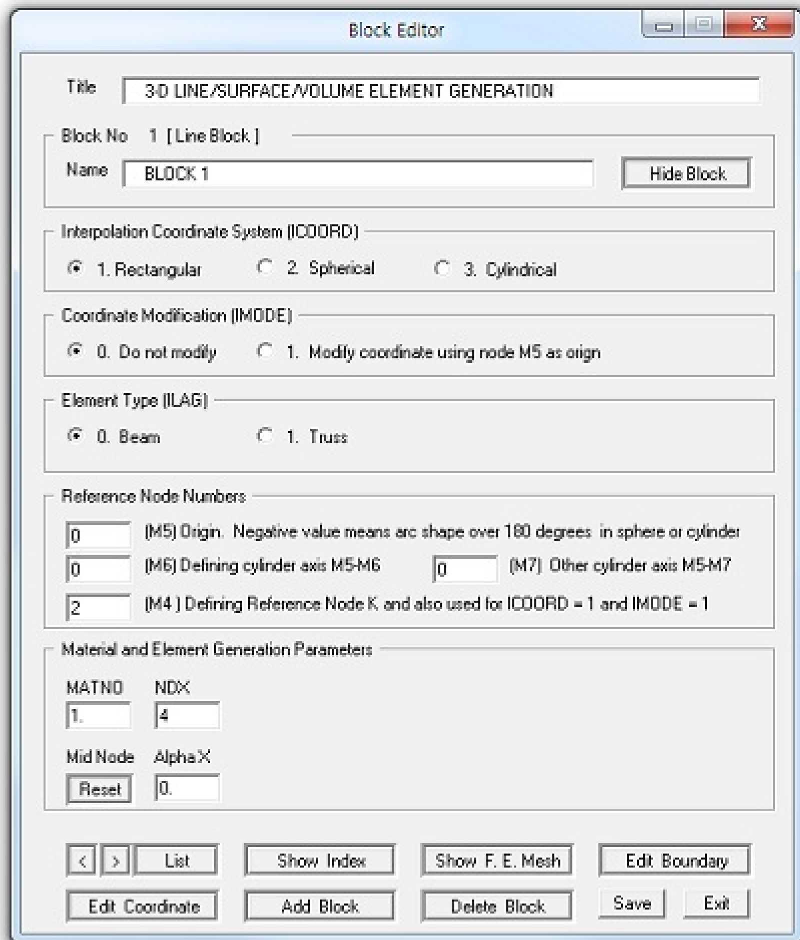


Figure 6.29 Block editor (Line Block)

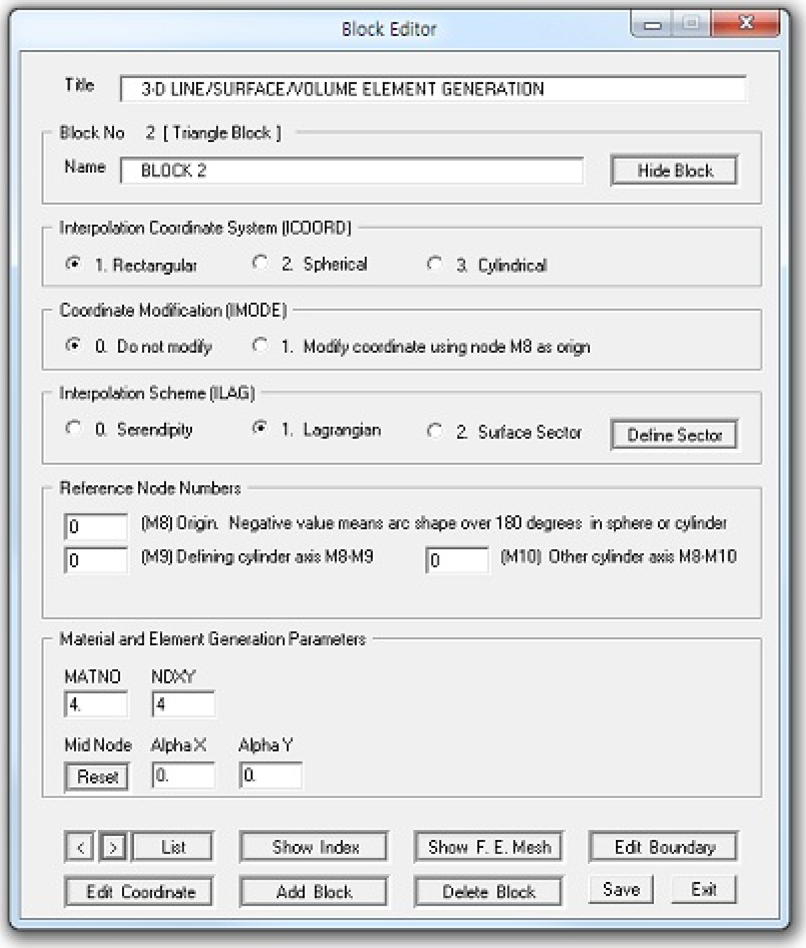


Figure 6.29 Block editor (Triangle Block)

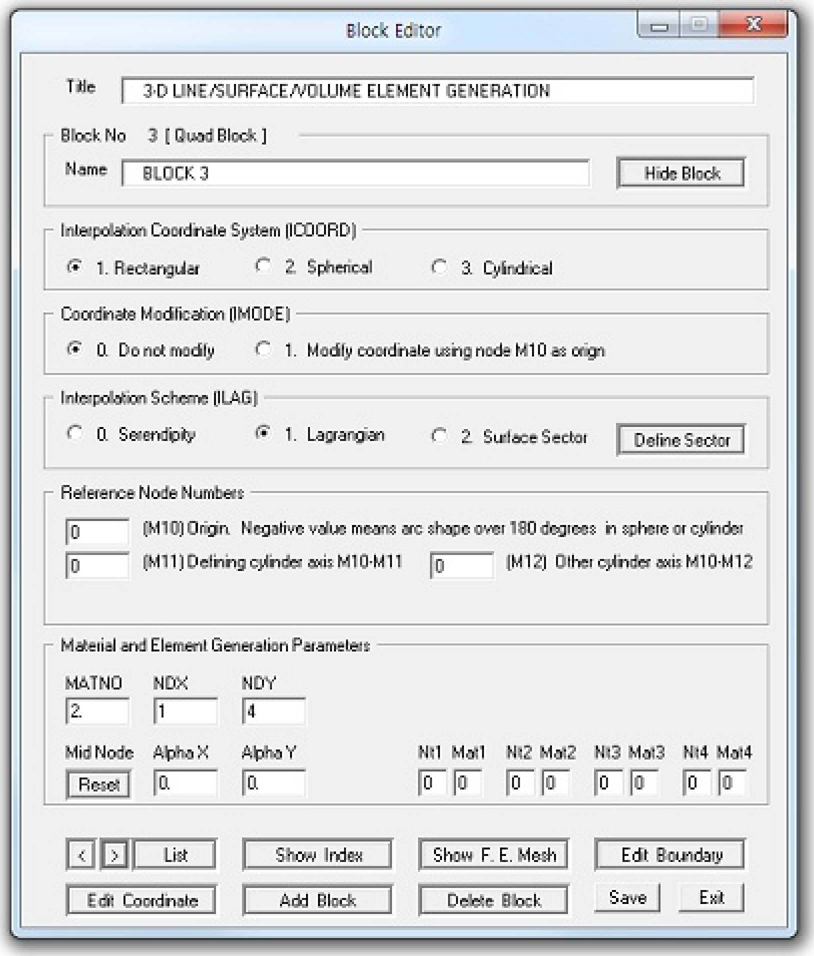


Figure 6.29 Block editor (Quad Block)

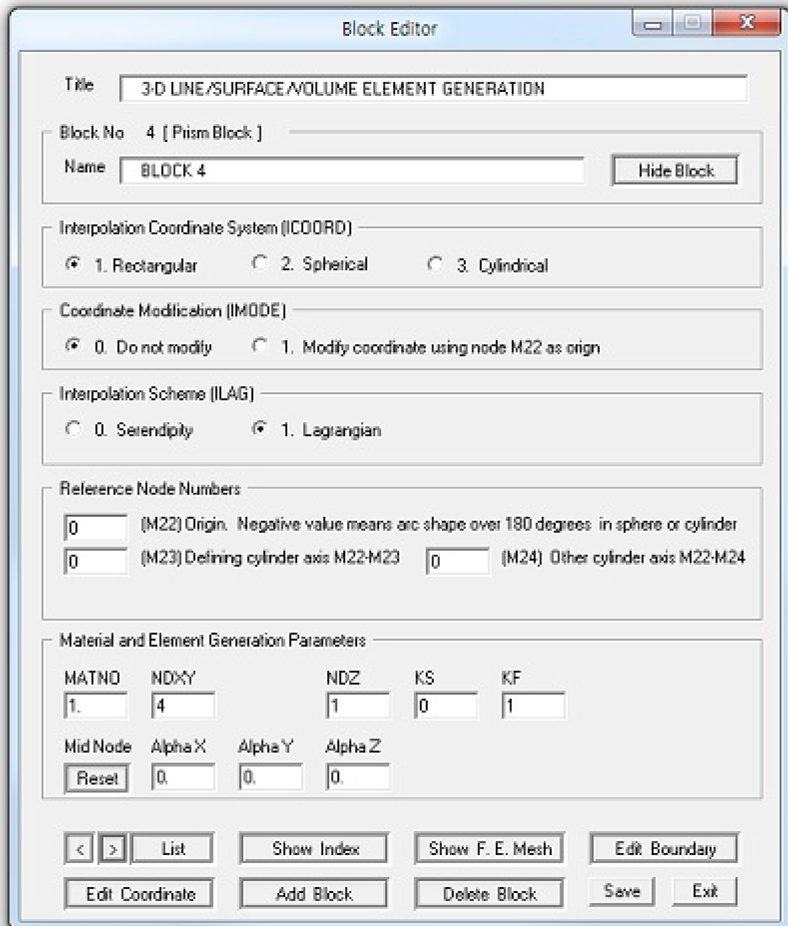


Figure 6.29 Block editor (Prism Block)

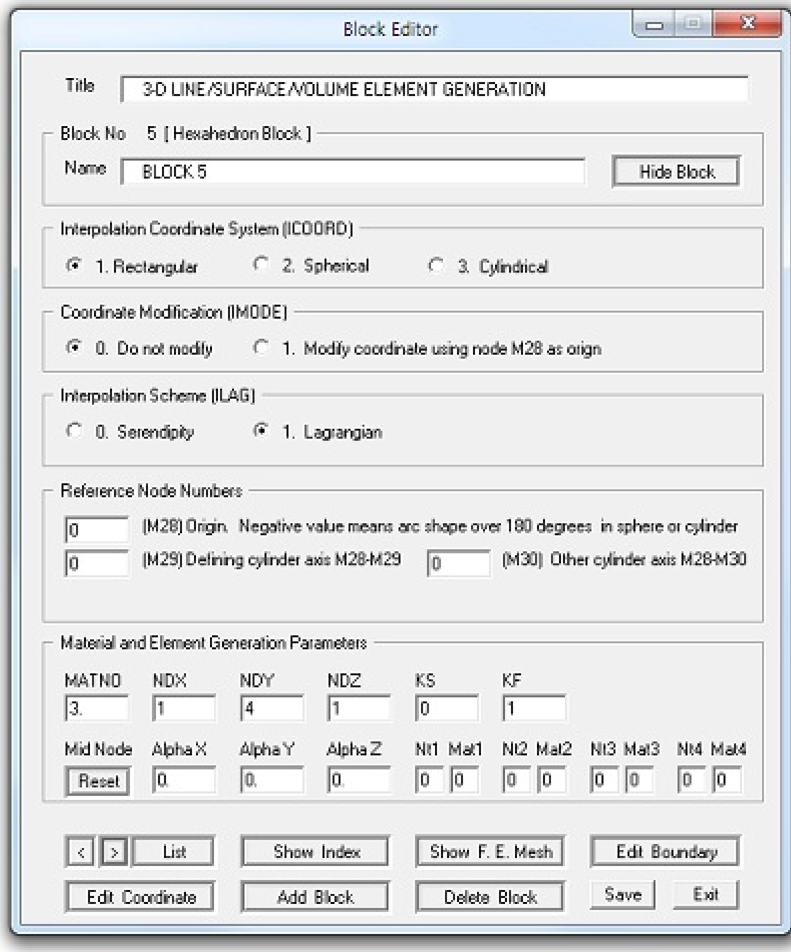


Figure 6.29 Block editor (Hexahedron Block)

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.

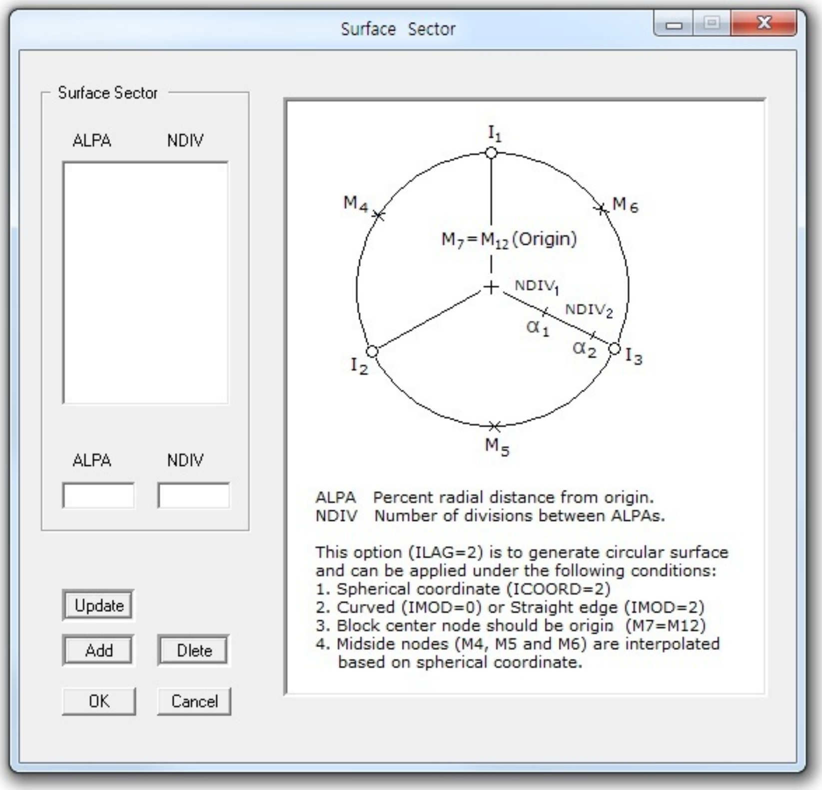


Figure 6.30 Surface sector (Triangle Block)

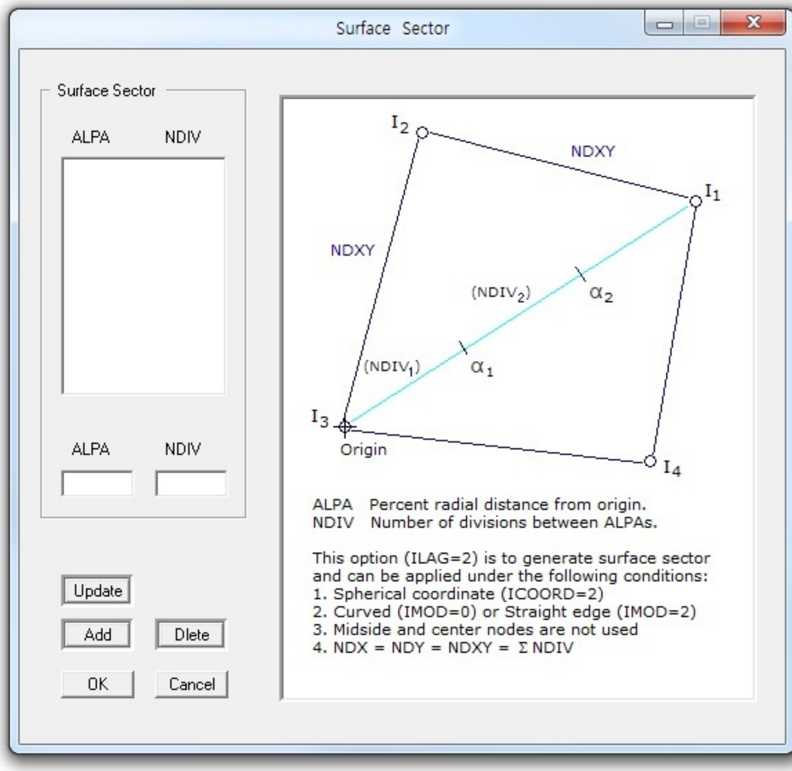


Figure 6.30 Surface sector (Quad Block)

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.

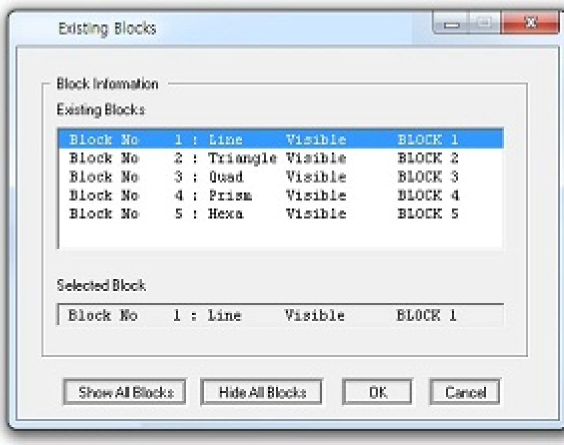


Figure 6.31 Block list

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.

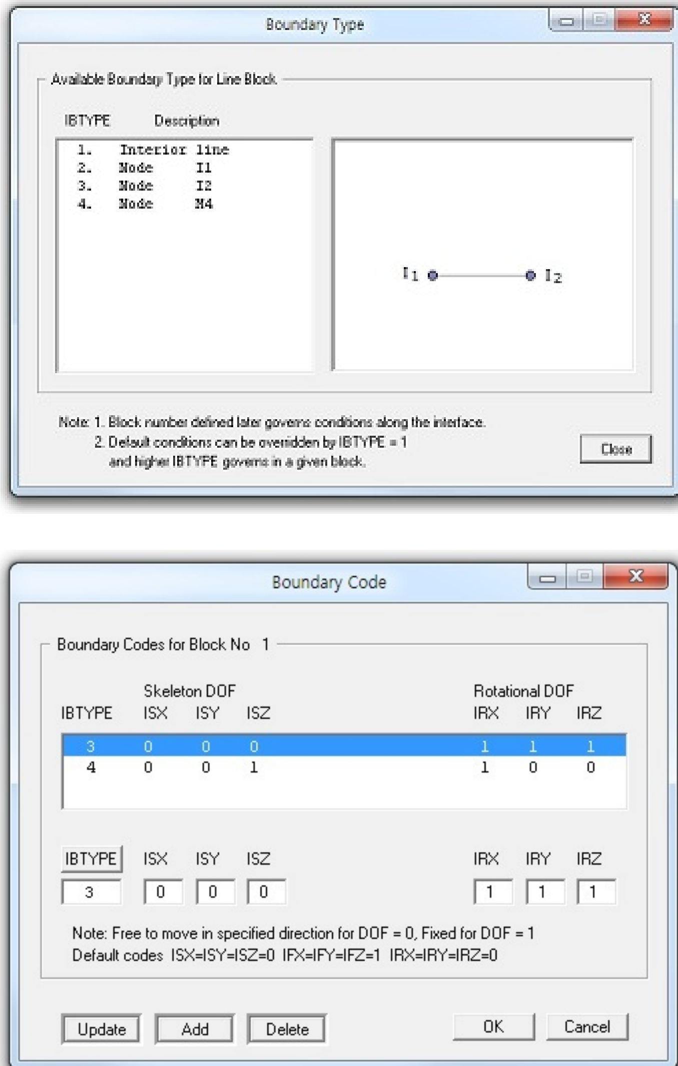


Figure 6.32 Boundary code (Line Block)

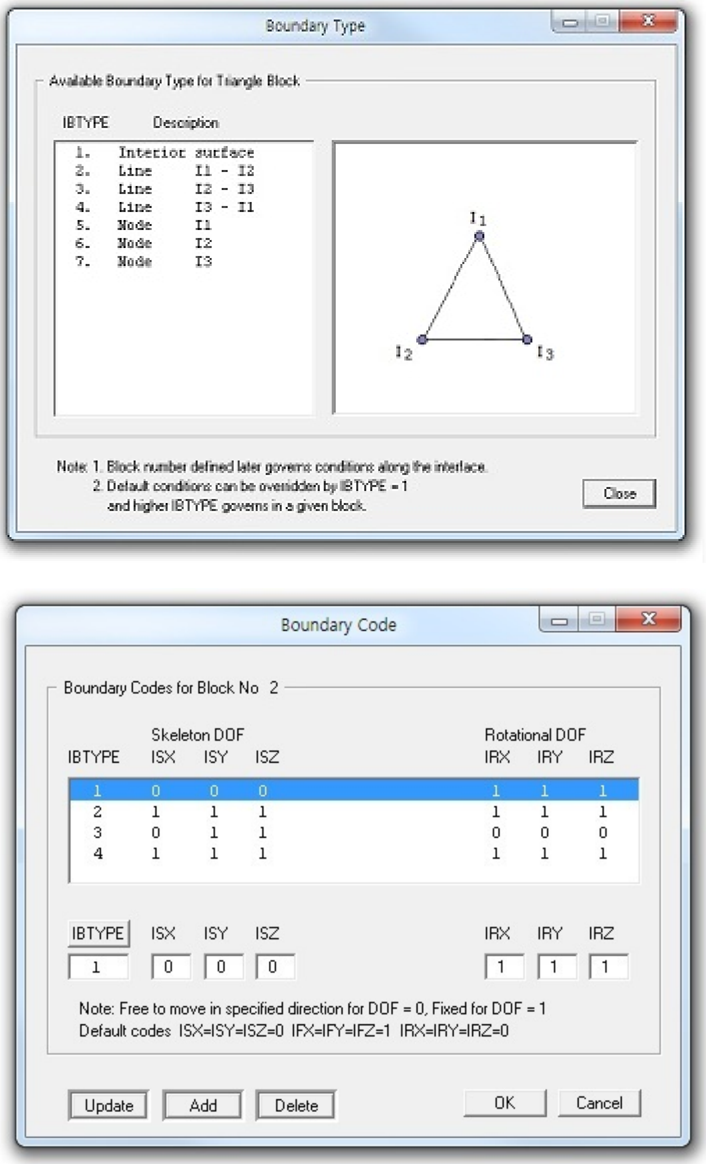


Figure 6.32 Boundary code (Triangle Block)

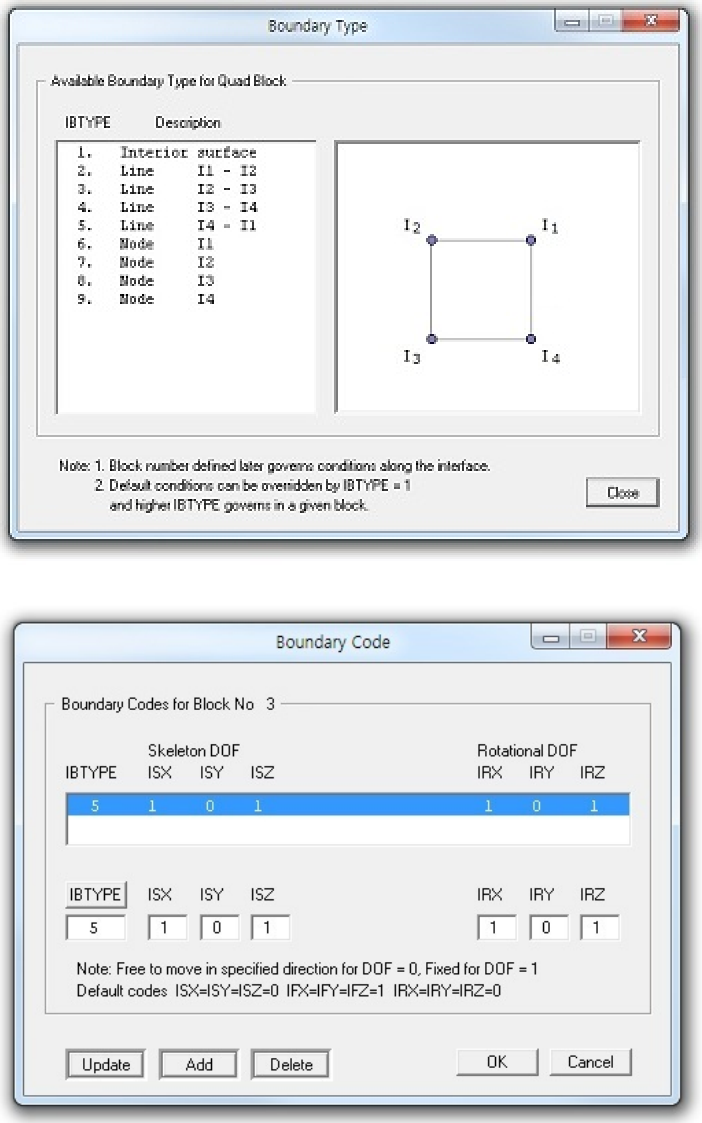


Figure 6.32 Boundary code (Quad Block)

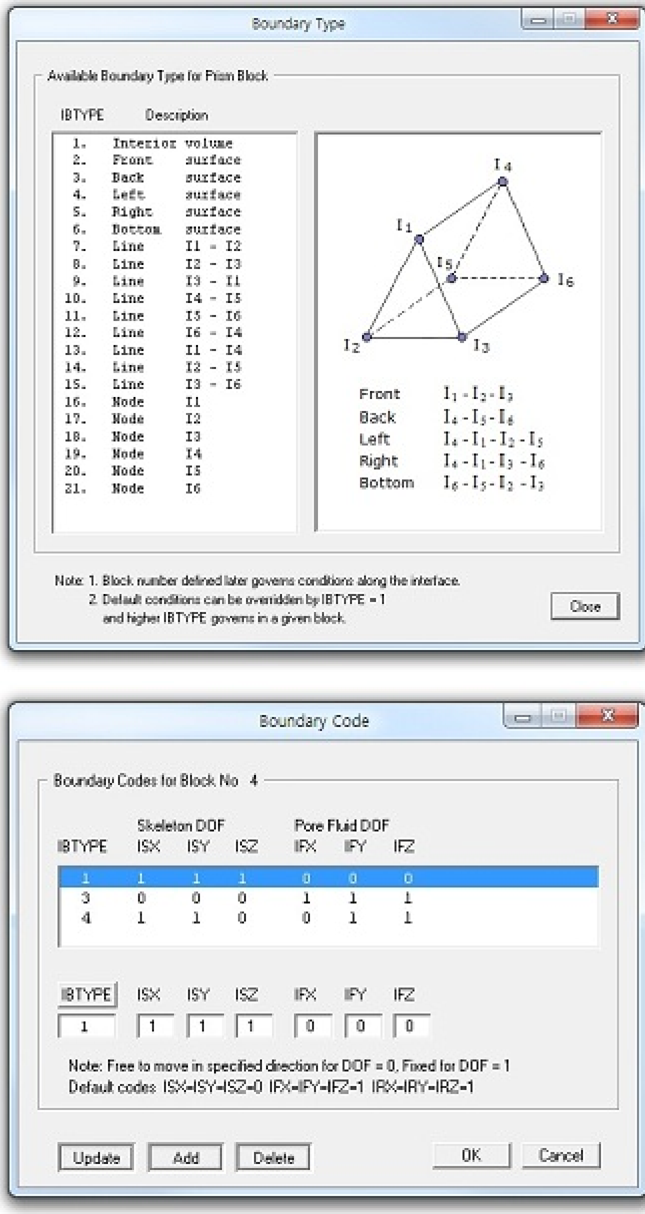


Figure 6.32 Boundary code (Prism Block)

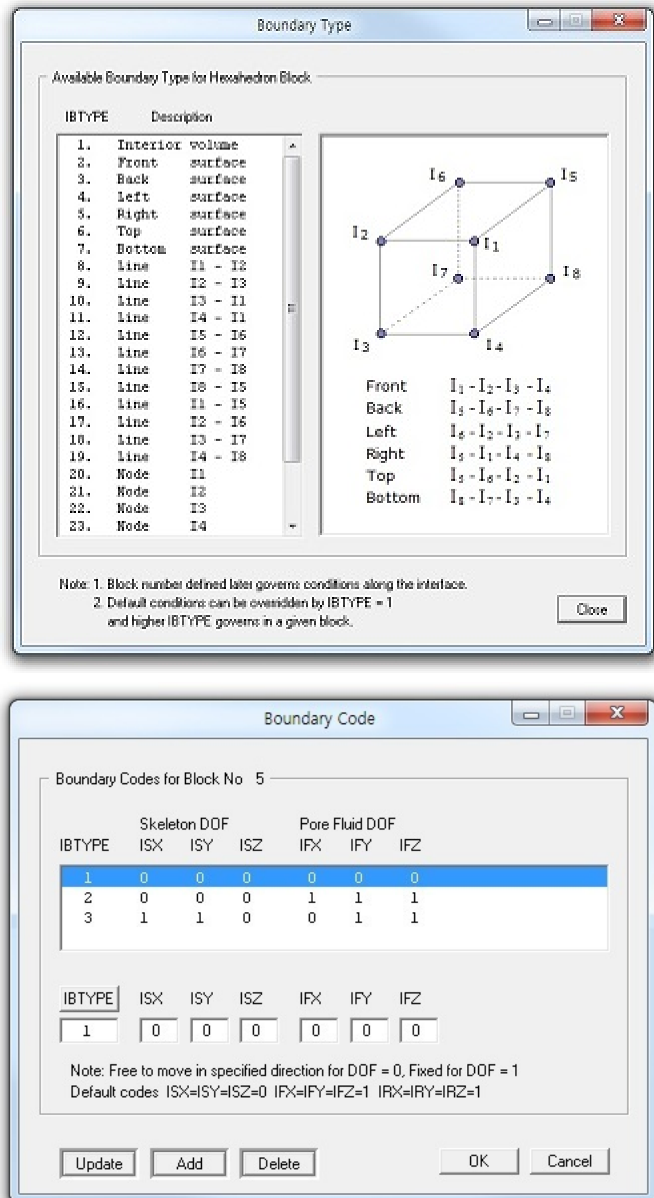


Figure 6.32 Boundary code (Hexahedron Block)

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.

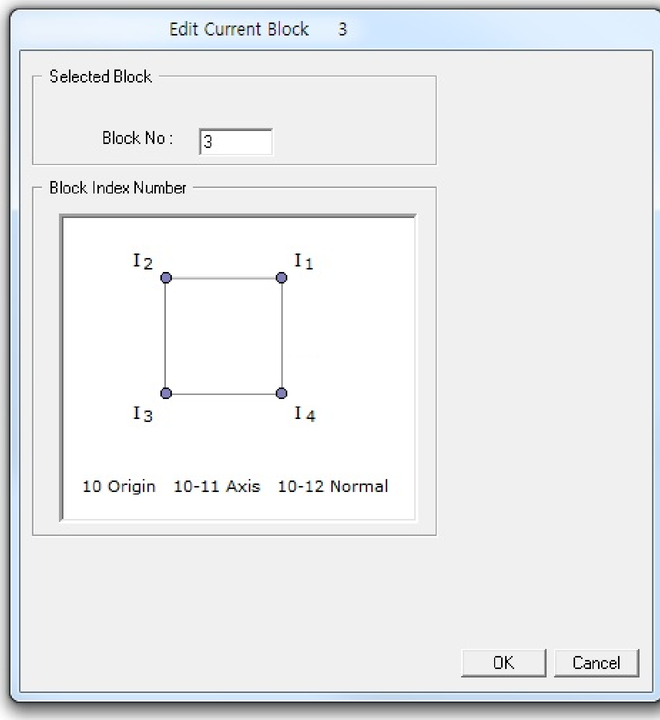


Figure 6.33 Edit current block (Selection Mode)

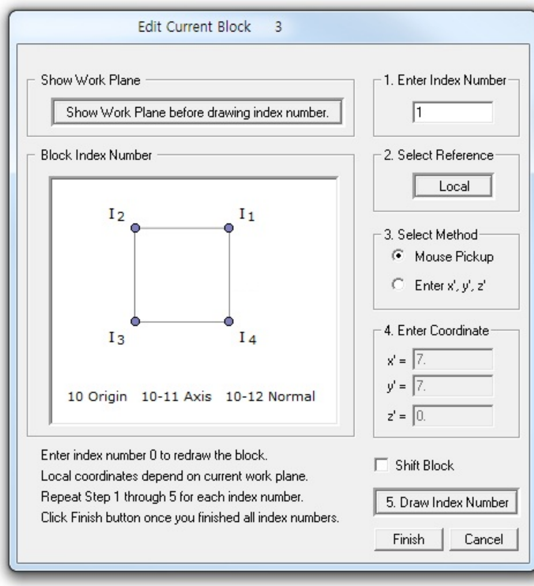
Edit Current Block dialog now shows input parameters required to edit the geometry of the block as shown in Figure 6.34.

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)



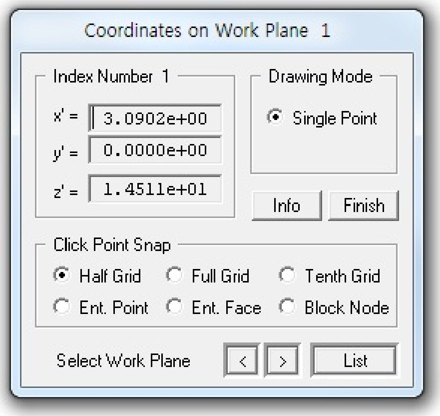


Figure 6.35 Coordinates on work plane

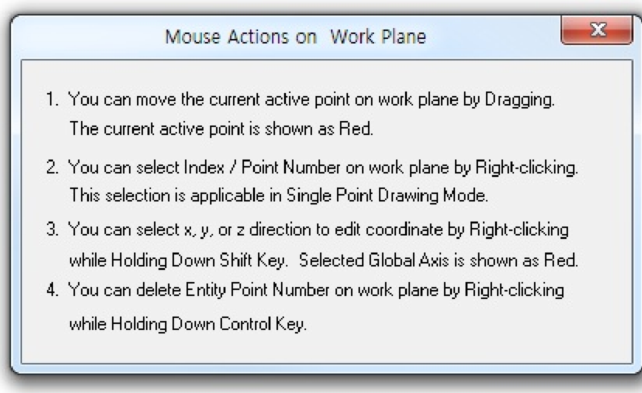


Figure 6.36 Mouse actions on work plane

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.

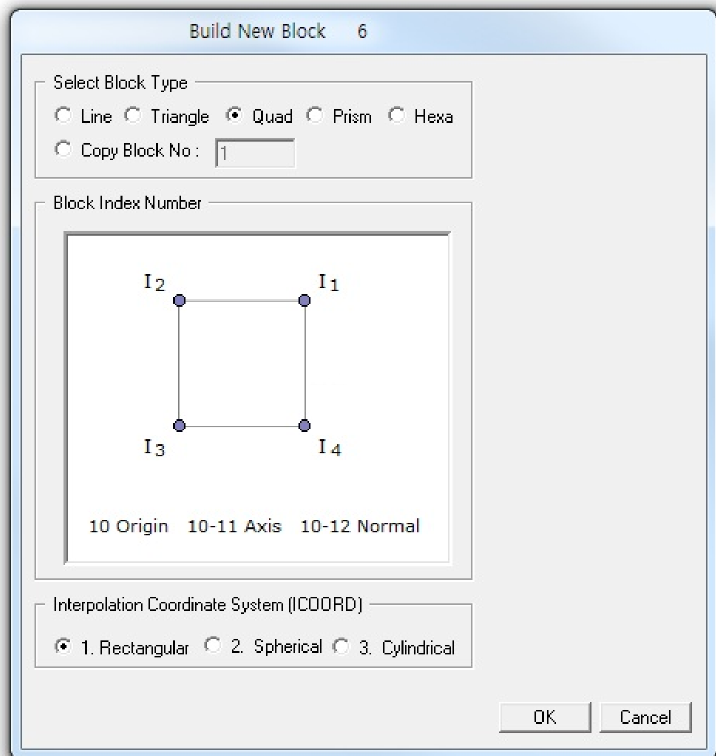


Figure 6.37 Build new block (Selection Mode)

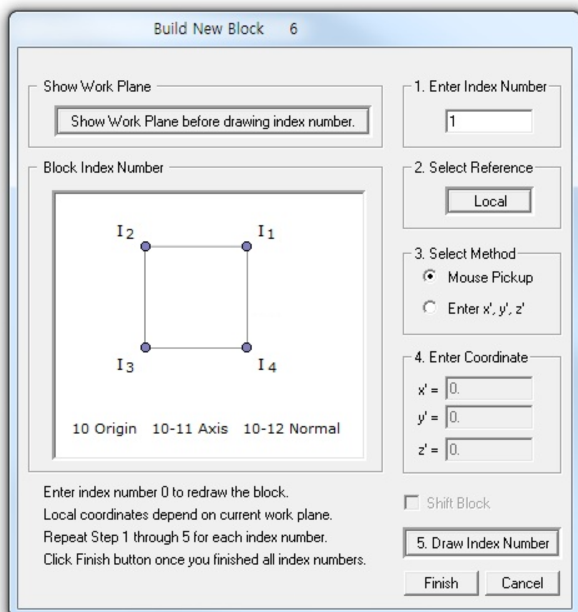
Build New Block dialog now shows input parameters required to build the geometry of new block as shown in Figure 6.38.

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)



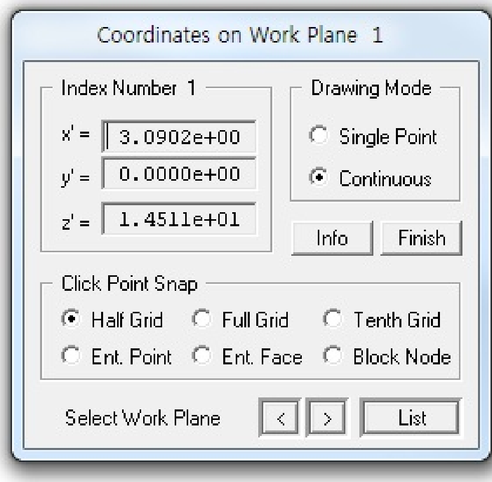


Figure 6.39 Coordinates on work plane

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.

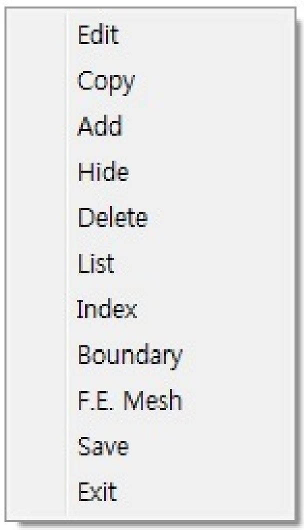


Figure 6.40 Popup menu for block

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.

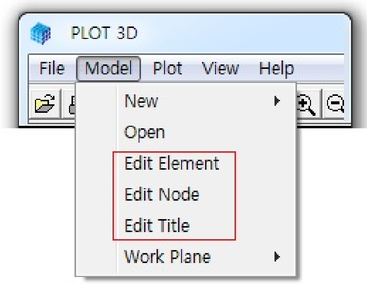


Figure 6.41 Menu for editing finite element mesh

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

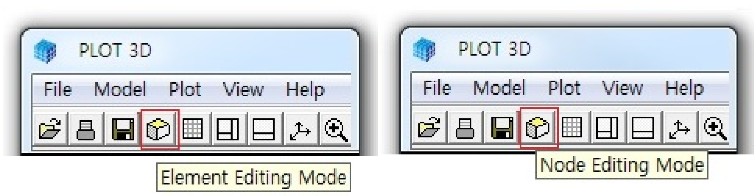
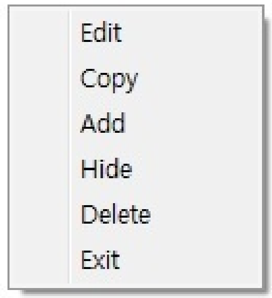


Figure 6.42 Toolbar for editing finite element mesh

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



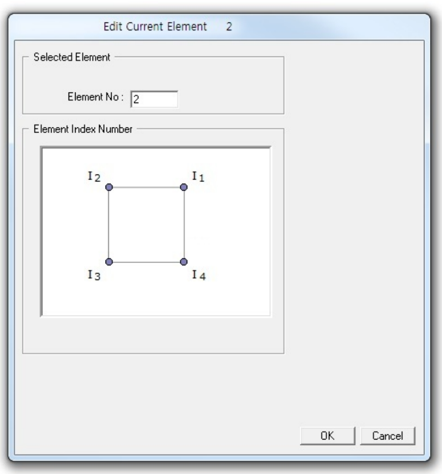
Element popup menu consists of six submenus:
Edit, Copy, Add, Hide, Delete and Exit.

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.

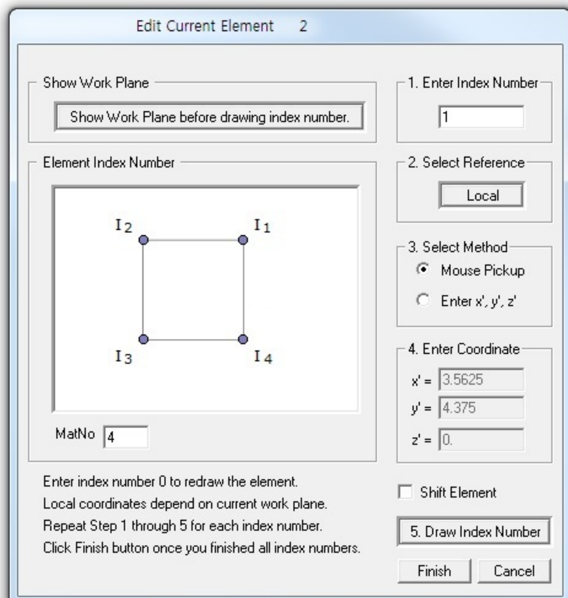
To edit element, follow five steps:

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)



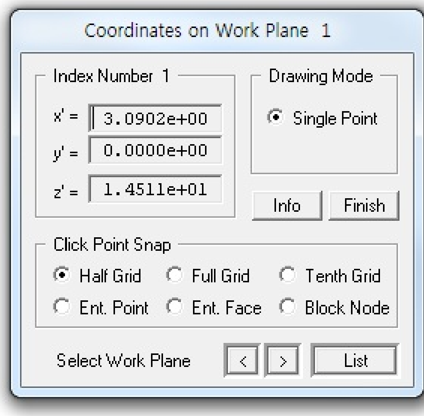


Figure 6.46 Coordinates on work plane (Edit Mode)

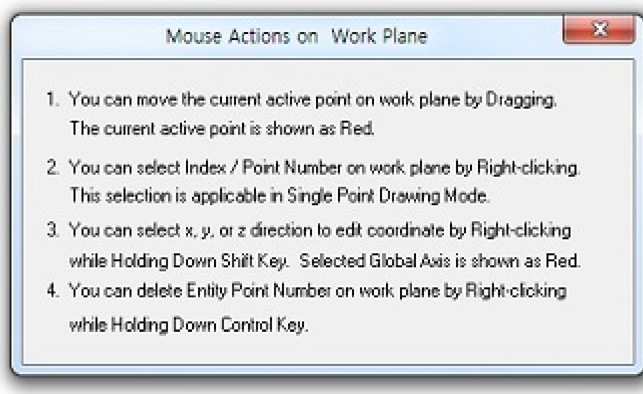


Figure 6.47 Mouse actions on work plane

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.

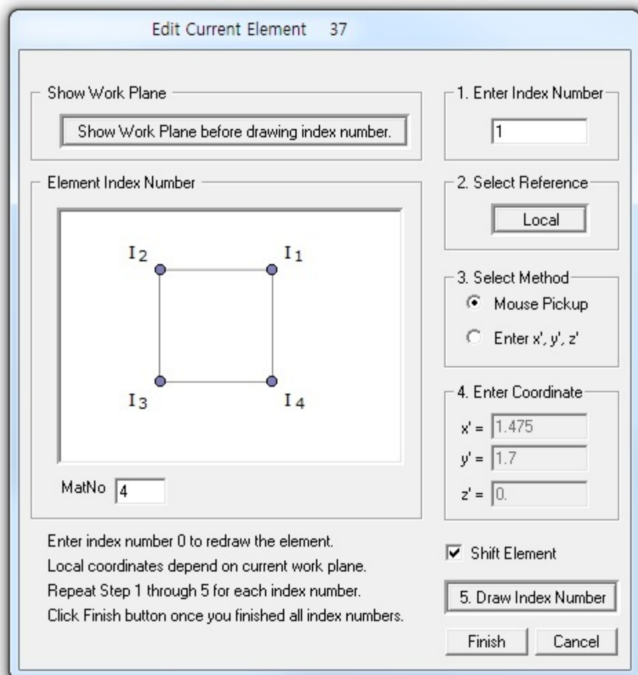


Figure 6.48 Edit current element (Copy Mode)

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.

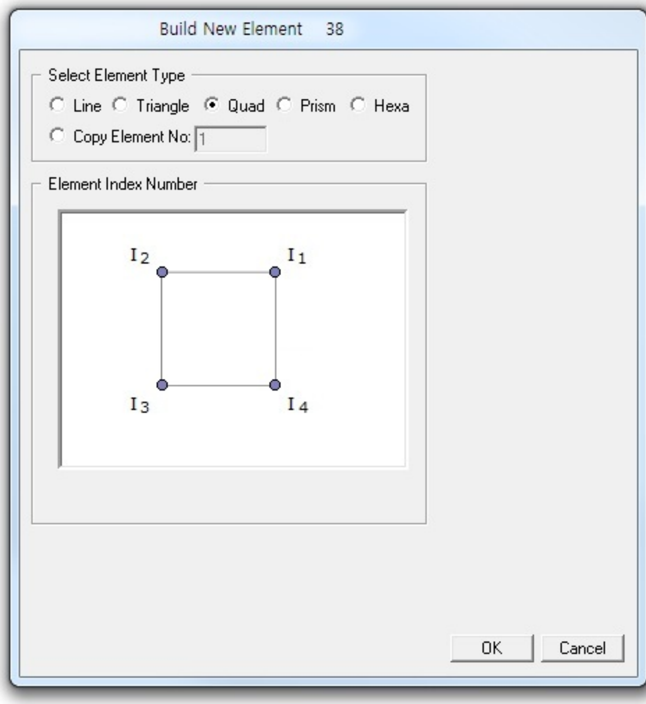


Figure 6.49 Build new element (Selection Mode)

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

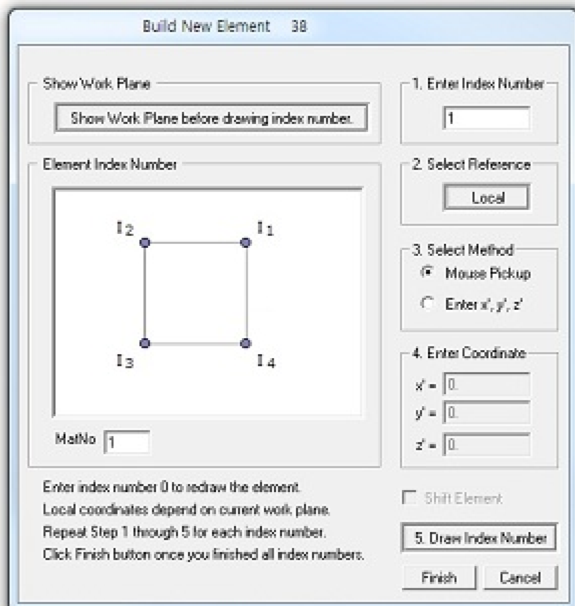
To build new element, follow five steps:

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)



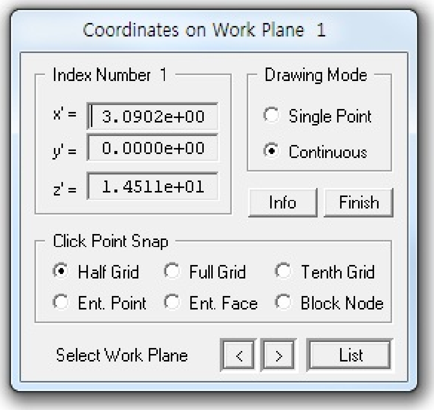


Figure 6.51 Coordinates on work plane (Add Mode)

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

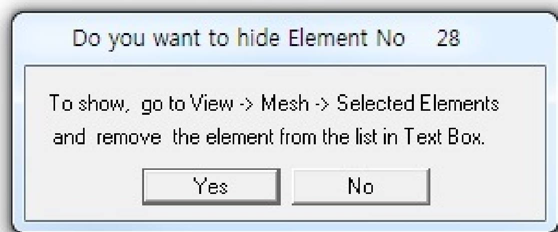


Figure 6.52 Instructions to show the hidden element

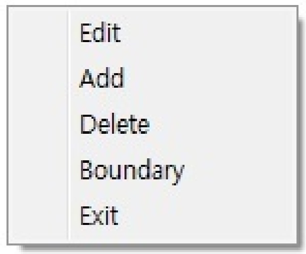
Delete
This is used to delete the selected element.

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.

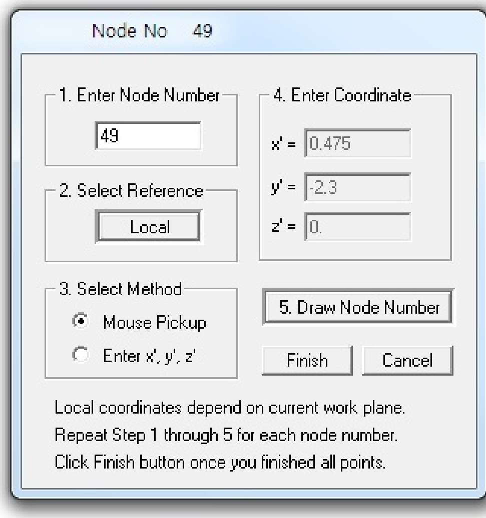


Figure 6.54 Edit current node dialog

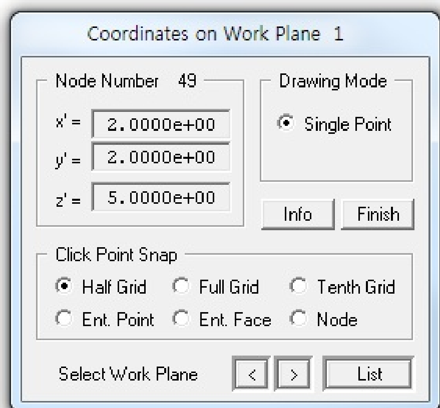


Figure 6.55 Coordinates on work plane (Edit Node)

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.

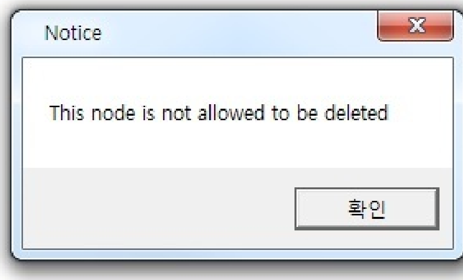
Delete

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

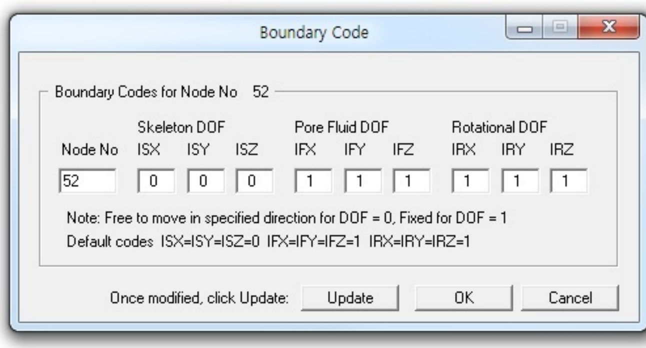


Boundary

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

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

General Information</td><td>1.1

TITLE

TITLE   Any title of (Max = 60 characters)</td></tr>
<tr><td>1.2

IP

IP = 0   Plane strain or plane stress
= 1   Axisymmetry</td></tr>
<tr><td>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</td></tr>
</table>

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

Card Group	Input Data and Definitions (Model 1)
3	3.1 BLNAME BLNAME Block name (up to 60 characters)
	3.2 IBLNO IBLNO Block number
	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
Data for Each Block	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

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

Data for Each Block</td><td>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

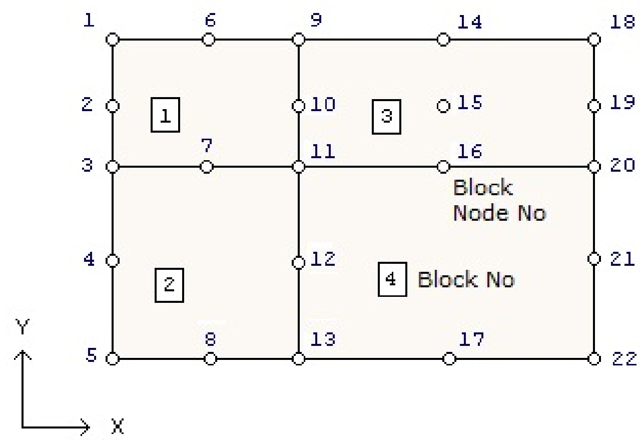
KS = 0 — Has solid phase
= 1 — No solid phase

KF = 0 — Has fluid phase
= 1 — No fluid phase

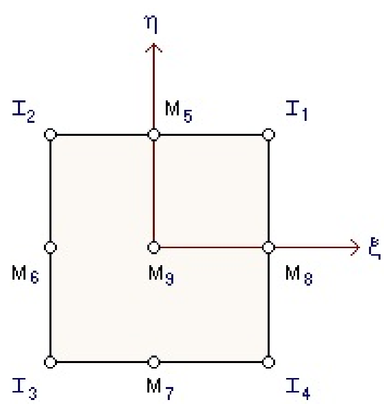
THICK — Thickness of element.
For plane strain, use THICK=1.0
DENSITY — Unit weight of element

IDH — Heat generation history ID number</td></tr>
</table>

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

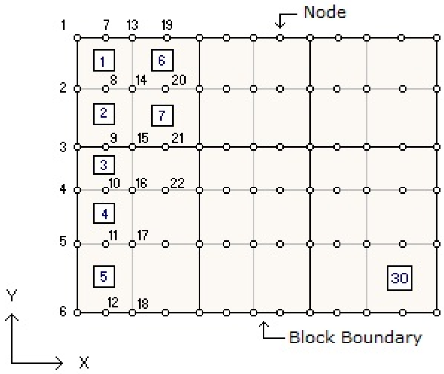


For Block Number 3

I_1 = 18	M_5 = 14
I_2 = 9	M_6 = 10
I_3 = 11	M_7 = 16
I_4 = 20	M_8 = 19
M_9 = 15	

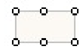
PRESMAP uses Serendipity interpolation if $M_9 = 0$
and Lagrangion interpolation if $M_9 \neq 0$

Figure 7.1 Block Specification and Block Index



NBX = 3
NBY = 2

MIDX = 1
MIDY = 0

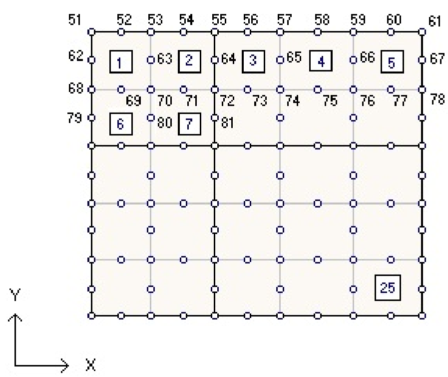


6 Node Element

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

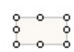
NSNODE = 1

☐ 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

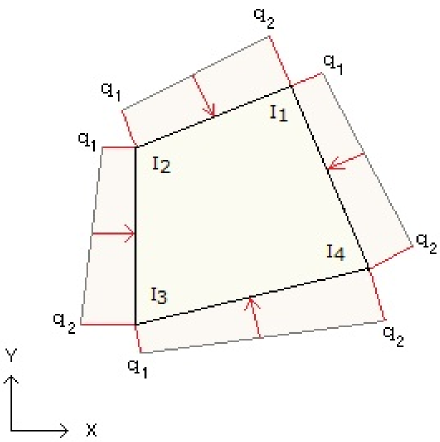
Boundary Type	Boundary Codes		
IB	IDX	IDY	IDT
0	0	0	1
1	1	0	1
2	0	1	1
3	1	1	1
4	0	0	0
5	1	0	0
6	0	1	0
7	1	1	0

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

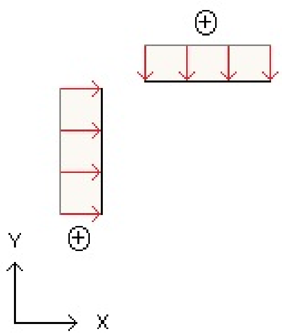
IDT = 0 Rotational degree of freedom is free
= 1 Rotational degree of freedom is fixed

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 General Information	1.1 TITLE TITLE — Any title (Max = 60 characters)
	1.2 IP IP = 0 — Plane strain or plane stress = 1 — Axisymmetry
	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	2.1 SUBNAME SUBNAME Subregion name (up to 60 characters)
	2.2 ISUBNO ISUBNO Subregion number
Data for Each Subregion	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

Card Group	Input Data and Definitions (Model 2)
2 Data for Each Subregion	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
	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

<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

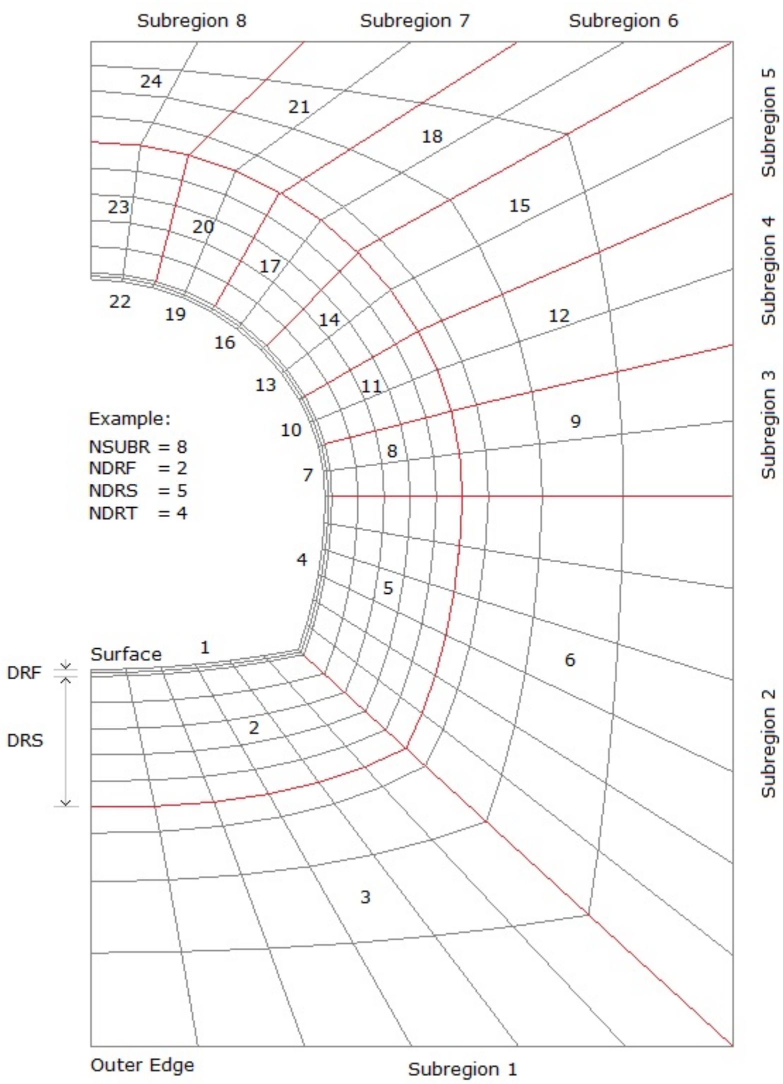


Figure 7.5 Subregion and block numbering sequence

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

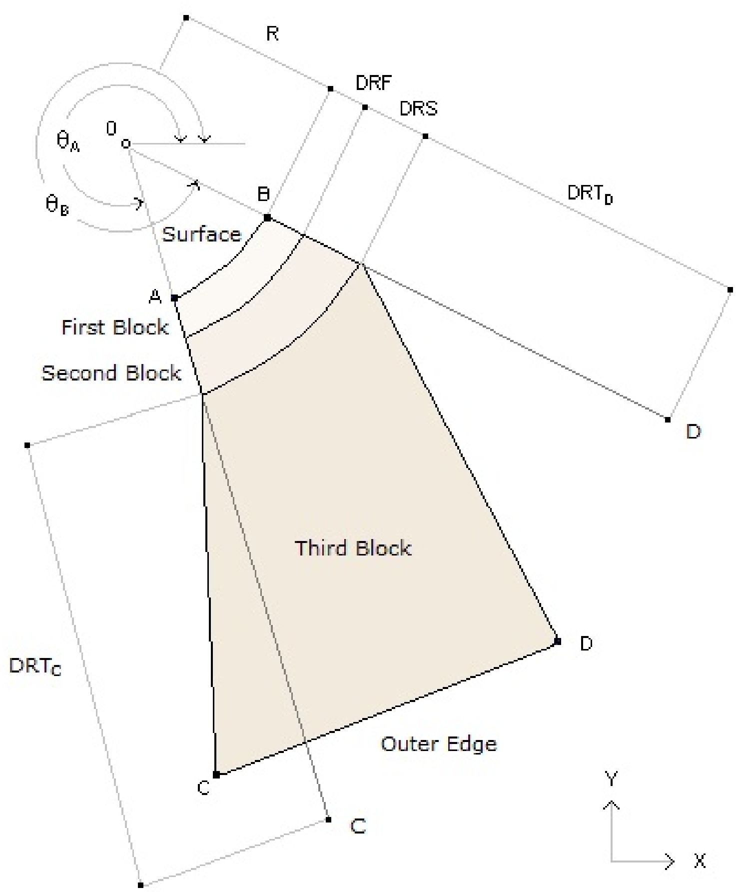


Figure 7.6 Typical Subregion for ISBTYPE = 0

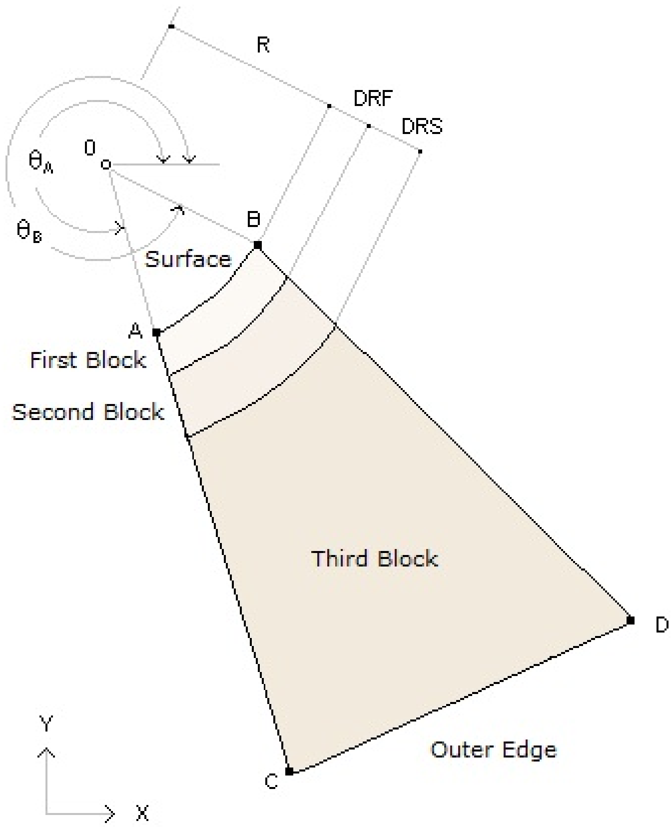
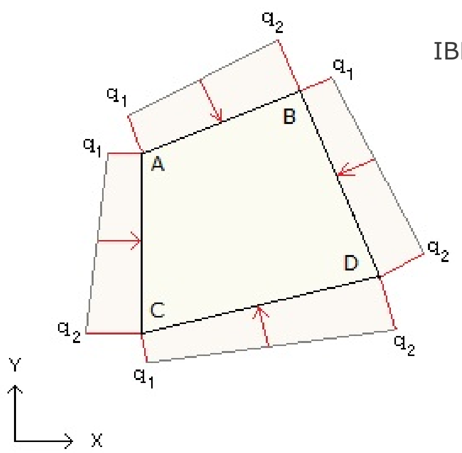
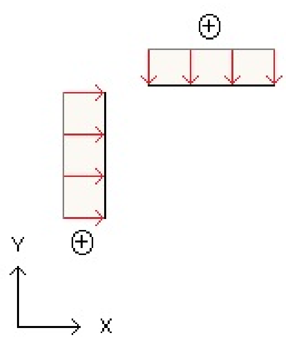


Figure 7.7 Typical Subregion for ISBTYPE = 1



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

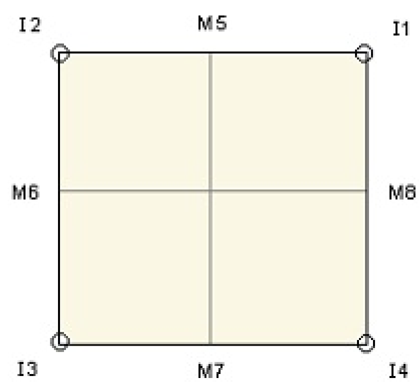
Card Group	Input Data and Definitions (Model 3)
3 Data for Each Block (see Figure 7.9)	3.1 IBLNO, IBLTYPE, MATNO, KS, KF (SMAP-2D) IBLNO, IBLTYPE, MATNO, DENSITY (SMAP-S2) IBLNO, IBLTYPE, MATNO, IDH (SMAP-T2) IBLNO Block number IBLTYPE Block type MATNO Material number KS = 0 Has solid phase = 1 No solid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

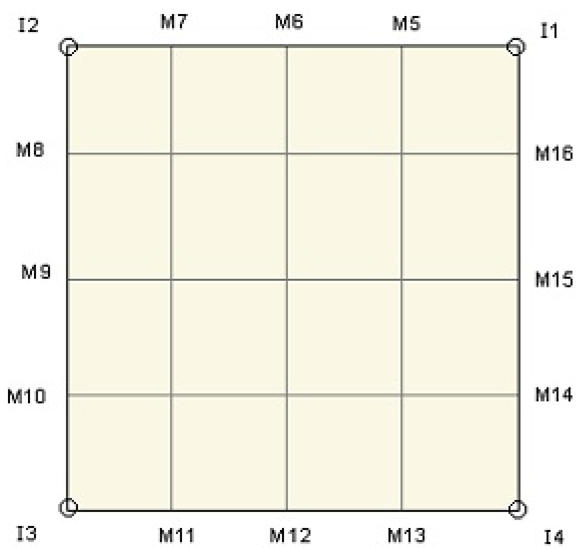
For IBLTYPE = 1

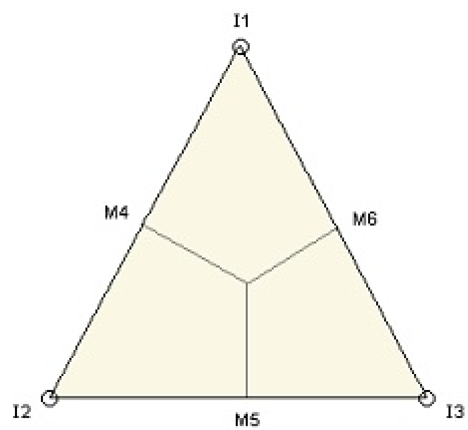
I_1, I_2, I_3, I_4,
M_5, M_6, M_7, M_8

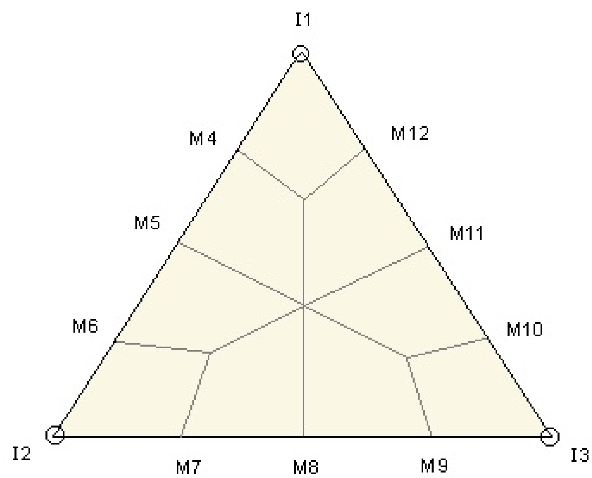
I_1, I_2, I_3, I_4	Corner	node number
M_5, M_6, M_7, M_8	Side	node number



Note: IBLTYPE = 1 generates 4 elements

Card Group	Input Data and Definitions (Model 3)
3 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

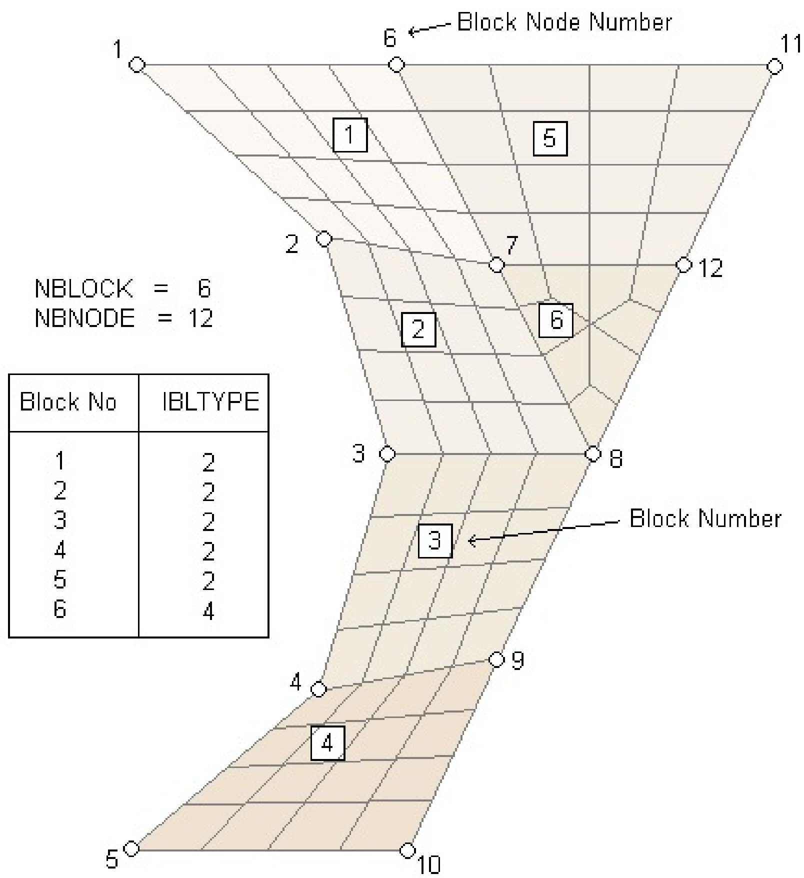


Figure 7.9 Block Node Number for Model 3 of PRESMAP-2D

PRESMAP-2D

Model 4

User's Manual

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

General Information</td><td>1.1

TITLE

TITLE   Any title (Max = 60 characters)</td></tr>
<tr><td>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</td></tr>
<tr><td>1.3

NSNEL, NSNODE, CMFAC

NSNEL   Starting element number
NSNODE   Starting node number
CMFAC   Coordinate magnification factor</td></tr>
<tr><td>2

Block Coordinates</td><td>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</td></tr>
</table>

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

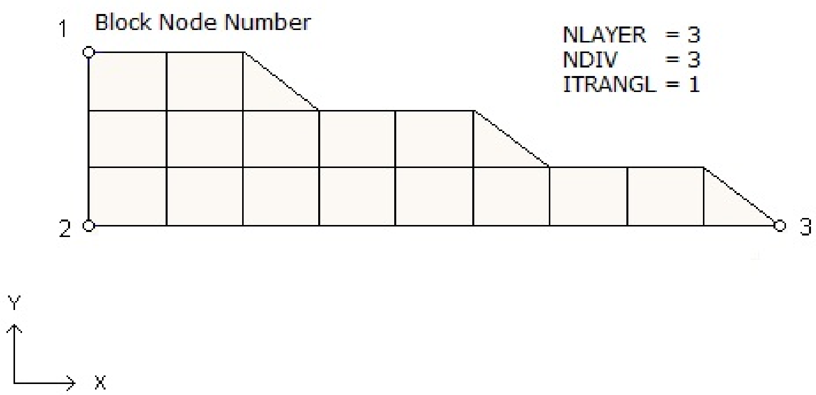


Figure 7.10 Block node number for Model 4 of PRESMAP-2D

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

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

Soil / Rock Layer Information</td><td>3.1

NLAYER

NLAYER — Total number of layers. Max = 10</td></tr>
<tr><td>3.2

NLAYER Cards:
LAYERNO₁, H₁, DD₁
LAYERNO₂, H₂, DD₂
- - -
- - -

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

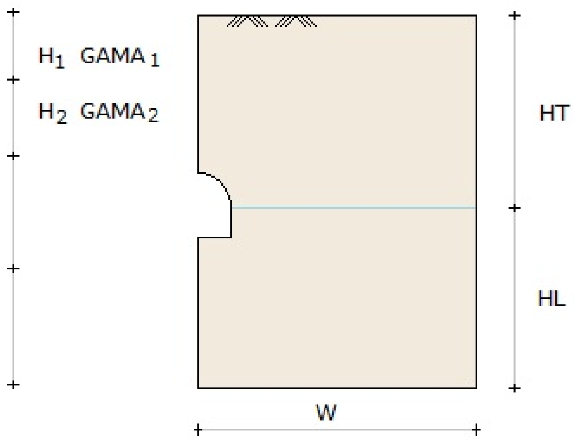
KF = 0 — Has fluid phase
= 1 — No fluid phase

See Figure 7.11</td></tr>
</table>

Card Group	Input Data and Definitions
4 Tunnel Dimension (Repeat this card group for the left tunnel when MODEL = 4)	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
	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
	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

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)

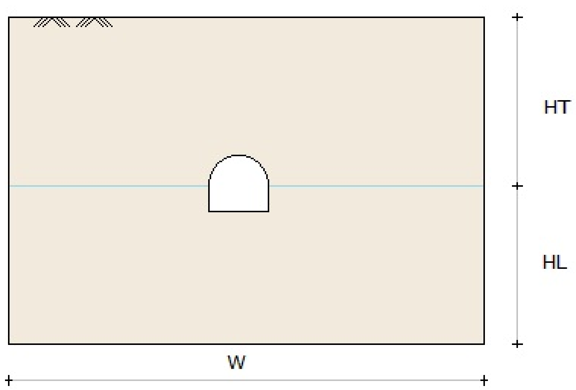
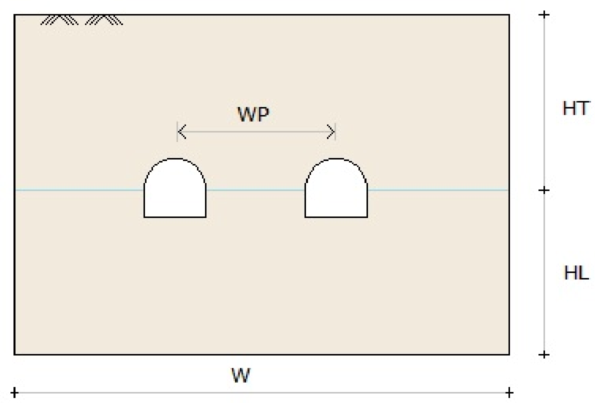


Figure 7.11 Schematic tunnel section view for MODEL = 1 and 2

MODEL = 3   Two Tunnel (Symmetric Section)



MODEL = 4   Two Tunnel (Unsymmetric Section)

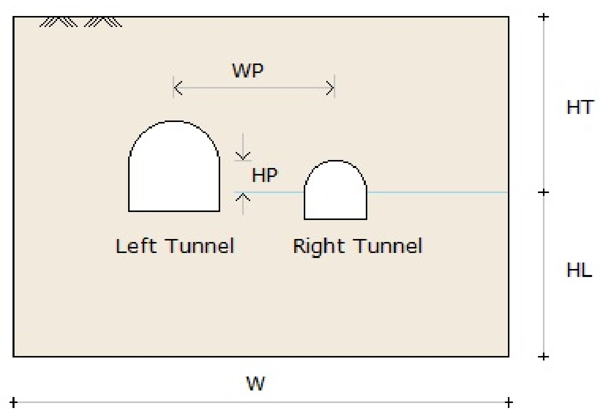
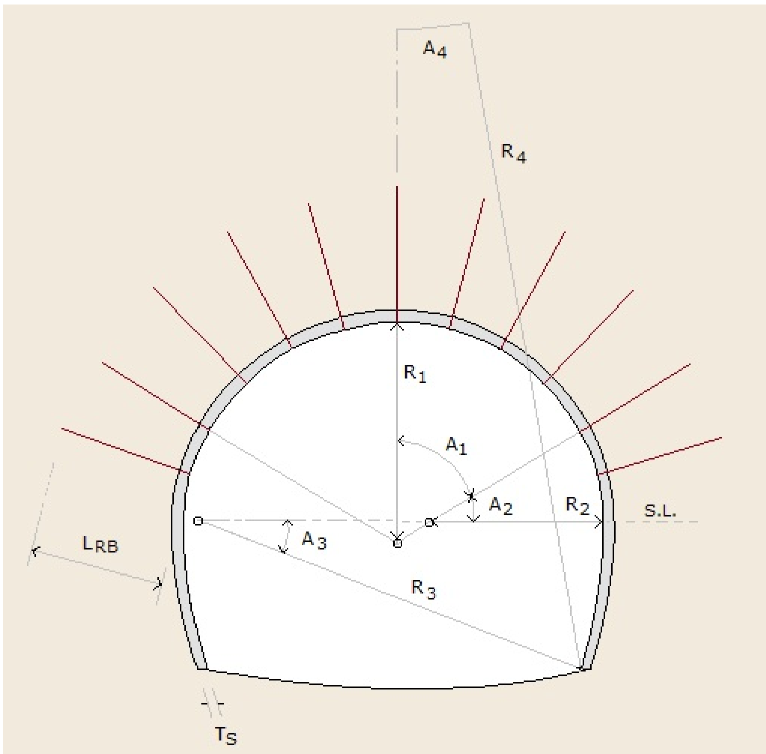


Figure 7.11   Schematic tunnel section view for MODEL = 3 and 4



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)

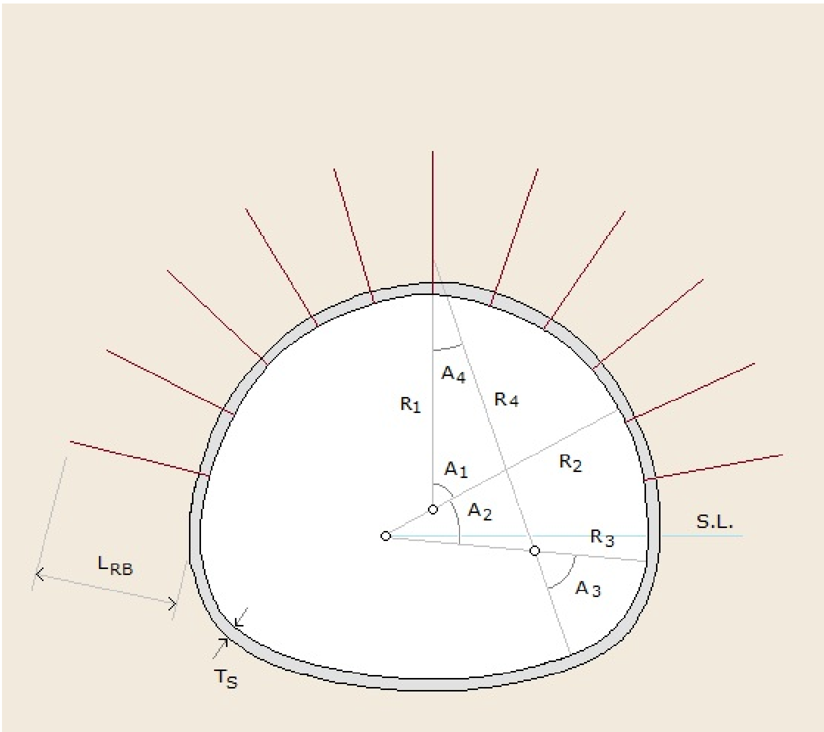


Figure 7.12 Tunnel dimension (A_1+A_2 > 90)

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.

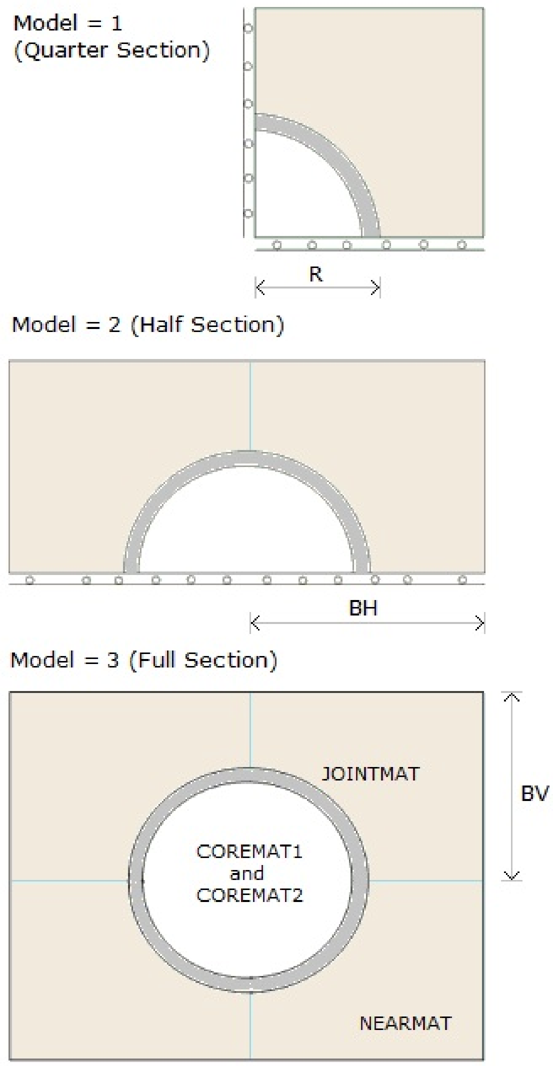


Figure 7.13 Model type for CIRCLE-2D

PRESMAP-GP

User's Manual

Card Group	Input Data and Definitions
1 General Information	1.1 TITLE TITLE Any title (Max = 80 characters) Note: Following two cards are required at the beginning StartPresmap VersionNo = 7.000
	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

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

<table>
<tr><th>Card Group</th><th>Input Data and Definitions</th></tr>
<tr><td>3

Data for Each Line Block [IBETYPE =1]</td><td>3.1
BLNAME

BLNAME   Block name (Max = 60 characters)</td></tr>
<tr><td></td><td>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</td></tr>
</table>

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 $\le$ min (NDX, NDY) and NT i + NT j $\le$ 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

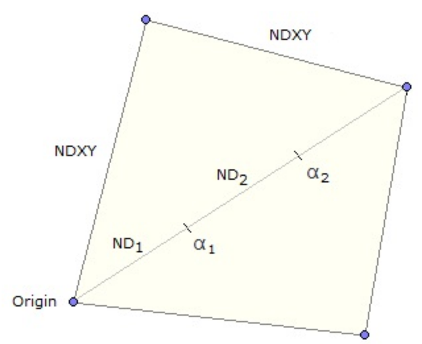
NSEG

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

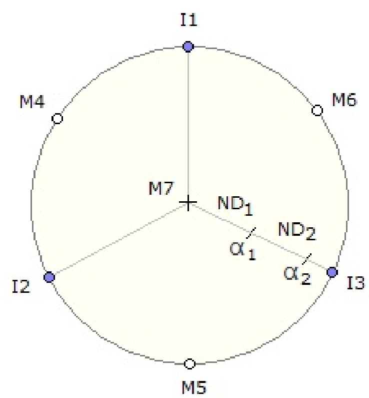
NSEG

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 Data for Each Hexahedron 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_{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	3.5
Data for Each Hexahedron Volume Block [IBETYPE =3]	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	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 Prism 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 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

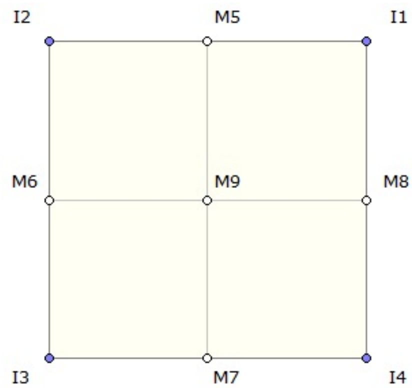
Note: Boundary Conditions

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

Line Block

Quad Surface Block



Triangle Surface Block

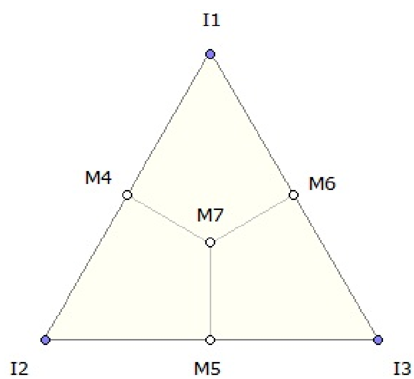


Figure 7.22 Block index for PRESMAP-GP

Hexahedron Volume Block

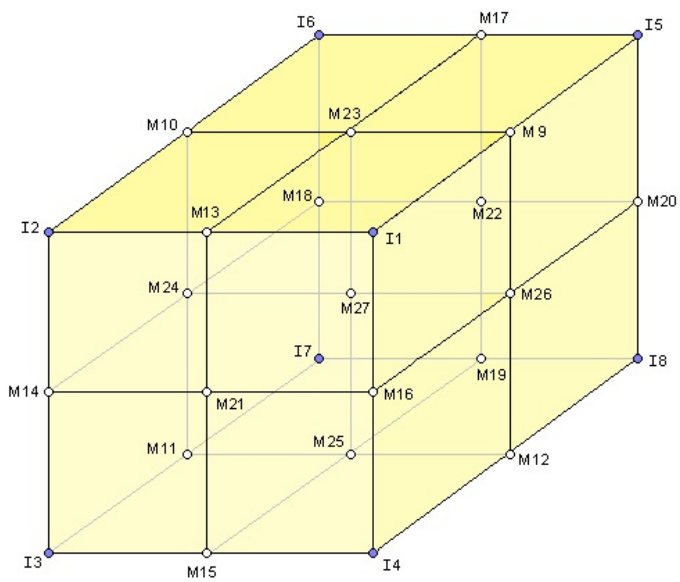


Figure 7.22 Block index for PRESMAP-GP (Continued)

Prism Volume Block

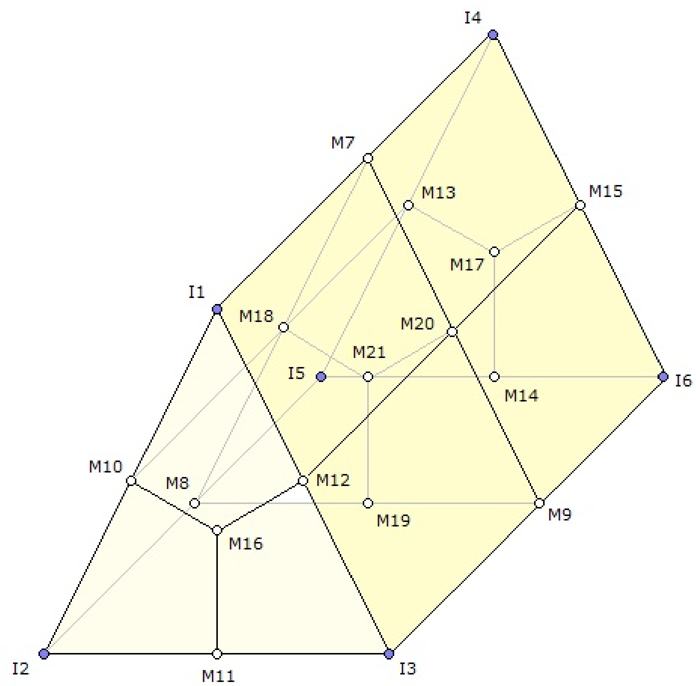


Figure 7.22 Block index for PRESMAP-GP (Continued)

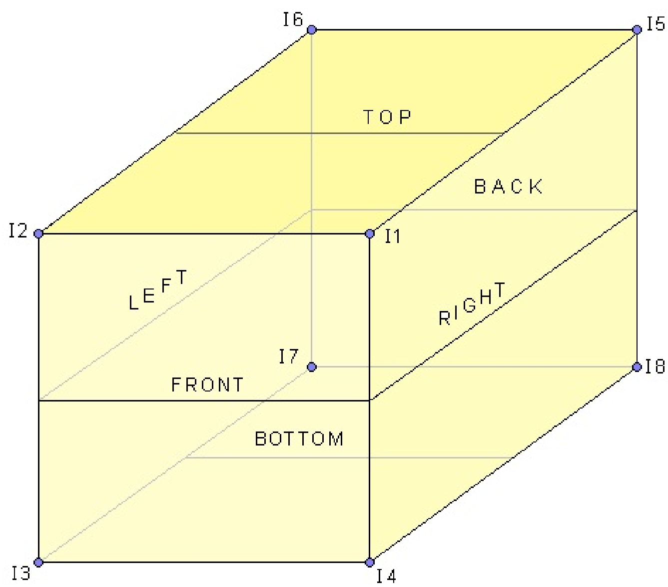


Figure 7.23 Boundary surface designation for Hexahedron Volume Block

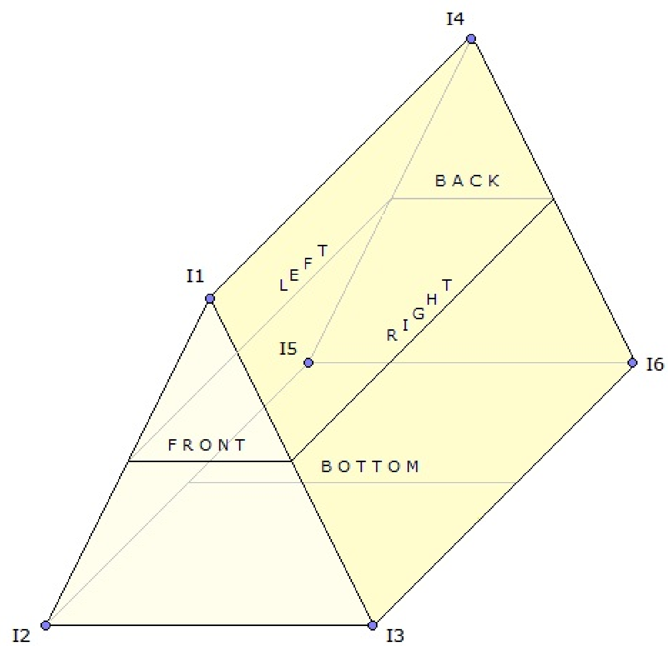


Figure 7.24 Boundary surface designation
for Prism Volume Block

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

<table>
<tr><th>Card Group</th><th>Input Data and Definitions</th></tr>
<tr><td>1
Mode Type</td><td>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 (SMAP-T2), 2 (SMAP-S2), 3 (SMAP-2D)</td></tr>
<tr><td>2
Adding Region B to Region A (IMOD = 0)</td><td>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.</td></tr>
<tr><td></td><td>2.2
INTERFACE
INTERFACE = 0 Interface is found automatically
= 1 Interface is specified by user</td></tr>
<tr><td></td><td>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 $NODA_i$</td></tr>
</table>

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 NSNEL New starting element number NSNODE New starting node number
	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

<table>
<tr><th>Card Group</th><th colspan="4">Input Data and Definitions</th></tr>
<tr><td rowspan="2">3

Modifying Existing Mesh (IMOD = 1)</td><td rowspan="2">Build User-Defined Curves and Material Zones (IEDIT = 4)</td><td rowspan="2">For Each Curve Group</td><td colspan="2">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</td></tr>
<tr><td>For Each Segment</td><td>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</td></tr>
</table>

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.

<table>
<tr><th>Card Group</th><th>Input Data and Definitions</th></tr>
<tr><td rowspan="5">4

Generate Base Mesh and then Modify (IMOD = 2) See Figure 8.1</td><td>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</td></tr>
<tr><td>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)</td></tr>
<tr><td>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</td></tr>
<tr><td>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</td></tr>
<tr><td>4.5

IGMOD

IGMOD = 0 Do not modify
= 1 Modify generated base mesh
If IGMOD = 1, go to Card 3.1</td></tr>
</table>

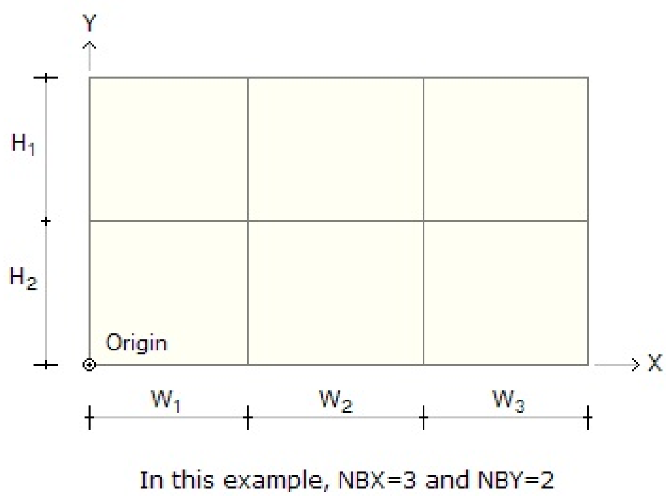


Figure 8.1 Layout of Base Mesh

Case	θ_b	θ_e
1	30 °	310 °
2	310 °	30 °
3	-50 °	30 °
4	30 °	-50 °

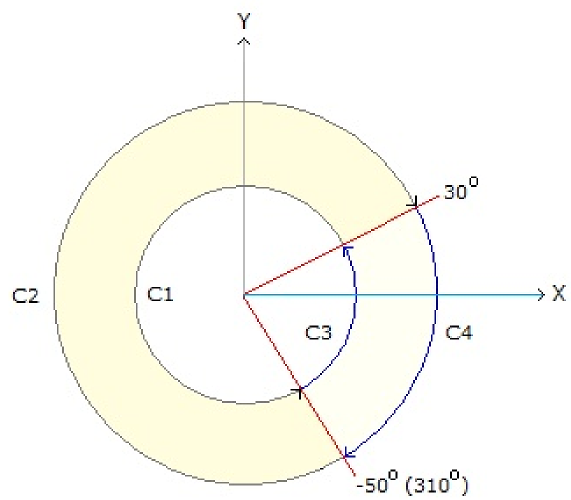


Figure 8.2 Examples of arc specification

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 four programs available:
EDIT, XY, CARDS, and 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 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.

9.2 Edit

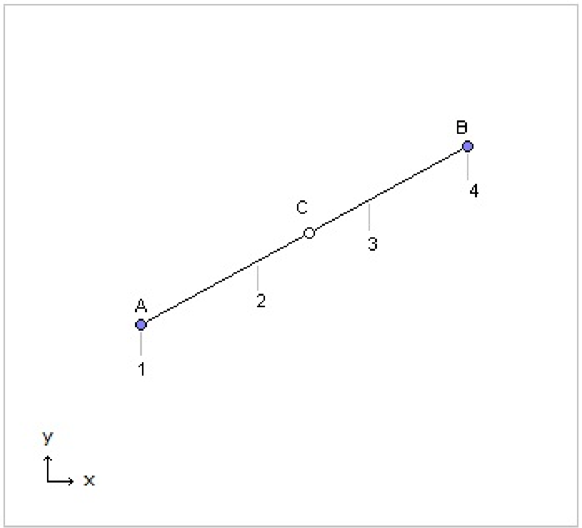
EDIT uses Windows text editor Wordpad to creat, modify, or list file.

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

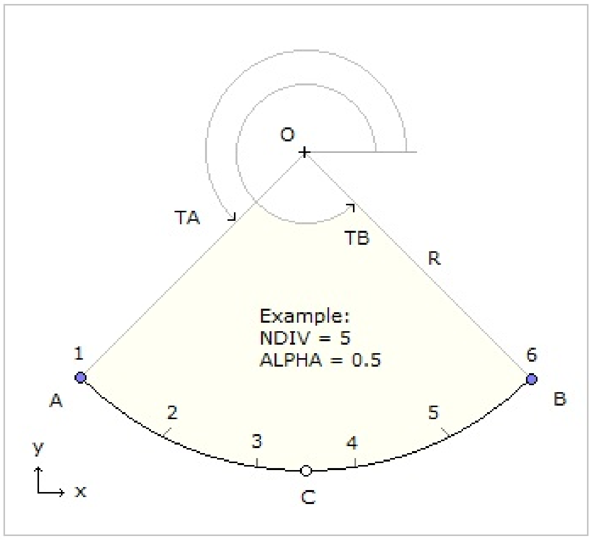


Example: NDIV = 3 and ALPHA = 0.5

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

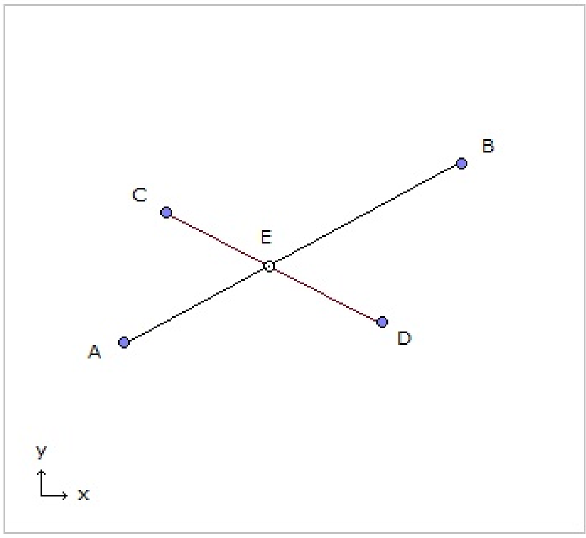


INPUT:
R, X_o, Y_o, TA, TB
NDIV, ALPHA

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

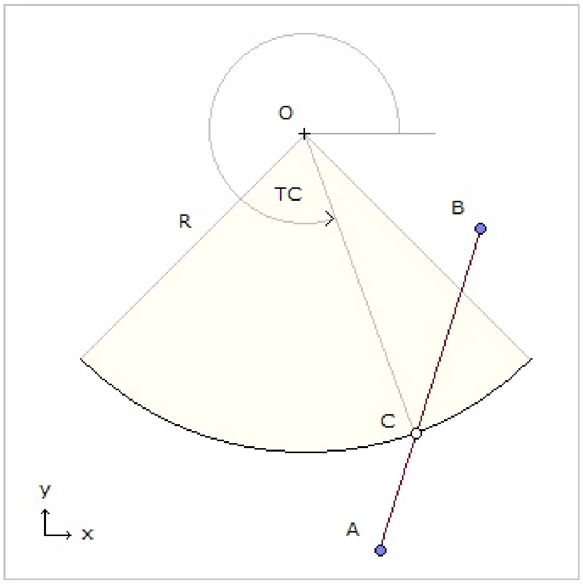
NF = 3 Compute Intersection Point of Two Straight Lines



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

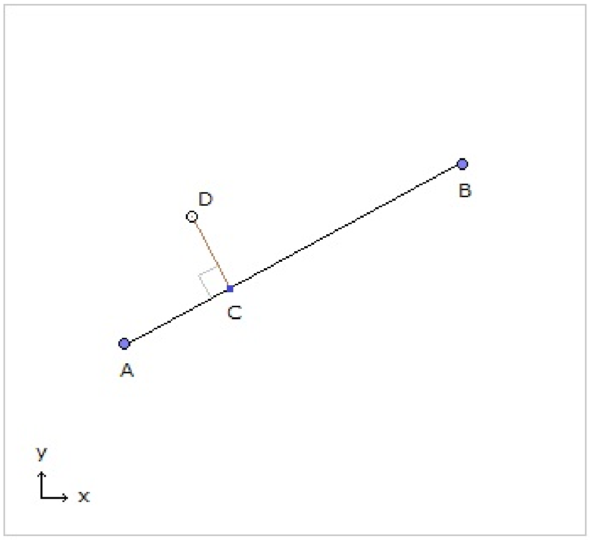
NF = 4 Compute Intersection point of Arc & Straight Line



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

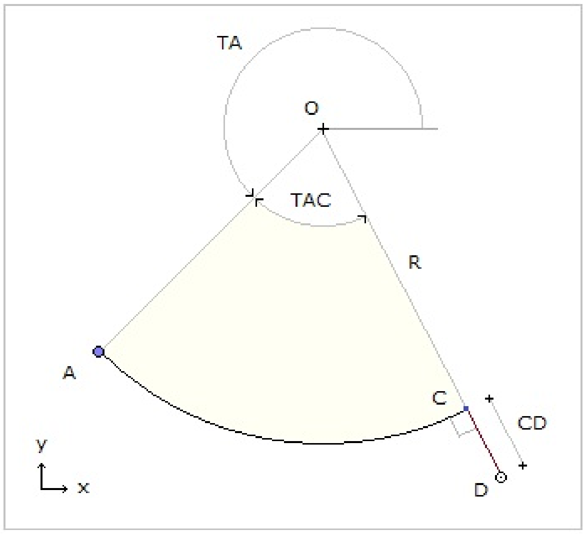
NF = 5 Compute Points Normal to Straight Line



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-S2 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 MCH

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.
MCH	Element stiffness/volume history number

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.

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-S2 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-S2 Mesh File

Following Mesh Files can be converted to SMAP-S2 Mesh File format:

- Mesh Files generated for two-dimensional SMAP programs (SMAP-2D and SMAP-T2)
- FEMAP (Version 4.1 - 4.5, neutral format)

Figure 10.1 shows File Conversion dialog box with Input Mesh File options.

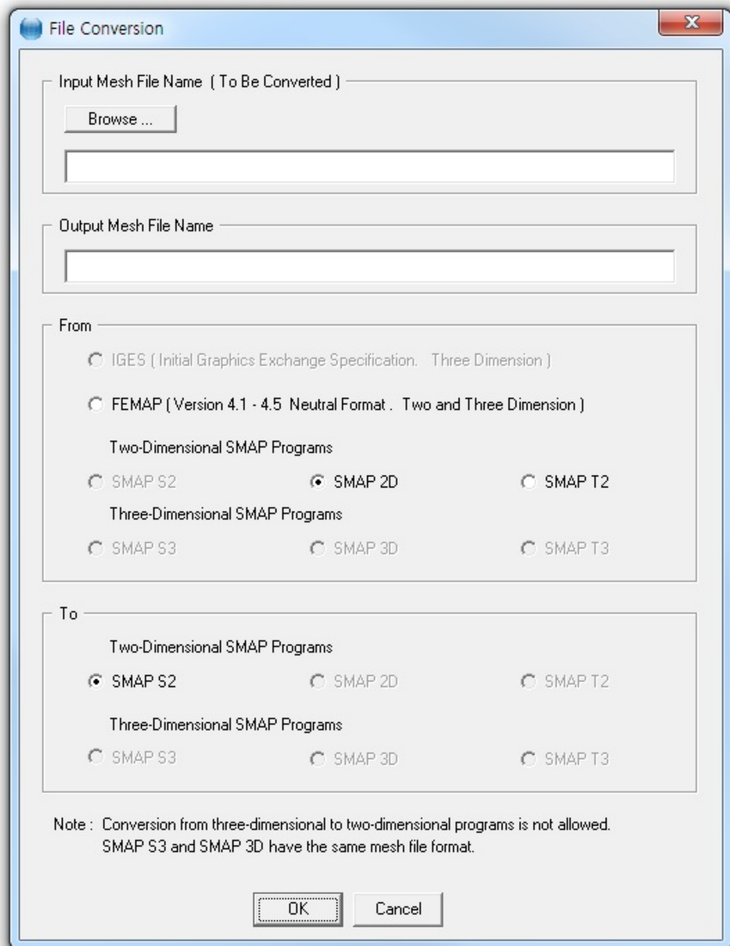


Figure 10.1 File Conversion dialog box

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

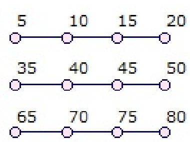
NUMNODG

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 $P_n = a_{no} + a_{nx}x + a_{ny}y$

Card Group	Input Data and Definitions (Pressure)	
4 Pressure History	4.1 NUMLH NUMLH Number of pressure histories (Max = 20)	
	4.2 For Each Pressure History	4.2.1 LHNO LHNO Pressure 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 Pressure intensity at time T_i

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

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.

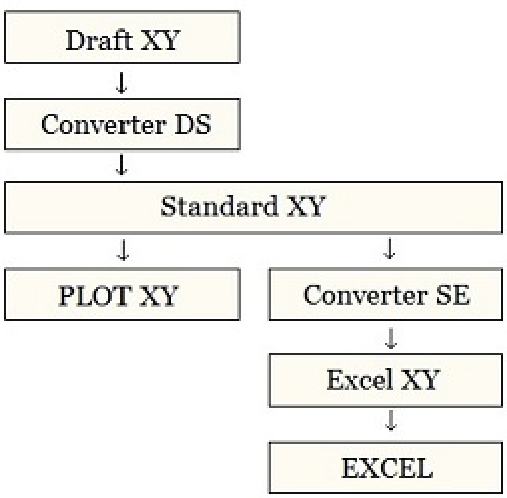


Figure 12.1 Flow diagram of plotting XY graph

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

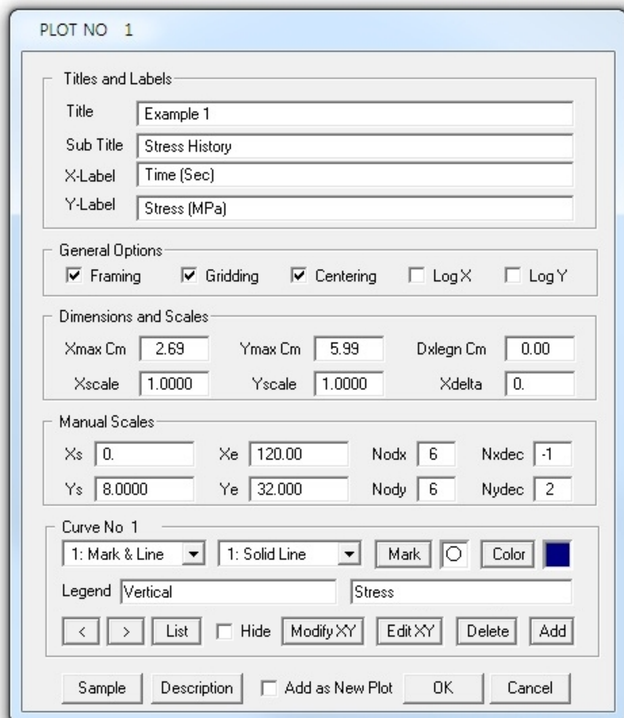
Edit Dialog in Figure 12.2 can be accessed by selecting the Edit menu in PLOT-XY.

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



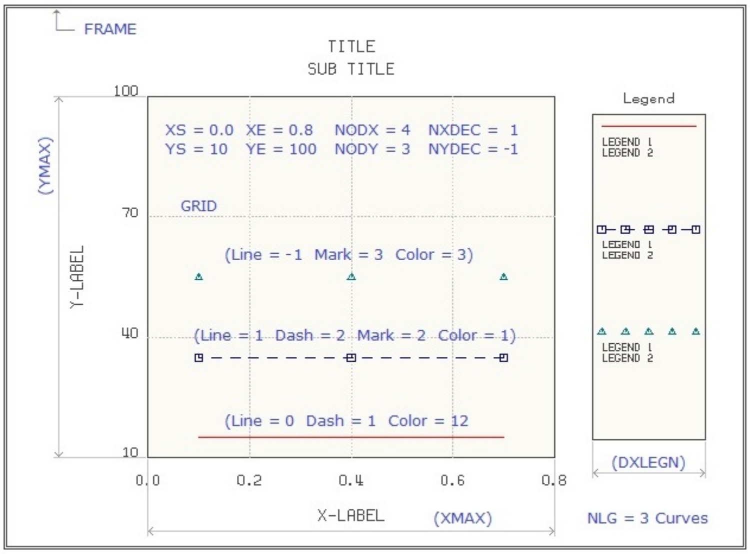


Figure 12.3 Sample graph

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

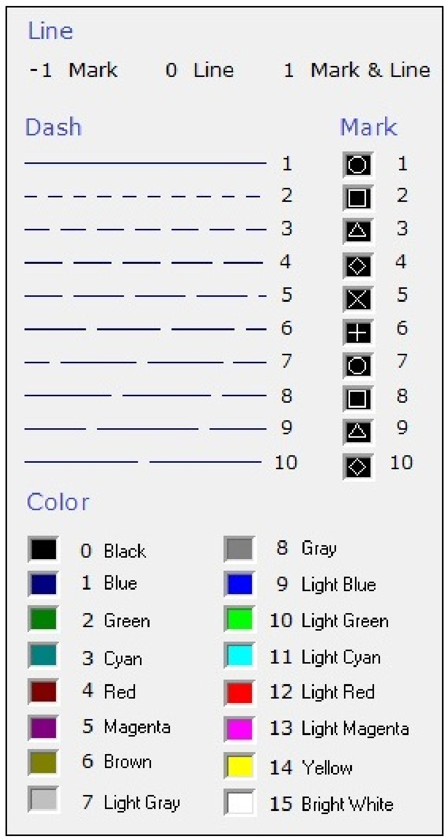
Refer to description in Sample Graph in Figure 12.3.

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

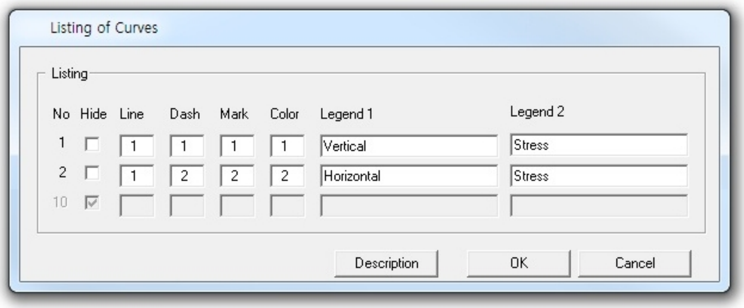


Figure 12.5a Listing of curves

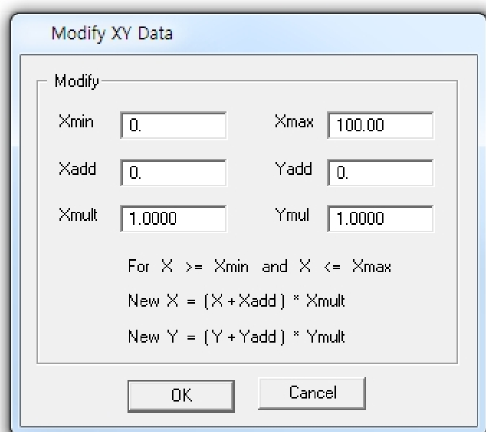


Figure 12.5b Modify current curve XY data

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.

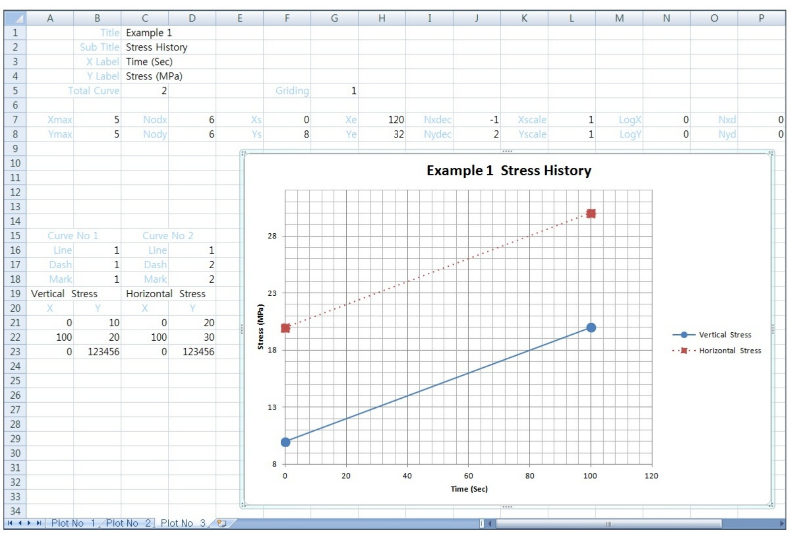


Figure 12.6 XY graph on Excel spread sheet

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 ≠ 0 and XS < 1 and XE > 1, XS is automatically scaled.

For LOGY = 1

NODY and NYD are not used.

If YSCALE ≠ 0 and YS < 1 and YE > 1, YS is automatically scaled.

Refer to samples in the following directory:
C:\Smap \SmapS2 \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.

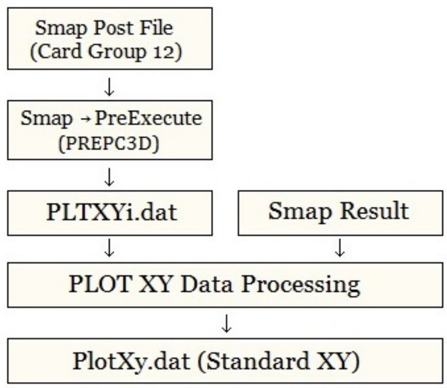


Figure 12.7 Processing SMAP results

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.

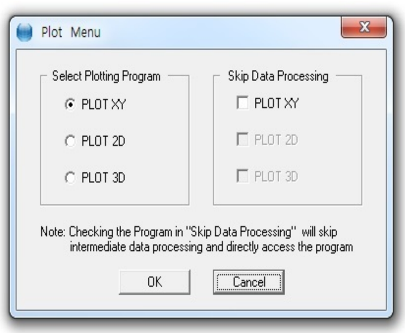


Figure 12.8
Plot menu dialog

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

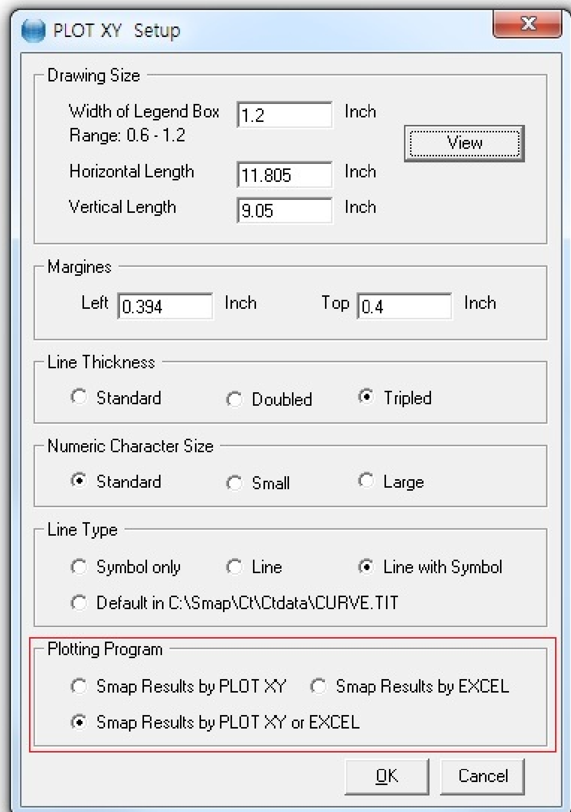


Figure 12.9 PLOT XY setup dialog

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.

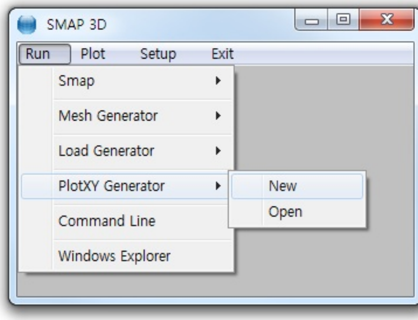
12.7.1 Accessing PlotXY Generator

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.

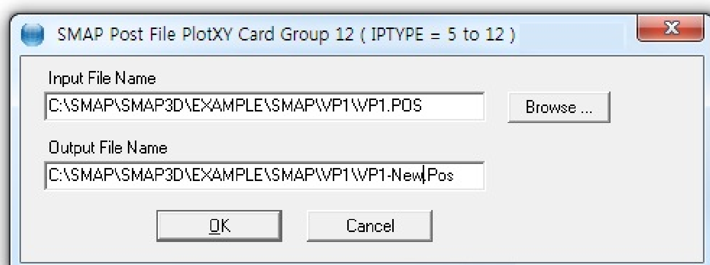


Figure 12.11 PlotXY input and output file dialog

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.

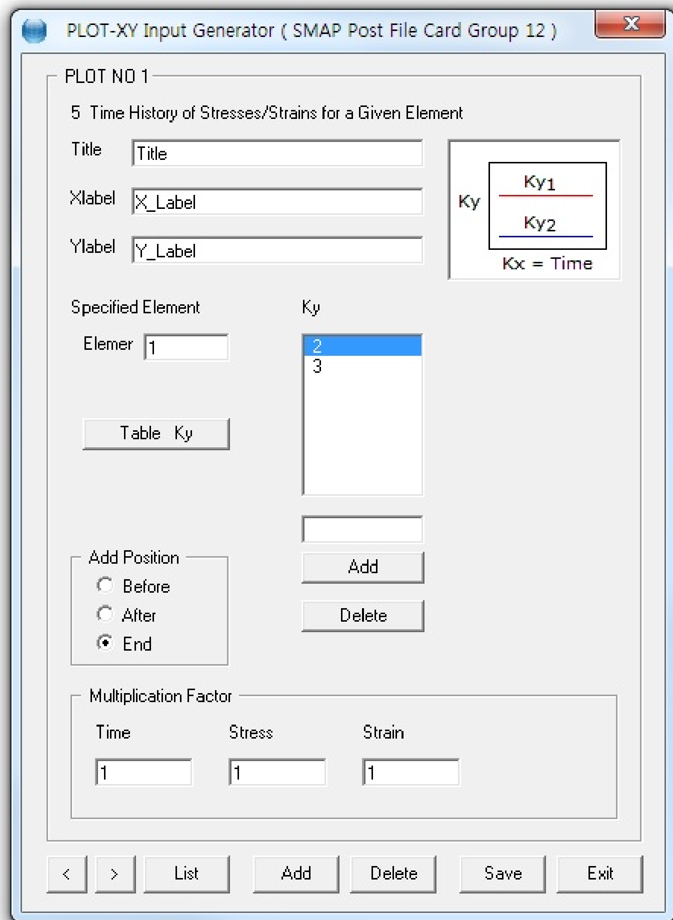


Figure 12.12 Time history for a given element

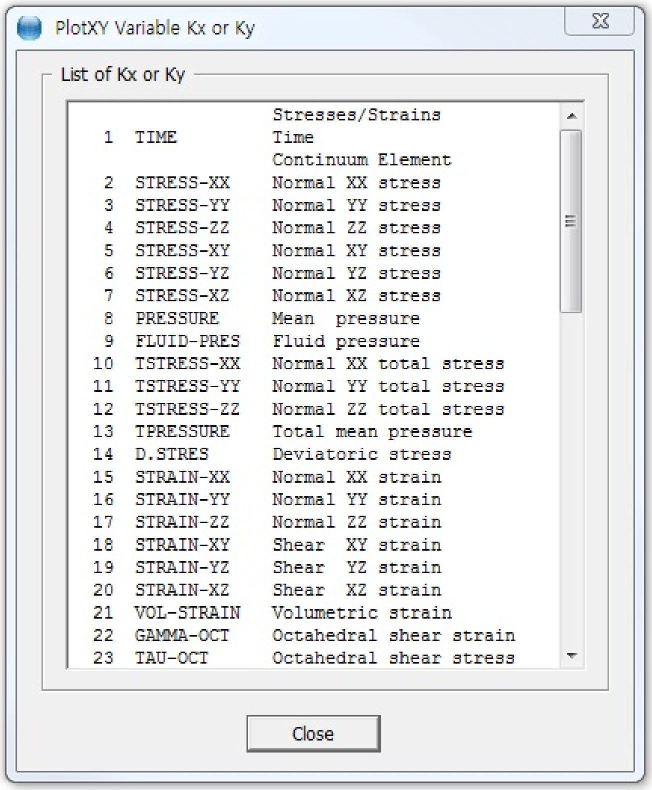


Figure 12.13 Available data for stresses / strains

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.

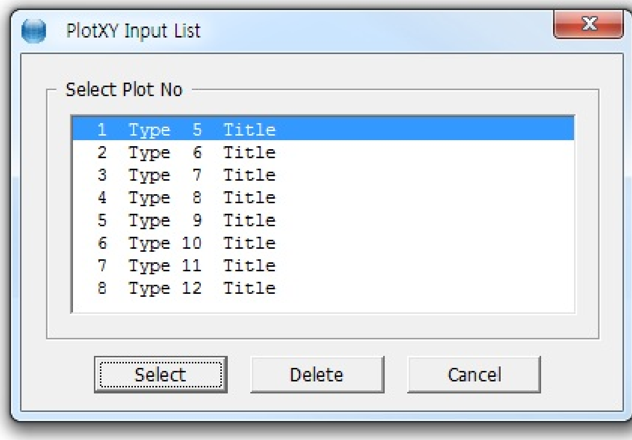


Figure 12.14 Listing of plots

Add shows new plot type to be added as in Figure 12.15.

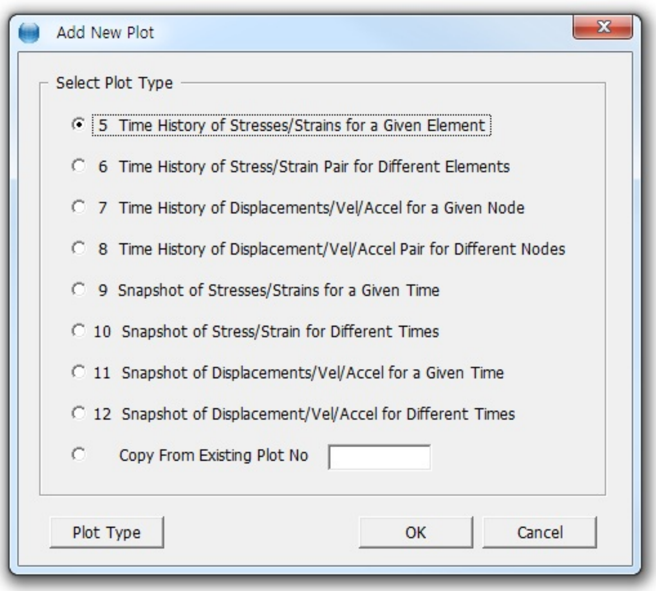


Figure 12.15 Add options for new plot

Plot Type in Add dialog illustrates graphically available plot types as shown schematically in Figure 12.16.

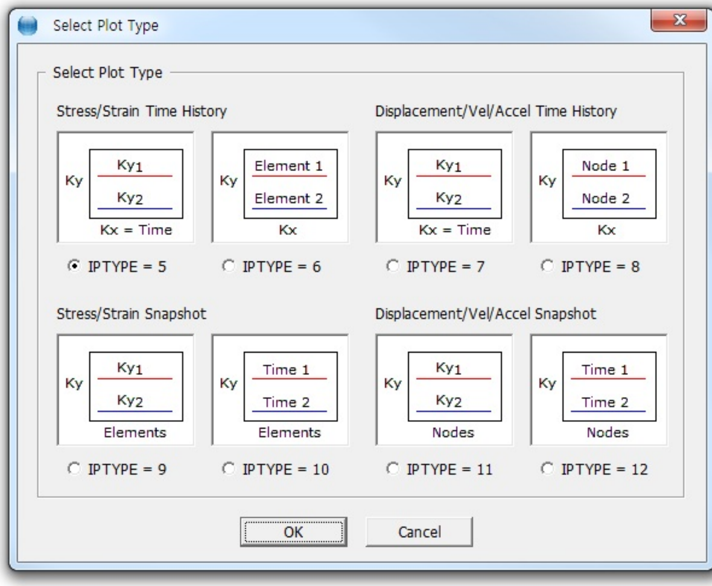


Figure 12.16 Available plot types

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.

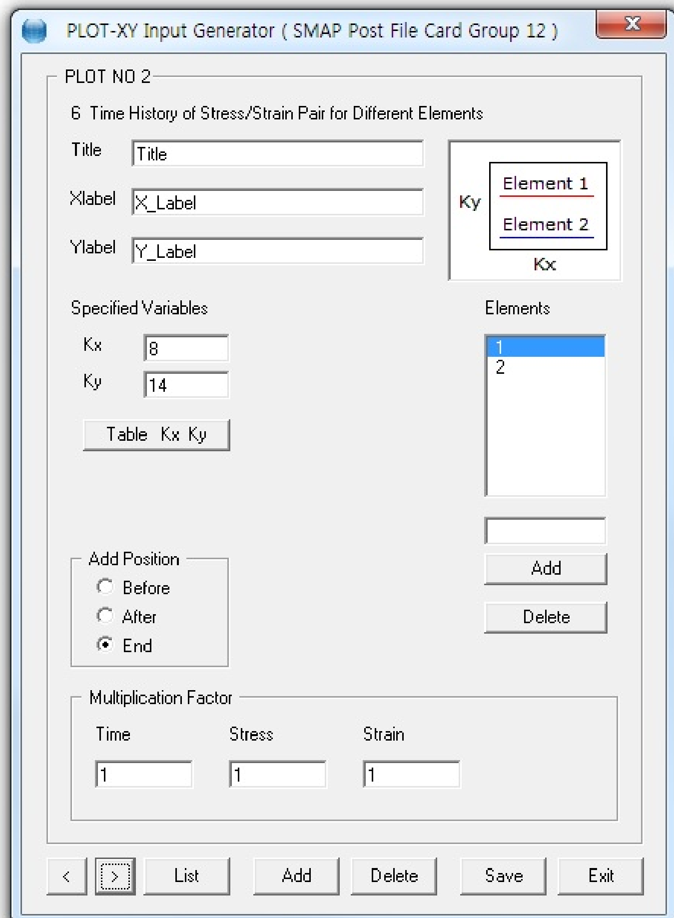


Figure 12.17 Time history for different elements

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.

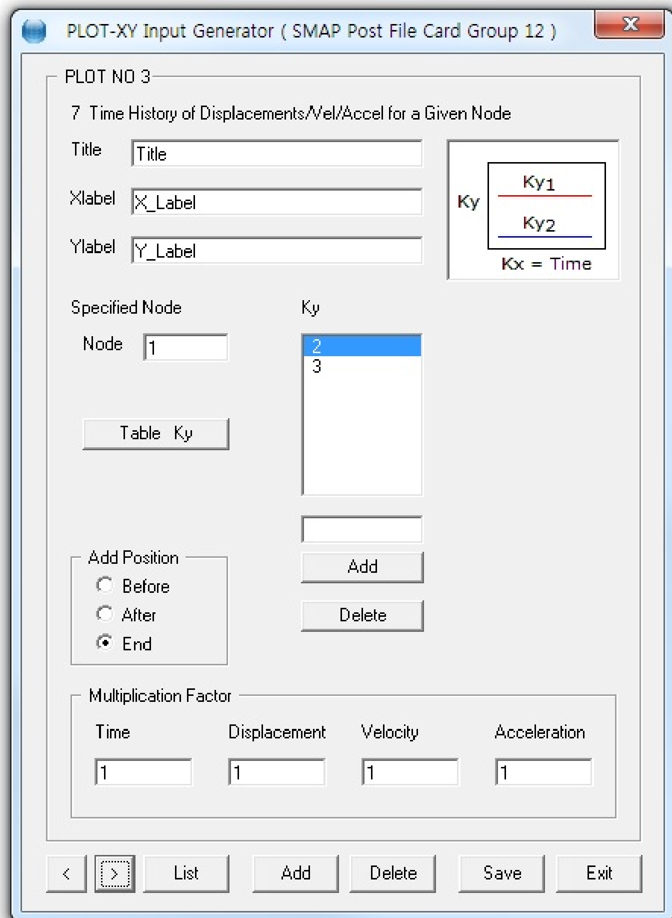


Figure 12.18 Time history for a given node

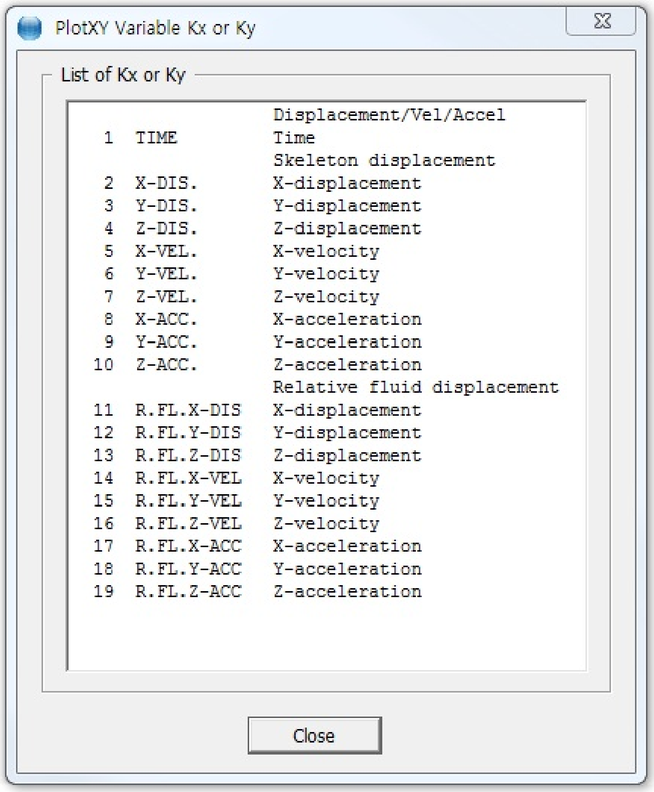


Figure 12.19 Available data for displacement/vel/accel

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.

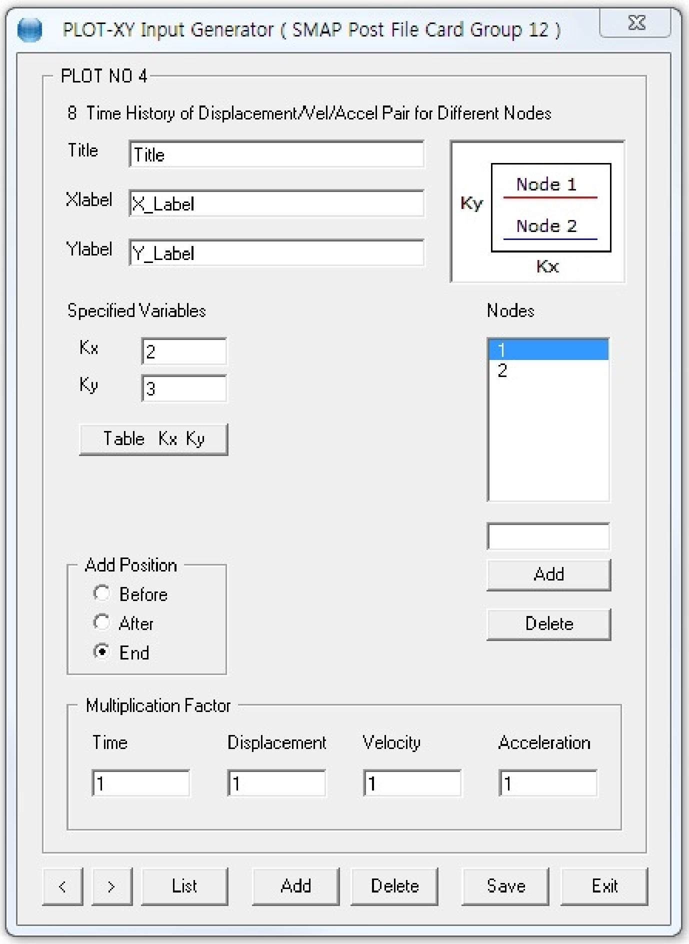


Figure 12.20 Time history for different nodes

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.

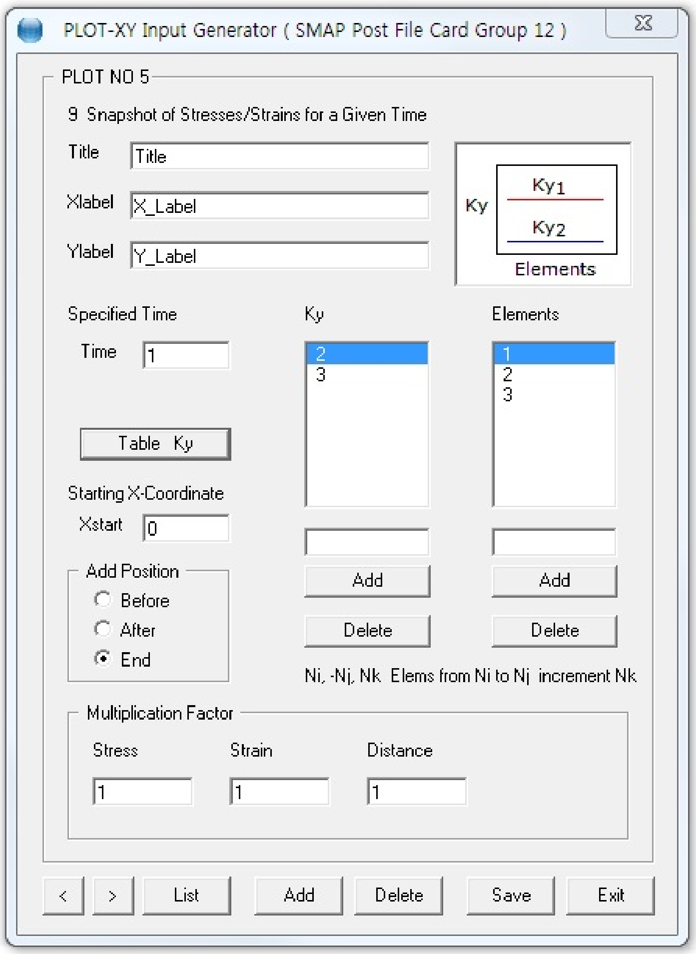


Figure 12.21 Stress/strain snapshot for a given time

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).

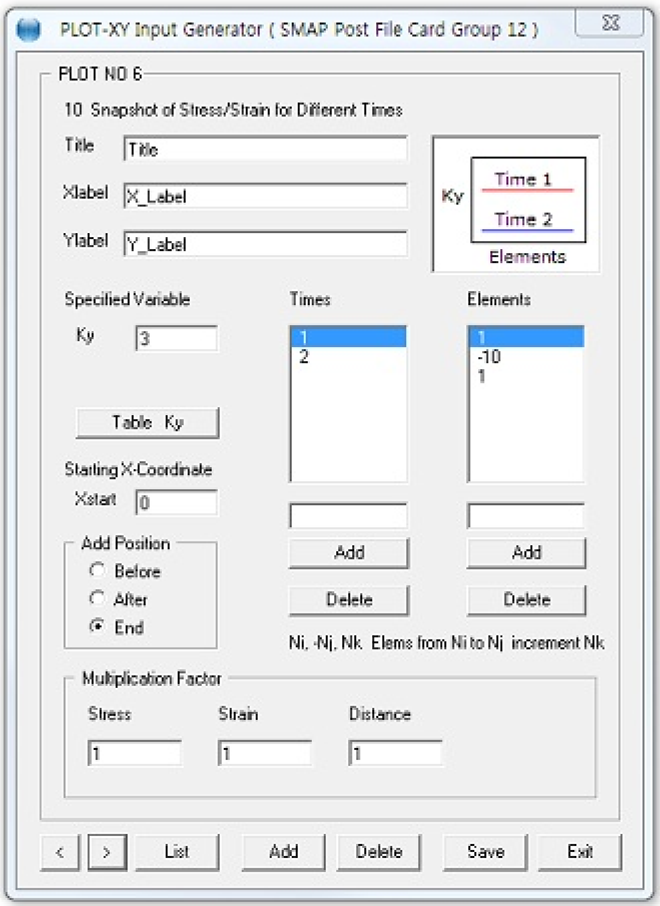


Figure 12.22 Stress/strain snapshot for different times

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.

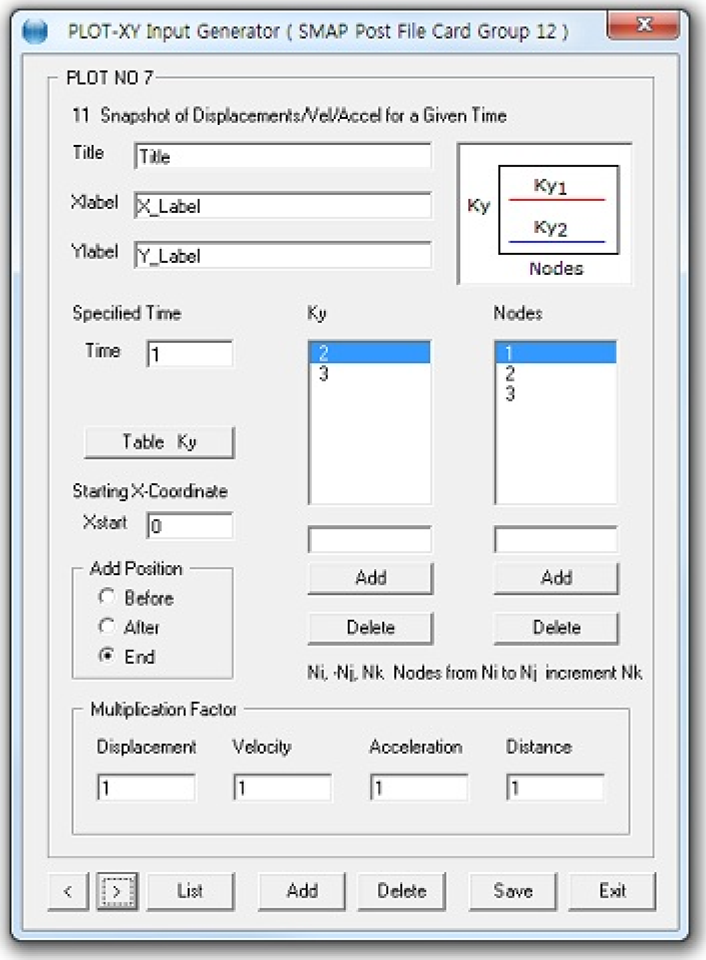


Figure 12.23 Displ/vel/accel snapshot for a given time

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).

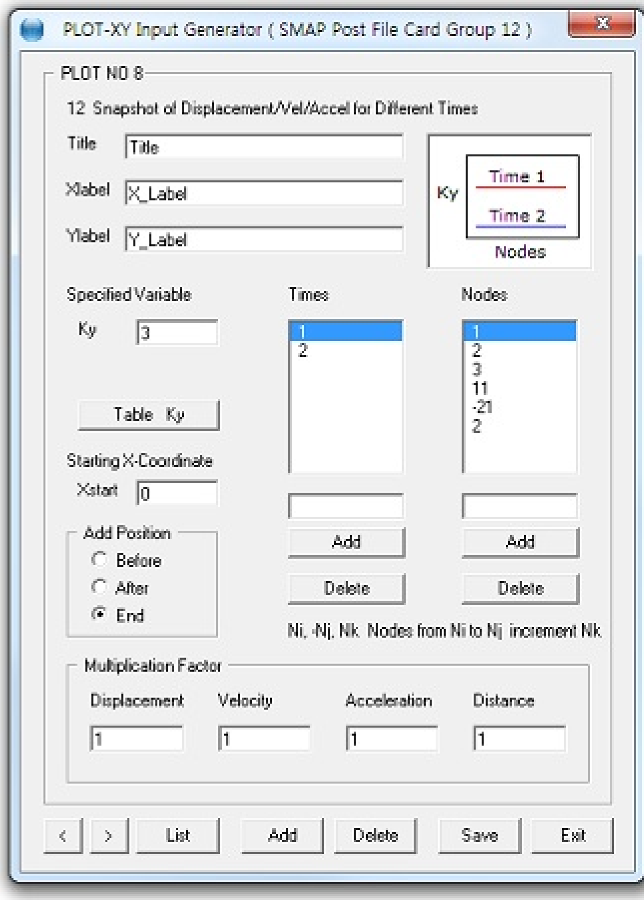


Figure 12.24 Displ/vel/accel snapshot for different times

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:

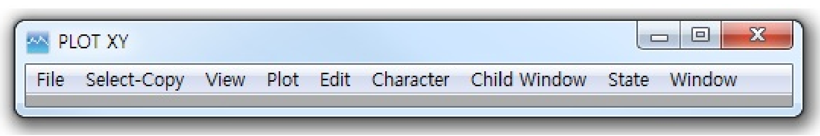
- Plot scatterplot data
  It reads the scatterplot data in text file and plots lines connecting each pair of data points.

- 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.

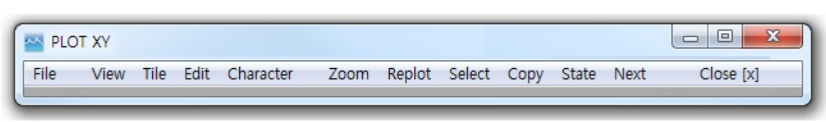
- Edit XY graph
  It reads XY data, edits titles and scales, adds user-defined additional curves.

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

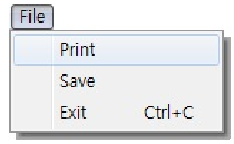


Express Style includes 12 menus which are rearranged so as to quickly access most frequently used menu items in practice.
For Express Style, specify 0 in C:\Smap\Ct\Ctdata\MenuStyle_XY.dat



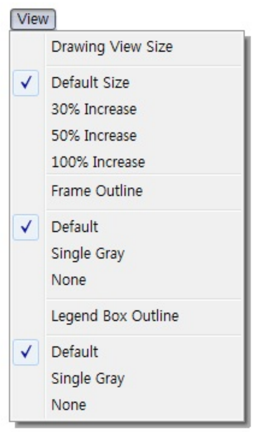
13.2 Menus

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.



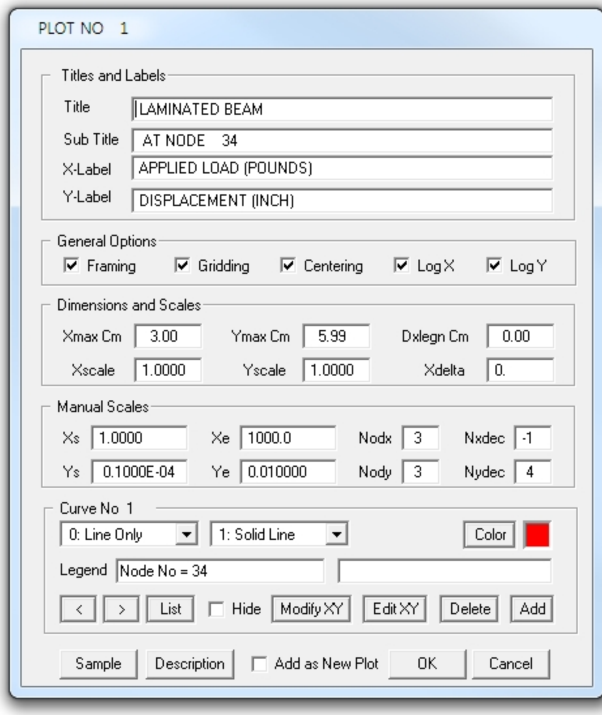
Select-Copy is mainly used to select and then copy the current view.

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

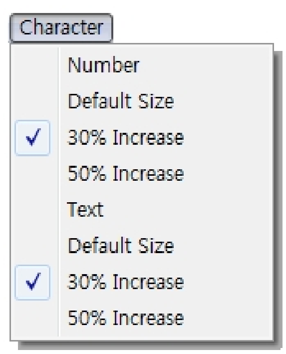


Plot has the following five sub menus.
Replot is to replot the currently focused child window. Zoom is to zoom the currently focused child window. Once this sub menu is selected, you can specify the rectangular zoom area by left mouse button down at the left top corner and then left mouse button up at the right bottom corner.
Hardcopy is to print the currently focused window.
Next is to plot the next graph.
Stop is to stop plotting.

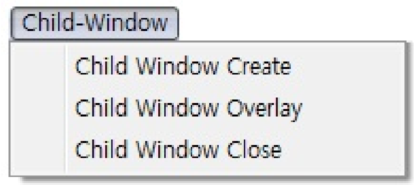
Edit opens following dialog to edit XY graph data.
It is described in detail in Section 12.3 in XY graph User's Manual.



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:

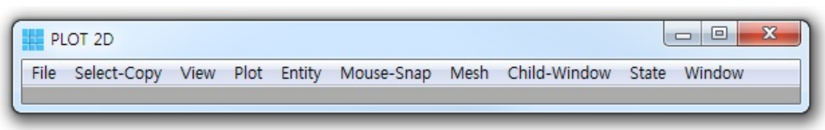
- **Plot finite element meshes**
  It reads the Mesh File and plots meshes along with node, element, boundary code, and material numbers.

- **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.

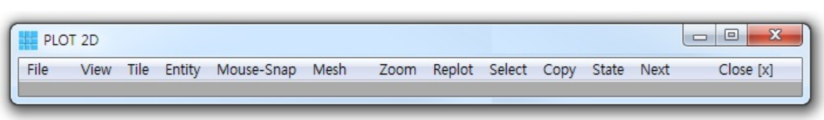
- **Edit finite element or group meshes**
  It reads finite element or group mesh files and edit these meshes.

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

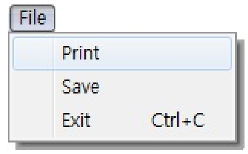


Express Style includes 13 menus which are rearranged so as to quickly access most frequently used menu items in practice.
For Express Style, specify 0 in C:\Smap\Ct\Ctdata\MenuStyle_2D.dat



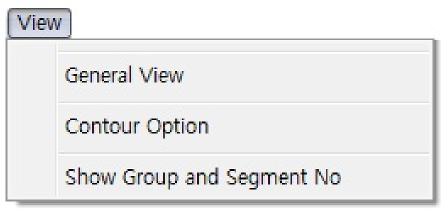
14.2 Menus

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.

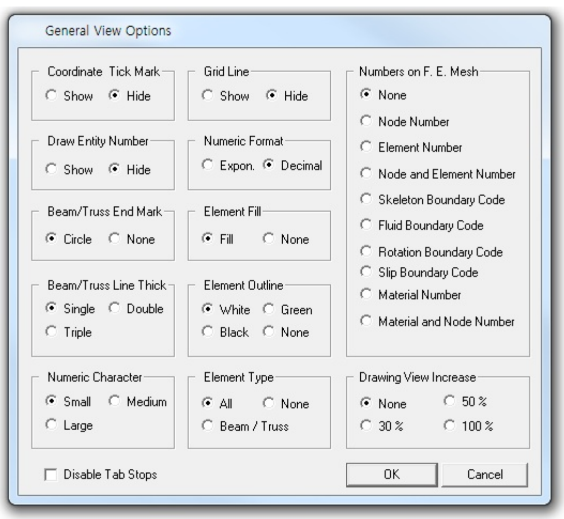


Select-Copy is mainly used to select and then copy the current view.

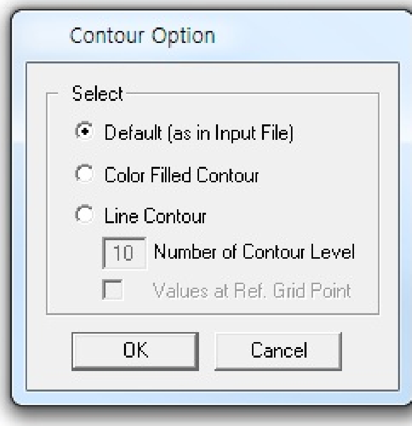
View has three sub menus; General View, Contour Option, and Show Group and Segment No.



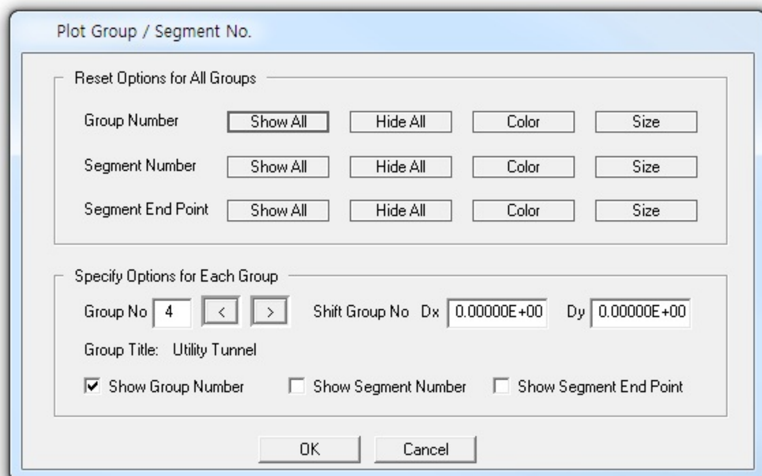
General View options affect all types of plots.



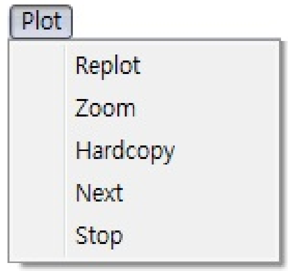
Contour Options affect contour plots of continuum element data for analysis results.



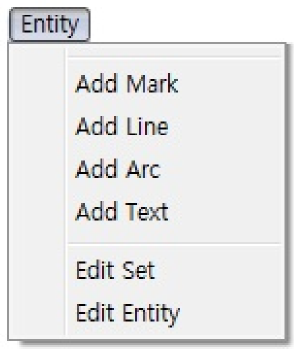
Show Group and Segment No is to show group and segment numbers when editing group meshes. It is described in detail in Section 5.3 in Group Mesh User's Manual.



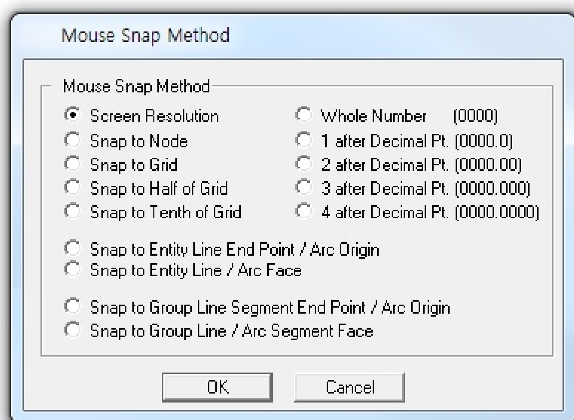
Plot has the following five sub menus.
Replot is to replot the currently focused child window. Zoom is to zoom the currently focused child window.
It zooms only mesh. Once this sub menu is selected, you can specify the rectangular zoom area by left mouse button down at the left top corner and then left 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.



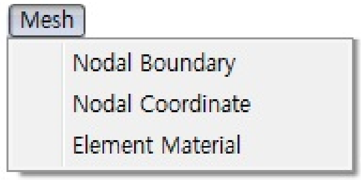
Entity is the graphical object which is mainly used to assist editing geometry of groups and elements. It has following six sub menus; Add Mark, Add Line, Add Arc, Add Text, Edit Set, and Edit Entity. It is described in detail in Section 5.7 in Group Mesh User's Manual.



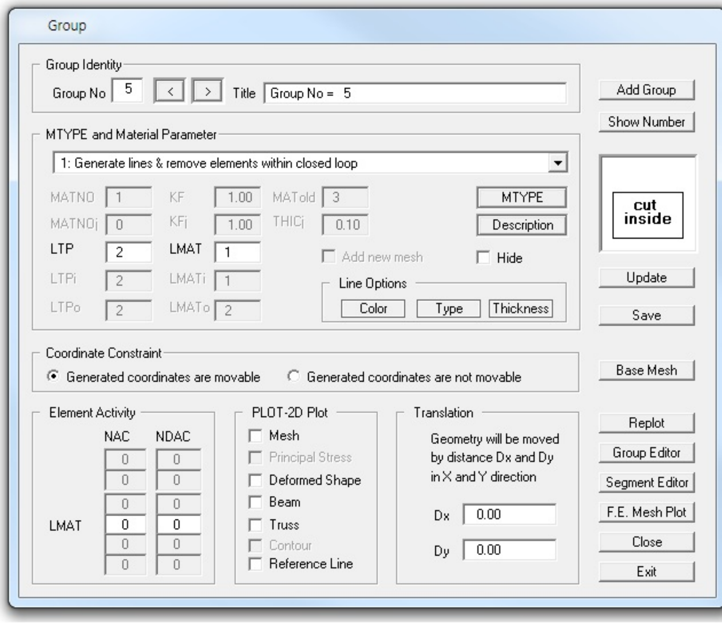
Mouse-Snap is to control the position of mouse cursor when you work for finite element mesh, group mesh, or entities. Mouse Snap Method helps you place the mouse cursor more accurately.



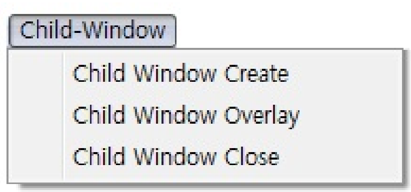
Mesh is used to directly modify finite element meshes. It has three sub menus; Nodal Boundary, Nodal Coordinate, and Element Material.
It is described in detail in Section 5.6 in Group Mesh User's Manual.



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.

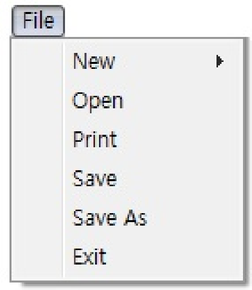
- **Compute intersections of surfaces**
  It reads the Mesh File containing shell elements for 3D surfaces and shows the locations of the computed intersections. The computed coordinates of intersections are saved in a file "Intersection.dat" which can be used for the construction of complicated 3D meshes.

PLOT-3D has 5 menus; File, Model, Plot, View and Help along with 25 toolbars.

15.2 Menus

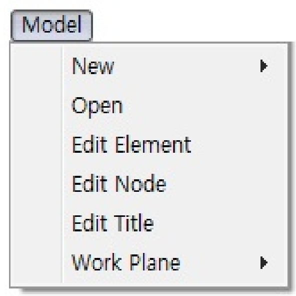
File has six sub menus.
New is used to build Finite Element Mesh or Block Mesh.
Open is used to open existing mesh file.
Print is to get the hard copy of the current view.
Save is to save the current mesh file or current view.

Save As is to save the current mesh file as another name.
Exit is to exit PLOT-3D.

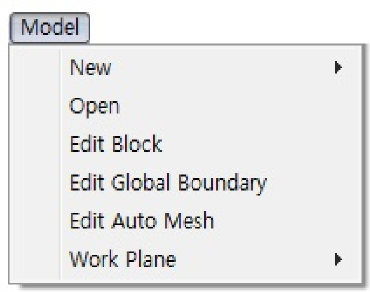


Model is mainly used to edit Finite Element or Block Mesh file.
For detailed description, refer to Block Mesh User's Manual in Section 6.

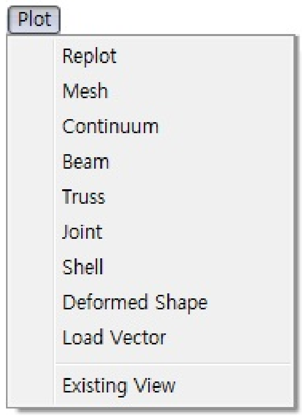
For editing Finite Element Mesh, 6 menus are shown.
New is to build new mesh file.
Open is to open existing mesh file.
Edit Element is to edit parameters related to element.
Edit Node is to edit parameters related to node.
Edit Title is to edit title.
Work Plane is to show prebuilt work planes.



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.

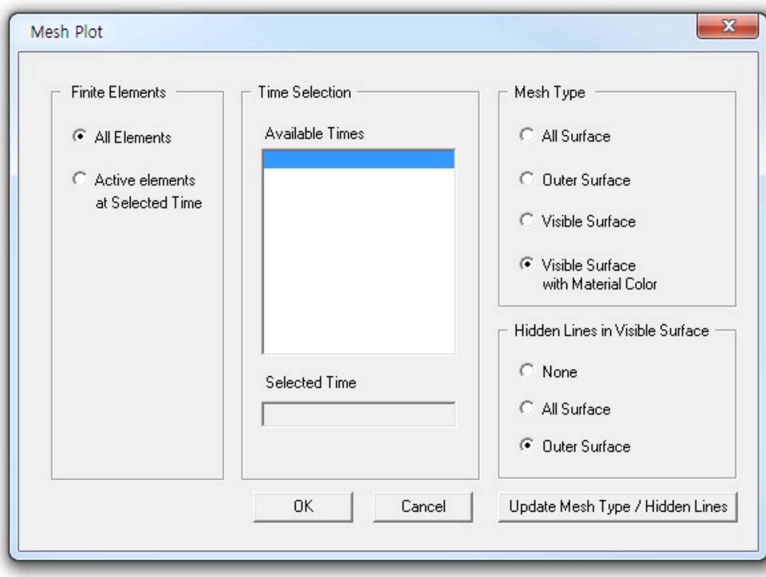


Plot is mainly used to plot Finite Element mesh and analysis results.
It has 10 sub menus; Replot, Mesh, Continuum, Beam, Truss, Joint, Shell, Deformed Shape, Load Vector, Existing View.
Joint plot is not available.

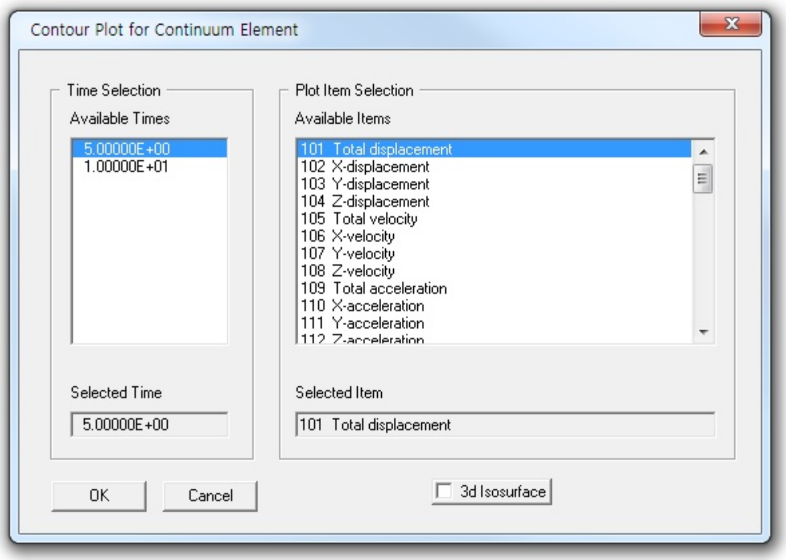


Replot is mainly used to refresh the current view.

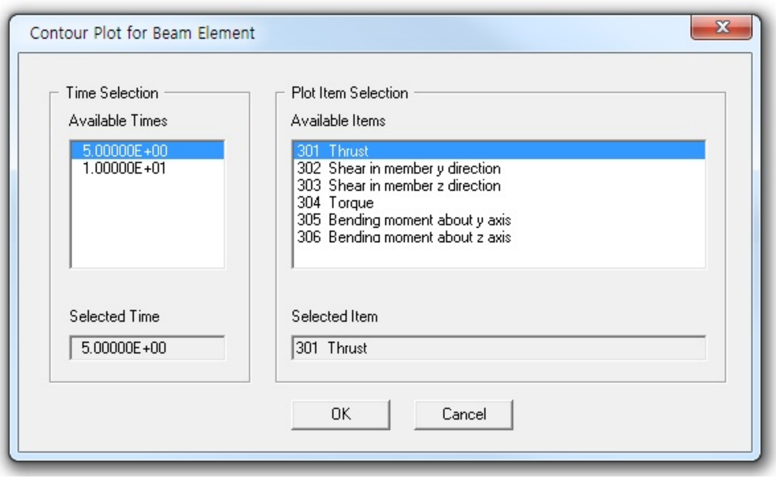
Mesh is to plot Finite Element meshes (Default plot type).
Mesh plot requires only Mesh File.



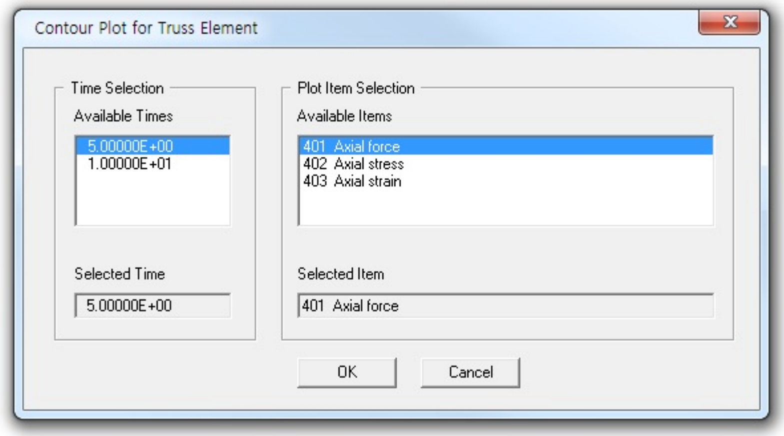
Continuum is to plot contours or principal stress vectors for continuum elements. By checking "3d Isosurface", iso surface will be shown.



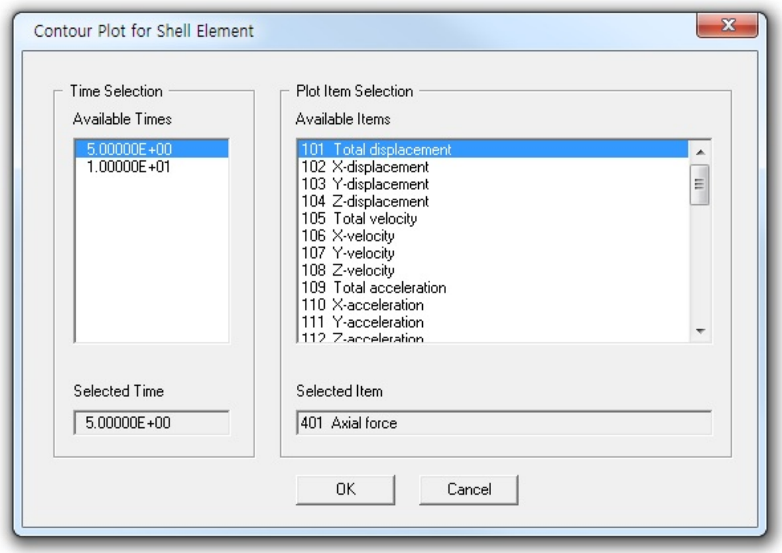
Beam is to plot section forces of beam elements.



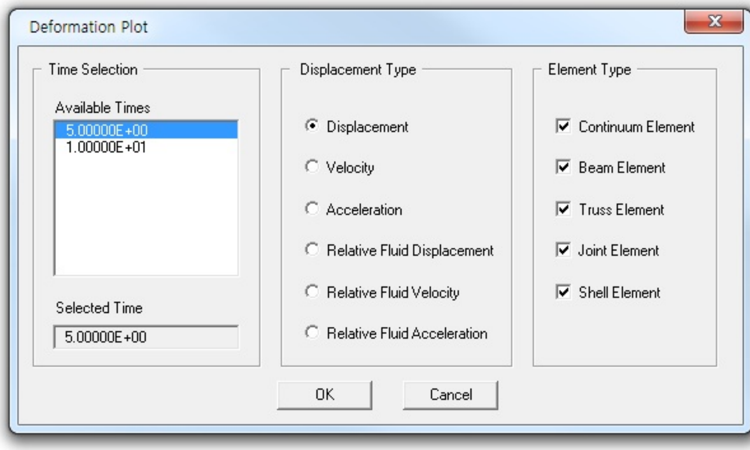
Truss is to plot axial force/stress/strain of truss elements.



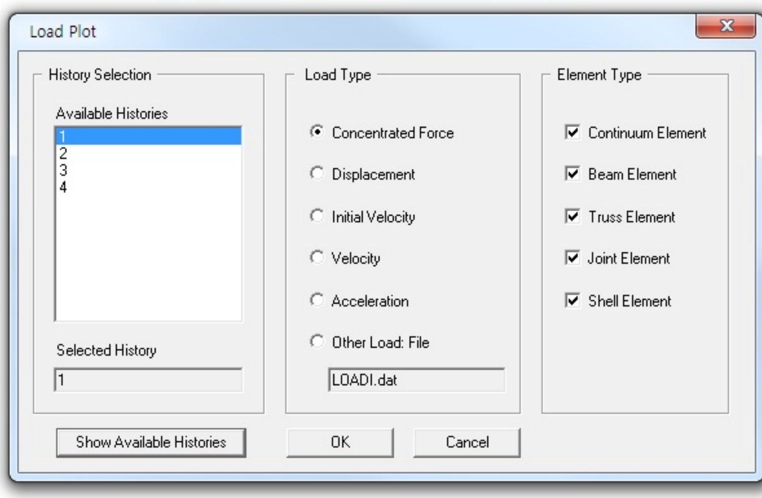
Shell is to plot contours or principal stress vectors for shell elements.



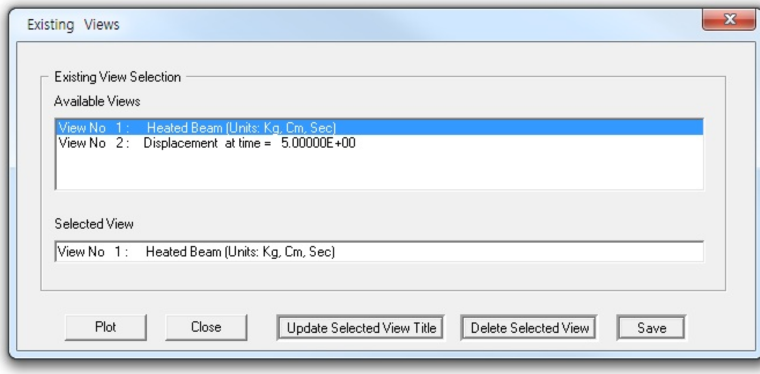
Deformed Shape is to plot the snap shot of all kinds of displacement/velocity/accelerations.
Note that deformed meshes can be combined with other plot types as discussed in "Displacement" option in view menu.



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.

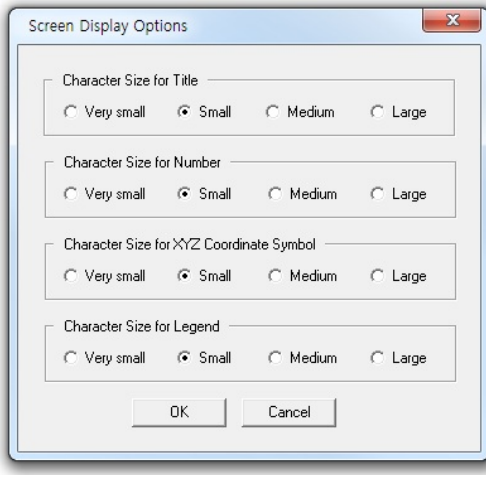


View is used to change the appearance of a selected plot.

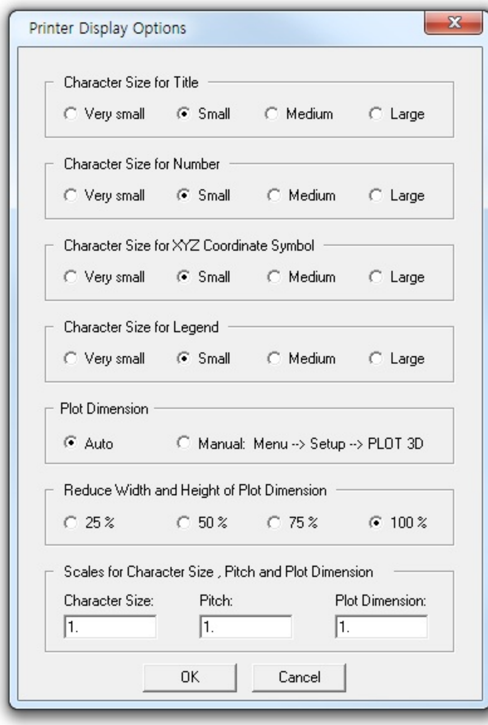
It has eleven sub menus; General, Screen, Printer, Contour, Clip Plane, Mesh, Beam, Truss, Principal Stress, Displacement, and Load Vector.

General view options affect most plot types.

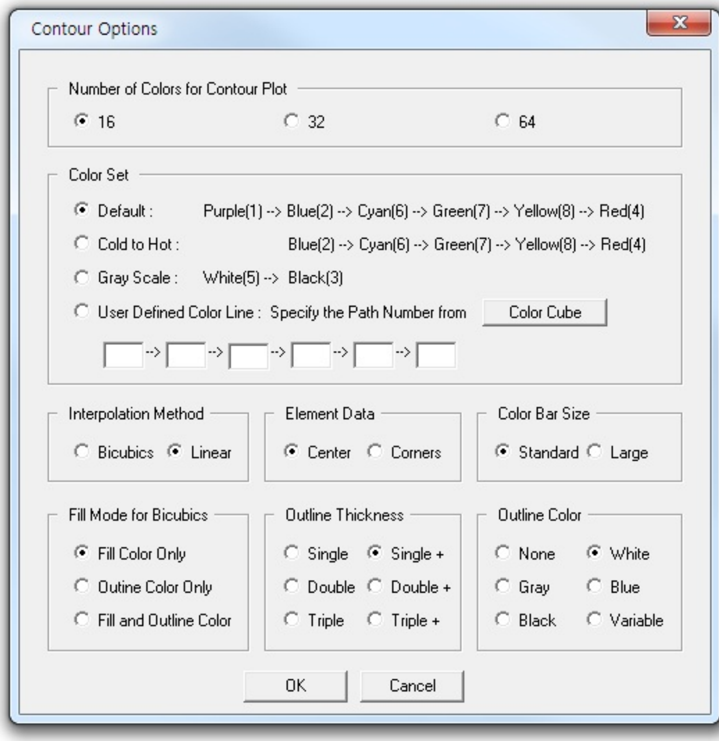
Screen display options affect character sizes shown on the monitor.



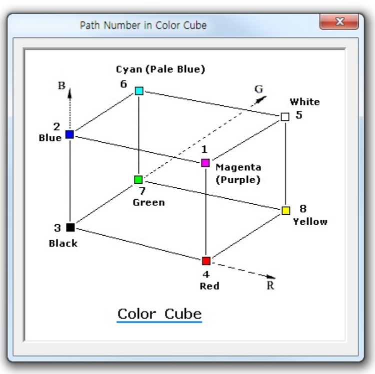
Printer display options affect character sizes and plot dimensions shown on the hard copy.



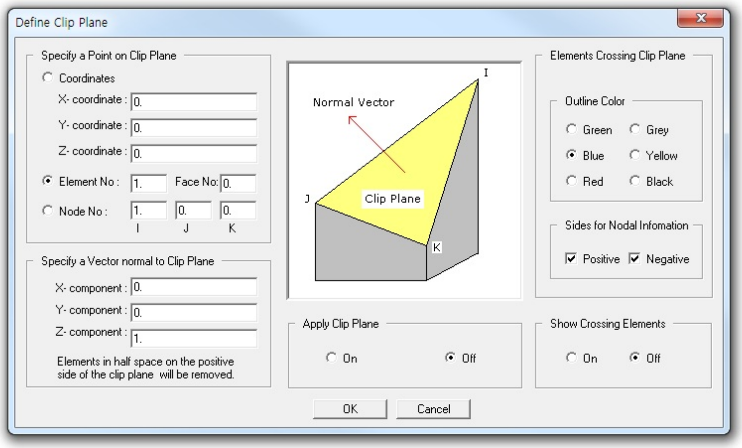
Contour options affect all types of plots involving contours.



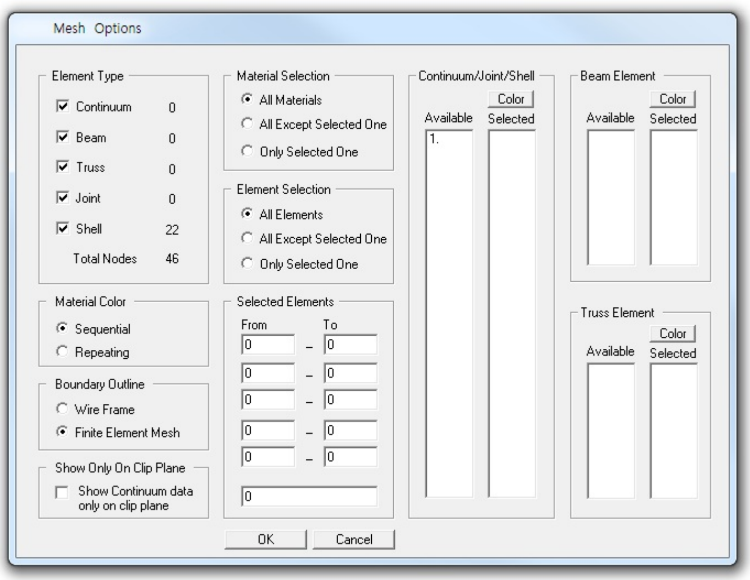
Color cube is to use for user defined color line.



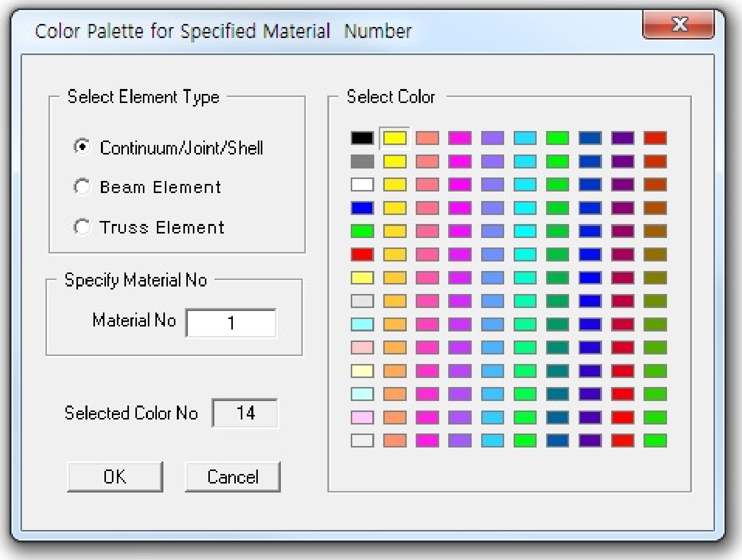
Clip plane defines parameters associated with the clip plane which cuts through the internal part of the 3D domain. When "Apply Clip Plane" is on, contours or deformed shapes are shown on such user defined plane.



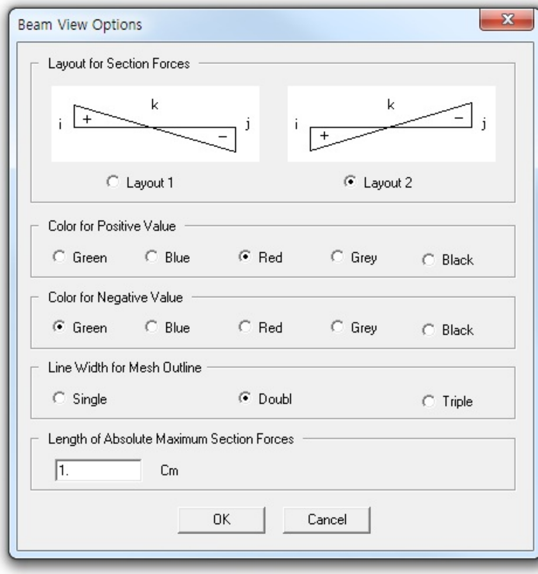
Mesh options affect all plot types. As one of useful features, it can select particular types of elements and materials.



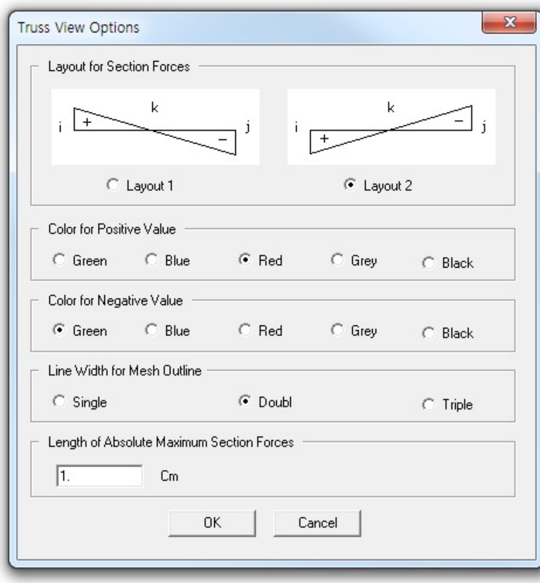
Color is to use for user defined mesh color.



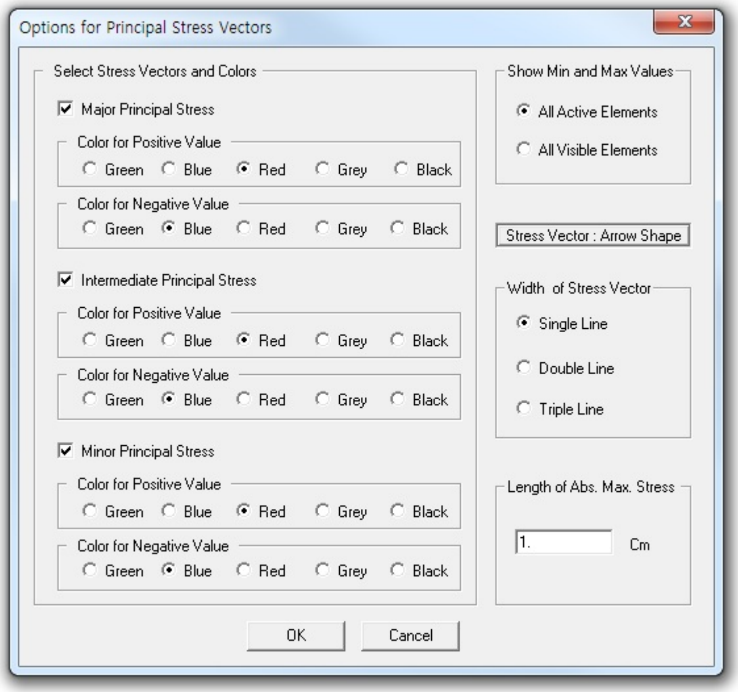
Beam view options affect only beam plot.



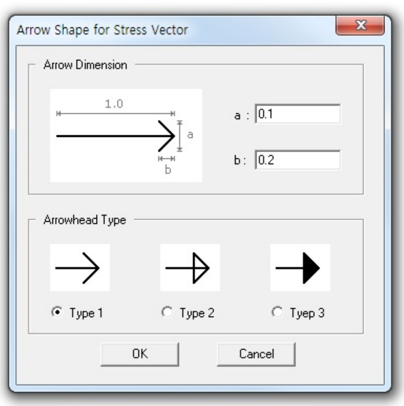
Truss view options affect only truss plot.



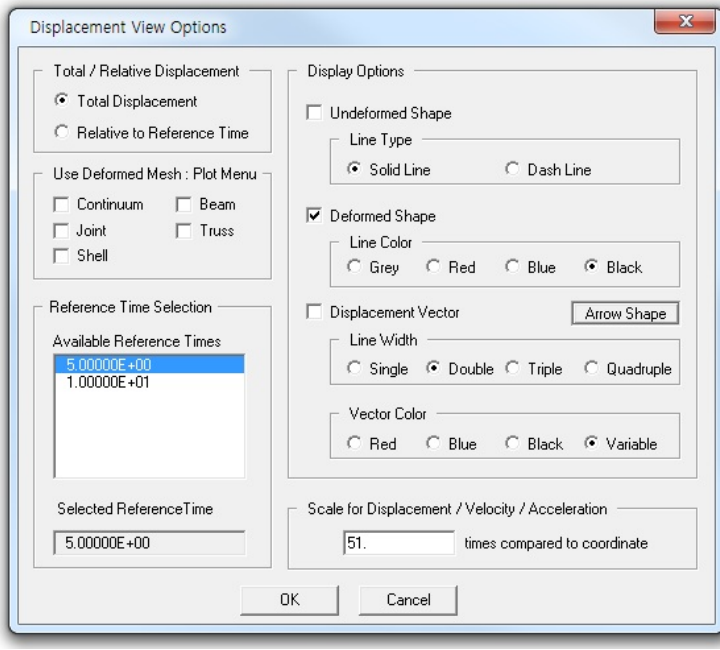
Principal Stress options affect only plots of principal stress vectors in continuum or shell elements.



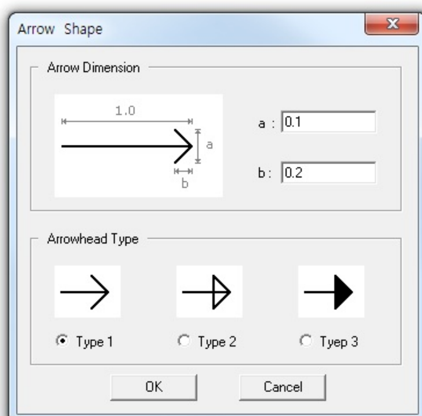
Users can specify the arrow shape for stress vector.



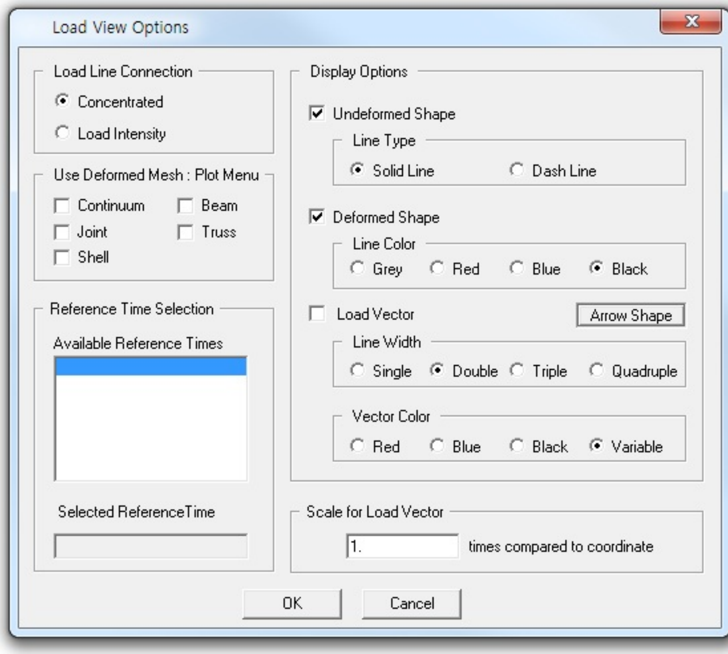
Displacement view options affect only deformed shape plot.
Continuum, Beam, Truss, and Shell plots can be displayed
over deformed mesh by checking types in "Use Deformed Mesh".



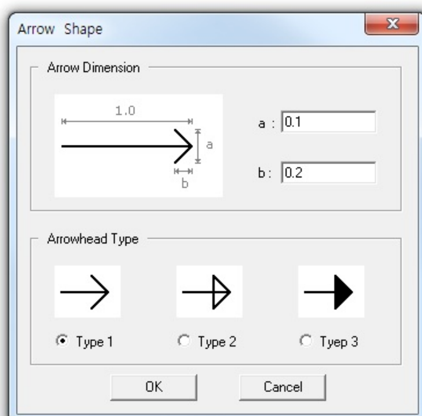
Users can specify the arrow shape
for displacement vector.



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

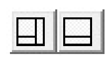
This button is used to edit finite element or block mesh.

Work Plane Toolbar

This button is to set work plane used for Model.

Layout Toolbar

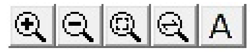
These buttons are used to show different layouts.
The first button divides the plot area into three parts; mesh, title, and legend. The second button divides the plot area into two parts; mesh and title.



XYZ Toolbar

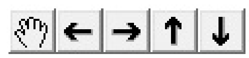
This button is used to locate position of XYZ coordinate symbol in the two part layout mode. Each time you click this button, the XYZ symbol moves counterclockwise along the corners of rectangle. XYZ button is also used to control the amount of movement, rotation, and zoom.

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 ® - S2

Structure Medium Analysis Program

2-D Static Analysis

Example Problems

LICENSE AGREEMENT

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

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

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

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

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

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

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

Contents

Introduction

Example Problems are mainly provided:

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

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

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

Section 4 illustrates SMAP-S2 main example problems as summarized in Table 1.1. First 8 problems are presented to demonstrate the accuracy and validity of SMAP-S2 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-S2 example problem

Problem Number	Project File Name	Run Time Pent. III 850	Description
1	VP1.dat	0.07 min.	Circular unlined tunnel Check: • Mohr-Coulomb model • Axisymmetric element
2	VP2.dat	0.07	William's toggled beam by continuum Check: • Geometric nonlinearity in continuum element
	VP2-1.dat		William's toggle by beam element
3	VP3.dat	0.30	Burn's and Siess' beam Check: • Concrete cracking & crushing • Steel yielding
4	VP4.dat	0.02	Buckled truss analysis Check: • Post-buckling of truss member
5	VP5.dat	0.01	Plane strain tunnel analysis Check: • Initial soil stress • Excavation & construction • Liner-soil interaction
6	VP6.dat	0.52	Laminated beam with slip interface Check: • Joint element • Joint model
7	VP7.dat	0.07	Buried pipe analysis
8	VP8.dat	0.02	Heated beam modeled by beam
	VP8-1.dat		Heated beam modeled by continuum

Table 1.1 List of SMAP-S2 example problem, continued

Problem Number	Project File Name	Run Time Pent. III 850	Description
9	VP9.dat	0.01 min.	Frames with hinge connection Units in Ton/Meter/Sec
	VP9-1.dat		Units in Pound/Inch/Sec NBLT = 0 NSPTB = 2 NSTYPE = 2
	VP9-2.dat		NBLT = 0 & NSPTB = 5
	VP9-3.dat		NBLT = 0 & NSPTB = -5
	VP9-4.dat		NBLT = 1 & NSPTB = 2
	VP9-5.dat		NBLT = 0 NSPTB = 2 NSTYPE = 20
	VP9-6.dat		Same as VP9-4 except variable E
	VP9-7.dat		NBLT = 2 NSPTB = 2
	VP9-8.dat		With Concentrated Load at Midspan NSTYPE = 16
10	VP10.dat		Embedded rebars with slip
11	VP11.dat		Pseudo dynamic embankment fill
12	VP12.dat		Excavation on nearby box frame
13	VP13.dat		Spring analysis
14	VP14.dat		Nonlinear truss analysis
15	VP15.dat		Frames with rotational spring connection
16	VP16.dat		Reinforced concrete cylinder
17	VP17.dat		Silo lining analysis

Pre-Processing Programs

Pre-Processing programs are mainly used to generate Mesh File described in Section 4.3 of SMAP-S2 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-S2, 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-S2 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-S2 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-S2
Example Problem

SMAP-S2 is the main-processing program which computes static response of two-dimensional problems. Input parameters of SMAP-S2 are described in detail in Section 4 of SMAP-S2 User's Manual.

Running SMAP-S2 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-S2 is outlined in Section 3.5 of User's Manual. Once you finished execution of SMAP-S2, 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-S2 User's Manual.

Table 1.1 in Section 1 shows the summary of SMAP-S2 example problems. First eight example problems are the verification problems. The main objective of these verification problems is to demonstrate the accuracy and validity of SMAP-S2.

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

C:\Smap\SmapS2\Example\Smap

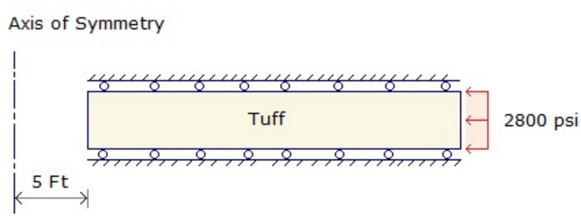
For each example problem, brief problem descriptions and partial graphical outputs will be presented in this section.

4.1 Circular Unlined Tunnel

As schematically shown in Figure 4.1, a ten-foot diameter unlined circular tunnel is subjected to a uniform free field stress of 2800 psi. Figure 4.2 shows the corresponding finite element model. A refined constant element length is used to model the first 36 inches of rock, while a coarser grid is used to extend the surrounding rock to 3000 inches from the tunnel centerline.

This problem is to verify one of the nonlinear material models available in SMAP-S2. The Generalized Mohr-Coulomb Model described in Card Group 5.3 in User's Manual can produce Drucker-Prager Model when the parameter K is equal to unity, where K is the ratio of the shear strength in triaxial extension to the shear strength in the triaxial compression at the same mean pressure. The same problem has been analyzed by the semi-analytical finite difference code FDAXP (Kim and Davister, 1987) for comparison.

Computed radial, tangential, and axial stress profiles are shown in Figures 4.3, 4.4, and 4.5, respectively. As seen, SMAP-S2 results agree very well with semi-analytical solution.

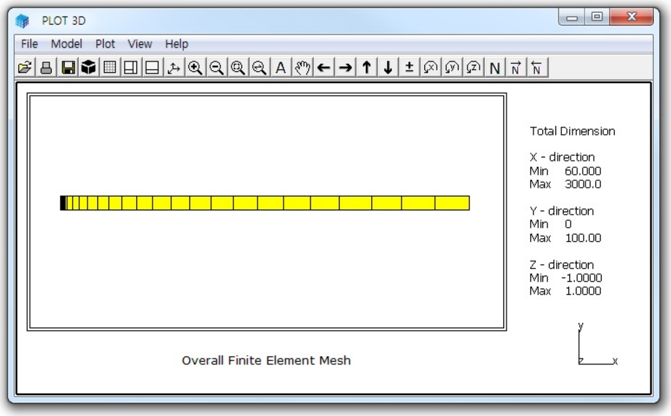


$E = 1.15 \times 10^{6}$ psi $\quad \nu = 0.33$

$\sigma_c = 1800$ psi (Unconfined Strength)

$\phi = 18^{\circ}$ (Friction Angle)

Figure 4.1 Schematic tunnel section (Not Scaled)



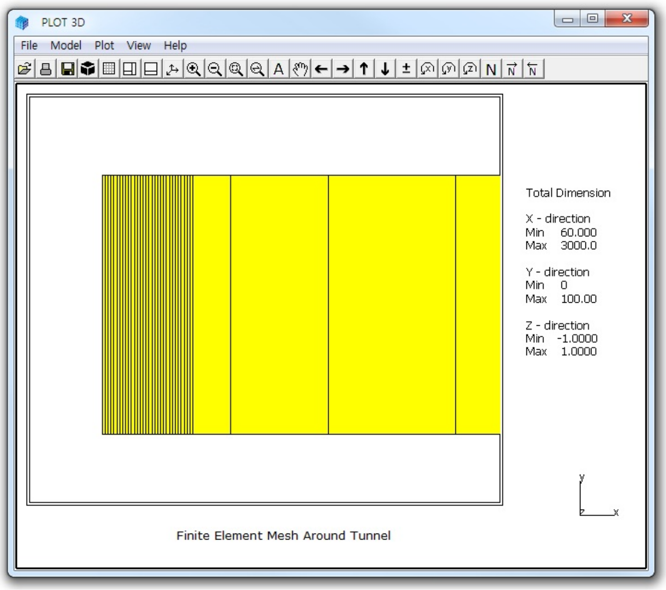


Figure 4.2 Finite element axisymmetric model

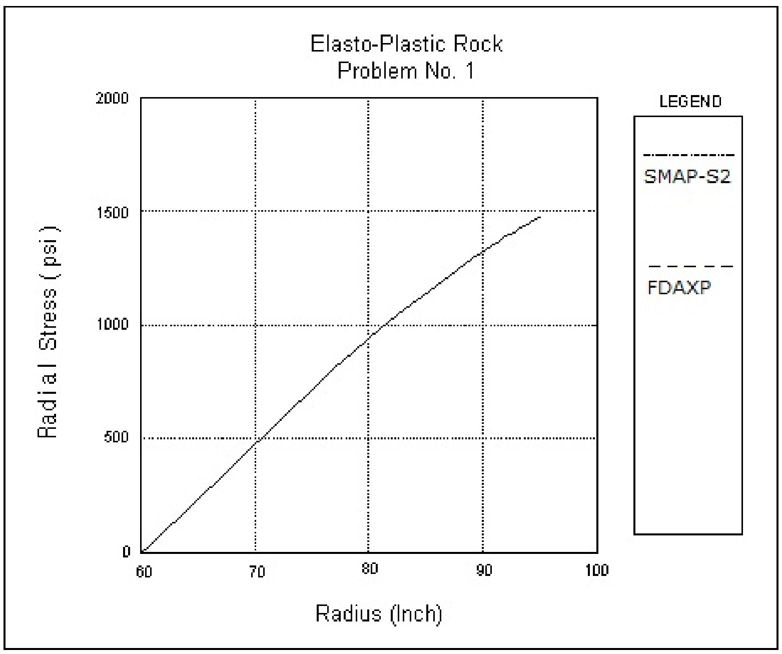


Figure 4.3 Radial stress profile

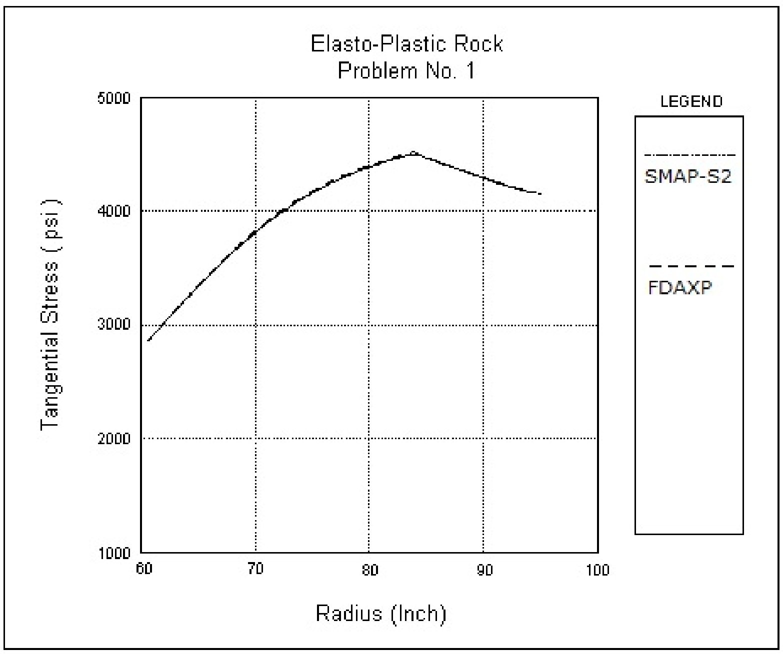


Figure 4.4 Tangential stress profile

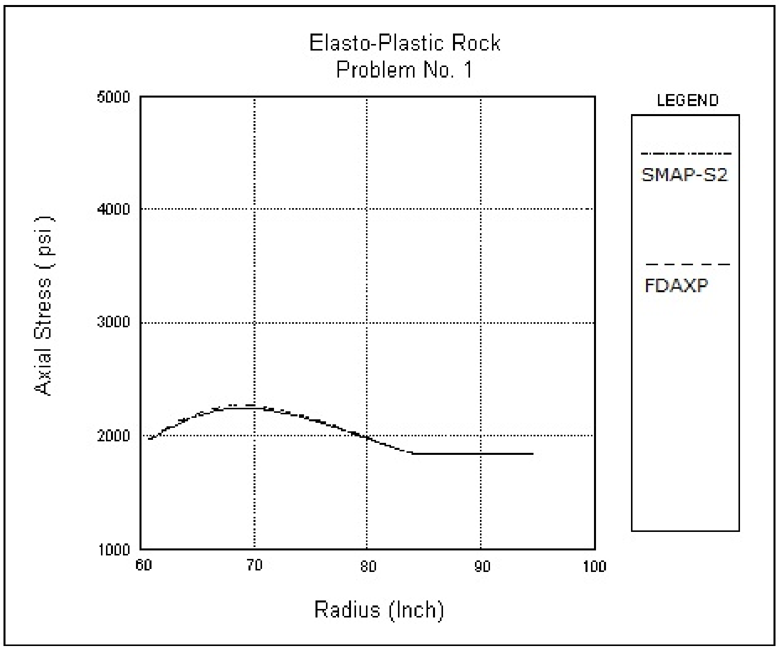


Figure 4.5 Axial stress profile

4.2 William's Toggled Beam Analysis

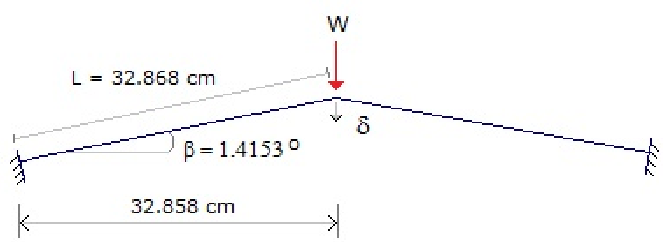
This classic problem of a rigidly jointed toggle is selected to verify the geometric nonlinear behavior of the continuum element.

For the toggle shown in Figure 4.6 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 4.7 and 4.8 show the load-deflection response at mid span and deformed shape at applied load of 16 kg, respectively.
SMAP-S2 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 4.9.



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

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

Figure 4.6 William's toggled beam (Not Scaled)

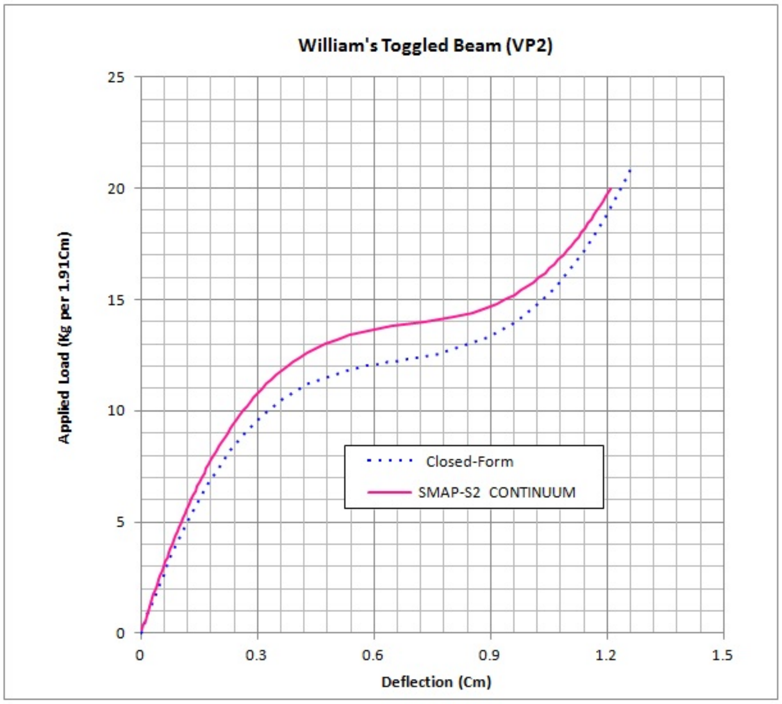


Figure 4.7 Load-deflection curve using continuum element

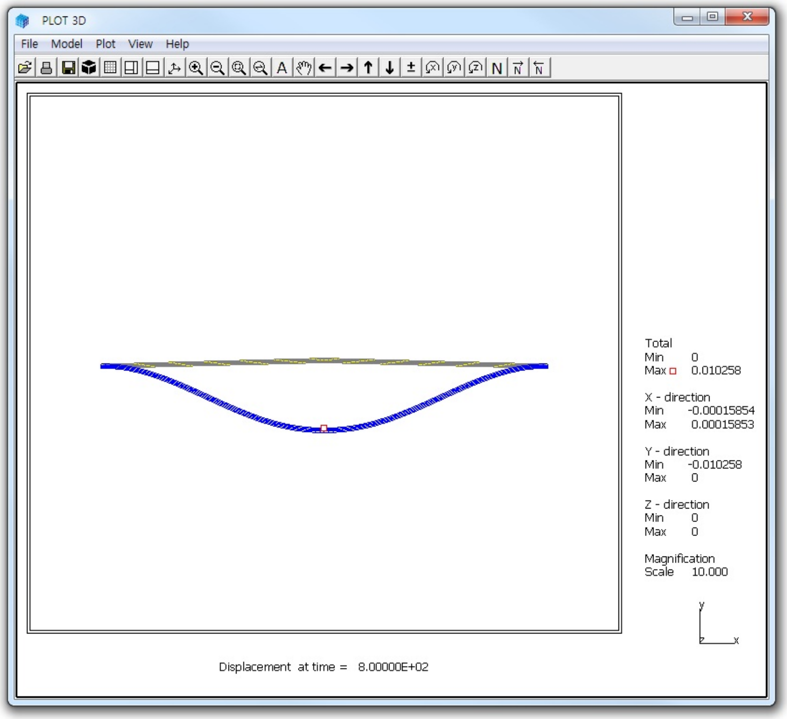


Figure 4.8 Deformed shape at applied load of 16 kg

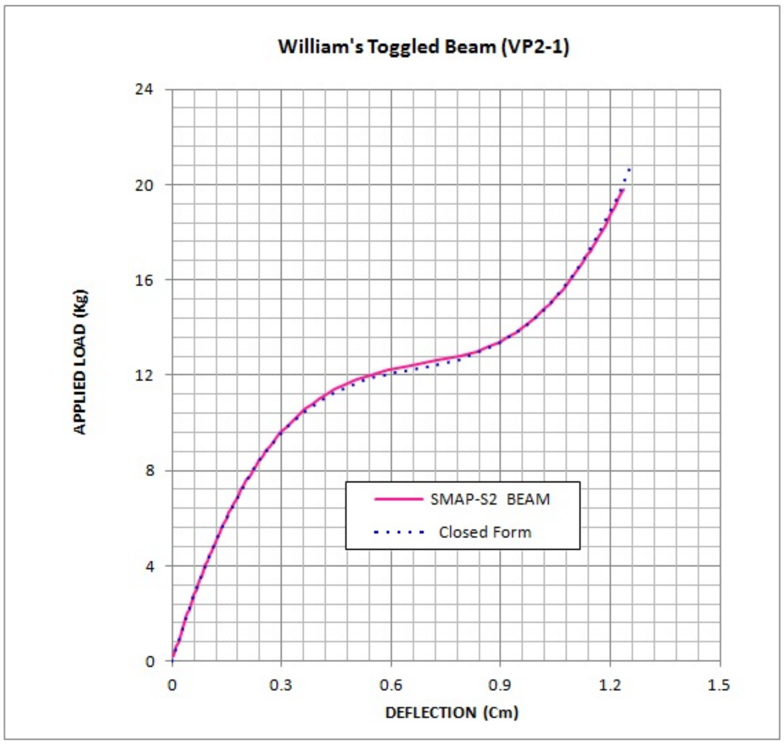


Figure 4.9 Load-deflection curve using beam element

4.3 Burn's and Siess' Beam Analysis

Burns and Siess' beam, shown in Figure 4.10 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

$$P_c = \frac{4M_c}{L} = \frac{4(28.71)}{12} = 9.57 \text{ kips}$$

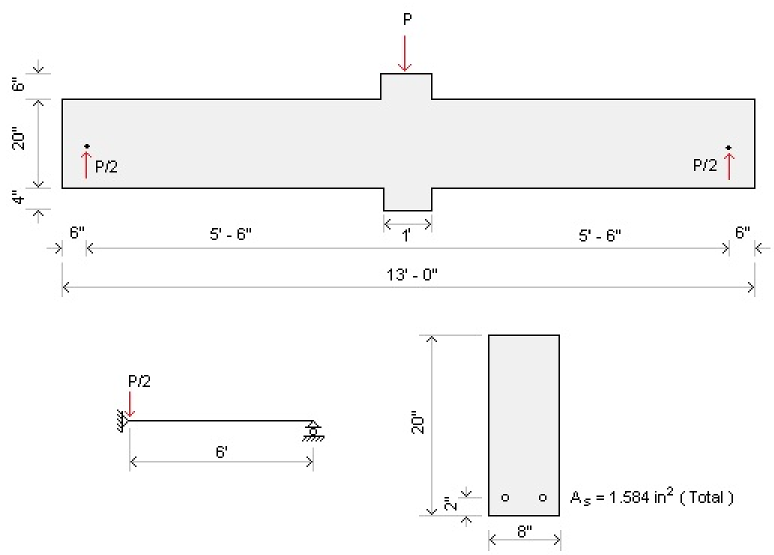
The maximum moment capacity, M_{max}, can be estimated by simplified ultimate strength theory as

$$M_{max} = A_s\, f_y\, (d - \frac{0.5\, A_s\, F_y}{0.85\, f_c^{/}\, b})$$

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 = 3.8×10^6 psi

υ = 0.15

f'_c = 4820 psi

f_t = 546 psi

φ = 40°

Steel Bar

E = 30×10^6 psi

υ = 0.3

f_y = 44,900 psi

Figure 4.10 Burns and Siess' beam

SMAP-S2 model is shown in Figure 4.11. A total of 22 beam elements is used to model right half of the structure. A constant concentrated load increment of 40 lbs is applied.

Figure 4.12 shows the load-deflection response at midspan. Figure 4.13 shows the deformed shapes at the applied loads; 20 and 32 kips. As seen, there is very good agreement between the experimental and SMAP-S2 results.

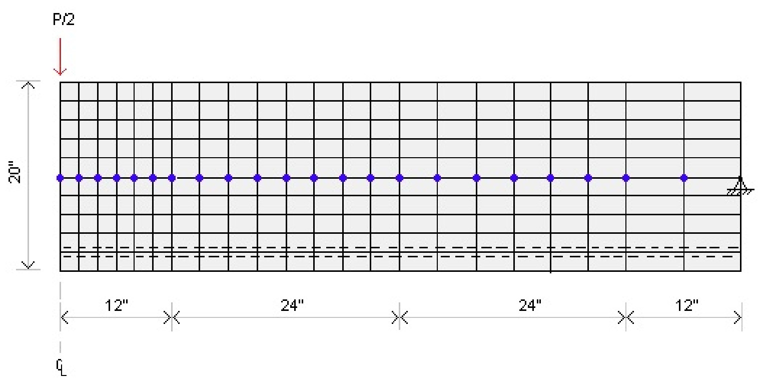


Figure 4.11 SMAP-S2 model of Burns and Siess' beam

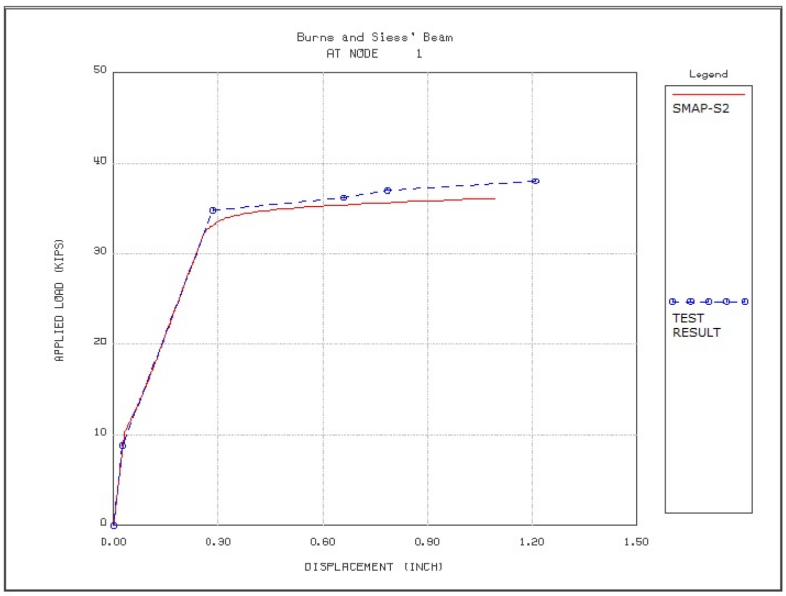


Figure 4.12 Load-deflection curve

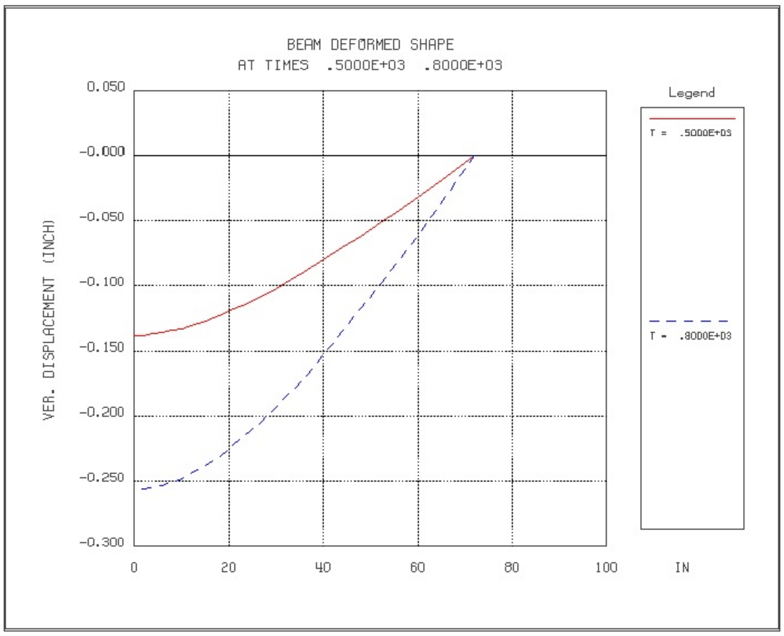
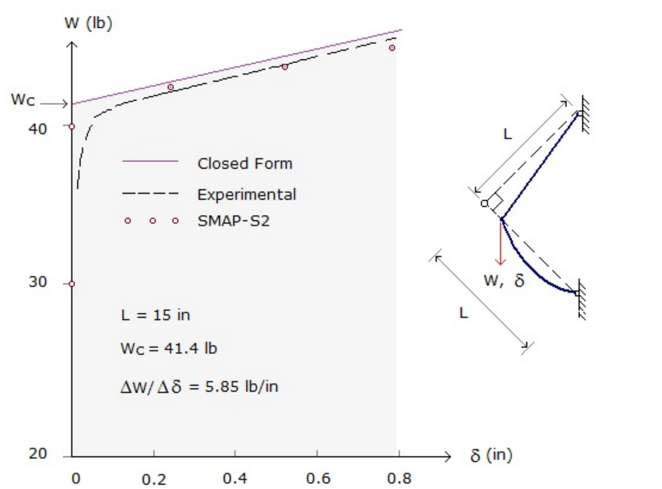


Figure 4.13 Deformed shapes at 20 and 32 kips

4.4 Buckled Truss

This truss problem, shown in Figure 4.14, consists of two identical members jointed at right angles. A vertical load, W, is applied monotonically until the structure buckles significantly. In fact, only the bottom member shortens and buckles while the top member elongates continuously.

Both theoretical closed form solution and experimental test result are presented by Britrec, 1973. As seen in Figure 4.14, SMAP-S2 results are close to the experimental test results.



E	=	3.28×10^{7} psi	A	=	0.0625 in^2
I	=	$2.03 \times 10^{-5} \text{ in}^4$	y_{max}	=	0.03 in
σ_y	=	36000 psi			

Figure 4.14 Buckled truss analysis

4.5 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-2D Example 4.16. Figure 4.15 shows schematic tunnel section view and material properties of soil and steel liner.

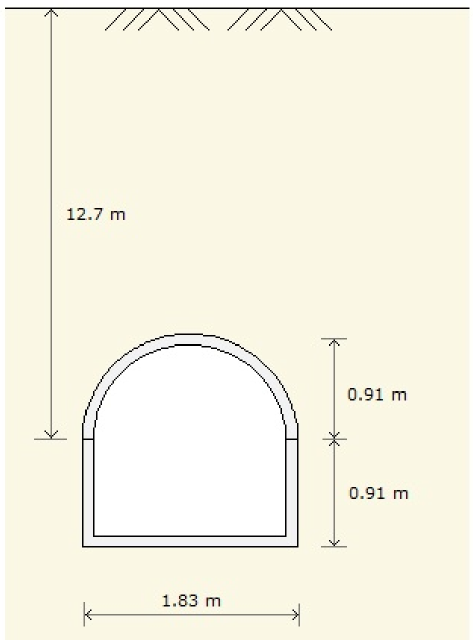
Figure 4.16 shows Finite element mesh. By symmetry, only the right half of the tunnel is modeled. Tunnel liner is modeled by beam elements as shown in Figure 4.17. Block mesh example 4 illustrates how to generate this mesh.

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 4.18 Tunnel deformed shape
Figure 4.19 Tunnel liner bending moment
Figure 4.20 Tunnel liner axial stress
Figure 4.21 Principal stress vector
Figure 4.22 Major principal stress distribution
Figure 4.23 Minor principal stress distribution

SMAP-S2 results are close to SMAP-2D results.



Soil

$E = 7{,}030\ t/m^2$
$\nu = 0.23$
$K_o = 0.3$

Steel Liner

$E = 20.4 \times 10^6\ t/m^2$
$\nu = 0.3$
$t = 2.54$ cm (Thickness)

Figure 4.15 Schematic tunnel section view (Not Scaled)

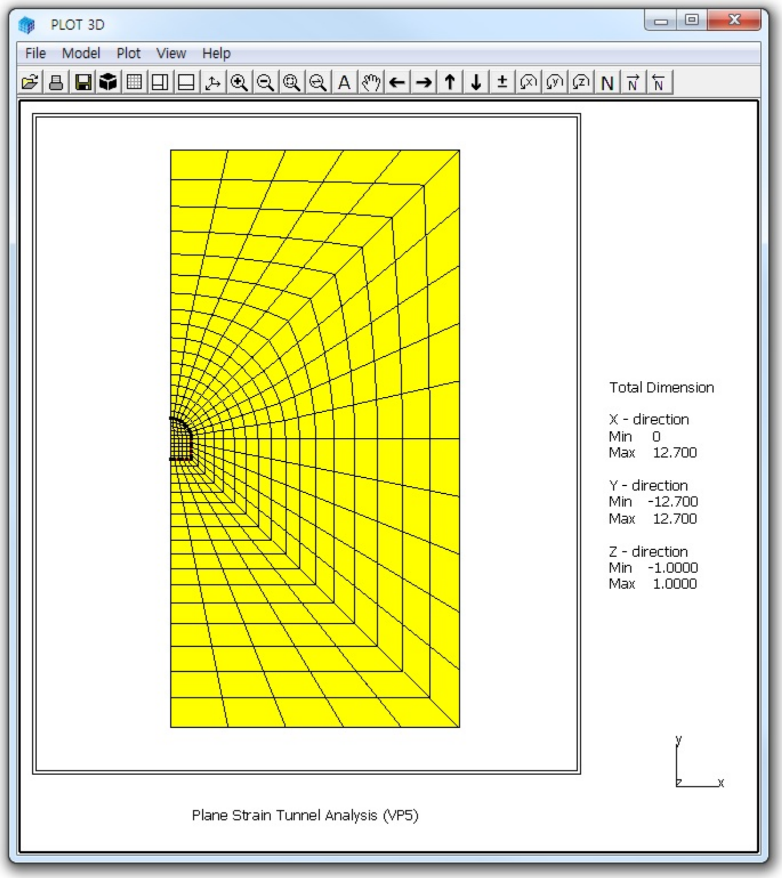


Figure 4.16 Finite element mesh

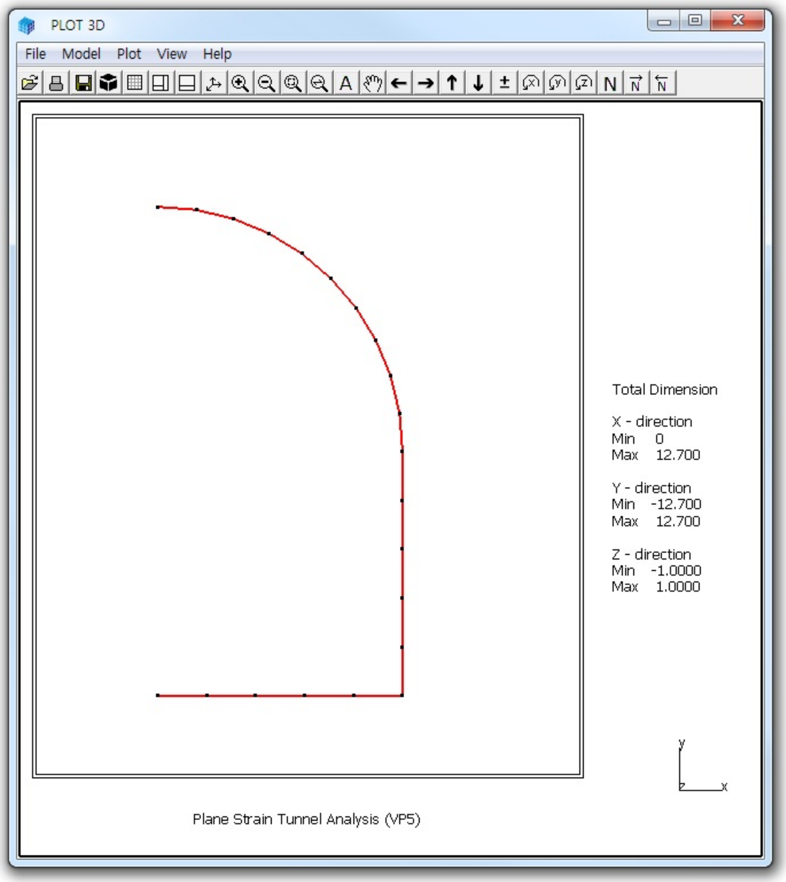


Figure 4.17 Finite element mesh for liner

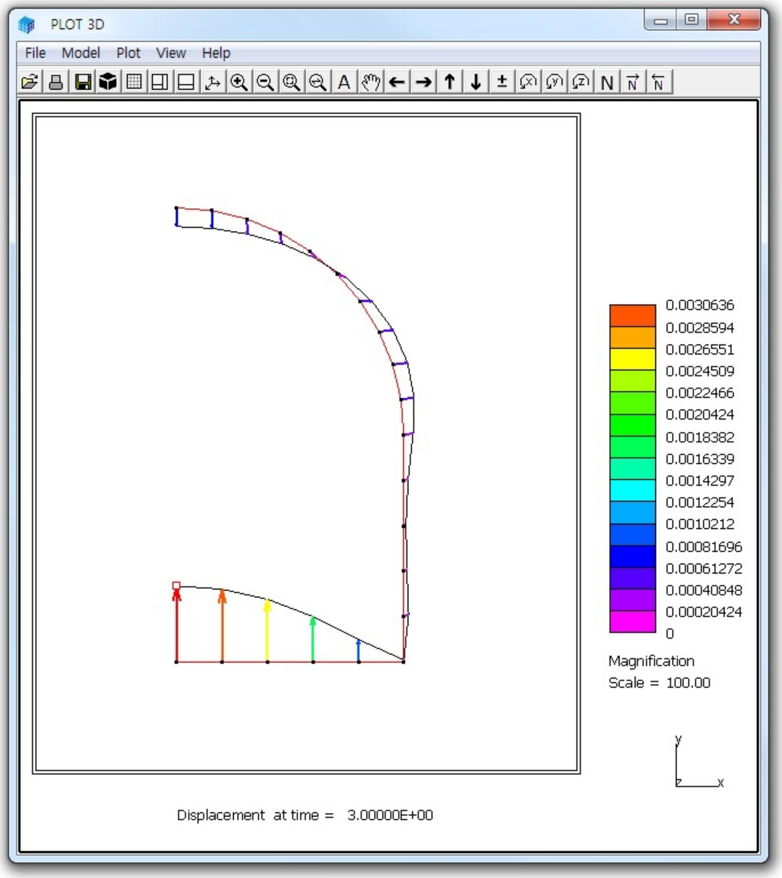


Figure 4.18 Tunnel deformed shape

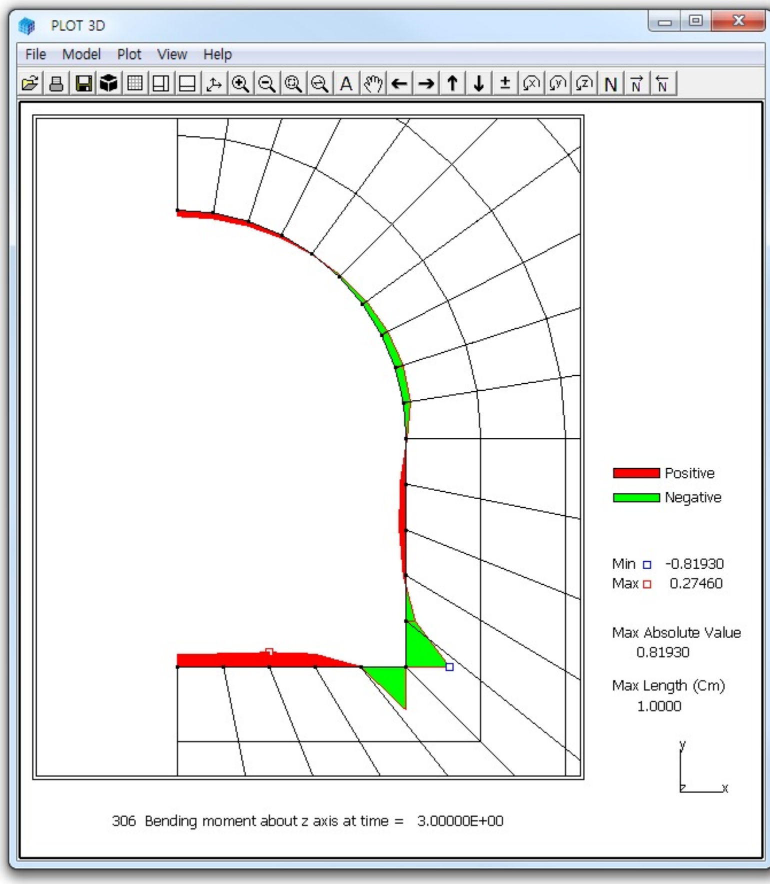


Figure 4.19 Tunnel liner bending moment

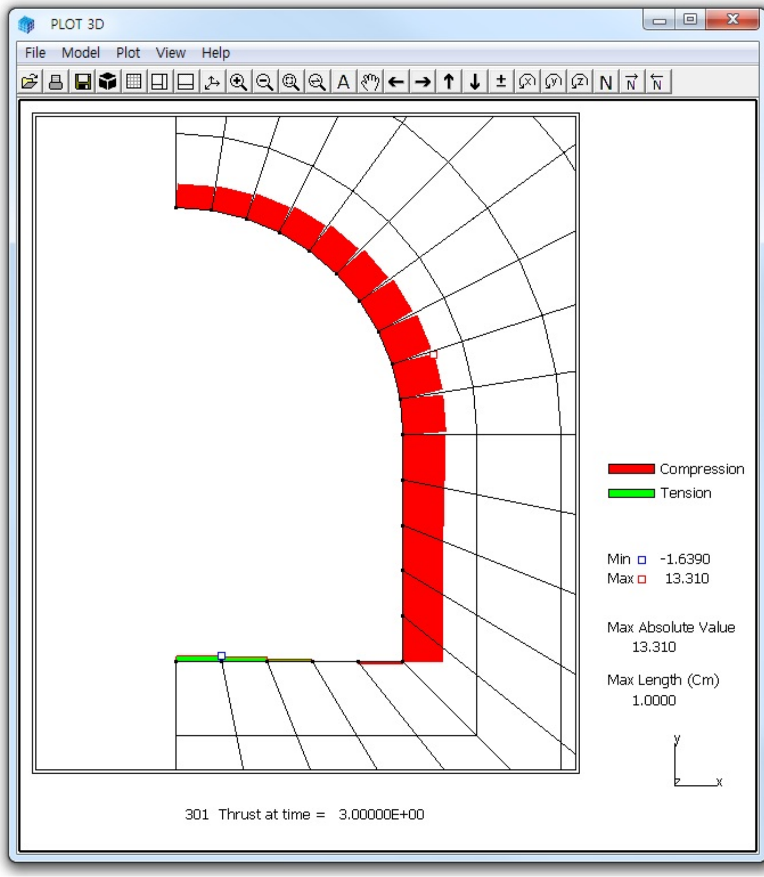


Figure 4.20 Tunnel liner axial force

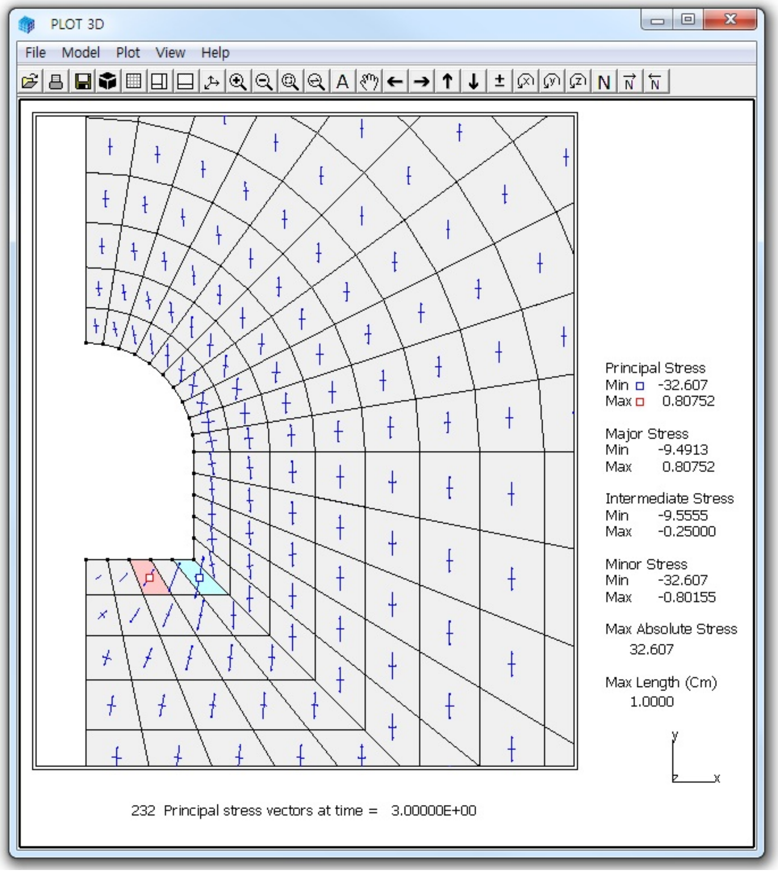


Figure 4.21 Principal stress vector

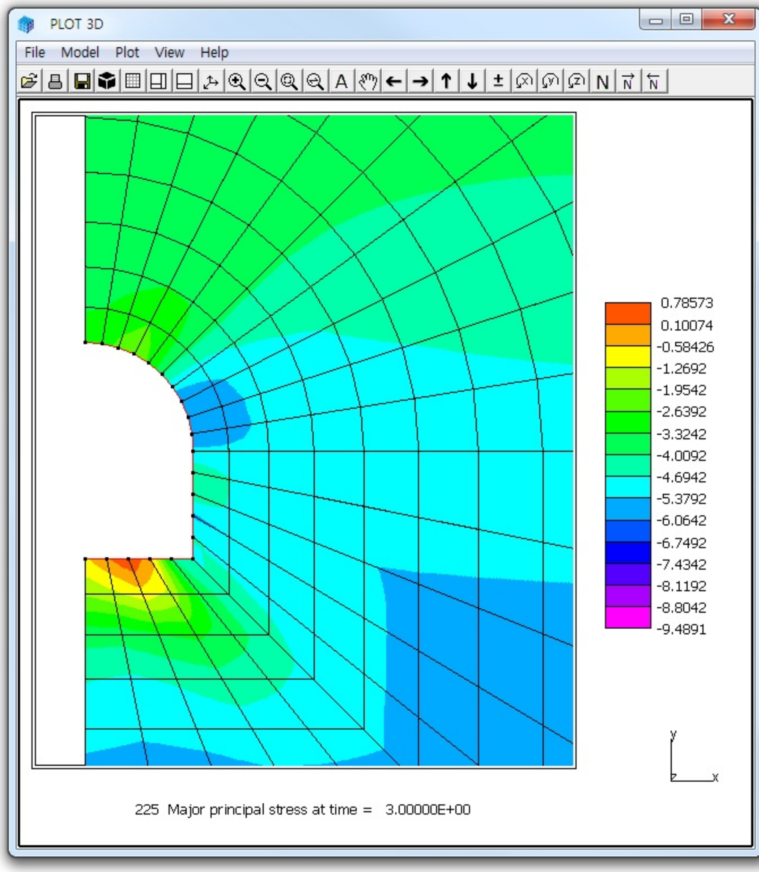


Figure 4.22 Major principal stress distribution

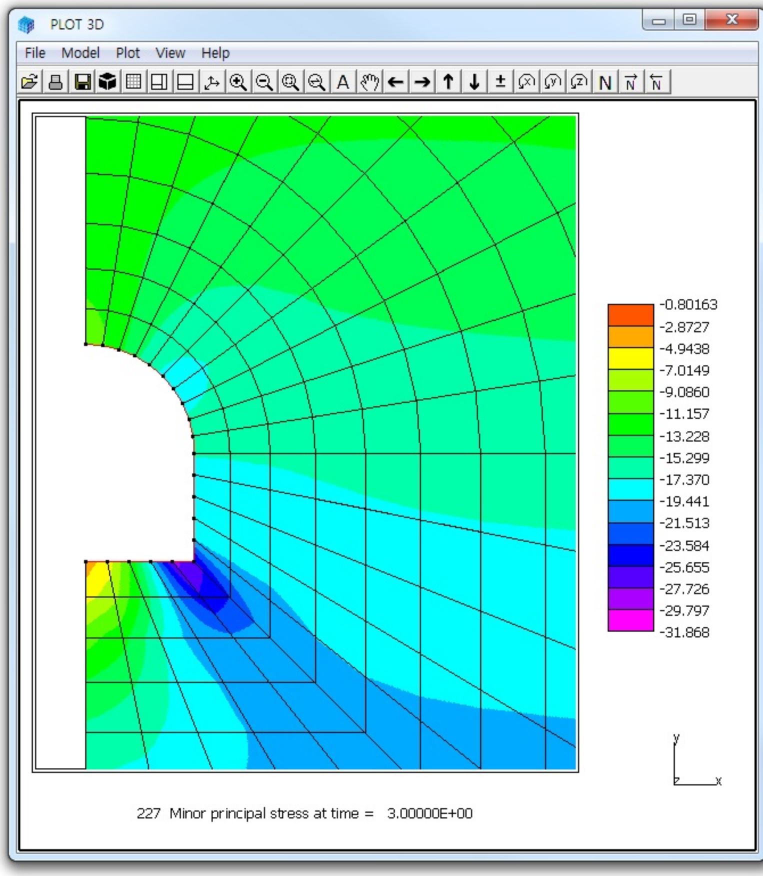


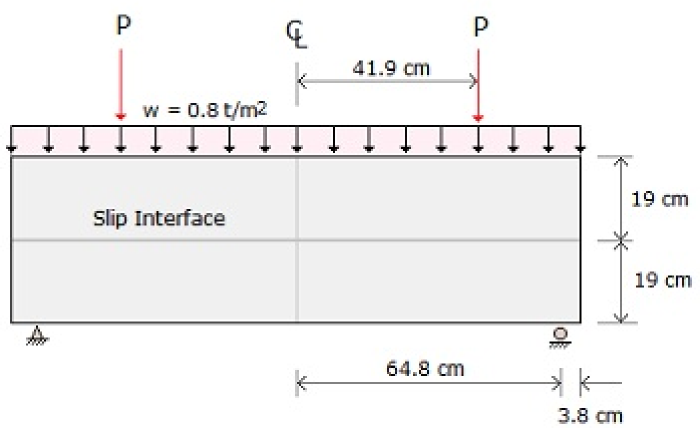
Figure 4.23 Minor principal stress distribution

4.6 Laminated Beam with Slip Interface

The problem is to check the joint element and the nonlinear joint model described in Section 3.2 in theory. Figure 4.24 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 4.25 and 26. 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-S2 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 4.27, the midspan deflections by SMAP-S2 are compared to the closed-form solution derived from beam theory (Agbabian Associates, 1981). Overall, SMAP-S2 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-S2 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 4.24 Laminated beam subjected to uniform and concentrated transverse loads

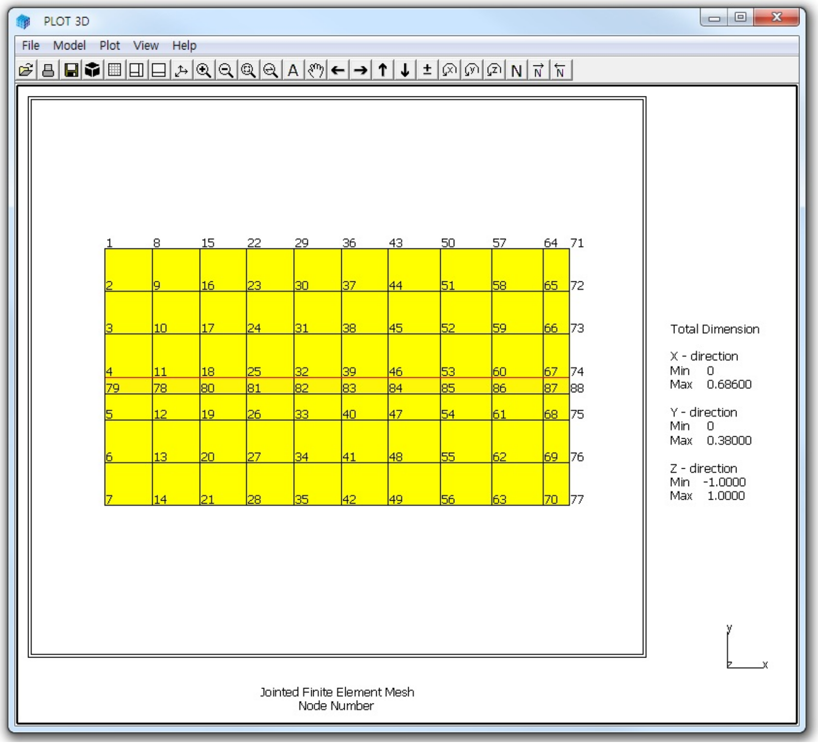


Figure 4.25 Node numbers of laminated beam

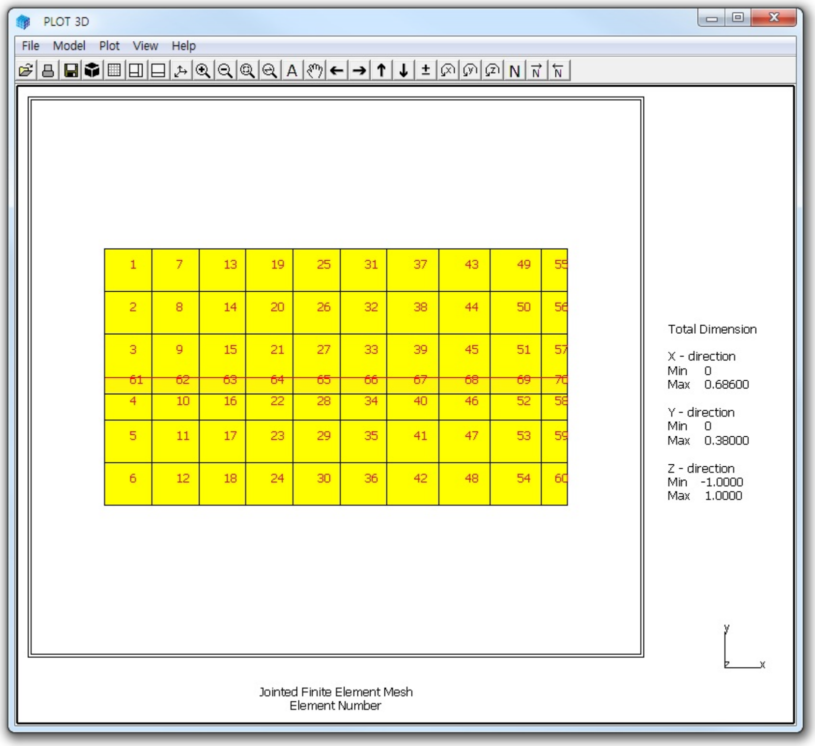


Figure 4.26 Element numbers of laminated beam

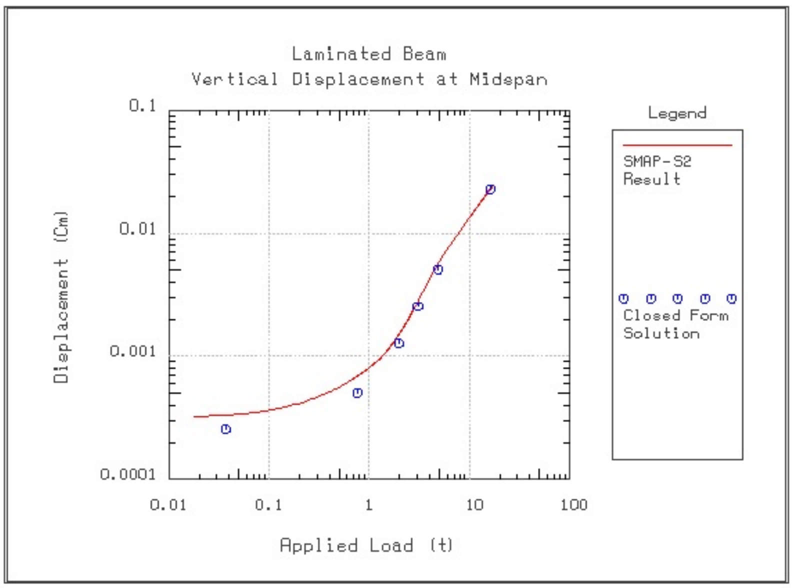


Figure 4.27 Vertical displacement at midspan

4.7 Buried Pipe Analysis

This example problem is to solve the static response of a buried pipe shown schematically in Figure 4.28, which consists of a circular steel pipe, compacted sand backfill, three lifts of embankment fill and sand dune.

Group Mesh Generator is used to generate the finite element meshes. A total of 9 groups are used to model this buried pipe as schematically shown in Figure 4.29. Refer to Group Mesh Example 4 for detailed descriptions to build the mesh.

Table 4.1 illustrates the sequence of construction with descriptions. Table 4.2 lists material properties used for the analysis.

Figure 4.30 shows finite element mesh generated from group mesh. Figure 4.31 shows finite element mesh near buried pipe.

Computed results at final step are presented in the following order:
Figure 4.32 Principal stress vector around pipe
Figure 4.33 Minor principal stress contour around pipe
Figure 4.34 Deformed shape of pipe
Figure 4.35 Bending moment of pipe
Figure 4.36 Axial force of pipe
Figure 4.37 Inner extreme fiber stress of pipe
Figure 4.38 Outer extreme fiber stress of pipe

Computed maximum extreme fiber stresses are 397 Kg/Cm^2 for tension and 509 Kg/Cm^2 for compression. Such maximum stresses are far below the yield strength of steel. Thus the buried pipe structure would be safe.

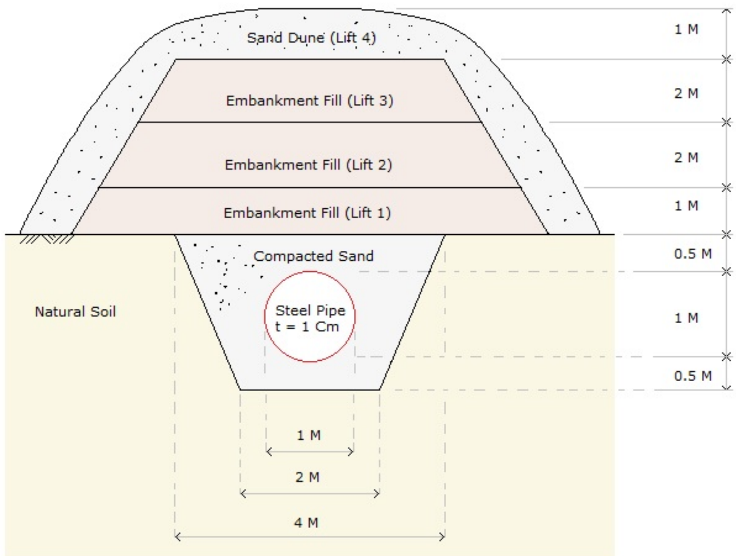


Figure 4.28 Schematic section view (Not Scaled)

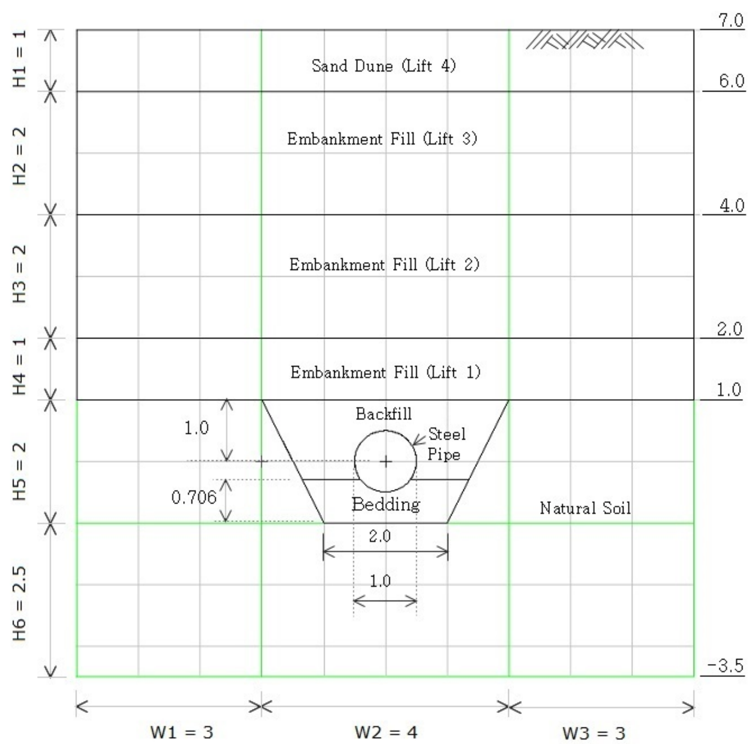


Figure 4.29 Group mesh

Table 4.1 Construction sequence

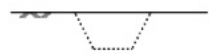
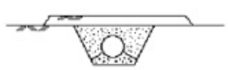
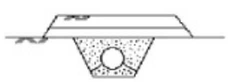
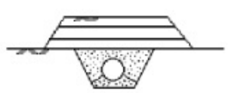
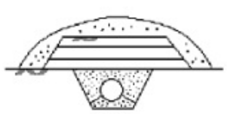
Step	Construction Sequence	Description	Element Activity
1,2		In situ K_o state	Active elements: Natural soil within trench
3		Excavate trench	Deactive elements: Natural soil within trench
4		Place bedding	Active elements: Compacted sand for bedding
5		Place steel pipe Fill the backfill	Active elements: 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 4.2 Material property

Material Number	Material Type	E (Kg/Cm^2)	ν	γ (Kg/Cm^3)
1	Natural Soil	600.	0.3	0.002
2	Compacted Sand	200.	0.3	0.0018
3	Embankment Fill	100.	0.3	0.0016
4	Sand Dune	50.	0.3	0.0015
5	Steel Pipe	$2. \times 10^6$	0.3	0.008

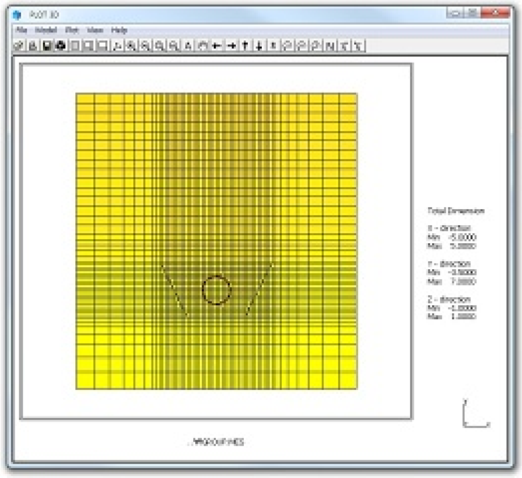


Figure 4.30 Finite element mesh

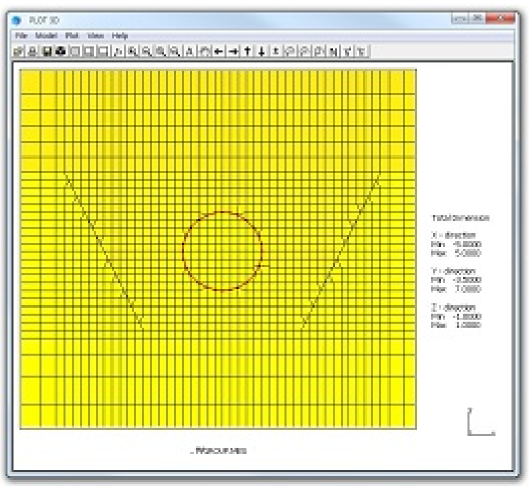


Figure 4.31 Finite element mesh near buried pipe

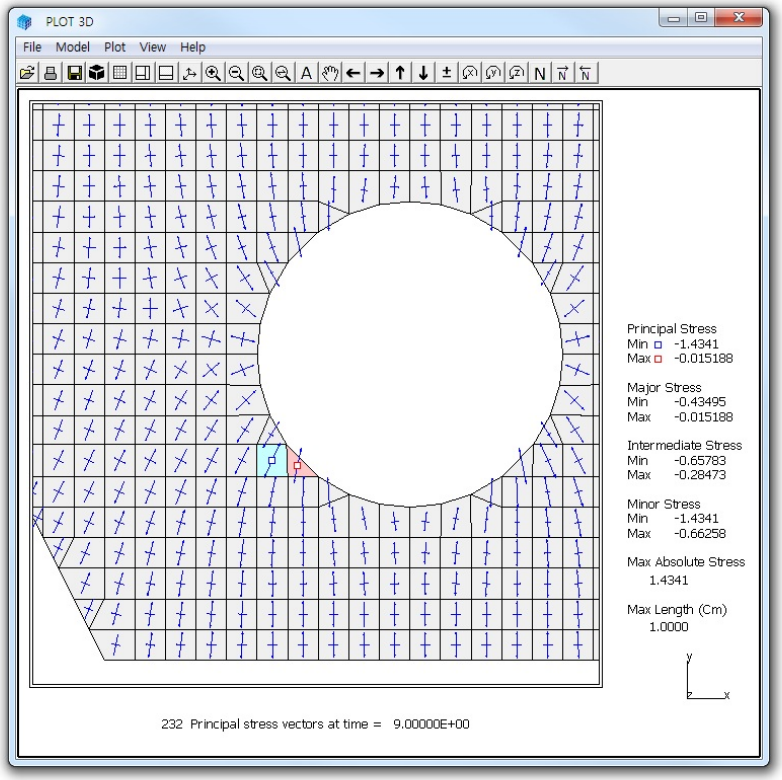


Figure 4.32 Principal stress vector around pipe

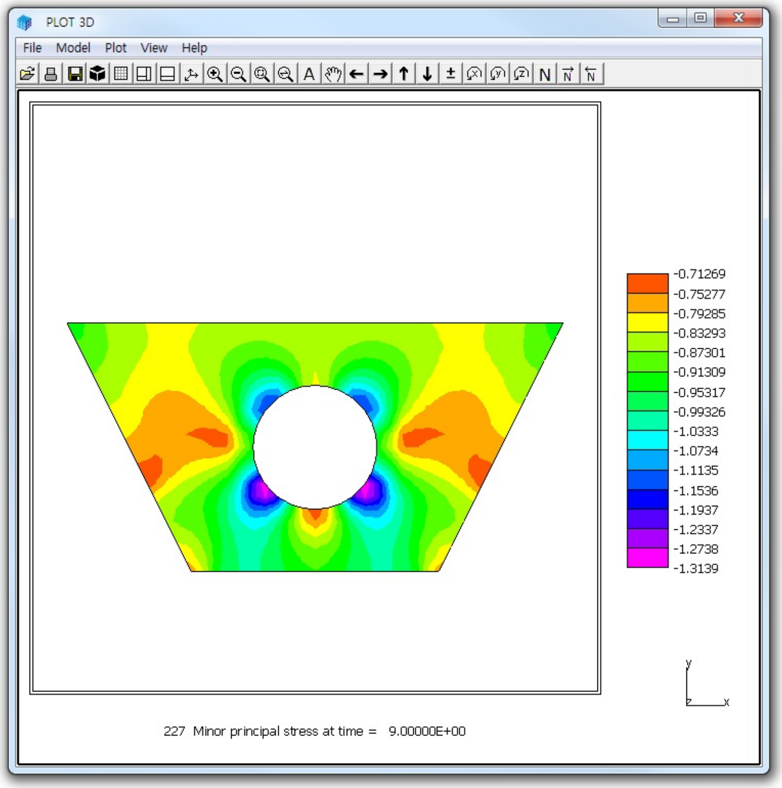


Figure 4.33 Minor principal stress contour around pipe

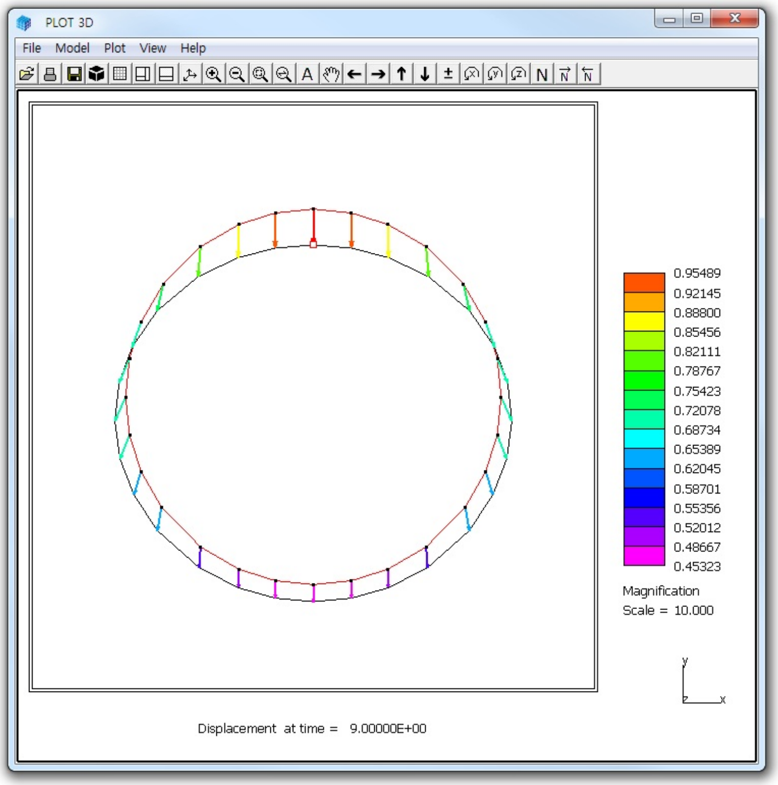


Figure 4.34 Deformed shape of pipe

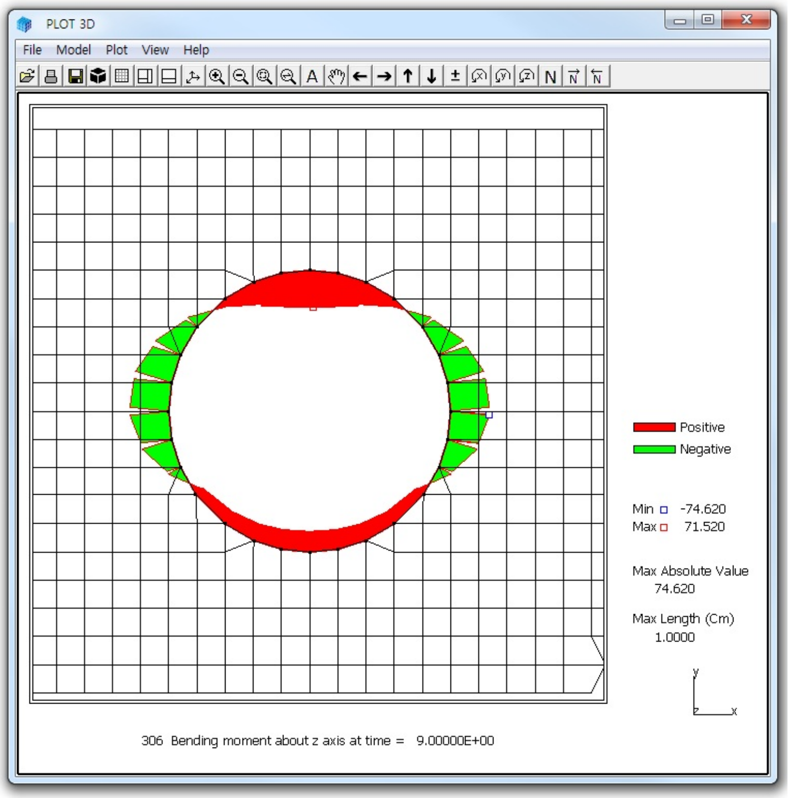


Figure 4.35 Bending moment of pipe

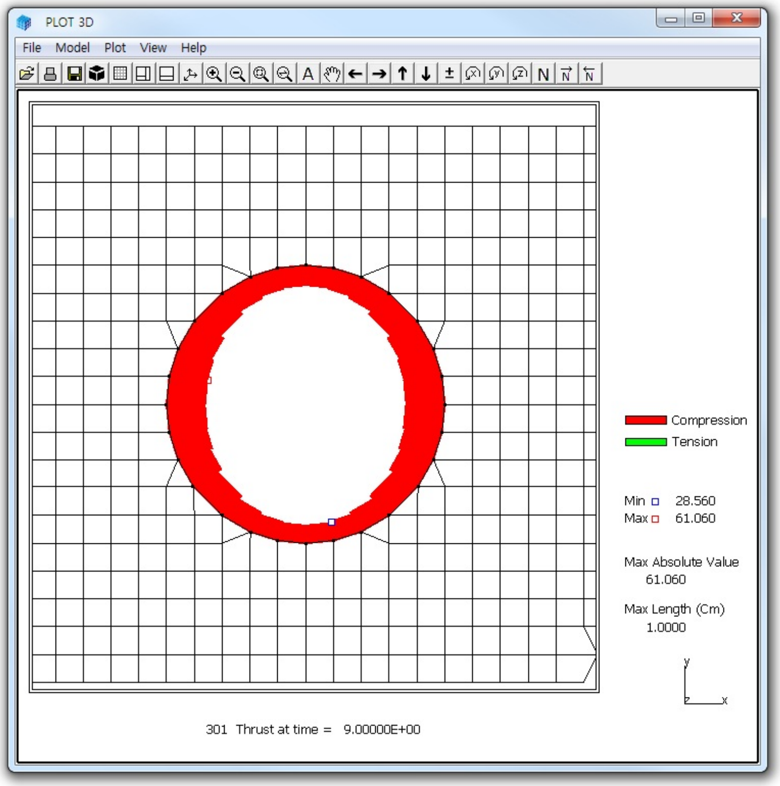


Figure 4.36 Axial force of pipe

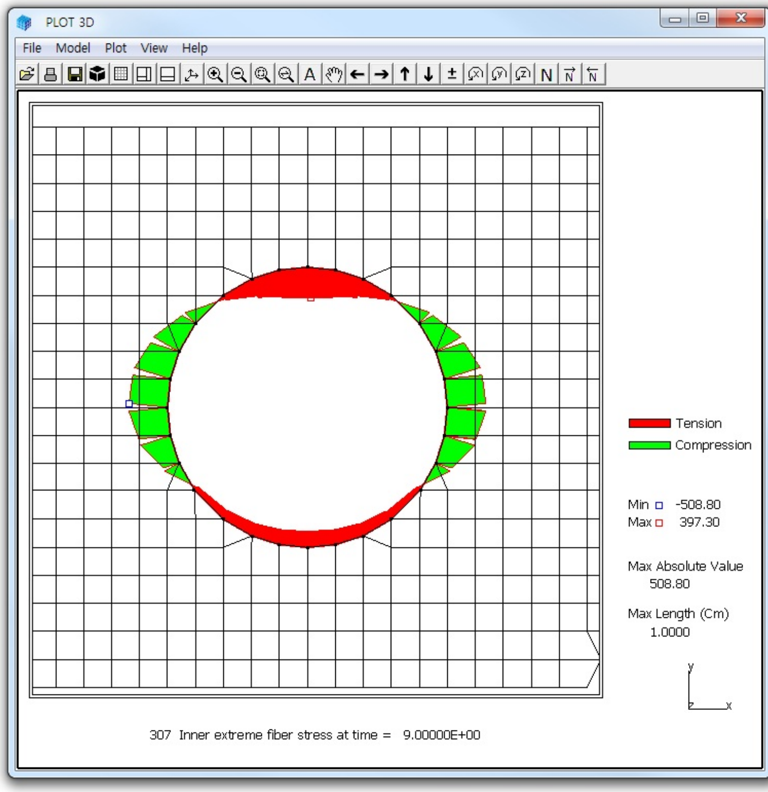


Figure 4.37 Inner extreme fiber stress of pipe

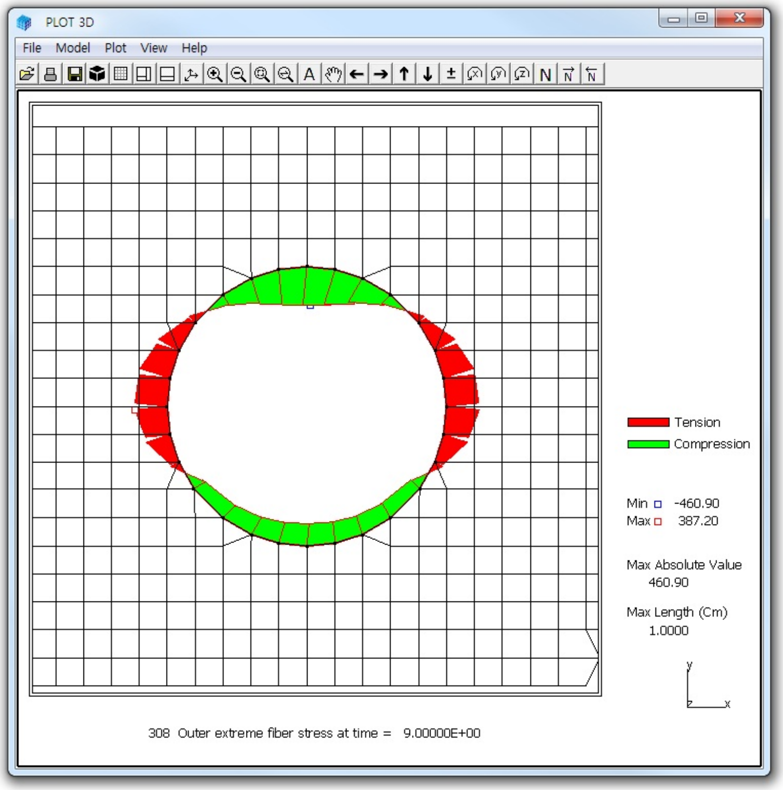


Figure 4.38 Outer extreme fiber stress of pipe

4.8 Heated Beam Analysis

A Simply supported plain concrete beam, shown schematically in Figure 4.39, 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.

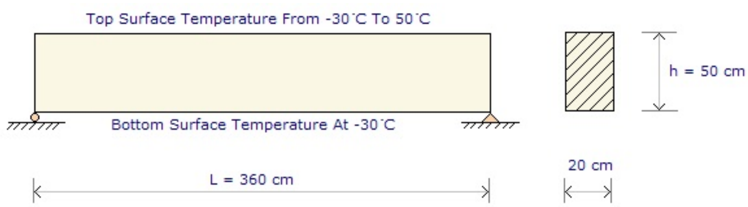


Figure 4.39 Heated beam subjected to temperature difference

By symmetry, only right half of the beam is modeled using a total of 22 beam elements as shown in Figure 4.40.

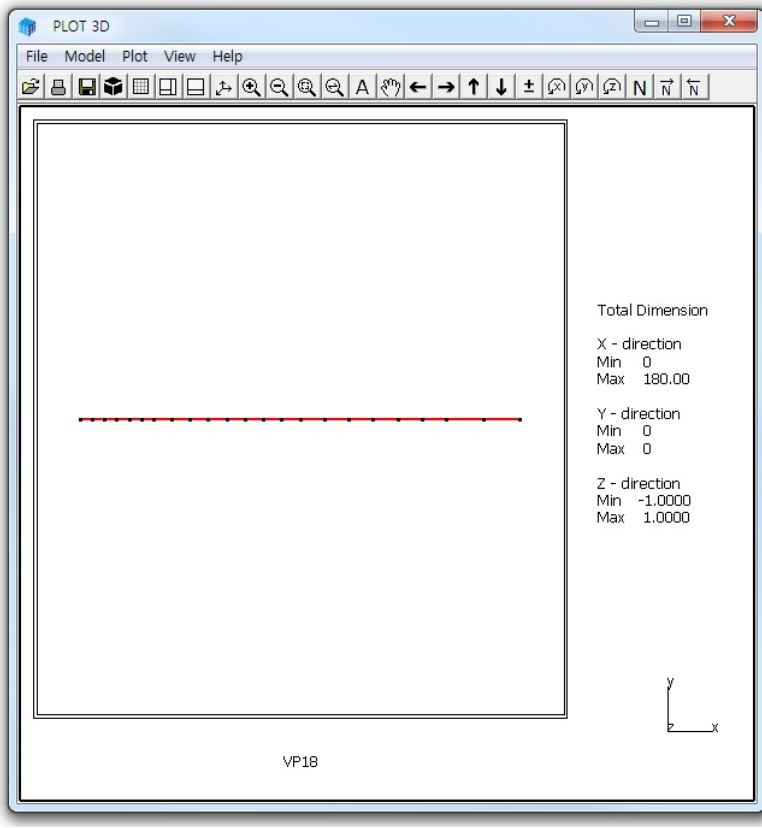


Figure 4.40 Finite element mesh

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 4.41 shows beam deflections. SMAP-S2 result gives excellent results for vertical displacement at center of the beam.

Theoretical solution = 0.8294 cm

SMAP-S2 solution = 0.82945 cm

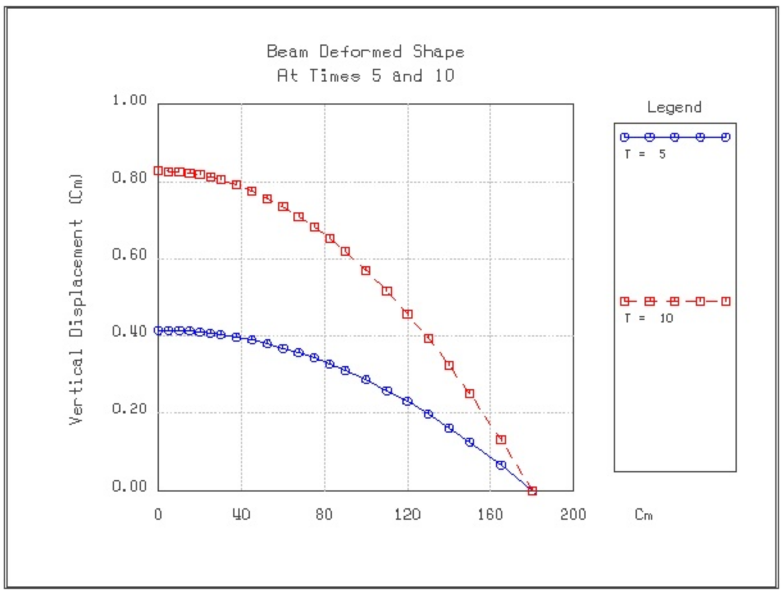


Figure 4.41 Beam deformed shapes

.

4.9 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 4.42.

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-S2 calculations are performed using the geometrical and material parameters listed in Figure 4.42. The frame is modeled by 10 beam elements as shown in Figure 4.43.

Figures 4.44 and 4.45 show beam deformed shape and bending moment diagram, respectively.

SMAP-S2 results show good agreement with the exact solutions.

Maximum deflection at the center (δ)

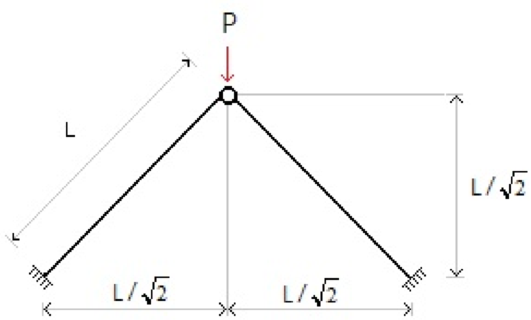
Exact solution = 0.01768 cm

SMAP-S2 (Beam) = 0.01767 cm

Maximum moment at fixed end (M_{max})

Exact solution = 0.1000 t-m

SMAP-S2 (Beam) = 0.1000 t-m



P	= 100 t	L	= 7.071 m
E	= 20×10^6 t/m^2	v	= 0.0
A	= 0.2 m^2	I	= 0.000667 m^4

Figure 4.42 Frames with hinge connection

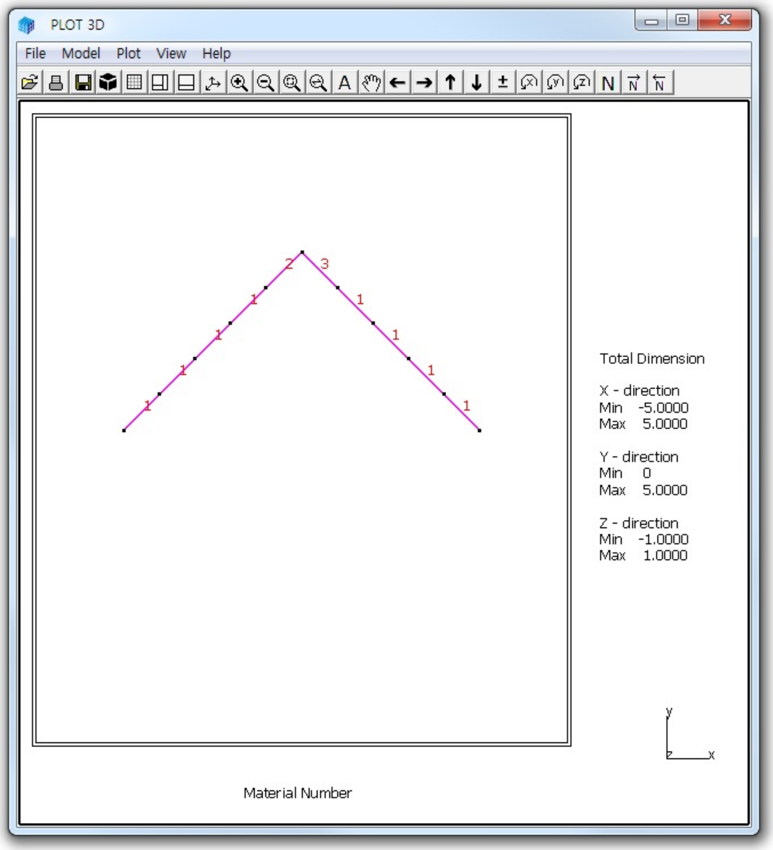


Figure 4.43 Beam element with material number

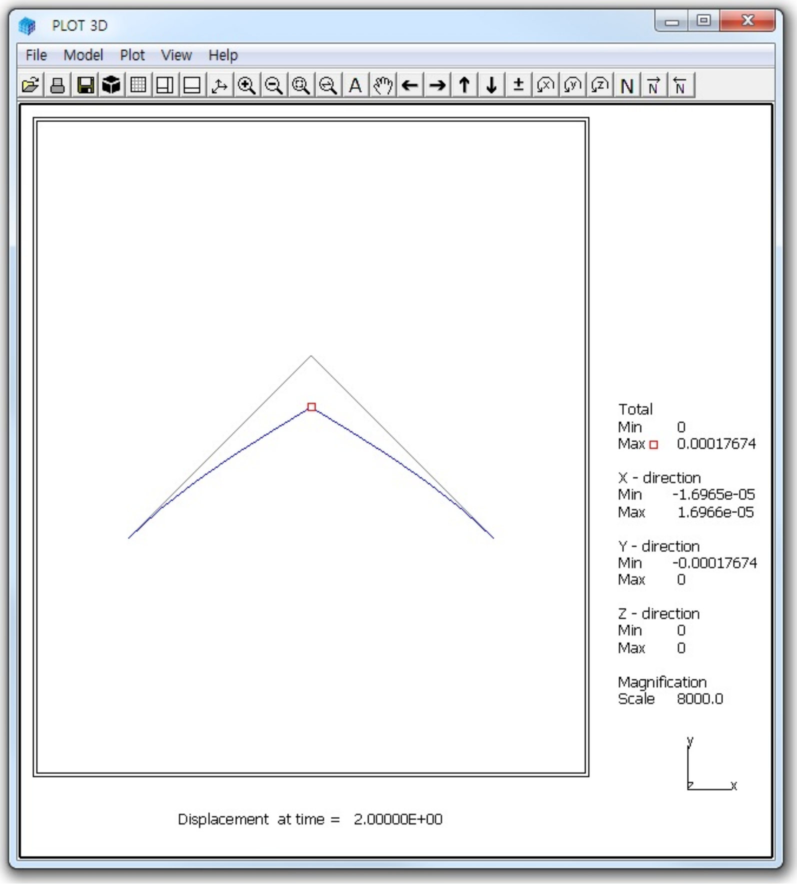


Figure 4.44 Beam deformed shape

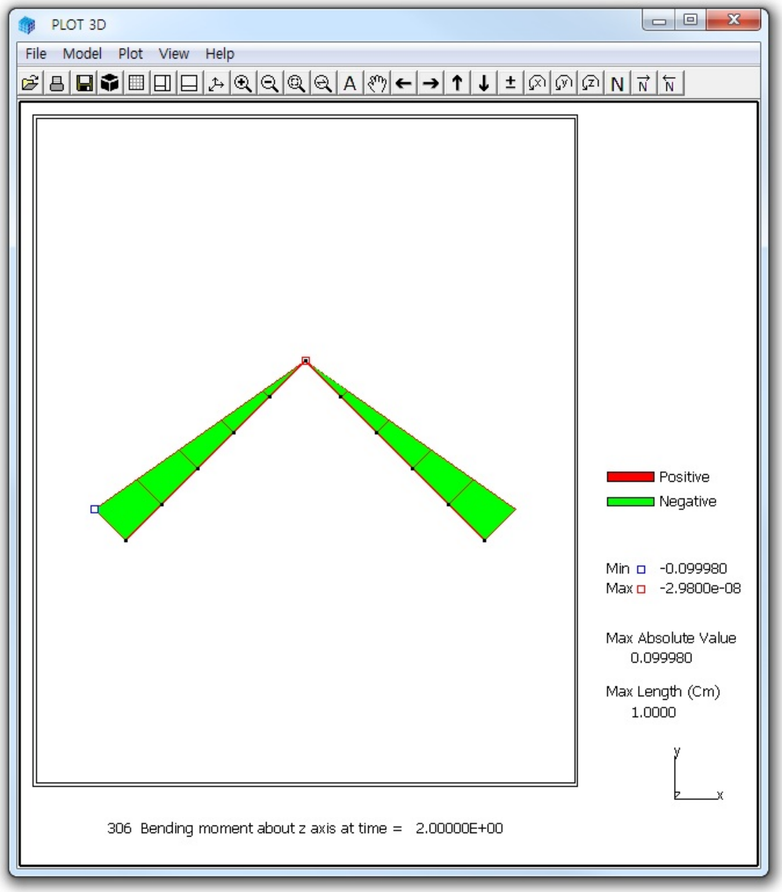


Figure 4.45 Beam bending moment diagram

4.10 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 4.46 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 4.47. It should be noted that the end points of embedded truss elements do not belong to the corner nodes of continuum elements.

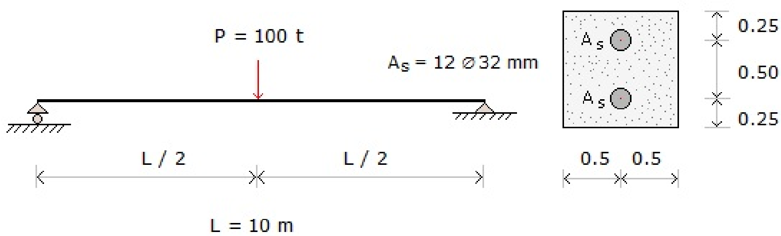
The computed center deflections are compared with the exact beam solution as shown in Table 4.3. SMAP-S2 results approach to the upper bound beam solution at lower cohesion and the lower bound beam solution at higher cohesion. At the intermediate cohesion, however, the computed deflection is in between upper and lower bound beam solutions, indicating some resistance from slip strength.

Figures 4.48 and 4.49 show the deformed shape and the axial stress distribution, respectively, from SMAP-S2 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 4.46 Embedded rebars with slip

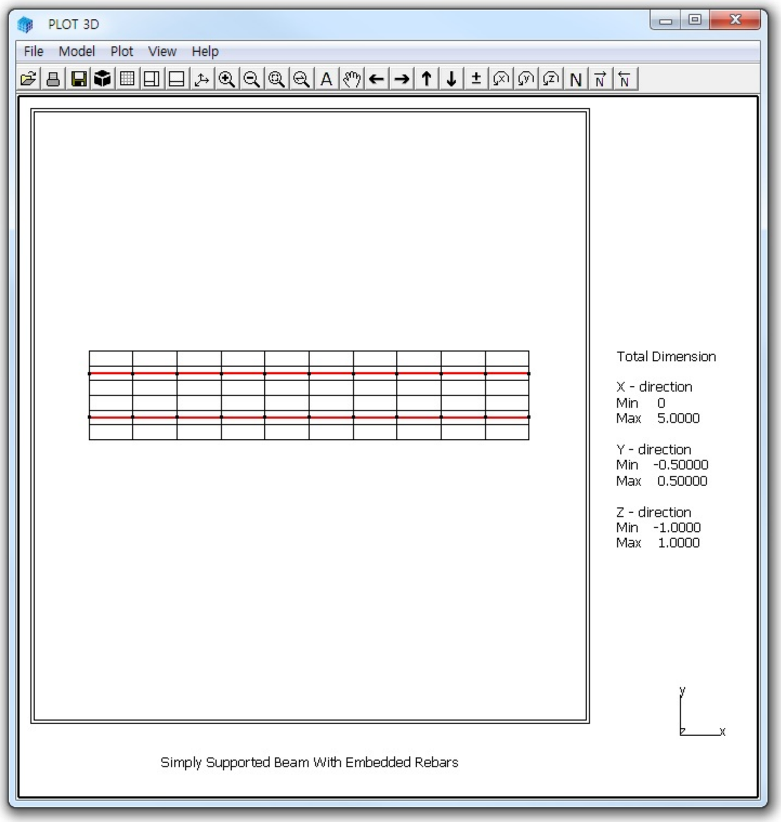


Figure 4.47 Finite element mesh

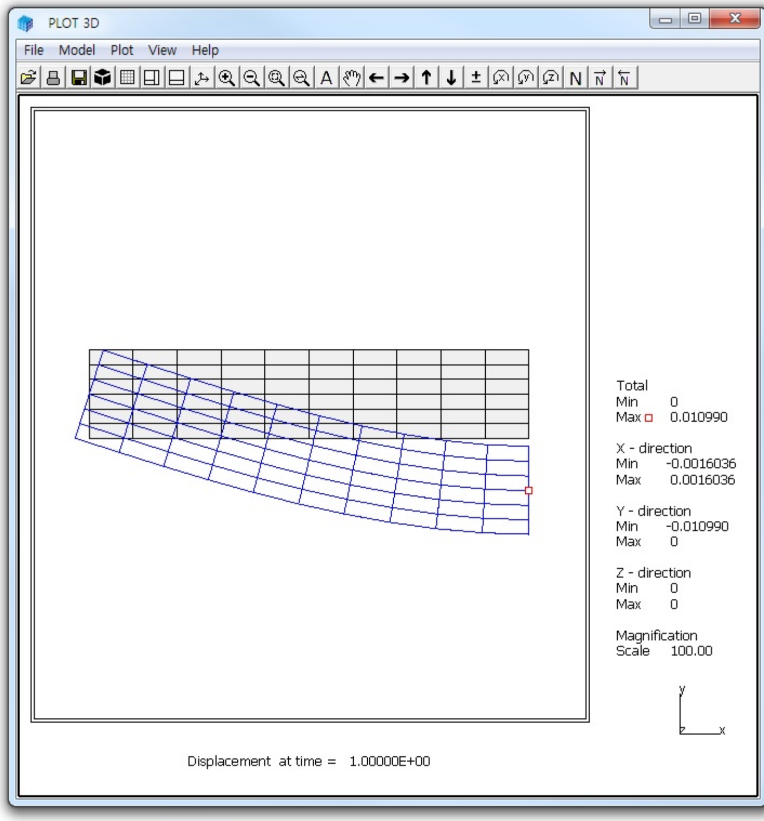


Figure 4.48 Deformed shape

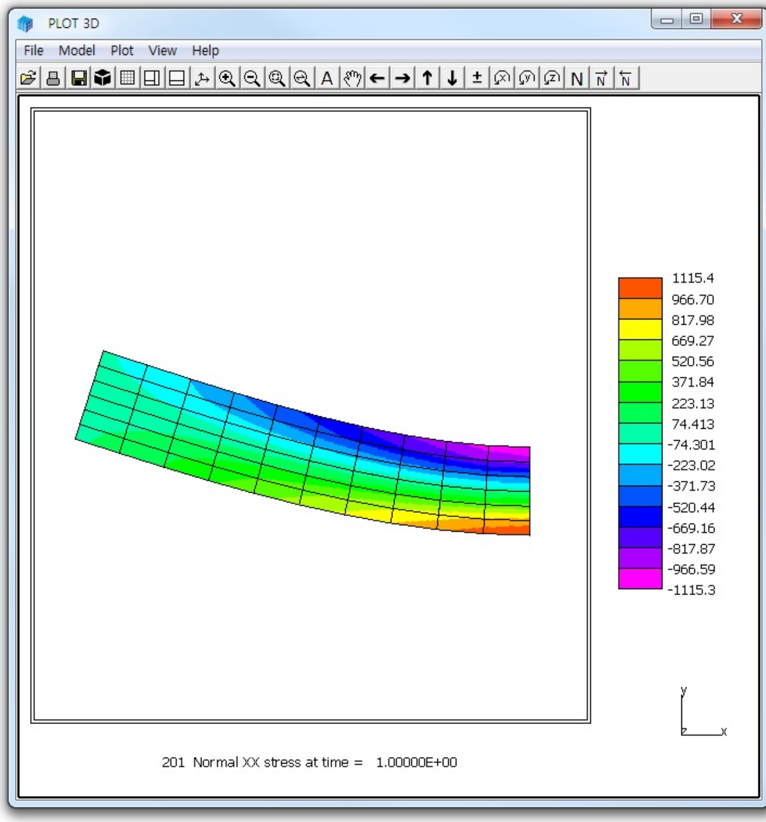


Figure 4.49 Axial stress distribution

4.11 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 4.50.

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 X component of unit gravity load (FRX).

Figure 4.51 shows the finite element mesh used for the analysis. Figures 4.52 and 4.53 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 results are very close to SMAP-2D results.

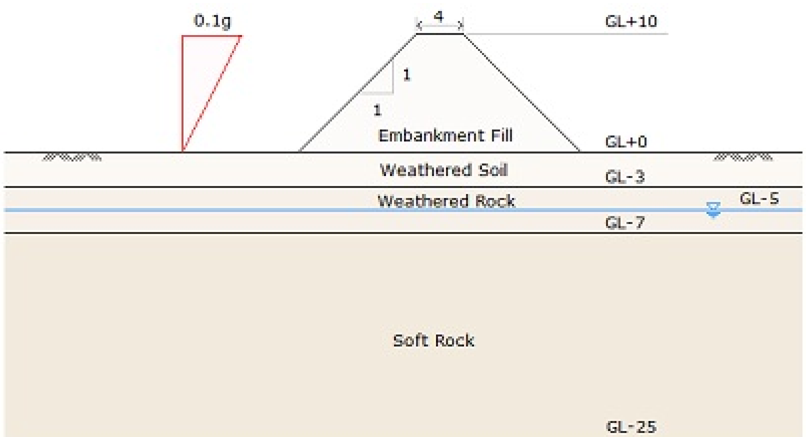


Figure 4.50 Schematic section view of embankment

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	2.0×10^3	0.33	30	3	20
Weathered Rock	1.90	0.43	5.0×10^3	0.30	35	30	30
Soft Rock	2.40	0.33	2.0×10^4	0.25	40	70	40
Embankment Fill	2.00	0.50	3.0×10^3	0.33	30	3	20

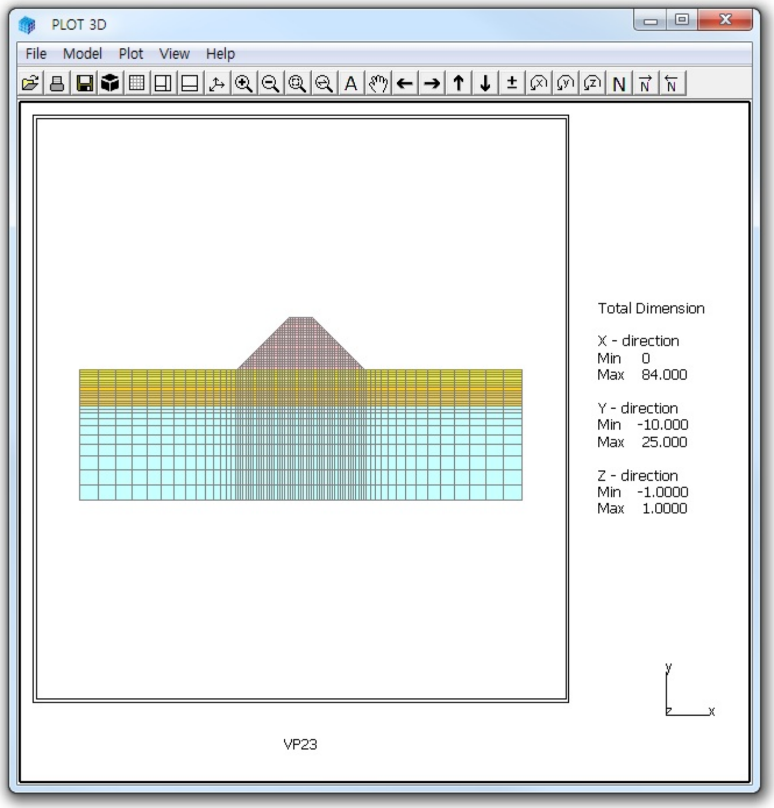


Figure 4.51 Finite element mesh

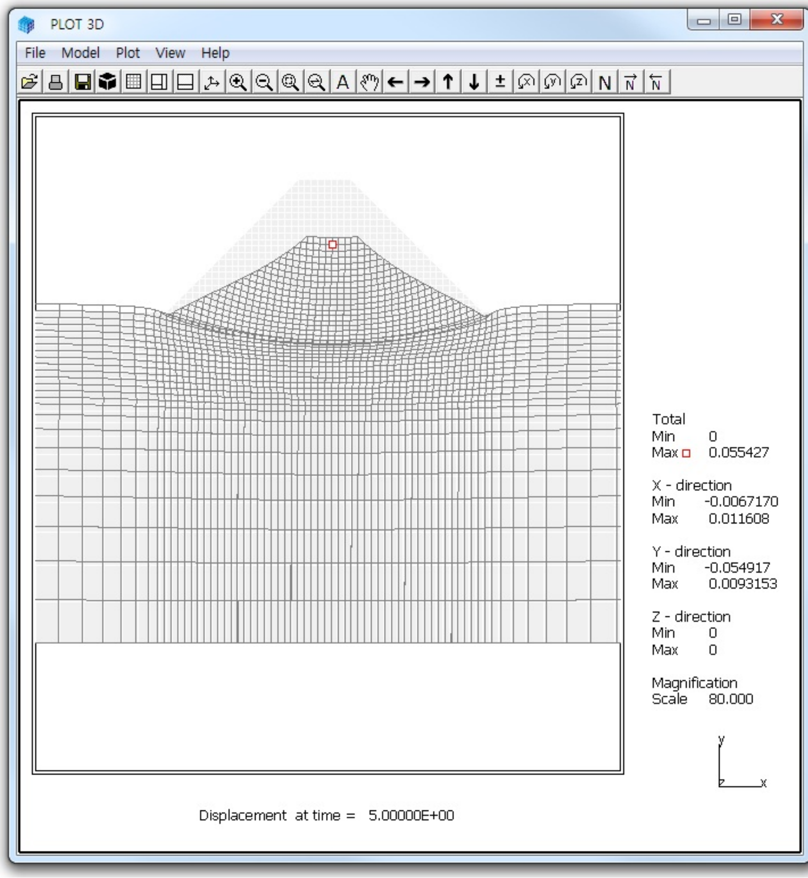


Figure 4.52 Deformed shape

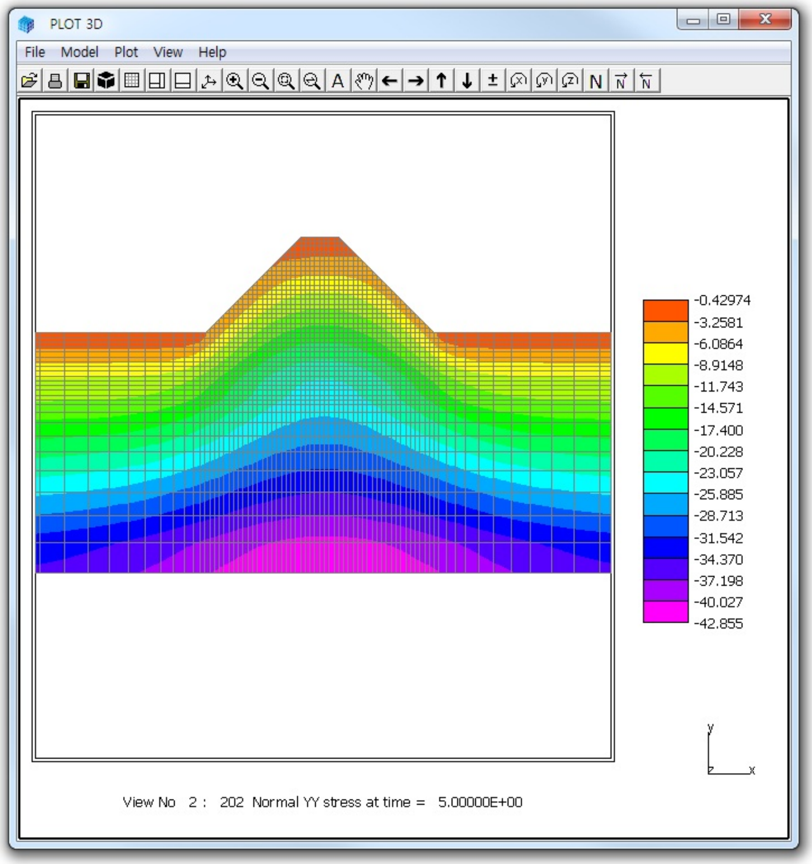


Figure 4.53 Vertical stress distribution

4.12 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 4.54 and 4.55. 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 4.56 shows all groups used for mesh generation.

Following graphical results are obtained from PLOT-3D:

Figure 4.57	Vertical stresses on deformed mesh at Step 10
Figure 4.58	Bending moments on deformed mesh at Step 10
Figure 4.59	Bending moments in RC box frame at Step 3
Figure 4.60	Bending moments in RC box frame at Step 10
Figure 4.61	Inner extreme fiber stress in concrete at Step 10
Figure 4.62	Outer extreme fiber stress in concrete at Step 10
Figure 4.63	Inner reinforcing bar stress at Step 10
Figure 4.64	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 320 t/m2 is much lower than the allowable compressive stress and the maximum tensile reinforcing bar stress of 1328 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.

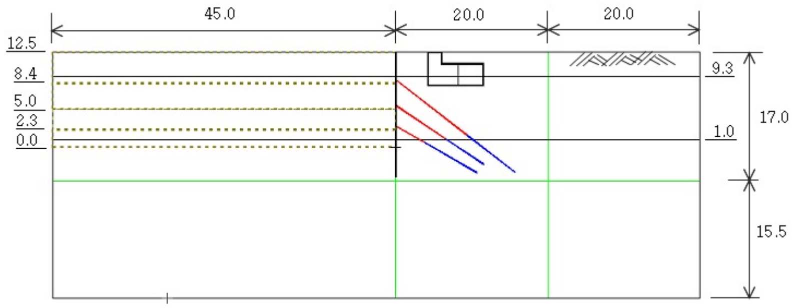


Figure 4.54 Schematic section view

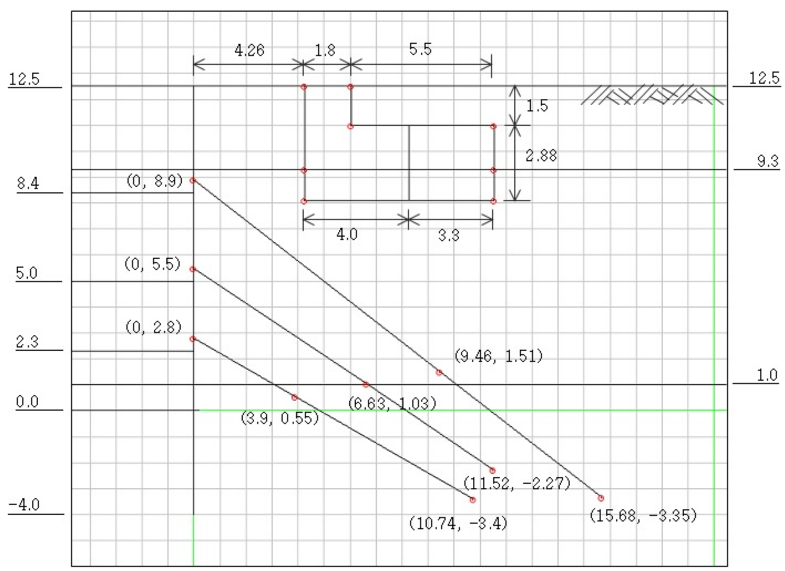


Figure 4.55 Cross section near box structure

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				

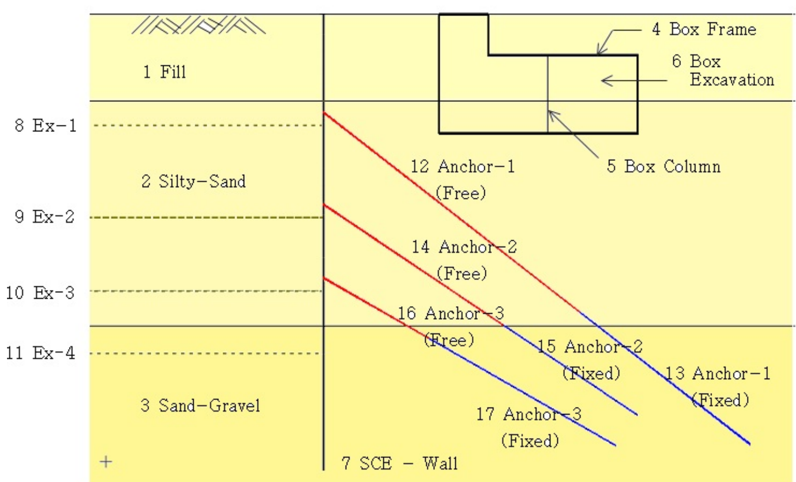


Figure 4.56 All groups used for mesh generation

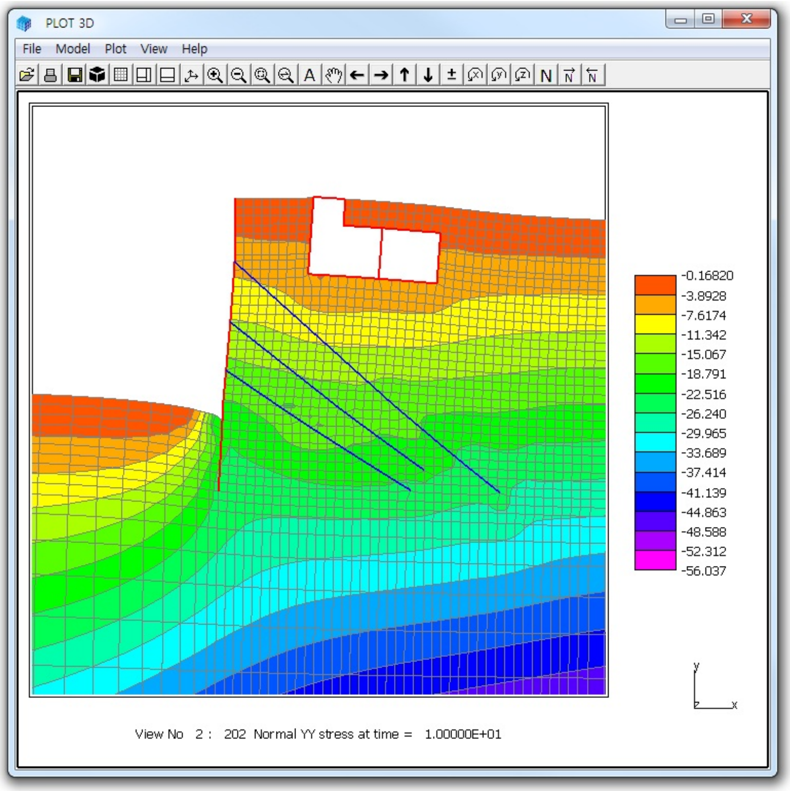


Figure 4.57 Vertical stresses on deformed mesh at Step 10

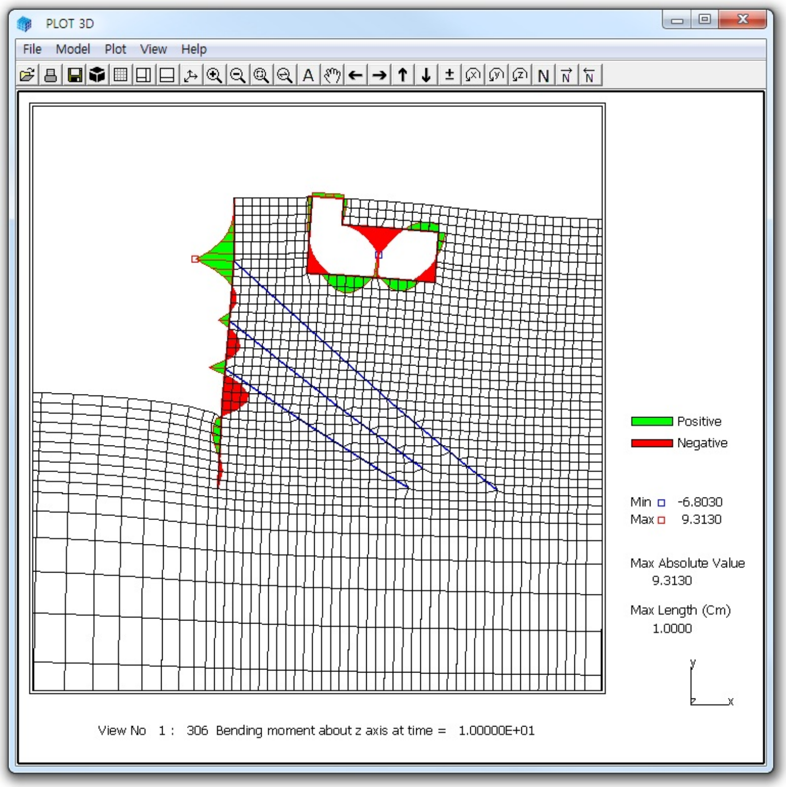


Figure 4.58 Bending moments on deformed mesh at Step 10

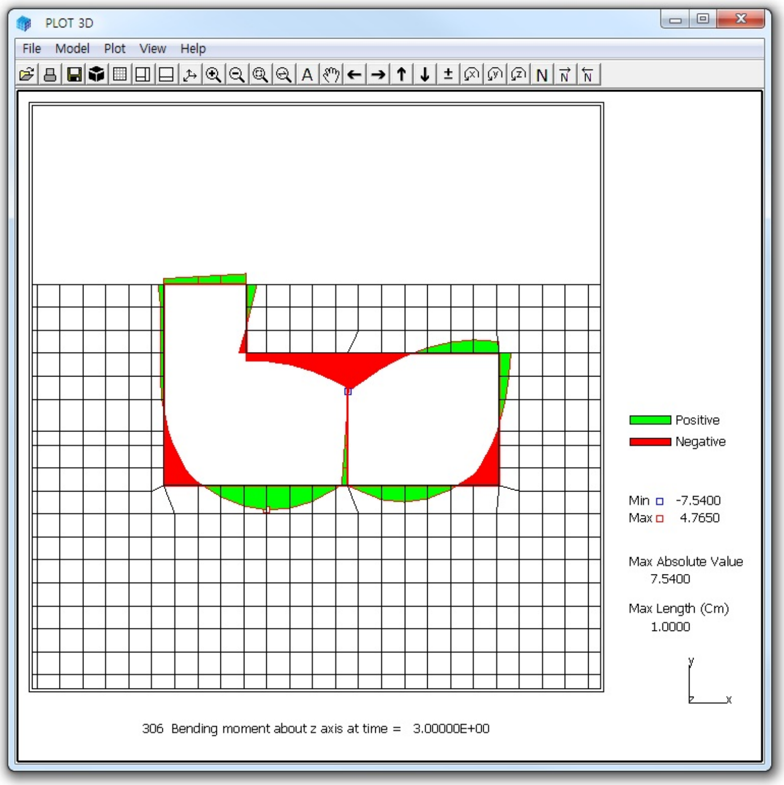


Figure 4.59 Bending moments in RC box frame at Step 3

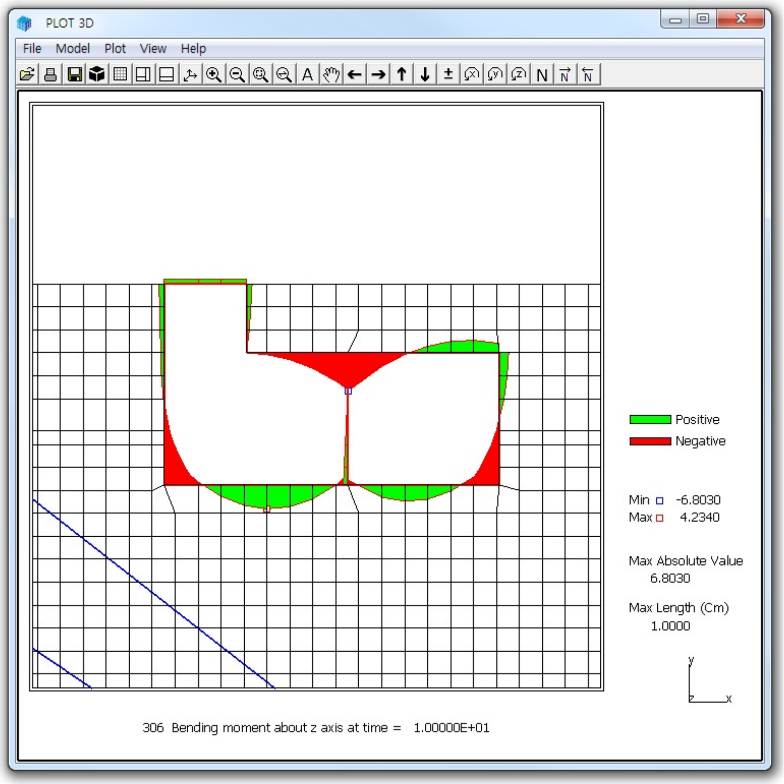


Figure 4.60 Bending moments in RC box frame at Step 10

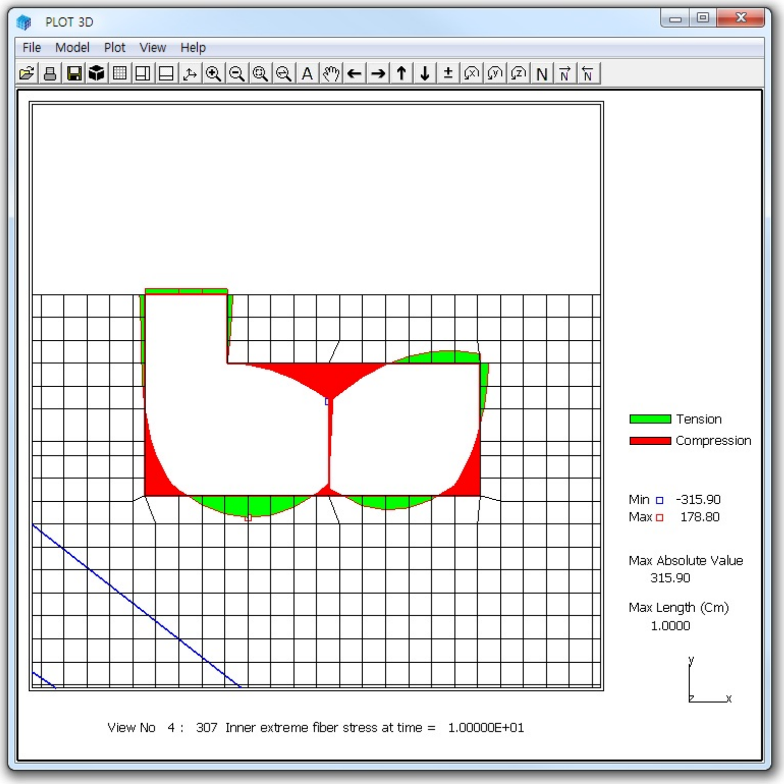


Figure 4.61 Inner extreme fiber stress in concrete at Step 10

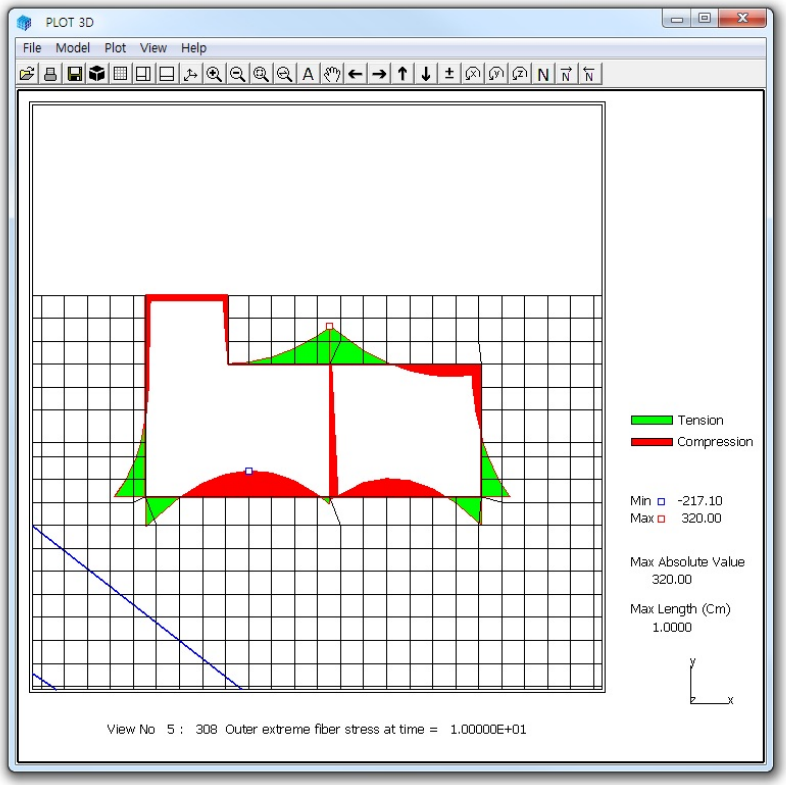


Figure 4.62 Outer extreme fiber stress in concrete at Step 10

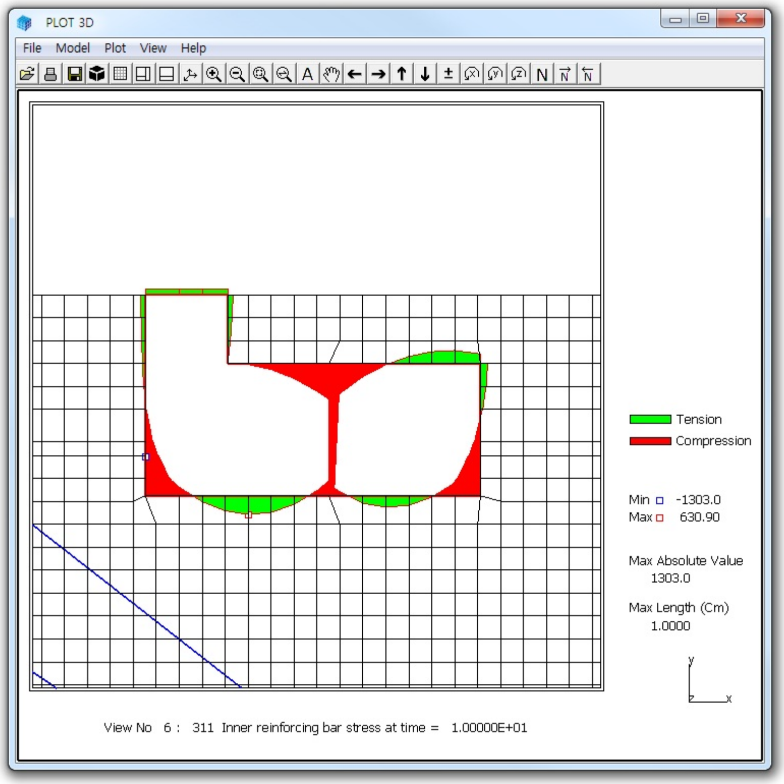


Figure 4.63 Inner reinforcing bar stress at Step 10

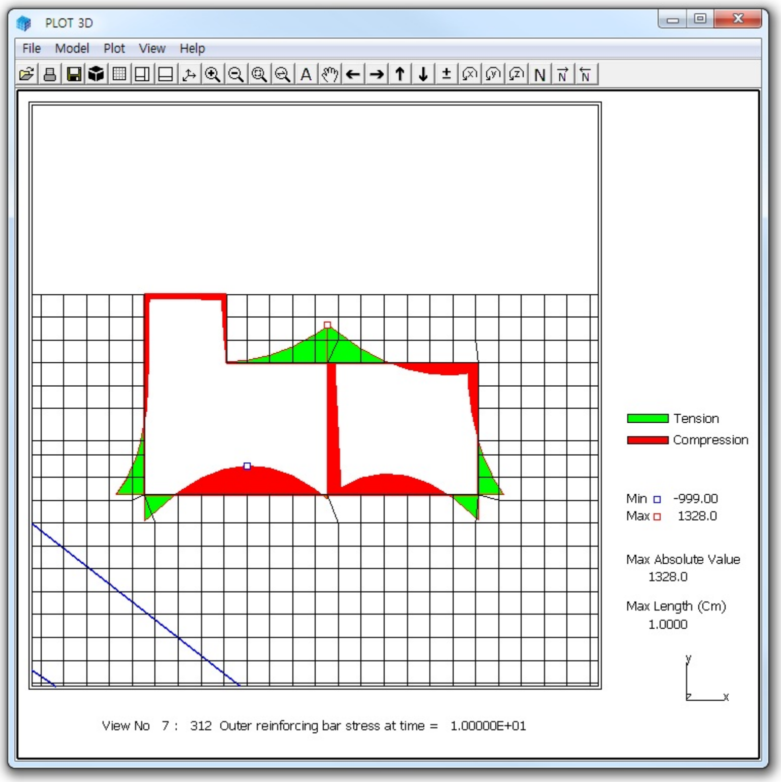


Figure 4.64 Outer reinforcing bar stress at Step 10

4.13 Spring Analysis

This example problem is to show how to model springs using special features in beam element in Card 6.5 of SMAP-S2 User's Manual.

The example is composed of two truss members connected by horizontal and vertical springs as shown in Figure 4.65. The structure is subjected to external horizontal and vertical nodal forces.

Figure 4.66 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 4.65, 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)

SMAPS2 results show exact as shown in Figures 4.67, and 4.68 for displacements and truss axial forces, respectively.

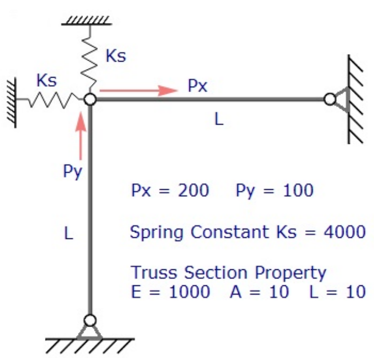


Figure 4.65 Truss members connected by springs

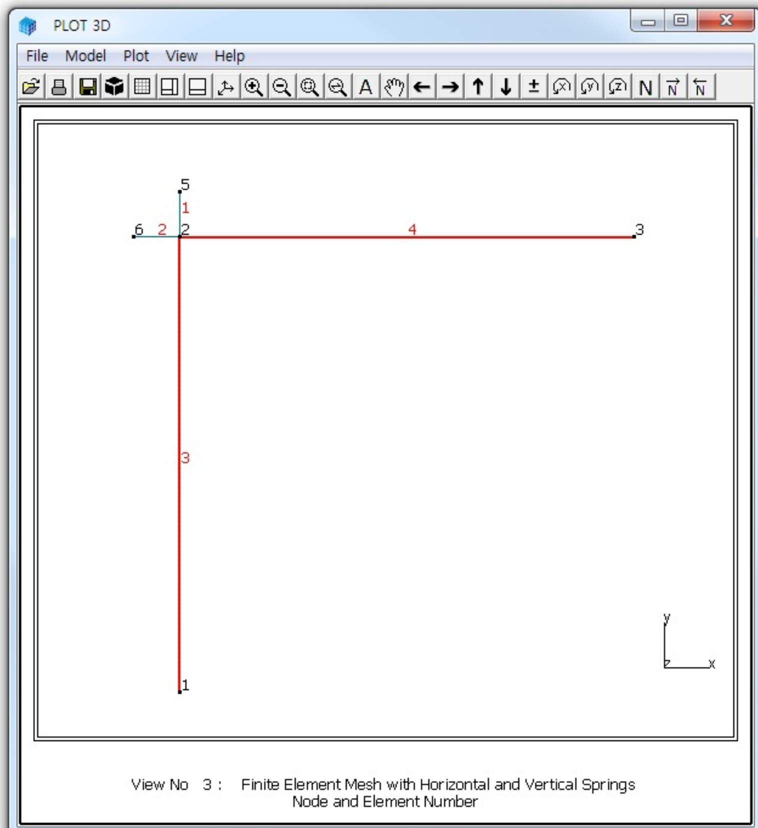


Figure 4.66 Finite element mesh

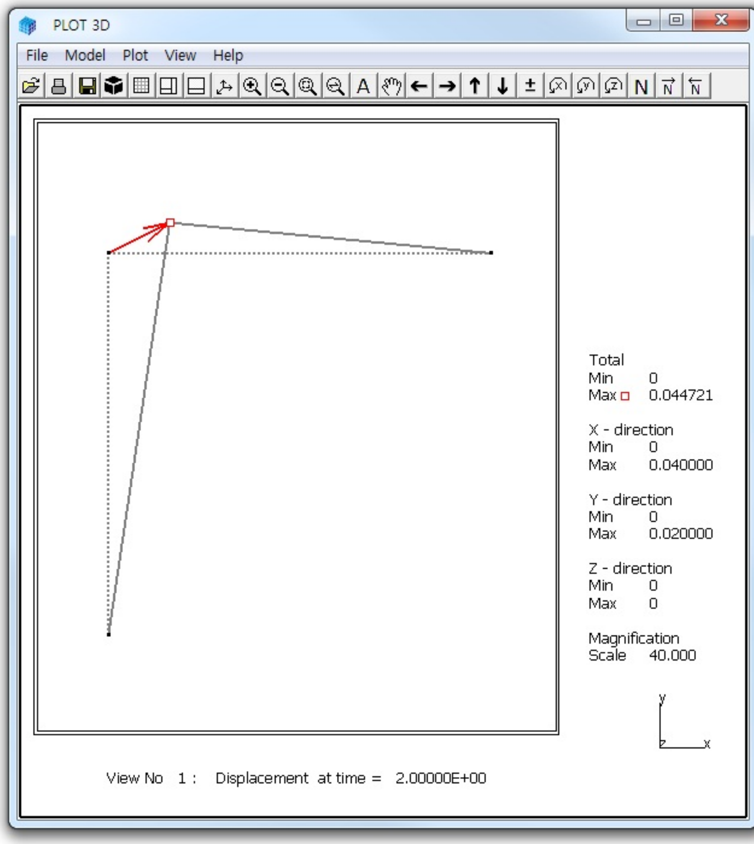


Figure 4.67 Deformed shape

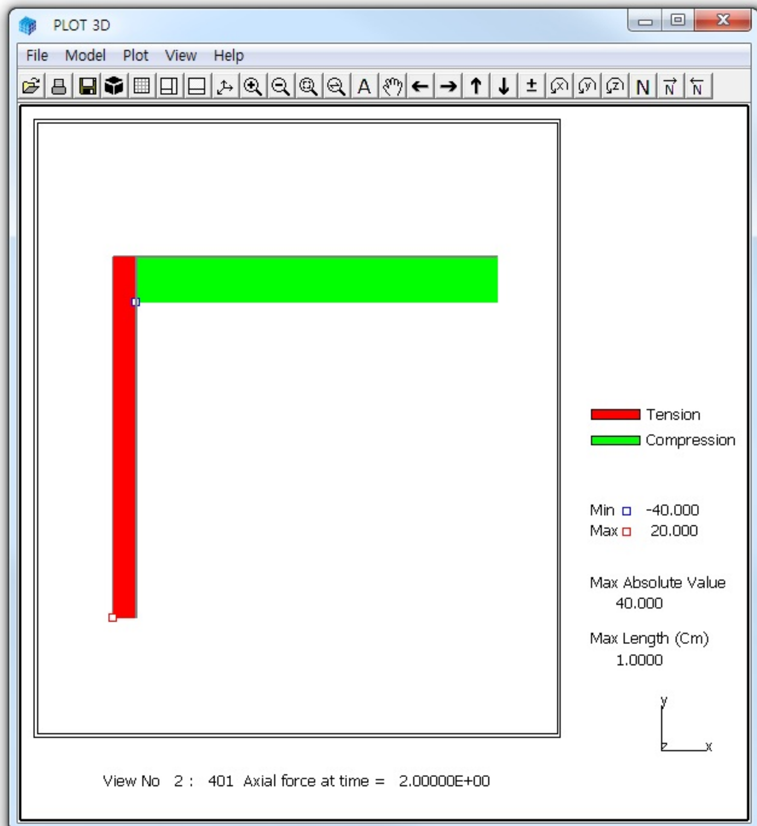


Figure 4.68 Truss axial force

4.14 Nonlinear Truss Analysis

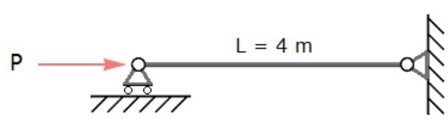
Truss elements in SMAP can consider nonlinear behavior such as yielding and post buckling as schematically illustrated in Figure 4.70. Following examples are to show how to use such material parameters in truss element in Card 7.4.3 of SMAP-S2 User's Manual.

Figure 4.69 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 4.71)
2. Compression and Tension Yielding (Figure 4.72)
3. Tension Yielding for No Compression Member (Figure 4.73)
4. Compression Yielding for No Tension Member (Figure 4.74)
5. Buckling for No Tension Member (Figure 4.75)
6. Initial Stress (See Case 6 at the end of example)

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



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

I_x = 2400 Cm^4 I_y = 887 Cm^4

E = 21000 kN/Cm^2 A = 91.73 Cm^2

σ_y = 32.5 kN/Cm^2 σ_{cr} = 12.52 kN/Cm^2

Figure 4.69 Truss member subjected to axial force

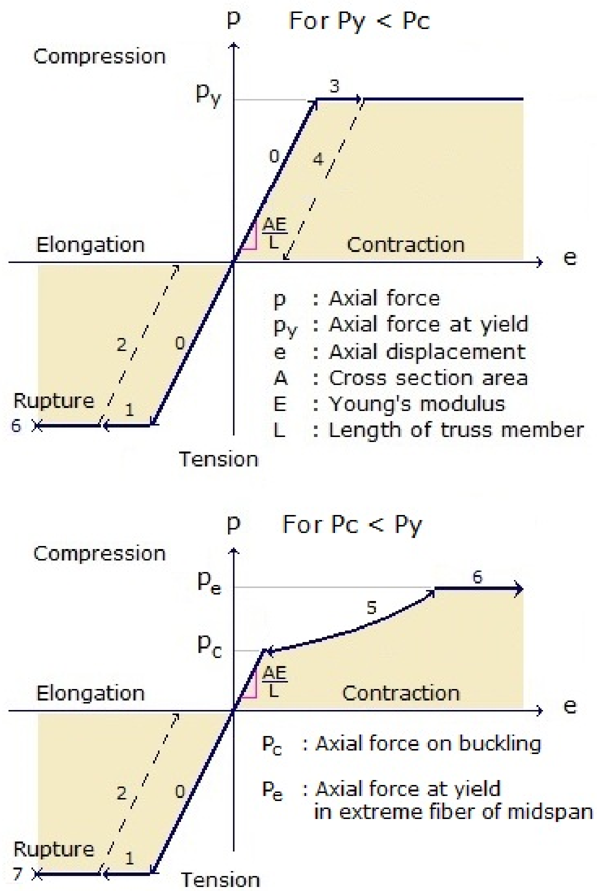


Figure 4.70 Nonlinear force displacement curve

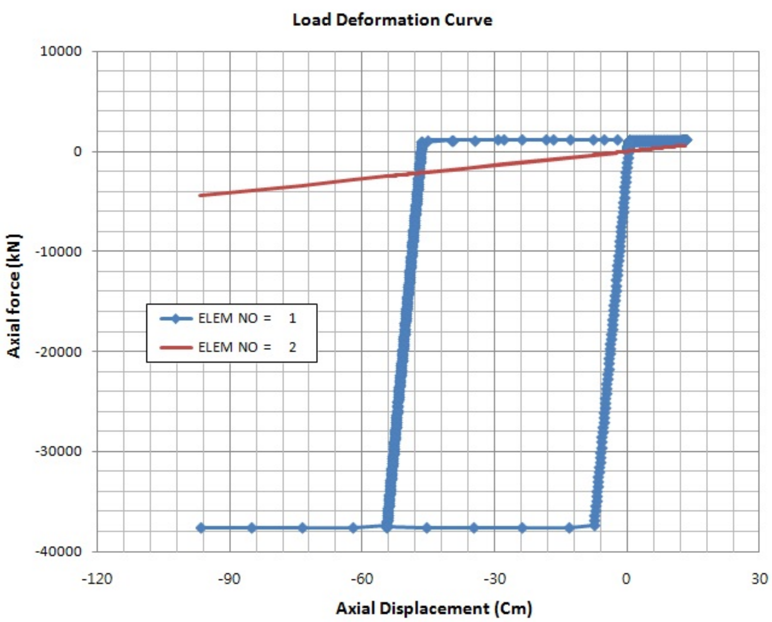


Figure 4.71 Buckling and Tension Yielding (VP14-1)

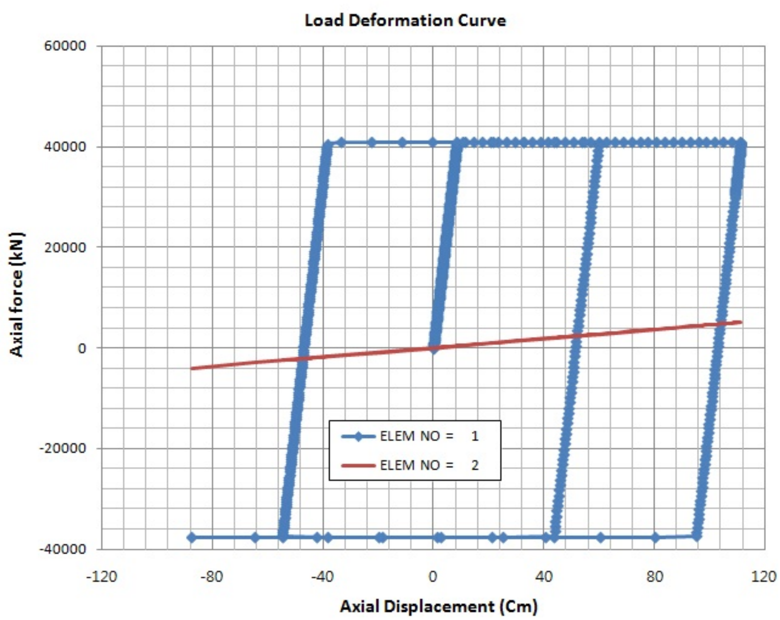


Figure 4.72 Compression and Tension Yielding (VP14-2)

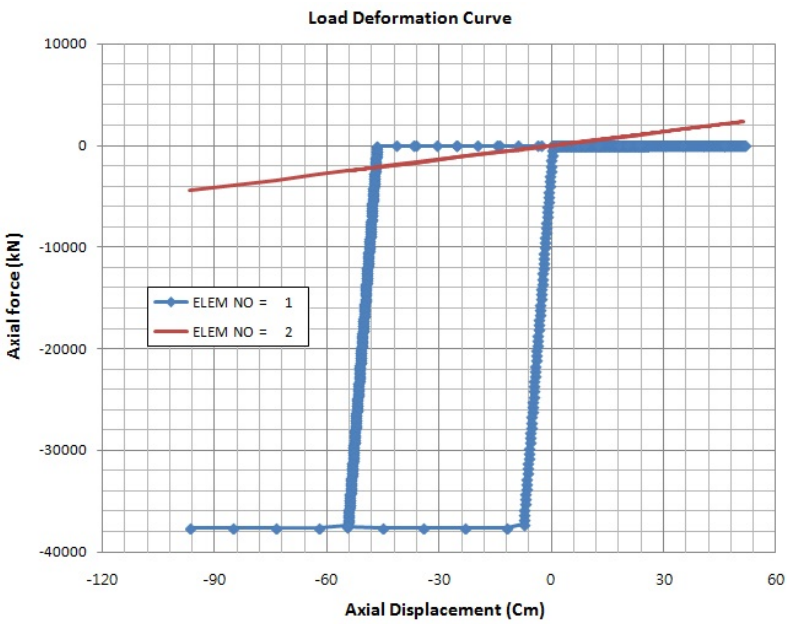


Fig 4.73 Tension Yielding for No Compression Member (VP14-3)

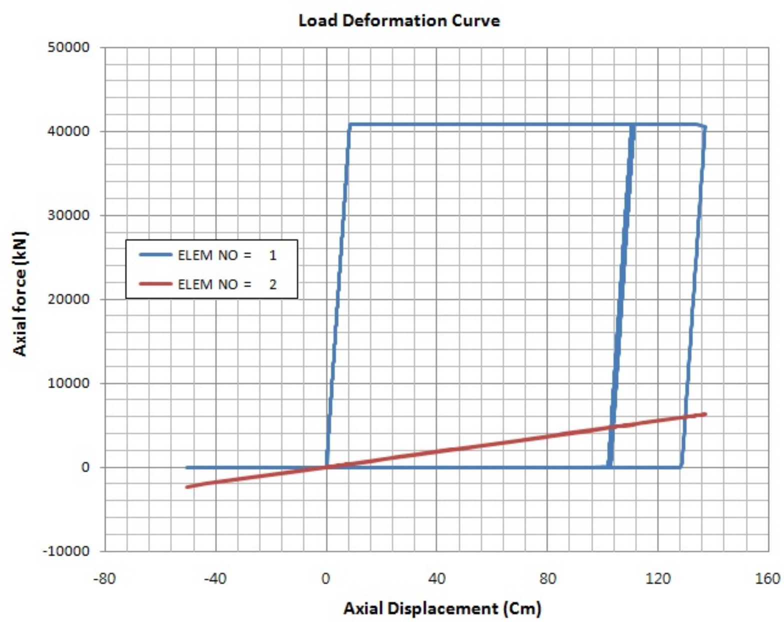


Fig 4.74 Compression Yielding for No Tension Member (VP14-4)

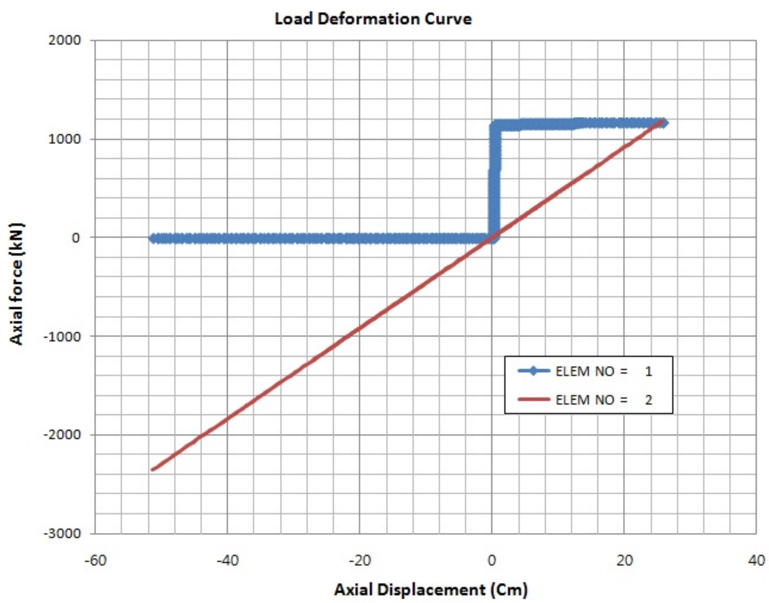


Figure 4.75 Buckling for No Tension Member (VP14-5)

Case 6 Initial Stress

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

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

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

Now, when Members 1 and 2 are connected,

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

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

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

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

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

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

From (3)

$$\varepsilon_1 = -0.000021654$$

And from (2) and (1)

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

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

SMAP results show exact solution once round-off for 4 digits.

4.15 Frames with Rotational Spring Connection

This example is the same as Example problem 9 except that it is connected by rotational spring and subjected to both moment and horizontal force at the connection as shown in Figure 4.76.

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_S2.pdf. It approaches to rigid connection when the rotational spring is rigid and hinge connection when the spring constant is very flexible.

Figures 4.77 to 4.81 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.

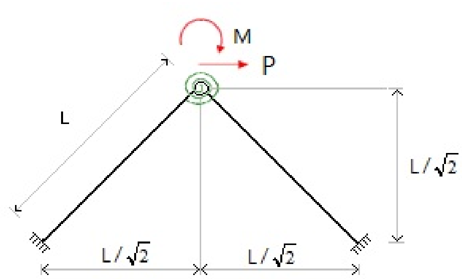


Figure 4.76 Frames with rotational spring connection

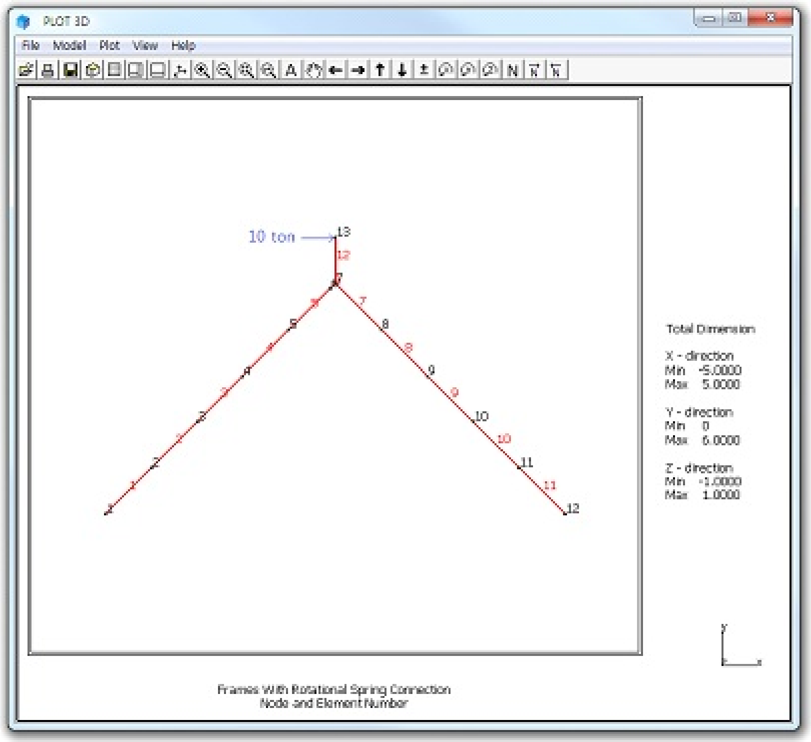


Figure 4.77 Finite element mesh

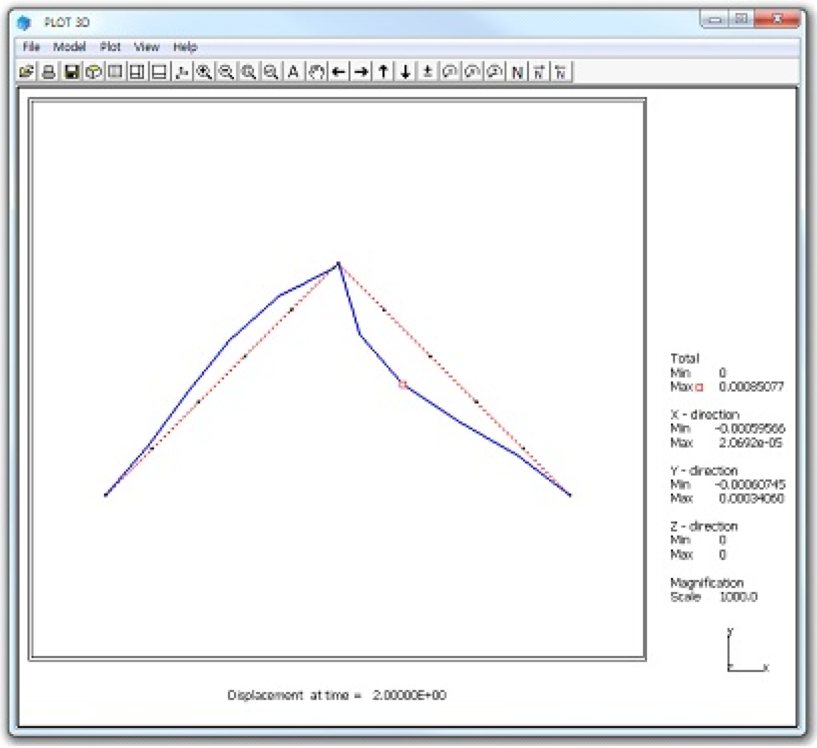


Figure 4.78 Deformed shape

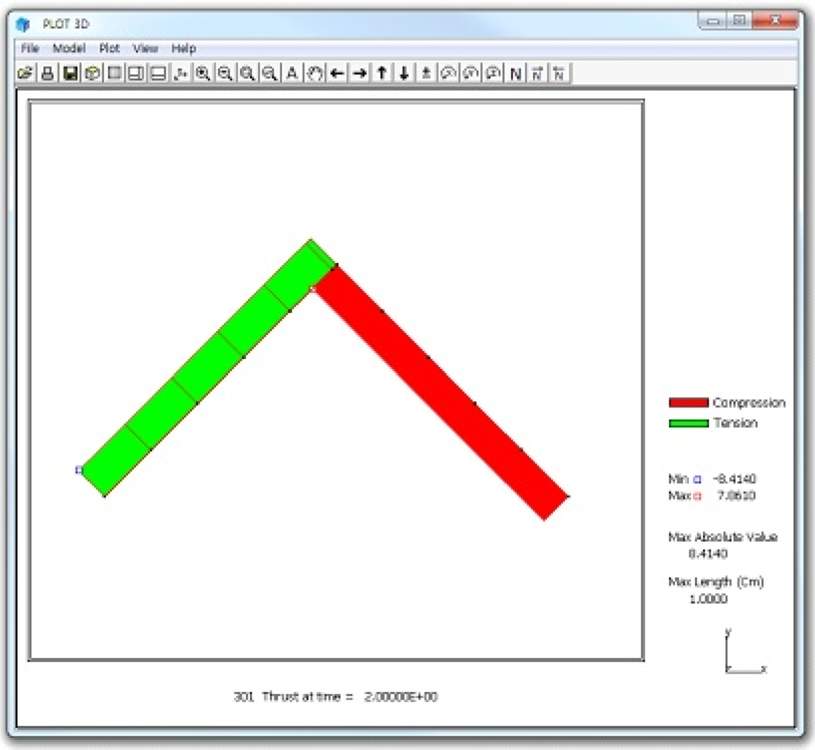


Figure 4.79 Thrust distribution

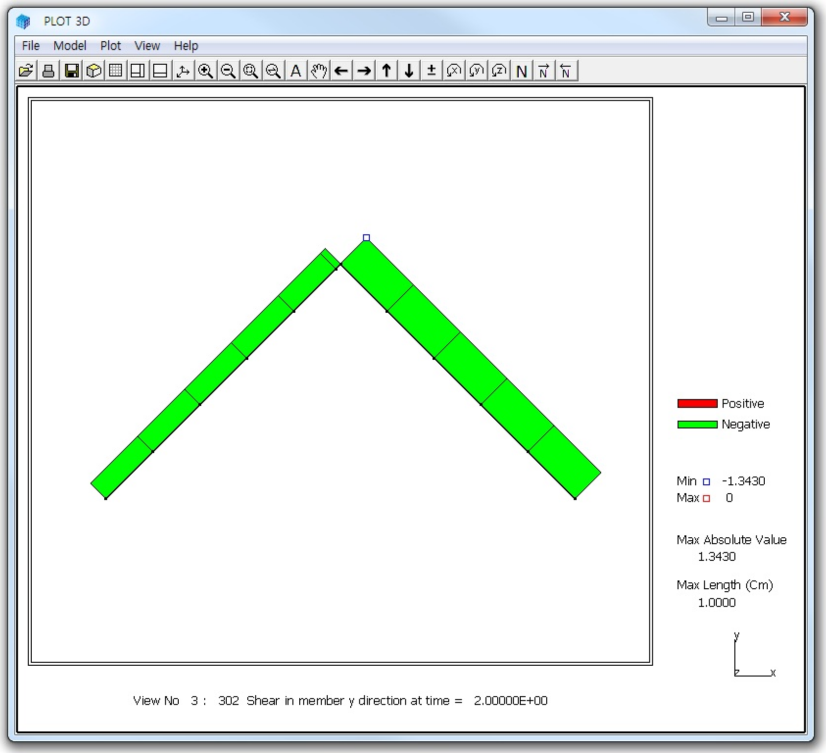


Fig 4.80 Shear distribution

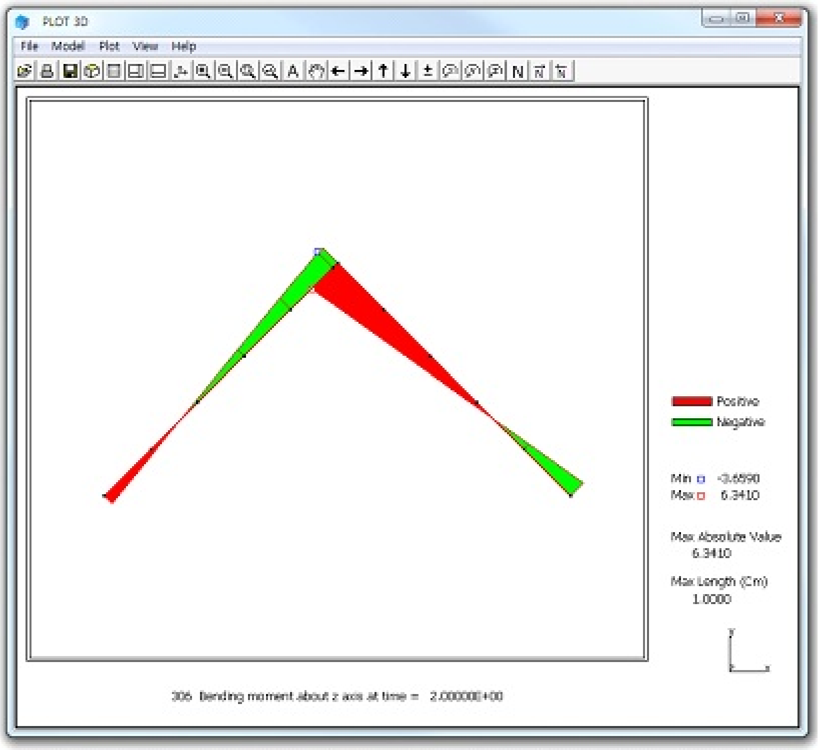


Fig 4.81 Bending moment distribution

4.16 Reinforced Concrete Cylinder

This example is to check the reinforced concrete cylinder subjected to uniformly distributed radial line loads as shown in Figure 4.82.
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_S2.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 4.83, the structure is modeled by beam elements which have capability of modeling axially symmetric reinforced shell.

Overall, SMAP-S2 results are very close to the exact solutions.
Refer to the following two files for detailed graphical outputs:
Reinforced_Cylinder_S2.pdf and Smap-S2_Vp16.pdf.

SMAP-S2 results for case 3 are compared with closed-form solutions:

Figure 4.84 Radial displacement profile
Figure 4.85 Meridian bending moment profile

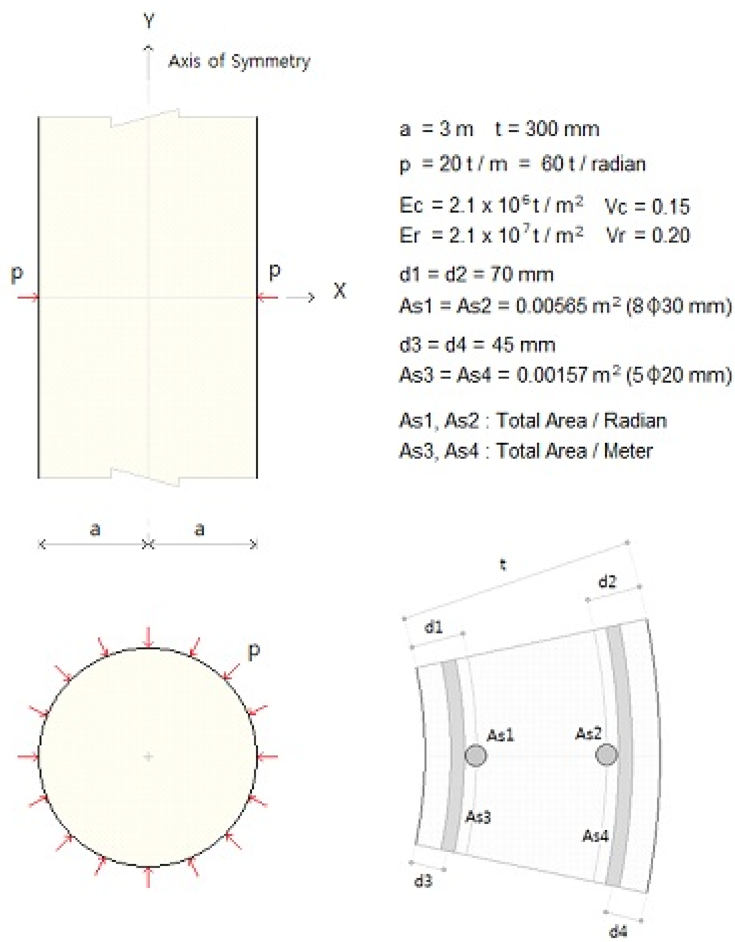


Figure 4.82 Reinforced cylinder section view

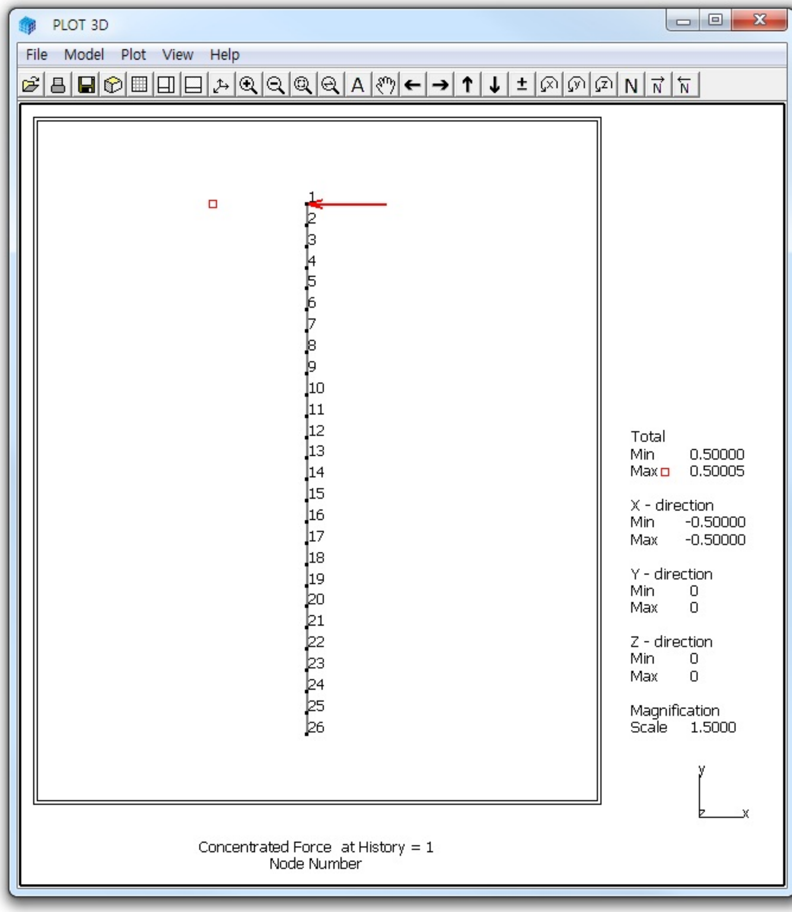


Figure 4.83 Finite element mesh with applied load

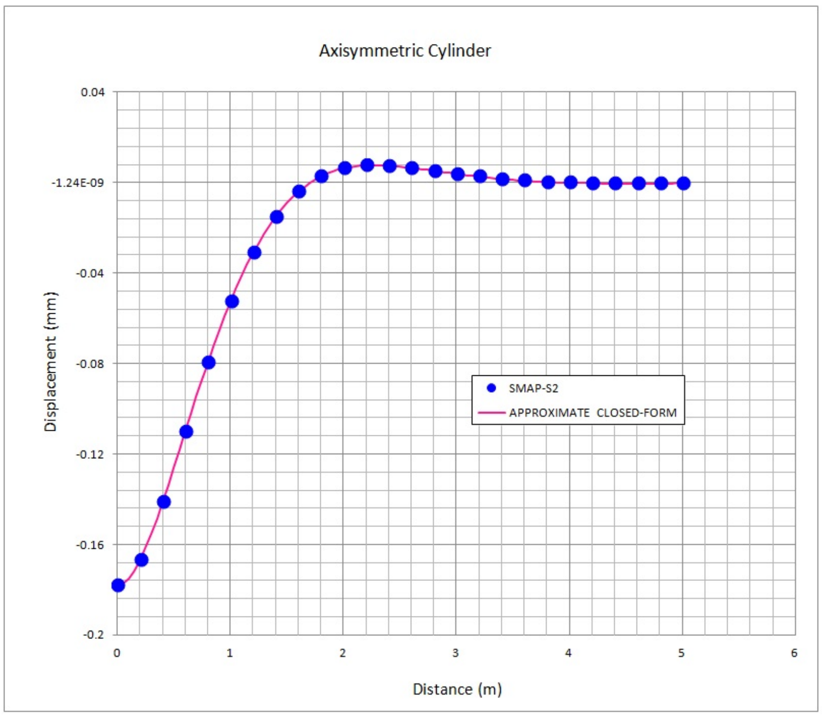


Figure 4.84 Radial displacement profile

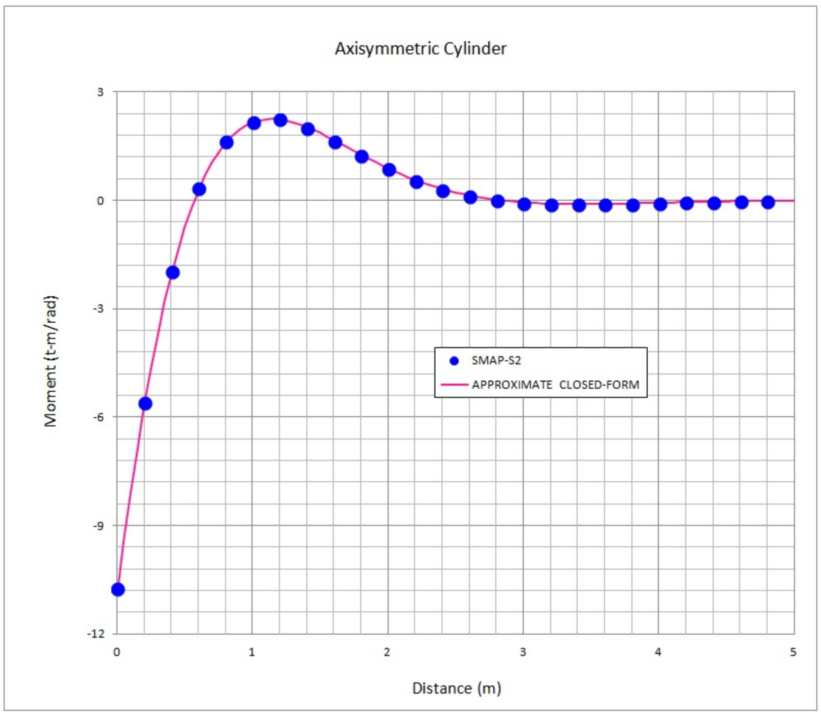


Figure 4.85 Meridian bending moment profile

4.17 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 4.86 and 4.87 shows 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 4.8 lists material properties and Figure 4.88 shows schematic view of detailed silo lining structure. Table 4.9 lists lining thickness and reinforcement. Figure 4.89 shows silo lining material numbers. Table 4.10 shows schematically the sequence of silo construction including residual water pressure applied at step 5. Figure 4.90 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 4.91	Deformed shape of silo lining
Figure 4.92	Dome deflection along A-B
Figure 4.93	Storage wall radial displacement along C-D
Figure 4.94	Dome lining inner hoop stress along A-B
Figure 4.95	Dome outer rebar meridian stress along A-B
Figure 4.96	Storage wall lining inner hoop stress along C-D
Figure 4.97	Storage wall outer rebar meridian stress along C-D

SMAP-S2 results are compared with SMAP-3D results to verify the validity of the solution. As shown, SMAP-S2 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.

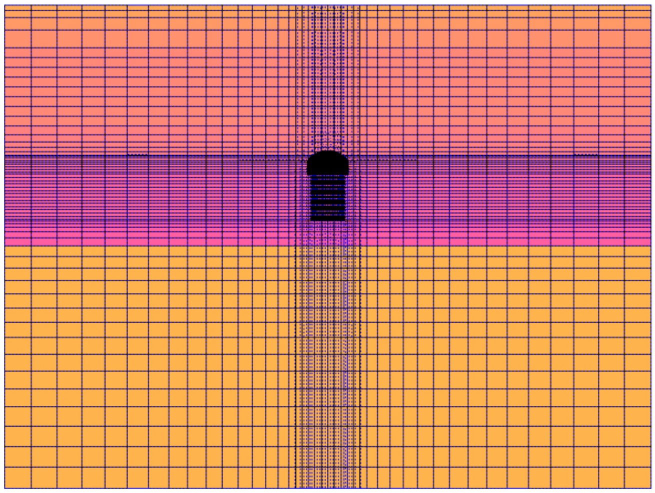


Figure 4.86 Finite element meshes

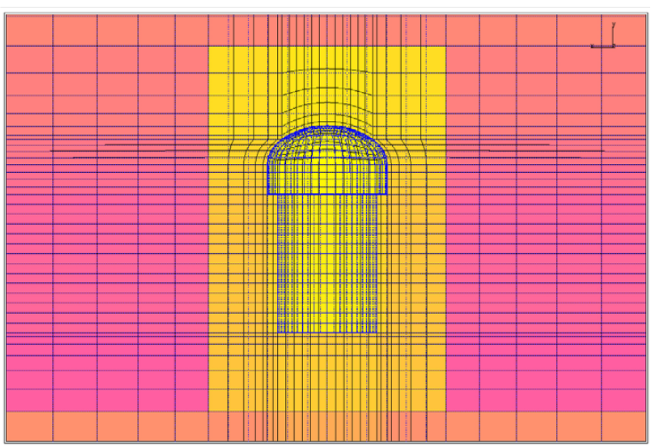


Figure 4.87 Finite element meshes around silo

Table 4.8 Material properties

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

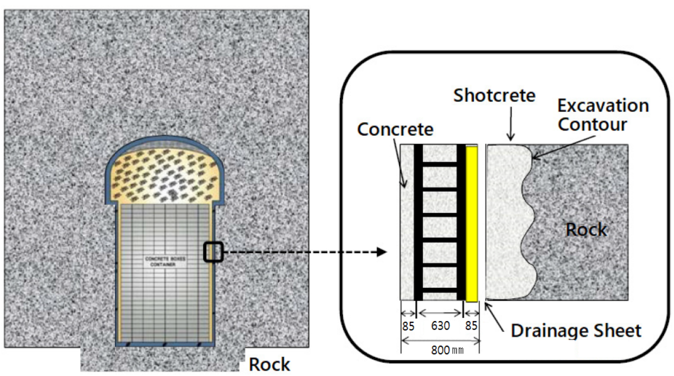


Figure 4.88 Schematic view of detailed silo lining structure

Table 4.9 Silo lining thickness and reinforcement

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

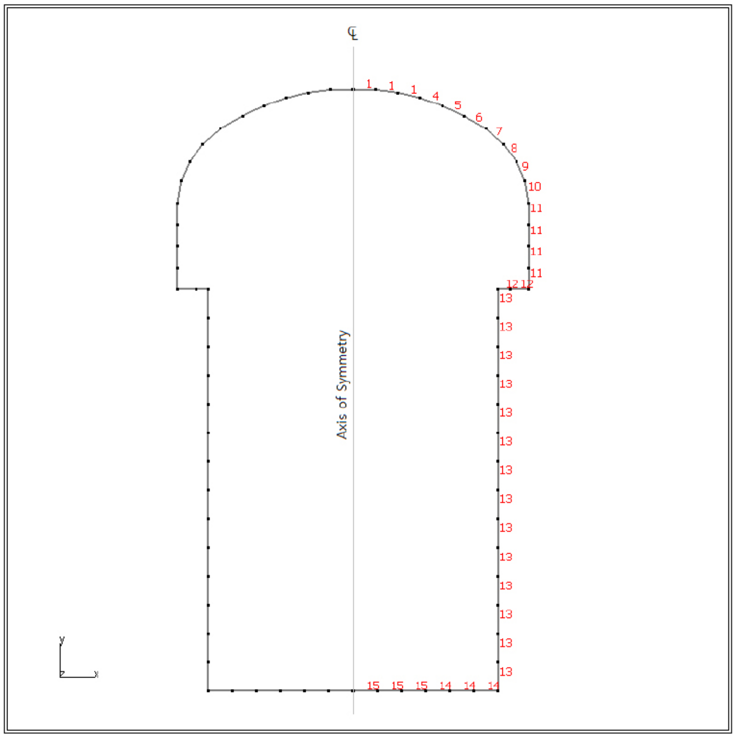
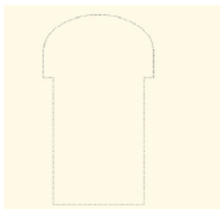
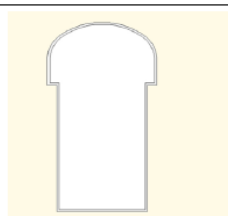
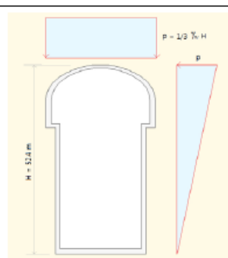


Figure 4.89 Silo lining material numbers

Table 4.10 Construction sequence

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

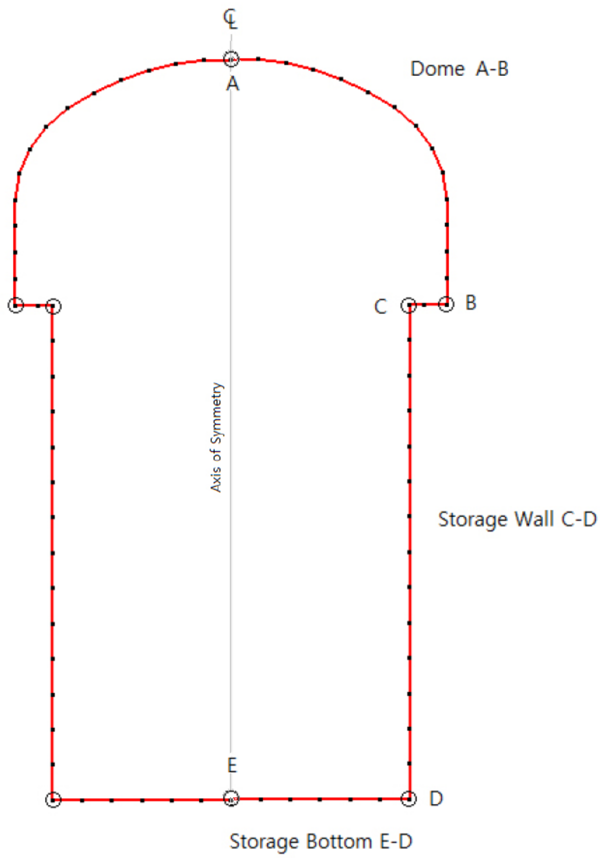


Figure 4.90 Key locations along silo lining

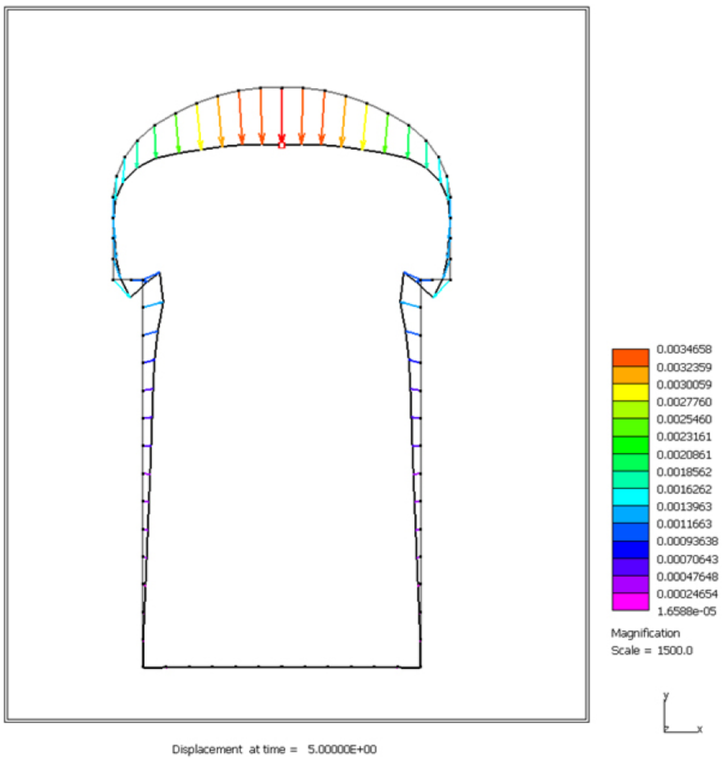


Figure 4.91 Deformed shape of silo lining

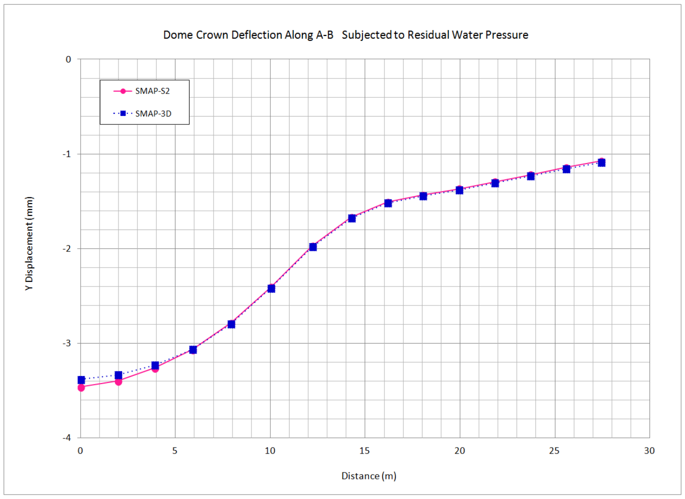


Figure 4.92 Dome deflection along A-B

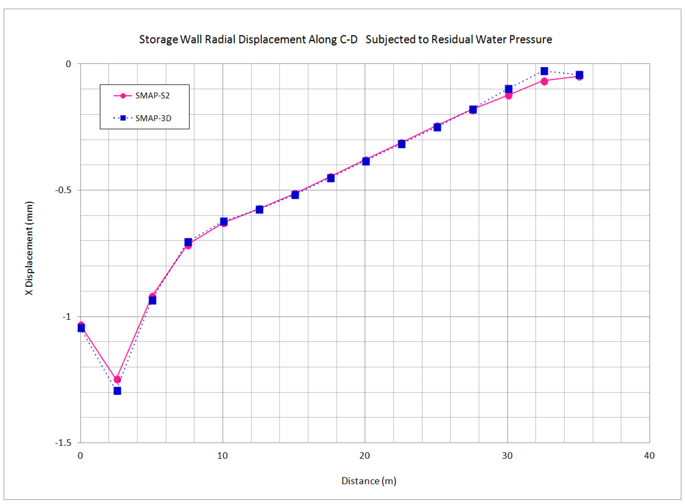


Figure 4.93 Storage wall radial displacement along C-D

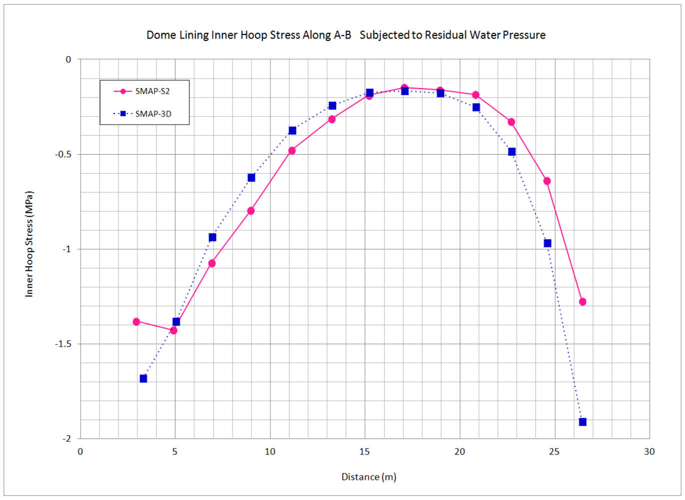


Figure 4.94 Dome lining inner hoop stress along A-B

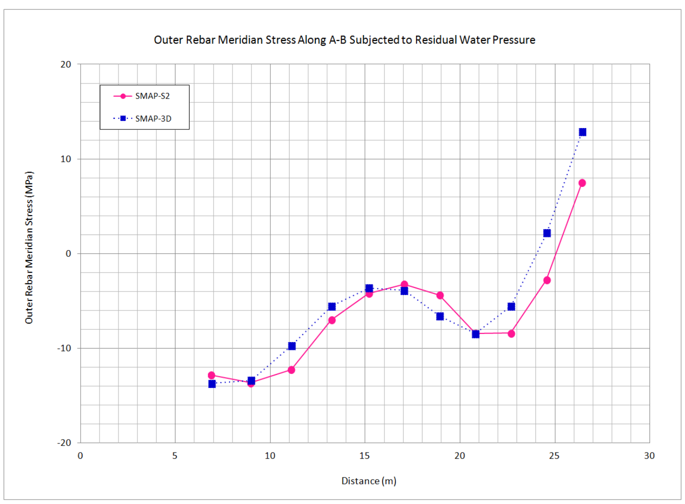


Figure 4.95 Dome outer rebar meridian stress along A-B

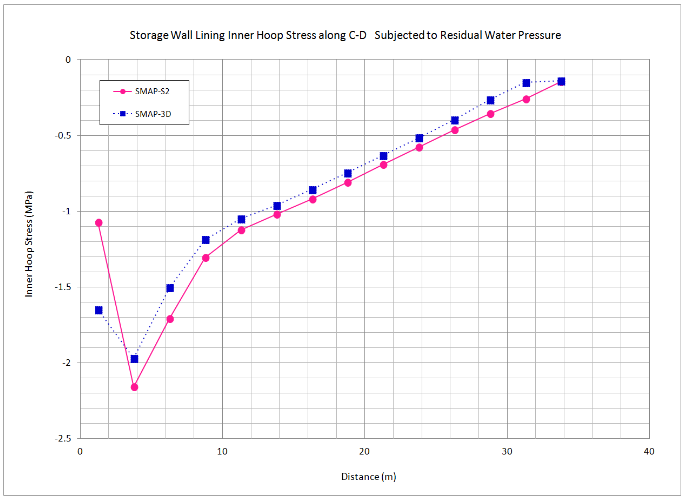


Figure 4.96 Storage wall lining inner hoop stress along C-D

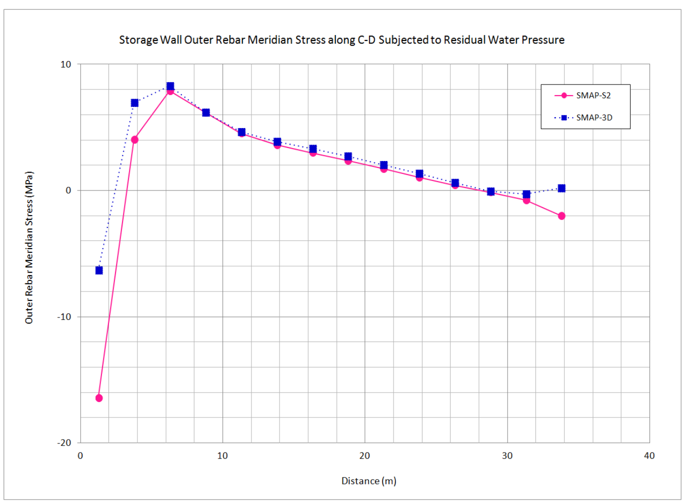


Figure 4.97 Storage wall outer rebar meridian stress along C-D

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.

Two example problems are presented:

1. Arch Tunnel
   Shows step by step procedure to create and modify group meshes.
2. 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.

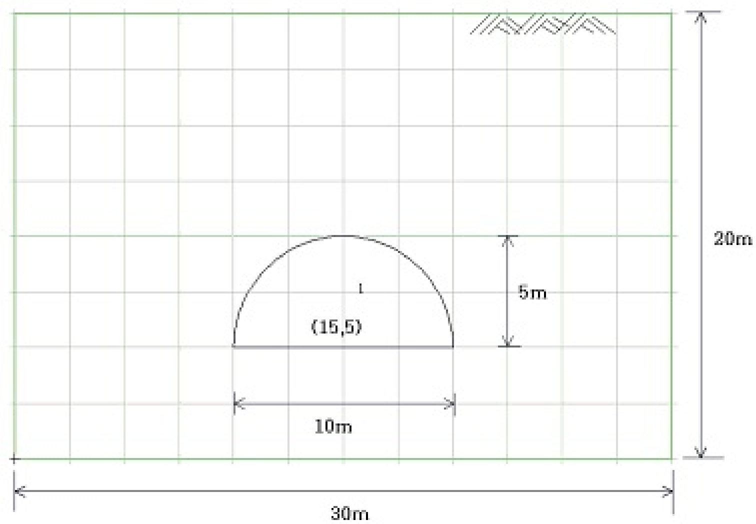


Figure 5.1 Arch tunnel (Part 1)

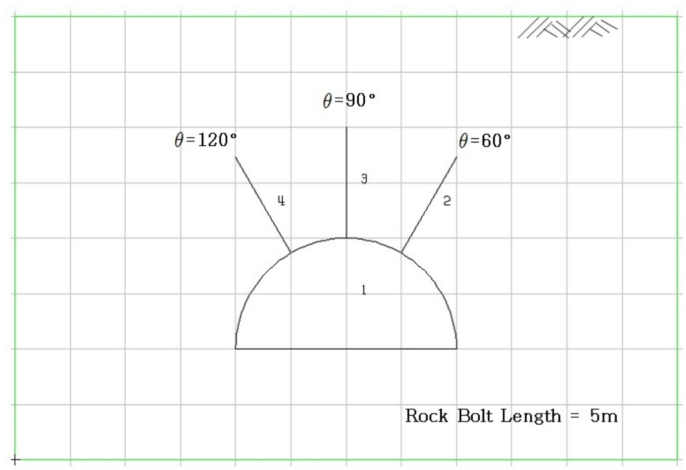


Figure 5.2 Arch tunnel with rock bolts (Part 2)

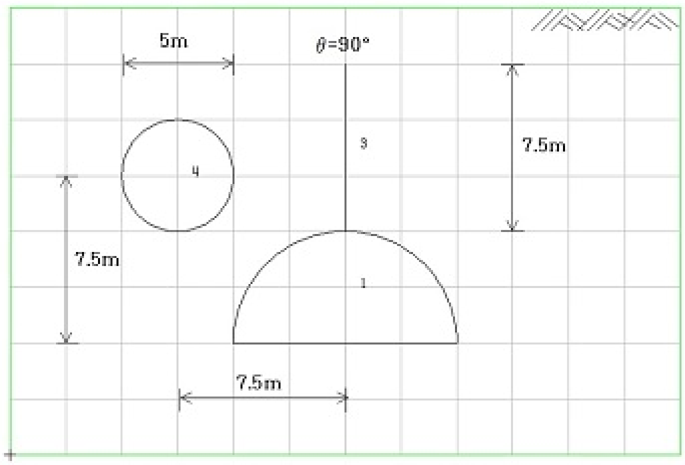


Figure 5.3 Arch tunnel with utility tunnel (Part 3)

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

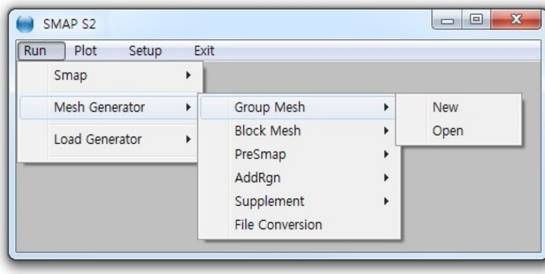


Figure 5.4 Accessing group mesh generator (New)

Step 2: Group Input (New)

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

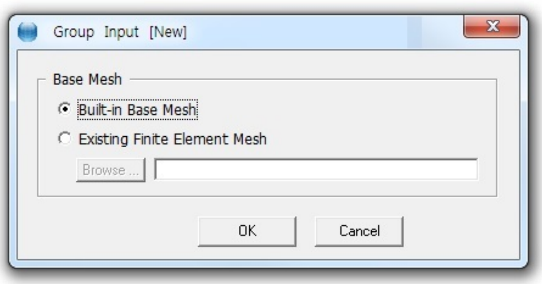


Figure 5.5 Group input (New)

Step 3: Group Menu and Dialog

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

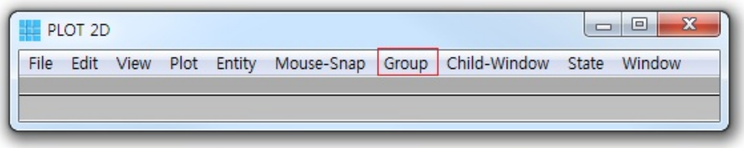


Figure 5.6 Group menu

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

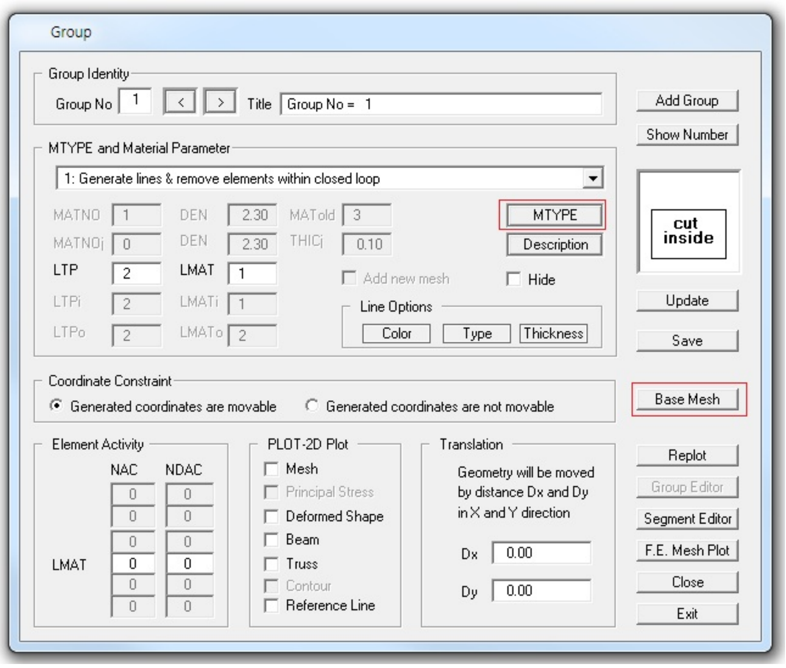


Figure 5.7 Group dialog with initial default values

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.

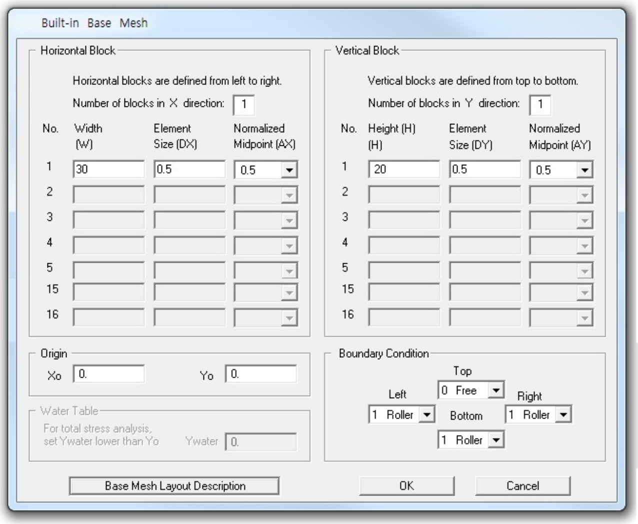


Figure 5.8 Built-in base mesh dialog

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

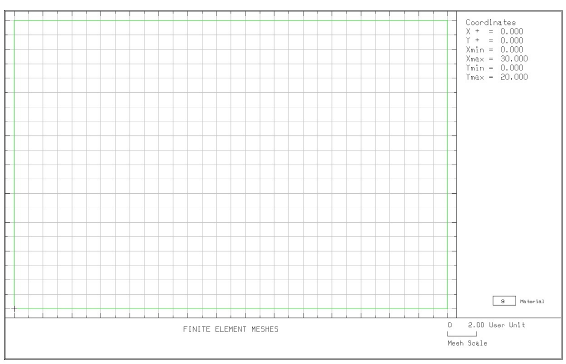


Figure 5.9 Base mesh on drawing board

Step 5: MTYPE

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

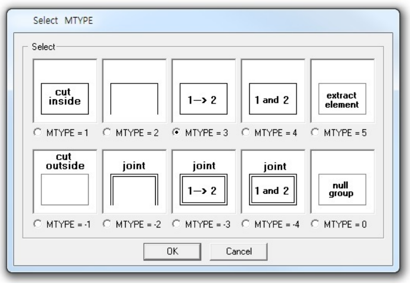


Figure 5.10 MTYPE dialog

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

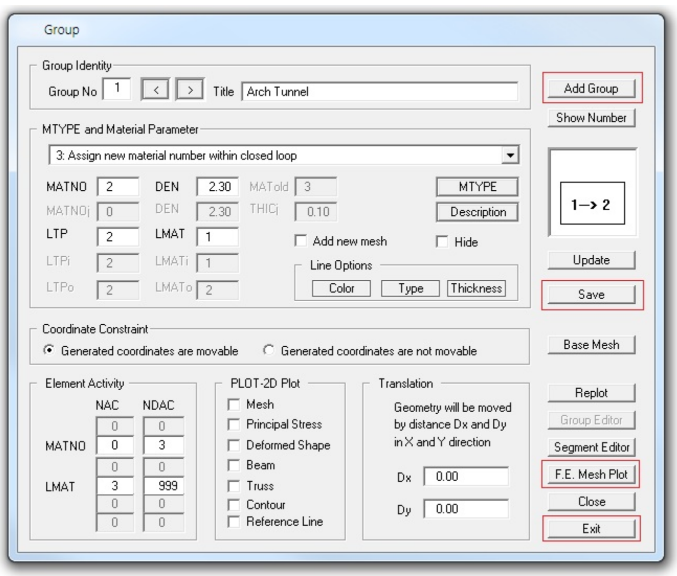


Figure 5.11 Group dialog with MTYPE = 3

Step 6: Mouse Snap

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

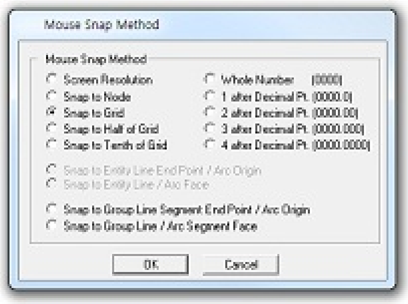


Figure 5.12
Mouse snap dialog

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)	LFUN=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.

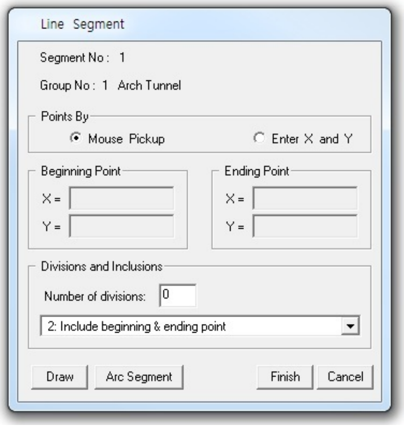


Figure 5.13 Line segment dialog

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

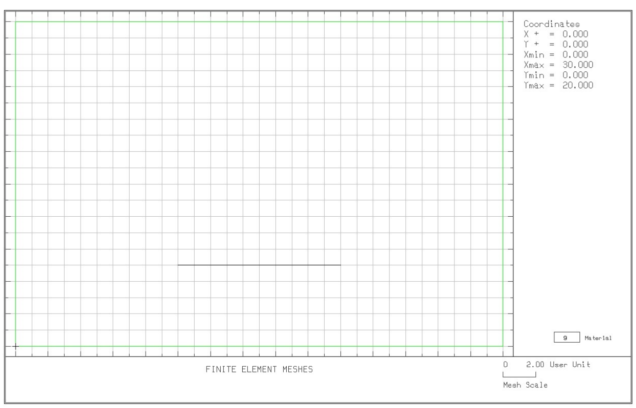


Figure 5.14 Line segment on drawing board

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.

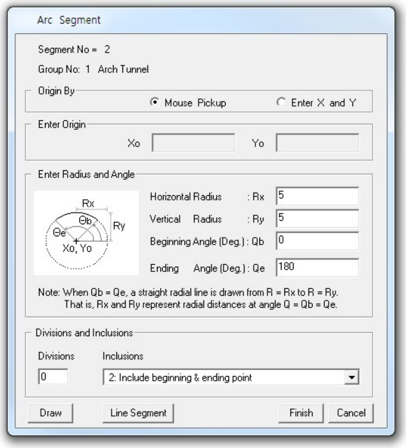


Figure 5.15 Arc segment dialog

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.

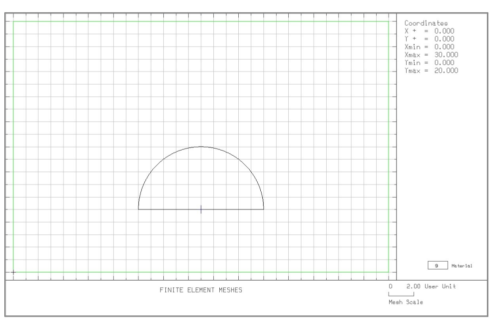


Figure 5.16 Arc segment on drawing board

Step 10: Save

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

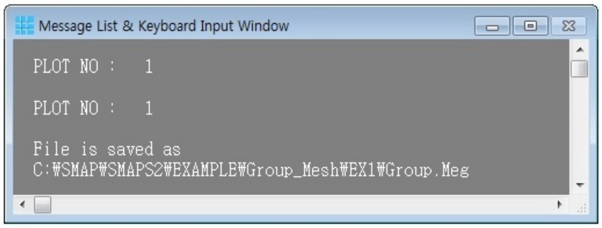


Figure 5.17 Message for file save

Step 11: Finite Element Mesh

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

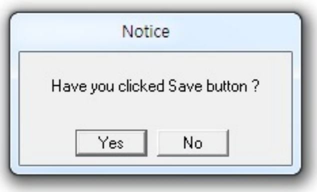


Figure 5.18 Notice for finite element mesh plot

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

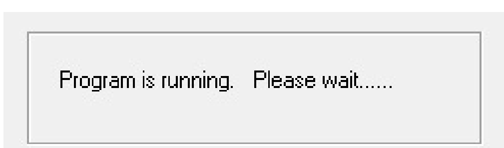


Figure 5.19 Notice while generating finite element mesh plot

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

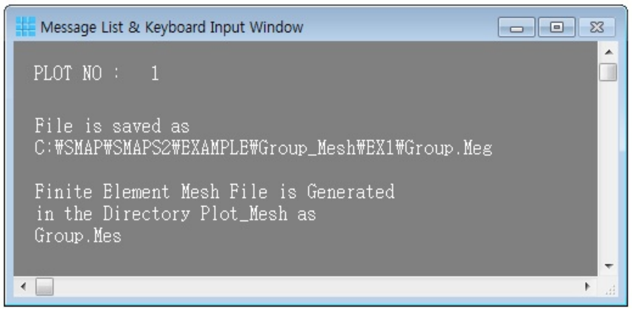


Figure 5.20 Message for finite element mesh file

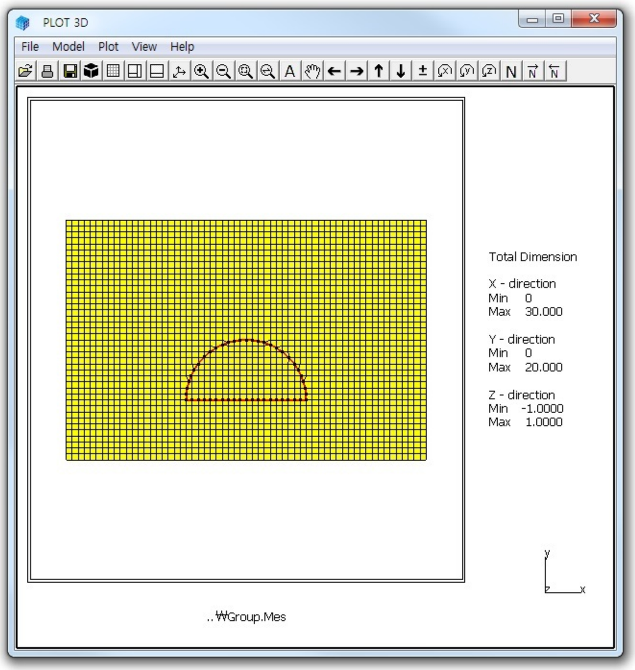


Figure 5.21 Finite element mesh plot

Step 12: Exit

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

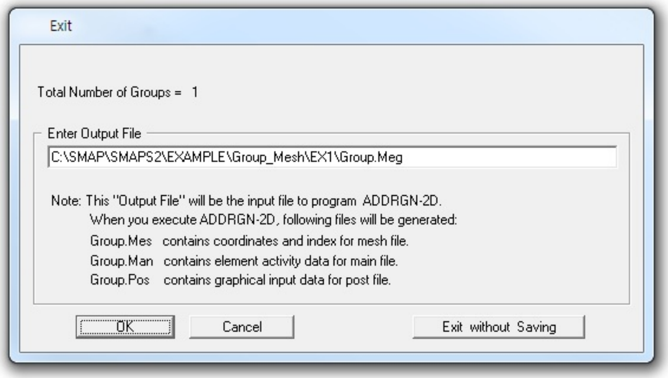


Figure 5.22 Exit dialog

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.

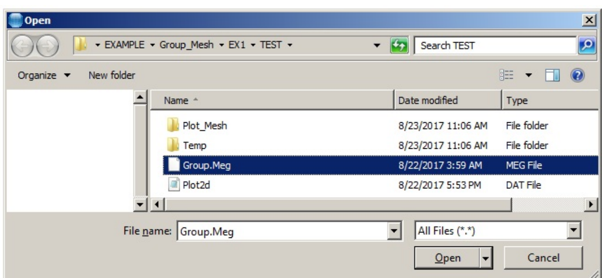


Figure 5.23 File open dialog

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 (LFUN)	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.

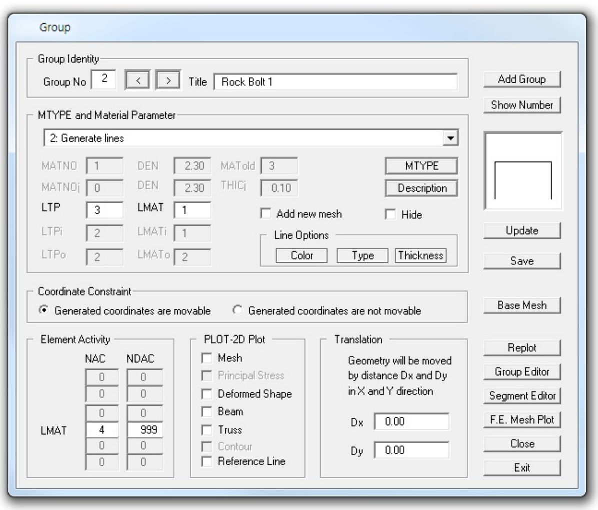


Figure 5.24 Group dialog for Rock Bolt 1

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.

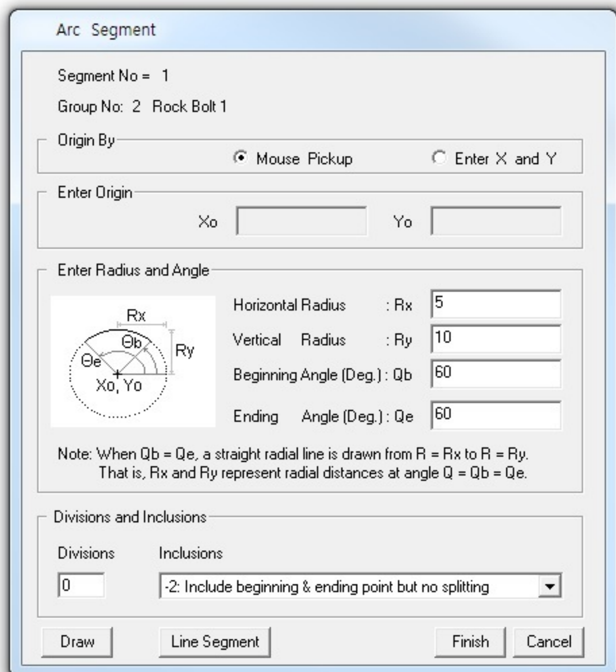


Figure 5.25 Arc segment dialog for Rock Bolt 1

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.

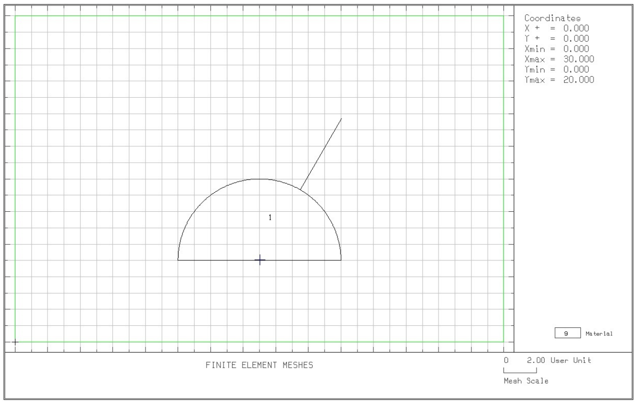


Figure 5.26 Rock Bolt 1 on drawing board

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.

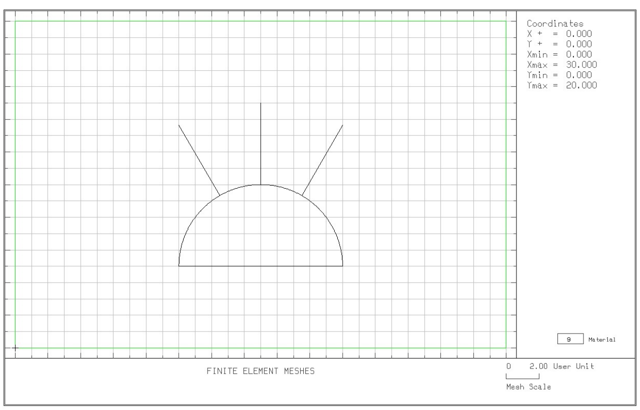


Figure 5.27 All three rock bolts on drawing board

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

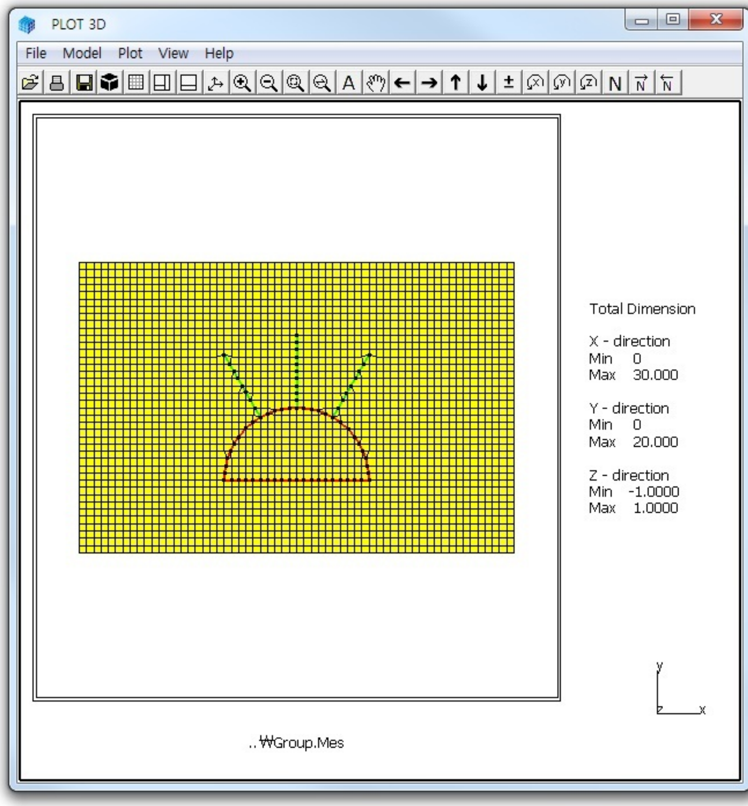


Figure 5.28 Finite element mesh plot

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

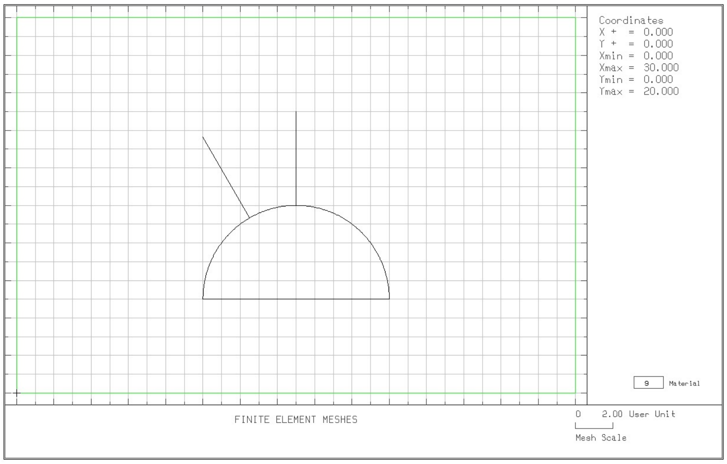


Figure 5.29 Rock Bolt 1 removed on drawing board

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.

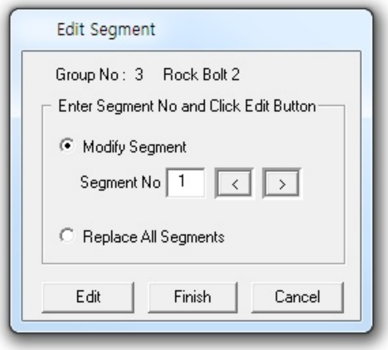


Figure 5.30
Edit segment dialog
for Group No 3

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.

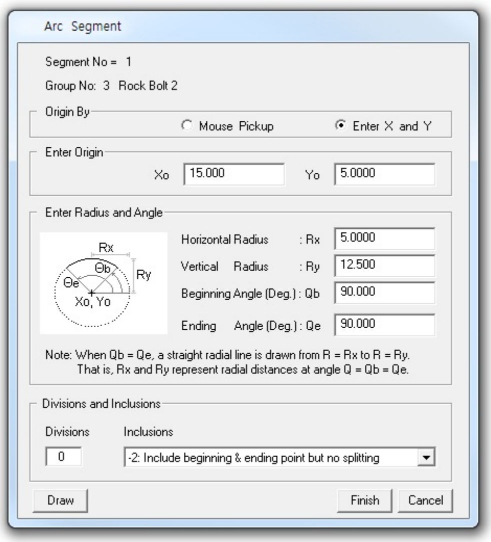


Figure 5.31
Arc segment dialog with
rock bolt length modified

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

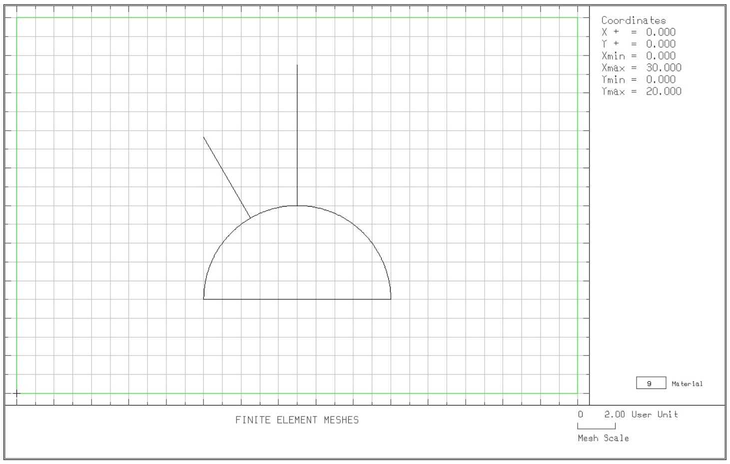


Figure 5.32 Longer Rock Bolt 2 on drawing board

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.

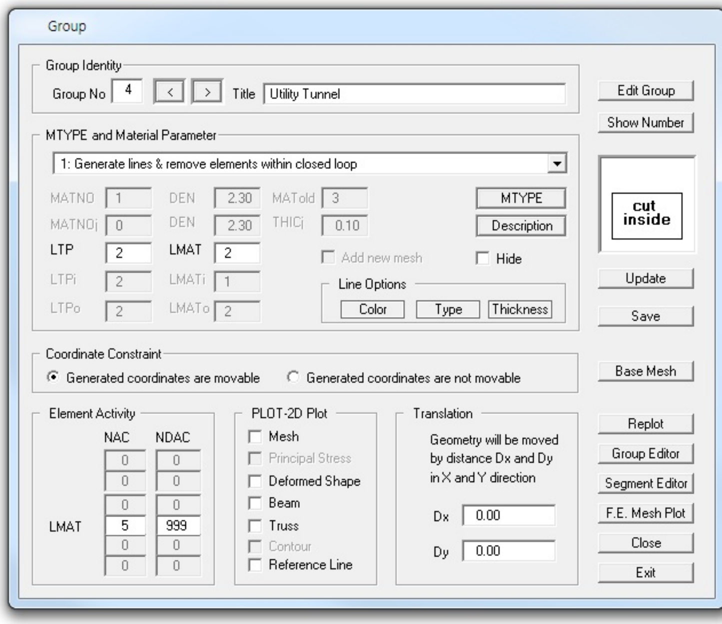


Figure 5.33 Group dialog for Utility Tunnel

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

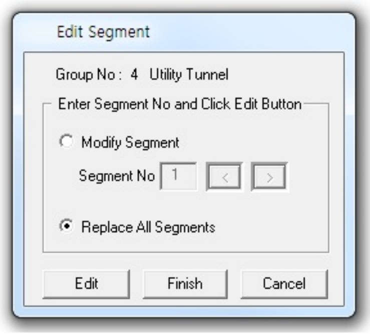


Figure 5.34 Edit segment dialog for Group No 4

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

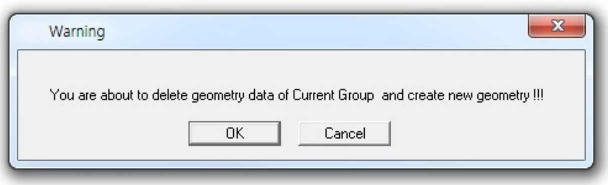


Figure 5.35 Warning message

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.

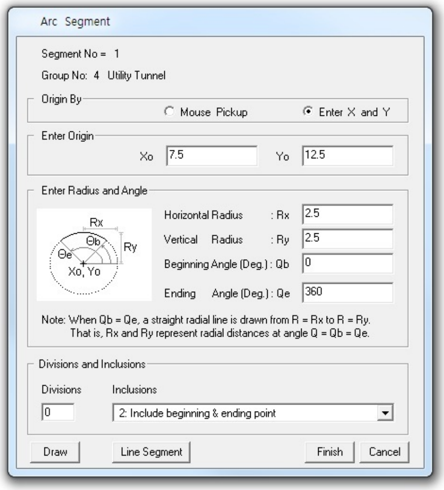


Figure 5.36 Arc segment dialog for Utility Tunnel

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

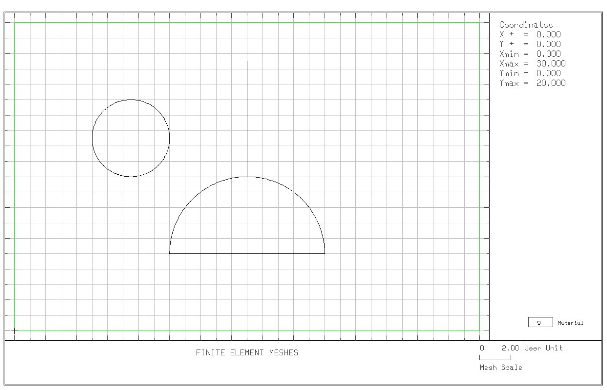


Figure 5.37 Arch and Utility Tunnels on drawing board

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

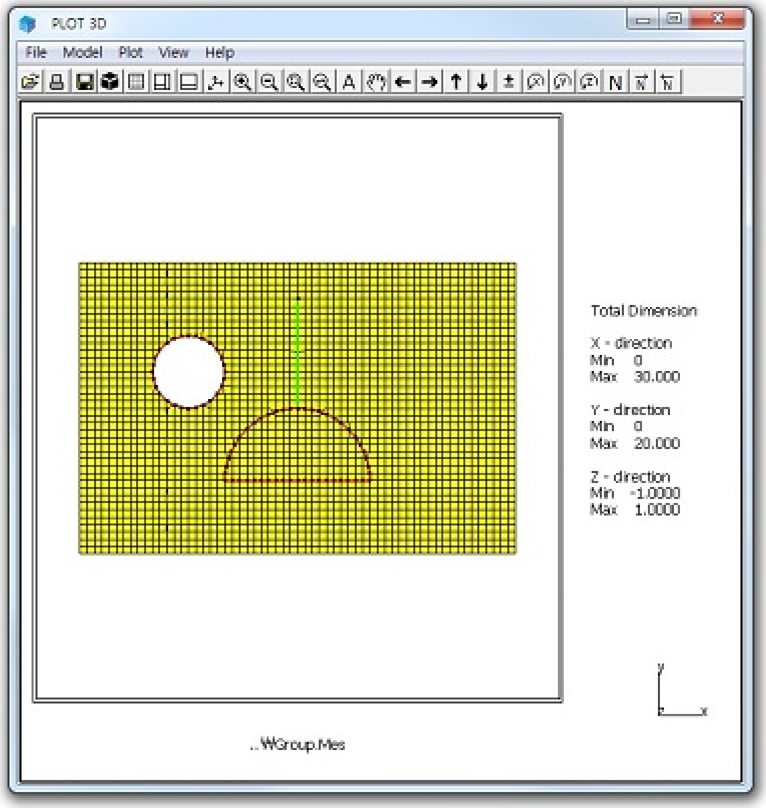


Figure 5.38 Finite element mesh plot

Step 29: Exit

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

5.2 Finite Element Mesh Modification

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

5.2.1 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 5.39.

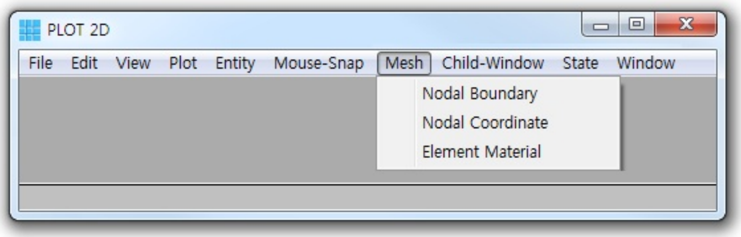


Figure 5.39 Menu for editing finite element mesh

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 5.40 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 5.41.

This modification involves following three works:

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

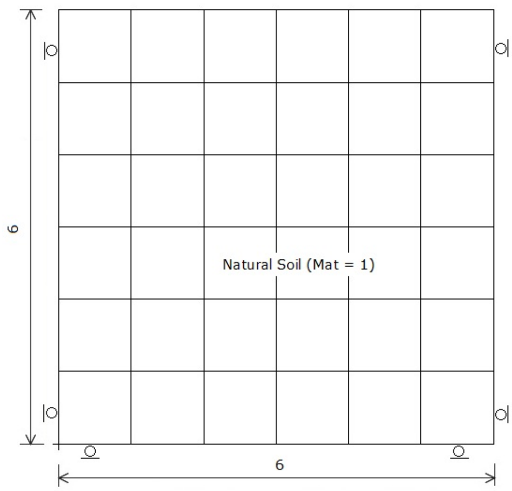


Figure 5.40 Existing finite element mesh

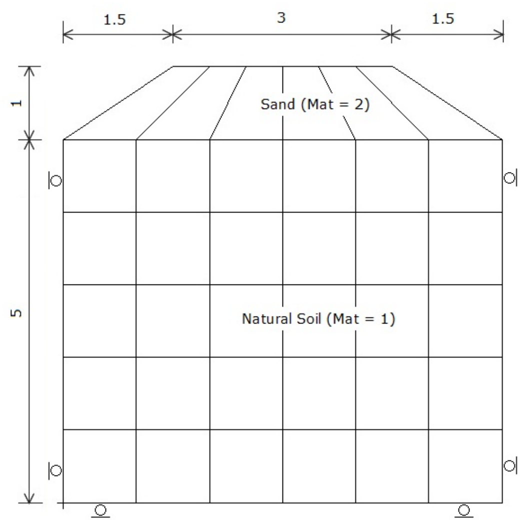


Figure 5.41 Modified finite element mesh

5.2.2 Change Top Surface Nodal Coordinates

Click Nodal Coordinate from the Mesh menu, then Edit Coordinate dialog in Figure 5.42 is displayed.

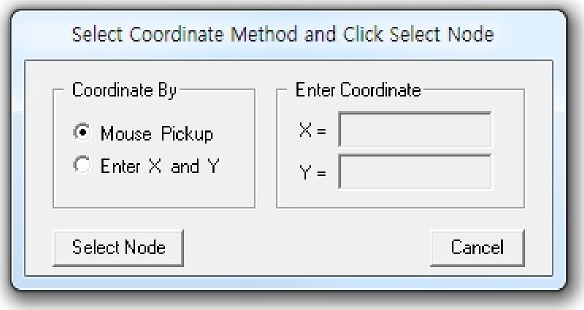


Figure 5.42 Edit coordinate dialog

For this example, Snap to Half of Grid in Figure 5.43 is the most convenient method for Mouse Pickup.

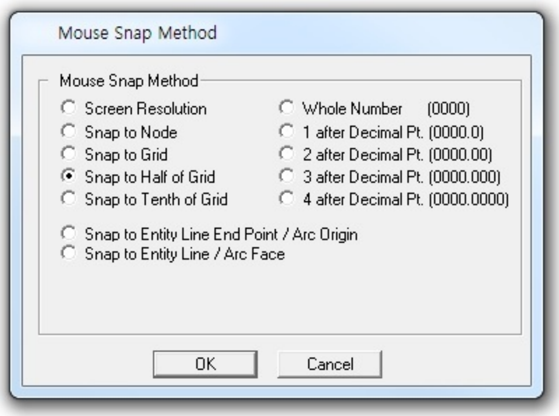


Figure 5.43 Mouse snap method

Click Select Node in Figure 5.42.
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 5.44.

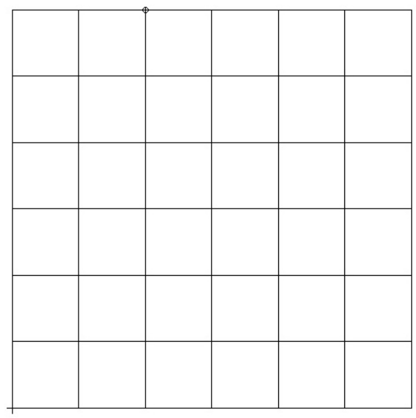


Figure 5.44 First selected node on drawing board

Now, move the first selected node by using drag-and-drop of Mouse Left Button as shown in Figure 5.45.

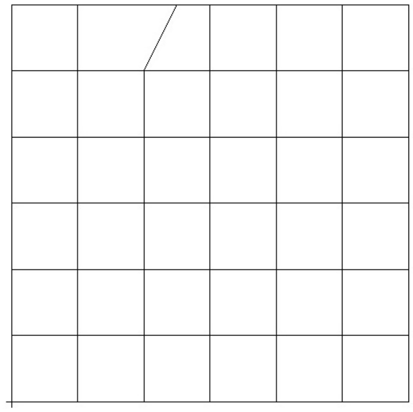


Figure 5.45 New position of first selected node

Select the next node by Mouse Right Click as shown in Figure 5.46.

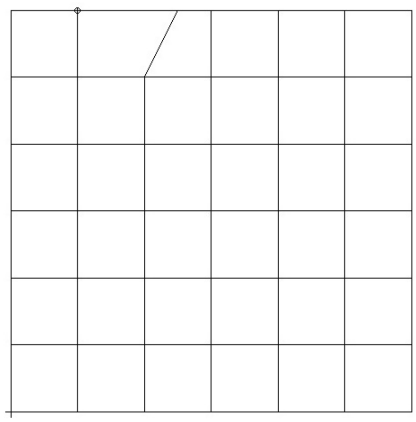


Figure 5.46 Second selected node on drawing board

Now, move the second selected node by using drag-and-drop of Mouse Left Button as shown in Figure 5.47.

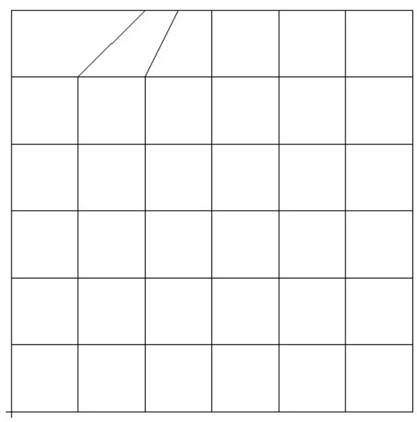


Figure 5.47 New position of second selected node

Repeat the same procedure for all other nodes on the top surface. Once finished, click Finish button in Figure 5.48.

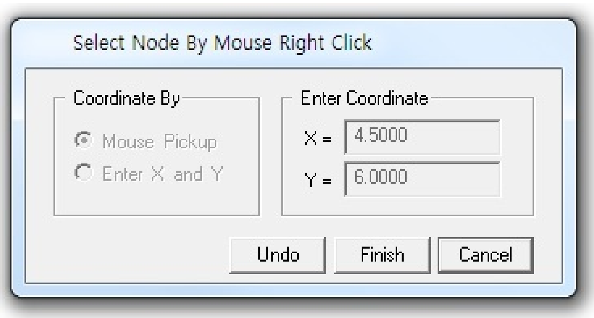


Figure 5.48 Edit coordinate dialog

Figure 5.49 shows final finite element mesh on the drawing board.

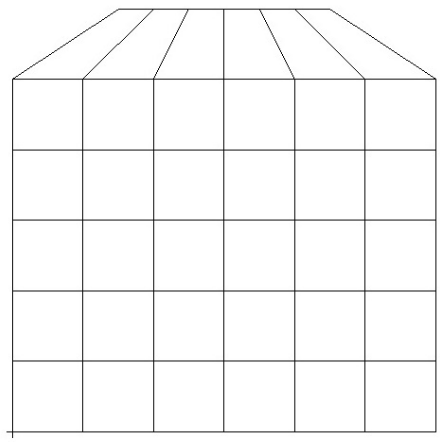


Figure 5.49 Final finite element mesh

5.2.3 Change Top Surface Nodal Boundaries

Click Nodal Boundary from the Mesh menu, then Edit Boundary Code dialog in Figure 5.50 is displayed.

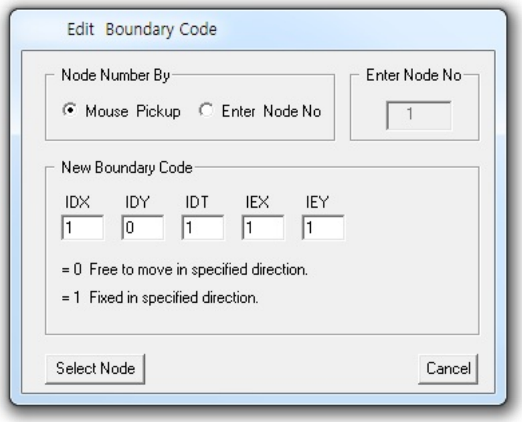


Figure 5.50
Edit boundary dialog

Click Select Node in Figure 5.50.
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 5.51.

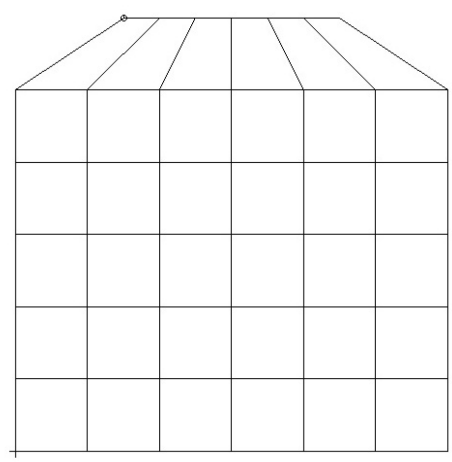
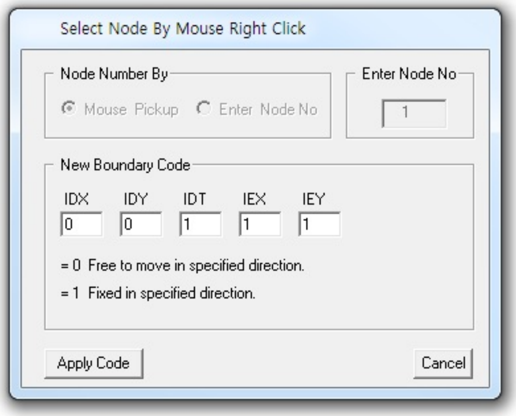


Figure 5.51
Selected node on drawing board

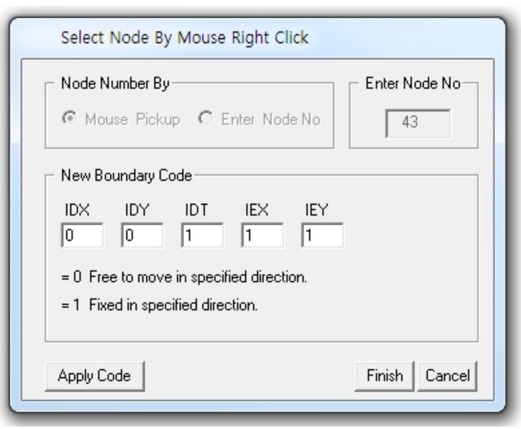
Change the boundary codes as in Figure 5.52 so that the top left node can be free to move in both horizontal and vertical directions and then click Apply Code button.

Figure 5.52
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 5.53.

Figure 5.53
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 5.54 and then click OK button. Modified skeleton boundary codes are shown in Figure 5.55.

Figure 5.54
General view
for skeleton boundary code

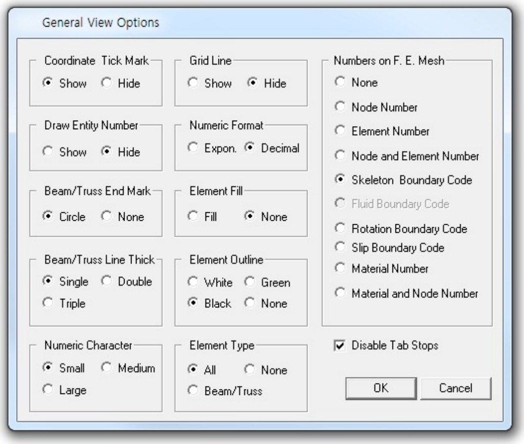
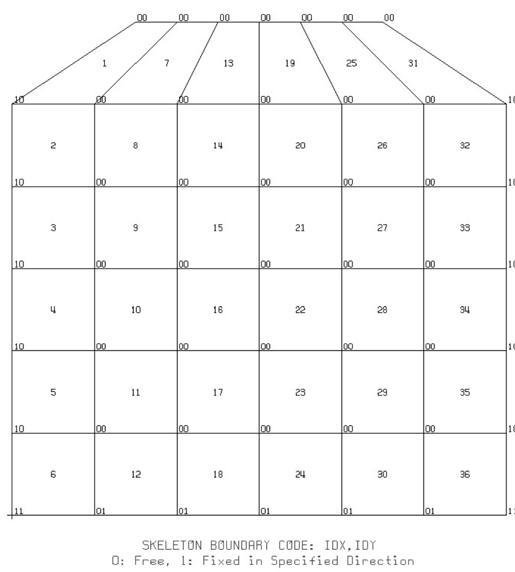


Figure 5.55
Modified skeleton
boundary code plot

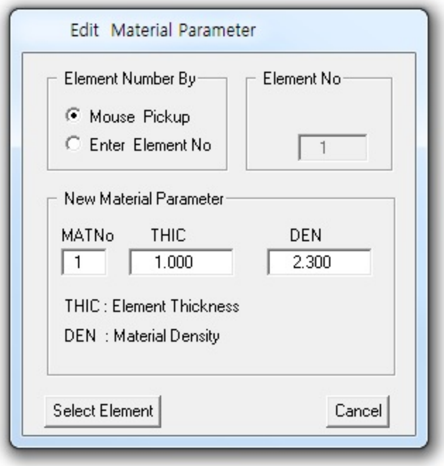


5.2.4 Change Top Layer Element Materials

Click Element Material from the Mesh menu, then Edit Material Parameter dialog in Figure 5.56 is displayed.

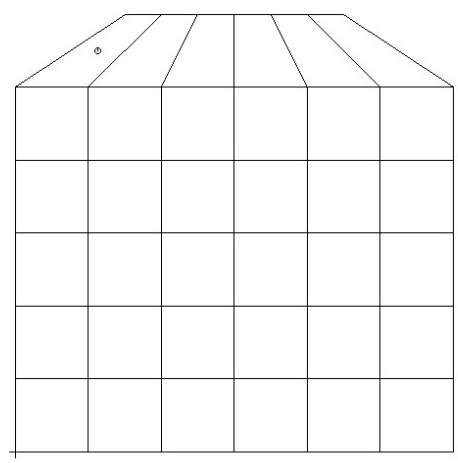
Figure 5.56

Edit element material dialog



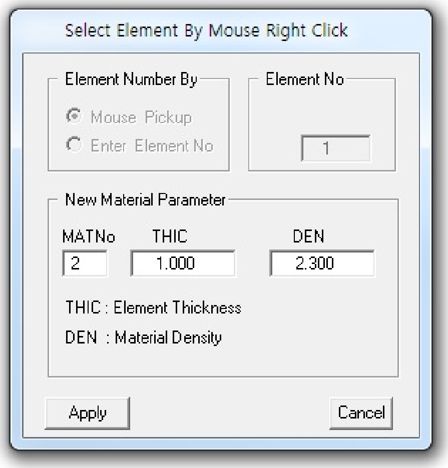
Click Select Element button.
Click the element on the top layer by Mouse Right Click.
Selected element is marked as an open circle as shown in Figure 5.57.

Figure 5.57
Selected element on drawing board



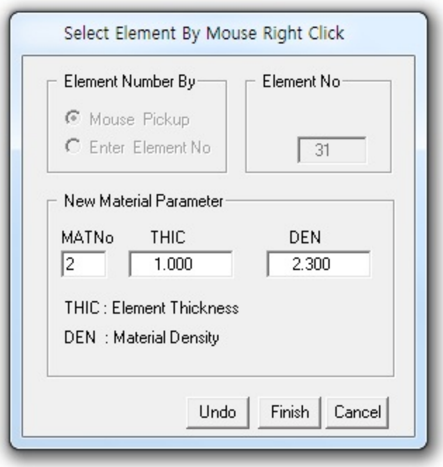
Change the material number as shown in Figure 5.58 and then click Apply button.

Figure 5.58
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 5.59.

Figure 5.59
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 5.60 and then click OK button. Modified material number is shown in Figure 5.61.

Figure 5.60
General view
for material number

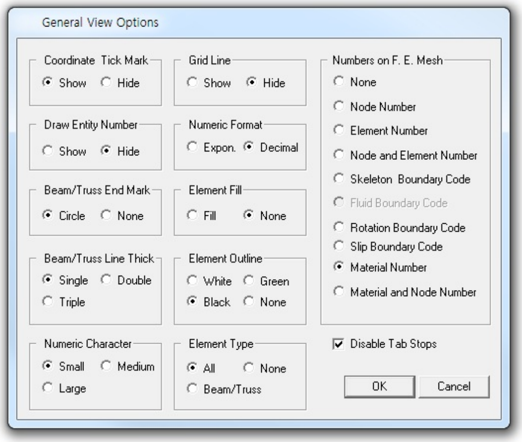
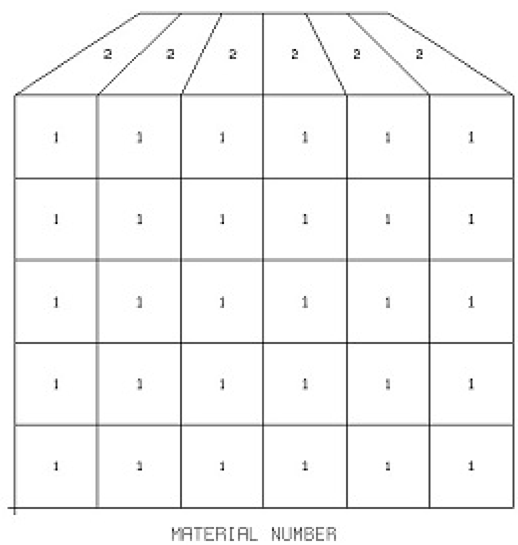


Figure 5.61
Modified material number
plot



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

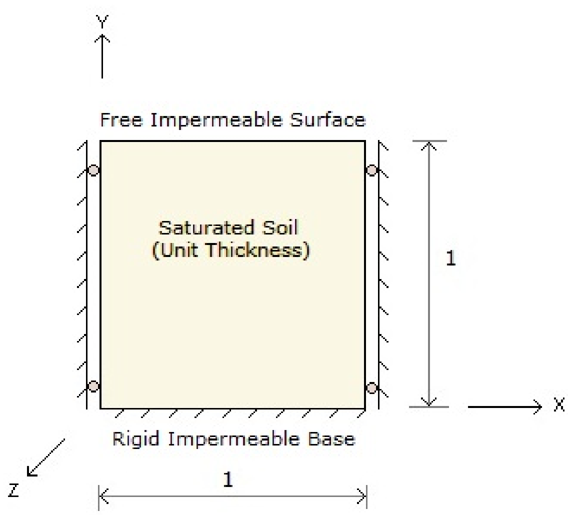


Figure 6.1 Single element in uniaxial strain condition

Step 1: Access Block Mesh Generator (New)

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

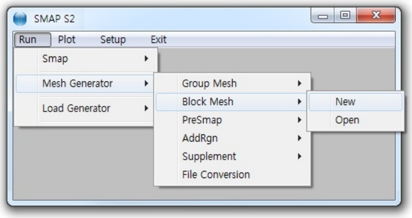


Figure 6.2 Accessing block mesh generator

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.

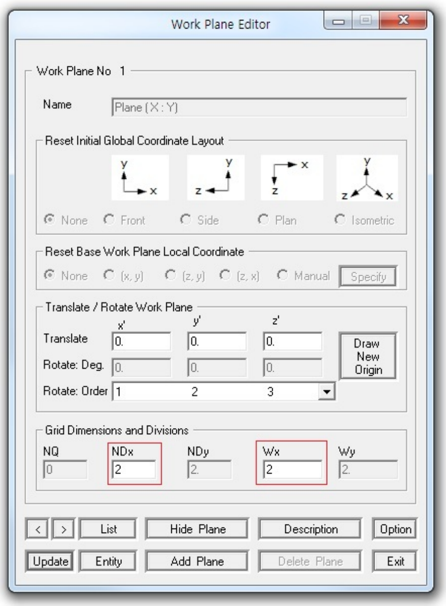


Figure 6.3 Work plane editor

Step 3: Build Cube Entity

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

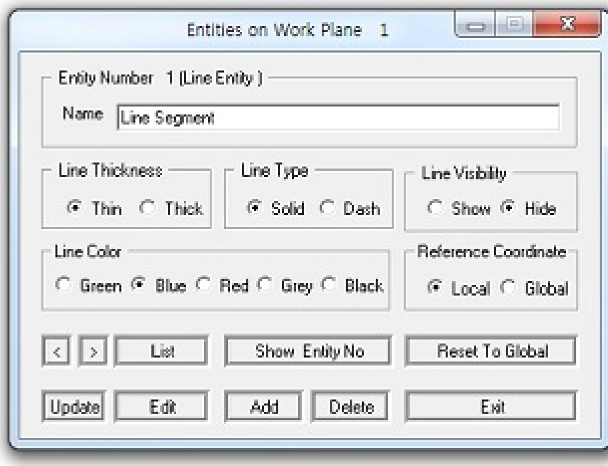


Figure 6.4 Entity editor

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

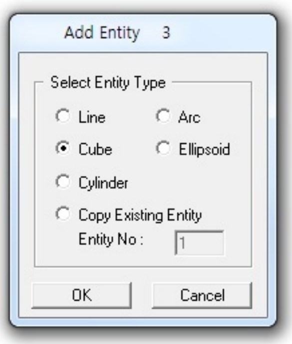


Figure 6.5 Entity type selection

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

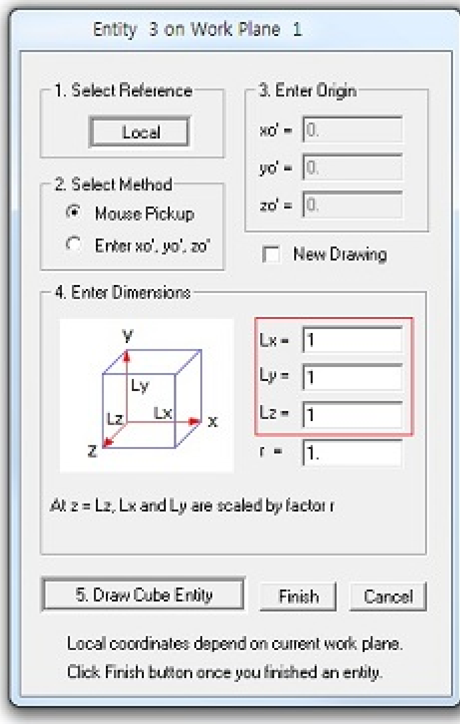


Figure 6.6 Cube entity

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.

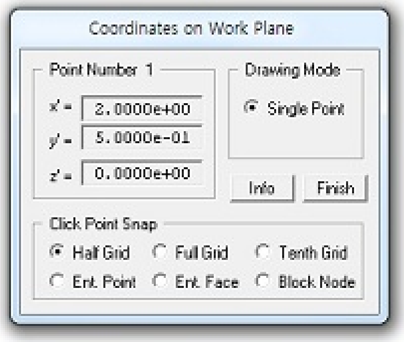


Figure 6.7 Coordinates on work plane

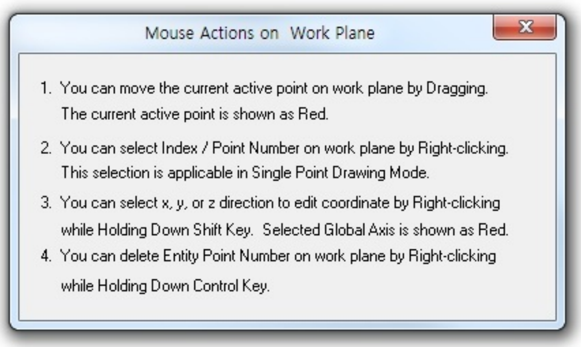


Figure 6.8 Mouse actions on work plane

9. Click Axis toolbar as shown in Figure 6.9.

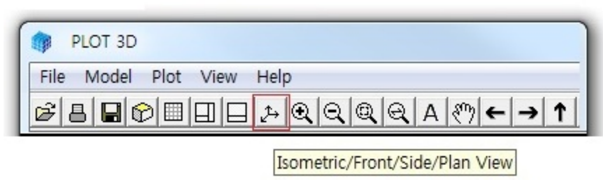


Figure 6.9 Axis toolbar

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

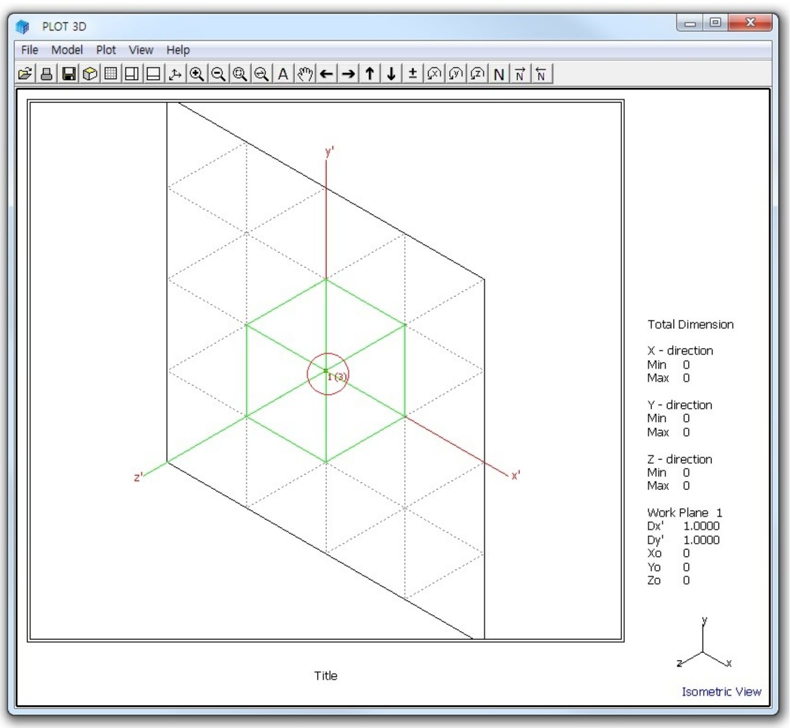


Figure 6.10 Cube entity on isometric work plane

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.

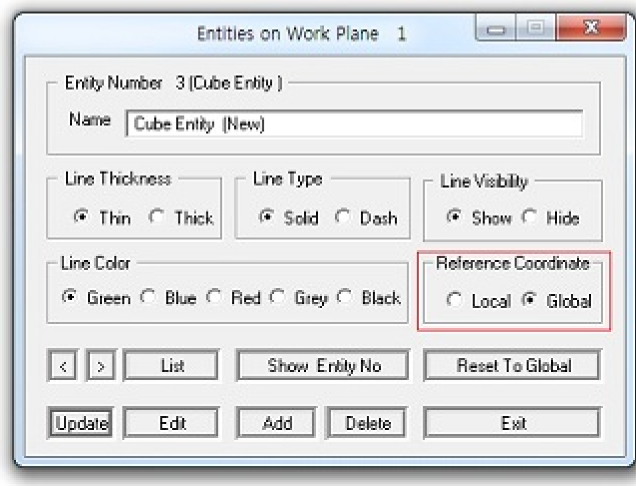


Figure 6.11 Entity editor

Step 4: Build Quad Block

1. Click Block Editor toolbar in Figure 6.12.

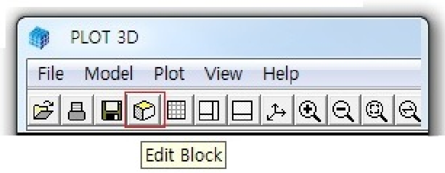


Figure 6.12 Block editor toolbar

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

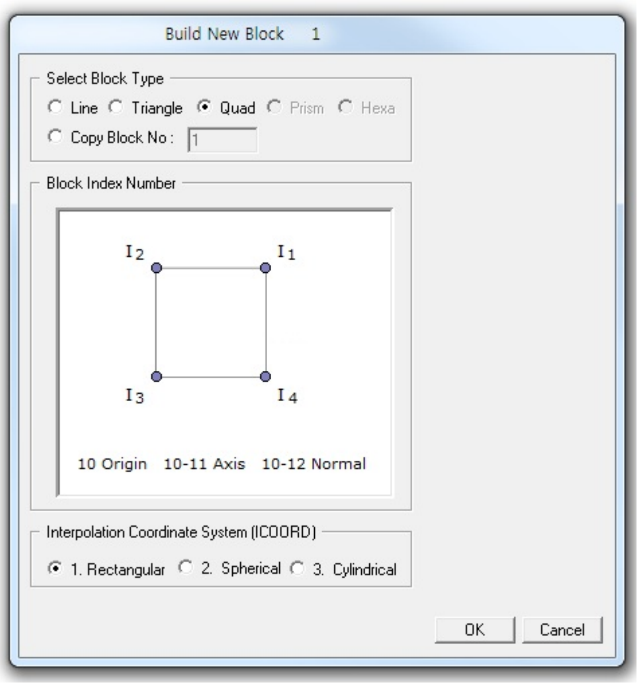


Figure 6.13 Block type selection

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

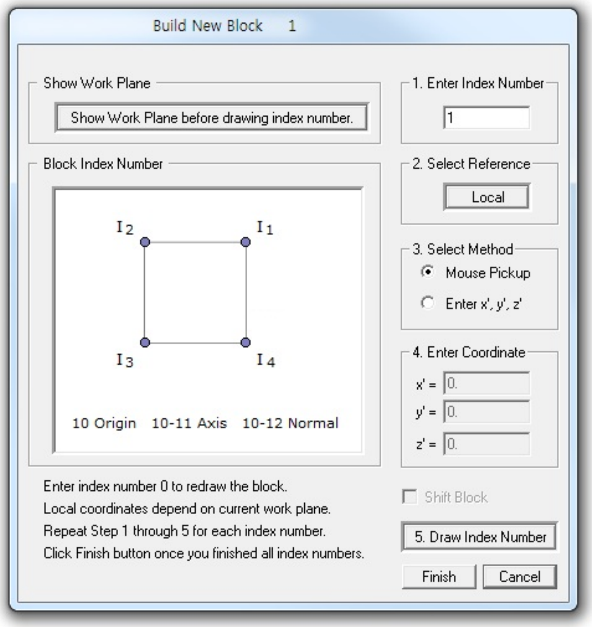


Figure 6.14 Quad block

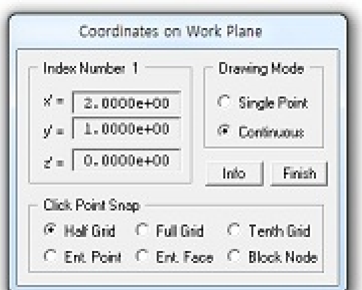


Figure 6.15 Coordinates on work plane

5. Click the points for index numbers for quad block as in Fig. 6.16.

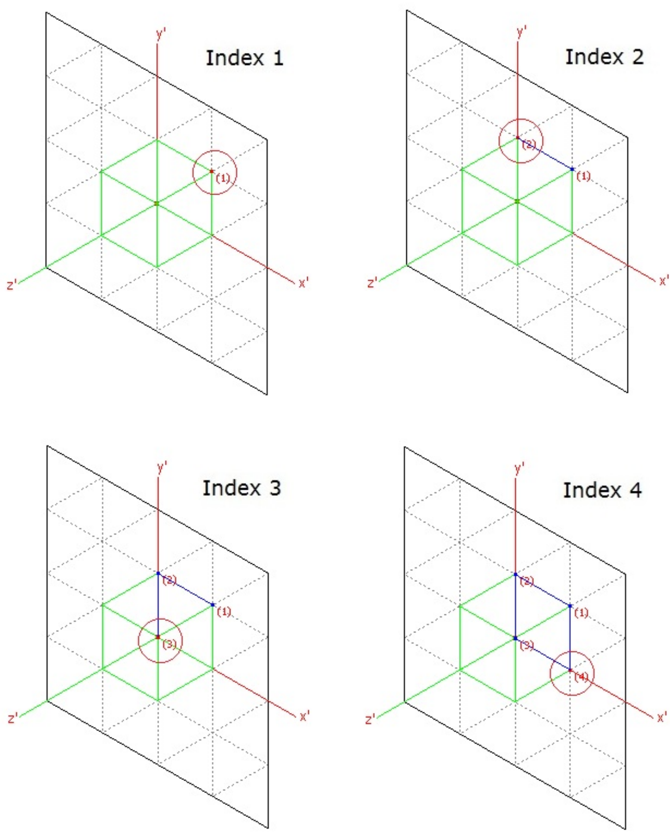


Figure 6.16 Index numbers for quad block

Now, the geometry of quad block is completed.

6. Click Finish in Figure 6.17 and then click Finish in Figure 6.14.

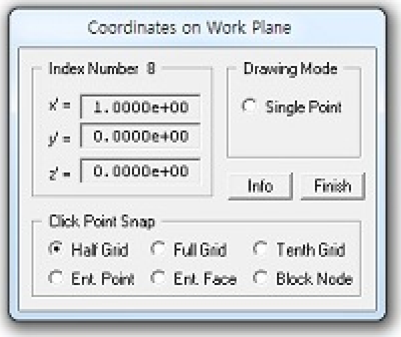


Figure 6.17 Coordinates on work plane

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.

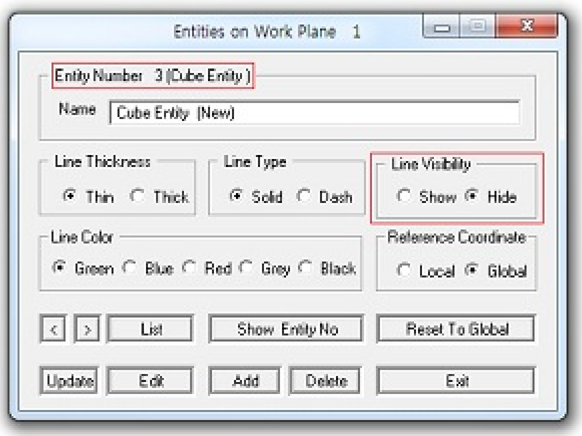


Figure 6.18 Entity editor

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.

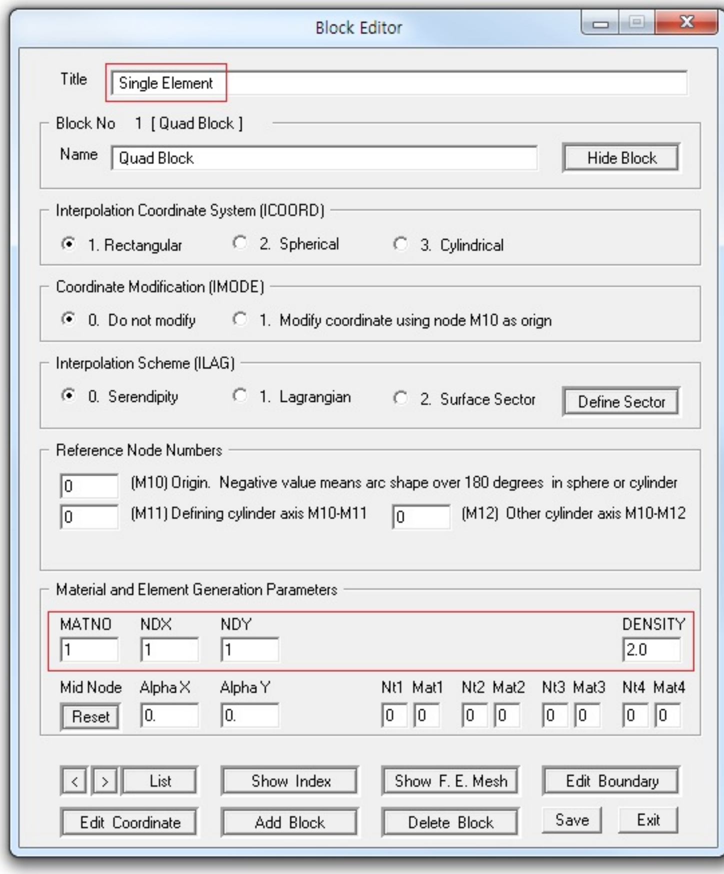


Figure 6.19 Block editor

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.

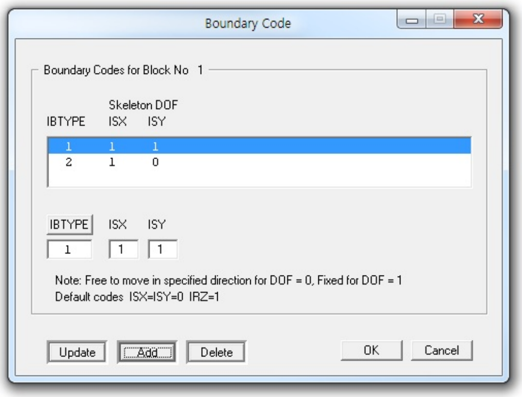


Figure 6.20 Boundary code editor

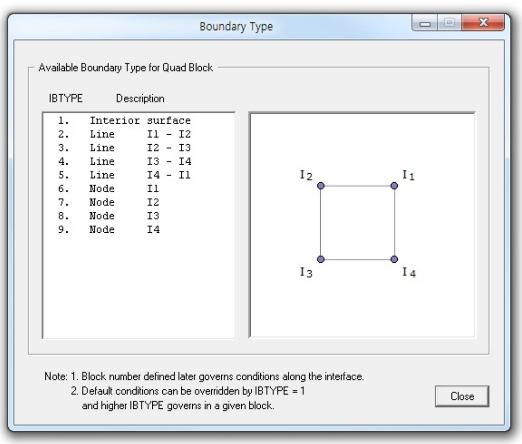


Figure 6.21 Boundary type for quad block

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.

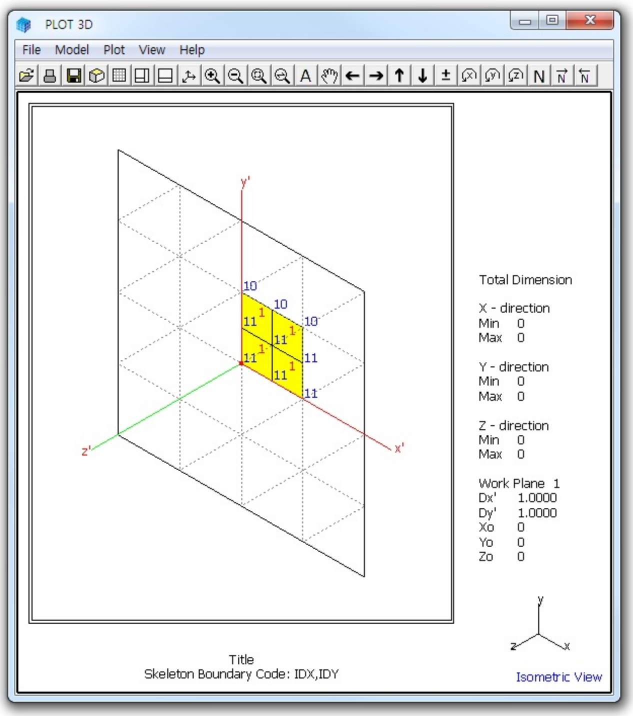


Figure 6.24 Skeleton boundary codes on drawing board

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.

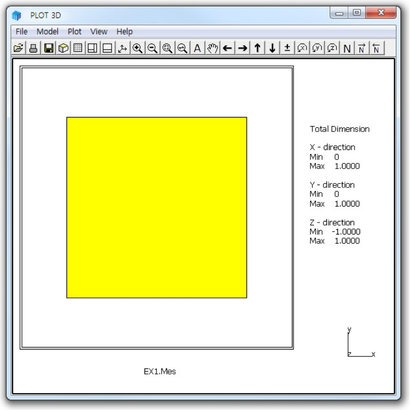


Figure 6.25 Finite element mesh

3. Follow same procedure to plot skeleton boundary code in Step 6.
4. Figure 6.26 shows skeleton boundary code for finite element mesh.

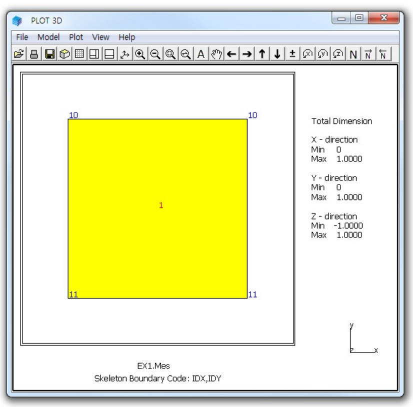


Figure 6.26 Skeleton boundary codes

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

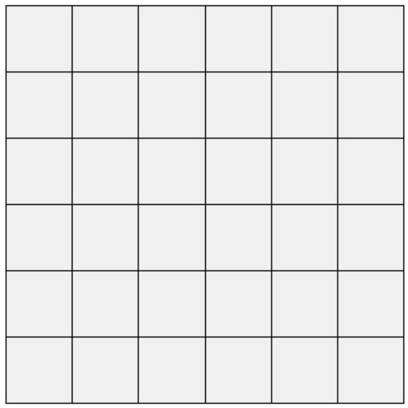


Figure 6.27 Square foundation with constant element size

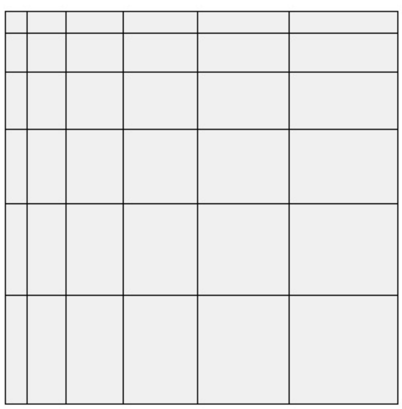


Figure 6.28 Square foundation with variable element size

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.

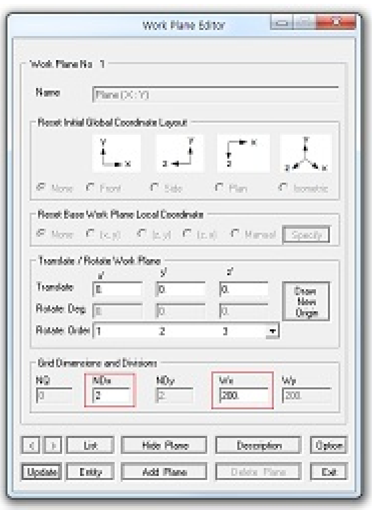


Figure 6.29 Work plane editor

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.

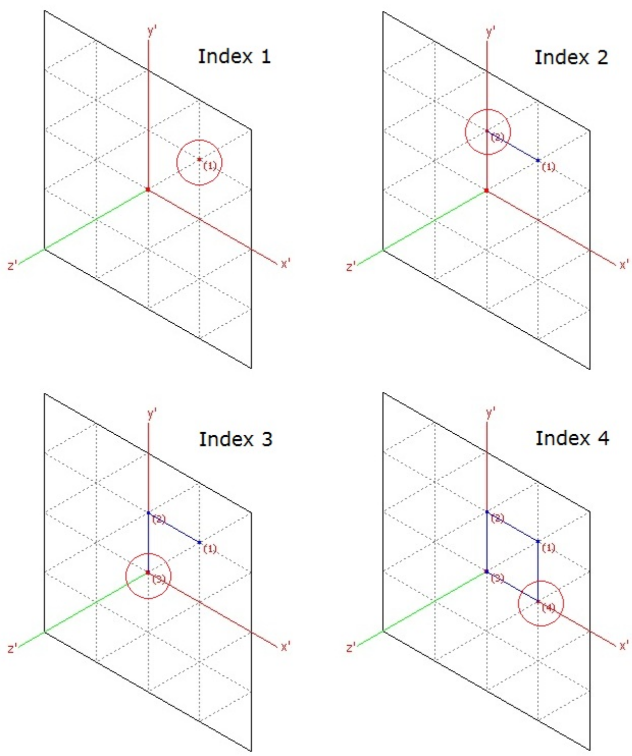


Figure 6.30 Index numbers for quad block

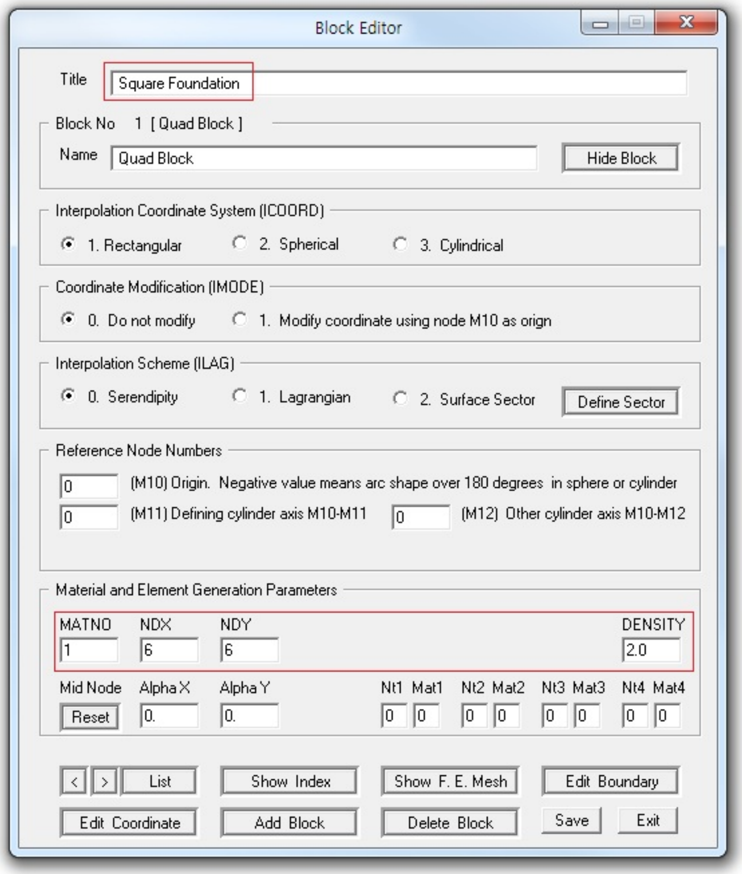


Figure 6.31 Block editor

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.

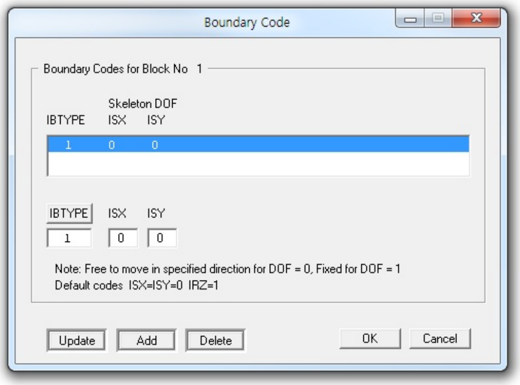


Figure 6.32 Boundary code editor

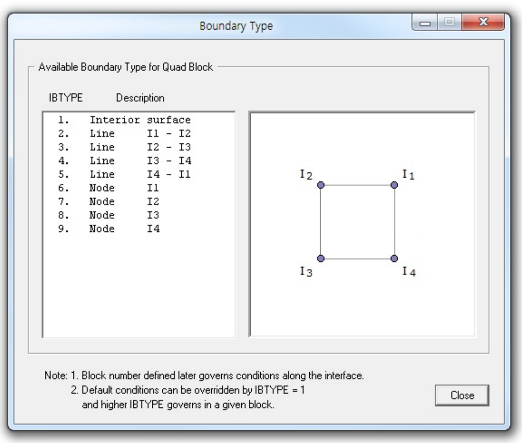


Figure 6.33 Boundary type for quad block

Step 5: Set Global Boundary Code

1. Select Model → Edit Global Boundary in Figure 6.34.

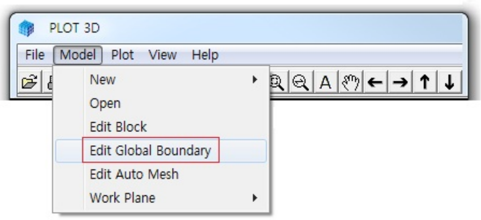


Figure 6.34 Edit global boundary menu

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.

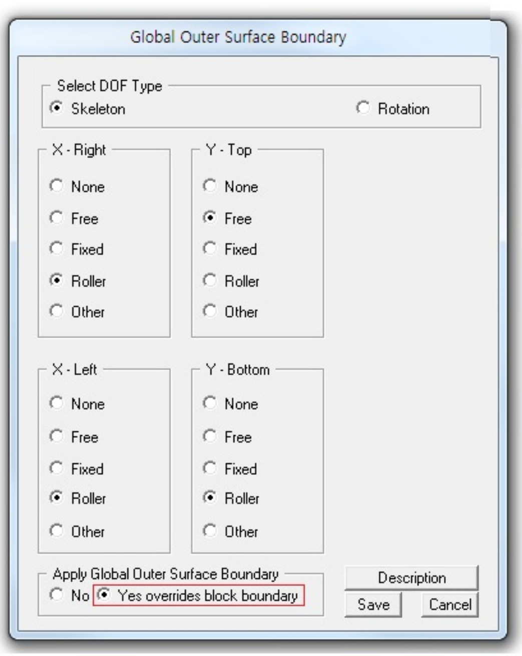


Figure 6.35 Global outer surface boundary

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.

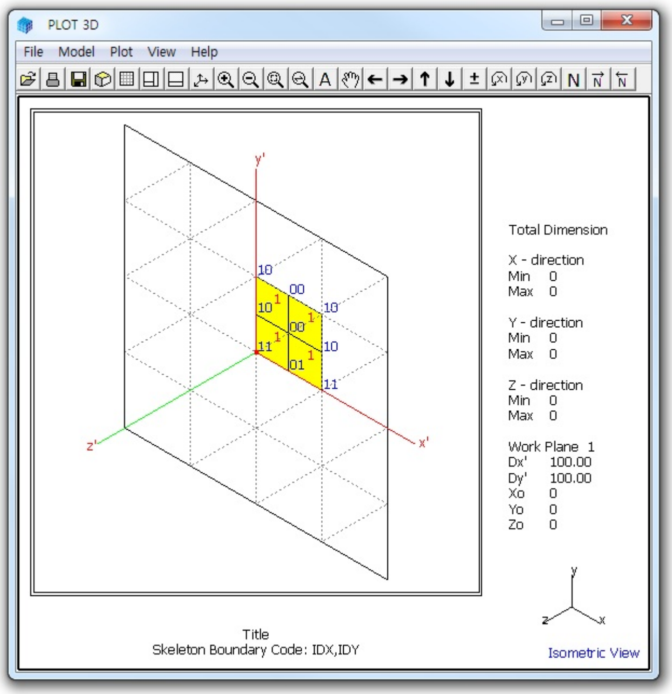


Figure 6.36 Skeleton boundary codes on drawing board

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.

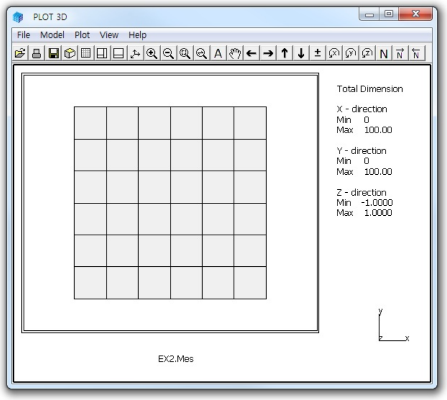


Figure 6.37 Finite element mesh

3. Follow same procedure to plot skeleton boundary code in Step 6.
4. Figure 6.38 shows skeleton boundary code for finite element mesh.

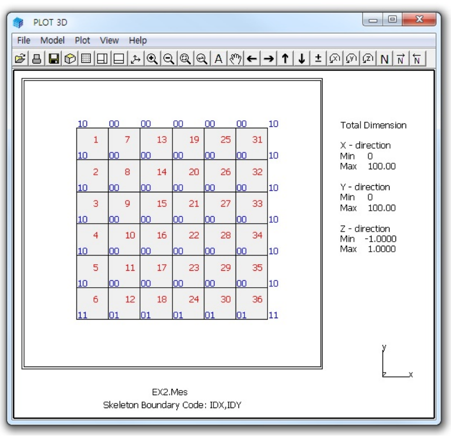


Figure 6.38 Skeleton boundary codes

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.

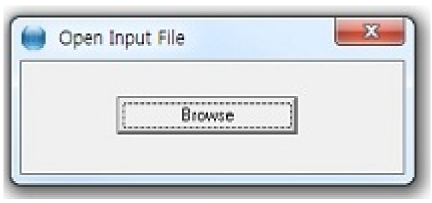


Figure 6.39 Open input file

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.

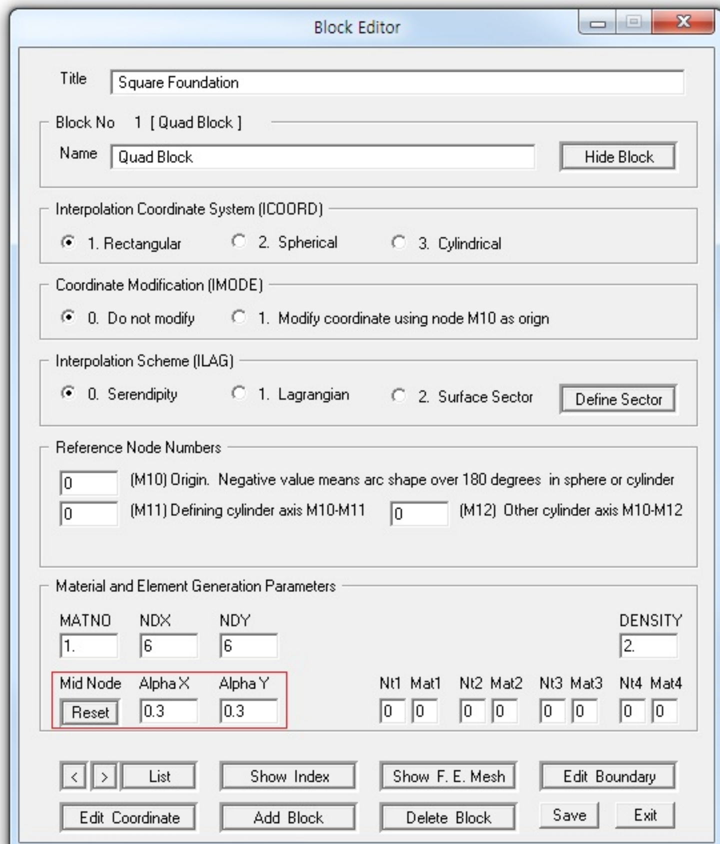


Figure 6.40 Block editor

5. Block mesh is shown in Figure 6.41.

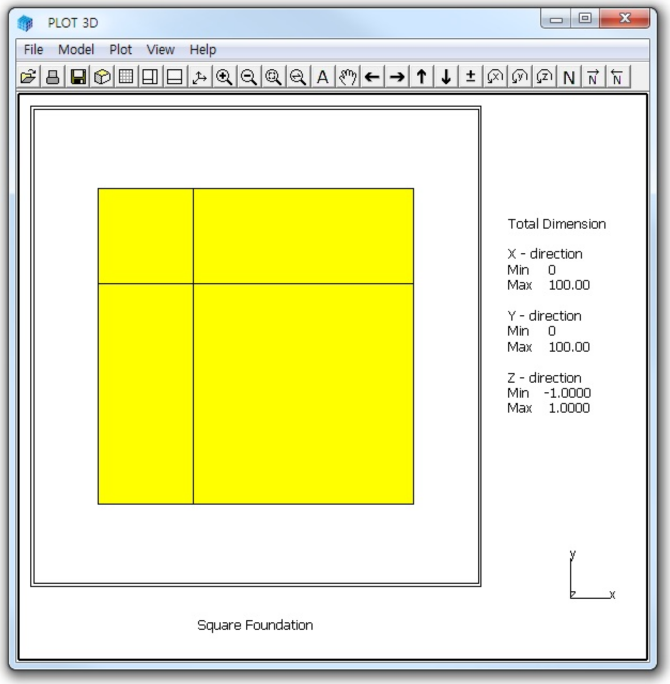


Figure 6.41 Block mesh on drawing board

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.

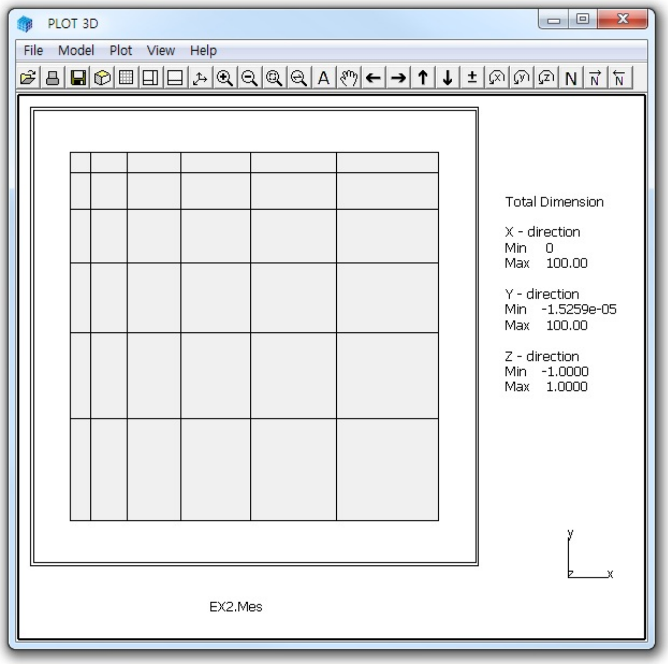


Figure 6.42 Finite element mesh

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-S2 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.

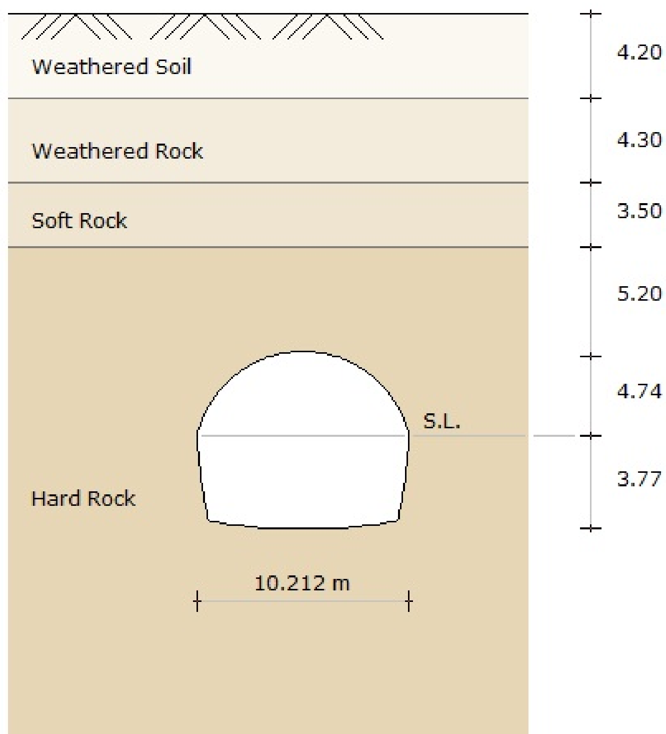
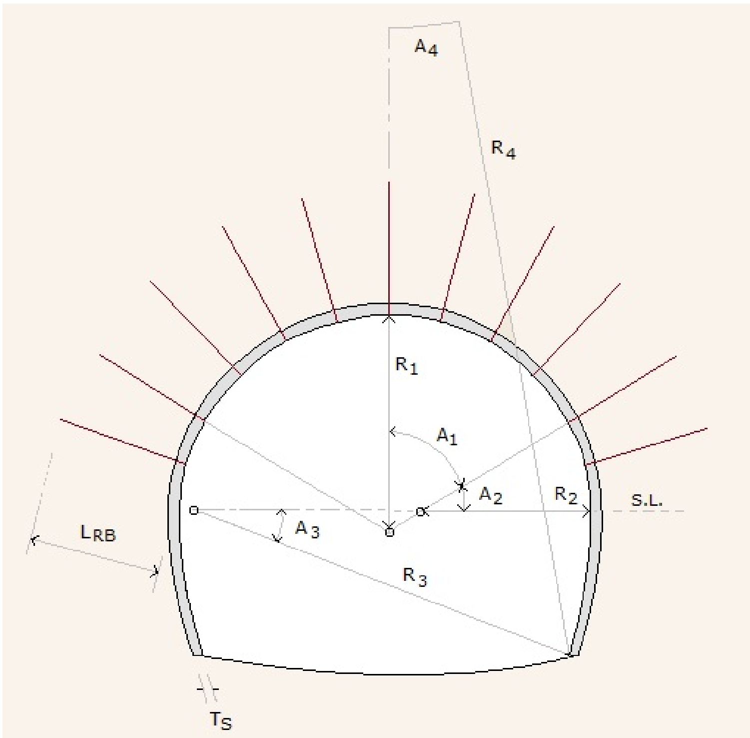


Figure 7.1 Geological condition



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

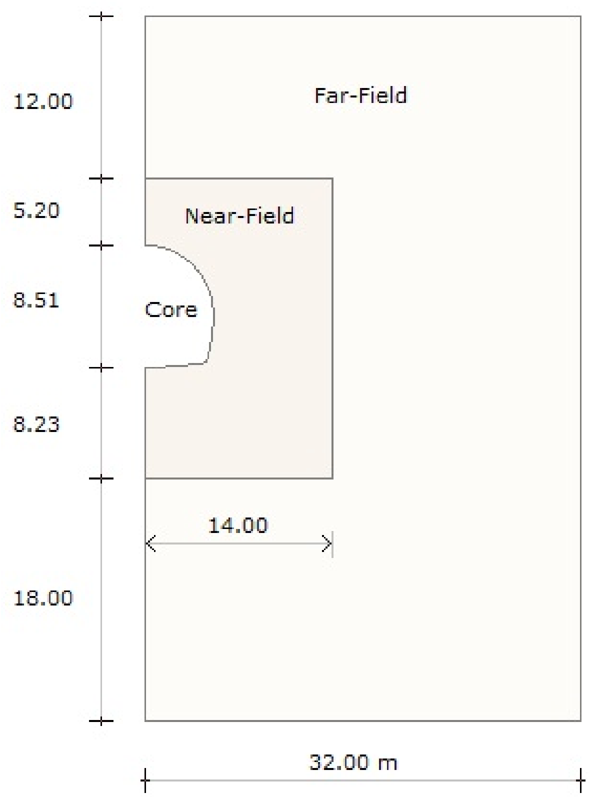


Figure 7.3 Core, Near-field, and Far-field regions

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-S2 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  THICK DENSITY
    4    2   2   1.0   2.55
* 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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  0
* =====================================================
```

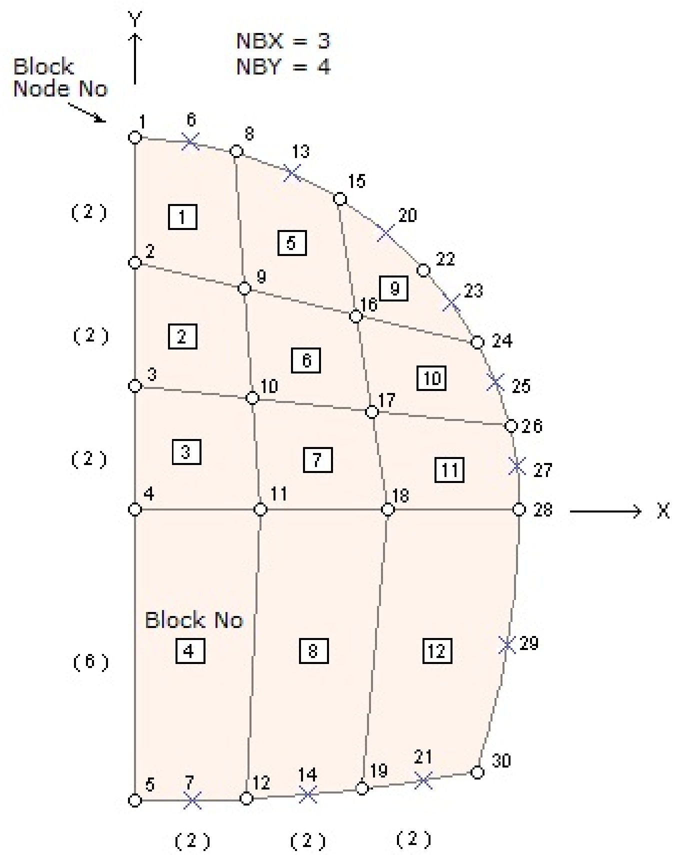


Figure 7.4 Core region block diagram

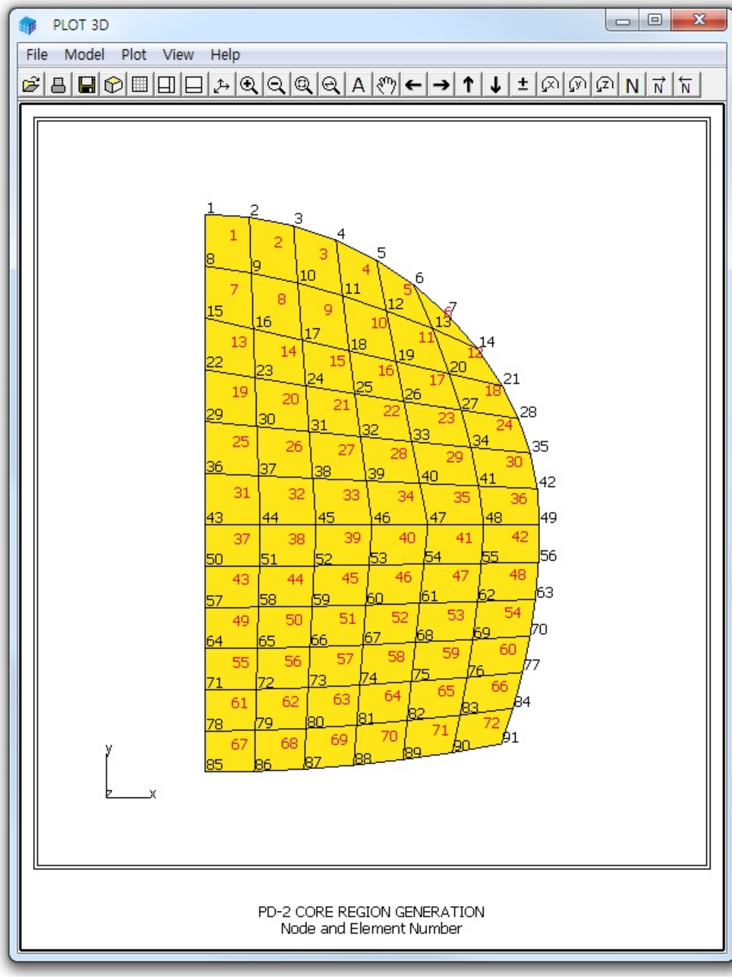


Figure 7.5 Generated element and node numbers for Core region

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  THICK  DENSITY
    1     6    1    1.0    1.9
* 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  1.0   1.9
  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  1.0   2.4
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   1.9
  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  1.0   1.9
  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  1.0   2.4
  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  1.0   2.55
  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  1.0   2.55
  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  1.0   2.55
  0
* ====================================================
* END OF DATA
```

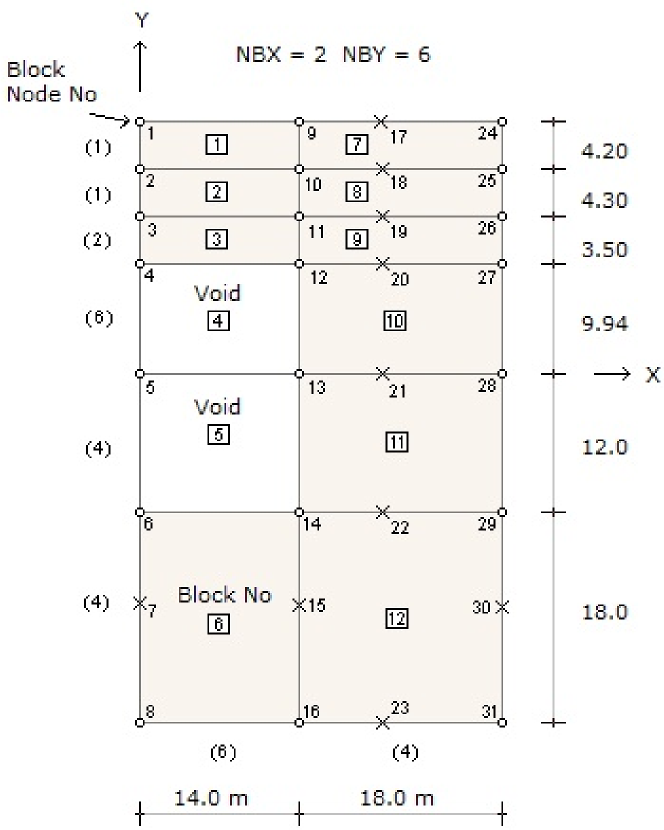


Figure 7.6 Far-field region block diagram

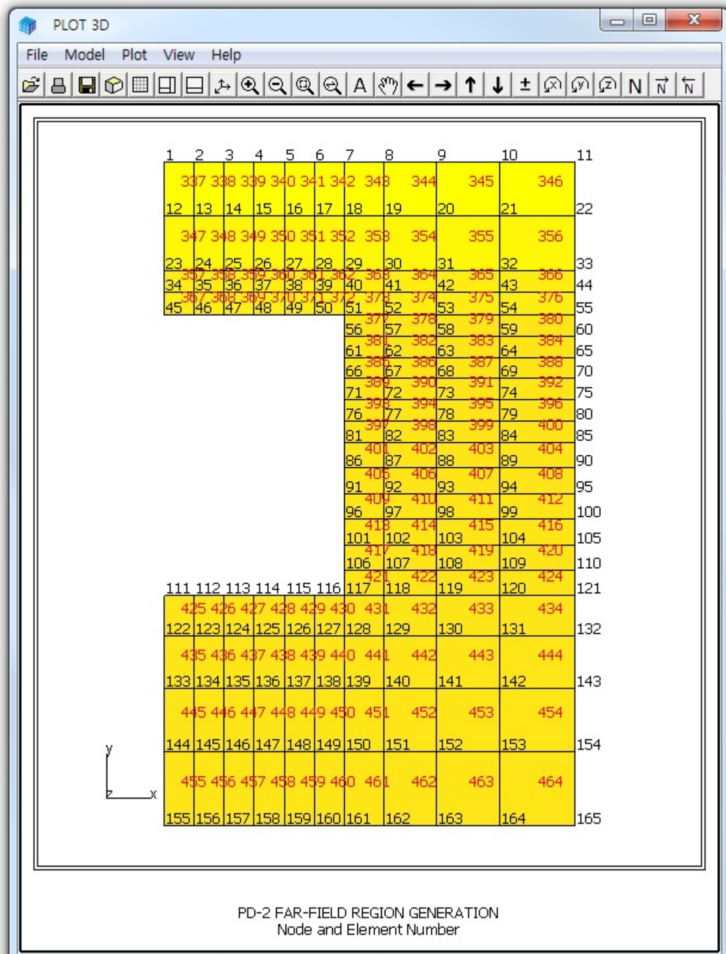


Figure 7.7 Generated element and node numbers for Far-field region

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
* IBb  IBa  IBc  IBd  IBab  IBac  IBcd  Ibbd
   0    1    1    0    0     1     0     0
* CARD 2.7
* MATNO1  DENSITY1
     4    2.55
* MATNO2  DENSITY2
     4    2.55
* MATNO3  DENSITY3
     4    2.55
* 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
      0  0  0  0  0  0  0  0
      4  2.55
      4  2.55
      4  2.55
      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
      0  0  0  0  0  0  0  0
      4  2.55
      4  2.55
      4  2.55
      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
      0  0  0  0  0  0  0  0
      4  2.55
      4  2.55
      4  2.55
      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
      0  0  0  0  0  0  0  0
      4  2.55
      4  2.55
      4  2.55
      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
      0  0  0  0  0  0  0  0
      4  2.55
      4  2.55
      4  2.55
      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
      0  0  0  0  0  0  0  0
      4  2.55
      4  2.55
      4  2.55
      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
      1  0  0  1  0  0  0  1
      4  2.55
      4  2.55
      4  2.55
      0
* =====================================================
```

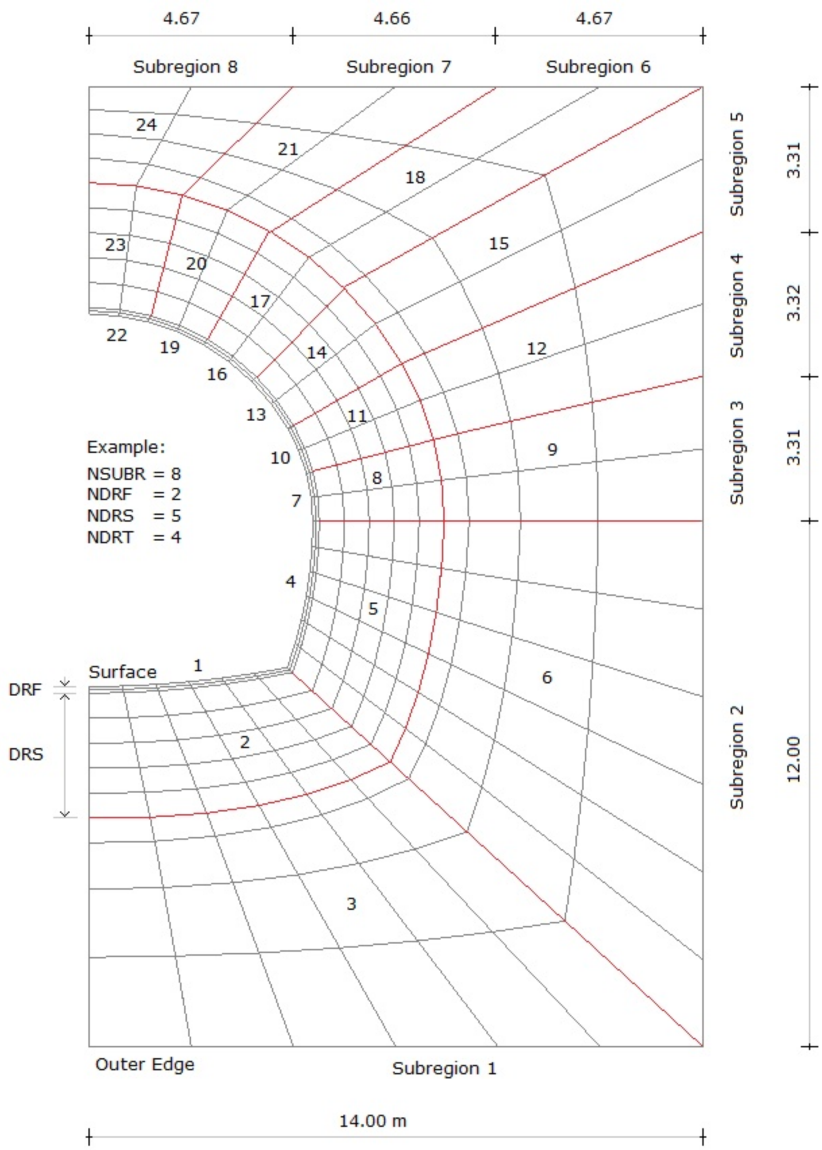


Figure 7.8 Subregions in Near-field region

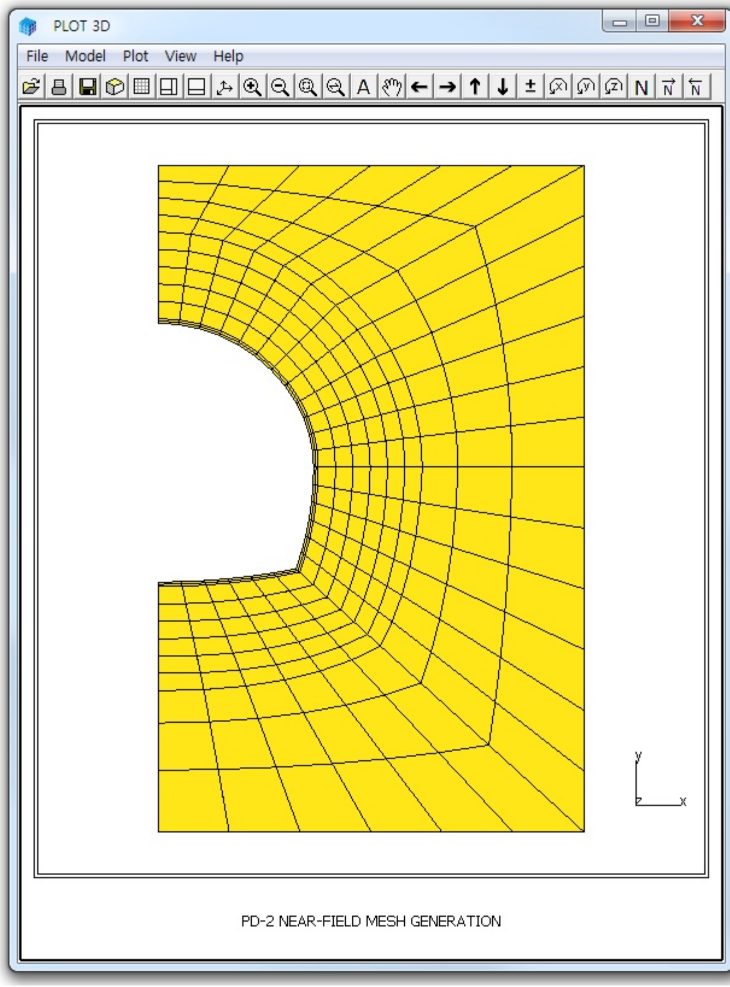


Figure 7.9 Generated mesh for Near-field region

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.

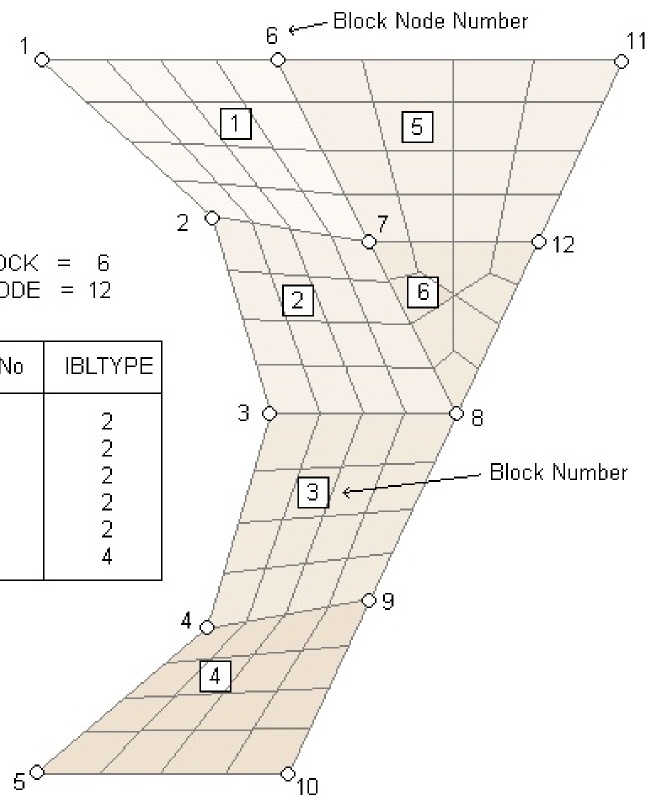
Figure 7.10 shows block diagram for Model 3 example problem. Block numbers 1 to 5 are 4 x 4 rectangular shape and Block number 6 is the 9-element triangular shape.

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
* 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  DENSITY
    1       2       2     0.0018
* 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  DENSITY
    2       2       2     0.0018
* 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  DENSITY
    3       2       2     0.0018
* 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  DENSITY
    4       2       2     0.0018
* 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  DENSITY
    5       2       2     0.0018
* 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  DENSITY
    6       4       2     0.0018
* 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
* =======================================================
* END OF DATA
```



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

Figure 7.10 Block diagram for Model 3 example problem

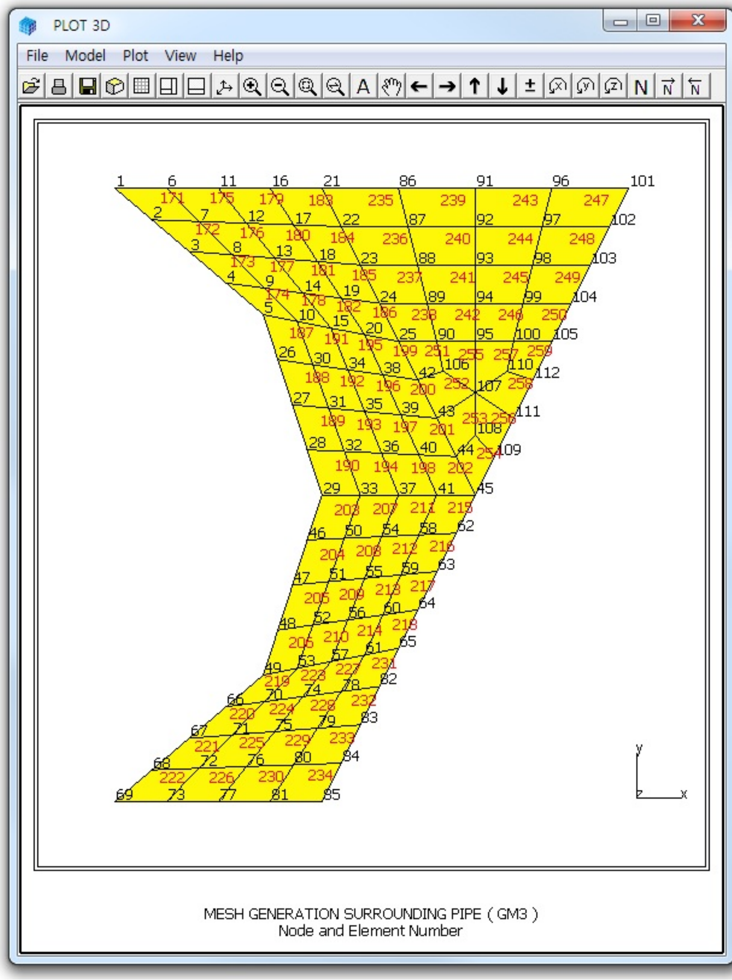


Figure 7.11 Generated element and node numbers for Model 3 example problem

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  DENSITY
  3      2.3
* END OF DATA
```

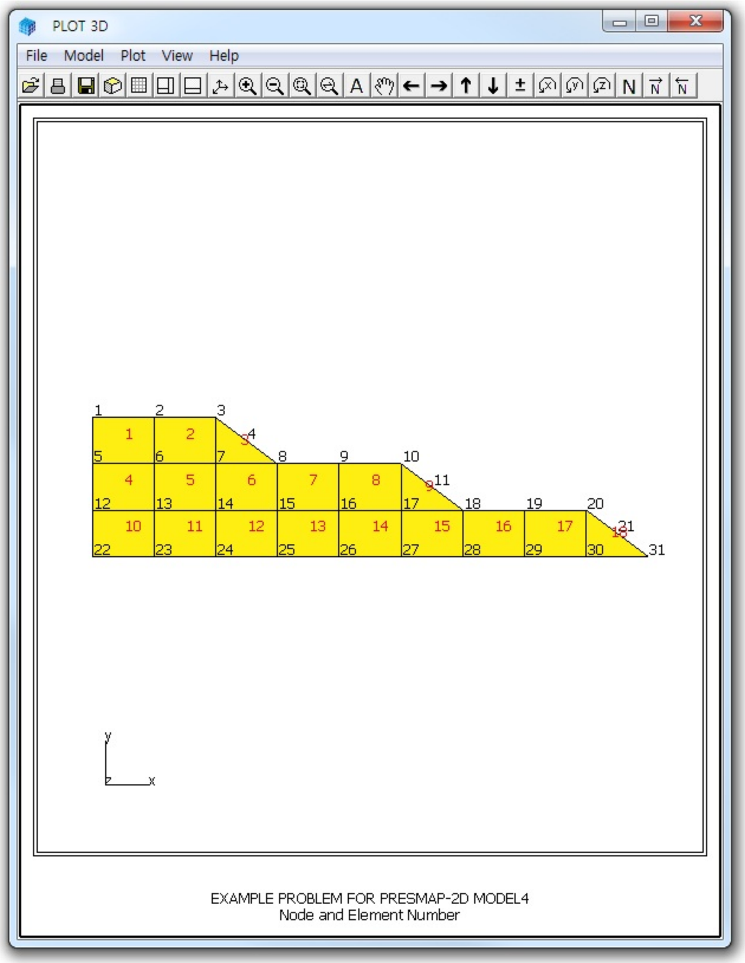


Figure 7.12 Generated element and node numbers for Model 4 example problem

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:

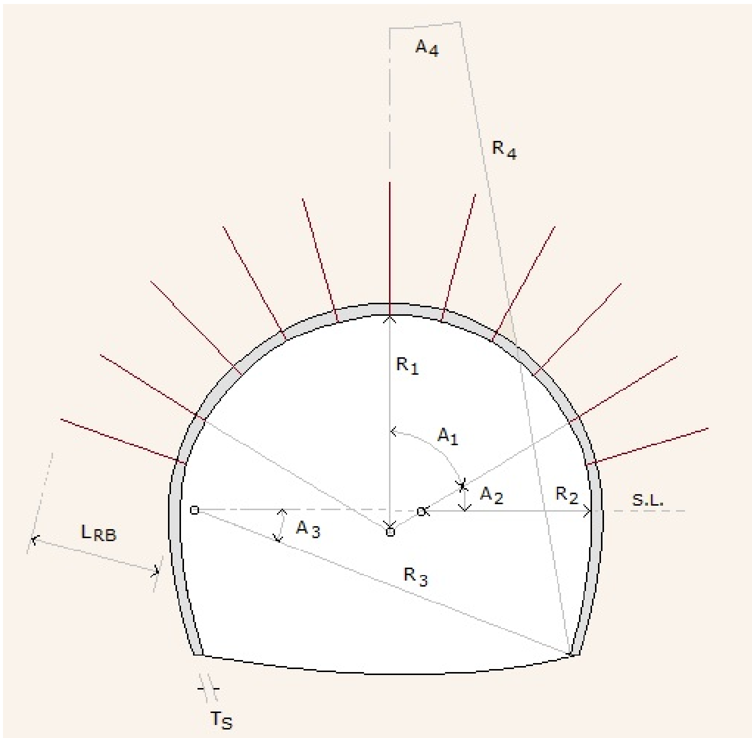
Output File	Mesh File including all elements (Continuum, Beam, and Truss). Output File is the user specified name.
BEAM.Dat	Mesh File including only beam elements.
TRUSS.Dat	Mesh File including only truss elements.
AD.Dat	Card Group 8 in Main File representing default element activities for upper and 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     GAMA
   1        4.2   1.9
   2        4.3   2.2
   3        3.5   2.2
   4        39.94 2.4
* 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   TL
  0        0.3  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
```

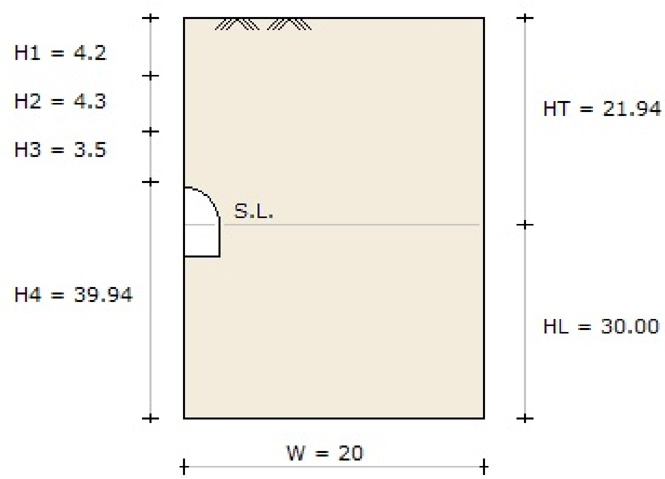


R_1	=	5.24 M	A_1 =	60°
R_2	=	4.24 M	A_2 =	30°
R_3	=	9.86 M	A_3 =	19.781°
R_4	=	23.86 M		

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

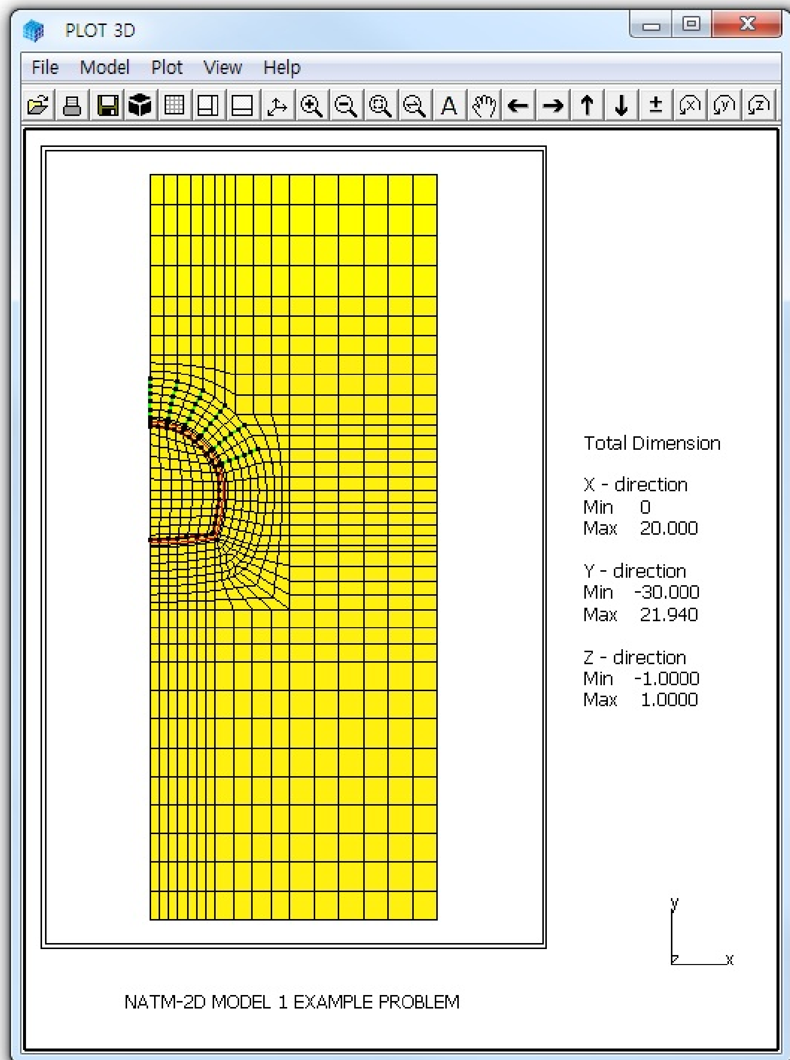


Figure 7.15 Generated finite element mesh for Model 1 example

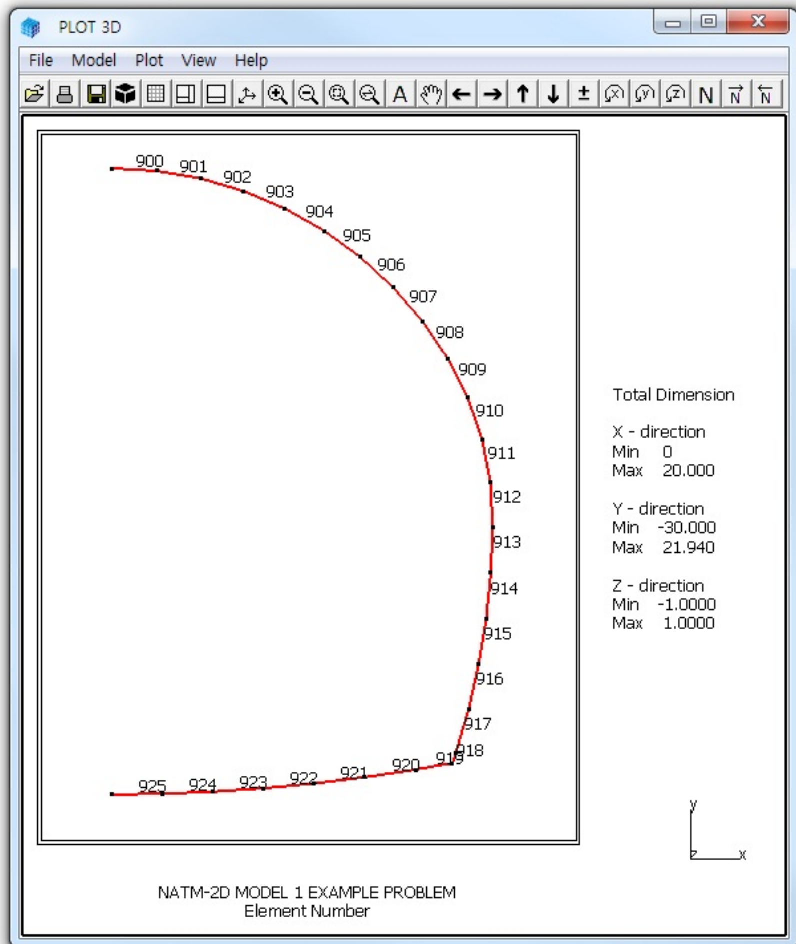


Figure 7.16 Generated beam element number for Model 1 example

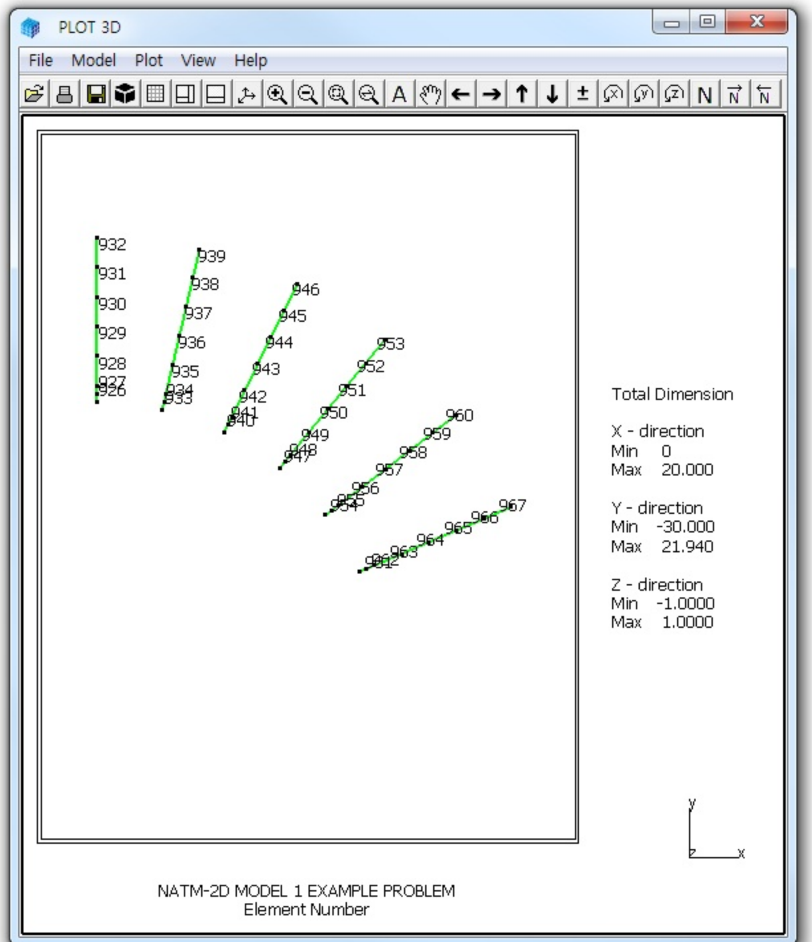
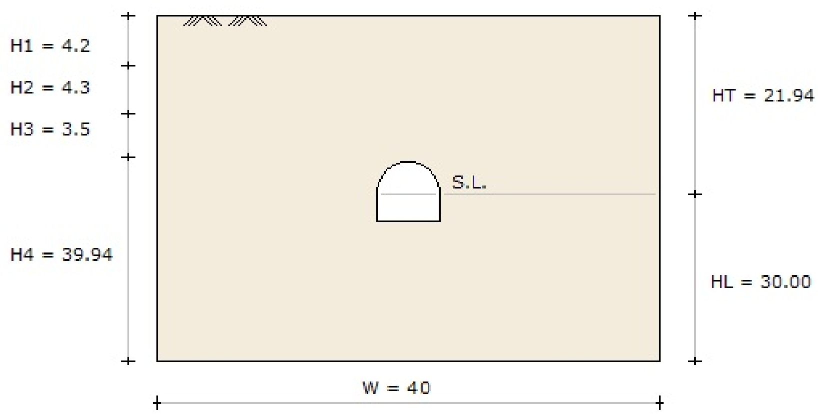


Figure 7.17 Generated truss element number for Model 1 example

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     GAMA
   1         4.2   1.9
   2         4.3   2.2
   3         3.5   2.2
   4         39.94 2.4
* 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   TL
  0        0.3  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

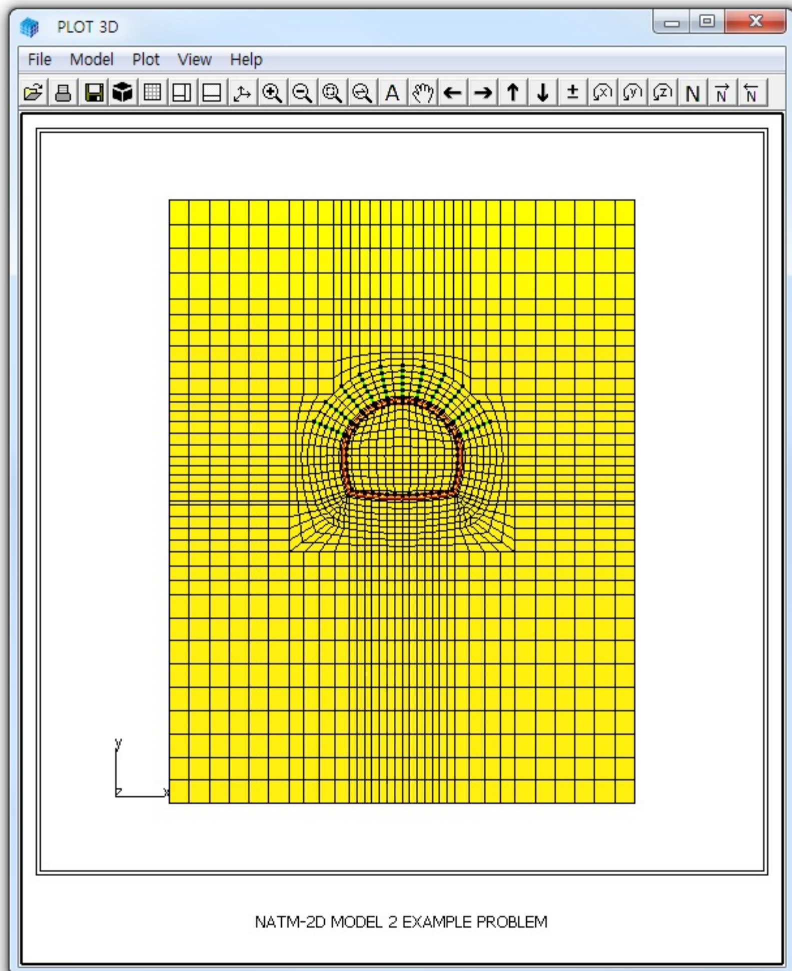
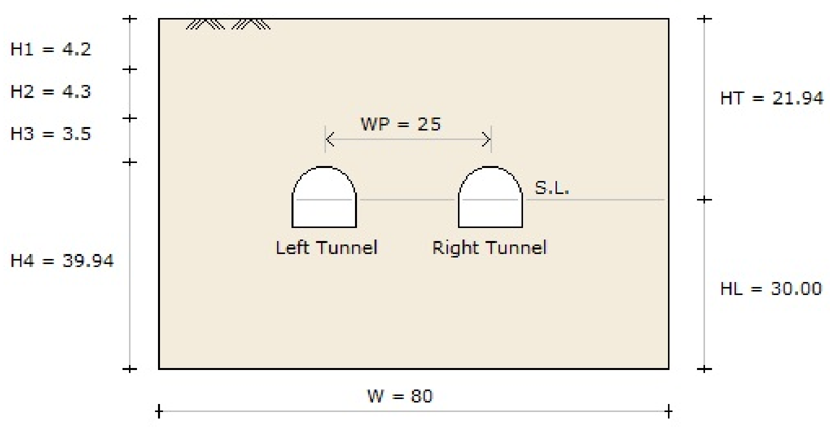


Figure 7.19 Generated finite element mesh for Model 2 example

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     GAMA
   1        4.2   1.9
   2        4.3   2.2
   3        3.5   2.2
   4        39.94 2.4
* 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

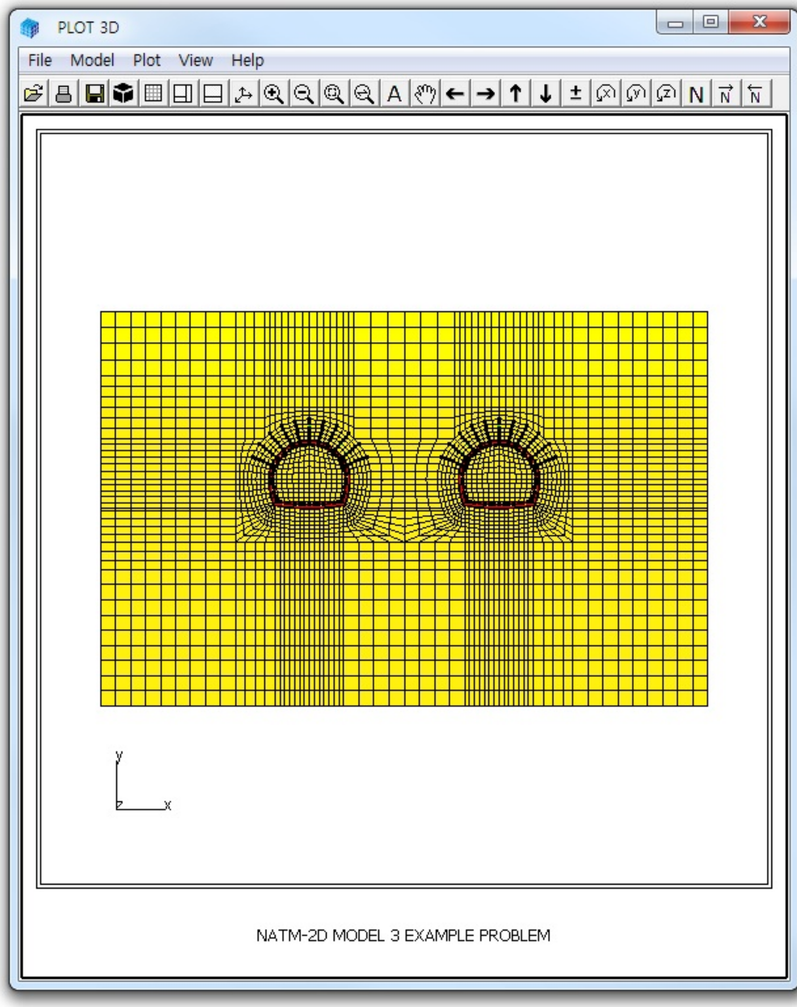
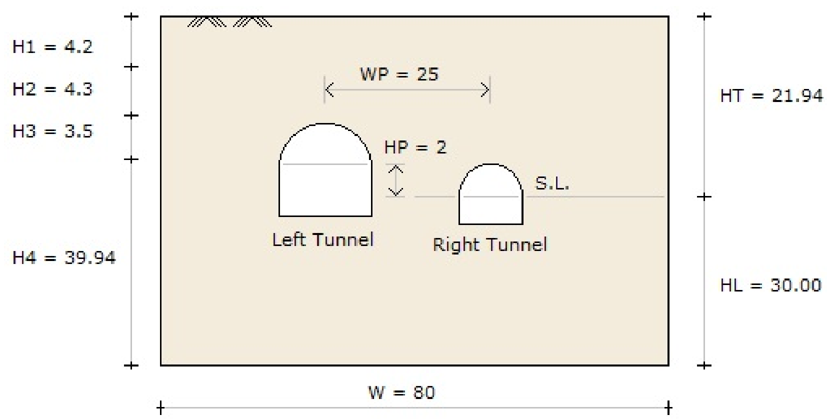


Figure 7.21 Generated finite element mesh for Model 3 example

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     GAMA
   1        4.2   1.9
   2        4.3   2.2
   3        3.5   2.2
   4        39.94 2.4
* 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

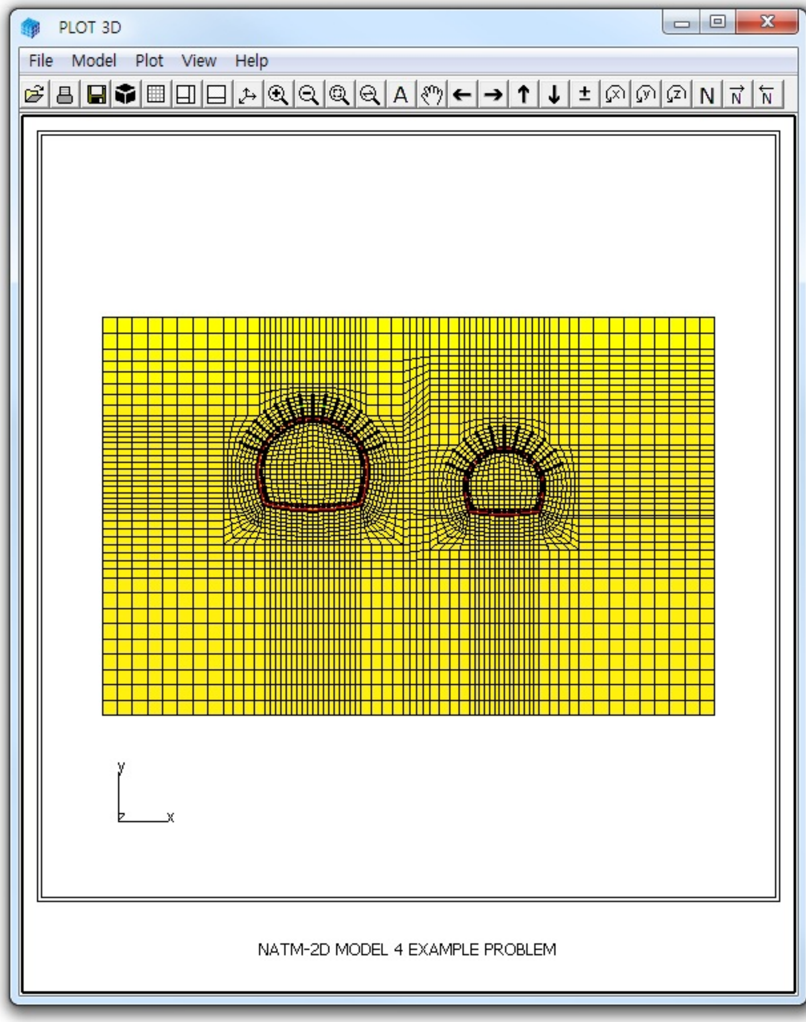


Figure 7.23 Generated finite element mesh for Model 4 example

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     GAMA
   1        4.2   1.9
   2        4.3   2.2
   3        3.5   2.2
   4        39.94 2.4
* 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
```

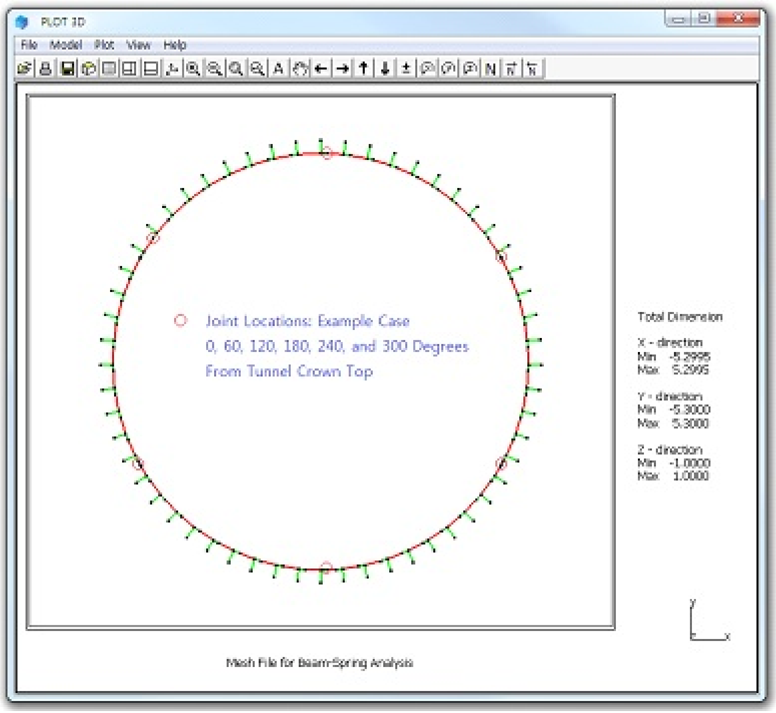


Figure 7.24 Finite element mesh for Model 2-1

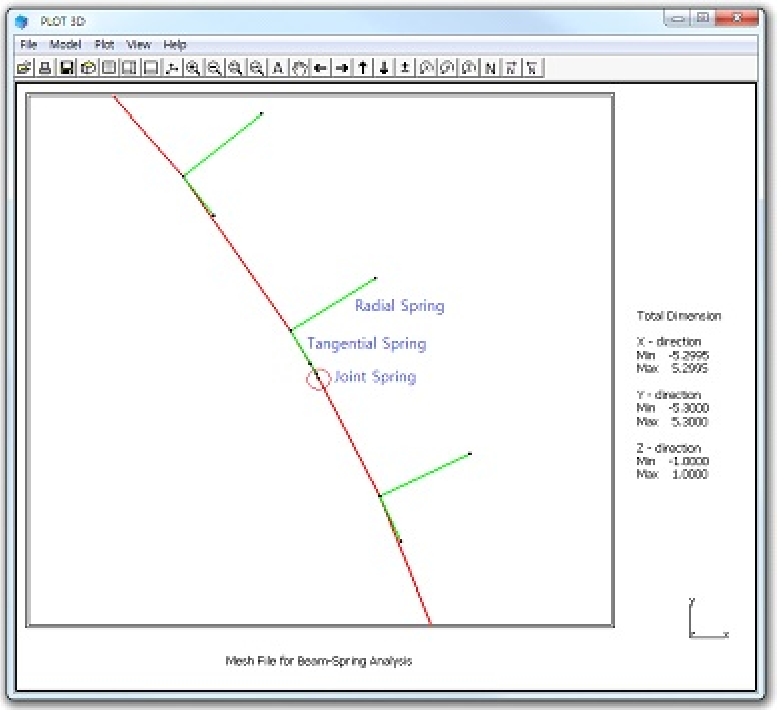


Figure 7.25 Detailed mesh around joint spring element

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

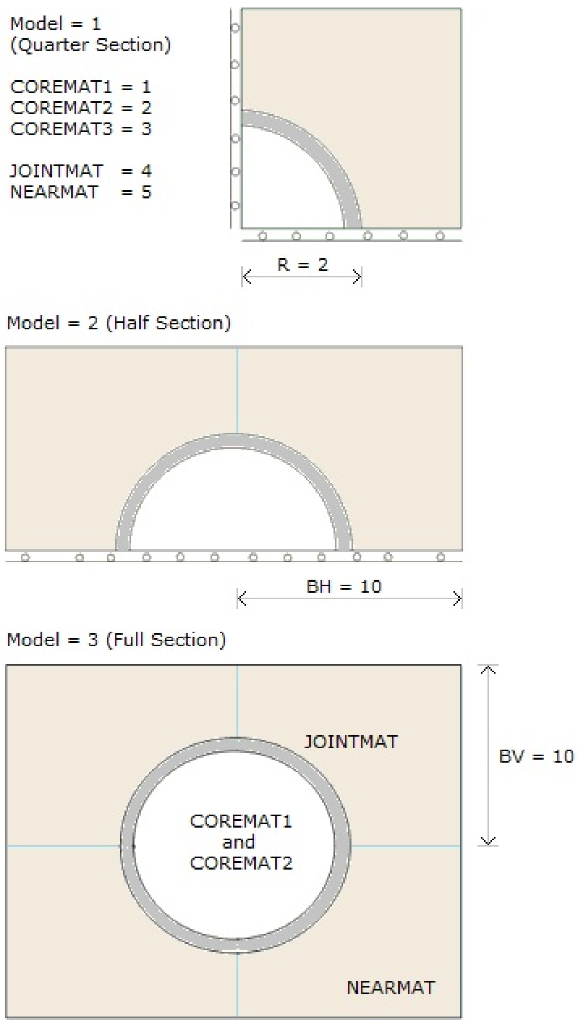


Figure 7.26 Schematic section views for CIRCLE-2D examples

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
```

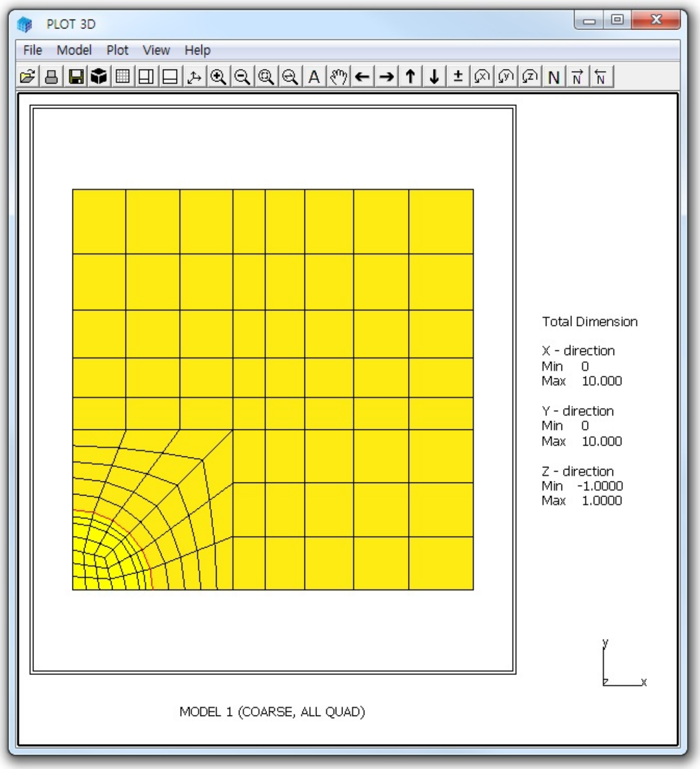


Figure 7.27 Generated finite element mesh for MODEL = 1

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
```

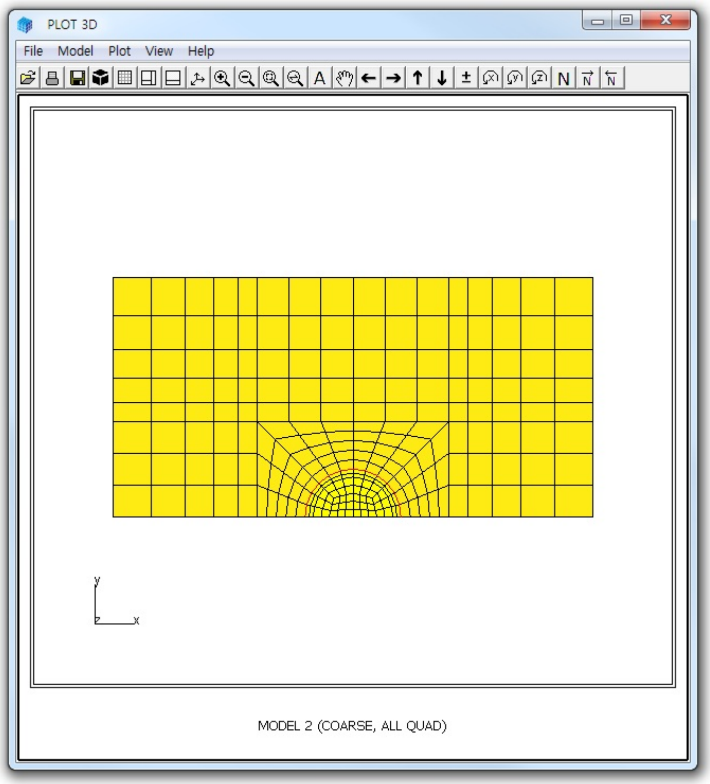


Figure 7.28 Generated finite element mesh for MODEL = 2

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
```

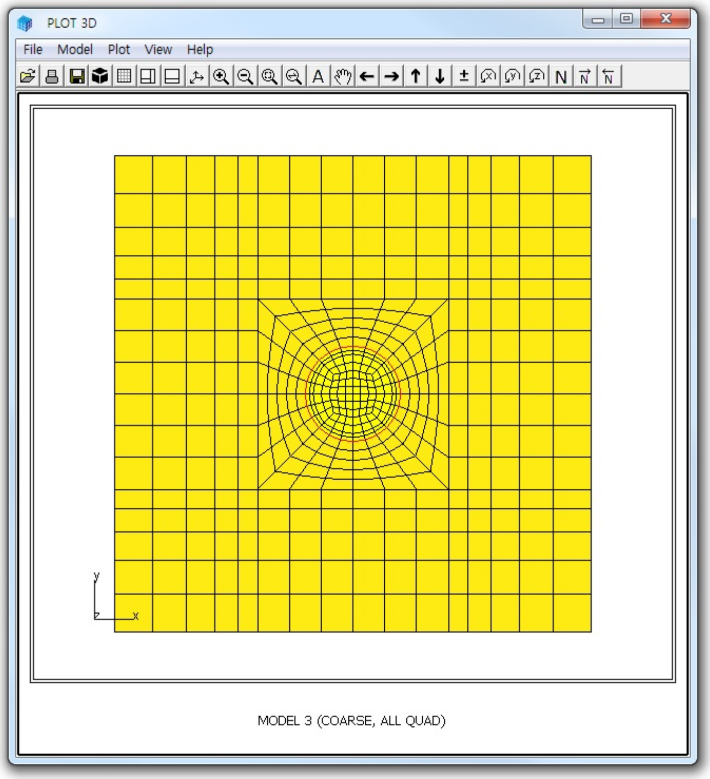


Figure 7.29 Generated finite element mesh for MODEL = 3

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 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         1         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
* THIC   DENSITY
  1.0    2.3
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
* THIC    DENSITY
  0.5     1.8
EndBlock
*========================================================================
EndOfLastBlock
```

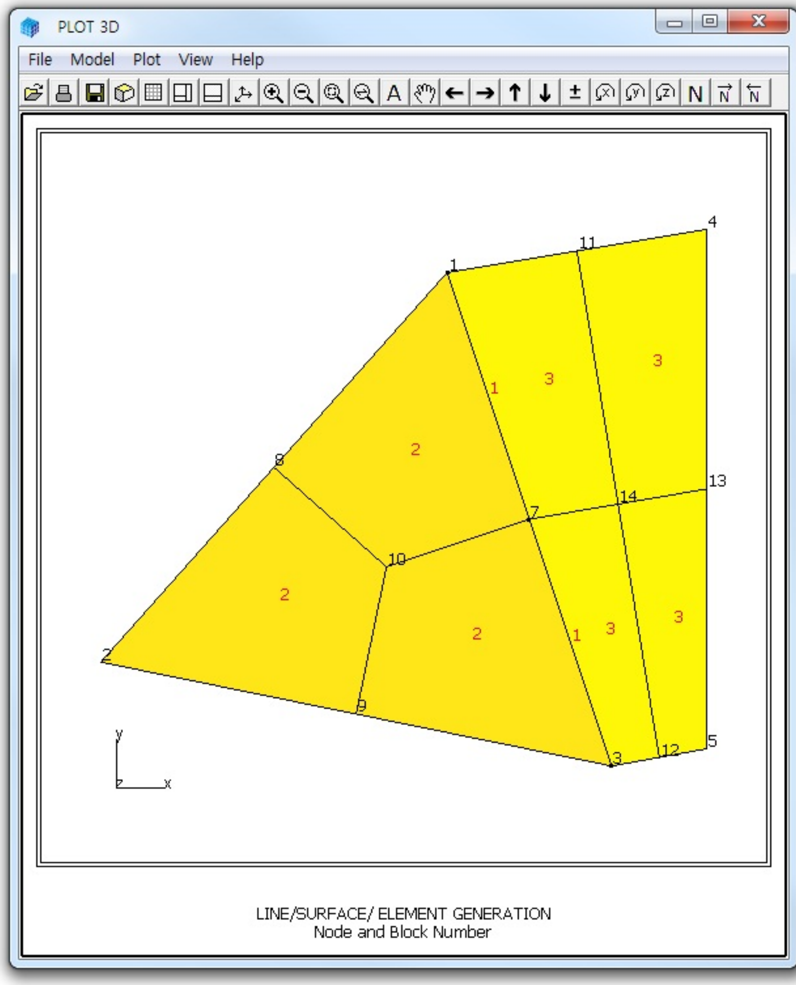


Figure 7.56 Node and block numbers for Example 1

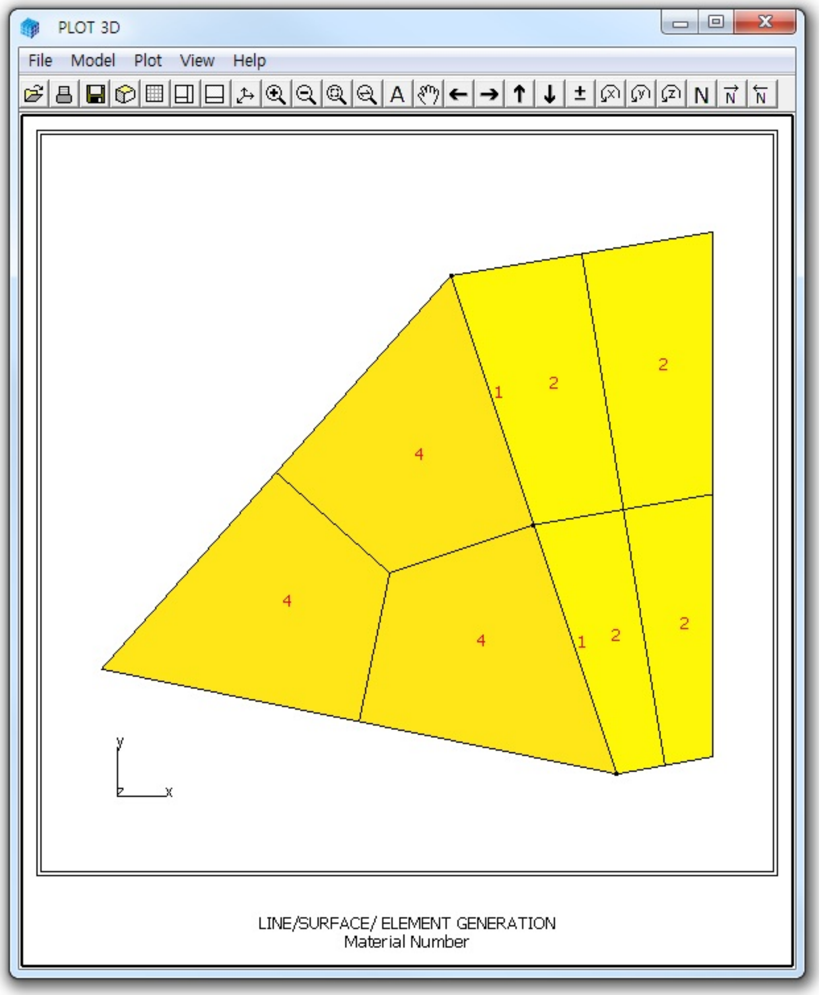


Figure 7.57 Material numbers

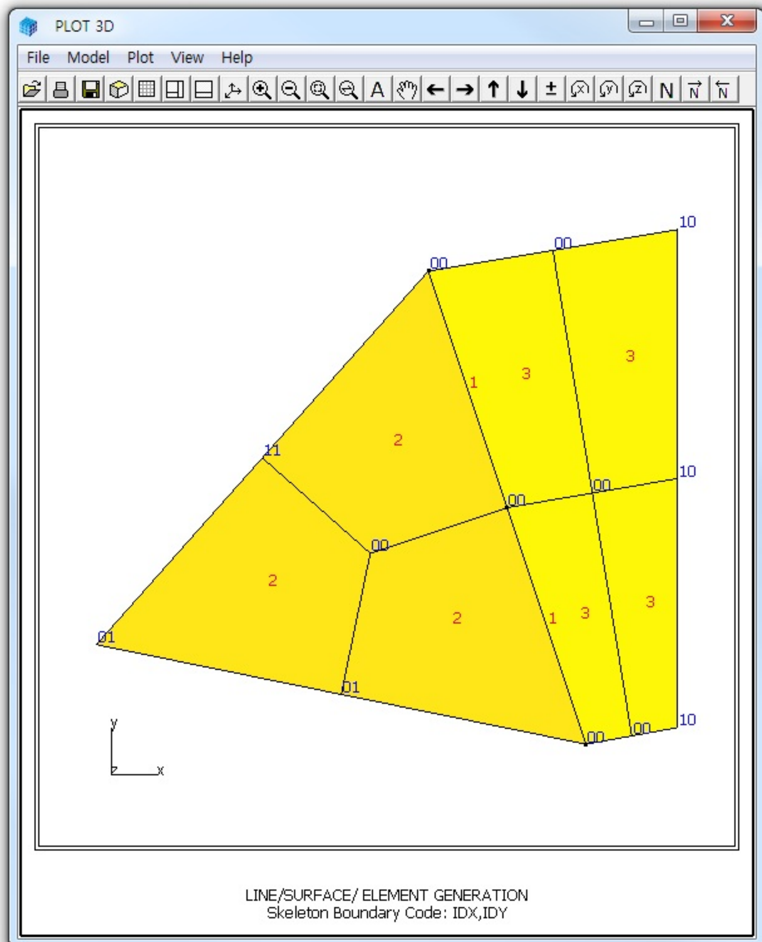


Figure 7.58 Skeleton boundary codes

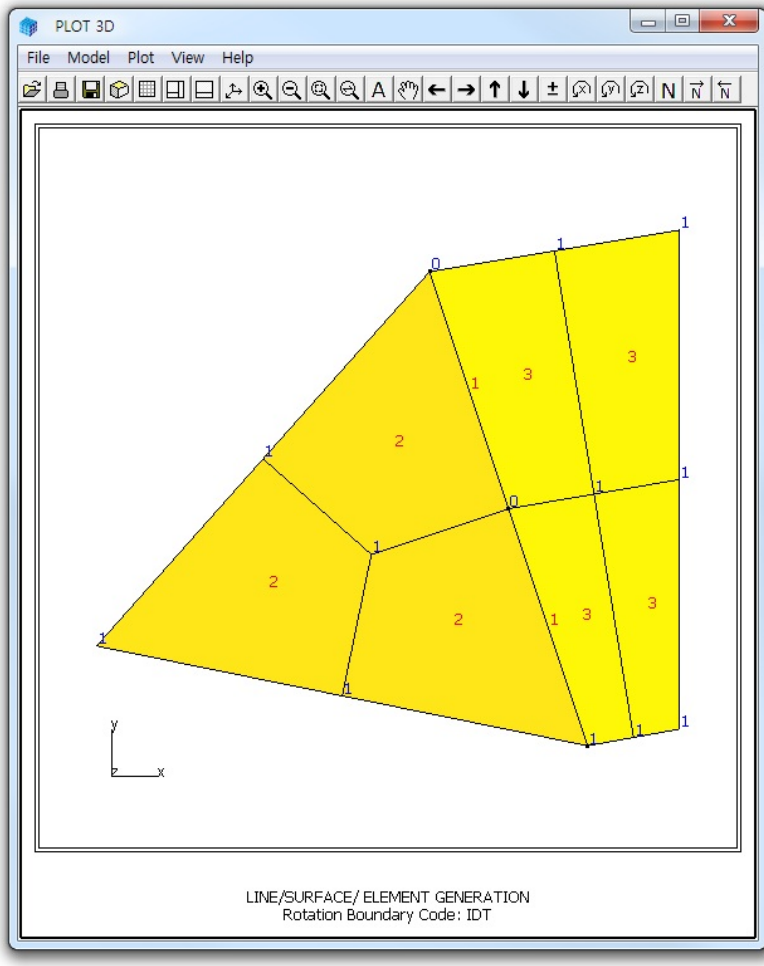


Figure 7.59 Rotation boundary codes

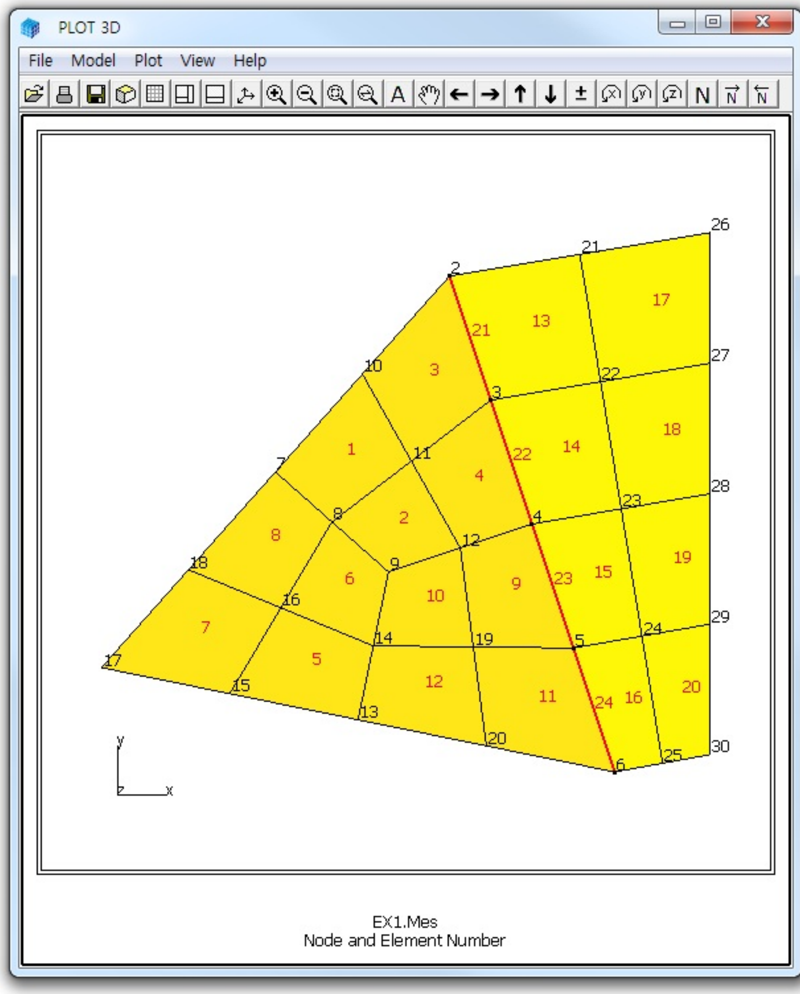


Figure 7.60 Node and element numbers

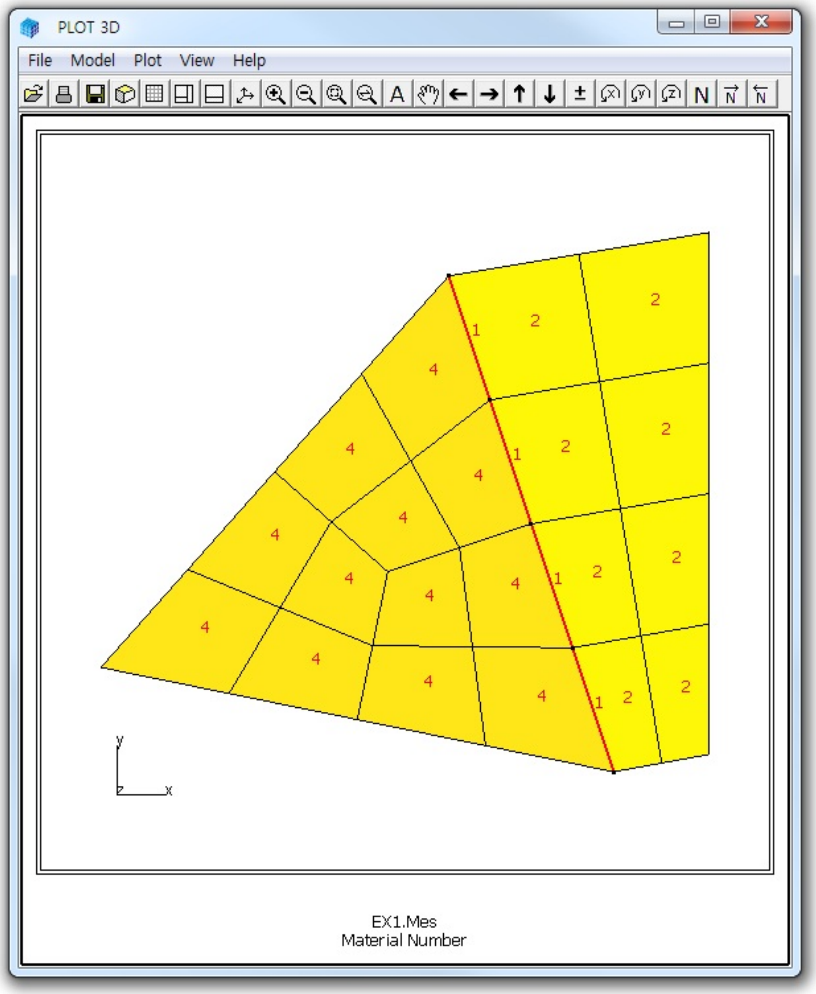


Figure 7.61 Material numbers

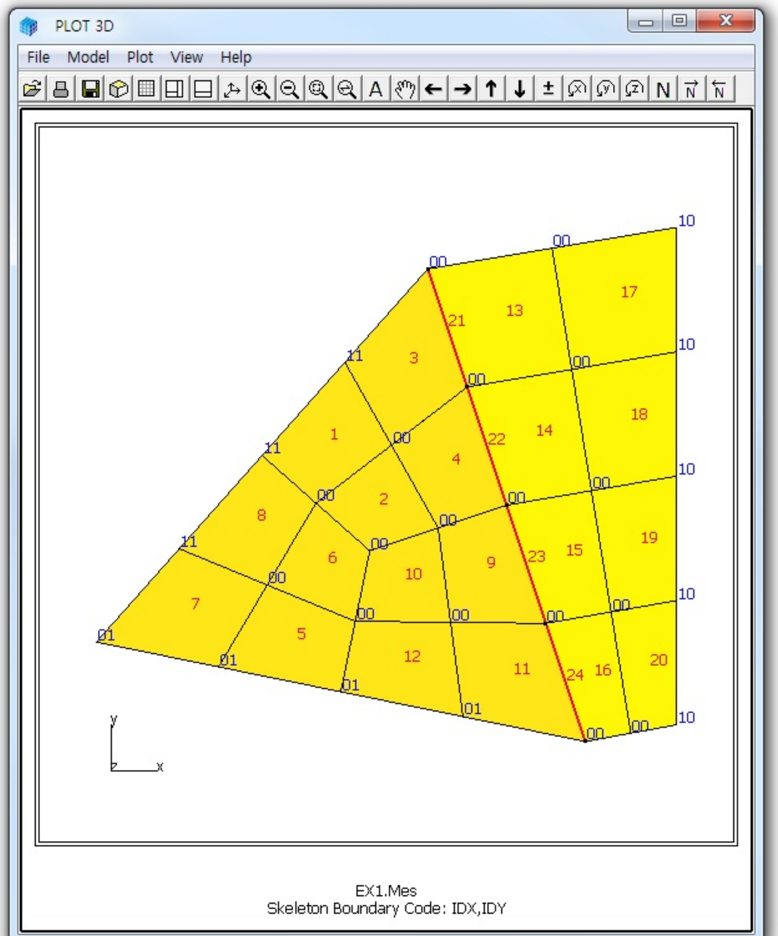


Figure 7.62 Skeleton boundary codes

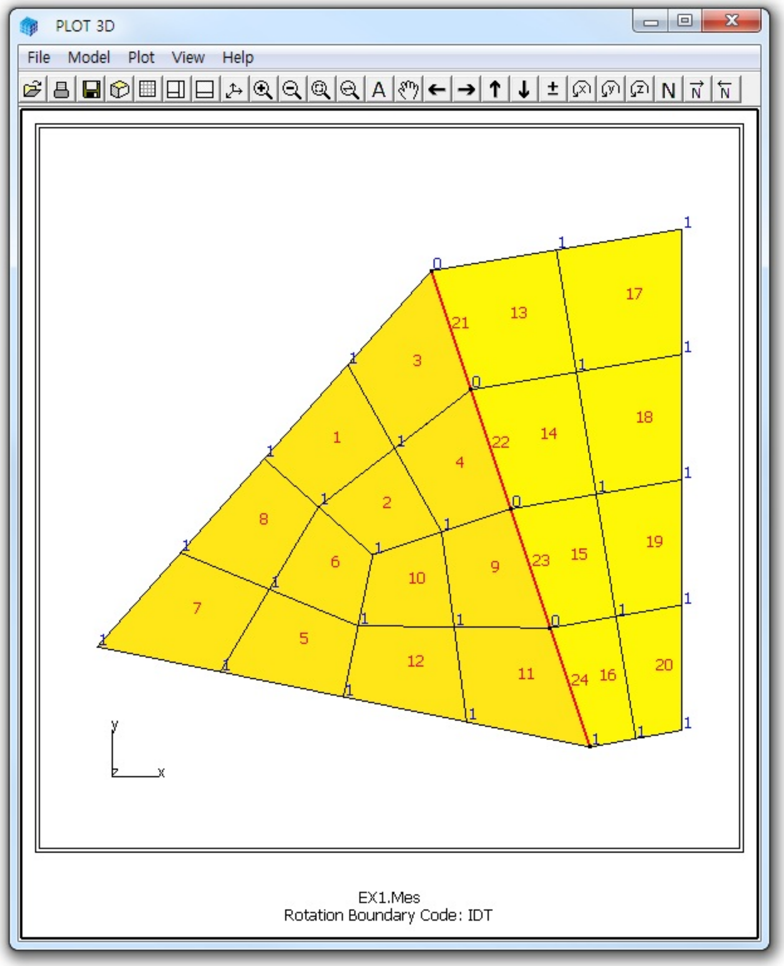


Figure 7.63 Rotation boundary codes

7.4.2 Example 2: Surface with Corner Triangles

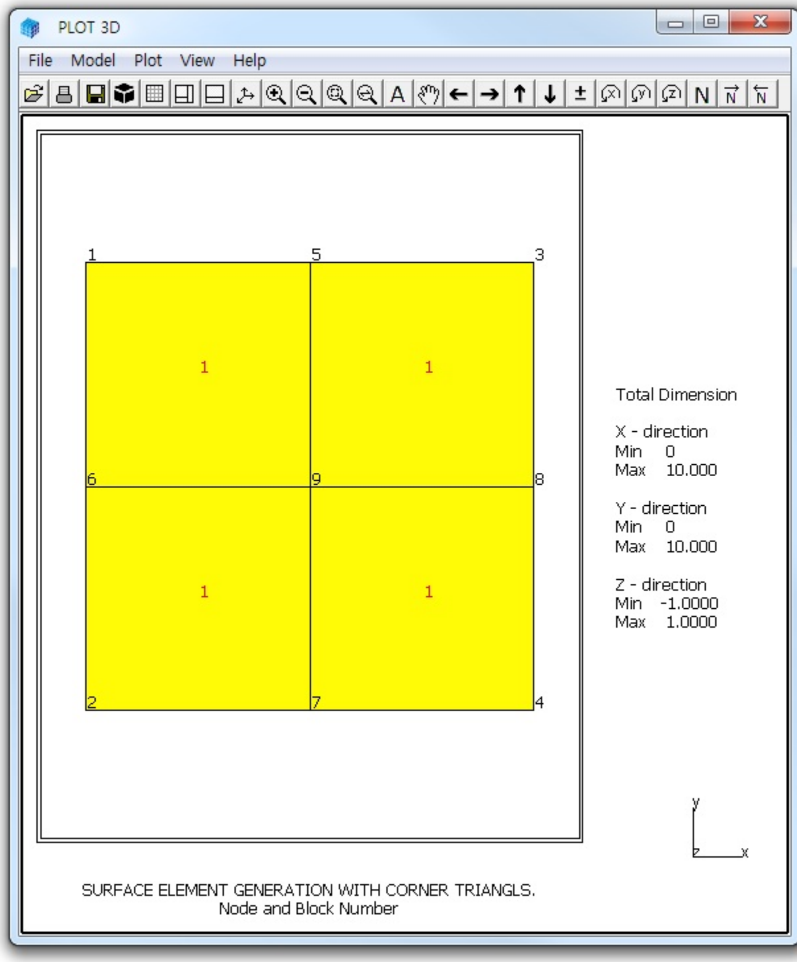


Figure 7.64 Block mesh for Example 2

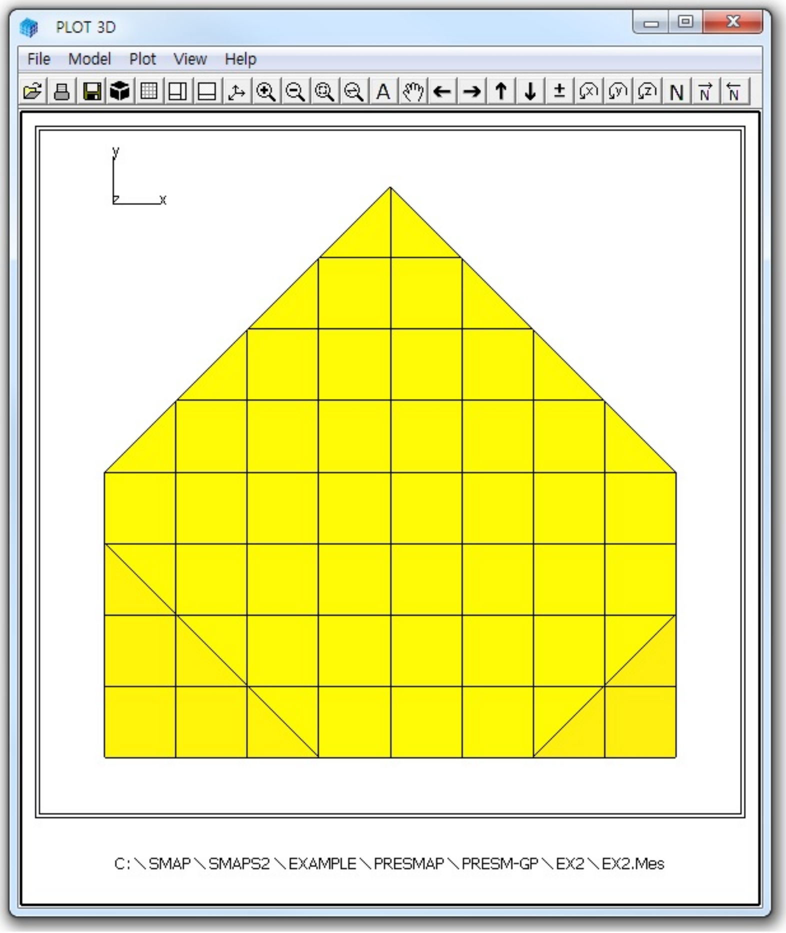


Figure 7.65 Finite element mesh for Example 2

7.4.3 Example 3: Circular Sector

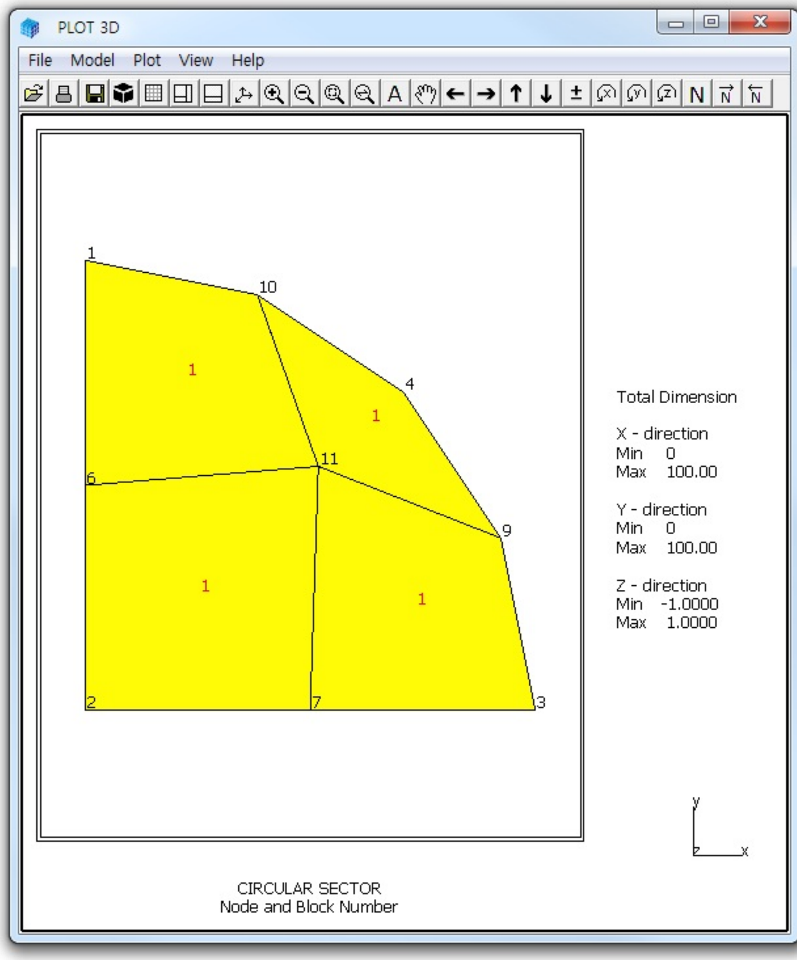


Figure 7.66 Block mesh for Example 3

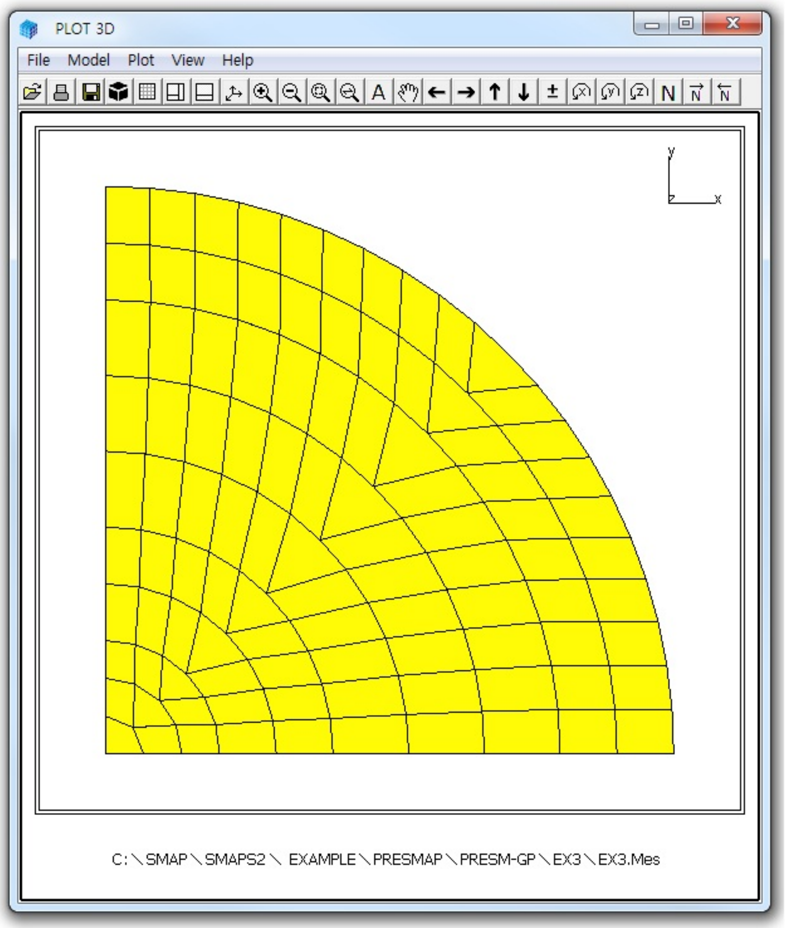


Figure 7.67 Finite element mesh for Example 3

7.4.4 Example 4: Straight Line Sector

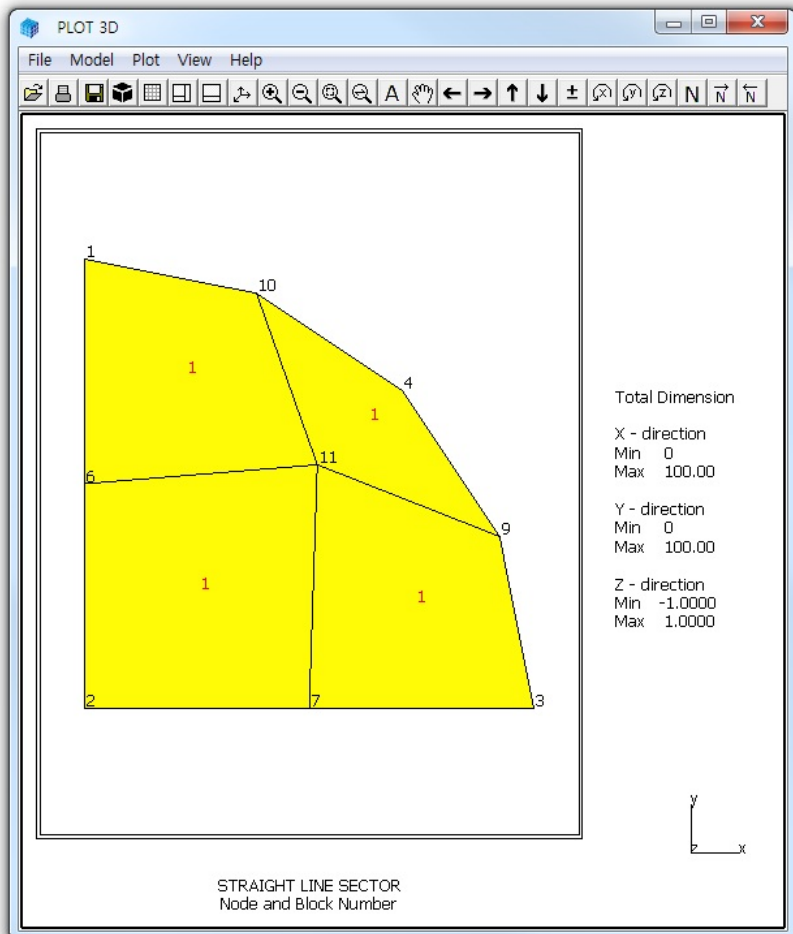


Figure 7.68 Block mesh for Example 4

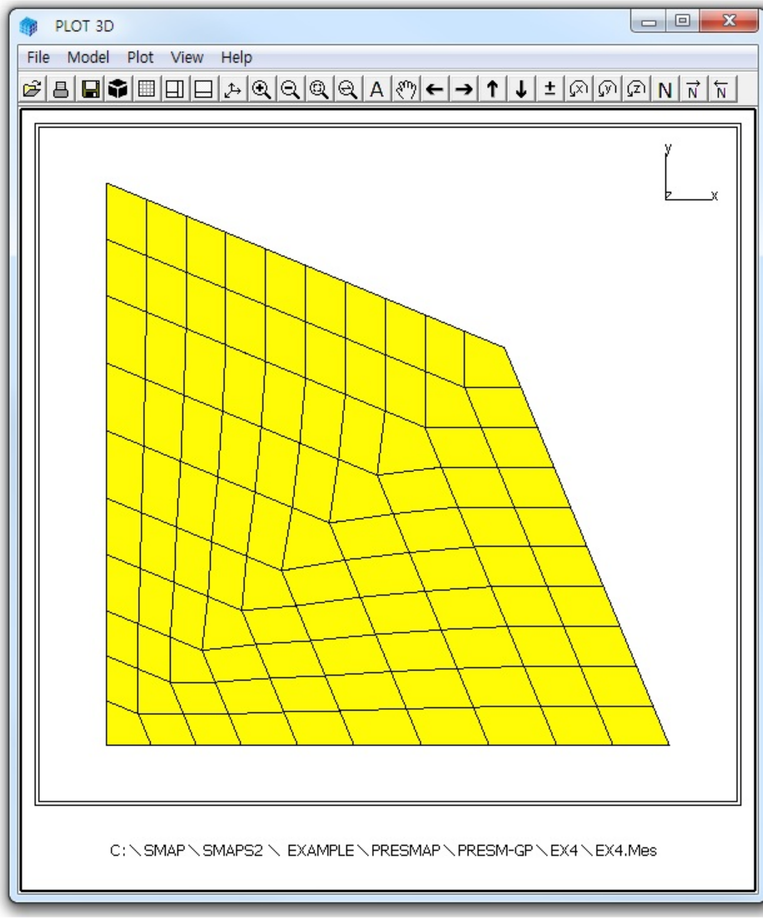


Figure 7.69 Finite element mesh for Example 4

7.4.5 Example 5: Surface and Line Element (1)

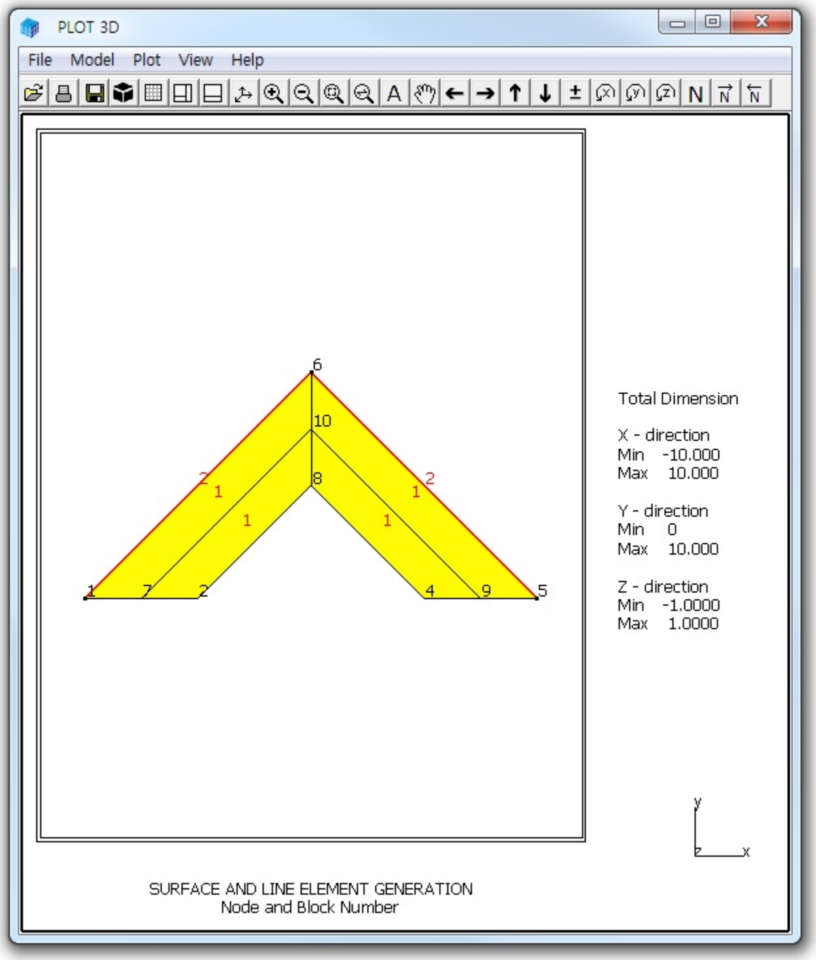


Figure 7.70 Block mesh for Example 5

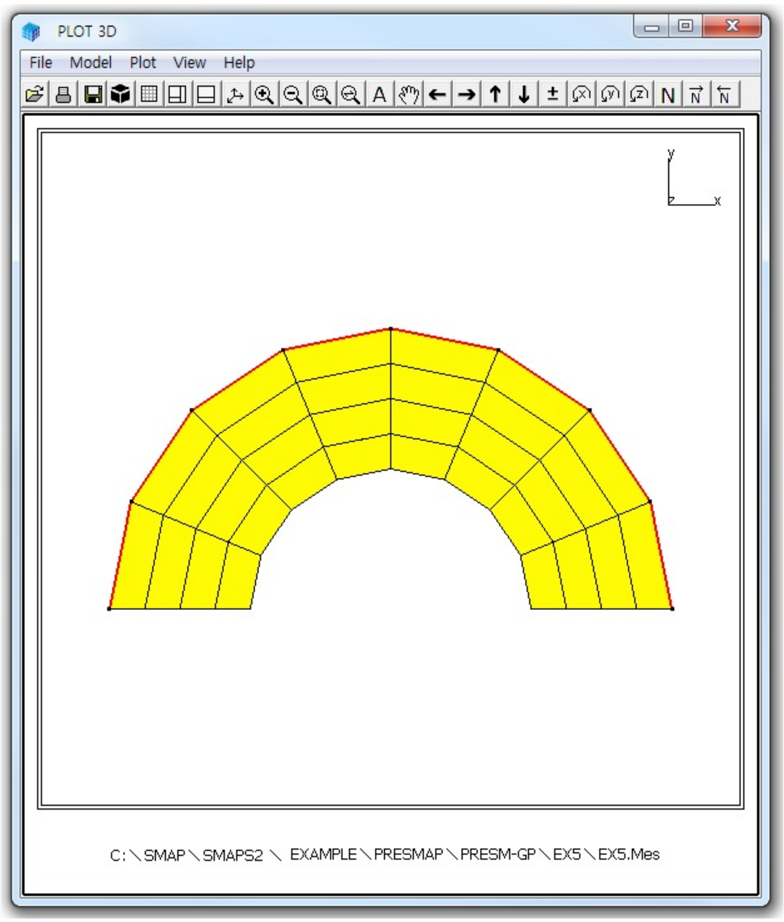


Figure 7.71 Finite element mesh for Example 5

7.4.6 Example 6: Surface and Line Element (2)

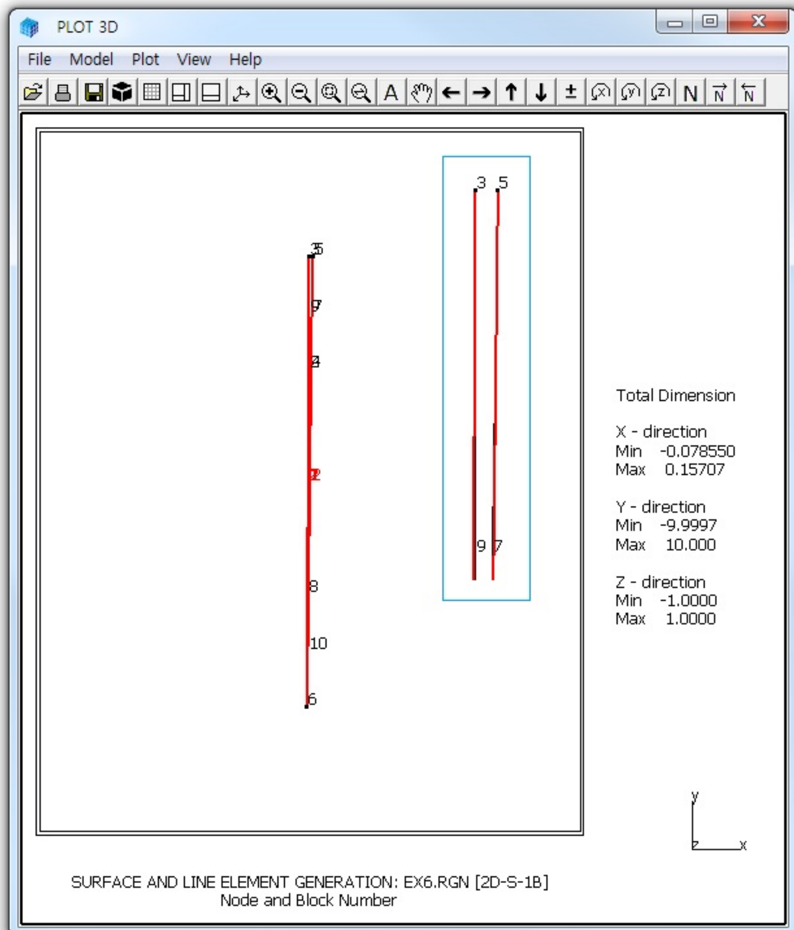


Figure 7.72 Block mesh for Example 6

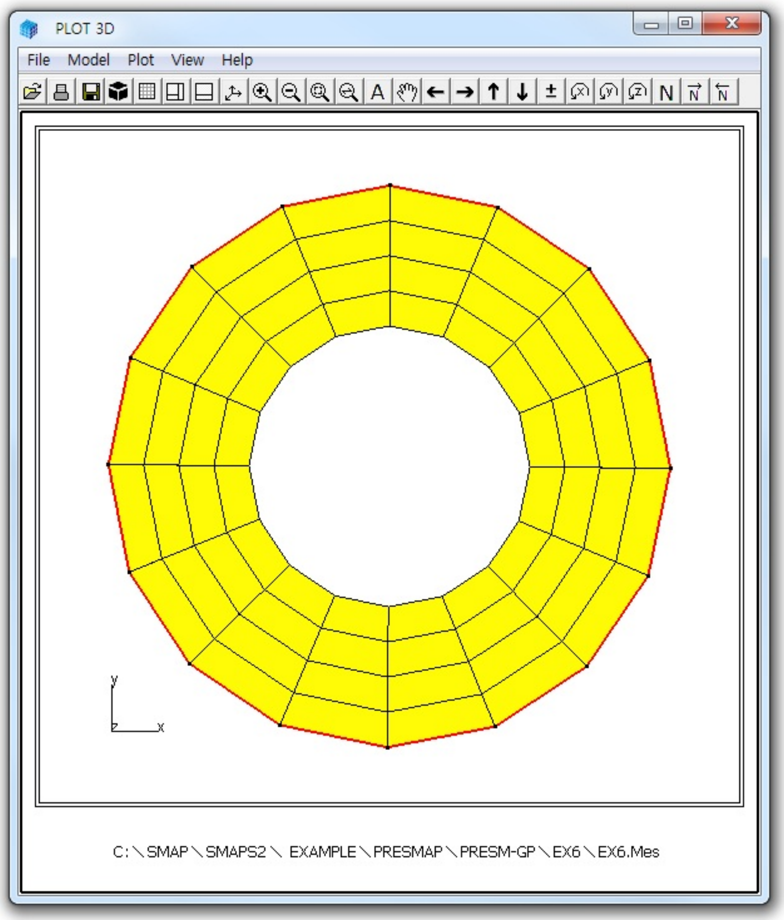


Figure 7.73 Finite element mesh for Example 6

7.4.7 Example 7: Surface and Line Element (3)

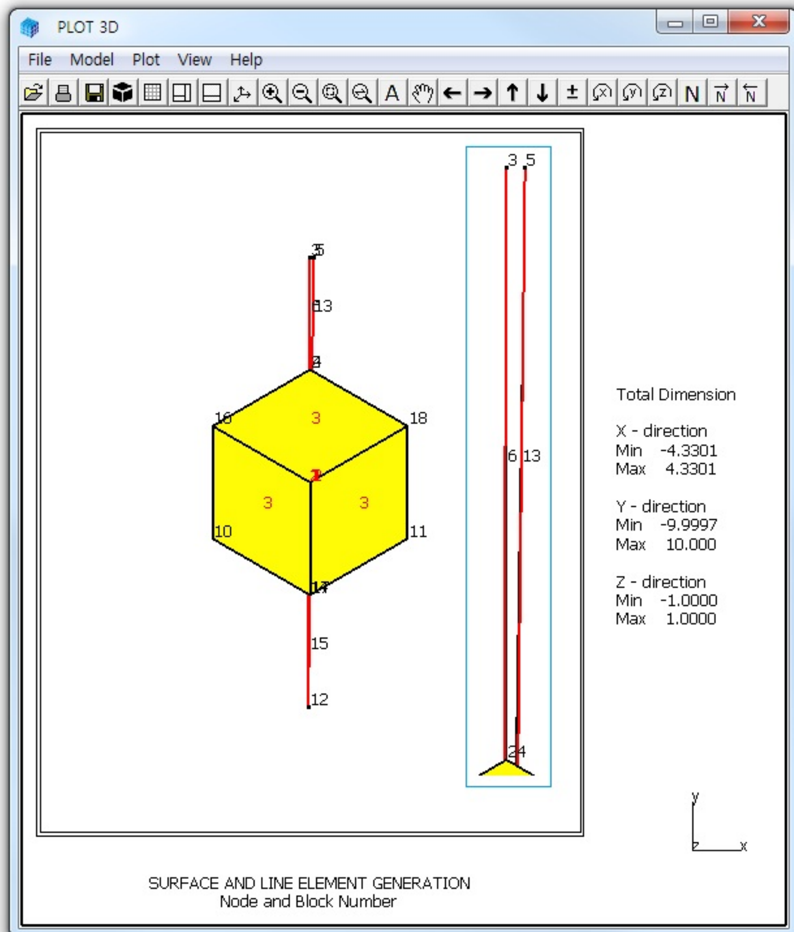


Figure 7.74 Block mesh for Example 7

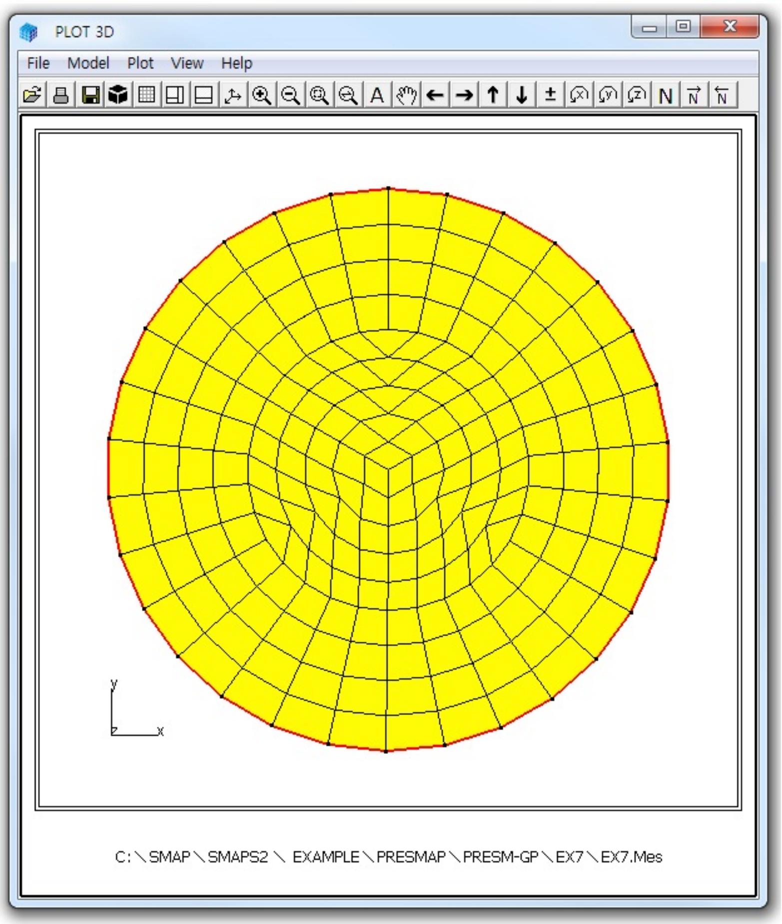


Figure 7.75 Finite element mesh for Example 7

7.4.8 Example 8: Cement-Soil Road

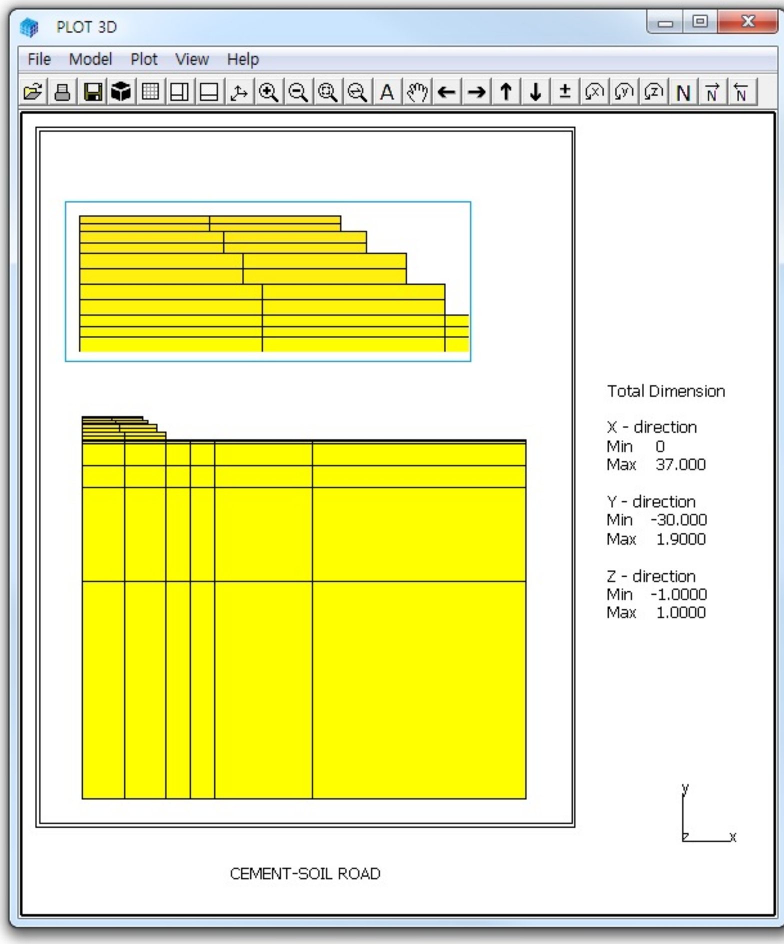


Figure 7.76 Block mesh for Example 8

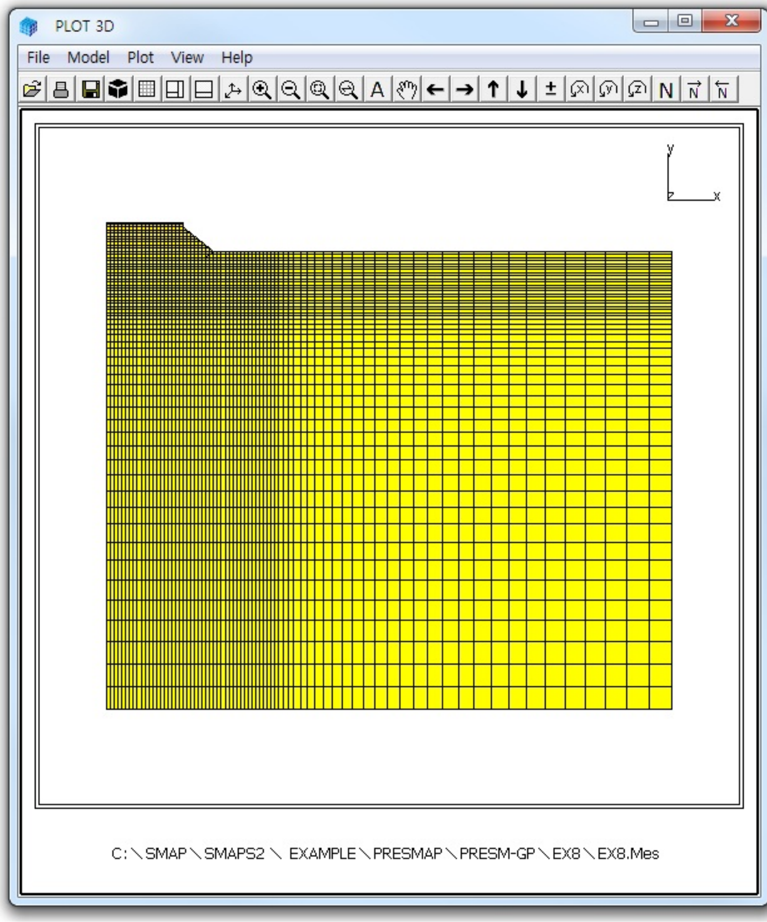


Figure 7.77 Finite element mesh for Example 8

7.4.9 Example 9: Tunnel in Spherical Geometry

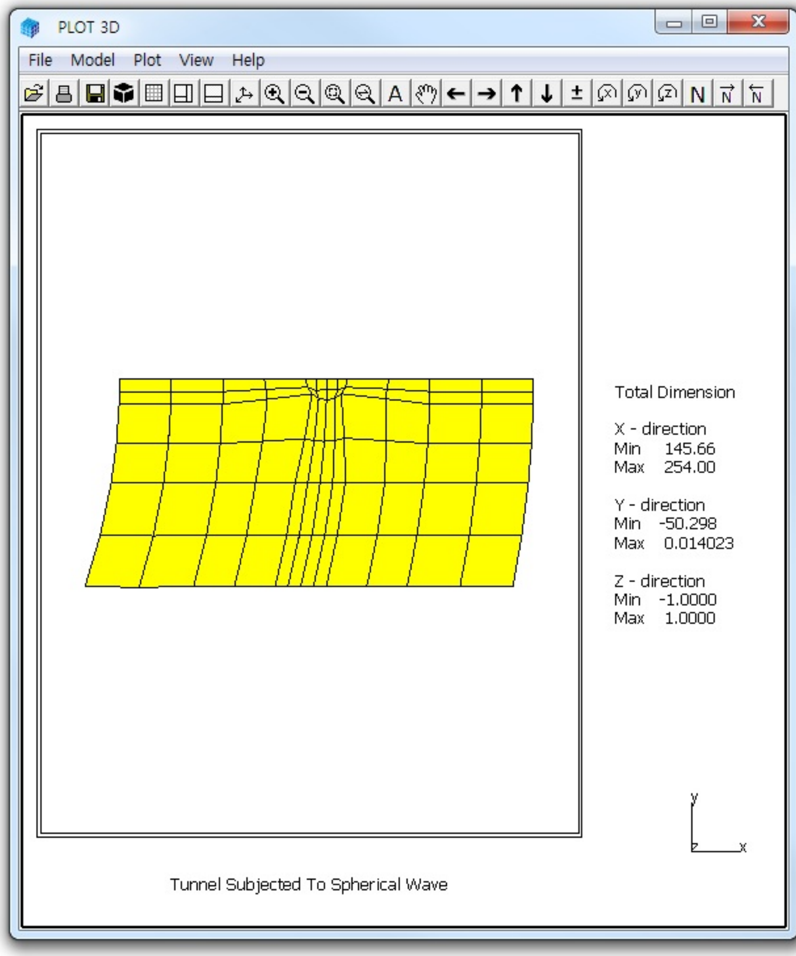


Figure 7.78 Block mesh for Example 9

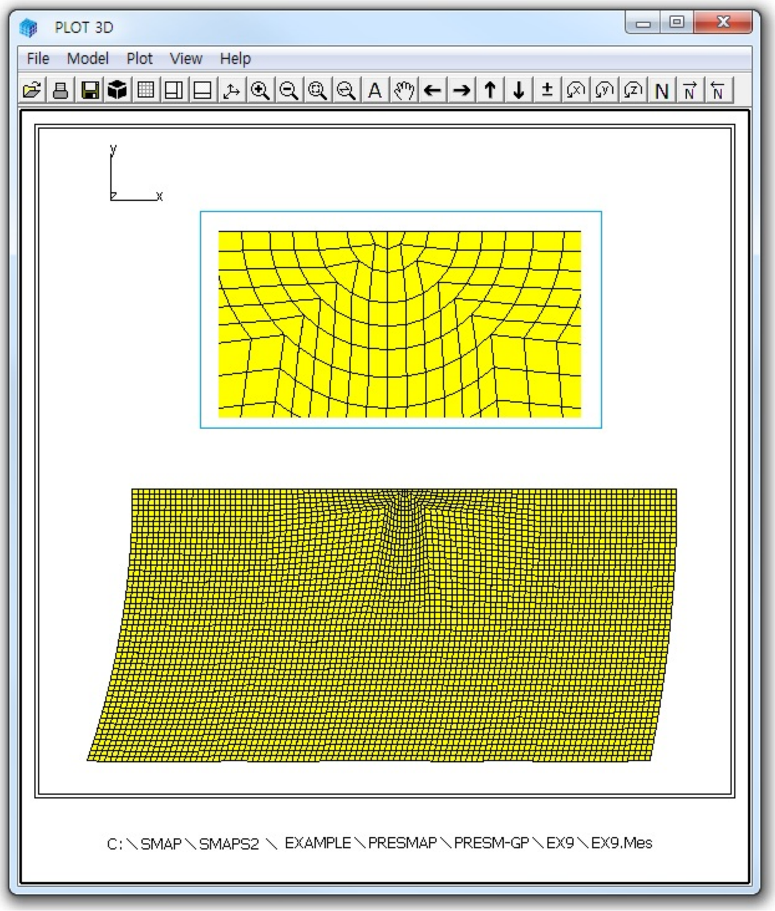


Figure 7.79 Finite element mesh for Example 9

7.4.10 Example 10: Horseshoe Tunnel

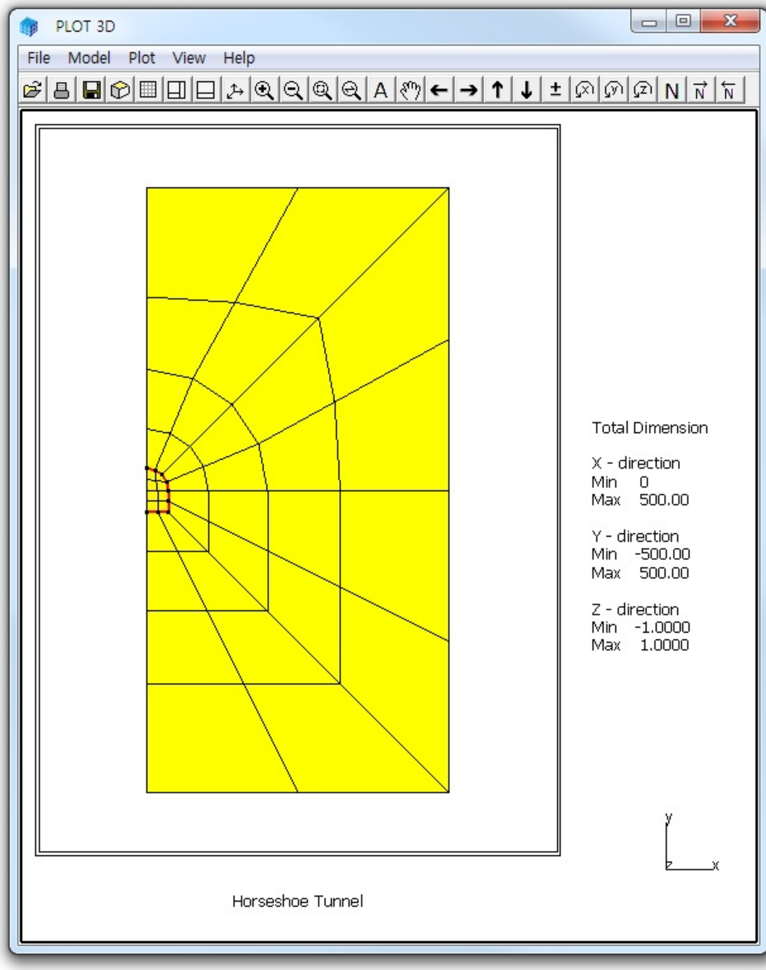


Figure 7.80 Block mesh for Example 10

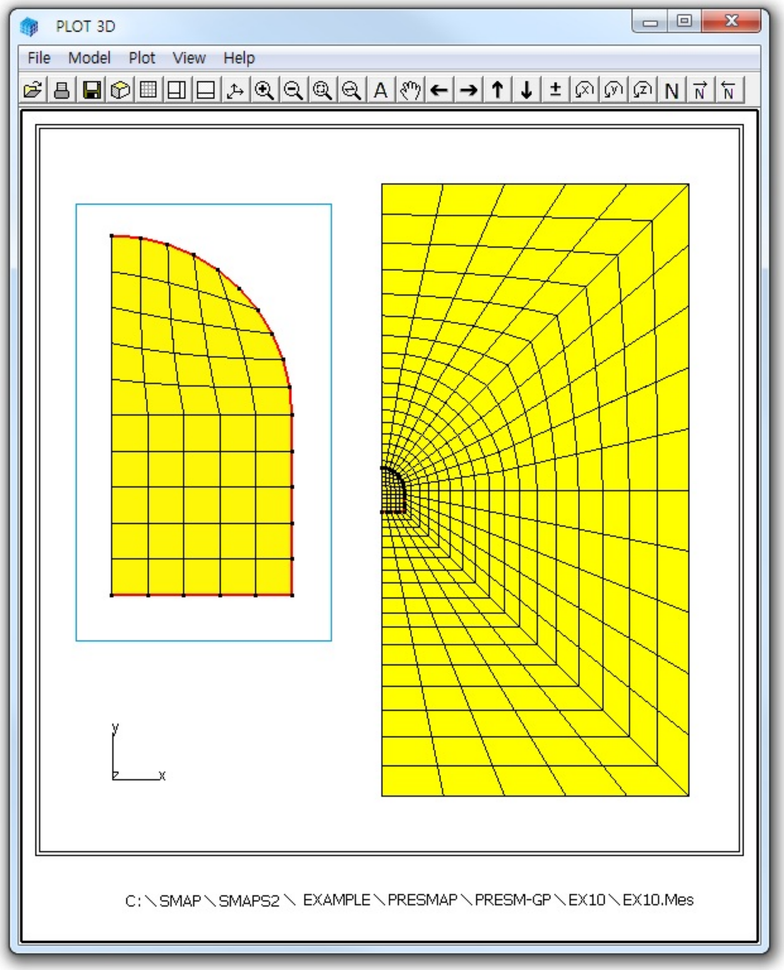


Figure 7.81 Finite element mesh for Example 10

7.4.11 Example 11: Wedge Surface Block

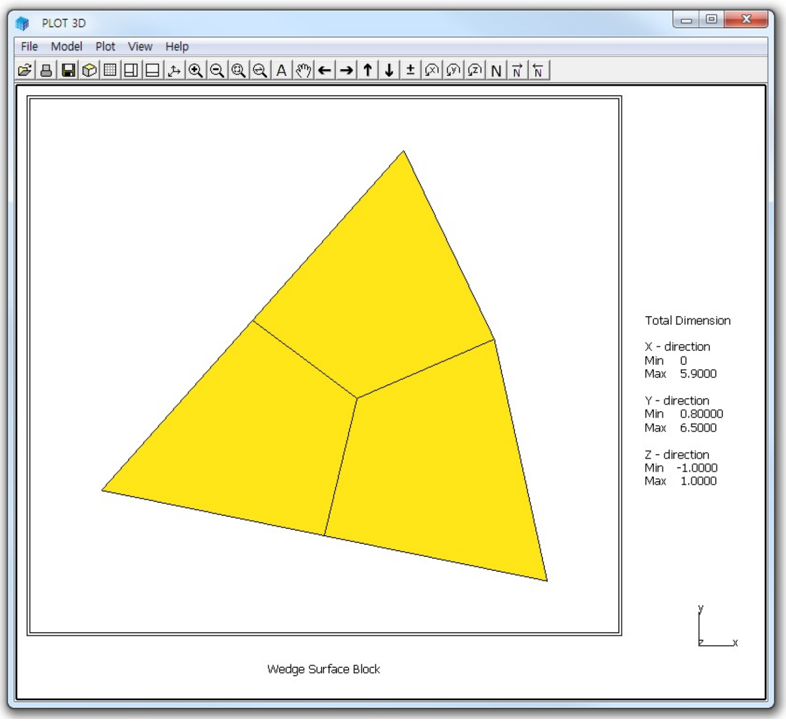


Figure 7.82 Block mesh for Example 11

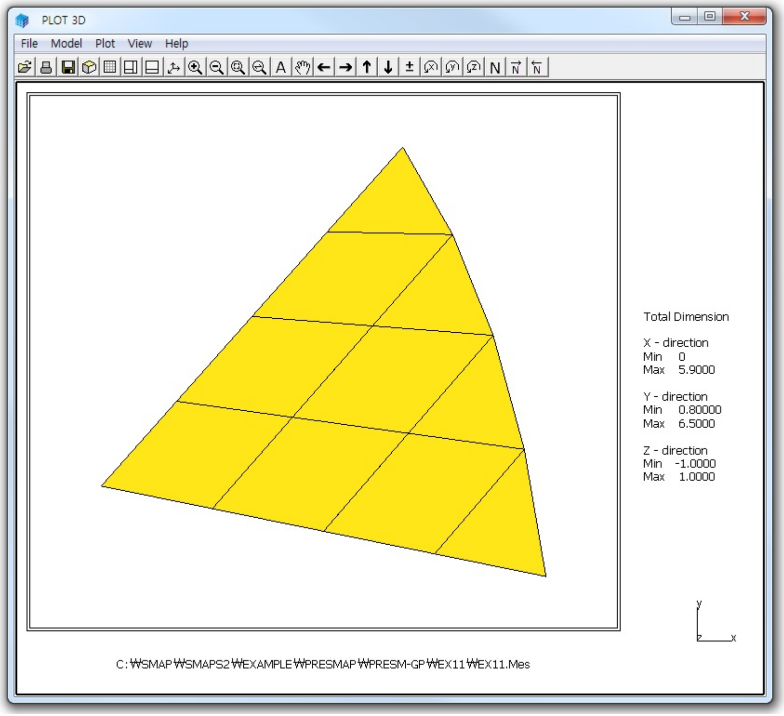


Figure 7.83 Finite element mesh for Example 11

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-S2 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
```

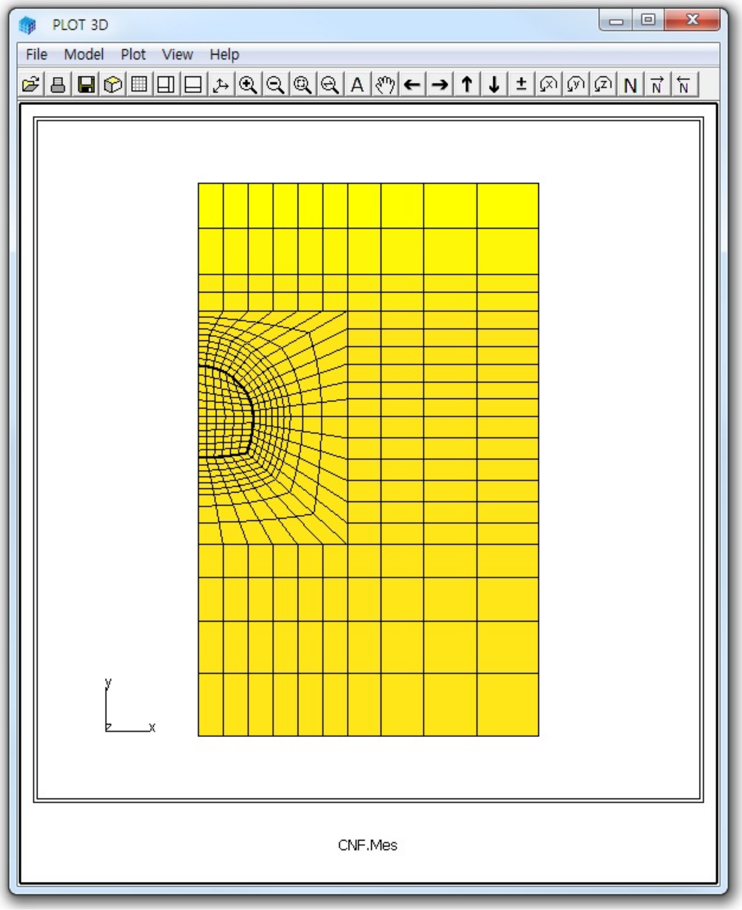


Figure 8.1 Final element meshes representing Core, Near-field, and Far-field regions, CNF.Mes

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
```

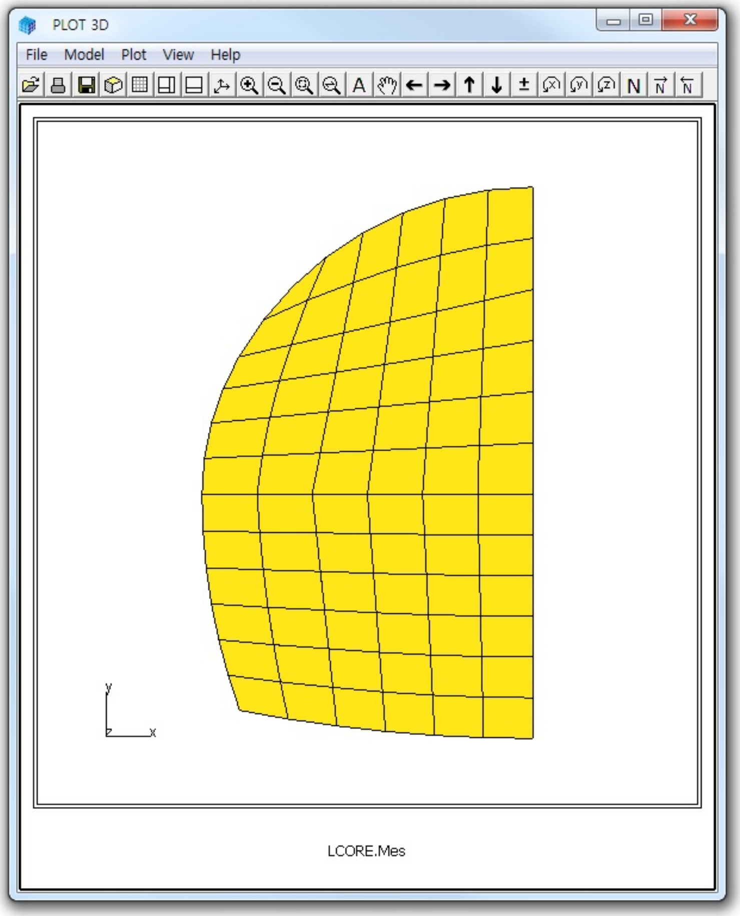


Figure 8.2 Left core mesh, LCORE.Mes

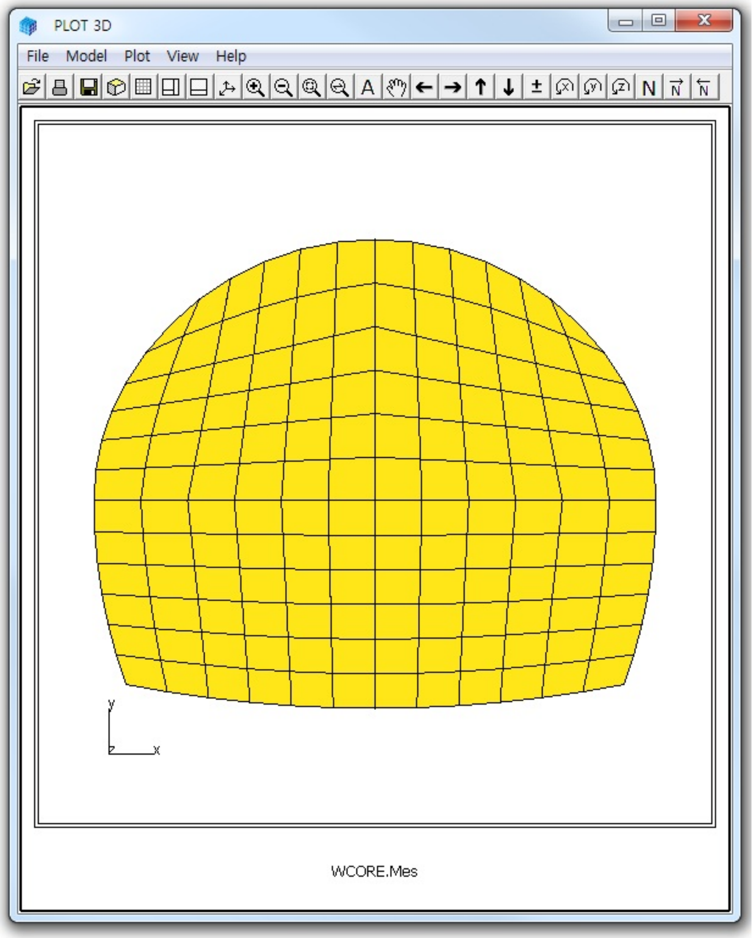


Figure 8.3 Combined whole core mesh, WCORE.Mes

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    2
* 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  DEN  LTPI  LMAT
     7   2.3   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 DEN MATNOJT DENJT  THICJT  LTPI, LMATI, LTPO, LMATO
     3 2.3    4      2.1     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
```

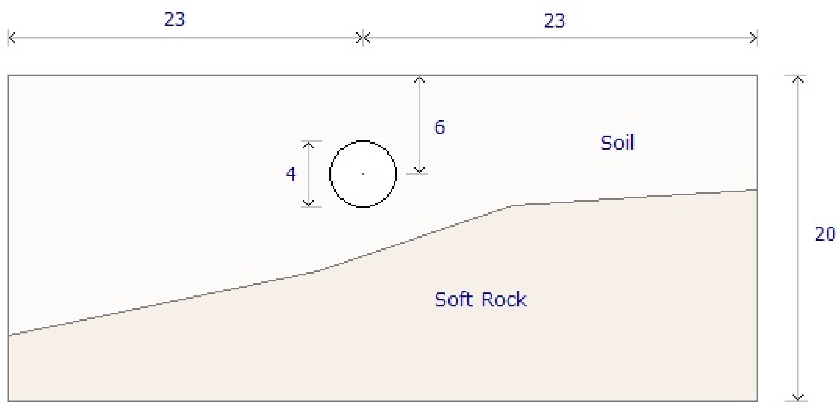


Figure 8.4 Schematic section view.

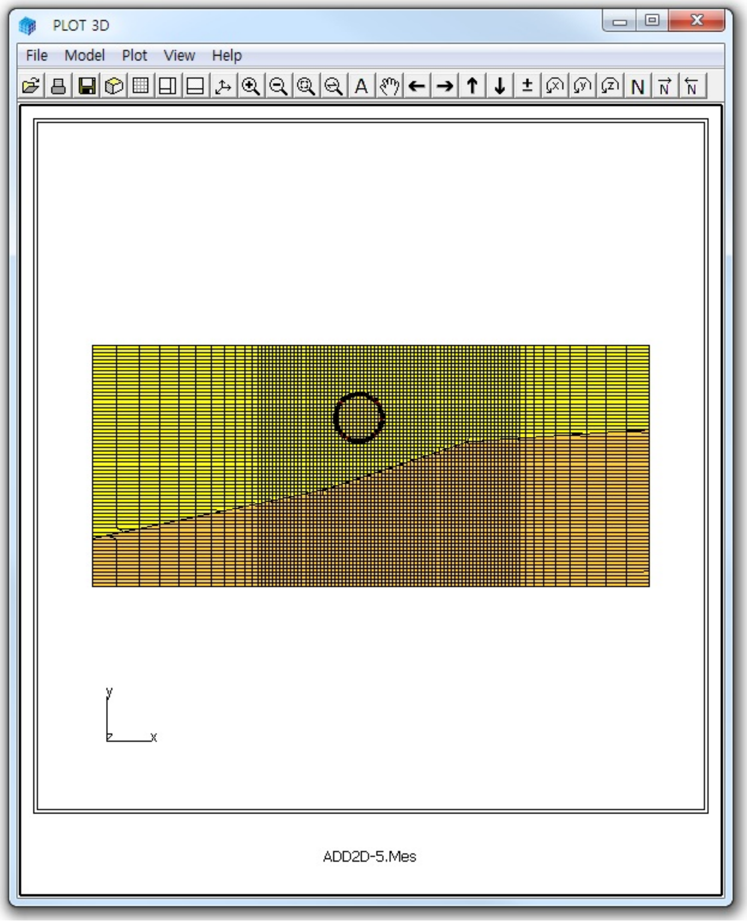


Figure 8.5 Overall finite element mesh

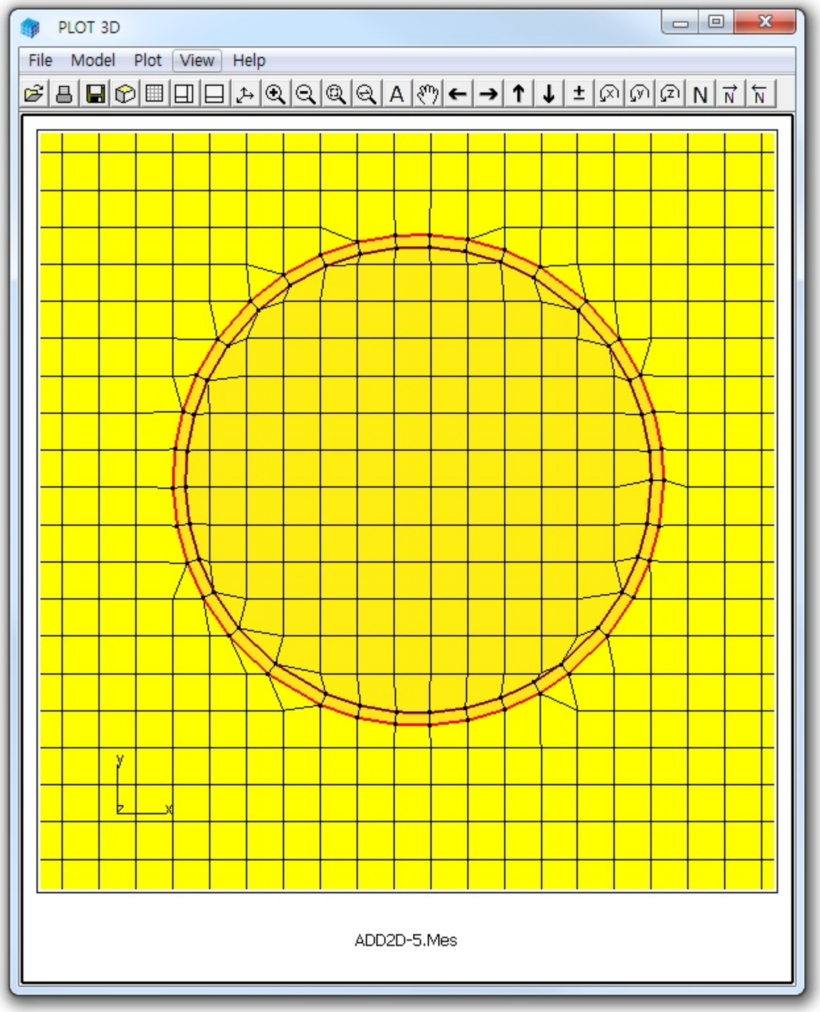


Figure 8.6 Finite element mesh around utility tunnel.

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-3D 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    2
* 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 DEN  LTPI  LMAT
     7  2.3    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 DENJT THICJT  LTPI, LMATI, LTPO, LMATO
    5      2.1   -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  DEN  LTPI  LMAT
     2   2.3    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 DEN  MATNOJT DENJT  THICJT  LTPI, LMATI, LTPO, LMATO
     3  2.3     4     2.1     -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 DEN  MATNOJT DENJT  THICJT  LTPI, LMATI, LTPO, LMATO
     3 2.3     4      2.1    -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  DEN MATj DENj  THICj  LTi LMi LTo LMo
    3  2.3   4  2.1  -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
```

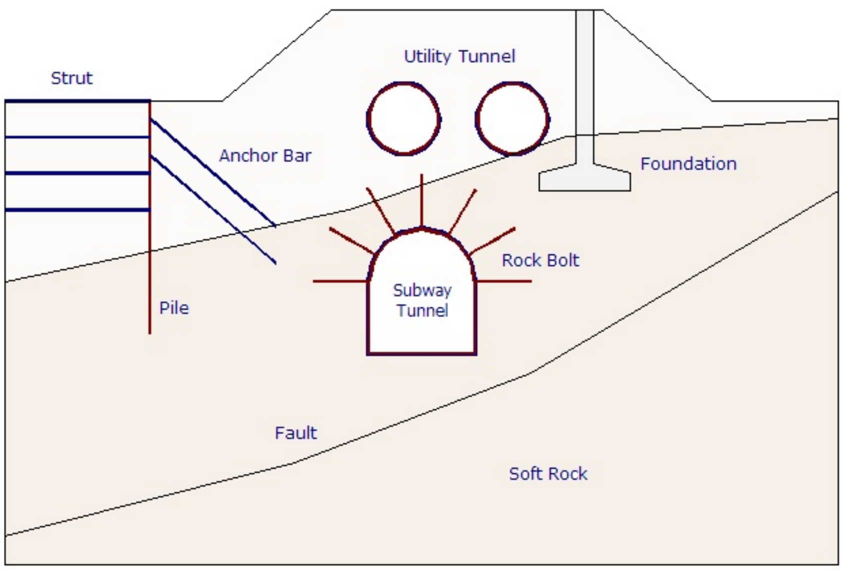


Figure 8.7 Schematic section view

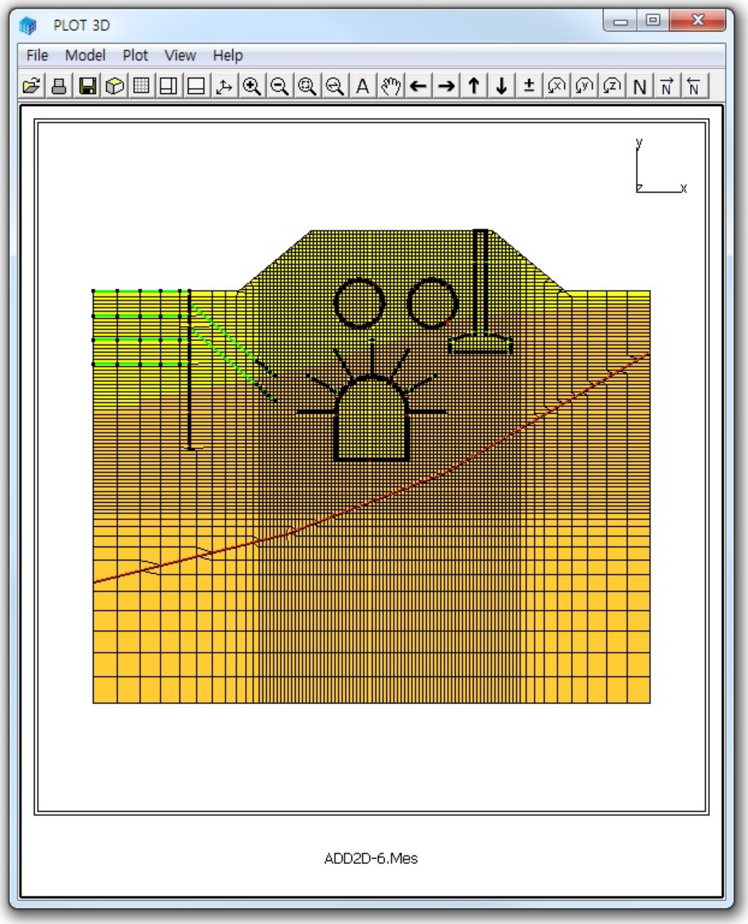


Figure 8.8 Overall finite element mesh

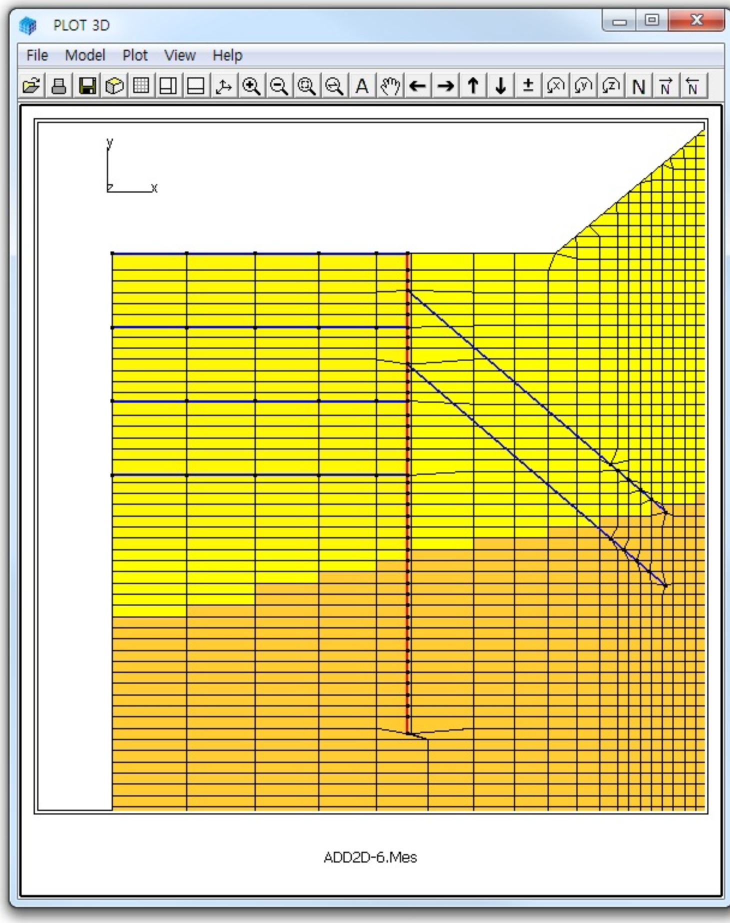


Figure 8.9 Finite element mesh around excavation zone

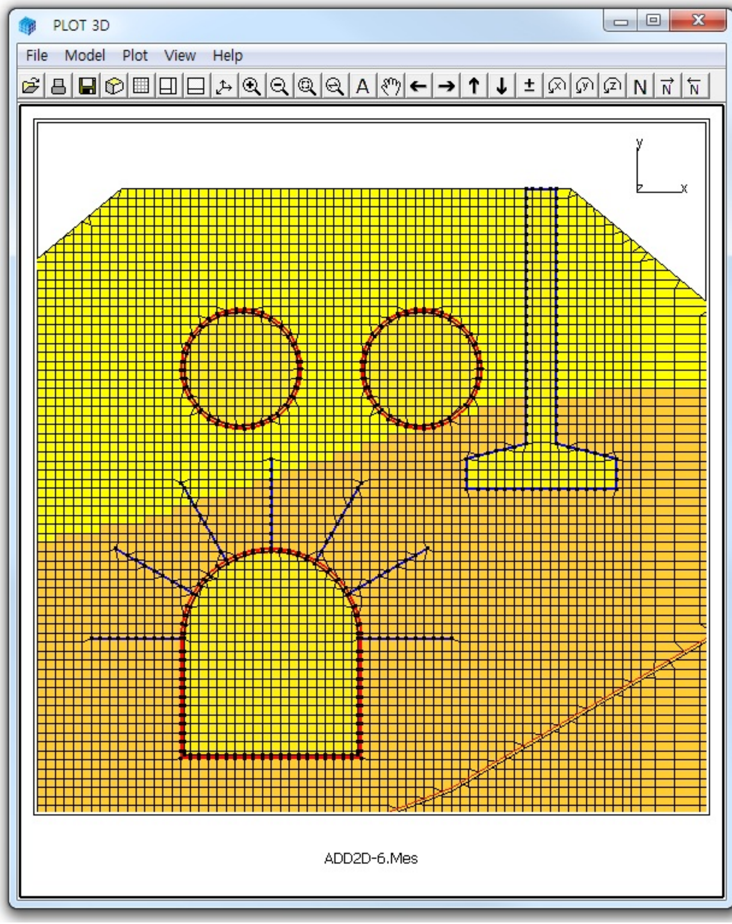


Figure 8.10 Finite element mesh around tunnels and foundation

SUPPLEMENT
Example Problem

SUPPLEMENT Menu contains supporting programs which are useful to prepare input data for pre- and main-processing programs of SMAP-S2.

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

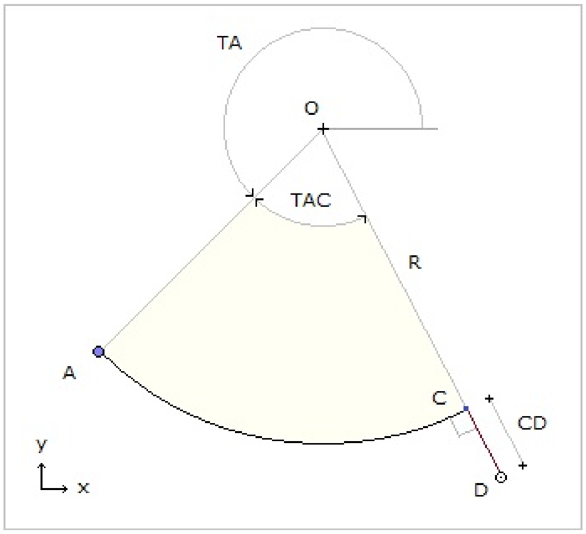
45.0 3.0

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-3D 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). 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. MeshS2.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. LoadS2.Dat in Table 10.2, has been prepared according to LOAD-2D User's Manual.

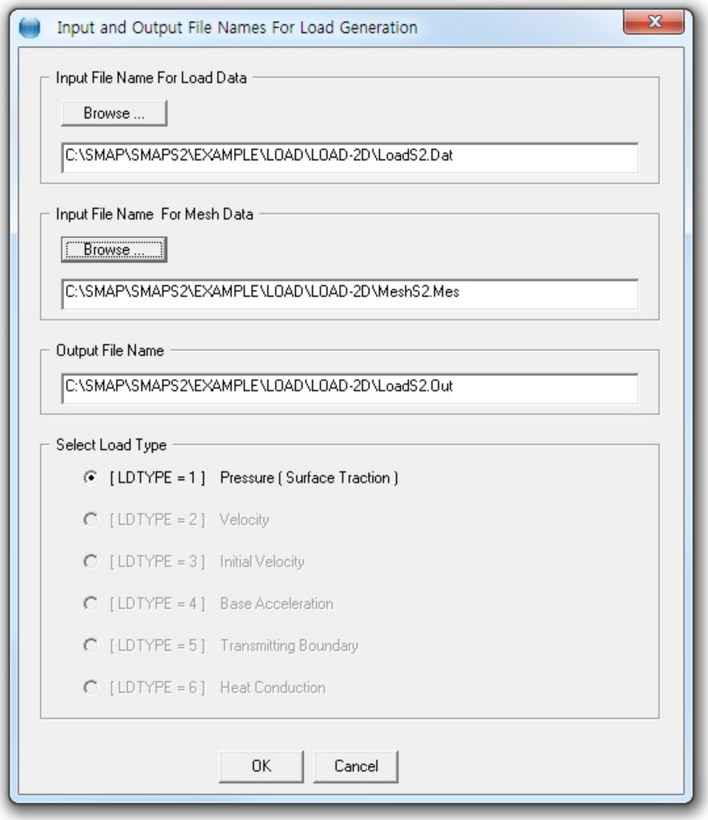
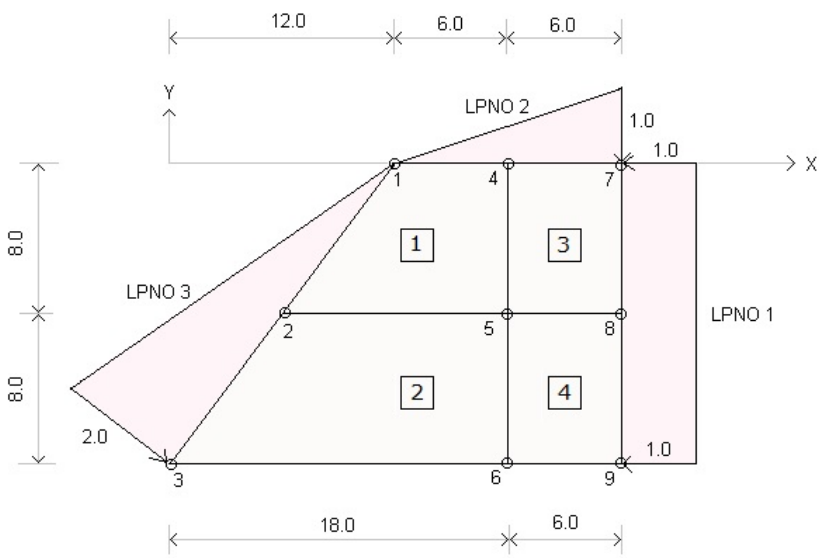


Figure 10.1 Load generation dialog



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

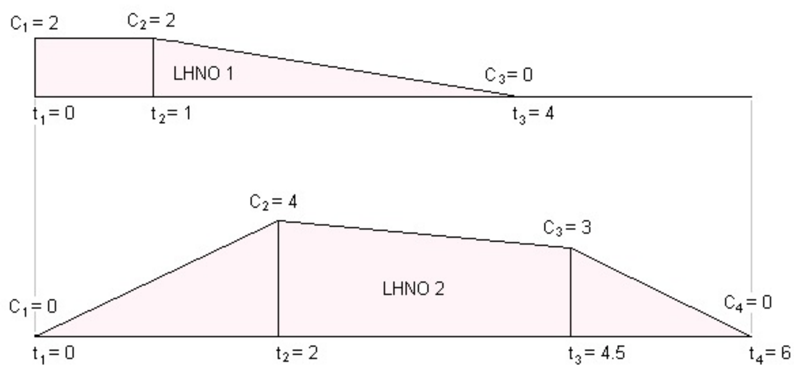


Figure 10.3 Load time histories for Example 1

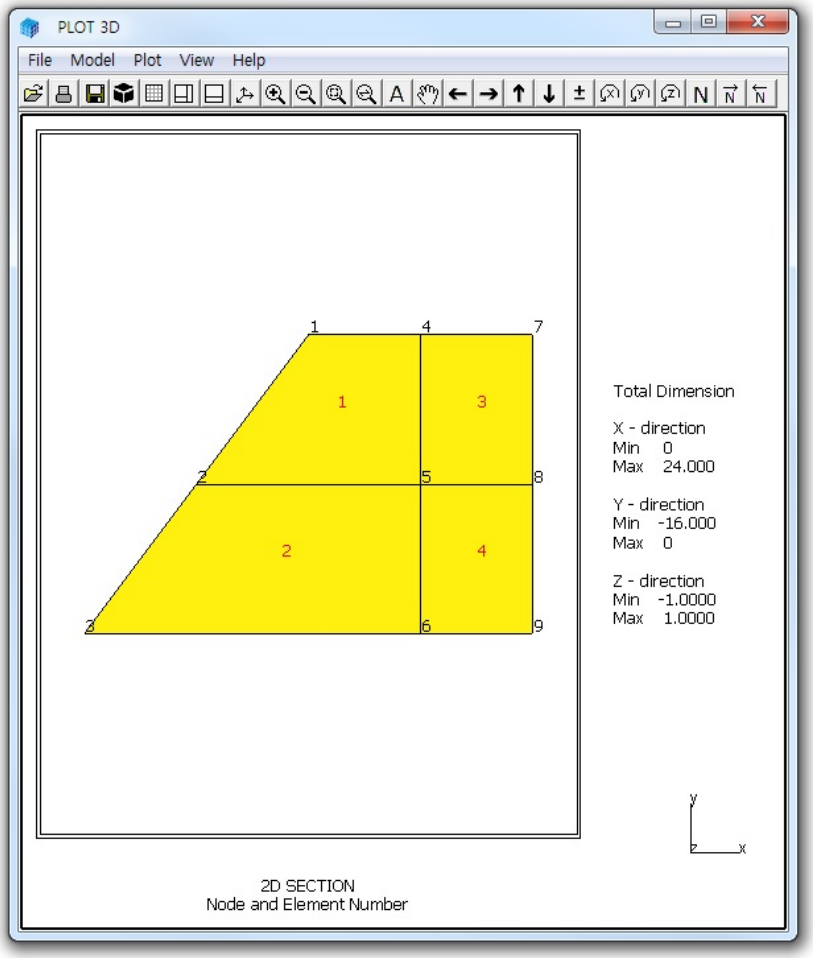


Figure 10.4 Finite element mesh for Example 1

Table 10.1 Listing of mesh data inut file MeshS2.Mes for Example 1

```
2D SECTION
NUMNP   NCONT   NBEAM  NTRUS
   9      4       0      0
NODAL COORDINATES
NODE  IDX  IDY  IDT      XC           YC
  1    1    0    1       12.           0.
  2    0    0    1        6.          -8.
  3    0    0    1        0.         -16.
  4    0    0    1       18.           0.
  5    0    0    1       18.          -8.
  6    0    0    1       18.         -16.
  7    0    0    1       24.           0.
  8    0    0    1       24.          -8.
  9    0    0    1       24.         -16.
ELEMENT INDEX
NEL   I1   I2   I3   I4   M5   M6   M7   M8 MATC THIC TBJWL
  1    4    1    2    5    0    0    0    0    4  1.  .0000E+00
  2    5    2    3    6    0    0    0    0    4  1.  .0000E+00
  3    7    4    5    8    0    0    0    0    4  1.  .0000E+00
  4    8    5    6    9    0    0    0    0    4  1.  .0000E+00
```

Table 10.2 Listing of load data input file LoadS2.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, LoadS2.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 format of Card Group 10 in SMAP-S2 main input.

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.

Table 10.3 Listing of load output file LoadS2.Out for Example 1

```
* CARD 10.1
* NLOAD
   12
* LOAD HISTORY NO:  1
* CARD 10.2
* 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 10.2
* 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 10.3
* NUMLH NUMSP
      2     6
*  CARD 10.4
*       STEP          Ci1          Ci2
  .00000E+00   .20000E+01   .00000E+00
  .10000E+01   .20000E+01   .20000E+01
  .20000E+01   .13333E+01   .40000E+01
  .40000E+01  -.59605E-07   .32000E+01
  .45000E+01   .00000E+00   .30000E+01
  .60000E+01   .00000E+00   .00000E+00
* END OF LOAD DATA
```

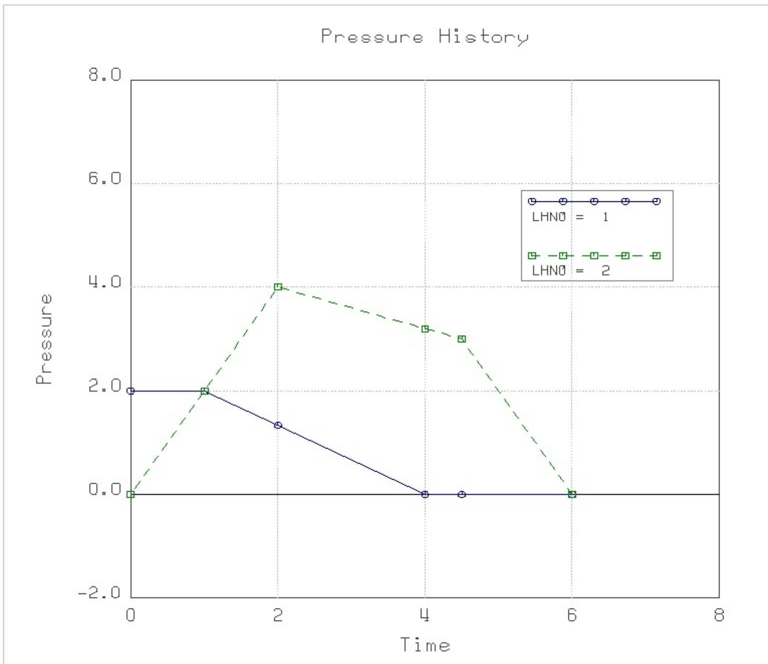


Figure 10.5 Generated load time histories for Example 1

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-S2 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

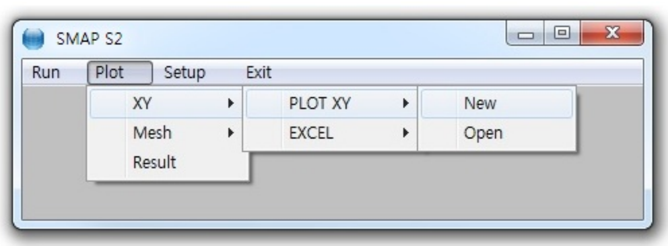


Figure 11.1 Accessing XY graph (New)

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
```

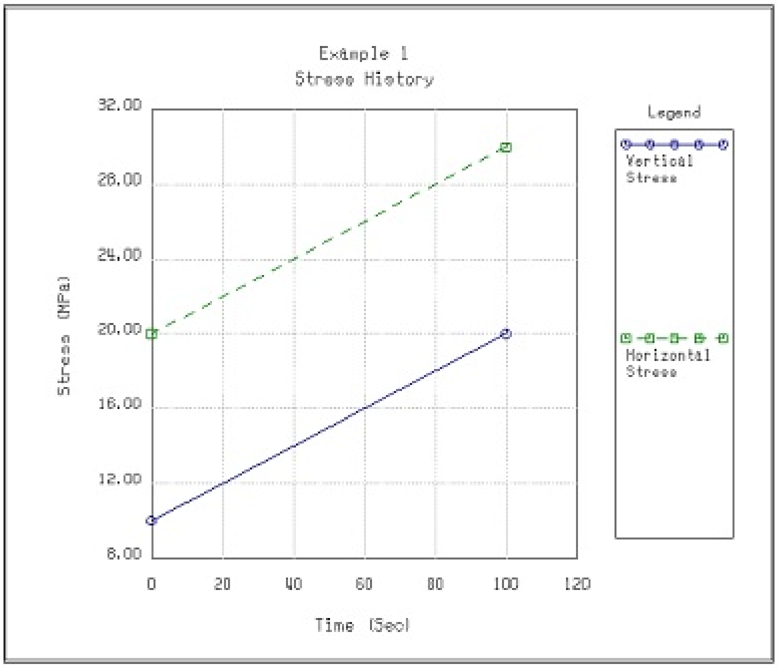


Figure 11.2 Modified graph on PLOT XY

Step 3: Modify XY Graph by Edit Dialog

Access Edit dialog by clicking the Edit menu in PLOT XY (Figure 11.3):

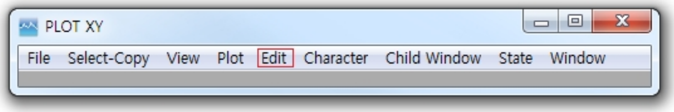


Figure 11.3 Edit menu in PLOT XY

Edit dialog will be displayed as shown in Figure 11.4.

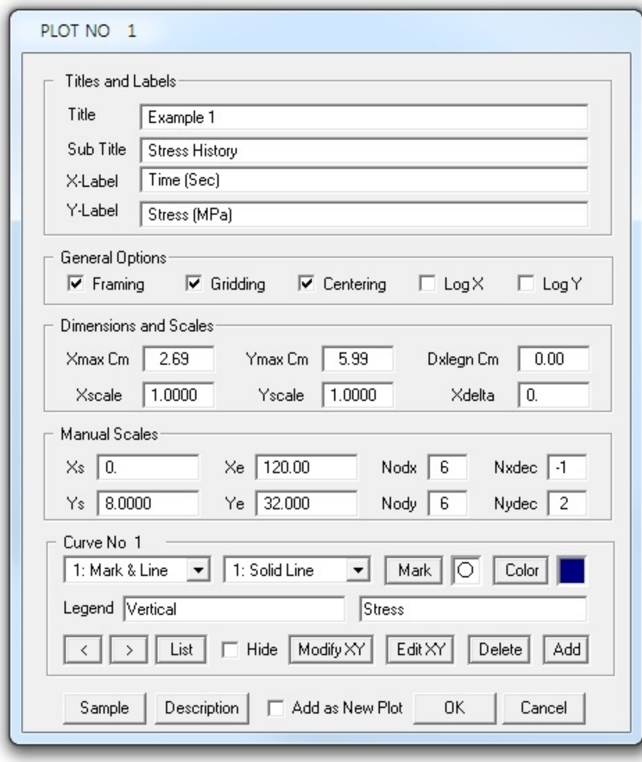


Figure 11.4 Edit dialog

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.

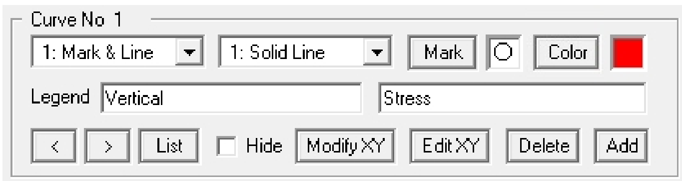


Figure 11.5 Updated red color in Edit dialog

Click OK button in Edit dialog.
Then updated plot will be displayed on PLOT XY as in Figure 11.6.

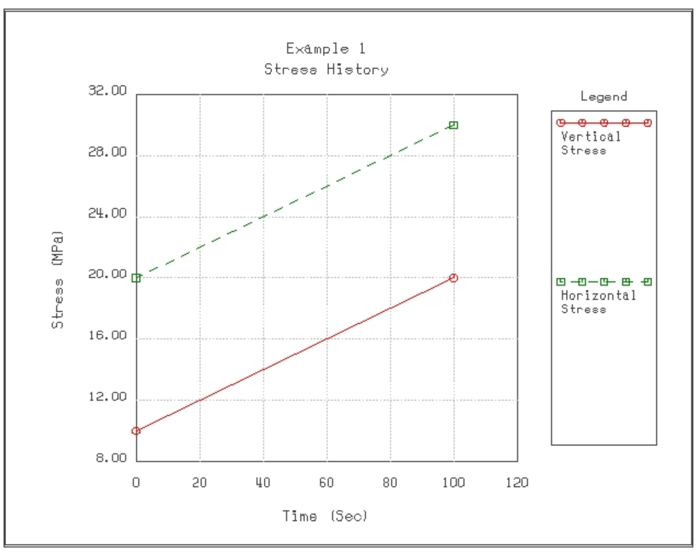


Figure 11.6 Updated first curve on PLOT XY

Step 4: Open XY Graph on Excel Spreadsheet

Access XY Graph by selecting following items in SMAP (Figure 11.7):
Plot → XY → EXCEL → Open

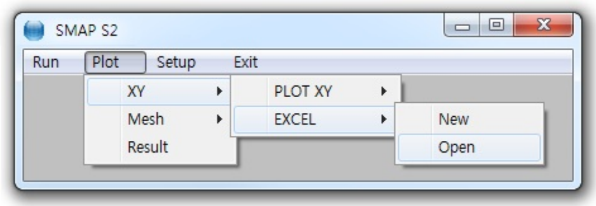


Figure 11.7 Accessing XY graph on Excel (Open)

Open XY.dat in the current working directory.
XY graph will be displayed on Excel Spreadsheet as in Figure 11.8.

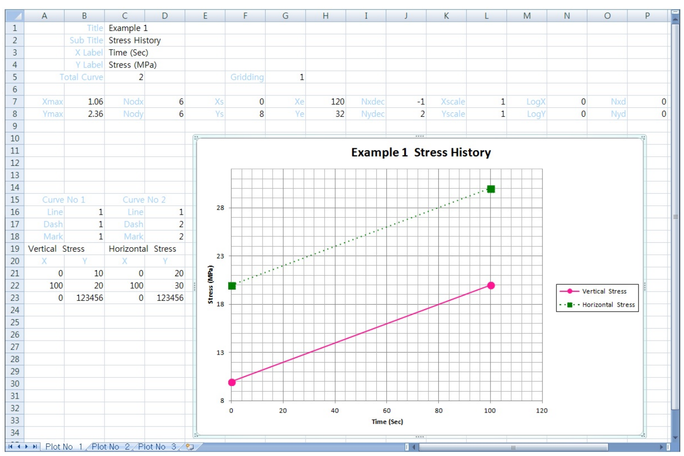


Figure 11.8 XY graph on Excel spreadsheet

Refer to more samples in the following directory:
C:\Smap \SmapS2 \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-S2 Example Problem 6 (Laminated Beam with Slip Interface).

This example consists of the following main actions:

- Execute SMAP-S2 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-S2 Example

Execute SMAP-S2 by selecting the following menu items in SMAP (Figure 11.9): Run → Smap → Execute

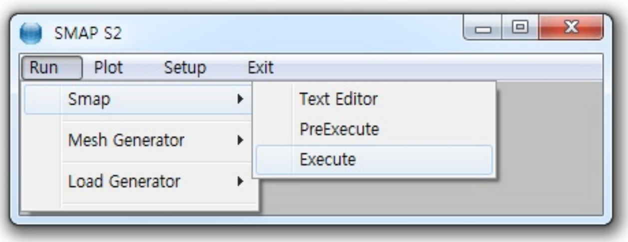


Figure 11.9 Execute SMAP-S2 example problem

Note that SMAP-S2 Example Problem 6 includes XY graph specified in Card Group 12 in SMAP Post File Vp6.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-S2 post file (File Vp6.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.

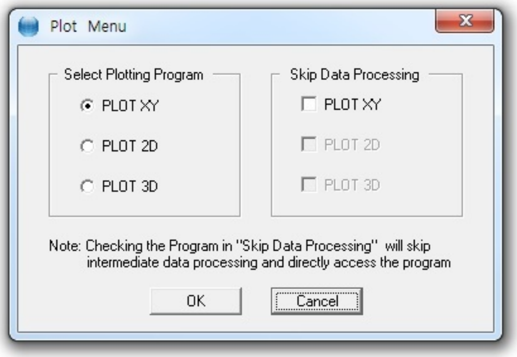


Figure 11.10 Plot menu dialog

Select PLOT XY in Select Plotting Program dialog in Figure 11.11.
Click OK button.

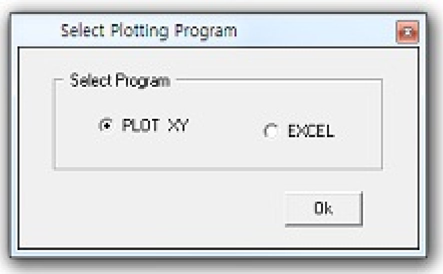


Figure 11.11 Select plotting program dialog

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

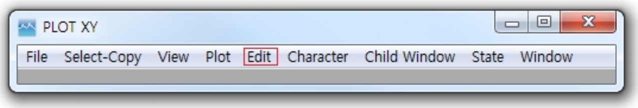


Figure 11.12 Edit menu in PLOT XY

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.

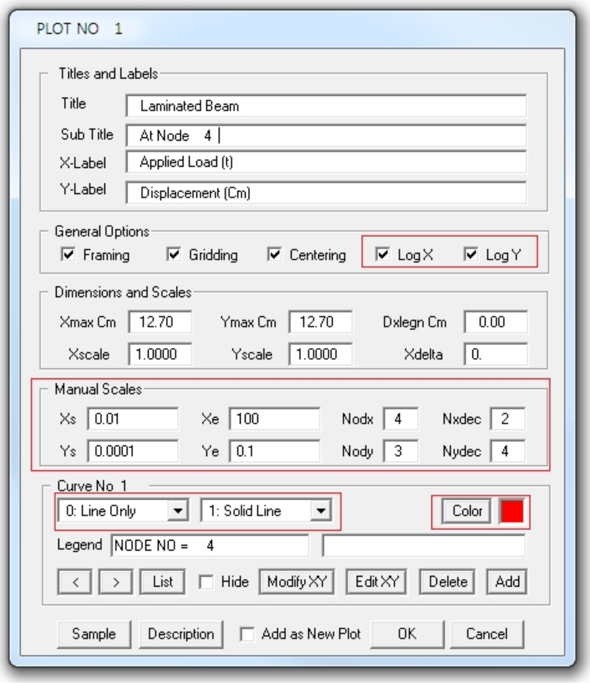


Figure 11.13 Edit dialog

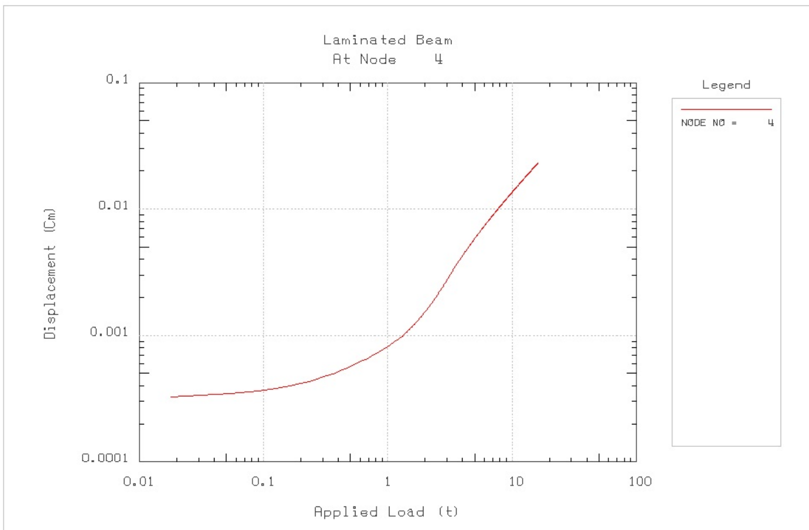


Figure 11.14 XY graph in log scales on PLOT XY

Refer to more samples in the following directory:
C:\Smap \SmapS2 \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

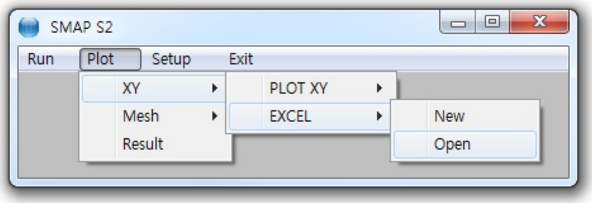


Figure 11.15 Accessing XY graph on Excel (Open)

Open PlotXy.dat in the current working directory.
XY graph will be displayed on Excel Spreadsheet as in Figure 11.16.

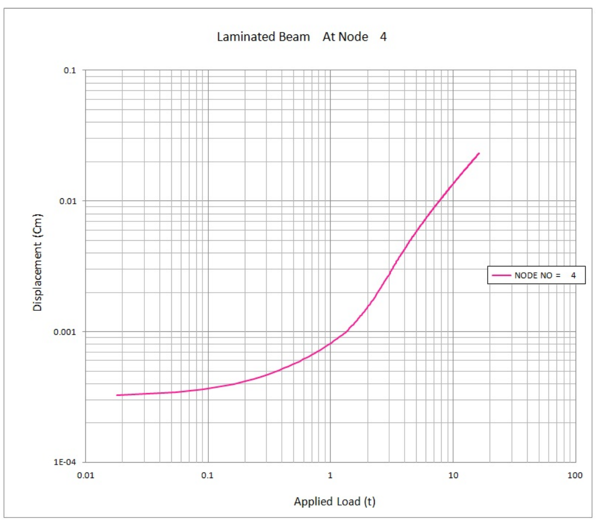


Figure 11.16 XY graph on Excel spreadsheet

Refer to more samples in the following directory:
C:\Smap \SmapS2 \Example \XY_Graph \Excel XY Graph Sample.pdf

Go to Edit > Preferences > Page Display > Uncheck Enhance Thin Lines

SMAP ® - S2

Structure Medium Analysis Program

2-D Static Analysis

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

This paper introduces the nonlinear finite element computer program SMAP-S2 developed by COMTEC RESEARCH. The program has specific applications for modeling geomechanical problems involving multi-staged excavation or embankment. Other practical applications of SMAP-S2 includes static mechanical analysis for underground structures such as tunnels, shafts, caverns, chambers, etc.

Section 2 describes finite element formulations. Based on the principle of virtual work, structural equilibrium equations are derived for continuum element, layered reinforced beam element, and truss element. This theoretical formulation takes account of the general nonlinearities associated with large deformation, material inelasticity, and boundary condition change.

Section 3 describes constitutive relations of nonlinear material models available in the program SMAP-S2. These nonlinear models include Generalized Hoek & Brown Model and Joint Model. The Generalized Hoek and Brown Model includes the empirically based Hoek and Brown failure equation for in situ rock mass as well as the classic Von Mises, Mohr-Coulomb, and Drucker-Prager failure equations.

The Hoek and Brown model is based on extensive field and laboratory data. The material parameters for this model are tabulated for several different rock types as a function of rock quality. This empirical-based model is very valuable for analyst to model in situ rock mass when the laboratory test data is not sufficient to define the strength properties of in situ rock mass. The Joint Model is useful to simulate rock joints, faults, and interfaces.

To see the validation of the computational algorithms of the computer program SMAP-S2, refer to SMAP-S2 Example Problems.

Finite Element Formulation

2.1 Introduction

In this section , we derive the structural equilibrium equations for continuum elements, layered reinforced beam elements, and truss elements. Theoretical formulations take account of the general nonlinearities associated with large deformation, material inelasticity, and boundary condition change.

2.2 Continuum Element Formulation

Structural static equilibrium equations can be derived by the principle of virtual work.

$$\int_V \{\delta\varepsilon\}^T \{\sigma\}\, dv = \int_S \{\delta u\}^T \{T\}\, ds - \int_V \{\delta u\}^T \rho \{b\}\, dv \tag{2.1}$$

Where ($\delta\varepsilon$) is the virtual strain corresponding to the virtual displacement $\{\delta u\}$. In the Updated Lagrangian formulation, Equation 2.1 can be approximated in the following incremental form:

$$\int_{V_{n-1}} \{\delta\Delta\varepsilon\}^T \{\sigma_n\}\, dv = \int_{S_o} \{\delta\Delta u\}^T \{T_n\}\, ds + \int_{V_{n-1}} \{\delta\Delta u\}^T \rho \{b\}\, dv \tag{2.2}$$

Equation 2.2 assumes that the direction and magnitude of the surface loading is independent of the configuration. The Green-Lagrangian strain increment vector $\{\Delta\varepsilon\}$ refers to the configuration at load step n-1 and can be decomposed into linear and nonlinear parts.

$$\{\Delta\varepsilon\} = \{\Delta\varepsilon^l\} + \{\Delta\varepsilon^{nl}\} \tag{2.3}$$

$$\Delta\varepsilon^l_{i,j} = \frac{1}{2}\left(\Delta u_{i,j} + \Delta u_{j,i}\right) \tag{2.4}$$

$$\Delta\varepsilon^{nl}_{i,j} = \frac{1}{2}\left(\Delta u_{k,i} + \Delta u_{k,j}\right) \tag{2.5}$$

The left hand term in Equation 2.2 can be expanded as follows:

$$\int_{v_{n-1}} \{\delta\Delta\varepsilon\}^T \{\sigma_n\}\, dv = \int_{v_{n-1}} \{\delta\Delta\varepsilon^l\}^T \{\Delta\sigma\}\, dv + \int_{v_{n-1}} \{\delta\Delta\varepsilon^{nl}\}^T \{\sigma_{n-1}\}\, dv$$

$$+ \int_{v_{n-1}} \{\delta\Delta\varepsilon^l\}^T \{\sigma_{n-1}\}\, dv \qquad (2.6)$$

The stress increment vector $\{\Delta\sigma\}$ in Equation 2.6 is related to the linear portion of the Green - Lagrangian strain increment vector.

$$\{\Delta\sigma\} = [D^{ep}]\, \{\Delta\varepsilon^l\} \qquad (2.7)$$

Field variables in the above equations can be discretized by the nodal variables. Within each element,

$$\{\Delta u\} = [N_{n-1}]\, \{\Delta\bar{u}\}$$

$$\{\Delta\varepsilon^l\} = [B^l_{n-1}]\, \{\Delta\bar{u}\}$$

$$\{\Delta\varepsilon_{n-1}\}^T \{\sigma_{n-1}\} = \{\Delta\bar{u}\}^T \left([B^{nl}_{n-1}]^T \{\sigma_{n-1}\} [B^{nl}_{n-1}] \right) \{\Delta\bar{u}\} \qquad (2.8)$$

Now substituting Equations 2.6 through 2.8 into 2.2,
we obtain the following linearized global equilibrium equation.

$$\left([K^l] + [K^{nl}] \right) \{\Delta\bar{u}\} = \{P_n\} - \{R_{n-1}\} \qquad (2.9)$$

where

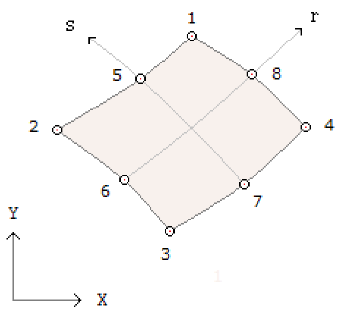
$$[K^{l}] = \Sigma \int_{v_{n-1}} [B^{l}_{n-1}]^T [D^{ep}] [B^{l}_{n-1}] \, dv \tag{2.10}$$

$$[K^{nl}] = \Sigma \int_{v_{n-1}} [B^{nl}_{n-1}]^T \{\sigma_{n-1}\} [B^{nl}_{n-1}] \, dv \tag{2.11}$$

$$\{P_n\} = \Sigma \int_{s_o} [N_o]^T \{T_n\} \, ds + \Sigma \int_{v_{n-1}} [N_{n-1}] \, \rho_{n-1} \{b\} \, dv \tag{2.12}$$

$$\{R_{n-1}\} = \Sigma \int_{v_{n-1}} [B^{l}_{n-1}]^T \{\sigma_{n-1}\} \, dv \tag{2.13}$$

Two dimensional continuum elements generally include plane strain, plane stress and axisymmetric conditions. Figure 2.1 shows 8 node isoparametric finite element and the corresponding shape functions incorporated into SMAP-S2.



$$\begin{Bmatrix} \Delta u_x \\ \Delta u_y \end{Bmatrix} = \begin{bmatrix} <h> & <o> \\ <o> & <h> \end{bmatrix} \begin{Bmatrix} \{\Delta \bar{u}_x\} \\ \{\Delta \bar{u}_y\} \end{Bmatrix}$$

$$\{\Delta \bar{u}_x\}^T = <\Delta \bar{u}_{x1}, \Delta \bar{u}_{x2}, \cdots, \Delta \bar{u}_{x8}>$$

$$\{\Delta \bar{u}_y\}^T = <\Delta \bar{u}_{y1}, \Delta \bar{u}_{y2}, \cdots, \Delta \bar{u}_{y8}>$$

$$<h> = <h_1, h_2, \cdots, h_8>$$

$h_1 = 1/4\ (1+r)\ (1+s) - 1/4\ (1-r^2)\ (1+s) - 1/4\ (1+r)\ (1-s^2)$

$h_2 = 1/4\ (1-r)\ (1+s) - 1/4\ (1-r^2)\ (1+s) - 1/4\ (1-r)\ (1-s^2)$

$h_3 = 1/4\ (1-r)\ (1-s) - 1/4\ (1-r)\ (1-s^2) - 1/4\ (1-r^2)\ (1-s)$

$h_4 = 1/4\ (1+r)\ (1-s) - 1/4\ (1-r^2)\ (1-s) - 1/4\ (1+r)\ (1-s^2)$

$h_5 = 1/2\ (1-r^2)\ (1+s)$

$h_6 = 1/2\ (1-r)\ (1+s^2)$

$h_7 = 1/2\ (1-r^2)\ (1-s)$

$h_8 = 1/2\ (1+r)\ (1-s^2)$

Figure 2.1 Two-dimensional 8-node isoparametric element

2.3 Beam Element Formulation

The Equation 2.9, derived for continuum element, can be applied to the beam elements. The influence of the geometric nonlinear stiffness matrix $[K^{nl}]$ in Equation 2.9 is negligible when the displacement increments are small. Thus the global static equilibrium equations for the beam in Updated Lagrangian formulation is given by

$$[K^{l}]\{\Delta\bar{u}\} = \{P_n\} - \{R_{n-1}\} \quad (2.14)$$

Structural stiffness matrix $[K^{l}]$ and internal resisting load vector $\{R_{n-1}\}$ in the global coordinate system can be obtained by rotating element stiffness matrix $[k^{*l}]$ and element internal resisting load vector $\{r^{*}_{n-1}\}$ in local coordinate system.

$$[K^{l}] = \Sigma\, [R_{n-1}]^T [K^{*l}] [R_{n-1}] \quad (2.15)$$

$$\{R_{n-1}\} = \Sigma\, [R_{n-1}]^T \{r^{*}_{n-1}\} \quad (2.16)$$

where

$$[K^{*l}] = \int_{v_{n-1}} [B^{*l}_{n-1}]^T [D^{*ep}] [B^{*l}_{n-1}]\, dv \quad (2.17)$$

$$\{r^{*}_{n-1}] = \int_{v_{n-1}} [B^{*l}_{n-1}]^T \{\sigma^{*}_{n-1}\}\, dv \quad (2.18)$$

The elasto-plastic matrix [D^{*ep}] in Equation 2.17 relates the local beam stress increment { $\Delta\sigma^*$ } to local beam strain increment { $\Delta\varepsilon^{*l}$ }.

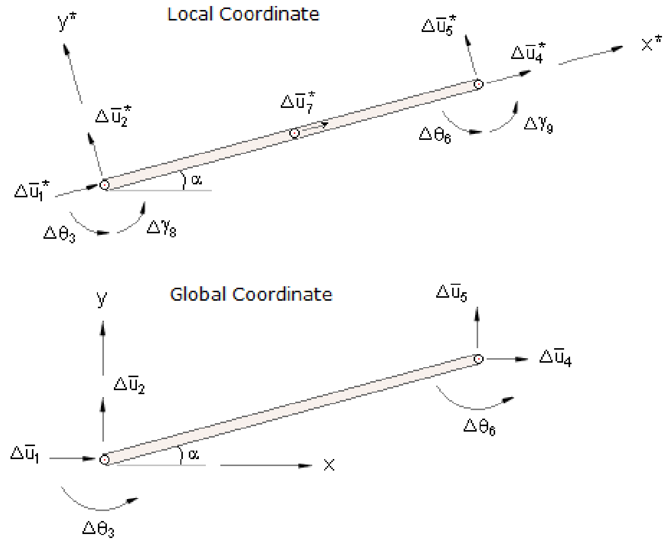
$$\{\Delta\sigma^*\} = [D^{*ep}]\,\{\Delta\varepsilon^{*l}\} \tag{2.19}$$

Displacement increments $\{\Delta u^*\}$ and strain increments $\{\Delta\varepsilon^{*1}\}$ can be expressed in terms of the element nodal displacement increments

$$\{\Delta u^*\} = [N_{n-1}]\,\{\Delta\bar{u}^*\} \tag{2.20}$$

$$\{\Delta\varepsilon^{*l}\} = [B^{*l}{}_{n-1}]\,\{\Delta\bar{u}^*\} \tag{2.21}$$

For 2-D plane beam element, element local and global degrees of freedom are shown in Figure 2.2 along with rotation matrix. Shape functions are given in Table 2.1. The shape functions in Table 2.1 assume cubic variation of flexure, quadratic variation of axial displacement and linear variation of shear rotations. It should be noted that the last three element nodal degrees of freedom $< \Delta\bar{u}_7, \Delta\gamma_8, \Delta\gamma_9 >$ are independent of the adjoining elements and they are eliminated by static condensation prior to assembling the element matrices.



$$\{ \Delta\bar{u}^* \} = [R_{n-1}] \{ \Delta\bar{u} \}$$

$$[k^I] = [R_{n-1}]^T [k^{*I}] [R_{n-1}]$$

$$[R_{n-1}] = \begin{bmatrix} [r] & [0] \\ [0] & [r] \end{bmatrix}$$

$$[r] = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 \\ -\sin\alpha & \cos\alpha & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

Figure 2.2 Degrees of freedom and rotation matrix for beam

$$\{\Delta u_x^{\star}\} = h_1\ \Delta\bar{u}_1^{\star} + h_7\ \Delta\bar{u}_7^{\star} + h_4\ \Delta\bar{u}_4^{\star} - y^{*}\ [\ h_{2,x^{\star}}\ \Delta\bar{u}_2^{\star} + h_{5,x^{\star}}\ \Delta\bar{u}_5^{\star}$$

$$+ h_{3,x^{\star}}\ (\Delta\theta_3 + \Delta\gamma_8) + h_{6,x^{\star}}\ (\Delta\theta_6 + \Delta\gamma_9) - (h_8\,\Delta\gamma_8 + h_9\,\Delta\gamma_9)\ ]$$

$$\{\Delta u_{y^{\star}}\} = h_2\ \Delta\bar{u}_2^{\star} + h_5\ \Delta\bar{u}_5^{\star} + h_3\,(\Delta\theta_3 + \Delta\gamma_8) + h_6\,(\Delta\theta_6 + \Delta\gamma_9)$$

$$\left\{ \begin{matrix} \Delta u_x^{*} \\ \Delta u_y^{*} \end{matrix} \right\} = \begin{bmatrix} N_{x^{\star}1}, & N_{x^{\star}2}, & ...., & N_{x^{\star}9} \\ N_{y^{\star}1}, & N_{y^{\star}2}, & ...., & N_{y^{\star}9} \end{bmatrix}_{n-1} \{\Delta\bar{u}^{\,*}\}$$

$$h_1 = \left(2\left(\frac{x^{*}}{L}\right) - 1\left(\frac{x^{*}}{L}\right) - 1\right)$$

$$h_2 = 1 - 3\left(\frac{x^{*}}{L}\right)^2 + 2\left(\frac{x^{*}}{L}\right)^3$$

$$h_3 = L\left(\left(\frac{x^{*}}{L}\right) - 2\left(\frac{x^{*}}{L}\right)^2 + \left(\frac{x^{*}}{L}\right)^3\right)$$

$$h_4 = \left(\frac{x^{*}}{L}\right)\left(2\left(\frac{x^{*}}{L}\right) - 1\right)$$

Table 2.1 Shape functions for beam element

$$h_5 = 3\left(\frac{x^*}{L}\right)^2 2\left(\frac{x^*}{L}\right)^3$$

$$h_6 = -L\left(\left(\frac{x^*}{L}\right)^2 - \left(\frac{x^*}{L}\right)^3\right)$$

$$h_7 = -4\left(\frac{x^*}{L}\right)\left(\left(\frac{x^*}{L}\right) - 1\right)$$

$$h_8 = 1 - \left(\frac{x^*}{L}\right)$$

$$h_9 = \left(\frac{x^*}{L}\right)$$

Table 2.1 Shape functions for beam element (continued)

In general, a beam element can be composed of composite sections with a set of reinforcing bars as shown in Figure 2.3. A beam section can be divided into top (flange), middle (web) and bottom (flange) subsections. Each subsection can be further divided into a number of layers. Numerical integration points are selected such that along the axis of beam, Gauss points are used and through the depth of the beam, the center points of each layer and each reinforcing bar are used as shown in Figure 2.3. The strain compatibilities along the boundary of subsections and between the plain body of beam and the reinforcing bar are assumed. Different material property sets can be specified to the top, middle and bottom subsections of beam. Element stiffness matrix $[k^{*I}]$ and element internal resisting force vector $\{r^{*}_{n-1}\}$ can be computed using the equations listed in Table 2.2.

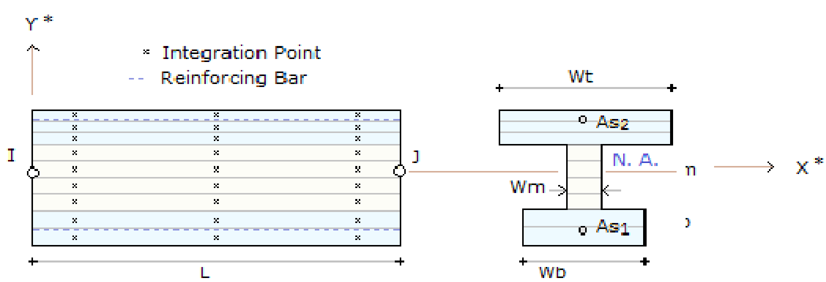


Figure 2.3 Schematic section view of layered reinforced beam

$$[k^*] = \sum_{j=1}^{N_g} H_j \left\{ \sum_{i=1}^{N_b} W_b \left(\frac{D_b}{N_b}\right) [B^{*l}(x_j^*,y_i^*)]^T_{n-1} [D_b^{*ep}(x_j^*,y_i^*)]_{n-1} [B^{*l}(x_j^*,y_i^*)]_{n-1} \right.$$

$$+ \sum_{i=N_b+1}^{N_b+N_m} W_m \left(\frac{D_m}{N_m}\right) [B^{*l}(x_j^*,y_i^*)]^T_{n-1} [D_m^{*ep}(x_j^*,y_i^*)]_{n-1} [B^{*l}(x_j^*,y_i^*)]_{n-1}$$

$$+ \sum_{i=N_b+N_m+1}^{N_b+N_m+N_t} W_t \left(\frac{D_t}{N_t}\right) [B^{*l}(x_j^*,y_i^*)]^T_{n-1} [D_t^{*ep}(x_j^*,y_i^*)]_{n-1} [B^{*l}(x_j^*,y_i^*)]_{n-1}$$

$$\left. + \sum_{i=1}^{N_r} A_{si} [B^{*l}(x_j^*,y_i^*)]^T_{n-1} [D_r^{*ep}(x_j^*,y_i^*)]_{n-1} [B^{*l}(x_j^*,y_i^*)]_{n-1} \right\}$$

$$\{r^*\}_{n-1} = \sum_{j=1}^{N_g} H_j \left\{ \sum_{i=1}^{N_b} W_b \left(\frac{D_b}{N_b}\right) [\, B^{*l}(x_j^*,y_i^*)\,]^T_{n-1} \{\, \sigma^*(x_j^*,y_i^*)\, \}_{n-1} \right.$$

$$+ \sum_{i=N_b+1}^{N_b+N_m} W_m \left(\frac{D_m}{N_m}\right) [\, B^{*l}(x_j^*,y_i^*)\,]^T_{n-1} \{\, \sigma^*(x_j^*,y_i^*)\, \}_{n-1}$$

$$+ \sum_{i=N_b+N_m+1}^{N_b+N_m+N_t} W_t \left(\frac{D_t}{N_t}\right) [\, B^{*l}(x_j^*,y_i^*)\,]^T_{n-1} \{\, \sigma^*(x_j^*,y_i^*)\, \}_{n-1}$$

$$\left. + \sum_{i=1}^{N_r} A_{si} [\, B^{*l}(x_j^*,y_i^*)\,]^T_{n-1} \{\, \sigma_r^*(x_j^*,y_i^*)\, \}_{n-1} \right\}$$

Table 2.2 Beam stiffness matrix and internal resisting load vector

H_j	Weight associated with j^{th} Gauss point
D_t	Depth of top (flange) subsection
D_m	Depth of middle (web) subsection
D_b	Depth of bottom (flange) subsection
W_t	Width of top (flange) subsection
W_m	Width of middle (web) subsection
W_b	Width of bottom (flange) subsection
A_{si}	Cross section area of i^{th} reinforcing bar
N_g	Number of Gauss points along the axis of beam
N_t	Number of layers for top (flange) subsection
N_m	Number of layers for middle (web) subsection
N_b	Number of layers for bottom (flange) subsection
N_r	Number of reinforcing bar set
x_j^*	x coordinate of J^{th} Gauss point
y_j^*	y coordinate of the centerline of i^{th} layer or i^{th} reinforcing bar.

Table 2.2 Continued

2.4 Truss Element Formulation

The same formulation (Equation 2.9) as derived for continuum element has been used for the truss element in SMAP-S2. It is rewritten here for clarity.

$$([K^{l}] + [K^{nl}]) \{\Delta\bar{u}\} = \{P_n\} - \{R_{n-1}\} \tag{2.22}$$

Since the truss element contains only the axial stress along the axis of the member, the element stiffness matrices and the internal resisting force vector can be readily formulated in the element local coordinate system and then be transformed to the global coordinate system. Thus

$$[K^{l}] = \sum [R_{n-1}]^T [K^{*l}] [R_{n-1}] \tag{2.23}$$

$$[K^{nl}] = \sum [R_{n-1}]^T [K^{*nl}] [R_{n-1}] \tag{2.24}$$

$$\{R_{n-1}\} = \sum [R_{n-1}]^T \{r^*{}_{n-1}\} \tag{2.25}$$

where

$$[k^{*l}] = \int_{v_{n-1}} <B^{*l}>^T_{n-1} E^{ep} <B^{*l}>_{n-1} dv \tag{2.26}$$

$$[K^{*nl}] = \int_{v_{n-1}} <B^{*nl}>^T_{n-1} \{\sigma_{x^*x^*}\}_{n-1} <B^{*nl}>_{n-1} dv \tag{2.27}$$

$$\{r^*{}_{n-1}\} = \int_{v_{n-1}} <B^{*nl}>^T_{n-1} \{\sigma_{x^*x^*}\}_{n-1} dv \tag{2.28}$$

The elasto-plastic Young's modulus E^{ep} in Equation 2.26 relates the axial stress increment $\Delta\sigma_{x^*x^*}$ to the axial strain increment $\Delta\epsilon^{l}_{x^*x^*}$.

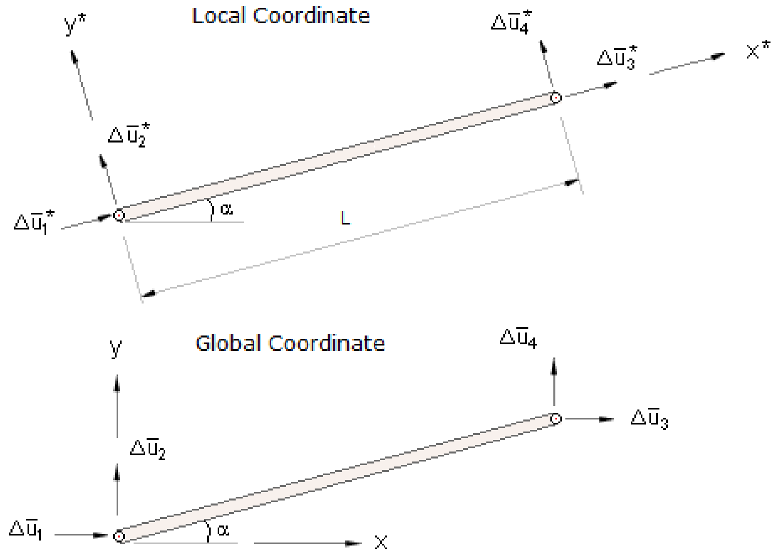
$$\Delta\sigma_{x^*x^*} = E^{ep}\,\Delta\epsilon^{l}_{x^*x^*} \tag{2.29}$$

Displacement increment $\{\Delta u^*\}$ and strain increment $\Delta\epsilon^{l}_{x^*x^*}$ can be expressed in terms of the element nodal displacement increments.

$$\{\Delta u^*\} = [N]_{n-1}\,\{\Delta\bar{u}^*\} \tag{2.30}$$

$$\Delta\epsilon^{l}_{x^*x^*} = <B^{*l}>\,\{\Delta\bar{u}^*\} \tag{2.31}$$

For 2-D plane truss element, element local and global degrees of freedom are shown in Figure 2.4 along with rotation matrix. Shape functions and element stiffness matrix in the local coordinate system are given in Table 2.3. It is assumed that the cross section area (A) remains constant during the deformation of truss member.



$$[R_{n-1}] = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 \\ -\sin\alpha & \cos\alpha & 0 & 0 \\ 0 & 0 & \cos\alpha & \sin\alpha \\ 0 & 0 & -\sin\alpha & \cos\alpha \end{bmatrix}$$

Figure 2.4 Degrees of freedom and rotation matrix for truss

Table 2.3 Shape functions and stiffness matrix for truss

$$\{\Delta u^*\} = [N]_{n-1}\,\{\Delta \bar{u}^*\}$$

$$\begin{Bmatrix} \Delta u_{x^*} \\ \Delta u_{y^*} \end{Bmatrix} = \begin{bmatrix} \left(1-\dfrac{x^*}{L}\right) & \dfrac{x^*}{L} & 0 & 0 \\ 0 & 0 & \left(1-\dfrac{x^*}{L}\right) & \dfrac{x^*}{L} \end{bmatrix} \begin{Bmatrix} \Delta\bar{u}_1^* \\ \Delta\bar{u}_2^* \\ \Delta\bar{u}_3^* \\ \Delta\bar{u}_4^* \end{Bmatrix}$$

$$[k^{*nl}] = \frac{\alpha_{x^*x^*}\,A}{L_{n-1}} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & -1 \\ -1 & 0 & 1 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

$$[k^{*l}] = \frac{E^{ep}\,A}{L_{n-1}} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

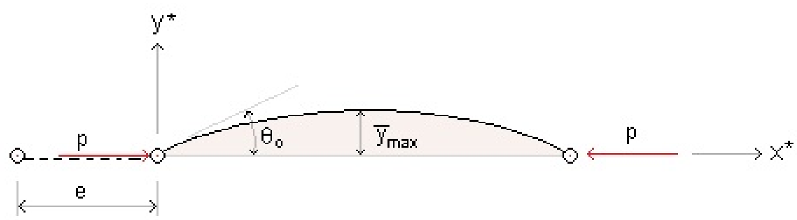


Figure 2.5 Laterally deflected truss member

Post Buckling Behavior

Figure 2.5 shows the shape of the laterally deflected truss member. The axial force (P)-axial contraction (e) relationship can be derived by the finite deflection theory.

$$\frac{P}{\left(\frac{EI}{L^2}\right)} = \alpha_0^2 \tag{2.32}$$

and

$$\frac{e}{L} = \left(1 - \frac{\beta_0}{\alpha_0}\right) \tag{2.33}$$

where

$$\alpha_0 = 2\int_0^{\frac{\pi}{2}} \frac{d\varphi}{\sqrt{1 - k^2 \sin^2\varphi}}$$

$$\beta_0 = 2\int_0^{\frac{\pi}{2}} \frac{(1 - 2k^2 \sin^2\varphi)\ d\varphi}{\sqrt{1 - k^2 \sin^2\varphi}} \tag{2.34}$$

$$k = \sin\frac{\theta_0}{2}$$

The nonlinear force-contraction curve can be developed by integrating Equation 2.34 for the given value of θ. It should be noted that the force-contraction relationship in Equation 2.32 and 2.33 is valid until the maximum extreme fiber stress ($\bar{\sigma}_{max}$) at the midspan reaches the uniaxial yield stress ($\bar{\sigma}_{yield}$) of the member. $\bar{\sigma}_{max}$ is given by

$$\bar{\sigma}_{max} = 2\,E\,\bar{y}_{max}\,\alpha_0\,k\,/\,L + P\,/\,A \qquad (2.35)$$

where $\bar{y}_{max}$ is the maximum distance from the neutral axis to the extreme fiber.

2.5. Multi-Staged Excavation and Construction Simulation

The sequence of excavation or construction events has a significant influence on the final states of stress and displacement fields. The source of nonlinearity in this case comes from the geometry or boundary change. Therefore the general nonlinear solution scheme can be used.

SMAP-S2 has an option of element activity to simulate the excavation or construction sequence. For those elements to be excavated, they are active initially and are deactive at a designated step when excavation occurs. For those elements to be constructed, such as support systems, they are deactive initially and are active at a designated step when the installation takes place. For those elements to be constructed temporarily, such as temporary support system, they are active at a step when installed and are deactive at a step when removed.

Nonlinear Material Models

3.1 Generalized Hoek & Brown Model

3.1.1 Introduction

SMAP-S2 employs Generalized Hoek and Brown Model developed by Kim, Piepenburg and Merkle (1986). The material model is based on theory of plasticity with the associated flow rule. The model in a generalized form can simulate Von Mises, Mohr-Coulomb, Drucker-Prager, and In situ rock (Hoek and Brown) models.

3.1.2 Failure Equation

The generalized form of failure equation is given by

$$F(p_i, q_i, \theta) = q - \{ (\alpha + \beta p)^n + \kappa \} R(\theta) = 0 \qquad (3.1)$$

where

$$P = \frac{1}{3} \sigma_{ii}$$

$$S_{ij} = \sigma_{ij} - p\,\delta_{ij}$$

$$J_2 = \frac{1}{2} s_{ij} s_{ij}$$

$$J_3 = \frac{1}{3} s_{ij} s_{jk} s_{ki}$$

$$q = \sqrt{3 J_2}$$

$$\theta = \frac{1}{3} \sin^{-1}\left(- \frac{27}{2} \frac{J_3}{q^3} \right)$$

3.1.3 Elastic Stress-Strain Relationship

The incremental elastic constitutive law can be expressed in the following matrix form:

$$\{d\sigma\} = [D^e] \{d\varepsilon^e\} \tag{3.2}$$

where $\{d\varepsilon^e\}$ is the elastic strain increment.

3.1.4 Yield Surface

The material model does not allow hardening. Thus the yield surface coincides with the failure surface as given by Equation 3.1. Materials behave elastically under the failure surface, and perfectly plastic on the failure surface. The expression for $R(\theta)$ in Equation 3.1 is given by

$$R(\theta) = \frac{x(\sqrt{3}\cos\theta + \sin\theta) + (2k-1)\left[(2+\cos 2\theta + \sqrt{3}\sin 2\theta)x + 5k^2 - 4k\right]^{\frac{1}{2}}}{\left[x(2+\cos 2\theta + \sqrt{3}\sin 2\theta) + (1-2k)^2\right]} \tag{3.3}$$

$$\left(-\frac{\pi}{6} \le \theta \le \frac{\pi}{6} \right)$$

where $x=(1-k^2)$ and k is the ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same mean pressure. As shown in Figure 3.1, R(θ) varies from a smooth-cornered triangle to a circle depending on the value of k.

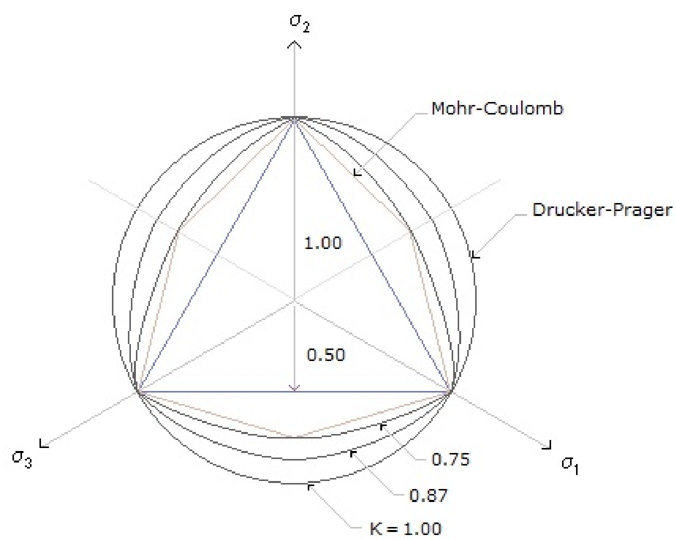


Figure 3.1 Shape of strength envelope, R(θ), on octahedral plane

3.1.5 Flow Rule

An associated flow rule is assumed. Thus the plastic strain increment, $\{d\varepsilon^p\}$ is in the direction normal to the failure surface, which will allow dilatancy at failure. In mathematical terms

$$\{d\varepsilon^p\} = d\lambda \{a\} \tag{3.4}$$

where $d\lambda$ is the non-negative proportionality constant and $\{a\}$ is the column vector of derivatives of the yield function with respect to the stress components.

$$\{a\} = \left\{ \frac{\partial F}{\partial \sigma} \right\}$$

3.1.6 Consistency Equation

During Yielding, the consistency equation forces the stress to move along the yield surface.

$$dF = \{a\}^T \{d\sigma\} = 0 \tag{3.5}$$

3.1.7 Incremental Elasto-Plastic Constitutive Law

Total strain is defined as the sum of elastic and plastic strains

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^p\} \tag{3.6}$$

Substituting Equation 3.6 into 3.2, we have

$$\{d\sigma\} = [D^e] \{d\varepsilon\} + \{d\varepsilon^p\} \tag{3.7}$$

From the flow rule defined in Equation 3.4, we can rewrite Equation 3.7 as

$$\{d\sigma\} = [D^{e}]\{d\varepsilon\} - d\lambda\,[D^{e}]\{a\} \tag{3.8}$$

Now substituting Equation 3.8 into Equation 3.5 and solving for dλ, we obtain

$$d\lambda = \frac{\{a\}^{T}\{D^{e}\}\{d\varepsilon\}}{\{a\}^{T}\{D^{e}\}\{a\}} \tag{3.9}$$

Backsubstituting this dλ into Equation 3.8, the stress increment is directly related to the total strain increment as follows:

$$\{d\sigma\} = [D^{ep}]\{d\varepsilon\} \tag{3.10}$$

Where incremental elasto-plastic constitutive matrix is expressed as

$$[D^{ep}] = [D^{e}] - \frac{[D^{e}]\{a\}\{a\}^{T}[D^{e}]}{\{a\}^{T}[D^{e}]\{a\}} \tag{3.11}$$

3.1.8 Determination of Material Parameters

The constant, n, determines the type of strength envelope in the p-q plane. Table 3.1 shows the material parameters and required tests. For n=1/2, the model represents the Hoek and Brown Model which is based on extensive field and laboratory data. The material parameters for this empirical model are tabulated in Table 3.2 for several different rock types as a function of rock quality.

Table 3.1 Material parameters and required tests

n	Required tests	Strength envelope in terms of major and minor principal stresses	Material constants for model (α, β, κ, and K)	Remarks
0	Uniaxial stress	$\sigma_1 - \sigma_3 = \bar{q}$	α and β are arbitrary $\kappa = \bar{q} - 1$ $K = 1$	Von Mises Model
1/2	Triaxial compression Triaxial extension	$\sigma_1 - \sigma_3 =$ $(m\,\sigma_c\,\sigma_3 + s\,\sigma_c^{\,2})^{1/2}$ σ_c : Unconfined compression strength of intact rock m and s are material constants depending on rock type and quality	$\alpha = (m^2/36+s)\,\sigma_c^{\,2}$ $\beta = m\,\sigma_c$ $\kappa = -1/6\;m\,\sigma_c$ $K = q_e/q_c$ q_e and q_c are triaxial extension and compression shear strengths respectively, measured at same mean pressure	Hoek and Brown Model Approximate values of m and s are available from the empirical data base by Hoek and Brown
1	Triaxial compression Triaxial extension	$\sigma_1 - \sigma_3 = (\sigma_1 + \sigma_3)$ $\sin\phi + 2 \cdot C \cdot \cos\phi$ ϕ : Internal friction angle c : Cohesion	$\alpha + \kappa =$ $C\;6\cos\phi\;/\;(3-\sin\phi)$ $\beta = 6\sin\phi/(3-\sin\phi)$ $K = q_e\;/\;q_c$	For Drucker-Prager Model, K= 1 For Smooth-cornered Mohr-Coulomb Model, $K = (3-\sin\phi)\;/\;(3+\sin\phi)$

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

3.2 Joint Model

3.2.1 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 3.2) as described in Card 5.4 of SMAP-S2 User's Manual.

3.2.2 Strain-Displacement Relation

Strains in the joint local coordinate are

$$\{\Delta\varepsilon\} = \begin{Bmatrix} \Delta\gamma_{xy}' \\ \Delta\varepsilon_{yy}' \end{Bmatrix} \tag{3.12}$$

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:

$$\{\Delta u'\} = [\beta]\ \{\Delta u\} \tag{3.13}$$

where

$$\{\Delta u'\} = \begin{Bmatrix} \Delta u_x' \\ \Delta u_y' \end{Bmatrix}$$

$$\{\Delta u\} = \begin{Bmatrix} \Delta u_x \\ \Delta u_y \end{Bmatrix}$$

[β] Coordinate transformation matrix

Strain-displacement relation in the local coordinate is given by

$$\{\Delta\epsilon'\} = \frac{1}{\delta}\{\Delta u'\} \tag{3.14}$$

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

$$\{\Delta u\} = [h]\{\Delta\bar{u}\} \tag{3.15}$$

Now, substituting Equations 3.13 and 3.15 into the Equation 3.14, we obtain

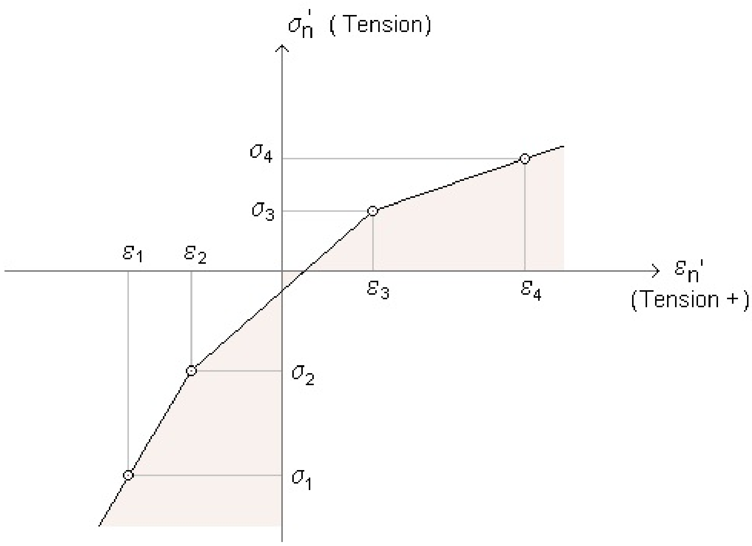
$$\{\Delta\epsilon'\} = [B]\{\Delta\bar{u}\} \tag{3.16}$$

$$[B] = \frac{1}{\delta}[\beta][h] \tag{3.17}$$

where

3.2.3 Normal Stress-Strain Relation

Normal Stress-strain relation is assumed to be nonlinear elastic as shown in the next page.



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 $\varepsilon_2 \leq \varepsilon_n' < \varepsilon_3$

$$E = \frac{\sigma_3 - \sigma_2}{\varepsilon_3 - \varepsilon_2} \tag{3.18}$$

For $\varepsilon_n' \geq \varepsilon_3$

$$E = \frac{\sigma_4 - \sigma_3}{\varepsilon_4 - \varepsilon_3}$$

3.2.4 Shear Stress-Strain Relation

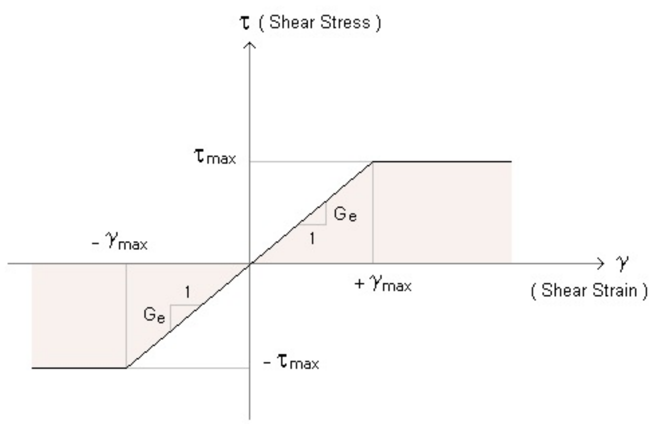
The shear strength of joint is assumed to follow Mohr-Coulomb failure criterion.

$$\tau_{max} = C - \sigma_n' \tan \varphi \tag{3.19}$$

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 the strength envelope as shown below:



Thus, shear modules (G) is computed as follow:

For $|\gamma| < \gamma_{max}$,

$$G = G_e \tag{3.20}$$

For $|\gamma| \geq \gamma_{max}$,

$$G = 0$$

Note that G_e is the elastic shear modulus.

3.2.5 Element Stiffness Matrix

Joint stress-strain relation can be given by

$$\{\Delta\sigma'\} = [C'] \{\Delta\varepsilon'\} \tag{3.21}$$

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:

$$[K] = \int_v [B]^T [C'] [B] \, dv \tag{3.22}$$

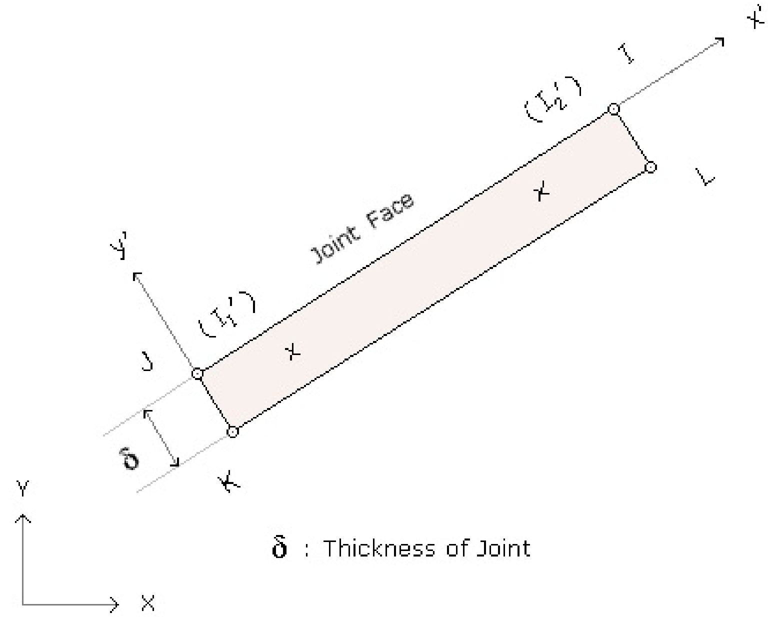


Figure 3.2 Local and Global Coordinates of Joint Element

References

Britvec, S.J. The Stability of Elastic Systems, University of Stuttgart, Pergamon Press Inc, 1973.

Burns, N.H. and Siess, C.P., Load-Deformation Characteristics of Beam-Column Connections in Reinforced Concrete, Civil Engineering Studies, SRS No. 234, Univerisity of Illinois, Urbana, Illinois, January 1962.

COMTEC RESEARCH, TUNA (Tunnel Analysis) User's Manual, Version 1, May 1990.

COMTEC RESEARCH, SMAP-S2 (Structure-Medium Analysis Program) User's Manual, Version 1, May 1991.

David E. Van Dillen, etc., Modernization of the BMINES Computer Code Vo. 1 User's Guide, Agbabian Associates, Sep. 1981.

Hoek, E. and E.T. Brown, Empirical Strength Criterion for Rock masses, Journal of the Geotechnical Engineering Division, ASCE, vol. 106, No. GT9, September 1980.

Kim, K.J. and M. Davister, Numerical Analysis of Nonlinear Liner-Medium Interaction, Volume II - Cylindrical Tunnel Subjected to Axisymmetric loading, Prepared for Defence Nuclear Agency, Contract No. DNA 001-84-c-0149, July, 1987.

Kim, K.J., D.D. Piepenburg, and D. Merkle, Influence of the Intermediate Principal Stress on Rock Tunnel Behavior, Applied Research Associates, Inc., Alexandria, VA, January 1986.

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.