SMAP ® - 3D

Structure Medium Analysis Program

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

User's Manual Version 7.06

COMTEC RESEARCH

Printed in the United States of America.

LICENSE AGREEMENT

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

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

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

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

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

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

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

Contents

Introduction

1.1 Overview

SMAP-3D is an advanced three-dimensional finite element computer program developed for the geometric and material nonlinear structure-medium interaction analysis. The program can be a powerful tool for the geomechanical analysis since it can solve static, consolidation and dynamic problems in dry, partially saturated or fully saturated soils and porous rock mass. The program has been designed to integrate the pre-, main-, and post-processors as shown at the end of this section.

1.2 Features

Features of SMAP-3D include:

- Three-dimensional isoparametric continuum element
  - Models soils, rocks and concrete media
  - Allows yielding and tension cut-off
  - Models dry, saturated and partially saturated porous media
- Joint element
  - Models faults, joints, and interfaces
  - Allows sliding and debonding

- Shell element
  - Models slabs, shear walls, tunnel linings, etc.
  - Considers membrane deformation, bending and torsional resistance
- Beam element
  - Models structural frames
  - Considers axial deformation, bending and torsional resistance
- Truss element
  - Models rock bolts and anchor bar
  - Allows yielding, buckling and post-buckling
- External loads
  - Pressure time history
  - Displacement/Velocity/Acceleration time history
  - Initial velocity
  - Gravity load
  - Base acceleration time history
- Special boundary condition
  - Skew boundary (Not Available)
  - Transmitting boundary
- Simulation of a sequence of excavation and construction
- Nonlinear material model
  - Von Mises model
  - Mohr-Coulomb model
  - In Situ Rock model
  - JWL Explosive Source model
  - Modified Cam-Clay model
  - Hyperbolic model
  - Engineering model
  - User defined model
- Large deformations
  - Use updated Lagrangian

1.3 Applications

Applications of SMAP-3D include:

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

Overview of SMAP-3D Program Structure

USER INPUT	User prepares Mesh, Main, and Post Files according to SMAP-3D 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-3D	Main-processor executing Mesh and Main Files to compute displacements, stresses and strains. Output files include:

CONTSS.DAT	Stresses/strains in continuum
SHELMEF.DAT	Shell member end forces
SHELSM.DAT	Shell stresses/moments
BEAMSF.DAT	Section forces in beam
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
- Principal stress distribution
- Section forces in beam elements
- Axial force/stress/strain in truss element
- Contours of stresses and factor of safety
- 3D iso surface of stresses and strains
- Time histories of displacements/stresses/strains

Installing SMAP -3D

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

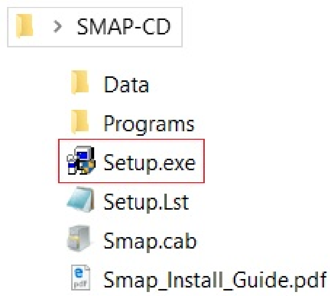
1. Uninstall if there are pre-existing SMAP programs.
   To uninstall SMAP programs, remove following program using Add/Remove in Control Panel:
      SMAP
   Delete following files if they are existing:
      C:\Program Files\Smap
      C:\Windows\Setup1.exe
   Rename or delete following folders if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

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

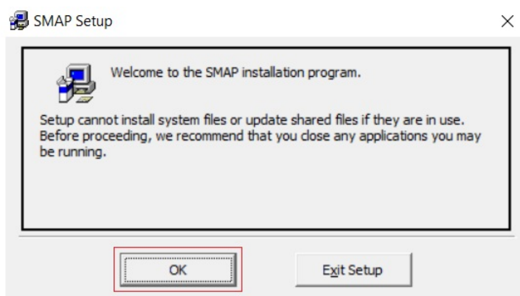
3. Run SMAP-CD.exe
   SMAP-CD folder will be created with SMAP installation programs

2-2 Installing SMAP-3D

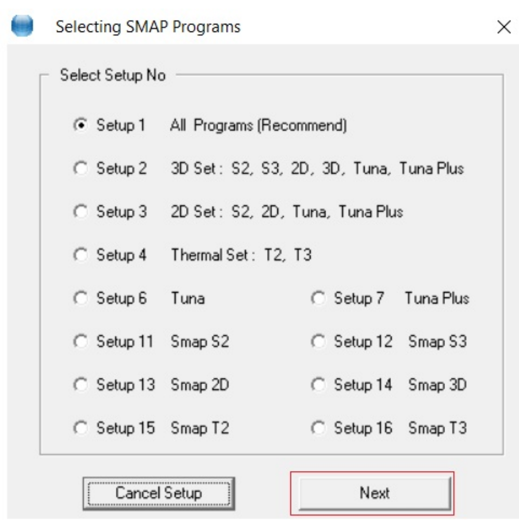
4. Double-click **Setup.exe**



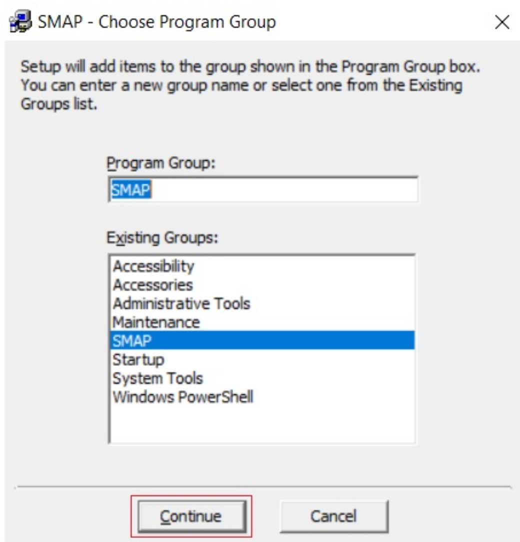
5. Click **OK**



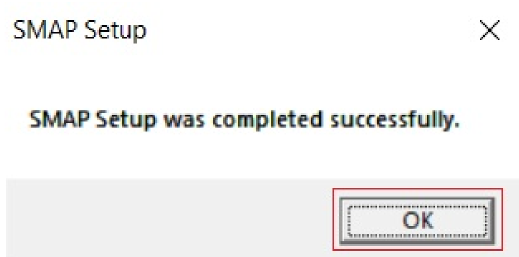
6. Click **Next**
   It will take few minutes.
   Wait until next step.



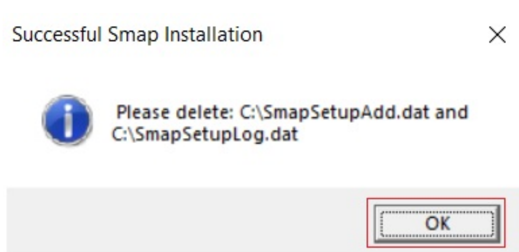
7. Click **Continue**



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

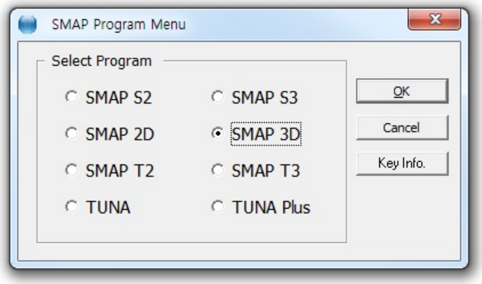
3.1 Introduction

Generally, SMAP-3D 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-3D is the one which computes static, consolidation and dynamic response of three-dimensional problems. Post-processing programs are used to show graphically the results from the main-processing program.

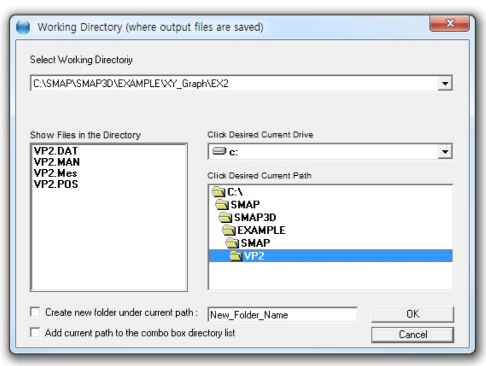
Accessing SMAP-3D Programs

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-3D** radio button and then click **OK** button.

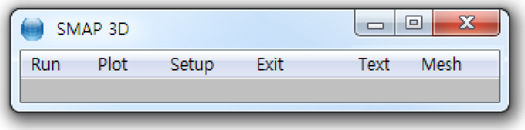


3. Next, you need to select **Working Directory**. Working Directory should be the existing directory where all the output files are saved. It is a good idea to have all your input files for the current project in this Working Directory. Click the disk drive, double-click the directory, and then **OK**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

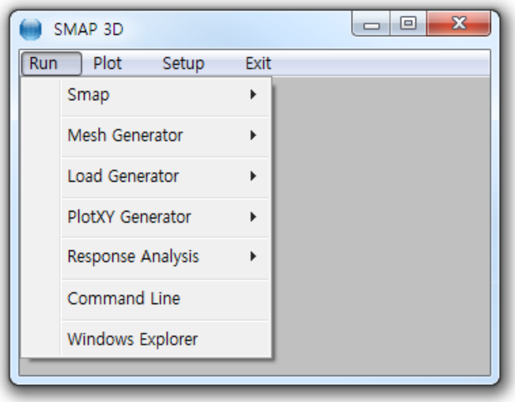


SMAP-3D Menu

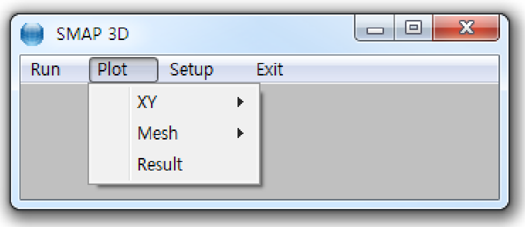
SMAP-3D provides following Main Menus; Run, Plot, Setup, Exit, Text and Mesh.



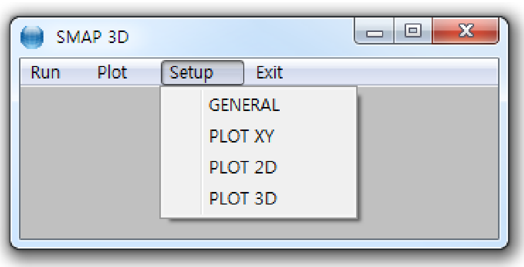
RUN executes main- and pre-processing programs and has following Sub Menus;
Smap,
Mesh Generator,
Load Generator,
PlotXY Generator,
Response Analysis,
Command Line and
Windows Explorer.



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



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



EXIT is used to end SMAP-3D.

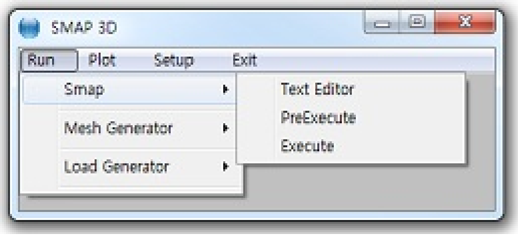
TEXT is used to edit Text files. **MESH** is used to plot F. E. Mesh files.

3.2 RUN Menu

3.2.1 SMAP

Once you have prepared the input files (Mesh, Main, and Post) according to the SMAP-3D User's Manual in Section 4, you are ready to execute SMAP-3D 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-3D main-processing program.

SMAP-3D Output Files

Once you execute SMAP-3D, generally you can obtain following output files:

CONTSS.DAT	Contains stresses/strains in continuum element
SHELSF.DAT	Contains shell member end section forces
SHELSM.DAT	Contains shell stresses/moments
SHELRB.DAT	Contains shell reinforcing bar axial stresses
BEAMSF.DAT	Contains section forces in beam element
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-3D Graphical Output

SMAP-3D Post-processing programs can generate the following graphical output:

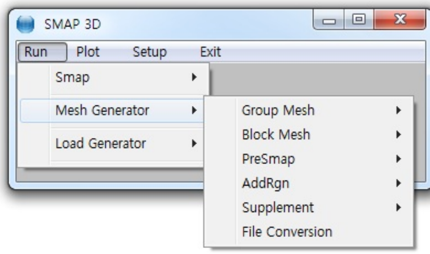
- Finite element mesh
- Deformed shape
- Principal stress distribution
- Section forces in beam element
- Extreme fiber stresses/strains in beam elements
- Axial force/stress/strain in truss element
- Contours of stresses, strains and factor of safety
- 3D iso surface of stresses and strains
- Time histories 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 and three dimensional finite element meshes that are used as Mesh File.

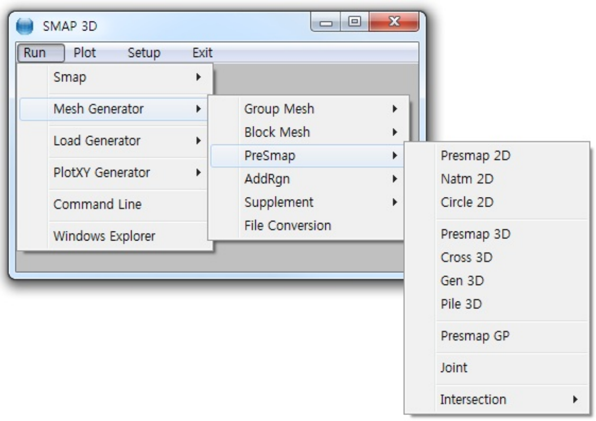
MESH GENERATOR Menu has the following Sub Menus; Group Mesh, Block Mesh, PreSmap, AddRgn, Supplement, and File Conversion.



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-3D 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-3D Example Problems describes in detail about running Block Mesh.

PRESMAP menu includes two and three dimensional pre-processing programs to generate finite element meshes: Section 7 in SMAP-3D Example Problems describes in detail about running PRESMAP Programs.



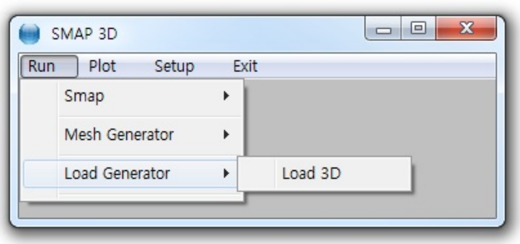
ADDRGN is the pre-processing program which has the following two basic functions: Combine two different meshes and modify existing meshes. Section 8 in SMAP-3D Example Problems describes in detail about running ADDRGN programs.

SUPPLEMENT contains supporting programs which are useful to prepare input data for pre- and main-processing programs. Section 9 in SMAP-3D Example Problems describes in detail about running SUPPLEMENT programs.

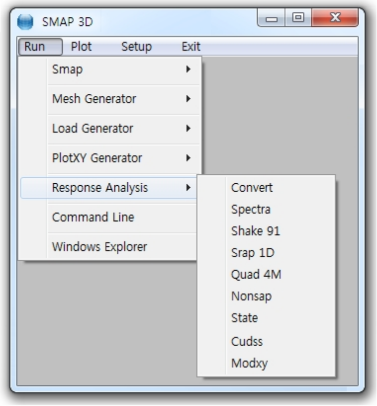
FILE CONVERSION is to convert Mesh File formats between different programs. IGES or FEMAP (Version 4.1- 4.5) can be converted to SMAP Mesh File format. Section 10 in SMAP-3D User's Manual describes in detail about running FILE CONVERSION program.

3.2.3 LOAD GENERATOR

LOAD GENERATOR includes the pre-processing program **LOAD-3D** which generates nodal values of external forces, specified velocities, initial velocities, accelerations and transmitting boundaries.
Section 10 in SMAP-3D Example Problems describes in detail about running **LOAD-3D** 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-3D User's Manual describes in detail about running PlotXY Generator program.

3.2.5 RESPONSE ANALYSIS

RESPONSE ANALYSIS runs following programs for seismic analysis:

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

3.2.6 COMMAND LINE

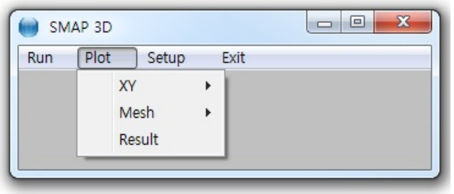
COMMAND LINE opens Windows Command Prompt at the current Working Directory. You can use a keyboard to navigate, access, and modify files and folders by entering commands. For example, COMMAND LINE is used when executing manually SMAP main solvers.

3.2.7 WINDOWS EXPLORER

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

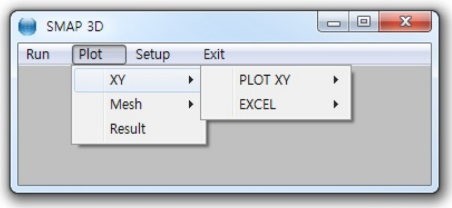
3.3 PLOT Menu

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



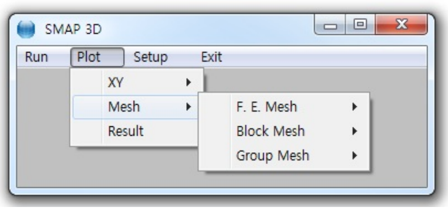
3.3.1 XY

XY graph can be displayed by PLOT-XY or EXCEL. Section 11 in SMAP-3D 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.

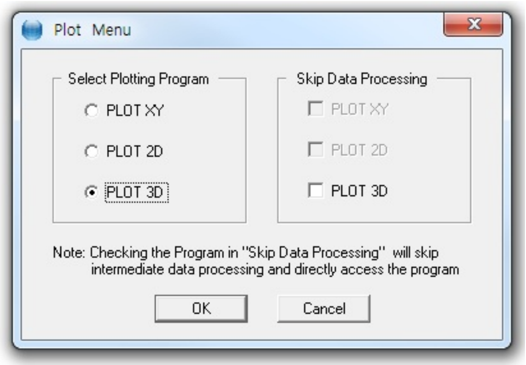
Block Mesh is used to open or build Block Mesh. Section 6 in SMAP-3D Example Problems describes in detail about running Block Mesh.

Group Mesh is used to open or build Group Mesh. Section 5 in SMAP-3D Example Problems describes in detail about running Group Mesh.

3.3.3 RESULT

Once you finished executing SMAP-3D 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.

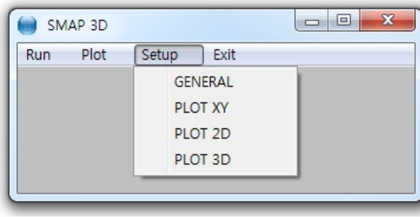
Note: When you first plot results, do not check the check box in Skip Data Processing. When you replot results, however, you can check the check box to skip intermediate data processing. This will save time and keep modified output data.

3.4 SETUP Menu

You need to run SETUP Menu

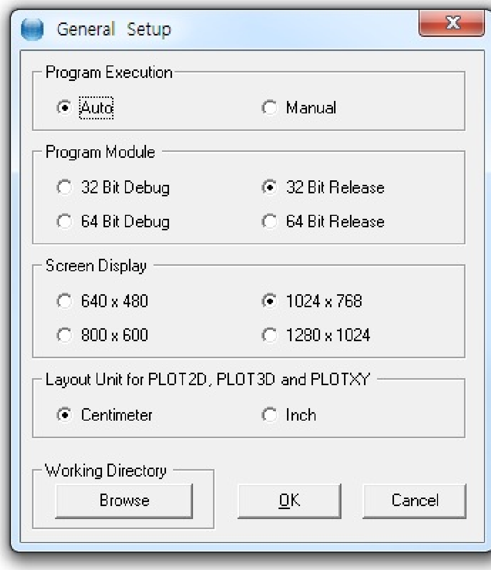
- To specify SMAP-3D main-processing program module.
- To adjust scales of graphical outputs from PLOT-XY, PLOT-2D, and PLOT-3D

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.

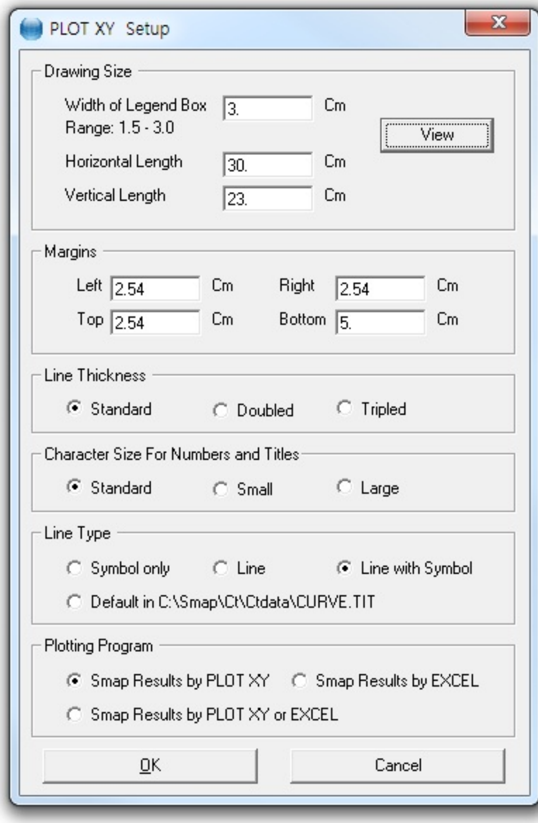
Screen Display has four options; 640x480, 800x600, 1024x768, and 1280x1024. This will affect the size of child window in PLOT-XY and PLOT-2D.

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.

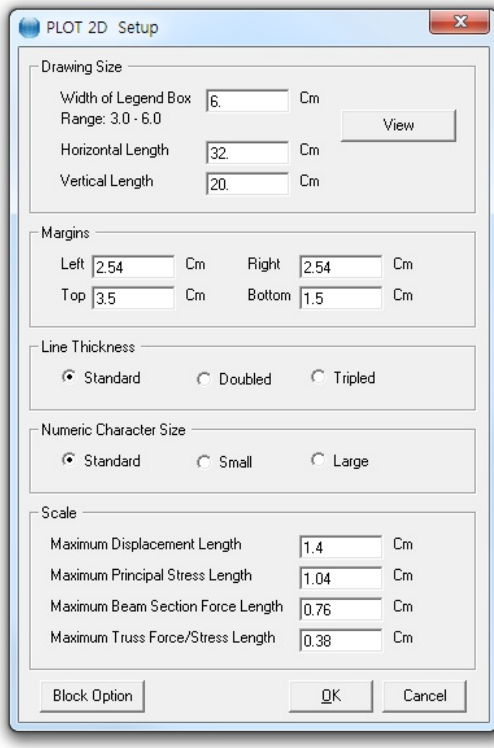
Plotting Program is used to specify default program to plot Smap results. It has three options; PLOT-XY, EXCEL, and PLOT-XY or EXCEL. Last option is to select either PLOT-XY or EXCEL at the time you plot results.

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.

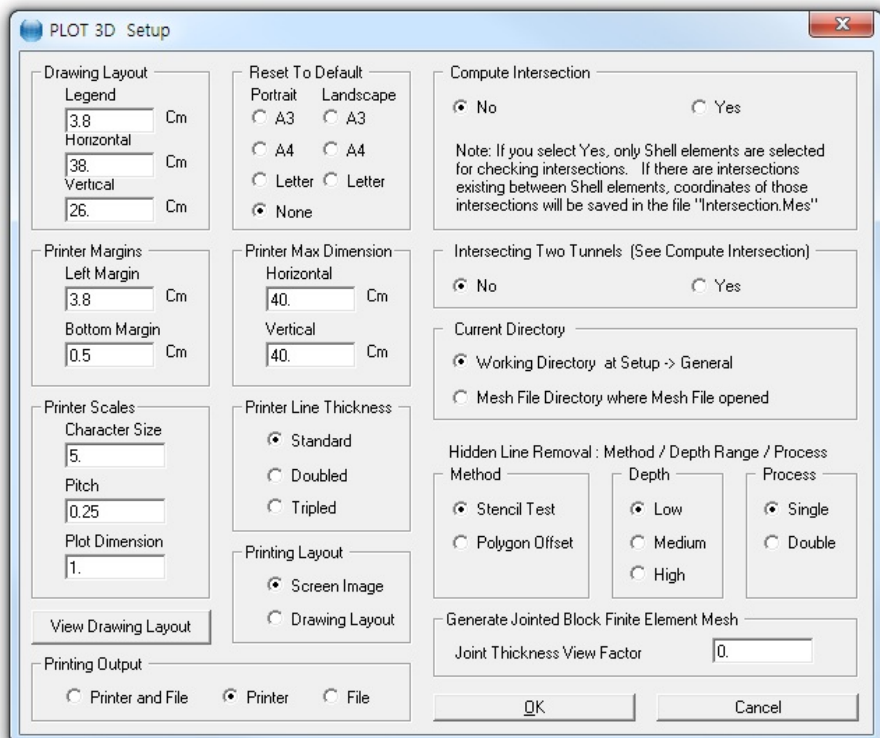
Block Option specifies options to generate either PRESMAP Output or Block Diagram.



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-3D

Occasionally, you need to execute SMAP-3D 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\Smap3D
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-3D 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_3D.dat" in the directory C:\Smap\Ct\Ctdata\Debug

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

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

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

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

   IOUTDEB =   0 : Debug information on screen.
               1 : Debug information on file,
                   Smap_3D.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-3D
User's Manual

4.1 Introduction

To run SMAP-3D 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 three-dimensional numerical analysis of static, consolidation, or dynamic problems.

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

4.2 Project File

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

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

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

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

4.3 Mesh File

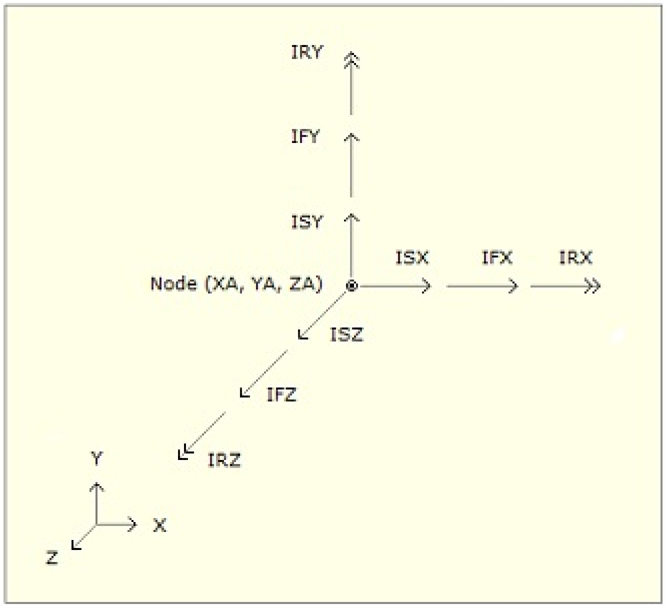
Mesh File contains nodal coordinates, boundary conditions, element indexes and material property numbers. This Mesh File is normally generated by Mesh Generator programs.

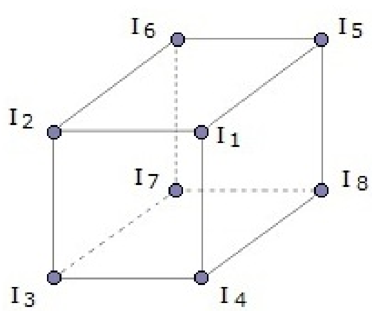
To plot Mesh File, select Mesh in Plot menu.

Mesh File

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

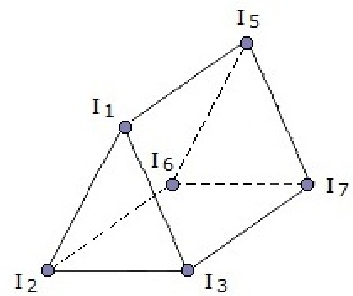
Card Group	Input Data and Definitions (Mesh File)
2 Coordinate	2.1 LABEL2A [Character string] LABEL2B [Character string] LABEL2A Label for coordinate LABEL2B Label for Card 2.2
	2.2 NUMNP Cards: NODE, ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ, IEX, IEY, IEZ, XA, YA, ZA - - - - - - - - - - - - NODE Node Number ISX Skeleton X DOF (Degree of Freedom) ISY Skeleton Y DOF ISZ Skeleton Z DOF IFX X DOF for relative pore fluid motion IFY Y DOF for relative pore fluid motion IFZ Z DOF for relative pore fluid motion IRX Rotational DOF about X axis for bending IRY Rotational DOF about Y axis for bending IRZ Rotational DOF about Z axis for bending IEX Slip X DOF IEY Slip Y DOF IEZ Slip Z DOF ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ, IEX, IEY, IEZ = 0 Free to move in specified direction = 1 Fixed in specified direction

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

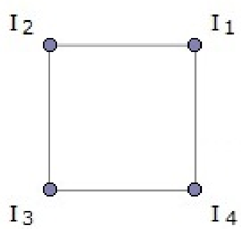
Card Group	Input Data and Definitions (Mesh File)
3 Continuum Element (If NCONT = 0, skip this card group)	3.1 LABEL3A [Character string] LABEL3B [Character string] LABEL3A Label for continuum element LABEL3B Label for Card 3.2
	3.2 NCONT Cards ┌ NEL, I_1, I_2, I_3, I_4, I_5, I_6, I_7, I_8, MATC, KS, KF, │ INTR, INTS, INTT, TBJWL └ - - - - - - - -  NEL Element number I_1,I_2,I_3,I_4,I_5,I_6,I_7,I_8 Element corner node numbers MATC Material property number

Card Group	Input Data and Definitions (Mesh File)
3 Continuum Element	3.2 KS = -1 Element has high explosive solid phase = 0 Element has solid phase 3D continuum = 1-6 Element has joint and KS represents face designation number. Refer to description in the following page. = 15 Element has SHELL element KF = 0 Element has fluid phase = 1 Element has no fluid phase INTR Use INTR = 2 INTS Use INTS = 2 INTT Use INTT = 2 TBJWL Detonation time (Required for KS = -1) Time from initial detonation to the detonation of this element.

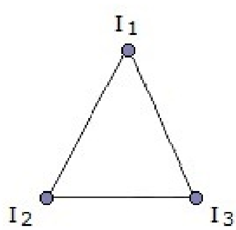
Wedge Element (I_4 = I_8 = 0)



Quadrilateral Shell Element ($I_5 = I_6 = I_7 = I_8 = 0$)



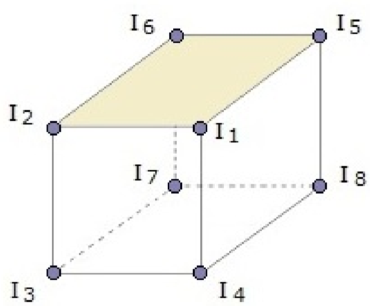
Triangular Shell Element ($I_4 = I_5 = I_6 = I_7 = I_8 = 0$)



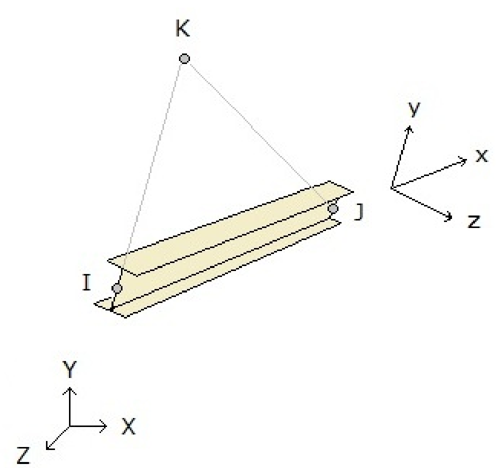
Joint Element Face Designation

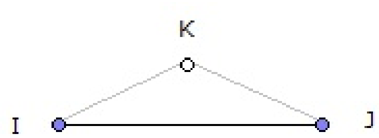
KS	I_1'	I_2'	I_3'	I_4'
1	5	6	2	1
2	6	7	3	2
3	7	8	4	3
4	8	5	1	4
5	1	2	3	4
6	6	5	8	7

For KS = 1



It should be noted that the thickness of joint element is determined
Not by gap between two faces (I_5 I_6 I_2 I_1 and I_8 I_7 I_3 I_4),
But by joint thickness (t) specified in Card 5.3.2.4.11 in Main File input
The nodal coordinates of I_5, I_6, I_2, I_1 represent the Location of Joint Face
but nodal coordinates of I_8, I_7, I_3, I_4 are used only For Plotting Purpose

Card Group	Input Data and Definitions (Mesh File)
4	4.1 LABEL4A [Character string] LABEL4B [Character string] LABEL4A Label for beam element LABEL4B Label for Card 4.2
Beam Element (If NBEAM = 0, skip this card group)	4.2 NBEAM Cards ┌ NEL, I, J, K, MSEC │ - - - - - └ - - - - - NEL Beam element number I, J Node number at beam end points K Reference node number MSEC Material section number  Local beam reference system Axis x and y are in plane defined by plane I, J, K nodes Axis x is defined by line I - J Axis z is perpendicular to the I, J, K plane

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

4.4 Main File

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

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

Main File consists of ten different card groups:

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

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

Card Group	Input Data and Definitions (Main File)
2 Analysis Type	2.1 NTCSF, NLNR, NGEN, IQUAD, NTEMP, ITDIS, MODAL NTCSF = 1 Static analysis = 2 Consolidation analysis = 3 Dynamic analysis (Implicit method) = 4 Dynamic analysis (Explicit method) = 5 Mode superposition analysis For NTCSF = -5, computes only natural frequencies and mode shapes NLNR = 0 Linear elastic material = 1 Nonlinear material NGEN = 0 Small displacement = 1 Large displacement (Updated Lagrangian) IQUAD = 0 No automatic generation = 1 Automatic generation of quadratic elements All linear continuum elements are automatically transformed into quadratic elements. For IQUAD = 16, transformed into 16 node hexahedral elements. NTEMP = 0 Thermal expansion is not considered = 1 Thermal properties and element temperatures are read from input file ELTEMP.DAT that should be located in working directory. See Table in the next page Output motions when base acceleration applied ITDIS = 0 Relative displacement & Relative velocity = 1 Total displacement & Total velocity Modal analysis options for NTCSF = 5 or -5 MODAL = 0 Subspace iteration method = 1 Determinant search method = 2 Jacobi iteration method

Input File ELTEMP.DAT

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

TITLE Project title (Max = 80 characters)</td></tr>
<tr><td colspan="2">2.1
LABEL 1 [Character string]
-
LABEL 6 [Character string]

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

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

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

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

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

Input File ELTEMP.DAT

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

Input File ELTEMP.DAT

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

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

Card Group	Input Data and Definitions (Main File)	
3 Computational Parameters	3.1.1 If NDTGR = 0, go to Card Group 3.1.3 ICYCLTIME ICYCLTIME = 0 Selection of time step is based on Cycle No = 1 Selection of time step is based on Time	
	3.1.2 For Each Time Step Group	3.1.2.1 STIME, ITYPE STIME Starting Cycle No for ICYCLTIME = 0 Starting Time (t_o) for ICYCLTIME = 1 For the first time group, use STIME = 0 ITYPE = 0 Constant time step = 1 Constant log time step = 2 Arbitrary specified time step If ITYPE = 0 DT DT Time step If ITYPE = 1 DT_1, CLDT DT_1 Starting time step CLDT Constant log time step $CLDT = \log_{10}(t_{i+1}-t_o) - \log_{10}(t_i - t_o)$ If ITYPE = 2 NUMDT $DT_1, \ldots, DT_{NUMDT}$ NUMDT Number of time step $DT_1, \ldots, DT_{NUMDT}$ Listing of specified time steps

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

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

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

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

Table 1 Values of β and θ for $\gamma = 1/2$ *

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

*$\gamma = 1/2$ indicates no damping

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

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

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

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="2">4
Coordinates</td><td rowspan="2">4.4
Boundary Code Change</td><td>4.4.1

NBNODE, NCLBCH, IFLCOD

NBNODE — Number of nodes where boundary codes are changed

NCLBCH — Cycle No where boundary codes are changed

IFLCOD = 0 — Read Card 4.4.2 here
= 1 — Read Card 4.4.2 from file NewBcode.dat starting with NBNODE as first card

If NBNODE = 0, go to next Card Group 4.5</td></tr>
<tr><td>4.4.2

NBNODE Cards:
$NODE_1$, ISX_1, ISY_1, ISZ_1, IFX_1, IFY_1, IFZ_1, IRX_1, IRY_1, IRZ_1
- - - - - - -
- - - - - - -

Refer to Card Group 2.2 in Mesh File in page 4-5 for description</td></tr>
</table>

Card Group		Input Data and Definitions (Main File)
4 Coordinates	4.5 Repeating Nodes	4.5.1 NREPEAT NREPEAT Number of repeating nodes If NREPEAT = 0, go to next Card Group 5.1
		4.5.2 NREPEAT Cards: NODER, NODEP - - - - - - NODER Repeating node NODEP Parent node Note : Repeating node NODER shares the same degrees of freedom as those of the corresponding parent node NODEP

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 IPFORM, NSPTC, IEDOF IPFORM Use IPFORM = 0 NSPTC = 0 Compute stresses and strains at integration points = 1 Compute stresses and strains at center of element For shell elements, use NSPTC = 0 IEDOF = 0 Do not include incompatible extra DOF = 1 Include incompatible extra DOF

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

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

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

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

5.3.2.3.2

Solid Grain Property
NG, BKG, SGG, CO, VO, S, PB

NG	= 0	Constant grain modulus
	= 1	Nonlinear grain modulus
BKG		Initial bulk modulus of grain
SGG		Initial specific gravity
CO		Initial wave velocity at relatively low pressure*
VO		Initial Poisson's ratio*
S		Experimentally determined constant relating loading wave velocity to peak particle velocity. Generally equal to about 1.5 for most rocks and minerals*
PB		Threshold pressure beyond which material tends to behave like a fluid

(*) Not used for NG = 0

Table 2 Permeability Constants

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

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.3 Fluid and Solid Grain Property (NF = 1)	5.3.2.3.2 Pore Fluid Property NW, BKF, SGF, SO, GAMMA, PAO, T NW = 0 Constant fluid modulus = 1 Nonlinear modulus (Fresh water) = 2 Nonlinear modulus (Sea water) BKF Initial bulk modulus of pore fluid SGF Initial specific gravity of pore fluid SO Initial degree of saturation* SO ≠1.0 invokes partial saturation model GAMMA Ratio of heat capacity *, $\gamma = C_p / C_v$ PAO Initial pore air pressure (Absolute)* T Not used (*) Not used for NW = 0

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property	5.3.2.4

MODELNO, DSRNMAX, MAXCYCL, Ko, NEHNO, NRHNO

MODELNO	= 1	Elastic Model
	= 2	Von Mises Model
	= 3	Mohr-Coulomb Model
	= 4	In Situ Rock Model
	= 5	Generalized Hoek and Brown Model
	= 6	Advanced Elasto-Plastic Model (N.A.)
	= 7	Single Hardening Plastic Model (N.A.)
	= 8	JWL High Explosive Model
	= 9	Modified Cam Clay Model
	= 10	Engineering Model
	= 11	Joint Model
	= 12	Duncan and Chang Hyperbolic Model
	= 13	Elastic Model for SHELL element
	= 14	User Defined Model
	= 15	User Defined Model
	= 16	User Defined Model
	= 17	User Defined Model
	= 18	User Defined Model
	= 21	PM4Sand Model (N.A.)
DSRNMAX	= 0.0	Do not apply strain sub cycling
	> 0.0	Maximum strain sub increment
MAXCYCL		Maximum number of strain sub cycling
Ko		Coefficient of earth pressure at rest
NEHNO		Young□s modulus multiplication factor history number in Card Group 9.2.3
NRHNO		Element volume multiplication factor history number in Card Group 9.2.3

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

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

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

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 3 (Mohr-Coulomb Model)	5.3.2.4.3

For MODELNO = 3 [Mohr-Coulomb Model]

E, v

φ, c, K, T, ST_n, ST_s

- E — Young's modulus
- v — Poisson's ratio
- φ — Internal frictional angle (°)

$$C = \frac{(1 - \sin\varphi)}{2\cos\varphi}\sigma_c$$

- C — Cohesion
- K — The ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same pressure
- T — Tensile strength
- ST_n — Factor used to divide stiffness normal to tensile crack
- ST_s — Factor used to divide shear modulus for the cracked zone

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

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

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

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

Table 3 Continued

Description of Rock Quality

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

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

5.3.2.4.5

For MODELNO = 5 [Generalized Hoek & Brown Model]

Elastic Parameters

E, v

- E Young's Modulus
- v Poisson's Ratio

Tensile Strength Parameters

NTCUT

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

For NTCUT = 1, otherwise go to next Card

T, St_n, St_s

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

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

Strength Parameters

A_1, A_2, A_3, A_4, A_5, A_6, A_7, A_8

1. Von Mises (A_1 = 0.0)

   $F = q - A_4 R(\theta)$

   $A_2 = A_3 = 0.0$

   $A_4 = A_6 = q_{VM} = \sigma$

   Refer to Card 5.3.2.4.2

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property	Skeleton Property for MODELNO = 5 (Generalized Hoek and Brown Model)	5.3.2.4.5 2. Hoek and Brown (A_1 = 0.5) $F = q - ((A_2 + A_3\,p)^{?} + A_4)\,R(\theta)$ $A_2 = (m^2 / 36 + s)\,\sigma_c^{\,2}$ $A_3 = m\,\sigma_c$ $A_4 = m\,\sigma_c / 6$ Refer to Card 5.3.2.4.4 3. Mohr-Coulomb (A_1 = 1.0) $F = q - ((A_2 + A_4) + A_3\,p)\,R(\theta)$ $A_2 + A_4 = 3\,\sigma_c\,(1 - \sin\varphi) / (3 - \sin\varphi)$ $A_3 = 6\sin\varphi / (3 - \sin\varphi)$ Refer to Card 5.3.2.4.3 4. Quadratic (A_1 = 2.0) $F = q - (A_2 + A_3\,p + A_4\,p^2)\,R(\theta)$ 5. Elliptic (A_1 = 3.0) $F = q - (A_3 + (A_6 - A_3)\,(1 - ((p - A_2)/A_4)^2)^{1/2})\,R(\theta)$ $A_5 = K$ (See notes in previous page) $A_6 = q_{VM}$ (Von Mises limit stress) The mean pressure (p_o) at which it reaches Von Mises limit stress (q_{VM}) is given by: For A1 = 0.0, $p_o = \infty$ For A1 = 0.5, $p_o = ((A_6 - A_4)^2 - A_2)/A_3$ For A1 = 1.0, $p_o = (A_6 - (A_2 + A_4))/A_3$ For A1 = 2.0, $p_o = (-A_3 + (A_3^{\,2} - 4A_4\,(A_2 - A_6))^{1/2})/(2A_4)$ For A1 = 3.0, $p_o = A_2$

Card Group	Input Data and Definitions (Main File)

5 Continuum Element

5.3 Material Property

Skeleton Property for MODELNO = 5 (Generalized Hoek and Brown Model)

5.3.2.4.5

$A_7 = p_{BD}$ Brittle-Ductile transition pressure
$A_8 = r_i$ Initial dilatancy parameter

Dilatancy parameter r is calculated as

For $p_{BD} > 0.0$ and $p < p_{BD}$
$r = r_i \ (1 - p / p_{BD})$

For $p_{BD} > 0.0$ and $p \geq p_{BD}$
$r = 0.0$

For $p_{BD} = 0.0$
$r = r_i$

Note: Derivatives of potential function (Q) are related to the yield function (F) as
$\partial Q / \partial p = (\partial F / \partial p) \ r$
$\partial Q / \partial q = (\partial F / \partial q)$
$\partial Q / \partial \theta = (\partial F / \partial \theta)$

For associated flow rule
use $A_7 = 0.0$, $A = 1.0$
and set ISYMSOL=1 in Card 1.1

For no plastic volume change
use $A_7 = 0.0$, $A_8 = 0.0$
and set ISYMSOL=2 in Card 1.1

For all non associated cases
set ISYMSOL=2 in Card 1.1

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

Pressure - Dependent Moduli

IBULK, ISHEAR

IBULK = 0 Constant bulk modulus
= 1 Nonlinear bulk modulus

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

Loading Bulk Modulus Definition

NLPC

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

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

P_i, B_{Li} Pressure and bulk modulus pairs

Unloading Bulk Modulus Definition

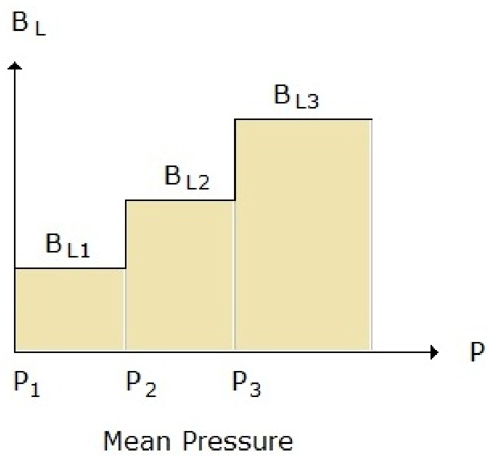
NUPC

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

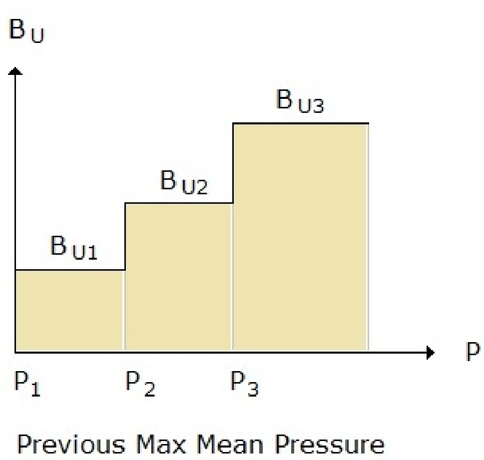
NUPC Cards
P_1, B_{U1}
P_2, B_{U2}
- -
P_n, B_{Un}

P_i, B_{Ui} Pressure and bulk modulus pairs

Loading Bulk Modulus as a Function of Mean Pressure



Unloading Bulk Modulus as a Function of Previous Max Pressure



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

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property	Skeleton Property for MODELNO = 7 (Single Hardening Elasto-Plastic Model)	5.3.2.4.7 For MODELNO = 7 [Single Hardening Elasto-Plastic Model] Not Available Precision Parameters NDVMIN, NDVMAX, DEEMAX, NUNLOAD, NDRIFT NDVMIN — Min number of plastic strain sub increment NDVMAX — Max number of plastic strain sub increment DEEMAX — Max plastic strain sub increment NUNLOAD = 0 — Smooth initial unloading = 1 — No smooth unloading NDRIFT = 0 — Drift correction = 1 — No drift correction Tensile Strength APEX, ATMO APEX — Tensile strength T ATMO — Atmospheric pressure P_a Elastic Constant AKUR, AN, APOI AKUR — Elastic Young's modulus constant Kur AN — Elastic Young's modulus exponent n APOI — Elastic Poisson's ratio v Isotropic Hardening NACRV AACC(I), AAPC(I), ABRK(I) I = 1, NACRV NACRV — Number of segments for isotropic hardening function AACC — Isotropic hardening constant C

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property	Skeleton Property for MODELNO = 7 (Single Hardening Elasto-Plastic Model)	5.3.2.4.7 AAPC Isotropic hardening constant P ABRK Break point in terms of W_p/P Failure Constant AK, AMY, AETA1 AK The ratio of triaxial extensive to compressive strength at a given pressure AMY Failure exponent m AETA1 Failure constant η_1 Yield Constant AY1, AH, ALPHA AY1 Yield constant ψ_1 AH Yield constant h ALPHA Yield constant α Potential Constant AY2, AMU AY2 Potential constant ψ_2 AMU Potential constant μ Unload/Reload Constant AHLAM, AHGAM, AHBET, APCO AHLAM (λ), AHGAM (γ), AHBET (β) These unload/reload constants are not used APCO Effective mean pressure at which yielding begins

Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property	Skeleton Property for MODELNO = 8 (JWL High Explosive Model)	5.3.2.4.8 For MODELNO = 8 [JWL High Explosive Model] Elastic Constant E, v Note: When using JWL model, specify NLNR = 1 and NGEN = 1 in Card 2 JWL Model Parameters A, B, R_1, R_2, ω, E_v A — JWL material constant (Megabar) B — JWL material constant (Megabar) R_1 — JWL material constant (Dimensionless) R_2 — JWL material constant (Dimensionless) ω — JWL material constant (Dimensionless) E_v — Chemical energy density of explosive (Megabar cc/cc) Burn Fraction Parameters C_d, B_s, XL C_d — Detonation velocity B_s — Constant used to spread the detonation front [Usually set B_s = 2.5] XL — Characteristic length of element. If XL = 0.0, program computes XL Note: If C_d = 0 and B_s = 0, XL represents LHNO (Pressure Load History Number) specified in Cards 9.2.3.1 through 9.2.3.5 and above JWL parameters are ignored

Card Group	Input Data and Definitions (Main File)

5 Continuum Element

5.3 Material Property Data

Skeleton Property for MODELNO = 9 (Modified Cam Clay Model)

5.3.2.4.9

For MODELNO = 9 [Modified Cam Clay Model]

Cam-Clay Material Parameters

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

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

Tensile Strength Parameters

NTCUT

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

For NTCUT = 1, otherwise go to next Card

T, ST_n, ST_s

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

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

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

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

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 10 (Engineering Model)	5.3.2.4.10 Unloading Modulus NUS PBU(i), BKU(i), POU(i) i = 1, NUS NUS — Number of unloading slopes PBU(i) — Pressure breakpoint between unloading slopes i and i+1 BKU(i) — Bulk modulus for unloading slope i POU(i) — Poisson's ratio for unloading slope i Note: Special case for NLS = 1 1. Loading and unloading modulus are assumed to be the same Input data for unloading Modulus is not considered 2. Tension cutoff is based on individual principal stress. The limit of tensile stress is equal to ST1 / 3

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 11 (Joint Model)	5.3.2.4.11 For MODELNO = 11 [Joint Model] Elastic Modulus and Thickness NM E, G, t, v NM = 0 Linear elastic joint = 1 Nonlinear joint = 2 Lumped nonlinear joint = 3 Contact nonlinear joint = 4 Thin Layer Element E Elastic Young's modulus G Elastic shear modulus t Joint thickness v Poisson's ratio (Used for NM = 4) Strength Parameters (Only for NM > 0) C, φ, r C Cohesion φ Friction angle (°) r = -1 Decoupled volume and shear = 0 No plastic volume change (N.A.) = 1 Associated flow rule (N.A.) = -2 Decoupled shear (N.A.)

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 11 (Joint Model)	5.3.2.4.11 Normal Stress-Strain Relation (Only for NM = 1,2,3) ε_1, σ_1 ε_2, σ_2 ε_3, σ_3 ε_4, σ_4 ε_i, σ_i Pair of strain and stress to define normal stress-strain relation (Tension is positive) Tensile Strength (Only for NM = 4) TENSTR TENSTR Tensile strength Note: 1. For t > 0.0, coordinateso of joint element is adjusted based on t 2. For t < 0.0, no adjustment of coordinates. Users input mesh should represent joint thickness t 3. For t = 0.0 and NM = 4, joint thickness by user's input coordinate 4. Lumped nonlinear joint (NM=2) has better performance than nonlinear joint (NM=1). Contact nonlinear joint (NM=3) has no resistance in shear.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	Skeleton Property for MODELNO = 12 (Duncan and Chang Hyperbolic Model)	5.3.2.4.12

For MODELNO = 12 [Duncan and Chang Hyperbolic Model]

A_1, A_2, A_3, A_4, A_5, R_f

A_1 = 1.0
A_2 = 1000.
A_3 = 6 sinφ / (3 - sinφ)
A_4 = 6 cosφ C / (3 - sinφ) - 1000
A_5 = 1.0
R_f = 0.7 ~ 0.9

Loading Bulk Modulus Definition

NLPC

NLPC — Number of volumetric strain/modulus/ Poisson's ratio pairs describing the virgin loading

NLPC Cards:
EBL_1, BKL_1, POL_1
EBL_2, BKL_2, POL_2
- - -
EBL_n, BKL_n, POL_n

EBL, BKL, POL — Refer to Card 5.3.2.4.10

Unloading Bulk Modulus Definition

NUPC

NUPC — Number of volumetric pressure/modulus /Poisson's ratio pairs describing the unloading

NUPC Cards:
PBU_1, BKU_1, POU_1
PBU_2, BKU_2, POU_2
- - -
PBU_n, BKU_n, POU_n

PBU, BKU, POU — Refer to Card 5.3.2.4.10

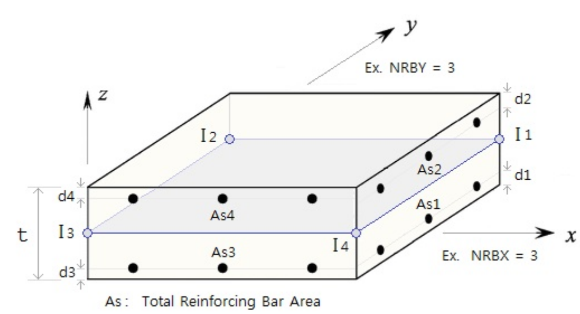
Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 13 (Elastic Model for SHELL Element)	5.3.2.4.13

For MODELNO = 13 [Elastic Model for SHELL Element]

E, v, t, FACIN, FACBD, MR_{12}, MR_{23}, MR_{34}, MR_{41}, IRB

For IRB =1 [Include Reinforcing Bars]
E_S, NRBX, NRBY, d_1, As_1, d_2, As_2, d_3, As_3, d_4, As_4

E, v	Young's modulus, Poisson's ratio
t	Shell thickness
FACIN	Multiplication factor for in-plane stiffness.
FACBD	Multiplication factor for bending stiffness.
	$E_{in\text{-}plane}$ = FACIN • E, $E_{bending}$ = FACBD • E.
	For only in-plane deformation, FACBD = 0.0
MR_{ij}	Moment release flag along the edge I_i - I_j
	MR = 0: No hinge MR = 1: Hinge
IRB	Include Reinforcing Bars. 0: No, 1: Yes
E_S	Young's modulus of reinforcing bars
NRBX	Number of reinforcing bars in x direction
NRBY	Number of reinforcing bars in y direction
d_i, As_i	Cover depth, reinforcing bar area



Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.3 Material Property Data	MODELNO = 14 (User Defined Model)	5.3.2.4.14 For MODELNO = 14 [User Defined Model] 60 Cards ┌ PROP (41) │ PROP (42) │ - │ - └ PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL14.FOR in the directory C:\SMAP\SMAP3D\PROGRAM\USER\MODEL-14. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL14.FOR. 2. MODEL14.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE14.BAT. 3. Text file LABEL14.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL14.DLL can be obtained once compiled. MODEL14.DLL should be saved in the directory C:\SMAP\SMAP3D\PROGRAM.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 15 (User Defined Model)	5.3.2.4.15 For MODELNO = 15 [User Defined Model] 60 Cards: PROP (41), PROP (42), -, -, PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL15.FOR in the directory C:\SMAP\SMAP3D\PROGRAM\USER\MODEL-15. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL15.FOR. 2. MODEL15.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE15.BAT. 3. Text file LABEL15.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL15.DLL can be obtained once compiled. MODEL15.DLL should be saved in the directory C:\SMAP\SMAP3D\PROGRAM.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 16 (User Defined Model)	5.3.2.4.16 For MODELNO = 16 [User Defined Model] 60 Cards ┌ PROP (41) │ PROP (42) │ - │ - └ PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL16.FOR in the directory C:\SMAP\SMAP3D\PROGRAM\USER\MODEL-16. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL16.FOR. 2. MODEL16.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE16.BAT. 3. Text file LABEL16.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL16.DLL can be obtained once compiled. MODEL16.DLL should be saved in the directory C:\SMAP\SMAP3D\PROGRAM.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 17 (User Defined Model)	5.3.2.4.17 For MODELNO = 17 [User Defined Model] 60 Cards: PROP (41) PROP (42) - - PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL17.FOR in the directory C:\SMAP\SMAP3D\PROGRAM\USER\MODEL-17. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL17.FOR. 2. MODEL17.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE17.BAT. 3. Text file LABEL17.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL17.DLL can be obtained once compiled. MODEL17.DLL should be saved in the directory C:\SMAP\SMAP3D\PROGRAM.

Card Group			Input Data and Definitions (Main File)
5 Continuum Element	5.3 Material Property Data	MODELNO = 18 (User Defined Model)	5.3.2.4.18 For MODELNO = 18 [User Defined Model] 60 Cards: PROP (41) PROP (42) - - PROP (100) PROP (41) - PROP (100): Material constants related to the User's Model. Note: 1. Users can use their own material model by modifying file MODEL18.FOR in the directory C:\SMAP\SMAP3D\PROGRAM\USER\MODEL-18. Input material constants and state variables to the User's Material Model are described in detail in source file MODEL18.FOR. 2. MODEL18.FOR can be compiled by Microsoft Fortran PowerStation 4.0 using the batch file MAKE18.BAT. 3. Text file LABEL18.DAT can be modified appropriately. 4. Dynamic Link Library file MODEL18.DLL can be obtained once compiled. MODEL18.DLL should be saved in the directory C:\SMAP\SMAP3D\PROGRAM.

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.3 Skeleton Property for MODELNO = 21 (PM4Sand Model)	5.3.2.4.21

For MODELNO = 21 [PM4Sand Model]

D_R G_o h_{po} p_a N_s S_{cheme} T_{antyp}

Secondary Parameters (Skip these cards for N_s = 1)

h_o e_{max} e_{min} n^b n^d A_{do}
z_{max} c_z C_e ϕ_{cv} v_o C_{GD}
C_{DR} c_{kaf} Q R m $F_{sed.min}$ p_{sed}

D_R	Apparent relative density (Fraction)
G_o	Shear modulus coefficient
h_{po}	Contraction rate parameter
p_a	Atmospheric pressure (10.33 for stress unit t/m^2)
N_s	Secondary parameter specification: 0 = Yes, 1 = No
S_{cheme}	Integration scheme (Use S_{cheme} = 0)
T_{antyp}	Drift correction method (Use T_{antyp} = 0)
h_o	Control parameter for ratio of plastic to elastic modulus
e_{max}	Maximum void ratio (Default 0.8)
e_{min}	Minimum void ratio (Default 0.5)
n^b	Control parameter for dilatancy & peak friction angle
n^d	Control parameter for transition from contr. to dilation
A_{do}	Bolton's dilatancy parameter
z_{max}	Maximum allowable fabric dilatancy tensor z
c_z	Control parameter when fabric effects get important
C_e	Control parameter for adjusting strain accumulation rate
ϕ_{cv}	Critical state effective friction angle (Default 33˚)
v_o	Poisson's ratio (Default 0.3)
C_{GD}	Factor for shear modulus degradation (Default 2.0)
C_{DR}	Control parameter for rotated dilatancy surface
c_{kaf}	Control parameter for effects of sustained shear stress
Q, R	Parameters for Bolton's empirical critical state line
m	Parameter defining size of yield stress (Default 0.01)
$F_{sed.min}$	Parameter for post-shaking elastic modulus reduction
p_{sed}	Mean effective stress for post-shaking reconsolidation

Set -1 for default values of secondary model parameters.

For description, refer to Boulanger, R. W. And ziotopoulou, k. PM4Sand (Version 3.1): A Sand Plasticity Model for Earthquake Engineering Applications, Report No UCD/CGM-17/01, Dept. of Civil & Env. Eng., U. of Cal., Davis, CA, 109 pp.

Card Group		Input Data and Definitions (Main File)
5 Continuum Element	5.5 Skew Boundary	5.5.1 NSKEW NSKEW Number of element sides on a skew boundary. Not available (Use NSKEW = 0)

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

<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">5.7
Element Surface</td><td colspan="2">5.7.1

NUMEST, MATEST

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

If NUMEST = 0, go to Card Group 6</td></tr>
<tr><td>For Each Material / Element Surface</td><td>5.7.2.1

(MATEST) Cards
MAT, KP, KH, KD, a_0, a_1, a_2, a_3

(NUMEST - MATEST) Cards
NEL, KP, KH, KD, a_0, a_1, a_2, a_3

MAT Material number
NEL Element number

KP Element surface designation number
KH Load history number specified in Cards 9.2.3.1 through 9.2.3.5.
If KH=0, constant static pressure/traction vector is acting all the time.

KD = 0 Uniformly distributed traction vector defined in local coordinate system
$P'_n = a_0$ $P_x=a_1$ $P_y=a_2$ $P_z=a_3$

= 1 Uniformly distributed traction vector defined in global coordinate system
$P'_n = a_0$ $P_X=a_1$ $P_Y=a_2$ $P_Z=a_3$
P'_n is static normal pressure.
(Compression is positive)

= 2 Linearly distributed static normal pressure
$P_{n4}= a_0$ at I_4' $P_{n1}= a_1$ at I_1'
$P_{n2}= a_2$ at I_2' $P_{n3}= a_3$ at I_3'</td></tr>
</table>

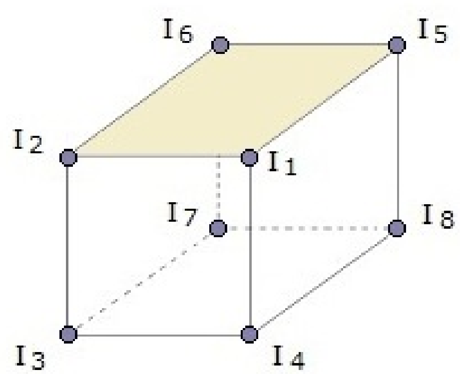
Card Group	Input Data and Definitions (Main File)		
5 Continuum Element	5.7 Element Surface	For Each Material / Element Surface	5.7.2.1 Linearly distributed surface tractions defined in global coordinate system = 3 q_X $q_{X4} = a_0$ at I_4' $q_{X1} = a_1$ at I_1' $q_{X2} = a_2$ at I_2' $q_{X3} = a_3$ at I_3' = 4 q_Y $q_{Y4} = a_0$ at I_4' $q_{Y1} = a_1$ at I_1' $q_{Y2} = a_2$ at I_2' $q_{Y3} = a_3$ at I_3' = 5 q_Z $q_{Z4} = a_0$ at I_4' $q_{Z1} = a_1$ at I_1' $q_{Z2} = a_2$ at I_2' $q_{Z3} = a_3$ at I_3' = 6 Static normal pressure given as functions of global X, Y and Z coordinates $P'_n = a_0 + a_1 X + a_2 Y + a_3 Z$ Global surface traction given as functions of global X, Y and Z coordinates = 7 q_X $q_X = a_0 + a_1 X + a_2 Y + a_3 Z$ = 8 q_Y $q_Y = a_0 + a_1 X + a_2 Y + a_3 Z$ = 9 q_Z $q_Z = a_0 + a_1 X + a_2 Y + a_3 Z$ Note1: Element traction is not available for KS = -1 (High Explosive Solid Element) Note2: (NEL1, -NEL2) generates the same surface traction from NEL1+1 to NEL2. This also applies to material based traction. Refer to description in next page

Element Surface Designation and Local Axes

KP	8-Node Brick				6-Node Wedge			
	I_1'	I_2'	I_3'	I_4'	I_1'	I_2'	I_3'	I_4'
1	5	6	2	1	5	6	2	1
2	6	7	3	2	6	7	3	2
3	7	8	4	3	7	5	1	3
4	8	5	1	4	0	0	0	0
5	1	2	3	4	1	2	3	0
6	6	5	8	7	6	5	7	0

For Shell element, use KP = 1.

For KP = 1



Card Group	Input Data and Definitions (Main File)
6	6.1 NBEAM NBEAM Total number of beam elements If NBEAM = 0, go to Card Group 7
	6.2 NBMST NBMST Use NBMST = 1
	6.3 NTNB NTNB Number of material property set for beam element
Beam Element	6.4 (For Each Msterial) 6.4.1 MATNO, MR, NEHNO, NFSHR, CTSy, CTSz, DAMP MATNO Material number MR Moment Release = 0 No hinge = 1 Hinge at node I = -1 Hinge at node J = 2 Hinge at node I and J = 3 Joint spring element Spring Element at Node I = 11 Axial spring (K_x = E A / L) = 12 Shear(y) spring (K_y = 12 E Iz / L^3) = 13 Shear(z) spring (K_z = 12 E Iy / L^3) = 14 Torsional spring (K_t = G J / L) = 15 Rotational(y) spring (K_{ry} = 4 E Iy / L) = 16 Rotational(z) spring (K_{rz} = 4 E Iz / L) Use Negative for Spring Element at Node J NEHNO Young's modulus multiplication factor history number in Card Group 9.2.3 NFSHR = 0 Neglect shear deformation = 1 Include shear deformation CTSy, CTSz Shear coefficient for Iy, Iz DAMP Critical damping ratio

Card Group			Input Data and Definitions (Main File)
6 Beam Element	6.4 For Each Msterial	6.4.2	**For MR ≠ 3** A, WL, RHO, E, G, J, I_y, I_z A — Cross section area WL — Weight per unit length of beam RHO — Mass density E — Young's modulus G — Shear modulus J — Torsional moment of inertia I_y — Moment of inertia about member y axis I_z — Moment of inertia about member z axis **For MR = 3** K_X, K_Y, K_Z, K_T, K_{RY}, K_{RZ} K_X — Axial spring stiffness K_Y — Shear(y) spring stiffness K_Z — Shear(z) spring stiffness K_T — Torsional spring stiffness K_{RY} — Rotational(y) spring stiffness K_{RZ} — Rotational(z) spring stiffness

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

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

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

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

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

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="3">9
Loads</td><td rowspan="3">9.1
Gravity Load</td><td>9.1.2.1
If NHFRX = 0, skip this card
A_0, A_1, A_2, A_3, Y_1, Y_2

A_i Distribution factor
Y_i Global Y coordinate

For $Y < Y_1$ $A_i = A_0$
For $Y > Y_2$ $A_i = A_3$
For others $A_i = A_1 + (Y - Y_1) * (A_2 - A_1) / (Y_2 - Y_1)$</td></tr>
<tr><td>9.1.2.2
If NHFRY = 0, skip this card
A_0, A_1, A_2, A_3, Y_1, Y_2

A_i Distribution factor
Y_i Global Y coordinate</td></tr>
<tr><td>9.1.2.3
If NHFRZ = 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</td></tr>
</table>

Card Group		Input Data and Definitions (Main File)
9 Loads	9.1 Specified Displacement	9.1.3 NUMDIS NUMDIS — Total number of degrees of freedom at which input displacement time histories are specified If NUMDIS = 0, go to Card Group 9.2.1
		9.1.4 For each of NUMDIS where displacement is specified NODE, IDOF, LHNO, DINT NODE — Node number Skeleton displacement IDOF = 1 x-direction = 2 y-direction = 3 z-direction Apparent relative fluid displacement = 4 x-direction = 5 y-direction = 6 Z-direction LHNO — Displacement history number corresponding to sequence of displacement specifications given in Card Group 9.1.5.3 DINT — Displacement intensity factor

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

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="2">9

Loads</td><td rowspan="2">9.2

Concentrated Nodal Force</td><td>9.2.1

NUMCON

NUMCON — Total number of degrees of freedom at which input concentrated force time histories are specified

If NUMCON = 0, skip the rest of this Card Group</td></tr>
<tr><td>9.2.2

For each of the NUMCON where load is applied

NODE, IDOF, LHNO, CINT

NODE — Node number

IDOF — Total force acting on a given node
= 1 x-direction
= 2 y-direction
= 3 z-direction

Fluid force acting on a given node
= 4 x-direction
= 5 y-direction
= 6 z-direction

LHNO — Load history number corresponding to sequence of load specifications given in Card Group 9.2.3.4 or 9.2.3.5

CINT — Load intensity factor</td></tr>
</table>

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

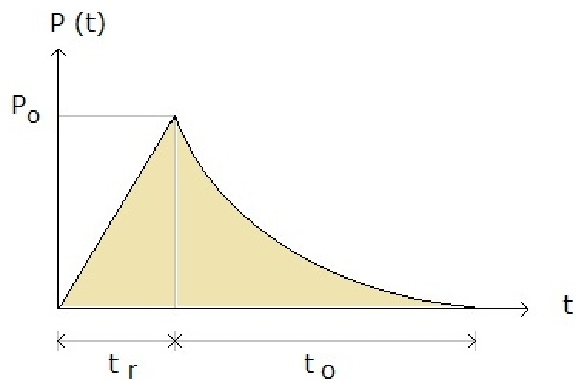
Card Group	Input Data and Definitions (Main File)		
9 Loads	9.2 Concentrated Nodal Force	NTFNC = 1 (Math Function)	9.2.3.5 For each of NUMCH loading time histories NFNC, a_1, a_2, a_3, a_4, a_5 NFNC = 1 Polynomial decaying load = 2 Exponential decaying load = 3 Trigonometric load a_1,a_2,a_3,a_4 Force function coefficients defined in the next page a_5 Starting time

Polynomial Decaying (NFNC = 1)

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

For $t_r \le t \le (t_r + t_o)$

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



Exponential Decaying (NFNC = 2)

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

Trigonometric (NFNC = 3)

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

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

Card Group	Input Data and Definitions (Main File)

9 Loads — 9.3 Specified Velocity

9.3.1

NUMVEL

NUMVEL Total number of degrees of freedom at which velocity histories are specified

If NUMVEL= 0, skip the rest of this Card Group

9.3.2

For each of the NUMVEL where velocity is specified

NODE, IDOF, LHNO, VINT

NODE Node number

Skeleton velocity

IDOF = 1 x - direction
= 2 y - direction
= 3 z - direction

Apparent relative fluid velocity

= 4 x-direction
= 5 y-direction
= 6 z-direction

LHNO Velocity history number corresponding to sequence of velocity specifications given in Card Group 9.3.3.4 or 9.3.3.5

VINT Velocity intensity factor

9.3.3.1

NTFNV, NUMVH

NTFNV = 0 User-specified arbitrary velocity
= 1 Velocity specified by math function

NUMVH Number of different input velocity time histories

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

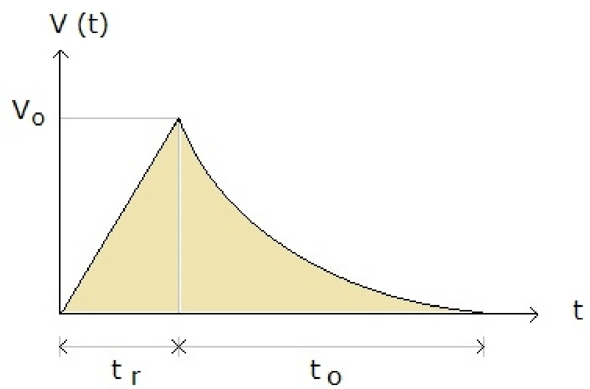
Card Group			Input Data and Definitions (Main File)
9 Loads	9.3 Specified Velocity	NTFNV = 1 (Math Function)	9.3.3.5 For each of NUMVH velocity time histories NFNV, a_1, a_2, a_3, a_4, a_5 NFNV = 1 Polynomial decaying velocity = 2 Exponential decaying velocity = 3 Trigonometric velocity a_1,a_2,a_3,a_4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

Polynomial Decaying (NFNV = 1)

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

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

For $t_r \le t \le (t_r + t_o)$



Exponential Decaying (NFNV = 2)

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

Trigonometric (NFNV = 3)

$t \le a_4$	$V(t) = a_1 \sin(a_2\, t) + a_3 \cos(a_2\, t)$
$t > a_4$	$V(t) = 0$

Card Group		Input Data and Definitions (Main File)
9 Loads	9.4 Initial Velocity	9.4.1 NINVEL NINVEL — Number of degrees of freedom where initial velocity is applied. If NINVEL= 0, skip the rest of this Card Group
		9.4.2 For each of the NINVEL where velocity is applied NODE, IDOF, VEL NODE — Node number Skeleton velocity IDOF = 1 x-direction = 2 y-direction = 3 z-direction Apparent relative fluid velocity = 4 x-direction = 5 y-direction = 6 z-direction VEL — Initial velocity

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Main File)</th></tr>
<tr><td rowspan="3">9
Loads</td><td rowspan="3">9.5
Specified Acceleration</td><td>9.5.1
NUMACC, MOTION, EYB, EDEN, ECP, ECS

NUMACC Total number of directions at which input acceleration time histories are specified

MOTION = 0 No EHS (Elastic Half Space)
= 1 EHS with base acceleration applied
= 2 EHS with base shear force applied
= 3 EHS with conventional base accel. applied

EYB, EDEN, ECP, ECS : Half space top y coordinate, Unit weight, Compression and Shear wave speeds used for elastic half space if MOTION is not zero

If NUMACC = 0, Skip the rest of this Card Group</td></tr>
<tr><td>9.5.2
For each of NUMACC where acceleration is specified
NODE, IDOF, LHNO, AINT

NODE Node number

IDOF = 1 x-direction skeleton acceleration
= 2 y-direction skeleton acceleration
= 3 z-direction skeleton acceleration

LHNO Acceleration history number corresponding to sequence of acceleration specifications given in Card Group 9.5.3.4 or 9.5.3.5

AINT Acceleration intensity factor

Note: For uniformly distributed acceleration, set all node numbers to zero. Output motions are relative to base motion for MOTION = 0 or 1</td></tr>
<tr><td>9.5.3.1
NTFNA, NUMAH

NTFNA = 0 User-specified arbitrary acceleration
= 1 Acceleration specified by math function

NUMAH Number of different input time histories</td></tr>
</table>

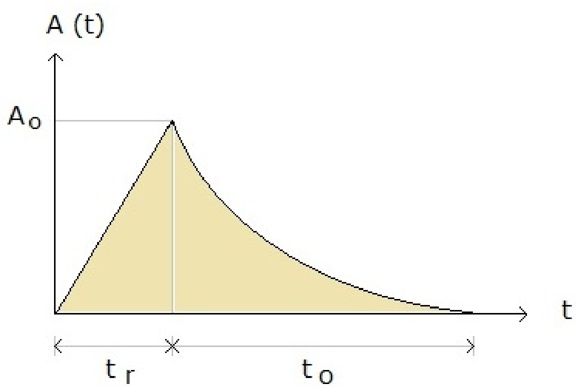
Card Group	Input Data and Definitions (Main File)			
9 Loads	Specified Acceleration	NTFNA = 0 (User-Specified Arbitrary Acceleration)	9.5.3.2 NUMATP, NATYPE, DTXA, TASTART, TAFAC, IACCM NUMATP Number of acceleration-time pairs NATYPE = 0 Constant time increment = 1 Specified times for all time histories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

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

Polynomial Decaying (NFNA = 1)

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

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



Exponential Decaying (NFNA = 2)

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

Trigonometric (NFNA = 3)

$t \le a_4$	$A(t) = a_1 \sin(a_2 t) + a_3 \cos(a_2 t)$
$t > a_4$	$A(t) = 0$

Card Group		Input Data and Definitions (Main File)
9 Loads	9.6 Transmitting Boundary	9.6.1 NODVIS, NELVIS, NOSVIS NODVIS Number of transmitting degrees of freedom NELVIS Number of continuum element surfaces NOSVIS Number of outer boundary surfaces (Max = 6)
		9.6.2 If NODVIS = 0, go to Card Group 9.6.3 NODE, IDOF, VISC For each of NODVIS NODE Node number IDOF = 1 Damping in x-direction = 2 Damping in y-direction = 3 Damping in z-direction VISC Constant which is proportional to the force on a given node (ρCA_c), equal to impedence times contributing area on the node $C = C_p$ for IDOF normal to the boundary $C = C_s$ for IDOF parallel to the boundary $C_{p,}$ C_s : Compression & shear wave speed
		9.6.3 If NELVIS = 0, go to next Card Group NEL, KT For each of the NELVIS NEL Element number KT Element surface designation number. Same as KP in Card Group 5.7.2.1
		9.6.4 If NOSVIS = 0, go to next Card Group NOS, VC For each of the NOSVIS NOS Outer surface number VC NOS = 1: VC = Y_{TOP} NOS = 2: VC = X_{LEFT} NOS = 3: VC = Y_{BOTOM} NOS = 4: VC = X_{RIGHT} NOS = 5: VC = Z_{FRONT} NOS = 6: VC = Z_{BACK}

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

4.5 Post File

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

Post File consists of three different card groups:

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

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

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

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

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

Snapshot

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

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

PLOT-2D

Post-Processor

Card Group	Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.1 NPTYPE, IHOR, IVER 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. IHOR, IVER Horizontal and Vertical coordinate flags (x=1, y=2, z=3, -x=-1, -y=-2, -z=-3) If NPTYPE = 0, Skip rest of Card Group 11

<table>
<tr><th>Card Group</th><th></th><th>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	11.4	11.4.5
PLOT-2D Plot Information	For NPTYPE = 3 (Deformed Shape)	

11.4.5

<u>Row and Line Plots (Repeat in any order)</u>

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

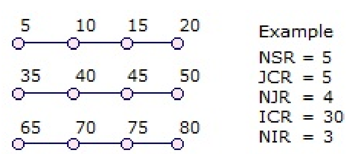
For Line Plot --> 2, IDISP
NPT
$NODE_1$, $NODE_2$, .., $NODE_{NPT}$

For End Plot --> 0, 0

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

<u>For Row Plot (Max = 1000)</u>

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



<u>For Line Plot (Max = 160)</u>

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)

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Post File)</th></tr>
<tr><td rowspan="5">11

PLOT-2D Plot Information</td><td rowspan="5">11.5

For NPTYPE = 4 (Beam Section Force / Extreme Fiber Stress / Strain)</td><td rowspan="3">For Each Beam Group</td><td>11.5.7.1

NBLIST

NBLIST = 0 Elements from NFBEAM to NLBEAM
= 1 Listing of individual elements</td></tr>
<tr><td>11.5.7.2

<u>For NBLIST = 0</u>
NFBEAM, NLBEAM

NFBEAM Starting beam element number
NLBEAM Ending beam element number</td></tr>
<tr><td>11.5.7.3

<u>For NBLIST = 1</u>
MBEAM
N_1, N_2, ..., N_{MBEAM}

MBEAM Number of beam element (Max=280)
N_1, N_2, N_{MBEAM} List of element number</td></tr>
<tr><td colspan="2">11.5.8

NRL

NRL Number of nodes to be connected by a Solid Line (Max=280)</td></tr>
<tr><td colspan="2">11.5.9

<u>If NRL = 0, Skip this Card</u>
$NODE_1$, $NODE_2$, ..., $NODE_{NRL}$

NODE Reference node numbers
If $NODE_i$ has negative sign, a New Line is drawn</td></tr>
</table>

Card Group	Input Data and Definitions (Post File)	
11 PLOT-2D Plot Information	11.6 For NPTYPE = 5 (Truss Axial Force / Stress / Strain)	11.6.1 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	For NPTYPE = 6 (Contours of Continuum Element Data)	11.7.1 TITLE TITLE Any title (Max = 70 characters)
PLOT-2D Plot Information		11.7.2 IUNIT IUNIT = 1 In, Pound = 2 Cm, Kg = 3 User-specified unit
		11.7.3 For IUNIT = 3 NCHR LABEL NCHRC LABELC NCHR Number of characters for mesh unit LABEL Name of mesh unit NCHRC Number of characters for contouring data LABELC Name of contouring data
		11.7.4 NLTIME, $TIME_{REF}$ $TIME_1$, $TIME_2$, ..., $TIME_{NLTIME}$ NLTIME Number of specified times (Max=1000) $TIME_{REF}$ Reference time TIME Specified time If $TIME_{REF}$ is not equal to 0.0, Contour plots at $TIME_i$ are relative to $TIME_{REF}$

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

Card Group			Input Data and Definitions (Post File)
11	11.7	11.7.7	NGROUP NGROUP = 0 Plot at all elements > 0 Plot at specified groups (Max=1000)
PLOT-2D Plot Information	For NPTYPE = 6 (Contours of Continuum Element Data)	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')

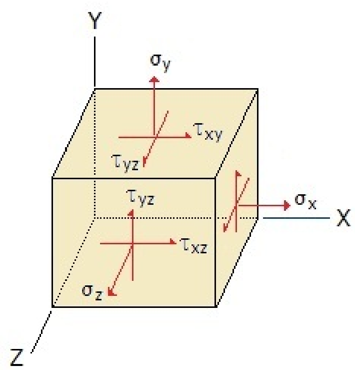


Figure PL-1 Sign Conventions for Continuum Stress

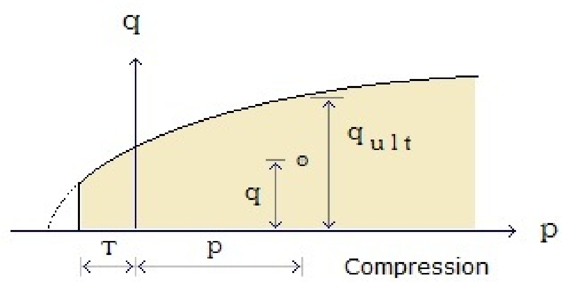


Figure PL-2 Definition of Safety Factor

Factor of Safety (FS) is defined as:

For elastic material FS = 10

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

FS is limited to $1 \leq$ F.S. ≤ 10

For p $\leq$ -T FS = 1

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

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

Yield Flag for Beam and Continuum Elements

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

Stress Status for Truss Element

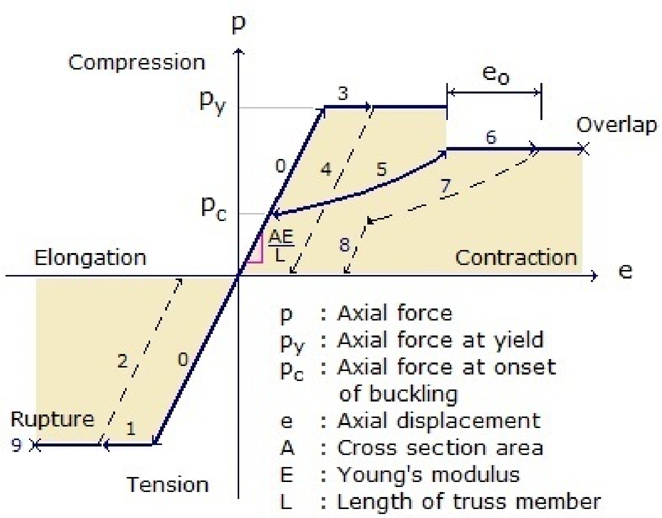


Figure PL-3 Description of Stress Status

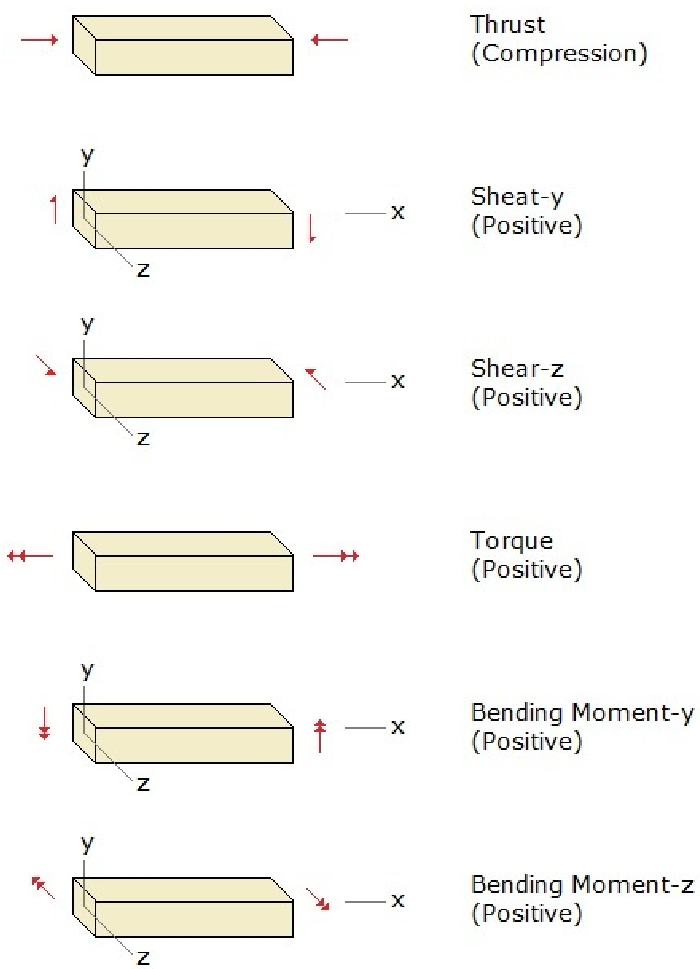


Figure PL-4 Sign Conventions for Beam

PLOT-XY

Post-Processor

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

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Post File)</th></tr>
<tr><td rowspan="4">12

PLOT-XY Information</td><td rowspan="4">12.2

For IPTYPE = 1 (Stress / Strain / Time History)</td><td>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</td></tr>
<tr><td>12.2.2

NOEL

NOEL Number of elements (Max 10)</td></tr>
<tr><td>12.2.3

LIST (I) I = 1, NOEL

LIST (I) List element numbers</td></tr>
<tr><td>12.2.4

NDPQ

NDPQ Number of different pair of variables</td></tr>
</table>

Card Group		Input Data and Definitions
12 PLOT-XY Information	12.2 For IPTYPE = 1 (Stress / Strain / Time History)	12.2.5 NDPQ Cards ⎡ K_{x1}, K_{y1} ⎢ K_{x2}, K_{y2} ⎢ - - ⎣ - - K_x,K_y Select from Table PL-1 Note: Shell rebar axial stresses & total bending moments K values in Table PL-1 are to select average values. To select values at corner i, assign K = 10 K_{Table} + i For top rebar x-dir. axial stress at element corner I_3 assign K = 10 (95) + 3 = 953
		12.2.6 TMFAC, STFAC, SNFAC Multiplication factor TMFAC Time STFAC Stress SNFAC Strain
		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)

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Post File)</th></tr>
<tr><td rowspan="3">12
PLOT-XY Information</td><td rowspan="3">12.3
For IPTYPE = 2 (Displacement / Velocity / Acceleration / Time History)</td><td>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</td></tr>
<tr><td>12.3.2
NODE

NODE Number of nodes (Max 10)</td></tr>
<tr><td>12.3.3
LIST (I), I = 1, NODE

LIST (I) List node numbers</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)	12.3.4 NDPQ NDPQ Number of different pair of variables
		12.3.5 NDPQ Cards: K_{x1}, K_{y1} / K_{x2}, K_{y2} / - - / - - K_x, K_y Select from Table PL-2
		12.3.6 TMFAC, SND, SNV, SNA, NC, ANGLE Multiplication factor TMFAC Time SND Displacement SNV Velocity SNA Acceleration NC = 0 No transfer = 1 Transfer from X-Y to polar coordinate = 2 Transfer from polar to X-Y coordinate ANGLE Rotation angle (Degree)
		12.3.7 IPLOT = 0: For each node IPLOT = 1: For each pair of variables TITLE (50 characters) X-LABEL (50 characters) Y-LABEL (50 characters)

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

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

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

IPLOT

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

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

NOTM

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

TLIST (I), I = 1, NOTM

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

NDPQ

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

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

K_y Select from Table PL-1

For K > 94, see Note in Card Group 12.2.5</td></tr>
</table>

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

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

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

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

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

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

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

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

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

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

Card Group		Input Data and Definitions
12 PLOT-XY Information	For IPTYPE = 10 (Snap Shot of a Stress/Strain for Different Times)	12.11.1 NOTM NOTM — Number of times (Max 10)
		12.11.2 TLIST (I), I = 1, NOTM TLIST (I) — List times in sequential order
		12.11.3 K_y K_y — Select from Table PL-1
		12.11.4 XSTART XSTART — Reference starting X-coordinate
		12.11.5 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 Node Number Specification (Max 800 Nodes) NRL N_1, N_2, N_{NRL} NRL Number of nodes N_1, N_2, .., N_{NRL} Node numbers N_i, $-N_{i+1}$, N_{i+2} From N_i to N_{i+1} with increment N_{i+2}
		12.13.6 SND, SNV, SNA, SDFAC Multiplication factor SND Displacement SNV Velocity SNA Acceleration SDFAC Distance
		12.13.7 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

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

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

Table PL-1 continued

K_x, K_y	Legend	Description	
		Beam Element (See Fig. PL-4)	
35	THRUST	Thrust	(F_x)
36	SHEAR-Y	Shear in y direction	(F_y)
37	SHEAR-Z	Shear in z direction	(F_z)
38	TORQUE	Torque	(T)
39	MOMENT-Y	Moment about y axis	(M_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})
49	STRAIN-FL	Left fiber strain	(ε_{fl})
50	STRESS-FL	Left fiber stress	(σ_{fl})
51	STRAIN-RL	Left reinf. bar strain	(ε_{rl})
52	STRESS-RL	Left reinf. bar stress	(σ_{rl})
53	STRAIN-RR	Right reinf. bar strain	(ε_{rr})
54	STRESS-RR	Right reinf. bar stress	(σ_{rr})
55	STRAIN-FR	Right fiber strain	(ε_{fr})
56	STRESS-FR	Right fiber stress	(σ_{fr})
		Truss Element	
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	($\sigma_{xy\ max\ bot}$)
		Rebar axial stress and total moment	
95	ASTRES-XT	Top rebar x direction axial stress	($\sigma_{xx\ top}$)
96	ASTRES-YT	Top rebar y direction axial stress	($\sigma_{yy\ top}$)
97	ASTRES-XB	Bot. rebar x direction axial stress	($\sigma_{xx\ bot}$)
98	ASTRES-YB	Bot. rebar y direction axial stress	($\sigma_{yy\ bot}$)
99	TMOMENT-XX	Total bending moment about x axis	($M_{xx\ total}$)
100	TMOMENT-YY	Total bending moment about y axis	($M_{yy\ total}$)
		Note: 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)

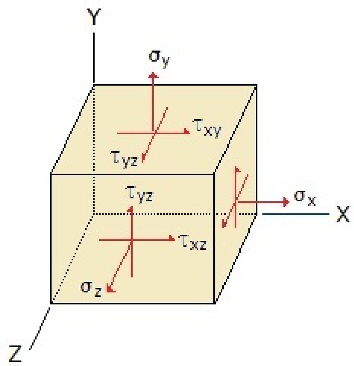


Figure PL-1 Sign Conventions for Continuum Stress

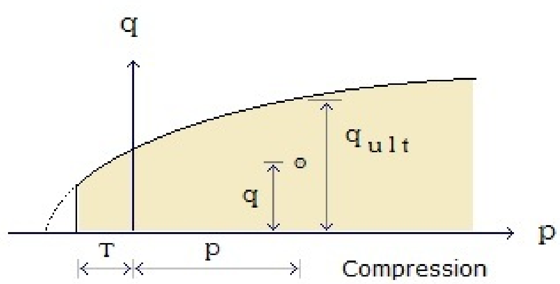


Figure PL-2 Definition of Safety Factor

<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

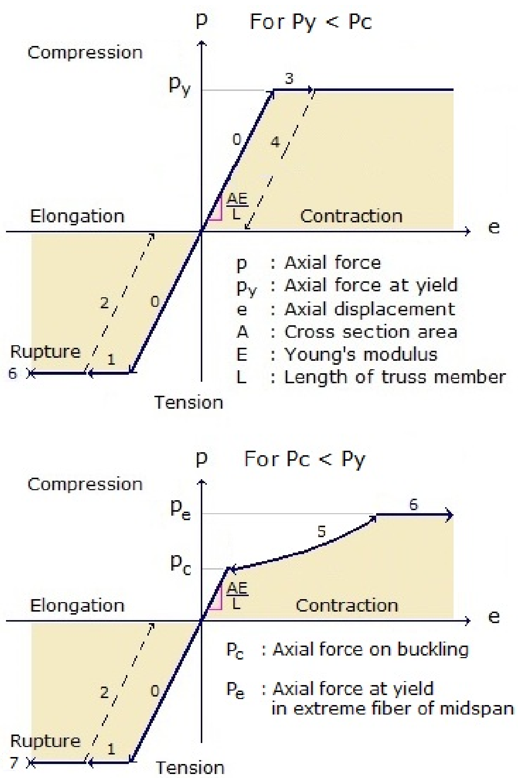


Figure PL-3 Description of Stress Status

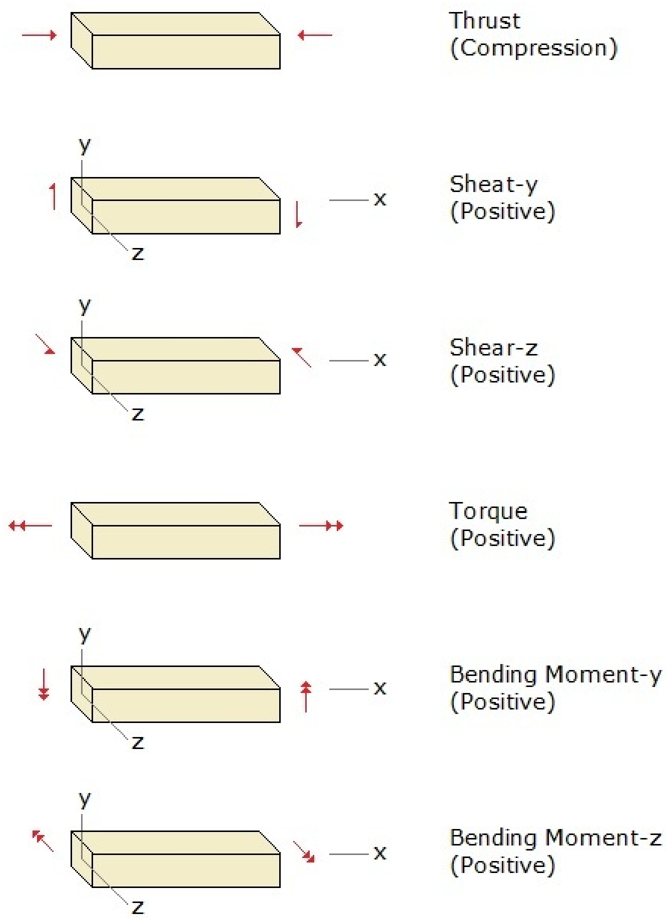
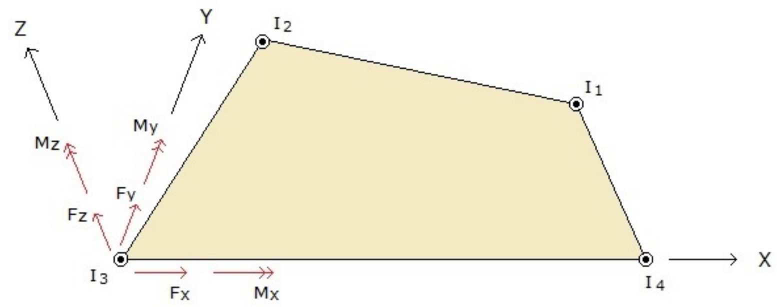
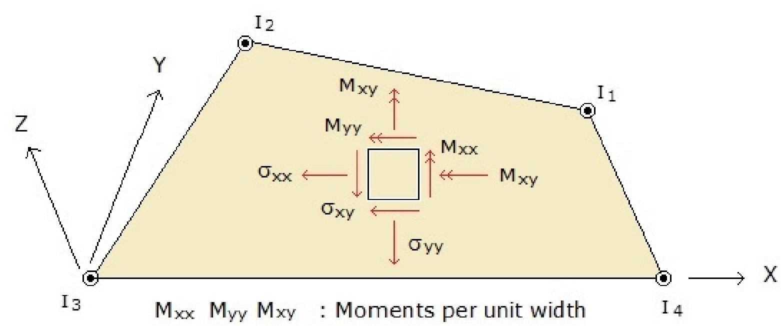


Figure PL-4 Sign Conventions for Beam



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



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

Figure PL-5 Sign Conventions for Shell

Group Mesh User's Manual

5.1 Introduction

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

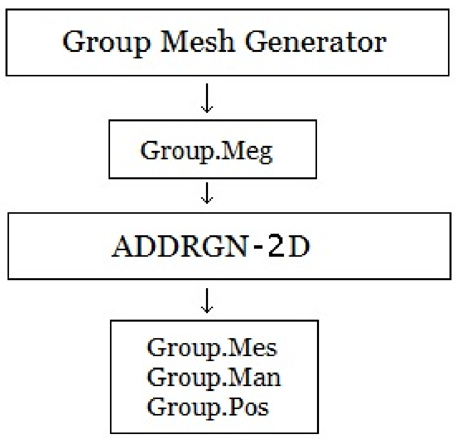


Figure 5.1 Flow diagram of group mesh generation

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

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

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

5.2 Group Mesh Generator

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

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

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

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

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

5.3 Group

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

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

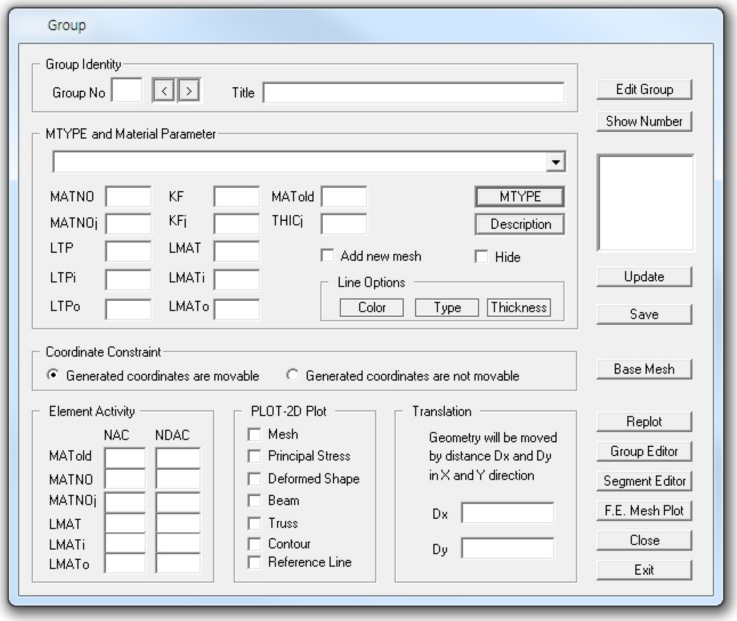


Figure 5.2 Group dialog.

5.3.1 Group Identity

Here, you type Group No and Title.

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

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

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

5.3.2 MTYPE and Material Parameter

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

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

Figure 5.3
MTYPE dialog

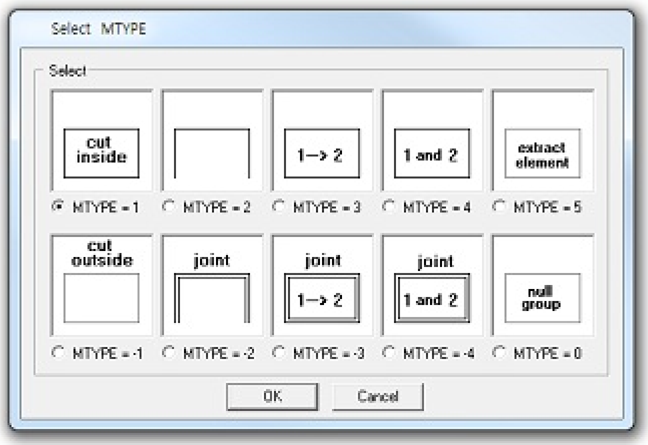
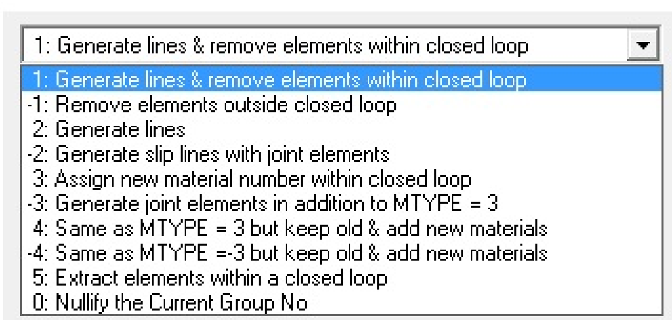


Figure 5.4
MTYPE list box



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

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

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

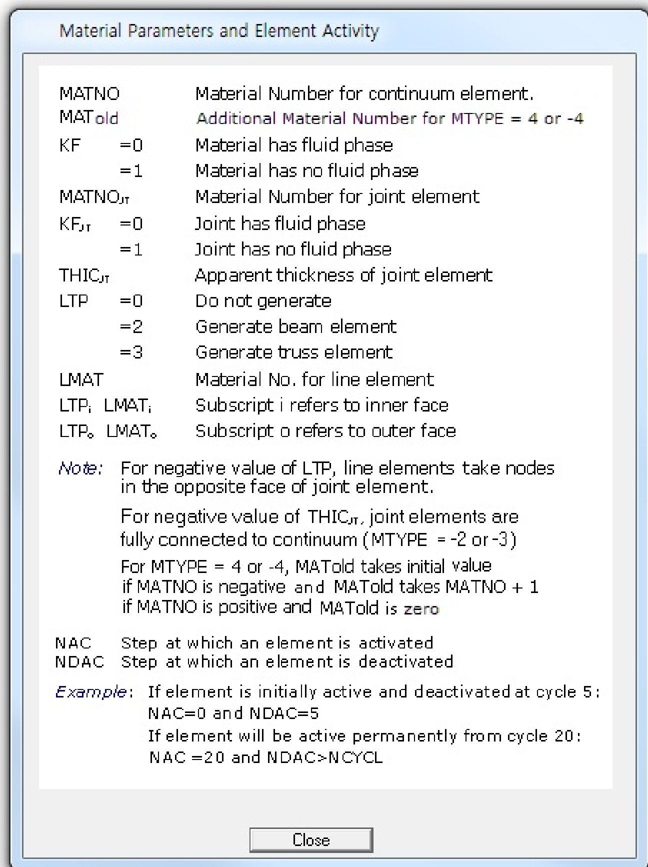


Figure 5.5 Material parameters & element activity (SMAP-2D)

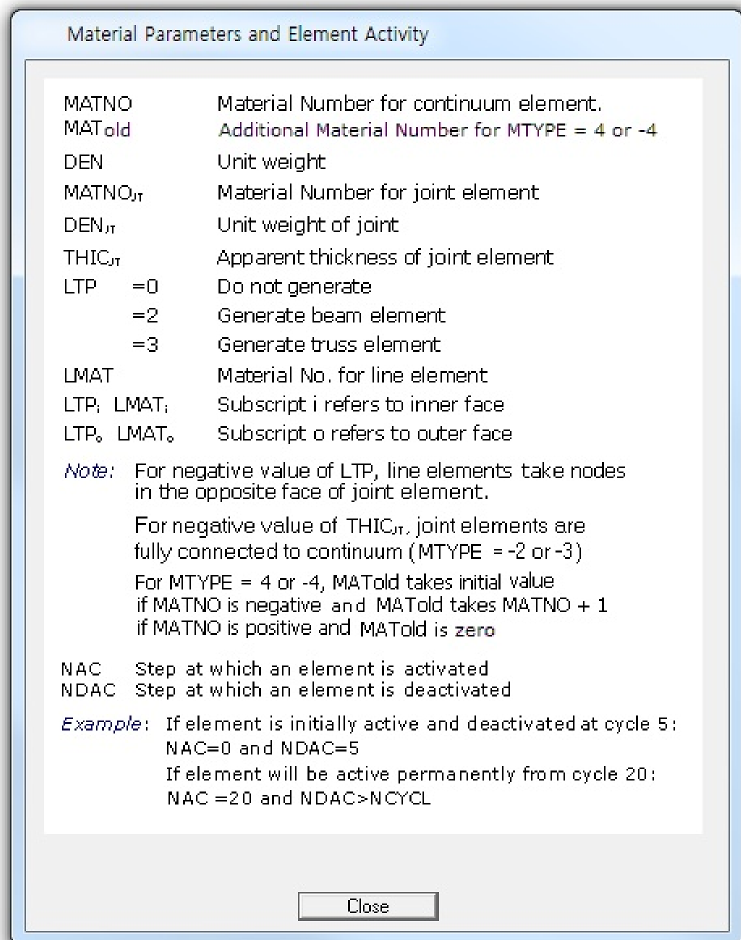


Figure 5.5 Material parameters & element activity (SMAP-S2)

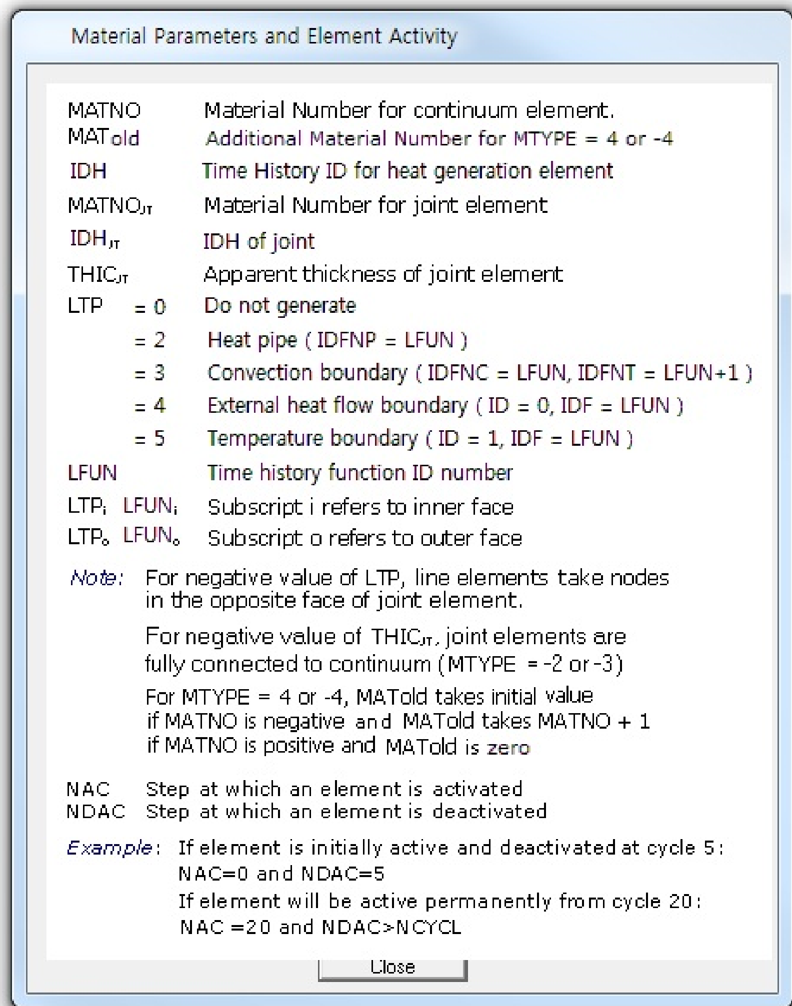


Figure 5.5 Material parameters & element activity (SMAP-T2)

5.3.3 Line Options

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

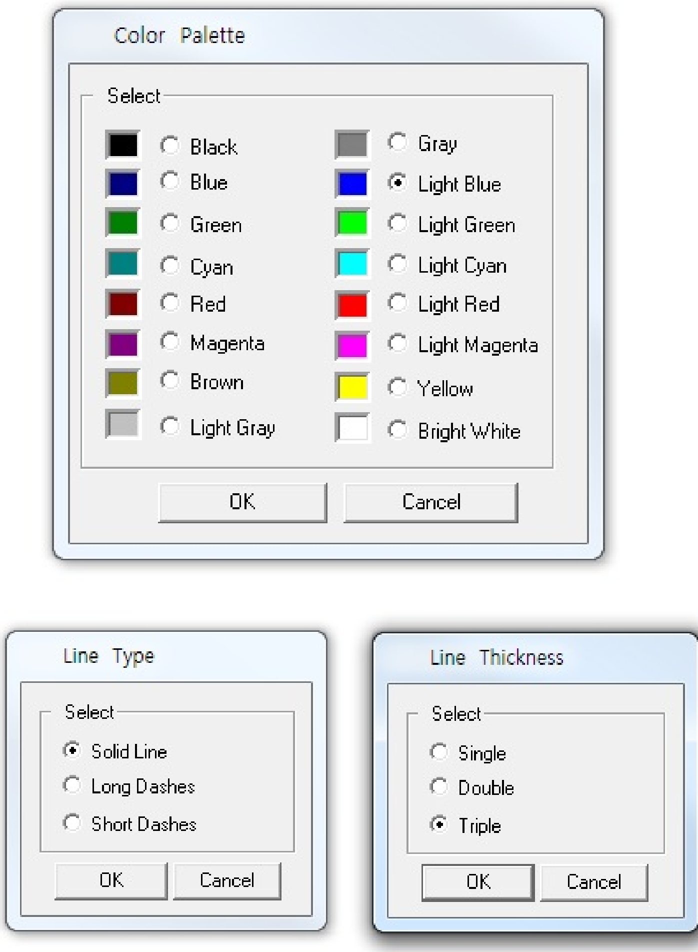


Figure 5.6 Line options.

5.3.4 Coordinate Constraint

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

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

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

5.3.5 Element Activity

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

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

5.3.6 PLOT-2D Plot

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

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

5.3.7 Translation

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

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

5.3.8 Command Buttons

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

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

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

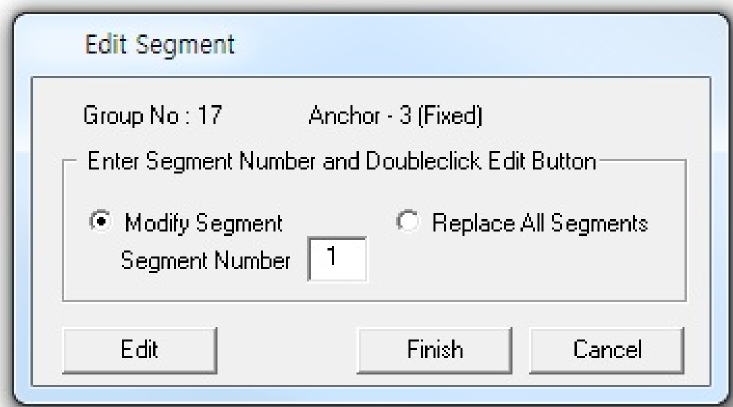


Figure 5.7 Edit segment dialog.

Show Number

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

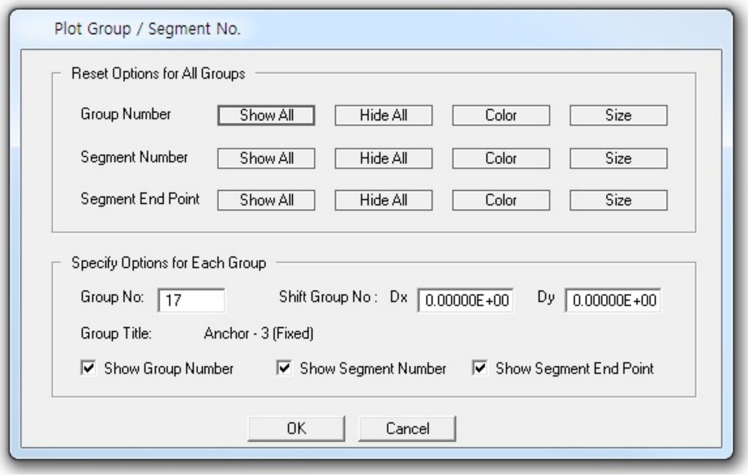


Figure 5.8 Plot Group / Segment No dialog.

Update

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

Save

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

Base Mesh

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

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

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

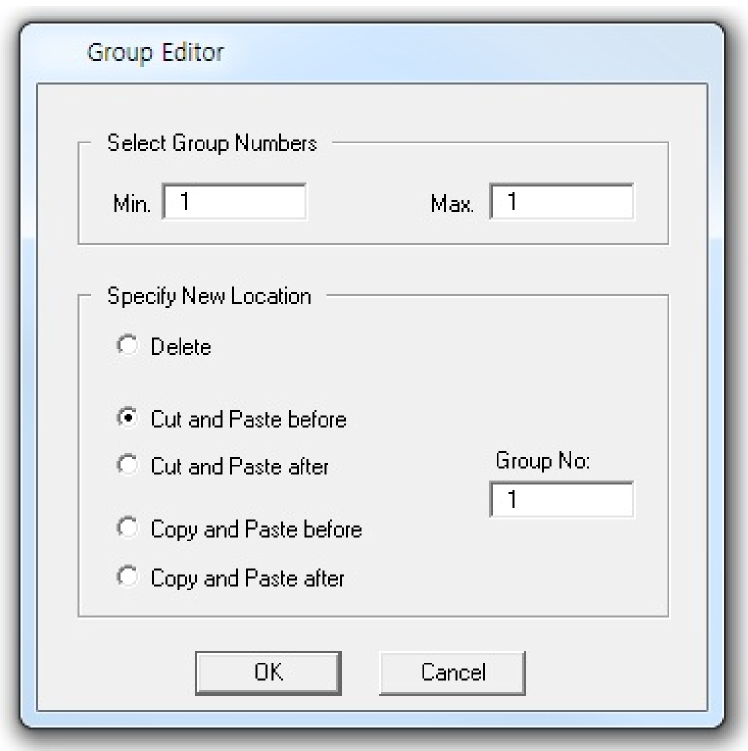


Figure 5.9 Group editor dialog.

Segment Editor

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

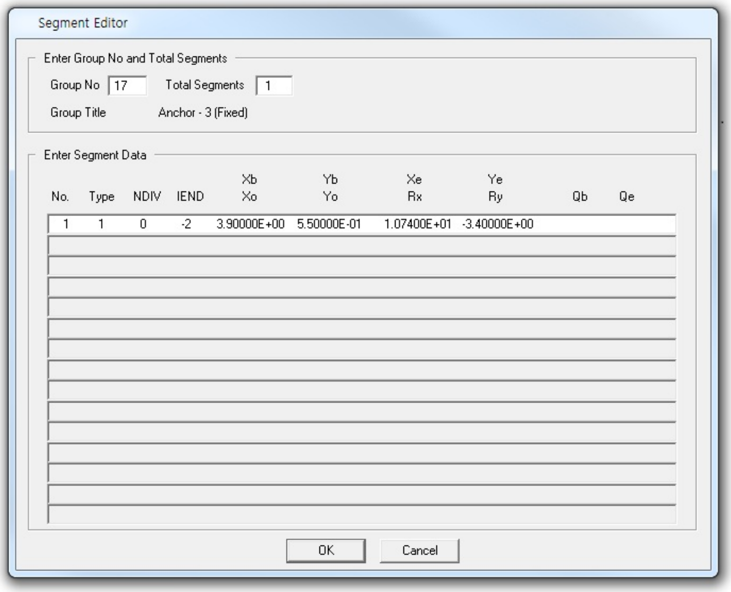


Figure 5.10 Segment editor dialog.

F. E. Mesh Plot

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

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

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

Close

This is used to close the Group dialog.

Exit

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

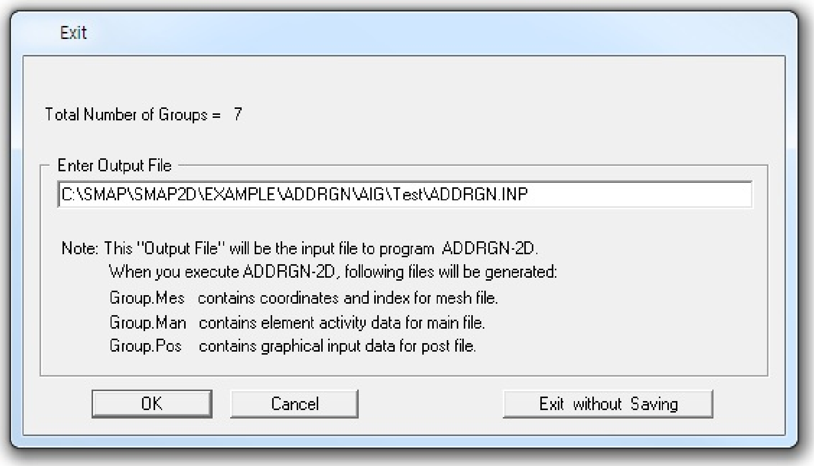


Figure 5.11 Exit dialog.

5.4 Base Mesh

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

5.4.1 Built-in Base Mesh

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

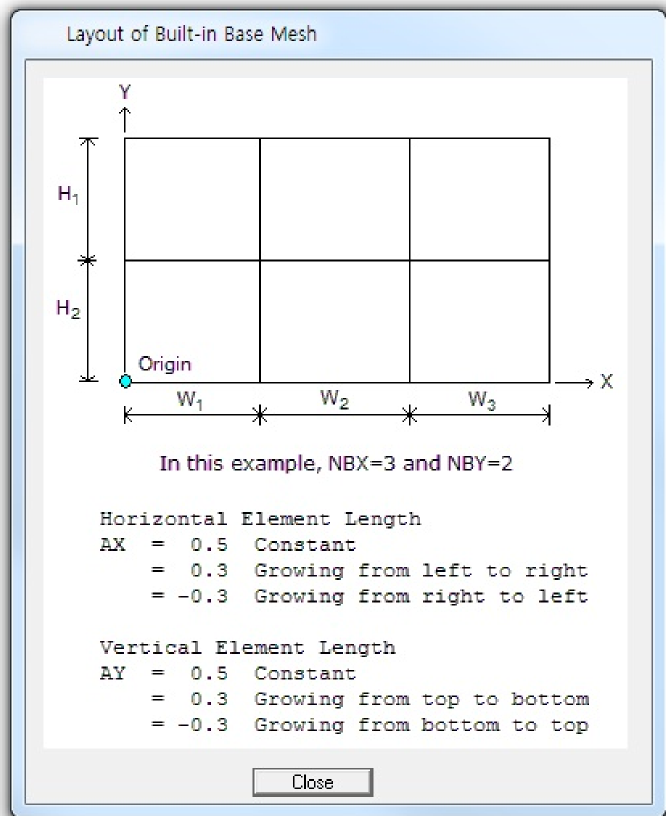


Figure 5.12 Layout of built-in base mesh.

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

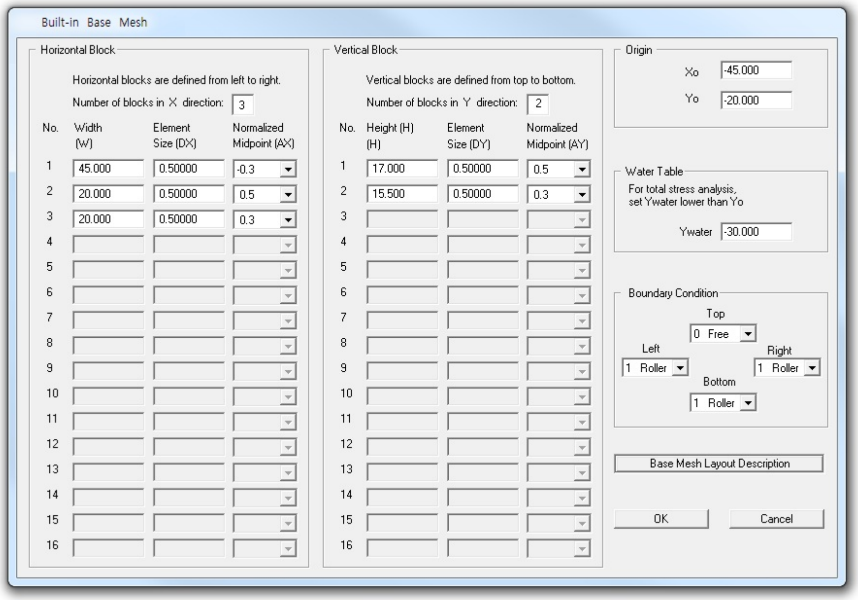


Figure 5.13 Built-in base mesh dialog.

5.4.2 Existing Finite Element Mesh

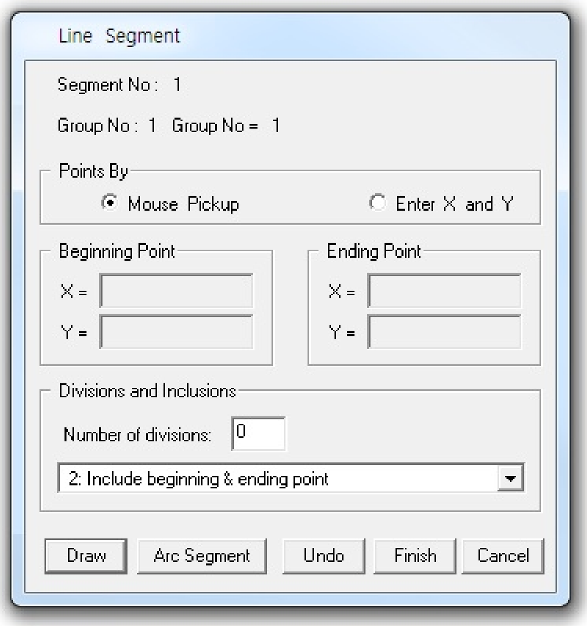
You can provide existing finite element mesh file to be used as base mesh. Group meshes will be built on this finite element mesh.

5.5 Segment

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

5.5.1 Line Segment

Figure 5.14
Line segment dialog.



Line Segment dialog is shown in Figure 5.14.

Segment No
Current segment number will be displayed automatically.

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

Point By
Select Mouse Pickup or Enter X and Y.

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

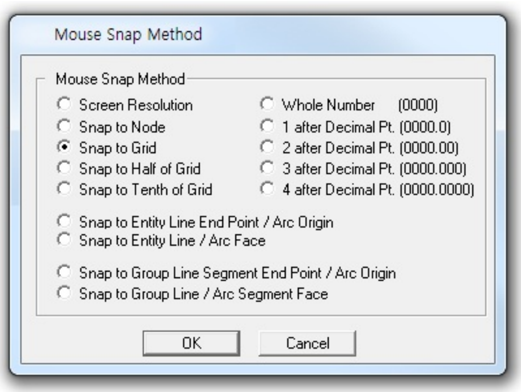
Divisions and Inclusions
Use following default values.
Number of divisions 0
Combo box selection 2: Include beginning & ending point

Draw
Draw line segment.

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

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

Figure 5.15
Mouse snap dialog



For Enter X and Y,

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

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

Arc Segment
Switch to arc segment.

Undo
Undo the changes you just made for line segment.

Finish
Finish and exit from drawing the current group.

Cancel
Cancel and exit from drawing the current group.

5.5.2 Arc Segment

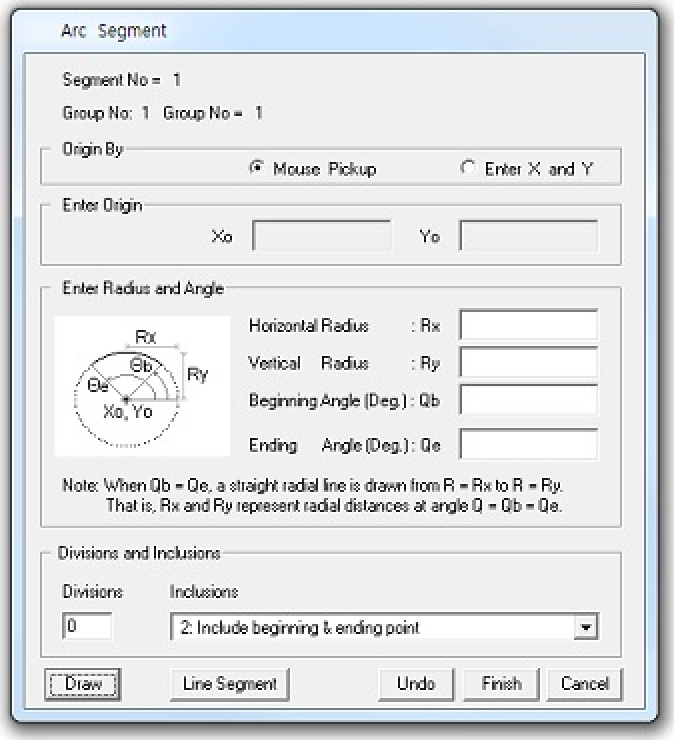


Figure 5.16 Arc segment dialog.

Arc Segment dialog is shown in Figure 5.16.

Segment No
Current segment number will be displayed automatically.

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

Origin By
Select Mouse Pickup or Enter X and Y.

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

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

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

Draw
Draw arc segment.

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

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

Refer to Note 1 & 2 in Section 5.5.1.

Line Segment
Switch to line segment.

Undo
Undo the changes you just made for arc segment.

Finish
Finish and exit from drawing the current group.

Cancel
Cancel and exit from drawing the current group.

5.6 Modifying Finite Element Meshes

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

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

Editing finite element meshes has three parts: Nodal Boundary, Nodal Coordinate and Element Material. These editing modes can be accessed from Mesh menu in PLOT-2D as shown in Figure 5.17.

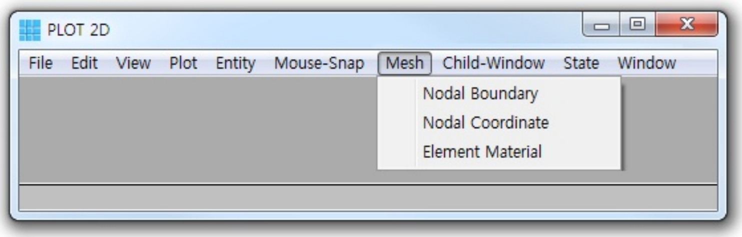


Figure 5.17 Menu for editing finite element mesh

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

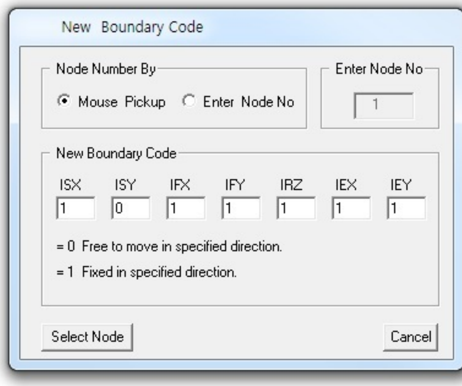
5.6.1 Edit Nodal Boundary

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

5.6.1.1 Mouse Pickup

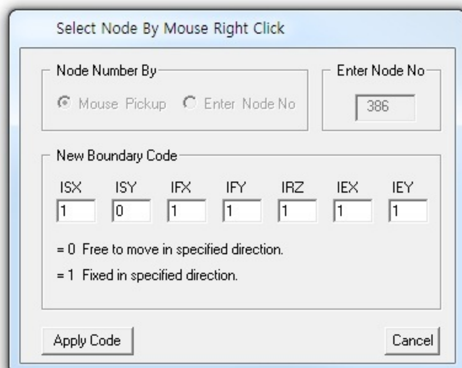
When you select Mouse Pickup mode as in Figure 5.18, you are supposed to select node number by mouse click. Click Select Node button.

Figure 5.18
Edit boundary
(Mouse Pickup)



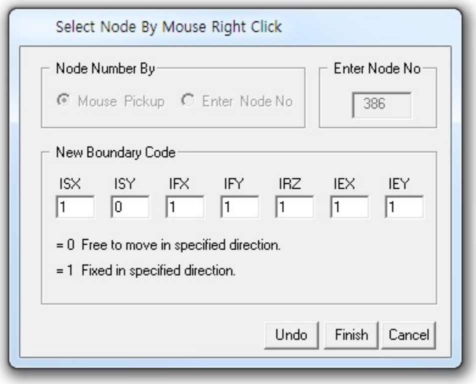
Click the node by Mouse Right Click, edit boundary codes and then click Apply Code button in Figure 5.19.

Figure 5.19
Edit boundary
(Apply Code)



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

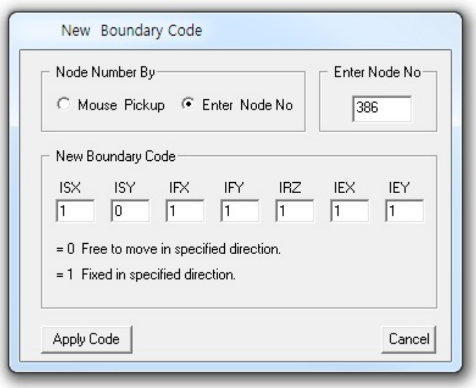
Figure 5.20
Edit boundary (Finish)



5.6.1.2 Enter Node No

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

Figure 5.21
Edit boundary (Enter Node No)



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

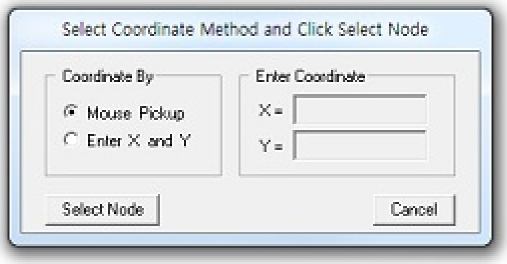
5.6.2 Edit Nodal Coordinate

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

5.6.2.1 Mouse Pickup

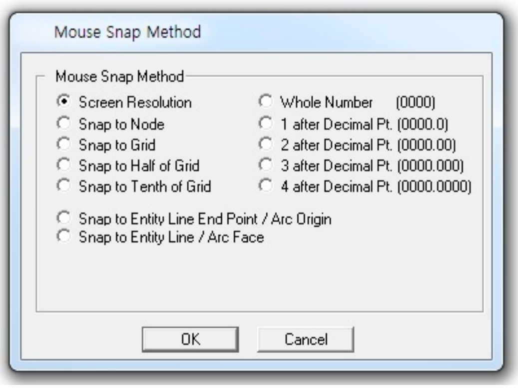
When you select Mouse Pickup mode as in Figure 5.22, you are supposed to select node number by mouse click. Click Select Node button.

Figure 5.22
Edit coordinate
(Mouse Pickup)



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

Figure 5.23
Mouse snap method



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

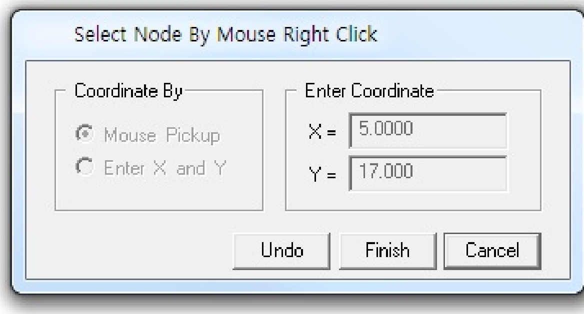


Figure 5.24 Edit coordinate (Finish)

5.6.2.2 Enter X and Y

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

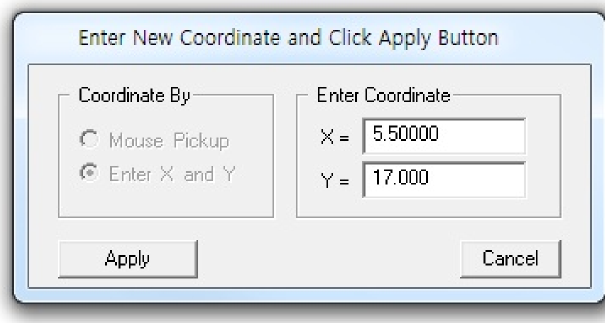


Figure 5.25 Edit coordinate (Enter X and Y)

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

5.6.3 Edit Element Material

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

5.6.3.1 Mouse Pickup

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

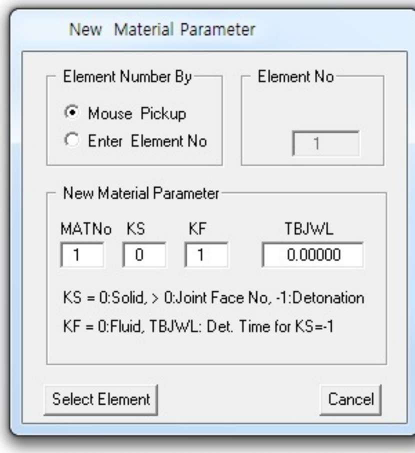


Figure 5.26
Edit element material
(Mouse Pickup)

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

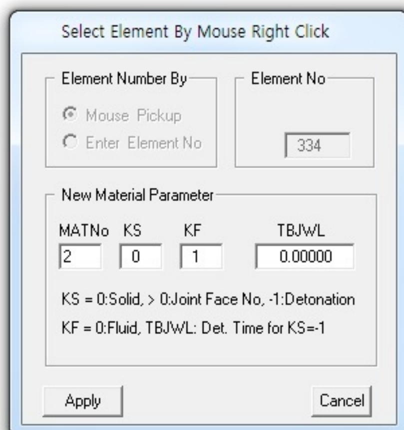


Figure 5.27
Edit element material
(Apply)

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

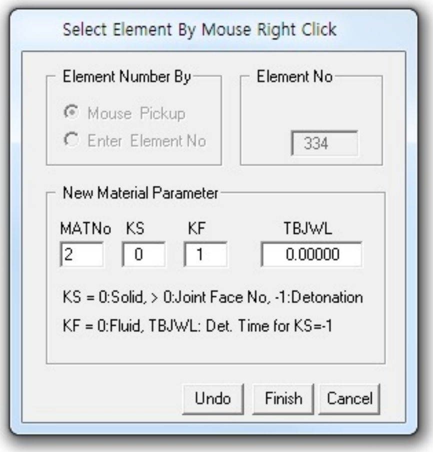


Figure 5.28
Edit element material
(Finish)

5.6.3.2 Enter Element No

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

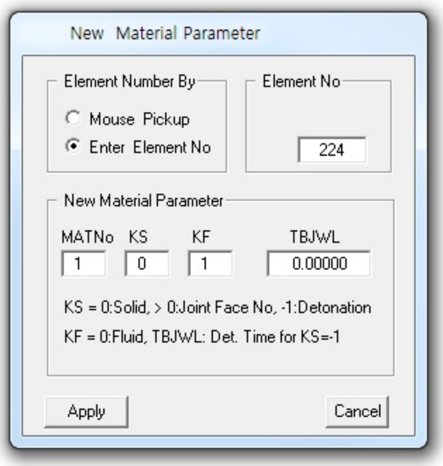


Figure 5.29
Edit element material
(Enter Element No)

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

5.7 Entities

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

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

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

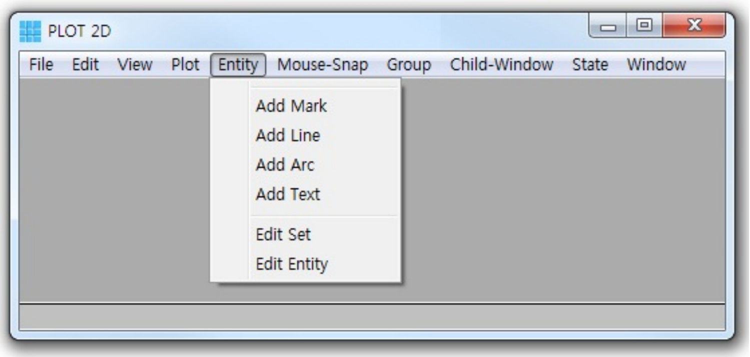


Figure 5.30 Entity menu

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

First four Add Entities are to build new entities.

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

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

5.7.1 Add Mark

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

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

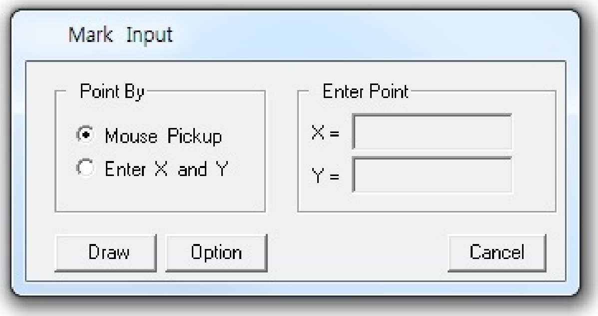


Figure 5.31
Mark input
(Mouse Pickup)

Option button is to show Mark Option in Figure 5.32.

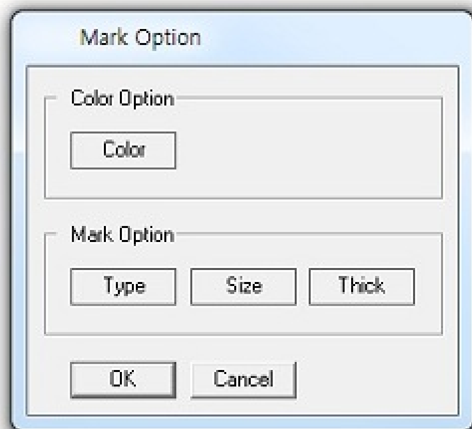
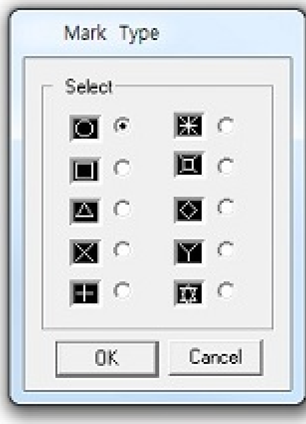


Figure 5.32
Mark option dialog

Available Mark Types are shown in Figure 5.33.

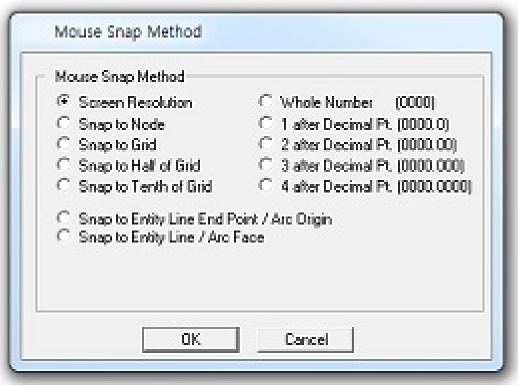
Figure 5.33 Mark type dialog



5.7.1.1 Mouse Pickup

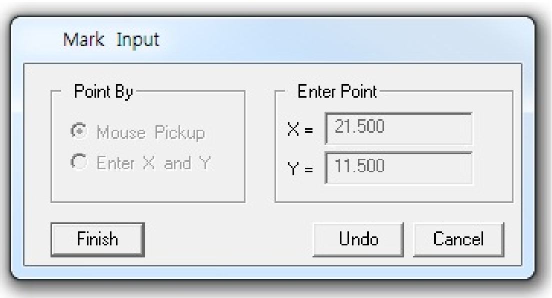
When you select Mouse Pickup mode as in Figure 5.31, you are supposed to select the mark center position by mouse click. Click Draw button and then move the position by Mouse Left Click. It is convenient to select an appropriate Mouse-Snap method in Figure 5.34 before moving the position.

Figure 5.34
Mouse snap method



Once finished, click Finish button in Figure 5.35.

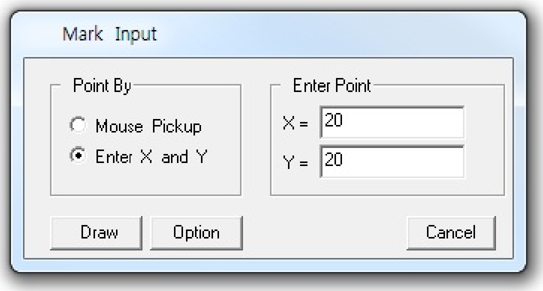
Figure 5.35
Mark input
(Finish)



5.7.1.2 Enter X and Y

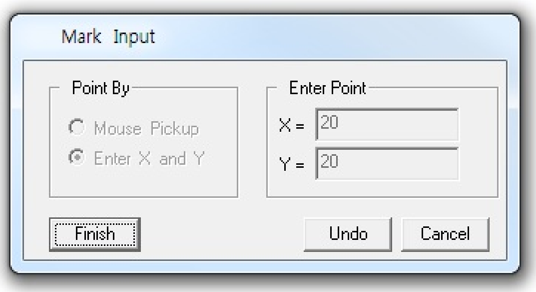
When you select Enter X and Y mode as in Figure 5.36, you are supposed to type in the coordinates of the mark center position. Click Draw button.

Figure 5.36
Mark input
(Enter X and Y)



Once finished, click Finish button in Figure 5.37.

Figure 5.37
Mark input
(Finish)

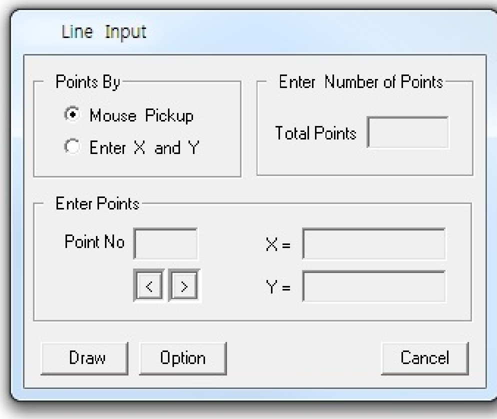


5.7.2 Add Line

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

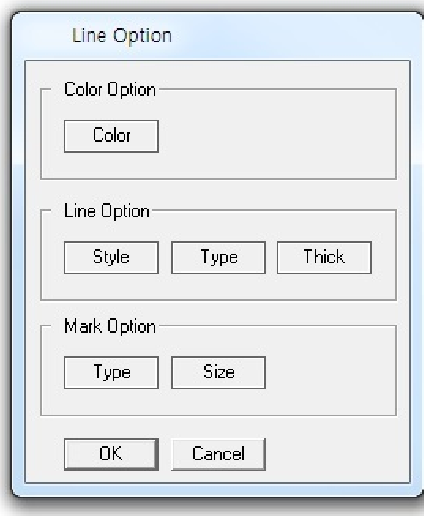
When you select Add Line submenu, Line Input dialog in Figure 5.38 is displayed.

Figure 5.38
Line input
(Mouse Pickup)



Option button is to show Line Option in Figure 5.39.

Figure 5.39
Line option dialog



Available Line Styles are shown in Figure 5.40.

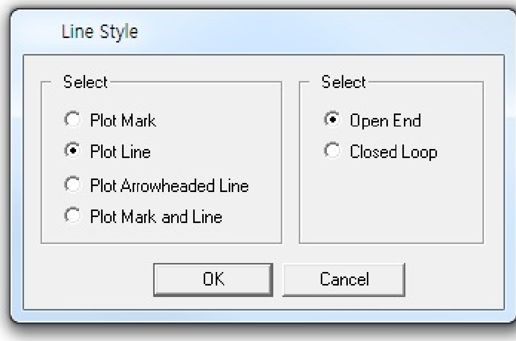


Figure 5.40 Line style dialog

Available Line Types are shown in Figure 5.41.

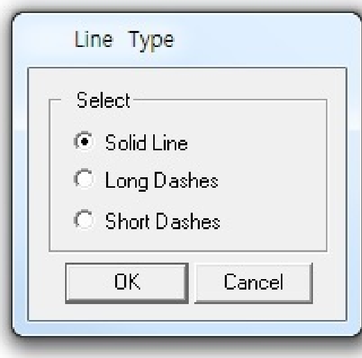


Figure 5.41 Line type dialog

5.7.2.1 Mouse Pickup

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

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

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

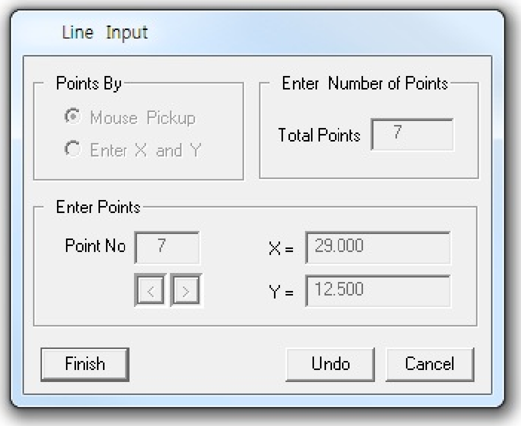
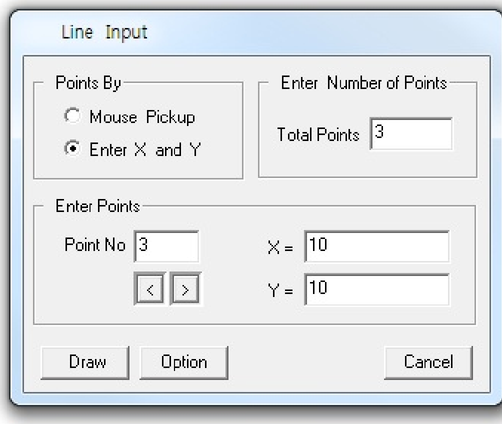


Figure 5.42 Line input (Finish)

5.7.2.2 Enter X and Y

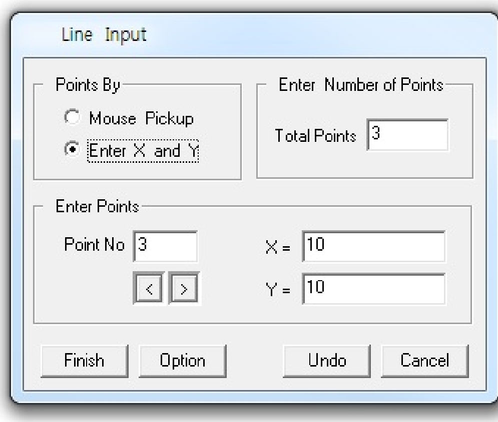
When you select Enter X and Y mode as in Figure 5.43, you are supposed to type the coordinates of the line. Click Draw button.

Figure 5.43
Line input
(Enter X and Y)



And then click Finish button in Figure 5.44.

Figure 5.44
Line input
(Finish)

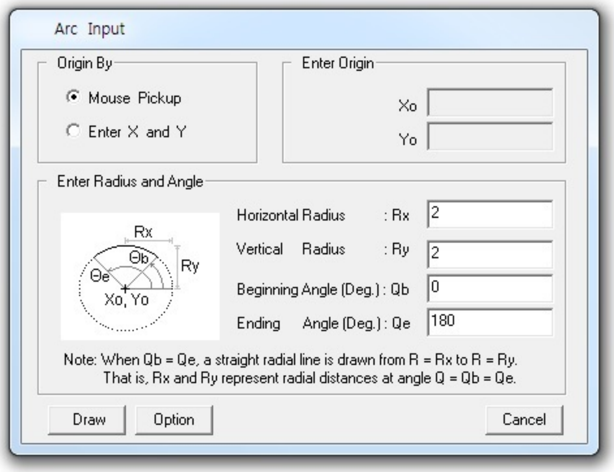


5.7.3 Add Arc

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

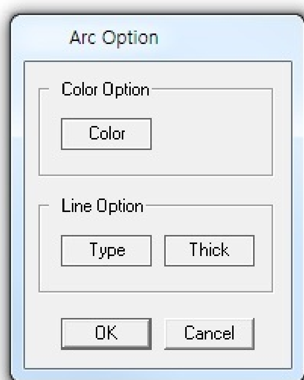
When you select Add Arc submenu, Arc Input dialog in Figure 5.45 is displayed.

Figure 5.45
Arc input
(Mouse Pickup)



Option button is to show Arc Option in Figure 5.46.

Figure 5.46 Arc option dialog



5.7.3.1 Mouse Pickup

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

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

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

Once finished, click Finish button in Figure 5.47.

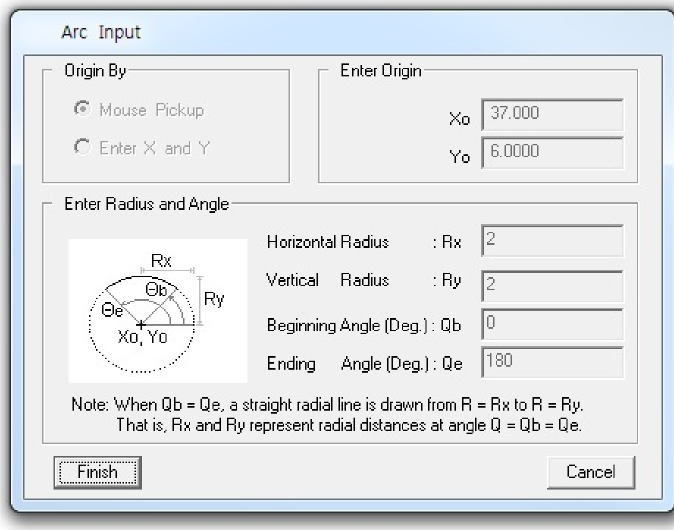


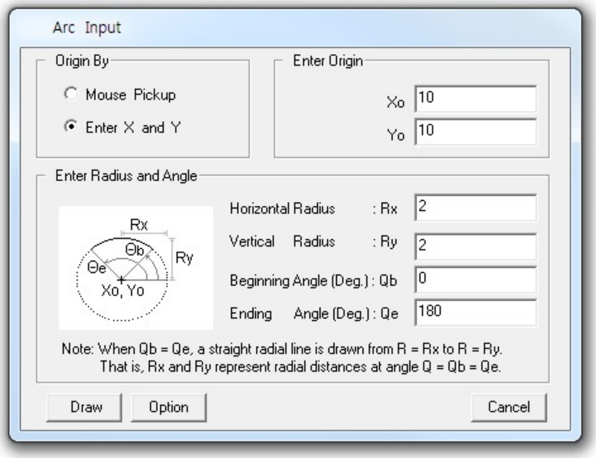
Figure 5.47 Arc input (Finish)

5.7.3.2 Enter X and Y

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

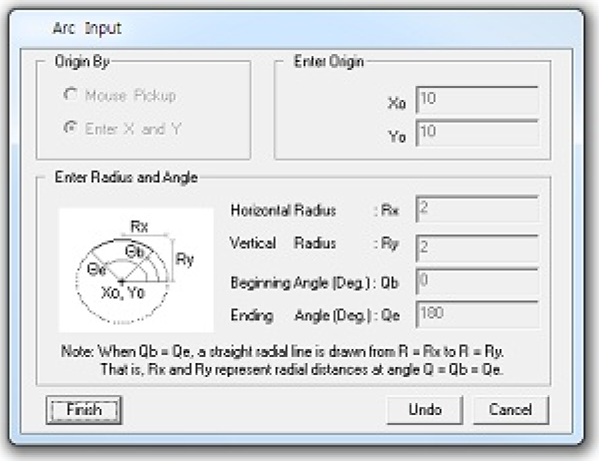
Type in Horizontal Radius, Vertical Radius, Beginning Angle and Ending Angle. And then click Draw button.

Figure 5.48
Arc input
(Enter X and Y)



Once finished, click Finish button in Figure 5.49.

Figure 5.49
Arc input
(Finish)

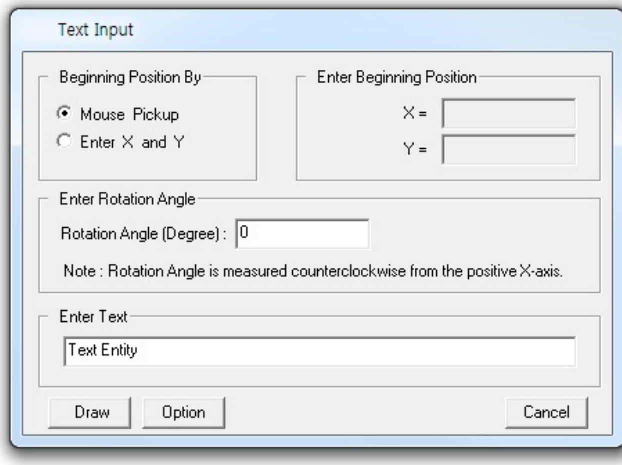


5.7.4 Add Text

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

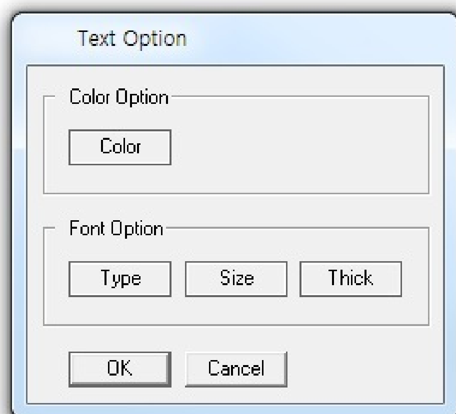
When you select Add Text submenu, Text Input dialog in Figure 5.50 is displayed.

Figure 5.50
Text input
(Mouse Pickup)



Option button is to show Text Option in Figure 5.51.

Figure 5.51
Text option dialog



Available Font Sizes are shown in Figure 5.52.

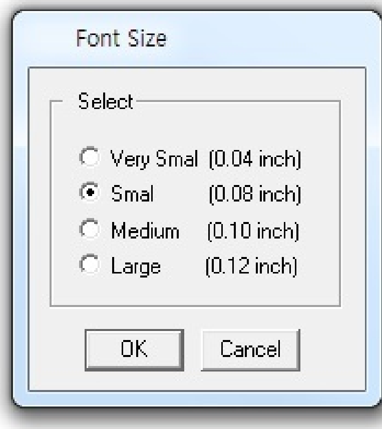


Figure 5.52 Font size dialog

5.7.4.1 Mouse Pickup

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

Type in Rotation Angle and Text.

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

Once finished, click Finish button in Figure 5.53.

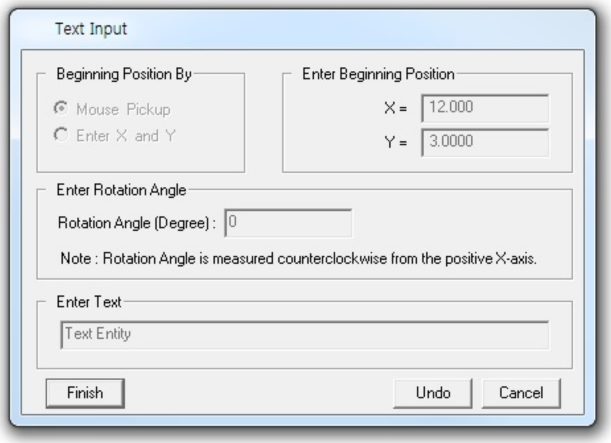


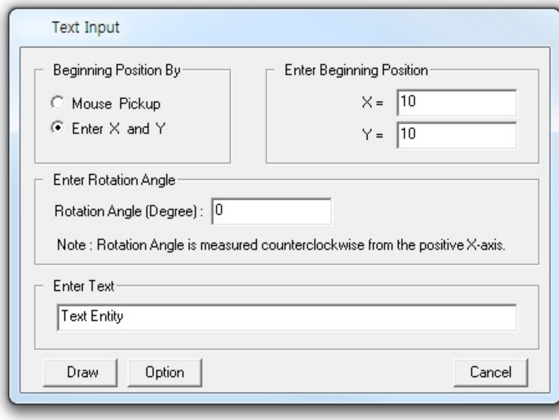
Figure 5.53 Text input (Finish)

5.7.4.2 Enter X and Y

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

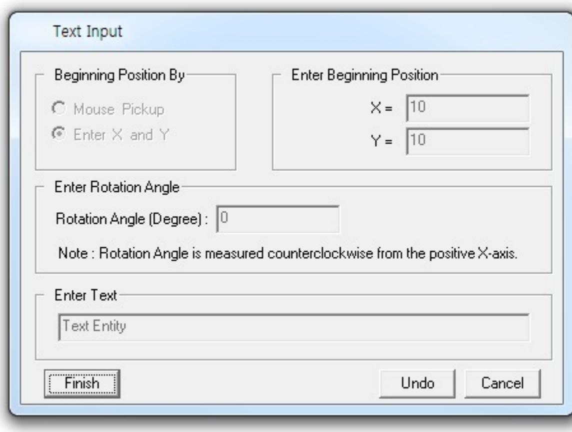
Type in Rotation Angle and Text. And then click Draw button.

Figure 5.54
Text input
(Enter X and Y)



Once finished, click Finish button in Figure 5.55.

Figure 5.55
Text input
(Finish)



5.7.5 Edit Set

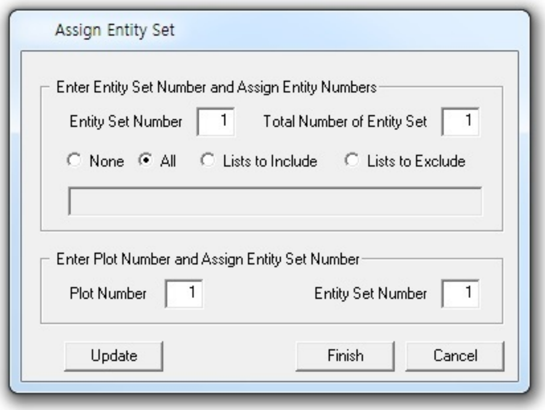
Edit Set is to assign Entity Set as shown in Figure 5.56.

Edit Set consists of two parts:

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

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

Figure 5.56
Assign entity set dialog



5.7.5.1 Enter Entity Set No & Assign Entity No

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

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

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

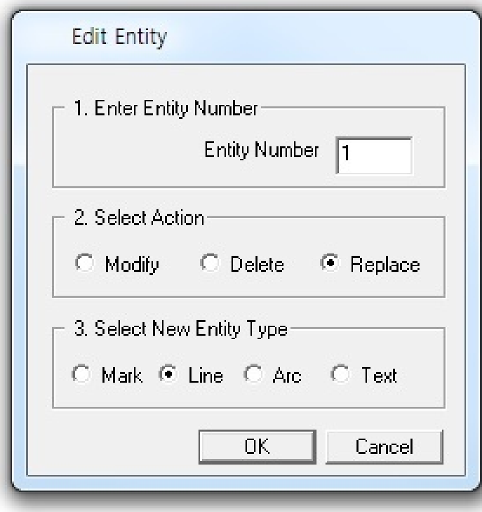
5.7.5.2 Enter Plot No & Assign Entity Set No

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

5.7.6 Edit Entity

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

Figure 5.57
Edit entity dialog



5.7.6.1 Modify

Modify is to modify the current entity.

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

5.7.6.2 Delete

Delete is to delete the current entity.

5.7.6.3 Replace

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

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

Block Mesh User's Manual

6.1 Introduction

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

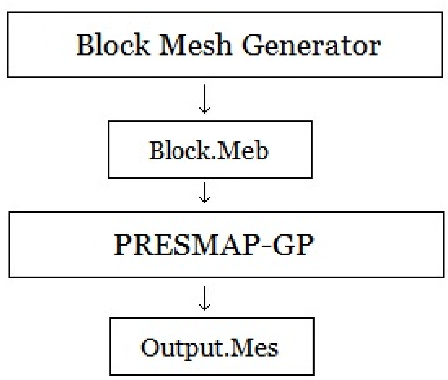


Figure 6.1 Flow diagram of block mesh generation

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

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

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

6.2 Block Mesh Generator

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

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

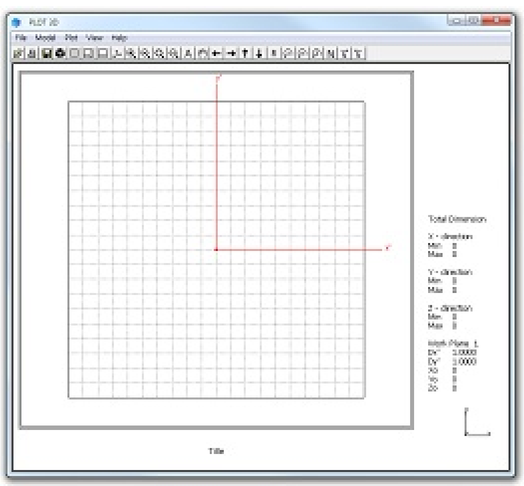


Figure 6.2 Prebuilt work plane on PLOT-3D

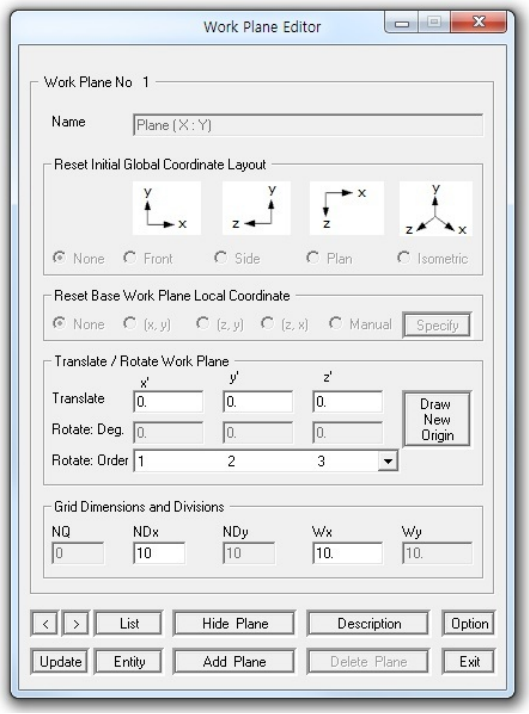


Figure 6.3 Prebuilt work plane editor

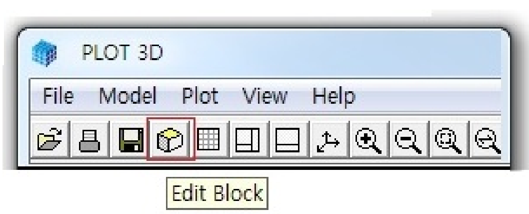


Figure 6.4 Block editor toolbar

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

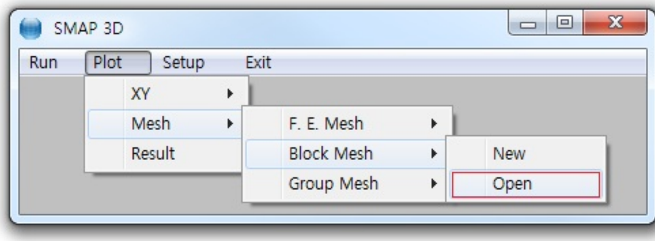


Figure 6.5 Open input file dialog

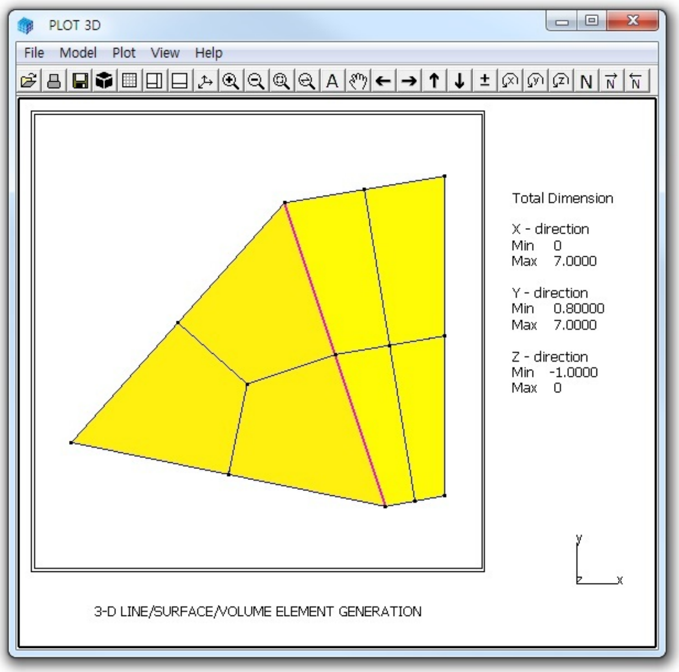


Figure 6.6 Block mesh on PLOT-3D

6.3 Work Plane

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

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

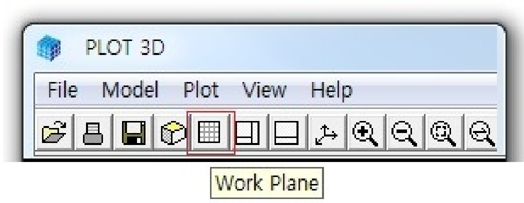


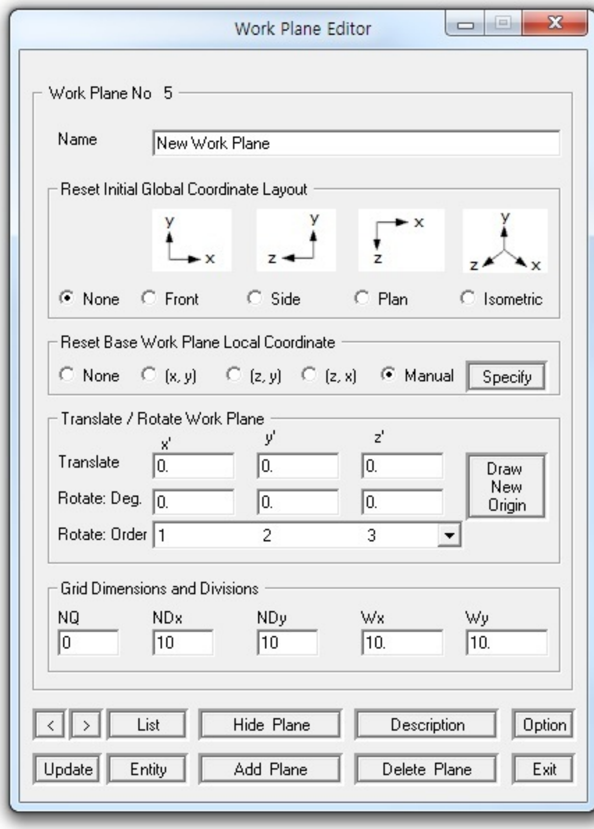
Figure 6.7 Work plane toolbar

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

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

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

Figure 6.8
Work plane editor



6.3.1 Name

Name is work plane name you can specify for identification.

6.3.2 Reset Initial Global Coordinate Layout

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

6.3.3 Reset Base Work Plane Local Coordinate

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

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

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

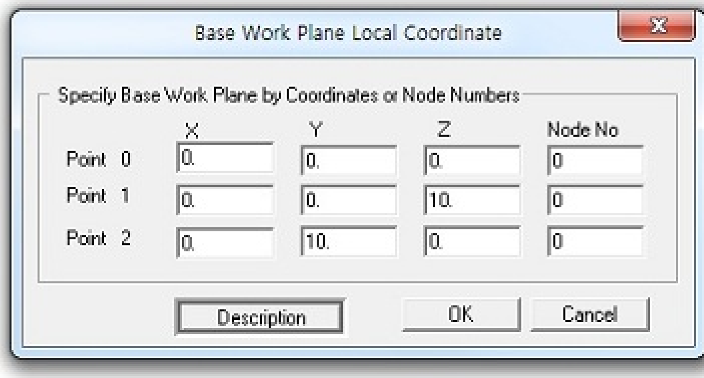


Figure 6.9 Base work plane local coordinate dialog

6.3.4 Translate / Rotate Work Plane

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

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

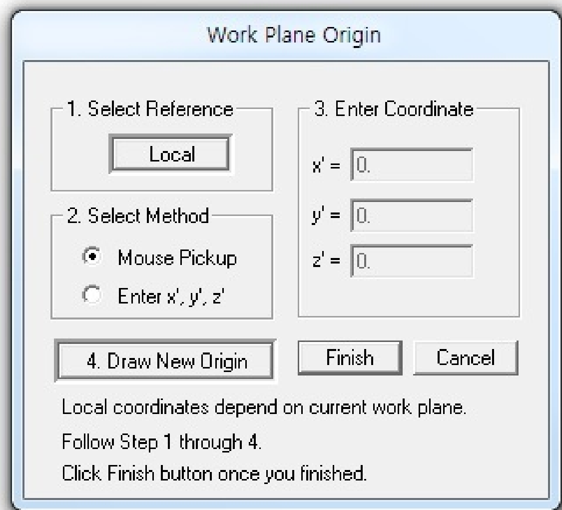


Figure 6.10 Work plane origin dialog

6.3.5 Grid Dimensions and Divisions

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

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

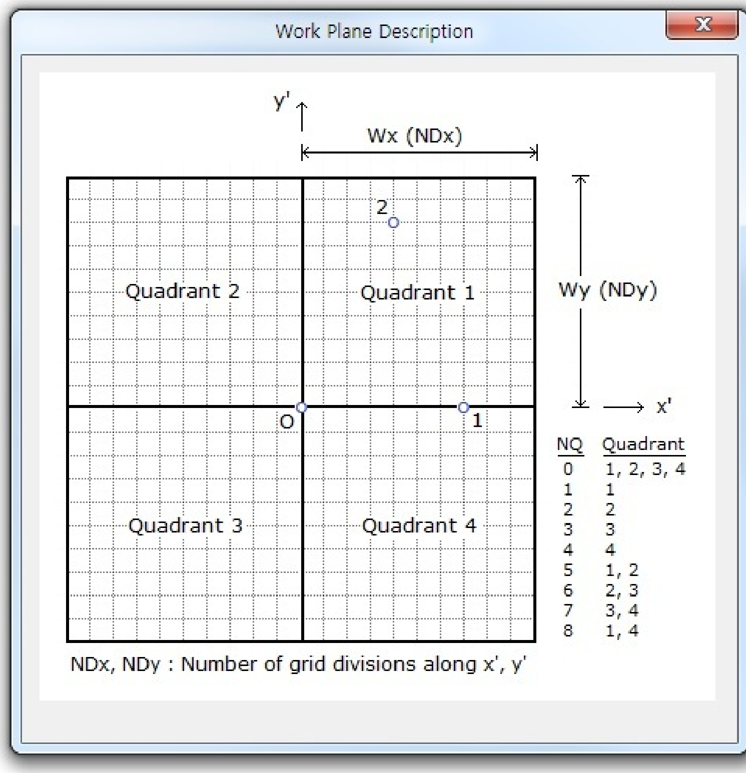


Figure 6.11 Work plane description

6.3.6 Command Buttons

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

List

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

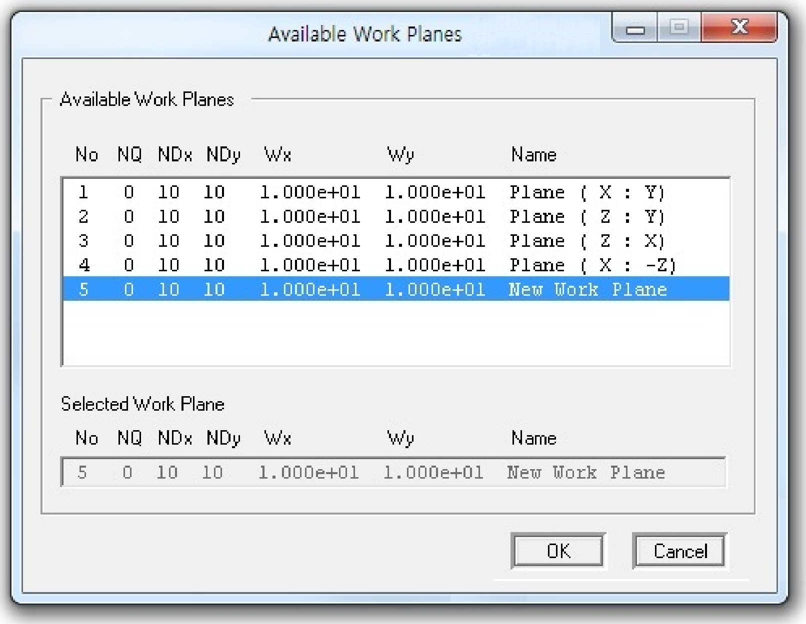


Figure 6.12 Work plane list

Hide Plane

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

Description

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

Option

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

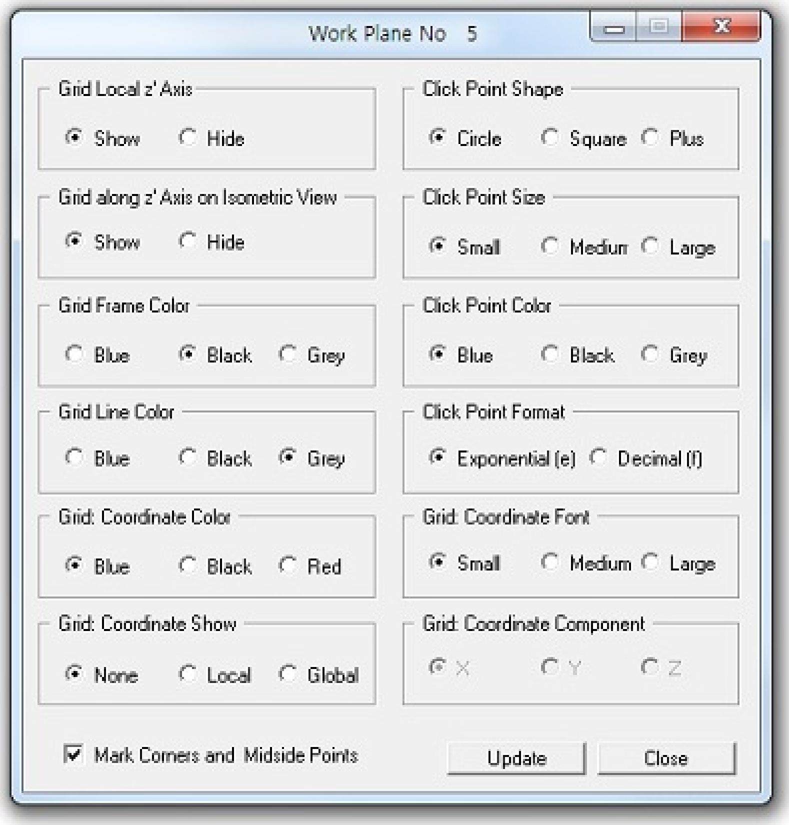


Figure 6.13 Work plane option dialog

Update

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

Entity

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

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

Add Plane

This is used to add new work plane.

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

Delete Plane

This is used to delete the current work plane.

Exit

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

6.3.7 Prebuilt Work Planes

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

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

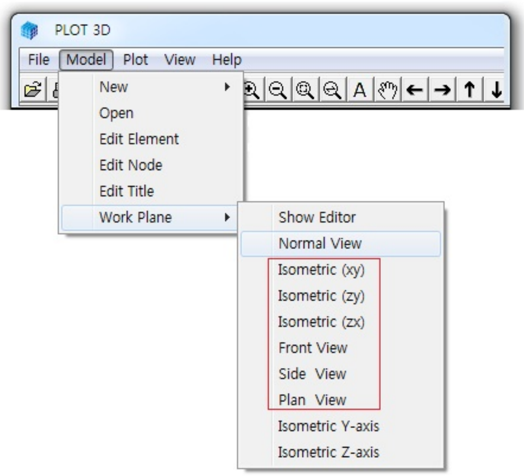


Figure 6.14 Prebuilt work plane menus

or by clicking Axis toolbar as shown in Figure 6.15.

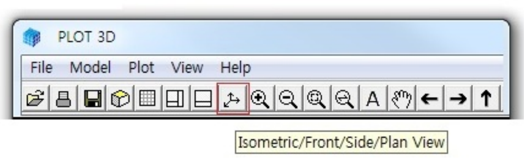


Figure 6.15 Axis toolbar

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

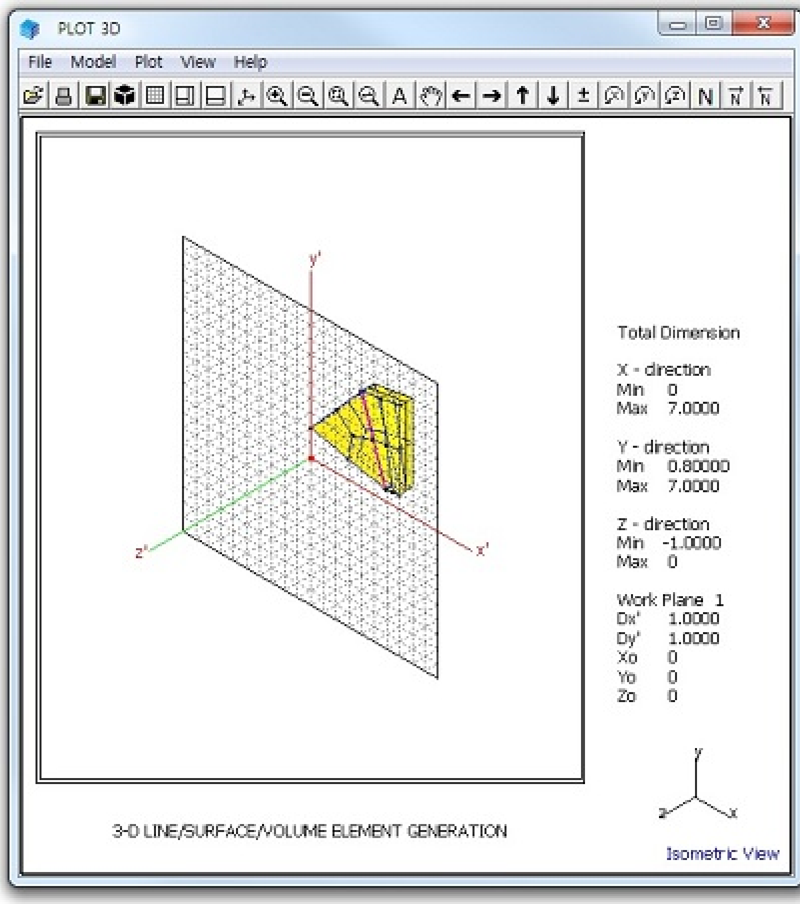


Figure 6.16 Prebuilt work plane: Isometric (xy) View

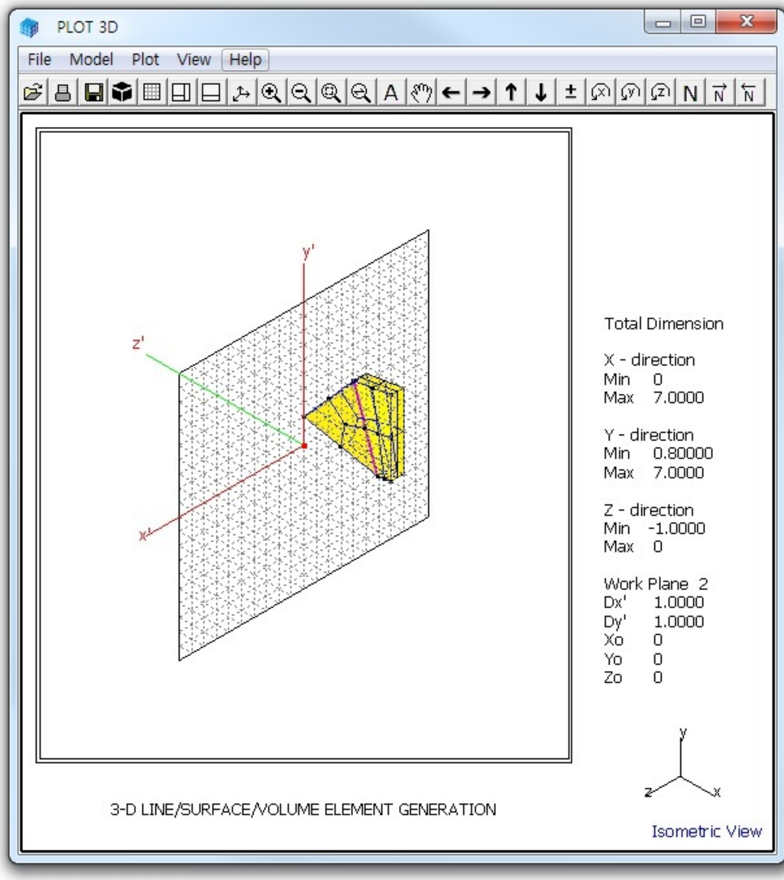


Figure 6.16 Prebuilt work plane: Isometric (zy) View

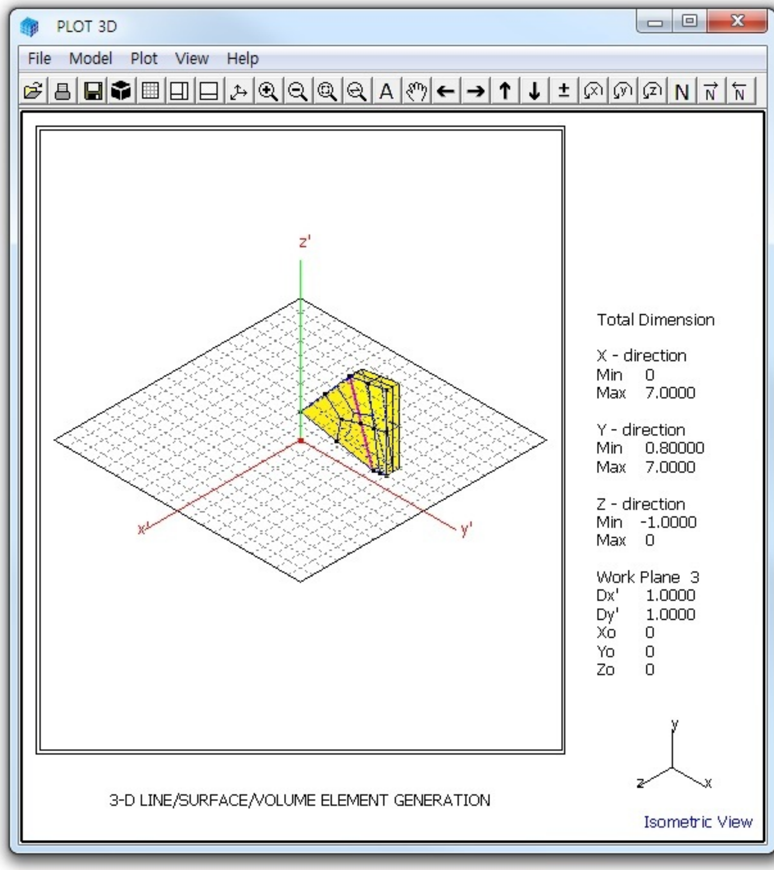


Figure 6.16 Prebuilt work plane: Isometric (zx) View

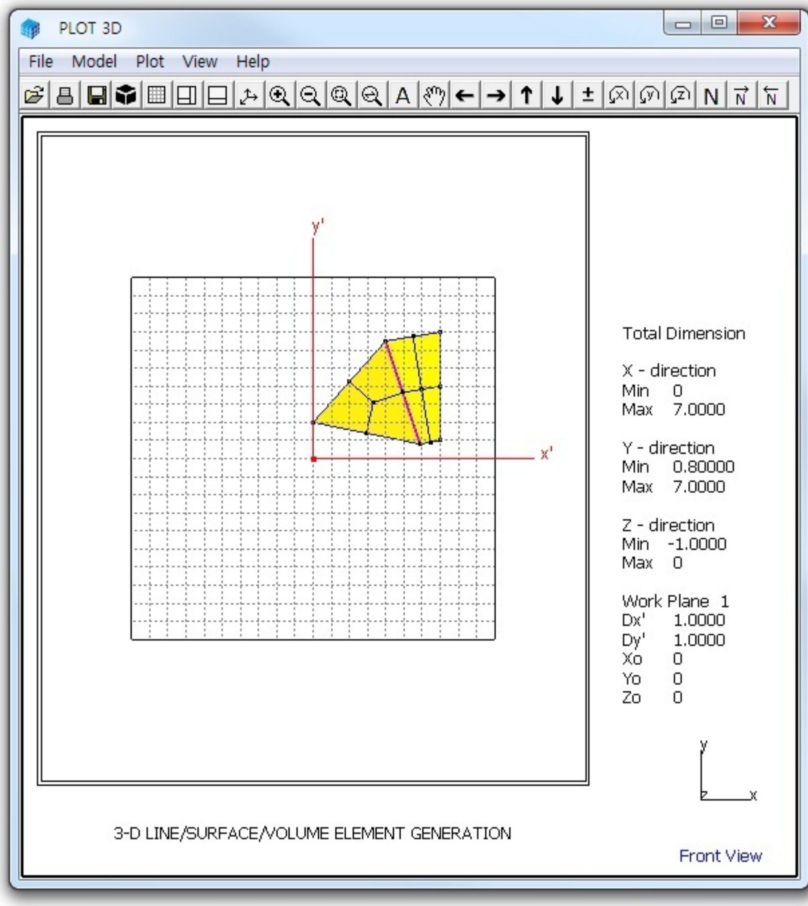


Figure 6.16 Prebuilt work plane: Front View

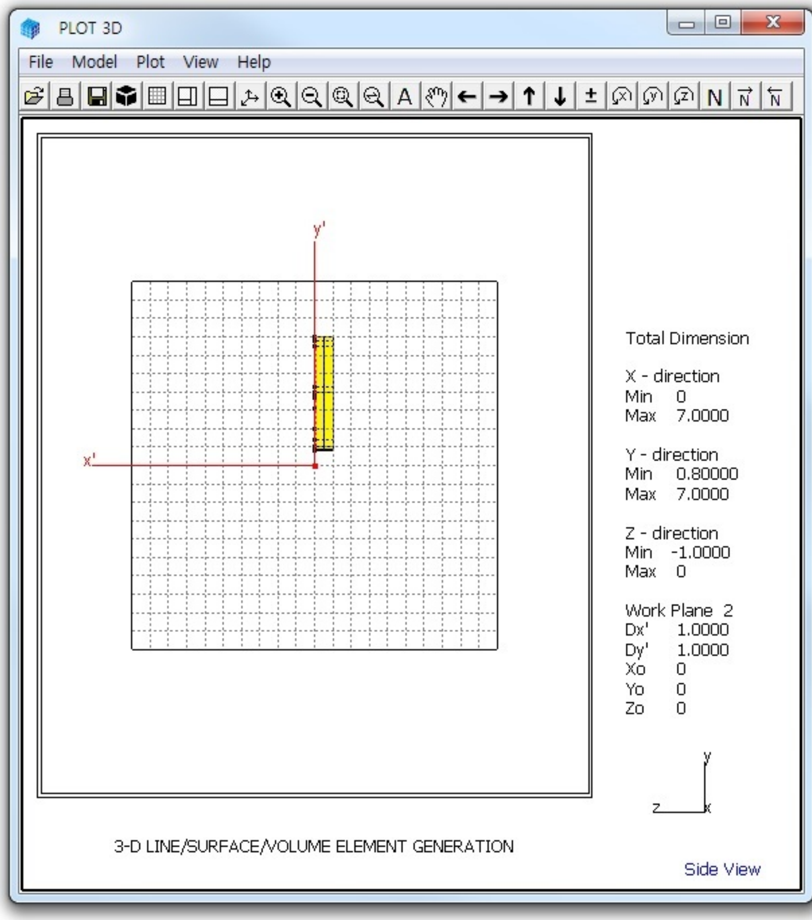


Figure 6.16 Prebuilt work plane: Side View

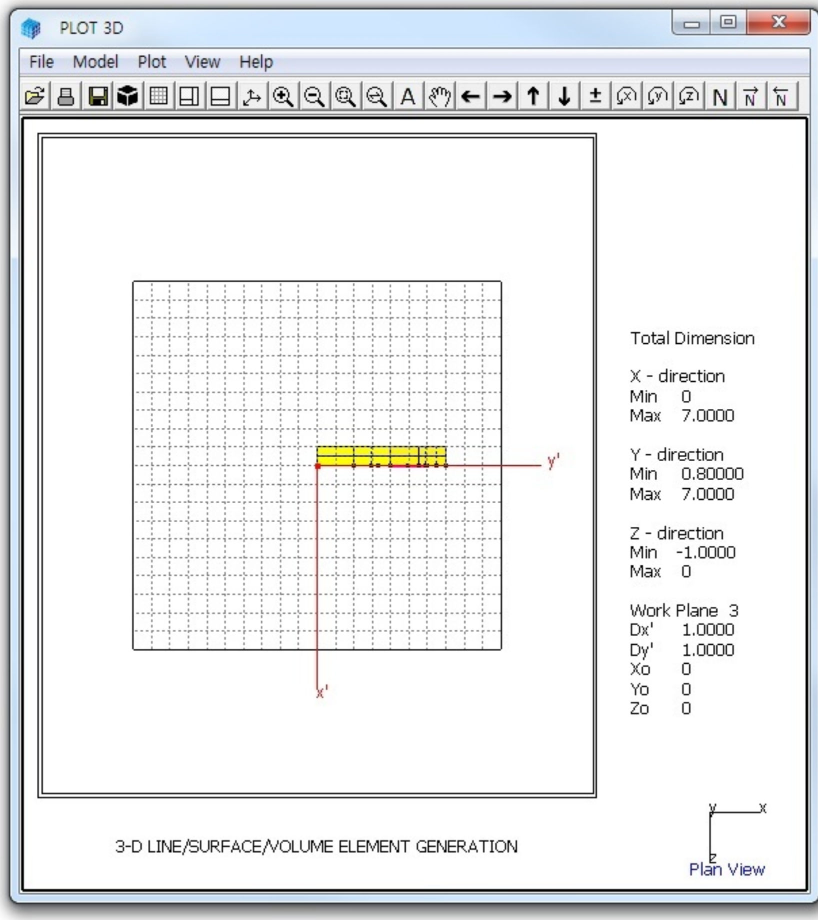


Figure 6.16 Prebuilt work plane: Plan View

6.4 Entities

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

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

Entity Editor dialog consists of following seven parts:

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

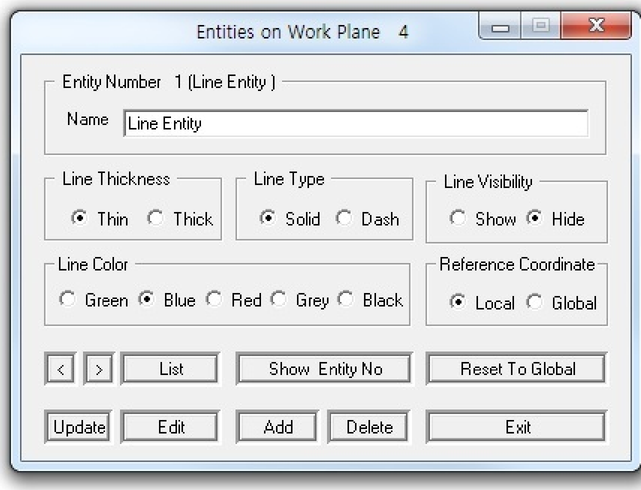


Figure 6.17 Entity editor dialog

6.4.1 Entity Number

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

6.4.2 Line Thickness

Two options are available: Thin and Thick.

6.4.3 Line Type

Two options are available: Solid and Dash.

6.4.4 Line Visibility

Two options are available: Show and Hide.

6.4.5 Line Color

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

6.4.6 Reference Coordinate

Two options are available: Local and Global.

6.4.7 Command Buttons

Command buttons are shown on Entity dialog in Figure 6.17.

List

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

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

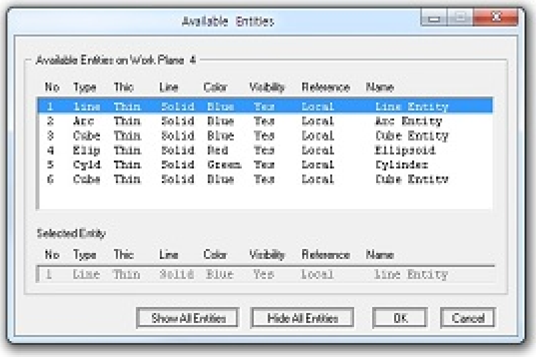


Figure 6.18
Entity list dialog

Show Entity No

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

Reset To Global

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

Update

This is used to update parameters of the current entity.

Edit

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

Add

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

Delete

This is used to delete the current entity.

Exit

This is used to exit from the Entity Editor dialog.

6.4.8 Popup Menu for Entity

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

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

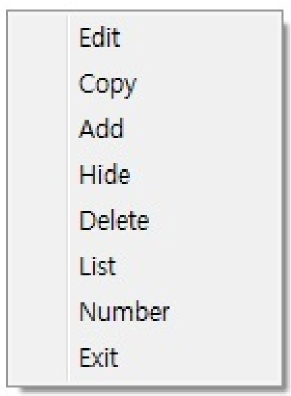


Figure 6.19 Popup menu for entity

6.4.9 Adding New Entity

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

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

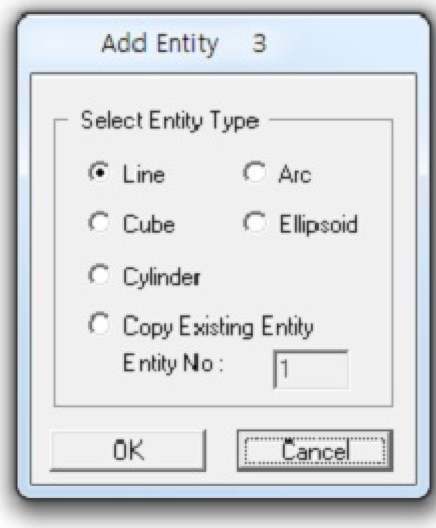


Figure 6.20 Entity type selection dialog

6.4.9.1 Line Entity

Line Entity dialog is shown in Figure 6.21.

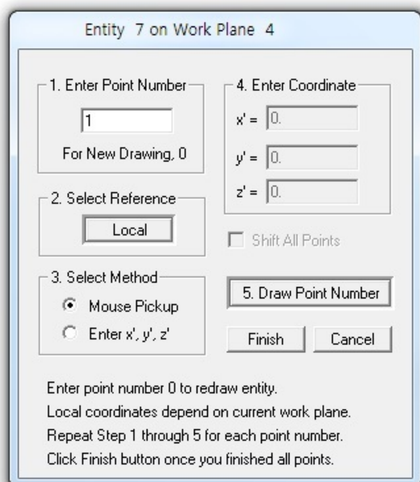
To draw Line Entity, follow five steps:

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

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

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

Figure 6.21
Line entity dialog



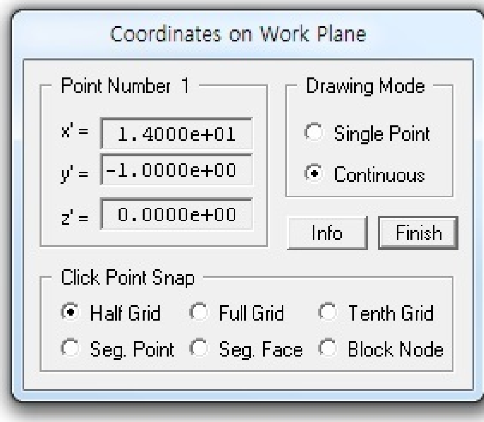


Figure 6.22 Coordinates on work plane

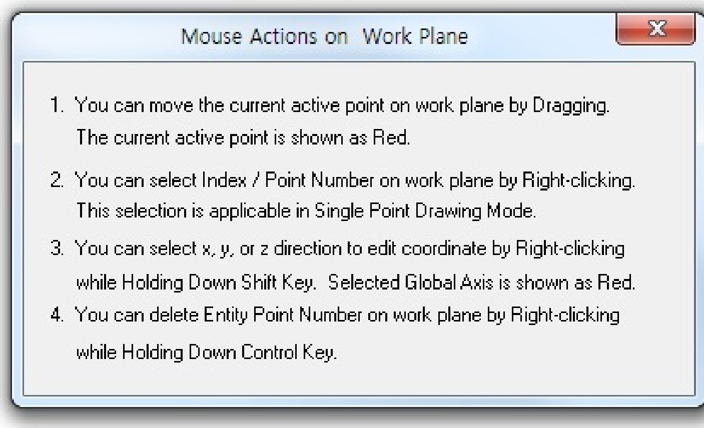


Figure 6.23 Mouse actions on work plane

6.4.9.2 Arc Entity

Arc Entity dialog is shown in Figure 6.24.

To draw Arc Entity, follow five steps:

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

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

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

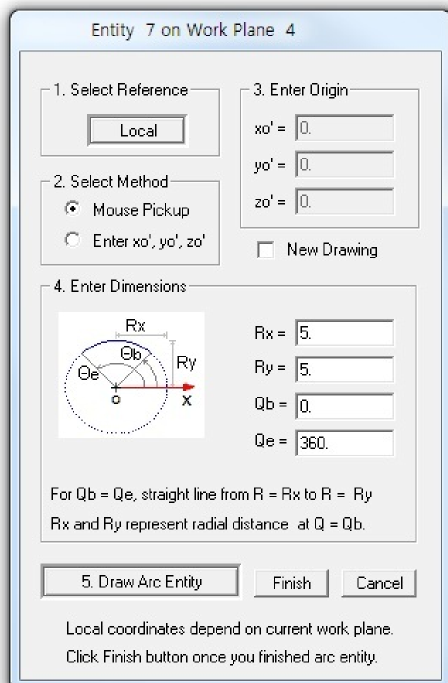


Figure 6.24
Arc entity dialog

6.4.9.3 Cube Entity

Cube Entity dialog is shown in Figure 6.25.

To draw Cube Entity, follow five steps:

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

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

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

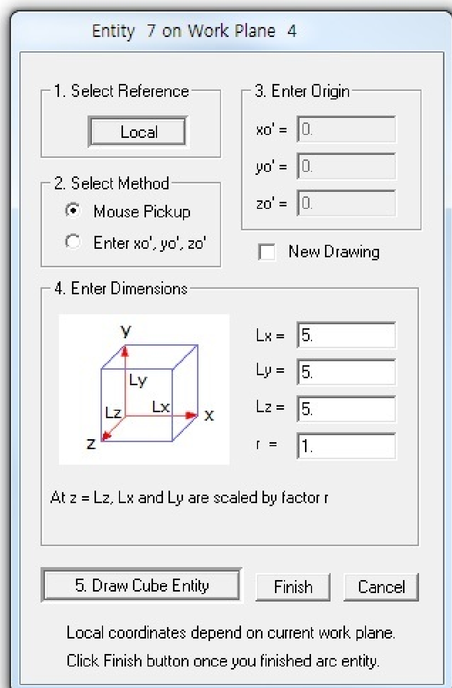


Figure 6.25
Cube entity dialog

6.4.9.4 Ellipsoid Entity

Ellipsoid Entity dialog is shown in Figure 6.26.

To draw Ellipsoid Entity, follow five steps:

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

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

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

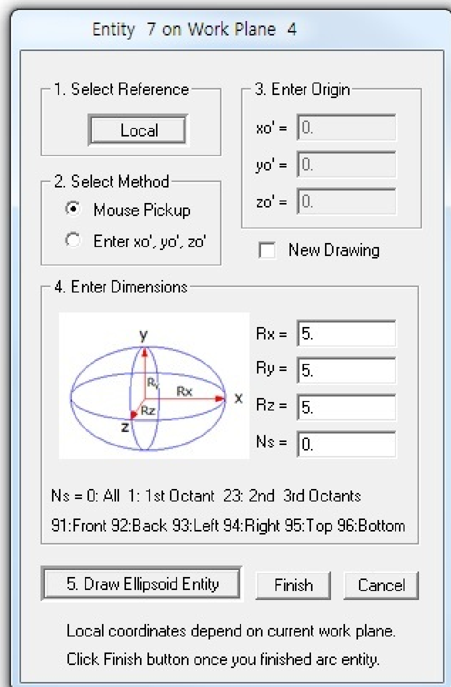


Figure 6.26
Ellipsoid entity dialog

6.4.9.5 Cylinder Entity

Cylinder Entity dialog is shown in Figure 6.27.

To draw Cylinder Entity, follow five steps:

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

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

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

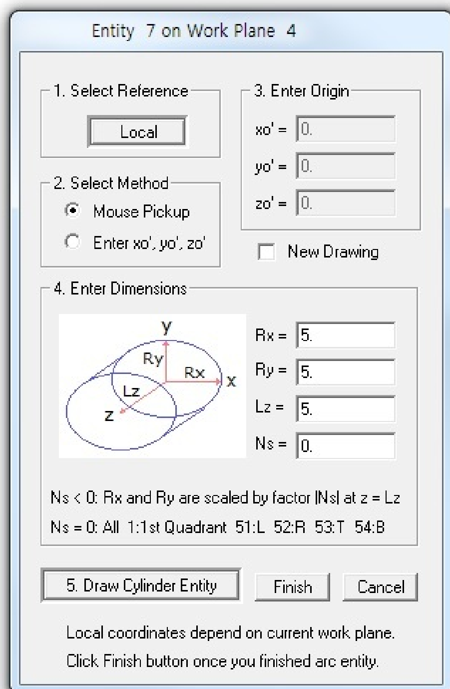


Figure 6.27
Cylinder entity dialog

6.5 Block

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

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

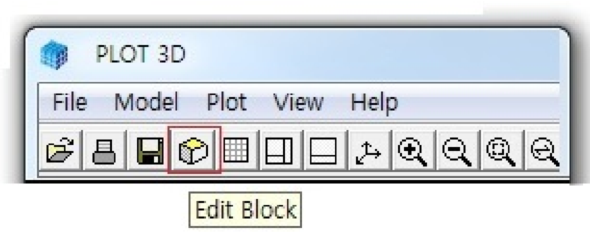


Figure 6.28 Block editor toolbar

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

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

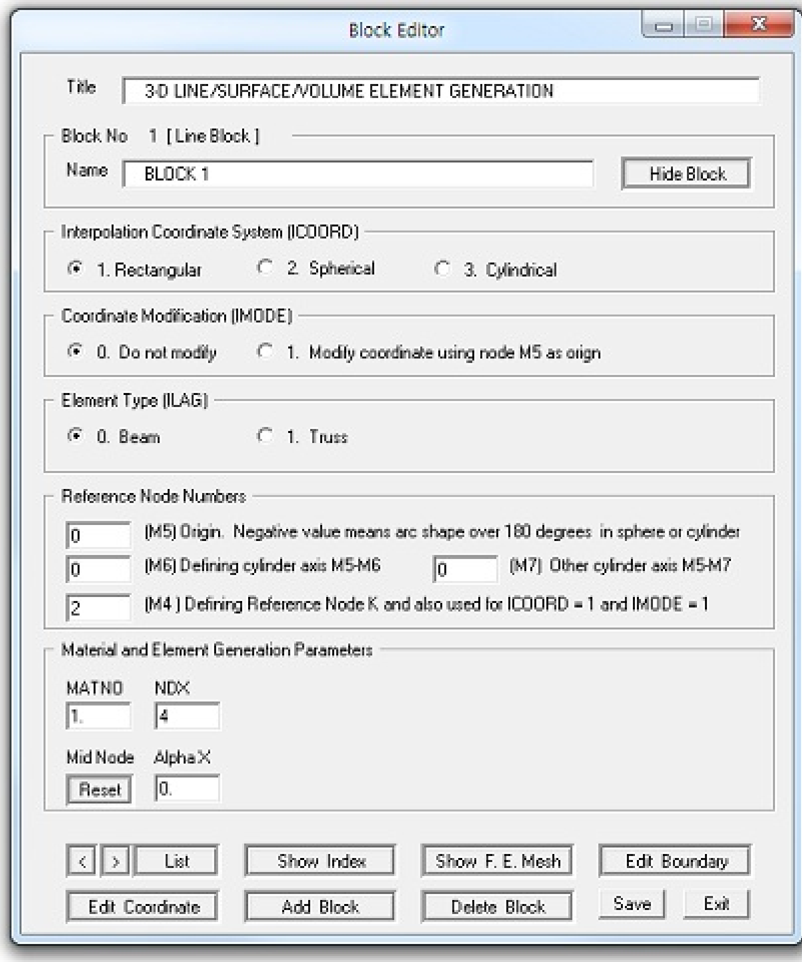


Figure 6.29 Block editor (Line Block)

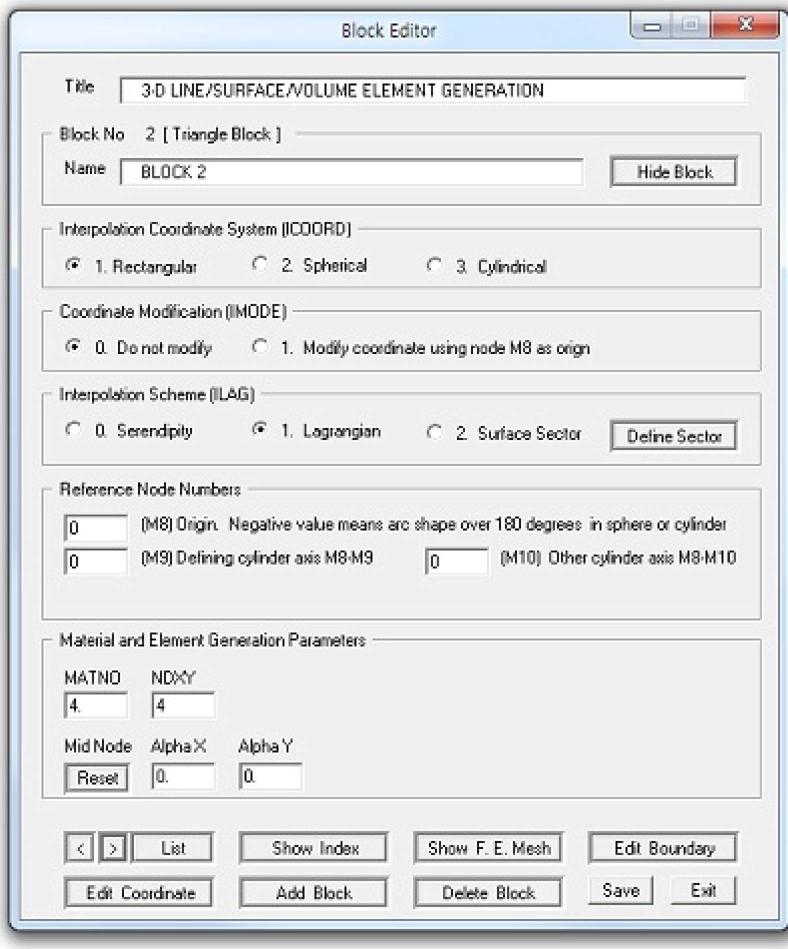


Figure 6.29 Block editor (Triangle Block)

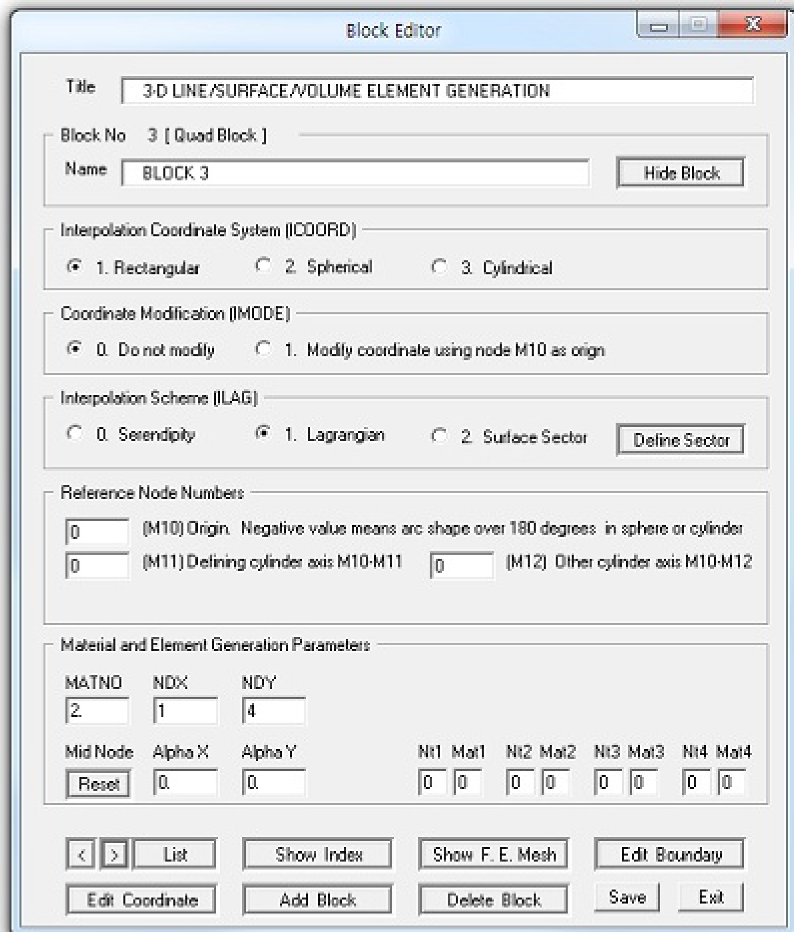


Figure 6.29 Block editor (Quad Block)

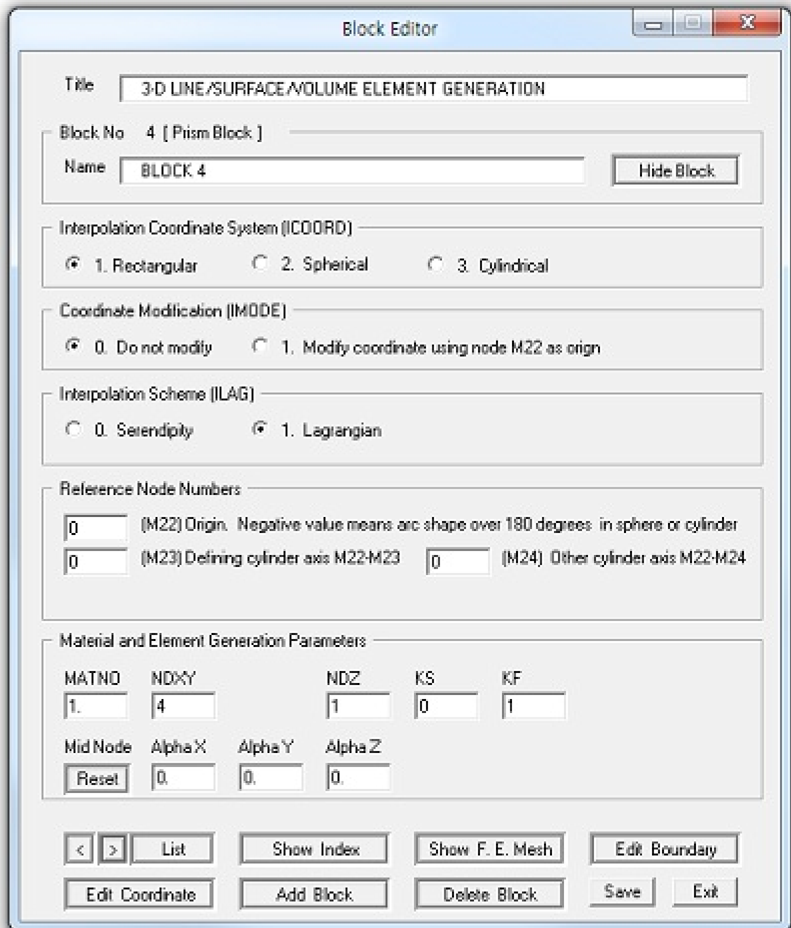


Figure 6.29 Block editor (Prism Block)

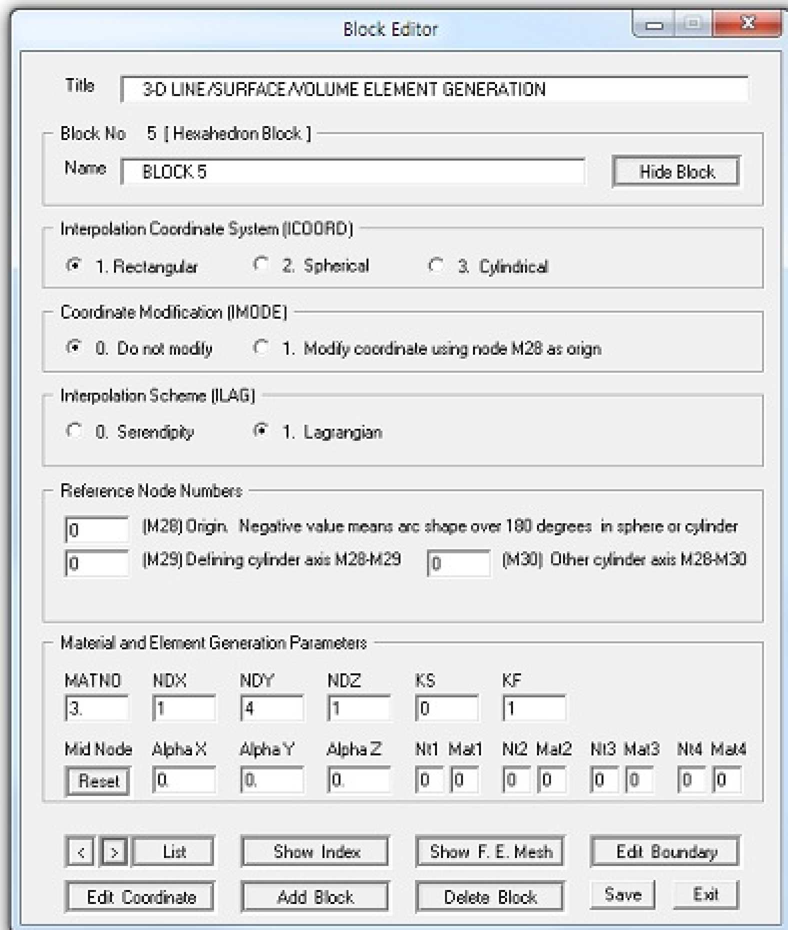


Figure 6.29 Block editor (Hexahedron Block)

6.5.1 Title

This is the title for the block mesh file.

6.5.2 Block Number

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

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

6.5.3 Interpolation Coordinate System

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

6.5.4 Coordinate Modification

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

6.5.5 Interpolation Scheme / Element Type

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

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

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

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

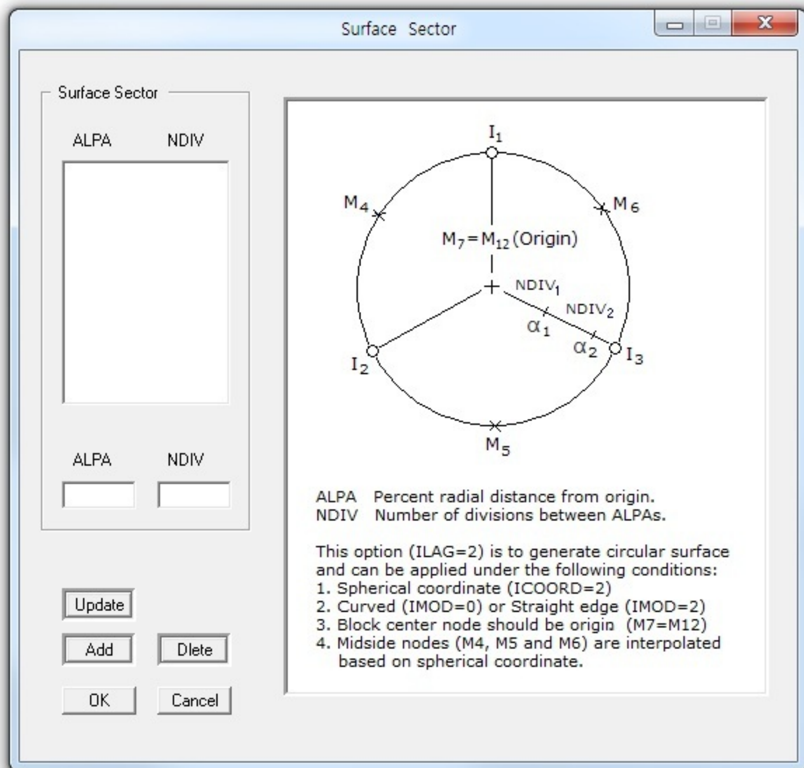


Figure 6.30 Surface sector (Triangle Block)

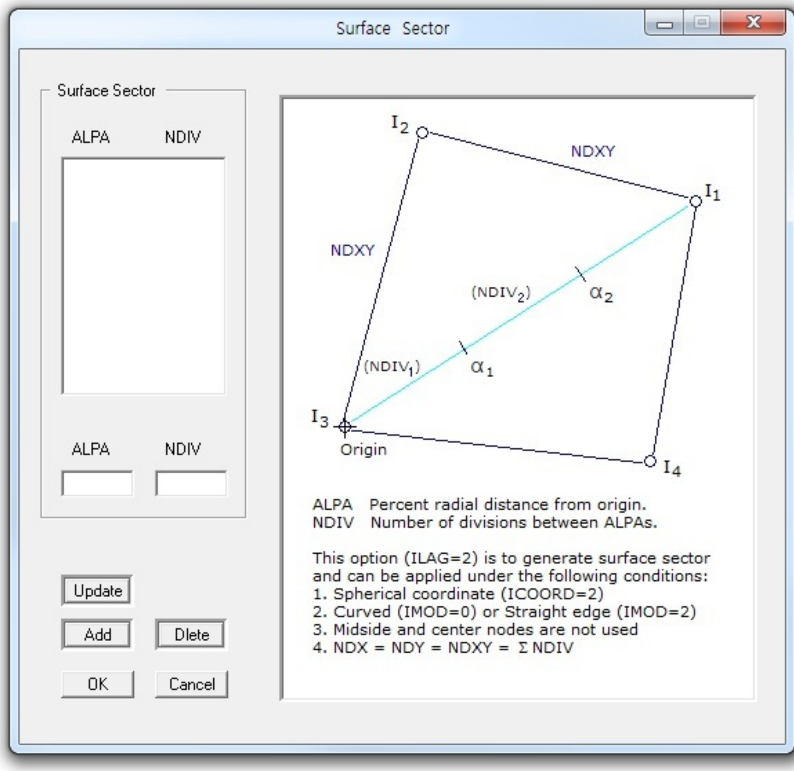


Figure 6.30 Surface sector (Quad Block)

6.5.6 Reference Node Numbers

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

6.5.7 Material & Element Generation Parameters

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

6.5.8 Command Buttons

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

List

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

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

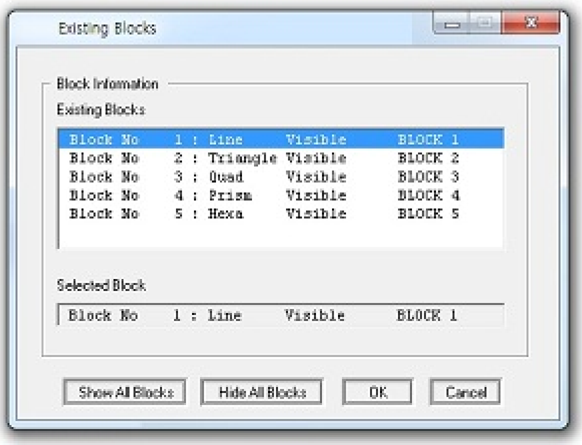


Figure 6.31 Block list

Show Index

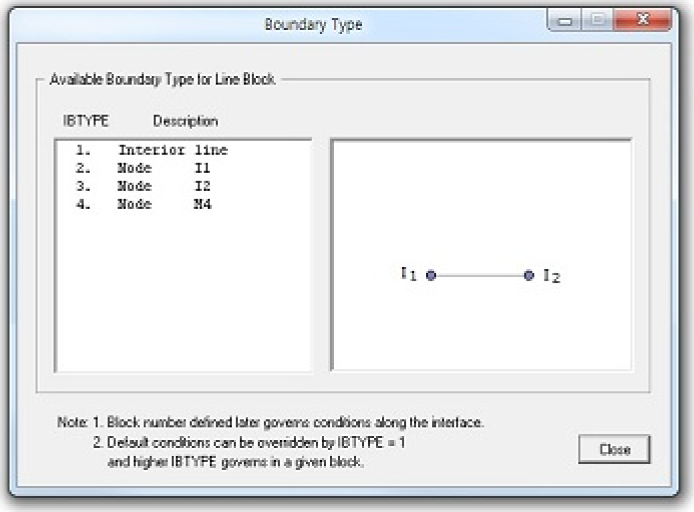
This is used to show block index numbers.

Show F. E. Mesh

This is used to execute block mesh and then plot the generated finite element mesh.

Edit Boundary

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



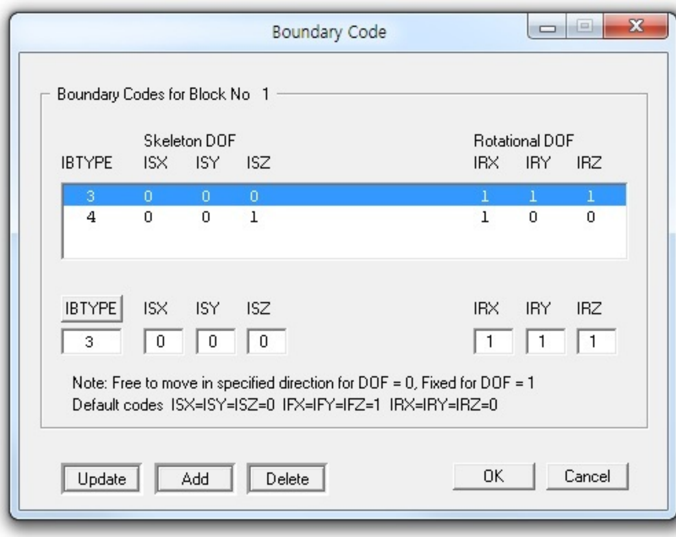


Figure 6.32 Boundary code (Line Block)

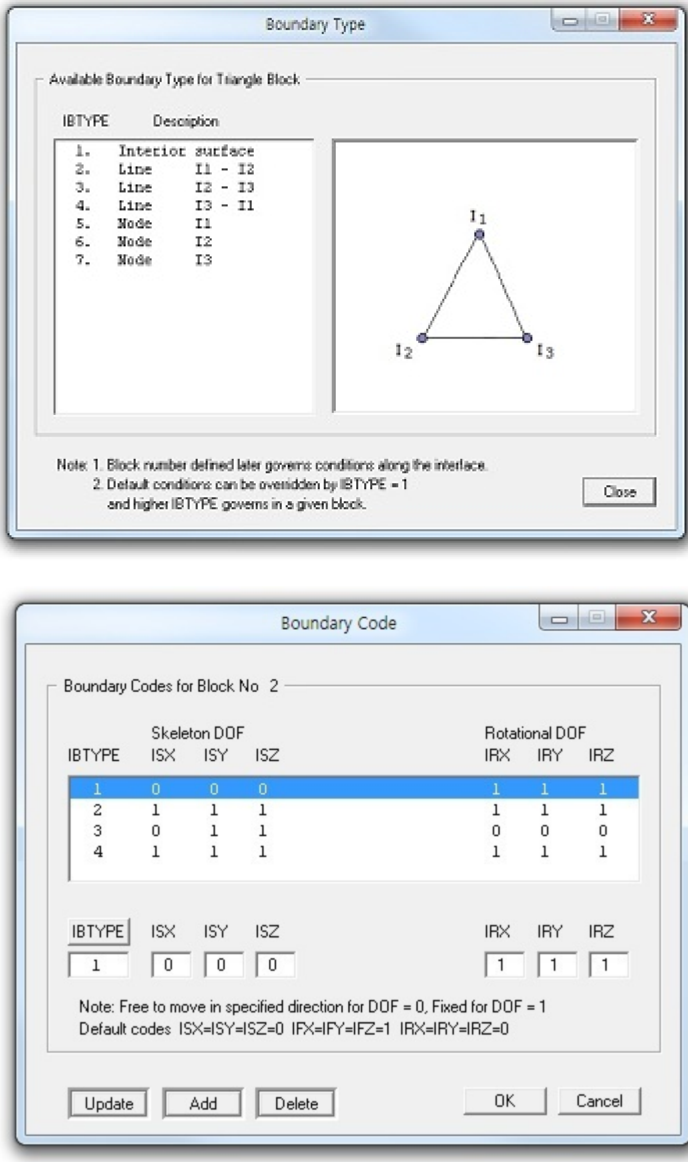


Figure 6.32 Boundary code (Triangle Block)

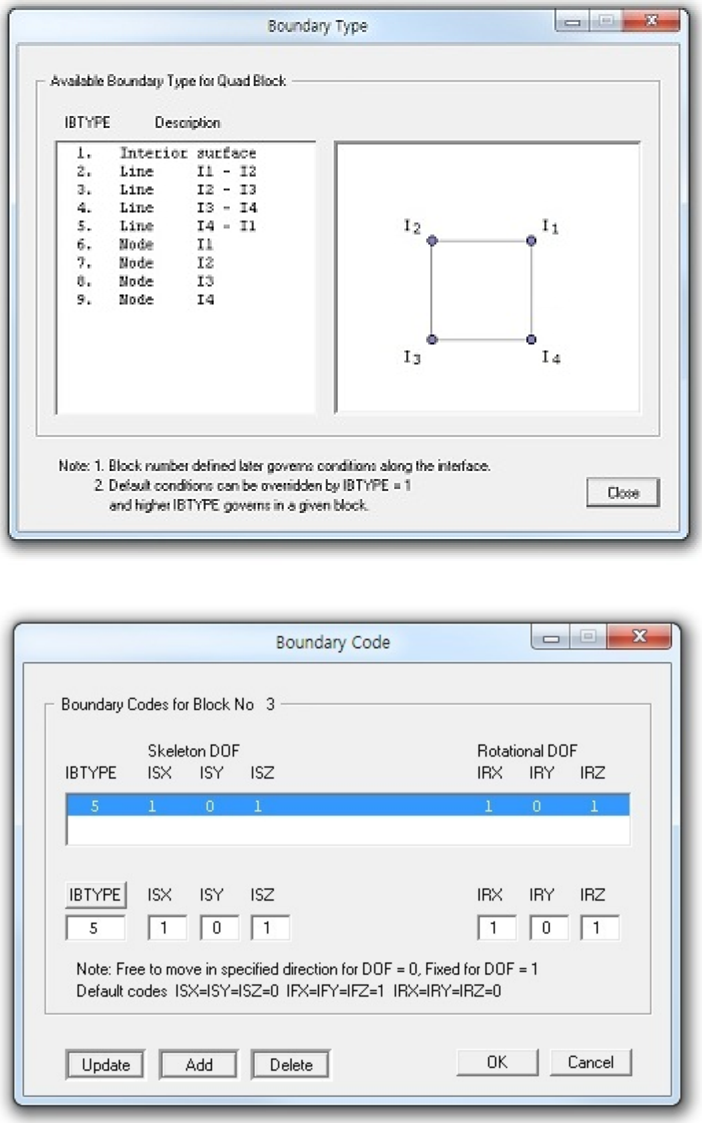


Figure 6.32 Boundary code (Quad Block)

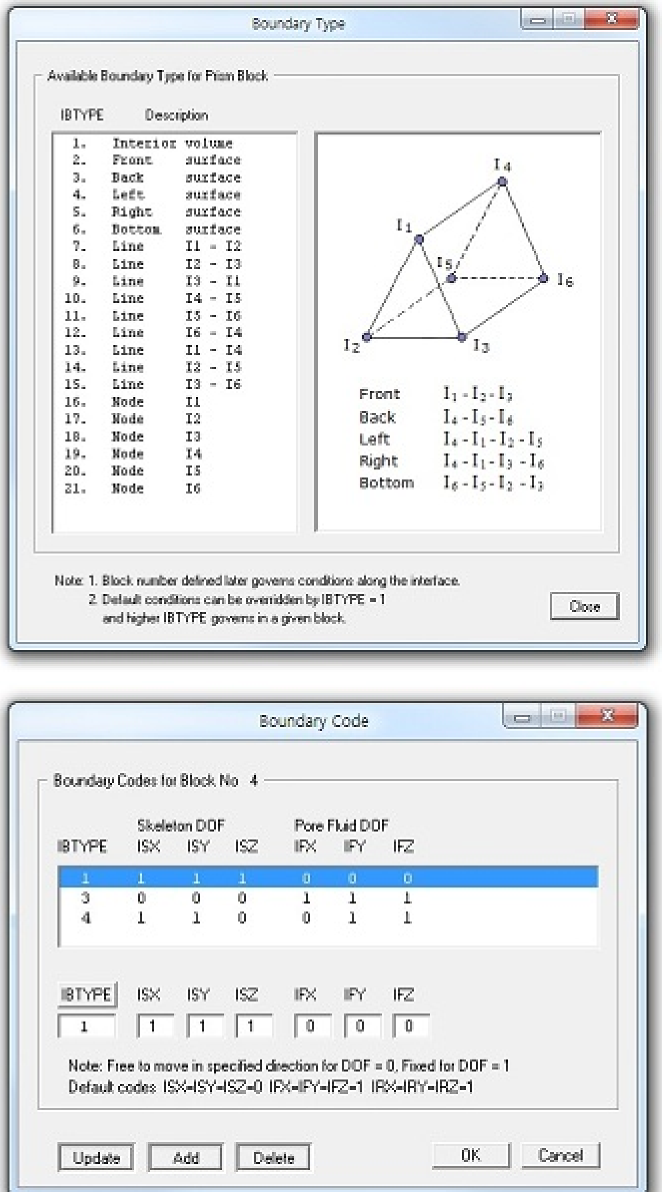


Figure 6.32 Boundary code (Prism Block)

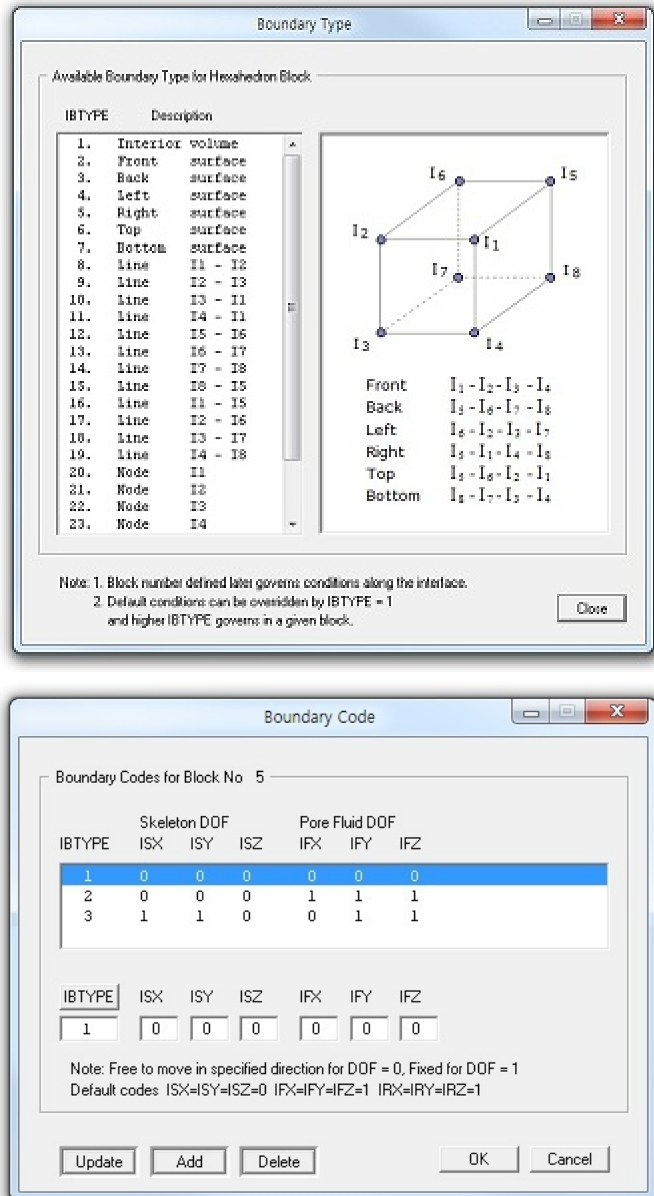


Figure 6.32 Boundary code (Hexahedron Block)

Edit Coordinate

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

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

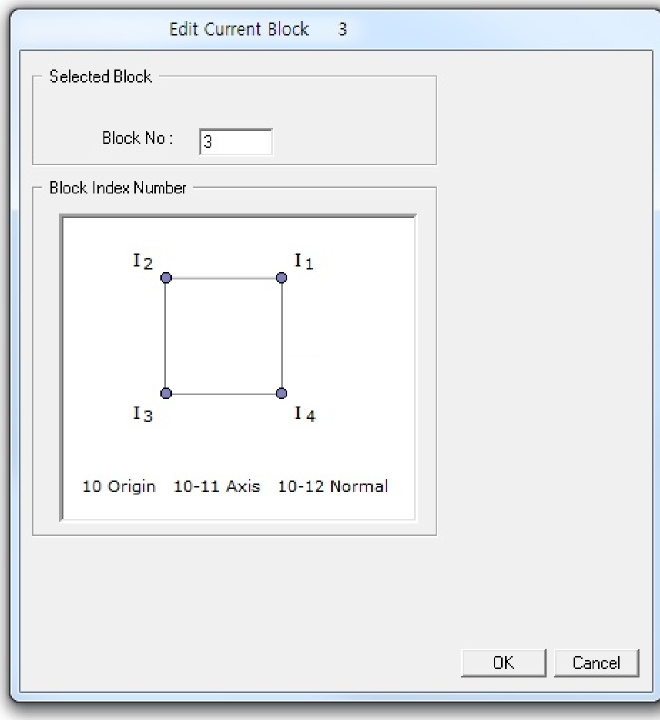


Figure 6.33 Edit current block (Selection Mode)

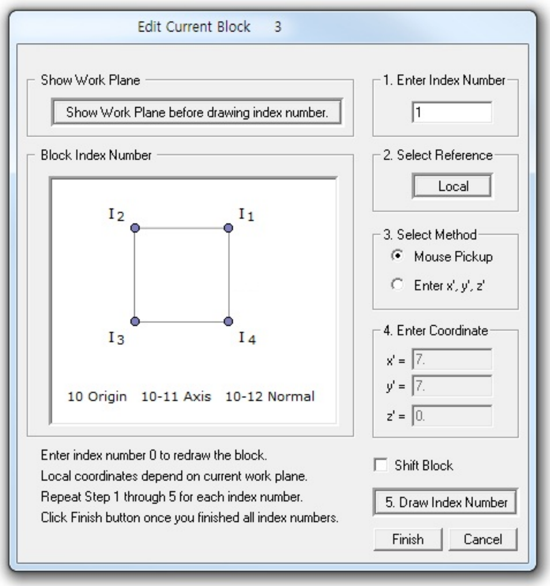
Edit Current Block dialog now shows input parameters required to edit the geometry of the block as shown in Figure 6.34.

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

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

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

Figure 6.34
Edit current block
(Edit Mode)



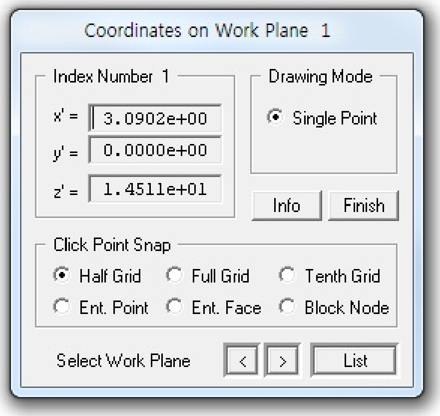


Figure 6.35 Coordinates on work plane

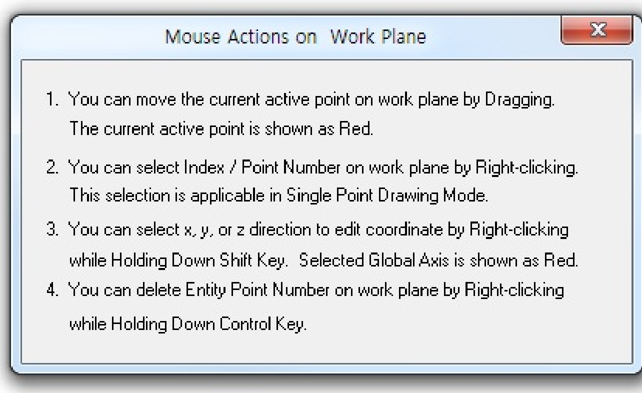


Figure 6.36 Mouse actions on work plane

Add Block

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

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

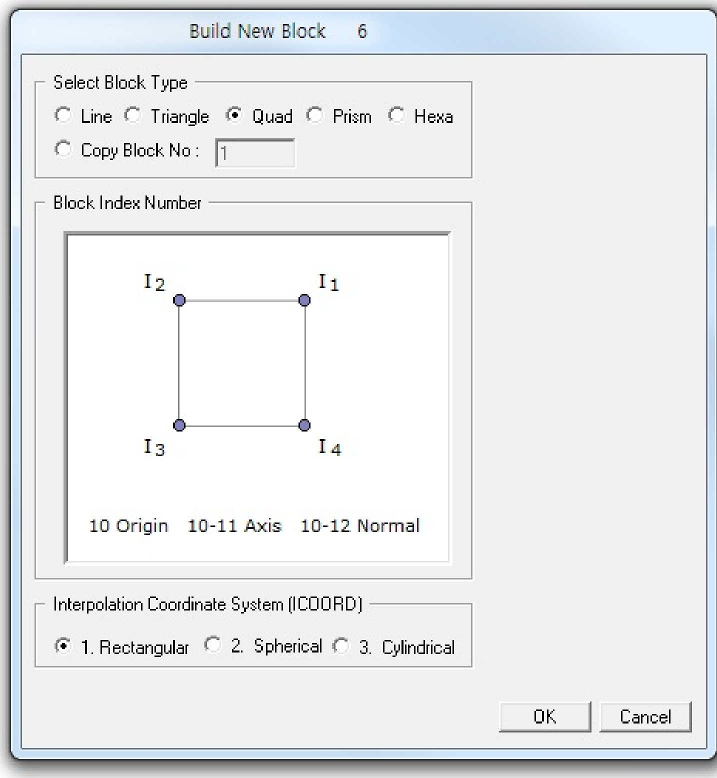


Figure 6.37 Build new block (Selection Mode)

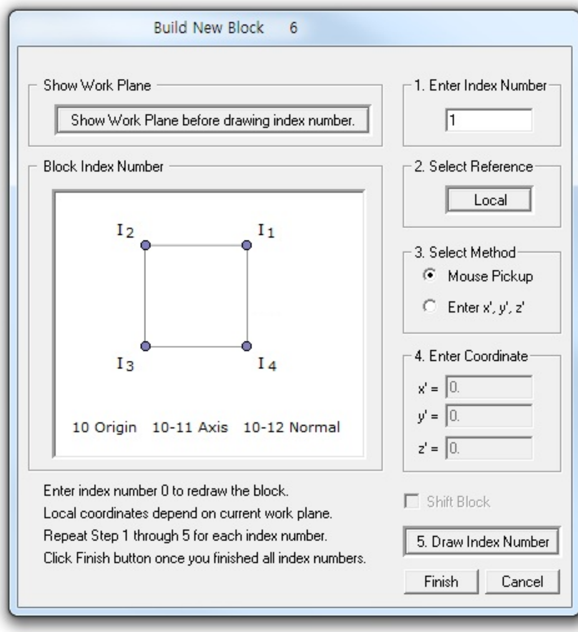
Build New Block dialog now shows input parameters required to build the geometry of new block as shown in Figure 6.38.

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

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

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

Figure 6.38
Build new block
(Build Mode)



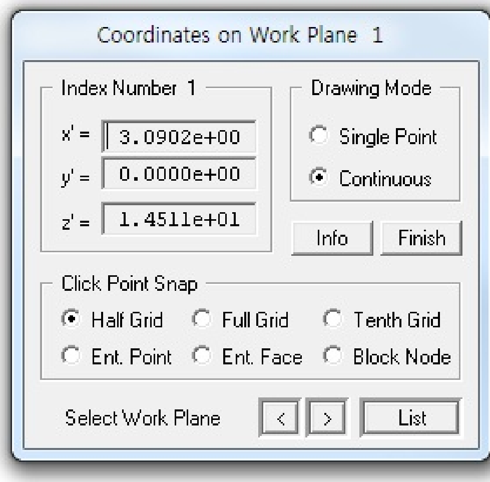


Figure 6.39 Coordinates on work plane

Delete Block

This is used to delete the current block.

Save

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

Exit

This is used to exit from the block editor.

6.5.9 Popup Menu for Block

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

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

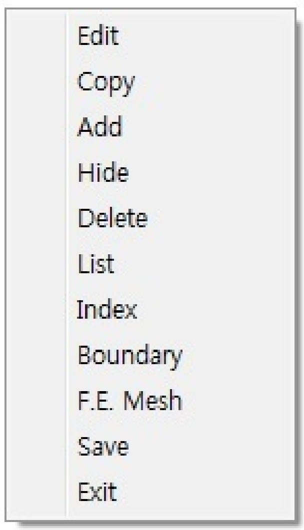


Figure 6.40 Popup menu for block

6.6 Modifying Finite Element Meshes

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

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

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

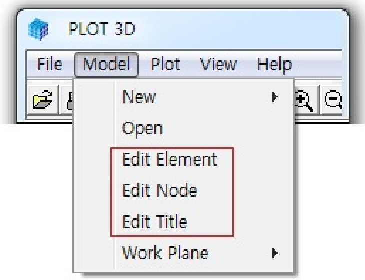


Figure 6.41 Menu for editing finite element mesh

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

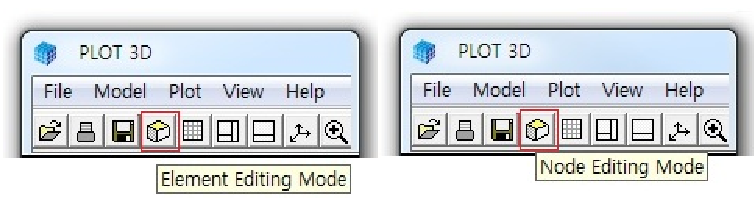
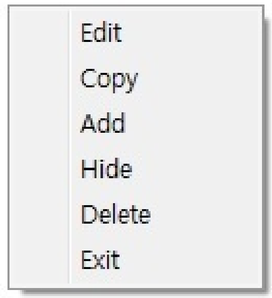


Figure 6.42 Toolbar for editing finite element mesh

6.6.1 Edit Element

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

Figure 6.43 Popup menu for element

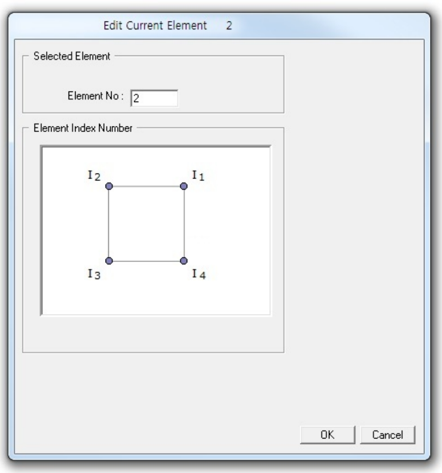


Element popup menu consists of six submenus:
Edit, Copy, Add, Hide, Delete and Exit.

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

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

Figure 6.44
Edit current element
(Selection Mode)



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

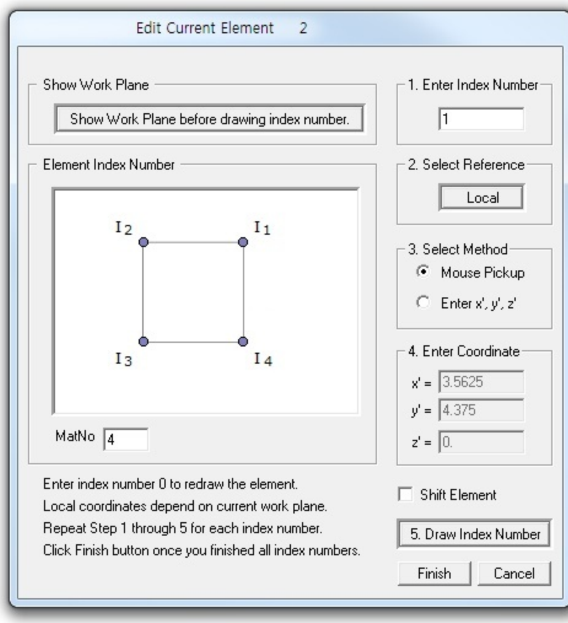
To edit element, follow five steps:

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

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

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

Figure 6.45
Edit current element
(Edit Mode)



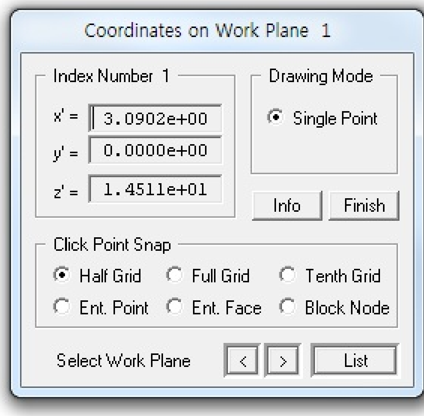


Figure 6.46 Coordinates on work plane (Edit Mode)

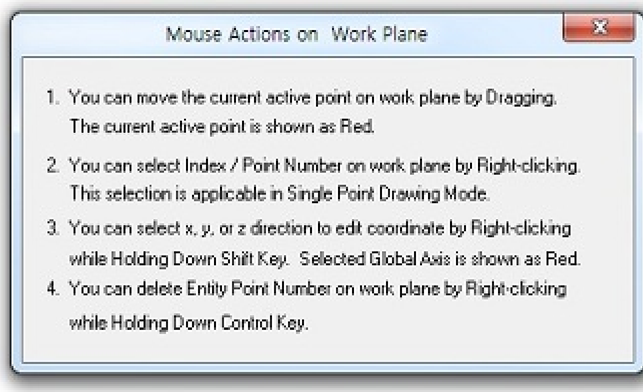


Figure 6.47 Mouse actions on work plane

Copy

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

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

Follow the same procedure as in Edit.

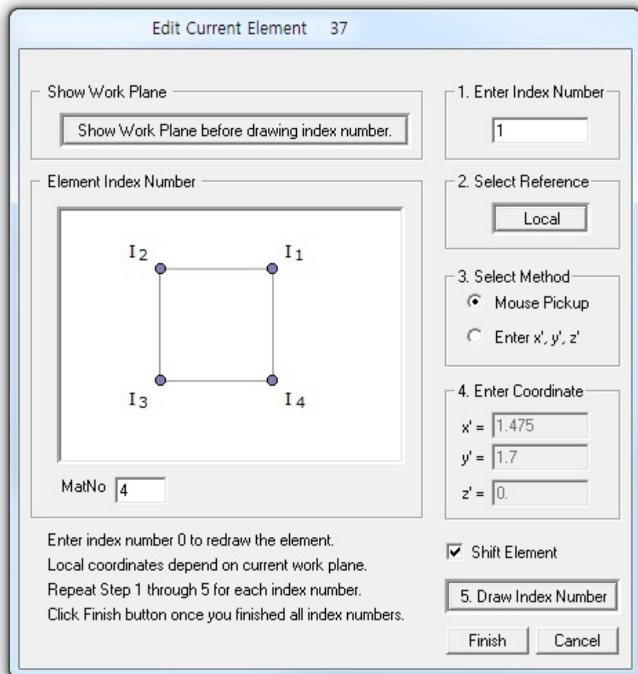


Figure 6.48 Edit current element (Copy Mode)

Add

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

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

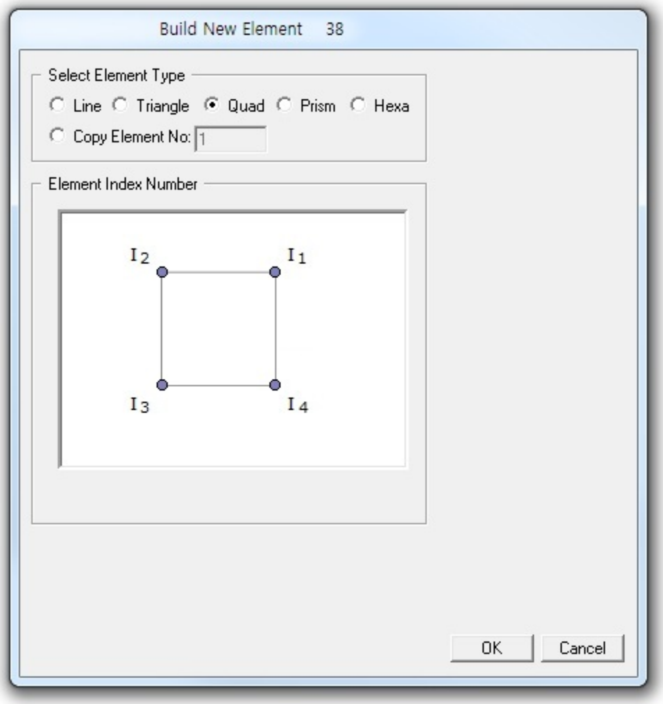


Figure 6.49 Build new element (Selection Mode)

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

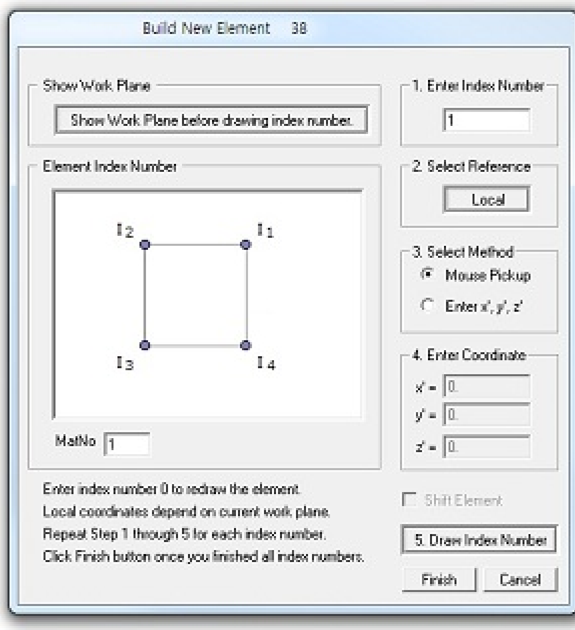
To build new element, follow five steps:

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

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

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

Figure 6.50
Build new element
(Edit Mode)



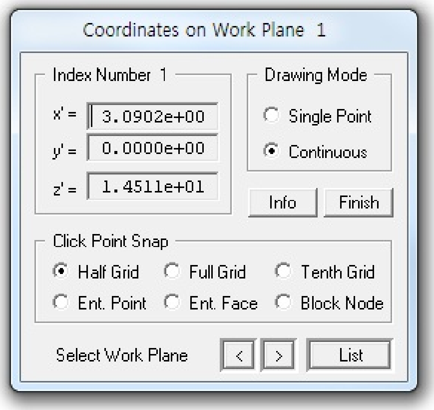


Figure 6.51 Coordinates on work plane (Add Mode)

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

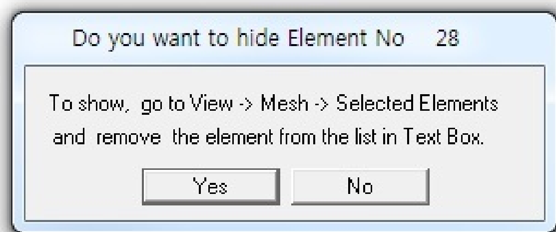


Figure 6.52 Instructions to show the hidden element

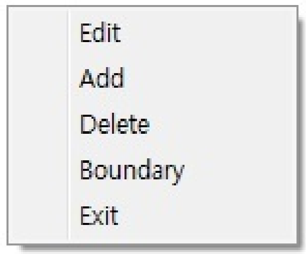
Delete
This is used to delete the selected element.

Exit
This is used to exit from the element editing mode.

6.6.2 Edit Node

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

Figure 6.53 Popup menu for node



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

Edit

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

Edit Current Node dialog is displayed in Figure 6.54.

To edit current node, follow five steps:

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

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

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

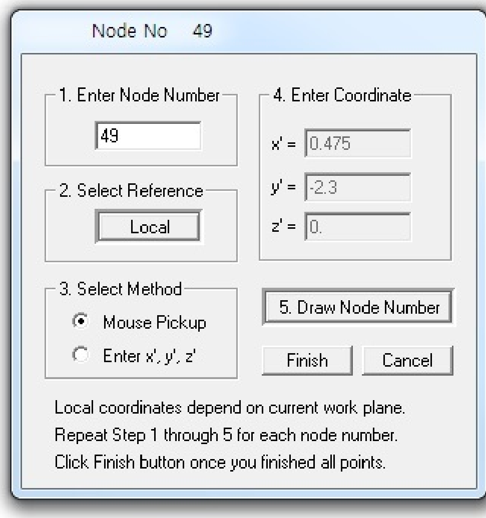


Figure 6.54 Edit current node dialog

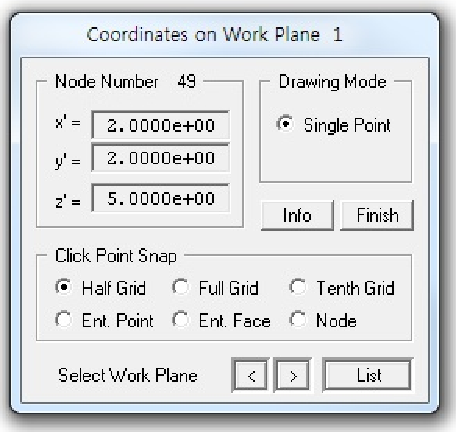


Figure 6.55 Coordinates on work plane (Edit Node)

Add

This is used to add new node.
Before adding, work plane should be displayed on the screen.

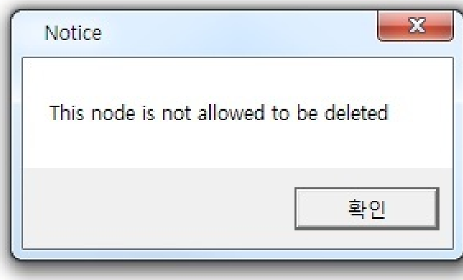
Build New Node dialog similar to Figure 6.54 is displayed.
Follow the same procedure as in Edit.

Delete

This is used to delete the selected node.

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

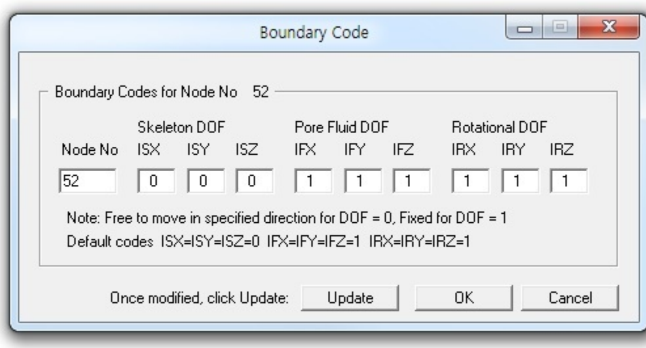
Figure 6.56
Notice on deleting connected nodes



Boundary

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

Figure 6.57
Boundary editor dialog



Exit

This is used to exit from the node editing mode.

6.6.3 Edit Title

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

Figure 6.58 Mesh title editor dialog

PRESMAP
User's Manual

7.1 Introduction

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

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

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

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

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

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

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

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

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

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

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

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

PRESMAP-2D
Model 1
User's Manual

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

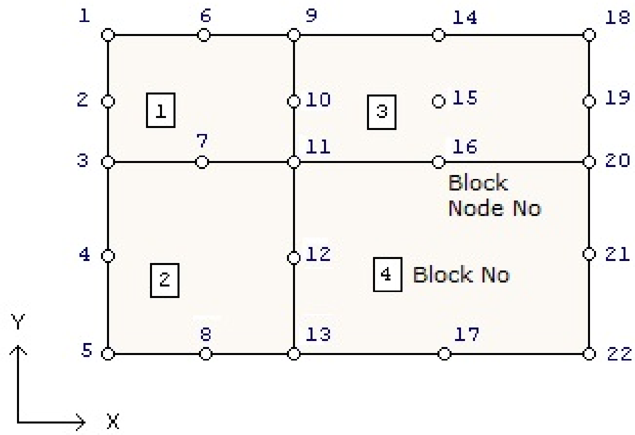
Card Group	Input Data and Definitions (Model 1)
1 General Information	1.4 NBX, NBY, MIDX, MIDY, NF, NSNODE See Figure 7.2 NBX — Number of blocks in x-direction NBY — Number of blocks in y-direction MIDX = 0 — Element has no side nodes in x-direction = 1 — Element has side nodes in x-direction MIDY =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

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

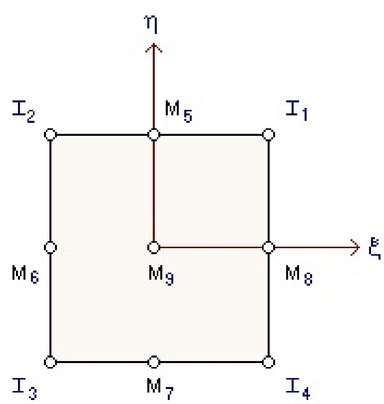
Card Group	Input Data and Definitions (Model 1)
3 Data for Each Block	3.1 BLNAME BLNAME Block name (up to 60 characters)
	3.2 IBLNO IBLNO Block number
	3.3 I_1, I_2, I_3, I_4, M_5, M_6, M_7, M_8, M_9 See Figure 7.1 I_1, I_2, I_3, I_4 Corner node number M_5, M_6, M_7, M_8 Side node number M_9 Center node number
	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

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

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



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

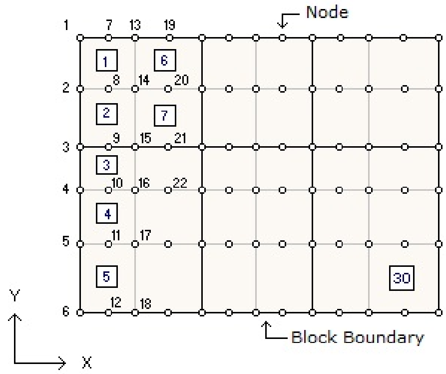


For Block Number 3

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

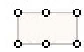
PRESMAP uses Serendipity interpolation if $M_9 = 0$
and Lagrangion interpolation if $M_9 \neq 0$

Figure 7.1 Block Specification and Block Index



NBX = 3
NBY = 2

MIDX = 1
MIDY = 0

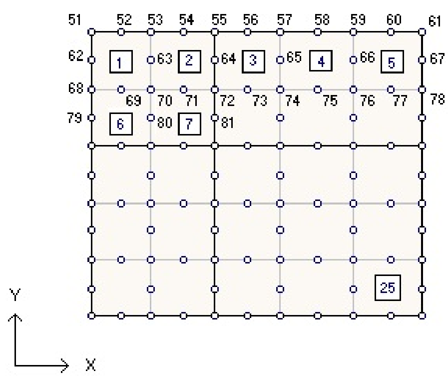


6 Node Element

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

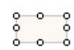
NSNODE = 1

□ 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

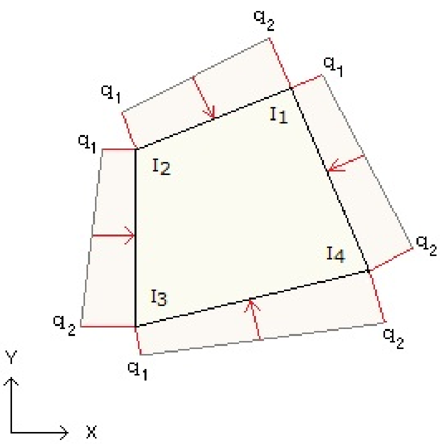
Boundary Type	Boundary Codes		
IB	IDX	IDY	IDT
0	0	0	1
1	1	0	1
2	0	1	1
3	1	1	1
4	0	0	0
5	1	0	0
6	0	1	0
7	1	1	0

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

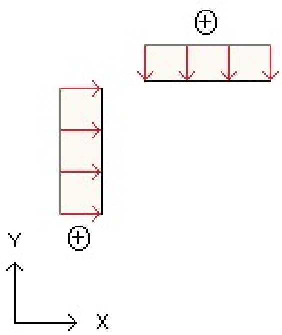
IDT = 0 Rotational degree of freedom is free
= 1 Rotational degree of freedom is fixed

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

<table>
<tr><th>Card Group</th><th>Input Data and Definitions (Model 2)</th></tr>
<tr><td rowspan="4">1

General Information</td><td>1.1

TITLE
TITLE — Any title (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

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

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

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

Card Group				Input Data and Definitions (Model 2)
2 Data for Each Subregion	2.5 Subregion Outer Edge	For ISBTYPE=0	Point C	2.5.1.1 LCTYPE LCTYPE = 0 X_C and Y_C are specified = 1 X_C is specified = 2 Y_C is specified = 3 DRT_C is specified
			Point C	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
			Point D	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

Card Group	Input Data and Definitions (Model 2)	
2 Data for Each Subregion	2.8 NFSIDE NFSIDE — Number of edge where boundary forces are specified	
	2.9 Force Data for Each Specified Edge (see Figure 7.8)	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
		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

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

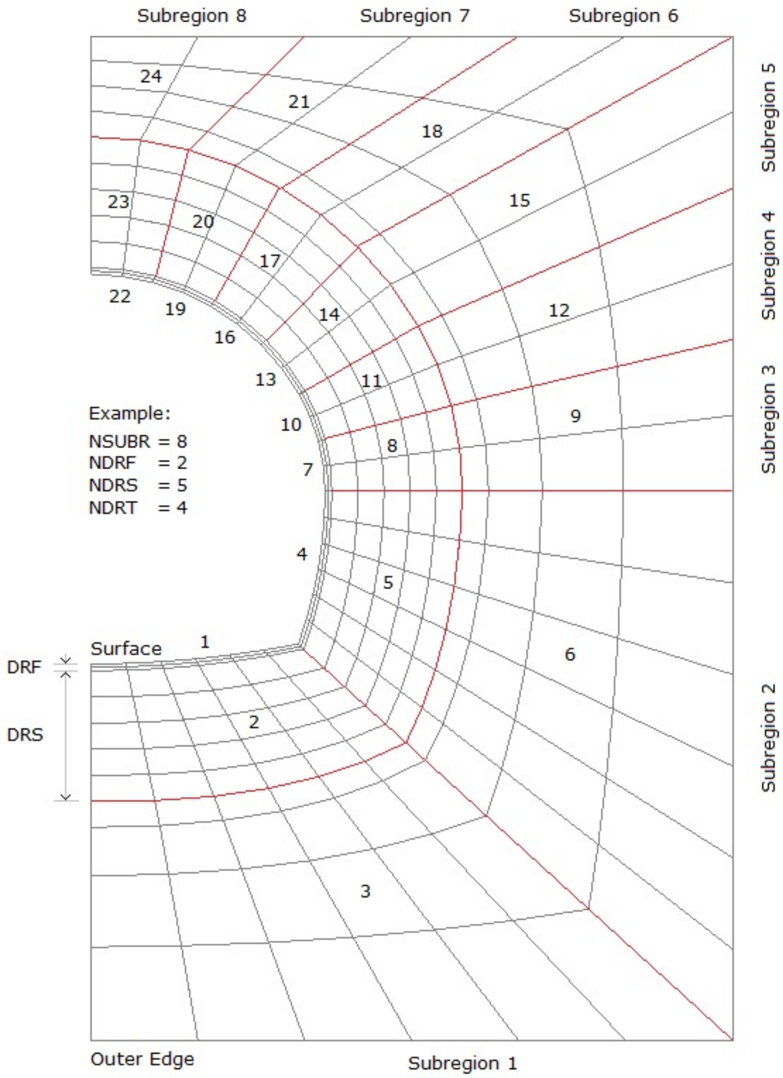


Figure 7.5 Subregion and block numbering sequence

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

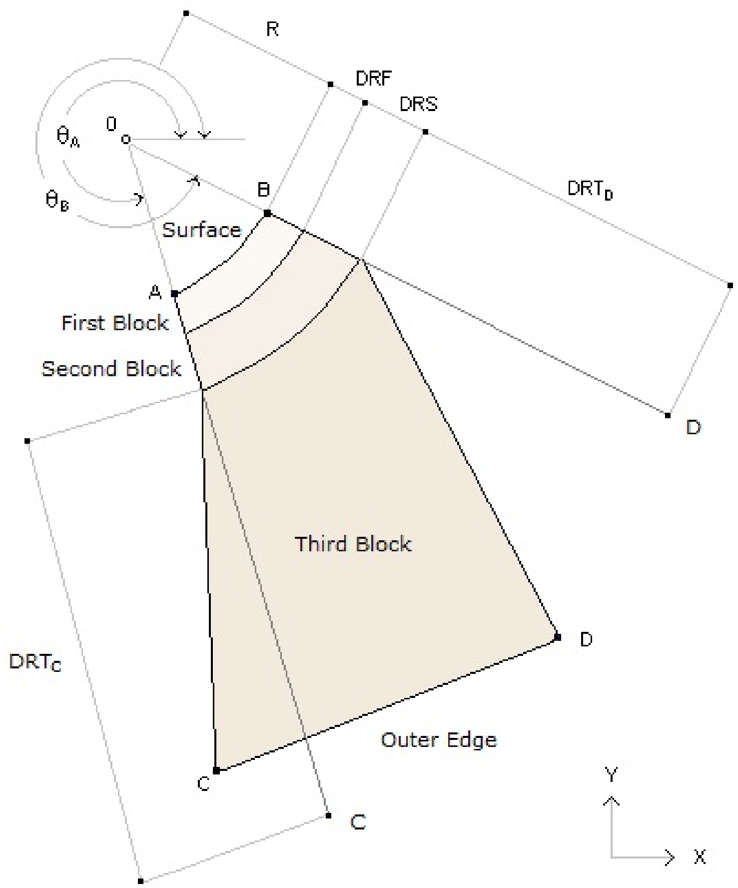


Figure 7.6 Typical Subregion for ISBTYPE = 0

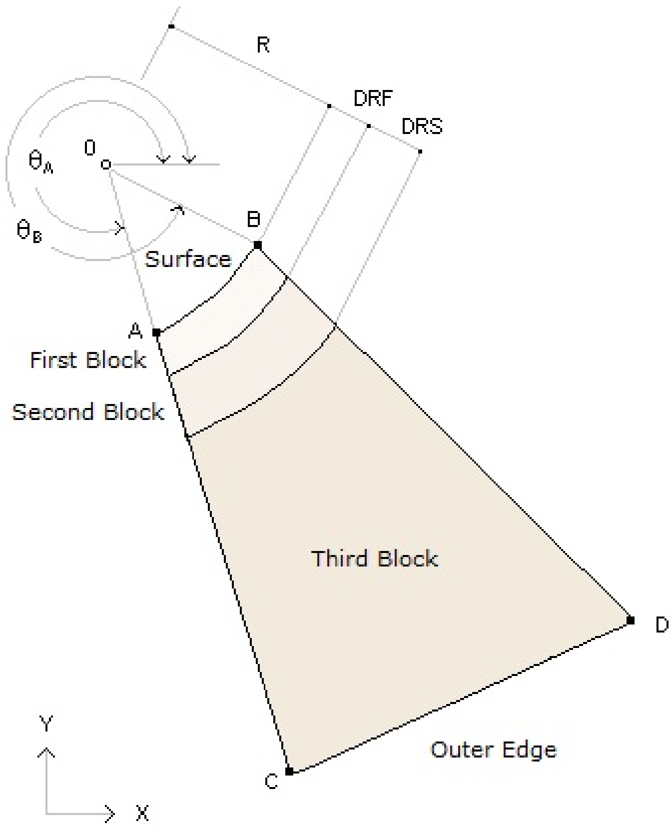
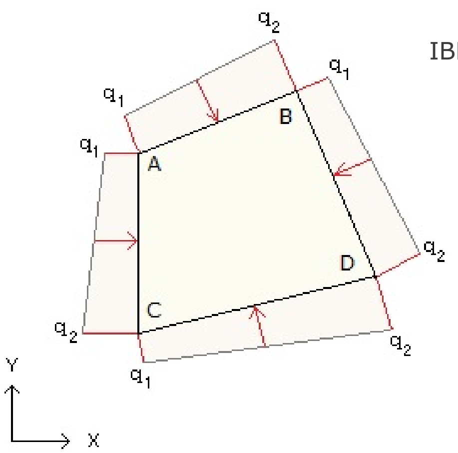
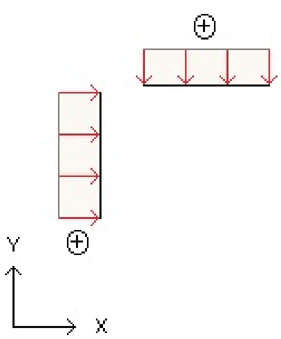


Figure 7.7 Typical Subregion for ISBTYPE = 1



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

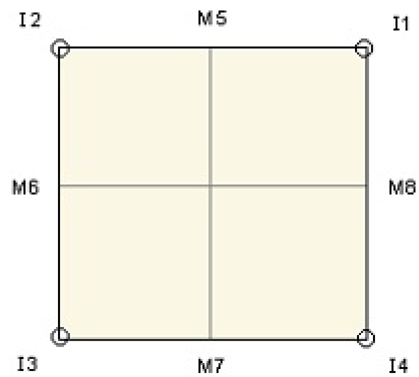
PRESMAP-2D

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

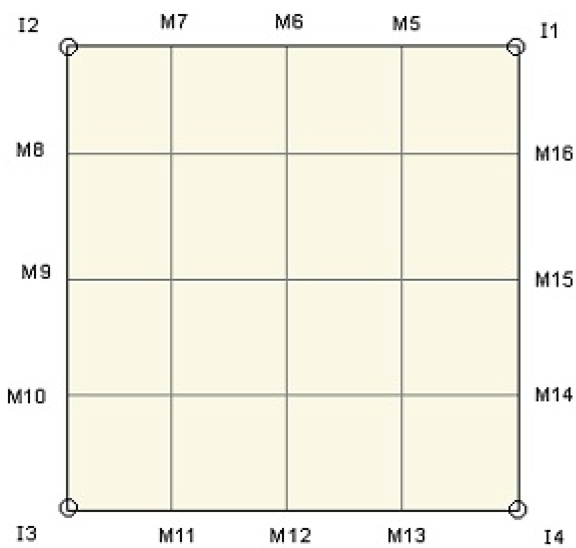
Card Group	Input Data and Definitions (Model 3)
3 Data for Each Block	3.2 For IBLTYPE = 1 I_1, I_2, I_3, I_4, M_5, M_6, M_7, M_8 I_1, I_2, I_3, I_4 Corner node number M_5, M_6, M_7, M_8 Side node number  Note: IBLTYPE = 1 generates 4 elements

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

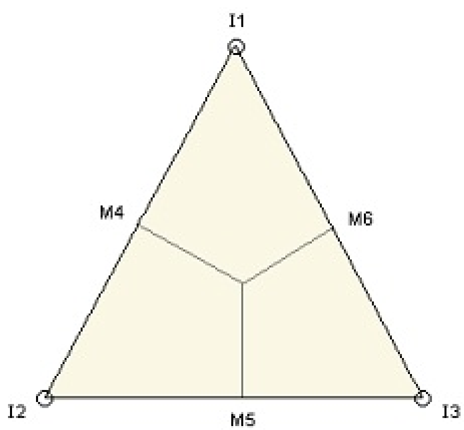
For IBLTYPE = 2

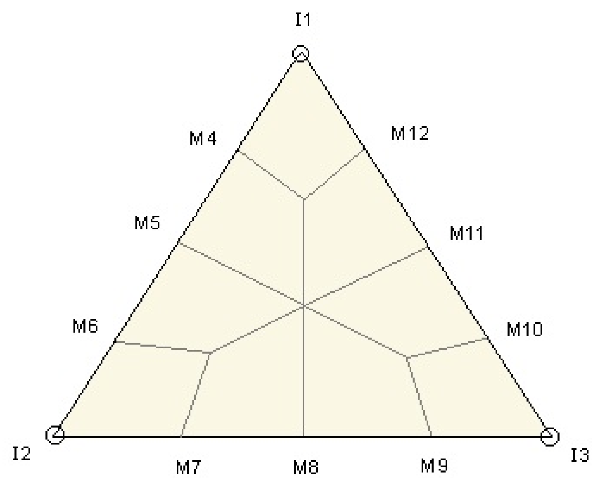
I_1, I_2, I_3, I_4,
M_5, M_6, M_7,
M_8, M_9, M_{10},
M_{11}, M_{12}, M_{13},
M_{14}, M_{15}, M_{16}

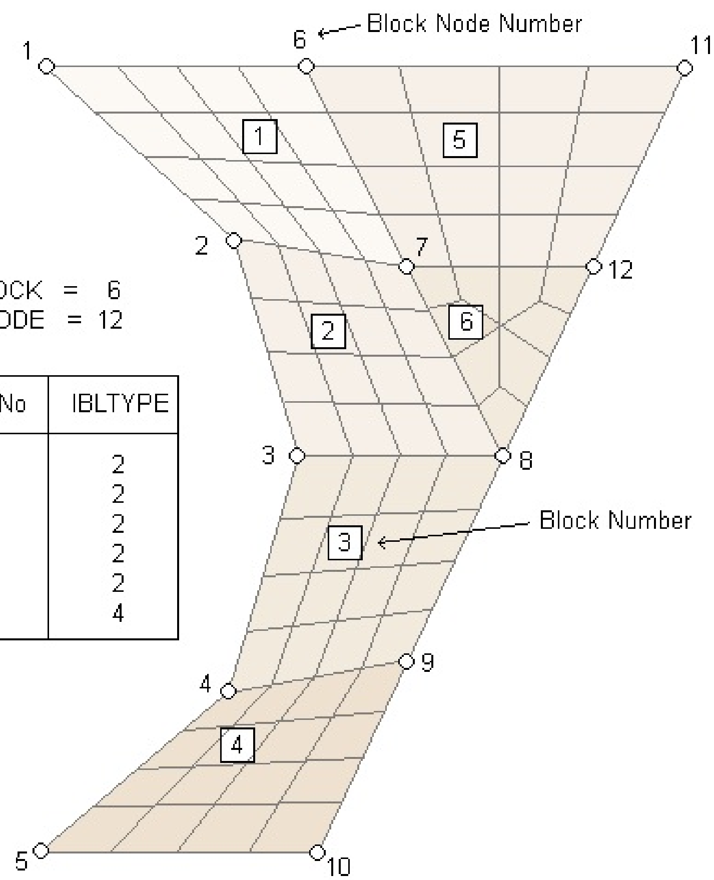
I_1, I_2, I_3, I_4	Corner node number
M_5, M_6,, M_{16}	Side node number



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



NBLOCK = 6
NBNODE = 12

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

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

PRESMAP-2D

Model 4

User's Manual

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

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

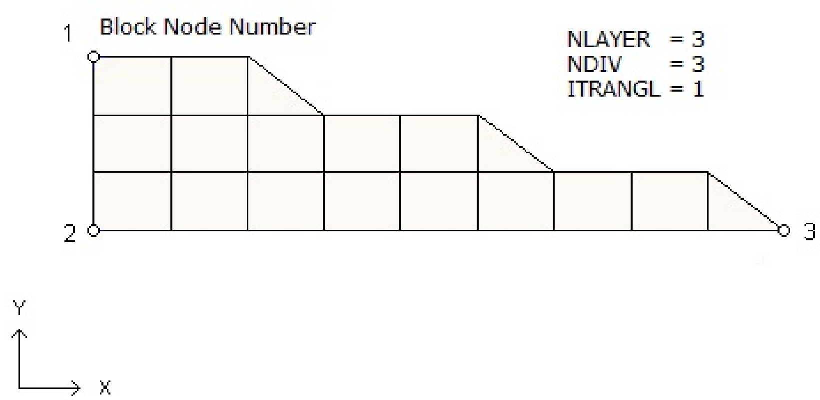


Figure 7.10 Block node number for Model 4 of PRESMAP-2D

NATM-2D

User's Manual

Card Group	Input Data and Definitions
1	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
General Information	1.3 MODEL, IGEN, IEXMESH, ILNCOUPL, IAUTO MODEL = 1 Single tunnel (Half section) = 2 Single tunnel (Full section) = 3 Two tunnels (Symmetric) = 4 Two tunnels (Unsymmetric) IGEN = 0 Generate whole mesh = 1 Generate core = 2 Generate surrounding IEXMESH = 0 No user supplied mesh = 1 Add generated mesh to user supplied mesh For Lining analysis ILNCOUPL= 0 Surrounding rock by continuum element = 1 Surrounding rock by spring element IAUTO = 0 Generate Mesh file = 1 Generate Mesh, Main and Post files Available only for SMAP-S2 See Figure 7.11

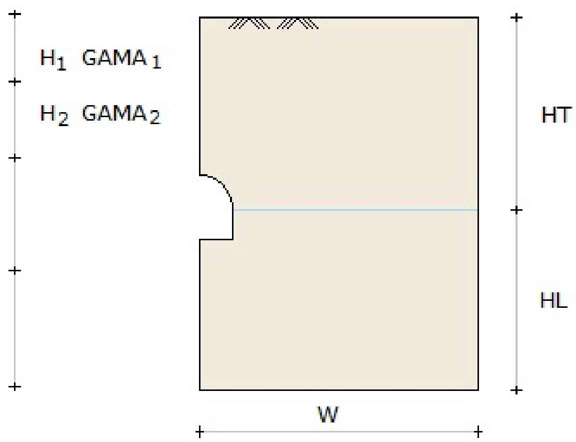
Card Group	Input Data and Definitions
2 Tunnel Analysis Boundary	2.1 MODEL = 1: HT, HL, W, DX, DY, NY = 2: HT, HL, W, DX, DY, NY = 3: HT, HL, W, WP, DX, DY, NY = 4: HT, HL, W, WP, HP, DX, DY, NY HT Tunnel depth HL Depth from springline to bottom boundary W Horizontal distance from left to right boundary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

Card Group	Input Data and Definitions
3 Soil / Rock Layer Information	3.1 NLAYER NLAYER Total number of layers. Max = 10
	3.2 NLAYER Cards: LAYERNO$_1$, H$_1$, DD$_1$ LAYERNO$_2$, H$_2$, DD$_2$ - - - - - - LAYERNO Soil/rock layer number H Thickness of soil/rock layer DD = GAMA SMAP-S2 = IDH SMAP-T2 = KF SMAP-2D GAMA Unit weight IDH Heat generation history ID number KF = 0 Has fluid phase = 1 No fluid phase See Figure 7.11

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)

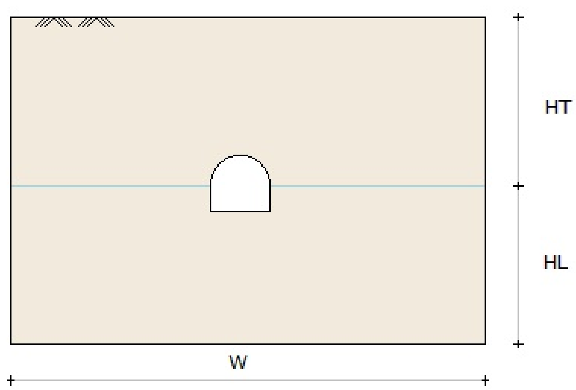
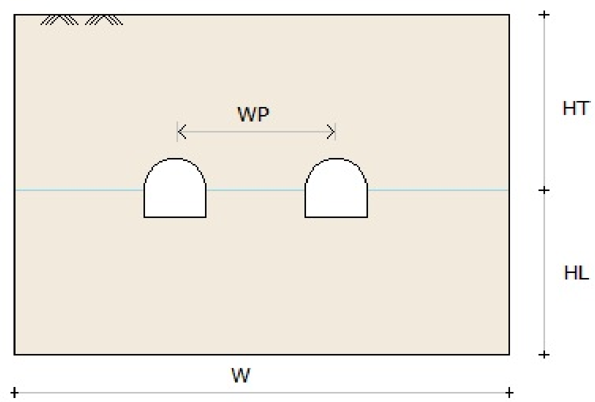


Figure 7.11 Schematic tunnel section view for MODEL = 1 and 2

MODEL = 3   Two Tunnel (Symmetric Section)



MODEL = 4   Two Tunnel (Unsymmetric Section)

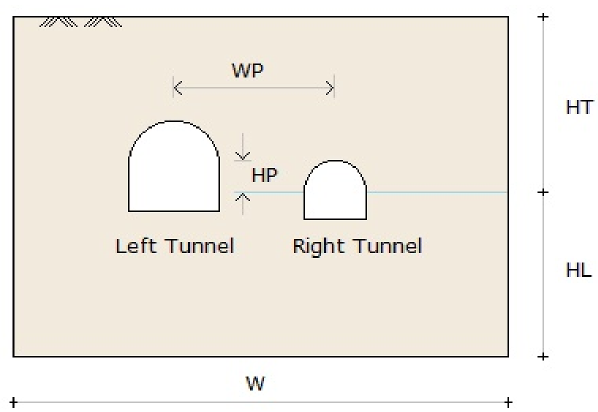
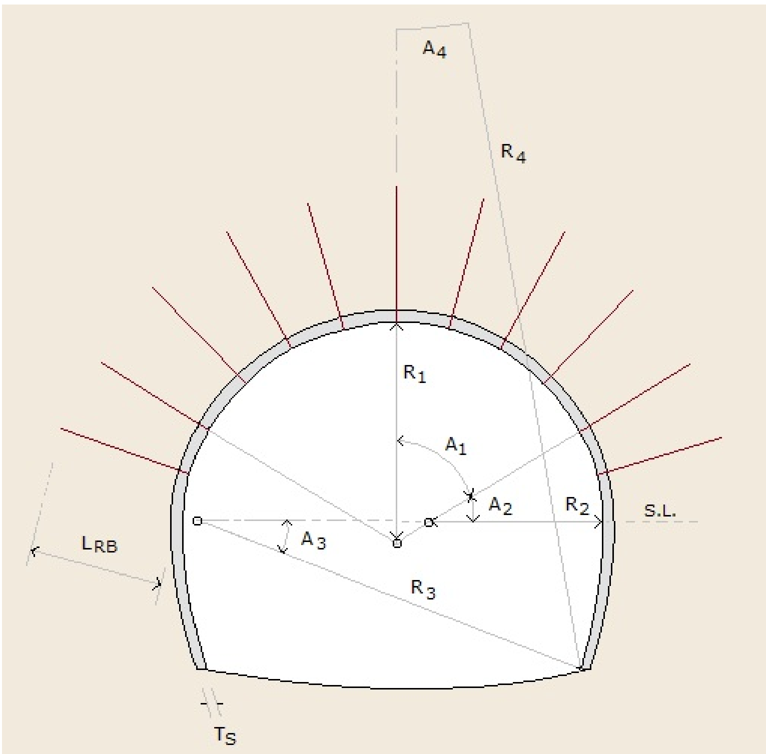


Figure 7.11   Schematic tunnel section view for MODEL = 3 and 4



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)

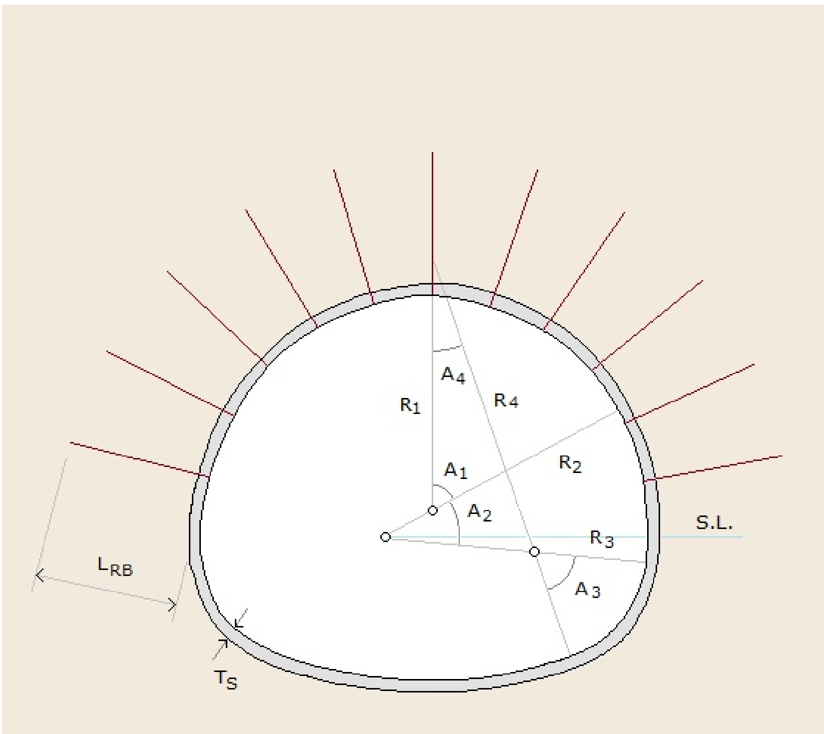


Figure 7.12 Tunnel dimension (A_1+A_2 > 90)

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.

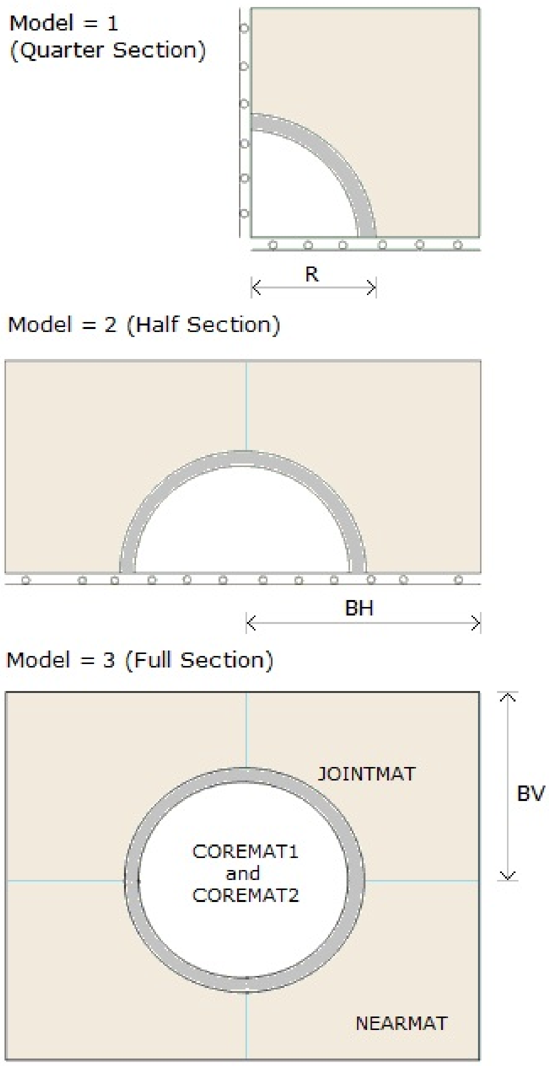


Figure 7.13 Model type for CIRCLE-2D

PRESMAP-3D

User's Manual

Card Group	Input Data and Definitions
1 General Information	1.1 TITLE TITLE — Any title (Max = 80 characters)
	1.3 NBLOCK, NBNODE, NSNODE, NSNEL, CMFAC NBLOCK — Number of blocks NBNODE — Number of block nodes NSNODE — Starting node number NSNEL — Starting element number CMFAC — Coordinate magnification factor Note: If NBLOCK is negative value, the output file contains plotting information for block diagram.
2 Block Coordinates	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	3.1 BLNAME BLNAME Block name (Max = 60 characters)
	3.2 ILAG ILAG = 0 Serendipity interpolation = 1 Lagrangian interpolation
	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} (only for ILAG=1) See Figure 7.13 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.

Card Group		Input Data and Definitions
3 Data for Each Block	3.4 See Figure 7.14	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-3D IBTYPE, ISX, ISY, ISZ, IFX, IFY, IFZ For SMAP-T3 IBTYPE, ID, IDF 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 and I_2 = 9 Line I_2 and I_3 = 10 Line I_3 and I_4 = 11 Line I_4 and I_1 = 12 Line I_5 and I_6 = 13 Line I_6 and I_7 = 14 Line I_7 and I_8 = 15 Line I_8 and I_5 = 16 Line I_1 and I_5 = 17 Line I_2 and I_6 = 18 Line I_3 and I_7 = 19 Line I_4 and I_8

Card Group	Input Data and Definitions	
3 Data for Each Block	See Figure 7.14	3.4.2 = 20 Node I_1 = 21 Node I_2 = 22 Node I_3 = 23 Node I_4 = 24 Node I_5 = 25 Node I_6 = 26 Node I_7 = 27 Node I_8 ISX Skeleton X DOF ISY Skeleton Y DOF ISZ Skeleton Z DOF IFX Pore fluid X DOF relative to skeleton IFY Pore fluid Y DOF relative to skeleton IFZ Pore fluid Z DOF relative to skeleton ISX, ISY, ISZ, IFX, IFY, IFZ = 0 Free to move in specified direction = 1 Fixed in specified direction Note: Default boundary conditions are ISX=ISY=ISZ=0 and IFX=IFY=IFZ=1 For SMAP-T3 ID = 0 Heat flow is specified = 1 Temperature is specified IDF = Time history identification number

Card Group		Input Data and Definitions
3 Data for Each Block		3.5 MATNO, NDX, NDY, NDZ, KS, KF For SMAP-S3/3D MATNO, NDX, NDY, NDZ, IDH For SMAP-T3 MATNO Material property number NDX Number of elements in x-direction NDY Number of elements in y-direction 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

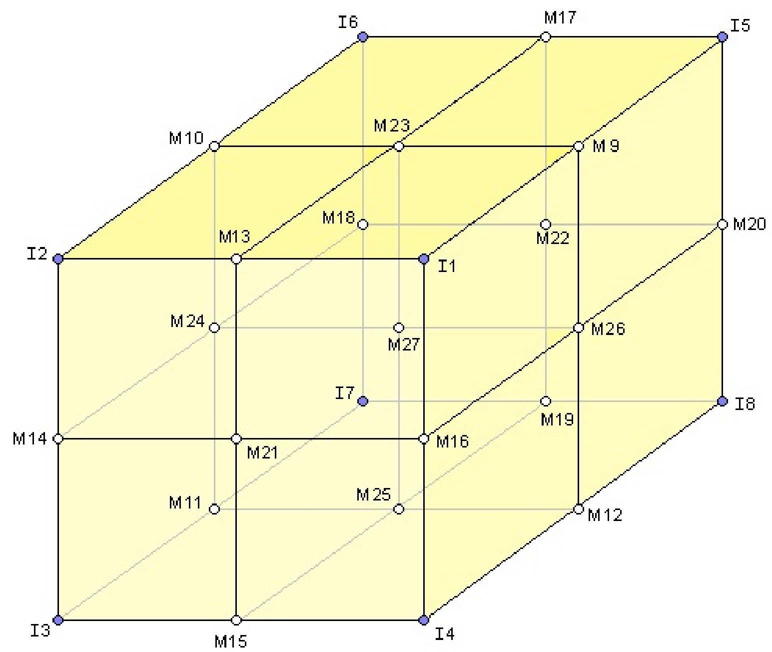


Figure 7.13 Block index for PRESMAP-3D

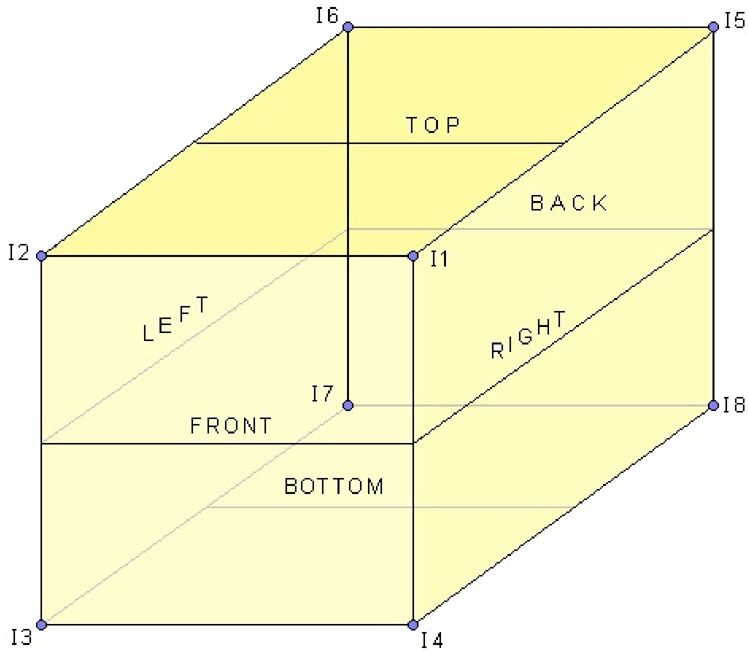


Figure 7.14 Boundary surface designation for PRESMAP-3D

CROSS-3D

User's Manual

Card Group	Input Data and Definitions
1 General Information	1.1 TITLE TITLE Any title (Max = 80 characters)
	1.2 MODELNO, KF, NSNODE, NSNEL, CMFAC (SMAP-3D) MODELNO, IH, NSNODE, NSNEL, CMFAC (SMAP-T3) MODELNO = 1 Identical size tunnels crossing at right angle at the same level. See Figure 7.15 and 7.16 = 2 Large and small tunnels crossing at right angle at the same level. See Figure 7.17 and 7.18 = 3 Lower and upper tunnels crossing at right angle with some clearance. See Figure 7.19 and 7.20 KF = 0 Element has fluid phase = 1 Element has no fluid phase IH Heat source ID number (SMAP-T3) NSNODE Starting node number NSNEL Starting element number CMFAC Coordinate magnification factor

Card Group	Input Data and Definitions
2 For MODELNO =1 (Identical Two Crossing Tunnels, See Figures 7.15 & 7.16)	2.1.1 XL, YB, YT, ZL, t XL, YB, YT, ZL — Problem dimensions (See Figure 7.15) t — Radial distance from tunnel surface to the boundary of near region. Default value is 20% of the tunnel width. Example, t = liner thickness
	2.1.2 IPART, NDR, NTBND, NTOPN IPART = 0 Whole region (from Y =-YB to Y = YT) = 1 Upper region (from Y = 0.0 to Y = YT) = 2 Lower region (from Y =-YB to Y = 0.0) NDR — Number of elements along radial distance (t) NTBND — Number of elements along the length (XL+YB+YT+ZL) NTOPN — Number of elements along the perimeter of tunnel opening from node 1 to node 5. See Figure 7.16

Card Group	Input Data and Definitions
2 For MODELNO =1 (Identical Two Crossing Tunnels, see Figures 7.15 & 7.16)	2.1.3 NTNODE NTNODE Cards: $NODE_1$, X_1, Y_1 / $NODE_2$, X_2, Y_2 / - - - / - - - NTNODE — Number of nodes to specify tunnel shape NODE — Node number X — X-coordinate Y — Y-coordinate Note: Nodes from 1 to 5 are required

Card Group	Input Data and Definitions
2 For MODELNO =2 (Large and Small Crossing Tunnels, See Figures 7.17 & 7.18)	2.2.1 XL, YB, YT, ZL, t_l, t_s XL, YB, YT, ZL — Problem dimensions (See Figure 7.17) t_l, t_s — Radial distance from tunnel surface to the boundary of near region. t_l is for large tunnel and t_s for small tunnel ($t_l \geq t_s$). Default value is 20% of the tunnel width. Example, t = liner thickness
	2.2.2 IPART, NDR, NTBND, NTOPNL, NTOPNS IPART = 0 — Whole region (from Y = -YB to Y = YT) = 1 — Upper region (from Y = 0.0 to Y = YT) = 2 — Lower region (from Y = -YB to Y = 0.0) NDR — Number of elements along the radial distance (t_l for large tunnel and t_s for small tunnel) NTBND — Number of elements along the length (XL+YB+YT+ZL) NTOPNL — Number of elements along the perimeter of large tunnel opening from node 1 to node 7 See Figure 7.18 NTOPNS — Number of elements along the perimeter of small tunnel opening from node 1 to node 5 See Figure 7.18

Card Group	Input Data and Definitions
2	2.2.3

For MODELNO =2 (Large and Small Crossing Tunnels, see Figures 7.17 & 7.18)

2.2.3

Large Tunnel Shape, See Figure 7.18

NTLNODE

NTLNODE Cards

$NODE_1$, X_1, Y_1
$NODE_2$, X_2, Y_2
- - -
- - -

NTLNODE	Number of nodes to specify large tunnel
NODE	Node number
X	X-coordinate
Y	Y-coordinate

Note: Nodes from 1 to 7 are required

2.2.4

Small Tunnel Shape, See Figure 7.18

NTSNODE

NTSNODE Cards

$NODE_1$, Z_1, Y_1
$NODE_2$, Z_2, Y_2
- - -
- - -

NTSNODE	Number of nodes to specify small tunnel
NODE	Node number
Z	Z-coordinate
Y	Y-coordinate

Note: Nodes from 1 to 5 are required

Card Group	Input Data and Definitions
2 For MODELNO =3 (Crossing Tunnels with Clearance, See Figures 7.19 & 7.20)	2.3.1 XL, YB, YC, YT, ZL, t_l, t_u XL, YB, YC, YT, ZL — Problem dimensions (See Figure 7.19) t_l, t_u — Radial distance from tunnel surface to the boundary of near region. t_l is for lower tunnel and t_u for upper tunnel. Default value is 20% of the tunnel width. Example, t = liner thickness.
	2.3.2. NDRL, NDRU, NTBND, NTOPNL, NTOPNU NDRL — Number of elements along the radial distance (t_l) for lower tunnel NDRU — Number of elements along the radial distance (t_u) for upper tunnel NTBND — Number of elements along the length (XL+YB+YC+YT+ZL) NTOPNL — Number of elements along the perimeter of lower tunnel opening from node 1 to node 5. See Figure 7.20 NTOPNU — Number of elements along the perimeter of upper tunnel opening from node 1 to node 5. See Figure 7.20

Card Group	Input Data and Definitions
2	2.3.3

For MODELNO =3 (Crossing Tunnels with Clearance, See Figures 7.19 & 7.20)

2.3.3

Lower Tunnel Shape, See Figures 7.20

NTLNODE

NTLNODE Cards

$NODE_1$, X_1, Y_1
$NODE_2$, X_2, Y_2
- - -
- - -

NTLNODE	Number of nodes to specify lower tunnel
NODE	Node number
X	X-coordinate
Y	Y-coordinate

Note: Nodes from 1 to 5 are required

2.3.4

Upper Tunnel Shape, See Figures 7.20

NTUNODE

NTUNODE Cards

$NODE_1$, Z_1, Y_1
$NODE_2$, Z_2, Y_2
- - -
- - -

NTUNODE	Number of nodes to specify upper tunnel
NODE	Node number
Z	Z-coordinate
Y	Y-coordinate

Note: Nodes from 1 to 5 are required

Card Group	Input Data and Definitions
3 Boundary Conditions	3.1 NBOUND NBOUND — Number of boundaries to be specified If NBOUND = 0, no data is required hereafter
	3.2 NBOUND Cards IBTYPE, ISX, ISY, ISZ, IFX, IFY, IFZ (SMAP-3D) IBTYPE, ID, IDF (SMAP-T3) IBTYPE = 1 Interior volume (overriding default) = 2 Front surface (Z = ZL) = 3 Back surface (Z = 0.0) = 4 Left surface (X = 0.0) = 5 Right surface (X = XL) = 6 Top surface For MODELNO = 1 or 2, Y = YT if IPART = 0 or 1 Y = 0.0 if IPART =2 For MODELNO = 3, Y = YT + YC = 7 Bottom surface For MODELNO = 1 or 2, Y = 0.0 if IPART = 1 Y =-YB if IPART = 0 or 2 For MODELNO = 3, Y =-YB ISX Skeleton X DOF ISY Skeleton Y DOF ISZ Skeleton Z DOF

Card Group	Input Data and Definitions
2 Boundary Conditions	3.2 IFX Pore fluiud X DOF relative to skeleton IFY Pore fluiud Y DOF relative to skeleton IFZ Pore fluiud Z DOF relative to skeleton ISX, ISY, ISZ, IFX, IFY, IFZ = 0 Free to move in specified direction = 1 Fixed in specified direction *Note:* Default boundary conditions are ISX=ISY=ISZ=0 and IFX=IFY=IFZ=1 For SMAP-T3 ID = 0 Heat flow is specified = 1 Temperature is specified IDF = Time history identification number

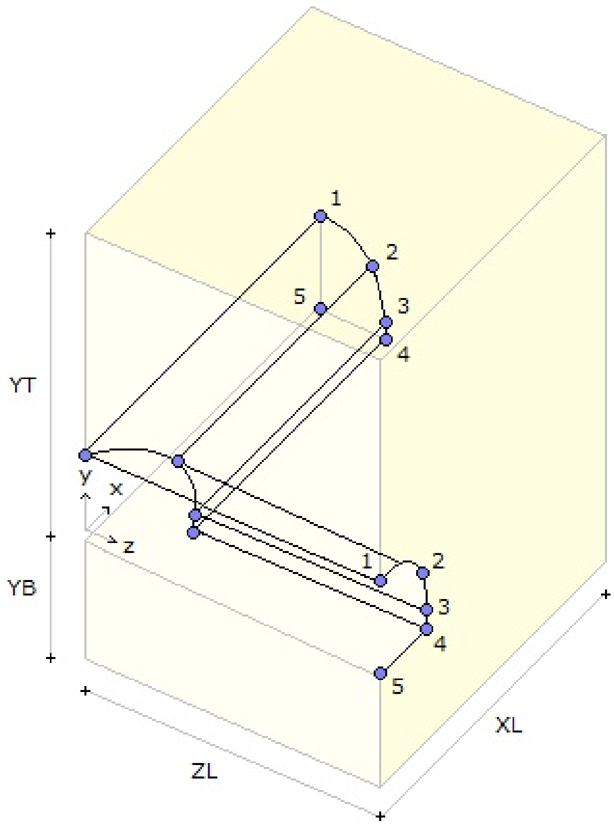


Figure 7.15 Schematic view of identical two crossing tunnels for MODELNO = 1

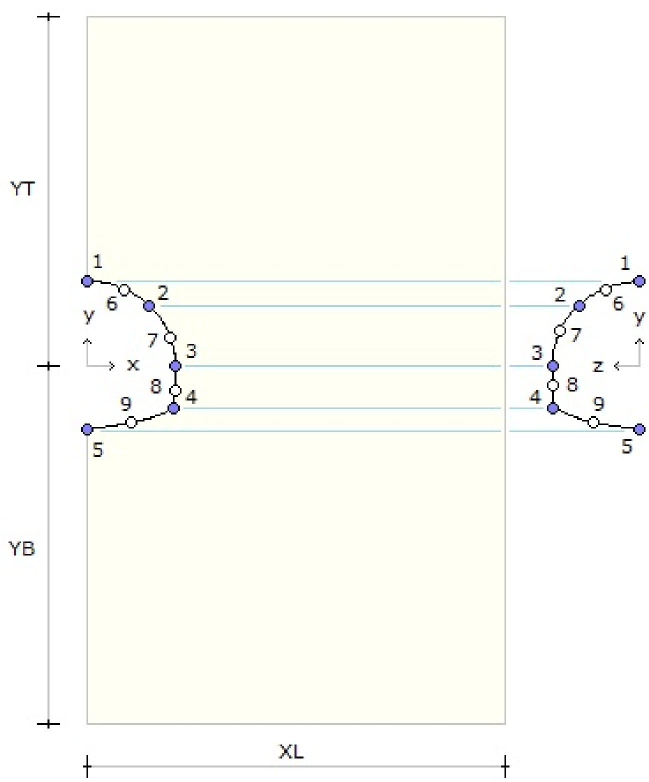


Figure 7.16 Node numbers defining tunnel shape for MODELNO = 1

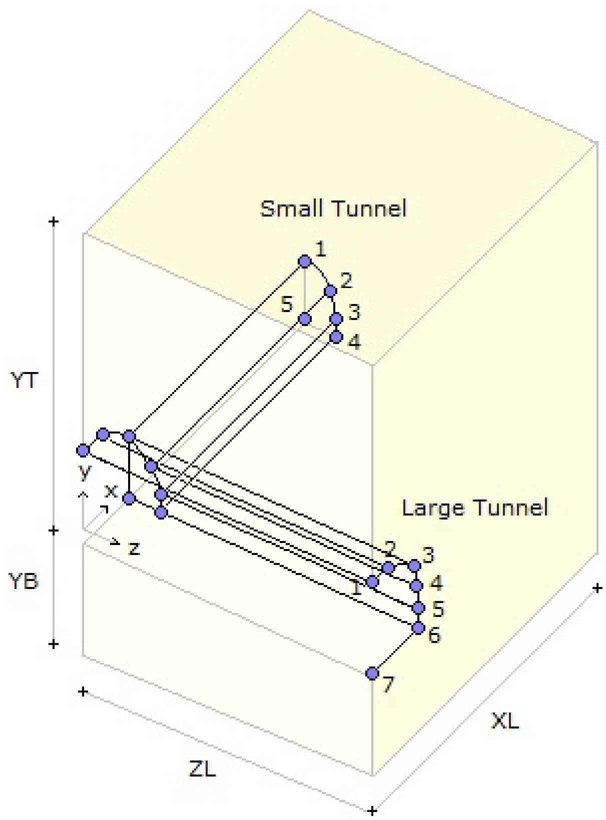


Figure 7.17 Schematic view of large and small crossing tunnels for MODELNO = 2

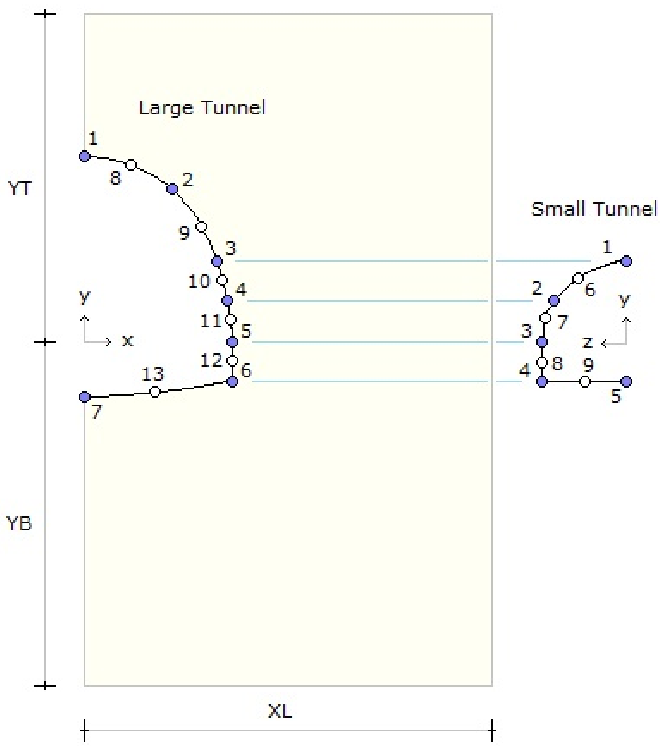


Figure 7.18 Node numbers defining tunnel shape
for MODELNO = 2

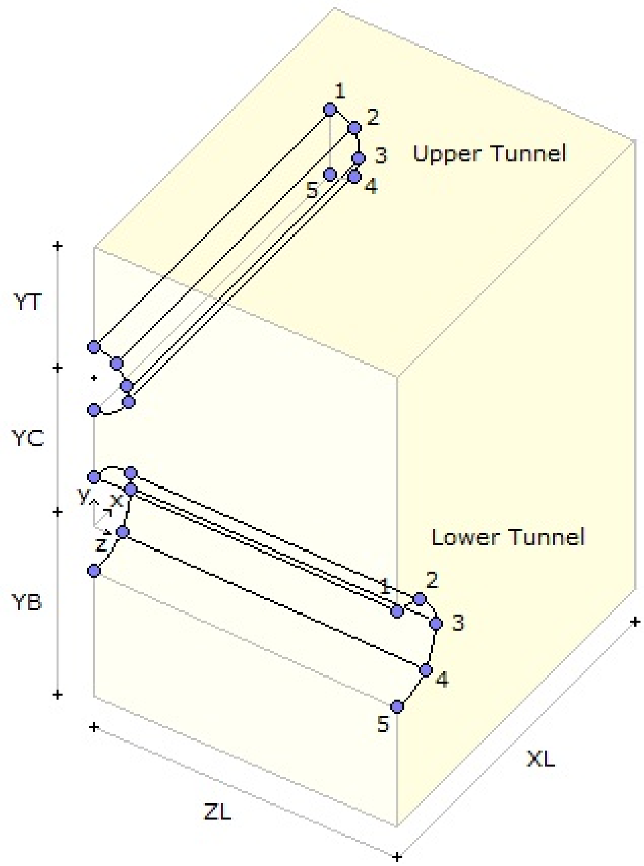


Figure 7.19 Schematic view of crossing tunnels with clearance for MODELNO = 3

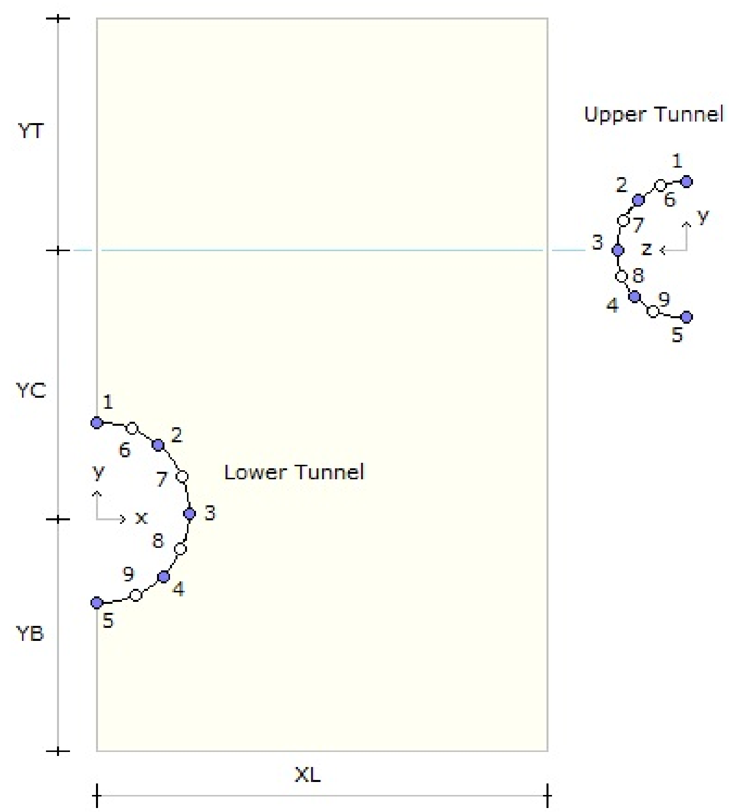


Figure 7.20 Node numbers defining tunnel shape for MODELNO = 3

GEN-3D

User's Manual

Card Group	Input Data and Definitions
1 General Information	1.1 TITLE TITLE Any title (Max = 60 characters)
	1.2 NBZ, NBNODE, NSNODE, NSNEL, IBOUND, IPLANE, CLOSE, CMFAC NBZ Number of blocks in z-direction NBNODE Number of block nodes in z-direction NSNODE Starting node number NSNEL Starting element number IBOUND = 0 Do not include control boundary (Default) = 1 Include boundary as wire frame (Truss) = 2 Include boundary as plane surface (Shell) = 3 Include boundary as frame and surface IPLANE = 0 Input 2D section in (X, Y) plane (Default) = 1 Input 2D section in (-Z, Y) plane = 2 Input 2D section in (X,-Z) plane = 3 Input 2D section in specified plane ICLOSE = 0 Open loop = 1 Closed loop First section represents last section CMFAC Coordinate magnification factor for 2D sec.
	1.2.1 If IBOUND = 0, skip this card X_{LEFT}, X_{RIGHT}, Y_{BOTTOM}, Y_{TOP}, Z_{BACK}, Z_{FRONT} X_{LEFT}, X_{RIGHT} X coordinates for left & right boundary Y_{BOTTOM}, Y_{TOP} Y coordinates for bottom & top boundary Z_{BACK}, Z_{FRONT} Z coordinates for back & front boundary

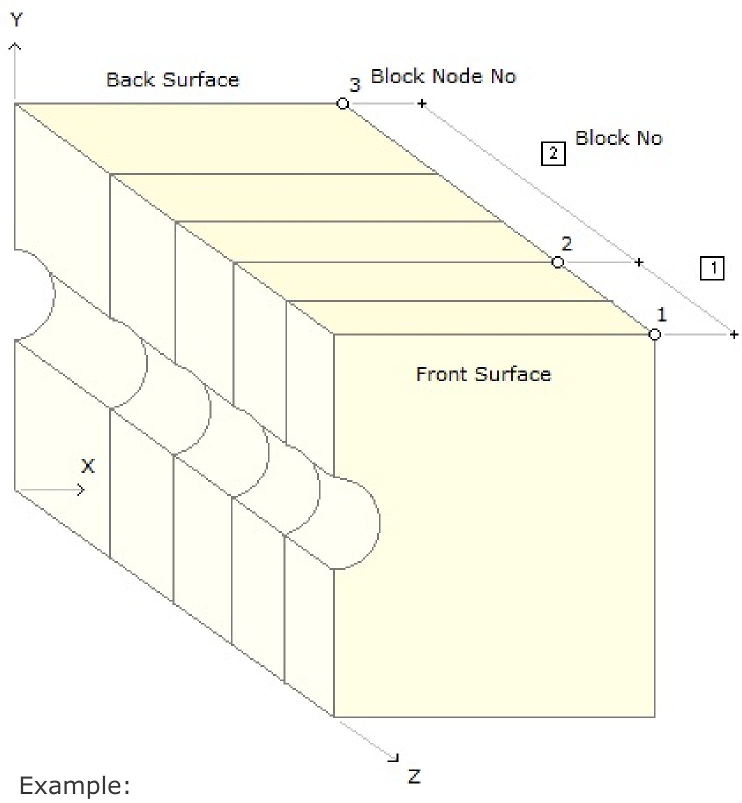
Card Group	Input Data and Definitions
1 General Information	1.2.2 Required only if IPLANE = 3 X_O, Y_O, Z_O X_a, Y_a, Z_a X_b, Y_b, Z_b X_O, Y_O, Z_O Coordinates defining local origin X_a, Y_a, Z_a Coordinates defining local x axis X_b, Y_b, Z_b Coordinates defining local y axis
	1.3 IBZ_{BASE} , IBZ_{FRONT} , IBZ_{BACK} See Figure 7.21 IBZ_{BASE} Base boundary code IBZ_{FRONT} Front surface boundary code IBZ_{BACK} Back surface boundary code IBZ ISZ IFZ 0 0 0 1 0 1 2 1 0 3 1 1 ISZ Z DOF for skeleton motion IFZ Z DOF for relative pore fluid motion ISZ,IFZ = 0 Free to move in specified direction. = 1 Fixed in specified direction. For SMAP-T3 ID = ISZ and IDF = IFZ ID = 0 Heat flow is specified = 1 Temperature is specified IDF Time history identification number

Card Group	Input Data and Definitions
2 Block Coordinate	2.1 NBNODE Cards: $NODE_1$, Z_1, X_1 $NODE_2$, Z_2, X_2 - - - - - - - - - NODE Node number Z Z coordinate X X coordinate Note: Z and X define the coordinates of center line

Card Group	Input Data and Definitions
3	3.1 BLNAME BLNAME Block name (up to 60 characters)
	3.2 IBLNO IBLNO Block number
Data for Each Block	3.3 I, J, LTYPE, IMATC, IMATB, IMATT, NIXCH (See Figure 7.21) I, J End node number of a block LTYPE = 0 Straight line = 1 Circular line IMATC Material number increment for Continuum IMATB Material number increment for Beam IMATT material number increment for Truss NIXCH Number of materials for index change
	3.4 NDZ, α, MC_1, MC_2, MC_3, MB, MT NDZ Number of elements in z-direction α = 0.5 Element length is constant = 0.3 Element length is growing from I to J = -0.3 Element length is growing from J to I MC Material number not to be modified for Continuum MB Material number not to be modified for Beam MT Material number not to be modified for Truss If MC/MB/MT has negative sign, that material will be removed
	3.5 Required only for LTYPE = 1 Z_o, X_o, R, θ_b, θ_e Z_o, X_o Coordinates of origin R Radius θ_b, θ_e Beginning and ending angle (°)

Card Group	Input Data and Definitions
3	3.6 Required only for NIXCH > 0 NIXCH Cards: MAT, NMAT, NI_1, NI_2, NI_3, NI_4, NI_5, NI_6, NI_7, NI_8 MAT — Material number NMAT — New material number NI_i — Index number increment at NI_i Note: Index change applied only for block first layer. If NMAT = -1, it assumes that new material property number 1 consists of joint elements whose joint face designates number KS = 6
4 Transmitting Boundary Generation	4.1 ITRANB ITRANB = 0 Do not generate transmitting boundary = 1 Generate transmitting boundary = 2 Generate element transmitting boundary If ITRANB = 0, rest of Cards are not used If ITRANB = 2, go to Card Group 4.4
4.2. Material Property	4.2.1 NTNC NTNC — Number of material property set
	4.2.21 NTNC Cards: MAT, RHO, CP, CS MAT — Material number RHO — Mass density CP — Compression wave speed CS — Shear wave speed

Card Group	Input Data and Definitions
4 Transmitting Boundary Generation	4.3 Nodal Transmitting Boundary Generation (Can be repeated in any order) For surface whose normal is x-direction, 1 NPT $N_1, N_2, \ldots, N_{NPT}$ For surface whose normal is y-direction, 2 NPT $N_1, N_2, \ldots, N_{NPT}$ For surface whose normal is z-direction (Front Surface) 3 For surface whose normal is z-direction (Back Surface) 4 For end of transmitting boundary generation, 0 NPT Number of nodes $N_1, N_2, \ldots, N_{NPT}$ Node numbers
	4.4 Element Transmitting Boundary Generation (Can be repeated in any order) For surface whose normal is X-Y plane 1 NPT $N_1, N_2, \ldots, N_{NPT}$ For front surface, 3 For back surface, 4 For end of transmitting boundary generation, 0 NPT Number of nodes $N_1, N_2, \ldots, N_{NPT}$ Node numbers



Example:

IPLANE = 0 (Input 2D section in X-Y plane)

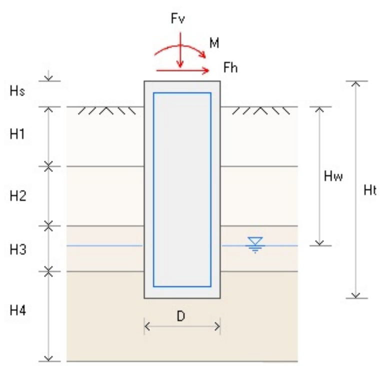
For Block No 1,	I = 1	J = 2	NDZ = 2	ɑ = 0.4
For Block No 2,	I = 2	J = 3	NDZ = 3	ɑ = 0.4

Total Number of Blocks, NBZ = 2
Total Number of Block Nodes, NBNODE = 3

Figure 7.21 Block index for GEN-3D

PILE-3D

User's Manual

Card Group	Input Data and Definitions
1 Title	1.1 Title Title Project title
	1.2 Stitle Stitle Project subtitle
2 Pile Properties	2.1 D, H_t, H_s, H_w, N_t D Pile diameter (m) H_t Pile length (m) H_s Pile length above ground surface (m) H_w Depth of water table (m) N_t Number of finite elements along the pile length  Fig. 1 Pile dimension
	2.2 E_p, v_p, t_p [Steel Pipe / Liner Plate] E_p Young's modulus (t/m^2) v_p Poisson's ratio t_p Thickness (mm) To exclude steel pipe, set t_p = 0.0

Card Group	Input Data and Definitions
2 Pile Properties	2.3 N_r, d_{top}, d_{bot} N_r Number of reinforcing bar layers d_{top} Top cover depth (cm) d_{bot} Bottom cover depth (cm) Note: Reinforcing bars are not considered, set N_r = 0
	2.4 D_b, d_b, N_b [For Each Longitudinal Reinforcing Bar Layer] D_b Diameter (mm) d_b Cover depth (mm) N_b Number of bars
	2.5 E_c, v_c, φ, c, T, γ [Concrete] E_c Young's modulus (t/m^2) v_c Poisson's ratio φ Internal friction angle (°) c Cohesion (t/m^2) T Tensile strength (t/m^2) γ Unit weight (t/m^3)
	2.6 E_r, σ_y [Reinforcing Bar] E_r Young's modulus (t/m^2) σ_y Yield stress (t/m^2)
	2.7 E_b, G_b, φ, c, T, t_b [Pile Base Interface] E_b Young's modulus normal to the base (t/m^2) G_b Shear modulus along the base (t/m^2) φ Friction angle along the base (°) c Cohesion along the base (t/m^2) T Tensile strength normal to the base (t/m^2) t_b Thickness (mm)

Card Group	Input Data and Definitions
3	Soil / Rock Layers

3.1

N_{LAYER}

- N_{LAYER} — Total number of layers

For Each Layer

3.2

L_{NO}, M_{NO}, H, γ_t, K_o

- L_{NO} — Layer number
- M_{NO} = 1 — Elastic model
  - 3 — Mohr-Coulomb model
  - 4 — In situ rock model
  - 9 — Modified Can-Clay model
  - 12 — Generalized decoupled hyperbolic model
- H — Thickness of layer (m)
- γ_t — Unit weight (t/m^3)
- K_o — Coefficient of earth pressure at rest

3.3

E_j, G_j, φ, c, T, t_j [Pile Side Interface, see Fig. 2]

- E_j — Young's modulus normal to interface (t/m^2)
- G_j — Shear modulus along the interface (t/m^2)
- φ — Friction angle along the interface (°)
- c — Cohesion along the interface (t/m^2)
- T — Tensile strength normal to interface (t/m^2)
- t_j — Thickness (mm)

3.4.1

E, v [M_{NO} = 1: Elastic]

- E — Young's modulus (t/m^2)
- v — Poisson's ratio

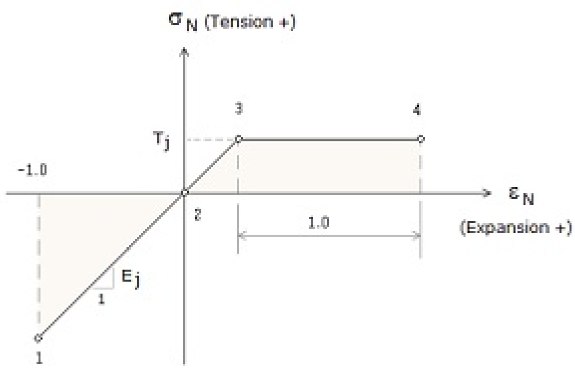
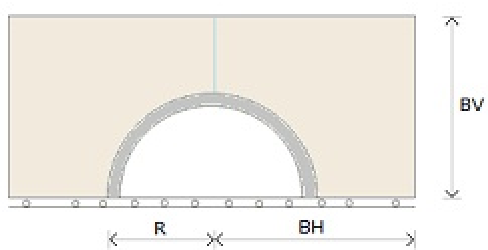
3.4.3

E, v, φ, c, T [M_{NO} = 3: Mohr-Coulomb]

- E — Young's modulus (t/m^2)
- v — Poisson's ratio
- φ — Internal friction angle (°)
- c — Cohesion (t/m^2)
- T — Tensile strength (t/m^2)

Card Group		Input Data and Definitions
3 Soil / Rock Layers	For Each Layer	3.4.4 E, v, m, s, σ_c, T [M_{NO} = 4: In Situ Rock] E Young's modulus (t/m^2) v Poisson's ratio m,s Hoek & Brown material parameters σ_c Unconfined strength of intact rock (t/m^2) T Tensile strength (t/m^2)
		3.4.9 P_c, e_o, v, C_c, C_r, T [M_{NO} = 9: Modified Cam-Clay] P_c Preconsolidation pressure (t/m^2) e_o Initial void ratio v Poisson's ratio C_c Virgin compression index C_r Swelling / recompression index M Strength parameter
		3.4.12 E, v, φ, c, E_u, K [M_{NO} = 12: Generalized Decoupled] E Loading Young's modulus (t/m^2) v Poisson's ratio φ Internal friction angle (°) c Cohesion (t/m^2) E_u Unloading Young's modulus (t/m^2) K The ratio of shear strength in triaxial extension over triaxial compression at the same mean pressure

Card Group	Input Data and Definitions
4 External Loads	4.1 F_V, F_H, M, N_{STEP} F_V Vertical force (t) Compression is positive F_H Horizontal force (t) M Moment (t-m) N_{STEP} Number of computational steps through which external loads are applied
5 Anchor Bolt	5.1 D_a, d_a, L_a, N_{bolt}, E_a, σ_{ya} D_a Diameter (mm) d_a Cover depth (mm) L_a Length embedded in pile (m) N_{bolt} Number of bolts E_a Young's modulus (t/m^2) σ_{ya} Yield stress (t/m^2) Note: For N_{bolt} = 0, Concrete below GS is not active
6 F. E. Mesh on Plan View	6.1 FineMesh, NearMesh, N_{DIV}, B_H, B_V [See Fig. 3] FineMesh = 0 Coarse mesh 1 Fine mesh NearMesh = 0 All quad mesh 1 Quad and triangle mesh N_{DIV} Number of divisions for outer zone B_H Horizontal dimension (m) B_V Vertical dimension (m)

Card Group	Input Data and Definitions
Notes	 Fig. 2 Joint Material Normal Stress-Strain Relation  Fig. 3 Finite Element Mesh on Plan View

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

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

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

Card Group	Input Data and Definitions	
3 Data for Each Line Block [IBETYPE =1]	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 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 Data for Each Quad Surface Block [IBETYPE =2]	3.5 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 $\leq$ min (NDX, NDY) and NT i + NT j $\leq$ 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 Data for Each Quad Surface Block [IBETYPE =2]	3.6

Only for ICOORD = 2 and ILAG = 2

NSEG

NSEG Cards ┌ $ALPA_1$, $NDIV_1$
│ $ALPA_2$, $NDIV_2$
└ - -

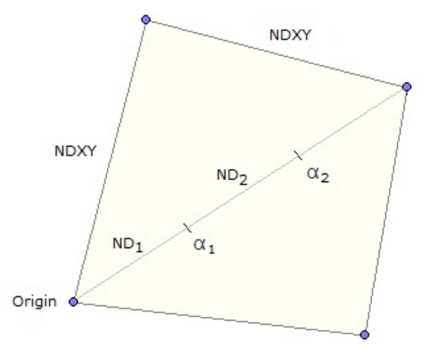
NSEG Number of segments

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

Card Group	Input Data and Definitions
3 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

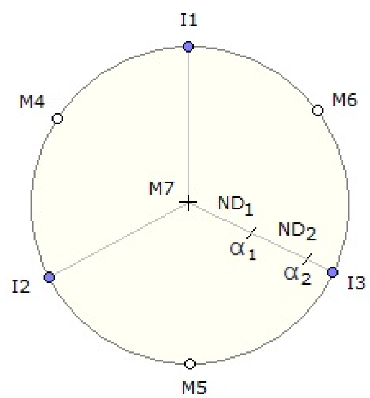
NSEG

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 Data for Each Hexahedron Volume Block [IBETYPE =3]	3.5 MATNO, NDX, NDY, NDZ, KS, KF (For ISMAP = 3) MATNO, NDX, NDY, NDZ, IDH (For ISMAP =-3) NT_1, NT_2, NT_3, NT_4 MAT_1, MAT_2, MAT_3, MAT_4 MATNO — Material property number NDX — Number of elements in I_2 - I_1 direction NDY — Number of elements in I_2 - I_3 direction NDZ — Number of elements in I_2 - I_6 direction KS = -1 — Element has high explosive solid phase = 0 — Element has solid phase > 0 — Element has joint and absolute value of KS represents face designation number. KF = 0 — Element has fluid phase = 1 — Element has no fluid phase IDH — Heat generation history ID number NT & MAT — See descriptions on page 7-92

Card Group	Input Data and Definitions
3 Data for Each Prism Volume Block [IBETYPE = -3]	3.1 BLNAME BLNAME Block name (Max = 60 characters)
	3.2 ICOORD, IMODE, ILAG Interpolation based on ICOORD = 1 Rectangular coordinate = 2 Spherical coordinate = 3 Cylindrical coordinate Modify generated coordinate IMODE = 0 Do not modify = 1 Modify using reference node (M_{22}) as origin for ICOORD = 1 Modify coordinate based on rectangular grid for ICOORD = 2 or 3 ILAG = 0 Serendipity interpolation = 1 Lagrangian interpolation

Card Group	Input Data and Definitions
3 Data for Each Prism Volume Block [IBETYPE =-3]	3.3 I_1, I_2, I_3, I_4, I_5, I_6 M_7, M_8, M_9, M_{10}, M_{11}, M_{12}, M_{13}, M_{14}, M_{15}, M_{16}, M_{17} M_{18}, M_{19}, M_{20}, M_{21} M_{22}, M_{23}, M_{24} See Figure 7.22 I_1 - I_6 Corner node number of a block M_7- M_{20} Side node number of a block M_{21} Center node number of a block For ICOORD = 2 or IMODE = 1 M_{22} Node number defining origin of spherical coordinate for ICOORD = 2, or node number defining reference origin to the whole volume for IMODE = 1 For ICOORD = 3 M_{22} Node number defining reference origin of cylindrical coordinate. M_{23} Node number defining cylinder axis M_{22}-M_{23} M_{24} Node number defining other local axis M_{22}-M_{24} which is normal to cylinder axis.

Card Group		Input Data and Definitions
3	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

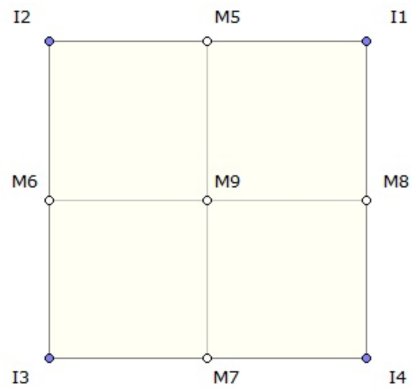
Note: Boundary Conditions

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

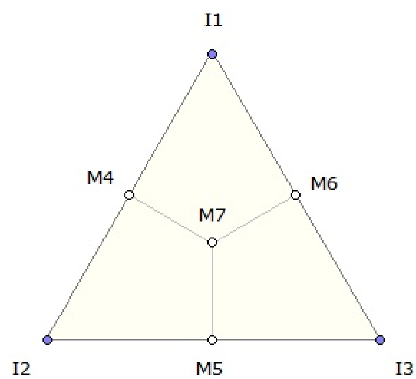


Figure 7.22 Block index for PRESMAP-GP

Hexahedron Volume Block

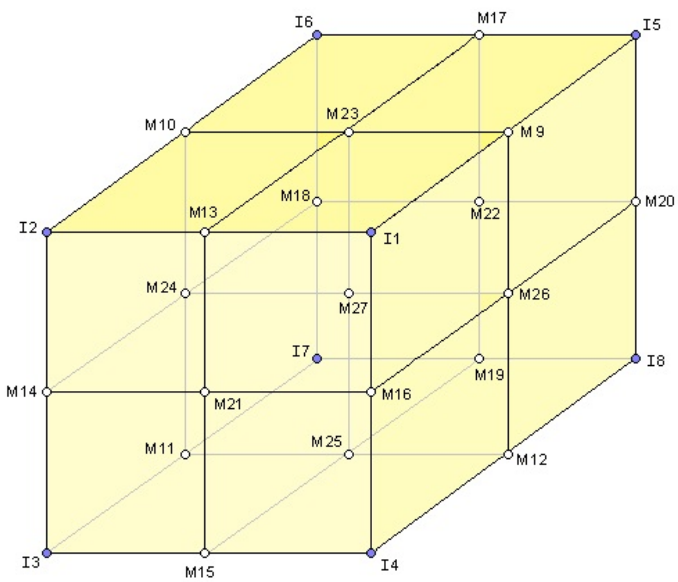


Figure 7.22 Block index for PRESMAP-GP (Continued)

Prism Volume Block

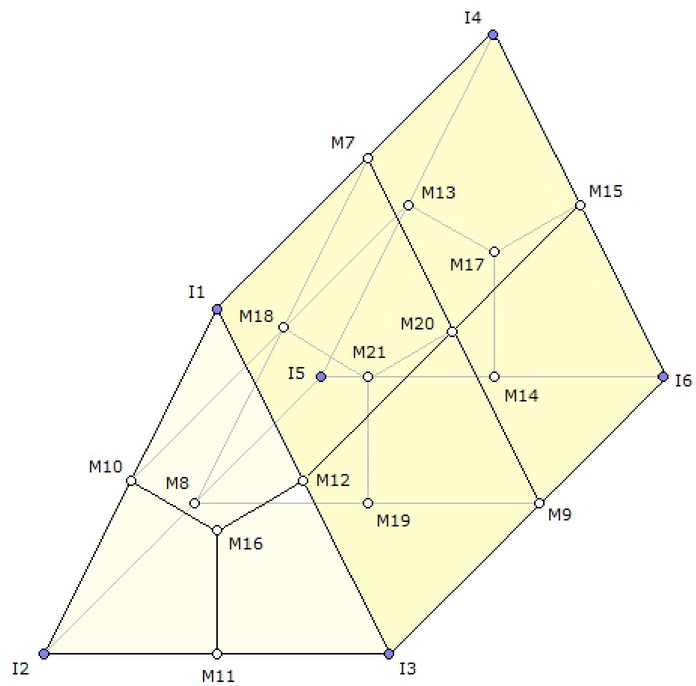


Figure 7.22 Block index for PRESMAP-GP (Continued)

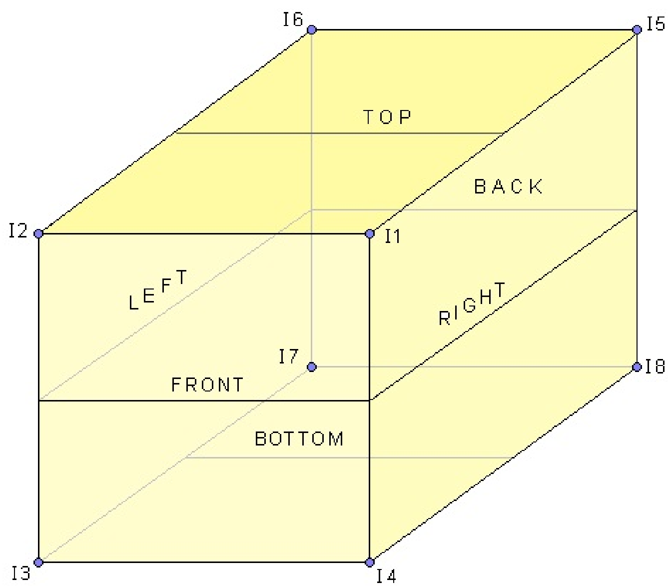


Figure 7.23 Boundary surface designation for Hexahedron Volume Block

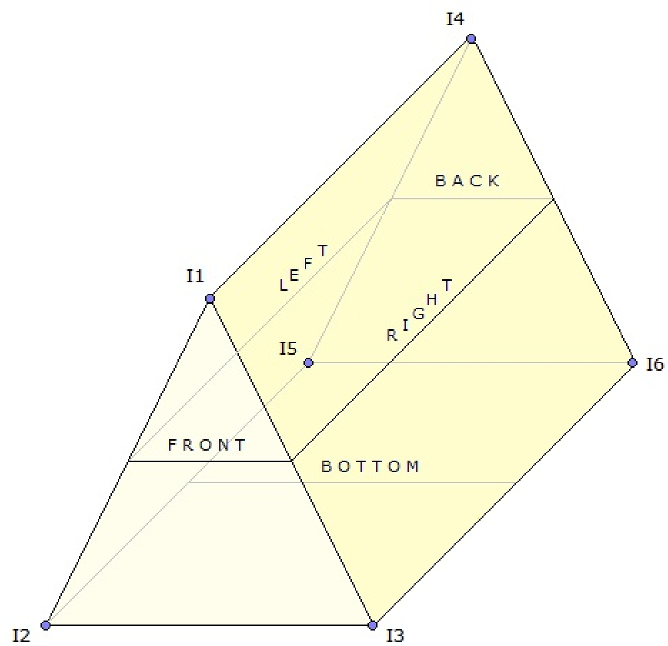


Figure 7.24 Boundary surface designation for Prism Volume Block

JOINT-2D

User's Manual

Card Group	Input Data and Definitions
1	1.1 TITLE TITLE Any title of up to 80 characters
General Information	1.2 AllJoint, ThicAJ AllJoint = 0 Generates Joint Elements along the All Interfaces between Continuum Elements. Cards 2, 3, and 4 are not used. = 1 Generates Joint Elements for the Material Numbers of Continuum Elements as specified in Cards 2 and 3. Card 4 is not used. = 2 Generates Joint Elements for the Element Surface Numbers of Continuum Elements as specified in Card 4. Cards 2 and 3 are ignored. ThicAJ Thickness used for AllJoint = 0. To Run JOINT-2D **Method 1** SMAP-2D > Run > Mesh Generator > PreSmap > Joint Specify input and output file names shown on the screen. **Method 2** 1. Select SMAP-2D > Setup > PLOT 3D Specify Joint Thickness View Factor which is greater than 0.0 Example: Joint Thickness View Factor = 1.0 2. Select SMAP-2D > Mesh > F.E. Mesh > Open This wil open Mesh File of Continuum Elements. Input file Joint.inp should exist in the Working Directory. Output File JointedMesh.Mes is shown in Working Directory.

Card Group		Input Data and Definitions
AllJoint = 1: Internal / Boundary Joint Generation	Internal Joint Generation	2.1 NumIJ, ThicIJ NumIJ — Number of continuum materials for Internal Joint. If NumIJ = 0, go to Card 3 ThicIJ — Thickness of Internal Joints
		2.2 NumIJ Cards: $MatIJ_1$ $InnerBeam_1$ $OuterBeam_1$; $MatIJ_2$ $InnerBeam_2$ $OuterBeam_2$; - - - MatIJ — Material property number of continuum element for Internal Joints (See Fig. 1) InnerBeam = 0 Do not include; = 1 Include Inner Beam element OuterBeam = 0 Do not include; = 1 Include Outer Beam element
	Boundary Joint Generation	3.1 NumBJ, ThicBJ, InterfaceJoint NumBJ — Number of continuum materials for Boundary Joint. If NumBJ = 0, go to Card 4 ThicBJ — Thickness of Boundary Joints. If negative, inside continuum elem. contacts joint face InterfaceJoint = 0 Do not include; = 1 Include Interface Joint Element
		3.2 NumBJ Cards: $MatBJ_1$ $InnerBeam_1$ $OuterBeam_1$; $MatBJ_2$ $InnerBeam_2$ $OuterBeam_2$; - - - MatBJ — Material property number of continuum element for Boundary Joints (See Fig. 1) InnerBeam = 0 Do not include; = 1 Include Inner Beam element OuterBeam = 0 Do not include; = 1 Include Outer Beam element

Card Group	Input Data and Definitions
4 AllJoint = 2 : Surface Joint Generation	4.1 NumSJG NumSJG — Number of Groups for Surface Joints If NumSJG = 0, end of data
	4.2 NumSJG Cards: $NumSJ_1$ $ThicSJ_1$; $NumSJ_2$ $ThicSJ_2$; - -; - - $NumSJ_i$ — Number of element surfaces in Group i $ThicSJ_i$ — Thickness of Surface Joint in Group i
	For Each Surface Joint Group 4.3 $NumSJ_i$ Cards: $ElementNo_1$ $SurfaceNo_1$; $ElementNo_2$ $SurfaceNo_2$; - -; - - ElementNo — Continuum Element No SurfaceNo — Continuum Element Surface No where Surface Joint is generated Note: To take new node number for corner contact element, set SurfaceNo = 0 Refer to page 4-67 of SMAP-2D User's Manual for Element Surface designation

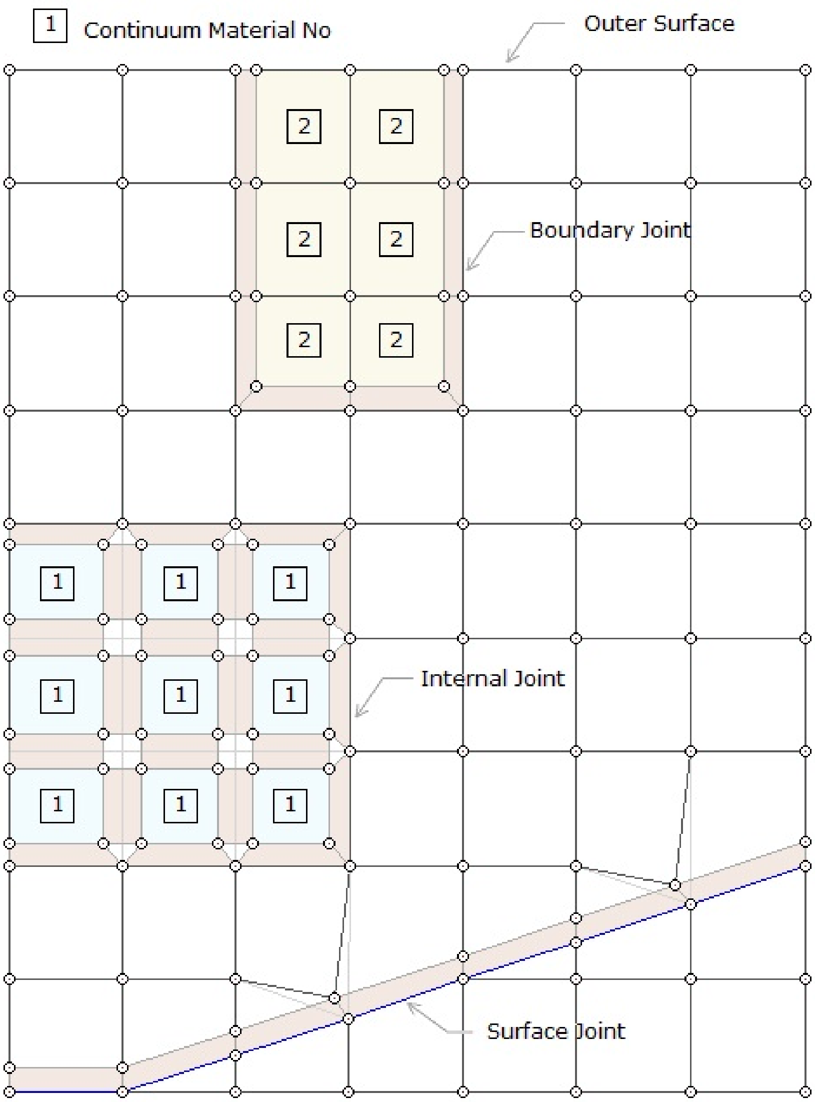


Figure 1 Joint element generation

JOINT-3D

User's Manual

Card Group	Input Data and Definitions
1	1.1 TITLE TITLE Any title of up to 80 characters
General Information	1.2 AllJoint, ThicAJ AllJoint = 0 Generates Joint Elements along the All Interfaces between Continuum Elements. Cards 2, 3, and 4 are not used. = 1 Generates Joint Elements for the Material Numbers of Continuum Elements as specified in Cards 2 and 3. Card 4 is not used. = 2 Generates Joint Elements for the Element Surface Numbers of Continuum Elements as specified in Card 4. Cards 2 and 3 are ignored. ThicAJ Thickness used for AllJoint = 0. To Run JOINT-3D **Method 1** SMAP-3D > Run > Mesh Generator > PreSmap > Joint Specify input and output file names shown on the screen. **Method 2** 1. Select SMAP-3D > Setup > PLOT 3D Specify Joint Thickness View Factor which is greater than 0.0 Example: Joint Thickness View Factor = 1.0 2. Select SMAP-3D > Mesh > F.E. Mesh > Open This wil open Mesh File of Continuum Elements. Input file Joint.inp should exist in the Working Directory. Output File JointedMesh.Mes is shown in Working Directory.

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions</th></tr>
<tr><td rowspan="4">AllJoint = 1: Internal / Boundary Joint Generation</td><td rowspan="2">Internal Joint Generation</td><td>2.1
NumIJ, ThicIJ

NumIJ Number of continuum materials for Internal Joint. If NumIJ = 0, go to Card 3

ThicIJ Thickness of Internal Joints</td></tr>
<tr><td>2.2
NumIJ Cards: $MatIJ_1$ $InnerShell_1$ $OuterShell_1$
$MatIJ_2$ $InnerShell_2$ $OuterShell_2$
- - -

MatIJ Material property number of continuum element for Internal Joints (See Fig. 1)

InnerShell = 0 Do not include
= 1 Include Inner Shell element

OuterShell = 0 Do not include
= 1 Include Outer Shell element</td></tr>
<tr><td rowspan="2">Boundary Joint Generation</td><td>3.1
NumBJ, ThicBJ, InterfaceJoint

NumBJ Number of continuum materials for Boundary Joint. If NumBJ = 0, go to Card 4

ThicBJ Thickness of Boundary Joints. If negative, inside continuum elem. contacts joint face.

InterfaceJoint = 0 Do not include
= 1 Include Interface Joint Element</td></tr>
<tr><td>3.2
NumBJ Cards: $MatBJ_1$ $InnerShell_1$ $OuterShell_1$
$MatBJ_2$ $InnerShell_2$ $OuterShell_2$
- - -

MatBJ Material property number of continuum element for Boundary Joints (See Fig. 1)

InnerShell = 0 Do not include
= 1 Include Inner Shell element

OuterShell = 0 Do not include
= 1 Include Outer Shell element</td></tr>
</table>

Card Group	Input Data and Definitions
4 AllJoint = 2 : Surface Joint Generation	4.1 NumSJG NumSJG — Number of Groups for Surface Joints If NumSJG = 0, end of data
	4.2 NumSJG Cards: $NumSJ_1$ $ThicSJ_1$ / $NumSJ_2$ $ThicSJ_2$ / - - / - - $NumSJ_i$ — Number of element surfaces in Group i $ThicSJ_i$ — Thickness of Surface Joint in Group i
	For Each Surface Joint Group 4.3 $NumSJ_i$ Cards: $ElementNo_1$ $SurfaceNo_1$ / $ElementNo_2$ $SurfaceNo_2$ / - - / - - ElementNo — Continuum Element No SurfaceNo — Continuum Element Surface No where Surface Joint is generated Note: To take new node number for corner contact element, set SurfaceNo = 0 Refer to page 4-68 of SMAP-3D User's Manual for Element Surface designation

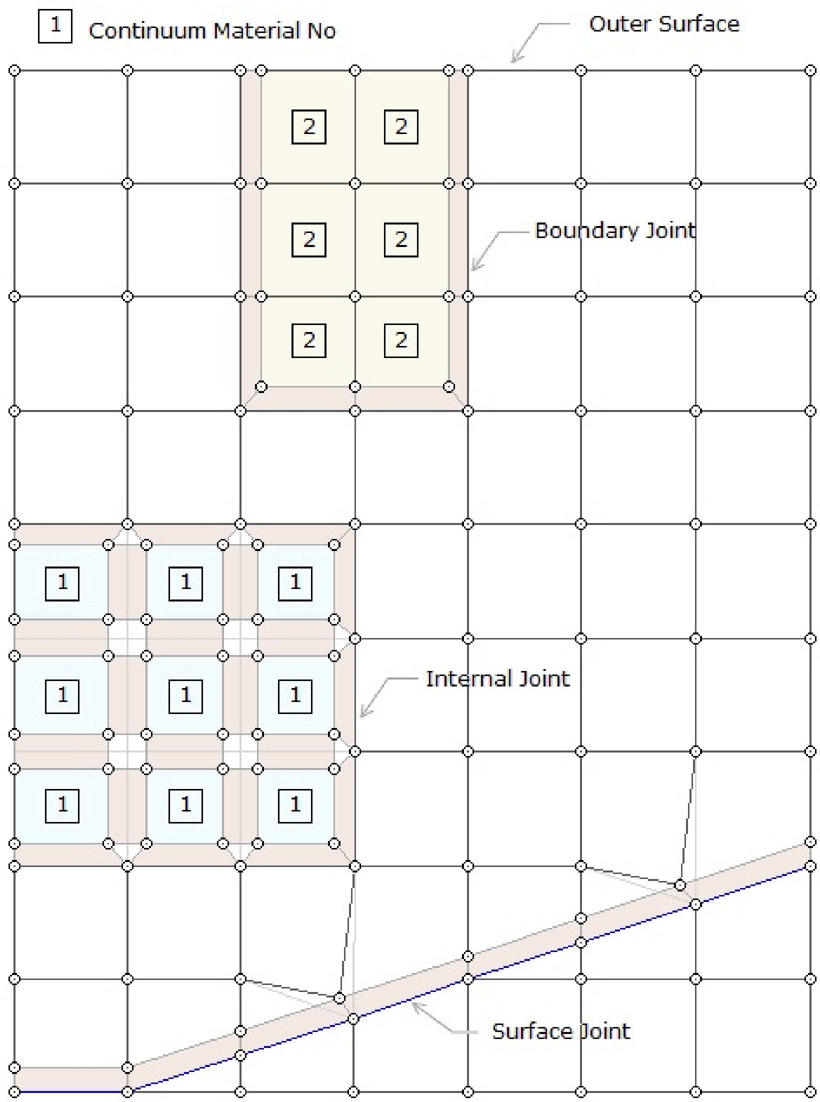


Figure 1 Joint element generation

INTERSECTION
User's Manual

Introduction

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.

There are two INTERSECTION programs provided in this manual; SHELL ELEMENT and TWO TUNNELS.

7.12.1 SHELL ELEMENT

SHELL ELEMENT intersection is the basic program which can be applied to find the line of intersection of three-dimensional surfaces.

First, you need to prepare a SMAP-3D mesh file composed of Shell Elements with different material numbers.

SHELL ELEMENT intersection can be accessed by selecting menu
Run → Mesh Generator → PreSmap → Intersection → Shell Element

or
Setup → PLOT 3D → Compute Intersection → Yes
and then open mesh file Plot → Mesh → F. E. Mesh → Open

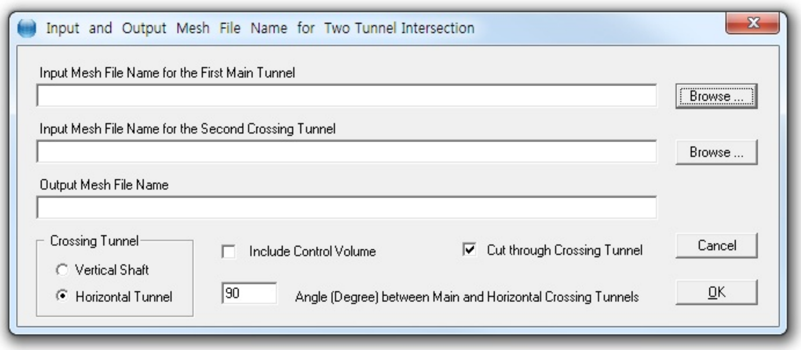
Note that computed coordinates of intersections are represented by Truss Elements.

7.12.2 TWO TUNNELS

TWO TUNNELS intersection is the special program where the second crossing tunnel cuts through the first main tunnel at some angle.

First, you need to prepare two SMAP-2D mesh files representing for cross sections of the first main and the second crossing tunnels. These cross sections are modeled by two-dimensional Beam Elements with different beam section numbers.

TWO TUNNELS intersection can be accessed by selecting menu
Run → Mesh Generator → PreSmap → Intersection → Two Tunnels



Browse the input files for Main and Crossing tunnels.
Select the Vertical Shaft or Horizontal Tunnel at some angles.

Main and Crossing tunnels are extended to three dimensional Shell elements using the default GEN-3D input file; ZI-A.dat and ZI-B.dat, respectively, in the sub directory Temp.

Note that output file Intersection.Mes contains Shell Elements representing both main and crossing tunnels.

For best appearance of generated meshes, you need to copy C:\SMAP\CT\CTDATA\DV-ADRGN.dat into Working Directory and then modify control parameters in Intersection for PLOT-3D.

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 (T2), 2 (S2), 3 (2D), 9 (W2)</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, otherwise element numbers are automatically reordered by program.</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	3.1 FILEA FILEM FILEA Input file name containing existing mesh FILEM Output file name to store modified mesh
	3.2 NSNEL, NSNODE, NBNEL, NTNEL NSNEL New starting continuum element number NSNODE New starting node number NBNEL New starting beam element number NTNEL New starting truss element number Note: NBNEL & NTNEL are used for IEDIT = 0, 1, 6
Modifying Existing Mesh (IMOD =1)	3.3 IEDIT, MC_1, MC_2, MC_3, MB, MT IEDIT = 0 Change coordinates = 1 Change boundary codes = 2 Cut elements = 3 Change material numbers = 4 Build user-defined curves and material zones = 6 Change element index order MC Continuum material number to be kept MB Beam material number to be kept MT Truss material number to be kept Note: MC, MB, and MT are applicable only for IEDIT = 2 and 3

Card Group	Input Data and Definitions	
3 Modifying Existing Mesh (IMOD = 1)	Changing Coordinates (IEDIT = 0)	3.3.1.1 X_o, Y_o, X_{oNew}, Y_{oNew} X_o, Y_o — Reference origin X_{oNew}, Y_{oNew} — New origin
		3.3.1.2 X_{scale}, Y_{scale} X_{scale}, Y_{scale} — Scale factors for X, Y coordinates Note: New coordinates $X_{(new)}$ and $Y_{(new)}$ are computed as follows: $X_{(new)} = X_{oNew} + (X-X_0)\ X_{scale}$ $Y_{(new)} = Y_{oNew} + (Y-Y_0)\ Y_{scale}$

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Changing Boundary Codes (IEDIT = 1)	3.3.2.1 IRANGE IRANGE = 0 Range specified by coordinates = 1 Range specified by node numbers = 2 Range specified by line strip = 3 Range specified by material numbers
		3.3.2.2.1 Required only for IRANGE = 0 X_{start}, Y_{start}, X_{end}, Y_{end} X_{start}, Y_{start} Coordinates for lower left boundary X_{end}, Y_{end} Coordiantes for upper right boundary
		3.3.2.2.2 Required only for IRANGE = 1, 2, 3 NODE NOD_1, NOD_2, ..., NOD_{NODE} NODE Number of nodes/materials to be specified NOD_i Node/Material number (Note 1 in page 8-7) Line strip is defined counterclockwise. For IRANGE = 3, Nodes refer to Material numbers.
		3.3.2.3 INSIDE (Not applicable for IRANGE= 3) INSIDE = 0 Apply inside of range = 1 Apply outside of range
		3.3.2.4 ISX, ISY, IFX, IFY, IRZ (SMAP-2D) IDX, IDY, IDT (SMAP-S2) ID, IDF (SMAP-T2) ISX, ISY X and Y DOF for skeleton motion IFX, IFY X and Y DOF for relative motion IRZ Z DOF for beam rotation IDX, IDY X and Y DOF for skeleton motion IDT Z DOF for beam rotation ID Heat flow (0), Temperature (1) specified IDF Time history identification number

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Cutting Elements (IEDIT = 2)	3.3.3.1 IRANGE IRANGE = 0 Range specified by coordinates = 1 Range specified by element numbers
		3.3.3.2.1 Required only for IRANGE = 0 X_{start}, Y_{start}, X_{end}, Y_{end} X_{start}, Y_{start} Coordinates for lower left boundary X_{end}, Y_{end} Coordinates for upper right boundary
		3.3.3.2.2 Required only for IRANGE = 1 NOEL NEL_1, NEL_2, ..., NEL_{NOEL} NOEL Number of elements to be specified NEL_i Element number (See Note 2)
		3.3.3.3 INSIDE INSIDE = 0 Apply inside of range = 1 Apply outside of range Note 1: NOD_1, $-NOD_2$ generates from NOD_1 to NOD_2 Note 2: NEL_1, $-NEL_2$ generates from NEL_1 to NEL_2

Card Group	Input Data and Definitions	
3 Modifying Existing Mesh (IMOD = 1)	Change Material No (IEDIT = 3)	3.3.4 IRANGE IRANGE = 0 Range specified by coordinates = 1 Range specified by element numbers
		3.3.4.1 Required only for IRANGE = 0 X_{start}, Y_{start}, X_{end}, Y_{end} X_{start}, Y_{start} Coordinates for lower left boundary X_{end}, Y_{end} Coordiantes for upper right boundary
		3.3.4.2 Required only for IRANGE = 1 NOEL NEL_1, NEL_2, ..., NEL_{NODE} NOEL Number of elements to be specified NEL_i Element number (See Note 2 in page 8-7)
		3.3.4.3 INSIDE INSIDE = 0 Apply inside of range = 1 Apply outside of range
		3.3.4.4 MATC, MATB, MATT MATC New continuum material number MATB New beam material number MATT New truss material number Note: When new material number is zero, keep the old material number

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	3.3.5 Build User-Defined Curves and Material Zones (IEDIT = 4)	3.3.5.1 NODE NOD_1, NOD_2, ..., NOD_{NODE} NODE Number of nodes which are not movable NOD_i Node number
		3.3.5.2 NOEL NEL_1, NEL_2, ..., NEL_{NOEL} NOEL Number of elements whose nodal coordinates are not movable NEL_i Element number
		3.3.5.3 IBOUND IBOUND = 0 Do not apply = 1 Nodal coordinates outside of rectangle are not movable Required only for IBOUND = 1 X_{LEFTt}, X_{RIGHT}, Y_{BOTTOM}, Y_{TOP} X_{LEFTt}, X_{RIGHT}, Y_{BOTTOM}, Y_{TOP} Coordinates of rectangle
		3.3.5.4 NGROUP, IGTITL X_{REF}, Y_{REF} NGROUP Number of curve groups. X_{REF}, Y_{REF} Coordinates of reference point IGTITL = 0 Do not specify = 1 Specify group title

Card Group			Input Data and Definitions
Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.1 GTITL (For IGTITL= 1) MTYPE, IGPOST, OVERLAY, GCOLOR, GLTYPE, GLTHIC, GHIDE GTITL Group title MTYPE = 1 Generate lines & remove within closed loop = -1 Remove elements outside closed loop = 2 Generate lines = -2 Generate slip lines with joint elements = 3 Assign new material number within the closed loop = -3 Assign new material number within the closed loop and generate slip lines with joint elements along the loop. MTYPE = 4 and -4 are the same as MTYPE=3 and -3, respectively, except that old material zone is not removed for MTYPE = 4 and -4. To make the group null, use MTYPE = 0. IGPOST Generate Post file for element activity (1) OVERLAY Overlaid over existing group mesh (1) GCOLOR Group color index number GLTYPE Group line type index number GLTHIC Group line thickness index number GHIDE Group hide (1)

Card Group			Input Data and Definitions
Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.1 For MTYPE = 1 or MTYPE = 2 LTP, LMAT For MTYPE =-2 $MATNO_{JT}$, DD_{JT}, $THIC_{JT}$, LTP_{I}, $LMAT_{I}$, LTP_{O}, $LMAT_{O}$ For MTYPE = 3 MATNO, DD, LTP, LMAT For MTYPE =-3 MATNO, DD, $MATNO_{JT}$, DD_{JT}, $THIC_{JT}$, LTP_{I}, $LMAT_{I}$, LTP_{O}, $LMAT_{O}$ For MTYPE = 4 MATNO, DD, LTP, LMAT, MATold For MTYPE =-4 MATNO, DD, $MATNO_{JT}$, DD_{JT}, $THIC_{JT}$, LTP_{I}, $LMAT_{I}$, LTP_{O}, $LMAT_{O}$, MATold DD = KF (SMAP-2D) = DEN (SMAP-S2) = IDH (SMAP-T2) DD_{JT} = KF_{JT} (SMAP-2D) = DEN_{JT} (SMAP-S2) = IDH_{JT} (SMAP-T2) For MTYPE = 4 or -4 MATold takes initial value if MATNO < 0 MATold takes MATNO + 1 if MATold = 0

Card Group			Input Data and Definitions
Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.1 MATNO — Material No for continuum element MATold — Additional MATNO for MTYPE = 4 or -4 KF = 0 Material has fluid phase = 1 Material has no fluid phase DEN — Unit weight IDH — Heat generation ID $MATNO_{JT}$ — Material No for joint element KF_{JT} = 0 Joint has fluid phase = 1 Joint has no fluid phase DEN_{JT} — Unit weight for joint element IDH_{JT} — Heat generation ID for joint element $THIC_{JT}$ — Apparent thickness of joint element LTP = 0 Do not generate = 2 Generate beam element Heat pipe (IDFNP=LFUN), T2 = 3 Generate truss element Convection (IDFNC=LFUN, IDFNT=LFUN+1), T2 = 4 External heat flow (ID=0, IDF=LFUN), T2 = 5 Temperature boun. (ID=1, IDF=LFUN), T2 LMAT — Material No for line element LTP_I, $LMAT_I$ — Subscript i refers to inner face LTP_o, $LMAT_o$ — Subscript o refers to outer face Note: For negative value of LTP, line elements take nodes in opposite face of joint element For negative value of $THIC_{JT}$, joint elements are fully connected to the surrounding continuum elements (MTYPE = -2 or -3)

Card Group			Input Data and Definitions
Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.1 Required only for IGPOST= 1 NAC, NDAC (MATold) NAC, NDAC (MATNO) NAC, NDAC ($MATNO_{JT}$) NAC, NDAC (LMAT) NAC, NDAC ($LMAT_{I}$) NAC, NDAC ($LMAT_{o}$) NAC Active step number NDAC Deactive step number Required only for IGPOST= 1 CHKBOX (Mesh) CHKBOX (Principal Stress) CHKBOX (Deformed Shape) CHKBOX (Beam) CHKBOX (Truss) CHKBOX (Contour) CHKBOX (Reference Line) CHKBOX = 0 Do not plot = 1 Plot the checked item Note: IGPOST= 1 will generate main file Group.man for element activity and post file Group.pos for PLOT-2D

Card Group			Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group	3.3.5.4.2 NPOINT, MOVE, IREF, X_{LO}, Y_{LO} NPOINT — Number of points defining X and Y coordinates of segments. Point numbering is counter-clockwise MOVE = 0 — Generated coordinates are movable = 1 — Generated coordinates are not movable IREF = 0 — Do not apply = 1 — Local Origin (X_{LO}, Y_{LO}) is relative to Reference Point in Card 3.3.5.4 X_{LO}, Y_{LO} — Coordinates of Local Origin NPOINT Cards: NP_1, X_1, Y_1 NP_2, X_2, Y_2 - - - - - - NP — Point number X — X-coordinate Y — Y-coordinate

Card Group					Input Data and Definitions
3	Modifying Existing Mesh (IMOD = 1)	Build User-Defined Curves and Material Zones (IEDIT = 4)	For Each Curve Group		3.3.5.4.3 NSEGMENT, GX, GY NSEGMENT — Number of segments If NSEGMENT is equal to NPOINT, the generated curve is closed loop. If NSEGMENT is less than NPOINT, the generated curve is open. GX, GY — Group No coordinates used in AIG
				For Each Segment	3.3.5.4.3.1 SEGNO, LTYPE, NDIV, IEND SEGNO — Segment No in sequential order LTYPE = 1 Straight line = 2 Elliptical line NDIV — Number of divisions. Use NIDV=0 for default divisions. Use negative value to consider intermediate points as line path only. IEND = 0 Include beginning and ending points but do not register contact information =-1 Include beginning point = 1 Include ending point = 2 Same as IEND=0 but register and split =-2 Same as IEND=2 but do not split = 3 This segment is only for reference line For LTYPE = 2 X_O, Y_O, R_X, R_Y, θ_b, θ_e X_O, Y_O — Arc Origin relative to (X_{LO},Y_{LO}) R_X, R_Y — Radius in X and Y axis, respectively θ_b, θ_e — Beginning and ending angle (˚) See Figure 8.2

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	3.6 Change Element Index Order (IEDIT = 6)	3.6.1 NumMATC MAT, I_1, I_2, I_3, I_4, MATC, KS, KF (SMAP-2D) MAT, I_1, I_2, I_3, I_4, MATC, THIC, DEN (SMAP-S2) MAT, I_1, I_2, I_3, I_4, MATC, IDH (SMAP-T2) NumMATC — Number of continuum materials MAT — Material number I_1, I_2, I_3, I_4 — Element corner index numbers MATC — New material property number KS, KF, THIC, DEN. IDH — Refer to Mesh File user manual
		3.6.2 NumSECB SEC, I, J, MSEC, K NumSECB — Number of beam sections SEC — Section number I, J — Element corner index numbers MSEC — New material section number K — New reference node number
		3.6.3 NumMATT MAT, I, J, MATT, K NumMATT — Number of truss materials MAT — Material number I, J — Element corner index numbers MATT — New material property number K — New reference node number Note: Index numbers are required as input. To keep the existing value, set it to -10.

Card Group	Input Data and Definitions
4 Generate Base Mesh and then Modify (IMOD = 2) See Figure 8.1	4.1 NBX, NBY, IB_LEFT, IB_RIGHT, IB_TOP, IB_BOTTOM NBX Number of blocks in X direction NBY Number of blocks in Y direction IB = 0 Free boundary = 1 Roller boundary
	4.2 X_O, Y_O, Y_{WT} X_O, Y_O Origin of X and Y coordinates Y_{WT} Y coordinate of water table (SMAP-2D) Initial temperature (SMAP-T2)
	4.3 NBX Cards [W_1, ΔX_1, α_{X1} W_2, ΔX_2, α_{X2} - - - - - - W_i Horizontal length of block Δx_i Minimum horizontal element length α_{Xi} = 0.5 Element length is constant = 0.3 Element length is growing from left to right =-0.3 Element length is growing from right to left
	4.4 NBY Cards [H_1, ΔY_1, α_{Y1} H_2, ΔY_2, α_{Y2} - - - - - - H_i Vertical length of block ΔY_i Minimum vertical element length α_Y = 0.5 Element length is constant = 0.3 Element length is growing from top to bottom =-0.3 Element length is growing from bottom to top
	4.5 IGMOD IGMOD = 0 Do not modify = 1 Modify generated base mesh If IGMOD = 1, go to Card 3.1

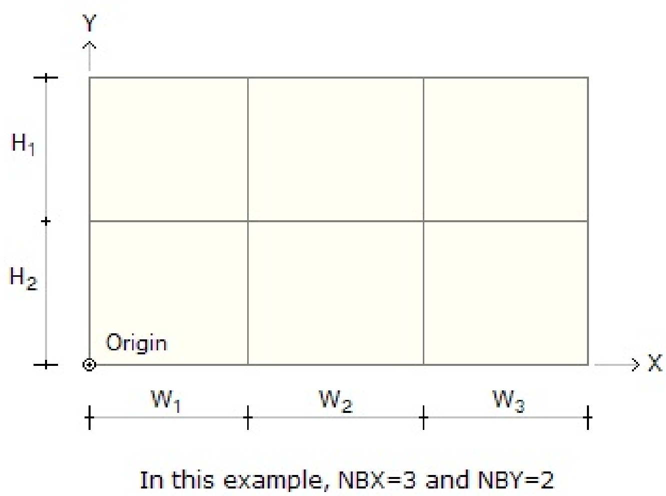


Figure 8.1 Layout of Base Mesh

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

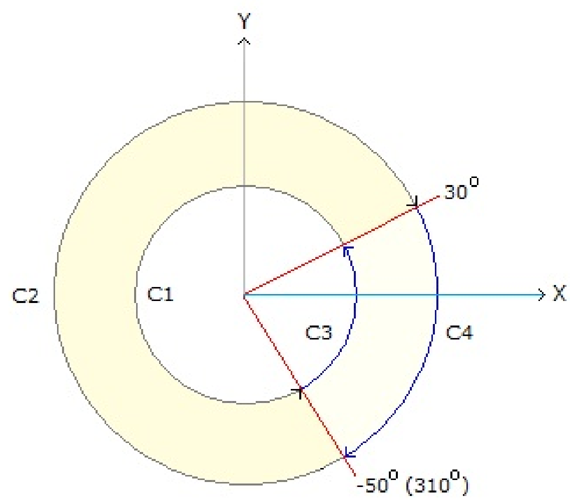


Figure 8.2 Examples of arc specification

ADDRGN-3D

User's Manual

Card Group	Input Data and Definitions
1 IMOD Type	1.1 IMOD IMOD = 0 Add Region B to Region A = 1 Modify existing mesh =-1 Same as IMOD = 0 except it uses DOF of Region B mesh along the interface
Adding Region B to Region A (IMOD = 0)	2.1 FILEA FILEB FILEC FILEA Input file name containing Region A mesh FILEB Input file name containing Region B mesh FILEC Output file name to store Combined Region mesh Note: When combining Region B mesh to Region A mesh, only Region B node numbers are changed. Element numbers for Region A and Region B should be continuous, otherwise element numbers are reordered automatically by program.

Card Group	Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	3.1 FILEA FILEM FILEA Input file name containing existing mesh FILEM Output file name to store modified mesh
	3.2 NSNEL, NSNODE, NBNEL, NTNEL NSNEL New starting continuum element number NSNODE New starting node number NBNEL New starting beam element number NTNEL New starting truss element number Note: NBNEL & NTNEL are used for IEDIT = 0, 1, 6
	3.3 IEDIT, MC_1, MC_2, MC_3, MB, MT IEDIT = 0 Change coordinates = 1 Change boundary codes = 2 Cut elements = 3 Change material numbers = -2 Cut elements in continuum blocks = -3 Change material numbers so as to match those in continuum blocks = 5 Add two layers of shell elements with joint elements in-between = 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 MC, MB, and MT are applicable for IEDIT = 2, 3, -2, and -3
	3.3.1 Required only for IEDIT = -2 or IEDIT = -3 FILEB FILEB Input file name containing continuum block mesh

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Changing Coordinates (IEDIT = 0)	3.3.1.1 X_o, Y_o, Z_o, X_{oNew}, Y_{oNew}, Z_{oNew} X_o, Y_o, Z_o — Reference origin X_{oNew}, Y_{oNew}, Z_{oNew} — New origin
		3.3.1.1 X_{scale}, Y_{scale}, Z_{scale} X_{scale}, Y_{scale}, Z_{scale} — Scale factors for X,Y, and Z coordinates. Note: New coordinates $X_{(new)}$, $Y_{(new)}$, $Z_{(new)}$ are computed as follows: $X_{(new)} = X_{oNew} + (X - X_0)\ X_{scale}$ $Y_{(new)} = Y_{oNew} + (Y - Y_0)\ Y_{scale}$ $Z_{(new)} = Z_{oNew} + (Z - Z_0)\ Z_{scale}$

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Changing Boundary Codes (IEDIT = 1)	3.3.2.1 IRANGE Range specified by IRANGE = 0 Coordinates = 1 Node numbers = 2 Polygon = 3 Plane = 4 Line strip = 5 Material numbers
		3.3.2.2.1 Required only for IRANGE = 0 X_{start}, Y_{start}, Z_{start}, X_{end}, Y_{end}, Z_{end} X_{start}, Y_{start}, Z_{start} Coordinates for lower left boundary X_{end}, Y_{end}, Z_{end} Coordinates for upper right boundary
		3.3.2.2.2 Required only for IRANGE = 1, 2, 3, 4, 5 NODE NOD_1, NOD_2, ..., NOD_{NODE} NODE Number of nodes/materials to be specified NOD_i Node/Material number (See Note 1) Polygon (IRANGE = 2) is defined counterclockwise Plane (IRANGE = 3) is defined by 3 nodes For IRANGE = 5, Nodes refer to Material numbers. 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)	Changing Boundary Codes (IEDIT = 1)	3.3.2.3 INSIDE (Not applicable for IRANGE= 5) INSIDE = 0 Apply inside of range = 1 Apply outside of range
		3.3.2.4 ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ (SMAP-3D) ID, IDF (SMAP-T3) ISX, ISY, ISZ X, Y, Z DOF for skeleton motion IFX, IFY, IFZ X, Y, Z DOF for relative fluid motion IRX, IRY, IRZ X, Y, Z DOF for rotation ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ = 0 Free to move in specified direction = 1 Fixed in specified direction ID = 0 External heat flow is specified = 1 Temperature is specified IDF Identification number for time dependent function If IDF = 0, external heat flow is zero at all times

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	Cutting Elements (IEDIT = 2)	3.3.3.1 IRANGE Range specified by IRANGE = 0 Coordinates = 1 Eement numbers
		3.3.3.2.1 Required only for IRANGE = 0 X_{start}, Y_{start}, $Z_{start,}$ X_{end}, Y_{end}, Z_{end} X_{start}, Y_{start}, Z_{start} Coordinates for lower left boundary X_{end}, Y_{end}, Z_{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 in page 8-24)
		3.3.3.3 INSIDE INSIDE = 0 Apply inside of range = 1 Apply outside of range

Card Group		Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	3.3.4 Change Material Numbers (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}, Z_{start}, X_{end}, Y_{end}, Z_{end} X_{start}, Y_{start}, Z_{start} Coordinates for lower left boundary X_{end}, Y_{end}, Z_{end} Coordinates 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 8-24)
		3.3.4.3 INSIDE INSIDE = 0 Apply inside of range = 1 Apply outside of range
		3.3.4.4 MATC, MATB, MATT New material number for MATC Continuum element MATB Beam element MATT Truss element Note: When new material number is zero, keep the old material number.

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

Modifying Existing Mesh (IMOD = 1)</td><td rowspan="3">3.5

Add Two Layers of Shell with Joint in-between (IEDIT = 5)</td><td colspan="2">3.5.1

$MATS_1$, MATJ, $MATS_2$, THICJ

$MATS_1$ — 1ST layer shell material number
MATJ — Joint material number
$MATS_2$ — 2nd layer shell material number
THICJ — Apparent thickness of joint element

Note: If the value of THICJ is negative,
joint elements are generated inward</td></tr>
<tr><td colspan="2">3.5.2

NSECTION, NUMNODE

NSECTION — Number of sections (Max=200)
NUMNODE — Number of nodes per section (Max=200)</td></tr>
<tr><td>For Each Section</td><td>3.5.3

NOD_1, NOD_2, . . . , $NOD_{NUMNODE}$

NOD_i — Node number

Note: List node numbers in counter clockwise
If NOD_1=$NOD_{NUMNODE}$, the loop is closed</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, I_5, I_6, I_7, I_8, MATC, KS, KF (SMAP-3D) MAT, I_1, I_2, I_3, I_4, I_5, I_6, I_7, I_8, MATC, IDH (SMAP-T3) NumMATC — Number of continuum materials MAT — Material number I_1,I_2,I_3,I_4,I_5,I_6,I_7,I_8 — Element corner index numbers MATC — New material property number KS, KF, IDH — Refer to Mesh File user manual
		3.6.2 NumSECB SEC, I, J, K, MSEC NumSECB — Number of beam sections SEC — Section number I, J — Element corner index numbers K — New reference node number MSEC — New material section 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.

Supplement Program

9.1 Introduction

Supplement programs contain supporting programs which are useful to prepare input data for pre-and main-processing programs and can be accessed through Run → Mesh Generater → Supplement menu.
Currently, there are five programs available:
EDIT, XY, CARDS, SHRINK FILE and CUDSS.

EDIT is used to run text editor.

XY computes coordinates of mid points, cross points, or normal points.

CARDS generates Element Activity data in Card Group 8 in Section 4.4 Main File.

SHRINK FILE removes extra blank spaces before carriage return.
This will reduce the size of the file.

CUDSS simulates cyclic undrained direct simple shear test.

9.2 EDIT

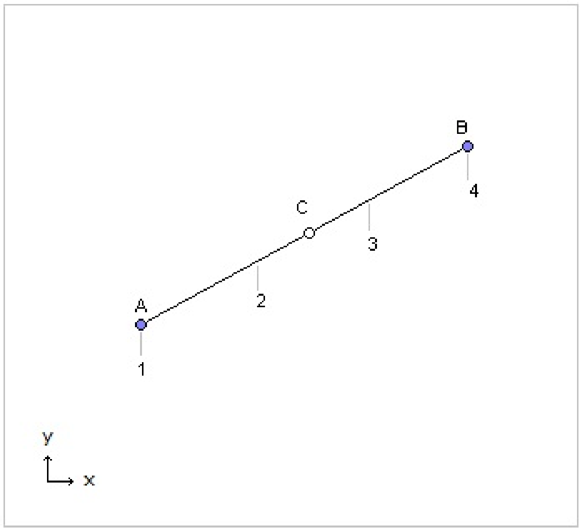
EDIT uses Windows text editor Wordpad to creat, modify, or list file.

9.3 XY

Program XY can be used to compute midpoints, intersection points and normal points of straight line and circular arc. The program is useful to construct the block diagrams of the problem geometry.

To run program XY, simply select XY from SUPPLEMENT Menu and follow instructions shown on the screen.

NF = 1 Compute Midpoint on Straight Line

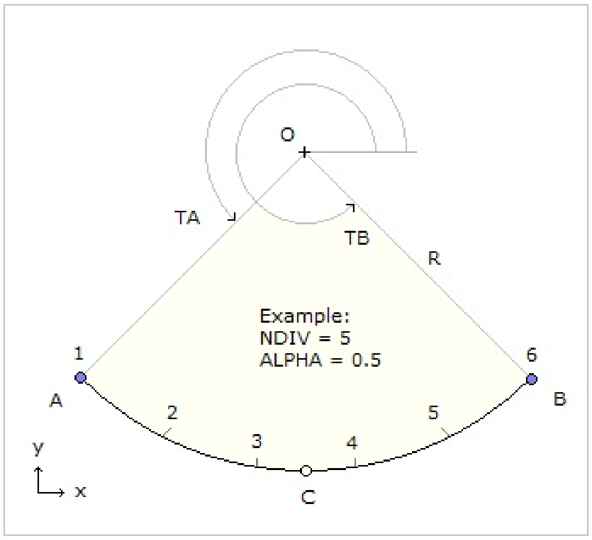


Example: NDIV = 3 and ALPHA = 0.5

INPUT:
XA, YA, XB, YB
NDIV, ALPHA

XA, YA = X and Y coordinates of A
XB, YB = X and Y coordinates of B
NDIV = Number of division
ALPHA = Geometric ratio

NF = 2 Compute Midpoint on Circular Arc



INPUT:

R, X_o, Y_o, TA, TB

NDIV, ALPHA

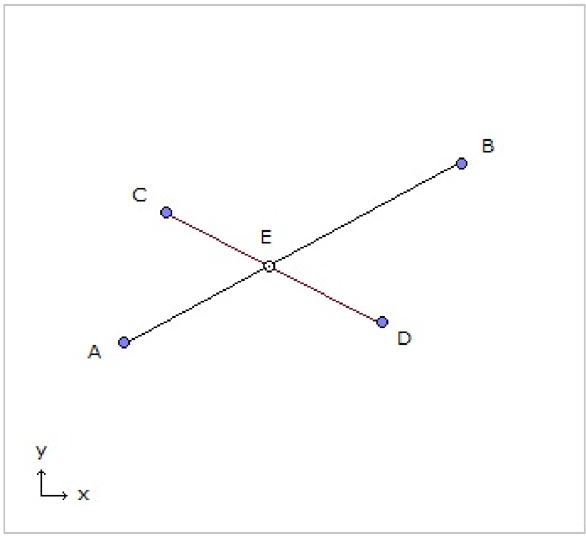
R	=	Radius
X_o, Y_o	=	X and Y coordinates of origin O
TA,TB	=	Angles (degrees) of A and B
NDIV	=	Number of division
ALPHA	=	Geometric ratio

If ALPHA = 0.5, midpoint C is located in half way between A and B

If ALPHA < 0.5, midpoint is close to A

If ALPHA > 0.5, midpoint is close to B

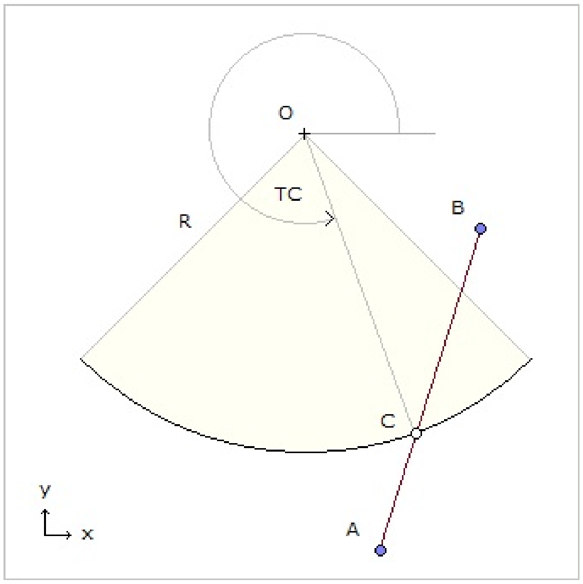
NF = 3 Compute Intersection Point of Two Straight Lines



INPUT:
XA, YA, XB, YB
XC, YC, XD, YD

XA, YA = X and Y coordinates of A
XB, YB = X and Y coordinates of B
XC, YC = X and Y coordinates of C
XD, YD = X and Y coordinates of D

NF = 4 Compute Intersection point of Arc & Straight Line



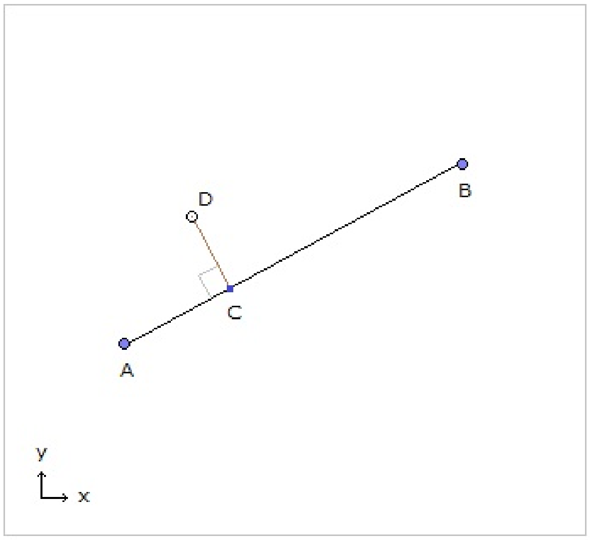
INPUT:

R, X_o, Y_o

XA, YA, XB, YB

R	=	Radius
X_o, Y_o	=	X and Y coordinates of origin O
XA, YA	=	X and Y coordinates of point A
XB, YB	=	X and Y coordinates of point B

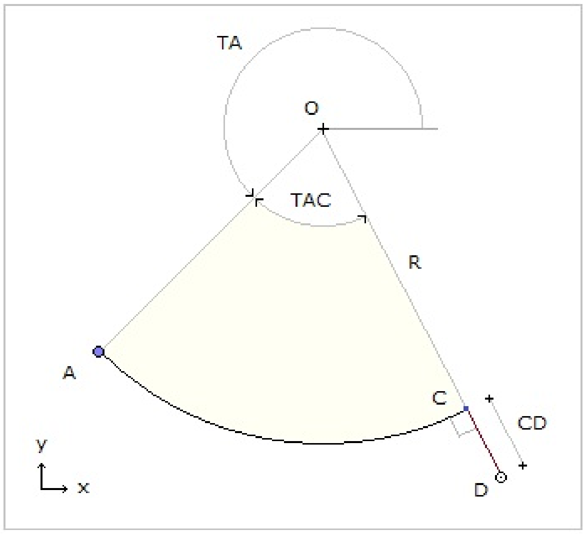
NF = 5 Compute Points Normal to Straight Line



INPUT:
XA, YA, XB, YB
AC, CD

XA, YA = X and Y coordinates of A
XB, YB = X and Y coordinates of B
AC = Distance between A and C
CD = Distance between C and D

NF = 6 Compute Points Normal to Circular Arc



INPUT:
R, X_o, Y_o, TA
TAC, CD

R	=	Radius
X_o, Y_o	=	X and Y coordinates of origin O
TA	=	Angle (degree) of A
TAC	=	Angle (degree) between A and C
CD	=	Distance between C and D

9.4 CARDS

Program CARDS is included to aid for users to prepare SMAP-3D input cards. Currently, there is only one routine which generates Element Activity in Card Group 8 in Section 4.4 Main File.

You are asked to type in following input data to generate element activity and deactivity;

NEL (start)   NEL(end)   NAC   NDAC

where

NEL (start)	Starting element number
NEL (end)	Ending element number
NAC	Load step at which elements from NEL(start) to NEL(end) are activated.
NDAC	Load step at which elements from NEL(start) to NEL(end) are deactivated.

Generated element activity data will be written in the output file you specified.

9.5 SHRINK FILE

SHRINK FILE is included to remove extra blank spaces before carriage return. This will reduce the size of the file where blank spaces are existing before the carriage return.

9.6 CUDSS

CUDSS is to simulate cyclic undrained direct simple shear test. PM4Sand material model is used to represent skeleton behavior, developed by Boulanger, R. W. And ziotopoulou, k. (Version 3.1).

Input File CUDSS.inp for PM4Sand Material Model

Card Group	Cyclic Undrained Direct Simple Shear Simulation
PM4Sand Material Model	1.0 Title Title Title
	2.0 σ_{vo}' K_o α_s σ_{vo}' Initial effective vertical stress K_o Coefficient of earth pressure at rest α_s Initial static shear stress ratio : $\alpha_s = \tau_s / \sigma_{vo}'$ where τ_s is initial static shear stress
	3.0 CSR γ_{max} CSR Cyclic stress ratio : $CSR = \tau_p / \sigma_{vo}'$ where τ_p is cyclic peak shear stress γ_{max} Maximum cutoff shear strain
	4.0 NCYCLE $\Delta\gamma$ NCYCLE Maximum number of cycles $\Delta\gamma$ Shear strain increment (Default 1.0e-05)

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

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-3D Mesh File is presented in detail in Section 4.3.

Three-dimensional Mesh Files can also be created by IGES (Initial Graphics Exchange Specification) or FEMAP (Version 4.1 - 4.5, neutral format) program which is developed by EDS.

In this section, we will briefly discuss Mesh File conversion under Mesh Generater → File Conversion menu:

10.2 Conversion to SMAP-3D Mesh File

Following Mesh Files can be converted to SMAP-3D Mesh File format:

- Mesh Files generated for two-dimensional SMAP programs (SMAP-S2, SMAP-2D, and SMAP-T2)
- Mesh Files generated for three-dimensional SMAP program (SMAP-T3)
- IGES (Initial Graphics Exchange Specification)
- FEMAP (Version 4.1 - 4.5, neutral format)

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

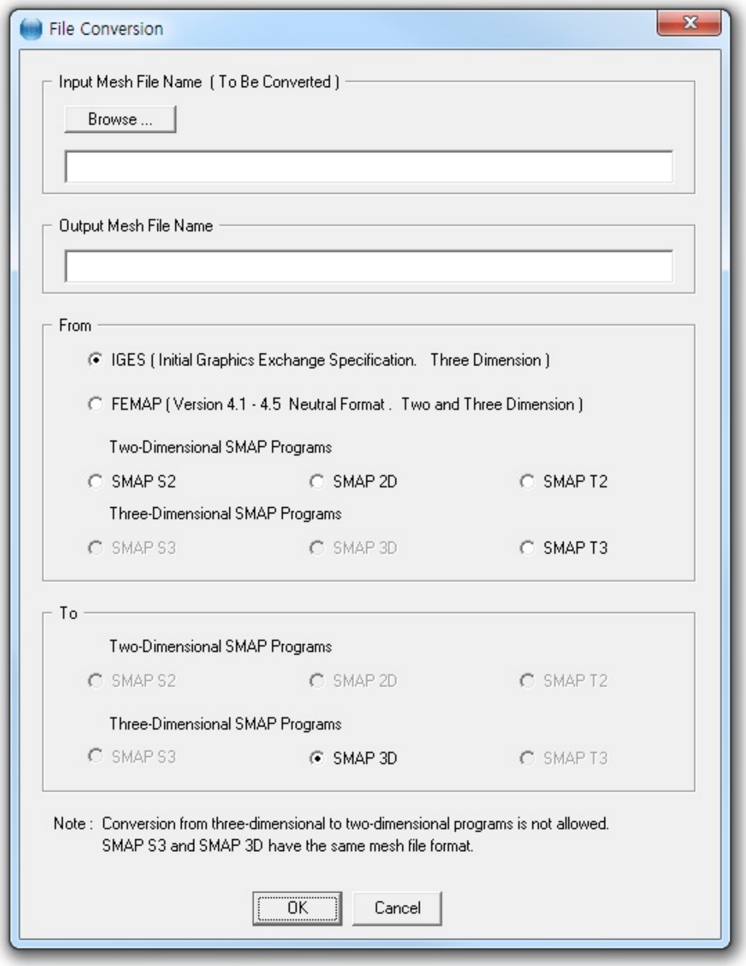


Figure 10.1 File Conversion dialog box

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
	2.2 For Each Loading Surface — LSTYPE = 0, 1, 2 — 2.2.2 NUMNODE NUMNODE — Number of nodes on this loading surface (Max = 9990)
	2.2 For Each Loading Surface — LSTYPE = 0, 1, 2 — 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)

Card Group 2 – Loading Surface; 2.2 – For Each Loading Surface

LSTYPE = 3 (Node Group)

2.2.4

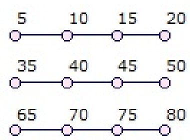
NUMNODG

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

LOAD-2D

LDTYPE = 2 [Velocity: SMAP-2D]

Card Group	Input Data and Definitions (Velocity)
1 Title & Element	1.1 TITLE TITLE — Any title (Max = 60 characters)
	1.2 NCTYPE NCTYPE = 0 Axisymmetric element Y-axis is axis of symmetry = 1 Plane strain element (Thickness=1.0) = 2 Plane stress element (Thickness=1.0) = 3 Spherically symmetric element (SMAP-2D)
2 Loading Surface	2.1 NUMLS NUMLS — Number of loading surfaces where velocities are specified (Max = 20)
	2.2 For Each Loading Surface 2.2.1 LSNO, LSTYPE LSNO — Loading surface number LSTYPE = 0 All specified nodes = 1 Line strip = 2 Points = 3 Node group = 4 Element group
	2.2 For Each Loading Surface — LSTYPE = 0, 1, 2 2.2.2 NUMNODE NUMNODE — Number of nodes on this loading surface (Max = 9990)
	2.2 For Each Loading Surface — LSTYPE = 0, 1, 2 2.2.3 NOD_1, NOD_2, ..., $NOD_{NUMNODE}$ NOD_i — Specified node Line strip (LSTYPE=1) is defined counterclockwise. For LSTYPE=1 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.

Card Group	Input Data and Definitions (Velocity)

2 Loading Surface — 2.2 For Each Loading Surface — LSTYPE = 3 (Node Group)

2.2.4

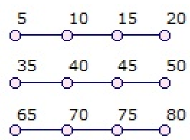
NUMNODG

NUMNODG — Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR — For Each Group

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



Example

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

LSTYPE = 4 (Element Group)

2.2.6

NUMNELG

NUMNELG — Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS — For Each Group

NSR	Starting element number of the first row
JCR	Element number increment in a row
NJR	Number of elements in a row
ICR	Element number increment for next row
NIR	Total number of rows
NS	Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

Example

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

Card Group		Input Data and Definitions (Velocity)
3 Velocity Function	3.1	NUMLV NUMLV Number of velocity functions (Max = 20)
	3.2 For Each Velocity Function	3.2.1 LVNO LVNO Velocity function number
		3.2.2 a_{xo}, a_{xx}, a_{xy} a_{xi} Coefficients defining velocity in x-direction $V_x = a_{xo} + a_{xx}\, x + a_{xy}\, y$
		3.2.3 a_{yo}, a_{yx}, a_{yy} a_{yi} Coefficients defining velocity in y-direction $V_y = a_{yo} + a_{yx}\, x + a_{yy}\, y$
		3.2.4 a_{no}, a_{nx}, a_{ny} a_{ni} Coefficients defining velocity normal to surface $V_n = a_{no} + a_{nx}\, x + a_{ny}\, y$

Card Group		Input Data and Definitions (Velocity)
4 Velocity History	4.1	NUMLH NUMLH Number of velocity histories (Max = 20)
	4.2 For Each Velocity History	4.2.1 LHNO LHNO Velocity history number
		4.2.2 NUMTP NUMTP Number of time points (Max = 1000)
		4.2.3 $T_1, T_2, \ldots, T_{NUMTP}$ T_i Specified time
		4.2.4 $C_1, C_2, \ldots, C_{NUMTP}$ C_i Velocity intensity at time T_i

Card Group	Input Data and Definitions (Velocity)
5 Velocity Specification	5.1 LSNO, LVNO, LHNO LSNO Loading surface number LVNO Velocity function number LHNO Velocity history number Repeat Card 5.1 until the last card (LSNO=0) is specified

LOAD-2D

LDTYPE = 3 [Initial Velocity: SMAP-2D]

Card Group	Input Data and Definitions (Initial Velocity)
1 Title & Element	1.1 TITLE TITLE Any title (Max = 60 characters)
	1.2 NCTYPE NCTYPE = 0 Axisymmetric element Y-axis is axis of symmetry = 1 Plane strain element (Thickness=1.0) = 2 Plane stress element (Thickness=1.0) = 3 Spherically symmetric element (SMAP-2D)
2 Loading Surface	2.1 NUMLS NUMLS Number of loading surfaces where initial velocities are specified (Max = 20)
	2.2 For Each Loading Surface 2.2.1 LSNO, LSTYPE LSNO Loading surface number LSTYPE = 0 All specified nodes = 1 Line strip = 2 Points = 3 Node group = 4 Element group
	LSTYPE = 0, 1, 2 2.2.2 NUMNODE NUMNODE Number of nodes on this loading surface (Max = 9990)
	2.2.3 NOD_1, NOD_2, ..., $NOD_{NUMNODE}$ NOD_i Specified node Line strip (LSTYPE=1) is defined counterclockwise. For LSTYPE=1 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.

Card Group	Input Data and Definitions (Initial Velocity)

2 — Loading Surface

2.2 — For Each Loading Surface

LSTYPE = 3 (Node Group)

2.2.4

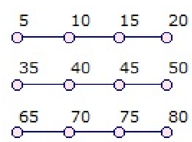
NUMNODG

NUMNODG — Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR — For Each Group

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



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

LSTYPE = 4 (Element Group)

2.2.6

NUMNELG

NUMNELG — Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS — For Each Group

NSR	Starting element number of the first row
JCR	Element number increment in a row
NJR	Number of elements in a row
ICR	Element number increment for next row
NIR	Total number of rows
NS	Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

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

Card Group		Input Data and Definitions (Initial Velocity)
3 Initial Velocity Function	3.1	NUMLIV NUMLIV Number of initial velocity functions (Max = 20)
	3.2 For Each Initial Velocity Function	3.2.1 LIVNO LIVNO Initial velocity function number
		3.2.2 a_{xo}, a_{xx}, a_{xy} a_{xi} Coefficients defining initial velocity in the x-direction $V_{ix} = a_{xo} + a_{xx}x + a_{xy}y$
		3.2.3 a_{yo}, a_{yx}, a_{yy} a_{yi} Coefficients defining initial velocity in the y-direction $V_{iy} = a_{yo} + a_{yx}x + a_{yy}y$
		3.2.4 a_{no}, a_{nx}, a_{ny} a_{ni} Coefficients defining initial velocity normal to the surface $V_{in} = a_{no} + a_{nx}x + a_{ny}y$

Card Group	Input Data and Definitions (Initial Velocity)
4 Initial Velocity Specification	4.1 LSNO, LIVNO LSNO Loading surface number LIVNO Initial velocity function Repeat Card 4.1 until the last card (LSNO=0) is specified

LOAD-2D

LDTYPE = 4 [Acceleration: SMAP-2D]

Card Group	Input Data and Definitions (Acceleration)
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 accelerations 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 (Acceleration)
2 Loading Surface	2.2 For Each Loading Surface	LSTYPE = 3 (Node Group)	2.2.4

2.2.4

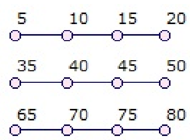
NUMNODG

NUMNODG — Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR   For Each Group

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



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

LSTYPE = 4 (Element Group)

2.2.6

NUMNELG

NUMNELG — Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS   For Each Group

- NSR — Starting element number of the first row
- JCR — Element number increment in a row
- NJR — Number of elements in a row
- ICR — Element number increment for next row
- NIR — Total number of rows
- NS — Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

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

Card Group		Input Data and Definitions (Acceleration)
3 Acceleration Function	3.1	NUMLA NUMLA — Number of acceleration functions (Max = 20)
	3.2 For Each Acceleration	3.2.1 LANO LANO — Acceleration function number
		3.2.2 a_{xo}, a_{xx}, a_{xy} a_{xi} — Coefficients defining acceleration in the x-direction $A_x = a_{xo} + a_{xx}x + a_{xy}y$
		3.2.3 a_{yo}, a_{yx}, a_{yy} a_{yi} — Coefficients defining acceleration in the y-direction $A_y = a_{yo} + a_{yx}x + a_{yy}y$
		3.2.4 a_{no}, a_{nx}, a_{ny} a_{ni} — Coefficients defining acceleration normal to the surface $A_n = a_{no} + a_{nx}x + a_{ny}y$

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Acceleration)</th></tr>
<tr><td rowspan="5">4
Acceleration History</td><td colspan="2">4.1
NUMLH
NUMLH Number of acceleration histories (Max = 20)</td></tr>
<tr><td rowspan="4">4.1
For Each Acceleration History</td><td>4.2.1
LHNO
LHNO Acceleration history number</td></tr>
<tr><td>4.2.2
NUMTP
NUMTP Number of time points (Max = 1000)</td></tr>
<tr><td>4.2.3
T_1, T_2, ..., T_{NUMTP}
T_i Specified time</td></tr>
<tr><td>4.2.4
C_1, C_2, ..., C_{NUMTP}
C_i Acceleration intensity at time T_i</td></tr>
</table>

Card Group	Input Data and Definitions (Acceleration)
5 Acceleration Specification	5.1 LSNO, LANO, LHNO LSNO Loading surface number LANO Acceleration function number LHNO Acceleration history number Repeat Card 5.1 until the last card (LSNO=0) is specified

LOAD-2D

LDTYPE = 5 [Transmitting Boundary: SMAP-2D]

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Transmitting Boundary)</th></tr>
<tr><td rowspan="2">1
Title & Element</td><td colspan="3">1.1
TITLE
TITLE — Any title (Max = 60 characters)</td></tr>
<tr><td colspan="3">1.2
NCTYPE
NCTYPE = 0 Axisymmetric element
Y-axis is axis of symmetry
= 1 Plane strain element (Thickness=1.0)
= 2 Plane stress element (Thickness=1.0)
= 3 Spherically symmetric element (SMAP-2D)</td></tr>
<tr><td rowspan="4">2
Loading Surface</td><td colspan="3">2.1
NUMLS
NUMLS — Number of loading surfaces where transmitting boundaries are specified (Max = 20)</td></tr>
<tr><td rowspan="3">2.2
For Each Loading Surface</td><td colspan="2">2.2.1
LSNO, LSTYPE
LSNO — Loading surface number
LSTYPE = 0 All specified nodes
= 1 Line strip = 2 Points
= 3 Node group = 4 Element group</td></tr>
<tr><td rowspan="2">LSTYPE = 0, 1, 2</td><td>2.2.2
NUMNODE
NUMNODE — Number of nodes on this loading surface (Max = 9990)</td></tr>
<tr><td>2.2.3
NOD_1, NOD_2, ..., $NOD_{NUMNODE}$
NOD_i — Specified node

Line strip (LSTYPE=1) is defined counterclockwise. For LSTYPE=1 and $NOD_{NUMNODE}$ < 0, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.</td></tr>
</table>

Card Group	Input Data and Definitions (Transmitting Boundary)

2 — Loading Surface

2.2 — For Each Loading Surface

LSTYPE = 3 (Node Group)

2.2.4

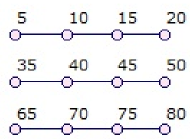
NUMNODG

- NUMNODG — Number of node groups on this loading surface (Max = 100)

2.2.5

NSR, JCR, NJR, ICR, NIR — For Each Group

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



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

2.2.6

NUMNELG

- NUMNELG — Number of element groups on this loading surface (Max = 100)

2.2.7

NSR, JCR, NJR, ICR, NIR, NS — For Each Group

- NSR — Starting element number of the first row
- JCR — Element number increment in a row
- NJR — Number of elements in a row
- ICR — Element number increment for next row
- NIR — Total number of rows
- NS — Element surface number (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

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

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Transmitting Boundary)</th></tr>
<tr><td rowspan="3">3
Material Property</td><td colspan="2">3.1
NUMMP

NUMMP Number of different material property (Max=5)</td></tr>
<tr><td rowspan="2">3.2
For Each Material Property</td><td>3.2.1
MATNO

MATNO Material property number</td></tr>
<tr><td>3.2.2
RO, E, V

RO Mass density
E Young's modulus
V Poisson's ratio</td></tr>
</table>

Card Group	Input Data and Definitions (Transmitting Boundary)
4 Transmitting Boundary Specification	4.1 LSNO, MATNO LSNO — Loading surface number MATNO — Material property number For MATNO = 0, loading surface is related to continuum element surface Refer to Card 9.6.3 in SMAP-2D User's Manual Repeat Card 4.1 until the last card (LSNO=0) is specified

LOAD-3D

LDTYPE = 1 [Pressure: SMAP-3D]

Card Group	Input Data and Definitions (Pressure)
1 Title	1.1 TITLE TITLE Any title (Max = 60 characters)
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 Polygon = 2 Plane = 3 Line strip = 4 Points = 5 Node group = 6 Element group
	LSTYPE = 0, 1, 2, 3, 4 — 2.2.2 NUMNODE NUMNODE Number of nodes on this loading surface (Max = 9990)
	LSTYPE = 0, 1, 2, 3, 4 — 2.2.3 NOD_1, NOD_2, ..., $NOD_{NUMNODE}$ NOD_i Specified node Polygon (LSTYPE=1) is defined counterclockwise. Plane (LSTYPE=2) is defined by 3 nodes. For LSTYPE=3 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 = 5 (Node Group)

2.2.4

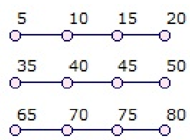
NUMNODG

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 = 6 (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_{xz} a_{xi} Coefficients defining surface traction in the x-direction. $P_x = a_{xo} + a_{xx}x + a_{xy}y + a_{xz}z$
		3.2.3 a_{yo}, a_{yx}, a_{yy}, a_{yz} a_{yi} Coefficients defining surface traction in the y-direction. $P_y = a_{yo} + a_{yx}x + a_{yy}y + a_{yz}z$
		3.2.4 a_{zo}, a_{zx}, a_{zy}, a_{zz} a_{zi} Coefficients defining surface traction in the z-direction. $P_z = a_{zo} + a_{zx}x + a_{zy}y + a_{zz}z$
		3.2.5 a_{no}, a_{nx}, a_{ny}, a_{nz} a_{ni} Coefficients defining surface traction normal to surface. Acting on actual surface $P_n = a_{no} + a_{nx}x + a_{ny}y + a_{nz}z$

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 $T_1, T_2, \ldots, T_{NUMTP}$ T_i Specified time
		4.2.4 $C_1, C_2, \ldots, C_{NUMTP}$ C_i Pressure intensity at time T

Card Group	Input Data and Definitions (Pressure)
5 Pressure Specification	5.1 LSNO, LPNO, LHNO LSNO Loading surface number LPNO Pressure function number LHNO Pressure history number Repeat Card 5.1 until the last card (LSNO=0) is specified

LOAD-3D

LDTYPE = 2 [Velocity: SMAP-3D]

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Velocity)</th></tr>
<tr><td>1
Title</td><td colspan="3">1.1
TITLE
TITLE Any title (Max = 60 characters)</td></tr>
<tr><td rowspan="4">2
Loading Surface</td><td colspan="3">2.1
NUMLS
NUMLS Number of loading surfaces where velocities are specified (Max = 20)</td></tr>
<tr><td rowspan="3">2.2
For Each Loading Surface</td><td colspan="2">2.2.1
LSNO, LSTYPE
LSNO Loading surface number
LSTYPE = 0 All specified nodes
= 1 Polygon = 2 Plane
= 3 Line strip = 4 Points
= 5 Node group
= 6 Element group</td></tr>
<tr><td rowspan="2">LSTYPE = 0, 1, 2, 3, 4</td><td>2.2.2
NUMNODE
NUMNODE Number of nodes on this loading surface (Max = 9990)</td></tr>
<tr><td>2.2.3
NOD_1, NOD_2, ..., $NOD_{NUMNODE}$
NOD_i Specified node

Polygon (LSTYPE=1) is defined counterclockwise.
Plane (LSTYPE=2) is defined by 3 nodes.
For LSTYPE=3 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.</td></tr>
</table>

Card Group	Input Data and Definitions (Velocity)

2 — Loading Surface

2.2 — For Each Loading Surface

LSTYPE = 5 (Node Group)

2.2.4

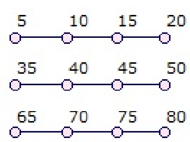
NUMNODG

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 = 6 (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 no. (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

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

Card Group		Input Data and Definitions (Velocity)
3 Velocity Function	3.1	NUMLV NUMLV Number of velocity functions (Max = 20)
	3.2 For Each Velocity Function	3.2.1 LVNO, LVTYPE LVNO Velocity function number LVTYPE = 0 Apply individual components Cards 3.2.2 - 3.2.4 = 1 Apply normal components Cards 3.2.5
		3.2.2 a_{xo}, a_{xx}, a_{xy}, a_{xz} a_{xi} Coefficients defining velocity in x-direction $V_x = a_{xo} + a_{xx}x + a_{xy}y + a_{xz}z$
		3.2.3 a_{yo}, a_{yx}, a_{yy}, a_{yz} a_{yi} Coefficients defining velocity in y-direction $V_y = a_{yo} + a_{yx}x + a_{yy}y + a_{yz}z$
		3.2.4 a_{zo}, a_{zx}, a_{zy}, a_{zz} a_{zi} Coefficients defining velocity in z-direction $V_z = a_{zo} + a_{zx}x + a_{zy}y + a_{zz}z$
		3.2.5 a_{no}, a_{nx}, a_{ny}, a_{nz} a_{ni} Coefficients defining velocity normal to surface $V_n = a_{no} + a_{nx}x + a_{ny}y + a_{nz}z$

Card Group	Input Data and Definitions (Velocity)	
4 Velocity History	4.1 NUMLH NUMLH Number of velocity histories (Max = 20)	
	4.2 For Each Velocity History	4.2.1 LHNO LHNO Velocity history number
		4.2.2 NUMTP NUMTP Number of time points (Max = 1000)
		4.2.3 $T_1, T_2, \ldots, T_{NUMTP}$ T_i Specified time
		4.2.4 $C_1, C_2, \ldots, C_{NUMTP}$ C_i Velocity intensity at time T_i

Card Group	Input Data and Definitions (Velocity)
5 Velocity Specification	5.1 LSNO, LVNO, LHNO LSNO Loading surface number LVNO Velocity function number LHNO Velocity history number Repeat Card 5.1 until the last card (LSNO=0) is specified

LOAD-3D

LDTYPE = 3 [Initial Velocity: SMAP-3D]

Card Group				Input Data and Definitions (Initial Velocity)
1 Title	1.1			TITLE TITLE Any title (Max = 60 characters)
2 Loading Surface	2.1			NUMLS NUMLS Number of loading surfaces where initial velocities are specified (Max = 20)
	2.2 For Each Loading Surface	2.2.1		LSNO, LSTYPE LSNO Loading surface number LSTYPE = 0 All specified nodes = 1 Polygon = 2 Plane = 3 Line strip = 4 Points = 5 Node group = 6 Element group
		LSTYPE = 0, 1, 2, 3, 4	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 Polygon (LSTYPE=1) is defined counterclockwise. Plane (LSTYPE=2) is defined by 3 nodes. For LSTYPE=3 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 (Initial Velocity)

2 Loading Surface

2.2 For Each Loading Surface

LSTYPE = 5 (Node Group)

2.2.4

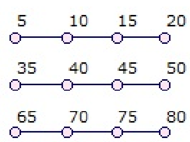
NUMNODG

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 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 = 6 (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 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 no. (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 (Initial Velocity)	
3	3.1 NUMLIV NUMLIV Number of initial velocity functions (Max = 20)	
Initial Velocity Function	3.2 For Each Initial Velocity Function	3.2.1 LIVNO, LIVTYPE LIVNO Initial velocity function number LIVTYPE = 0 Apply individual components Cards 3.2.2 - 3.2.4 = 1 Apply normal components Cards 3.2.5
		3.2.2 a_{xo}, a_{xx}, a_{xy}, a_{xz} a_{xi} Coefficients defining initial velocity in the x-direction. $V_{ix} = a_{xo} + a_{xx}x + a_{xy}y + a_{xz}z$
		3.2.3 a_{yo}, a_{yx}, a_{yy}, a_{yz} a_{yi} Coefficients defining initial velocity in the y-direction. $V_{iy} = a_{yo} + a_{yx}x + a_{yy}y + a_{yz}z$
		3.2.4 a_{zo}, a_{zx}, a_{zy}, a_{zz} a_{zi} Coefficients defining initial velocity in the z-direction. $V_{iz} = a_{zo} + a_{zx}x + a_{zy}y + a_{zz}z$
		3.2.5 a_{no}, a_{nx}, a_{ny}, a_{nz} a_{ni} Coefficients defining initial velocity normal to the surface. $V_{in} = a_{no} + a_{nx}x + a_{ny}y + a_{nz}z$

Card Group	Input Data and Definitions (Initial Velocity)
4 Initial Velocity Specification	4.1 LSNO, LIVNO LSNO Loading surface number LIVNO Initial velocity function Repeat Card 4.1 until the last card (LSNO=0) is specified

LOAD-3D

LDTYPE = 4 [Acceleration: SMAP-3D]

<table>
<tr><th>Card Group</th><th colspan="3">Input Data and Definitions (Acceleration)</th></tr>
<tr><td>1
Title</td><td colspan="3">1.1
TITLE
TITLE Any title (Max = 60 characters)</td></tr>
<tr><td rowspan="4">2
Loading Surface</td><td colspan="3">2.1
NUMLS
NUMLS Number of loading surfaces where accelerations are specified (Max = 20)</td></tr>
<tr><td rowspan="3">2.2
For Each Loading Surface</td><td colspan="2">2.2.1
LSNO, LSTYPE
LSNO Loading surface number
LSTYPE = 0 All specified nodes
= 1 Polygon = 2 Plane
= 3 Line strip = 4 Points
= 5 Node group
= 6 Element group</td></tr>
<tr><td rowspan="2">LSTYPE = 0, 1, 2, 3, 4</td><td>2.2.2
NUMNODE
NUMNODE Number of nodes on this loading surface (Max = 9990)</td></tr>
<tr><td>2.2.3
NOD_1, NOD_2, ..., $NOD_{NUMNODE}$
NOD_i Specified node

Polygon (LSTYPE=1) is defined counterclockwise.
Plane (LSTYPE=2) is defined by 3 nodes.
For LSTYPE=3 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.</td></tr>
</table>

Card Group	Input Data and Definitions (Acceleration)

2 — Loading Surface

2.2 — For Each Loading Surface

LSTYPE = 5 (Node Group)

2.2.4

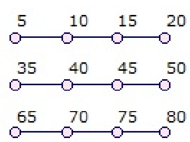
NUMNODG

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 = 6 (Element Group)

2.2.6

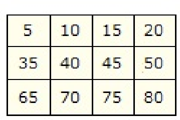
NUMNELG

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 no. (See Mesh File Card 3.2)



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

Card Group	Input Data and Definitions (Acceleration)	
3 Acceleration Function	3.1 NUMLA NUMLA Number of acceleration functions (Max = 20)	
	3.2 For Each Acceleration Function	3.2.1 LANO, LATYPE LANO Acceleration function number LATYPE = 0 Apply individual components (Cards 3.2.2 - 3.2.4) = 1 Apply normal components (Cards 3.2.5)
		3.2.2 a_{xo}, a_{xx}, a_{xy}, a_{xz} a_{xi} Coefficients defining acceleration in the x-direction $A_x = a_{xo} + a_{xx}x + a_{xy}y + a_{xz}z$
		3.2.3 a_{yo}, a_{yx}, a_{yy}, a_{yz} a_{yi} Coefficients defining acceleration in the y-direction. $A_y = a_{yo} + a_{yx}x + a_{yy}y + a_{yz}z$
		3.2.4 a_{zo}, a_{zx}, a_{zy}, a_{zz} a_{zi} Coefficients defining acceleration In the z-direction. $A_z = a_{zo} + a_{zx}x + a_{zy}y + a_{zz}z$
		3.2.5 a_{no}, a_{nx}, a_{ny}, a_{nz} a_{ni} Coefficients defining acceleration normal to the surface. $A_n = a_{no} + a_{nx}x + a_{ny}y + a_{nz}z$

Card Group	Input Data and Definitions (Acceleration)		
4 Acceleration History	4.1 NUMLH NUMLH Number of acceleration histories (Max = 20)		
	4.1 For Each Acceleration History	4.2.1 LHNO LHNO Acceleration history number	
		4.2.2 NUMTP NUMTP Number of time points (Max = 1000)	
		4.2.3 $T_1, T_2, \ldots, T_{NUMTP}$ T_i Specified time	
		4.2.4 $C_1, C_2, \ldots, C_{NUMTP}$ C_i Acceleration intensity at time T_i	

Card Group	Input Data and Definitions (Acceleration)
5 Acceleration Specification	5.1 LSNO, LANO, LHNO LSNO Loading surface number LANO Acceleration function number LHNO Acceleration history number Repeat Card 5.1 until the last card (LSNO=0) is specified

LOAD-3D

LDTYPE = 5 [Transmitting Boundary: SMAP-3D]

Card Group	Input Data and Definitions (Transmitting Boundary)
1 Title	1.1 TITLE TITLE Any title (Max = 60 characters)
2 Loading Surface	2.1 NUMLS NUMLS Number of loading surfaces where transmitting boundaries are specified (Max = 20)
	2.2 For Each Loading Surface — 2.2.1 LSNO, LSTYPE LSNO Loading surface number LSTYPE = 0 All specified nodes = 1 Polygon = 2 Plane = 3 Line strip = 4 Points = 5 Node group = 6 Element group
	LSTYPE = 0, 1, 2, 3, 4 — 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 Polygon (LSTYPE=1) is defined counterclockwise. Plane (LSTYPE=2) is defined by 3 nodes. For LSTYPE=3 and $NOD_{NUMNODE} < 0$, absolute value of $NOD_{NUMNODE}$ is the reference node defining normal to the Line strip.

Card Group			Input Data and Definitions (Transmitting Boundary)
2 Loading Surface	2.2 For Each Loading Surface	LSTYPE = 5 (Node Group)	2.2.4

2.2.4

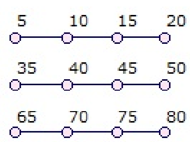
NUMNODG

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 = 6 (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 no. (See Mesh File Card 3.2)

5	10	15	20
35	40	45	50
65	70	75	80

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

<table>
<tr><th>Card Group</th><th colspan="2">Input Data and Definitions (Transmitting Boundary)</th></tr>
<tr><td rowspan="3">3
Material Property</td><td colspan="2">3.1
NUMMP

NUMMP Number of different material property (Max = 20)</td></tr>
<tr><td rowspan="2">3.2
For Each Material Property</td><td>3.2.1
MATNO

MATNO Material property number</td></tr>
<tr><td>3.2.2
RO, E, V

RO Mass density
E Young's modulus
V Poisson's ratio</td></tr>
</table>

Card Group	Input Data and Definitions (Transmitting Boundary)
4 Transmitting Boundary Specification	4.1 LSNO, MATNO LSNO Loading surface number MATNO Material property number Repeat Card 4.1 until the last card (LSNO=0) is specified

XY Graph
User's Manual

12.1 Introduction

XY Graph is a two-dimensional graph consisting of lines connecting each pair of data points, which can be plotted by PLOT XY or EXCEL. Figure 12.1 shows schematic flow diagram of plotting simple form of Draft XY data in Table 12.1.

This Draft XY is changed into Standard XY by Converter DS. Then Standard XY can be plotted by directly PLOT XY or by EXCEL with the aid of Converter SE.

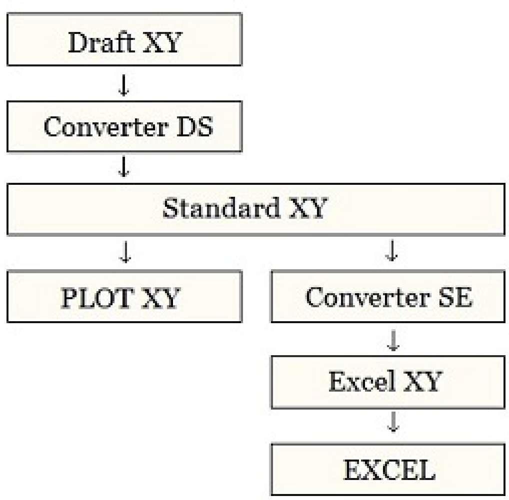


Figure 12.1 Flow diagram of plotting XY graph

Table 12.1 Draft XY Data Format

Card Group	Input Data and Definitions		
First Plot	Title	Title Sub Title X-Label Y-Label	(Max 50 Characters) (Max 50 Characters) (Max 50 Characters) (Max 50 Characters)
	First Curve	X_1 Y_1 X_2 Y_2 - - X_n Y_n 0.0 123456 Legend 1 Legend 2	 (End of Curve) (Max 20 Characters) (Max 20 Characters))
	Second Curve	X_1 Y_1 X_2 Y_2 - - X_n Y_n 0.0 123456 Legend 1 Legend 2	 (End of Curve) (Max 20 Characters) (Max 20 Characters)
	Last Curve	X_1 Y_1 X_2 Y_2 - - X_n Y_n 0.0 123456 Legend 1 Legend 2 0.0 987654	 (End of Curve) (Max 20 Characters) (Max 20 Characters) (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

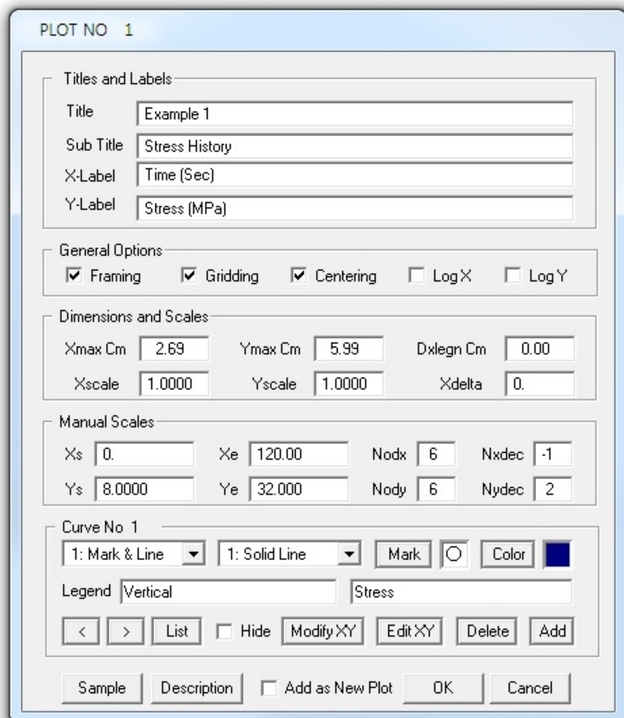
Edit Dialog in Figure 12.2 can be accessed by selecting the Edit menu in PLOT-XY.

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



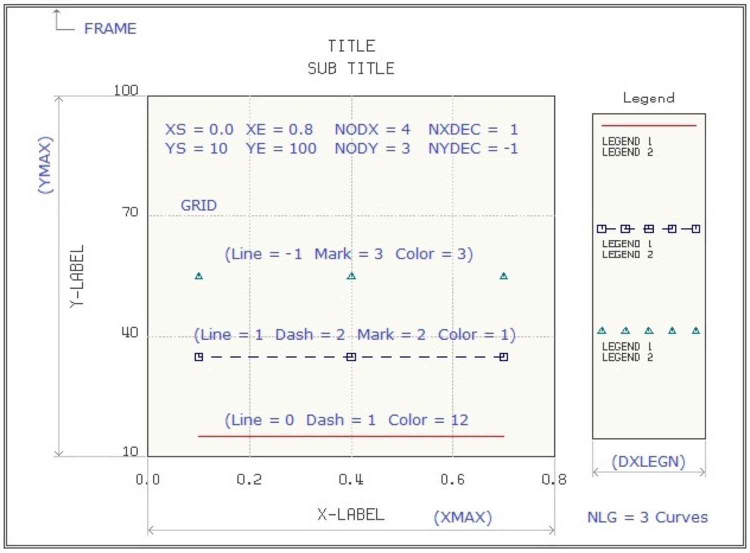


Figure 12.3 Sample graph

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

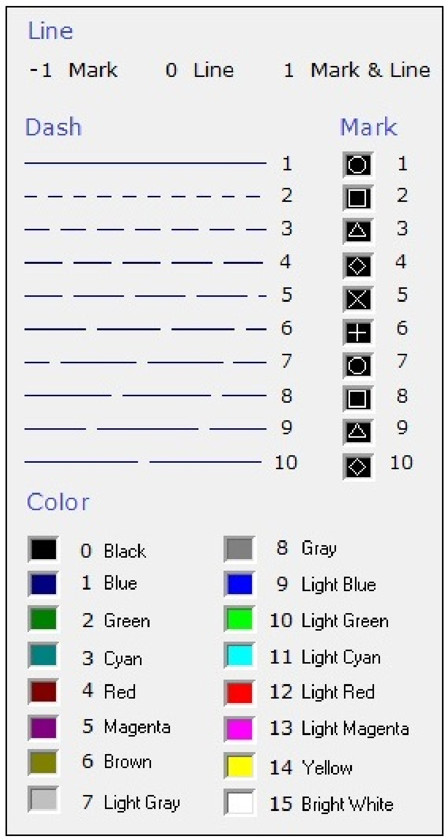
Refer to description in Sample Graph in Figure 12.3.

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

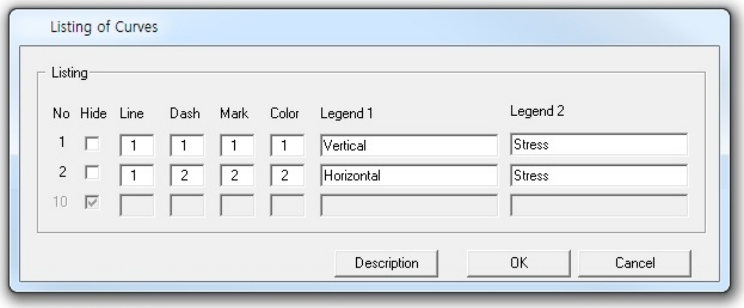


Figure 12.5a Listing of curves

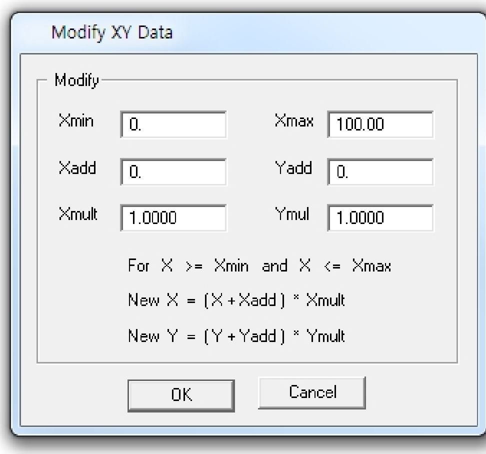


Figure 12.5b Modify current curve XY data

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.

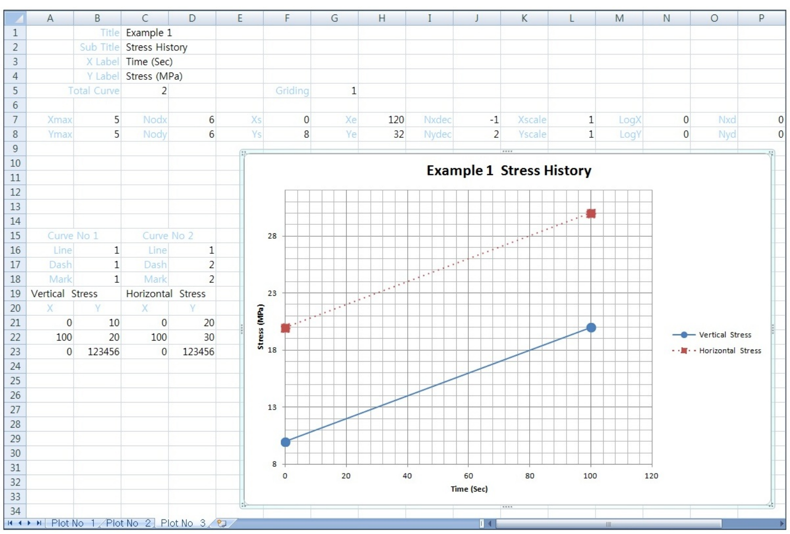


Figure 12.6 XY graph on Excel spread sheet

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 \Smap3D \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.

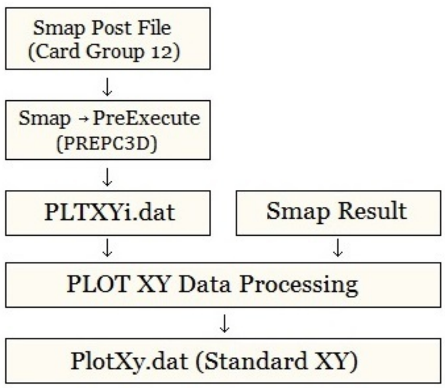


Figure 12.7 Processing SMAP results

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.

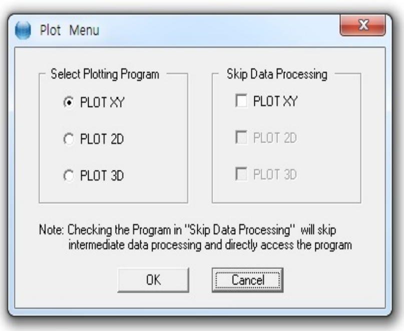


Figure 12.8
Plot menu dialog

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

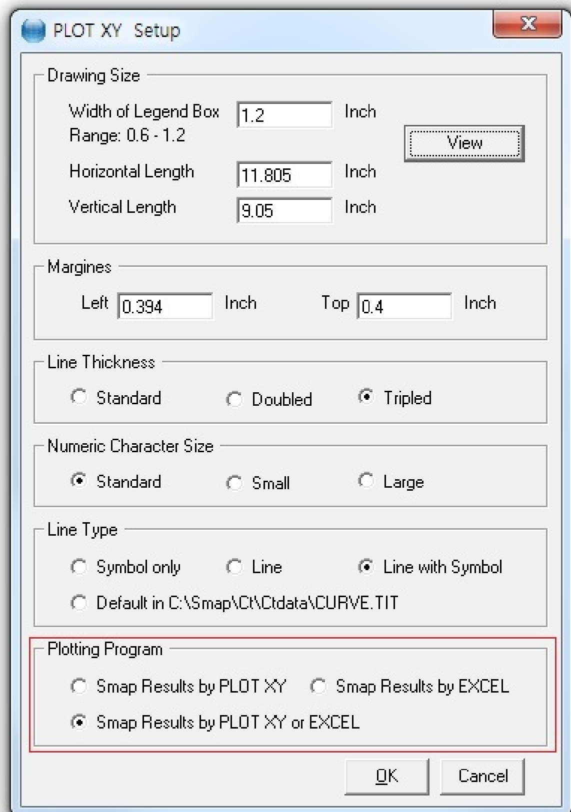


Figure 12.9 PLOT XY setup dialog

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.

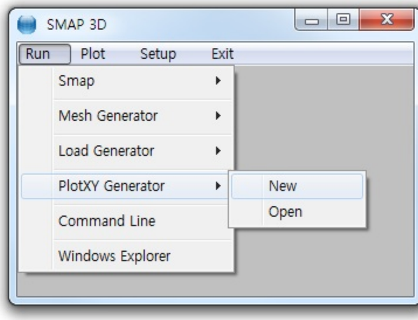


Figure 12.10
Menu for PlotXY Generator

Open is used to edit existing Post File. You can specify different output Post File name as shown in Figure 12.11.

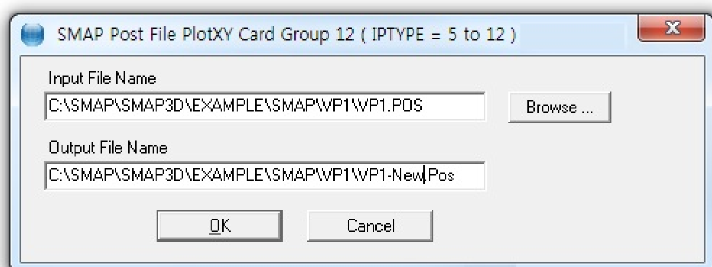


Figure 12.11 PlotXY input and output file dialog

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.

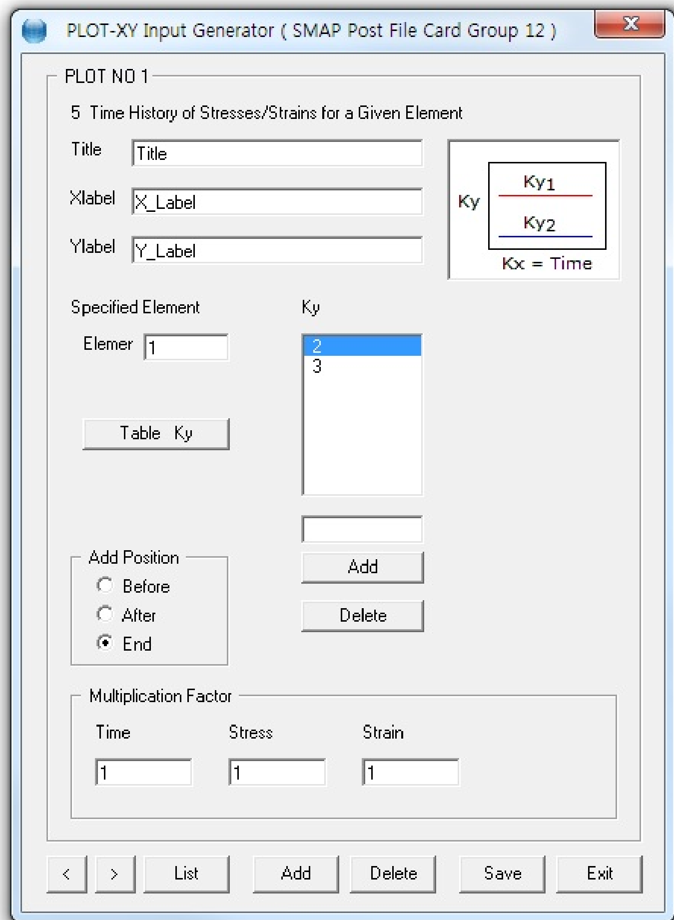


Figure 12.12 Time history for a given element

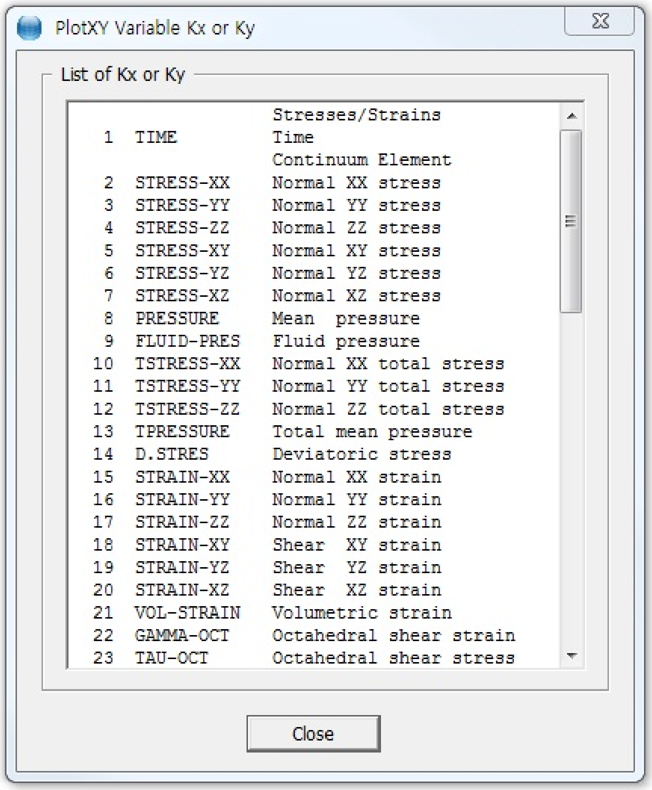


Figure 12.13 Available data for stresses / strains

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.

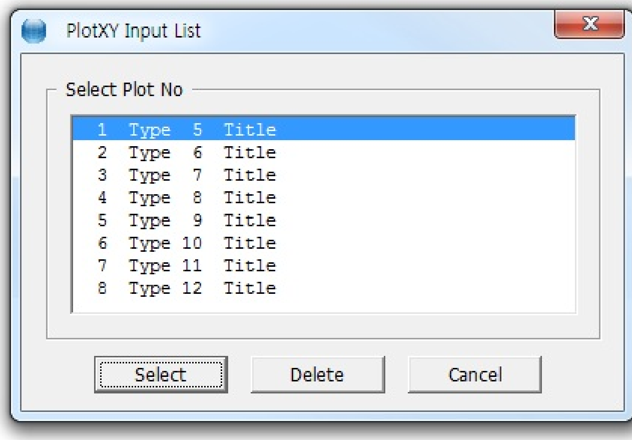


Figure 12.14 Listing of plots

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

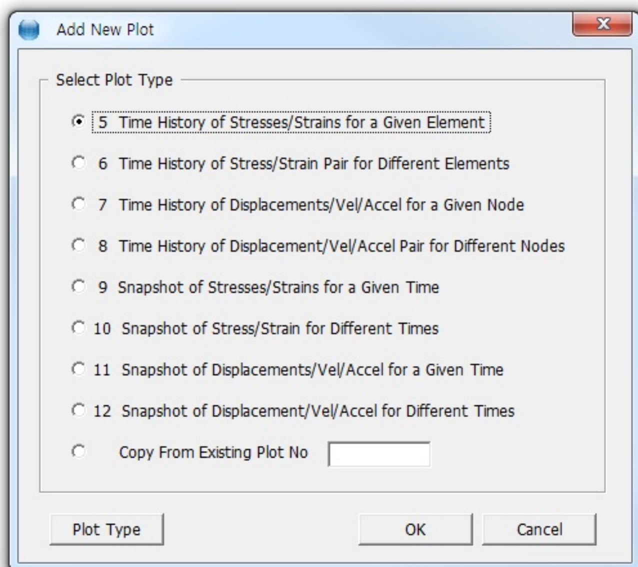


Figure 12.15 Add options for new plot

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

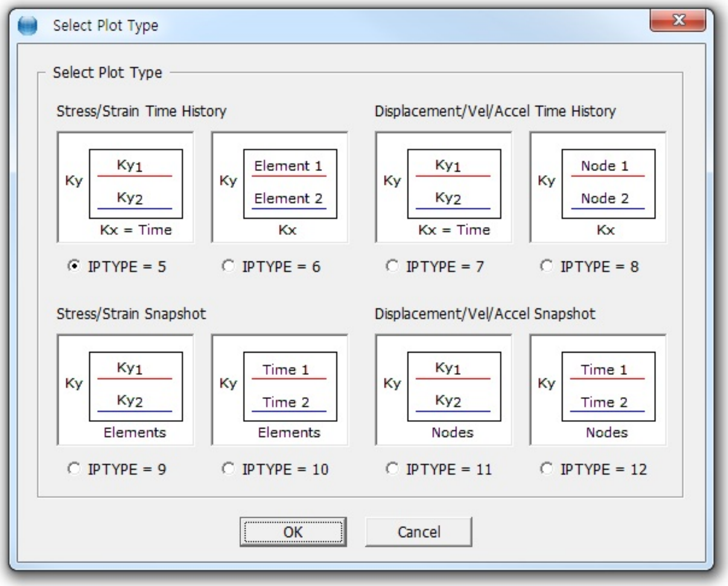


Figure 12.16 Available plot types

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.

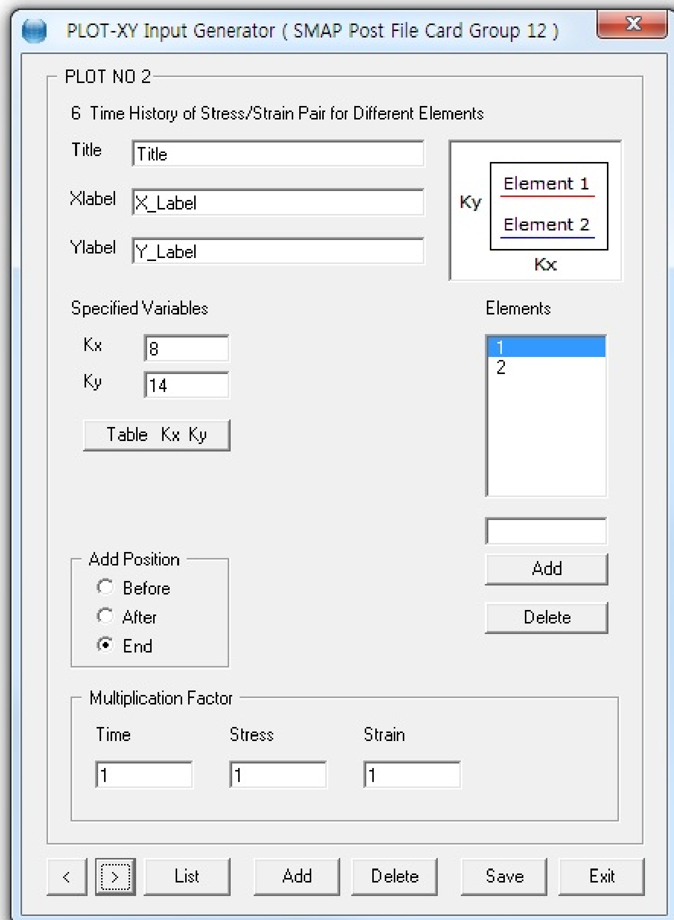


Figure 12.17 Time history for different elements

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.

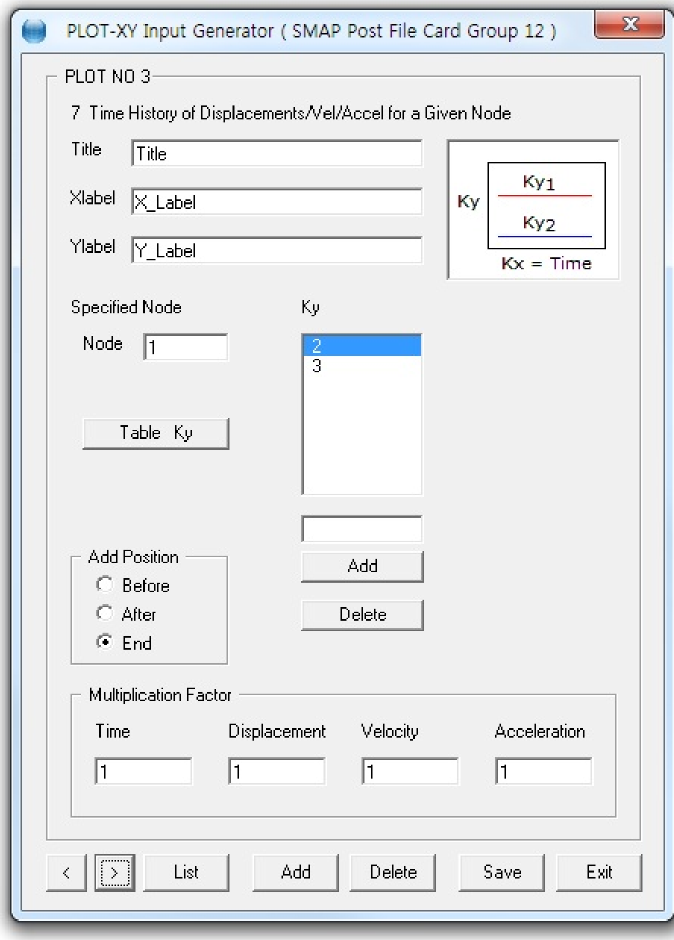


Figure 12.18 Time history for a given node

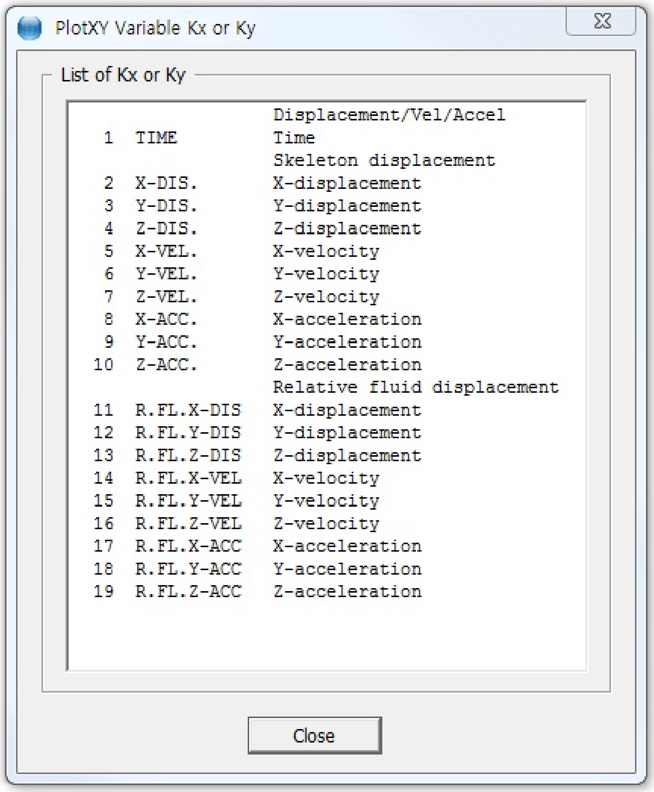


Figure 12.19 Available data for displacement/vel/accel

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.

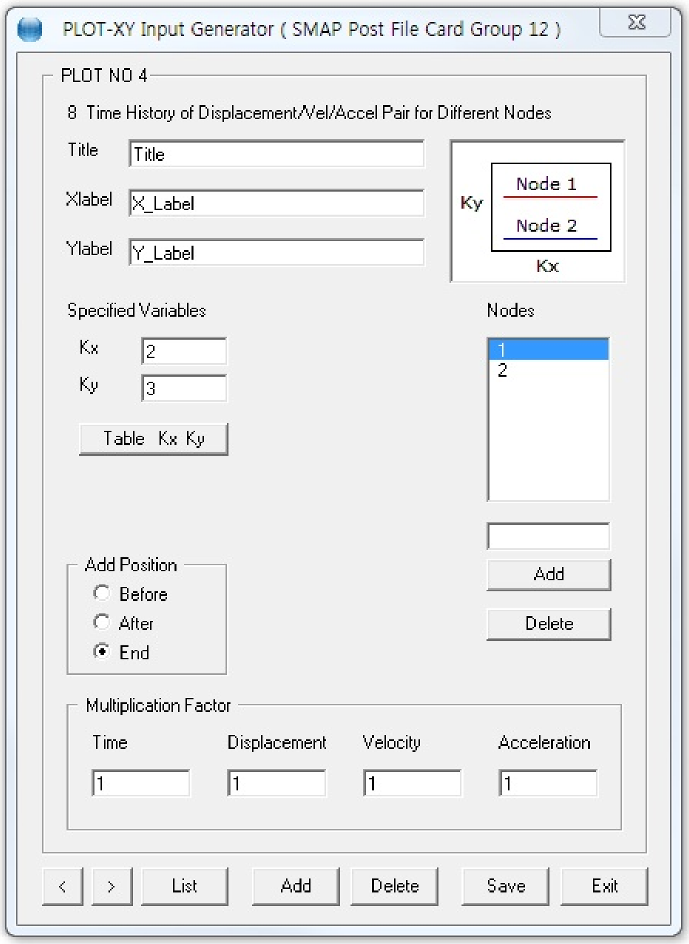


Figure 12.20 Time history for different nodes

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.

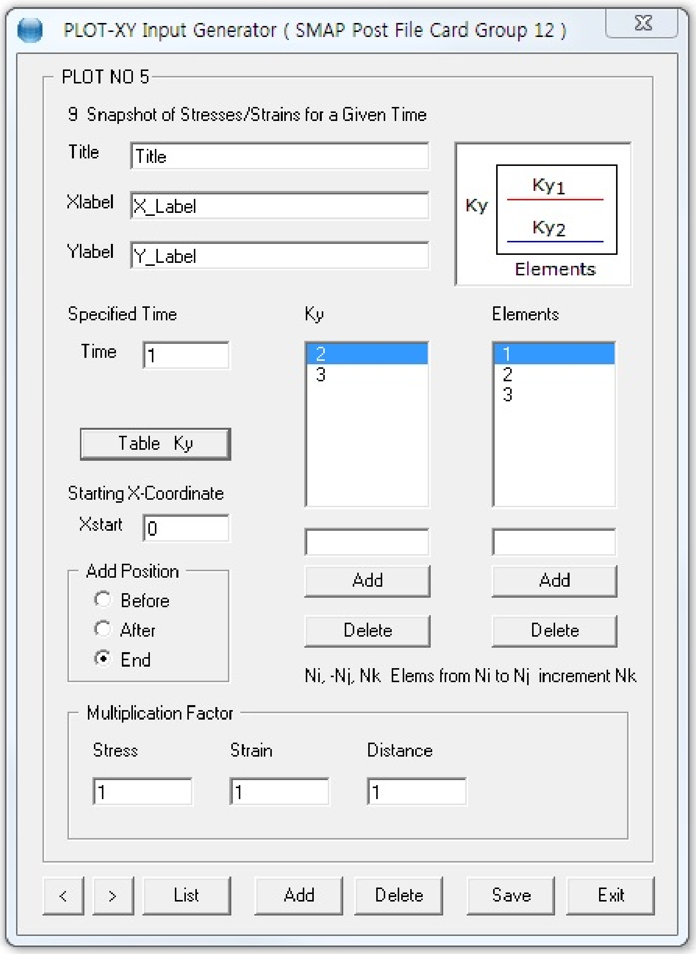


Figure 12.21 Stress/strain snapshot for a given time

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).

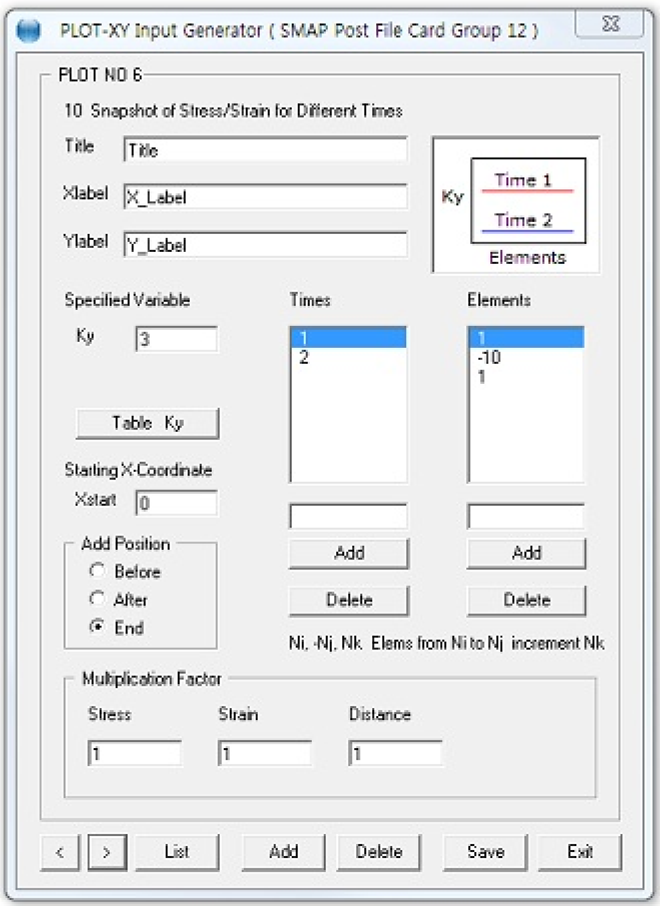


Figure 12.22 Stress/strain snapshot for different times

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.

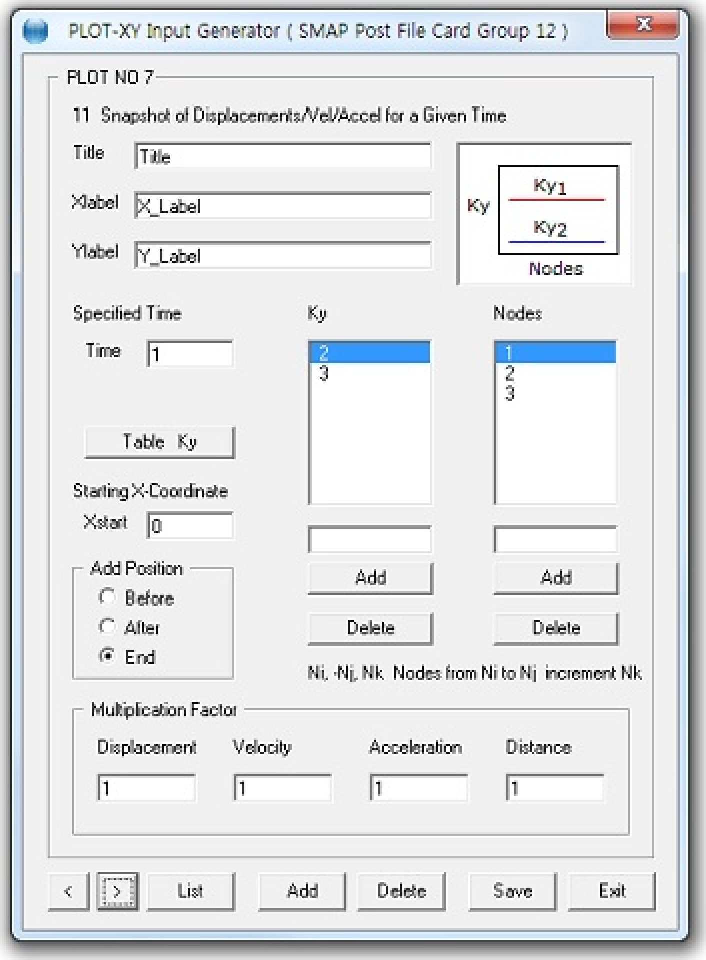


Figure 12.23 Displ/vel/accel snapshot for a given time

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).

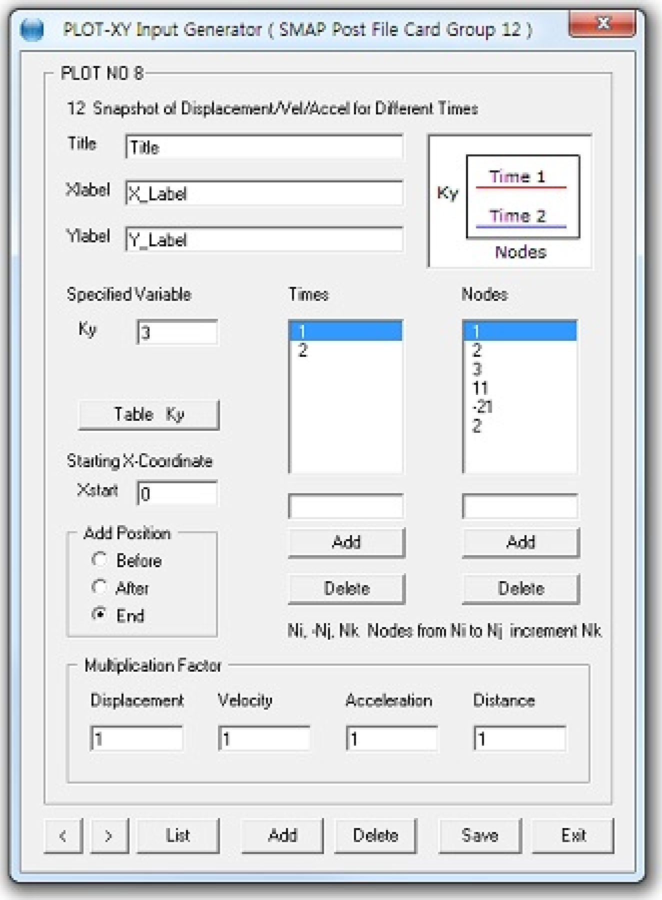


Figure 12.24 Displ/vel/accel snapshot for different times

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:

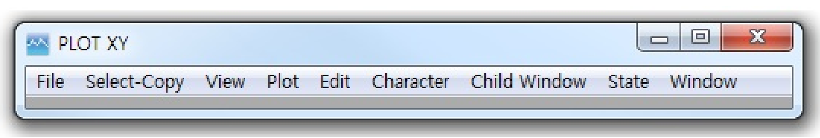
- Plot scatterplot data
  It reads the scatterplot data in text file and plots lines connecting each pair of data points.

- 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.

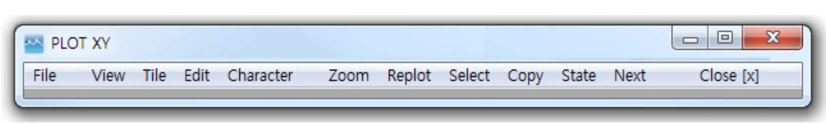
- Edit XY graph
  It reads XY data, edits titles and scales, adds user-defined additional curves.

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

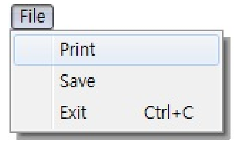


Express Style includes 12 menus which are rearranged so as to quickly access most frequently used menu items in practice.
For Express Style, specify 0 in C:\Smap\Ct\Ctdata\MenuStyle_XY.dat



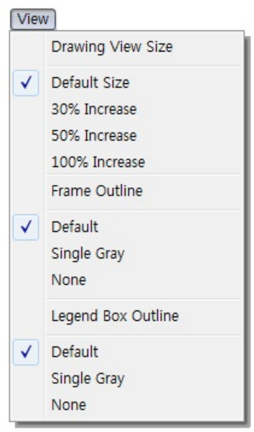
13.2 Menus

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.



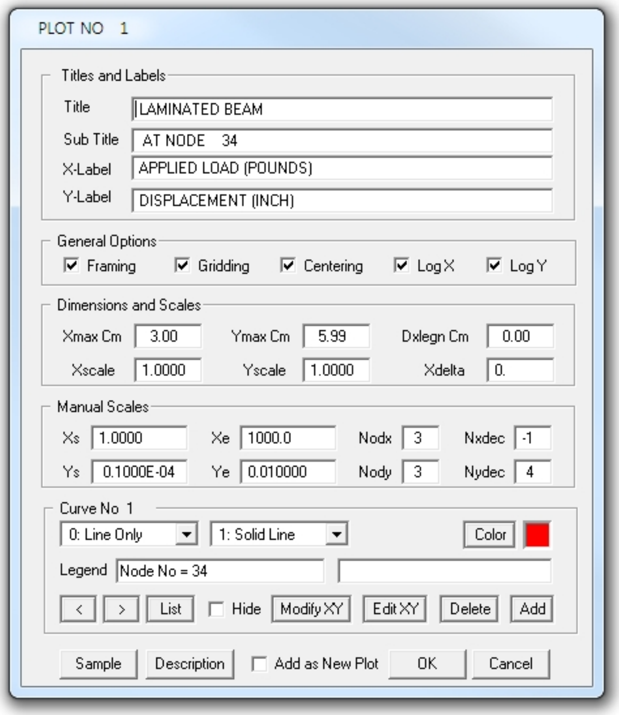
Select-Copy is mainly used to select and then copy the current view.

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

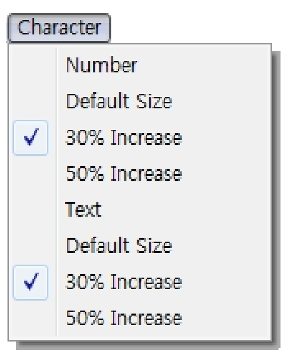


Plot has the following five sub menus.
Replot is to replot the currently focused child window. Zoom is to zoom the currently focused child window. Once this sub menu is selected, you can specify the rectangular zoom area by left mouse button down at the left top corner and then left mouse button up at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

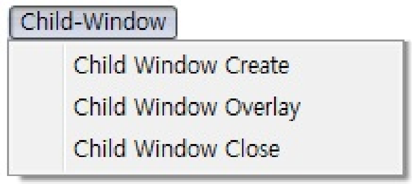
Edit opens following dialog to edit XY graph data.
It is described in detail in Section 12.3 in XY graph User's Manual.



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:

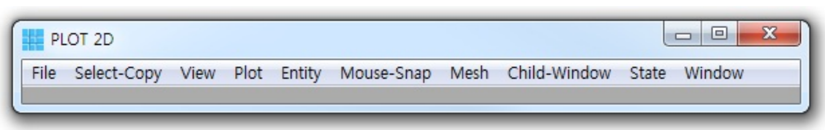
- **Plot finite element meshes**
  It reads the Mesh File and plots meshes along with node, element, boundary code, and material numbers.

- **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.

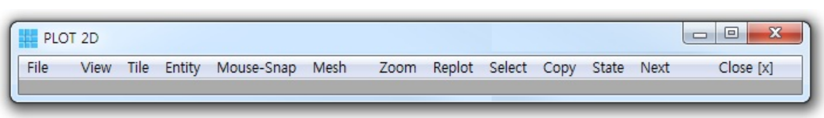
- **Edit finite element or group meshes**
  It reads finite element or group mesh files and edit these meshes.

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

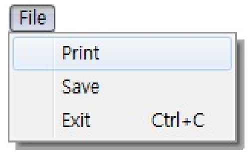


Express Style includes 13 menus which are rearranged so as to quickly access most frequently used menu items in practice.
For Express Style, specify 0 in C:\Smap\Ct\Ctdata\MenuStyle_2D.dat



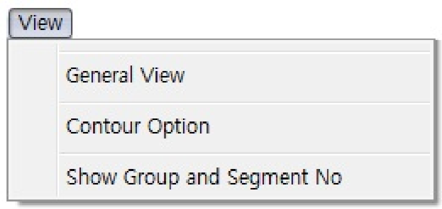
14.2 Menus

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.

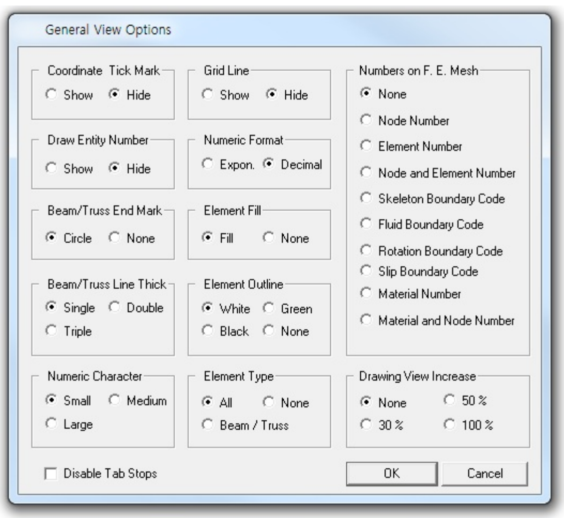


Select-Copy is mainly used to select and then copy the current view.

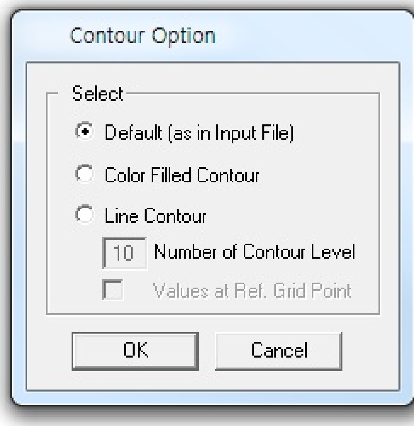
View has three sub menus;
General View, Contour Option,
and Show Group and Segment No.



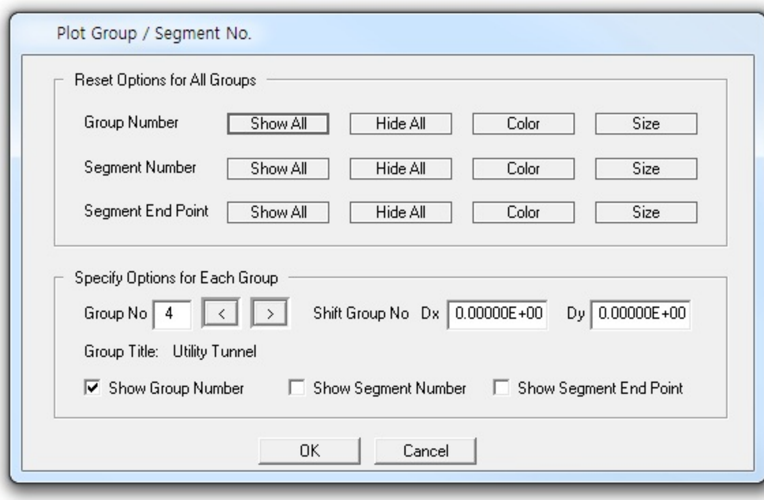
General View options affect all types of plots.

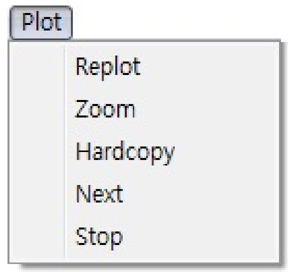


Contour Options affect contour plots of continuum element data for analysis results.

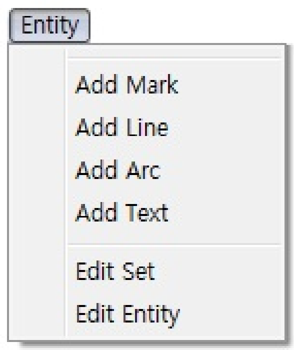


Show Group and Segment No is to show group and segment numbers when editing group meshes. It is described in detail in Section 5.3 in Group Mesh User's Manual.



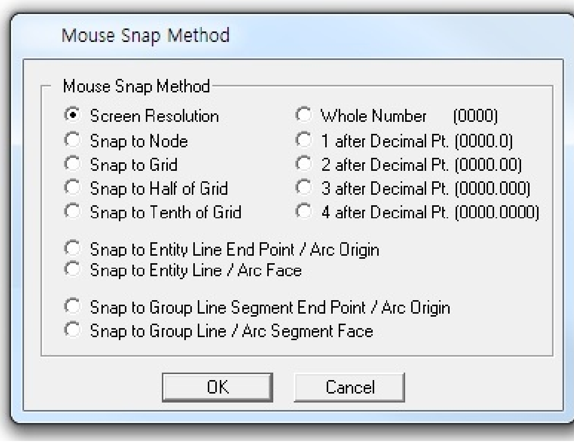


Plot has the following five sub menus.
Replot is to replot the currently focused child window. Zoom is to zoom the currently focused child window.
It zooms only mesh. Once this sub menu is selected, you can specify the rectangular zoom area by left mouse button down at the left top corner and then left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

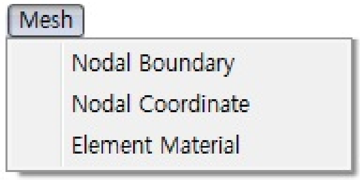


Entity is the graphical object which is mainly used to assist editing geometry of groups and elements. It has following six sub menus; Add Mark, Add Line, Add Arc, Add Text, Edit Set, and Edit Entity. It is described in detail in Section 5.7 in Group Mesh User's Manual.

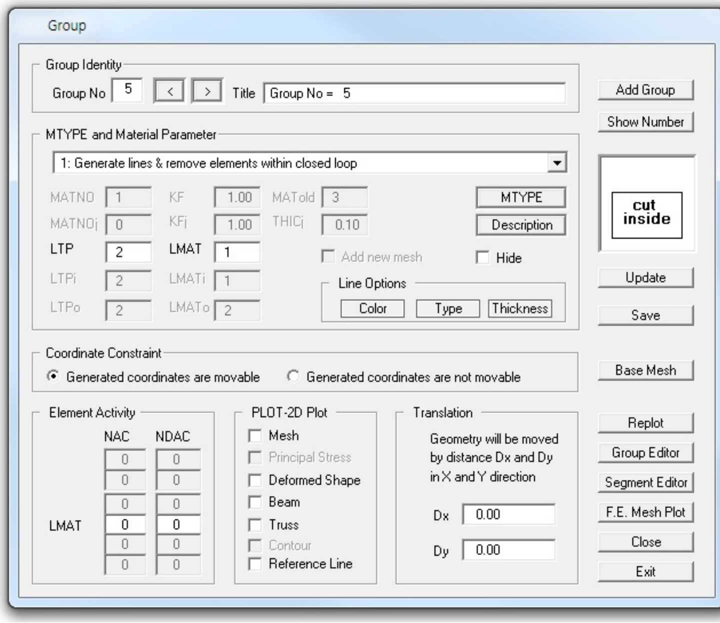
Mouse-Snap is to control the position of mouse cursor when you work for finite element mesh, group mesh, or entities. Mouse Snap Method helps you place the mouse cursor more accurately.



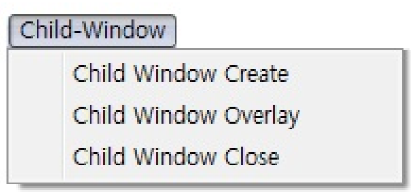
Mesh is used to directly modify finite element meshes. It has three sub menus; Nodal Boundary, Nodal Coordinate, and Element Material.
It is described in detail in Section 5.6 in Group Mesh User's Manual.



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:

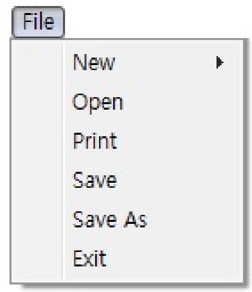
- **Plot finite element meshes**
  It reads the Mesh File and plots meshes along with node, element, boundary code, and material numbers.

- **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.

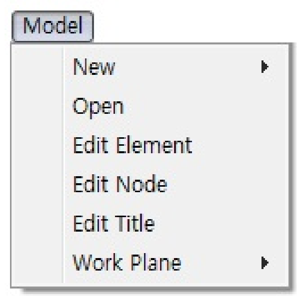
15.2 Menus



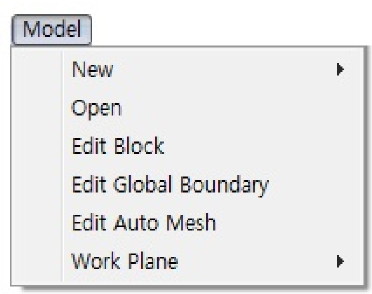
File has six sub menus.
New is used to build Finite Element Mesh
or Block Mesh.
Open is used to open existing mesh file.
Print is to get the hard copy of the current view.
Save is to save the current mesh file or current view.

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.

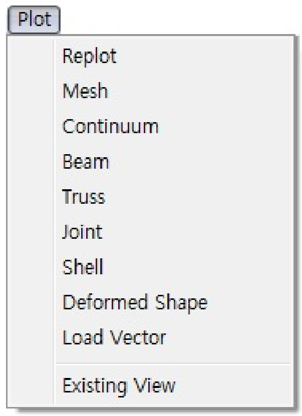


For editing Finite Element Mesh,
6 menus are shown.
New is to build new mesh file.
Open is to open existing mesh file.
Edit Element is to edit parameters
related to element.
Edit Node is to edit parameters related to node.
Edit Title is to edit title.
Work Plane is to show prebuilt work planes.



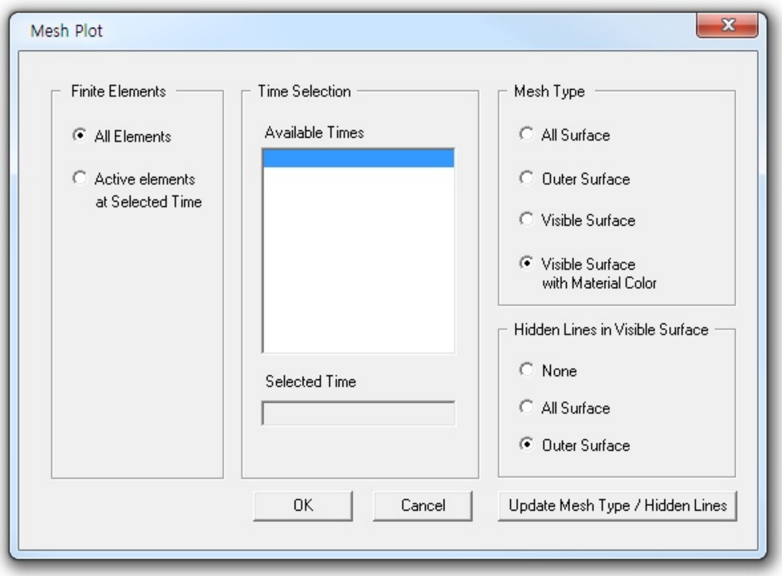
For editing Block Mesh,
6 menus are shown.
New is to build new mesh file.
Open is to open existing mesh file.
Edit Block is to edit parameters
related to block.
Edit Global Boundary is to edit parameters
related to boundary.
Edit Auto Mesh is to edit parameters
related to auto mesh.
Work Plane is to 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.

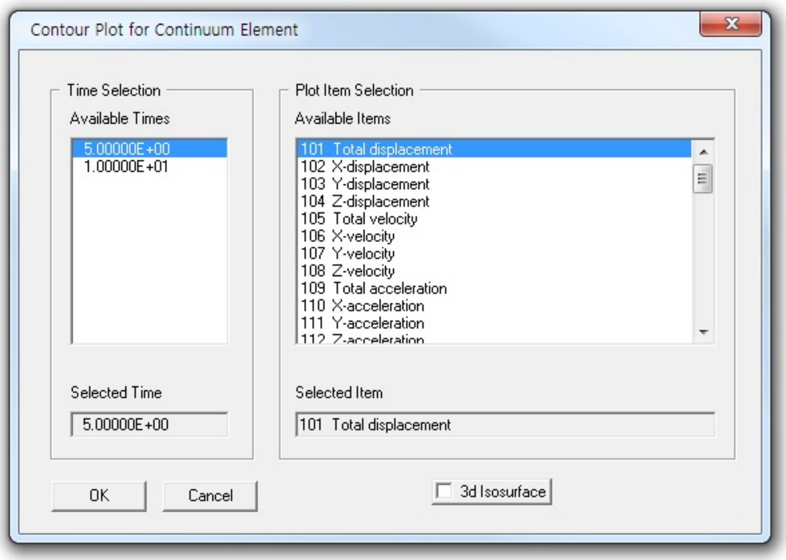


Replot is mainly used to refresh the current view.

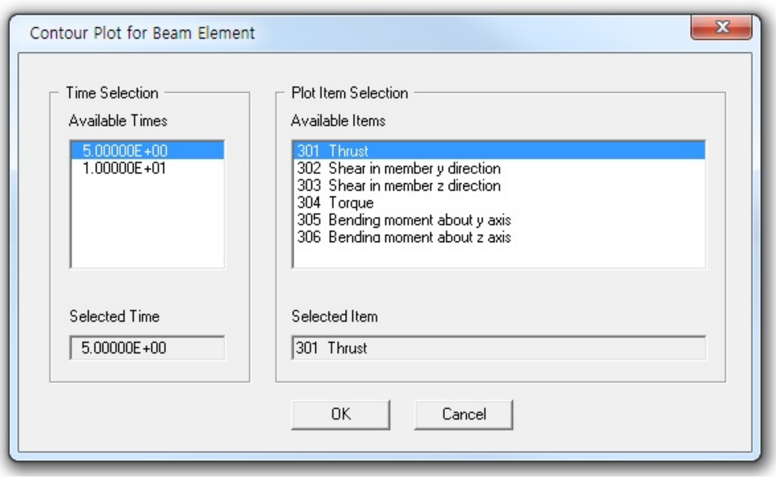
Mesh is to plot Finite Element meshes (Default plot type).
Mesh plot requires only Mesh File.



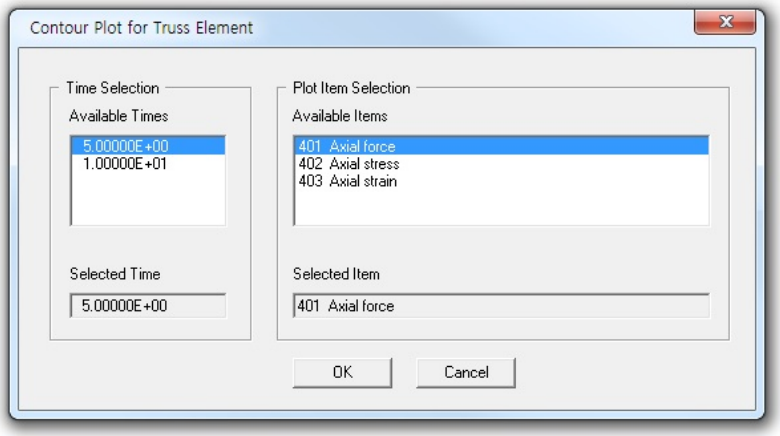
Continuum is to plot contours or principal stress vectors for continuum elements. By checking "3d Isosurface", iso surface will be shown.



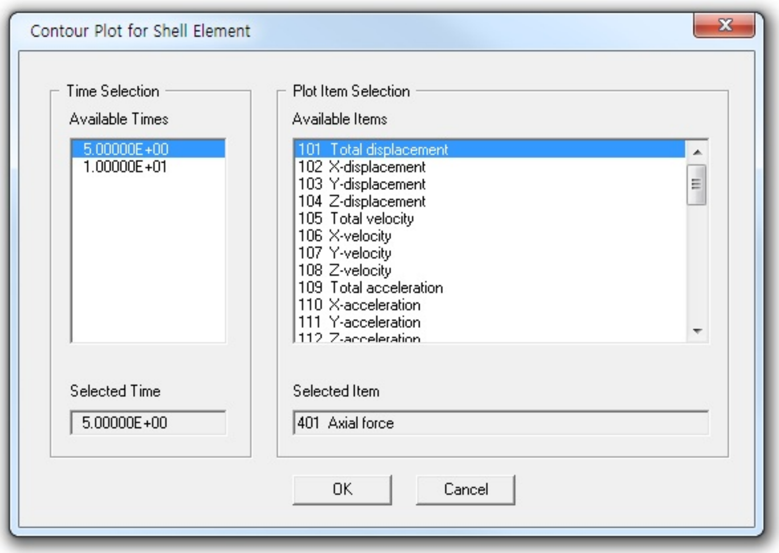
Beam is to plot section forces of beam elements.



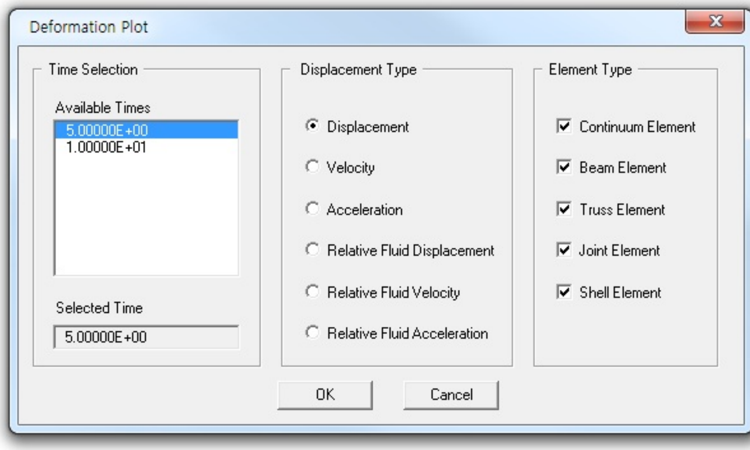
Truss is to plot axial force/stress/strain of truss elements.



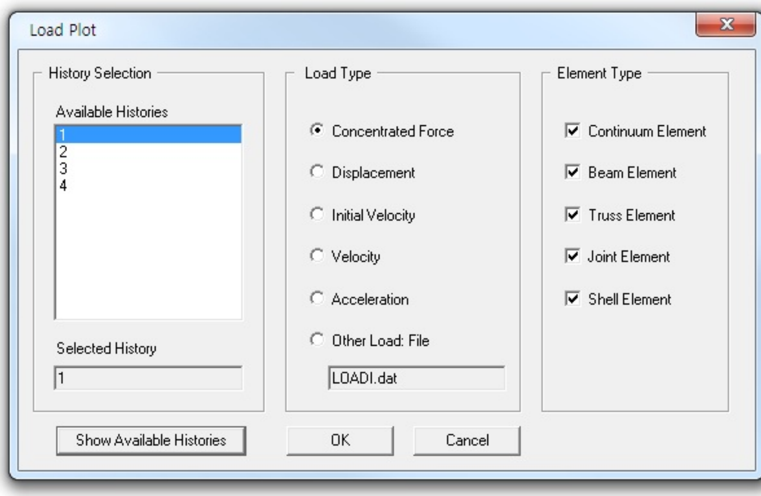
Shell is to plot contours or principal stress vectors for shell elements.



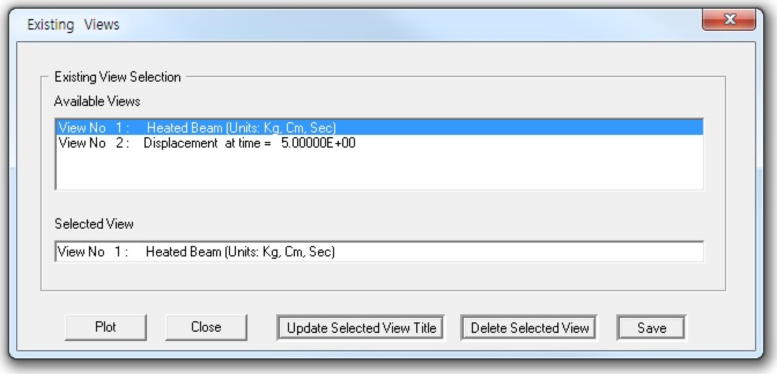
Deformed Shape is to plot the snap shot of all kinds of displacement/velocity/accelerations.
Note that deformed meshes can be combined with other plot types as discussed in "Displacement" option in view menu.



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.

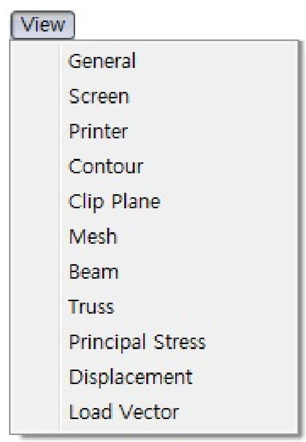


Existing View is to replot the saved views.



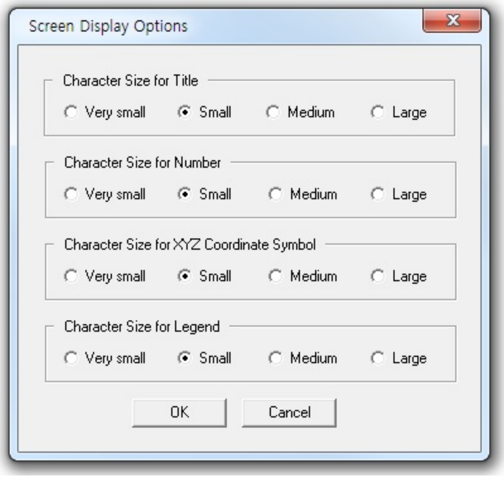
View is used to change the appearance of a selected plot.

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



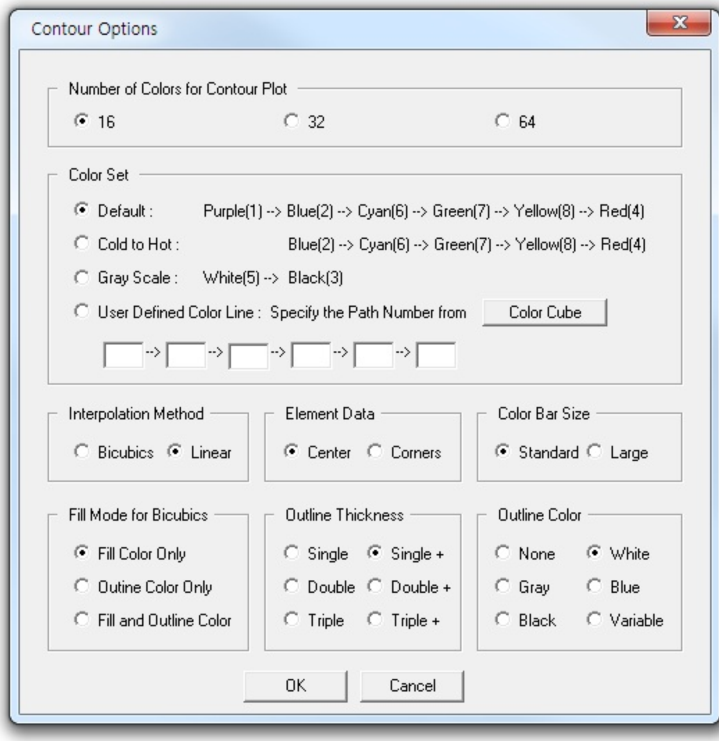
General view options affect most plot types.

Screen display options affect character sizes shown on the monitor.

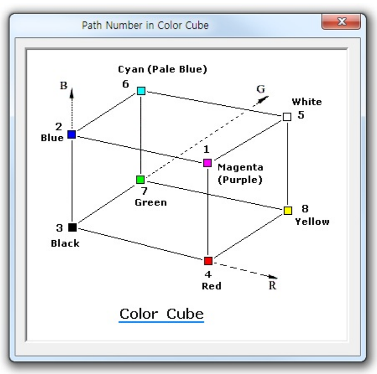


Printer display options affect character sizes and plot dimensions shown on the hard copy.

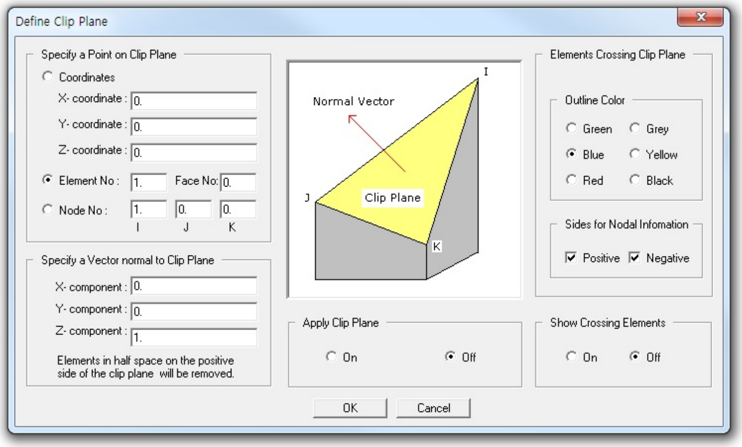
Contour options affect all types of plots involving contours.



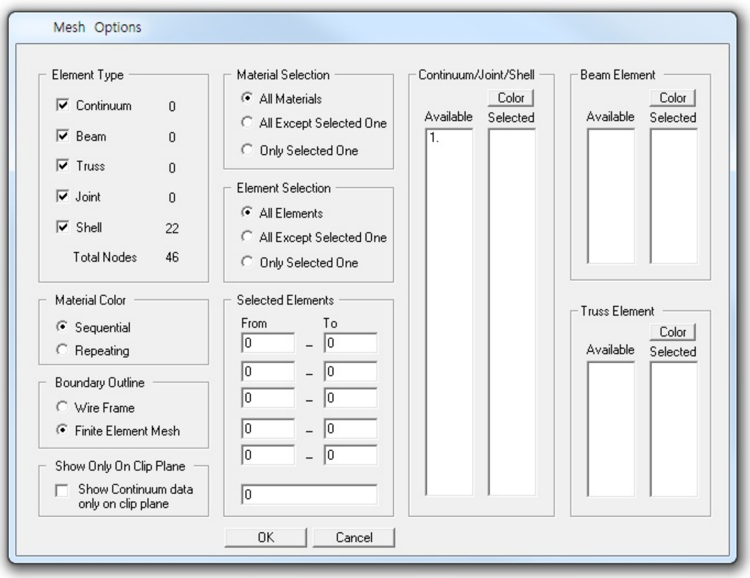
Color cube is to use for user defined color line.



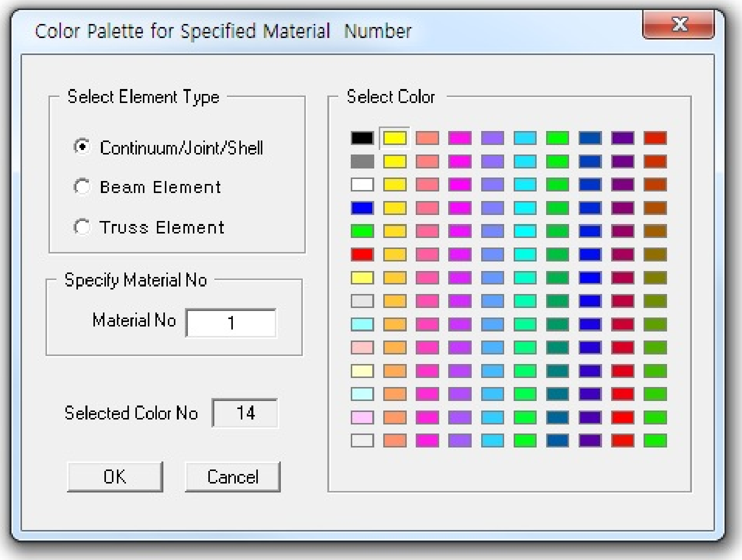
Clip plane defines parameters associated with the clip plane which cuts through the internal part of the 3D domain. When "Apply Clip Plane" is on, contours or deformed shapes are shown on such user defined plane.



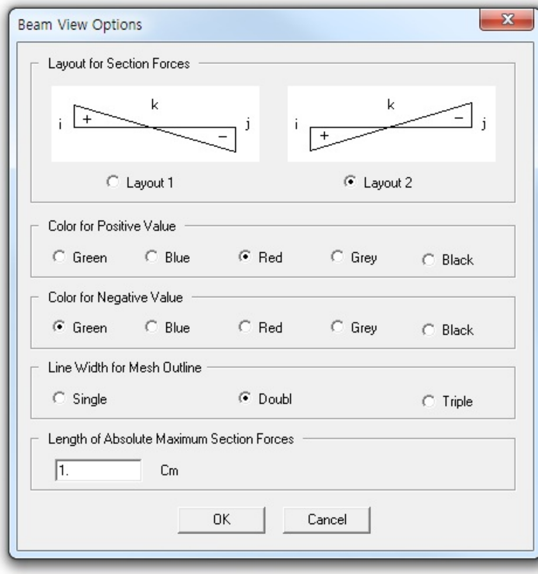
Mesh options affect all plot types. As one of useful features, it can select particular types of elements and materials.



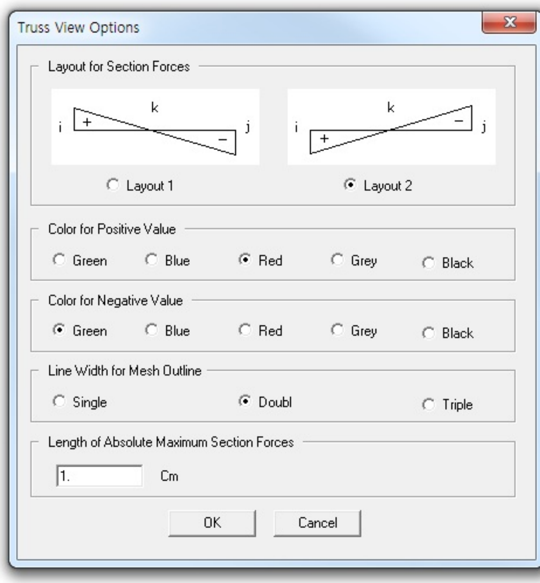
Color is to use for user defined mesh color.



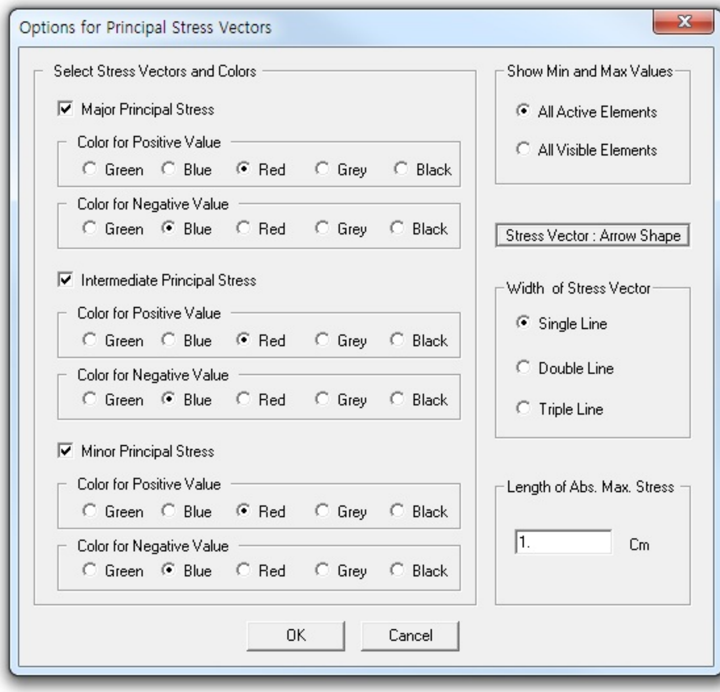
Beam view options affect only beam plot.



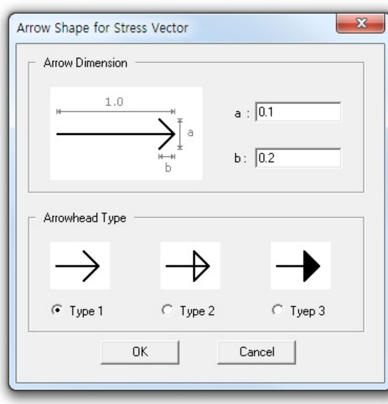
Truss view options affect only truss plot.



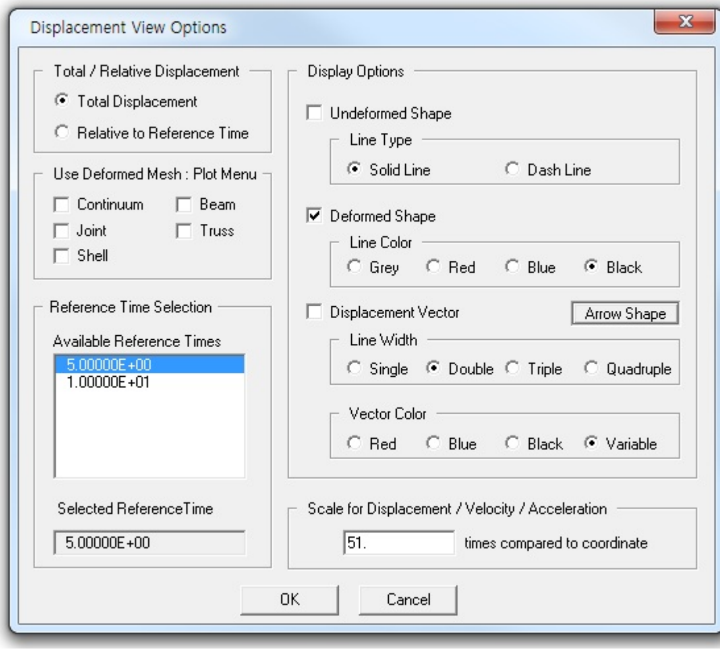
Principal Stress options affect only plots of principal stress vectors in continuum or shell elements.



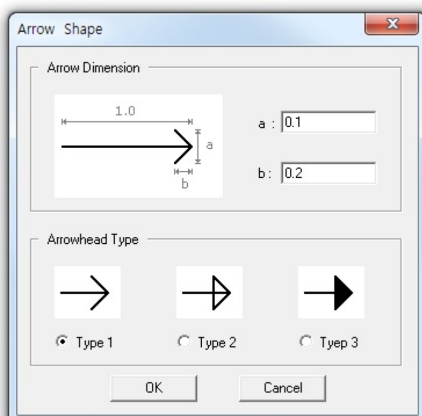
Users can specify the arrow shape for stress vector.



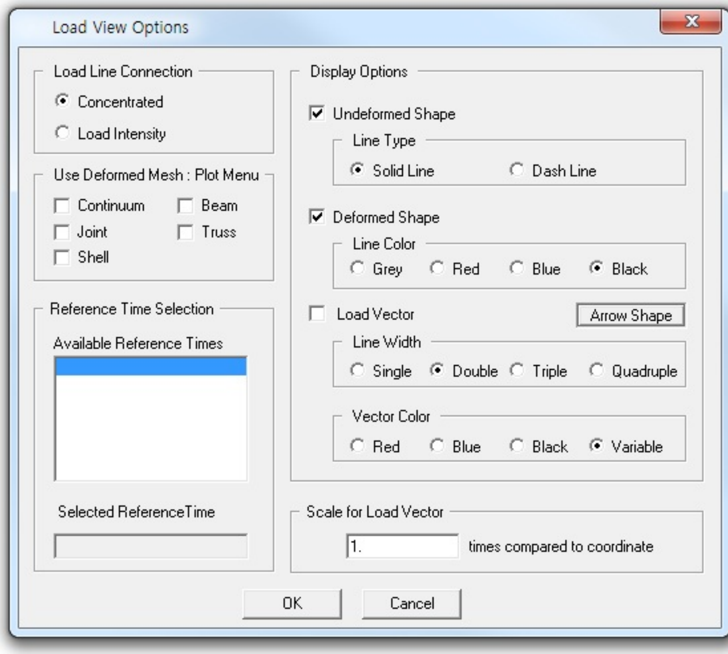
Displacement view options affect only deformed shape plot. Continuum, Beam, Truss, and Shell plots can be displayed over deformed mesh by checking types in "Use Deformed Mesh".



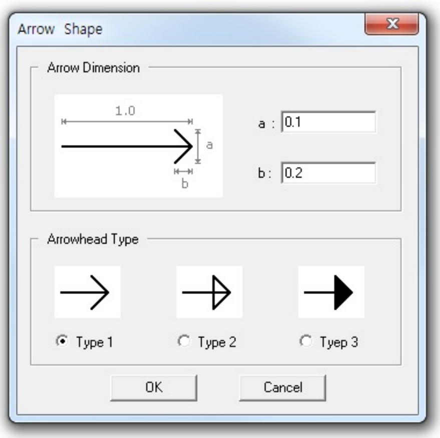
Users can specify the arrow shape for displacement vector.



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

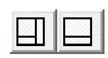
This button is used to edit finite element or block mesh.

Work Plane Toolbar

This button is to set work plane used for Model.

Layout Toolbar

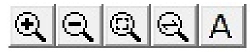
These buttons are used to show different layouts.
The first button divides the plot area into three parts; mesh, title, and legend. The second button divides the plot area into two parts; mesh and title.



XYZ Toolbar

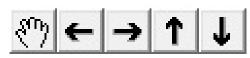
This button is used to locate position of XYZ coordinate symbol in the two part layout mode. Each time you click this button, the XYZ symbol moves counterclockwise along the corners of rectangle. XYZ button is also used to control the amount of movement, rotation, and zoom.

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.

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

SMAP ® - 3D

Structure Medium Analysis Program

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

Theory

Printed in the United States of America.

LICENSE AGREEMENT

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

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

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

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

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

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

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

Contents

Introduction

1.1 Introduction

SMAP-3D, which is an upgraded follow-on to the original MPDAP (Multi-Phase Dynamic Analysis Program), is a three-dimensional finite element computer program which has been continuously improved based on theoretical and experimental works since 1982. The program has been used to study fundamental mechanics of saturated porous medium. The program can be a powerful tool for the geomechanical analysis since it can solve static, consolidation and dynamic problems in dry, partially saturated or fully saturated soils and porous rock mass. The program considers material, geometric and boundary condition nonlinearities. Next two sections describe theoretical backgrounds of program SMAP-3D.

Section 2 describes theoretical formulations of nonlinear two-phase medium. Nonlinear compressibility equations are derived in detail for grains, saturated pore water, and partially saturated pore water. Field equations representing fundamental mechanics of two-phase medium are presented.

These field equations include effective stress law, constitutive equation for skeleton deformation, continuity equation of pore fluid, equation of motion for the bulk mixture, and equation of motion for pore fluid. Then, these field equations are discretized in space and expressed in incremental forms. Finally, global equilibrium equations are derived by principle of virtual work and then linearized to be solved by linear equation solver.

Section 3 describes constitutive relations of various nonlinear material models available in the program SMAP-3D. These nonlinear models include; Generalized Hoek and Brown Model, Single Hardening Elasto-Plastic Model, JWL High Explosive Model, Modified Cam Clay Model with Creep, Engineering Model, Joint Model, and Generalized Decoupled Hyperbolic Model.

To see the validation of the computational algorithms of the computer program SMAP-3D, refer to SMAP-3D Example Problems.

Finite Element Formulation of Nonlinear Two-Phase Medium

2.1 Introduction

Biot introduced fundamental analytical work describing the behavior of saturated porous media in a series of papers extending over many years (e.g. 1956, 1962a and 1962b). Other investigators have applied Biot's analytic results using techniques which approximate his equations with varying degrees of accuracy and sophistication (e.g. Ghaboussi and Wilson 1972, Mengi and McNiven, 1977). Theoretical formulations incorporated in the code SMAP-3D are the extension of Biot's two-phase theory to nonlinear region. These nonlinear two-phase theories have been developed over a decade under the sponsorship of Air Force Office of Scientific Research (e.g. Kim and Blouin 1984, Kim et al. 1986, 1987 and 1988).

In this section, the fundamental equations implemented in the code SMAP-3D are described. First the individual material components compressibility models are described in Subsection 2.2 for the solid grain, Subsection 2.3 for the pore water and Subsection 2.4 for the partially saturated water. Nonlinear material models of the skeleton are presented in Section 3. Field equations described in Subsection 2.5 include effective stress law, constitutive equation for skeleton deformation, continuity equation of pore fluid flow, equation of motion for the bulk mixture and equation of motion for pore fluid.

These field equations are described in terms of nodal values and expressed in incremental form in Subsection 2.6. Finally, global equilibrium equations for the two-phase medium are formulated in Subsection 2.7 and linearized to be solved by linear equation solver in Subsection 2.8.

2.2 Grain Model

To model the nonlinear response of the solid grains to both the applied pore pressure and effective stress, analytic expressions for the deformation of solids at high pressure are employed. High pressure data for many rocks and minerals show a linear relationship between loading wave velocity and particle velocity (e.g. Allen, 1967).
The loading wave velocity can be expressed as:

$$c_L = c_o + S\, v_p \tag{2.1}$$

where:

- c_L = Loading wave velocity
- c_o = The initial wave velocity at relatively low pressure
- v_p = Peak particle velocity
- S = Experimentally determined constant relating c_L to v_p (generally equal to about 1.5 for most dense rocks and minerals)

Conservation of mass and momentum on either side of the wave front yields the familiar relationships:

$$\sigma_p = \rho_o\, c_L\, v_p \tag{2.2}$$

$$M = \rho_o\, c_L^2 \tag{2.3}$$

where:

σ_p = Peak axial stress

ρ_o = Initial material density

M = Constrained secant modulus = σ_p / ε_p

ε_p = Peak axial strain corresponding to the peak stress σ_p

Substitution of Equation 2.1 into 2.2 gives:

$$\sigma_p = \rho_o c_o v_p + \rho_o S v_p^2 \tag{2.4}$$

and solving for peak particle velocity as a function of peak stress yields

$$v_p = \frac{f(\sigma_p)}{2 \rho_o S} \tag{2.5}$$

where

$$f(\sigma_p) = (\rho_o^2 c_o^2 + 4 \rho_o S \sigma_p)^{1/2} - \rho_o c_o \tag{2.6}$$

Substitution of Equation 2.1, 2.5, and 2.6 into Equation 2.3 gives:

$$M = F(\sigma_p) = \rho_o c_o^2 + c_o f(\sigma) + \frac{f^2(\sigma)}{4\rho_o} \tag{2.7}$$

The tangent constrained modulus, M_t, used in the numerical model is defined as the slope of the stress strain curve by:

$$M_t = \frac{d\sigma}{d\varepsilon} \tag{2.8}$$

From Equation 2.7 and the definition of constrained modulus, M:

$$\varepsilon_p = \frac{\sigma_p}{F(\sigma_p)} \tag{2.9}$$

Differentiating Equation 2.9 with respect to σ_p and inverting gives the tangent constrained modulus as

$$M_t = \frac{F^2(\sigma_p)}{F(\sigma_p) - \sigma_p F'(\sigma_p)} \tag{2.10}$$

Differentiating Equations 2.6 and 2.7 with respect to σ_p yields:

$$F'(\sigma_p) = c_o f'(\sigma_p) + \frac{f(\sigma_p) f'(\sigma_p)}{2 \rho_o} \tag{2.11}$$

and

$$f'(\sigma_p) = \frac{2 \rho_o S}{(\rho_o^2 c_o^2 + 4 \rho_o S \sigma_p)^{1/2}} \tag{2.12}$$

Hence, Equations 2.5 through 2.12 can be used to define high pressure constrained stress strain and modulus relationships for the solid grains.

For two phase, coupled calculations, the volumetric relationships for the solid grains should be specified in terms of the bulk modulus, K_g, rather than in terms of the constrained modulus. At high pressures, the shear strength of the grain materials becomes insignificant compared to the applied stress and the materials tend to behave like fluids. At these pressures, the tangent bulk modulus equals the tangent constrained modulus with Poisson's ratio equal to 0.5. Beneath some threshold pressure, p_b, Poisson's ratio begins to decrease from 0.5 at p_b to an initial value of Poisson's ratio, ν_o, at a low value of mean stress. We have used a simple relationship to approximate the influence of mean stress on Poisson's ratio for the solid grains:

$$K_g = g(p) M_t \tag{2.13}$$

The ratio of the bulk modulus to the tangent constrained modulus, g(p) at pressures less than p_b is given by:

$$g(p) = \frac{2}{3} \frac{(1 - 2v_o)}{(1 - v_o)} \frac{p}{p_b} + \frac{(1 + v_o)}{3(1 - v_o)} \tag{2.14}$$

For pressures greater than p_b;

$$g(p) = 1 \tag{2.15}$$

Poisson's ratio can be computed as a function of the modulus ratio at a given pressure as:

$$v = \frac{3\,g(p) - 1}{1 + 3\,g(p)} \tag{2.16}$$

2.3 Pore Water Model

The model for the nonlinear, elastic compressibility of the pore water is derived from an equation of state reported by Ahrens (1988) and attributed to Bakanova, et. al. (1976). This equation relates the shock velocity in water to the peak particle velocity. In the lower pressure regime, a quadratic relation is used while a linear relation is used in the higher pressure regime. The transition point between the two regimes is defined in terms of a peak particle velocity at the transition, V_{pt}. Bakanova's equations can be expressed as:

$$v_s \le v_{pt}: \qquad c = c_1 + S_1 v_p + S_2 v_p^2 \tag{2.17}$$

$$v_s > v_{pt}: \qquad c = c_2 + S_3 v_p \tag{2.18}$$

where:

c	=	Shock propagation velocity in the fluid
v_p	=	Peak fluid particle velocity
c_1, S_1, S_2	=	Constants used to fit data below the transition
c_2, S_3	=	Constants used to fit data above the transition

Equation 2.18 can also be expressed in terms of the shock velocity at the transition point, c_t. Substituting v_{pt} into Equation 2.18 yields:

$$c_2 = c_t - S_3 v_{pt} \tag{2.19}$$

Substituting 2.19 into 2.18 produces this expression for the shock velocity above the transition:

$$v_s > v_{pt} : \quad c = c_t + S_3 (v_p - v_{pt}) \tag{2.20}$$

where:

c_t	=	Shock velocity at the transition
v_{pt}	=	Peak particle velocity at the transition (Model constant)

At the transition point, the shock velocity from Equations 2.17 and 2.20 should be equal to preserve continuity. Setting Equations 2.17 and 2.20 equal at $v_p = v_{pt}$ gives:

$$c_t = c_1 + S_1 v_{pt} + S_2 v_{pt}^2 \tag{2.21}$$

thereby defining c_t in terms of the model constants. Equations 2.17, 2.20, and 2.21 (with the constants c_1, S_1, S_2, and S_3) define the shock velocity as a function of peak particle velocity.

To derive a bulk modulus for water as a function of pressure, we first need an expression for peak particle velocity as a function of pressure.

Conservation of mass and momentum on either side of the wave front yields the familiar relationship from shock physics:

$$\pi_p = \rho_o c v_p \tag{2.22}$$

where:

π_p = Pore fluid pressure

ρ_o = Mass density of fluid

Substitution of Equation 2.17 into 2.22 yields an expression for the transition fluid pressure (π_{pt}):

$$\pi_{pt} = \rho_o v_{pt} (c_1 + S_1 v_{pt} + S_2 v_{pt}^2) \tag{2.23}$$

For water, the transition pressure is greater than 30,000 MPa. Below the transition pressure, substitution of Equation 2.17 into 2.22 will give:

$$v_p^3 + \frac{S_1}{S_2} v_p^2 + \frac{c_1}{S_2} v_p - \frac{\pi_p}{\rho_o S_2} = 0 \tag{2.24}$$

This cubic equation can be solved to yield an expression for v_p as a function of fluid pressure below the transition pressure π_{pt} : where:

$$v_p = m \cos\left[\frac{1}{3}\cos^{-1}\left(\frac{3\beta}{\alpha m}\right) + \frac{4\pi}{3}\right] - \frac{S_1}{3 S_2} \tag{2.25}$$

where

$$\alpha = \frac{c_1}{S_2} - \frac{1}{3}\left(\frac{S_1}{S_2}\right)^2 \tag{2.26}$$

$$\beta = \frac{-\pi_p}{\rho_o S_2} - \frac{1}{3}\left(\frac{S_1}{S_2}\right)\left(\frac{c_1}{S_2}\right) + \frac{2}{27}\left(\frac{S_1}{S_2}\right)^3 \qquad (2.27)$$

$$m = 2\sqrt{\frac{-\alpha}{3}} \qquad (2.28)$$

Above the transition pressure, substitution of Equation 2.20 into 2.22 yields a quadratic equation:

$$v_p^2 + \left(\frac{c_t - S_3 v_{pt}}{S_3}\right) v_p - \frac{\pi_p}{\rho_o S_3} = 0 \qquad (2.29)$$

Solving this equation for v_p as a function of fluid pressure gives v_p for pressures above the transition pressure π_{pt} :

$$v_p = -\left(\frac{c_t - S_3 v_{pt}}{2S_3}\right) + \left[\left(\frac{c_t - S_3 v_{pt}}{2S_3}\right)^2 + \frac{\pi_p}{\rho_o S_3}\right]^{\frac{1}{2}} \qquad (2.30)$$

The elastic bulk modulus of water (K_w) is defined as:

$$K_w = \frac{d\pi_p}{d\varepsilon_v} = \frac{d\pi_p \,/\, dv_p}{d\varepsilon_v \,/\, dv_p} \qquad (2.31)$$

where ε_v is the volume strain corresponding to the pressure π_p.
Taking the derivative of Equation 2.22:

$$\frac{d\pi_p}{dv_p} = \rho_o \left(c' v_p + c\right) \qquad (2.32)$$

The volume strain is given by:

$$\varepsilon_v = \frac{v_p}{c} \tag{2.33}$$

and taking the derivative yields:

$$\frac{d\varepsilon_v}{dv_p} = \frac{c - v_p\, c'}{c^2} \tag{2.34}$$

Substitution of Equations 2.32 and 2.34 into 2.31 gives an expression for the bulk modulus in terms of the shock and peak particle velocities:

$$K_w = \frac{\rho_o\, c^2\, (c + v_p\, c')}{c - v_p\, c'} \tag{2.35}$$

The derivatives of the shock velocity with respect to the peak particle velocity are given by:

$$\pi_p \leq \pi_{pt} : \qquad c' = S_1 + 2\, S_2\, v_p \tag{2.36}$$

$$\pi_p > \pi_{pt} : \qquad c' = S_3 \tag{2.37}$$

The material constant values for this model are given in Table 2.1 for fresh water and sea water. The fresh water values are from Bakanova, et. al. (1976) as reported by Ahrens (1988). Parameters for sea water were fit to compressibility data described by Kim, et. al. (1986) and attributed to Britt (1985).

2.4 Partially Saturated Pore Water Model

When rock or soil is unsaturated, compression of the pore water and solid grains is nearly insignificant when compared with the compression of pore air. Under these conditions, material behavior is governed mostly by the skeleton model. With sufficient compression, the pore air gets squeezed out and the material becomes saturated. Rischbieter, et. al. (1977) demonstrated that even a minute amount of entrapped air drastically alters the pore pressure response in multiphase porous materials. To simulate this behavior, the pore fluid model is modified to account for the compressibility of pore air and converges to a saturated condition. Note that this model is invoked only when the initial saturation is less than 100%.

The compressibility of the air-water mixture, C_{aw}, is defined as:

$$C_{aw} = \frac{d\varepsilon_{v,aw}}{d\pi_p} \qquad (2.38)$$

where π_p is the fluid pressure. The volumetric strain in the air-water mixture, $\varepsilon_{v,aw}$, is the sum of volume strain in the air and water. Using the definition of the initial saturation, it can be shown that:

$$\varepsilon_{v,aw} = (1 - S_o)\,\varepsilon_{v,a} + S_o\,\varepsilon_{v,w} \qquad (2.39)$$

where:

$\varepsilon_{v,aw}$ = Volume strain of air-water mixture
$\varepsilon_{v,a}$ = Volume strain of air bubbles
$\varepsilon_{v,w}$ = Volume strain of water (from Equation 2.33)
S_o = Initial saturation

From Equations 2.38 and 2.39 we can get an expression for the compressibility of the air-water mixture:

$$C_{aw} = (1 - S_o)\, C_a + S_o\, C_w \tag{2.40}$$

Since the compressibility is the inverse of the bulk modulus, Equation 2.40 can be expressed as:

$$\frac{1}{K_{aw}} = \frac{1 - s_o}{K_a} + \frac{s_o}{K_w} \tag{2.41}$$

where:

- K_{aw} = Bulk modulus of air-water mixture
- K_a = Equivalent bulk modulus of air bubbles in the fluid
- K_w = Bulk modulus of water (from Equation 2.35)

The volume strain and the equivalent bulk modulus of the air bubbles in the pore fluid are derived here using the adiabatic ideal gas law (γ-law). The model has been shown to be applicable when the degree of pore water saturation is above approximately 85% where the pore air is thought to exist as small bubbles within the fluid (occluded state).

The model is derived from the adiabatic ideal gas law:

$$\pi_a \cdot V_a^{\gamma} = \pi_{ao} \cdot V_{ao}^{\gamma} \tag{2.42}$$

where

- π_{ao} Initial air pressure (absolute pressure)
- π_a Current air pressure (absolute pressure)
- V_{ao} Initial air volume
- V_a Current air volume
- γ Ratio of heat capacity (c_p/c_v)

The volume strain of air can be defined in terms of engineering strain:

$$\varepsilon_{v,a} = 1 - \left(\frac{V_a}{V_{ao}}\right) \tag{2.43}$$

Substituting Equation 2.42 into Equation 2.43, we can express the volume strain of air bubble in terms of air pressure:

$$\varepsilon_{v,a} = 1 - \left[\frac{\pi_{ao}}{\pi_a}\right]^{\frac{1}{\gamma}} \tag{2.44}$$

Neglecting the influence of surface tension,

$$\pi_a = \pi + p_a \tag{2.45}$$

where

π	Current pore water pressure (gage pressure)
P_a	Reference atmospheric pressure

Substitution of Equation 2.45 into Equation 2.44 yields

$$\varepsilon_{v,\,a} = 1 - \left(\frac{\pi_a}{\pi + P_a}\right)^{\frac{1}{\gamma}} \tag{2.46}$$

Tangent bulk modulus of air bubbles can be defined as

$$K_a = \frac{d\pi_a}{d\varepsilon_{v,a}} \tag{2.47}$$

Differentiating Equation 2.46 with respect to π,

$$\frac{d\varepsilon_{v,a}}{d\pi_a} = \frac{1}{\gamma \cdot \pi_{ao}}\left(\frac{\pi_{ao}}{\pi + P_a}\right)^{\left(1 + \frac{1}{\gamma}\right)} \tag{2.48}$$

Substitution of Equation 2.48 into Equation 2.47 yields

$$K_a = \gamma \cdot \Pi_{ao} \left[\frac{\Pi + P_a}{\Pi_{ao}} \right]^{\left(1 + \frac{1}{\gamma}\right)} \qquad (2.49)$$

Equations 2.35 and 2.49, when substituted into Equation 2.41, define the compressibility of the pore air-water mixture. The model does not employ an explicit expression for the saturation point, where the air bubbles no longer exist. However, Equation 2.49 implies that the stiffness of the pore air increases with the pressure. As the pressure increases, the contribution of the air to the net compressibility of the mixture becomes insignificant when compared to the compressibility of the water. This, in essence, results in fully saturated behavior but with a smooth model transition during collapse of the air bubbles. An example pressure-volume curve for water with an initial air content of 5% is shown in Figure 2.1. Notice that the mixture becomes pressure saturated at a volume strain of about 5%.

Table 2.1 Fluid compressibility model constants
(See Section 2.3 for definitions of constants)

Parameter	Unit	Fresh Water	Sea Water
ρ_0	kg /m^3	1002.8	1026
c_1	m/s	1500	1522
S_1	-	2.00	1.97
S_2	s/m	-1.07×10^{-4}	-0.898×10^{-4}
S_3	-	1.144	1.123
v_{pt}	m/s	4000	4573
c_t	m/s	7788	8653
π_{pt}	MP$_a$	31,240	40,600

2.5 Field Equations

Effective Stress Law

Terzaghi's effective stress equation is fundamental to the development of the fully coupled model. It relates the total applied stress, σ, to the pore pressure, π, and the effective stress, σ', according to

$$\sigma_{ij} = \sigma'_{ij} + \delta_{ij}\,\pi \tag{2.50}$$

where

σ_{ij} = Total stress
σ'_{ij} = Effective stress
δ_{ij} = Kronecker's delta
δ_{ij} = 0 if $i \neq j$
δ_{ij} = 1 if $i = j$

Constitutive Equation for Skeleton Deformation

The deformation of the porous skeleton is related to the applied effective stress and the pore pressure acting on the solid grains. The stress-strain relationship is given by

$$\{d\sigma'\} = [D^{ep}]\left(\{d\varepsilon\} - \frac{1}{3K_g}\{1\}\,d\pi\right) \tag{2.51}$$

The last term in Equation 2.51 is the strain in the skeleton resulting from compression of the solid grains by the pore pressure.

Continuity Equation of Pore Fluid Flow

The continuity equation for pore fluid flow is derived from mass conservation relationships. The volumetric strain of the pore fluid, ε_f, is given by

$$d\varepsilon_f = -\frac{d\rho_f}{\rho_f} = C_f\,d\pi \tag{2.52}$$

where

C_f = Pore fluid compressibility

π = Pore fluid pressure

The volume strain of the solid grains, ε_g , is give by

$$d\varepsilon_g = -\frac{d\rho_g}{\rho_g} = C_g\, d\pi + \frac{C_g}{1-n}\, dp' \tag{2.53}$$

where

C_g = Bulk compressibility of solid grains

p' = Effective mean pressure

The dry density, ρ_d , is given by

$$\rho_d = \frac{m_g}{V_t} = (1-n)\,\rho_g \tag{2.54}$$

where m_g is the mass of the solid grains in skeleton volume V_t . The change in dry density is given by

$$d\rho_d = -\rho_d\, d\varepsilon_v \tag{2.55}$$

where ε_v is the volumetric strain of the skeleton. Differentiating Equation 2.54 with respect to n and ρ_g gives

$$d\rho_d = (1-n)\, d\rho_g - \rho_g\, dn \tag{2.56}$$

Equating 2.55 and 2.56 yields

$$d\varepsilon_v = \frac{dn}{1-n} - \frac{d\rho_g}{\rho_g} \tag{2.57}$$

Conservation of mass for the pore fluid within a specified initial volume of saturated porous material is given by

$$n \ \rho_f \ V_t \ = \ \bar{n} \ \bar{\rho}_f \ \bar{V}_t \tag{2.58}$$

where as illustrated in Figure 2.2, the terms to the left of the equal sign represent the fluid mass under the initial conditions and the terms to the right represent the same fluid mass under deformed conditions.

Equation 2.58 may be expressed in infinitesimal incremental form as

$$n \ \rho_f \ V_t \ = \ (n + dn) \ (\rho_f + d\rho_f) \ (1 + d\epsilon_F) \ V_t \tag{2.59}$$

where
ϵ_F = Volumetric diffusion of pore fluid as depicted in Figure 2.2

Solving Equation 2.59 for $d\epsilon_F$ and discarding second order terms yields

$$d\epsilon_F \ = \ - \frac{dn}{n} - \frac{d\rho_f}{\rho_f} \tag{2.60}$$

Equation 2.60 is combined with Equation 2.57 by elimination of dn to yield

$$(1-n) \ d\epsilon_v + n \ d\epsilon_F + (1-n) \frac{d\rho_g}{\rho_g} + n \frac{d\rho_f}{\rho_f} \ = \ 0 \tag{2.61}$$

Combining Equations 2.52 and 2.53 with 2.61 gives

$$n \ (d\epsilon_F - d\epsilon_v) + d\epsilon_v - \frac{1}{K_m} d\pi - c_g \ dp' \ = \ 0 \tag{2.62}$$

where K_m is the bulk modulus of the solid/fluid mixture which is expressed by

$$K_m \ = \ \frac{1}{n \ C_f + (1-n) \ C_g} \tag{2.63}$$

The change in effective mean pressure is given by

$$dp' = K_s \,(d\epsilon_v - C_g \, d\pi) \tag{2.64}$$

Substituting Equation 2.64 into 2.62 gives

$$n\,(d\epsilon_F - d\epsilon_v) + (1 - C_g\, K_s)\, d\epsilon_v + \left(C_g^2\, K_s - \frac{1}{K_m}\right) d\pi = 0 \tag{2.65}$$

or

$$n\,(d\epsilon_F - d\epsilon_v) = \left(\alpha - \frac{C_g^2}{9}\,\{1\}^T\,[D^{ep}]\,\{1\}\right) d\pi - \left(\{1\}^T - \frac{C_g}{3}\,\{1\}^T\,[D^{ep}]\right)\{d\epsilon\} \tag{2.66}$$

Equation 2.66 can be expressed in the following convenient form:

$$d\pi = \bar{m}_2 \cdot d\epsilon_v + \bar{m} \cdot n\,(d\epsilon_F - d\epsilon_v) \tag{2.67}$$

where

$$\bar{m} = \frac{1}{\left[\dfrac{1}{K_m} - \dfrac{K_s^{ep}}{K_g^2}\right]} \tag{2.68}$$

$$\bar{m}_2 = \left[1 - \frac{K_s^{ep}}{K_g}\right] \cdot \bar{m} \tag{2.69}$$

Equation of Motion for the Bulk Mixture

The differential equation of motion governing the bulk mixture is expressed by equating the stress gradient to the inertial resistance as

$$\sigma_{ij,\,j} \;=\; (1-n)\ \rho_s\ \ddot{u}_i \;+\; n\ \rho_f\ \ddot{U}_i \tag{2.70}$$

$\sigma_{ij,j}$ is the total stress gradient applied to an infinitesimal element of saturated material at some given time. $\sigma_{ij,j}$ is expressed in tensor and represents the stress gradient in each of three mutually perpendicular coordinates (e.g. see Mendleson, 1968). For instance, in the x direction,

$$\sigma_{xj,\,j} \;=\; \frac{\partial\sigma_{xx}}{\partial x} + \frac{\partial\sigma_{xy}}{\partial y} + \frac{\partial\sigma_{xz}}{\partial z} \;=\; (1-n)\ \rho_s\ \ddot{u}_x \;+\; n\ \rho_f\ \ddot{U}_x \tag{2.71}$$

The term (1-n) ρ_s is the mass of the soil skeleton per unit volume of saturated material, where n is the porosity and ρ_s is the mass density of the solid grains. u_i is the displacement of the skeleton in the i direction and $\ddot{u}$ is the acceleration of the skeleton in the i direction. The term n ρ_f is the mass of pore fluid per unit volume of saturated material where ρ_f is the mass density of the pore fluid. U_i is the absolute displacement of the pore fluid in the i direction.

The bulk mass density of the saturated material, ρ, is given by

$$\rho \;=\; (1-n)\ \rho_s \;+\; n\ \rho_f \tag{2.72}$$

Substitution of the value for (1-n) ρ_s from Equation 2.72 into Equation 2.70 gives

$$\sigma_{ij,\,j} \;=\; (\rho \;-\; n\ \rho_f)\ \ddot{u}_i \;+\; n\ \rho_f\ \ddot{U}_i \tag{2.73}$$

A term w_i is introduced which is the apparent fluid displacement in the i direction relative to the soil skeleton and is given by

$$w_i \;=\; n\ (U_i \;-\; u_i) \tag{2.74}$$

In seepage problems, w_i , is referred to as the discharge displacement. It describes the discharge of fluid through a soil mass of unit area. The discharge velocity, or apparent relative velocity, $\dot{w}_I$, between the soil particles and pore water is the velocity of water in a discharge duct of unit area needed to maintain the actual relative velocity in the porous soil of the same unit area. The actual relative velocity between the skeleton and the pore water is given by $\dot{w}_i / n$. Finally, $\ddot{w}_i$ is the apparent relative acceleration between the soil skeleton and pore water given by

$$\ddot{w}_i = n\,(\ddot{U}_i - \ddot{u}_i) \tag{2.75}$$

Equation 2.73 can be expressed in terms of the apparent relative fluid acceleration as simply

$$\sigma_{ij,\,j} = \rho\,\ddot{u}_i + \rho_f\,\ddot{w}_i \tag{2.76}$$

Equation of Motion for Pore Fluid

The finite element code SMAP-3D is capable of calculating the flow of pore fluid between elements. The flow of fluid with respect to the skeleton is controlled by Forchheimer's permeability model as described in a series of reports to the Air Force Office of Scientific Research (Kim, et. al., 1986, 1987, 1988; Blouin et. al., 1990, 1991). The Forchheimer model, as described by Kim, et. al. (1988) can be expressed as:

$$\pi_{,i} = \frac{\rho_f\, g}{k}\,\dot{w}_i + \frac{\beta_f}{k^{1/2}}\,\dot{w}_i^2 + \rho_f\,\ddot{U}_i \tag{2.77}$$

where

$\pi_{,i}$	=	Pore pressure gradient
g	=	Acceleration of gravity
ρ_f	=	Mass density of pore fluid
k	=	Darcy's coefficient of permeability (function of skeleton and fluid properties)
β_f	=	Ward's turbulent flow coefficient (function of skeleton and fluid properties)
$\dot{w}$	=	Apparent flow velocity relative to the skeleton
$\ddot{U}$	=	Absolute acceleration of pore fluid

The first term in Equation 2.77 is simply Darcy's law while the velocity squared term was apparently first proposed by Forchheimer (1901). The first two terms represent the frictional component of the pressure gradient while the last term accounts for the inertial effect of fluid flow.

Equation 2.77 can also be written in the form:

$$\pi_{,i} = \frac{\rho_f \, g}{k'} \dot{w}_i + \rho_f \ddot{U}_i \tag{2.78}$$

where k' represents an equivalent permeability coefficient given by:

$$k' = \frac{k}{1 + \frac{\beta_f}{\rho_f \, g} \sqrt{k} \, |\dot{w}_i|} \tag{2.79}$$

Hence, the flow of pore fluid in the soil skeleton is governed by Equations 2.78 and 2.79 and the flow coefficients k and β_f which can be determined from laboratory test data. Using the Equation 2.75, Equation 2.78 can be expressed in terms of skeleton and apparent relative fluid motions given by

$$\pi_{,i} = \frac{\rho_f}{n} \ddot{w}_i + \rho_f \ddot{u}_i + k' \dot{w}_i \tag{2.80}$$

More recently, Blouin and his coworkers (1991) have proposed a refined expression of the Forchheimer model that, while remaining equivalent to Equation 2.77, clarifies the distinction between fluid-related and skeleton-related permeability properties. This new expression is:

$$\pi_{,i} = \frac{\mu}{\alpha} \dot{w}_i + \frac{\rho_f}{\beta} \dot{w}_i^2 + \rho_f \ddot{U}_i \tag{2.81}$$

where

μ	=	Dynamic viscosity of the fluid
α, β	=	Flow coefficients that are properties of the porous skeleton only

The conversions between the different permeability parameters are obtained from equating the corresponding terms of Equations 2.77 and 2.81 to obtain:

$$k = \frac{\alpha \rho_f g}{\mu} \tag{2.82}$$

$$\beta_f = \frac{k^{\frac{1}{2}} \rho_f}{\beta} \tag{2.83}$$

While the parameters α and β and Equation 2.81 form the preferred expression for the permeability model, the current implementation of the model in our numerical codes follow the form of Equations 2.77 through 2.80.

2.6 Spatial Discretization and Incremental Relationships of Field Variables

Within each element, field variables can be discretized into element nodal values.

$$\begin{aligned} \{\Delta w\} &= [N]\,\{\Delta \underline{w}\}_e \\ \{\Delta \varepsilon\} &= [B]\,\{\Delta \underline{u}\}_e \\ \Delta w_{i,i} &= \{1\}^T\,[B]\,\{\Delta \underline{w}\}_e \\ \{\Delta u\} &= [N]\,\{\Delta \underline{u}\}_e \end{aligned} \tag{2.84}$$

Stress vector at time step n can be expressed as:

$$\{\sigma_n\} = \{\sigma_{n-1}\} + \{\Delta\sigma'\} + \{1\}\,\Delta\pi \tag{2.85}$$

Combining Equations 2.50, 2.51, 2.67 and 2.84 yields

$$\begin{aligned}\{\Delta\sigma\} = {} & ([D^{ep}]\,[B] + \bar{m}_1\,\{1\}\,\{1\}^T\,[B])\,\{\Delta\underline{u}\} \\ & + \bar{m}_2\,\{1\}\,\{1\}^T\,[B]\,\{\Delta\underline{w}\}\end{aligned} \tag{2.86}$$

where

$$\bar{m}_1 = \left[1 - \frac{K_s^{ep}}{K_g}\right]^2 \cdot \bar{m} \tag{2.87}$$

Equation 2.67 can be rewritten in incremental form as:

$$\Delta\pi = \bar{m}_2 \cdot \Delta u_{i,i} + \bar{m} \cdot \Delta w_{i,i} \tag{2.88}$$

2.7 Global Equilibrium Equations

Two global equilibrium equations are derived, first in terms of field variables and then discretized using nodal variables.

The first equates the total internal stresses plus the inertia forces to the applied boundary traction. Letting the solid skeleton movement be the virtual displacement, δu, the following global equilibrium equation for the bulk mixture is established:

$$\begin{aligned}\int_v \{\delta\varepsilon\}^T\,\{\sigma\}\,dv = {} & \int_s \{\delta u\}^T\,\{T\}\,ds - \int_v \{\delta u\}^T\,\rho\,\{\ddot{u}\}\,dv \\ & - \int_v \{\delta u\}^T\,\rho_f\,\{\ddot{w}\}\,dv\end{aligned} \tag{2.89}$$

where

δε is the virtual strain corresponding to virtual displacement δu.

The second equates the applied pore pressure on the boundary to the internal pore pressure plus the flow resistance force plus the inertia force on the pore fluid. Taking the apparent relative fluid movement as the virtual displacement, δw, the internal virtual work done by the pore pressure should be equal to the external virtual work. That is,

$$\int_v (\delta w_{i,i})^T \pi \cdot dv = \int_s \{\delta w\}^T \hat{n}\, ds - \int_v \{\delta w\}^T [r] \cdot \{\dot{w}\}\, dv - \int_v \{\delta w\}^T \rho_f \{\ddot{u}\}\, dv - \int_v \{\delta w\}^T \frac{1}{n} \rho_f \{\ddot{w}\}\, dv \tag{2.90}$$

Replacing the field variables in Equation 2.89 and 2.90 by the discretized nodal variables using Equation 2.84 gives the following global equilibrium equation at time step n:

$$\begin{bmatrix} M_m & M_c \\ M_c^T & M_f \end{bmatrix} \begin{Bmatrix} \underline{\ddot{u}}_n \\ \underline{\ddot{w}}_n \end{Bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & H \end{bmatrix} \begin{Bmatrix} \underline{\dot{u}}_n \\ \underline{\dot{w}}_n \end{Bmatrix} + \begin{bmatrix} K_t + EE & C \\ C^T & E \end{bmatrix} \begin{Bmatrix} \Delta \underline{u}_n \\ \Delta \underline{w}_n \end{Bmatrix} = \begin{Bmatrix} F_n \\ G_n \end{Bmatrix} - \begin{Bmatrix} R^s_{n-1} + R^f_{n-1} \\ R^f_{n-1} \end{Bmatrix} \tag{2.91}$$

where

$$M_m = \Sigma \int_v [N]^T \rho [N]\, dv$$

$$M_c = \Sigma \int_v [N]^T \rho_f [N]\, dv$$

$$M_f = \Sigma \int_v [N]^T \frac{1}{n} \rho_f [N]\, dv$$

$$H = \Sigma \int_v [r] [N]^T [N]\, dv$$

$$K_t = \Sigma \int_v [B]^T [D^{ep}] [B]\, dv$$

$$EE = \Sigma \int_v \bar{m}_1 [B]^T \{1\} \{1\}^T [B] \, dv$$

$$C = \Sigma \int_v \bar{m}_2 [B]^T \{1\} \{1\}^T [B] \, dv$$

$$F_n = \Sigma \int_s [N]^T \{T\} \, ds + \Sigma \int_v [N]^T \rho \{b\} \, dv$$

$$E = \Sigma \int_v \bar{m} [B]^T \{1\} \{1\}^T [B] \, dv$$

$$G_n = \Sigma \int_s [N]^T \hat{\pi}_n \, ds + \Sigma \int_v [N]^T \rho_f \{b\} \, dv$$

$$R^s_{n-1} = \Sigma \int_v [B]^T \{\sigma'_{n-1}\} \, dv$$

$$R^f_{n-1} = \Sigma \int_v [B]^T \{1\} \pi_{n-1} \, dv$$

$[r]$ = Inverse of permeability matrix

$\{b\}$ = Component of body force vector

Equation 2.91 can be rewritten in the simpler form:

$$[M] \{\ddot{\underline{d}}_n\} + [D] \{\dot{\underline{d}}_n\} + [K] \{\Delta \underline{d}_n\} = \{P_n\} - \{R_{n-1}\} \tag{2.92}$$

2.8 Linearized Global Equilibrium Equations

Introducing a time integration method which incorporates both Newmark's β method and Wilson's θ method, the generalized acceleration vector is expressed as

$$\{ \ddot{\underline{d}}_n \} = C_1 \{ \Delta \underline{d}_n \} + C_2 \{ \dot{\underline{d}}_{n-1} \} + C_3 \{ \ddot{\underline{d}}_{n-1} \} \tag{2.93}$$

where

$$C_1 = \frac{1}{\beta \, \theta^3 \, \Delta t^2}$$

$$C_2 = - \frac{1}{\beta \, \theta^2 \, \Delta t} \tag{2.94}$$

$$C_3 = 1 - \frac{1}{2 \, \beta \, \theta}$$

and the generalized velocity vector is expressed as

$$\{ \dot{\underline{d}}_n \} = B_1 \{ \Delta \underline{d}_n \} + B_2 \{ \dot{\underline{d}}_{n-1} \} + B_3 \{ \ddot{\underline{d}}_{n-1} \} \tag{2.95}$$

where

$$B_1 = \frac{\gamma}{\beta \, \theta^3 \, \Delta t}$$

$$B_2 = 1 - \frac{\gamma}{\beta \, \theta^2} \tag{2.96}$$

$$B_3 = \Delta t - \frac{\gamma}{2 \, \beta \, \theta} \Delta t$$

Substituting Equations 2.93 and 2.95 into Equation 2.92 and rearranging, we can obtain the following linearized global equilibrium equations which can be solved simultaneously at each step:

$$[\tilde{K}]\,\{\Delta \underline{d}_n\} = \{\tilde{P}_n\} \tag{2.97}$$

where the generalized stiffness matrix is given by

$$[\tilde{K}] = C_1\,[M] + B_1\,[D] + [K] \tag{2.98}$$

and the generalized force vector is given by

$$\begin{aligned}\{\tilde{P}_n\} = \{P_n\} - \{R_{n-1}\} - [M]\,(C_2\,\{\dot{\underline{d}}_{n-1}\} + C_3\,\{\ddot{\underline{d}}_{n-1}\}) \\ - [D]\,(B_2\,\{\dot{\underline{d}}_{n-1}\} + B_3\,\{\ddot{\underline{d}}_{n-1}\})\end{aligned} \tag{2.99}$$

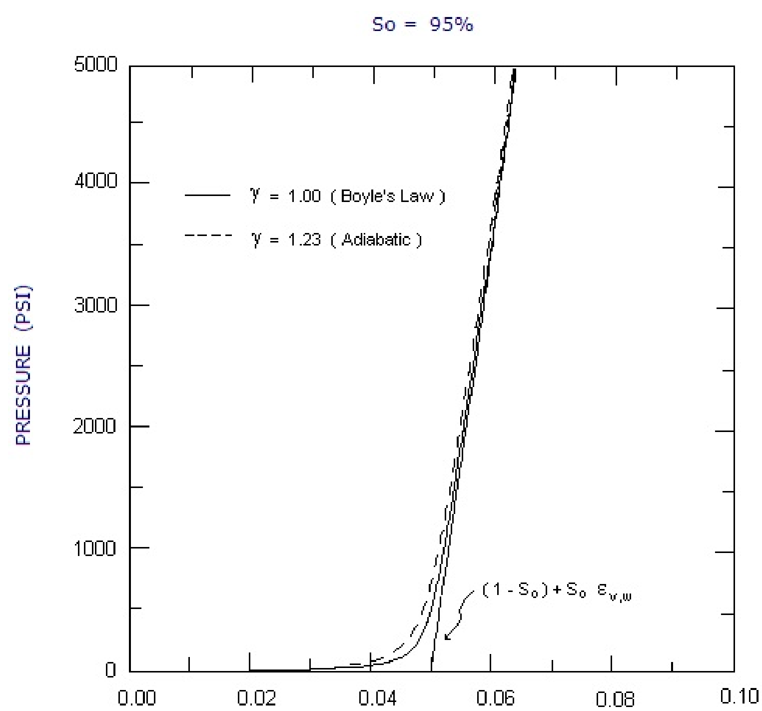


Figure 2.1 Prediction of air-water compressibility

Conservation of Fluid Mass

$$n \ \ \rho_f \, v_t = n' \, \rho_f' \, v_t'$$

v_t	=	Apparent fluid volume before compression
v_t'	=	$(1+\varepsilon_F)\, v_t$: apparent fluid volume after compression
ε_v	=	Volumetric strain of porous skeleton
ε_F	=	Volumetric diffusion of pore fluid

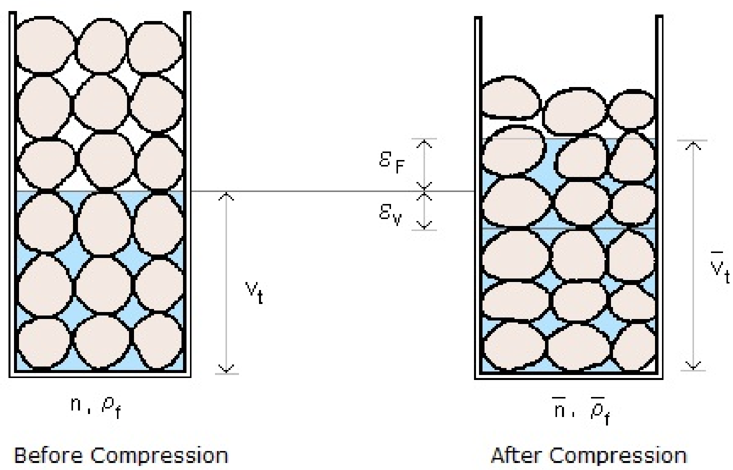


Figure 2.2 Schematic illustration of conservation of pore fluid mass in saturated porous materials

Nonlinear Material Models

3.1 Generalized Hoek and Brown Model

3.1.1 Introduction

Generalized Hoek and Brown Model represents the skeleton constitutive relations of soils or porous materials. In its generalized form, the model includes the empirically based Hoek and Brown failure equation as well as the classical Von Mises, Mohr-Coulomb, and Drucker-Prager failure equations. As one of the useful features, the model can use empirical data base for the strength of in situ rock mass when the in situ strength data are not available.

In this section, the 3-dimensional elasto-plastic matrix is derived for the Generalized Hoek and Brown Model. The model is elastic below the failure surface and perfectly plastic along the failure surface with the volumetric and deviatoric behaviors dependent upon one another once the failure surface is reached.

And the failure equation is expressed in terms of the alternate stress invariant (p, q, and θ) given by

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

$$
\begin{aligned}
S_{ij} &= \sigma_{ij} - p \cdot \delta_{ij} \\
J_2 &= \frac{1}{2} S_{ij} S_{ij} \\
J_3 &= \frac{1}{3} S_{ij} S_{jk} S_{ki} \\
q &= \sqrt{3J_2} \\
\theta &= \frac{1}{3} \mathrm{Sin}^{-1} \left(-\frac{27}{2} \frac{J_3}{q^3} \right)
\end{aligned} \tag{3.1}
$$

where σ_{ij} is the total stress tensor and S_{ij} is the deviatoric stress tensor.

3.1.2 Elastic Stress-Strain Relationship

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

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

where

$\{d\sigma\}$	Stress increment
$[D^e]$	Elastic stress-strain matrix
$\{d\varepsilon^e\}$	Elastic strain increment

3.1.3 Failure Surface

The failure surface is described by the following equation:

$$F(p, q, \theta) = q - ((\alpha + \beta p)^n + \kappa)\, R(\theta) = 0 \tag{3.3}$$

The expression for R(θ) in Equation 3.3 is given by

$$R(\theta) = \frac{x\,(\sqrt{3}\,\cos\theta + \sin\theta) + (2k-1)\,[(2+\cos2\theta+\sqrt{3}\,\sin2\theta)\,x + 5k^2-4k]^{1/2}}{[\,x\,(2+\cos2\theta+\sqrt{3}\,\sin2\theta) + (1-2k)^2\,]} \tag{3.4}$$

where

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

x = $(1-k^2)$

k = the ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same mean pressure

The function R(θ) describes the shape of the yield surface, as projected in the π plane (octahedral plane). Figure 3.1 and 3.2 show the influence of the parameter k on the shape of the yield surface. k is the ratio of the shear strength in triaxial extension to the shear strength in triaxial compression at the same mean pressure. k is a measure of the influence of the intermediate principal stress on the yield surface and can vary from 0.5 to 1.0. When k is equal to unity, R(θ) is circular, indicating a Drucker-Prager or Von Mises failure model. When k is less than unity, R(θ) is a smooth cornered approximation to the Mohr-Coulomb failure envelope.

The parameter n in Equation 3.3 determines the shape of the yield surface in the p-q plane. For n=0, the shear strength is constant with respect to the mean pressure and the strength envelope reduces to the Von Mises or Tresca yield surface. For n=1/2, the strength envelope represents Hoek and Brown (1982) failure surface. This nonlinear failure model is a multidimensional generalization of the original one-dimensional axisymmetric Hoek and Brown model which is based on extensive laboratory and field data (Kim, Piepenburg and Merkle, 1986).

For n = 1, shear strength is linearly proportional to the mean pressure and the strength envelope in the p-q plane is representative of the Drucker-Prager or Mohr-Coulomb failure surface.

The parameters α, β and κ of Equation 3.3 define the failure envelope in the p-q plane. They can be determined from laboratory tests. Recommended relationships for determining these parameters for Von Mises, Hoek and Brown and Mohr-Coulomb type materials are listed in Table 3.1. The empirical material parameters for n=1/2 are tabulated in Table 3.2 for several different rock types as a function of rock quality. Detailed description of rock quality is shown in Table 3.3.

3.1.4 Flow Rule

A variable dilatancy potential function, G, is defined such as

$$\frac{\partial G}{\partial p} = \left(\frac{\partial F}{\partial p}\right) r$$
$$\frac{\partial G}{\partial q} = \frac{\partial F}{\partial q} \qquad (3.5)$$
$$\frac{\partial G}{\partial \theta} = \frac{\partial F}{\partial \theta}$$

where r is a dilatancy parameter ($0 \le r \le 1$)

r = 0 No plastic volume change
= 1 Associated flow

Thus, in general,

$$\{d\varepsilon^p\} = d\lambda \{g\} \qquad (3.6)$$

where

$$\{g\} = \left\{\frac{\partial G}{\partial \sigma}\right\}$$

3.1.5 Consistency Equation

During yielding , the consistency equation forces the stress to move along the failure surface

$$dF = \{a\}^T \{d\sigma\} = 0 \tag{3.7}$$

where

$$\{a\} = \left\{\frac{\partial F}{\partial \sigma}\right\} \tag{3.8}$$

3.1.6 Incremental Elasto-Plastic Constitutive Law

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

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^p\} \tag{3.9}$$

Substituting Equation 3.9 into 3.2, we have

$$\{d\sigma\} = [D^e] (\{d\varepsilon\} - \{d\varepsilon^p\}) \tag{3.10}$$

From the flow rule defined in Equation 3.6, we can rewrite Equation 3.10 as

$$\{d\sigma\} = [D^e] \{d\varepsilon\} - d\lambda [D^e] \{g\} \tag{3.11}$$

Substituting Equation 3.11 into 3.7 and solving for $d\lambda$, we obtain

$$d\lambda = \frac{\{a\}^T [D^e] \{d\varepsilon\}}{\{a\}^T [D^e] \{g\}} \tag{3.12}$$

Back substituting Equation 3.12 into Equation 3.11, the stress increment is directly related to the total strain increment as follows:

$$\{d\sigma\} = [D^{ep}]\{d\varepsilon\} \tag{3.13}$$

where

$$[D^{ep}] = [D^{e}] - \frac{[D^{e}]\{g\}\{a\}^T[D^{e}]}{\{a\}^T[D^{e}]\{g\}} \tag{3.14}$$

3.1.7 Calculation of {a}

Differentiating the yield function with respect to p, q, and θ, we have

$$\begin{aligned}
\frac{\partial F}{\partial p} &= -n(\alpha + \beta p)^{n-1}\cdot\beta\cdot R(\theta) \\
\frac{\partial F}{\partial q} &= 1 \\
\frac{\partial F}{\partial \theta} &= -\{(\alpha + \beta p)^{n} + \kappa\}\frac{\partial R(\theta)}{\partial \theta}
\end{aligned} \tag{3.15}$$

where

$$\begin{aligned}
\frac{\partial R}{\partial \theta} &= \frac{1}{R_D}\left[\frac{\partial R_N}{\partial \theta} - R(\theta)\frac{\partial R_D}{\partial \theta}\right] \\
R_N &= x(\sqrt{3}\cos\theta + \sin\theta) + (2k-1)\left[(2+\cos 2\theta + \sqrt{3}\sin 2\theta)x + 5k^2 - 4k\right]^{1/2} \\
R_D &= x(2+\cos 2\theta + \sqrt{3}\sin 2\theta) + (1-2k)^2 \\
\frac{\partial R_N}{\partial \theta} &= x(\cos\theta - \sqrt{3}\sin\theta) + \frac{x(2k-1)(\sqrt{3}\cos 2\theta - \sin 2\theta)}{\left[x(2+\cos 2\theta + \sqrt{3}\sin 2\theta) + 5k^2 - 4k\right]^{1/2}} \\
\frac{\partial R_D}{\partial \theta} &= 2x(\sqrt{3}\cos 2\theta - \sin 2\theta)
\end{aligned} \tag{3.16}$$

The derivative of the yield function with respect to stress can be written in general 3-dimensional condition as

$$\{a\} = \frac{\partial F}{\partial p}\left\{\frac{\partial p}{\partial \sigma}\right\} + \frac{\partial F}{\partial q}\left\{\frac{\partial q}{\partial \sigma}\right\} + \frac{\partial F}{\partial \theta}\left\{\frac{\partial \theta}{\partial \sigma}\right\} \tag{3.17}$$

where

$$\left\{\frac{\partial p}{\partial \sigma}\right\} = \frac{1}{3} < 1\ 1\ 1\ 0\ 0\ 0 >^T$$

$$\left\{\frac{\partial \theta}{\partial \sigma}\right\} = \frac{9}{2q^3 \cos 3\theta}\left(\frac{3J_3}{q}\left\{\frac{\partial q}{\partial \sigma}\right\} - \left\{\frac{\partial J_3}{\partial \sigma}\right\}\right)$$

$$\left\{\frac{\partial q}{\partial \sigma}\right\} = \frac{3}{2q} < S_x\ \ S_y\ \ S_z\ \ 2\sigma_{xy}\ \ 2\sigma_{yz}\ \ 2\sigma_{xz} >^T$$

$$\left\{\frac{\partial J_3}{\partial \sigma}\right\} = \begin{Bmatrix} S_y S_z - \sigma_{yz}^2 + \frac{1}{9}q^2 \\ S_x S_z - \sigma_{xz}^2 + \frac{1}{9}q^2 \\ S_x S_y - \sigma_{xy}^2 + \frac{1}{9}q^2 \\ 2(-S_z\sigma_{xy} + \sigma_{yz}\sigma_{xz}) \\ 2(-S_x\sigma_{yz} + \sigma_{xz}\sigma_{xy}) \\ 2(-S_y\sigma_{xz} + \sigma_{xy}\sigma_{yz}) \end{Bmatrix}$$

$$\{\sigma\}^T = < \sigma_x\ \ \sigma_y\ \ \sigma_z\ \ \sigma_{xy}\ \ \sigma_{yz}\ \ \sigma_{xz} >$$

$$\{\varepsilon\}^T = < \varepsilon_x\ \ \varepsilon_y\ \ \varepsilon_z\ \ \gamma_{xy}\ \ \gamma_{yz}\ \ \gamma_{xz} >$$

$$\gamma_{xy} = 2\,\varepsilon_{xy} \qquad \gamma_{yz} = 2\,\varepsilon_{yz} \qquad \gamma_{xz} = 2\,\varepsilon_{xz}$$

Table 3.1 Material Constants in Grenerlized Hoek and Brown Model

	n = 0 Von Mises or Tresca	n = 1/2 Hoek and Brown	n = 1 Mohr-Coulomb or Drucker-Prager
α	N/A	$\left(\frac{m^2}{36} + s\right) \sigma_c^2$	1000
β	N/A	$m\ \sigma_c$	$\frac{6 \sin\phi}{(3 - \sin\phi)}$
κ	$q' - 1$	$\frac{1}{6}\ m\ \sigma_c$	$\frac{3(1 - \sin\phi)}{(3 - \sin\phi)} \sigma_c - 1000$

q' = $\sigma_1 - \sigma_3$
where σ_1 and σ_3 are major and minor pricipal stresses at failure.

σ_c = Unconfined compressive strength

ϕ = Internal friction angle

m,s = Hoek and Brown's material constants as tabulated in Table 3.2.

Table 3.2 Hoek and Brown Material Parameters (m, s)

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

Table 3.3 Description of Rock Quality in Table 3.2

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

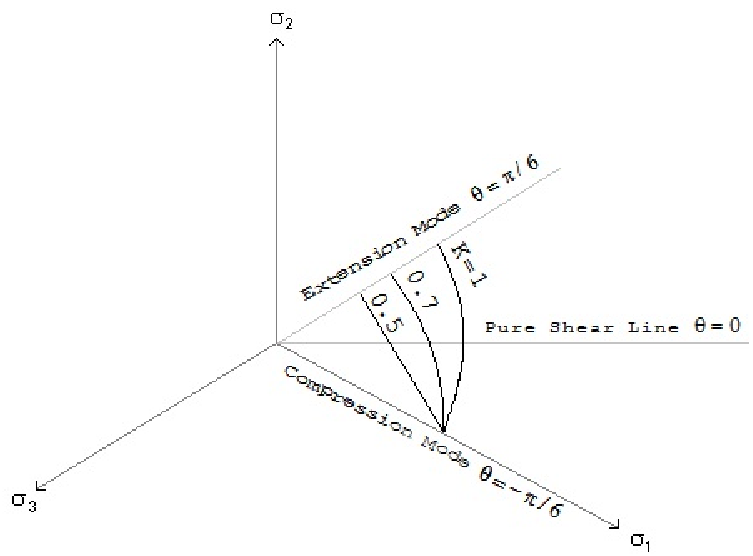


Figure 3.1 Shape of strength envelope, R(θ), on octahedral plane

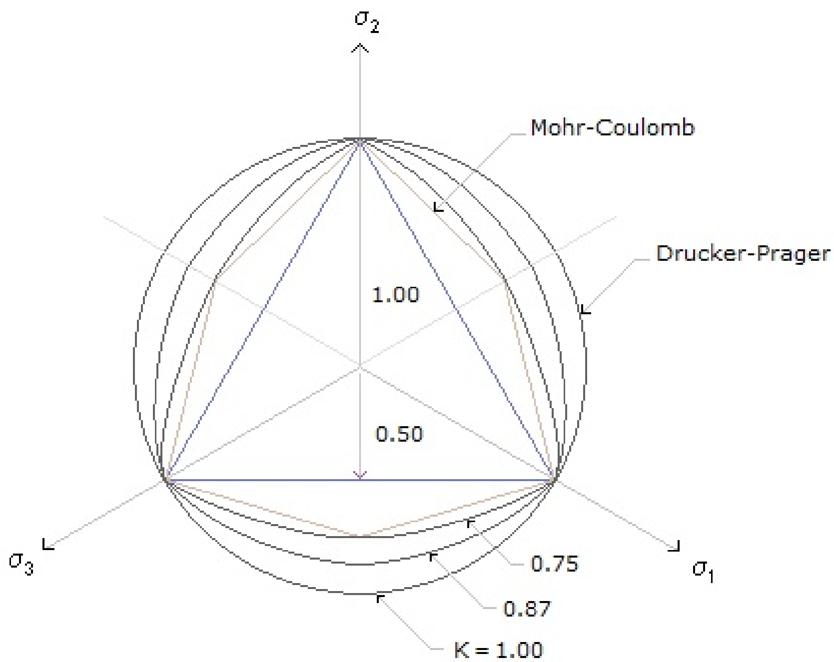


Figure 3.2 Shape of strength envelope, R(θ), on octahedral plane

3.2 Single Hardening Elasto-Plastic Model

3.2.1 Introduction

The Single Hardening Elasto-Plastic Model is a 3 invariant, single hardening surface, material model representing the drained response of the porous skeleton. The Single Hardening Model is the simplified version of the existing three invariant model which has been continuously upgraded since 1985 and reported by Merkle and Dass (1985), Dass and Merkle (1986), and Blouin, Chitty, Rauch, and Kim (1990). The major change from the existing three invariant model is the replacement of two hardening yield surfaces by the single hardening yield surface developed by Lade (1990). The advantages of this new model over the three invariant model include requirement of a lesser number of material constants, simple procedures of material parameter determination, and computational efficiency.

The Single Hardening Elasto-Plastic Model is a non-associated, isotropic, work hardening, elasto-plastic model with a single hardening yield surface bounded by a failure envelope, as shown in Figure 3.3. The yield surface has the shape of a teardrop with its pointed apex at the origin in principal stress space. The failure surface is a hyperboloid with its apex on the hydrostatic axis in the principal stress space. The shape of both yield and failure surfaces in the π-plane, perpendicular to the hydrostatic axis (see Figure 3.3), is a triple ellipse in polar coordinates.

3.2.2 Notations

Positive signs are used throughout this section to represent compression. Only those symbols which are not explicitly defined in the main text will be described below.

P_a	Atmospheric pressure
$\{\varepsilon\}$	Total strain vector
$\{\varepsilon_e\}$	Elastic strain vector
$\{\varepsilon_p\}$	Plastic strain vector associated with yield surface
$\{\varepsilon_u\}$	Plastic strain vector associated with failure surface

$\{\sigma\}$	Stress vector
σ_{oct}	Octahedral normal stress
τ_{oct}	Octahedral shear stress
ν	Poisson's ratio

3.2.3 Total Strain Formulation

In general, the total strain is comprised of the elastic component and two plastic components related to the yield and failure surfaces as given respectively by:

$$\{d\varepsilon\} = \{d\varepsilon_e\} + \{d\varepsilon_p\} + \{d\varepsilon_u\} \tag{3.18}$$

3.2.4 Elastic Response

At stress states inside the yield surfaces, the skeleton response is treated as nonlinear elastic and governed by the previous maximum peak stress. Two options are available for modeling the elastic response within the framework of the three invariant skeleton model: the modified elastic model and the Lade and Nelson elastic model. In both options, Poisson's ratio is assumed to remain constant.

Modified elastic model

During virgin unloading, the elastic bulk modulus is given by:

$$K = \frac{K_{ur}\,P_a}{3(1-2\nu)}\left[\frac{\sigma_{oct}}{P_a}\right]^n \geq K_i \tag{3.19}$$

where K_{ur} and n are material constants obtained in the parameter fitting. K_i represents the initial bulk modulus at low pressures and is necessary for modeling the behavior of rock-type materials that have a definite initial elastic behavior. In uncemented soils, K_i can be taken as a very small value. The initial bulk modulus is also used to determine the initial position of the yield surface by defining the initial elastic range.

During unloading or reloading, the skeleton modulus is described by one of two segments as depicted in Figure 3.4. Between the previous peak mean stress, $\sigma_{oct,max}$, and the transition into the nonlinear segment at $\sigma_{oct,\ b,}$ the elastic bulk modulus is constant and is given by:

$$K = K_1 = \frac{K_{ur}\ P_a}{3(1-2\nu)}\left[\frac{\sigma_{oct,\ max}}{P_a}\right]^n \tag{3.20}$$

The transition into the nonlinear segment occurs at:

$$\sigma_{oct,b} = \lambda\ \sigma_{oct,\ max} \tag{3.21}$$

where λ is a model parameter. At mean stresses less than $\sigma_{oct,\ b}$ the nonlinear bulk modulus is given by:

$$K = K_1\left[1 - \gamma\left[\frac{\beta}{\gamma}\right]^{\frac{\sigma_{oct}}{\sigma_{oct,b}}}\right] \tag{3.22}$$

Referring to Figure 3.4, the model parameters γ and β are given by:

$$\gamma = 1 - \frac{K_o}{K_1} \tag{3.23}$$

where K_0 is the bulk modulus at zero pressure and

$$\beta = 1 - \frac{K_*}{K_1} \tag{3.24}$$

where K_* is the bulk modulus at one quarter of the transition pressure $\sigma_{oct,b}$. While this formulation allows for relatively accurate curve fitting of observed soil response, the model has three disadvantages:

1. For certain closed-loop stress/strain paths, the model may violate the energy conservation principle;
2. Unloading at low pressures could potentially generate expansive volumetric strains; and
3. At the transition pressure, $\sigma_{oct,b}$ the modulus is not continuous.

Lade and Nelson elastic model

The second elastic model option is based on a relationship derived by Lade and Nelson (1987). This formulation is continuous and was derived from the energy conservation principle. Lade and Nelson's model can be expressed as:

$$K = \frac{K_{ur} P_a}{3^{n+1}(1-2\nu)} \left[\left[\frac{3\sigma_{oct}}{P_a} \right]^2 + \frac{6(1+\nu)}{1-2\nu} \frac{J_2^1}{P_a^2} \right]^{\frac{n}{2}} \geq K_i \qquad (3.25)$$

where the parameters K_{ur}, n, and K_i are the same as used in Equation 3.19. Since this model is fit strictly using the slope of an initial unload curve, it can be difficult to closely match the observed characteristics of an unload cycle.

Fitting

Poisson's ratio (ν) for a given material can be determined in a number of ways using unload/reload data which represent the elastic response of the skeleton. Lade and Nelson (1987) recommended obtaining Poisson's ratio directly from strain measurements in triaxial compression unload/reload cycles, right after stress reversal at hydrostatic conditions where:

$$\nu = -\frac{\varepsilon_r}{\varepsilon_a} = \frac{1}{2}\left[1 - \frac{\varepsilon_v}{\varepsilon_a}\right] \qquad (3.26)$$

where

ε_a	Axial strain
ε_r	Radial strain
ε_v	Volume strain

In addition, since the elastic response is completely defined by any two independent elastic parameters, Poisson's ratio can be obtained from the bulk modulus, (K), measured in a hydrostatic compression unload, and any other elastic modulus. For example, a triaxial compression unload yields the shear modulus (G), an unconfined compression unload gives the

Young's modulus (E), and an uniaxial strain unload produces the constrained modulus (M). Any one of these parameters can be used with the bulk modulus to obtain Poisson's ratio:

$$\nu = \frac{3K - 2G}{2(3K + G)} \tag{3.27}$$

$$\nu = \frac{3K - E}{6K} \tag{3.28}$$

$$\nu = \frac{3K - M}{3K + M} \tag{3.29}$$

To obtain the elastic model parameters K_{ur} and n, Equation 3.19 is rewritten in the form:

$$\log\left[\frac{3K(1-2\nu)}{P_a}\right] = \log K_{ur} + n \log\left[\frac{\sigma_{oct}}{P_a}\right] \tag{3.30}$$

Values of K and σ_{oct} from the initial unloading response at various pressures in the hydrostatic compression test, are then plotted as log $(3K(I\text{-}2\nu)/P_a)$ versus log (σ_{oct}/P_a). A least squares linear regression is then applied in log-log space. The parameter n is the slope of this line, while K_{ur} is the intercept where (σ_{oct}/P_a) is 1.0. The parameters λ, γ, and β for the modified elastic unload model are determined from a single unload/reload cycle in the hydrostatic compression test as depicted in Figure 3.4. The parameters are computed using Equations 3.21, 3.23, and 3.24.

3.2.5 Failure Surface

The failure surface is a hyperboloid with its apex on the hydrostatic axis in principal stress space as shown in Figure 3.3. The shape of the failure surface in the π -plane, perpendicular to the hydrostatic axis is a triple ellipse in polar coordinates.

The failure criteria are given by:

$$f_u = \frac{\tau_{oct}}{R(\theta)}\left(\frac{m}{P_a} + \frac{1}{\bar{\sigma}_{oct}}T\right) - \eta_1 = 0 \tag{3.31}$$

where

$$R(\theta) = \frac{2K}{(1 + K) + (1 - K)\,\text{Sin}3\theta} \tag{3.32}$$

$$\bar{\sigma}_{oct} = \sigma_{oct} + T \tag{3.33}$$

σ_{oct}	Octahedral normal stress
τ_{oct}	Octahedral shear stress
θ	Lode angle
T	Tensile strength
K	The ratio of extensive to compressive strength at given mean pressure

m and η_1 are the failure constants which can be determined from the following fitting procedure. In triaxial compression mode, $R(\theta) = 1$ and Equation 3.31 reduces to:

$$\frac{\bar{\sigma}_{oct}}{\tau_{oct}} = \frac{1}{\eta_1} + \frac{m}{\eta_1}\left(\frac{\bar{\sigma}_{oct}}{P_a}\right) \tag{3.34}$$

By plotting the failure stress points from each triaxial compression test in terms of $\bar{\sigma}_{oct}/\tau_{oct}$ versus $\bar{\sigma}_{oct}/P_a$, a straight line fit will yield an intercept of $1/\eta_1$ and a slope of m/η_a. Then the parameter η_1 is obtained simply by taking the inverse value of intercept and the parameter m is obtained by multiplying the slope by η_1.

3.2.6 Plastic Response Related to Yield Surface

Both yield and potential equations are based on Lade's single hardening model (Lade, 1990) which replaces previous two yield surface model (Lade, 1977).

To be consistent with the failure equation described in the previous subsection, however, Lade's equations were modified such that the shape of both yield and potential surfaces in the π-plane consists of triple ellipse given by Equation 3.32.

The yield equation is composed of the stress function (f_p') and the hardening function (f_p'').

$$f_p = f_p'(I_1, J_2, \theta) - f_p''(W_p) = 0 \tag{3.35}$$

The stress function is given by:

$$f_p' = \left(\psi_1 \frac{\bar{I}_1^{\,3}}{\bar{I}_3} - \frac{\bar{I}_1^{\,2}}{\bar{I}_2}\right)\left[\frac{\bar{I}_1}{p_a}\right]^h - e^q \tag{3.36}$$

where the stress quantities I_1, I_2, and I_3 are defined by:

$$\bar{I}_1 = I_1 + 3T \tag{3.37}$$

$$\bar{I}_2 = \left(\frac{J_2}{R(\theta)^2}\right) - \frac{\bar{I}_1^{\,3}}{3} \tag{3.38}$$

$$\bar{I}_3 = 2\left[\frac{J_2}{3R(\theta)^2}\right]^{\frac{3}{2}} - \frac{\bar{I}_1}{3}\left(\frac{J_2}{R(\theta)^2}\right) + \frac{\bar{I}_1^{\,3}}{27} \tag{3.39}$$

Note that I_1 is the first invariant of the total stress tensor, J_2 is the second invariant of deviatoric stress tensor and $R(\theta)$ is given by Equation 3.32. The variable q in Equation 3.36 is related to the shear stress level S as:

$$q = \frac{\alpha \cdot s}{1 - (1 - \alpha) \cdot s} \tag{3.40}$$

The shear stress level is defined as:

$$S = \frac{\frac{\tau_{oct}}{R_{(\theta)}}\left(\frac{m}{P_a} + \frac{1}{\bar{\sigma}_{oct}}\right)}{\eta_1} \tag{3.41}$$

The parameter q has the value of zero along the hydrostatic axis and unity along the failure surface. Thus, the material constants which are specific to the stress function are ψ_1, h and α. Determination of these material constants will be described at the end of this subsection.

The hardening function is given by:

$$f_p'' = \left[\frac{W_p}{D \cdot P_a}\right]^{\frac{h}{p}} \tag{3.42}$$

where the plastic work is expressed as:

$$W_p = \int \{\sigma\}^T \{d\epsilon_p\} \tag{3.43}$$

and the constant D is related to the isotropic hardening constants (C and P) as:

$$D = \frac{C}{(27\psi_1 + 3)^{P/h}} \tag{3.44}$$

The constant ψ_1 in Equation 3.36 and 3.44 is assumed to depend on the type of material.

Material Type	ψ_1
Sand	0.018
Clay	0.006
Mortar	0.004
Sandstone	0.0013
Concrete	0.0015
Reinforced Concrete	0.0007

It should be noted that the values of ψ_1 in the above table are based on Lade's data (Kim and Lade, 1988) but ψ_1 does not have any influence on the shape of yield surfaces on the π-plane.

Isotropic hardening constants (C and P) can be determined by fitting to the isotropic compression test. For the isotropic compression loading, Equation 3.35 reduces to:

$$\frac{W_p}{P_a} = C\left[\frac{I_1}{P_a}\right]^p \qquad (3.46)$$

Taking the logarithm of both sides of Equation 3.46 yields

$$\log_{10}\left(\frac{W_p}{P_a}\right) = \log_{10}C + P\log_{10}\left(\frac{I_1}{P_a}\right) \qquad (3.47)$$

so that the parameters C and P can be found from a log-log plot of (W_p/P_a) versus (I_1/P_a).

Yield constants (h and α) can be obtained by fitting to the triaxial compression test data. Along the isotropic and triaxial compression stress paths, Equation 3.35 has the same form as the Lade's single hardening yield equation so that the same procedure as described by Lade and Kim, 1988 can be used to determine yield constants (h and α).

The yield constant h can be obtained from:

$$h = \frac{\log_{10}\left(\frac{\left(\Psi_1 \frac{I_{1F}^3}{I_{3F}} - \frac{I_{1F}^2}{I_{2F}}\right)e}{27\psi_1 + 3}\right)}{\log_{10}\left(\frac{I_{1H}}{I_{1F}}\right)} \quad (3.48)$$

where I_{IF}, I_{2F}, and I_{3F} are the first, second and third invariant of the total stress tensor, respectively, at the failure point of triaxial compression test; I_{1H} is the first invariant of the total stress tensor in the hydrostatic compression test, measured at the same plastic work as for the failure point of triaxial compression test.

The yield constant α can be determined from

$$\alpha = \frac{1}{4} \frac{q_{80}}{1 - q_{80}} \quad (3.49)$$

Where q_{80} is value of q at the stress level S= 0.8 and is obtained from

$$q = l_n \frac{\left[\frac{W_p}{D P_a}\right]^{\frac{h}{p}}}{\left(\psi_1 \frac{I_1^{\,3}}{I_3} - \frac{I_1^{\,2}}{I_2}\right)\left[\frac{I_1}{P_a}\right]^h} \quad (3.50)$$

The potential equation is expressed in terms of stress invariants as

$$g_p = \left(\psi_1 \frac{\bar{I}_1^{\,3}}{\bar{I}_3} - \frac{\bar{I}_1^{\,2}}{\bar{I}_2} + \psi_2\right)\left[\frac{\bar{I}_1}{P_a}\right]^{\mu} \quad (3.51)$$

Material constants (ψ_2 and μ) which are specific to the potential surface can be determined in the same way as for Lade's single hardening model (Kim and Lade, 1988).

For the triaxial compression test, the potential constants (ψ_2 and μ) are related to the stresses and the plastic strains as

$$\xi_y = \frac{1}{\mu}\xi_x - \psi_2 \tag{3.52}$$

where

$$\xi_x = \psi_1 \frac{I_1^{\ 3}}{I_3} - \frac{I_1^{\ 2}}{I_2} \tag{3.53}$$

$$\xi_y = \frac{1}{1+\upsilon_p}\left(\frac{I_1^{\ 3}}{I_2^2}(\sigma_a + \sigma_r + 2\upsilon_p\sigma_r) + \psi_1\frac{I_1^{\ 4}}{I_3^{\ 2}}(\sigma_a\sigma_r + \upsilon_p\sigma_r^2)\right)$$

$$- 3\Psi_1\frac{I_1^{\ 3}}{I_3} + 2\frac{I_1^{\ 2}}{I_2} \tag{3.54}$$

and

$$\upsilon_p = -\frac{\varepsilon_r^p}{\varepsilon_a^p} \tag{3.55}$$

Note that σ_a and σ_r are the axial and radial stress, respectively, and ε_a^p and ε_r^p are the axial and radial plastic strain, respectively. As described in Equation 3.52, the constants ψ_2 and μ now can be determined by the least square fit of a series of ξ_x and ξ_y data set.

The potential surface in Equation 3.51 is mainly used to compute the direction of the plastic strain increment during yielding.

$$\{d\varepsilon_p\} = d\lambda_p\left\{\frac{\partial g_p}{\partial \sigma_{ij}}\right\} \tag{3.56}$$

where $d\lambda_p$ is the scalar quantity.

3.2.7 Plastic Response Along the Failure Surface

When materials are hardening along the failure surface, the yield surface is crossing over the failure surface so that the state of stresses should satisfy both yield and failure equations. For simplicity, it has been assumed that there are no plastic volume changes and no strain softenings associated with the failure surface. However, there will be plastic volume changes associated with the yield surface along the failure surface. The exact method to derive the elasto-plastic stress-strain matrix $[D_{ep}]$ is presented by Merkle and Dass (1985).

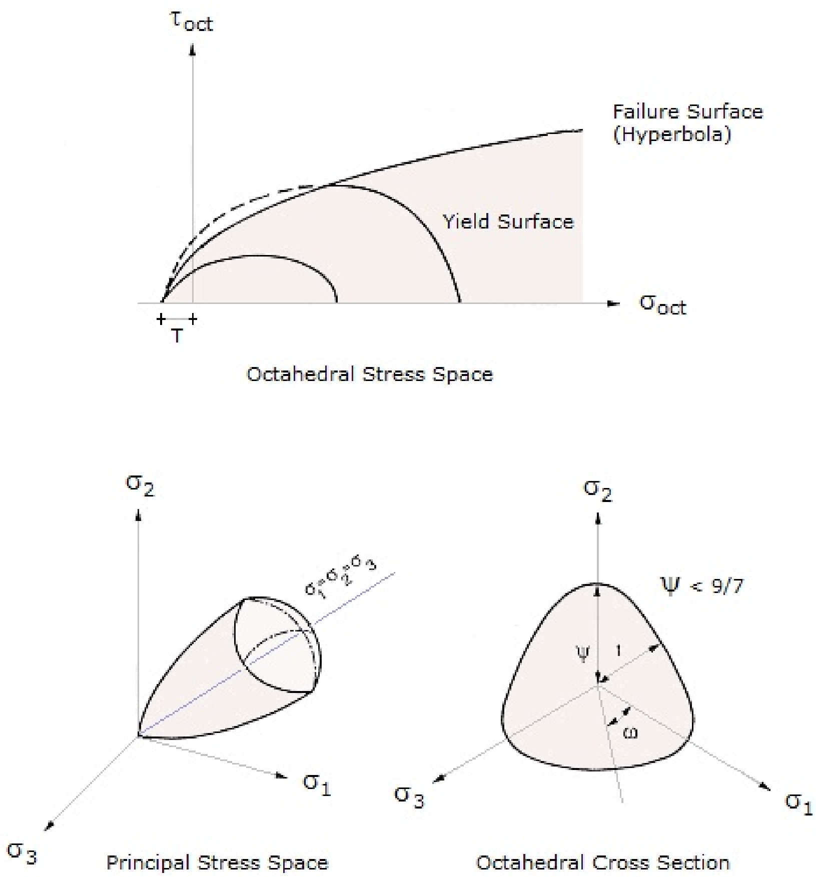


Figure 3.3 Yield and Failure Surfaces used in the Single Hardening Elasto-Plastic Model

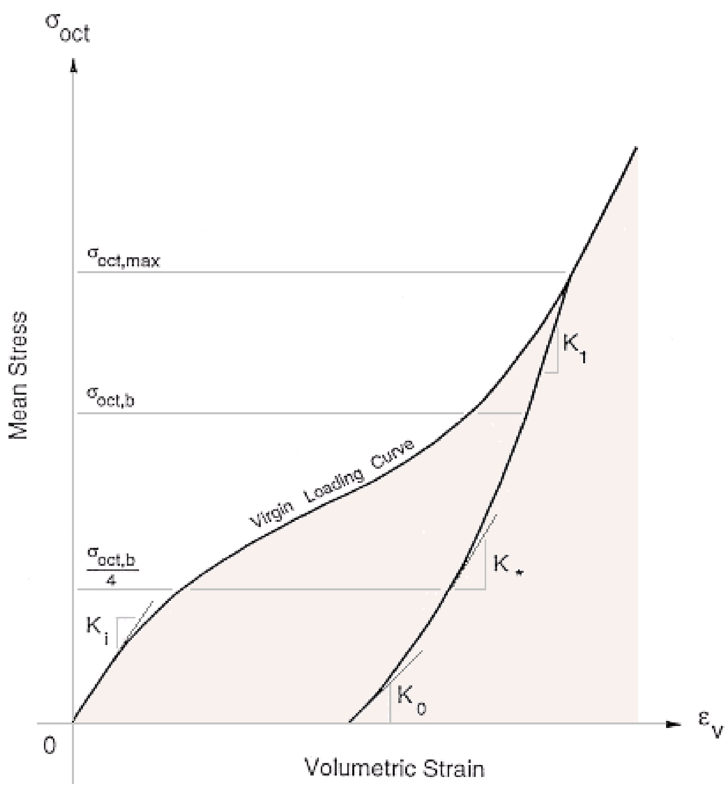


Figure 3.4 Fitting of modified elastic formulation to unload/reload hydrostatic compression response

3.3 JWL High Explosive Model

JWL High Explosive Model represents JWL equation of state (Jones, Wilkens, and Lee, 1968).

JWL is a relatively simple pressure-volume-energy equation of state developed to predict the adiabatic expansion of detonation products. JWL equation of state is given by:

$$\bar{P} = A\left(1-\frac{\omega}{R_1 V}\right)e^{-R_1 V} + B\left(1-\frac{\omega}{R_2 V}\right)e^{-R_1 V} + \frac{\omega E}{V} \tag{3.57}$$

where $\bar{p}$ is the pressure, V is the relative volume (ρ_o/ρ), and E is the internal energy density. And A, B, R_1, R_2, and ω are material constants.

To simulate progress of chemical reaction, Burn Fraction (BF) is used.

$$BF = \frac{(t - t_b)\, C_d}{B_s\, \ell} \tag{3.58}$$

where

t	Current time
t_b	Detonation time
C_d	Detonation velocity
B_s	Constant used to spread the detonation front (usually 2.5)
ℓ	Element characteristic length

The value of Burn Fraction is limited as follows:

For $t \leq t_b$,

$$BF = 0 \tag{3.59}$$

For $t > t_b + B_s \ell / C_d$,

$$BF = 1 \tag{3.60}$$

Thus, the actual pressure (P) developing in the element is obtained by combining Equations 3.57 and 3.58. That is

$$P = BF \cdot \bar{P} \tag{3.61}$$

The following JWL model parameters represents the properties of typical ANFO:

A	20 GPa
B	0.2 GPa
R_1	3.7
R_2	0.9
ω	0.2
E	7.08 GPa (Initial chemical energy)
C_d	3048 m/s
ρ_o	830 Kg/m^3 (Initial density)

3.4 Modified Cam Clay Model with Creep

3.4.1 Introduction

Long term deformations of embankments on saturated clay soils are generally associated with consolidation and creep effects. Consolidation settlements are primarily due to the expulsion of pore water while creep strains are time-dependent deformations taking place under constant stress.

This section presents detailed derivation of constitutive relations when both plastic and creep strains develop. Modified Cam Clay Model is used to represent the elasto-plastic behavior of clay soils. In computing creep strains, averaging scheme is introduced such that volumetric scaling governs at low stress ratio and deviatoric scaling governs at high stress ratio.

In this section, the elasto-plastic matrix is derived for the Modified Cam Clay Model incorporating Singh-Mitchell creep equations.

3.4.2 Yield and Failure Equations

The failure surface (critical state line) of the Modified Cam Clay Model is given by

$$q = M P' \tag{3.62}$$

where M is the failure constant and P′ and q are the alternate stress invariants given by

$$\begin{aligned} P' &= \frac{1}{3}\sigma'_{ij}\ \delta_{ij} \\ q &= \sqrt{3J_2} \\ J_2 &= \frac{1}{2} S_{ij} S_{ij} \\ S_{ij} &= \sigma'_{ij} - P'\delta_{ij} \end{aligned} \tag{3.63}$$

The yield surface of the Modified Cam Clay Model is given by

$$F = \frac{q^2}{M^2} + P'(P' - P'_o) = 0 \tag{3.64}$$

where P_o' is the preconsolidation pressure which grows not only with plastic volumetric strain but also with time.

3.4.3 Elastic Stress-Strain Relationship

The elastic tangent bulk modulus is based on the recompression or swelling response in isotropic compression test.

$$B_k = \frac{2.3\,(1 + e_o)}{C_r} P' \tag{3.65}$$

Where

e_o Initial void ratio

C_r Recompression or swelling index

Assuming the constant Poisson's ratio (υ), the elastic tangent shear modulus would be

$$G = 3.45 \frac{(1-2v)}{(1+v)} \frac{(1+e_o)}{C_r} p' \qquad (3.66)$$

With these effective mean pressure dependent elastic tangent moduli, the incremental elastic constitutive law can be expressed in the following matrix form:

$$\{d\sigma'\} = [D^e]\{d\varepsilon^e\} \qquad (3.67)$$

where

$\{d\sigma'\}$	Effective stress increment
$[D^e]$	Elastic stress-strain matrix
$\{d\varepsilon^e\}$	Elastic strain increment

3.4.4 Plastic Strain Increment

Plastic strain increment is assumed to be normal to the yield surface.

$$\{d\varepsilon^p\} = d\lambda_p \left\{ \frac{\partial F}{\partial \sigma'} \right\} \qquad (3.68)$$

where

$\{d\varepsilon^p\}$	Plastic strain increment
$d\lambda_p$	Proportional constant for plastic strain
$\{\partial F/\partial \sigma'\}$	Derivative of yield surface with respect to stress

3.4.5 Creep Strain Increment

Creep strain increment is assumed to be normal to the equivalent yield surface.

$$\{d\varepsilon^c\} = d\lambda_c \left\{ \frac{\partial F_e}{\partial \sigma'} \right\} dt \tag{3.69}$$

where

$\{d\varepsilon^c\}$	Creep strain increment
$d\lambda_c$	Proportional constant for creep strain
$\{\partial F_e / \partial \sigma'\}$	Derivative of equivalent yield surface with respect to stress
dt	Time increment

Note that the equivalent yield surface is defined as

$$F_e = \frac{q^2}{M^2} + P'(P' - P_e') = 0 \tag{3.70}$$

where

$$P_e' = \frac{q^2}{M^2 P'} + P' \tag{3.71}$$

The proportional constant for creep strain can be computed by averaging both volumetric and deviatoric scaling.

$$d\lambda_c = \frac{\eta}{M} d\lambda_d + \left(1 - \frac{\eta}{M}\right) d\lambda_v \tag{3.72}$$

where

η	Stress ratio (q/p')
$d\lambda_v$	Volumetric scaling factor
$d\lambda_d$	Deviatoric scaling factor

The volumetric scaling factor is based on the secondary consolidation curve.

$$d\lambda_v = \frac{C_\alpha}{2.3\,(1+e_o)\,t_v}\left(\frac{\partial F_e}{\partial P'}\right)^{-1} \tag{3.73}$$

The volumetric age (t_v) in Equation 3.73 is given by

$$t_v = t_{vi}\left(\frac{P'_o}{P'_e}\right)^{\frac{C_c - C_r}{C_\alpha}} \tag{3.74}$$

where

t_{vi}	Reference volumetric time
C_c	Virgin compression index
C_α	Secondary compression coefficient

The deviatoric scaling factor is based on Singh-Mitchell creep equation (1968).

$$d\lambda_d = \sqrt{\frac{3}{2}}\;A e^{\alpha\eta}\left(\frac{t_{di}}{t}\right)^m\left(\frac{\partial F_e}{\partial \sigma'_{ij}}\frac{\partial F_e}{\partial \sigma'_{ij}} - \frac{1}{3}\frac{\partial F_e}{\partial P'}\right)^{-1/2} \tag{3.75}$$

Where

t_{di}	Reference deviatoric time
t	Current time
A	Singh-Mitchell creep parameter
α	Singh-Mitchell creep parameter
m	Singh-Mitchell creep parameter

3.4.6 Total Strain Increment

Total strain increment consists of elastic, plastic and creep strains.

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^p\} + \{d\varepsilon^c\} \tag{3.76}$$

3.4.7 Consistency Equation

During the subsequent yielding, the equation forces the stress increment to move on the subsequent yield loci.

$$dF = \left\{\frac{\partial F}{\partial \sigma'}\right\}^T \{d\sigma'\} + \frac{\partial F}{\partial P_o'} dP_o' = 0 \tag{3.77}$$

The preconsolidation pressure increment (dP_o') is related to the plastic volumetric strain increment ($d\varepsilon_v^p$) and the time increment (dt).

$$dP_o' = \frac{2.3\,(1 + e_o)}{(c_c - c_r)} p_o' \, d\varepsilon_v^p + \frac{c_\alpha}{(c_c - c_r)} \frac{p_o'}{t_v} dt \tag{3.78}$$

From Equation 3.68 the plastic volumetric strain increment ($d\varepsilon_v^p$) can be expressed in terms of $d\lambda_p$

$$d\varepsilon_v^p = d\lambda_p \frac{\partial F}{\partial P'} \tag{3.79}$$

3.4.8 Evaluation of $d\lambda_p$

The elastic strain increment in Equation 3.67 can be expressed in terms of $d\lambda_p$ by combining Equations 3.76 and 3.67.

$$\{d\sigma'\} = [D^e] \left(\{d\varepsilon\} - d\lambda_p \left\{\frac{\partial F}{\partial \sigma'}\right\} - \{d\varepsilon^c\} \right) \tag{3.80}$$

Substituting Equations 3.78, 3.79 and 3.80 into the Consistency Equation 3.77 and solving for $d\lambda_p$.

$$d\lambda_p = \frac{\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e] (\{d\varepsilon\} - \{d\varepsilon^c\}) + p_n}{\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e] \left\{\frac{\partial F}{\partial \sigma'}\right\} - p_d} \qquad (3.81)$$

Where

$$P_n = \frac{\partial F}{\partial P_o'} \frac{P_o'}{t_v} \frac{c_\alpha}{(c_c - c_r)} dt$$

$$P_d = \frac{\partial F}{\partial P_o'} \frac{\partial F}{\partial P'} \frac{(1 + e_o)}{2.3(c_c - c_r)} P_o'$$

3.4.9 Effective Stress Increment

The effective stress increment can be obtained by backsubstituting Equation 3.81 into 3.80.

$$\{d\sigma'\} = [D^{ep}]\{d\varepsilon\} - \{d\sigma_c'\} \qquad (3.82)$$

where the incremental elasto-plastic matrix is expressed as

$$[D^{ep}] = [D^e] - \frac{[D^e]\left\{\frac{\partial F}{\partial \sigma'}\right\}\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e]}{\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e] \left\{\frac{\partial F}{\partial \sigma'}\right\} - P_d} \qquad (3.83)$$

and the stress increment associated with creep is given by

$$\{d\sigma_c'\} = [D^{ep}]\{d\varepsilon^c\} + \frac{[D^e]\left\{\frac{\partial F}{\partial \sigma'}\right\} P_n}{\left\{\frac{\partial F}{\partial \sigma'}\right\}^T [D^e]\left\{\frac{\partial F}{\partial \sigma'}\right\} - P_d} \tag{3.84}$$

3.4.10 Evaluation of Derivatives

$$\frac{\partial F}{\partial P'} = 2P' - P_o'$$

$$\left\{\frac{\partial P'}{\partial \sigma'}\right\} = \frac{1}{3} < 1\ 1\ 1\ 0\ 0\ 0 >^T$$

$$\frac{\partial F}{\partial q} = \frac{2q}{M^2}$$

$$\left\{\frac{\partial q}{\partial \sigma'}\right\} = \frac{3}{2q}\{S_{ij}\}$$

$$\frac{\partial F}{\partial P_o'} = -P'$$

3.5 Engineering model

3.5.1 Introduction

The Engineering Model is hypoelastic-perfectly plastic in shear and hypoelastic in compression. A hypoelastic material is one for which the stress increments are homogeneous linear functions of the strain increments. In general, the coefficients in the linear functions depend on the stress. The principal advantages of the Engineering Model are ease of fitting to laboratory or in situ test data, simplicity of shear plasticity formulation, and the simple form of compressive hysteresis, which most soils exhibit. Its principal disadvantages are lack of hysteresis in pure shear at constant volume below the failure surface, and lack of dilatancy because the plastic strain increments are assumed to be normal to the hydrostatic axis. The Engineering Model is completely described by a pressure-volume strain curve for hydrostatic compression and a two-invariant failure surface.

3.5.2 Hydrostatic Response

The hydrostatic response is represented by the incremental elastic (hypoelastic) bulk modulus as a function of current compressive volumetric strain (ε_v), maximum past compressive volumetric strain (ε_{vm}) and compressive volumetric strain increment ($d\varepsilon_v$) as shown in Figure 3.5a.

$$K = K(\varepsilon_v, \varepsilon_{vm}, d\varepsilon_v) \tag{3.85}$$

Poisson's ratio is also defined for each hydrostat segment.

$$\upsilon = \upsilon(\varepsilon_v, \varepsilon_{vm}, d\varepsilon_v) \tag{3.86}$$

The corresponding hypoelastic constrained compression and shear moduli are then computed from the following expressions respectively:

$$M = \frac{3K(1 - \upsilon)}{(1 + \upsilon)} \tag{3.87}$$

and

$$G = \frac{3K(1 - 2\upsilon)}{2(1 + \upsilon)} \tag{3.88}$$

3.5.3 Plastic Shear Response

The failure surface is composed of three segments of conical surfaces as shown in Figure 3.5b, each having an equation of the form:

$$f(I_1, \sqrt{J_2}) = \sqrt{J_2} - (a + bI_1) = 0 \tag{3.89}$$

The material is assumed to behave incrementally elastically when the stress point lies below the failure surface. When the stress point moves along the failure surface, the material response is assumed to be hypoelastic-perfectly plastic in shear. The plastic strain increments are assumed to be normal to the hydrostatic axis so that there would be no plastic volume changes associated with the failure surface.

The derivation of elasto-plastic stress-strain matrix $[D_{ep}]$ is given by Merkle and Dass (1985).

3.5.4 Parameter Determination

The Engineering Model parameters can be obtained by fitting a series of straight lines to shear strength, hydrostatic compression, and constrained compression or K_0 test data.

The parameters of shear strength envelope shown in Figure 3.5b can be determined by fitting to shear strength data in drained triaxial compression tests which arc expressed as functions of I_1, and $\sqrt{J_2}$.

Note that the values of I_1 and $\sqrt{J_2}$ at the failure points of triaxial compression are computed in terms of σ_{af} and σ_r.

$$I_1 = \sigma_{af} + 2\sigma_r \tag{3.90}$$

and

$$\sqrt{J_2} = \frac{|\sigma_{af} - \sigma_r|}{\sqrt{3}} \tag{3.91}$$

where σ_{af} is the axial stress at failure and σ_r is the confining stress.

The hypoelastic bulk modulus and Poisson's ratio in Equations 3.85 and 3.86 respectively can be determined from hydrostatic and constrained compression tests. Then the Poisson's ratio is computed from Equation 3.87.

$$\upsilon = \frac{3K - M}{3K + M} \tag{3.92}$$

When K_o test data are available from constrained compression tests conducted in a triaxial cell, in which the confining stress is measured, Poisson's ratio can be directly computed from:

$$\upsilon = \frac{K_o}{1 + K_o} \tag{3.93}$$

where

$$K_o = \frac{d\sigma_r}{d\sigma_a} \tag{3.94}$$

When only uniaxial compression test data are available, it is customary to assume Poisson's ratio and to compute bulk modulus from

$$K = \frac{M(1 + \upsilon)}{3(1 - \upsilon)} \tag{3.95}$$

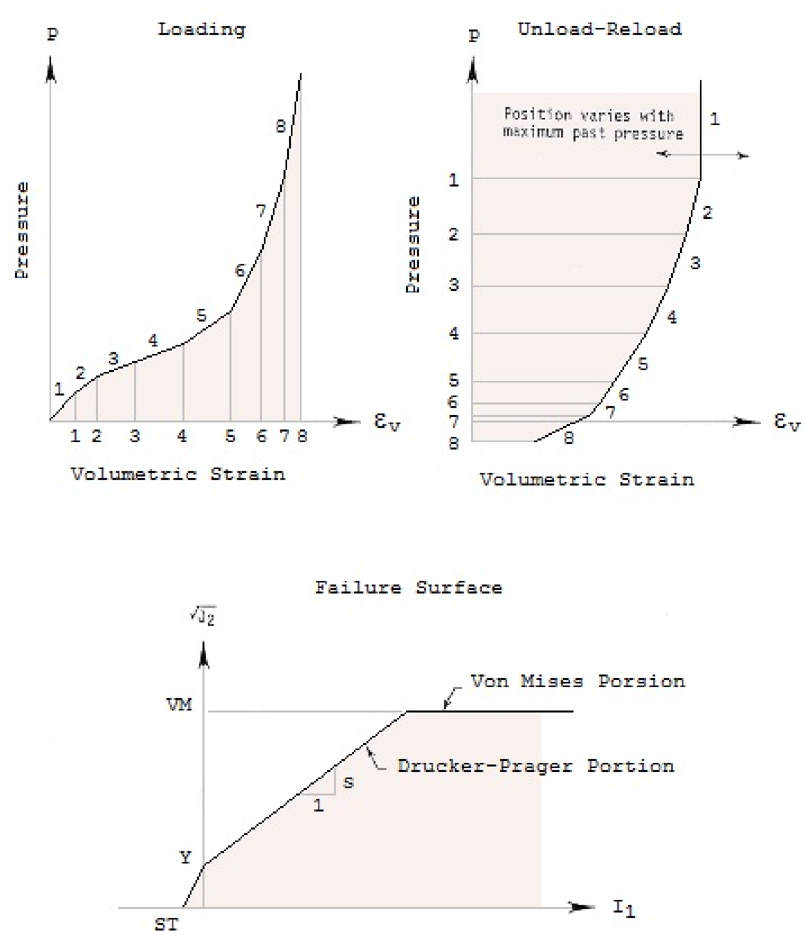


Figure 3.5 Engineering Model

3.6 Joint Model

3.6.1 Introduction

Joint Model is often used to represent rock joints, faults, and interfaces. Along the joint face, slipping takes place when the shear stress exceeds shear strength and debonding occurs when adjacent two blocks are not in contact.

Joint Model is to be used as a material model for the joint element (see Figure 3.6) as described in Card 5.4 of SMAP-3D User's Manual.

3.6.2 Strain-Displacement Relation

Strains in the joint local coordinate are

$$\{\Delta\varepsilon\} = \begin{Bmatrix} \Delta\gamma'_{zx} \\ \Delta\gamma'_{zy} \\ \Delta\varepsilon'_{zz} \end{Bmatrix} \tag{3.96}$$

where

$\Delta\gamma_{zx}'$	Shear strain increment in the plane $z'x'$
$\Delta\gamma_{zy}'$	Shear strain increment in the plane $z'y'$
$\Delta\varepsilon_{zz}'$	Normal strain increment

Local displacement increment, $\{\Delta u'\}$, is related to the global displacement increment, $\{\Delta u\}$, as follows:

$$\{\Delta u'\} = [\beta]\,[\Delta u] \tag{3.97}$$

where

$$\{\Delta u'\} = \begin{Bmatrix} \Delta u'_x \\ \Delta u'_y \\ \Delta u'_z \end{Bmatrix} \qquad \{\Delta u\} = \begin{Bmatrix} \Delta u_x \\ \Delta u_y \\ \Delta u_z \end{Bmatrix}$$

$[\beta]$ Coordinate transformation matrix

Strain-displacement relation in the local coordinate is given by

$$\{\Delta\epsilon'\} = \frac{1}{\delta}\{\Delta u'\} \tag{3.98}$$

where δ is the thickness of joint. And global displacement increment can be expressed in terms of global nodal displacement increment, $\{\Delta\bar{u}\}$, using the shape function matrix, [h], as

$$\{\Delta u\} = [h]\,\{\Delta\bar{u}\} \tag{3.99}$$

Now, Substituting Equations 3.97 and 3.99 into the Equation 3.98, we obtain

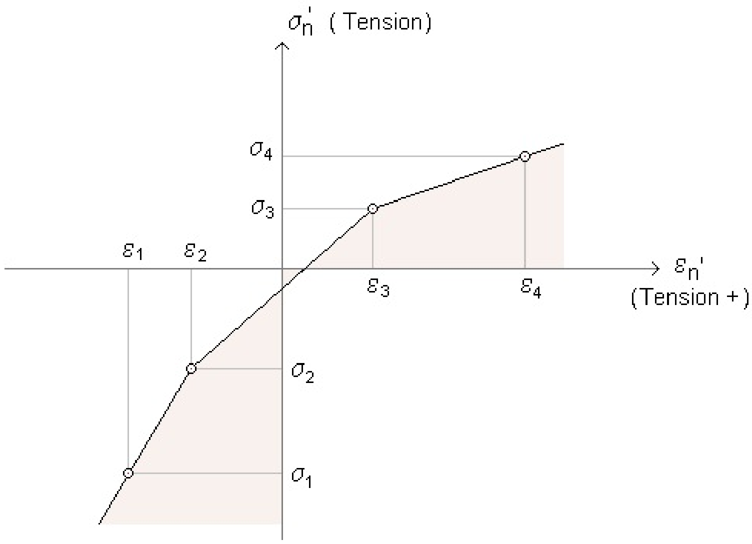
$$\{\Delta\epsilon'\} = [B]\,\{\Delta\bar{u}\} \tag{3.100}$$

where

$$[B] = \frac{1}{\delta}[\beta]\,[h] \tag{3.101}$$

3.6.3 Normal Stress-Strain Relation

Normal Stress-strain relation is assumed to be nonlinear elastic as shown below



Thus, Young's modulus (E) is computed as follows:

For $\varepsilon_n' < \varepsilon_2$

$$E = \frac{\sigma_2 - \sigma_1}{\varepsilon_2 - \varepsilon_1}$$

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

$$E = \frac{\sigma_3 - \sigma_2}{\varepsilon_3 - \varepsilon_2} \quad (3.102)$$

For $\varepsilon_n' \geq \varepsilon_3$

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

3.6.4 Shear Stress-Strain Relation

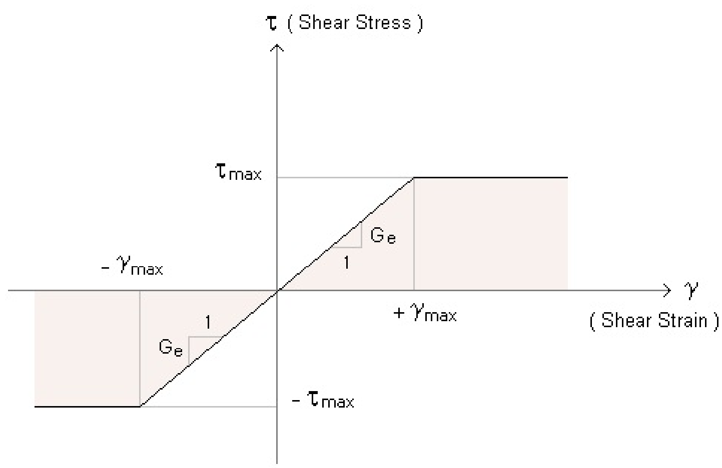
The shear strength of joint is assumed to follow Mohr-Coulomb failure criterion.

$$\tau_{max} = C - \sigma_n' \tan \varphi \qquad (3.103)$$

where

τ_{max}	Maximum shear stress
C	Cohesion
φ	Friction angle
σ_n'	Normal stress (Tension is positive)

Shear stress-strain relation is assumed to be elastic below the strength envelope and perfectly plastic along strength envelope as shown below:



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

$$\text{For } |\gamma| < \gamma_{max} \quad G = G_e \qquad \text{For } |\gamma| \geq \gamma_{max} \quad G = 0 \qquad (3.104)$$

Note that G_e is the elastic shear modulus.

3.6.5 Element Stiffness Matrix

Joint stress-strain relation can be given by

$$\{\Delta\sigma'\} = [C'] \{\Delta\varepsilon'\} \qquad (3.105)$$

where

$$\{\Delta\sigma'\} = \begin{Bmatrix} \Delta\tau'_{zx} \\ \Delta\tau'_{zy} \\ \Delta\sigma'_{zz} \end{Bmatrix}$$

$$[C'] = \begin{bmatrix} G & 0 & 0 \\ 0 & G & 0 \\ 0 & 0 & E \end{bmatrix}$$

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:

$$[K] = \int_v [B]^T [C'] [B] \, dv \qquad (3.106)$$

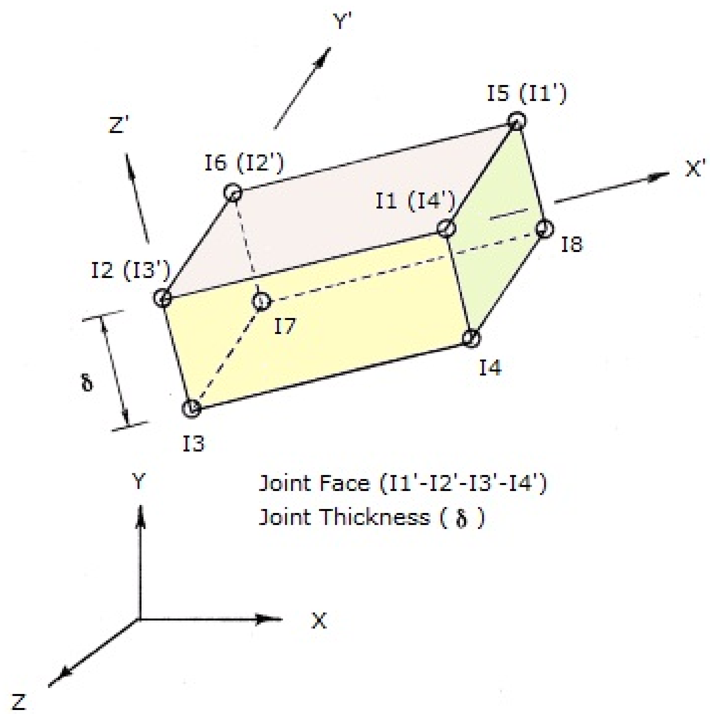


Figure 3.6 Local and Global Coordinates of Joint Element

3.7 Generalized Decoupled Hyperbolic Model

3.7.1 Introduction

GDHM (Generalized Decoupled Hyperbolic Model) is the decoupled material model which is the generalized form of the original hyperbolic model (Duncan and Chang, 1970). Main features of GDHM include:

- Hoek and Brown in situ rock strength model and
- Strength envelope expressed as a function of Lode angle on octahedral plane

3.7.2 Stress-Strain Relation

It is assumed that volumetric behavior is not coupled with deviatoric behavior. Thus, we have two independent equations:

$$\begin{aligned} dp &= K \cdot d\varepsilon_v \\ d\tau_{oct} &= 2 \cdot G \cdot d\gamma_{oct} \end{aligned} \tag{3.107}$$

where

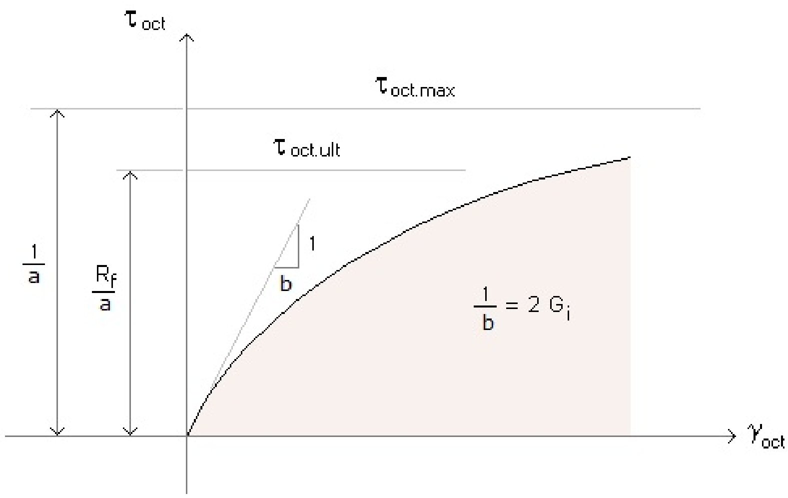
p	Mean pressure
ε_v	Volumetric strain
τ_{oct}	Octahedral shear stress
γ_{oct}	Octahedral shear strain
K	Bulk modulus
G	Shear modulus

At constant mean pressure and constant Lode angle, the shear stress-strain relation is assumed to be hyperbolic.

That is

$$\tau_{oct} = \frac{\gamma_{oct}}{b + a\,\gamma_{oct}} \tag{3.108}$$

As shown in the following figure, τ_{oct} approaches to the maximum shear stress, $\tau_{oct.max}$, as γ_{oct} goes to infinity. And the slope of the Equation 3.109 at γ_{oct} = 0 is equal to 1/b. Most triaxial compression tests, however, show that ultimate shear strength, $\tau_{oct.ult}$, is reached at finite value of γ_{oct} and $\tau_{oct.ult}$ is approximately 70~90% of $\tau_{oct.max}$.



That is,

$$\frac{1}{a} = \tau_{oct.max} = \frac{1}{R_f}\tau_{oct.ult} \qquad (3.109)$$

$$\frac{1}{b} = 2\ G_i \qquad (3.110)$$

where

R_f Material constant (0.7~0.9)
G_i Initial shear modulus

Differentiating Equation 3.109 with respect to γ_{oct},

$$\frac{d\tau_{oct}}{d\gamma_{oct}} = \frac{b}{(b + a\gamma_{oct})^2} \tag{3.111}$$

Solving for γ_{oct} from Equation 3.108,

$$\gamma_{oct} = \frac{b\,\tau_{oct}}{(1 - a\,\tau_{oct})} \tag{3.112}$$

Now, substituting Equation 3.113 into 3.112, we obtain the following loading shear modulus:

$$G = G_i \left(1 - \frac{\tau_{oct}}{\left(\frac{1}{a}\right)}\right)^2 \tag{3.113}$$

3.7.3 Shear Strength Equation

Kim (1984) presented the following generalized form of ultimate shear strength equation:

$$\tau_{oct.ult} = \frac{\sqrt{2}}{3}\,[(\alpha + \beta\,p)^n + \kappa]\;R(\theta) \tag{3.114}$$

The strength parameters (n, α, β, κ) are tabulated in Table 3.1 and R(θ) is given by the Equation 3.4. Note that the strength equation represents Von Mises Model when n =1, Hoek and Brown In Situ Rock Model when n=1/2, and Mohr-Coulomb or Drucker-Prager Model when n=1.

Substituting Equation 3.110 into 3.115,

$$\frac{1}{a} = \frac{1}{R_f}\,\frac{\sqrt{2}}{3}\,[(\alpha + \beta p)^n + \kappa]\;R(\theta) \tag{3.115}$$

Now, combining Equations 3.114, 3.115 and 3.116, the generalized loading shear modulus, G_i is given by

$$G = G_i \left[1 - \frac{R_f \, \tau_{oct}}{\tau_{oct.ult}} \right]^2 \qquad (3.116)$$

The initial shear modulus, G_i, in Equation 3.117 may be obtained from the following empirical equations:

For cohesive soil (Hardin and Black, 1968)

$$G_i = 1230 \frac{(2.973 - e)^2}{(1 + e)} \bar{\sigma}_{oct}^{1/2} \; OCR^K \qquad (3.117)$$

where

e	Void ratio
OCR	Overconsolidation ratio
K	Parameter expressed as a function of plasticity index. Note that stress unit is psi

For cohesionless soil (Janbu, 1963)

$$G_i = \frac{1}{2(1 + \nu)} K_{ur} \, P_a \left(\frac{\sigma_3}{P_a} \right)^n \qquad (3.118)$$

where

v	Poisson's ratio
P_a	Atmospheric pressure
K_{ur}, n	Material constants
σ_3	Confining pressure

It should be noted that Equation 3.117 can be degenerated to the original Duncan and Chang's hyperbolic model when Mohr-Coulomb strength envelope is specified along the triaxial compression mode.

That is,

$$\tau_{oct} = \frac{\sqrt{2}}{3}(\sigma_1 - \sigma_3) \qquad (3.119)$$

$$\tau_{oct.ult} = \frac{\sqrt{2}}{3}(\sigma_1 - \sigma_3)_{ult}$$

$$= \frac{6\ Sin\varphi}{(3 - Sin\varphi)} P + \frac{6\ Cos\varphi}{(3 - Sin\varphi)} C \qquad (3.120)$$

where

$$P = \frac{1}{3}(\sigma_1 + 2\ \sigma_3) \qquad (3.121)$$

Substituting Equation 3.122 into 3.121 and solving for σ_1, we obtain

$$\sigma_1 = \frac{(1 + Sin\varphi)}{(1 - Sin\varphi)} \sigma_3 + \frac{2\ Cos\varphi}{(1 - Sin\varphi)} C \qquad (3.122)$$

Backsubstituting σ_1 in Equation 3.123 into the right hand side of Equation 3.121,

$$\tau_{oct.ult} = \frac{\sqrt{2}}{3} \frac{(2\ Sin\varphi\ \sigma_3 + 2\ Cos\varphi\ C)}{(1 - Sin\varphi)} \qquad (3.123)$$

Now, substituting Equations 3.120 and 3.124 into 3.117, we can obtain the following Duncan and Chang Hyperbolic Model (1970):

$$G = G_i \left[1 - \frac{R_f\ (1 - Sin\varphi)\ (\sigma_1 - \sigma_3)}{2\ Sin\varphi\ \sigma_3 + 2\ Cos\varphi\ C} \right]^2 \qquad (3.124)$$

References

Ahrens, T.J., Equation of State of Earth Media, Report to Defense Nuclear Agency, DNA-TR-88-265, Washington, D.C. November 1988.

Allen, R.T., Equation of State of Rocks and Minerals, Defense Atomic Support Agency, DA49-146-XZ-462, Project No. 532, March 1967.

Bakanova, A.A., V.N. Zubarev, Y.N. Sutulov, and R.F. Trunin, Thermodynamic Properties of Water at High Pressures and Temperatures, Soviet Phys. JETP, 41, 544, 1976.

Biot, M.A., Theory of Propagation of Elastic Waves in Fluid Saturated Porous Solid. I, II, Journal of Acoustical Society of America, Vol. 28, pp 168-191, 1956.

Biot, M.A., Mechanics of Deformation and Acoustic Propagtion in Porous media, Journal of Applied Physics, Vol. 33, pp 1482-1498, 1962A.

Biot, M.A., Generalized Thoery of Acoustic Propagation in Porous Dissipative Media, Journal of Acoustical Society of America, Vol. 34, pp 1254-1264, 1962B.

Blouin, S.E., and K.J. Kim, Undrained Compressibility of Saturated Soil, DNA-TR-87-42, Defense Nuclear Agency, Washington, D.C., February 1984.

Blouin, S.E., D.E. Chitty, A.F. Rauch, and K.J. Kim, Dynamic Response of Multiphase Porous Media, Annual Technical Report 1, Report to U.S. Air Force Office of Scientific Research, Washington D.C., March 1990.

Blouin, S.E., D.E. Chitty, A.F. Rauch, and K.J. Kim, Dynamic Response of Multiphase Porous Media, Annual Technical Report 2, Report to U.S. Air Force Office of Scientific Research, Washington D.C., June 1991

Borja, Ronaldo I., Generalized Creep and Stress Relaxation Model for Clays, Journal of Geotechnical Engineering, Vol. 118, No. 11, November 1992.

Britt, J.R., Behavior of Water in Low Pressure Region, unpublished manuscript, 1985.

Dass, W.C. and D.H. Merkle, Computational Aspects of the ARA Three Invariant Constitutive Model, Report to U.S. Air Force Office of Scientific Research Washington, D.C., May 1986.

D avid E. Van Dillen, etc., Modernization of the BMINES Computer Code Vo. I: User's Guide, Agbabian Associates, Sep. 1981.

Duncan, J.M., and C. Y. Chang, Nonlinear Analysis of Stress and Strain in Soils, J. Soil Mech. Found. Div. ASCE, vol. 96, no. SM5, pp. 1629-1653, September, 1970.

E.L.Lee, H.C.Hornig, and J.K.Kury, Adiabatic Expansion of High Explosive Detonation Products, UCRL-50422, Lawrence Livermore National Laboratory, California, May 2, 1968.

Hardin, B.O. and W.L Black, Vibration Modulus of Normally Consolidated Clays, J. Soil Mech. Found. Div., ASCE, vol. 94, no. SM2, pp. 353-369, March 1968.

Forchheimer, P., Wasserbewegung durch Boden, Z. Ver. Deutsch Ing., 45 pp. 1782-1788, 1901.

Ghaboussi, J., and E. L. Wilson, Variational Formulation of Dynamics of Fluid-Saturated Porous Elastic Solids, Journal of the Engineering Mechanics Division, ASCE, Vol. 98, pp 947-963, 1972.

Gibson, R.E., The Progress of Consolidation in a Clay Layer Increasing in Thickness with Time, Geotechnique, Vol. 8, No. 4, pp. 171-182.

Hoek, E. and E.T. Brown, Underground Excavations in Rock, The Institution of Mining and Metallurgy, London, England, 1982.

Janbu, Nilmar, Soil Compressibility as Determined by Oedometer and Triaxial Tests, Proc. Eur. Conf. Soil Mech. Found. Eng., Wiesbaden, 1963, vol. 1, pp. 19-25.

Karshenas, M. and J. Ghaboussi, Modeling and Finite Element Analysis of Soil Behavior, Civil Engineering Studies, Geotechnical Research Series No. 17, University of Illinois, Urbana, 1979.

Kim, K. J., Finite Element Analysis of Nonlinear Consolidation, Ph.D. Thesis, University of Illinois at Urbana-Champaign, 1982.

Kim, K.J. and S.E. Blouin, Response of Saturated Parous Nonlinear Materials to Dynamic Loadings, Report to Air Force Office of Scientific Research, Washington, DC, F49620-81-C-0014 (May 1984).

Kim, K.J., S.E. Blouin, and D.A. Timian, Experimental and Theoretical Response of Multiphase Porous Media to Dynamic Loads, Annual Report No. 1 to Air Force Office of Scientific Research, Washington, D.C., 1986.

Kim, K.J., S.E. Blouin, and D.A. Timian, Experimental and Theoretical Response of Multiphase Porous Media to Dynamic Loads, Annual Report No. 2 to Air Force Office of Scientific Research, Washington, D.C., 1987.

Kim, K.J., S.E. Blouin, D.E. Chitty, and D.H. Merkle, Experimental and Theoretical Response of Multiphase Porous Media to Dynamic Loads, Final Report to Air Force Office of Scientific Research, Washington, D.C., 1988.

Kim, M.K. and P.V. Lade, Single Hardening Constitutive Model for Frictional Materials, 1. Plastic Potential Function, Computers and Geotechnics, 5(4), 307- 324, 1988.

Lade, P.V., Elasto-plastic Stress-Strain Theory for Cohesionless Soil with Curved Yield Surfaces, Int. J. Solids Struct., 13, 1019-1035, 1977.

Lade, P.V. and R.B. Nelson, Modelling the Elastic Behavior of Granular Materials, International Journal for Numerical and analytical Methods in Geomechanics, Vol. II, pp.521-542, 1987.

Lade, P.V., Single-Hardening Model with Application to NC Clay, J. Geotech. Engrg., ASCE, 116 (3), 394-414, 1990.

Lade, P.V. and M.K. Kim, Single Hardening Constitutive Model for Frictional Materials, III. Comparisons with Experimental Data, Computers and Geotechnics, 6(1), 30-47. 1988b.

Mendelson, A., Plasticity: Theory and Application, The MacMillan Company, New York (1968).

Mengi, Y., and H.D. McNiven, Fluid-filled Porous Media to a Transient Input, Journal of Acoustical Society of America, Vol. 61, pp 84-94. 1977.

Merkle, D.H. and W.C. Dass, Fundamental Properties of Soils for Complex Dynamic Loadings: Development of a Three Invariant Constitutive Model, Report to the U.S. Air Force Office of Scientific Research, Washington, D.C.,April 1985.

Piepenburg, D.D., K.J. Kim and M.D. Davister, Numerical Analysis of Nonlinear Liner-Medium Interaction. Tunnels Subjected to Biaxial Loading Vol. III, Technical Report to Defense Nuclear Agency, Washington, D.C., DNA-TR-86-138-V3, December 1986.

Rischbieter, F., et al., Studies of Soil Liquefaction by Shock Wave Loading, Fifth International Symposium on Military applications of Blast Simulation, Vol. III, Royal Swedish Fortifications Administration, Stockholm, Sweden, May 1977.

Singh, Awtar and James K. Mitchel, Generalized Stress-Strain-Time Functions for Soils, Journal of the Soil Mechanics and Foundation Division, ASCE, 21-46, 1968.