

SMAP[®] - W2

Structure Medium Analysis Program

2-D Seepage Analysis

User's Manual Version 7.07

COMTEC RESEARCH

Copyright @2025 by COMTEC RESEARCH

All right reserved. No part of this manual may be reproduced in any form or by any means without a written permission of 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**1. Introduction**

1.1	Overview	1-1
1.2	Features	1-1
1.3	Applications.	1-3

2. Installing SMAP-W2

2.1	Minimum System Requirements.	2-1
2.2	Installation Procedure	2-1

3. Running Programs

3.1	Introduction	3-1
3.2	RUN Menu	3-4
3.2.1	SMAP	3-4
3.2.2	Mesh Generator.	3-6
3.2.3	Load Generator.	3-7
3.2.4	PlotXY Generator	3-8
3.2.5	Command Line	3-8
3.2.6	Windows Explorer	3-8
3.2.7	Swcc Analysis	3-8
3.3	PLOT Menu	3-9
3.3.1	XY.	3-9
3.3.2	MESH.	3-9
3.3.3	RESULT.	3-10
3.4	SETUP Menu	3-11
3.4.1	General Setup.	3-11
3.4.2	PLOT-XY Setup.	3-13
3.4.3	PLOT-2D Setup.	3-15
3.4.4	PLOT-3D Setup.	3-16
3.5	Manual Procedure to Run SMAP-W2	3-17
3.6	Debugging SMAP-W2 Main-Processing Program.	3-18

4. SMAP-W2 User's Manual

4.1	Introduction	4-1
4.2	Project File	4-2
4.3	Mesh File	4-3
4.4	Main File	4-7
4.5	Post File	4-21
4.5.1	PLOT-2D	4-95
4.5.2	PLOT-XY.	4-117

5. Group Mesh User's Manual

5.1	Introduction	5-1
5.2	Group Mesh Generator.	5-2
5.3	Group	5-2
5.4	Base Mesh	5-16
5.5	Segment	5-18
5.6	Modifying Finite Element Meshes	5-23
5.7	Entities	5-30

6. Block Mesh User's Manual

6.1	Introduction.	6-1
6.2	Block Mesh Generator.	6-2
6.3	Work Plane.	6-5
6.4	Entities	6-20
6.5	Block.	6-31
6.6	Modifying Finite Element Meshes	6-53

7. PRESMAP User's Manual

7.1	Introduction	7-1
7.2	PRESMAP-2D	
7.2.1	MODEL 1	7-3
7.2.2	MODEL 2	7-15
7.2.3	MODEL 3	7-27
7.2.4	MODEL 4	7-35
7.3	NATM-2D.	7-39
7.4	CIRCLE-2D.	7-51
7.9	PRESMAP-GP.	7-97

8. ADDRGN User's Manual

8.1	Introduction	8-1
8.2	ADDRGN-2D	8-5

9. SUPPLEMENT Program

9.1	Introduction	9-1
9.2	EDIT	9-1
9.3	XY	9-1
9.4	CARDS	9-8
9.5	SHRINK FILE	9-8

10. File Conversion

10.1 Introduction	10-1
10.2 Conversion to SMAP-W2 Mesh File	10-1

11. LOAD User's Manual

11.1 Introduction	11-1
11.2 LOAD-2D	
11.2.7 LDTYPE = 7 Seepage for SMAP-W2	11-41

12. XY Graph User's Manual

12.1 Introduction	12-1
12.2 New Graph	12-3
12.3 Edit Dialog	12-6
12.4 Existing Graph	12-10
12.5 Excel XY Graph	12-11
12.6 SMAP Result	12-13
12.7 PlotXY Generator.	12-15

13. PLOT-XY User's Manual

13.1 Introduction	13-1
13.2 Menus	13-2

14. PLOT-2D User's Manual

14.1 Introduction	14-1
14.2 Menus	14-2

15. PLOT-3D User's Manual

15.1 Introduction	15-1
15.2 Menus	15-2
15.3 Toolbars.	15-18

16. SWCC User's Manual

16. Swcc	16-1
--------------------	------

17. SMAP-F2 User's Manual

17.1 Introduction	17-1
17.2 Project file	17-2
17.3 Mesh File.	17-3
17.4 Main File.	17-7

Introduction

1.1 Overview

SMAP-W2 is an advanced two-dimensional finite element computer program developed for the steady-state and transient seepage analyses. The program has been applied to various types of seepage problems. 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-W2 include:

- Two-dimensional plane or axisymmetric isoparametric element.
 - Models unsaturated and saturated soils and rocks.
 - Allows nonlinear hydraulic conductivities and water capacities with consideration of analytical models of van Genuchten and Fredlund and Xing as well as user specified tables.
- Earth dam mesh generator
 - Builds finite element meshes for the typical earth dams including cores and cut-off walls.

- Boundary conditions
 - Prescribed total head or pressure
 - Prescribed water flow
 - Potential seepage face
- Water generation sources
- Relaxation method for convergence of solution during iteration

1.3 Applications

Applications of SMAP-W2 include:

- Slope stability
- Earth dam safety
- Foundation design
- Ground water management
- Construction dewatering
- Contaminant transport

Overview of SMAP-W2 Program Structure

USER INPUT	User prepares Mesh, Main, and Post Files according to SMAP-W2 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-W2	Main-processor executing Mesh and Main Files to compute water flow. Output files include: CONTSS.DAT Water pressures in elements DISPLT.DAT Total head, pressure & flow rate at nodes
PLOT-XY PLOT-2D PLOT-3D	Post-processors executing Post File for graphical output: • Finite element mesh • Contours of total head and water pressure • Time histories of total head and water pressure • Profiles of flow velocity

Installing SMAP -W2

2.1 Minimum System Requirements

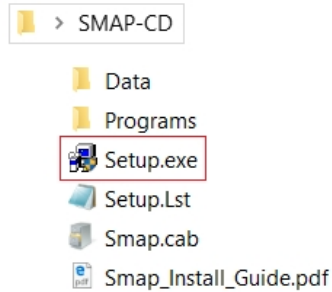
- ✓ Windows 64 bit operating system
- ✓ Intel Pentium 4 or AMD processors
- ✓ 4 GB Ram with 30 GB free space in Drive C
- ✓ SVGA monitor

2.2 Installation Procedure

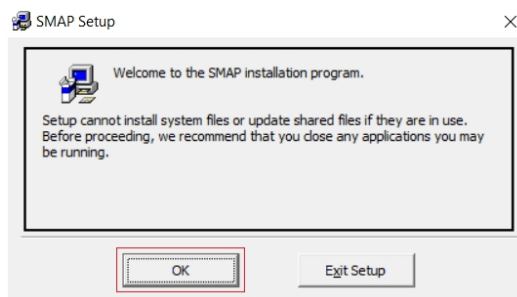
1. Uninstall if there are pre-existing SMAP programs.
To uninstall SMAP programs, remove following program using Add/Remove in Control Panel:
 SMAP
Delete following files if they exist:
 C:\Program Files\Smapi
 C:\Windows\Setup1.exe
Rename or delete following folders if they exist:
 C:\SMAP
 C:\SmapiKey
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-W2

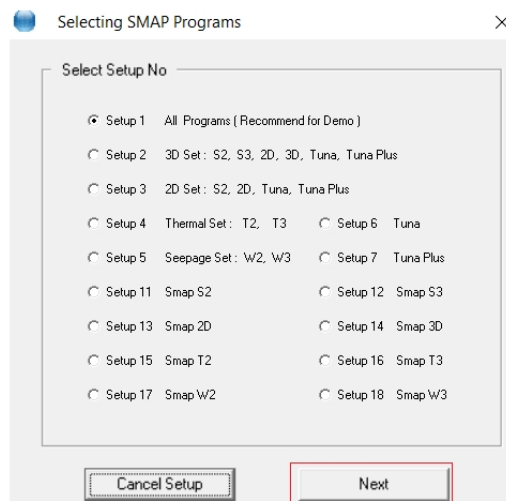
4. Double-click **Setup.exe**



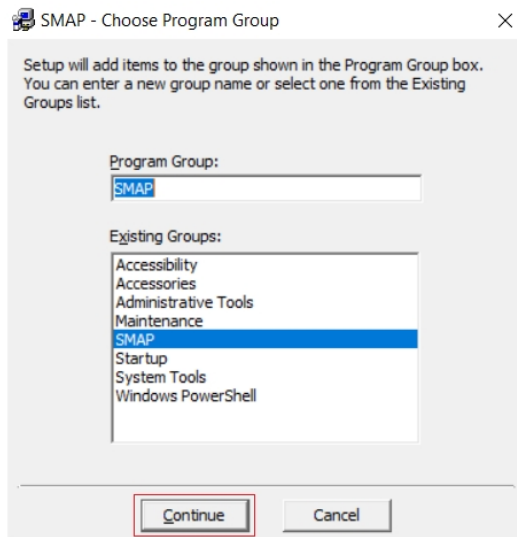
5. Click **OK**



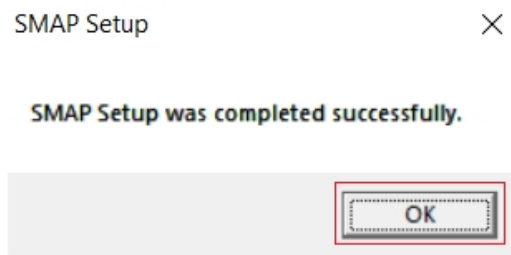
6. Click **Next**
It will take few minutes.
Wait until next step.



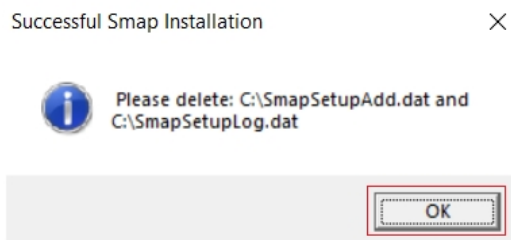
7. Click **Continue**



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

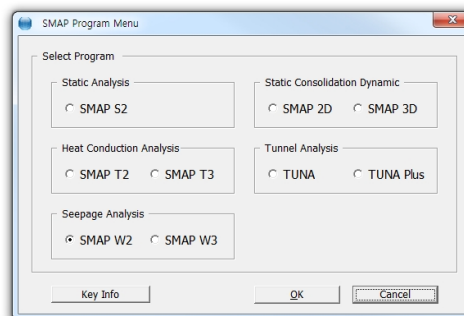
Running Programs

3.1 Introduction

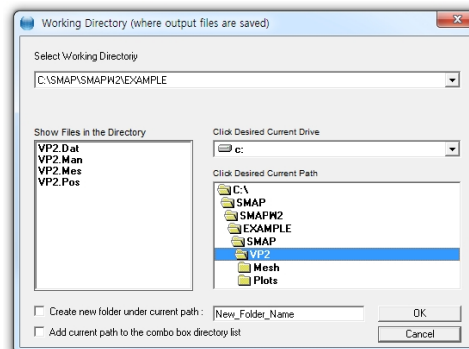
Generally, SMAP-W2 consists of pre-, main-, and post-processing programs. Pre-processing programs are mainly used to automatically generate Mesh Files which will contain nodal coordinates, boundary conditions, and element indexes. Main-processing program of SMAP-W2 is the one which computes water flow of two-dimensional problems. Post-processing programs are used to show graphically the results from the main-processing program.

Accessing SMAP-W2 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-W2** 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**.

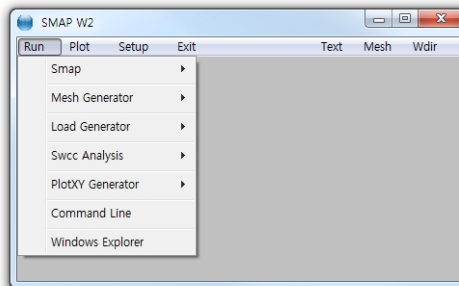
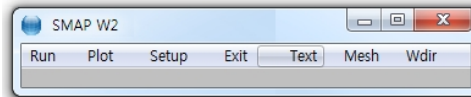


SMAP-W2 Menu

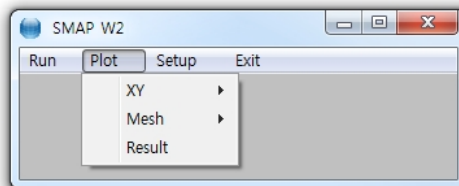
SMAP-W2 provides following Main Menus; Run, Plot, Setup, Exit, Text, Mesh and Wdir.

RUN executes main- and pre-processing programs and has following Sub Menus;

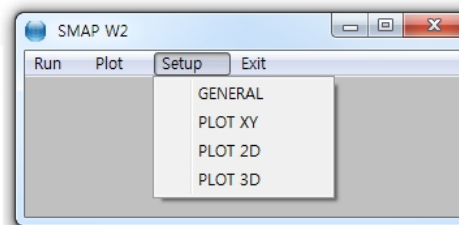
Smep,
Mesh Generator,
Load Generator,
Swcc Analysis,
PlotXY Generator,
Command Line and
Windows Explorer.



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



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



EXIT is used to end SMAP-W2.

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

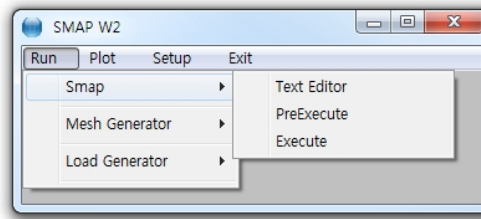
WDIR is used to change Working Directory.

3.2 RUN Menu

3.2.1 SMAP

Once you have prepared the input files (Mesh, Main, and Post) according to the SMAP-W2 User's Manual in Section 4, you are ready to execute SMAP-W2 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-W2 main-processing program.

SMAP-W2 Output Files

Once you execute SMAP-W2, generally you can obtain following output files:

CONTSS.DAT	Contains water pressures in elements
DISPLT.DAT	Contains total head, pressure, and flow rates in nodes

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

SMAP-W2 Graphical Output

SMAP-W2 Post-processing programs can generate the following graphical output:

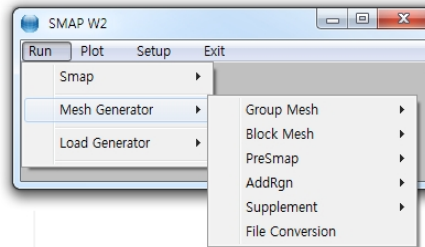
- Finite element mesh
- Contours of total head and water pressure
- Profiles of nodal flow velocity vectors
- Time histories of total head, water pressure and flow rate

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

3.2.2 MESH GENERATOR

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

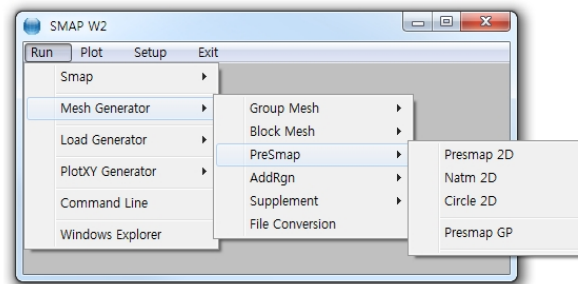
MESH GENERATOR Menu has the following Sub Menus; Group Mesh, Block Mesh, PreSmap, AddRgn, Supplement, and File Conversion.



GROUP MESH is a two-dimensional CAD program specially designed to build group mesh which can be used to generate finite element mesh with the aid of program ADDRGN-2D. Section 5 in SMAP-W2 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 PRESMA-P. Section 6 in SMAP-W2 Example Problems describes in detail about running Block Mesh.

PRESMAP menu includes two dimensional pre-processing programs to generate finite element meshes: Section 7 in SMAP-W2 Example Problems describes in detail about running PRESMA-P Programs.



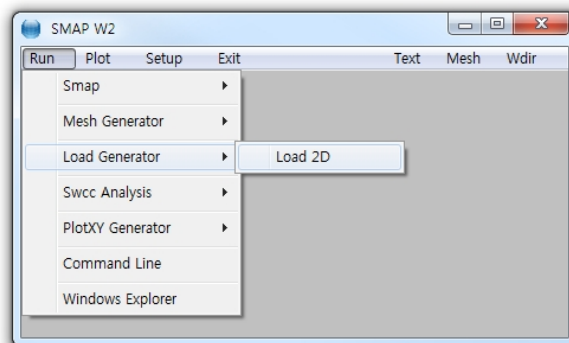
ADDRGN is the pre-processing program which has the following two basic functions: Combine two different meshes and modify existing meshes. Section 8 in SMAP-W2 Example Problems describes in detail about running ADDRGN programs.

SUPPLEMENT contains supporting programs which are useful to prepare input data for pre- and main-processing programs. Section 9 in SMAP-W2 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-W2 User's Manual describes in detail about running FILE CONVERSION program.

3.2.3 LOAD GENERATOR

LOAD GENERATOR includes the pre-processing program **LOAD-2D** which can be used to generate initial head, specified total head, specified pressure head boundary, external water flow, and potential seepage face boundary. Section 10 in SMAP-W2 Example Problems describes in detail about running **LOAD-2D** program.



3.2.4 PlotXY GENERATOR

PlotXY GENERATOR is the graphical user interface which is mainly used to generate or edit [Simplified Time History](#) and [Simplified Snapshot](#) of Card Group 12 in [SMAP Post File](#). Section 12.7 in SMAP-W2 User's Manual describes in detail about running [PlotXY Generator](#) program.

3.2.5 COMMAND LINE

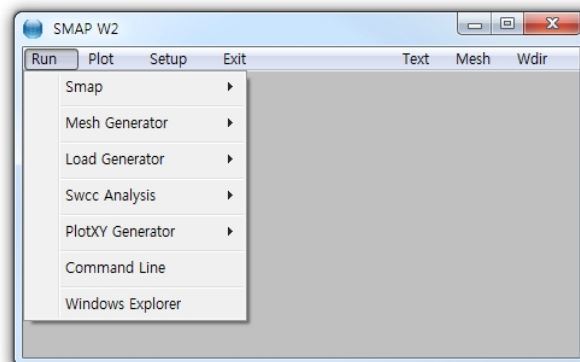
COMMAND LINE opens [Windows Command Prompt](#) at the current Working Directory. you can use a keyboard to navigate, access, and modify files and folders by entering commands. For example, COMMAND LINE is used when executing manually [SMAP](#) main solvers.

3.2.6 WINDOWS EXPLORER

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

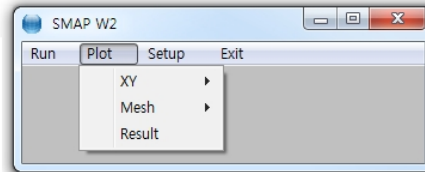
3.2.7 SWCC ANALYSIS

SWCC ANALYSIS opens the input file as prepared according to [SWCC User's Manual](#) and shows graphical outputs of specific moisture capacity, hydraulic conductivity, etc as a function of soil suction.



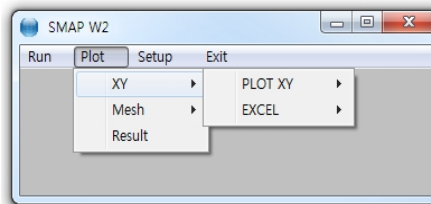
3.3 PLOT Menu

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



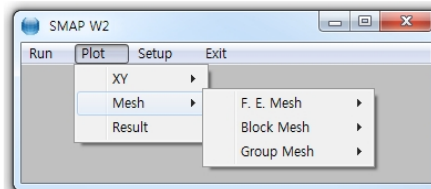
3.3.1 XY

XY graph can be displayed by PLOT-XY or EXCEL. Section 11 in SMAP-W2 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.

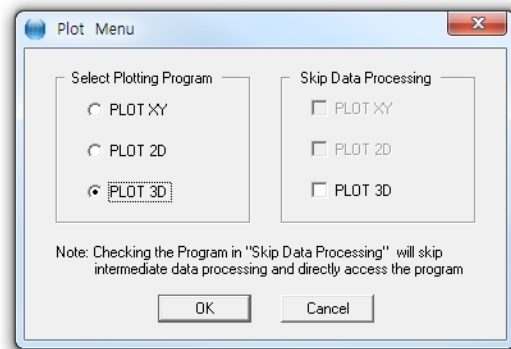
Block Mesh is used to open or build Block Mesh. Section 6 in SMAP-W2 Example Problems describes in detail about running Block Mesh.

Group Mesh is used to open or build Group Mesh. Section 5 in SMAP-W2 Example Problems describes in detail about running Group Mesh.

3.3.3 RESULT

Once you finished executing SMAP-W2 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 computed results and snapshots of computed result vs. distance. Refer to PLOT-XY User's Manual in Section 13.

PLOT-2D reads Card 11 in Post File and plots contours of total head and water pressures. 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 total head, water pressure and iso surface. Refer to PLOT-3D User's Manual in Section 15.

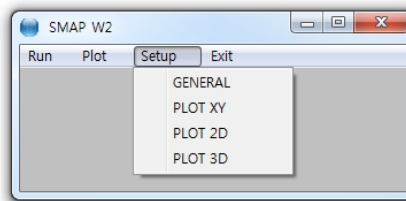
Note: When you first plot results, do not check the check box in Skip Data Processing. When you replot results, however, you can check the check box to skip intermediate data processing. This will save time and keep modified output data.

3.4 SETUP Menu

You need to run SETUP Menu

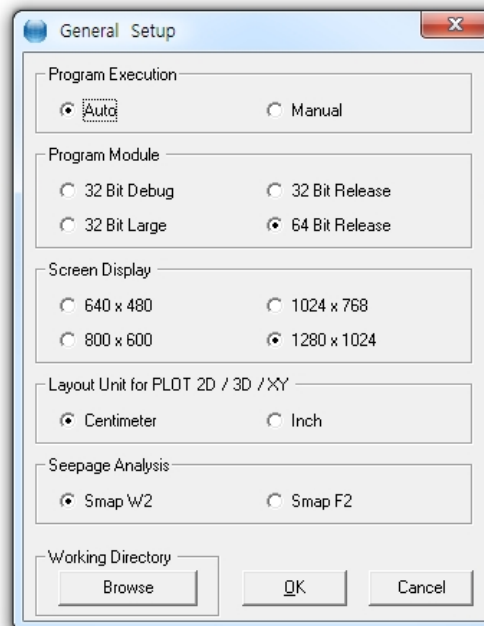
- To specify SMAP-W2 main-processing program module.
- To adjust scales of graphical outputs from PLOT-XY, PLOT-2D, and PLOT-3D

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



3.4.1 General Setup

General Setup has six different items; Program Execution, Program Module, Screen Display, Layout Unit, Seepage Analysis 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.

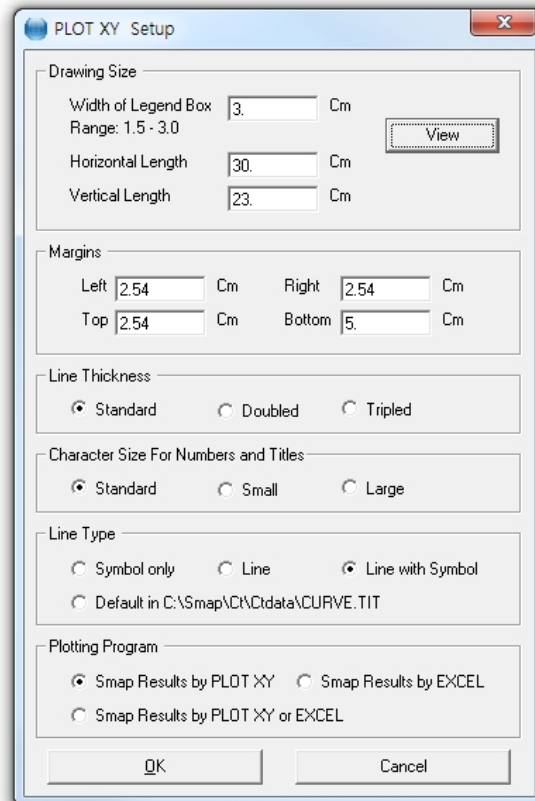
Layout Unit is used for PLOT-XY, PLOT-2D, and PLOT-3D. You can select either Centimeter or Inch in specifying plot scales and dimensions.

Seepage Analysis is used to select main seepage analysis program. You can select either SMAP-W2 or SMAP-F2.

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.



The screenshot shows the 'PLOT XY Setup' dialog box with the following settings:

- Drawing Size:**
 - Width of Legend Box: 3.0 Cm (Range: 1.5 - 3.0)
 - Horizontal Length: 30.0 Cm
 - Vertical Length: 23.0 Cm
 - View button
- Margins:**
 - Left: 2.54 Cm
 - Right: 2.54 Cm
 - Top: 2.54 Cm
 - Bottom: 5.0 Cm
- Line Thickness:**
 - ☒ Standard
 - ☐ Doubled
 - ☐ Tripled
- Character Size For Numbers and Titles:**
 - ☒ Standard
 - ☐ Small
 - ☐ Large
- Line Type:**
 - ☐ Symbol only
 - ☐ Line
 - ☒ Line with Symbol
 - ☐ Default in C:\Smapi\CD\data\CURVE.TIT
- Plotting Program:**
 - ☒ Smapi Results by PLOT XY
 - ☐ Smapi Results by EXCEL
 - ☐ Smapi Results by PLOT XY or EXCEL

Buttons: OK, Cancel

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.

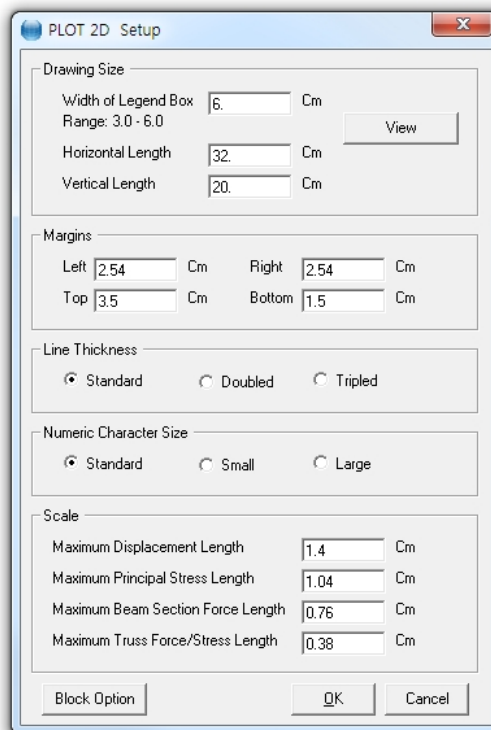
Plotting Program is used to specify default program to plot Smap results. It has three options; PLOT-XY, EXCEL, and PLOT-XY or EXCEL. Last option is to select either PLOT-XY or EXCEL at the time you plot results.

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.



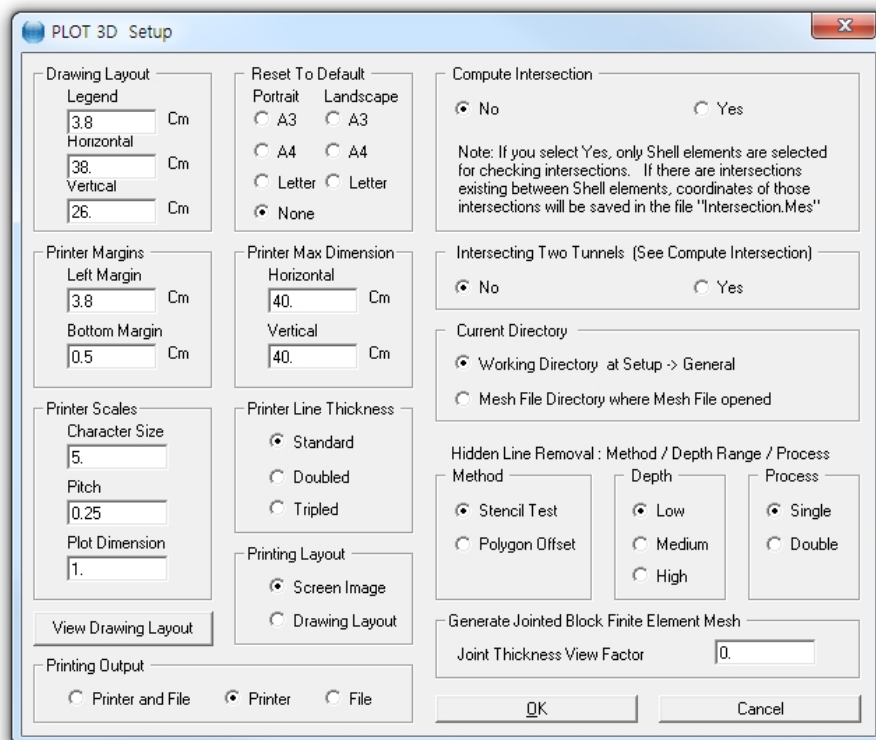
The screenshot shows the 'PLOT 2D Setup' dialog box with the following settings:

- Drawing Size:**
 - Width of Legend Box: 6.00 Cm (Range: 3.0 - 6.0)
 - Horizontal Length: 32.00 Cm
 - Vertical Length: 20.00 Cm
 - View button
- Margins:**
 - Left: 2.54 Cm
 - Right: 2.54 Cm
 - Top: 3.50 Cm
 - Bottom: 1.50 Cm
- Line Thickness:**
 - ☒ Standard
 - ☐ Doubled
 - ☐ Tripled
- Numeric Character Size:**
 - ☒ Standard
 - ☐ Small
 - ☐ Large
- Scale:**
 - Maximum Displacement Length: 1.40 Cm
 - Maximum Principal Stress Length: 1.04 Cm
 - Maximum Beam Section Force Length: 0.76 Cm
 - Maximum Truss Force/Stress Length: 0.38 Cm
- Block Option:** (button)
- Buttons:** OK, Cancel

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 for SMAP-3D. And It can also generate jointed block finite element meshes for SMAP-3D. 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-W2

Occasionally, you need to execute SMAP-W2 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\SmapW2**
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-W2 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_W2.dat" in the directory C:\Smap\Ct\Ctdata\Debug

```
1,      100
IDEBUG, ENDPASS

IDEBUG  =  0 : Do not print debug information.
          1 : Print general debug information.
          2 : Print detailed debug information.

ENDPASS      : Ending pass number.
              No printing debug information after ENDPASS.
```

Debug information is printed on the file Smap_W2.deb in the Working Directory \Temp

This debug 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.

SMAP-W2 User's Manual

4.1 Introduction

To run SMAP-W2 main-processing program, you need to prepare a Project File which contains Mesh File name, Main File name and Post File name.

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

Main File contains all the other data required for the two-dimensional numerical analysis of seepage 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
  C:\Example\VP2.Mes
Main File Name
  C:\Example\VP2.Man
Post File Name
  C:\Example\VP2.Pos
```

4.3 Mesh File

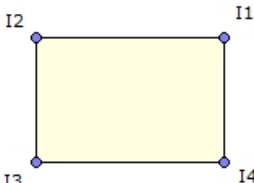
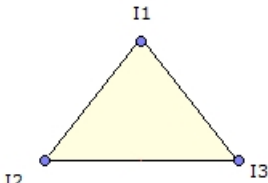
Mesh File contains nodal coordinates, boundary conditions, element indexes and material property numbers. This Mesh File is normally generated by Mesh Generator programs.

To plot Mesh File, select Mesh in Plot menu.

Mesh File

Card Group	Input Data and Definitions (Mesh File)	
1	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 (N.A.) NTRUSS Total number of truss elements (N.A.)

Card Group	Input Data and Definitions (Mesh File)	
2	2.1	LABEL2A [Character string] LABEL2B [Character string] LABEL2A Label for coordinate LABEL2B Label for Card 2.2
	2.2	NUMNP [NODE, ID, IDF, X, Y, H, CF Cards [- - - - - - - - - - - - - - NODE Node number ID = 0 Specified external water flow = 1 Specified total head = 2 Specified pressure head = 3 Specified potential seepage face = 4 Same as ID = 3 but head based on hydrograph IDF Identification number of time dependent function at Card 9.4. Absolute IDF is used. If ID = 1 and IDF < 0, total head is specified based on hydrograph. Above water table, zero flux is specified. If ID < 3 and IDF = 0, reset to ID = 0 and external water flow is zero at all times. If ID = 3, review potential seepage face assign zero flow or zero pressure head If ID = 4, review potential seepage face and assign zero flow or total head based on hydrograph X, Y X and Y coordinates, respectively. H Initial total head CF Coefficient of time dependent function for water flow or total/pressure head at the node.

Card Group	Input Data and Definitions (Mesh File)	
3	3.1	LABEL3A [Character string] LABEL3B [Character string] LABEL3A Label for continuum element LABEL3B Label for Card 3.2
	3.2	NCONT Cards NEL, I₁, I₂, I₃, I₄, MATNO, IDH - - - - - - - - - - - - - - I2 I1 I3 I4  I1 I2 I3  NEL Element number I₁, I₂, I₃, I₄ Element corner node numbers If I₄ = 0, degenerates to triangular element MATNO Materia number IDH Time dependent function ID number for water generation rate (volume per time) at Card 9.4 If IDH < 0, absolute value of IDH refers to time history ID number for specified pressure head.

4.4 Main File

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

Main File contains all the other data required for the two-dimensional numerical analysis of seepage problems.

Main File consists of seven different card groups:

- Title
- Analysis Parameters
- Computational Parameters
- Continuum Element
- Boundary Conditions
- Material Property Data
- Requested Output

Card Group	Input Data and Definitions (Main File)	
1	1.1	
Title	TITLE	
	TITLE	Title (Max 50 characters)
2	2.1	
Analysis Parameters	IP, ILUMP, IBATCH, ISST, IMSEEP, IDFVEL, IFVOL, ISFACE	
	IP	= 0 Axisymmetric geometry about Y-axis = 1 Plane geometry
	ILUMP	= 0 Consistent capacity matrix = 1 Lumped capacity matrix
	IBATCH	= 0 Interactive terminal job = -1 Same as IBATCH = 0 with short beep sound when calculation is finished
	ISST	= 0 Steady-state solution = 1 Transient solution
	IMSEEP	= 0 General seepage face review = 1 Bottom up seepage face review (N.A.)
		Darcy nodal flow vector
	IDFVEL	= 0 Galerkin residual method = 1 Averaging element corner vector
	IFVOL	= 0 Nodal flow volume by global equation = 1 Nodal flow volume by element equation
	ISFACE	= 0 Initial seepage face specified by flux = 1 Initial seepage face specified by head

Card Group	Input Data and Definitions (Main File)
3	3.1 Cycles, Time Step and Iteration Method NCYCL, NSTCL, NITER, MCYCL, TOLER, IRANGE, IRNODE NCYCL Number of solution cycles (Total number of time steps) NSTCL Number of cycles for initial steady-state analysis followed by transient analysis. Used for ISST = 1 NITER Number of maximum iteration MCYCL Number of maximum sub cycles to review potential seepage face. (Default 10) TOLER Tolerance for convergence, defined as max. difference of total heads between current and previous iterations. For negative value, the ratio of total head increment to current total head (Default 0.001) IRANGE = 0 NITER is applied throughout NCYCL NITER is applied during the ranges specified in Card 3.3 = 1 Based on Cycle No = 2 Based on Time IRNODE = 0 NITER is applied throughout all nodes = 1 NITER is applied for selected nodes specified in Card 3.4

Card Group	Input Data and Definitions (Main File)	
3	3.2	
	Solution Iteration Parameter SIP SIP = 0.5 Under relaxation Recommend when solution is fluctuating = 1.0 Exact relaxation = 2.0 Over relaxation	
	Time Steps	3.2.1 IDELT, DT IDELT = 0 Constant time step (DT) = 1 Every step uses new step at Card 3.2.2 = -1 Every 100 steps use new step at Card 3.2.2 DT Constant time step for IDELT = 0
		3.2.2 If IDELT = 0, go to Card Group 3.3 NUMDT DT₁ DT₂ DT₃ ---- DT_{NUMDT} NUMDT Number of specified time steps DT_i Specified time step

Card Group	Input Data and Definitions (Main File)	
3	3.3	If IRANGE = 0, go to Card Group 3.4 NRANGE NRANGE Number of specified ranges where NITER is applied (Max=100)
	For Each Range	3.3.1 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	3.4	If $IRNODE = 0$, go to Card Group 4.1 NRNODE NRNODE Number of specified nodes where NITER is applied (Max=1000)
	3.4.1	$NODE_1, \quad NODE_2, \quad NODE_{NRNODE}$ $NODE_i$ Node numbers to be used for checking convergence

Card Group	Input Data and Definitions (Main File)	
4	4.1	NUMNP NUMNP Total number of nodal points
	4.2	CMFAC, XREF CMFAC Coordinate multiplication factor XREF Reference number to be added to all X coordinate

Coordinates

Card Group	Input Data and Definitions (Main File)		
5	5.1	NCONT : Total number of continuum element	
	5.2	5.2.1	NMAT : Number of material property set
		5.2.2.1	TITLE : Material name (Max 50 characters)
		5.2.2.2	MATNO, MATFN, MW, GW, KH, KV, THETA MATNO Material identification number MATFN = 0 Saturated material < 11 Pressure dependent function ID for nonlinear hydraulic conductivity and specific moisture at Card 9.5 = 11 van Genuchten (1980) = 12 Fredlund and Xing (1994) = 13 Brutsaert (1966) & Gardner (1958) MW Specific moisture capacity at saturation GW Unit weight of water Saturated permeability KH Permeability in major principal direction KV Permeability in minor principal direction THETA Angle (degree) between x axis and major principal direction

Card Group	Input Data and Definitions (Main File)		
5	5.2		5.2.2.3 For MATFN = 11 or 12 VWCsat, VWCres, a, n, m, Uswcc, scfac, Gwm, Pr VWCsat Saturated volumetric water content VWCres Residual volumetric water content Not used for MATFN = 12 a, n, m Parameters defining unsaturated hydraulic conductivity and water storage For MATFN = 11, default $m = 1 - 1/n$ Uswcc Unit in fitting SWCC model 0: pressure unit, 1: pressure head unit scfac Conversion factor from user input unit to SWCC model unit in terms of length Gwm Unit weight of water in SWCC model used for Uswcc = 0 Pr Suction at VWCres. Used for MATFN = 12 When Pr > 0, assumes that suction in SWCC model is expressed in terms of kPa For MATFN = 13 VWCsat, VWCres, A₁, n₁, A₂, n₂, scfac A₁, n₁ Parameters defining unsaturated hydraulic conductivity, Brutsaert (1966) A₂, n₂ Parameters defining unsaturated water storage, Gardner (1958) scfac Conversion factor from user input unit to model unit in terms of length

Card Group	Input Data and Definitions (Main File)				
9	9.4	9.4.1	NTIMF, NTIM, ITIMF		
		NTIMF Number of time functions (Max=25)			
		NTIM Number of time points (Max=11000)			
		ITIMF = 0 Time functions (Card 9.4.2) from Main File			
		= 1 Time functions from TimeFunction.dat			
		First 3 lines represent headers			
		If NTIM = 0, go to Card group 9.5			
Boundary Conditions	Time Dependent Function Specifications	9.4.2			
		TIME ₁ FN _{1, 1} FN _{2, 1} - - - FN _{NTIMF, 1}			
		TIME ₂ FN _{1, 2} FN _{2, 2} - - - FN _{NTIMF, 2}			
		- - - - - - -			
		TIME _{NTIM} FN _{1, NTIM} FN _{2, NTIM} - - - FN _{NTIMF, NTIM}			
		TIME _i Specified time			
		FN _{i, j} Value of function i at specified time j			

Card Group	Input Data and Definitions (Main File)																															
9	9.5	9.5.1 NWPRF, NWPR, IWPf, INTP NWPRF Number of pressure functions (Max = 25) NWPR Number of pressure points (Max=11000) IWPf = 0 Pressure functions (Card 9.5.2) from Main File = 1 Pressure functions from PresFunction.dat First 7 lines represent headers = 2 Same as IWPf = 1 but specific moisture is computed from volumetric water content Interpolations between WPRS and (K, MV) INTP = 0 Linear scales = 1 Semi-logarithmic scales (Arithmetic WPRS) = 2 Logarithmic scales If NWPRF = 0 or IWPf > 0 , go to Card group 10 9.5.2 <table> <tr> <td>WPRS₁</td><td>K_{1, 1}</td><td>MV_{1, 1}</td></tr> <tr> <td></td><td>K_{2, 1}</td><td>MV_{2, 1}</td></tr> <tr> <td></td><td>-</td><td>-</td></tr> <tr> <td>WPRS₂</td><td>K_{1, 2}</td><td>MV_{1, 2}</td></tr> <tr> <td></td><td>K_{2, 2}</td><td>MV_{2, 2}</td></tr> <tr> <td></td><td>-</td><td>-</td></tr> <tr> <td>WPRS_{NWPR}</td><td>K_{1, NWPR}</td><td>MV_{1, NWPR}</td></tr> <tr> <td></td><td>K_{2, NWPR}</td><td>MV_{2, NWPR}</td></tr> <tr> <td></td><td>-</td><td>-</td></tr> <tr> <td></td><td>K_{NWPRF, NWPR}</td><td>MV_{NWPRF, NWPR}</td></tr> </table> WPRS_i Specified water pressure (suction as positive) K_{i, j} Permeability at water pressure WPRS_j MV_{i, j} Specific moisture (MW) at WPRS_j for IWPf = 1 Volumetric water content at WPRS_j for IWPf = 2	WPRS ₁	K _{1, 1}	MV _{1, 1}		K _{2, 1}	MV _{2, 1}		-	-	WPRS ₂	K _{1, 2}	MV _{1, 2}		K _{2, 2}	MV _{2, 2}		-	-	WPRS _{NWPR}	K _{1, NWPR}	MV _{1, NWPR}		K _{2, NWPR}	MV _{2, NWPR}		-	-		K _{NWPRF, NWPR}	MV _{NWPRF, NWPR}
WPRS ₁	K _{1, 1}	MV _{1, 1}																														
	K _{2, 1}	MV _{2, 1}																														
	-	-																														
WPRS ₂	K _{1, 2}	MV _{1, 2}																														
	K _{2, 2}	MV _{2, 2}																														
	-	-																														
WPRS _{NWPR}	K _{1, NWPR}	MV _{1, NWPR}																														
	K _{2, NWPR}	MV _{2, NWPR}																														
	-	-																														
	K _{NWPRF, NWPR}	MV _{NWPRF, NWPR}																														

Card	Input Data and Definitions (Main File)
10 Requested Output	10.1 NTPRNT NTPRNT Number of cycles between output data print
	10.2.1 NHPEL NHPEL Number of elements at which water pressure time histories are requested
	10.2.2 If NHPEL = 0, skip the following Card NEL₁ NEL₂ ... NEL_{NHPEL} NEL Element number to be printed
	10.3.1 NHPMT NHPMT Number of nodes at which total head / pressure time histories are requested
	10.3.2 If NHPMT = 0, skip the following Card NODE₁ NODE₂ ... NODE_{NHPMT} NODE Node number to be printed
	10.4.1 NTIME NTIME Number of times at which total head / pressure profiles are requested
	10.4.2 If NTIME = 0, skip the following Card TIME₁ TIME₂ ... TIME_{NTIME} TIME Times to be printed aaa

Card	Input Data and Definitions (Main File)
10 Requested PLOT-XY Output File: FLOW.Lin	10.5.1 NPHD NPHD Number of water pressure lines
	10.5.2 If NPHD = 0, skip the following Card WP ₁ WP ₂ ... WP _{NPHD} WP Water pressure to be plotted
	10.6.1 NTHD NTHD Number of equipotential lines For NTHD < 0, PLOT-XY plots without legends
	10.6.2 If NTHD = 0, skip the following Card TH ₁ TH ₂ ... TH _{NTHD} TH Total heads to be plotted
	10.7.1 NFLN ISPL DELV NFLN Number of flow lines For NFLN < 0, PLOT-XY plots without legends ISPL Split quad. element into triangles (0: No, 1: Yes) DELV Vector length to trace flow path Default: one half of smallest element length
	10.7.2 If NFLN = 0, skip the following Card NOD ₁ NOD ₂ ... NOD _{NFLN} NOD Node numbers from which flow line starts

Card Group	Input Data and Definitions (Main File)	
10	10.8 NFSEC, IBFQ NFSEC Number of flow sections (Max=100) Compute boundary flow IBFQ = 0 Based on surfaces from global solution = 1 Based on nodes from global solution = 2 Based on surfaces from global flow velocity If NFSEC = 0, go to Card Group 11	
	For Each Flow Section	10.8.1 NODES NOD₁ NOD₂ - - - NOD_{NODES} NODES Number of nodes defining a flow section NOD_i Node number

Requested Output File: FlowSection.out

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
- Total head / water pressure distribution

Card Group 12 contains the input data for following plots:

Time history

- Total head / water pressure

Snapshot

- Total head / water pressure vs. distance

Card Groups 13 is no longer supported.

These plots can be performed automatically by using PLOT-3D.

PLOT-2D
Post-Processor

Card Group	Input Data and Definitions (Post File)
11 PLOT-2D Plot Information	11.1 NPTYPE NPTYPE = 0 End of plotting output = 1 Finite element mesh / element number = 2 Contours of element water pressure If NPTYPE = 0, Skip rest of Card Group 11

Card Group	Input Data and Definitions (Post File)	
11	11.2	11.2.1 TITLE TITLE Any title (Max = 70 characters)
		11.2.2 IUNIT IUNIT = 1 Inch = 2 Cm = 3 User-specified unit
		11.2.3 <u>For IUNIT = 3</u> NCHR LABEL NCHR Number of characters for mesh unit LABEL Name of mesh unit

PLOT-2D Plot Information

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

Card Group

11

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 boundary codes
= 5 Plot material numbers
= -5 Plot node and material numbers

11.2.5

NGROUP

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

11.2.6

If NGROUP = 0, Skip this Card

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

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

PLOT-2D Plot Information

Card Group	Input Data and Definitions (Post File)	
11	For NPTYPE = 2 (Contours of Element Water Pressure)	11.3.1 TITLE TITLE Any title (Max = 70 characters)
		11.3.2 IUNIT IUNIT = 1 In, Fahrenheit = 2 Cm, Centigrade = 3 User-specified unit
		11.3.3 <u>For IUNIT = 3</u> NCHR LABEL NCHRC LABELC NCHR Number of characters for mesh unit LABEL Name of mesh unit NCHRC Number of characters for pressure unit LABELC Name of water pressure unit
		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, 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 = 2 (Contours of Element Water Pressure)	11.3.5 NCTS NCTS Variable to be plotted. Use NCTS = 51
		11.3.6 DELTA, IRES, IRGP, IENL, R_x , R_y DELTA = -DELTA Line contour, absolute value of DELTA is desired contour interval = 0 Color-filled contour = 2 Smoothed color-filled contour IRES = 0 Draft copy = 1 Fine copy IRGP = 0 Values at ref. grid points are not added = 1 Values at ref. grid points are added IENL = 0 Standard view = 2 Laplacian & spline interpolation scheme = 3 Davis distance to a power interpolation <u>For IENL= 2</u> R_x Weight factor applied to spline function If $R_x = 0.0$, only Laplacian interpolation is used R_y is not used <u>For IENL= 3</u> R_y Power applied to $1/(\text{distance}^{**}\text{power})$ 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	11.3	11.3.7 NGROUP NGROUP = 0 Plot at all elements > 0 Plot at specified groups (Max=1000)
		11.3.8 <u>If NGROUP = 0, Skip this Card</u> NGROUP Cards [NSS, NEE, NIC, NNN - - - - - - - - Refer to Card Group 11.2.6
		11.3.9 NRL NRL Number of nodes to be connected by a Solid Line (Max=5000)
		11.3.10 <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

PLOT-2D Plot Information

For NPTYPE = 2 (Contours of Element Water Pressure)

PLOT-XY
Post-Processor

Card Group	Input Data and Definitions (Post File)
12	12.1 IPTYPE IPTYPE 0 End of plotting output Standard Time history 1 Element Water Pressure 2 Nodal Total Head / Water Pressure / Flow Rate / Flow Velocity Standard Snapshot 3 Element Water Pressure vs. Distance 4 Nodal Total Head / Water Pressure / Flow Rate / Flow Velocity vs. Distance Simplified Time history 5 Water Pressure for a Given Element 6 Water Pressure for Different Elements 7 Total Head / Pressure for a Given Node 8 Total Head / Pressure for Different Nodes Simplified Snapshot 9 Element Water Pressure for a Given Time 10 Element Water Pressure for Different Times 11 Nodal Total Head / Pressure for a Given Time 12 Nodal Total Head / Pressure 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.

PLOT-XY Information

Card Group	Input Data and Definitions (Post File)	
12	12.2	12.2.1 IPLOT IPLOT = 0 For each specified element, Number of different pair of variables = 1 For each specified pair of variables, Number of different element data
		12.2.2 NOEL NOEL Number of elements (Max 10)
		12.2.3 LIST (I) I = 1, NOEL LIST (I) List element numbers
		12.2.4 NDPQ NDPQ Number of different pair of variables Use NDPQ = 1

Card Group	Input Data and Definitions	
12	12.2 PLOT-XY Information For IPTYPE = 1 (Element Water Pressure Time History)	12.2.5 NDPQ Cards K_{x1} K_{x2} - - K_{y1} K_{y2} - - K_x Use $K_x = 1$ $K_y = 51$ Element water pressure
		12.2.6 TMFAC, TPFAC TMFAC TPFAC Multiplication factor Time Water pressure
		12.2.7 IPLOT = 0: For each element IPLOT = 1: For each pair of variables TITLE X - LABEL Y - LABEL (50 characters) (50 characters) (50 characters)

Card Group	Input Data and Definitions (Post File)	
12	12.3	12.3.1 IPLOT IPLOT = 0 For each specified node, Number of different pair of variables = 1 For each specified pair of variables, Number of different node data
		12.3.2 NODE NODE Number of nodes (Max 10)
		12.3.3 LIST (I), I = 1, NODE LIST (I) List node numbers

Card Group	Input Data and Definitions (Post File)									
12	12.3	12.3.4								
		NDPQ								
		NDPQ Number of different pair of variables Use NDPQ = 1								
		12.3.5								
		NDPQ <table><tr><td rowspan="4">Cards</td><td rowspan="4">[</td><td>K_{x1r}</td><td>K_{y1}</td></tr><tr><td>K_{x2r}</td><td>K_{y2}</td></tr><tr><td>-</td><td>-</td></tr><tr><td>-</td><td>-</td></tr></table> K_x Use $K_x = 1$ $K_y =$ 41: Total head 42: Water pressure 43: Flow rate 44: Flow velocity 45, 46, 47: Flow velocity X, Y, Z	Cards	[	K_{x1r}	K_{y1}	K_{x2r}	K_{y2}	-	-
Cards	[	K_{x1r}			K_{y1}					
		K_{x2r}			K_{y2}					
		-			-					
		-	-							
12.3.6										
TMFAC, TPFAC										

PLOT-XY Information

For IPTYPE = 2 (Nodal Total Head / Pressure Time History)

Card Group	Input Data and Definitions (Post File)	
12	12.4	12.4.1 IPLOT IPLOT = 0 For each specified time, Number of different variables = 1 For each specified variable, Number of different time data
		12.4.2 NOTM NOTM Number of times (Max 10)
		12.4.3 TLIST (I), I = 1, NOTM TLIST (I) List times in sequential order
		12.4.4 NDPQ NDPQ Number of different variables Use NDPQ = 1
		12.4.5 NDPQ Cards $\begin{matrix} \lceil & K_{y1} \\ & K_{y2} \\ & - \\ \lfloor & \end{matrix}$ K_y Use K_y = 51

PLOT-XY Information

For IPTYPE = 3 (Element Water Pressure vs Distance)

Card Group	Input Data and Definitions (Post File)	
12	12.4	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 <u>Element Number Specification (Max 800 Elements)</u> 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 $N_1, N_2, \dots, N_{NRL}$ Element numbers NSTAR Starting element numbers NINCR Element number increment NPONT Number of element

Card Group	Input Data and Definitions (Post File)	
12	12.4	12.4.8 TPFAC, SDFAC TPFAC SDFAC Multiplication factor Water pressure Distance
		12.4.9 IPLOT = 0: For each specified time IPLOT = 1: For each variable TITLE X-LABEL Y-LABEL (50 characters) (50 characters) (50 characters)

PLOT-XY Information

For IPTYPE = 3 (Element Water Pressure vs Distance)

Card Group	Input Data and Definitions (Post File)	
12	12.5	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 Use NDPQ = 1
		12.5.5 NDPQ ⎧ K_{y1} K_{y2} Cards ⎩ - K_y = 41: Total head 42: Water pressure 43: Flow rate 44: Flow velocity 45, 46, 47: Flow velocity X, Y, Z

Card Group	Input Data and Definitions (Post File)	
12	12.5	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

PLOT-XY Information

For IPTYPE = 4 (Nodal Total Head / Pressure vs Distance)

Card Group	Input Data and Definitions (Post File)	
12	12.5	12.5.7 <u>Node Number Specification (Max 800 nodes)</u> For Arbitrary Order > 1 NRL $N_1, N_2, \dots, N_{NRL}$ For Sequential Order > 2 NSTAR, NINCR, NPONT For End Generation > 0 NRL Number of nodes $N_1, N_2, \dots, N_{NRL}$ Node numbers NSTAR Starting node number NINCR Node number increment NPONT Number of nodes 12.5.8 TPFAC, SDFAC Multiplication factor TPFAC Total head / Water pressure / $K_y = 43 \sim 47$ SDFAC 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 Water Pressure for a Given Element)	12.6.1 NEL NEL Element number
		12.6.2 NDQ NDQ Number of different quantities Use NDQ = 1
		12.6.3 K_{y1} $K_y = 51$ Element water pressure
		12.6.4 TMFAC, TPFAC Multiplication factor TMFAC Time TPFAC Water pressure
		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 Water Pressure 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 $K_x = 1$ and $K_y = 51$
		12.7.4 TMFAC, TPFAC TMFAC Multiplication factor Time TPFAC Water pressure
		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 Total Head / Pressure for a Given Node)	12.8.1 NOD NOD Node number
		12.8.2 NDQ NDQ Number of different quantities Use NDQ = 1
		12.8.3 NDQ Cards [] K_{y1} K_{y2} - - K_y = 41: Total head 42: Water pressure 43: Flow rate 44: Flow velocity
		12.8.4 TMFAC, TPFAC TMFAC TPFAC Multiplication factor Time Total head / Water pressure / K_y = 43 ~ 44
		12.8.5 TITLE X - LABEL Y - LABEL (50 characters) (50 characters) (50 characters)

Card Group	Input Data and Definitions	
12	PLOT-XY Information For IPTYPE = 8 (Time History of Total Head / Pressure 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 Use $K_x = 1$ $K_y =$ 41: Total head 42: Water pressure 43: Flow rate 44: Flow velocity
		12.9.4 TMFAC, TPFAC Multiplication factor TMFAC Time TPFAC Total head / Water pressure / $K_y = 43 \sim 44$
		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 Water Pressure for a Given Time)	12.10.1 TIME TIME Specified time
		12.10.2 NDQ NDQ Number of different quantities Use NDQ = 1
		12.10.3 NDQ Cards $\left[\begin{array}{l} K_{y1} \\ K_{y2} \\ - \end{array} \right]$ K_y Use $K_y = 51$
		12.10.4 XSTART XSTART Reference starting X-coordinate
		12.10.5 <u>Element Number Specification (Max 800 Elements)</u> NRL N_1, N_2, N_{NRL} NRL Number of elements $N_1, N_2, \dots, N_{NRL}$ 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 TPFAC, SDFAC TPFAC Multiplication factor SDFAC Water pressure 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 Water Pressure 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 Use $K_y = 51$
		12.11.4 XSTART XSTART Reference starting X-coordinate
		12.11.5 <u>Element Number Specification (Max 800 Elements)</u> NRL N_1, N_2, N_{NRL} NRL Number of elements $N_1, N_2, \dots, N_{NRL}$ 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 TPFAC, SDFAC TPFAC Multiplication factor TPFAC Water pressure SDFAC Distance SDFAC
		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 Total Head / Pressure for a Given Time)	12.12.1 TIME TIME Specified time
		12.12.2 NDQ NDQ Number of different quantities Use NDQ = 1
		12.12.3 NDQ $\left[\begin{array}{l} K_{y1} \\ \vdots \\ L \end{array} \right]$ Cards $\left[\begin{array}{l} \\ \vdots \\ \end{array} \right]$ $K_y =$ 41: Total head 42: Water pressure 43: Flow rate 44: Flow velocity
		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 $N_1, N_2, \dots, N_{NRL}$ 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 TPFAC, SDFAC TPFAC Multiplication factor SDFAC Total head / Water pressure / $K_y = 43 \sim 44$ 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 Total Head / Pressure 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 $K_y =$ 41: Total head 42: Water pressure 43: Flow rate 44: Flow velocity
		12.13.4 XSTART XSTART Reference starting X-coordinate
		12.13.5 <u>Node Number Specification (Max 800 Nodes)</u> NRL N_1, N_2, N_{NRL} NRL Number of nodes $N_1, N_2, \dots, N_{NRL}$ 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 TPFAC, SDFAC Multiplication factor TPFAC Total head / Water pressure / $K_y = 43 \sim 44$ SDFAC Distance
		12.13.7 TITLE (50 characters) X - LABEL (50 characters) Y - LABEL (50 characters)

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](#).

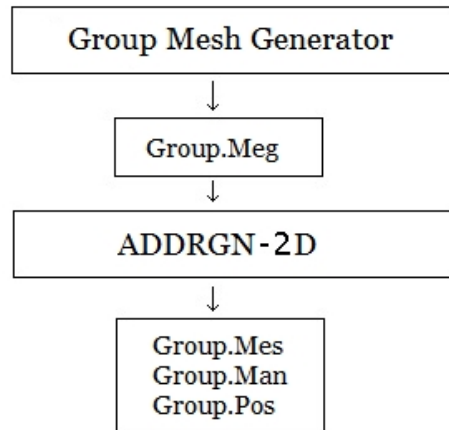


Figure 5.1 Flow diagram of group mesh generation

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

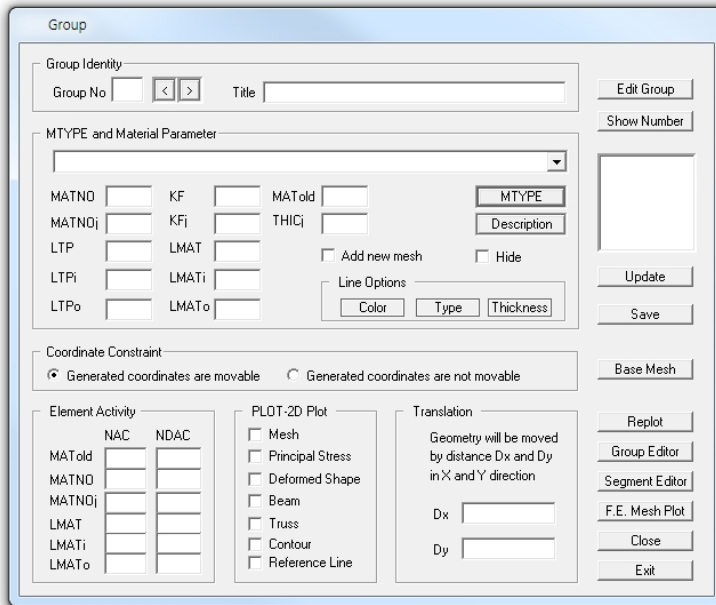


Figure 5.2 Group dialog.

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

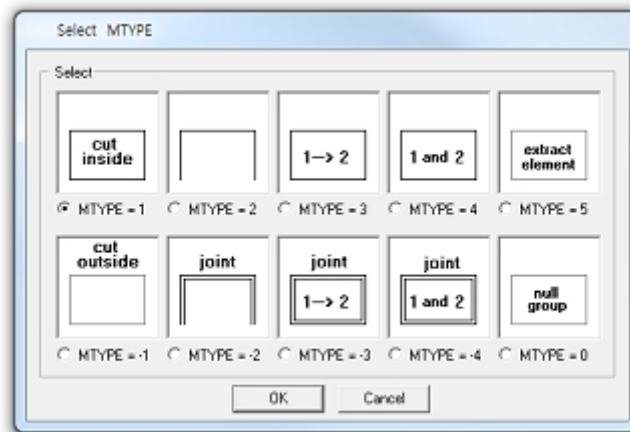
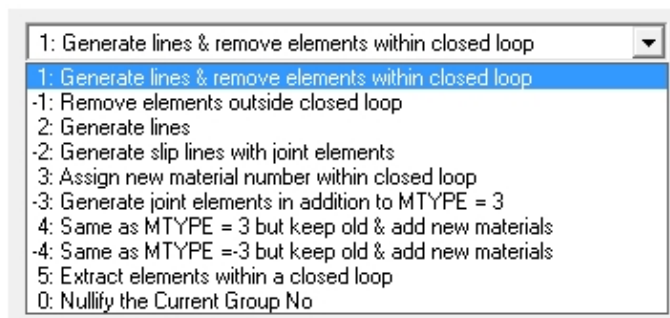


Figure 5.4
MTYPE list box



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.

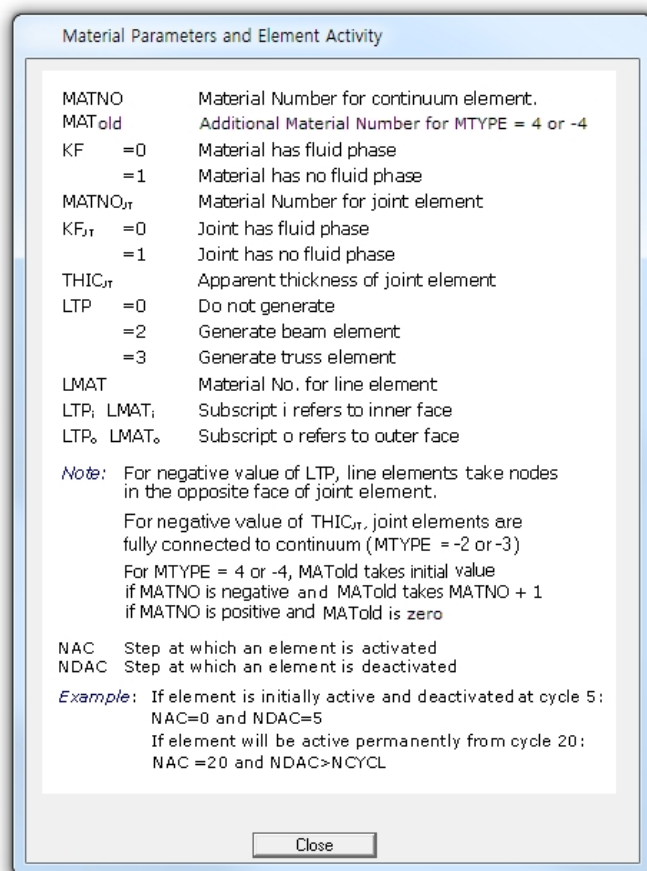


Figure 5.5 Material parameters & element activity ([SMAP-2D](#))

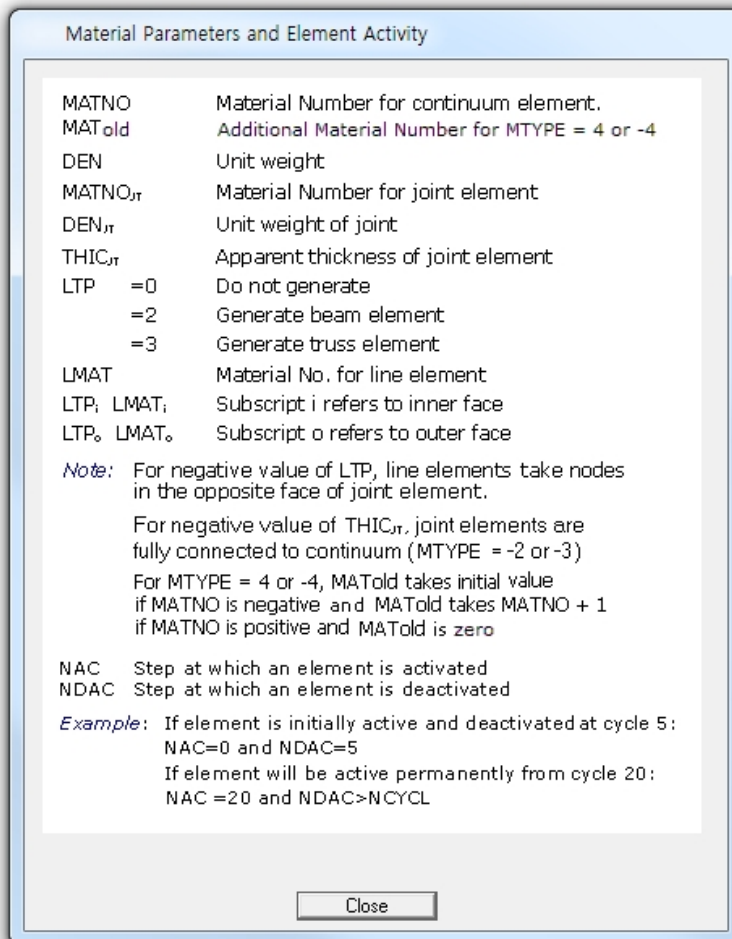


Figure 5.5 Material parameters & element activity (SMAP-S2)

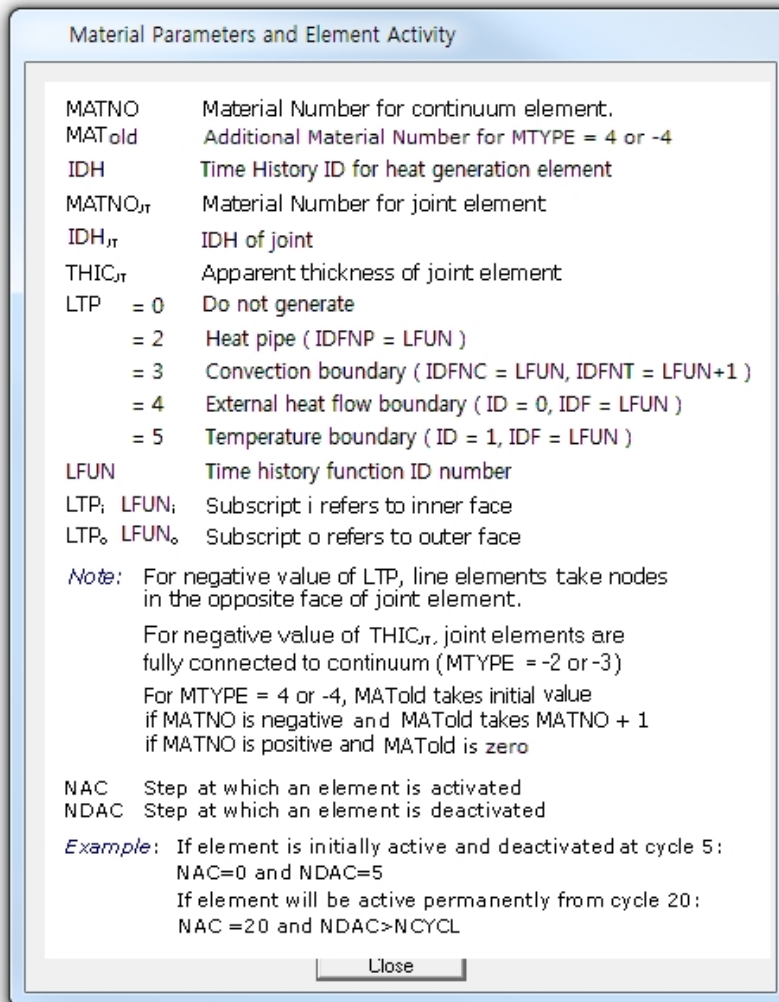


Figure 5.5 Material parameters & element activity (SMAP-T2)

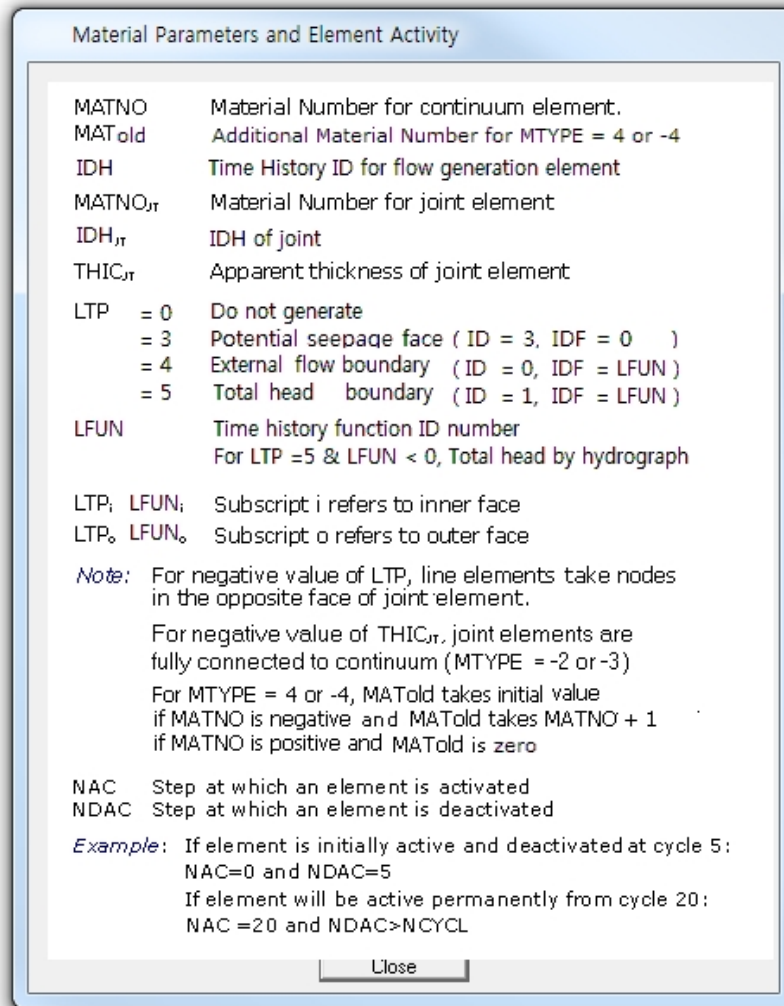


Figure 5.5 Material parameters & element activity ([SMAP-W2](#))

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.

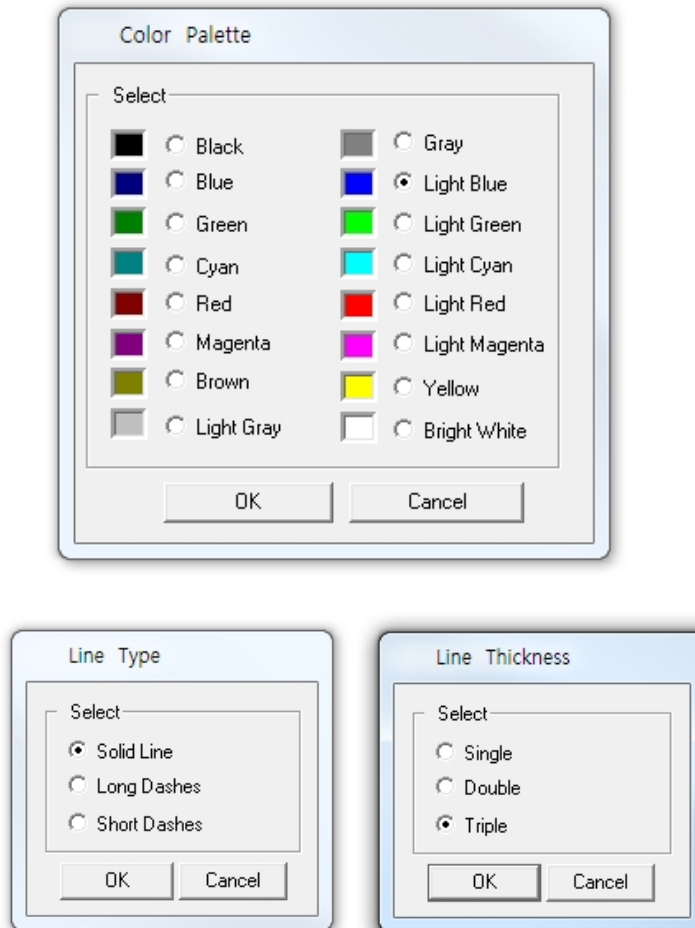


Figure 5.6 Line options.

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.

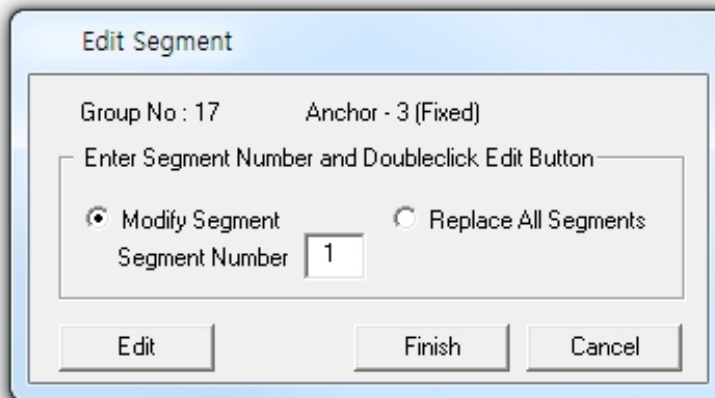


Figure 5.7 Edit segment dialog.

[Show Number](#)

This is used to show group and segment numbers.

[Plot Group / Segment No](#) dialog in Figure 5.8 will be displayed.

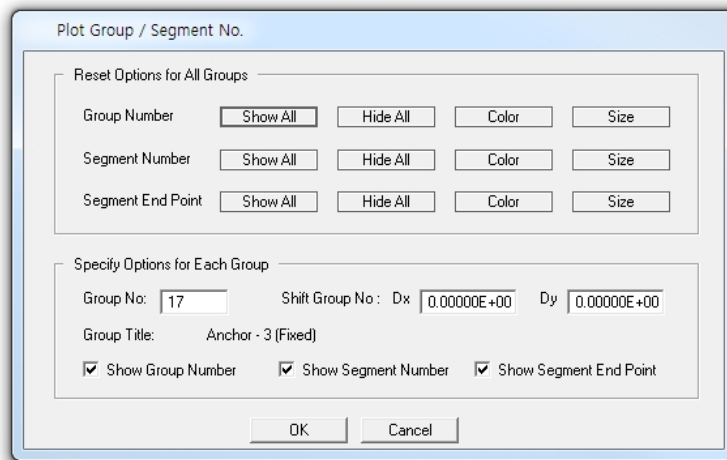


Figure 5.8 Plot Group / Segment No dialog.

[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.

[illegible]

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.

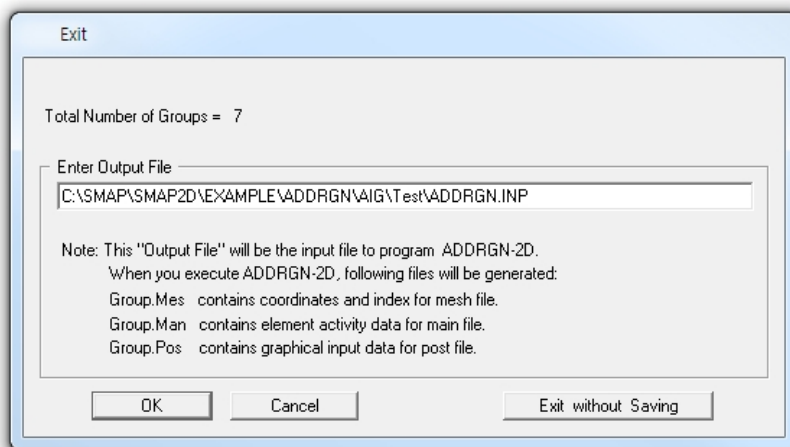


Figure 5.11 Exit dialog.

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.

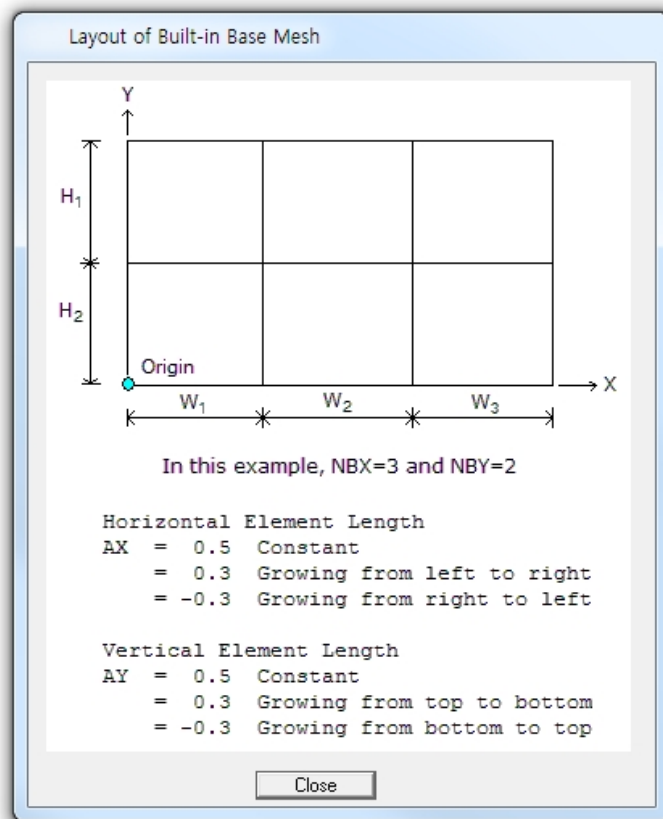


Figure 5.12 Layout of built-in base mesh.

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

Built-in Base Mesh

Horizontal Block
Horizontal blocks are defined from left to right.
Number of blocks in X direction: 3

No.	Width (w)	Element Size (DX)	Normalized Midpoint (AX)
1	45.000	0.50000	-0.3
2	20.000	0.50000	0.5
3	20.000	0.50000	0.3
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			
15			
16			

Vertical Block
Vertical blocks are defined from top to bottom.
Number of blocks in Y direction: 2

No.	Height (H)	Element Size (DY)	Normalized Midpoint (AY)
1	17.000	0.50000	0.5
2	15.500	0.50000	0.3
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			
15			
16			

Origin
Xo: -45.000
Yo: -20.000

Water Table
For total stress analysis, set Ywater lower than Yo
Ywater: -30.000

Boundary Condition
Top: 0 Free
Left: 1 Roller
Right: 1 Roller
Bottom: 1 Roller

Base Mesh Layout Description

OK Cancel

Figure 5.13 Built-in base mesh dialog.

5.4.2 Existing Finite Element Mesh

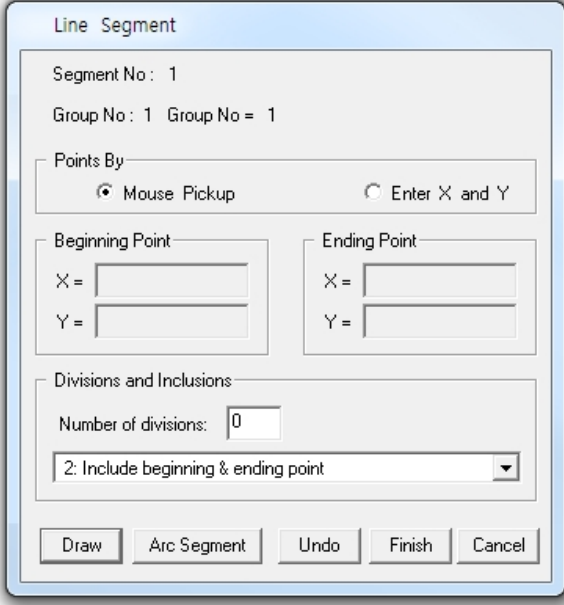
You can provide existing finite element mesh file to be used as base mesh. Group meshes will be built on this finite element mesh.

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.

The image shows a 'Line Segment' dialog box. At the top, it has a title bar 'Line Segment'. Below the title bar, there are two labels: 'Segment No : 1' and 'Group No : 1 Group No = 1'. Underneath, there is a section 'Points By' with two radio buttons: 'Mouse Pickup' (which is selected) and 'Enter X and Y'. Below this, there are two columns of input fields. The left column is labeled 'Beginning Point' and has 'X =' and 'Y =' labels followed by empty text boxes. The right column is labeled 'Ending Point' and also has 'X =' and 'Y =' labels followed by empty text boxes. Below these columns is a section 'Divisions and Inclusions' with a label 'Number of divisions:' followed by a text box containing the number '0'. Below that is a dropdown menu with the text '2: Include beginning & ending point'. At the bottom of the dialog box, there are five buttons: 'Draw', 'Arc Segment', 'Undo', 'Finish', and 'Cancel'.

[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**,

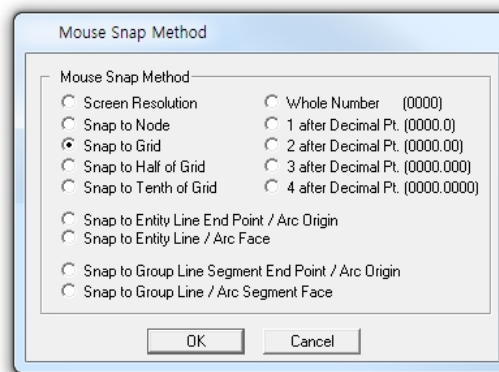
1. Click **Draw** button.
2. Move the mouse to the point and click left mouse button. Or hold down left mouse button, move the mouse and release the button at the point.

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

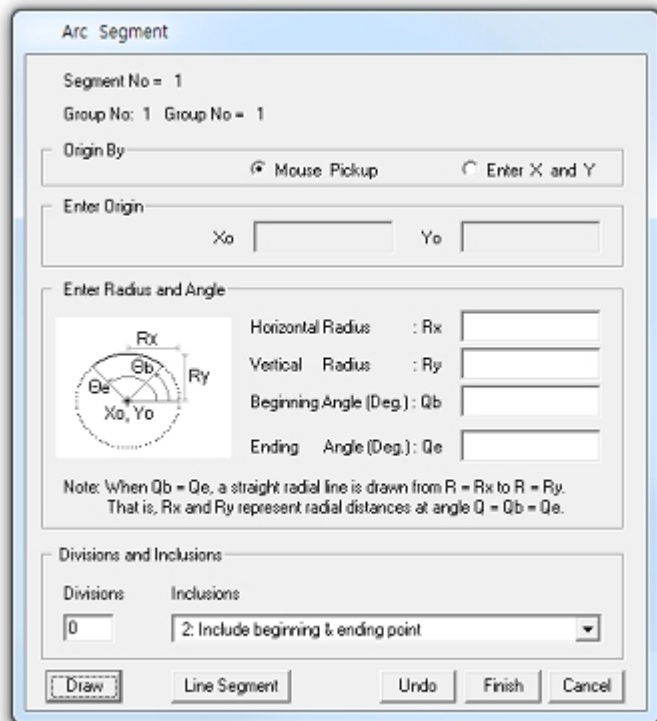


Figure 5.16 Arc segment dialog.

[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.

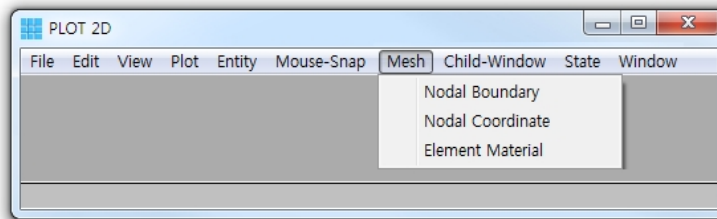


Figure 5.17 Menu for editing finite element mesh

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**)

The dialog box is titled "New Boundary Code". It has two radio buttons under "Node Number By": "Mouse Pickup" (selected) and "Enter Node No". To the right of the radio buttons is a text box labeled "Enter Node No" containing the value "1". Below this is a section titled "New Boundary Code" containing a grid of seven checkboxes: ISX, ISY, IFX, IFY, IRZ, IEX, and IEY. The values for these checkboxes are 1, 0, 1, 1, 1, 1, and 1 respectively. Below the grid, there is a legend: "= 0 Free to move in specified direction." and "= 1 Fixed in specified direction." At the bottom of the dialog are two buttons: "Select Node" and "Cancel".

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**)

The dialog box is titled "Select Node By Mouse Right Click". It has two radio buttons under "Node Number By": "Mouse Pickup" (selected) and "Enter Node No". To the right of the radio buttons is a text box labeled "Enter Node No" containing the value "386". Below this is a section titled "New Boundary Code" containing a grid of seven checkboxes: ISX, ISY, IFX, IFY, IRZ, IEX, and IEY. The values for these checkboxes are 1, 0, 1, 1, 1, 1, and 1 respectively. Below the grid, there is a legend: "= 0 Free to move in specified direction." and "= 1 Fixed in specified direction." At the bottom of the dialog are two buttons: "Apply Code" and "Cancel".

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**)

Select Node By Mouse Right Click

Node Number By: ☒ Mouse Pickup ☐ Enter Node No

Enter Node No:

New Boundary Code

ISX	ISY	IFX	IFY	IRZ	IEX	IEY
1	0	1	1	1	1	1

= 0 Free to move in specified direction.
= 1 Fixed in specified direction.

Undo Finish Cancel

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**)

New Boundary Code

Node Number By: ☐ Mouse Pickup ☒ Enter Node No

Enter Node No:

New Boundary Code

ISX	ISY	IFX	IFY	IRZ	IEX	IEY
1	0	1	1	1	1	1

= 0 Free to move in specified direction.
= 1 Fixed in specified direction.

Apply Code Cancel

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

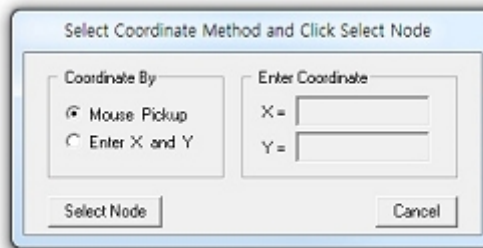
5.6.2 Edit Nodal Coordinate

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

5.6.2.1 Mouse Pickup

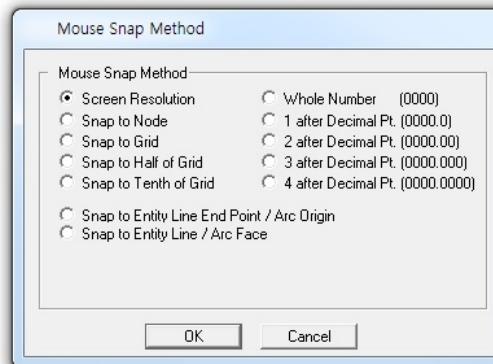
When you select **Mouse Pickup** mode as in Figure 5.22, you are supposed to select node number by mouse click. Click **Select Node** button.

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.

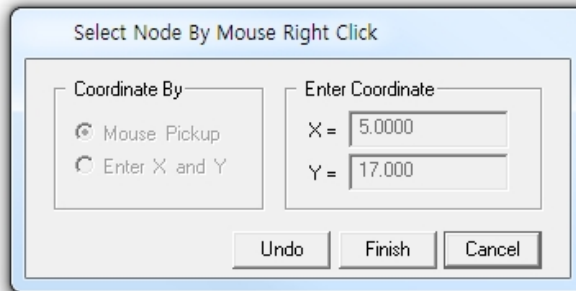


Figure 5.24 Edit coordinate (**Finish**)

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.

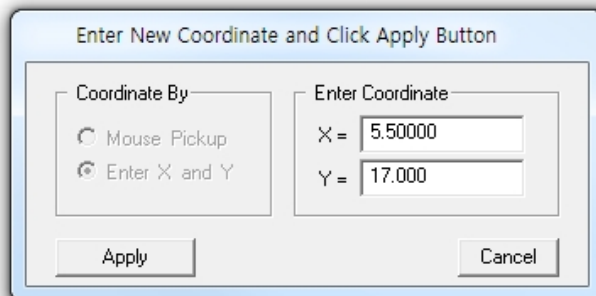


Figure 5.25 Edit coordinate (**Enter X and Y**)

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**)

New Material Parameter

Element Number By: ☒ Mouse Pickup ☐ Enter Element No

Element No: 1

New Material Parameter

MATNo	KS	KF	TBJwL
1	0	1	0.00000

KS = 0:Solid, > 0:Joint Face No, -1:Detonation
KF = 0:Fluid, TBJwL: Det. Time for KS=1

Select Element Cancel

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**)

Select Element By Mouse Right Click

Element Number By: ☒ Mouse Pickup ☐ Enter Element No

Element No: 334

New Material Parameter

MATNo	KS	KF	TBJwL
2	0	1	0.00000

KS = 0:Solid, > 0:Joint Face No, -1:Detonation
KF = 0:Fluid, TBJwL: Det. Time for KS=1

Apply Cancel

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**)

Select Element By Mouse Right Click

Element Number By: ☐ Mouse Pickup ☐ Enter Element No

Element No: 334

New Material Parameter

MATNo	KS	KF	TBJWL
2	0	1	0.00000

KS = 0:Solid, > 0:Joint Face No., -1:Detonation
KF = 0:Fluid, TBJWL: Det. Time for KS=-1

Undo Finish Cancel

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**)

New Material Parameter

Element Number By: ☐ Mouse Pickup ☒ Enter Element No

Element No: 224

New Material Parameter

MATNo	KS	KF	TBJWL
1	0	1	0.00000

KS = 0:Solid, > 0:Joint Face No., -1:Detonation
KF = 0:Fluid, TBJWL: Det. Time for KS=-1

Apply Cancel

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.

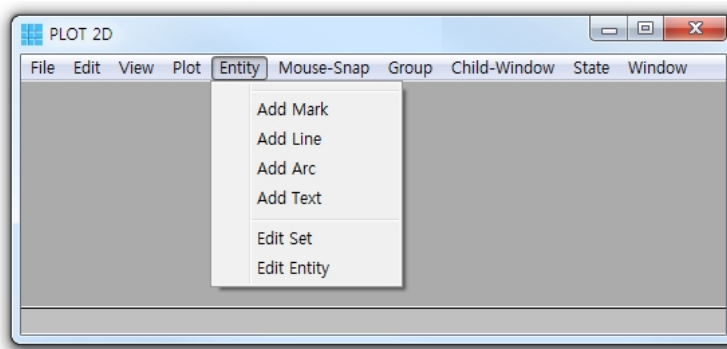


Figure 5.30 Entity menu

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.

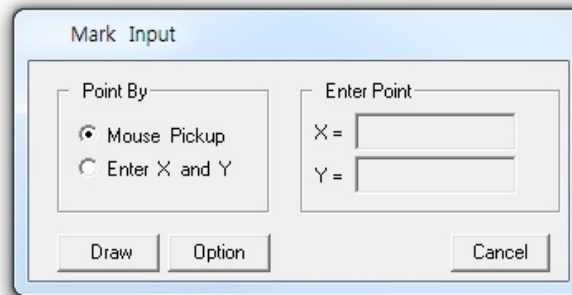
Edit Entity is to modify, delete or replace the selected entity.

5.7.1 Add Mark

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

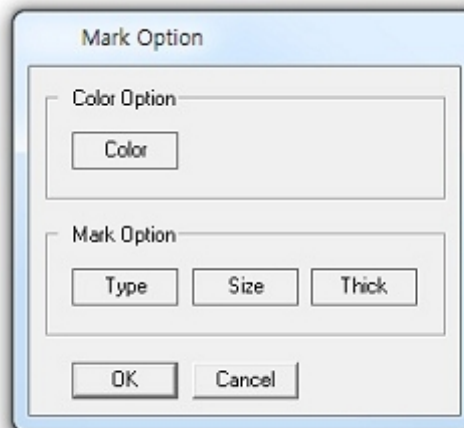
When you select **Add Mark** submenu, **Mark Input** dialog in Figure 5.31 is displayed.

Figure 5.31
Mark input
(Mouse Pickup)



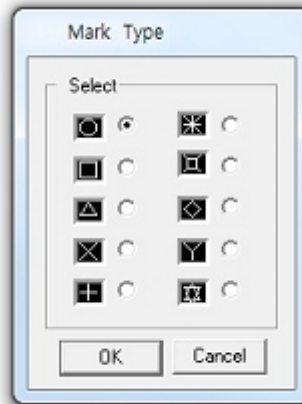
Option button is to show **Mark Option** in Figure 5.32.

Figure 5.32
Mark option dialog



Available [Mark Types](#) are shown in Figure 5.33.

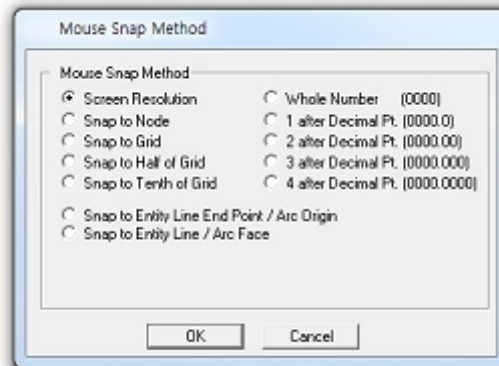
Figure 5.33 Mark type dialog



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**)

Mark Input

Point By

☒ Mouse Pickup

☐ Enter X and Y

Enter Point

X = 21.500

Y = 11.500

Finish Undo Cancel

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**)

Mark Input

Point By

☐ Mouse Pickup

☒ Enter X and Y

Enter Point

X = 20

Y = 20

Draw Option Cancel

Once finished, click **Finish** button in Figure 5.37.

Figure 5.37
Mark input
(**Finish**)

Mark Input

Point By

☐ Mouse Pickup

☒ Enter X and Y

Enter Point

X = 20

Y = 20

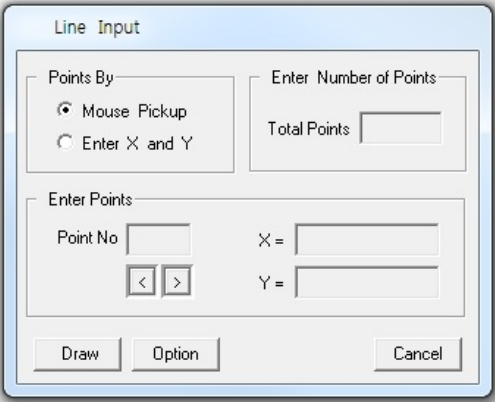
Finish Undo Cancel

5.7.2 Add Line

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

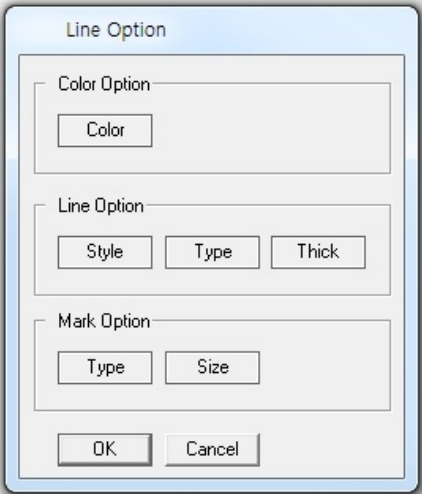
When you select Add Line submenu, Line Input dialog in Figure 5.38 is displayed.

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.

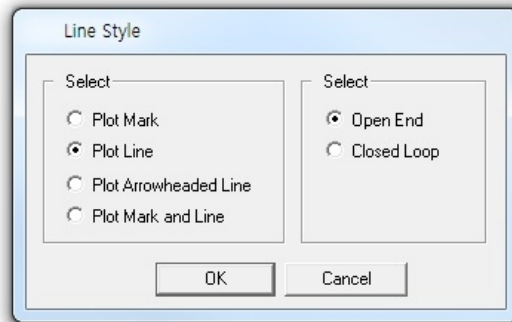


Figure 5.40 Line style dialog

Available [Line Types](#) are shown in Figure 5.41.

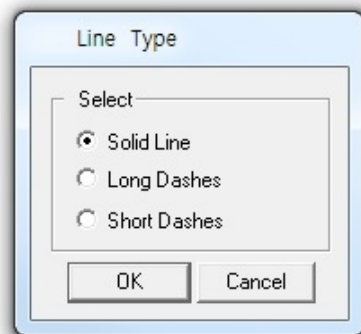


Figure 5.41 Line type dialog

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.

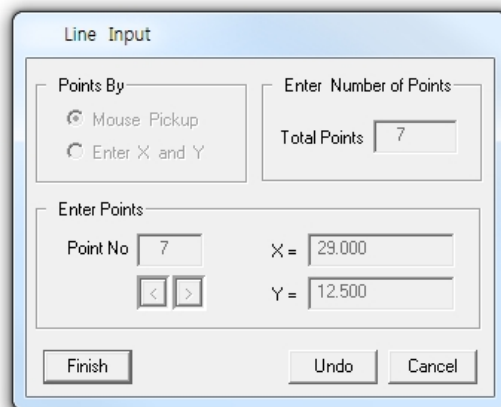
The image shows a 'Line Input' dialog box with a light blue title bar. Inside, there are three main sections. The first section, 'Points By', contains two radio buttons: 'Mouse Pickup' (which is selected) and 'Enter X and Y'. The second section, 'Enter Number of Points', has a label 'Total Points' followed by a text box containing the number '7'. The third section, 'Enter Points', contains a 'Point No' text box with '7', two arrow buttons ('<' and '>'), and two coordinate input fields: 'X =' with '29.000' and 'Y =' with '12.500'. At the bottom, there are three buttons: 'Finish', 'Undo', and 'Cancel'.

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**)

The dialog box is titled "Line Input". It contains two main sections. The first section, "Points By", has two radio buttons: "Mouse Pickup" (unselected) and "Enter X and Y" (selected). The second section, "Enter Number of Points", has a label "Total Points" and a text input field containing the number "3". Below these is the "Enter Points" section, which includes a "Point No" label and a text input field containing "3", followed by left and right arrow buttons. To the right of this are two text input fields labeled "X =" and "Y =", both containing the value "10". At the bottom, there are three buttons: "Draw", "Option", and "Cancel".

And then click **Finish** button in Figure 5.44.

Figure 5.44
Line input
(**Finish**)

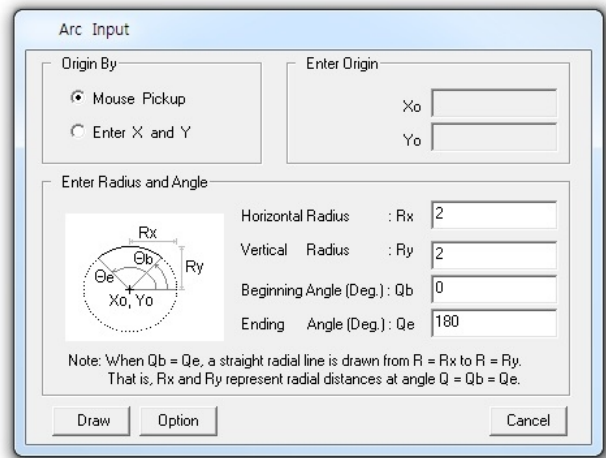
This dialog box is identical to Figure 5.43, but the "Draw" button at the bottom has been replaced by a "Finish" button. The "Option", "Undo", and "Cancel" buttons remain in their original positions.

5.7.3 Add Arc

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

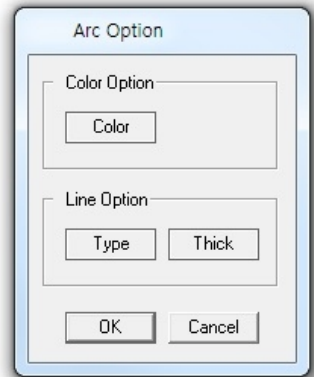
When you select Add Arc submenu, Arc Input dialog in Figure 5.45 is displayed.

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.

Arc Input

Origin By:

☒ Mouse Pickup

☐ Enter X and Y

Enter Origin:

Xo 37.000

Yo 6.0000

Enter Radius and Angle:

Horizontal Radius : Rx 2

Vertical Radius : Ry 2

Beginning Angle (Deg.) : Qb 0

Ending Angle (Deg.) : Qe 180

Note: When $Qb = Qe$, a straight radial line is drawn from $R = Rx$ to $R = Ry$. That is, Rx and Ry represent radial distances at angle $Q = Qb = Qe$.

Finish Cancel

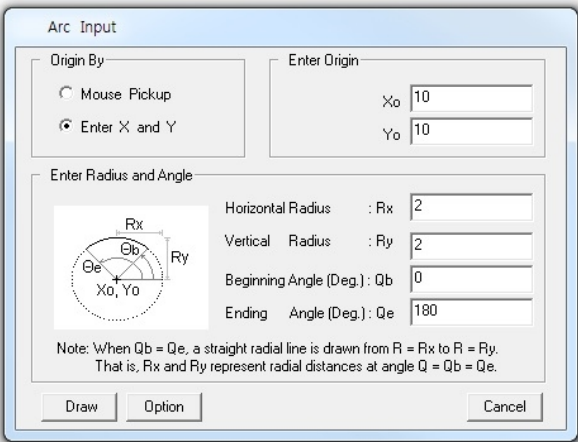
Figure 5.47 Arc input (**Finish**)

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.

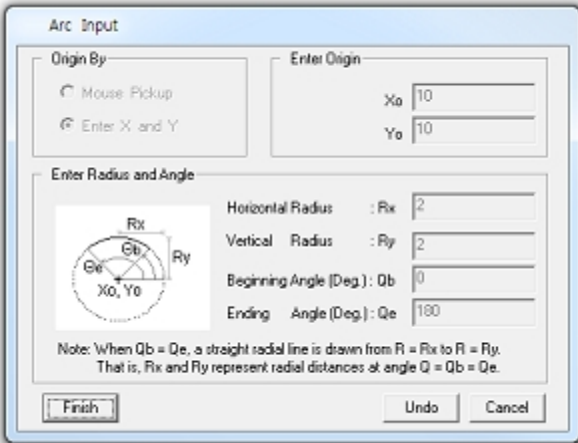
Type in **Horizontal Radius**, **Vertical Radius**, **Beginning Angle** and **Ending Angle**. And then click **Draw** button.

Figure 5.48
Arc input
(Enter X and Y)



Once finished,
click **Finish** button
in Figure 5.49.

Figure 5.49
Arc input
(Finish)

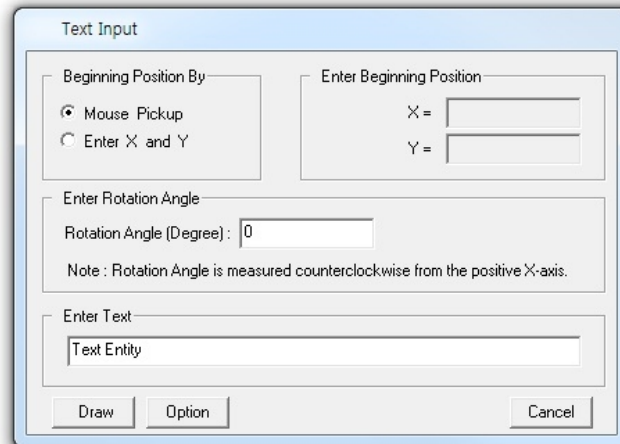


5.7.4 Add Text

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.

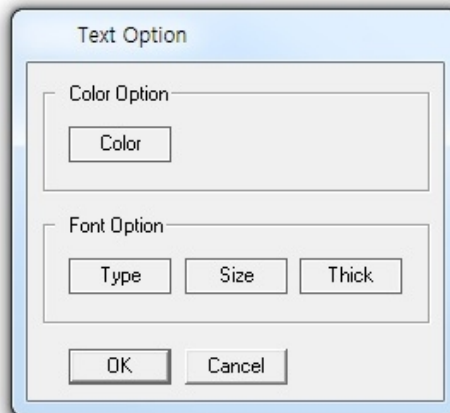
Figure 5.50
Text input
(Mouse Pickup)



The **Text Input** dialog box is shown. It has a title bar labeled "Text Input". Inside, there are three main sections. The first section is "Beginning Position By" with two radio buttons: "Mouse Pickup" (selected) and "Enter X and Y". To the right of this is a section titled "Enter Beginning Position" with input fields for "X =" and "Y =". The second section is "Enter Rotation Angle" with a text input field for "Rotation Angle (Degree):" containing the value "0". Below this is a note: "Note : Rotation Angle is measured counterclockwise from the positive X-axis." The third section is "Enter Text" with a text input field containing "Text Entity". At the bottom are three buttons: "Draw", "Option", and "Cancel".

Option button is to show **Text Option** in Figure 5.51.

Figure 5.51
Text option dialog



The **Text Option** dialog box is shown. It has a title bar labeled "Text Option". Inside, there are two main sections. The first section is "Color Option" with a "Color" button. The second section is "Font Option" with three buttons: "Type", "Size", and "Thick". At the bottom are two buttons: "OK" and "Cancel".

Available [Font Sizes](#) are shown in Figure 5.52.

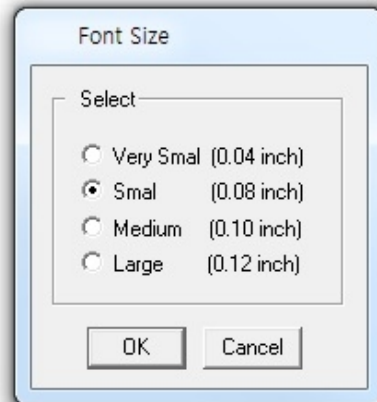


Figure 5.52 Font size dialog

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.

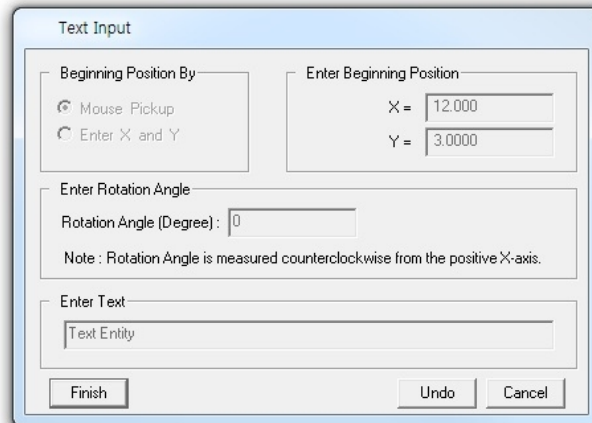
The image shows a 'Text Input' dialog box with a light blue title bar. It contains three main sections: 'Beginning Position By' with radio buttons for 'Mouse Pickup' (selected) and 'Enter X and Y'; 'Enter Beginning Position' with input fields for 'X =' (12.000) and 'Y =' (3.0000); and 'Enter Rotation Angle' with a 'Rotation Angle (Degree):' input field set to '0'. A note below states: 'Note : Rotation Angle is measured counterclockwise from the positive X-axis.' At the bottom is an 'Enter Text' section with a 'Text Entity' input field. At the very bottom are three buttons: 'Finish', 'Undo', and 'Cancel'.

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**)

The 'Text Input' dialog box is shown with the 'Beginning Position By' section set to 'Enter X and Y' (selected with a radio button). The 'Enter Beginning Position' section has 'X = 10' and 'Y = 10' entered. The 'Enter Rotation Angle' section has 'Rotation Angle (Degree): 0' entered. The 'Enter Text' section has 'Text Entity' entered. The 'Draw' button is highlighted.

Once finished, click **Finish** button in Figure 5.55.

Figure 5.55
Text input
(**Finish**)

The 'Text Input' dialog box is shown with the 'Beginning Position By' section set to 'Enter X and Y' (selected with a radio button). The 'Enter Beginning Position' section has 'X = 10' and 'Y = 10' entered. The 'Enter Rotation Angle' section has 'Rotation Angle (Degree): 0' entered. The 'Enter Text' section has 'Text Entity' entered. The 'Finish' button is highlighted.

5.7.5 Edit Set

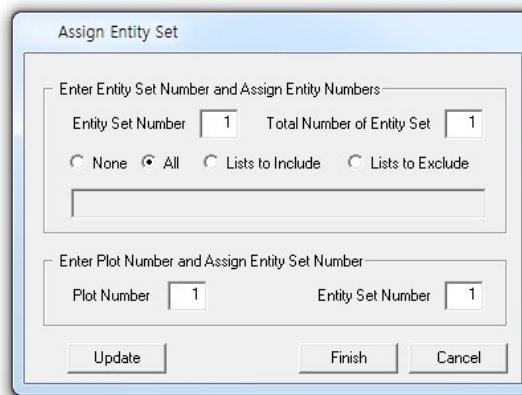
Edit Set is to assign **Entity Set** as shown in Figure 5.56.

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



The dialog box is titled "Assign Entity Set". It contains two main sections. The first section is titled "Enter Entity Set Number and Assign Entity Numbers" and includes input fields for "Entity Set Number" (value 1) and "Total Number of Entity Set" (value 1). Below these are four radio buttons: "None", "All" (which is selected), "Lists to Include", and "Lists to Exclude". There is an empty text box below the radio buttons. The second section is titled "Enter Plot Number and Assign Entity Set Number" and includes input fields for "Plot Number" (value 1) and "Entity Set Number" (value 1). At the bottom of the dialog are three buttons: "Update", "Finish", and "Cancel".

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.

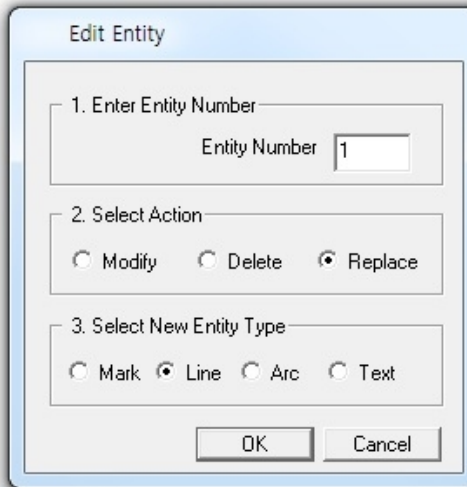
5.7.5.2 Enter Plot No & Assign Entity Set No

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

The image shows a software dialog box titled "Edit Entity". It contains three numbered sections. Section 1, "1. Enter Entity Number", has a text input field labeled "Entity Number" with the value "1". Section 2, "2. Select Action", contains three radio buttons: "Modify", "Delete", and "Replace", with "Replace" being selected. Section 3, "3. Select New Entity Type", contains four radio buttons: "Mark", "Line", "Arc", and "Text", with "Line" being selected. At the bottom of the dialog are "OK" and "Cancel" buttons.

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](#).

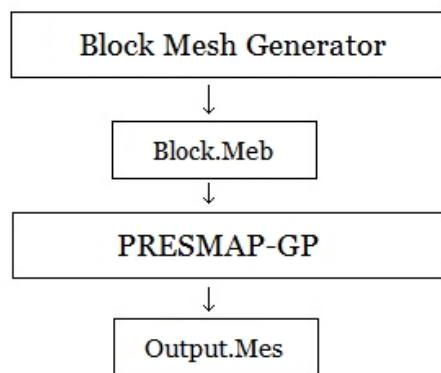


Figure 6.1 Flow diagram of block mesh generation

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.

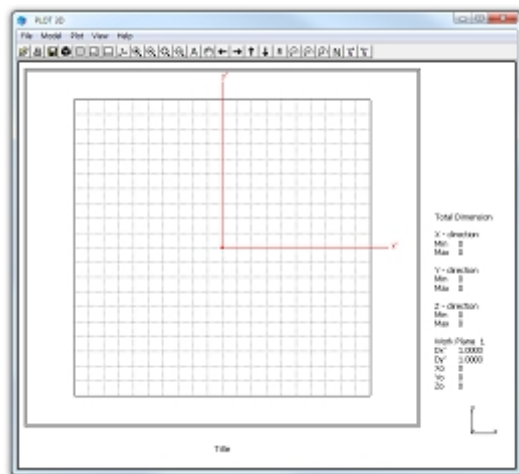


Figure 6.2 Prebuilt work plane on PLOT-3D

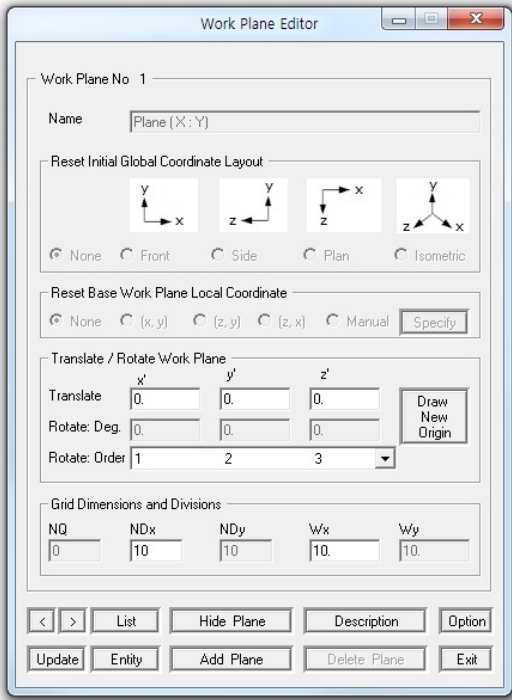


Figure 6.3 Prebuilt work plane editor

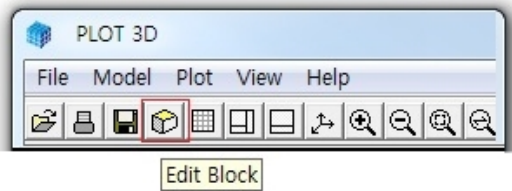


Figure 6.4 Block editor toolbar

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.

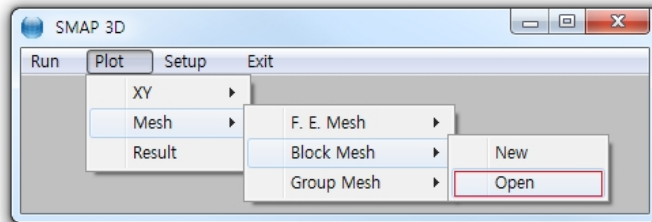


Figure 6.5 Open input file dialog

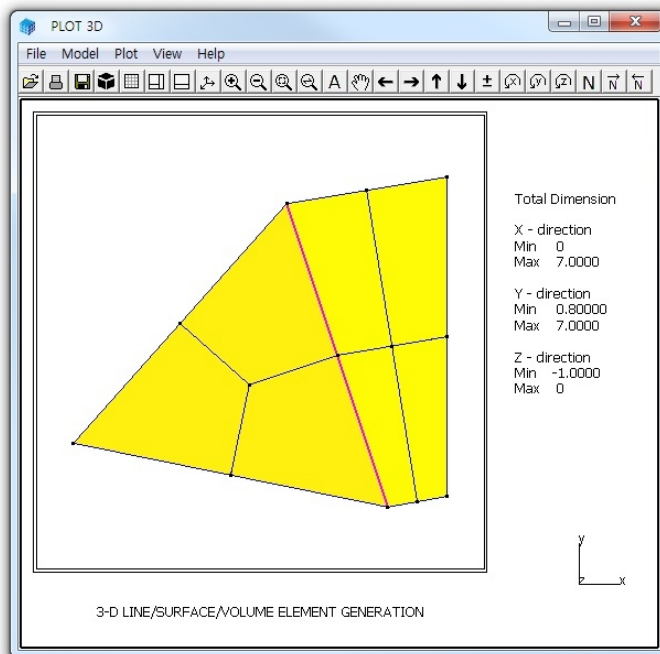


Figure 6.6 Block mesh on PLOT-3D

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.

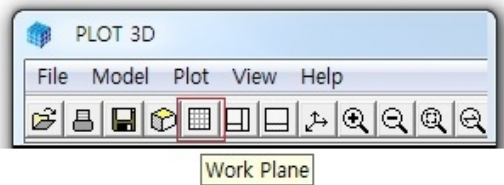


Figure 6.7 Work plane toolbar

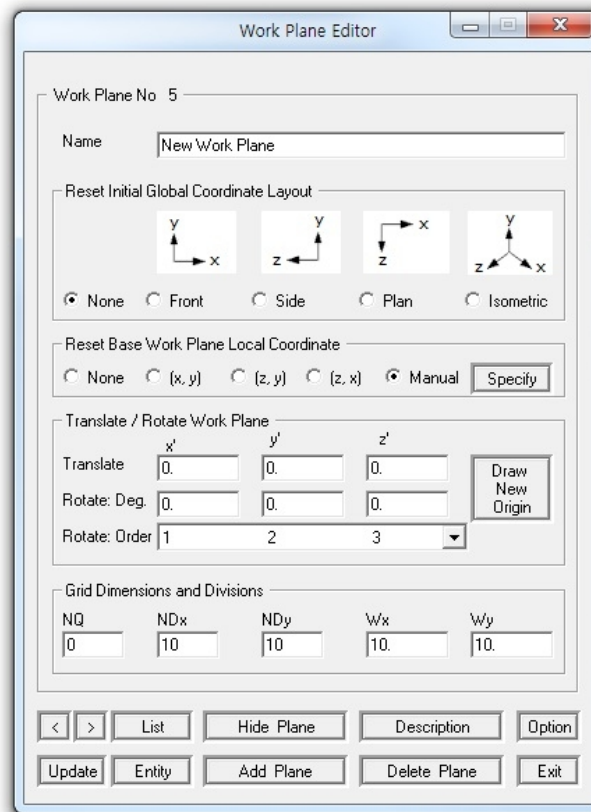
Work Plane Editor dialog in Figure 6.8 consists of following six parts:

- 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.

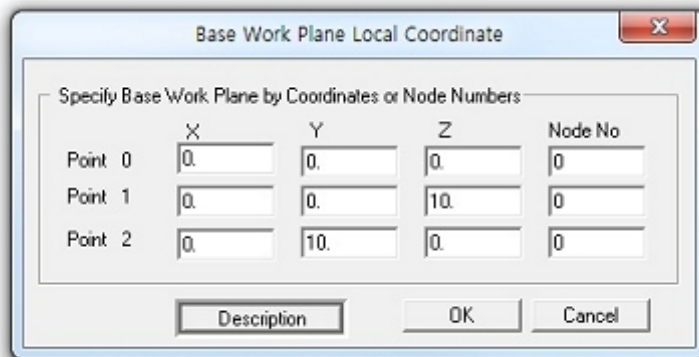
6.3.3 Reset Base Work Plane Local Coordinate

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.



The dialog box is titled "Base Work Plane Local Coordinate" and contains a section titled "Specify Base Work Plane by Coordinates or Node Numbers". This section contains a table with four columns: X, Y, Z, and Node No. There are three rows labeled Point 0, Point 1, and Point 2. Each row has input fields for X, Y, and Z, and a dropdown menu for Node No. Below the table are three buttons: Description, OK, and Cancel.

	X	Y	Z	Node No
Point 0	0.	0.	0.	0
Point 1	0.	0.	10.	0
Point 2	0.	10.	0.	0

Description OK Cancel

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.

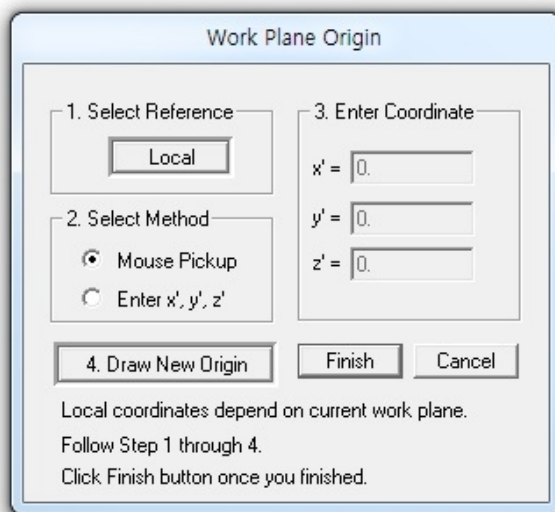


Figure 6.10 Work plane origin dialog

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.

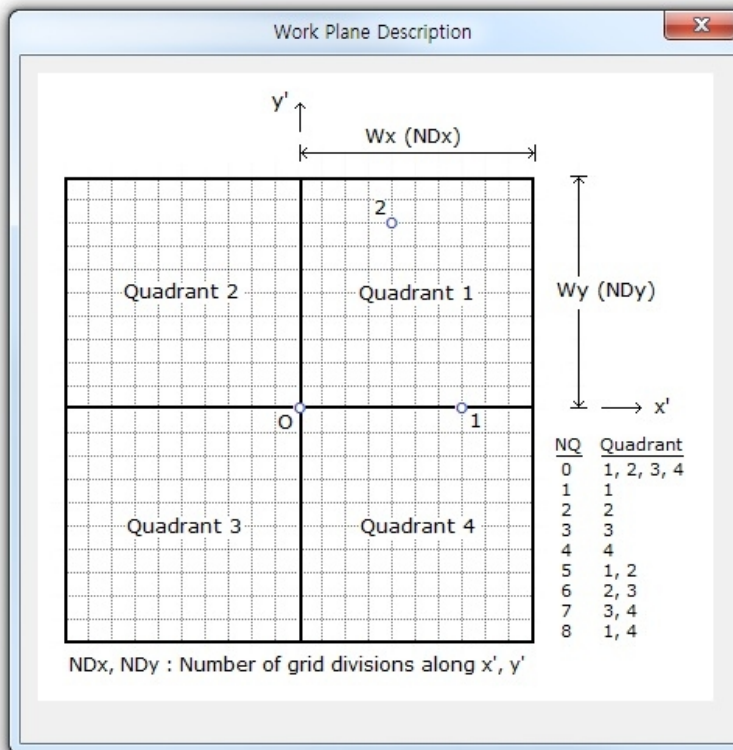


Figure 6.11 Work plane description

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.

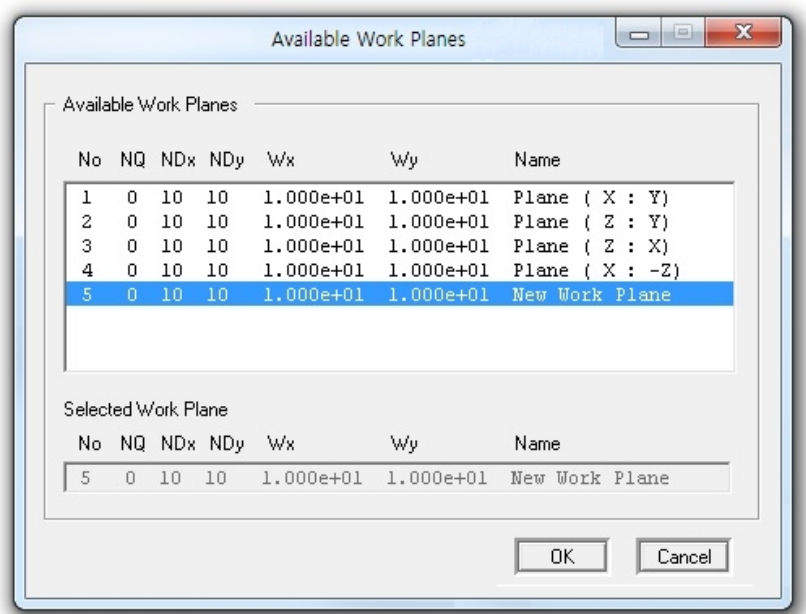


Figure 6.12 Work plane list

[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](#)

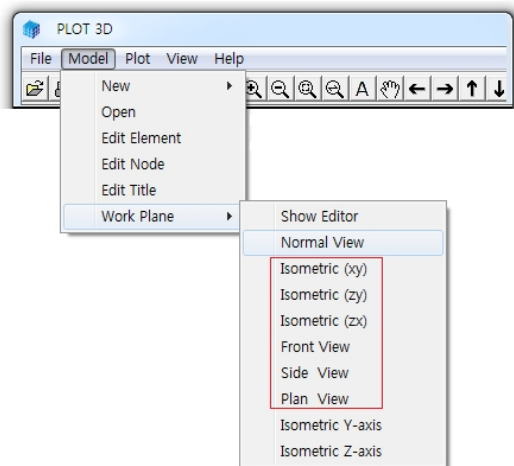


Figure 6.14 Prebuilt work plane menus

or by clicking [Axis](#) toolbar as shown in Figure 6.15.

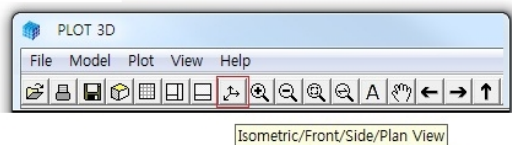


Figure 6.15 Axis toolbar

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.

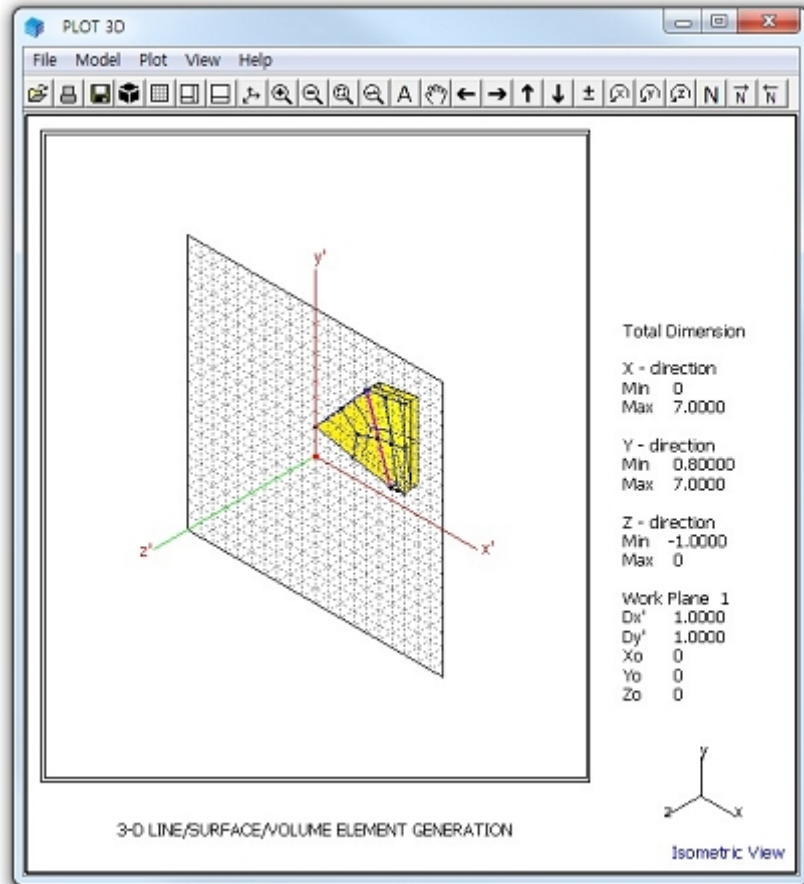


Figure 6.16 Prebuilt work plane: [Isometric \(xy\)](#) View

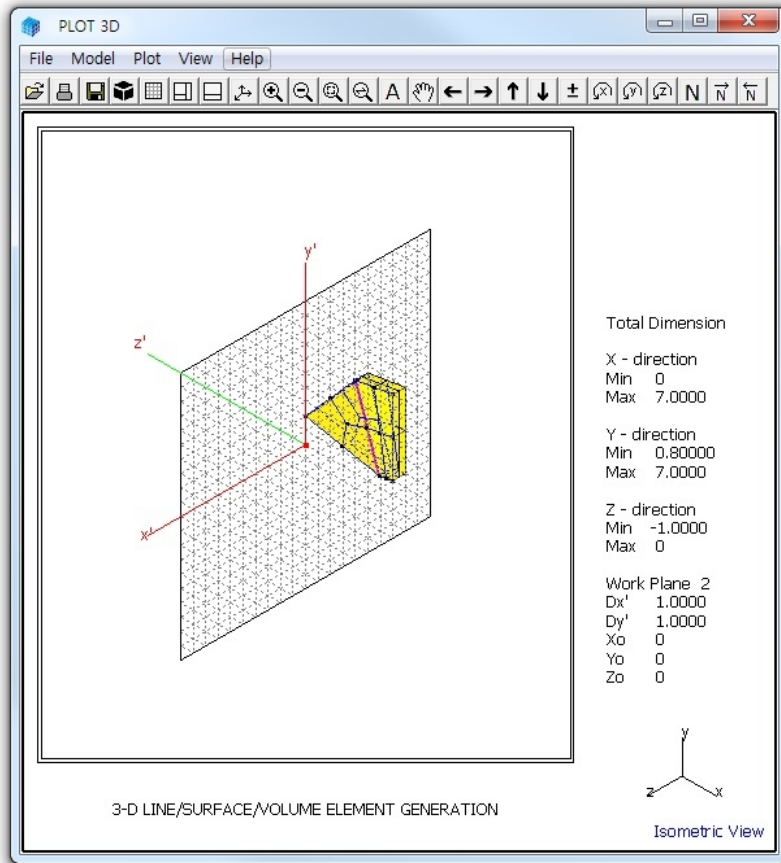


Figure 6.16 Prebuilt work plane: Isometric (zy) View

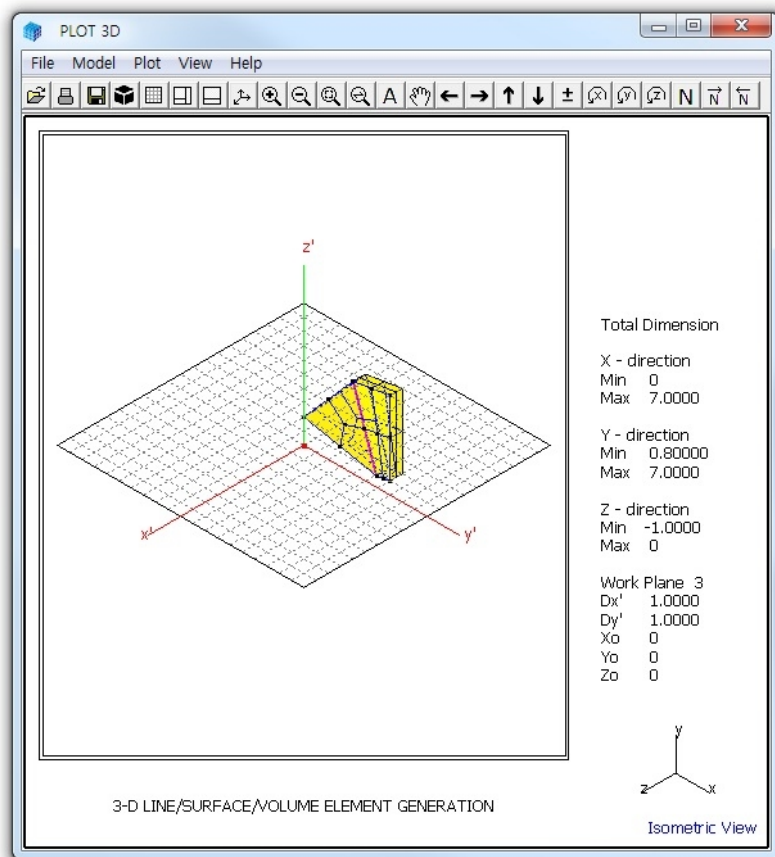
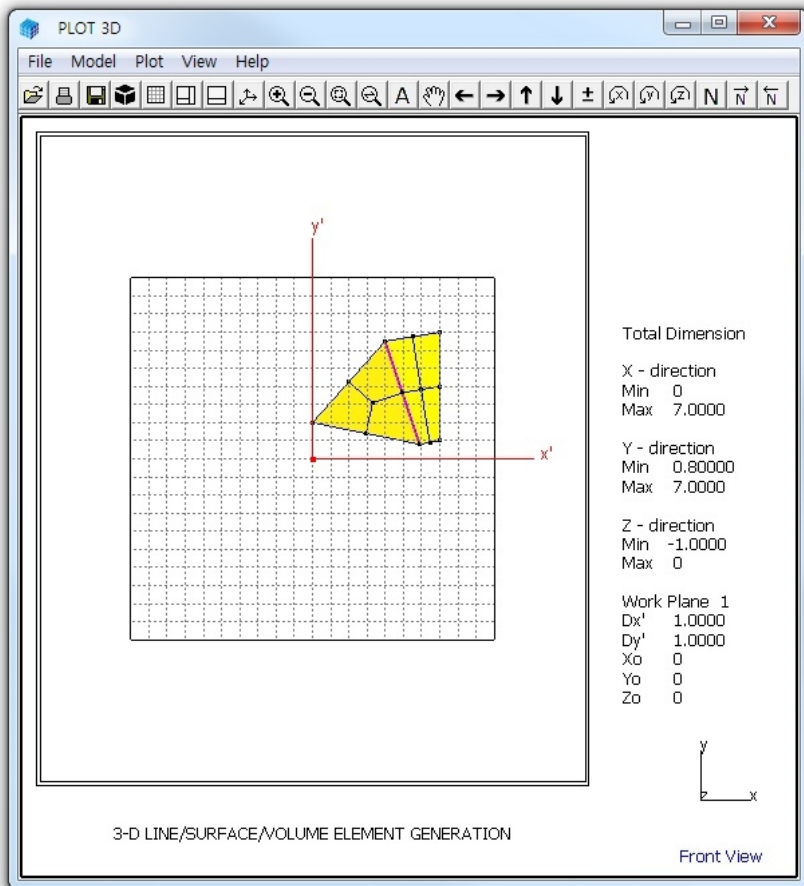


Figure 6.16 Prebuilt work plane: Isometric (zx) View

Figure 6.16 Prebuilt work plane: **Front View**

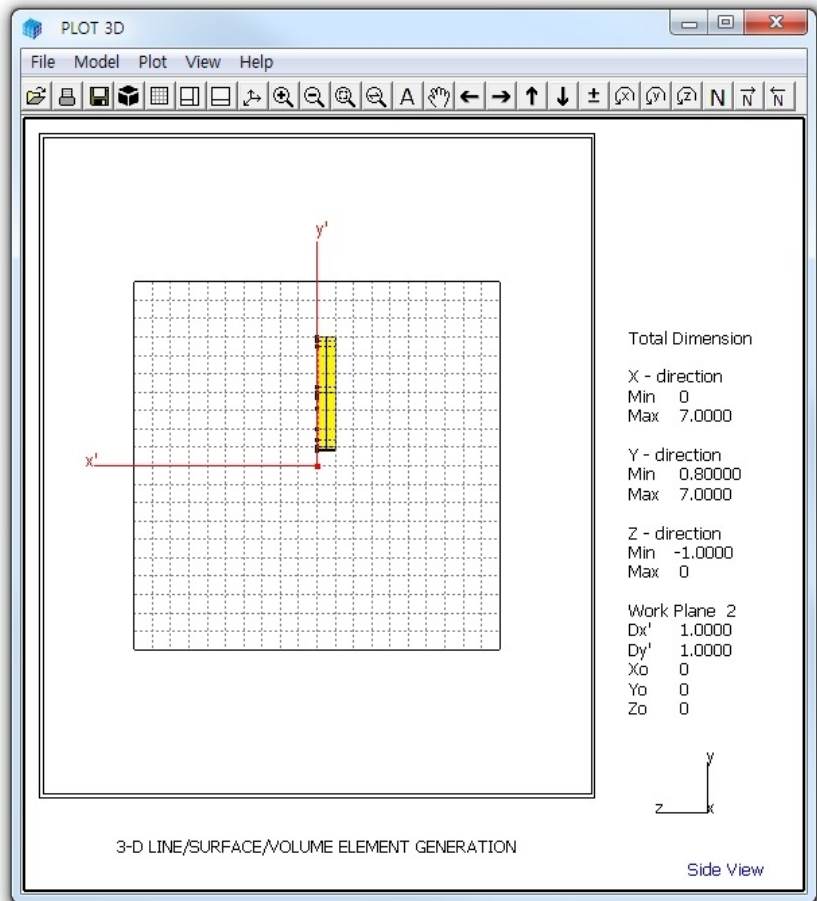
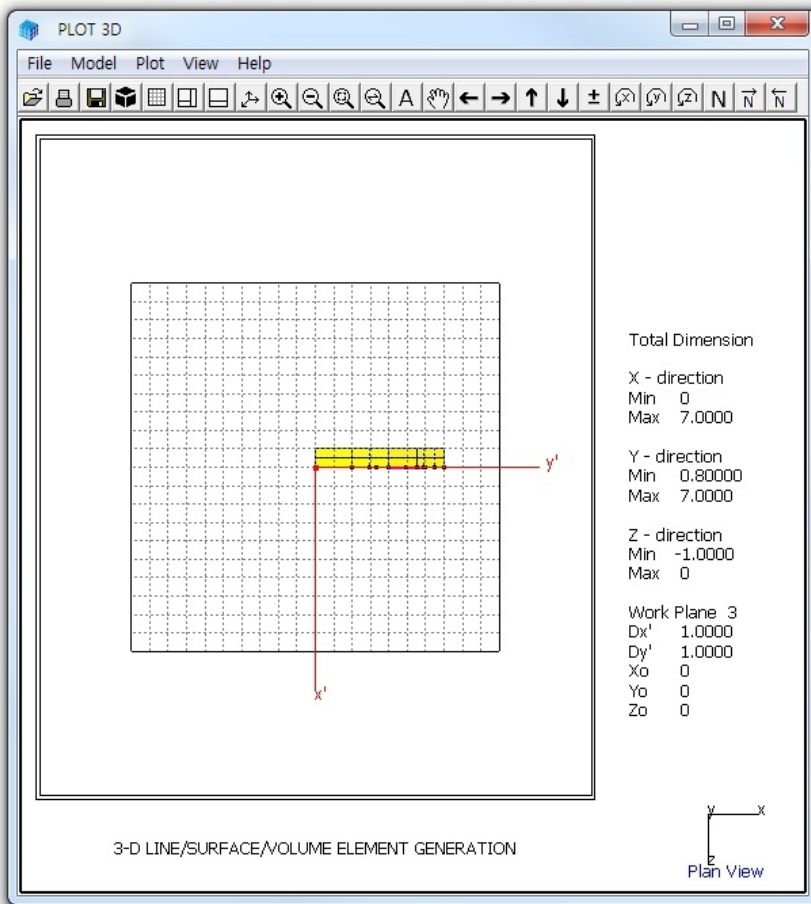


Figure 6.16 Prebuilt work plane: Side View

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

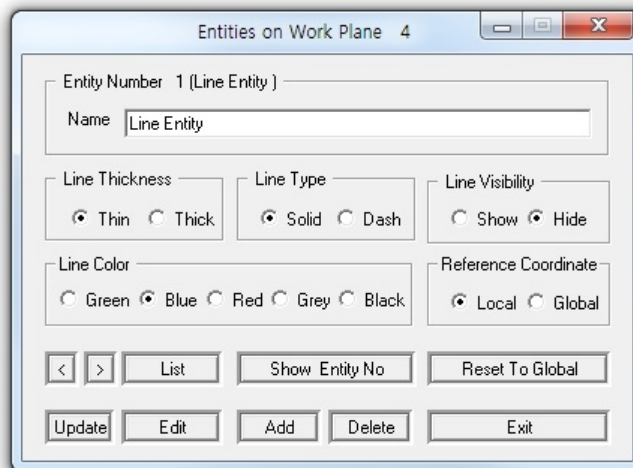


Figure 6.17 Entity editor dialog

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.

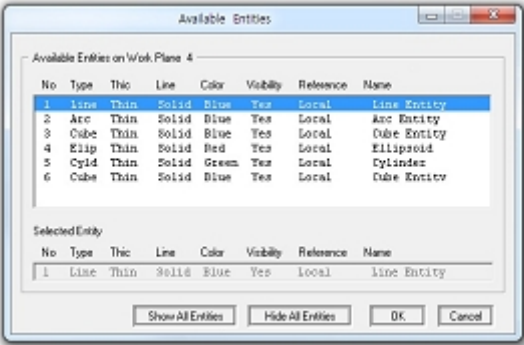


Figure 6.18
Entity list dialog

[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.

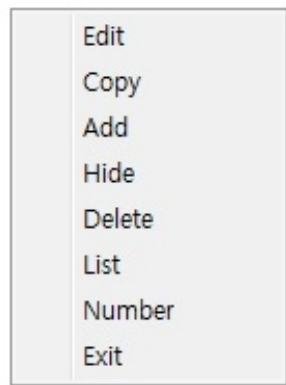


Figure 6.19 Popup menu for entity

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.](#)

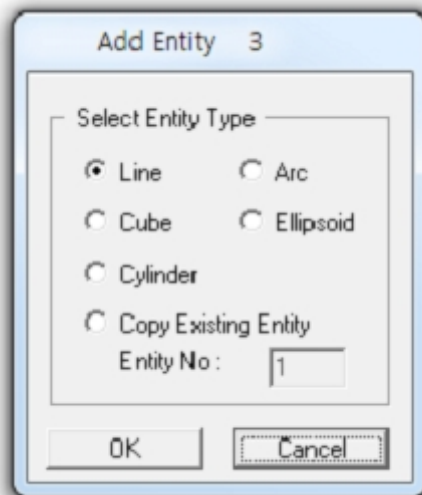


Figure 6.20 Entity type selection dialog

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

Entity 7 on Work Plane 4

1. Enter Point Number
1
For New Drawing, 0

2. Select Reference
Local

3. Select Method
☒ Mouse Pickup
☐ Enter x' , y' , z'

4. Enter Coordinate
 $x' =$ 0
 $y' =$ 0
 $z' =$ 0
☐ Shift All Points

5. Draw Point Number

Finish Cancel

Enter point number 0 to redraw entity.
Local coordinates depend on current work plane.
Repeat Step 1 through 5 for each point number.
Click Finish button once you finished all points.

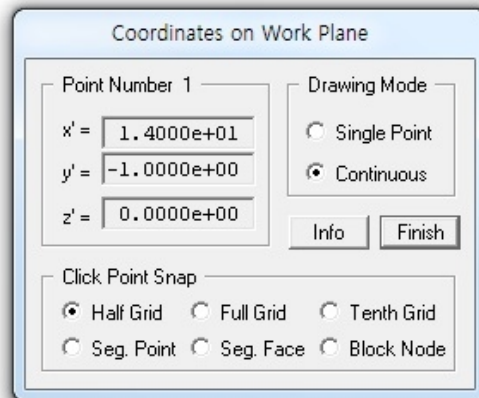


Figure 6.22 Coordinates on work plane

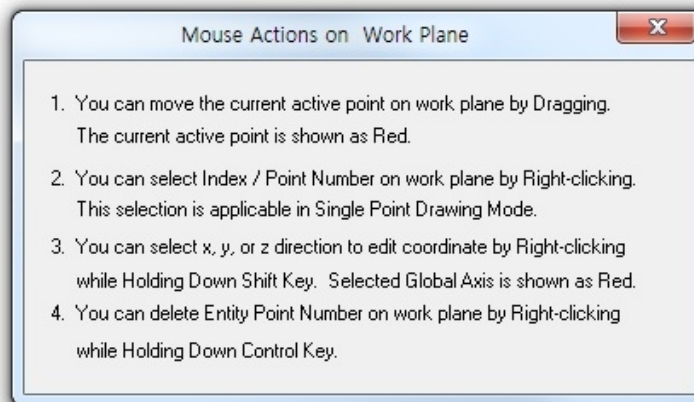


Figure 6.23 Mouse actions on work plane

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.

Entity 7 on Work Plane 4

1. Select Reference
Local

2. Select Method
☒ Mouse Pickup
☐ Enter xo', yo', zo'

3. Enter Origin
xo' = 0.
yo' = 0.
zo' = 0.
☐ New Drawing

4. Enter Dimensions

Rx = 5.
Ry = 5.
Qb = 0.
Qe = 360.

For Qb = Qe, straight line from R = Rx to R = Ry
Rx and Ry represent radial distance at Q = Qb.

5. Draw Arc Entity Finish Cancel

Local coordinates depend on current work plane.
Click Finish button once you finished arc 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:

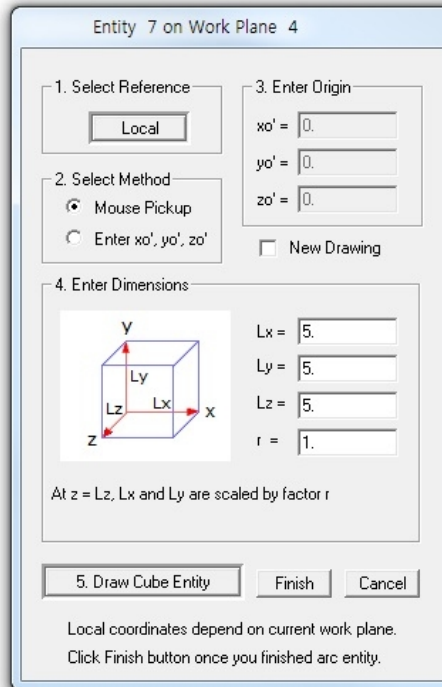
1. Select Reference
2. Select Method
3. Enter Origin
4. Enter Dimensions
5. Draw Cube Entity

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.

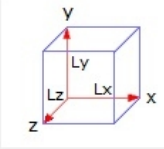


Entity 7 on Work Plane 4

1. Select Reference
Local

2. Select Method
☒ Mouse Pickup
☐ Enter xo', yo', zo'

3. Enter Origin
xo' = 0.
yo' = 0.
zo' = 0.
☐ New Drawing

4. Enter Dimensions

Lx = 5.
Ly = 5.
Lz = 5.
r = 1.
At z = Lz, Lx and Ly are scaled by factor r

5. Draw Cube Entity Finish Cancel

Local coordinates depend on current work plane.
Click Finish button once you finished arc 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.

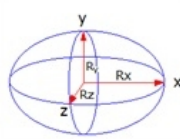
Figure 6.26
Ellipsoid entity dialog

Entity 7 on Work Plane 4

1. Select Reference
Local

2. Select Method
☒ Mouse Pickup
☐ Enter xo', yo', zo'

3. Enter Origin
xo' = 0.
yo' = 0.
zo' = 0.
☐ New Drawing

4. Enter Dimensions

Rx = 5.
Ry = 5.
Rz = 5.
Ns = 0.
Ns = 0: All 1: 1st Octant 23: 2nd 3rd Octants
91: Front 92: Back 93: Left 94: Right 95: Top 96: Bottom

5. Draw Ellipsoid Entity Finish Cancel

Local coordinates depend on current work plane.
Click Finish button once you finished arc entity.

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.

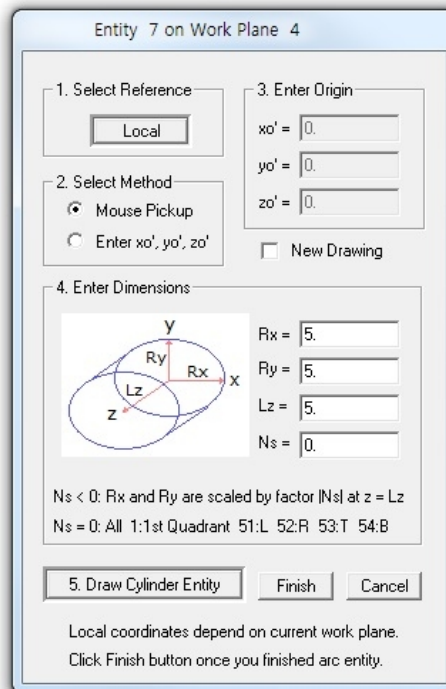


Figure 6.27
Cylinder entity dialog

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.

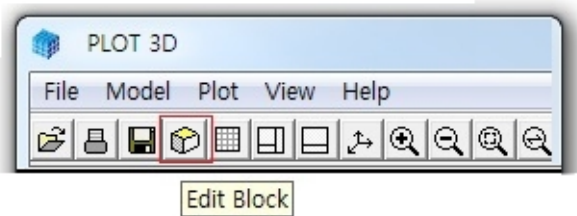


Figure 6.28 Block editor toolbar

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

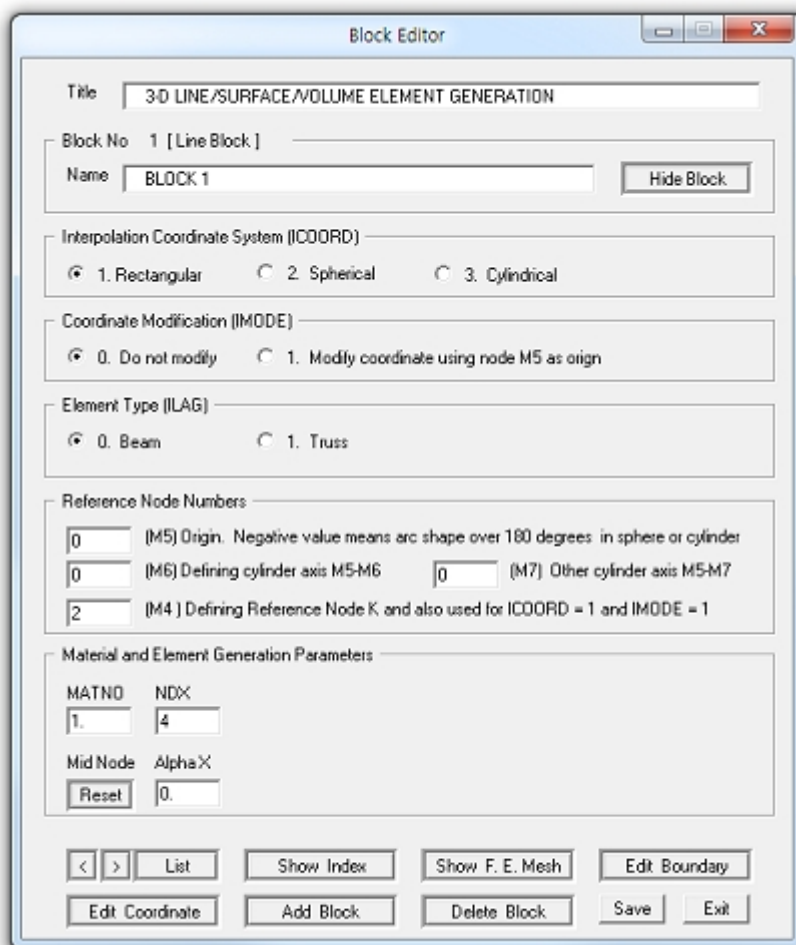


Figure 6.29 Block editor (Line Block)

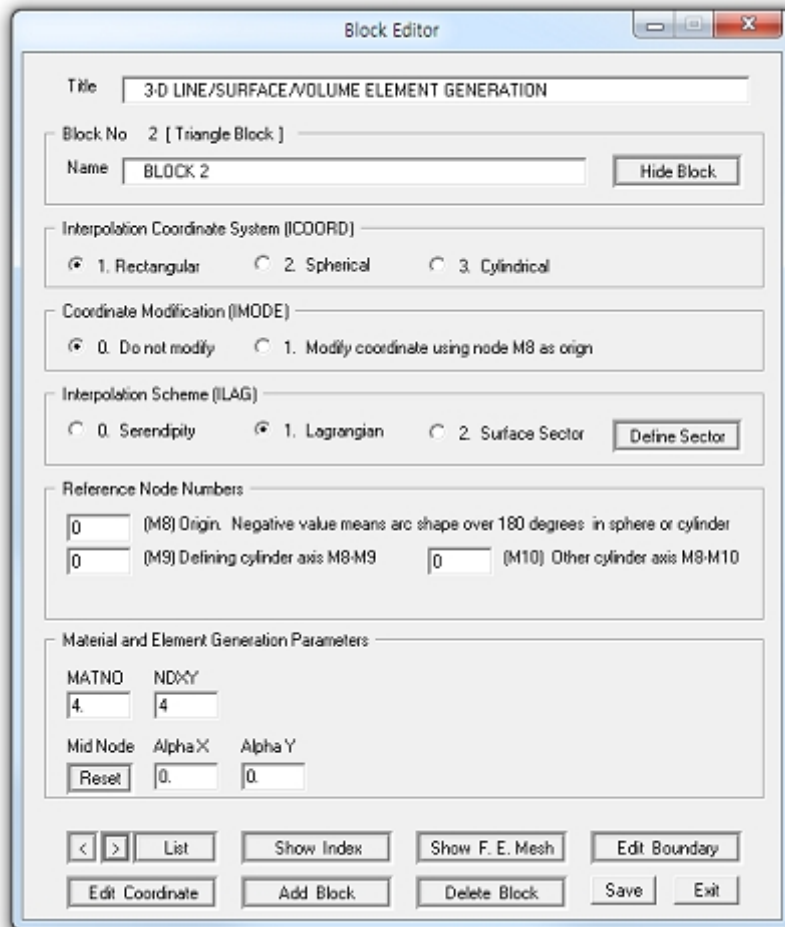


Figure 6.29 Block editor (Triangle Block)

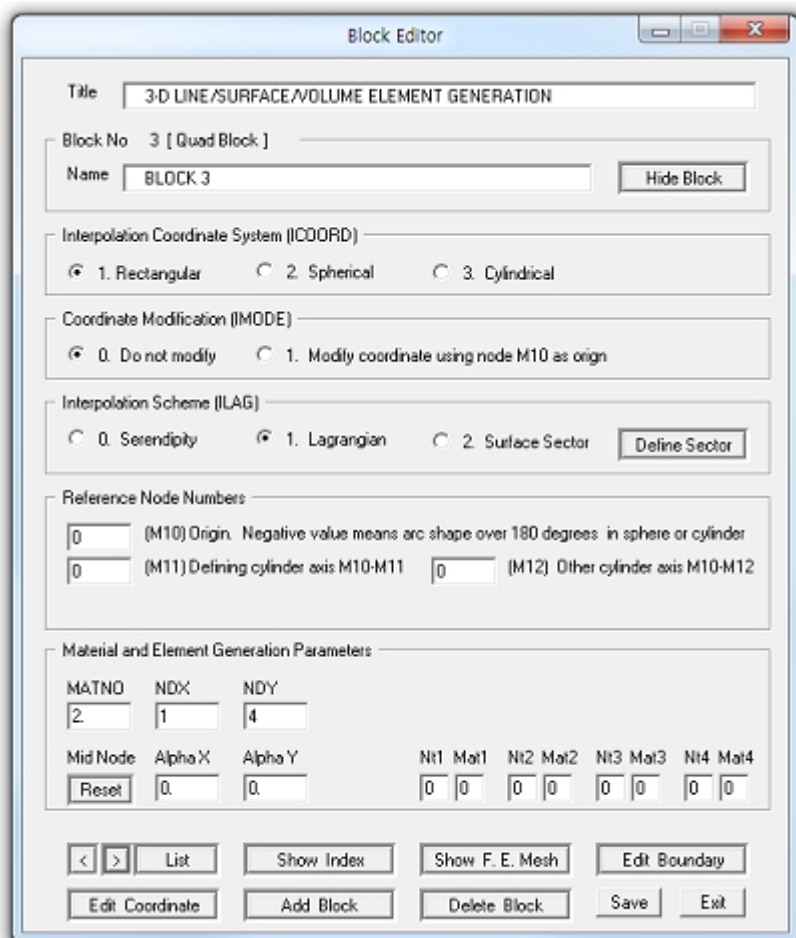


Figure 6.29 Block editor (Quad Block)

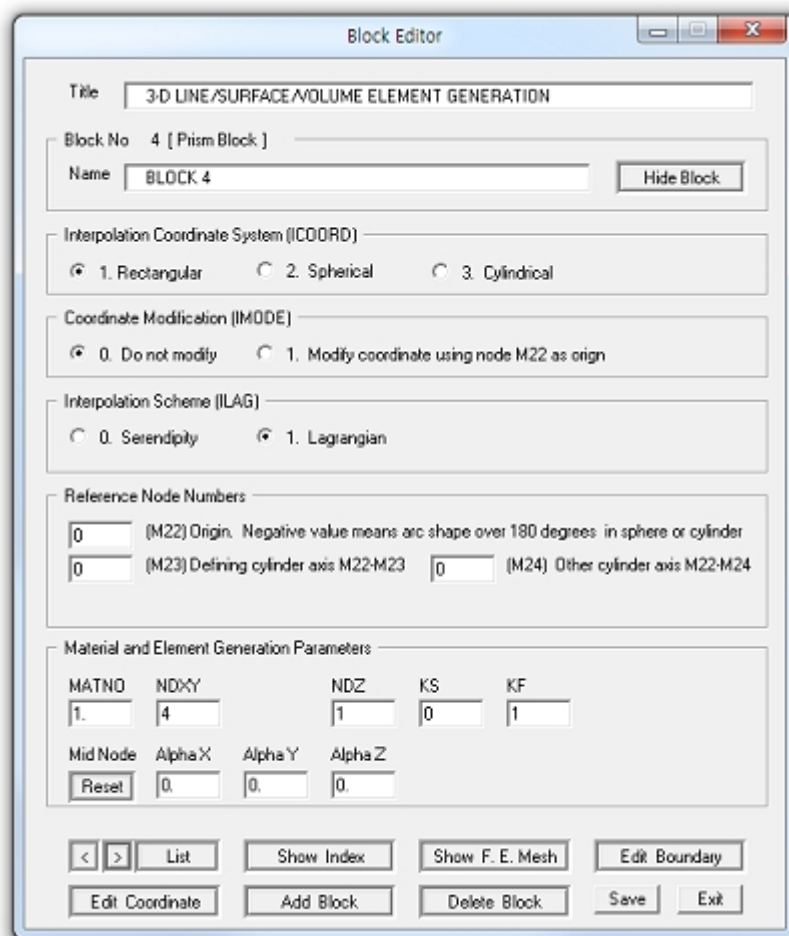


Figure 6.29 Block editor (Prism Block)

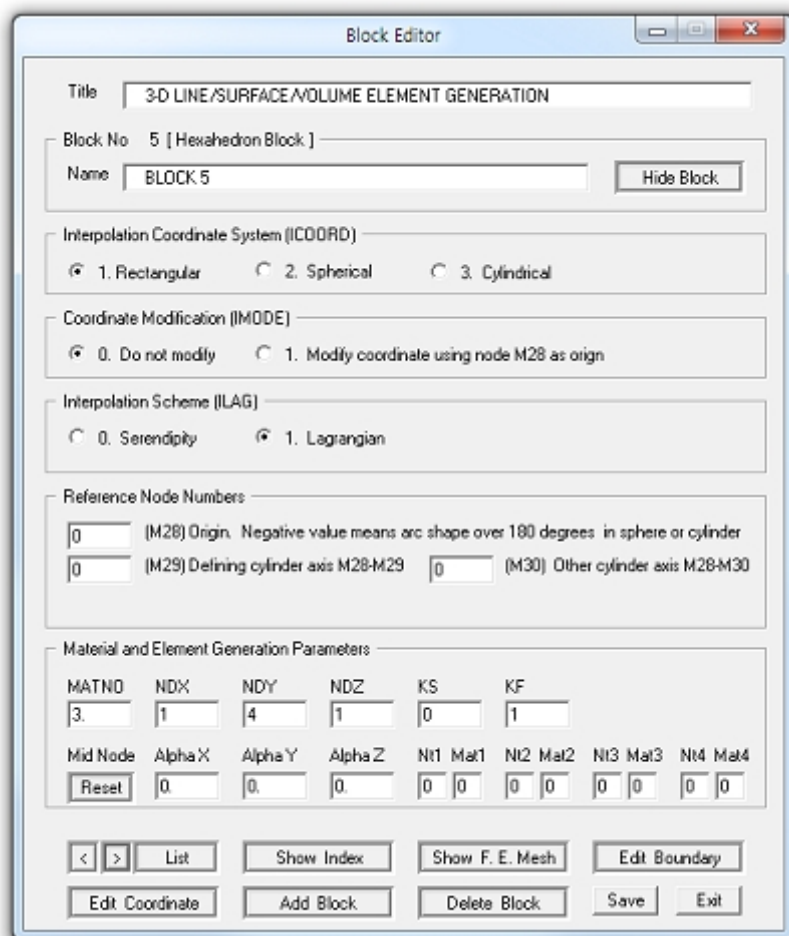


Figure 6.29 Block editor (Hexahedron Block)

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.

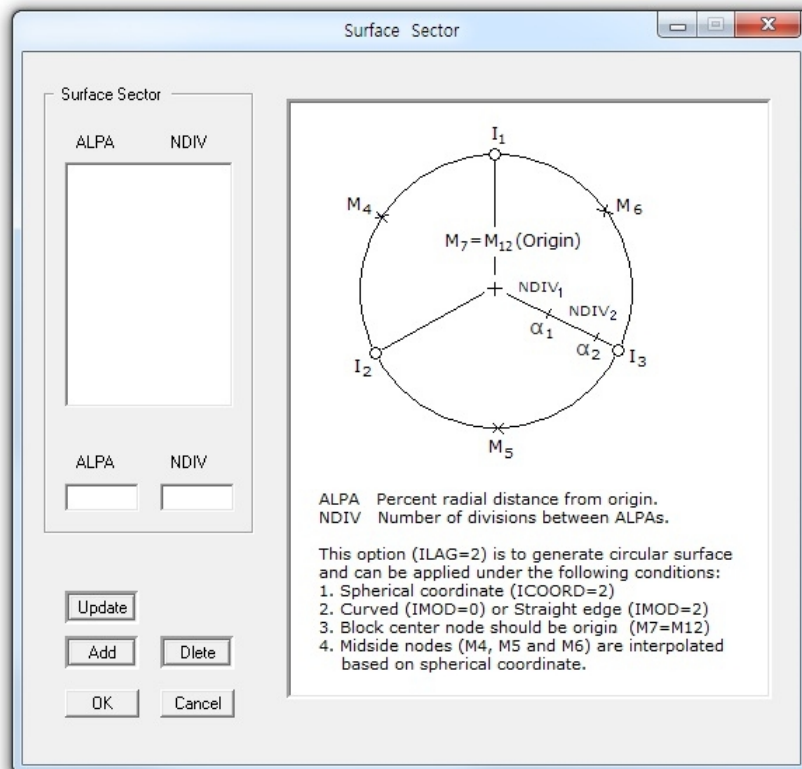


Figure 6.30 Surface sector (Triangle Block)

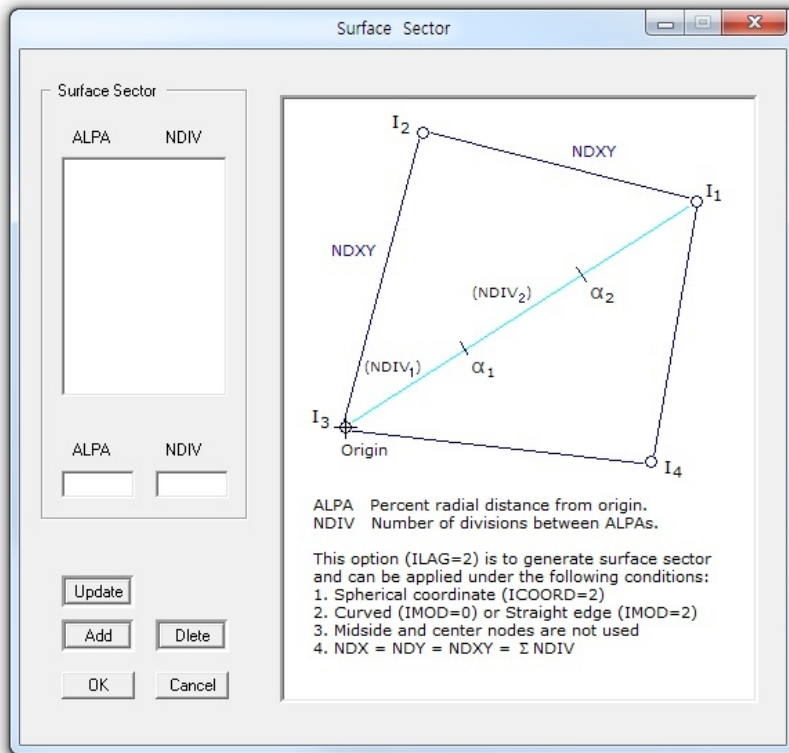


Figure 6.30 Surface sector (Quad Block)

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.

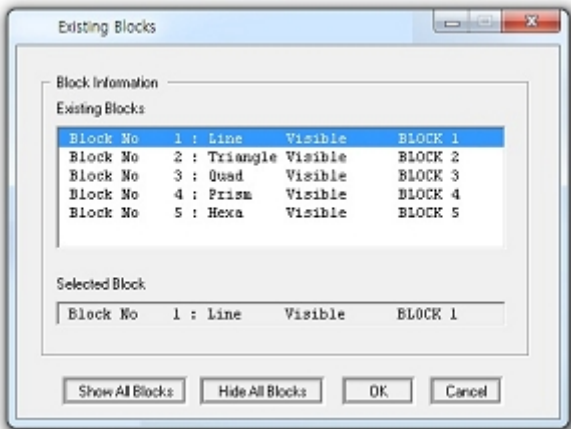


Figure 6.31 Block list

[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.

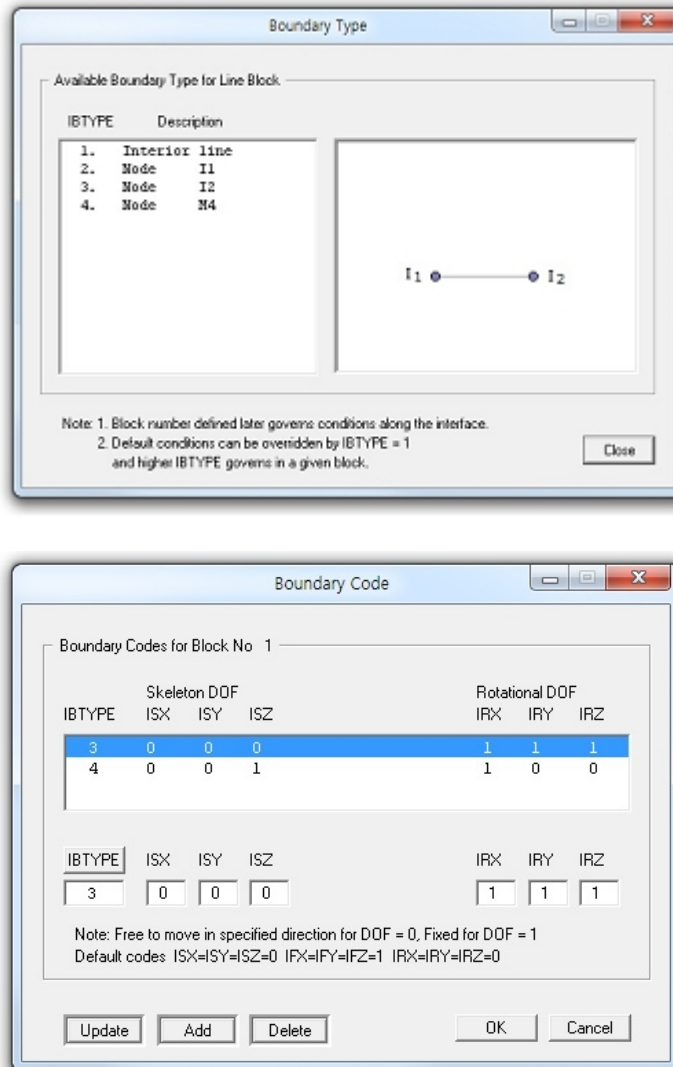


Figure 6.32 Boundary code (Line Block)

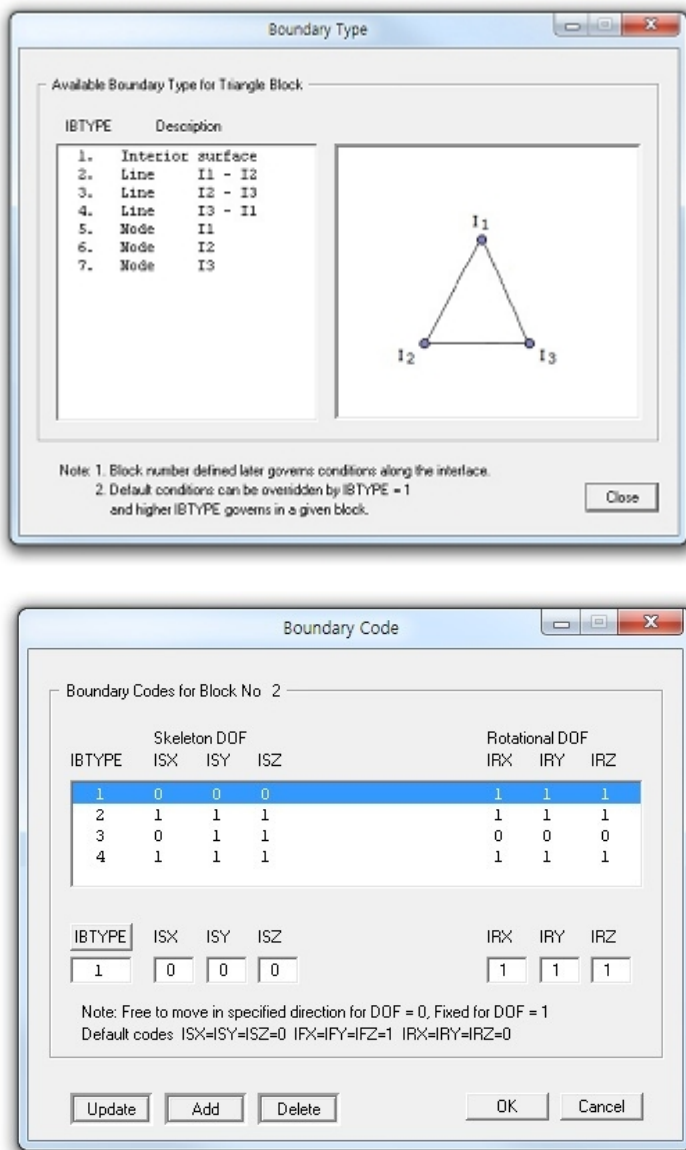


Figure 6.32 Boundary code (Triangle Block)

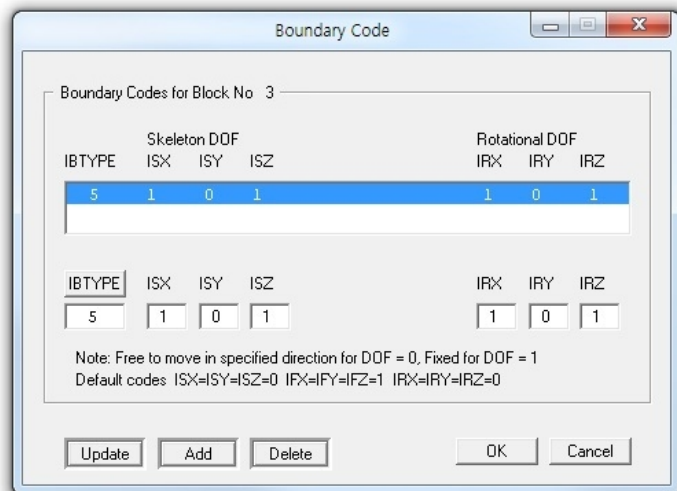
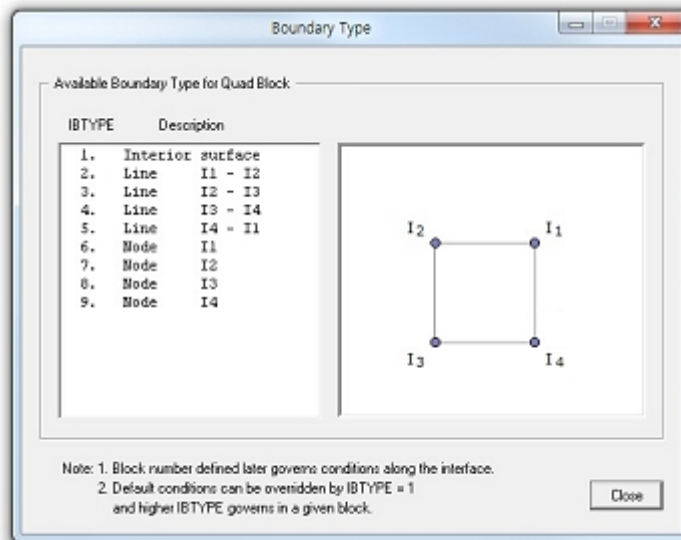


Figure 6.32 Boundary code (Quad Block)

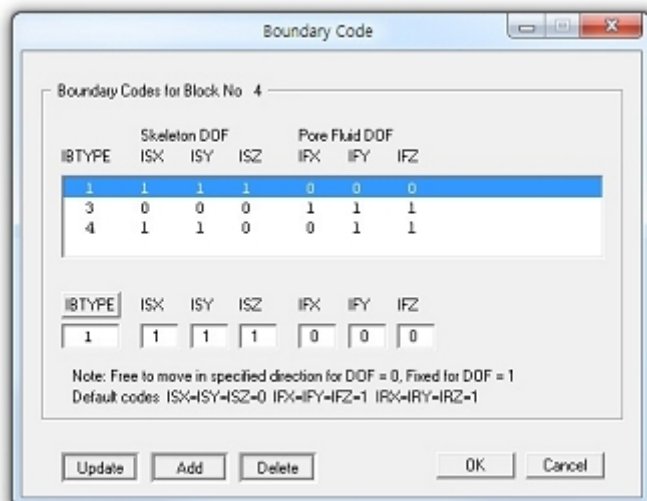
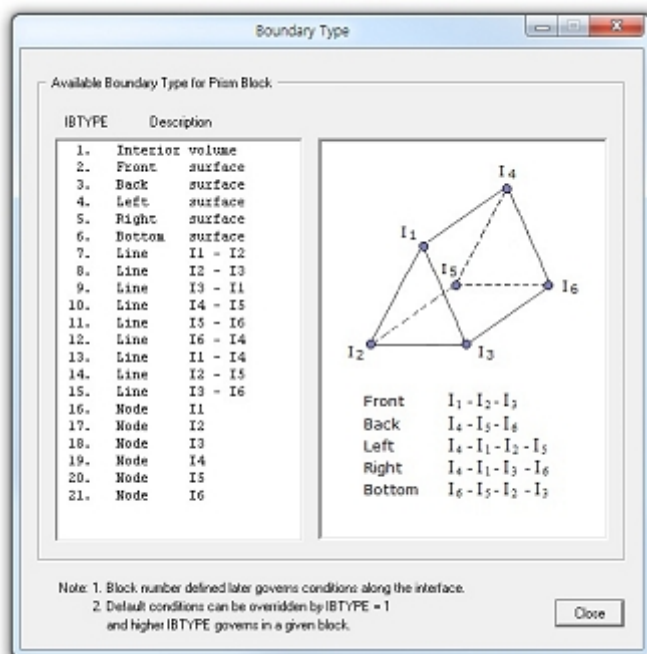


Figure 6.32 Boundary code (Prism Block)

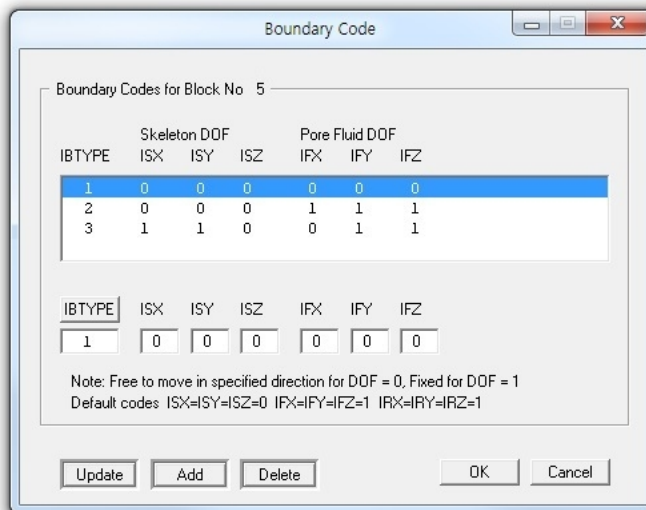
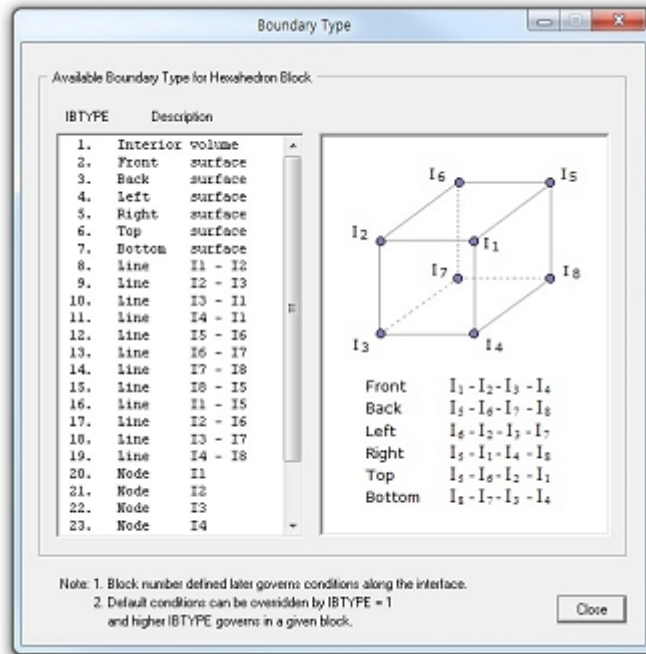


Figure 6.32 Boundary code (Hexahedron Block)

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.

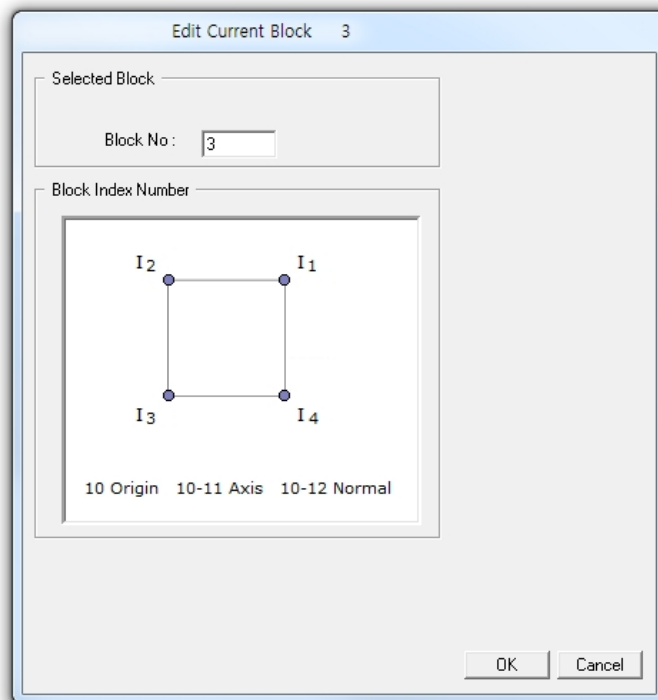


Figure 6.33 Edit current block (**Selection Mode**)

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

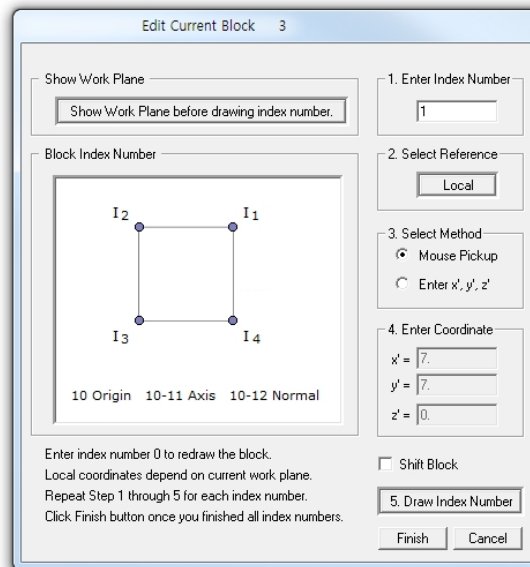
To edit block, follow five steps:

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)



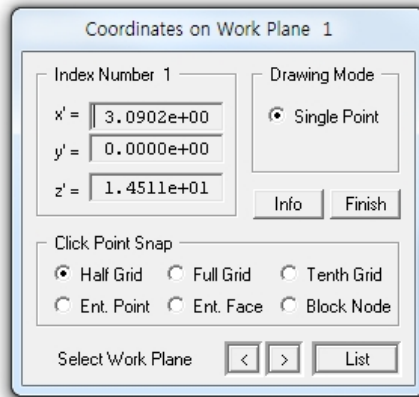


Figure 6.35 Coordinates on work plane

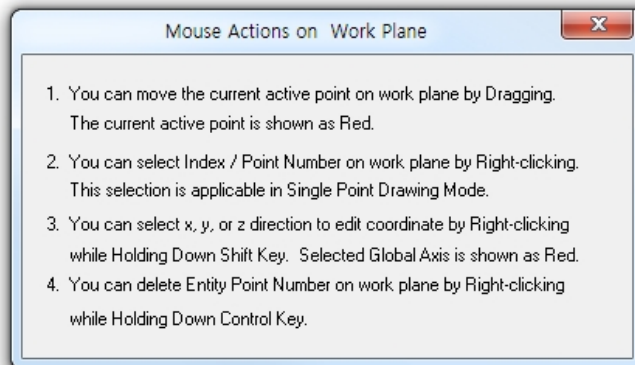


Figure 6.36 Mouse actions on work plane

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.

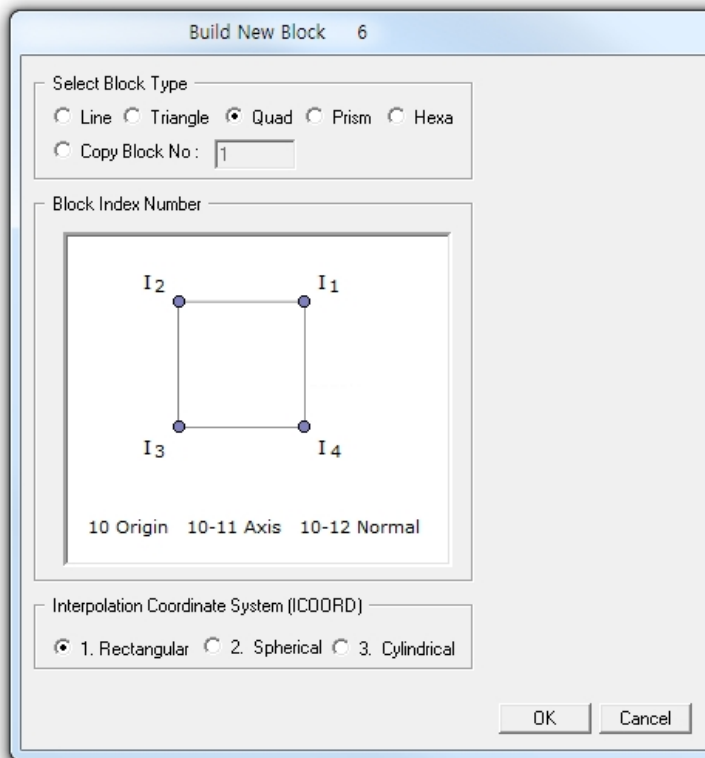


Figure 6.37 Build new block (Selection Mode)

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

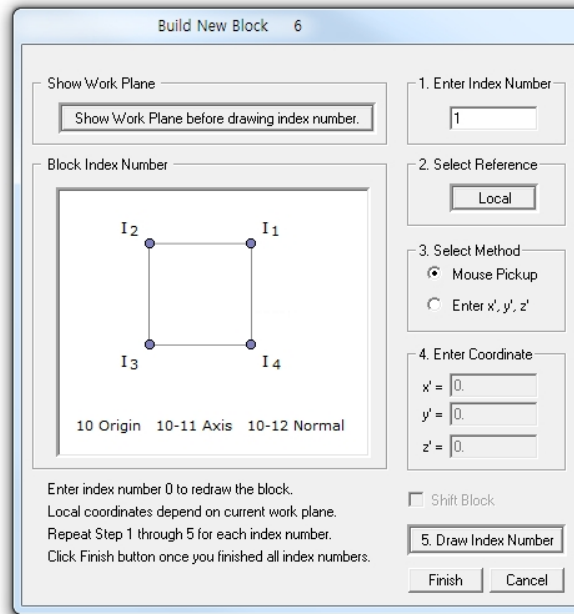
To build new block, follow five steps:

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)



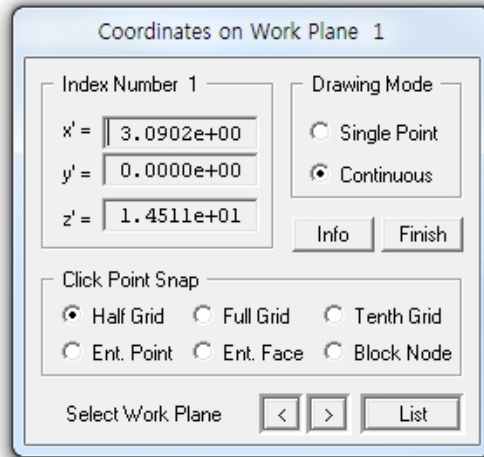


Figure 6.39 Coordinates on work plane

[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.

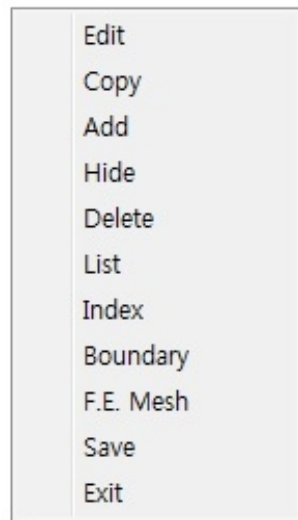


Figure 6.40 Popup menu for block

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.

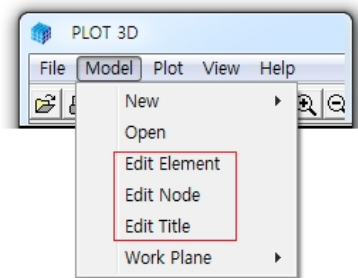


Figure 6.41 Menu for editing finite element mesh

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

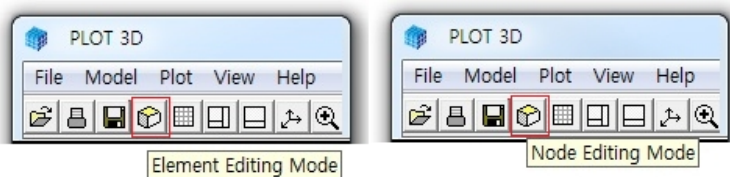
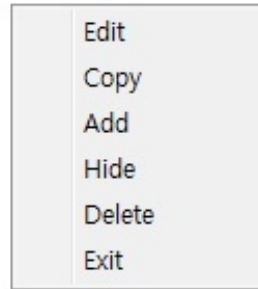


Figure 6.42 Toolbar for editing finite element mesh

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



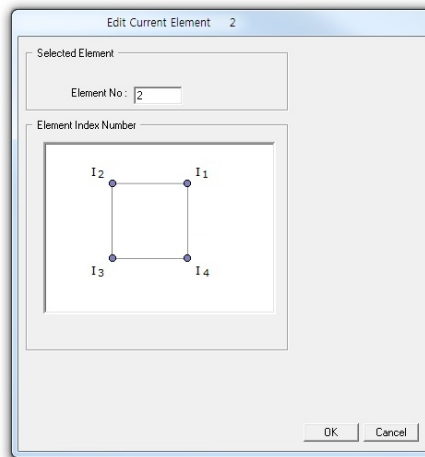
Element popup menu consists of six submenus:
[Edit](#), [Copy](#), [Add](#), [Hide](#), [Delete](#) and [Exit](#).

[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)

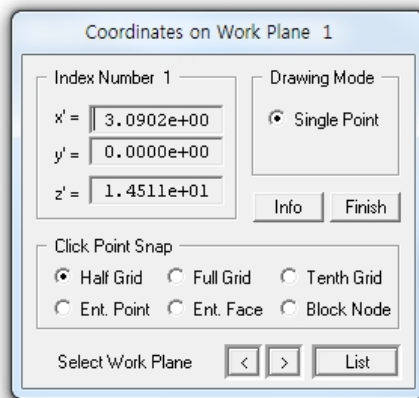


Figure 6.46 Coordinates on work plane ([Edit Mode](#))

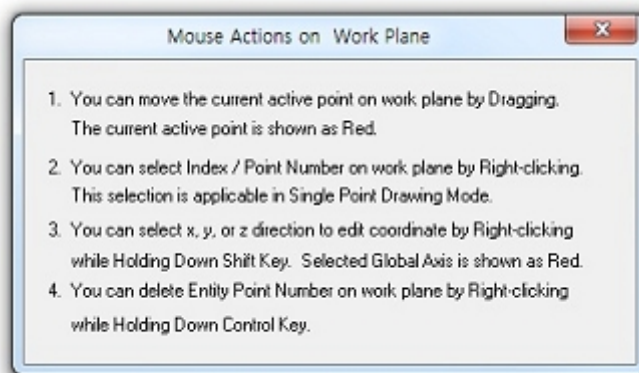


Figure 6.47 Mouse actions on work plane

[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](#).

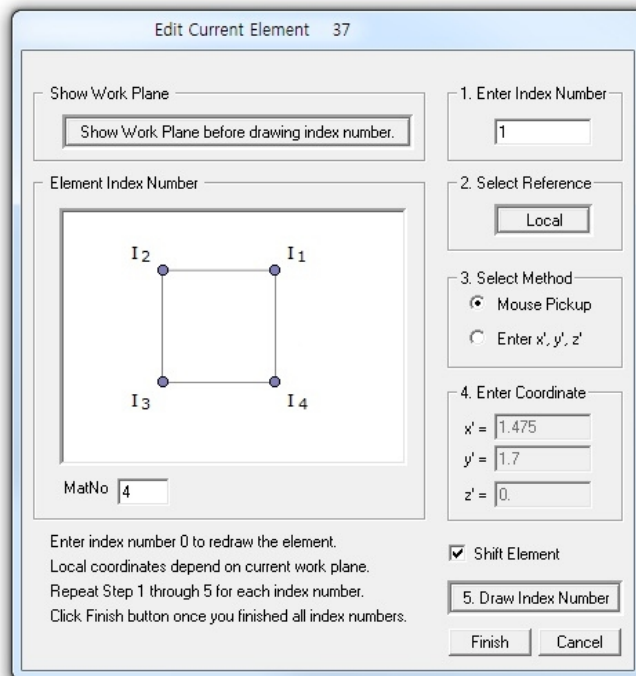


Figure 6.48 Edit current element ([Copy Mode](#))

[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.

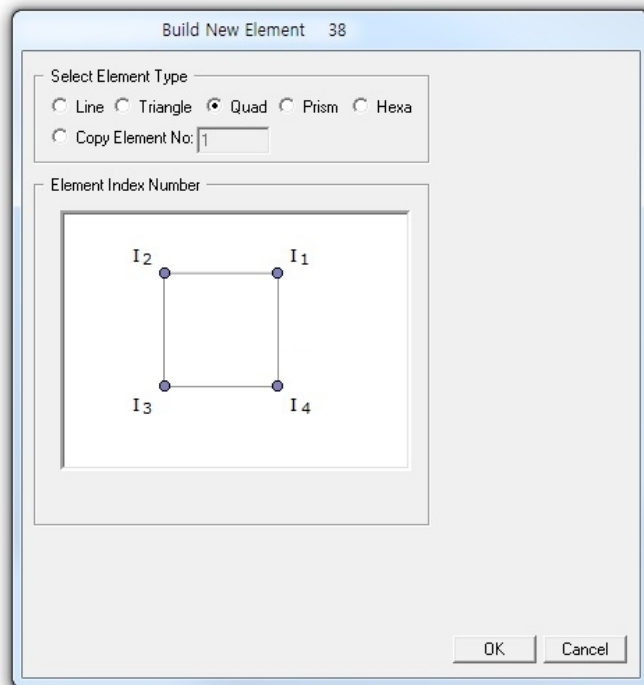


Figure 6.49 Build new element ([Selection Mode](#))

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

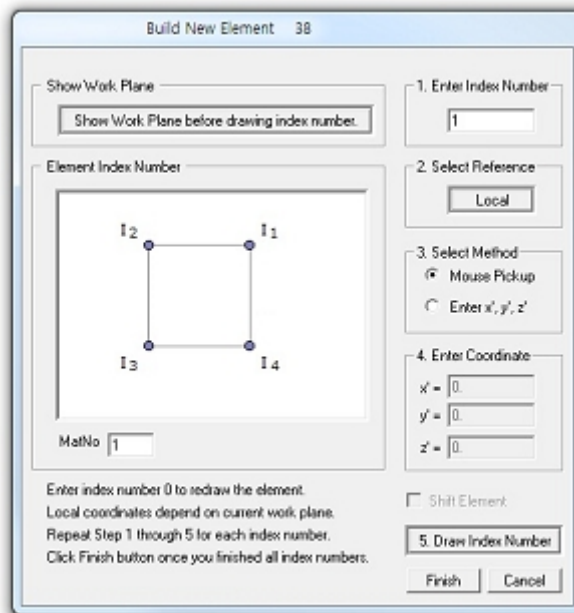
To build new element, follow five steps:

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)



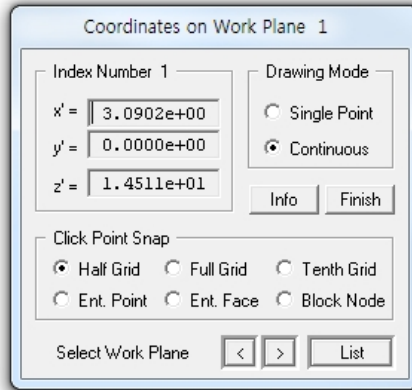


Figure 6.51 Coordinates on work plane ([Add Mode](#))

[Hide](#)

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

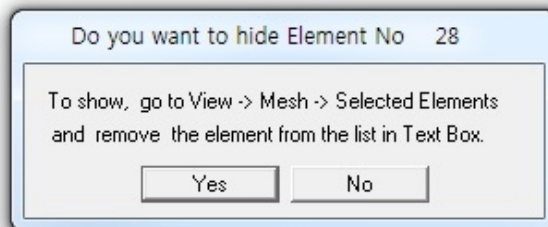


Figure 6.52 Instructions to show the hidden element

[Delete](#)

This is used to delete the selected element.

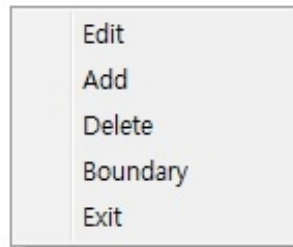
[Exit](#)

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.

Node No 49

1. Enter Node Number
49

2. Select Reference
Local

3. Select Method
☒ Mouse Pickup
☐ Enter x', y', z'

4. Enter Coordinate
x' = 0.475
y' = -2.3
z' = 0.

5. Draw Node Number

Finish Cancel

Local coordinates depend on current work plane.
Repeat Step 1 through 5 for each node number.
Click Finish button once you finished all points.

Figure 6.54 Edit current node dialog

Coordinates on Work Plane 1

Node Number 49

Drawing Mode
☒ Single Point

x' = 2.0000e+00
y' = 2.0000e+00
z' = 5.0000e+00

Info Finish

Click Point Snap
☒ Half Grid ☐ Full Grid ☐ Tenth Grid
☐ Ent. Point ☐ Ent. Face ☐ Node

Select Work Plane < > List

Figure 6.55 Coordinates on work plane ([Edit Node](#))

[Add](#)

This is used to add new node.

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

[Build New Node](#) dialog similar to Figure 6.54 is displayed.

Follow the same procedure as in [Edit](#).

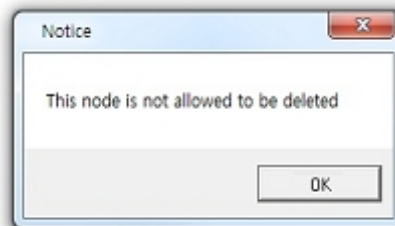
[Delete](#)

This is used to delete the selected node.

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

Figure 6.56

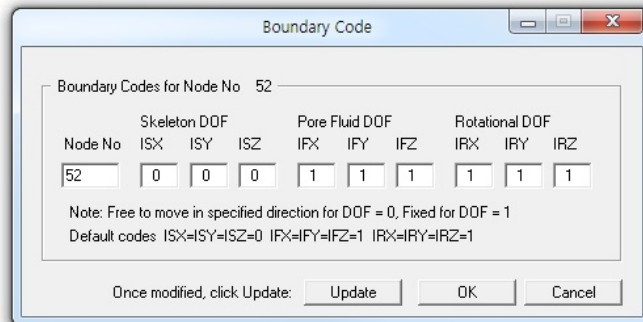
Notice on deleting connected nodes

[Boundary](#)

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

Figure 6.57

Boundary editor dialog

[Exit](#)

This is used to exit from the node editing mode.

6.6.3 Edit Title

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

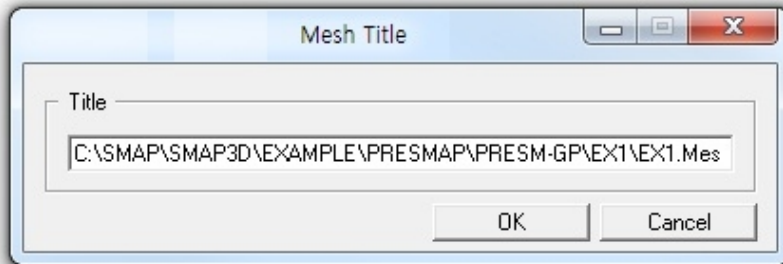


Figure 6.58 Mesh title editor dialog

PRESMAP

User's Manual

7.1 Introduction

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

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

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

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

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

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

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

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

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

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

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

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

PRESMAP-2D
Model 1
User's Manual

Card Group	Input Data and Definitions (Model 1)	
1	1.1	TITLE TITLE Any title of (Max = 60 characters)
	1.2	IP IP = 0 Plane strain or plane stress = 1 Axisymmetry
	1.3	NBLOCK, NBNODE, NSNEL, CMFAC (SMAP-S2/2D) NBLOCK, NBNODE, NSNEL, CMFAC, THI (SMAP-T2/W2) See Figure 7.1 NBLOCK Number of blocks NBNODE Number of block nodes NSNEL Starting element number CMFAC Coordinate magnification factor THI Initial temperature / total head

Card Group	Input Data and Definitions (Model 1)
1	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	2.1		
		┌	NODE ₁ , X ₁ , Y ₁
	NBNODE		NODE ₂ , X ₂ , Y ₂
	Cards		- - -
		└	- - -
	NODE		Node number
	X		X-coordinate
	Y		Y-coordinate

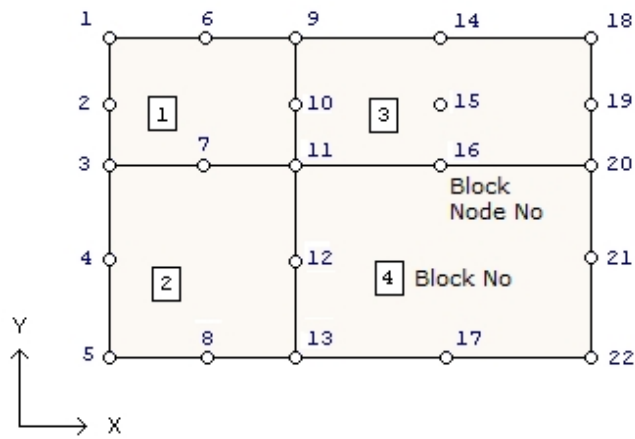
Card Group	Input Data and Definitions (Model 1)
3	3.1 BLNAME BLNAME Block name (up to 60 characters)
	3.2 IBLNO IBLNO Block number
	3.3 $I_1, I_2, I_3, I_4, M_5, M_6, M_7, M_8, M_9$ See Figure 7.1 I_1, I_2, I_3, I_4 Corner node number M_5, M_6, M_7, M_8 Side node number M_9 Center node number
	3.4 IBASE, $IB_1, IB_2, IB_3, IB_4, IB_5, IB_6, IB_7, IB_8$ (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	3.5 MATNO, NDX, NDY, KS, KF (SMAP-2D) MATNO, NDX, NDY, THICK, DENSITY (SMAP-S2) MATNO, NDX, NDY, IDH (SMAP-T2/W2) 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 / water 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
	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

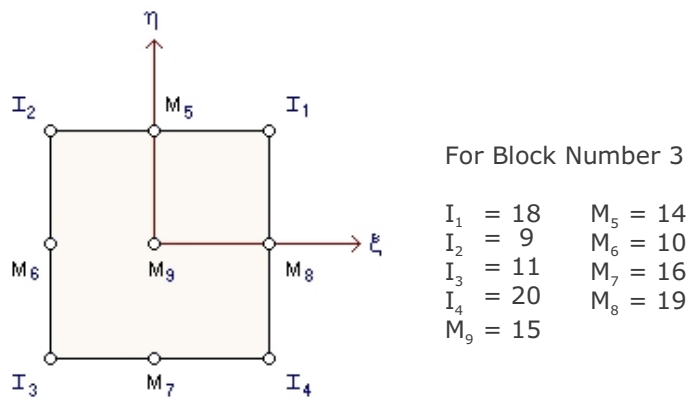
Data for Each Block

Force Data for Each Specified Side (see Figure 7.4)



NBLOCK = 4, NBNODE = 22

Block number should be in order from top to bottom and left to right



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

Figure 7.1 Block Specification and Block Index

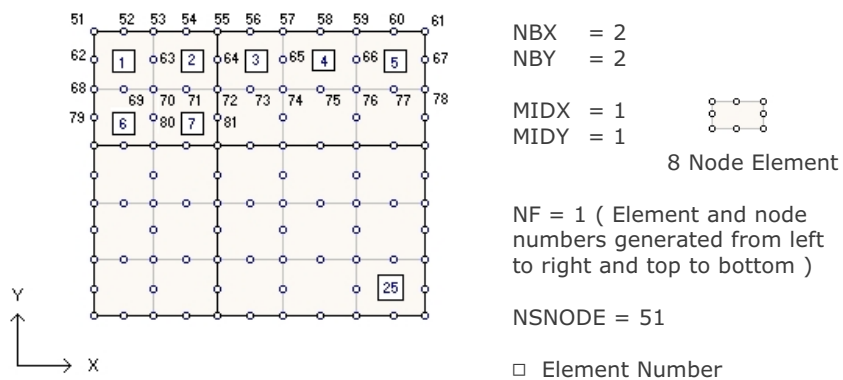
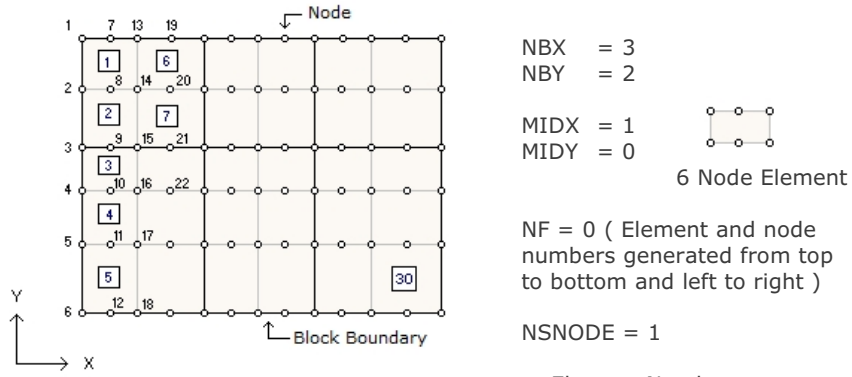


Figure 7.2 Element and Node Numbering Sequence for Model 1 of PRESMAP-2D

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

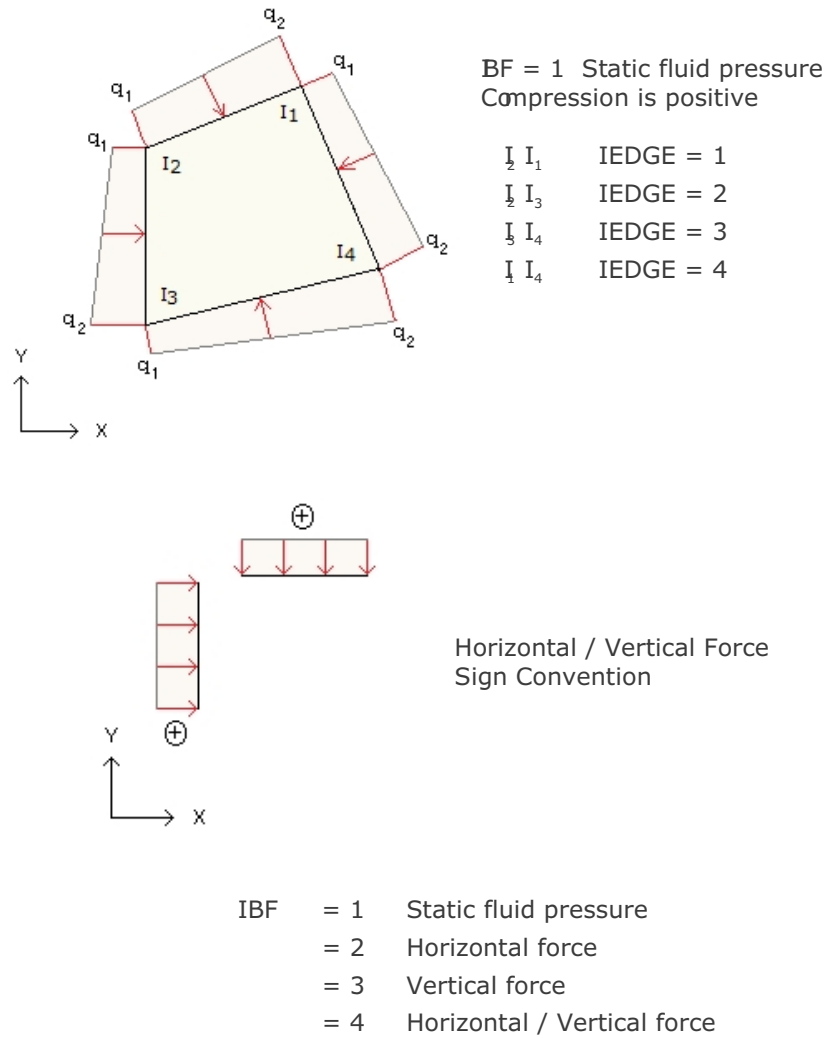


Figure 7.4 Boundary Force/Pressure Sign Conventions

PRESMAP-2D
Model 2
User's Manual

Card Group	Input Data and Definitions (Model 2)	
1	1.1	TITLE TITLE Any title (Max = 60 characters)
	1.2	IP IP = 0 Plane strain or plane stress = 1 Axisymmetry
	1.3	NSNEL, NSNODE, NF, CMFAC (SMAP-S2/2D) NSNEL, NSNODE, NF, CMFAC, THI (SMAP-T2/W2) 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 THI Initial temperature / total head
	1.4	NSUBR, NDRF, NDRS, NDRT, DRF, DRS See Figure 7.5 NSUBR Number of subregions NDRF Number of divisions in the first row block NDRS Number of divisions in the second row block NDRT Number of divisions in the third row block DRF Length of the first row block DRS Length of the second row block

Card Group	Input Data and Definitions (Model 2)
2	2.1 SUBNAME SUBNAME Subregion name (up to 60 characters)
	2.2 ISUBNO ISUBNO Subregion number
	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	2.4	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 $R, X_O, Y_O, \theta_A, \theta_B$ 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
	Subregion Surface (Figure 7.6 and 7.7)		

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

Card Group	Input Data and Definitions (Model 2)		
2	2.5	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
Data for Each Subregion	Subregion Outer Edge	For ISBTYP = 1	

Card Group	Input Data and Definitions (Model 2)
2	2.6 IBASE₁, IBASE₂, IBASE₃ (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 IBASE₁, IBASE₂, IBASE₃ 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: MATNO₁, KS₁, KF₁ (SMAP-2D) MATNO₁, DENSITY₁ (SMAP-S2) MATNO₁, IDH₁ (SMAP-T2/W2) 2nd Block: - - 3rd Block: - - MATNO₁ Material property number of first block KS₁, KF₁ Solid and fluid phase flag of first block DENSITY₁ Unit weight of first block IDH₁ Heat / water 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	2.8	NFSIDE NFSIDE Number of edge where boundary forces are specified
	2.9	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

Data for Each Subregion

Force Data for Each Specified Edge (see Figure 7.8)

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

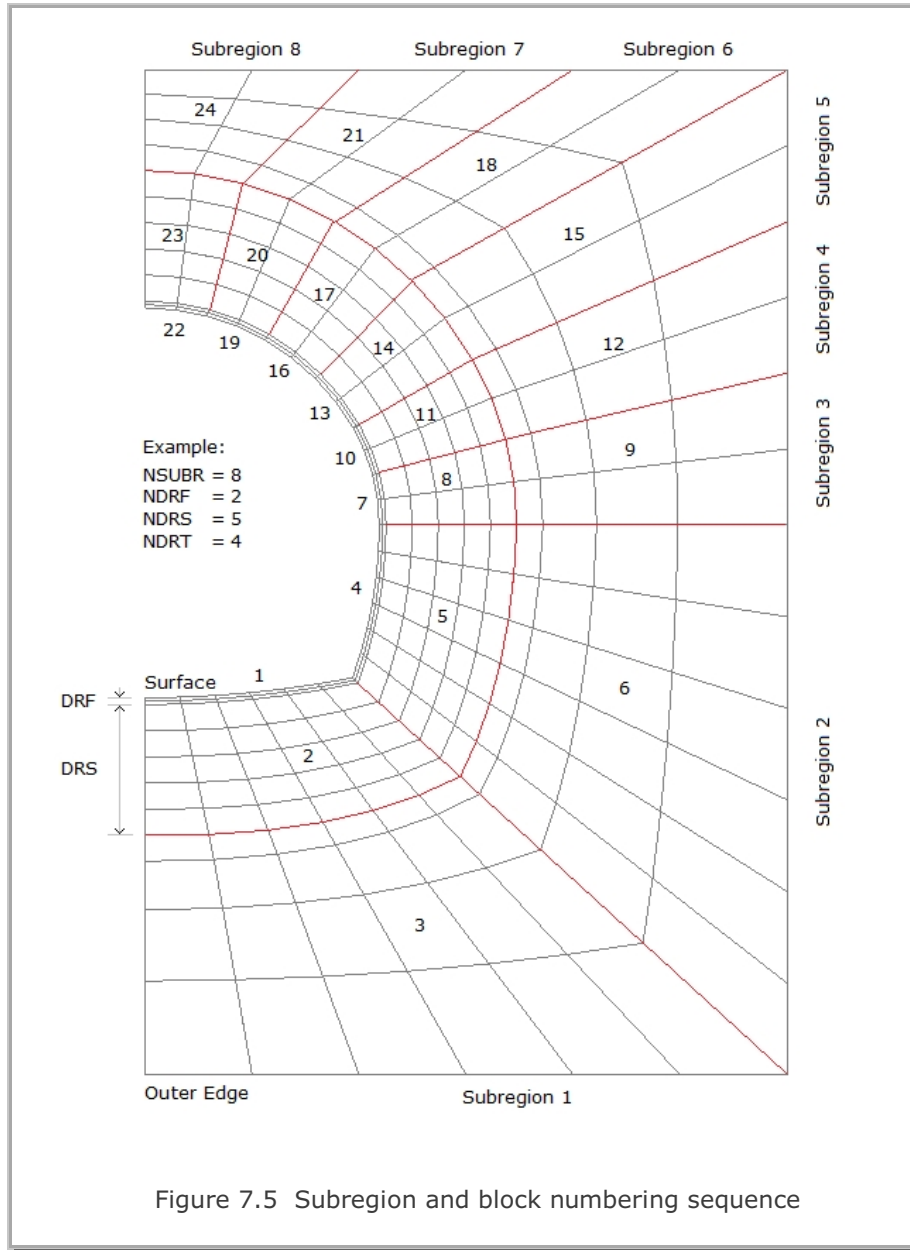


Table 7.1 Subregion parameters in Example Figure 7.5

Subregion	ISBTYP	LSFTYP	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

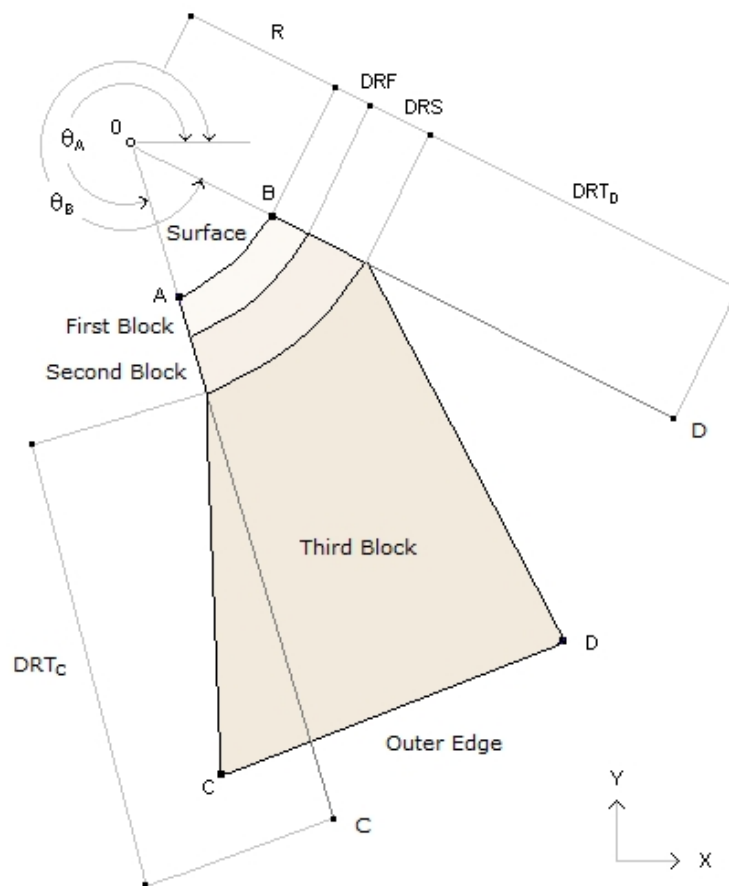


Figure 7.6 Typical Subregion for ISBTYP = 0

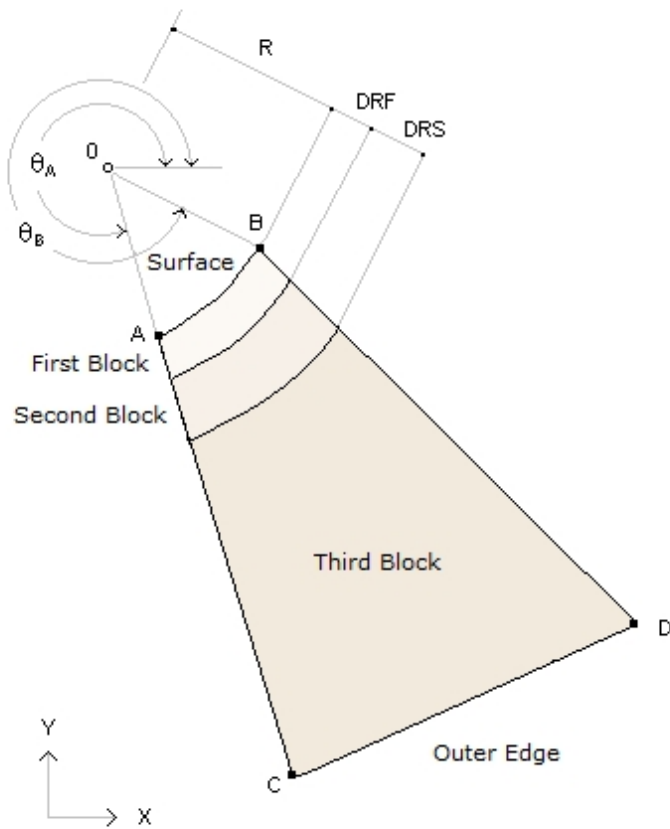


Figure 7.7 Typical Subregion for ISBTYP = 1

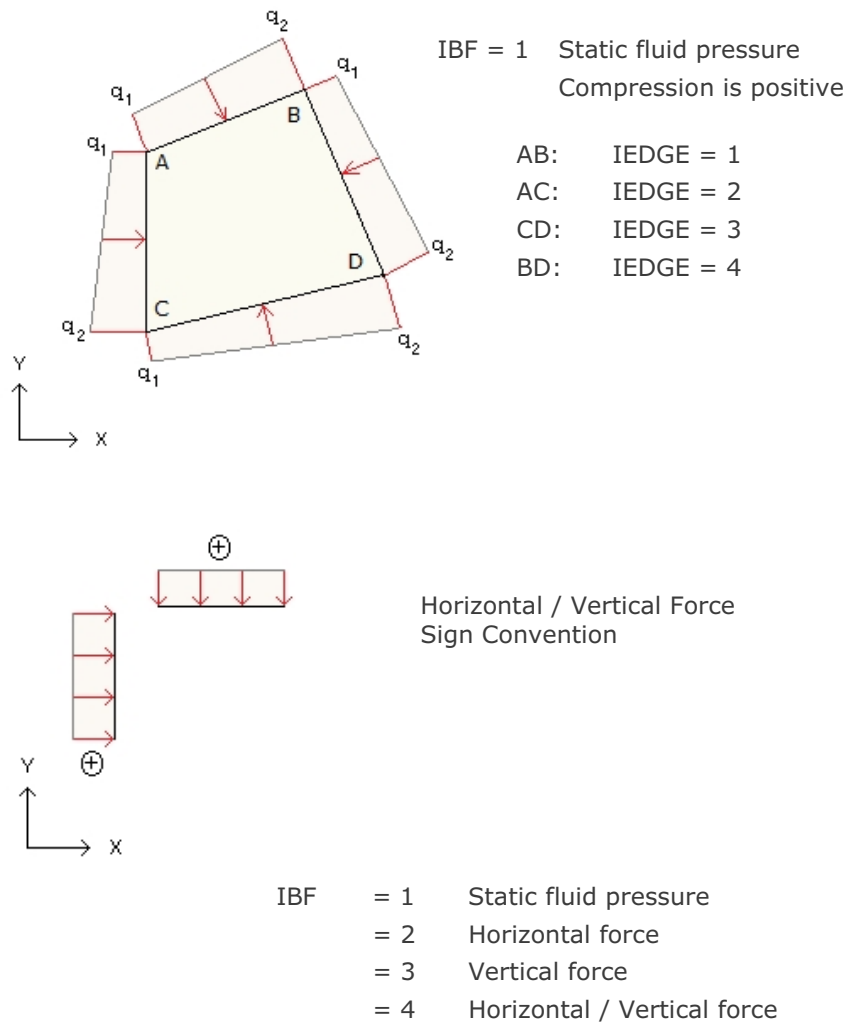


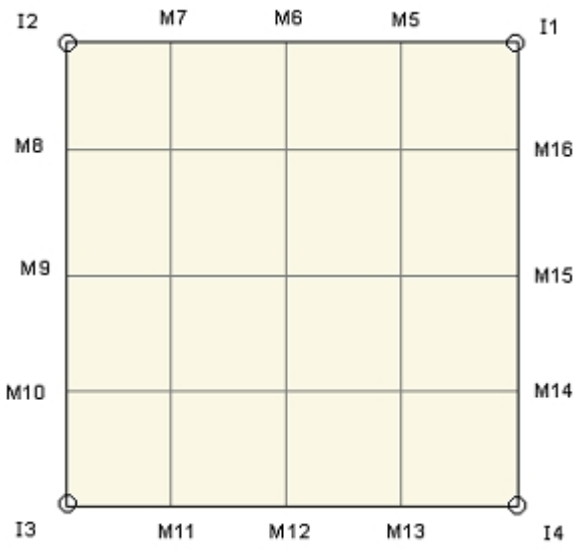
Figure 7.8 Boundary Force/Pressure Sign Conventions

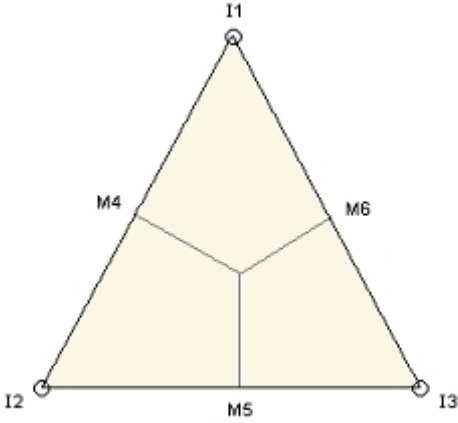
PRESMAP-2D
Model 3
User's Manual

Card Group	Input Data and Definitions (Model 3)		
1	General Information	1.1	
		TITLE	
		TITLE	Any title (Max = 60 characters)
1.2	IP	Plane geometry	
		IP = 0	Axisymmetry geometry
		= 1	
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		NODE ₁ , X ₁ , Y ₁	
Cards		NODE ₂ , X ₂ , Y ₂	
		-	-
		-	-
	NODE	Node number	
	X	X-coordinate	
	Y	Y-coordinate	

Card Group	Input Data and Definitions (Model 3)
3	3.1 IBLNO, IBLTYPE, MATNO, KS, KF (SMAP-2D) IBLNO, IBLTYPE, MATNO, DENSITY (SMAP-S2) IBLNO, IBLTYPE, MATNO, IDH (SMAP-T2/W2) 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 / water generation history ID number

Card Group	Input Data and Definitions (Model 3)
3	3.2 For IBLTYPE = 1 I₁, I₂, I₃, I₄, M₅, M₆, M₇, M₈ I₁, I₂, I₃, I₄ Corner node number M₅, M₆, M₇, M₈ Side node number Note: IBLTYPE = 1 generates 4 elements

Card Group	Input Data and Definitions (Model 3)
3	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 Data for Each Block  Note: IBLTYPE = 2 generates 16 elements

Card Group	Input Data and Definitions (Model 3)
3	For IBLTYPE = 3 I₁, I₂, I₃, M₄, M₅, M₆ I₁, I₂, I₃ M₄, M₅, M₆ Corner node number Side node number  Note: IBLTYPE = 3 generates 3 elements

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

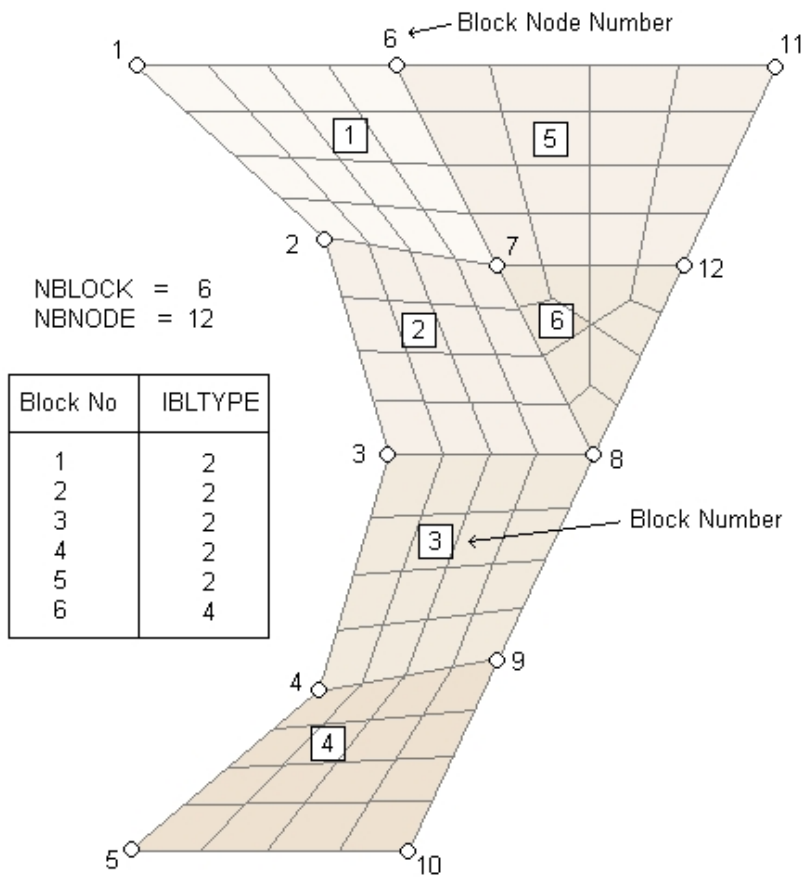


Figure 7.9 Block Node Number for Model 3 of PRESMAP-2D

PRESMAP-2D
Model 4
User's Manual

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	3.1
Material Parameters	MATNO, KS, KF (SMAP-2D)
	MATNO, DENSITY (SMAP-S2)
	MATNO, IDH (SMAP-T2/W2)
	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 / water generation history ID number

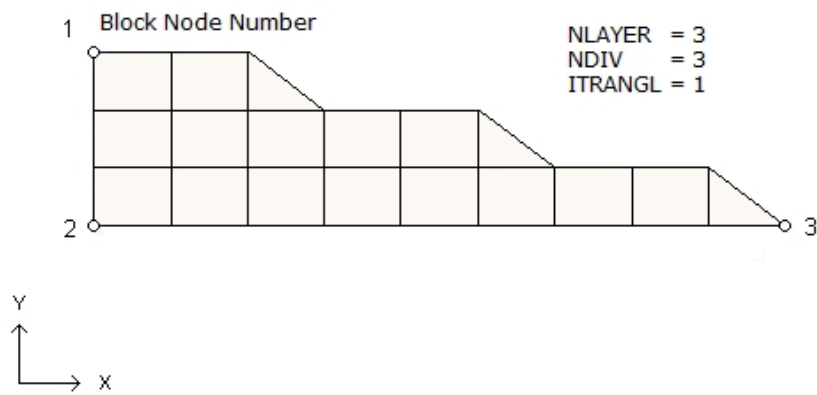


Figure 7.10 Block node number for Model 4 of PRESMAP-2D

NATM-2D
User's Manual

Card Group	Input Data and Definitions															
1	1.1 TITLE TITLE Any title (Max = 60 characters)															
	1.2 IUNIT <table><tr><th>IUNIT</th><th>Length</th><th>Force</th><th>Pressure</th><th>Unit Weight</th></tr><tr><td>1</td><td>in</td><td>lb</td><td>lb/in²</td><td>lb/in³</td></tr><tr><td>2</td><td>m</td><td>ton</td><td>ton/m²</td><td>ton/m³</td></tr></table>	IUNIT	Length	Force	Pressure	Unit Weight	1	in	lb	lb/in ²	lb/in ³	2	m	ton	ton/m ²	ton/m ³
	IUNIT	Length	Force	Pressure	Unit Weight											
1	in	lb	lb/in ²	lb/in ³												
2	m	ton	ton/m ²	ton/m ³												
1.3 MODEL, IGEN, IEXMESH, ILNCOUPL, IAUTO MODEL = 1 Single tunnel (Half section) = 2 Single tunnel (Full section) = 3 Two tunnels (Symmetric) = 4 Two tunnels (Unsymmetric) IGEN = 0 Generate whole mesh = 1 Generate core = 2 Generate surrounding IEXMESH = 0 No user supplied mesh = 1 Add generated mesh to user supplied mesh For Lining analysis ILNCOUPL= 0 Surrounding rock by continuum element = 1 Surrounding rock by spring element IAUTO = 0 Generate Mesh file = 1 Generate Mesh, Main and Post files Available only for SMAP-S2 See Figure 7.11																

Card Group	Input Data and Definitions
2	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	3.1		
	NLAYER		
Soil / Rock Layer Information	NLAYER	Total number of layers. Max = 10	
	3.2		
	NLAYER	┌ LAYERNO ₁ ,	H ₁ , DD ₁
		├ LAYERNO ₂ ,	H ₂ , DD ₂
	Cards	├ -	- -
		└ -	- -
	LAYERNO	Soil/rock layer number	
	H	Thickness of soil/rock layer	
	DD = GAMA	SMAP-S2	
	= IDH	SMAP-T2/W2	
= KF	SMAP-2D		
GAMA	Unit weight		
IDH	Heat / water generation history ID number		
KF = 0	Has fluid phase		
= 1	No fluid phase		
	See Figure 7.11		

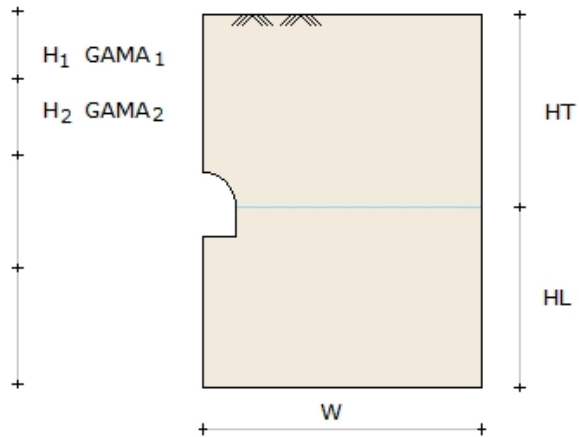
Soil / Rock Layer Information

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 shotcrete 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	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 $AJ_1, AJ_1, \dots, AJ_{NUMSJ}$ AJ_i Angle (degrees) from crown top ($AJ_i \leq 180$)

Water Pressure and Loosening Load

MODEL = 1 Single Tunnel (Half Section)



MODEL = 2 Single Tunnel (Full Section)

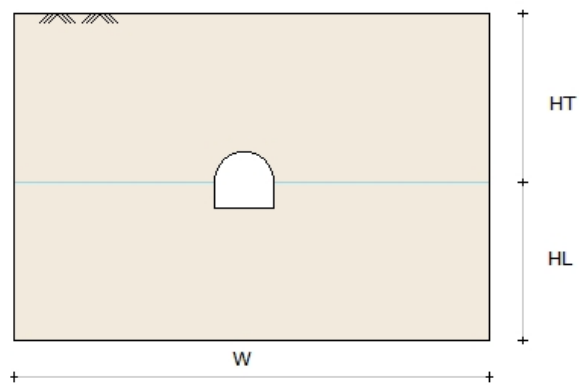
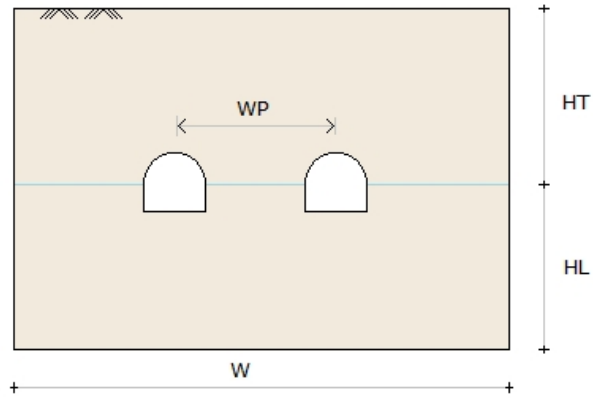


Figure 7.11 Schematic tunnel section view for MODEL = 1 and 2

MODEL = 3 Two Tunnel (Symmetric Section)



MODEL = 4 Two Tunnel (Unsymmetric Section)

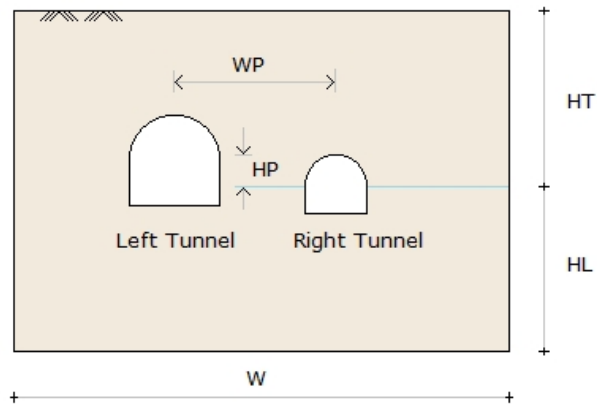
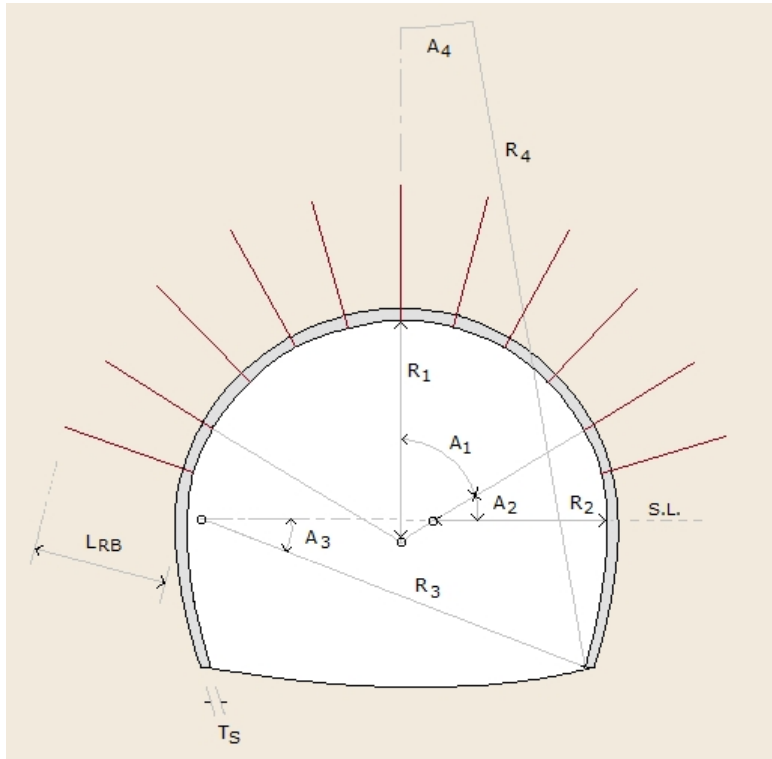


Figure 7.11 Schematic tunnel section view for MODEL = 3 and 4



$R_4 = 0$: 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 ($A_1 + A_2 = 90$)

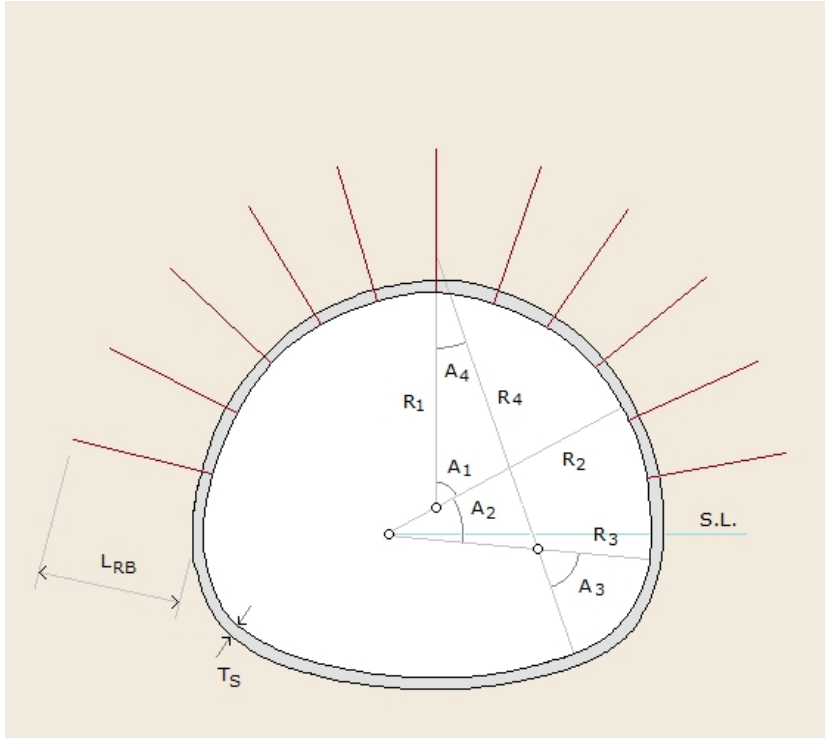


Figure 7.12 Tunnel dimension ($A_1 + A_2 > 90$)

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	3.1 COREMAT₁, COREMAT₂, COREMAT_{2j}, JOINTMAT, NEARMAT COREMAT₁ Material No for Core 1 COREMAT₂ 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 COREMAT₁ and COREMAT₂ have the common interface with NEARMAT and JOINTMAT, respectively. When material number for COREMAT₁ or JOINTMAT is zero, meshes corresponding to that material will not be generated.

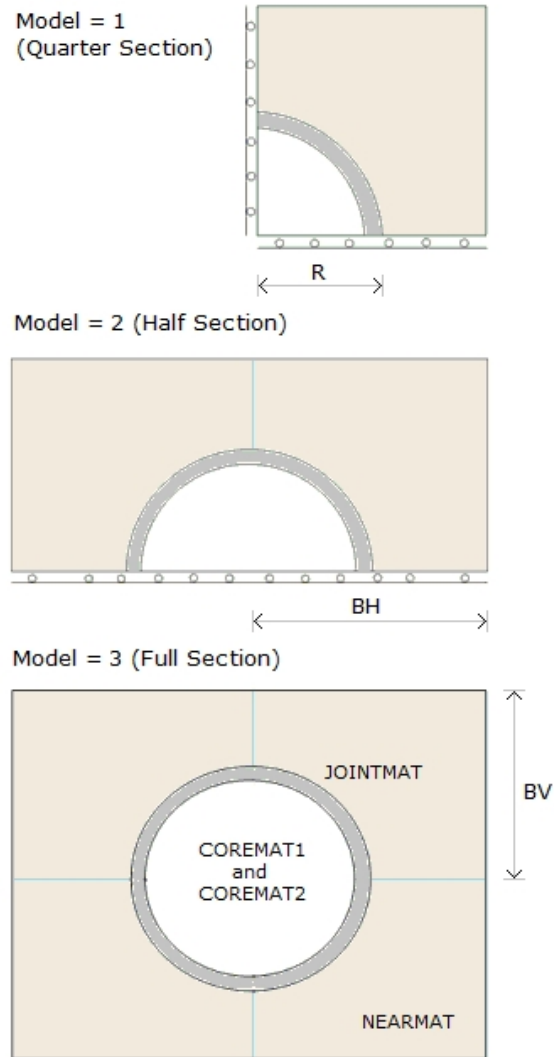


Figure 7.13 Model type for CIRCLE-2D

PRESMAP-GP
User's Manual

Card Group	Input Data and Definitions									
1	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 Mesh generation program (ISMAP) <table> <tr> <td>SMAP-S2 (1)</td><td>SMAP-2D (2)</td></tr> <tr> <td>SMAP-S3 (3)</td><td>SMAP-3D (3)</td></tr> <tr> <td>SMAP-T2 (-2)</td><td>SMAP-T3 (-3)</td></tr> <tr> <td>SMAP-W2 (9)</td><td>SMAP-W3 (10)</td></tr> <tr> <td>SMAP-C2 (11)</td><td>SMAP-C3 (12)</td></tr> </table> 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	SMAP-S2 (1)	SMAP-2D (2)	SMAP-S3 (3)	SMAP-3D (3)	SMAP-T2 (-2)	SMAP-T3 (-3)	SMAP-W2 (9)	SMAP-W3 (10)	SMAP-C2 (11)
SMAP-S2 (1)	SMAP-2D (2)									
SMAP-S3 (3)	SMAP-3D (3)									
SMAP-T2 (-2)	SMAP-T3 (-3)									
SMAP-W2 (9)	SMAP-W3 (10)									
SMAP-C2 (11)	SMAP-C3 (12)									

Card Group	Input Data and Definitions
1	1.3 Six cards starting from right, left, top, bottom, front, back <u>For SMAP-S2 / S3 / 2D / 3D / C2 / C3</u> ISG, ISX, ISY,ISZ, IFG, IFX, IFY,IFZ, IRG, IRX, IRY,IRZ <u>For SMAP-T2 / T3</u> ITG, IDF, T, CF <u>For SMAP-W2 / W3</u> IHG, IDF, H, 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 IHG = 0 None = 1 Water Flow = 2 Total head = 3 Potential seepage face IDF Time function identification number T Initial temperature H Initial total head CF Time function coefficient
	1.4 ELMIN, MAXNEL ELMIN Minimum element length MAXNEL Maximum number of elements

Card Group	Input Data and Definitions			
2	2.1			
Block Coordinate	NBNode	┌	Node ₁ , X ₁ , Y ₁ , Z ₁	
			Node ₂ , X ₂ , Y ₂ , Z ₂	
	Cards		- - - -	
		└	- - - -	
	Node		Node number	
	X		X-coordinate	
	Y		Y-coordinate	
	Z		Z-coordinate	

Card Group	Input Data and Definitions
3	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

Data for Each Line Block [IBETYPE = 1]

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
	3.4.2	<u>NBOUND cards</u> <u>For SMAP-S2 / S3 / 2D / 3D / C2 / C3</u> IBTYPE, ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ <u>For SMAP-T2 / T3 / W2 / W3</u> IBTYPE, ID, IDF, TH, 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 <u>For SMAP-T2 / T3 / W2 / W3</u> ID = 0 Heat / Water flow is specified = 1 Temperature / Head is specified = 3 Potential seepage face is specified IDF Time function identification number TH Initial temperature / Initial total head CF Time function coefficient

Data for Each Line Block [IBTYPE = 1]

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

Data for Each Quad Surface Block [IBETYPE = 2]

Card Group	Input Data and Definitions
3	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 $ILAG = 1$ <u>For $ICOORD = 2$</u> M_{10} Node number defining origin of spherical coordinate <u>For $ICOORD = 3$</u> 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

Data for Each Quad Surface Block [IBETYPE = 2]

Card Group	Input Data and Definitions	
3	3.4	3.4.1 NBOUND Number of boundaries to be specified If NBOUND = 0, go to Card group 3.5
	3.4.2	<u>NBOUND cards</u> <u>For SMAP-S2 / S3 / 2D / 3D / C2 / C3</u> IBTYPE, ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ <u>For SMAP-T2 / T3 / W2 / W3</u> IBTYPE, ID, IDF, TH, 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 <u>For SMAP-T2 / T3 / W2 / W3</u> ID = 0 Heat / Water flow is specified = 1 Temperature / Head is specified = 3 Potential seepage face is specified IDF Time function identification number TH Initial temperature / Initial total head CF Time function coefficient

Data for Each Quad Surface Block [IBETYPE = 2]

Card Group	Input Data and Definitions	
3	3.5	MATNO, NDX, NDY NT₁, NT₂, NT₃, NT₄ MAT₁, MAT₂, MAT₃, MAT₄ THICK, DENSITY (For ISMAP = 1) KS, KF (For ISMAP = 2, 11) IDH (For ISMAP = -2, -3, 9, 10) MATNO Material property number NDX Number of elements in I₂ to I₁ direction NDY Number of elements in I₂ to I₃ direction NT For NT i is greater than zero, a triangle at block node i with NT i divisions along the triangle base. NT i ≤ min (NDX, NDY) and NT i + NT j ≤ min (NDX, NDY) where i = 1, 2, 3, 4 j = 2, 3, 4, 1 MAT_i Material property number for the triangle at block node i. Zero value of MAT will remove the triangle. THICK Thickness of element. For plane strain, use THICK = 1.0 DENSITY Unit weight of element KS = -1 Element has high explosive solid phase = 0 Element has solid phase > 0 Element has joint and absolute value of KS represents face designation number. KF = 0 Element has fluid phase = 1 Element has no fluid phase IDH Heat / Water generation history ID number

Data for Each Quad Surface Block [IBETYPE = 2]

Card Group	Input Data and Definitions
3	3.6 Only for ICOORD = 2 and ILAG = 2 NSEG NSEG Cards ALPA₁, ALPA₂, - NDIV₁ NDIV₂ -

NSEG

ALPA

NDIV

Number of segments

Percent radial distance from origin

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.

2.

3.

4.

ICOORD = 2 (Spherical Coordinate)

IMOD = 0 Curved edge

= 2 Straight edge

Midside and center nodes are not used.

NDX = NDY = NDXY = $\sum NDIV_i$

NDXY

NDXY

NDXY

ND₂

ND₁

α_2

α_1

Origin

Card Group	Input Data and Definitions
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_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

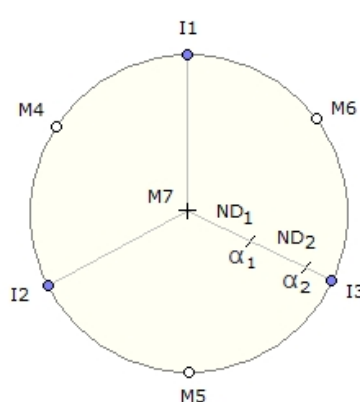
Data for Each Triangle Surface Block [IBETYPE = -2]

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

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
	3.4.2	<u>NBOUND cards</u> <u>For SMAP-S2 / S3 / 2D / 3D / C2 / C3</u> IBTYPE, ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ <u>For SMAP-T2 / T3 / W2 / W3</u> IBTYPE, ID, IDF, TH, 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 <u>For SMAP-T2 / T3 / W2 / W3</u> ID = 0 Heat / Water flow is specified = 1 Temperature / Head is specified = 3 Potential seepage face is specified IDF Time function identification number TH Initial temperature / Initial total head CF Time function coefficient

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

Card Group	Input Data and Definitions	
3	3.5	
Data for Each Triangle Surface Block [IBETYPE = -2]	MATNO, NDXY	
	THICK, DENSITY	(For ISMAP = 1)
	KS, KF	(For ISMAP = 2, 11)
	IDH	(For ISMAP = -2, -3, 9, 10)
	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 / Water generation history ID number

Card Group	Input Data and Definitions																		
3	3.6 Only for ICOORD = 2 and ILAG = 2 NSEG <table><tr><td>NSEG</td><td>[</td><td>ALPA₁,</td><td>NDIV₁</td></tr><tr><td>Cards</td><td>[</td><td>ALPA₂,</td><td>NDIV₂</td></tr><tr><td></td><td>[</td><td>-</td><td>-</td></tr></table> <table><tr><td>NSEG</td><td>Number of segments</td></tr><tr><td>ALPA</td><td>Percent radial distance from origin</td></tr><tr><td>NDIV</td><td>Number of divisions between ALPA_{i-1} and ALPA_i</td></tr></table> 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 (M₇=M₈) 4. Midside nodes (M₄, M₅ and M₆) are interpolated based on spherical coordinate 	NSEG	[	ALPA ₁ ,	NDIV ₁	Cards	[	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
NSEG	[	ALPA ₁ ,	NDIV ₁																
Cards	[	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																		

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

Card Group	Input Data and Definitions
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

Data for Each Hexahedron Volume Block [IBETYPE = 3]

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 3.4.2 <u>NBOUND cards</u> <u>For SMAP-S3 / 3D / C3</u> IBTYPE, ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ <u>For SMAP-T3 / W3</u> IBTYPE, ID, IDF, TH, 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 I₁ - I₂ = 9 Line I₂ - I₃ = 10 Line I₃ - I₄ = 11 Line I₄ - I₁ = 12 Line I₅ - I₆ = 13 Line I₆ - I₇ = 14 Line I₇ - I₈ = 15 Line I₈ - I₅ = 16 Line I₁ - I₅ = 17 Line I₂ - I₆ = 18 Line I₃ - I₇ = 19 Line I₄ - I₈ = 20 Node I₁ = 21 Node I₂ = 22 Node I₃ = 23 Node I₄ = 24 Node I₅

Card Group	Input Data and Definitions
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-T3 / W3 ID = 0 Heat / Water flow is specified = 1 Temperature / Head is specified = 3 Potential seepage face is specified IDF Time function identification number TH Initial temperature / Initial total head CF Time function coefficient

Data for Each Hexahedron Volume Block [IBTYPE = 3]

Card Group	Input Data and Definitions
3	3.5 MATNO, NDX, NDY, NDZ, KS, KF (For ISMAP = 3, 12) MATNO, NDX, NDY, NDZ, IDH (For ISMAP = -3, 10) NT₁, NT₂, NT₃, NT₄ MAT₁, MAT₂, MAT₃, MAT₄ MATNO Material property number NDX Number of elements in I₂ - I₁ direction NDY Number of elements in I₂ - I₃ direction NDZ Number of elements in I₂ - I₆ 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 / Water generation history ID number NT & MAT See descriptions on page 7-92

Data for Each Hexahedron Volume Block [IBETYPE = 3]

Card Group	Input Data and Definitions
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

Data for Each Prism Volume Block [IBETYPE = -3]

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
		3.4.2 NBOUND cards For SMAP-S3 / 3D / C3 IBTYPE, ISX, ISY, ISZ, IFX, IFY, IFZ, IRX, IRY, IRZ For SMAP-T3 / W3 IBTYPE, ID, IDF, TH, CF IBTYPE = 1 Interior volume = 2 Front surface = 3 Back surface = 4 Left surface = 5 Right surface = 6 Bottom surface = 7 Line I₁ - I₂ = 8 Line I₂ - I₃ = 9 Line I₃ - I₁ = 10 Line I₄ - I₅ = 11 Line I₅ - I₆ = 12 Line I₆ - I₄ = 13 Line I₁ - I₄ = 14 Line I₂ - I₅ = 15 Line I₃ - I₆ = 16 Node I₁ = 17 Node I₂ = 18 Node I₃ = 19 Node I₄ = 20 Node I₅ = 21 Node I₆ See Figure 7.24

Data for Each Prism Volume Block [IBTYPE = -3]

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 <u>For SMAP-T3 / W3</u> ID = 0 Heat / Water flow is specified = 1 Temperature / Head is specified = 3 Potential seepage face is specified IDF Time function identification number TH Initial temperature / Initial total head CF Time function coefficient
	3.5 MATNO, NDXY, NDZ, KS, KF (For ISMAP = 3, 12) MATNO, NDXY, NDZ, IDH (For ISMAP = -3, 10) 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 / Water 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 Coincident Nodes

RLIMIT

When the distance between two adjacent nodes is less than RLIMIT, those two nodes are assumed to be coincident.

2. Variables Controlling 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 Controlling 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

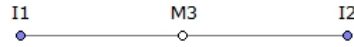
CDCLOSE = 359.1 CDTOL = 0.001 CDZERO = 0.001

Note: Boundary Conditions

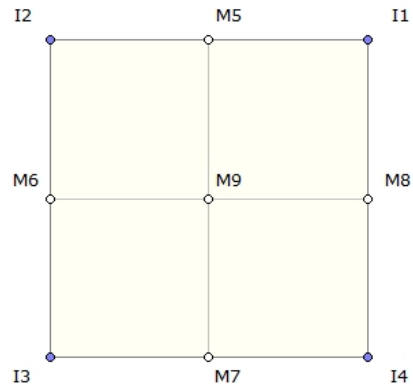
Boundary conditions at nodes are generated based on following rules:

1. Default conditions are applied first based on block type
2. Default conditions can be overridden by specifying IBTYPE = 1
3. Higher IBTYPE overrides lower IBTYPE in a given block
4. Each block number defined later governs conditions along the block interface

Line Block



Quad Surface Block



Triangle Surface Block

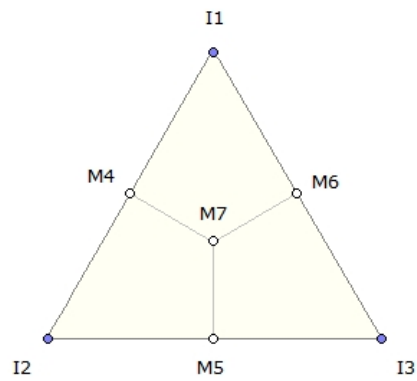


Figure 7.22 Block index for PRESMAP-GP

Hexahedron Volume Block

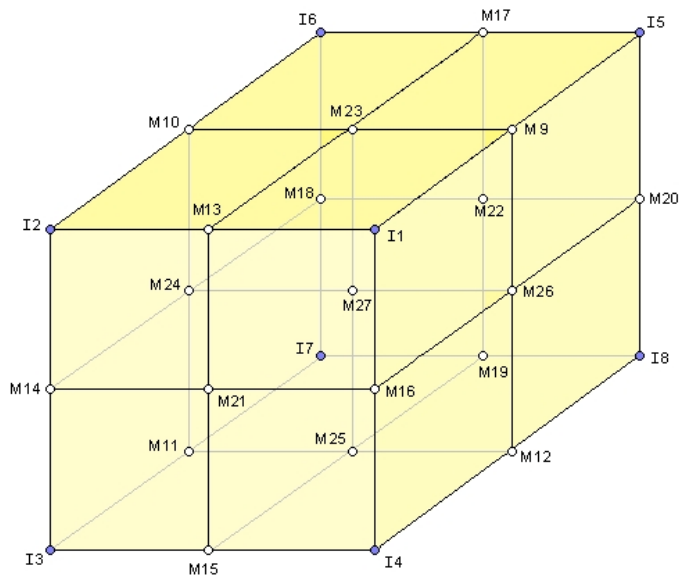


Figure 7.22 Block index for PRESMAP-GP (Continued)

Prism Volume Block

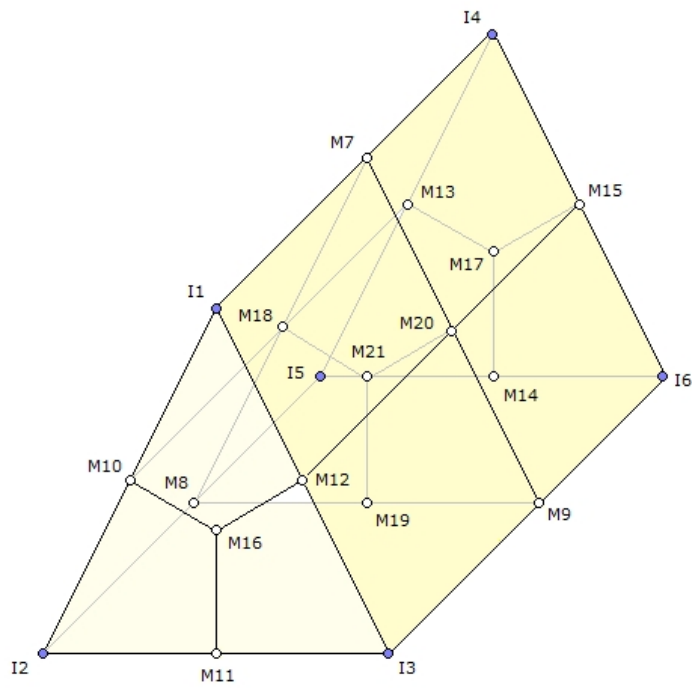


Figure 7.22 Block index for PRESMAP-GP (Continued)

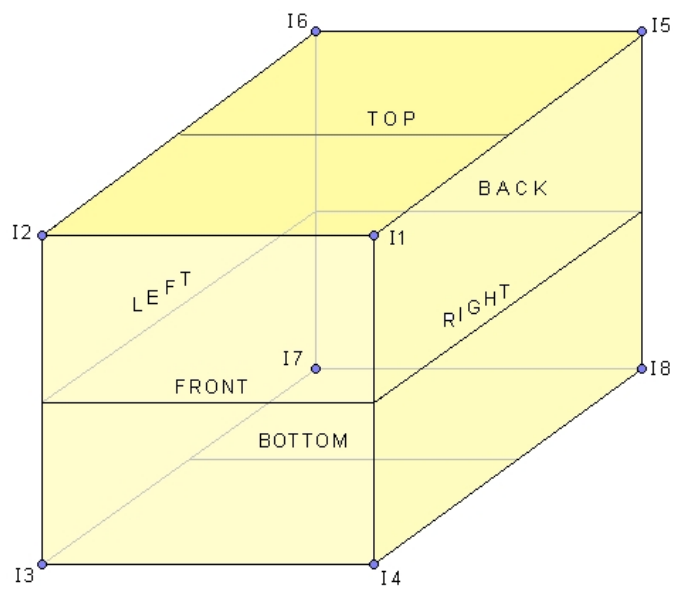


Figure 7.23 Boundary surface designation for Hexahedron Volume Block

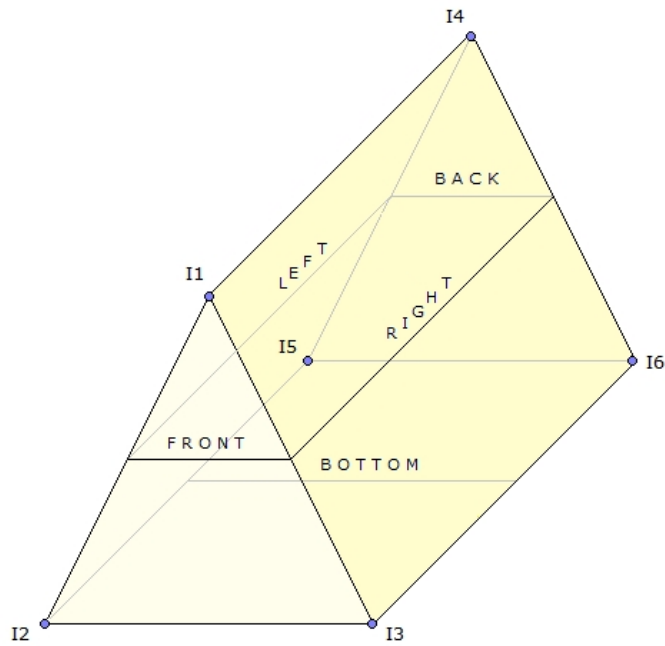


Figure 7.24 Boundary surface designation for Prism Volume Block

ADDRGN User's Manual

8.1 Introduction

ADDRGN is the pre-processing program which has the following two basic functions:

- Combine two different meshes
- Modify existing meshes

A problem geometry can be composed of a number of regions.

Parts of the problem geometry can be generated using the PRESMAP programs described in Section 7. Then ADDRGN is used to combine two different regions (Region A and Region B). When Region B is added to Region A to make Combined Region, following restrictions are applied:

- Element numbers for Region A and Region B should be continuous
- Only those node numbers for Region B are modified to be consistent with the Region A, but element numbers for both regions do not change.

Though the program ADDRGN combines only two regions at a time, users can apply ADDRGN many times to assemble all the different regions.

ADDRGN can also be used to modify the existing meshes:

- Change coordinates
- Change boundary codes
- Cut elements
- Change material numbers

ADDRGN-2D deals with two dimensional meshes and ADDRGN-3D deals with three dimensional meshes.

ADDRGN-2D has an additional powerful feature which is very useful to generate meshes for complicated underground structures. This special feature modifies the existing meshes such that new structures can be easily added by simply specifying the geometries and material properties of structures. It can even generate a base mesh and then add new structures (IMOD=2).

ADDRGN-2D
User's Manual

Card Group	Input Data and Definitions
1	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 = 3 Generate earth dam mesh: EarthDam.Meg =-1 Same as IMOD = 0 except it uses DOF of Region B mesh along the interface JK 1 (T2), 2 (S2), 3 (2D), 9 (W2)
2	2.1 FILEA FILEB FILEC FILEA Input file name containing Region A mesh FILEB Input file name containing Region B mesh FILEC Output file name to store Combined Region mesh When combining Region B mesh to Region A mesh, only Region B node numbers are changed. Element numbers for Region A and Region B should be continuous, otherwise element numbers are automatically reordered by program. 2.2 INTERFACE INTERFACE = 0 Interface is found automatically = 1 Interface is specified by user 2.3 Required only for INTERFACE = 1 NODE NODA₁, NODA₂, ..., NODA_{NODE} NODB₁, NODB₂, ..., NODB_{NODE} NODE Number of interface nodes. NODA_i Interface node numbers in Region A NODB_i Interface node numbers in Region B Note: NODB_i should be the same location as NODA_i

Card Group	Input Data and Definitions
3 Modifying Existing Mesh (IMOD = 1)	3.1 FILEA FILEM FILEA Input file name containing existing mesh FILEM Output file name to store modified mesh
	3.2 NSNEL, NSNODE, NBNEL, NTNEL NSNEL New starting continuum element number NSNODE New starting node number NBNEL New starting beam element number NTNEL New starting truss element number Note: NBNEL & NTNEL are used for IEDIT = 0, 1, 6
	3.3 IEDIT, MC₁, MC₂, MC₃, MB, MT IEDIT = 0 Change coordinates IEDIT = 1 Change boundary codes IEDIT = 2 Cut elements IEDIT = 3 Change material numbers IEDIT = 4 Build user-defined curves and material zones IEDIT = 5 Scale built-in right triangle IEDIT = 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. FILEA is not used for IEDIT = 5.

Card Group	Input Data and Definitions	
3	Modifying Existing Mesh (IMOD = 1)	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: $X_{(new)} = X_{oNew} + (X - X_o) X_{scale}$ $Y_{(new)} = Y_{oNew} + (Y - Y_o) Y_{scale}$

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} Coordinates for upper right boundary
		3.3.2.2.2 Required only for IRANGE = 1, 2, 3 NODE $NOD_1, NOD_2, \dots, NOD_{NODE}$ 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/W2) 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 Flow (0), Temperature / Head (1) specified IDF Time history identification number

Card Group	Input Data and Definitions	
3	Cutting Elements (IEDIT = 2)	3.3.3.1 IRANGE IRANGE = 0 Range specified by coordinates IRANGE = 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 $NEL_1, NEL_2, \dots, NEL_{NOEL}$ 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 INSIDE = 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} 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 8-7)
		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	3.3.5	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 _{LEFT} , X _{RIGHT} , Y _{BOTTOM} , Y _{TOP} X _{LEFT} , X _{RIGHT} , Y _{BOTTOM} , Y _{TOP} Coordinates of rectangle
		3.3.5.4 NGROUP, IGTITL X _{REF} , Y _{REF} , Hi NGROUP Number of curve groups. X _{REF} , Y _{REF} Coordinates of reference point Hi Initial total head (Used for SMAP-W2) 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 <u>For MTYPE = 1 or MTYPE = 2</u> LTP, LMAT <u>For MTYPE = -2</u> MATNO_{JT}, DD_{JT}, THIC_{JT}, LTP_I, LMAT_I, LTP_O, LMAT_O <u>For MTYPE = 3</u> MATNO, DD, LTP, LMAT <u>For MTYPE = -3</u> MATNO, DD, MATNO_{JT}, DD_{JT}, THIC_{JT}, LTP_I, LMAT_I, LTP_O, LMAT_O <u>For MTYPE = 4</u> MATNO, DD, LTP, LMAT, MATold <u>For MTYPE = -4</u> 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/W2) DD_{JT} = KF_{JT} (SMAP-2D) = DEN_{JT} (SMAP-S2) = IDH_{JT} (SMAP-T2/W2) 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/Water 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 flow (ID=0, IDF=LFUN), T2/W2 = 5 Temper / Head bnd. (ID=1, IDF=LFUN), T2/W2 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 <u>Required only for IGPOST= 1</u> 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 <u>Required only for IGPOST= 1</u> CHKBOX (Mesh) CHKBOX (Principal Stress) CHKBOX (Deformed Shape) CHKBOX (Beam) CHKBOX (Truss) CHKBOX (Contour) CHKBOX (Reference Line) CHKBOX = 0 Do not plot CHKBOX = 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)	3.3.5.4.2
		For Each Curve Group	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 [NP₁, X₁, Y₁ NP₂, X₂, Y₂ - - - - - -] 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)	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
			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 NDIV=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 <u>For LTYPE = 2</u> X_{O_r} Y_{O_r} R_{X_r} R_{Y_r} θ_{b_r} θ_{e_r} X_{O_r} Y_{O_r} Arc Origin relative to (X_{LO_r}, Y_{LO_r}) R_{X_r} R_{Y_r} Radius in X and Y axis, respectively θ_{b_r} θ_{e_r} Beginning and ending angle (°) See Figure 8.2

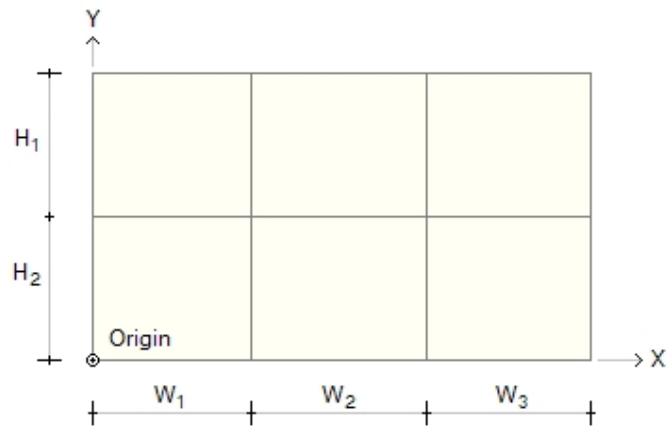
Card Group	Input Data and Definitions	
3 Modifying Existing Mesh (IMOD = 1)	Scale Buile-in Right Triangle (IEDIT = 5)	3.3.6.1 MODEL Select built-in right triangle model MODEL = 1 Slope 1:1 (L_1.0.Mes) = 2 Slope 1.5:1 (L_1.5.Mes) = 3 Slope 2:1 (L_2.0.Mes) = 4 Slope 2.5:1 (L_2.5.Mes) = 5 Slope 3:1 (L_3.0.Mes) Built-in right triangles are found in the directory: c:\Smap\Ct\Ctdata\Slope
		3.3.6.2 MATNO, rHw, Hi MATNO Material number rHw Ratio of water level to dam height Hi Initial total head (Used for SMAP-W2)
		3.3.6.3 Coordinates of new triangle $X_1, Y_1, X_2, Y_2, X_3, Y_3$ X_1, Y_1 Top corner X_2, Y_2 Bottom left corner X_3, Y_3 Bottom right corner

Card Group	Input Data and Definitions	
3	3.6	3.6.1
		3.6.2
		3.6.3

Modifying Existing Mesh (IMOD = 1)	Change Element Index Order (IEDIT = 6)	3.6.1 NumMATC MAT, I₁, I₂, I₃, I₄, MATC, KS, KF (SMAP-2D) MAT, I₁, I₂, I₃, I₄, MATC, THIC, DEN (SMAP-S2) MAT, I₁, I₂, I₃, I₄, MATC, IDH (SMAP-T2/W2) NumMATC MAT I₁, I₂, I₃, I₄ MATC KS, KF, THIC, DEN. IDH Number of continuum materials Material number Element corner index numbers New material property number Refer to Mesh File user manual
		3.6.2 NumSECB SEC, I, J, MSEC, K NumSECB SEC I, J MSEC K Number of beam sections Section number Element corner index numbers New material section number New reference node number
		3.6.3 NumMATT MAT, I, J, MATT, K NumMATT MAT I, J MATT K Number of truss materials Material number Element corner index numbers New material property number 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
5	5.1 NLAYER, ICORE, IWALL, IMESH, NW_{MG}, H_i NLAYER Number of subsoil layers For not including subsoil, set NLAYER = 0 ICORE 0 : Do not include, 1 : Include core IWALL 0 : Do not include, 1 : Include cut-off wall IMESH 0 : Default, 1 : Slope format (See IEDIT = 5) NW_{MG} Number of divisions within W_{MG} (0 : Default) H_i Initial total head (Used for SMAP-W2)
	5.2 H, H_W, W_{UT}, W_{UG}, W_{MT}, W_{MG}, W_{DT}, W_{DG} H, H_W Dam height and Water elevation W_{UT}, W_{UG} Upstream shell width at top and ground level W_{MT}, W_{MG} Main shell width at top and ground level W_{DT}, W_{DG} Downstream shell width at top and ground level
	5.3 Required only for ICORE = 1 XA_1, YA_1, XA_2, YA_2 XA_3, YA_3 XA_4, YA_4 XA_i, YA_i X and Y coordinates of core
	5.4 Required only for NLAYER > 0 W_{UB}, W_{DB} H_1, H_2, H_{NLAYER} W_{UB}, W_{DB} Upstream and downstream boundary width H_i Subsoil layer thickness
	5.5 Required only for IWALL = 1 and NLAYER > 0 XB_1, YB_1, XB_2, YB_2 XB_3, YB_3 XB_4, YB_4 XB_i, YB_i X and Y coordinates of cut-off wall

Generate Earth Dam (IMOD = 3) See Figure 8.3



In this example, $NBX=3$ and $NBY=2$

Figure 8.1 Layout of Base Mesh

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

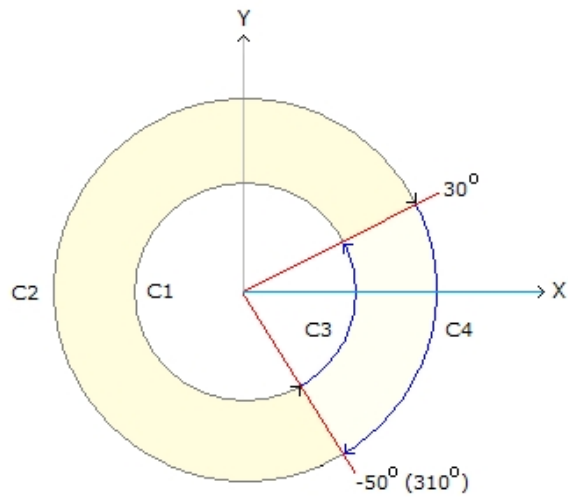


Figure 8.2 Examples of arc specification

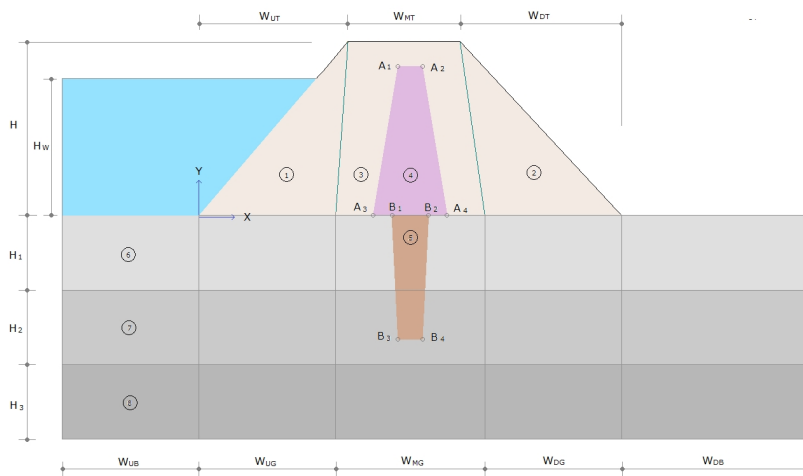


Figure 8.3 Schematic section view of earth dam

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 Generator → 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

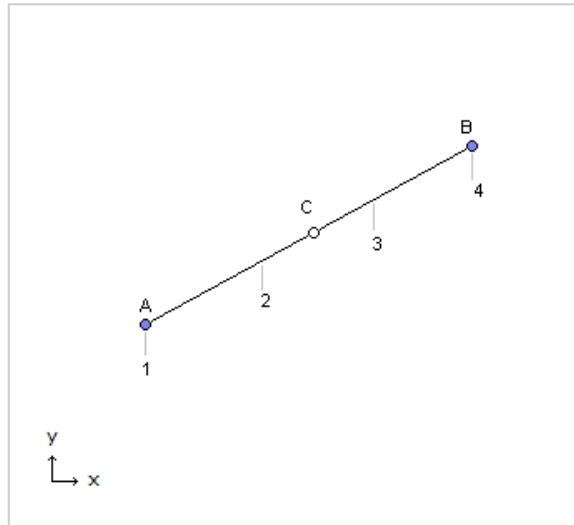
EDIT uses Windows text editor **Wordpad** to creat, modify, or list file.

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

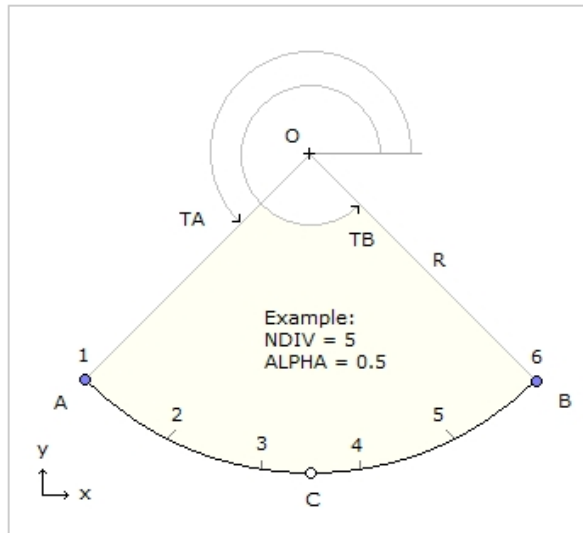
XA, YA = X and Y coordinates of A

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
 $NDIV$, $ALPHA$

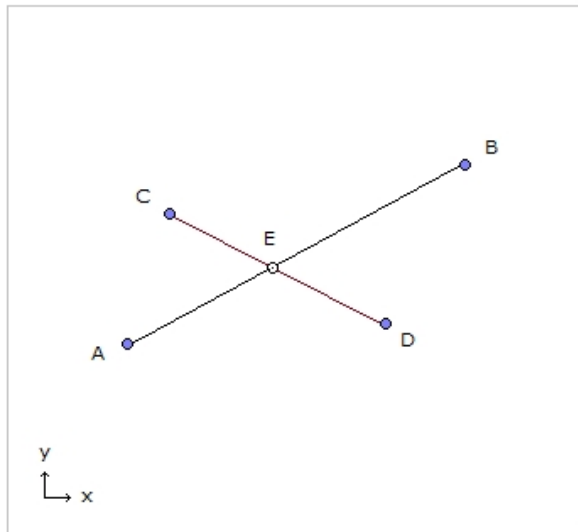
R = Radius
 X_o, Y_o = X and Y coordinates of origin O
 TA, TB = Angles (degrees) of A and B
 $NDIV$ = Number of division
 $ALPHA$ = Geometric ratio

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

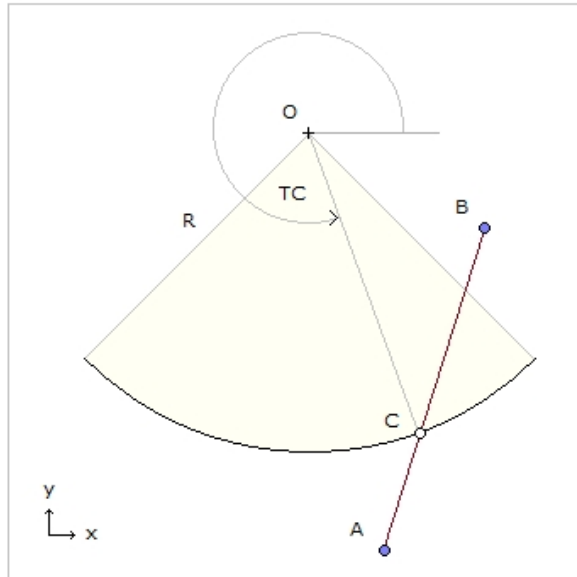


INPUT:

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

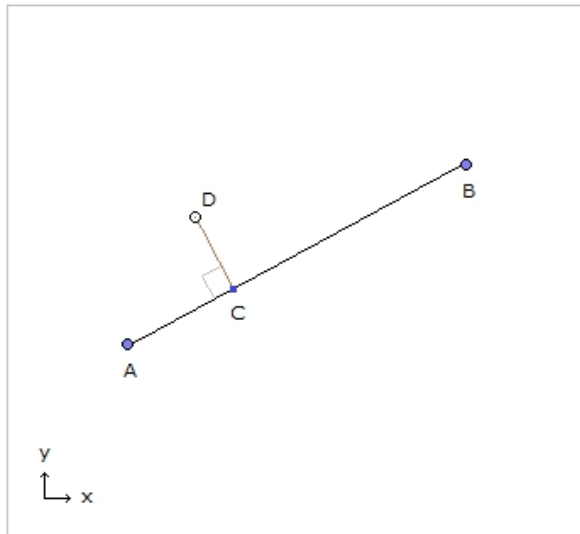


INPUT:

$R,$ $X_o,$ 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

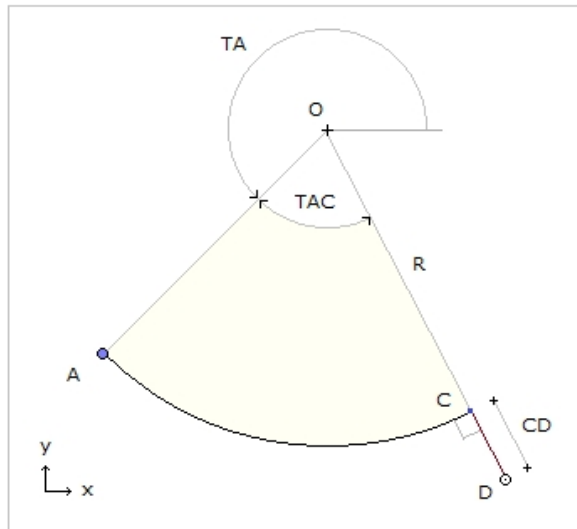


INPUT:

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.

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-W2 Mesh File is presented in detail in Section 4.3.

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

10.2 Conversion to SMAP-W2 Mesh File

Following Mesh Files can be converted to SMAP-W2 Mesh File format:

- Mesh Files generated for two-dimensional SMAP programs (SMAP-S2, SMAP-2D and SMAP-W2)
- FEMAP (Version 4.1 - 4.5, neutral format)

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

10-2 File Conversion

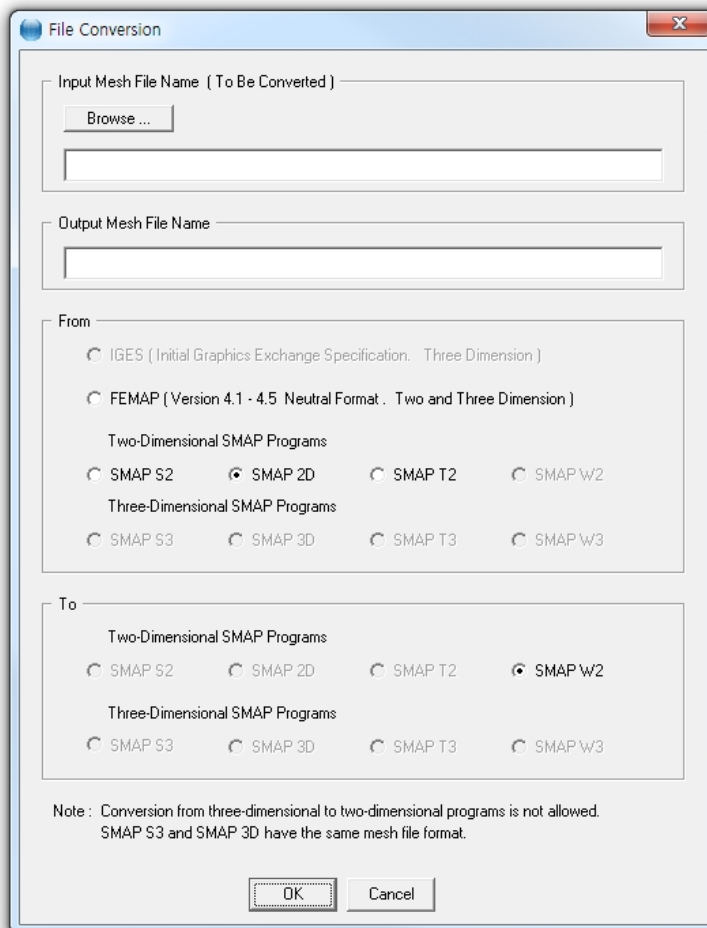


Figure 10.1 File Conversion dialog box

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 PRESMAPI/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 = 7 [Seepage: SMAP-W2]

Card Group	Input Data and Definitions (Seepage)		
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 element (Thickness=1.0)
2	Loading Surface	2.1	NUMLS NUMLS Number of loading surfaces where hydraulic boundary conditions are specified (Max = 20) Loading surface is saved in LoadingSurface.out
		2.2	LSNO, LSTYPE LSNO Loading surface number LSTYPE = 0 All specified nodes = 1 Line strip = 2 Points = 3 Node group = 4 Element group
		2.2.2	NUMNODE NUMNODE Number of nodes on this loading surface (Max = 9990)
		2.2.3	NOD₁, NOD₂, ..., 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.

Card Group	Input Data and Definitions (Seepage)												
2	2.2	Loading Surface											
		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) <table border="1" style="border-collapse: collapse; text-align: center;"> <tr><td>5</td><td>10</td><td>15</td><td>20</td></tr> <tr><td>35</td><td>40</td><td>45</td><td>50</td></tr> <tr><td>65</td><td>70</td><td>75</td><td>80</td></tr> </table> Example NSR = 5 JCR = 5 NJR = 4 ICR = 30 NIR = 3	5	10	15	20	35	40	45	50	65	70
5	10	15	20										
35	40	45	50										
65	70	75	80										

Card Group	Input Data and Definitions (Seepage)	
3	3.1	NUMLF NUMLF Number of loading functions (Max = 20)
	3.2	3.2.1 LFNO LFNO Loading function number
	3.2.2	a_0, a_1, a_2 a_i Coefficients defining loading function (F) $F = a_0 + a_1x + a_2y$

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

Card Group	Input Data and Definitions (Seepage)
5	5.1 IBTYPE IBTYPE = 0 End of data = 1 Initial head = 4 External water flow boundary = 5 Head boundary = 6 Potential seepage boundary For IBTYPE = 1 (Initial Head) LFNO_IH, LHED_IH LFNO_IH Loading function number for initial head LHED_IH 0: Total head 1: Water pressure head

Initial Head & Boundary Condition Specification

Card Group	Input Data and Definitions (Seepage)
5	5.1 <u>For IBTYPE = 4 (External Water Flow Boundary)</u> LSNO_EW, LFNO_EW, LHNO_EW External water flow boundary LSNO_EW Loading surface number LFNO_EW Loading function number for (ID=0, CF) LHNO_EW Loading history number for (IDF) <u>For IBTYPE = 5 (Head Boundary)</u> LSNO_HB, LFNO_HB, LHNO_HB, LHED_HB Head boundary LSNO_HB Loading surface number LFNO_HB Loading function number for (ID=1, CF) LHNO_HB Loading history number for (IDF) LHED_HB 0: Total head 1: Water pressure head <u>For IBTYPE = 6 (Potential Seepage Boundary)</u> LSNO_PS, LFNO_PS, LHNO_PS Potential Seepage boundary LSNO_PS Loading surface number LFNO_PS Loading function number for (ID=4, CF) LHNO_PS Loading history number for (IDF) For IDF = 0, ID = 3 with zero flux For IDF > 0, ID = 4 with hydrograph Note: For IBTYPE = 1, 4, 5, or 6, specified initial head or boundary conditions are saved in mesh file (NewMeshFile.Mes)

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.

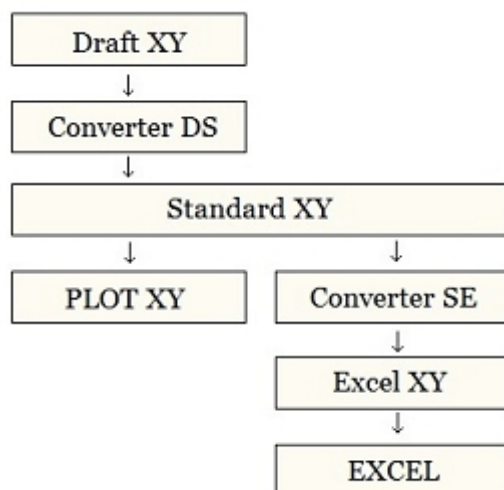


Figure 12.1 Flow diagram of plotting XY graph

Table 12.1 Draft XY Data Format

Card Group	Input Data and Definitions	
First Plot	Title	(Max 50 Characters)
	Sub Title	(Max 50 Characters)
	X-Label	(Max 50 Characters)
	Y-Label	(Max 50 Characters)
	First Curve	X_1 Y_1 X_2 Y_2 - - X_n Y_n 0.0 123456 (End of Curve) Legend 1 (Max 20 Characters) Legend 2 (Max 20 Characters))
First Plot	Second Curve	X_1 Y_1 X_2 Y_2 - - X_n Y_n 0.0 123456 (End of Curve) Legend 1 (Max 20 Characters) Legend 2 (Max 20 Characters)
	Last Curve	X_1 Y_1 X_2 Y_2 - - 0.0 -123456 (End of Line Segment) X_1 Y_1 X_2 Y_2 - - X_n Y_n 0.0 123456 (End of Curve) Legend 1 (Max 20 Characters) Legend 2 (Max 20 Characters) 0.0 987654 (End of Plot)
Next Plot	Next Plot can be added using the same format as the First Plot	

12.2 New Graph

[XY Graph](#) can be created by performing the following steps:

Step 1:

Select the following menu items in [SMAP](#):

[Plot](#) → [XY](#) → [PLOT XY](#) → [New](#)

Step 2:

Once selected, initial default file [XY.dat](#) will be opened by [Notepad](#) as listed in Table 12.2.

Edit this default file according to the format of [Draft XY Data](#) in Table 12.1. And then save and exit.

Step 3:

[Draft XY.dat](#) is automatically changed into [Standard Form](#) by [Converter DS](#) as listed in Table 12.3. Modified graph will be displayed on [PLOT XY](#) drawing board.

Step 4:

[XY Graph](#) can be further modified by [Edit Dialog](#) explained in detail in the next Section 12.3.

Table 12.2 Draft XY Data (Initial Default File [XY.dat](#))

```

Plot No. 1
Sub Title 1
XLabel-1
YLabel-1
0      10
100    20
.000000E+00 .123456E+06
Curve 1
Legend
10,    20
90,    30
.000000E+00 .123456E+06
Curve 2
Legend
.000000E+00 .987654E+06
Plot No. 2
Sub Title 2
XLabel-2
YLabel-2
0      100
1000   200
.000000E+00 .123456E+06
Curve 1
Legend
100    200
900    300
.000000E+00 .123456E+06
Curve 2
Legend
.000000E+00 .987654E+06
Plot No. 3
Sub Title 3
XLabel-3
YLabel-3
0      100
1000   200
.000000E+00 .123456E+06
Curve 1
Legend
200,   200
900,   300
.000000E+00 .123456E+06
Curve 2
Legend
.000000E+00 .987654E+06

```

Table 12.3 Standard XY Data (Initial Default File [XY.dat](#))

```

*****
*                               PLOT NO: 1                               *
*****
C Following data can be modified for plotting configuration
  TITLE (50 CHAR) = Plot No. 1
  SUB-TITLE (50 CHAR) = Sub Title 1
  XLABEL (50 CHAR) = XLabel-1
  YLABEL (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

PLOT NO 1

Titles and Labels

Title: Example 1
 Sub Title: Stress History
 X-Label: Time (Sec)
 Y-Label: Stress (MPa)

General Options

☒ Framing ☒ Gridding ☒ Centering ☐ Log X ☐ Log Y

Dimensions and Scales

Xmax Cm: 2.69 Ymax Cm: 5.99 Dxlegn Cm: 0.00
 Xscale: 1.0000 Yscale: 1.0000 Xdelta: 0.

Manual Scales

Xs: 0. Xe: 120.00 Nodx: 6 Nxdec: -1
 Ys: 8.0000 Ye: 32.000 Nody: 6 Nydec: 2

Curve No 1

1: Mark & Line 1: Solid Line Mark ☐ Color ☒
 Legend: Vertical Stress

< > List ☐ Hide Modify XY Edit XY Delete Add

Sample Description ☐ Add as New Plot OK Cancel

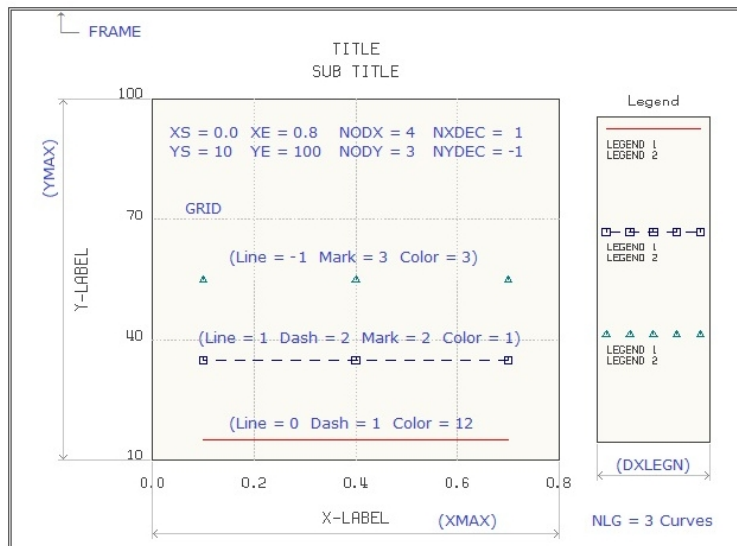


Figure 12.3 Sample graph

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

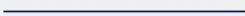
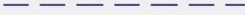
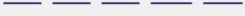
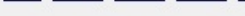
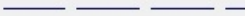
Refer to description in [Sample Graph](#) in Figure 12.3.

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](#) to hide the current curve.

Figure 12.4
Curve options

Line		
-1 Mark	0 Line	1 Mark & Line
Dash		
	1	 1
	2	 2
	3	 3
	4	 4
	5	 5
	6	 6
	7	 7
	8	 8
	9	 9
	10	 10
Color		
 0 Black	 8 Gray	
 1 Blue	 9 Light Blue	
 2 Green	 10 Light Green	
 3 Cyan	 11 Light Cyan	
 4 Red	 12 Light Red	
 5 Magenta	 13 Light Magenta	
 6 Brown	 14 Yellow	
 7 Light Gray	 15 Bright White	

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

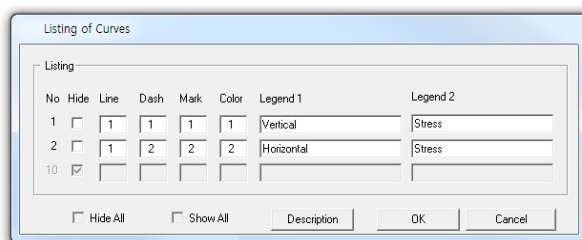


Figure 12.5a Listing of curves

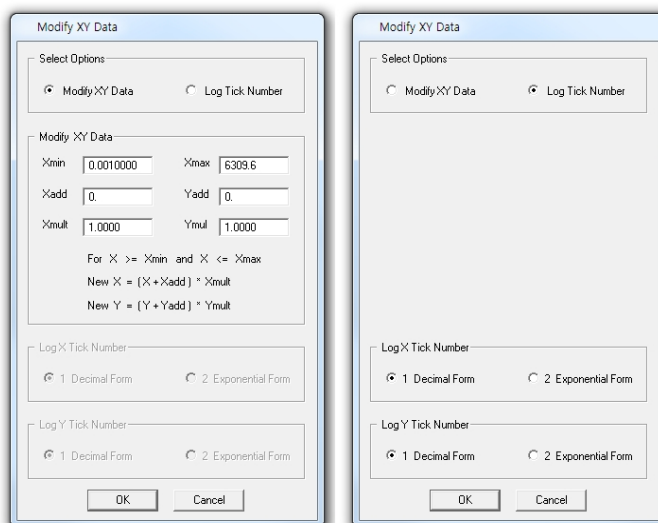


Figure 12.5b Modify current curve XY data

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.

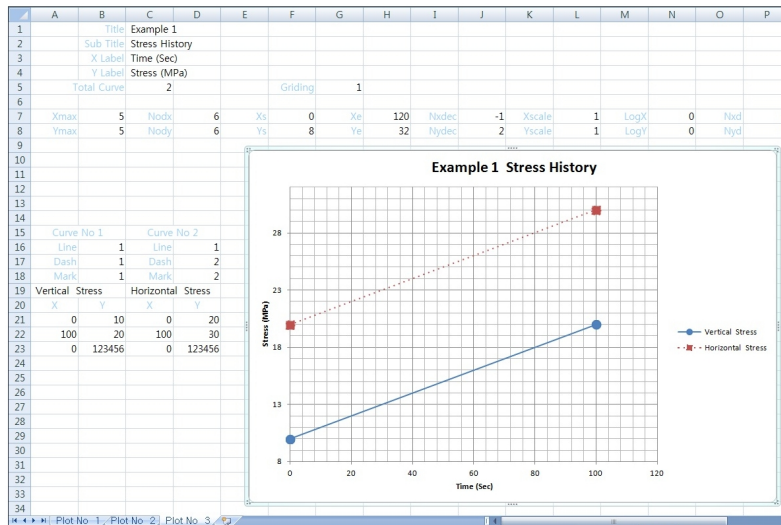


Figure 12.6 XY graph on Excel spreadsheet

Notes on Excel XY Graph

Excel XY Graph can be influenced by the following input parameters in [Standard Form](#):

Note 1: Input Parameters Not Considered

Following parameters are not considered:

Plot dimensions: [XMAX](#) , [YMAX](#)

Number of digits after decimal point: [NXDEC](#), [NYDEC](#)

Note 2: Automatic Scaling (Xscale = 0, Yscale = 0)

For [XSCALE](#) = 0

X axis is automatically scaled and [XS](#), [XE](#) and [NODX](#) are not used.

For [YSCALE](#) = 0

Y axis is automatically scaled and [YS](#), [YE](#) and [NODY](#) are not used.

Note 3: Logarithmic Scaling (Logx = 1, Logy = 1)

For [LOGX](#) = 1

[NODX](#) and [NXD](#) are not used.

If [XSCALE](#) $\neq$ 0 and [XS](#) < 1 and [XE](#) > 1, [XS](#) is automatically scaled.

For [LOGY](#) = 1

[NODY](#) and [NYD](#) are not used.

If [YSCALE](#) $\neq$ 0 and [YS](#) < 1 and [YE](#) > 1, [YS](#) is automatically scaled.

Refer to samples in the following directory:

C:\Smap\Smap3D\Example\XY_Graph\Excel XY Graph Sample.pdf

C:\Smap\Response\ModXY\EX2\Log Tick Number.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.

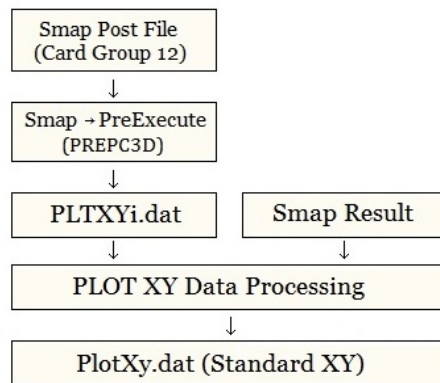


Figure 12.7 Processing SMAP results

SMAP Results can be plotted by performing following steps:

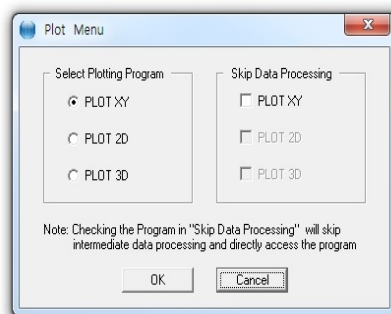
Step 1:

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

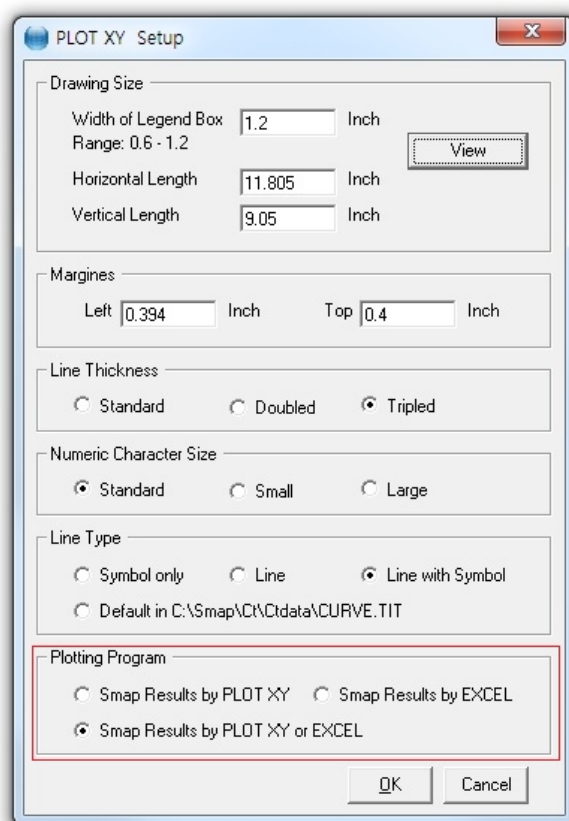


Figure 12.9 PLOT XY setup dialog

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

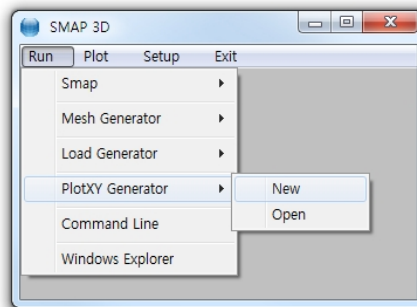
PlotXY Generator can be accessed by selecting the following item in **SMAP** main menu as in Figure 12.10.

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.

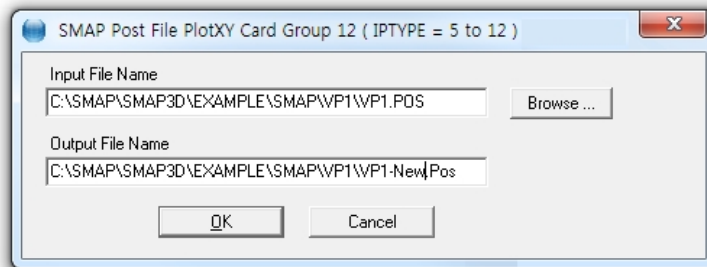


Figure 12.11 PlotXY input and output file dialog

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.

PLOT-XY Input Generator (SMAP Post File Card Group 12)

PLOT NO 1

5 Time History of Stresses/Strains for a Given Element

Title

X_Label

Y_Label

Specified Element

Elemen

Ky

Table Ky

Add Position

☐ Before

☐ After

☒ End

Add

Delete

Multiplication Factor

Time

Stress

Strain

Ky

Ky1

Ky2

Kx = Time

< > List Add Delete Save Exit

Figure 12.12 Time history for a given element

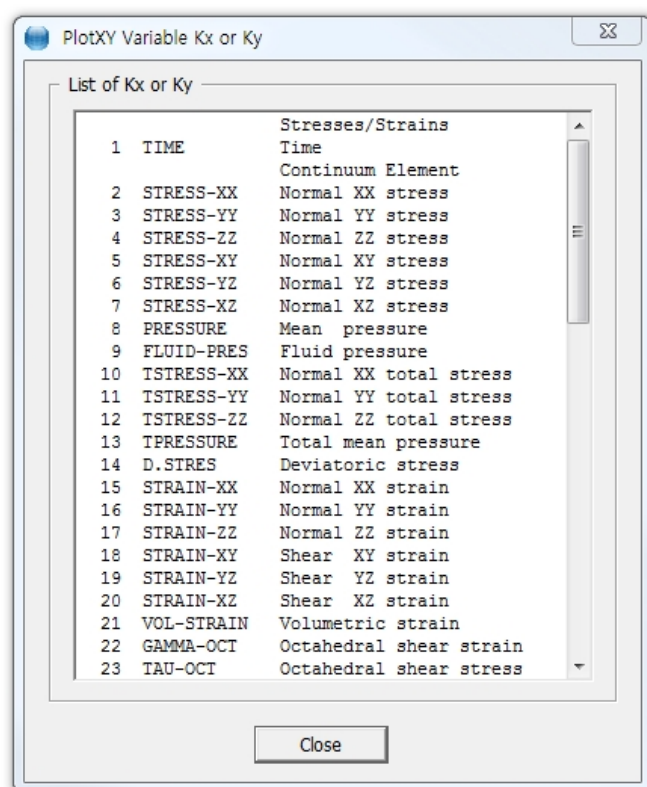


Figure 12.13 Available data for stresses / strains

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.

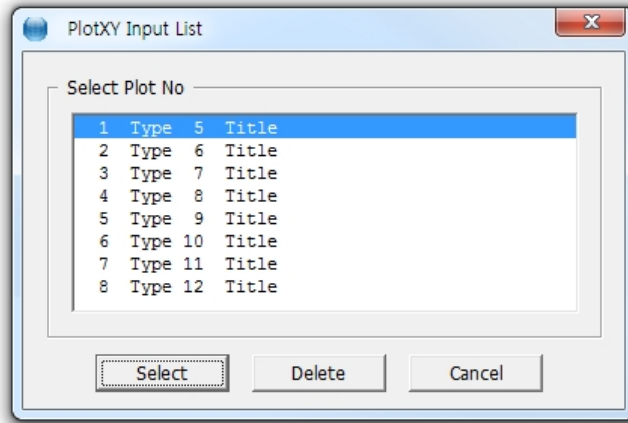


Figure 12.14 Listing of plots

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

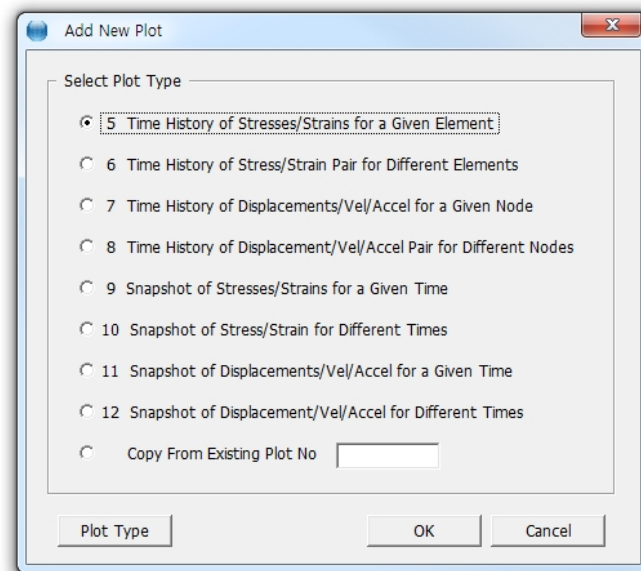


Figure 12.15 Add options for new plot

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

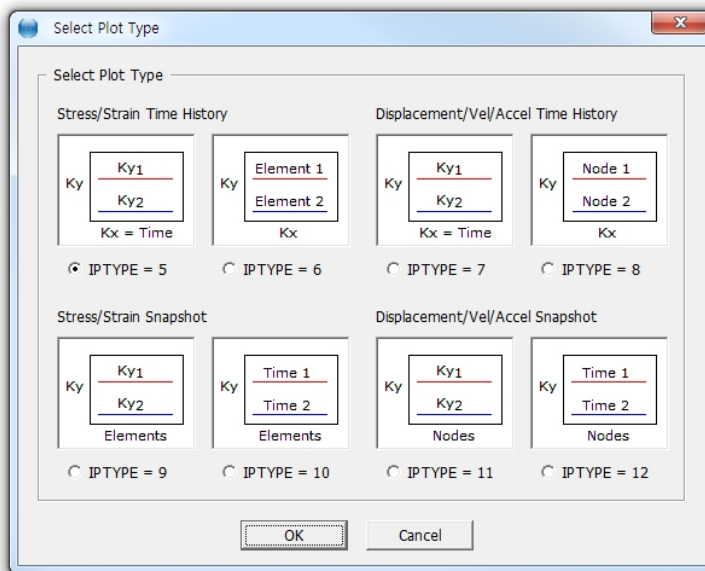


Figure 12.16 Available plot types

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.

The screenshot shows the 'PLOT-XY Input Generator (SMAP Post File Card Group 12)' dialog box. The title bar includes a close button. The main area is titled 'PLOT NO 2' and '6 Time History of Stress/Strain Pair for Different Elements'. It contains several input fields and controls:

- Title:** A text box containing 'Title'.
- Xlabel:** A text box containing 'X_Label'.
- Ylabel:** A text box containing 'Y_Label'.
- Specified Variables:** Two text boxes for 'Kx' (containing '8') and 'Ky' (containing '14'). Below them is a button labeled 'Table Kx Ky'.
- Elements:** A list box containing '1' and '2'. Above it, a small box shows 'Element 1' and 'Element 2' with 'Kx' and 'Ky' labels. Below the list box are 'Add' and 'Delete' buttons.
- Add Position:** A group box with three radio buttons: 'Before', 'After', and 'End' (which is selected).
- Multiplication Factor:** A group box with three text boxes for 'Time', 'Stress', and 'Strain', each containing the value '1'.

At the bottom of the dialog are several buttons: '<', '>', 'List', 'Add', 'Delete', 'Save', and 'Exit'.

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.

PLOT-XY Input Generator (SMAP Post File Card Group 12)

PLOT NO 3

7 Time History of Displacements/Vel/Accel for a Given Node

Title

Xlabel

Ylabel

Specified Node

Node

Ky

2
3

Table Ky

Add Position

☐ Before

☐ After

☒ End

Add

Delete

Multiplication Factor

Time	Displacement	Velocity	Acceleration
1	1	1	1

Ky1
Ky2
Kx = Time

< > List Add Delete Save Exit

Figure 12.18 Time history for a given node

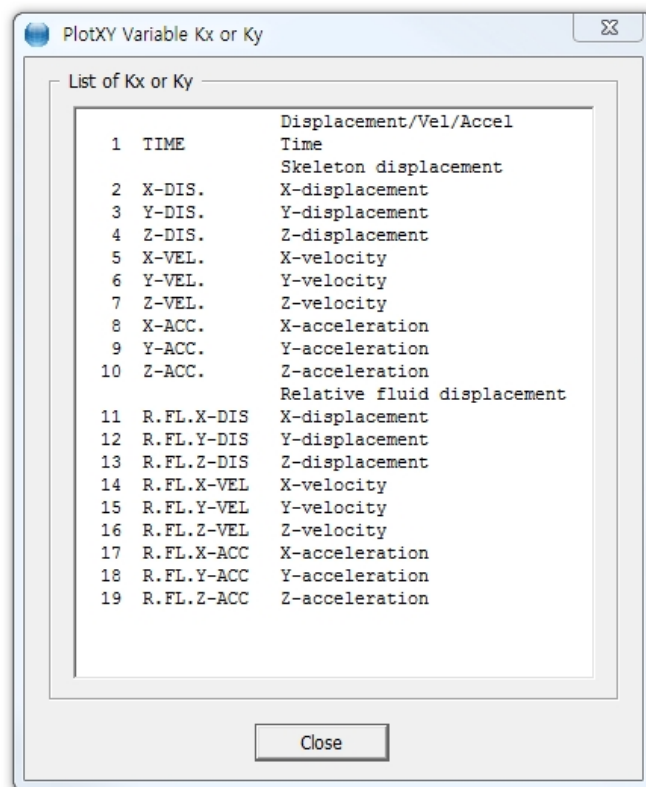


Figure 12.19 Available data for displacement/vel/accel

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.

PLOT-XY Input Generator (SMAP Post File Card Group 12)

PLOT NO 4

8 Time History of Displacement/Vel/Accel Pair for Different Nodes

Title

Xlabel

Ylabel

Specified Variables

Kx

Ky

Table Kx Ky

Nodes

1

2

Add Position

☐ Before

☐ After

☒ End

Multiplication Factor

Time

Displacement

Velocity

Acceleration

< > List Add Delete Save Exit

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](#).

PLOT-XY Input Generator (SMAP Post File Card Group 12)

PLOT NO 5

9 Snapshot of Stresses/Strains for a Given Time

Title:

Xlabel:

Ylabel:

Specified Time: Time

Ky:

Elements:

Table Ky

Starting X-Coordinate: Xstart

Add Position:
☐ Before
☐ After
☒ End

Multiplication Factor:
 Stress:
 Strain:
 Distance:

Buttons: < > List Add Delete Save Exit

Ni, -Nj, Nk Elems from Ni to Nj increment Nk

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).

PLOT-XY Input Generator (SMAP Post File Card Group 12)

PLOT NO 6

10 Snapshot of Stress/Strain for Different Times

Title

Xlabel

Ylabel

Specified Variable
Ky

Table Ky

Starting X-Coordinate
Xstart

Add Position
☐ Before
☐ After
☒ End

Multiplication Factor
 Stress Strain Distance

Times

1
2

Elements

1
-10
1

Ni, -Nj, Nk. Elems from Ni to Nj increment Nk

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](#).

PLOT-XY Input Generator (SMAP Post File Card Group 12)

PLOT NO 7

11 Snapshot of Displacements/Vel/Accel for a Given Time

Title:

Xlabel:

Ylabel:

Specified Time:

Ky:

Nodes:

Table Ky:

Starting X-Coordinate:

Add Position:
☐ Before
☐ Alter
☒ End

Multiplication Factor:
Displacement:
Velocity:
Acceleration:
Distance:

Ni, Nj, Nk Nodes from Ni to Nj increment Nk

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).

PLOT-XY Input Generator (SMAP Post File Card Group 12)

PLOT NO 8

12 Snapshot of Displacement/Vel/Accel for Different Times

Title

X_Label

Y_Label

Specified Variable

Ky

Table Ky

Starting X-Coordinate

Xstart

Add Position

☐ Before

☐ After

☒ End

Multiplication Factor

Displacement

Velocity

Acceleration

Distance

Times

1

2

Nodes

1

2

3

11

-21

2

Add

Delete

Ni, -Nj, Nk Nodes from Ni to Nj increment Nk

< > List Add Delete Save Exit

Figure 12.24 Displ/vel/accel snapshot for different times

PLOT-XY User's Manual

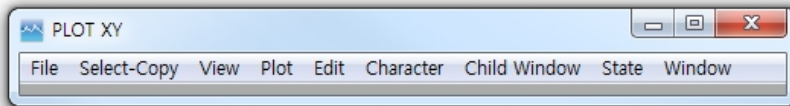
13.1 Introduction

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.

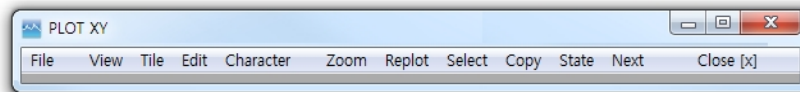
PLOT-XY has two menu styles, General and Express.

General Style includes 9 menus consisting of all menu items available. For General Style, specify 1 in `C:\Smap\Ct\Ctdata\MenuStyle_XY.dat`



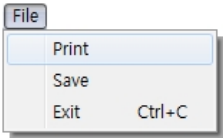
Express Style includes 12 menus which are rearranged so as to quickly access most frequently used menu items in practice.

For Express Style, specify 0 in `C:\Smap\Ct\Ctdata\MenuStyle_XY.dat`



13.2 Menus

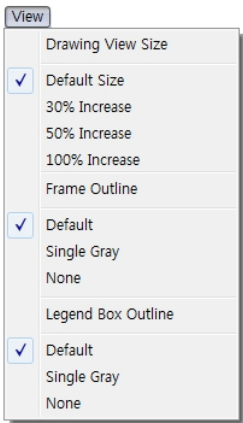
File has three sub menus.
Print is to get the hard copy of the current view.
Save is to save the current view.
Exit is to exit PLOT-XY.



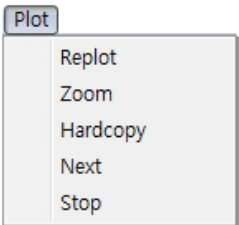
Select-Copy is mainly used to select and then copy the current view.



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.

The dialog box is titled "PLOT NO 1" and contains several sections for configuring the plot:

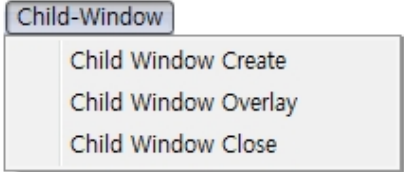
- Titles and Labels:**
 - Title: LAMINATED BEAM
 - Sub Title: AT NODE 34
 - X-Label: APPLIED LOAD (POUNDS)
 - Y-Label: DISPLACEMENT (INCH)
- General Options:**
 - ☒ Framing
 - ☒ Gridding
 - ☒ Centering
 - ☒ Log X
 - ☒ Log Y
- Dimensions and Scales:**
 - Xmax Cm: 3.00
 - Ymax Cm: 5.99
 - Dxlegn Cm: 0.00
 - Xscale: 1.0000
 - Yscale: 1.0000
 - Xdelta: 0.
- Manual Scales:**
 - Xs: 1.0000
 - Xe: 1000.0
 - Nodx: 3
 - Nxdec: -1
 - Ys: 0.1000E-04
 - Ye: 0.010000
 - Nody: 3
 - Nydec: 4
- Curve No. 1:**
 - 0: Line Only
 - 1: Solid Line
 - Color: [Red]
- Legend:** Node No = 34
- Buttons:** < > List ☐ Hide Modify XY Edit XY Delete Add
- Footer:** Sample Description ☐ Add as New Plot OK Cancel

Character is used to change sizes of number and text fonts. Default sizes are specified in PLOT-XY setup menu.

The "Character" dialog box shows settings for font sizes:

- Number:**
 - Default Size
 - ☒ 30% Increase
 - 50% Increase
- Text:**
 - Default Size
 - ☒ 30% Increase
 - 50% Increase

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

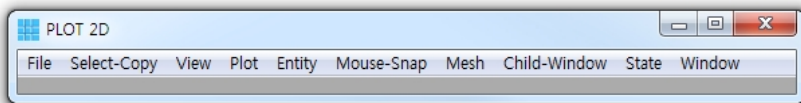
14.1 Introduction

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.

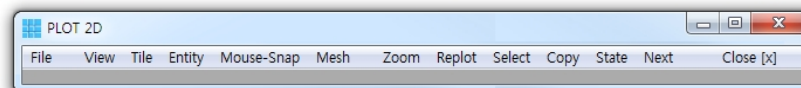
PLOT-2D has two menu styles, General and Express.

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`

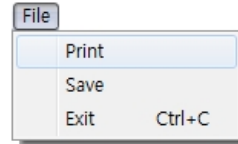


14.2 Menus

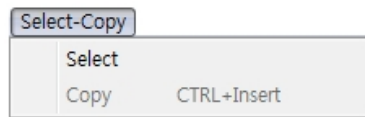
File has three sub menus.

Print is to get the hard copy of the current view.

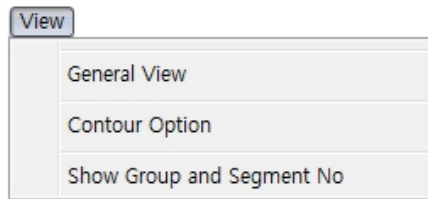
Save is to save the current mesh file. **Exit** is to exit PLOT-2D.



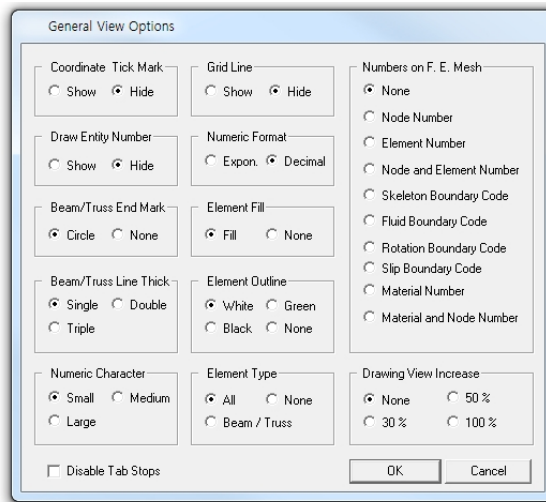
Select-Copy is mainly used to select and then copy the current view.



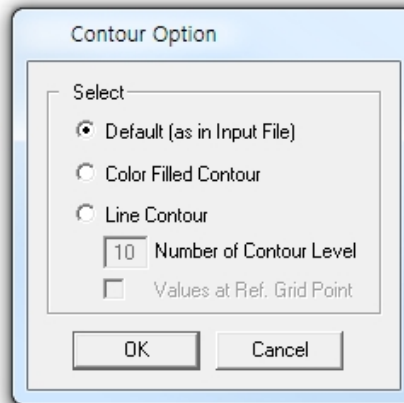
View has three sub menus;
General View, Contour Option,
and Show Group and Segment No.



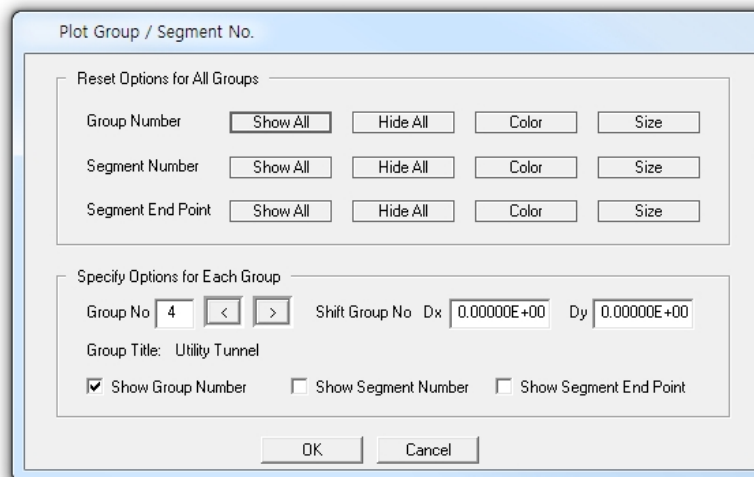
General View options
affect all types of plots.



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.

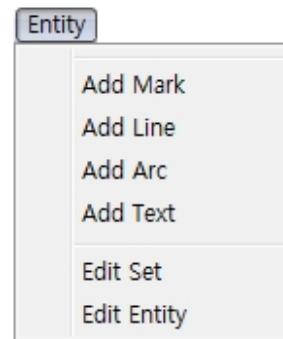
Hardcopy is to print the currently focused window.

Next is to plot the next graph.

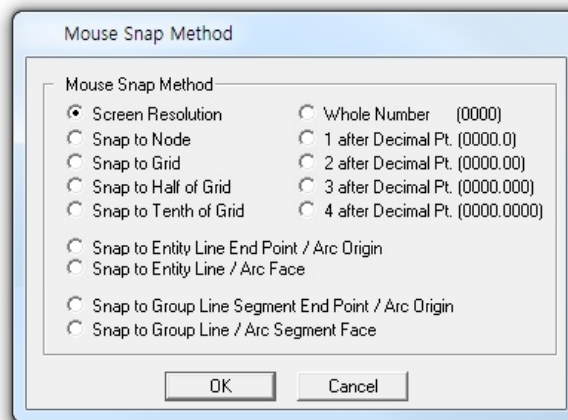
Stop is to stop plotting.



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.

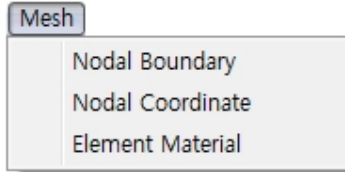


Mouse-Snap is to control the position of mouse cursor when you work for finite element mesh, group mesh, or entities. Mouse Snap Method helps you place the mouse cursor more accurately.



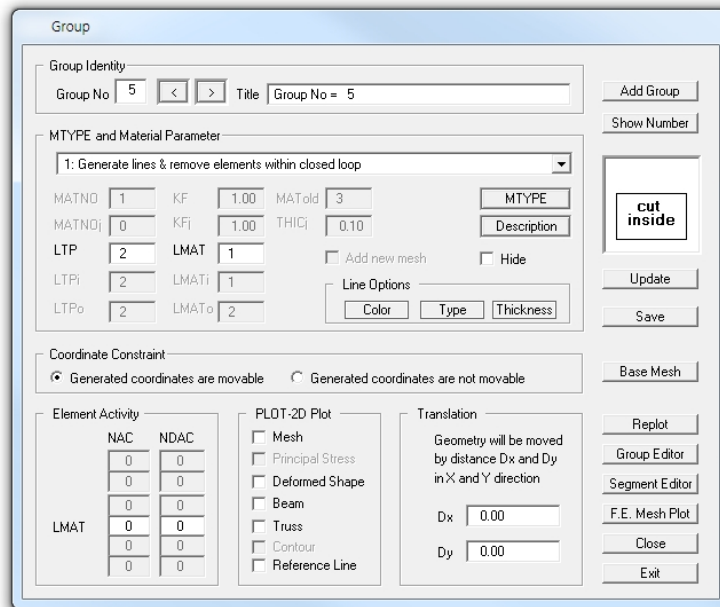
Mesh is used to directly modify finite element meshes. It has three sub menus; Nodal Boundary, Nodal Coordinate, and Element Material.

It is described in detail in Section 5.6 in Group Mesh User's Manual.

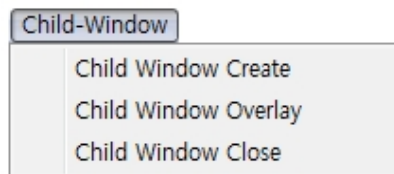


Group is used to build or edit group mesh.

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, temperatures, total heads 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.

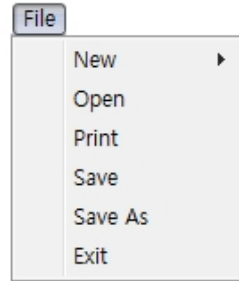
Open is used to open existing mesh file.

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.

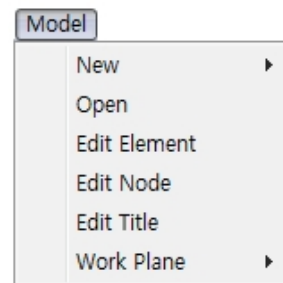
Open is to open existing mesh file.

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.

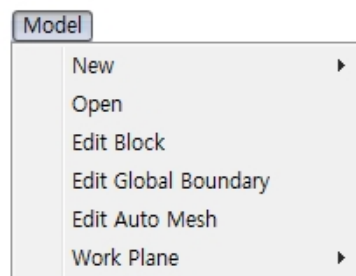
Open is to open existing mesh file.

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.

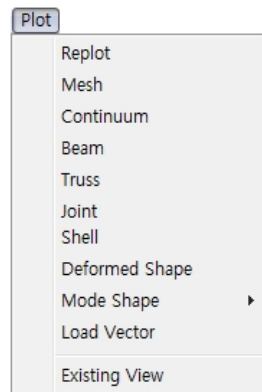
Work Plane is to show prebuilt work planes.



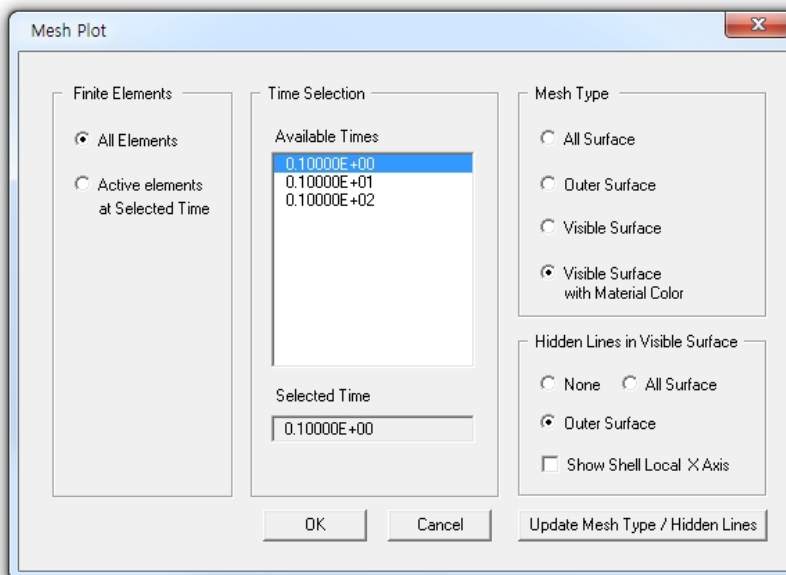
Plot is mainly used to plot Finite Element mesh and analysis results.

It has 11 sub menus; Replot, Mesh, Continuum, Beam, Truss, Joint, Shell, Deformed Shape, Mode Shape, Load Vector, Existing View. Joint plot is not available.

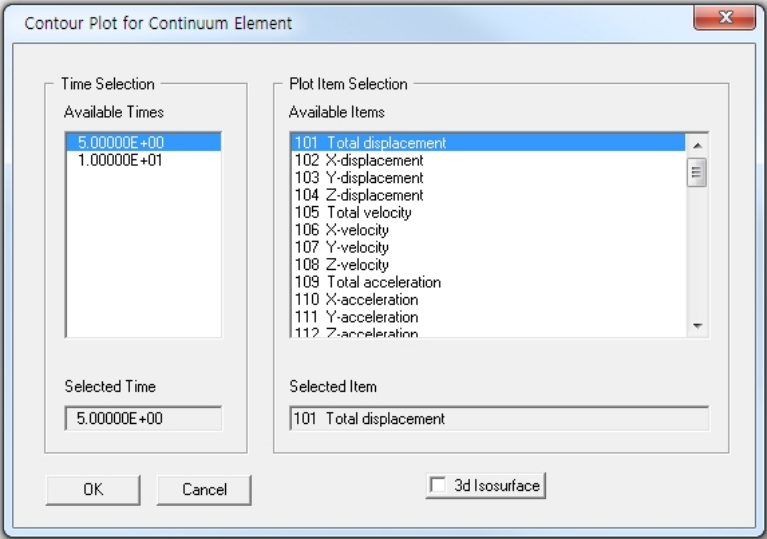
Replot is mainly used to refresh the current view.



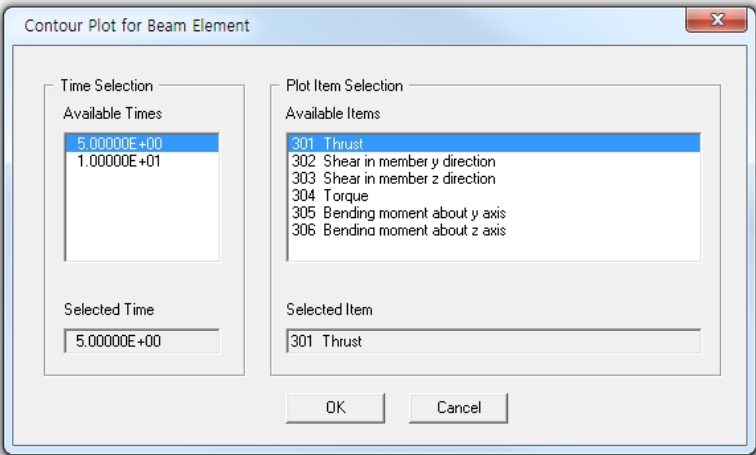
Mesh is to plot Finite Element meshes (Default plot type). Mesh plot requires only Mesh File.



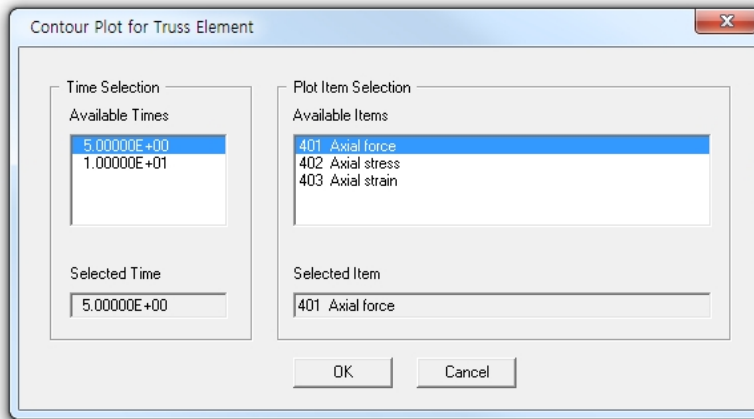
Continuum is to plot contours or principal stress vectors for continuum elements. By checking "3d Isosurface", iso surface will be shown.



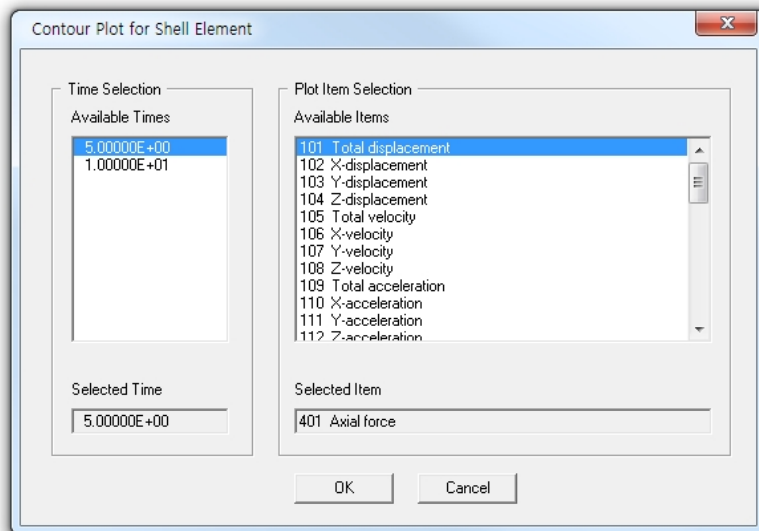
Beam is to plot section forces of beam elements.



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

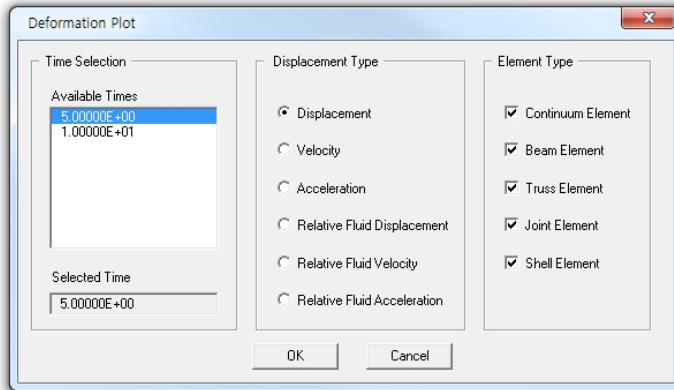


Shell is to plot contours or principal stress vectors for shell elements.



Deformed Shape is to plot the snap shot of all kinds of displacement/velocity/accelerations.

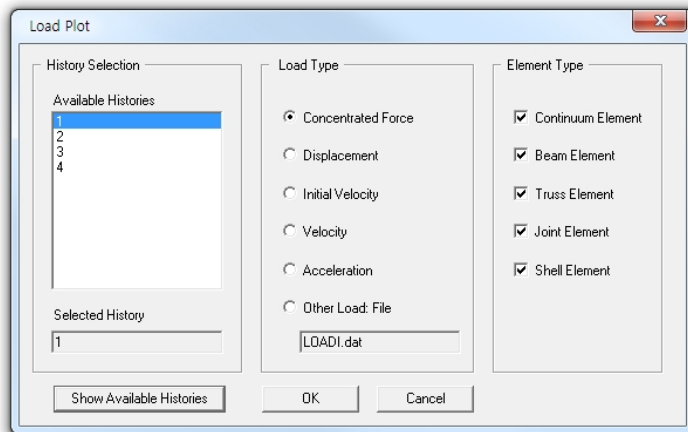
Note that deformed meshes can be combined with other plot types as discussed in "Displacement" option in view menu.



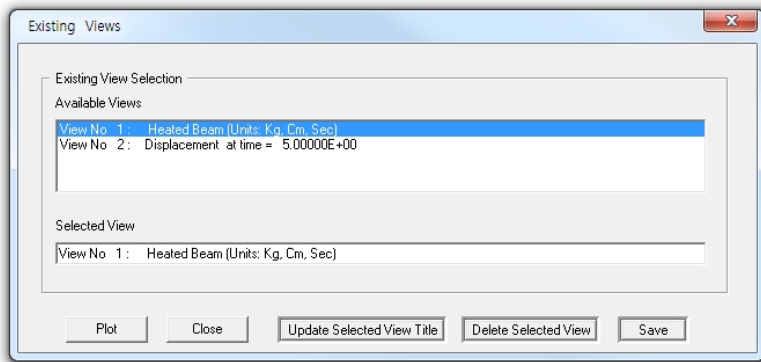
Mode shape is to plot mode shapes of modal analyses for SMAP-2D/3D

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.

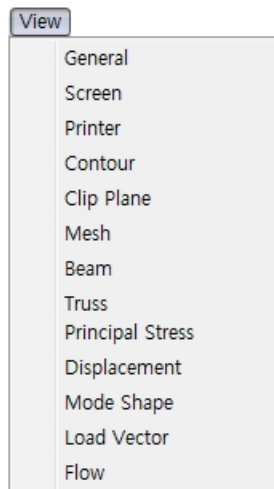


Existing View is to replot the saved views.



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

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



General view options affect most plot types.

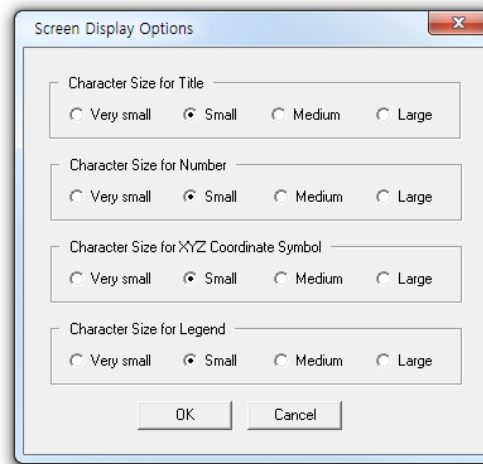


The dialog box titled "General View Options" contains the following settings:

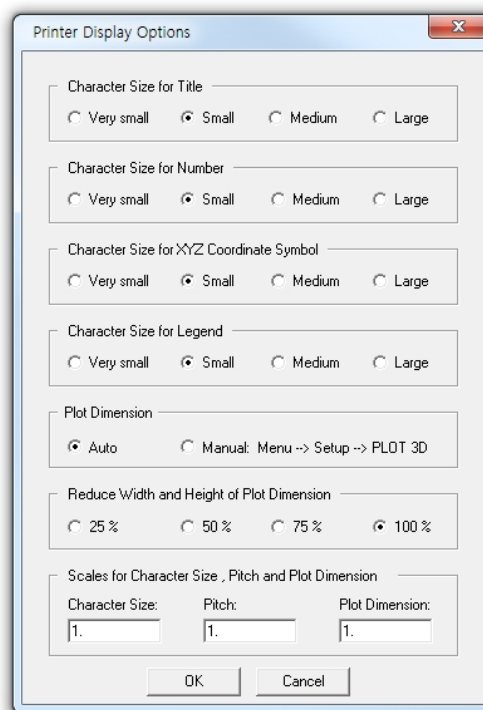
- Legend Number Format:** ☐ Exponential (e) ☒ Decimal Floating (f)
- Continuum Element Outline:** ☐ White ☐ Blue ☐ Red ☐ Grey ☒ Black
- Beam Element Outline / Seepage Phreatic Surface:** ☐ Green ☐ Blue ☒ Red ☐ Grey ☐ Black
- Truss Element Outline / Seepage Water Pressure Line:** ☒ Green ☐ Blue ☐ Red ☐ Grey ☐ Black
- Joint Element Outline / Seepage Total Head Line:** ☐ White ☐ Blue ☐ Red ☐ Grey ☒ Black
- Shell Element Outline / Seepage Flow Line:** ☐ White ☒ Blue ☐ Red ☐ Grey ☐ Black
- Node No:** ☐ Green ☐ Blue ☐ Red ☐ Grey ☒ Black
- Boundary Code:** ☐ Green ☒ Blue ☐ Red ☐ Grey ☐ Black
- Element No. / Material No:** ☐ Green ☐ Blue ☒ Red ☐ Grey ☐ Black
- Index No. / Shell Local X Axis on Element Top Surface:** ☐ Green ☐ Blue ☒ Red ☐ Grey ☐ Black
- Color on Clip Plane:** ☒ Default ☐ Yellow / Red ☐ Blue ☐ Grey / Green
- Show At Right Mouse Button Click:** ☒ None ☐ Element Index ☐ Node ☐ Element
- Show Unreferenced Nodes: Not Connected to Elements:** ☒ None ☐ Mark with Node Number ☐ Mark only
- Numbers & Current Mesh File:** ☒ None
 ☐ Node No ☐ Element No
 ☐ Node & Element No
- Boundary Codes:** ☐ Water Flow ☐ Func His No
 ☐ Initial Head ☐ Func Coeff
- ☐ Material No
 ☐ Material & Node No
 ☐ Data Value ☐ Flow Data
 ☐ X ☐ Y ☐ Z Coordinate
 ☐ Current Mesh File Name
- Show Mid Node & New B. Code:** ☐ Mid Node ☐ New Boundary
- Element Number Range:** Minimum: 1 Maximum: 100000
- Node Number Range:** Minimum: 1 Maximum: 100000
- Mark Nodal Points:** ☒ Shell ☒ Beam ☒ Truss
- Min and Max Values:** ☒ Mark min and max points
 ☐ Add XYZ axes
- Reset All View Options:** ☐ Yes ☒ No

Buttons: OK, Cancel

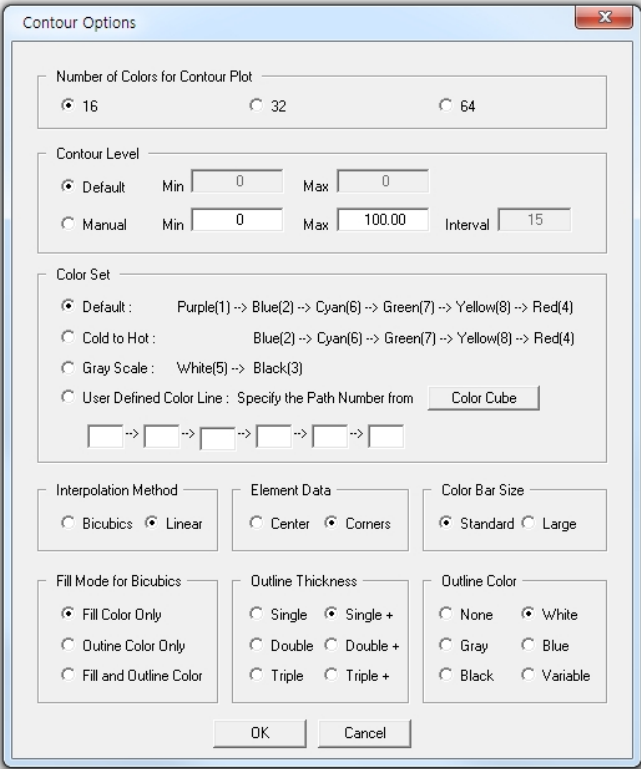
Screen display options affect character sizes shown on the monitor.



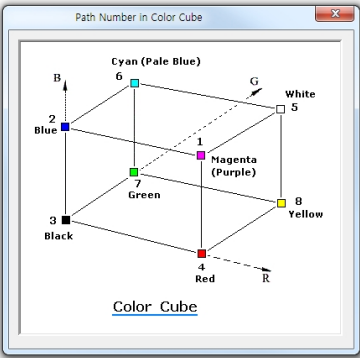
Printer display options affect character sizes and plot dimensions shown on the hard copy.



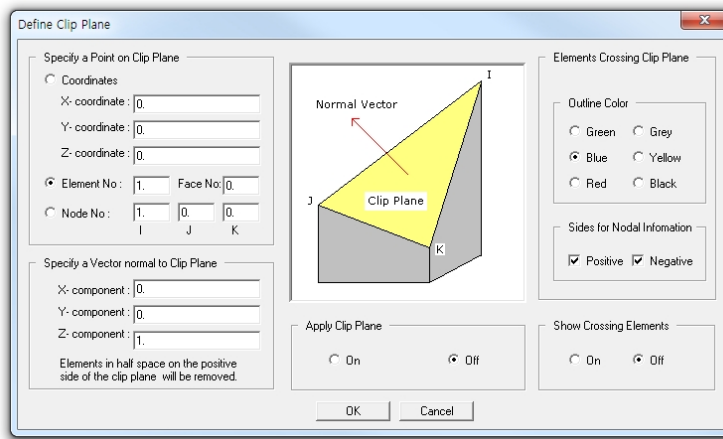
Contour options affect all types of plots involving contours.



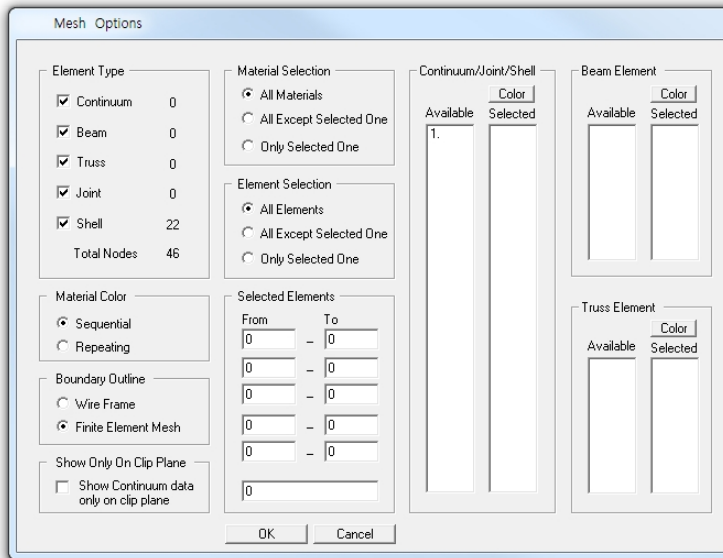
Color cube is to use for user defined color line.



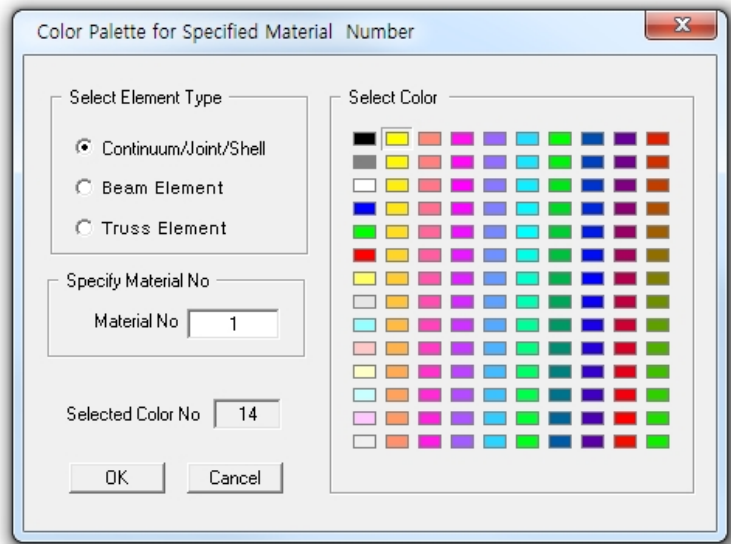
Clip plane defines parameters associated with the clip plane which cuts through the internal part of the 3D domain. When "Apply Clip Plane" is on, contours or deformed shapes are shown on such user defined plane.



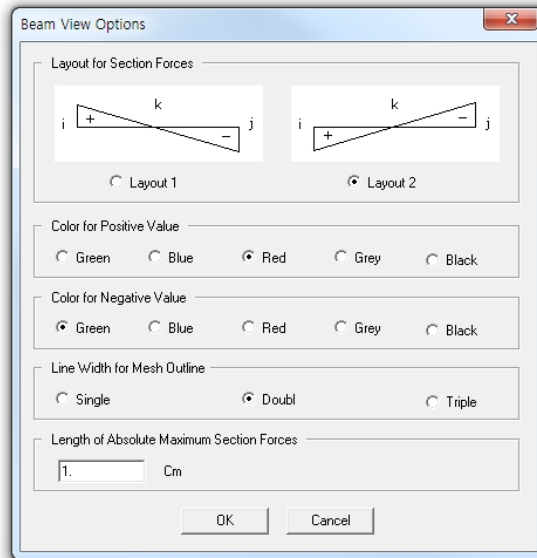
Mesh options affect all plot types. As one of useful features, it can select particular types of elements and materials.



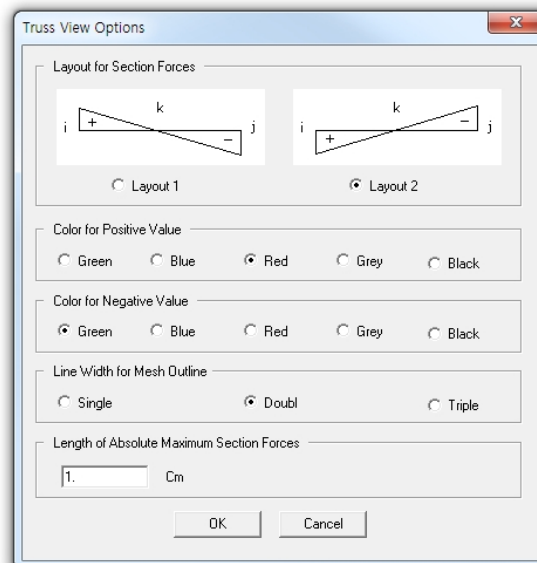
Color is to use for user defined mesh color.



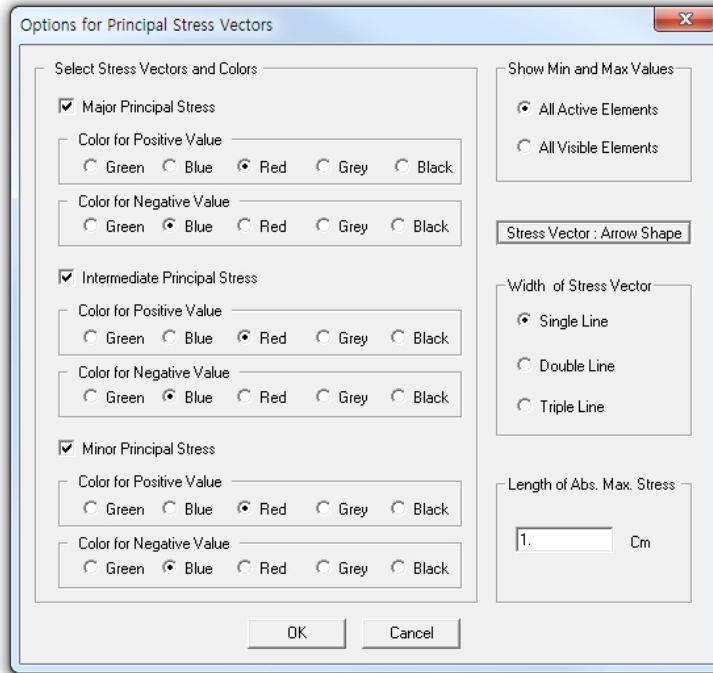
Beam view options
affect only beam plot.



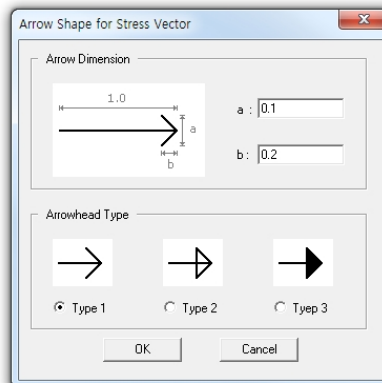
Truss view options
affect only truss plot.



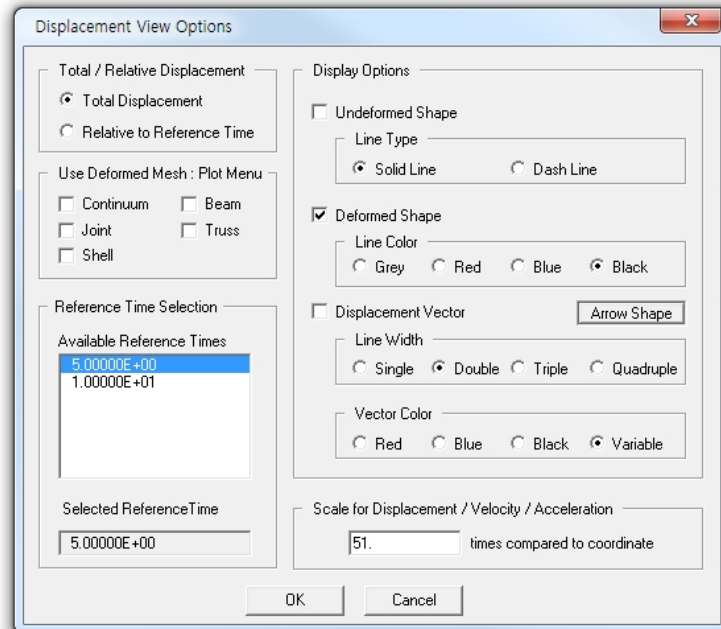
Principal Stress options affect only plots of principal stress vectors in continuum or shell elements.



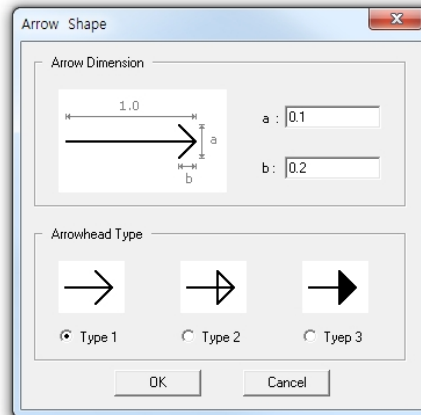
Users can specify the arrow shape for stress vector.



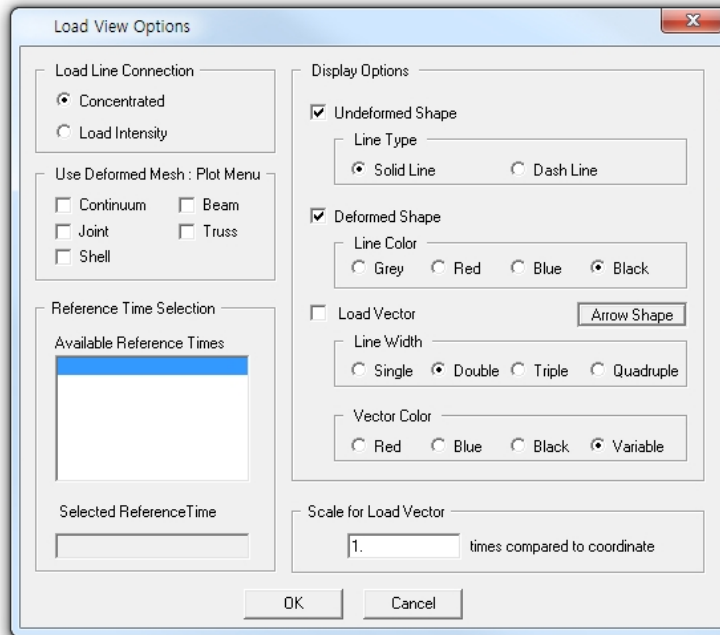
Displacement view options affect only deformed shape plot. Continuum, Beam, Truss, and Shell plots can be displayed over deformed mesh by checking types in "Use Deformed Mesh".



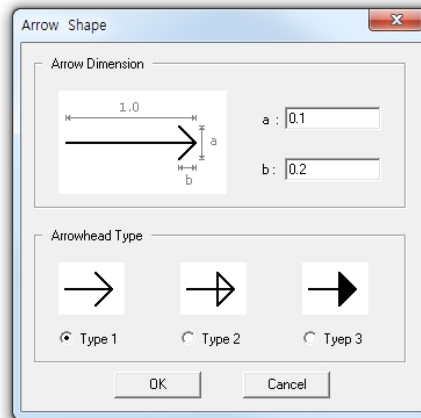
Users can specify the arrow shape for displacement vector.



Load Vector view options affect only load vector plot.
Load vectors can be displayed over deformed mesh by checking "Deformed Shape" in Display Options

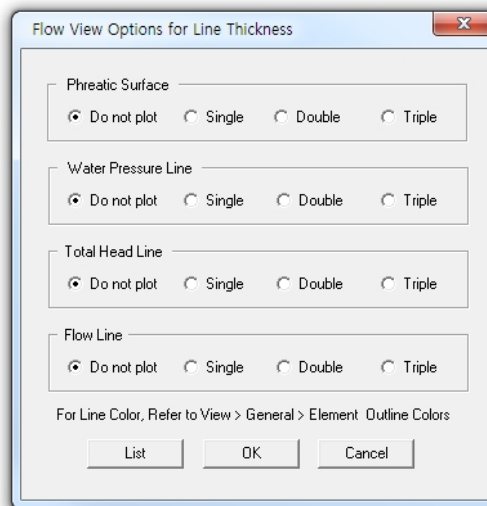


Users can specify the arrow shape for load vector.



Flow view options affect only SMAP-W2 and SMAP-W3 plots. You can control visibility and line thickness of

- Phreatic surface
- Water pressure line
- Total head line
- Flow line



Line colors of flow data can be selected in General view options.



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.



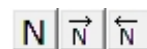
Number Toolbar

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.



16. SWCC

SWCC program supports following three SWCC models:

- van Genuchten (1980)
- Fredlund and Xing (1994)
- Brutsaert (1966) and Gardner (1958)

Graphical output file:

SWCC.Lin

Provides following seven plots as a function of soil suction:

- Volumetric water content
- Normalized volumetric water content
- Specific moisture capacity
- Relative hydraulic conductivity
- Absolute hydraulic conductivity in log scale
- Absolute hydraulic conductivity in arithmetic scale
- Volumetric water content vs absolute hydraulic conductivity

Text output files:

PresfunctionM.dat

Generates following data as a function of soil suction:

- Permeability and specific moisture capacity

PresfunctionV.dat

Generates following data as a function of soil suction:

- Permeability and volumetric water content

PresfunctionA.dat

Generates following data as a function of soil suction head:

- Permeability, volumetric water content and water capacity

Note:

Graphical file SWCC.Lin can be plotted by PLOT-XY.

Text output files can be used as input data for SMAP-F2/W2/W3.

Input Data and Definitions (Main File)	
SWCC Input Data	1.0 TITLE TITLE Title (Max 50 characters)
	2.0 MATFN, MW, GW, KSAT, XSmin, XEmax, DLX MATFN = 11 van Genuchten (1980) = 12 Fredlund and Xing (1994) = 13 Brutsaert (1966) and Gardner (1958) MW Specific moisture capacity at saturation GW Unit weight of water KSAT Saturated permeability XSmin Min suction (Default XSmin = 1.0e-3) XEmax Max suction (Default XEmax = 1.0e+6) DLX Increment of suction in common log scale (Default DLX = 0.02, Min increment = 0.001)

Input Data and Definitions (Main File)	
SWCC Input Data	3.
	For MATFN = 11 or 12
	VWCsat, VWCres, a, n, m, Uswcc, SCFac, Gwm, Pr
	VWCsat Saturated volumetric water content
	VWCres Residual volumetric water content Not used for MATFN = 12
	a, n, m Parameters defining unsaturated hydraulic conductivity and water storage. For MATFN = 11, default $m = 1 - 1/n$ For $n = m = 0$, Simple Gardner (1958) is used.
	Uswcc Unit in fitting SWCC model 0: pressure unit, 1: pressure head unit
	SCFac Conversion factor from user input unit to SWCC model unit in terms of length
	Gwm Unit weight of water in SWCC model used for Uswcc = 0
	Pr Suction at VWCres. Used for MATFN = 12 When $Pr > 0$, assumes that suction in SWCC model is expressed in terms of kPa
	For MATFN = 13
	VWCsat, VWCres, A_1 , n_1 , A_2 , n_2 , SCFac
	A_1 , n_1 Parameters defining unsaturated hydraulic conductivity, Brutsaert (1966)
	A_2 , n_2 Parameters defining unsaturated water storage, Gardner (1958)
	SCFac Conversion factor from user input unit to SWCC model unit in terms of length

SMAP-F2

User's Manual

17.1 Introduction

SMAP-F2 is two-dimensional seepage analysis program which is based on FEMWATER by G. T. Yeh and D. S. Ward, 1979.

The main modifications and expansions include: axial symmetry, triangular finite element and soil-water characteristic curves such as Fredlund and Xing (1994) and Brutsaert (1966).

SMAP-F2 is mainly used to verify the basic functions implemented in the program SMAP-W2 and SMAP-W3.

To run SMAP-F2 main-processing program, you need to prepare a Project File which contains Mesh File name, Main File name and Post File name. [Go to Smap W2 > Setup > Seepage Analysis > Select Swap F2 and click OK button.](#)

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

Main File contains all the other data required for the two-dimensional numerical analysis of seepage problems.

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

17.2 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
C:\Example\VP2.Mes
Main File Name
C:\Example\VP2.Man
Post File Name
C:\Example\VP2.Pos

Note: Both Mesh and Post files of SMAP-F2 are the same formats as SMAP-W2. That is, you can use SMAP-W2 Mesh file for SMAP-F2 analysis.

17.3 Mesh File

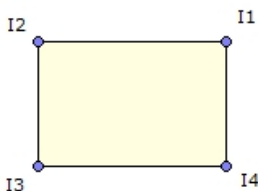
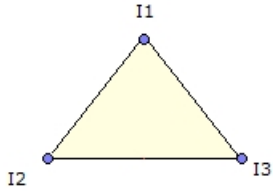
Mesh File contains nodal coordinates, boundary conditions, element indexes and material property numbers. This Mesh File is normally generated by Mesh Generator programs.

To plot Mesh File, select Mesh in Plot menu.

Mesh File

Card Group	Input Data and Definitions (Mesh File)	
1	1.1	TITLE [Character string] TITLEProject title
	1.2	LABEL1 [Character string] LABEL1Label 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 (N.A.) NTRUSS Total number of truss elements (N.A.)

Card Group	Input Data and Definitions (Mesh File)								
2	2.1								
	LABEL2A [Character string] LABEL2B [Character string] LABEL2A Label for coordinate LABEL2B Label for Card 2.2								
Coordinate	2.2								
	NUMNP Cards	[	NODE,	ID,	IDF,	X,	Y,	H,	CF
			-	-	-	-	-	-	-
			-	-	-	-	-	-	-
	NODE		Node number						
	ID = 0		Specified external water flow						
	= 1		Specified total head						
	IDF		Identification number of time dependent function at Card 9.4. Absolute IDF is used.						
			If IDF = 0,						
			external water flow is zero at all times.						
		If ID = 1 and IDF < 0,							
		total head is specified based on hydrograph.							
		Above water table, zero flux is specified.							
	X, Y		X and Y coordinates, respectively.						
	H		Initial total head						
	CF		Coefficient of time dependent function for water flow or total head at the node.						

Card Group	Input Data and Definitions (Mesh File)
3	3.1 LABEL3A [Character string] LABEL3B [Character string] LABEL3A Label for continuum element LABEL3B Label for Card 3.2 3.2 NCONT Cards NEL, I₁, I₂, I₃, I₄, MATNO, IDH - - - - - - - - - - - - - - I2 I1 I3 I4  I1 I2 I3  NEL Element number I₁, I₂, I₃, I₄ Element corner node numbers If I₄ = 0, degenerates to triangular element (N.A.) MATNO Materia number IDH Time dependent function ID number for water generation rate (volume per time) at Card 9.4 If IDH < 0, absolute value of IDH refers to time history ID number for specified pressure head.

17.4 Main File

Mesh File in the previous section 17.3 contains the geometrical data of the structure to be analyzed.

Main File contains all the other data required for the two-dimensional numerical analysis of seepage problems.

Main File consists of seven different card groups:

- Title
- Analysis Parameters
- Computational Parameters
- Continuum Element
- Boundary Conditions
- Material Property Data
- Requested Output

Card Group	Input Data and Definitions (Main File)		
General Information	1	1.0	
		NPROB	
		NPROB	Problem number
	1.1	TITLE	
		TITLE	Title (Max 50 characters)
		1.2	
	IBUG, ICHNG, ITHD		
	IBUG	= 0 Do not print	
		= 1 Print iteration information	
	ICHNG	= 0 Do not print	
	= 1 Print boundary change information		
ITHD	= 0 Pressure head as solution main variable		
	= 1 Total head as solution main variable		

Card Group	Input Data and Definitions (Main File)
2	2.1
Basic Integer Parameters	NNP, NEL, NMAT, NCM, NTI, KSS, KSP
	NNP Number of nodes
	NEL Number of elements
	NMAT Number of materials
	NCM Number of elements with material properties to be modified (N.A.)
	NTI Number of time increments
	KSS = 0 Steady-state solution
	= 1 Transient solution
	KSP = 0 Soil property by analytic function
	= 1 Soil property by tabular data

Card Group	Input Data and Definitions (Main File)	
2	2.2	
Basic Integer Parameters	NSPPM, NSTR, KCP, KGRAV, NSTCL, MAXIT, MAXCY, NMPPM	
	NSPPM	Number of points for soil property data for analytical function or tabular data. Set NSPPM = 9 for KSP = 0
	NSTR = 0	Do not store
	NSTR = 1	Store auxiliary output (N.A.)
	KCP = 0	Soil conductivity as input data
	KCP = 1	Soil permeability as input data (N.A.)
	KGRAV = 0	Gravity is included
	KGRAV = 1	Gravity is not included (N.A.)
	NSTCL	Number of cycles for initial steady-state analysis followed by transient analysis. Used for KSS = 1
	MAXIT	Maximum iterations per time step
	MAXCY	Maximum cycles per rainfall-seepage boundary condition adjustments
	NMPPM	Number of parameters per material model. Set NMPPM = 7

Card Group	Input Data and Definitions (Main File)
2	2.3
Basic Integer Parameters	ILUMP, IMID, IDCYF, IPLAN ILUMP = 0 Do not lump = 1 Lump storage matrix (Recommend) IMID = 0 Backward difference for time derivation = 1 Mid difference for time derivation (N.A.) IDCYF = 0 Darcy flux by Yeh and Wad 1979 = 1 Darcy flux by Reeves and Dugid 1975 IPLAN = 0 Plane geometry = 1 Axisymmetric geometry about Y-axis

Card Group	Input Data and Definitions (Main File)
3	3.1
Basic Real Parameters	DELT, CHNG, DELMAX, TMAX, FE, SIP
	DELT Time increment
	CHNG Multiplier for time increment
	DELMAX Maximum value of DELT
	TMAX Maximum value of simulation time
	FE Angle between coordinate axes and Principal directions of conductivity tensor in degrees
	Solution iteration parameter
	SIP = 0.5 Under relaxation Recommend for fluctuated solution
	= 1.0 Exact relaxation
	= 2.0 Over relaxation

Card Group	Input Data and Definitions (Main File)
3 Basic Real Parameters	3.2 TOLA, TOLB, GW, GRAV, VISC, W TOLA Steady state convergence criteria TOLB Transient state convergence criteria GW Unit weight of water GRAV Acceleration of gravity VISC Dynamic viscosity of water Time derivative weighting W = 0.5 Crank-Nicolson = 1.0 Backward difference (Recommend) Note: Convergence tolerance (TOLA, TOLB) is defined as the maximum difference between new and old iterations. For negative value, the ratio of total head increment to current total head.

Card Group	Input Data and Definitions (Main File)
4	4.1 KPRO, NKPR KPRO = 0 Do not print steady state output = 1 Print FLOW, FRATE, TFLOW = 2 Print H along with KPRO = 1 = 3 Print HT along with KPRO = 2 = 4 Print TH along with KPRO = 3 = 5 Print VX and VY along with KPRO = 4 NKPR Number of times to print transient output Use NKPR = 0
	4.2 KDSKO, NDSKD KDSKO Auxiliary output control for steady state. Use KDSKO = 0 NDSKD Number of times to print transient output Use NDSKD = 0

Card Group	Input Data and Definitions (Main File)
5	5.1 For each material ALP, BETAP, POR, SATKx, SATKy, VWCsat, uMODEL ALP Soil compressibility [L^2 / F] BETAP Water compressibility (N.U.) POR Porosity SATKx Conductivity in x-direction SATKy Conductivity in y-direction VWCsat Saturated volumetric water content Material model for KSP = 0 uMODEL = 0 Saturated material = 1 van Genuchten (1984) = 2 Fredlund and Xing (1994) = 3 Gardner (1958) for permeability Brutsaent (1966) for water capacity Note: For description of equations used for unsaturated material, refer to Section 3.1 SWCC Models in SMAP-W2 Theory.

Card Group	Input Data and Definitions (Main File)
6	6.1 For each soil material For uMODEL = 1 or 2 WCR, WCS, n, alpha, m, Pr, uSWCC, SCFac, Gwm WCR Residual volumetric water content Not used for uMODEL = 2 WCS Saturated volumetric water content n, alpha, m SWCC model parameters For n = m = 0, Simple Gardner is used. Pr Suction at WCR. Used for uMODEL = 2 When Pr > 0, assumes that suction in SWCC model is expressed in terms of kPa uSWCC = 0 Pressure unit in SWCC model = 1 Pressure head unit in SWCC model SCFac Conversion factor from user unit to SWCC model unit in terms of length. For example, user unit is feet and SWCC unit is meter, then SCFac = 0.3048 Gwm Unit weight of water in SWCC model. Used for uSWCC = 0. For uMODEL = 3 WCR, WCS, n₁, A₁, n₂, A₂, uSWCC, SCFac, Gwm n₁, A₁ Gardner (1958) permeability parameters n₂, A₂ Brutsaent (1966) capacity parameters

Card Group	Input Data and Definitions (Main File)
7	7.1 IWPF, INTP IWPF = 0 Pressure head functions (Card 7.2) from Main File = 1 Pressure head functions from PresFunction.dat First 7 lines represent headers = 2 Same as IWPF = 1 but CA is computed from TH and input CA is not used. Interpolations between H and (TH, AK, CA) INTP = 0 Linear scales = 1 Semi-logarithmic scales (Arithmetic H) = 2 Logarithmic scales If IWPF > 0 , go to Card group 12
	7.2 For each soil material 1, H ₁ , TH ₁ , AK ₁ , CA ₁ 2, H ₂ , TH ₂ , AK ₂ , CA ₂ - - - - - NSPPM, H _{NSPPM} , TH _{NSPPM} , AK _{NSPPM} , CA _{NSPPM} H Water pressure head TH Volumetric water content at H AK Conductivity at H CA Water capacity [dTH / dH] at H

Soil Parameter in Tabular Form (KSP = 1)

Card Group	Input Data and Definitions (Main File)
12	12.1
Integer Parameters for Steady or Transient State	NBC, NST, NRFPR, NRFPAR, NRSEL, NRSN, NSTF
	NBC Number of specified head (Set NBC = 0). Program computes NBC from Mesh File
	NST Number of specified flux (Set NST = 0). Program computes NST from Mesh File
	NRFPR Number of rainfall profiles
	NRFPAR Number of parameters in each rainfall profiles
	NRSEL Number of rainfall-seepage elements
	NRSN Number of rainfall-seepage nodes
	NSTF Intensity history number for specified flux. Program computes NSTF from Mesh File

Card Group	Input Data and Definitions (Main File)	
13	13.1	
Rainfall Profiles (NRFPR > 0)	For each rainfall profile	
	$T_{1,}$	RF_1
	$T_{2,}$	RF_2
	-	-
	$T_{NRFPR,}$	RF_{NRFPR}
	T	Time
	RF	Rainfall

Card Group	Input Data and Definitions (Main File)
14	14.1
Rainfall Types and Ponding Depth (NRSN > 0)	For each rainfall-seepage node
	$N_{1,}$ $IRTYP_{1,}$ $HCON_1$ $N_{2,}$ $IRTYP_{2,}$ $HCON_2$ - - -
	$N_{NRSN,}$ $IRTYP_{NRSN,}$ $HCON_{NRSN}$
	N Node number IRTYP Rainfall profile number HCON Ponding depth

Card Group	Input Data and Definitions (Main File)
15	15.1
Rainfall - Seepage Surface Elements (NRSEL > 0)	For each rainfall-seepage surface element
	$N_{1,}$ $IS1_{1,}$ $IS2_{1,}$
	$N_{2,}$ $IS1_{2,}$ $IS2_{2,}$
	- - -
	$N_{NRSEL,}$ $IS1_{NRSEL,}$ $IS2_{NRSEL}$
	N Element number IS1 First node of element side IS2 Second node of element side

Card	Input Data and Definitions (Main File)
24 Requested Output	24.1 NTPRNT NTPRNT Number of cycles between output data print
	24.2.1 NHPEL NHPEL Number of elements at which water pressure time histories are requested
	24.2.2 If NHPEL = 0, skip the following Card NEL ₁ NEL ₂ ... NEL _{NHPEL} NEL Element number to be printed
	24.3.1 NHPMT NHPMT Number of nodes at which total head / pressure time histories are requested
	24.3.2 If NHPMT = 0, skip the following Card NODE ₁ NODE ₂ ... NODE _{NHPMT} NODE Node number to be printed
	24.4.1 NTIME NTIME Number of times at which total head / pressure profiles are requested
	24.4.2 If NTIME = 0, skip the following Card TIME ₁ TIME ₂ ... TIME _{NTIME} TIME Times to be printed

Card	Input Data and Definitions (Main File)
24	24.5.1 NPHD NPHD Number of water pressure lines
	24.5.2 If NPHD = 0, skip the following Card WP ₁ WP ₂ ... WP _{NPHD} WP Water pressure to be plotted
	24.6.1 NTHD NTHD Number of equipotential lines For NTHD < 0, PLOT-XY plots without legends
	24.6.2 If NTHD = 0, skip the following Card TH ₁ TH ₂ ... TH _{NTHD} TH Total heads to be plotted
	24.7.1 NFLN ISPL DELV NFLN Number of flow lines For NFLN < 0, PLOT-XY plots without legends ISPL Split quad. element into triangles (0: No, 1: Yes) DELV Vector length to trace flow path Default: one half of smallest element length
	24.7.2 If NFLN = 0, skip the following Card NOD ₁ NOD ₂ ... NOD _{NFLN} NOD Node numbers from which flow line start

Requested PLOT-XY Output File: FLOW.Lin

Card Group	Input Data and Definitions (Main File)	
24	24.8 NFSEC, IBFQ NFSEC Number of flow sections (Max=100) Compute boundary flow IBFQ = 0 Based on surfaces from global solution = 1 Based on nodes from global solution = 2 Based on surfaces from global flow velocity If NFSEC = 0, go to Card Group 11	
	For Each Flow Section	24.8.1 NODES NOD₁ NOD₂ - - - NOD_{NODES} NODES Number of nodes defining a flow section NOD_i Node number

Requested Output File: FlowSection.out

SMAP[®] - W2

Structure Medium Analysis Program

2-D Seepage Analysis

Example Problems

Contents

1. Introduction	1-1
2. Pre-Processing Programs	2-1
3. Main- and Post-Processing Programs	3-1
4. SMAP-W2 Example Problems	
4.1 1-D Transient Linear Consolidation	4-2
4.2 1-D Transient Infiltration	4-13
4.3 Square Dam Under Rapid Drawdown	4-24
4.4 Steady-State Radial Flow Toward Well	4-37
4.5 Transient Seepage Through Generic Levee	4-48
4.6 Steady-State Seepage Through Earth Dam	4-60
5. Group Mesh Example	
5.1 Arch Tunnel	5-2
5.1.1 Part 1: Creating Arch Tunnel	5-5
5.1.2 Part 2: Adding Rock Bolts	5-15
5.1.3 Part 3: Adding Utility Tunnel	5-20
5.2 Finite Element Mesh Modification	
5.2.1 Overview	5-27
5.2.2 Change Top Surface Nodal Coordinates	5-29
5.2.3 Change Top Surface Nodal Boundaries	5-33
5.2.4 Change Top Layer Element Materials	5-36
6. Block Mesh Example	
6.1 Single Element	6-2
6.2 Square Earth Dam	6-18
6.2.1 Part 1: Creating Square Earth Dam	6-20
6.2.2 Part 2: Modifying Square Earth Dam	6-28
7. PRESMAP Example	
7.1 PRESMAP-2D	7-1
7.1.1 Model 1	7-2
7.1.1.1 Core Region Mesh Generation	7-6
7.1.1.2 Far-Field Region Mesh Generation	7-13
7.1.2 Model 2	7-20
7.1.3 Model 3	7-28
7.1.4 Model 4	7-33

7.2	NATM-2D.	7-36
7.2.1	Model 1 Single Tunnel (Half Section).	7-37
7.2.2	Model 2 Single Tunnel (Full Section)	7-43
7.2.3	Model 3 Two Tunnel (Symmetric Section)	7-46
7.2.4	Model 4 Two Tunnel (Unsymmetric Section).	7-49
7.3	CIRCLE-2D.	7-52
7.4	PRESMAP-GP	
7.4.1	Ex1 2D Line/Surface Blocks	7-57
7.4.2	Ex2 Surface with Corner Triangles	7-74
7.4.3	Ex3 Circular Sector.	7-76
7.4.4	Ex4 Straight Line Sector.	7-78
7.4.5	Ex5 Surface and Line Element (1).	7-80
7.4.6	Ex6 Surface and Line Element (2).	7-82
7.4.7	Ex7 Surface and Line Element (3).	7-84
7.4.8	Ex8 Cement Soil Road	7-86
7.4.9	Ex9 Tunnel in Spherical Geometry.	7-88
7.4.10	Ex10 Horseshoe Tunnel	7-90
7.4.11	Ex11 Wedge Surface Block	7-92
8.	ADDRGN Example	
8.1	ADDRGN-2D	8-1
8.1.1	Combining Meshes	8-2
8.1.2	Modifying Mesh.	8-5
8.1.3	Generating Mesh	8-9
9.	SUPPLEMENT Example	
9.1	XY Example Problem	9-1
9.2	CARDS Example Problem.	9-4
10.	LOAD Example	
10.1	LOAD-2D	10-1
11.	XY Graph Example	
11.1	New Graph	11-2
11.2	SMAP Result	11-9
12.	SWCC Example	
12.	SWCC Example Problem	12-1

Introduction

Example Problems are mainly provided:

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

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

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

Section 4 illustrates SMAP-W2 main example problems as summarized in Table 1.1. These example problems are presented to demonstrate the accuracy and validity of SMAP-W2 main- processing program.

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

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

Section 7 illustrates PRESMA-PGP examples which are used to generate two dimensional Mesh Files.

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

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

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

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

Section 12 illustrates SWCC examples. SWCC plots soil-water characteristic curves for van Genuchten (1980), Fredlund & Xing (1994) and Brutsaert (1966) & Gardner (1958).

Table 1.1 List of SMAP-W2 example problem

Problem Number	Project File Name	Description
1	VP1.dat	1-D Transient Linear Consolidation
2	VP2.dat	1-D Transient Infiltration
3	VP3.dat	Square Dam Under Rapid Drawdown
4	VP4.dat	Steady-State Radial Flow toward Well
5	VP5.dat	Transient Seepage Through Generic Levee
6	VP6.dat	Steady-State Seepage Through Earth Dam

Pre-Processing Programs

Pre-Processing programs are mainly used to generate Mesh File described in Section 4.3 of SMAP-W2 User's Manual. The Mesh File represents the geometry of the structure to be analyzed. This file contains information about nodal coordinates, element indexes, material property numbers, and boundary codes. In SMAP-W2, you may generate such Mesh Files using the following method:

Method

First, generate 2D Mesh File using Group Mesh Generator, Block Mesh Generator, or 2D PRESMA. Then combine or modify these Mesh Files using ADDRGN-2D if you need to do it.

1. Generate 2D Mesh File

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

2. Combine or modify Mesh File

ADDRGN-2D

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

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

Main- and Post-Processing Programs

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

Mesh Files can be generated using Pre-Processing programs as outlined in the previous Section 2. Main and Post Files can be created according to Section 4.4 and 4.5, respectively, in SMAP-W2 User's Manual. Normally, they can copy existing Main or Post Files which are similar to the problem to be analyzed and modify those files using Text Editor.

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

Method 1

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

1. Prepare All Input Files

Mesh, Main and Post Files

2. Get Analysis Results

RUN → SMAP → EXECUTE

3. View Graphical Output

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

Method 2

Prepare Mesh, Main, and Blank Post Files. Run **EXECUTE** menu to get analysis results. Now, prepare Post File according to Section 4.5 in SMAP-W2 User's Manual. Run **PRE EXECUTE** menu to obtain intermediate plotting information files. And then run **PLOT** menu to view graphical output of analysis results. Note that Blank Post File consists of following 3 lines:

```
[ 0
| 0
L 0, 4.5
```

1. Prepare Mesh and Main Files

[Mesh, Main and Blank Post Files](#)

2. Get Analysis Results

[RUN → SMAP → EXECUTE Menu](#)

3. Prepare Post File

[Post File in Section 4.5 of User's Manual](#)

4. Get Plotting Information Files

[RUN → SMAP → PreEXECUTE](#)

5. View Graphical Output

[PLOT → RESULT → PLOT-XY, PLOT-2D, PLOT-3D](#)

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

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

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

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

PLOT-3D does not need any Post File.

This program plots following three dimensional snapshots:

- Finite element mesh
- Darcy flow velocity vector
- Contours of total head, water pressure, and flow rate
- 3D iso surface

SMAP-W2 Example Problem

SMAP-W2 is the main-processing program which computes seepage analysis of two-dimensional problems. Input parameters of SMAP-W2 are described in detail in Section 4 of SMAP-W2 User's Manual.

Running SMAP-W2 is described in Section 3.2.1 of User's Manual and can be selected in the following order:

RUN → SMAP → EXECUTE

Manual procedure to run SMAP-W2 is outlined in Section 3.5 of User's Manual. Once you finished execution of SMAP-W2, you can obtain graphical outputs by selecting:

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

PLOT Menu is described in Section 3.3 of SMAP-W2 User's Manual.

Table 1.1 in Section 1 shows the summary of SMAP-W2 example problems. These example problems are the verification problems. The main objective of these verification problems is to demonstrate the accuracy and validity of SMAP-W2.

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

C:\Smap\SmapW2\Example\Smap

graphical outputs will be presented in this section. References used in example problems are listed at [Sec. 4 in SWAP-W2 Theory](#).

4.1 1-D Transient Linear Consolidation

The first verification problem concerns one-dimensional transient linear consolidation through saturated soil which is the same problem as in Verification Problem 2 in SMAP-2D Example. A soil column shown in Figure 4.1 is assumed that it is liquefied initially and pore water takes all the weight. Thus the excess pore water pressure is dissipated through the top free surface over time. Soil properties used for SMAP-W2 are shown in Figure 4.1.

The exact solution is described in detail in VP2 in SMAP-2D Example.

Table 4.1 shows the listing of main input file VP1.Man. Two different finite element meshes are considered to see the influence of element types. The quad mesh in Figure 4.2 consists of 20 four-node quadrilateral elements and the triangle mesh in Figure 4.3 consists of 40 three-node triangular elements. Boundary codes and initial total heads are shown in Figures 4.4 and 4.5, respectively.

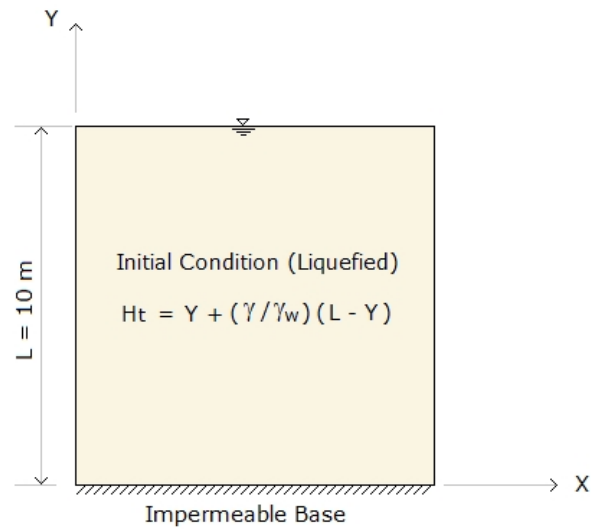
Following are the partial listing of graphical outputs:

Figure 4.6 Water pressure contour at $T = 0.05$

Figure 4.7 Water pressure contour at $T = 0.5$

Figure 4.8 Water pressure profiles at $T = 0.05$ and 0.5

Computed total head profiles are compared with the exact solution by Terzaghi. Results from all cases match very closely with the exact solution.



$$k = 0.001 \text{ m/d} \quad E = 1000 \text{ t/m}^2 \quad \nu = 0.3$$

$$\gamma_w = 1.0 \text{ t/m}^3 \quad n = 0.3 \quad G_s = 2.7$$

$$M = (1 - \nu) E / ((1 + \nu) (1 - 2\nu)) = 1346 \text{ t/m}^2$$

$$M_w = 1 / M = 7.429 \times 10^{-4} \text{ m}^2 / \text{t}$$

$$\gamma = \gamma_w (n + G_s (1 - n)) = 2.19 \text{ t/m}^3$$

$$C_v = k M / \gamma_w = 1.346 \text{ m}^2 / \text{d}$$

$$T = C_v t_m / L^2 = 0.0136 t_m$$

Figure 4.1 1-D transient linear consolidation for VP1

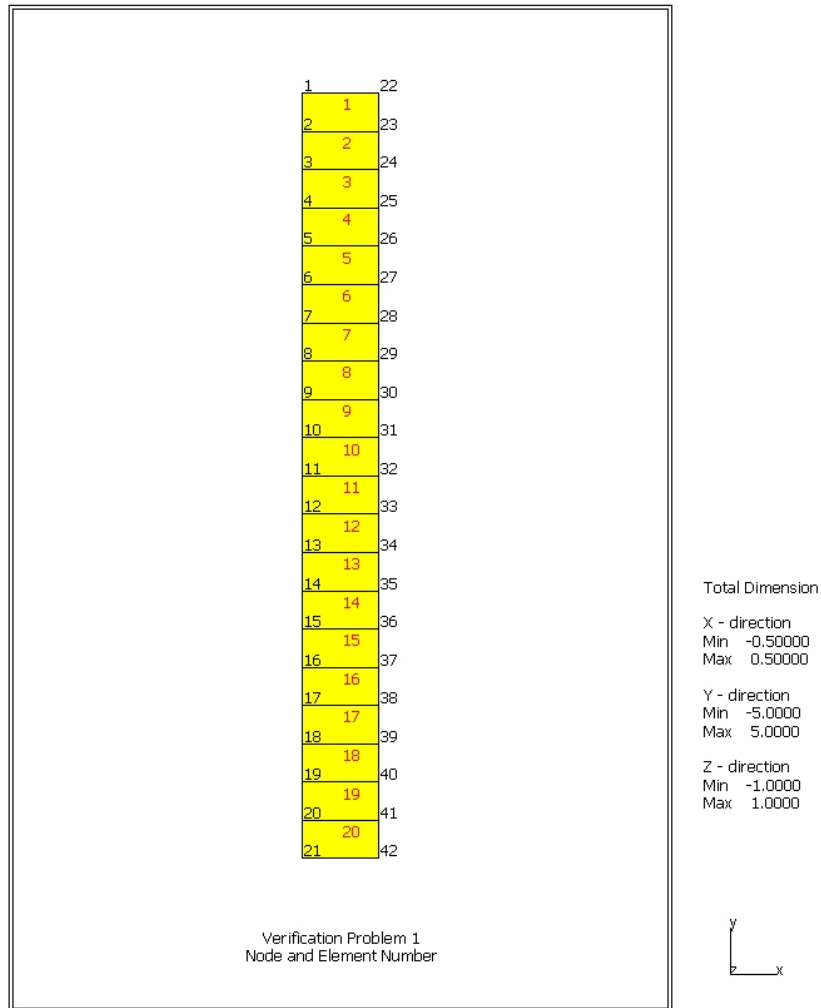


Figure 4.2 Quad finite element mesh

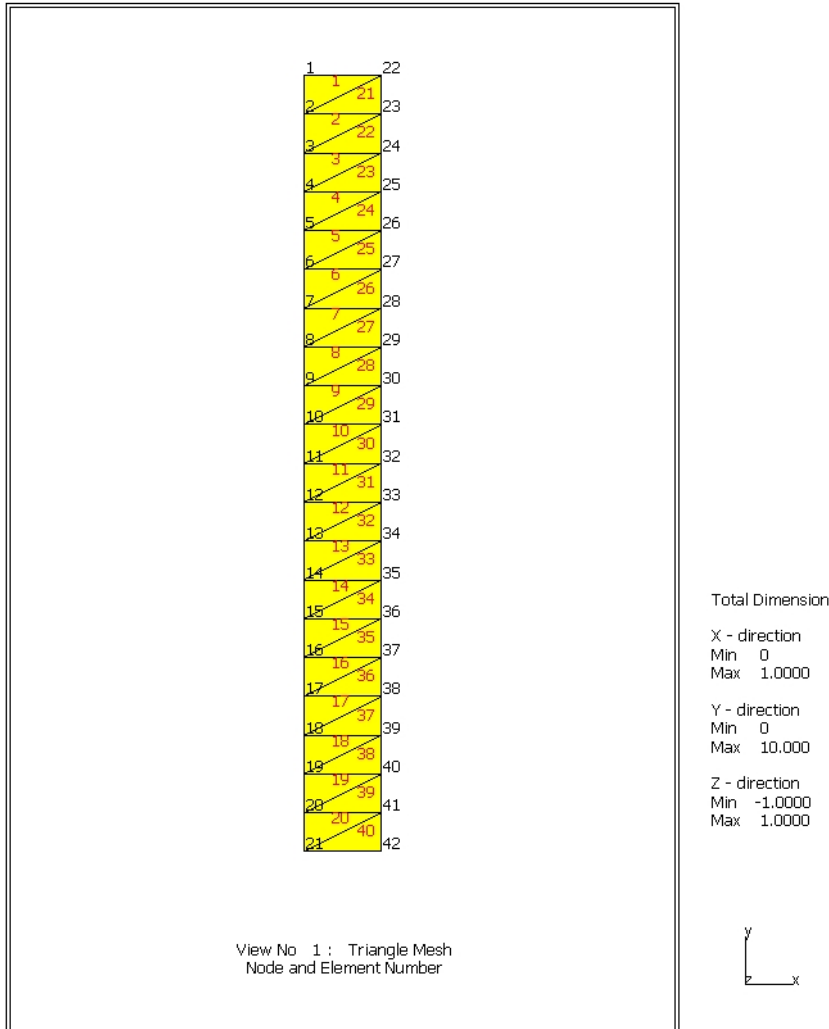


Figure 4.3 Triangle finite element mesh

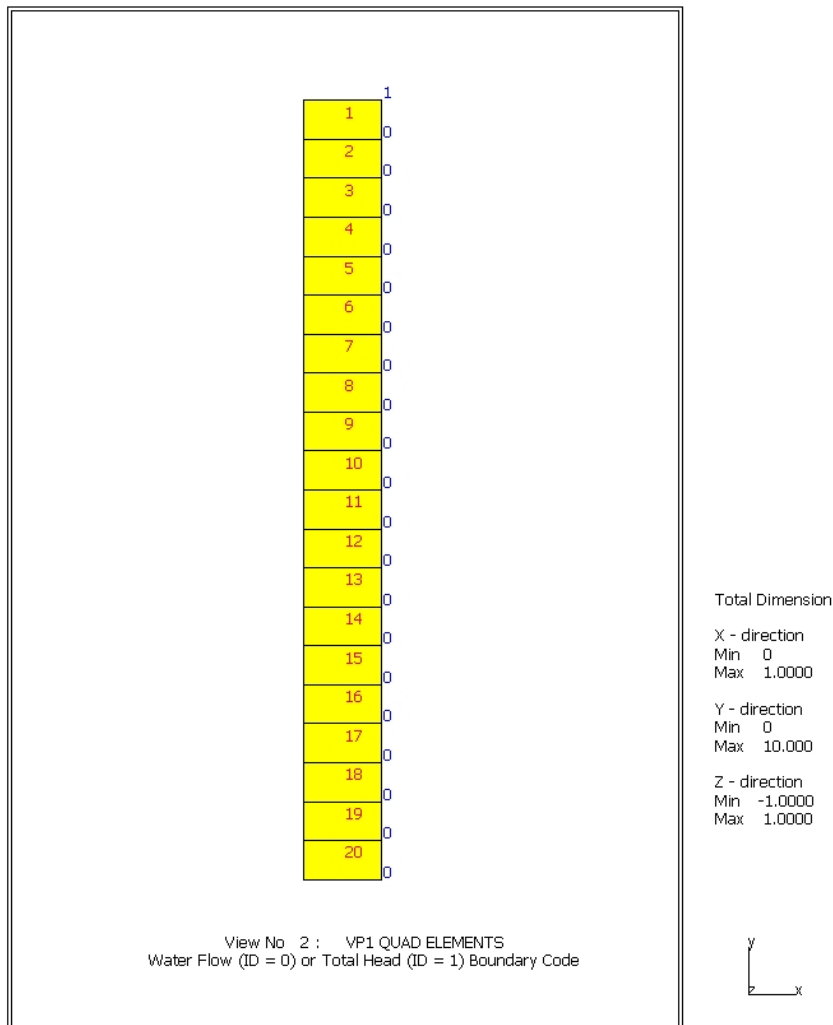


Figure 4.4 Boundary codes

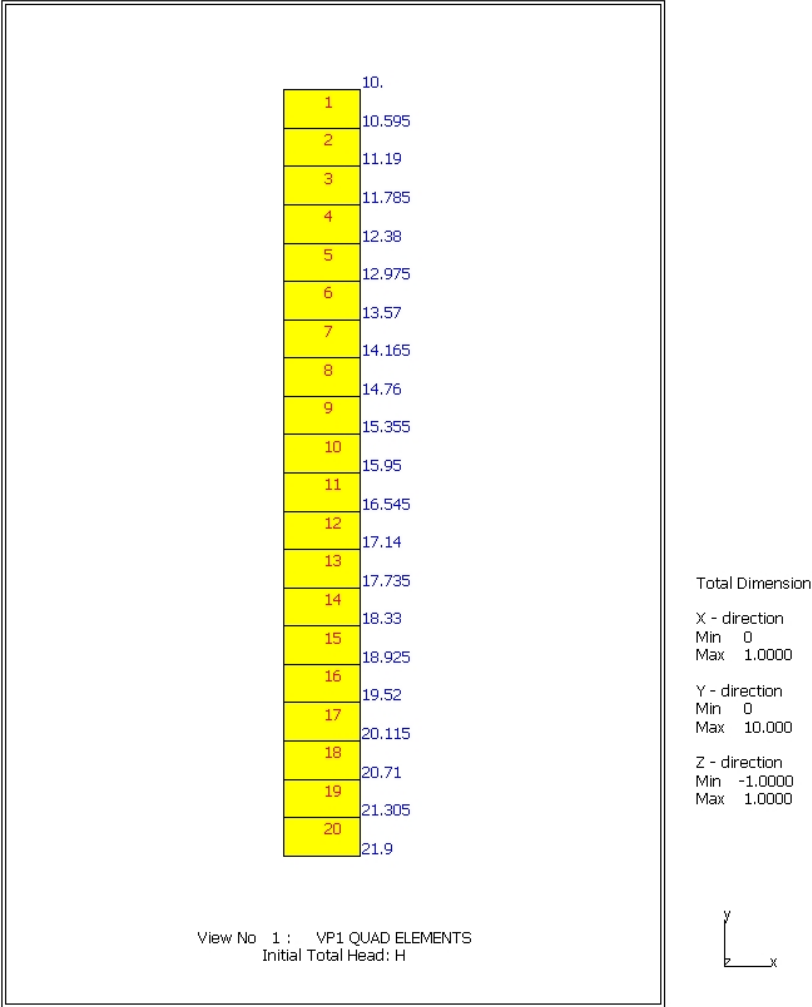


Figure 4.5 Initial total heads

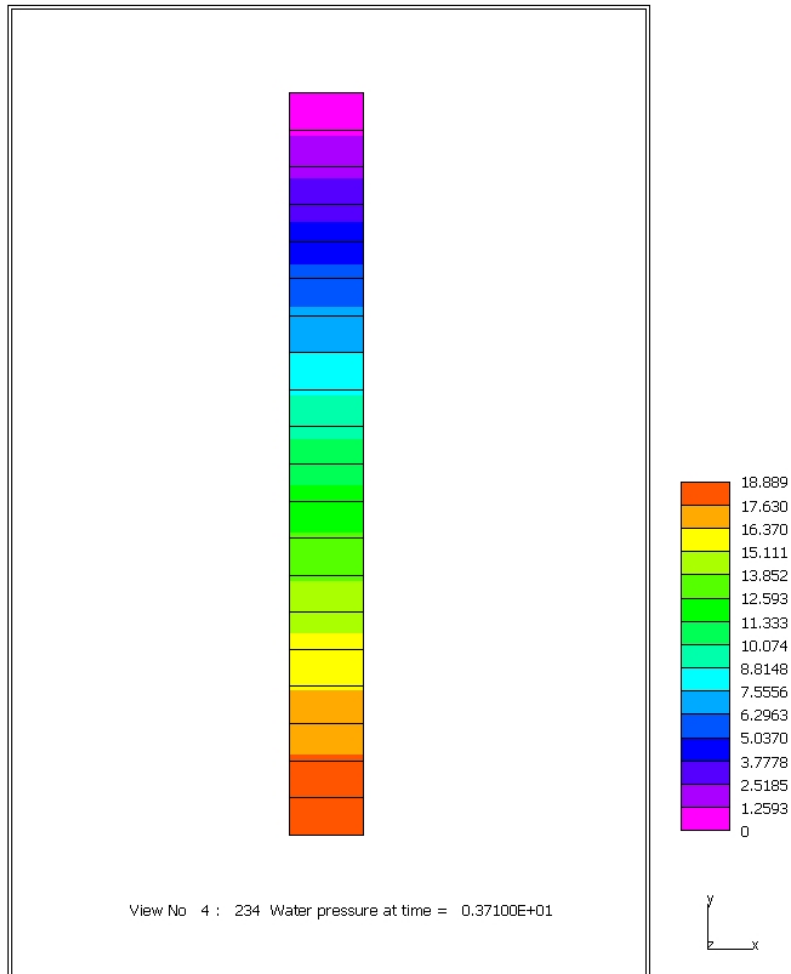


Figure 4.6 Water pressure contour at $T = 0.05$

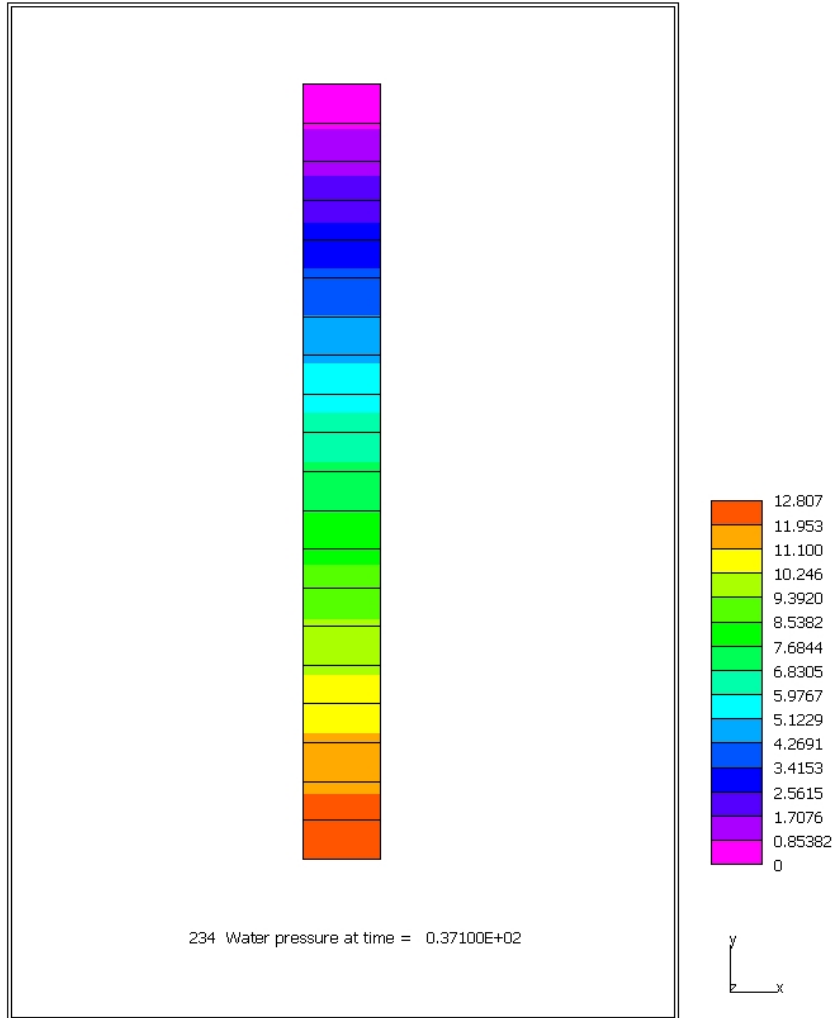


Figure 4.7 Water pressure contour at $T = 0.5$

4-10 SMAP-W2 Example Problem

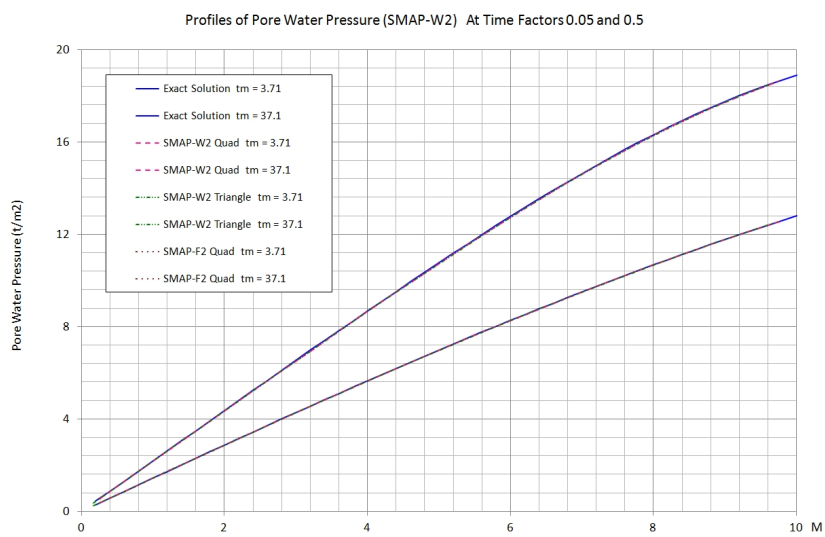


Figure 4.8 Water pressure profiles at $T = 0.05$ and 0.5

Table 4.1 Listing of main input file VP1.Man

```

* CARD 1.1
* TITLE
  Verification Problem 1
* CARD 2.1
* IP  ILUMP  IBATCH  ISST  IMSEEP  IDFVEL  IFVOL  ISFACE
  1    1      -1      1      0        0        1      1
* CARD 3.1
* NCYCL  NSTCL  NITER  MCYCL  TOLER  IRANGE  IRNODE
  3720    0      1      1      0.01    0        0
* CARD 3.2
* SIP
  1.0
* CARD 3.2.1
* IDELT  DT
  0      0.01
* CARD 4.1
* NUMNP
  42
* CARD 4.2
* CMFAC  XREF
  1.0    0.0
* CARD 5.1
* NCONT
  20
* CARD 5.2.1
* NMAT
  1
* CARD 5.2.2.1
* TITLE
  MATERIAL 1
* CARD 5.2.2.2
* MATNO  MATFN  MW(m2/t)  GW(t/m3)  KH(m/d)  KV(m/d)  THETA
  1      0      7.429E-04  1.        0.001   0.001   0.
* CARD 9.4.1
* NTIME  NTIM
  1      2
* CARD 9.4.2
* TIME  FN1
  0.0    10.
  100.   10.

```

4-12 SMAP-W2 Example Problem

```
* CARD 9.5.1
* NWPRF      NWPR      IWPF      INTP
  0          0          0          0
* CARD 10.1
* NTPRNT
  1
* CARD 10.2.1
* NHPEL
  1
* CARD 10.2.2
* NEL1
  20
* CARD 10.3.1
* NHPMT
  1
* CARD 10.3.2
* NODE1
  21
* CARD 10.4.1
* NTIME
  2
* CARD 10.4.2
* TIME1      TIME2
  3.71       37.1
* CARD 10.5.1
* NPHD
  0
* CARD 10.6.1
* NTHD
  0
* CARD 10.7.1
* NFLN      ISPL      DELV
  0          1          0.0
* CARD 10.8
* NFSEC      IBFQ
  0          1
* END OF DATA
```

4.2 1-D Transient Infiltration

The second verification problem concerns a one-dimensional transient infiltration through unsaturated soil. A soil column shown in Figure 4.9, initially unsaturated with uniformly distributed residual water content, is subjected to constant infiltration at the top of soil column while it keeps constant residual water content at the bottom of column. Thus water is flowing down through unsaturated soil. Soil properties used for SMAP-W2 are shown in Figure 4.9.

Both unsaturated volumetric water content and unsaturated hydraulic conductivity are modeled by simple Gardner (1958) equation which is described in detail in Section 3.1.5 in SMAP-W2 Theory.

Table 4.2 shows the listing of main input file VP2.Man. Three different finite element meshes are considered to see the influence of element sizes and element types. The standard mesh in Figure 4.10 consists of 125 four-node quadrilateral elements, triangle mesh in Figure 4.11 consists of 250 three-node triangular elements and double mesh in Figure 4.12 consists of 250 four-node quadrilateral elements.

Following are the partial listing of graphical outputs:

Figure 4.13 Total head profile at time 1 day

Figure 4.14 Total head profile at time 5 day

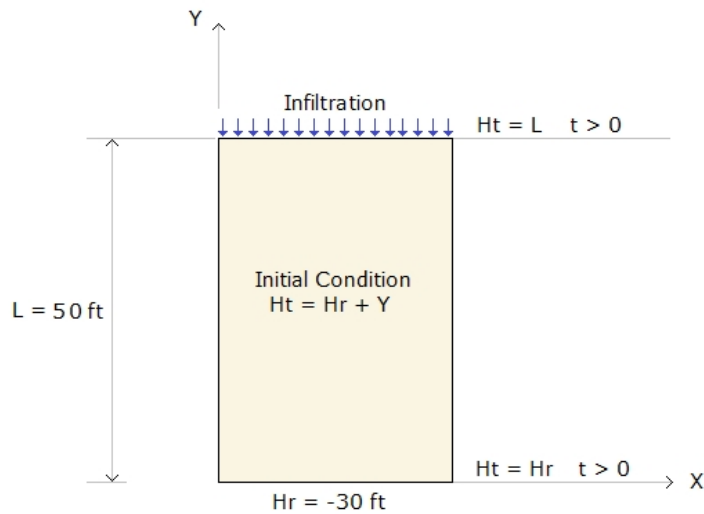
Figure 4.15 Water pressure contour at time 1 day

Figure 4.16 Water pressure contour at time 5 day

Computed total head profiles are compared with the analytic solution by Tracy et al. (2016), ERDC TR-16-8. Results from standard and triangle meshes show slight deviation at time = 1 day but they are very close to analytic solution at time = 5 day. Results from double mesh match very closely with the analytic solution even at early time.

For other analyses with different optional parameters, refer to the file [VP2 Optional Analyses W2.pdf](#).

4-14 SMAP-W2 Example Problem



Saturated Soil Properties :

$$k = 0.1 \text{ ft/d} \quad M_w = 1 \times 10^{-5} \text{ ft}^2/\text{lb}$$

Unsaturated Volumetric Water Content (Gardner, 1958)

$$\theta_s = 0.45 \quad \theta_r = 0.15 \quad \alpha = 0.1 \text{ ft}^{-1}$$

Unsaturated Hydraulic Conductivity (Gardner, 1958)

$$k_s = 0.1 \text{ ft/d} \quad \alpha = 0.1 \text{ ft}^{-1}$$

Figure 4.9 1-D transient infiltration for VP2

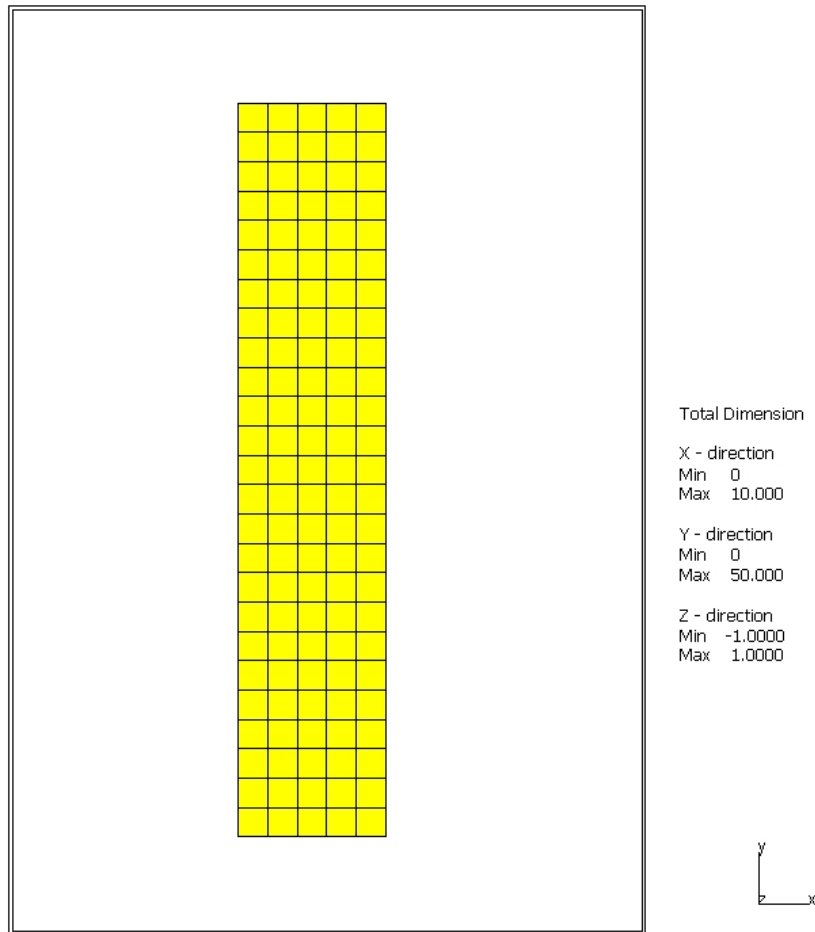
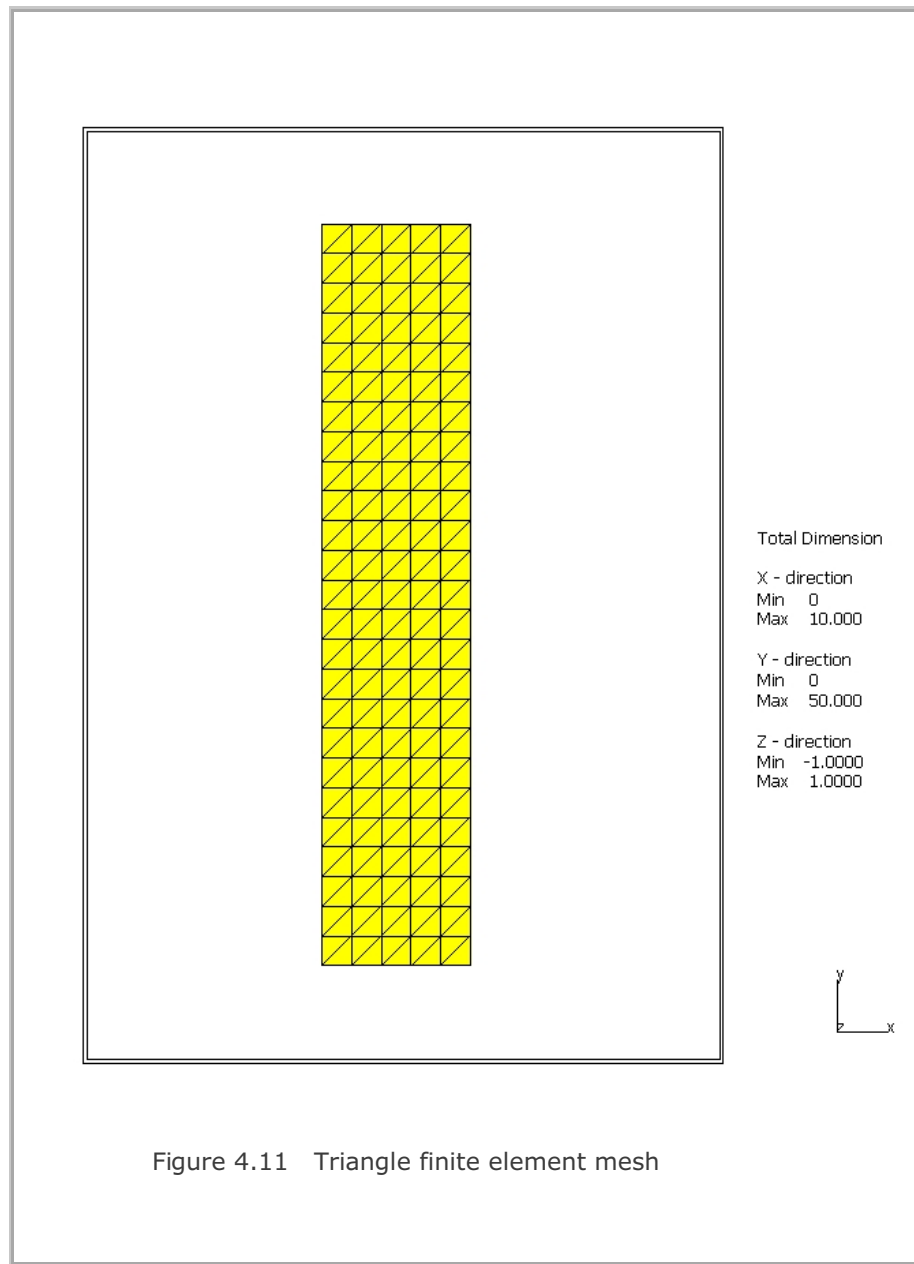


Figure 4.10 Standard finite element mesh



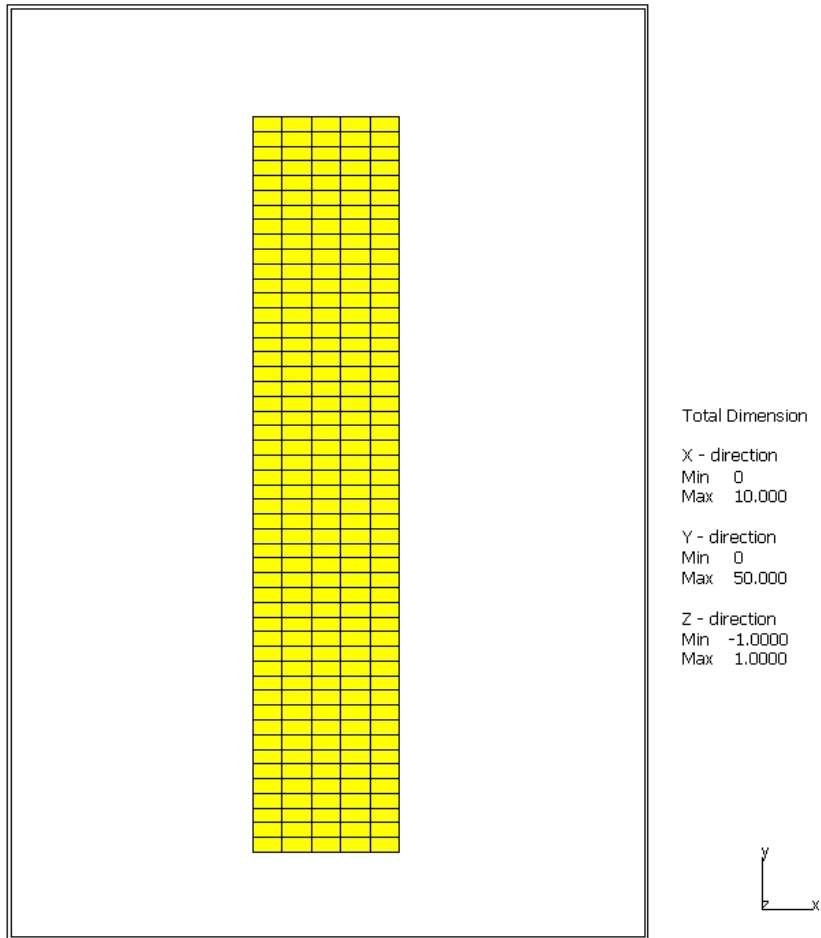


Figure 4.12 Double finite element mesh

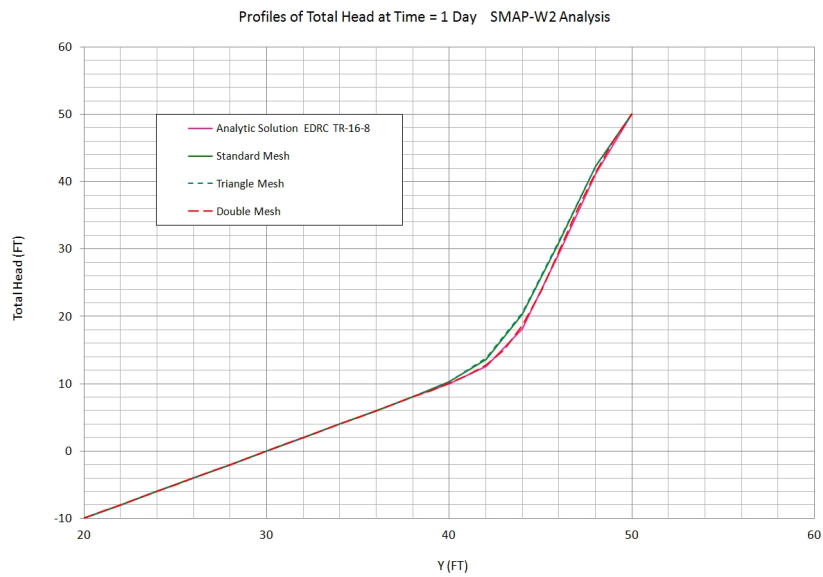


Figure 4.13 Total head profile at time 1 day

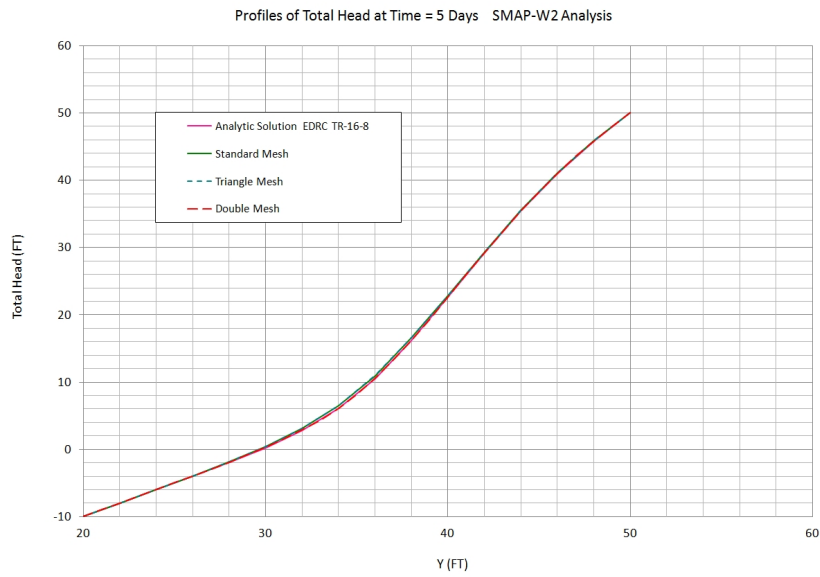


Figure 4.14 Total head profile at time 5 day

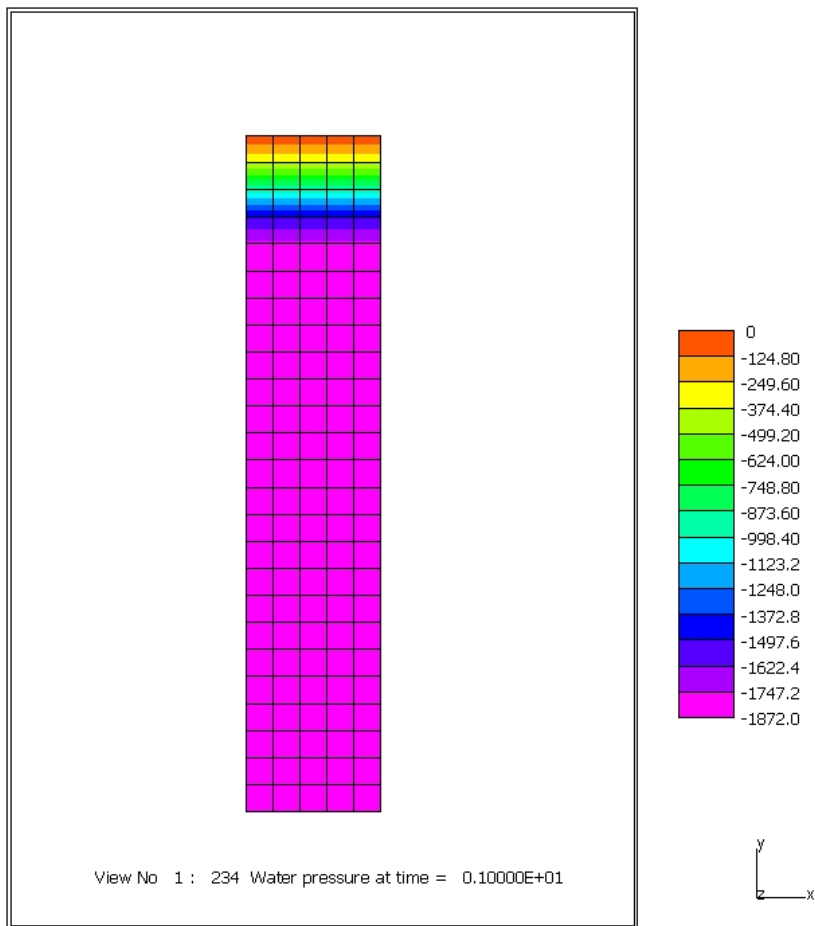


Figure 4.15 Water pressure contour at time 1 day

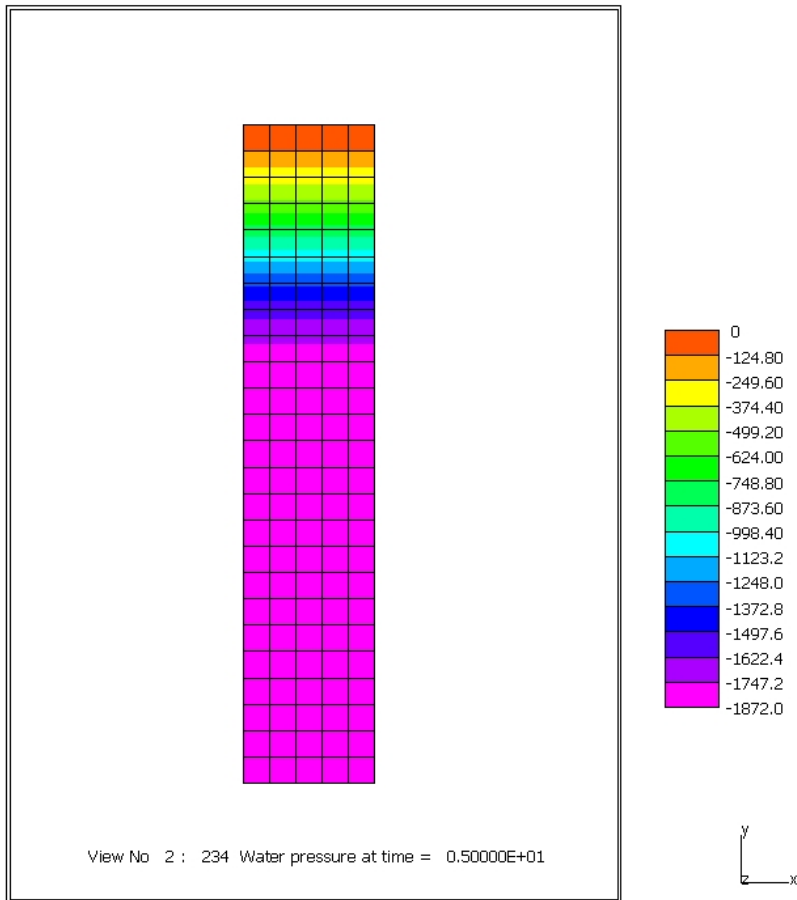


Figure 4.16 Water pressure contour at time 5 day

Table 4.2 Listing of main input file VP2.Man

```

* CARD 1.1
* TITLE
  Verification Problem 2
* CARD 2.1
* IP  ILUMP  IBATCH  ISST  IMSEEP  IDFVEL  IFVOL  ISFACE
  1    1      -1      1      0        0        1      1
* CARD 3.1
* NCYCL  NSTCL  NITER  MCYCL  TOLER  IRANGE  IRNODE
  5000    0      20      1      0.01    0        0
* CARD 3.2
* SIP
  1.0
* CARD 3.2.1
* IDELT  DT
  0      0.001
* CARD 4.1
* NUMNP
  156
* CARD 4.2
* CMFAC  XREF
  1.0    0.0
* CARD 5.1
* NCONT
  125
* CARD 5.2.1
* NMAT
  1
* Simple Gardner Equation (1958) applied for both K and Mw
* To simulate example in Section 6 in ERDC TR-16-8
* CARD 5.2.2.1
* TITLE
  MATERIAL 1
* CARD 5.2.2.2
* MATNO  MATFN  MW(ft2/lb)  GW(lb/ft3)  KH(ft/d)  KV(ft/d)  THETA
  1      11      1.0E-05      62.4        0.1        0.1        0.
* CARD 5.2.2.3
* VWCsat  VWCres  alpha(1/ft)  n  m  Uswcc  SCFac  Gwm  Pr
  0.45    0.15    0.1          0.  0.  1      1.      62.4  0.
* CARD 9.4.1
* NTIMF  NTIM
  2      2

```

```

* CARD 9.4.2
* TIME      FN1      FN2
  0.0      50.      -30.
 1000.     50.      -30.
* CARD 9.5.1
* NWPRF     NWPR      IWPF      INTP
  0         0         0         0
* CARD 10.1
* NTPRNT
  1
* CARD 10.2.1
* NHPEL
  4
* CARD 10.2.2
* NEL1  NEL2  NEL3  NEL4
  2     6    11    16
* CARD 10.3.1
* NHPMT
  4
* CARD 10.3.2
* NOD1  NOD2  NOD3  NOD4
  2     6    11    16
* CARD 10.4.1
* NTIME
  2
* CARD 10.4.2
* TIME1,  TIME2,  ...
  1       5
* CARD 10.5.1
* NPHD
  0
* CARD 10.6.1
* NTHD
  0
* CARD 10.7.1
* NFLN      ISPL      DELV
  0         1         0.0
* CARD 10.8
* NFSEC      IBFQ
  0          1
* END OF DATA

```

4.3 Square Dam Under Rapid Drawdown

The third verification problem concerns a square dam under rapid drawdown of water table. A square dam shown in Figure 4.17, initially fully saturated, is subjected to instantaneous drawdown of left side water table from top surface to base. Saturated and unsaturated properties used for SMAP-W2 are shown in Figure 4.17.

Unsaturated volumetric water content is modeled by Brutsaert (1966) as shown in Figures 4.18 and 4.19. Unsaturated hydraulic conductivity is modeled by Gardner (1958) as shown in Figure 4.20. Both models are described in Section 3.1.4 in SMAP-W2 Theory.

Table 4.3 shows the listing of main input file VP3.Man. 256 equal sized 4-node elements have been used to model the square dam as shown in the finite element mesh in Figure 4.21. The left boundary of the dam represents potential seepage face which is varying during transient state while the right boundary keeps constant total head.

Following are the partial listing of graphical outputs:

Figure 4.22 Water pressure contour at time 1.89 day

Figure 4.23 Water pressure contour at steady state

Figure 4.24 Darcy flow velocity at time 1.89 day

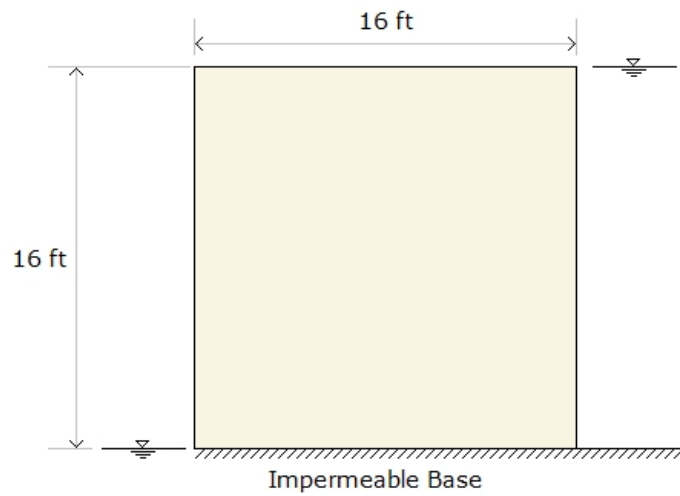
Figure 4.25 Darcy flow velocity at steady state

Figure 4.26 Phreatic surfaces during transient and steady states

Zero water pressure contour represents the phreatic surface which is the boundary between the saturated and unsaturated zones. This surface is moving down with time and remains constant over time. Flow velocity field shows very low velocity in unsaturated zone.

Computed phreatic surfaces are compared with the published studies by France et al. (1971) and Bathe and Khoshgoftaar (1979) in Figure 4.26. During the transient state, SMAP-W2 results are close to their results except the seepage face. However, SMAP-W2 result is very close to their results at steady state.

For other analyses with different optional parameters, refer to the file [VP3 Optional Analyses W2.pdf](#).



Saturated Soil Properties :

$$k = 0.3 \text{ ft/d} \quad n = 0.3 \quad M_w = 3 \times 10^{-5} \text{ ft}^2/\text{lb}$$

Unsaturated Volumetric Water Content (Brutsaert, 1966)

$$\theta_s = 0.3 \quad \theta_r = 0.0 \quad A_1 = 0.05 \quad n_1 = 1.5$$

Unsaturated Hydraulic Conductivity (Gardner, 1958)

$$k_s = 0.3 \text{ ft/d} \quad A_2 = 0.0001 \quad n_2 = 7.0$$

Figure 4.17 Square dam under rapid drawdown for VP3

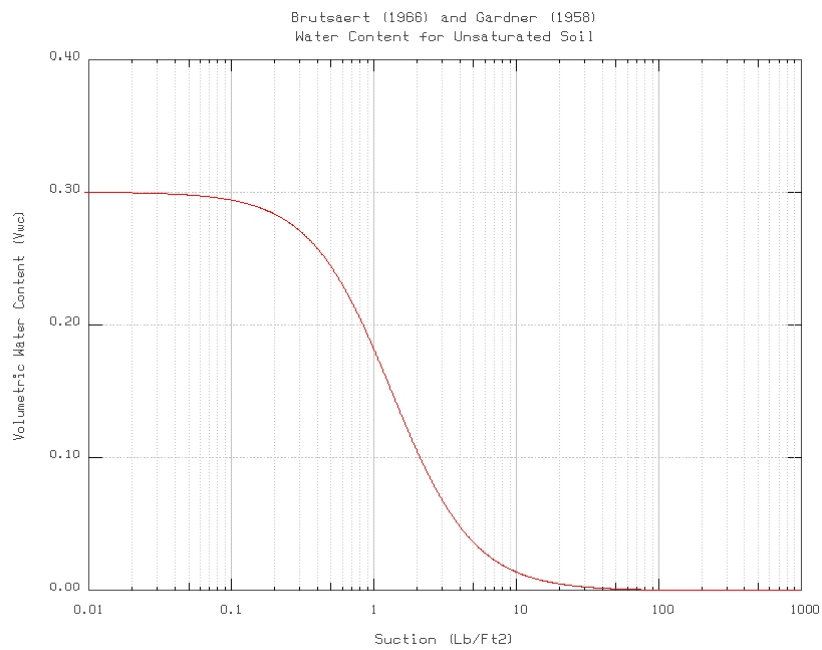


Figure 4.18 Unsaturated volumetric water content

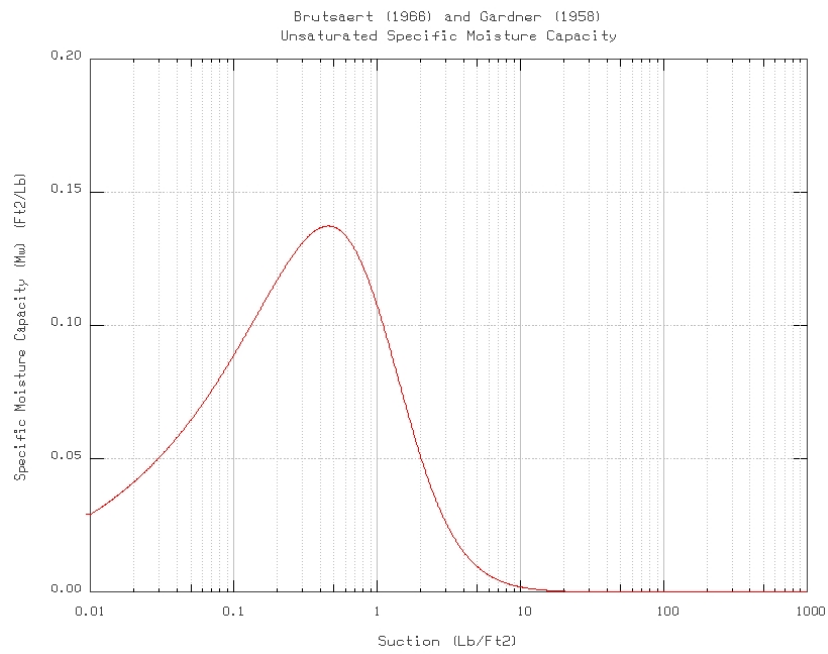


Figure 4.19 Unsaturated specific moisture capacity

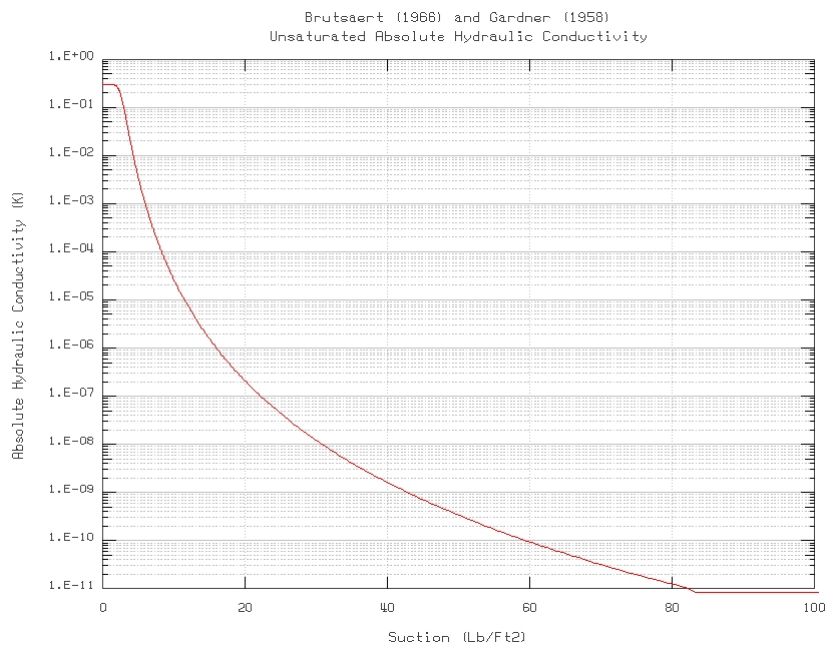


Figure 4.20 Unsaturated hydraulic conductivity

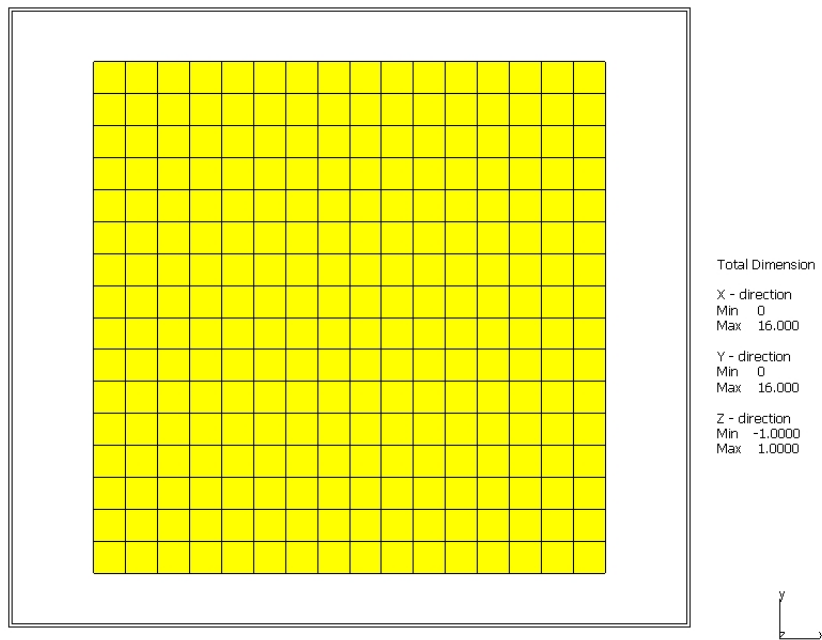


Figure 4.21 Finite element mesh for VP3

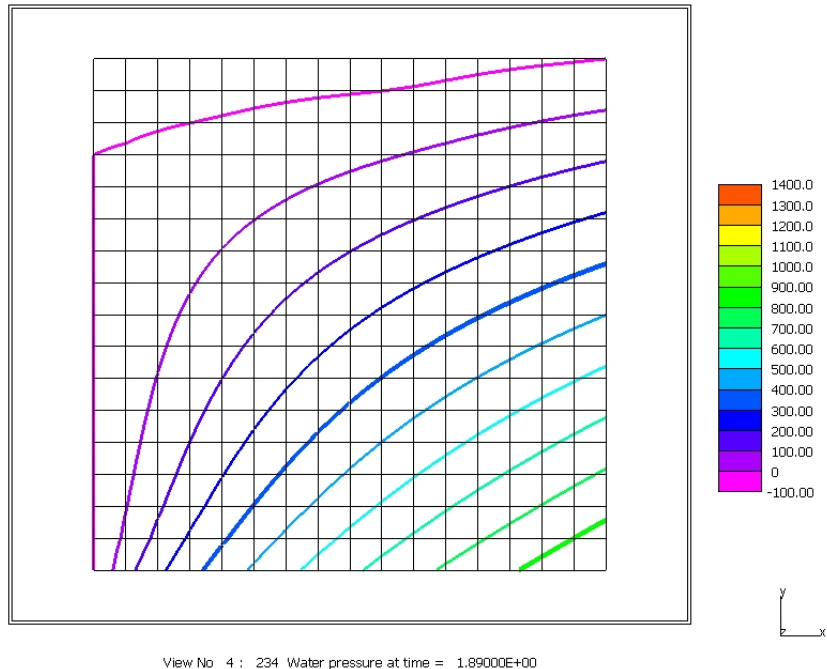


Figure 4.22 Water pressure contour at time 1.89 day

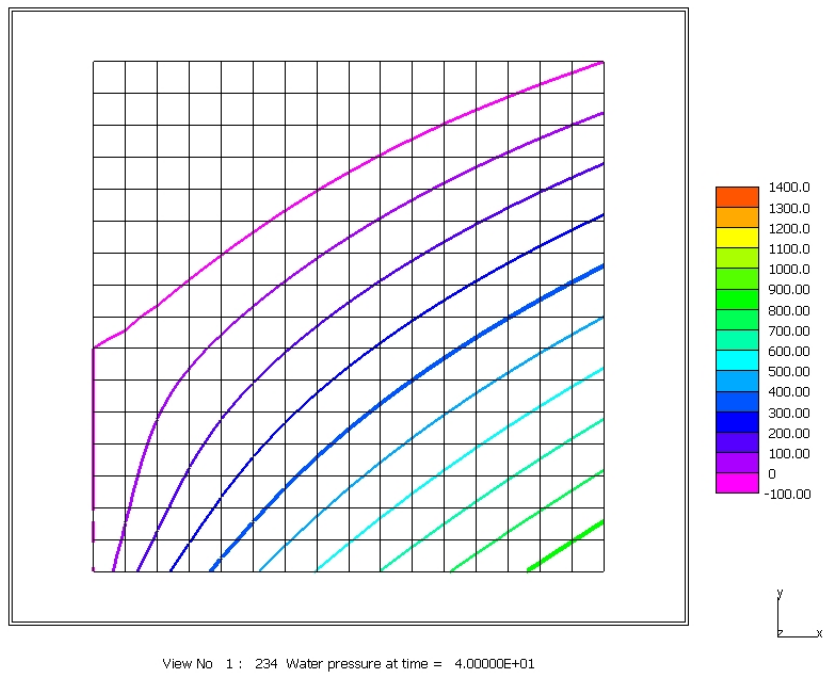


Figure 4.23 Water pressure contour at steady state

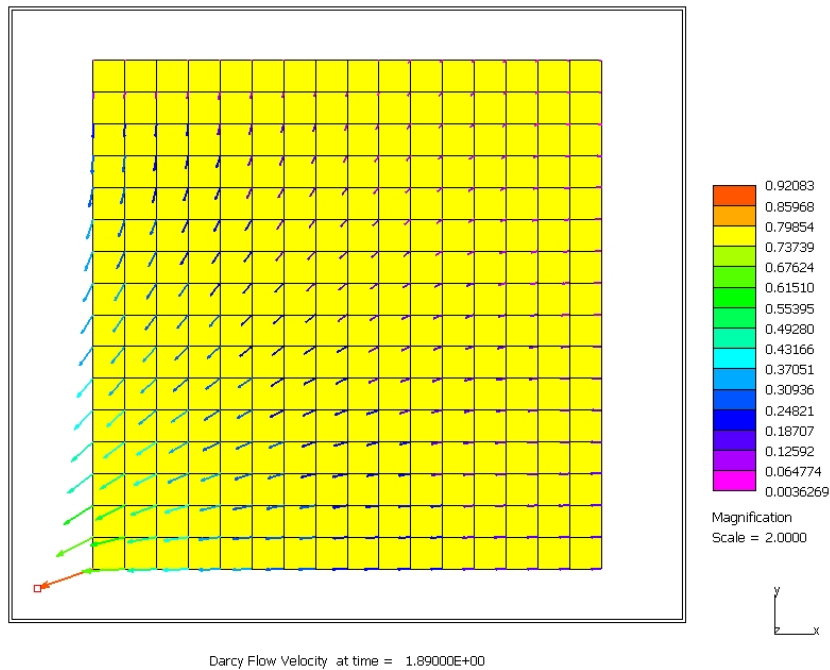


Figure 4.24 Darcy flow velocity at time 1.89 day

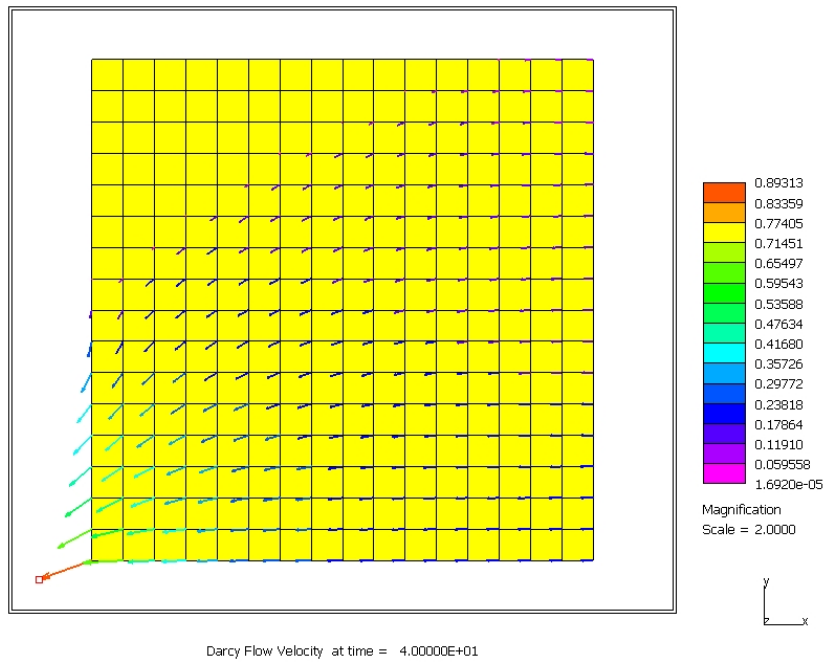


Figure 4.25 Darcy flow velocity at steady state

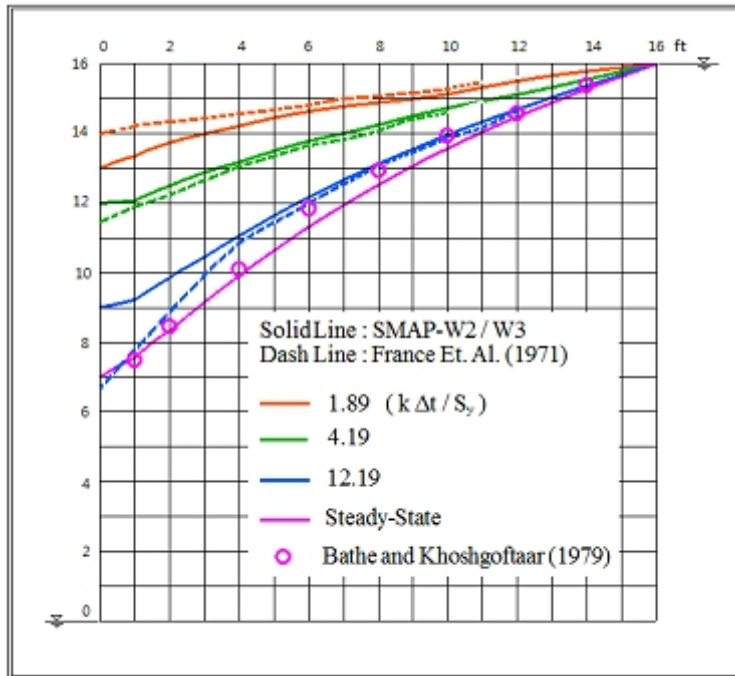


Figure 4.26 Phreatic surfaces during transient and steady states

Table 4.3 Listing of main input file VP3.Man

```

* CARD 1.1
* TITLE
  Verification Problem 3
* CARD 2.1
* IP  ILUMP  IBATCH  ISST      IMSEEP  IDFVEL  IFVOL  ISFACE
  1    1      -1      1        0        0      1      1
* CARD 3.1
* NCYCL  NSTCL  NITER  MCYCL      TOLER  IRANGE  IRNODE
  4000   0      20     10        0.01   0        0
* CARD 3.2
* SIP
  0.5
* CARD 3.2.1
* IDELT  DT
  0      0.01
* CARD 4.1
* NUMNP
  289
* CARD 4.2
* CMFAC  XREF
  1.0    0.0
* CARD 5.1
* NCONT
  256
* CARD 5.2.1
* NTNC
  1
* MATFN = 13
* Gardner (1958) For Unsaturated Hydraulic Conductivity, K
* Brutsaert(1966) For Unsaturated Specific Moisture, Mw
* CARD 5.2.2.1
* TITLE
  MATERIAL 1
* CARD 5.2.2.2
* MATNO  MATFN MW(ft2/lb) GW(lb/ft3) KH(ft/d) KV(ft/d) THETA
  1      13    3.0E-05  62.4      0.3      0.3      0.
* CARD 5.2.2.3
* VWCsat  VWCres  A1      n1      A2      n2      SCFac (ft -> m)
  0.3     0.0     0.05    1.5     0.0001  7.0     0.3048
* CARD 9.4.1
* NTIME  NTIM
  2
* CARD 9.4.2
* TIME   FN1      FN2
  0.0    16.      0.
  1000.  16.      0.

```

4-36 SMAP-W2 Example Problem

```
* CARD 9.5.1
* NWPRF      NWPR      IWPF      INTF
  1          5          0          0
* CARD 9.5.2
* WPRS       K          MW
-2000      0.3      3.0E-05
  0         0.3      3.0E-05
  10        0.3E-03  1.0E-01
  100       0.3E-03  1.0E-01
  2000      0.3E-03  1.0E-02
* CARD 10.1
* NTPRNT
  1
* CARD 10.2.1
* NHPEL
  8
* CARD 10.2.2
* NEL1
  1  2  3  4  5  6  7  8
* CARD 10.3.1
* NHPMT
  11
* CARD 10.3.2
* NODE1
  1  2  3  4  5  6  7  8  9  10  11
* CARD 10.4.1
* NTIME
  6
* CARD 10.4.2
* TIME1,    TIME2,    ...
  1.89      4.19      7.19      12.19      20.      40.
* CARD 10.5.1
* NPHD
  0
* CARD 10.6.1
* NTHD
  0
* CARD 10.7.1
* NFLN      ISPL      DELV
  0          1          0.0
* CARD 10.8
* NFSEC      IBFQ
  0          1
* END OF DATA
```

4.4 Steady-State Radial Flow Toward Well

The fourth verification problem concerns a steady-state radial water flow toward a well. An axially symmetric well shown in Figure 4.27, initially fully saturated, is subjected to drawdown of left side water table from top surface to the some level h_o . This example represents laboratory experimental test conducted by Hall (1955) for standard Ottawa sands. Saturated and unsaturated properties used for SMAP-W2 are shown in Figure 4.27.

Model parameters for unsaturated soil in Figure 4.27 are the same as those used by Taylor and Luthin (1961) where they assumed the same functions for both relative volumetric water content and relative hydraulic conductivity. Here, it simply converted to the model equations for Brutsaert (1966) and Gardner (1958).

Table 4.4 shows the listing of main input file VP4.Man. 800 four-node elements have been used to model this example problem. Figure 4.28 shows the key section dimensions used to generate finite element meshes. [VP4 Block Mesh 2D.pdf](#) illustrates the detailed step-by-step procedures for mesh generation. Generated finite element mesh is shown in Figure 4.29.

Following are the partial listing of graphical outputs:

Figure 4.30 Water pressure contour at steady state

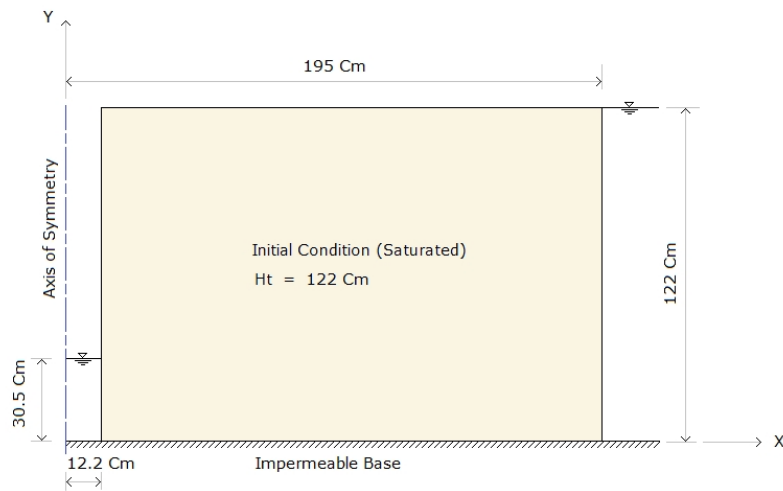
Figure 4.31 Total head contour at steady state

Figure 4.32 Darcy flow velocity at steady state

Figure 4.33 Phreatic surface at steady state

Zero water pressure contour represents the phreatic surface which is the boundary between the saturated and unsaturated zones. As shown in Figure 4.32, this phreatic surface does not represent the flow line when we consider unsaturated soil condition. Flow velocity field shows very low velocity in unsaturated zone.

Computed phreatic surfaces are compared with test result by Hall (1955) in Figure 4.33. SMAP-W2 result is close to test result at steady state. Table 4.5 shows results of computed flows compared with theoretical Dupuit-Thiem equation and Hall's test result.



Saturated Soil Properties :

$$k = 397 \text{ m/d} \quad n = 0.43$$

Unsaturated Volumetric Water Content (Brutsaert, 1966)

$$\theta_s = 0.43 \quad \theta_r = 0.0 \quad A_1 = 0.00189 \quad n_1 = 3.0$$

Unsaturated Hydraulic Conductivity (Gardner, 1958)

$$k_s = 397 \text{ m/d} \quad A_2 = 0.00189 \quad n_2 = 3.0$$

Figure 4.27 Radial flow to axially symmetric well for VP4

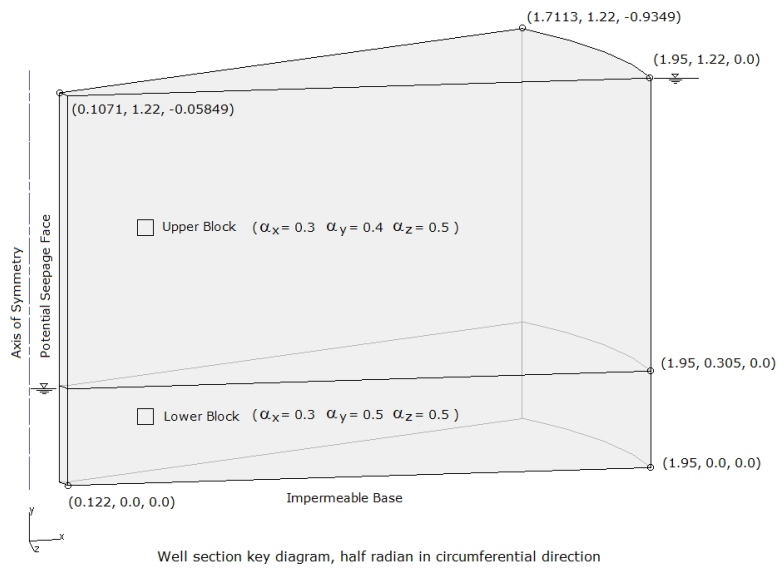


Figure 4.28 Well section key dimensions

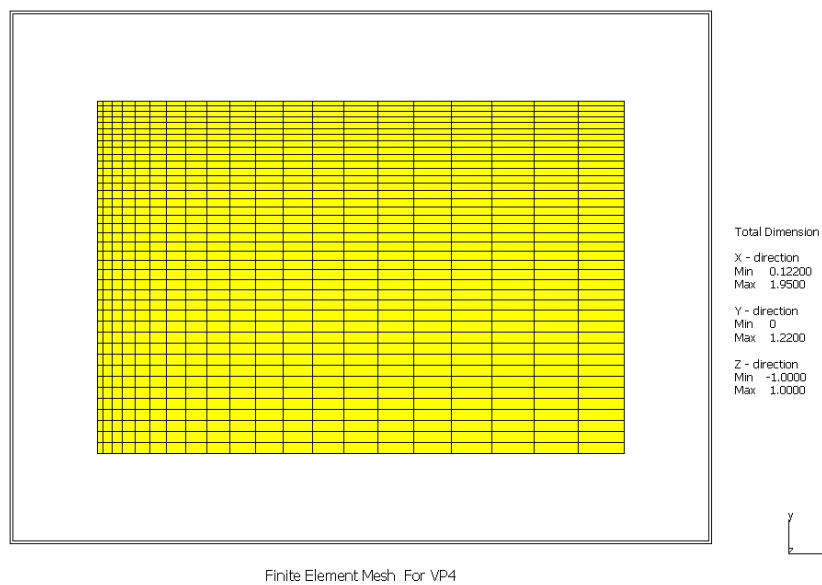


Figure 4.29 Finite element mesh

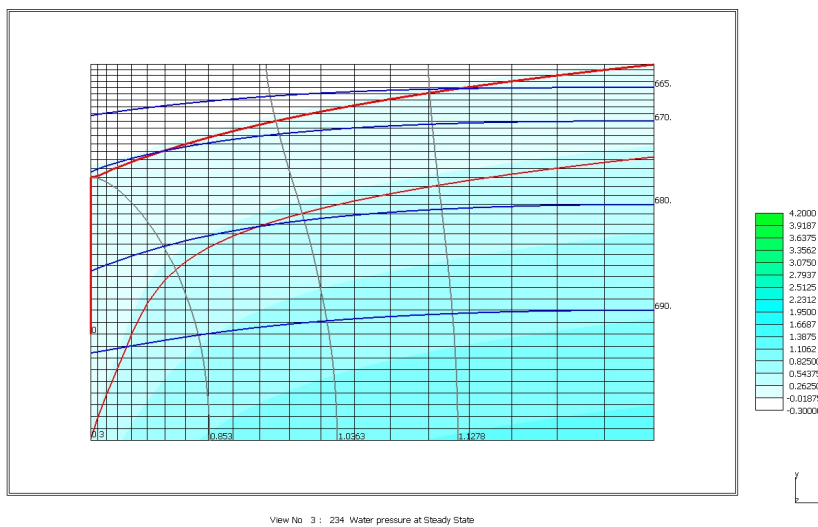


Figure 4.30 Water pressure contour at steady state

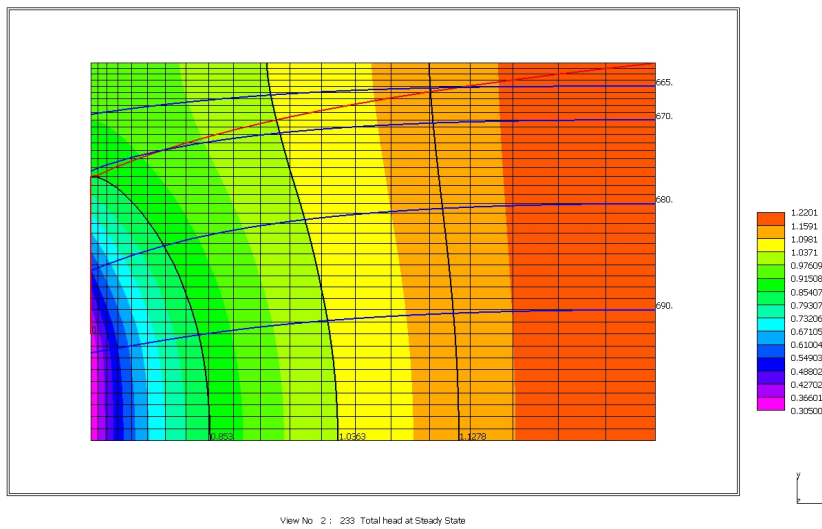


Figure 4.31 Total head contour at steady state

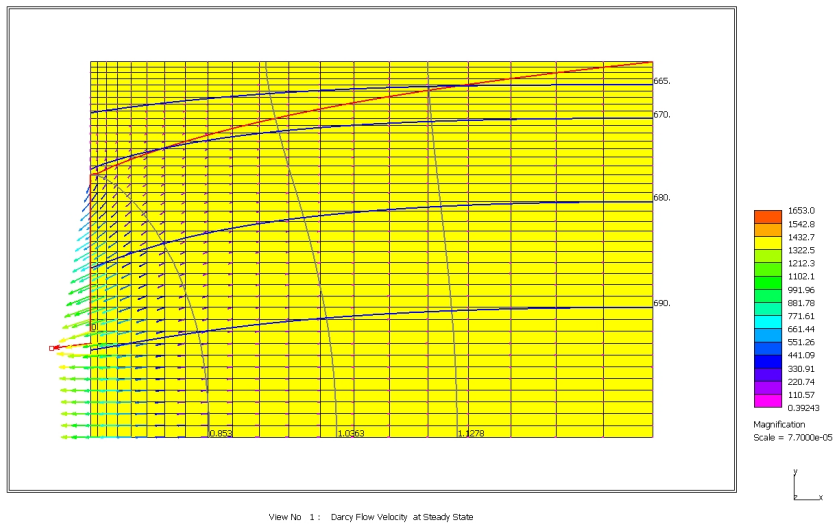


Figure 4.32 Darcy flow velocity at steady state

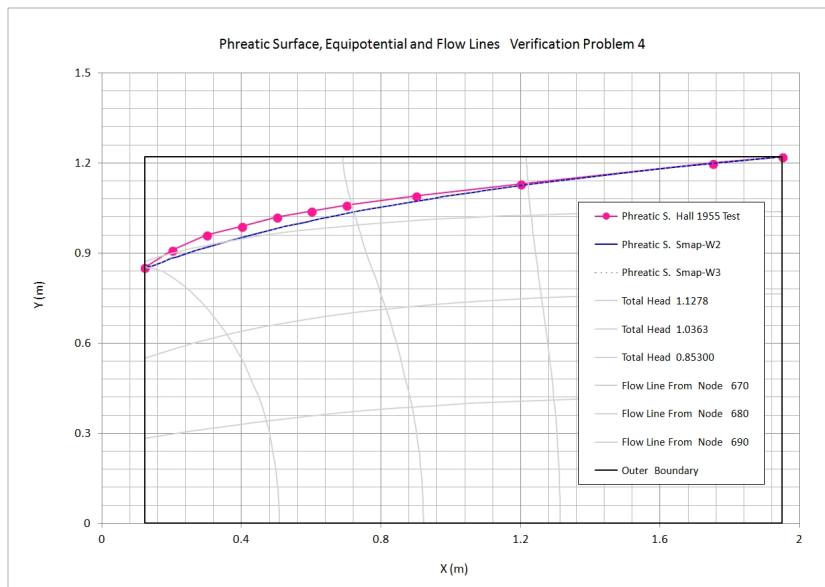


Figure 4.33 Phreatic surface at steady state

Table 4.4 Listing of main input file VP4.Man

```

* CARD 1.1
* TITLE
Verification Problem 4 (Brutsaert & Gardner)
* CARD 2.1
* IP ILUMP IBATCH ISST IMSEEP IDFVEL IFVOL ISFACE
0 1 -1 1 0 0 1 1
* CARD 3.1
* NCYCL NSTCL NITER MCYCL TOLER IRANGE IRNODE
1000 0 20 10 0.01 0 0
* CARD 3.2
* SIP
0.5
* CARD 3.2.1
* IDELT DT
0 0.01
* CARD 4.1
* NUMNP
861
* CARD 4.2
* CMFAC XREF
1.0 0.0
* CARD 5.1
* NCONT
800
* CARD 5.2.1
* NTNC
1
* MATFN = 13
* Gardner (1958) For Unsaturated Hydraulic Conductivity, K
* Brutsaert(1966) For Unsaturated Specific Moisture, Mw
* Model Parameters from Taylor and Luthin (1961)
* Converted to Brutsaert & Gardner
* CARD 5.2.2.1
* TITLE
MATERIAL 1
* CARD 5.2.2.2
* MATNO MATFN MW(m2/t) GW(t/m3) KH(m/d) KV(m/d) THETA
1 13 3.0E-05 1.0 397 397 0.
* CARD 5.2.2.3
* VWCsat VWCres A1 n1 A2 n2 SCFac
0.43 0.0 0.00189 3. 0.00189 3. 1.
* CARD 9.4.1
* NTIME NTIM
2 2
* CARD 9.4.2
* TIME FN1 FN2
0.0 0.305 1.22
1000. 0.305 1.22
* CARD 9.5.1
* NWPRF NWPR IWPFI INTP
1 5 0 0

```

4-46 SMAP-W2 Example Problem

```
* CARD 9.5.2
* WPRS      K      MW
-2000      0.3      3.0E-05
0          0.3      3.0E-05
10         0.3E-03   1.0E-01
100        0.3E-03   1.0E-01
2000       0.3E-03   1.0E-02
* CARD 10.1
* NTPRNT
1
* CARD 10.2.1
* NHPEL
8
* CARD 10.2.2
* NEL1
1  2  3  4  5  6  7  8
* CARD 10.3.1
* NHPMT
11
* CARD 10.3.2
* NODE1
1  2  3  4  5  6  7  8  9  10  11
* CARD 10.4.1
* NTIME
2
* CARD 10.4.2
* TIME1      TIME2
1.          5.
* CARD 10.5.1
* NPHD
2
* CARD 10.5.2
* WP1        WP1
0.0          0.3
* CARD 10.6.1
* NTHD
3
* CARD 10.6.2
* TH1        TH2      TH3
1.1278      1.0363    0.853
* CARD 10.7.1
* NFLN        ISPL      DELV
4           1          0.1
* CARD 10.7.2
* NOD1        NOD2      NOD3      NOD4
665          670        680        690
* CARD 10.8
* NFSEC        IBFQ
0             1
* END OF DATA
```

Table 4.5 Results of computed flow

Dupuit-Thiem Equation :

$$Q_{DT} = \frac{n \cdot k}{\ln\left(\frac{R}{r_o}\right)} (H^2 - h_o^2) = \frac{n \cdot (397)}{\ln\left(\frac{1.95}{0.122}\right)} (1.22^2 - 0.305^2)$$

$$Q_{DT} = 627.9 \text{ m}^3/\text{d per } 2\pi = 99.93 \text{ m}^3/\text{d per radian}$$

Hall 1955 Test A-6 Result :

$$Q_H = 345 \text{ Cm}^3/\text{s per } 15^\circ = 113.7 \text{ m}^3/\text{d per radian}$$

SMAP Result :

Analyses	Discharge Face (r = 0.122 m)	Internal Section (r = 0.6704 m)	Entrance Face (r = 1.95 m)
SMAP-F2 Quad Elem.	104.75 (100.08)	104.35	104.69 (107.25)
SMAP-F2 Triangle Elem.	104.87 (98.71)	104.36	104.76 (107.41)
SMAP-W2 Quad Elem.	104.75 (100.08)	104.35	104.69 (107.25)
SMAP-W2 Triangle Elem.	104.83 (98.73)	104.38	104.78 (107.44)
SMAP-W3 Hexa Elem.	104.85 (100.06)	104.35	104.80 (107.23)
SMAP-W3 Prism Elem.	105.02 (98.69)	104.36	104.86 (107.39)

Values in () are computed from global flow velocity: IBFQ = 2 in Card 10.8

4.5 Transient Seepage Through Generic Levee

The fifth verification problem concerns a transient seepage flow through a generic levee shown in Figure 4.34. It consists of three layers of lean clay, silty sand top stratum and pervious uniform sand aquifer. The river elevation changes with time as shown in the hydrograph in Figure 4.35. This example was solved by Tracy et al. (2016), ERDC TR-16-8 using three different computer programs; SEEP/W, SLIDE, and SEEP2D. SMAP-W2 analyses used the same model parameters as reported in the above reference.

Three different analyses are performed; homogeneous, steady-state and transient analyses. The homogeneous analysis is to check the generated mesh and boundary conditions by assuming the same hydraulic conductivity for all three layers. The steady-state analysis is conducted by applying the peak river elevation of 17.5 ft. And the transient analysis is conducted using the hydrograph in Figure 4.35.

Table 4.6 shows the listing of main input file VP5.Man. This example problem is modeled by 5512 two-dimensional elements mixed with quad and some triangle along the levee slopes. [VP5 Group Mesh.pdf](#) illustrates the detailed step-by-step procedures for mesh generation. Generated finite element mesh around levee is shown in Figure 4.36.

Following are the partial listing of graphical outputs:

Figure 4.37 Homogeneous result with phreatic and other surfaces

Figure 4.38 Steady-state result with phreatic and other surfaces

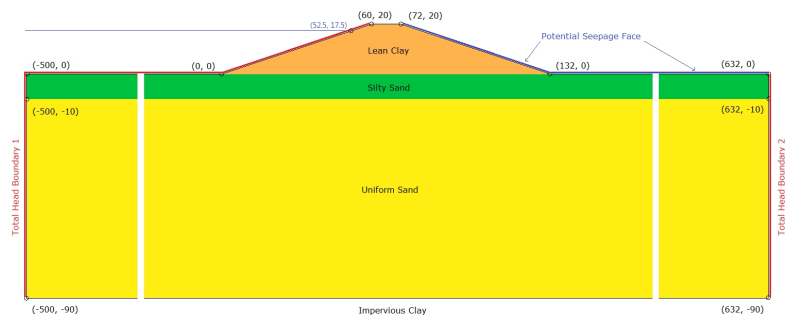
Figure 4.39 Transient phreatic surface at 14 days

Figure 4.40 Total head variation at 10 feet below toe

Figure 4.41 Flow rate through landside ground surface

Overall, SMAP-W2 results are very close to those results in the above reference (ERDC TR-16-8) except flow rate variation in Figure 4.41. At 20 days, SMAP-W2 shows about 15 percent lower peak flow rate compared to ERDC result.

[VP5 Result Summary.pdf](#) includes more detailed transient results at key locations compared with the ERDC report.



Unsaturated soils modeled by van Genuchten (1980)

Material Type	k_s [ft/d]	M_w [ft ³ /lb]	θ_s	θ_r	a [ft ² /lb]	n
Lean Clay	0.0028	10^{-5}	0.5	0.05	0.004885	1.23
Silty Sand	0.28	10^{-5}	0.35	0.035	0.04975	2.24
Uniform Sand	28	10^{-5}	-	-	-	-

Figure 4.34 Generic levee section for VP5

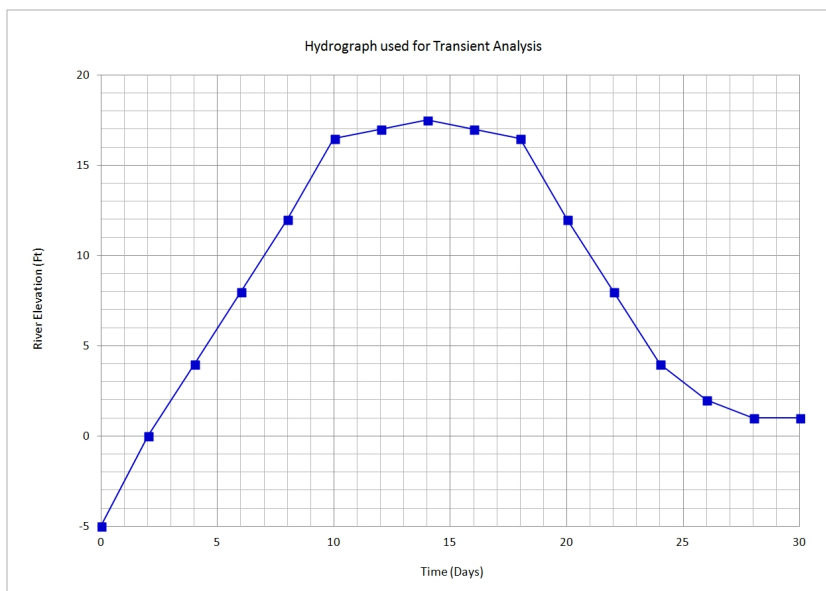


Figure 4.35 Hydrograph used for transient analysis

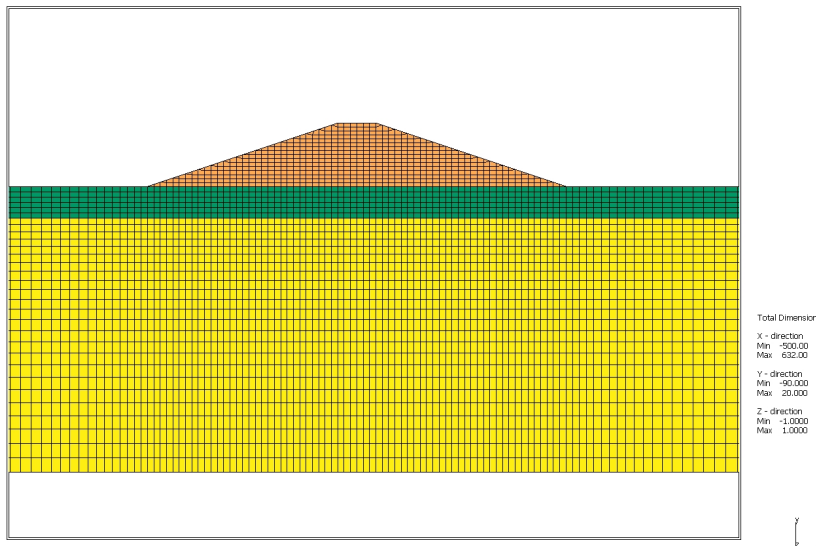
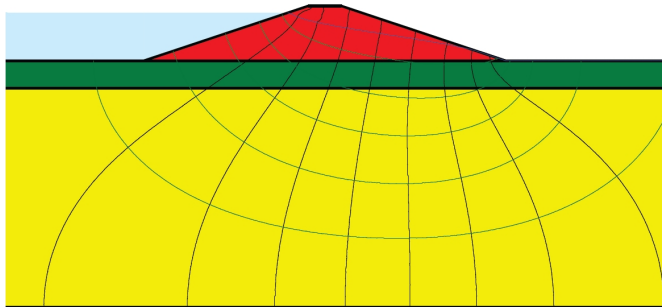


Figure 4.36 Finite element mesh around levee

ERDC TR-16-8 Homogeneous Soils



SMAP-W2 Homogeneous Soils

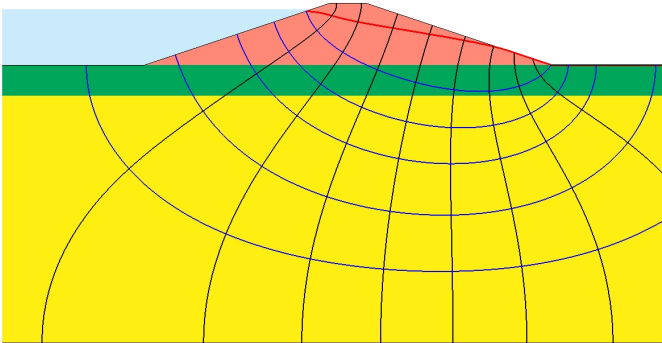


Figure 4.37 Homogeneous result with phreatic and other surfaces

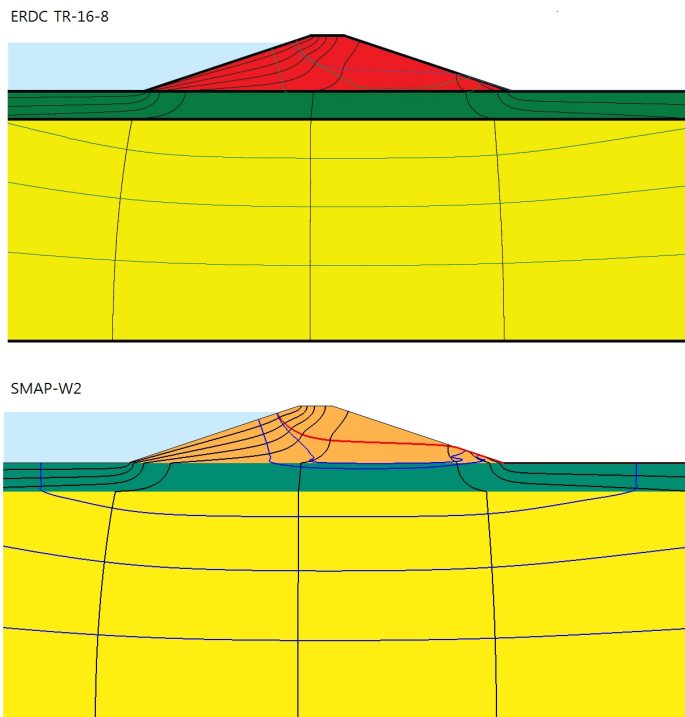


Figure 4.38 Steady-state result with phreatic and other surfaces

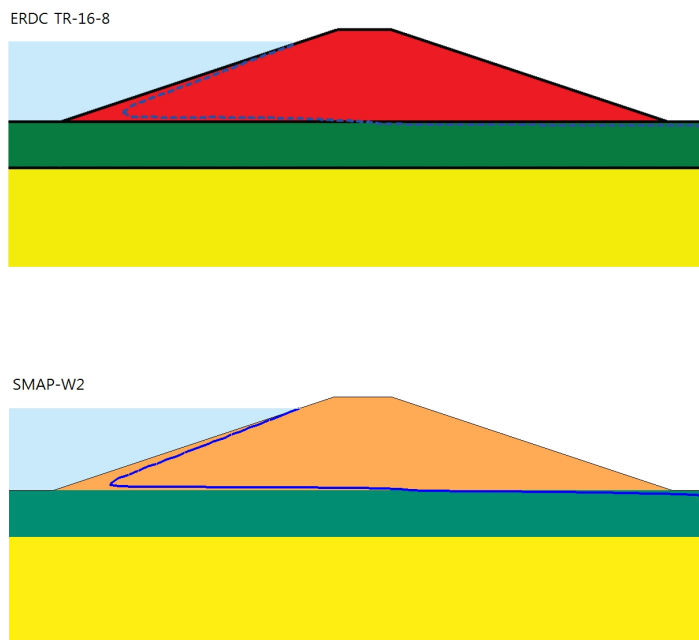


Figure 4.39 Transient result with phreatic surface at 14 days

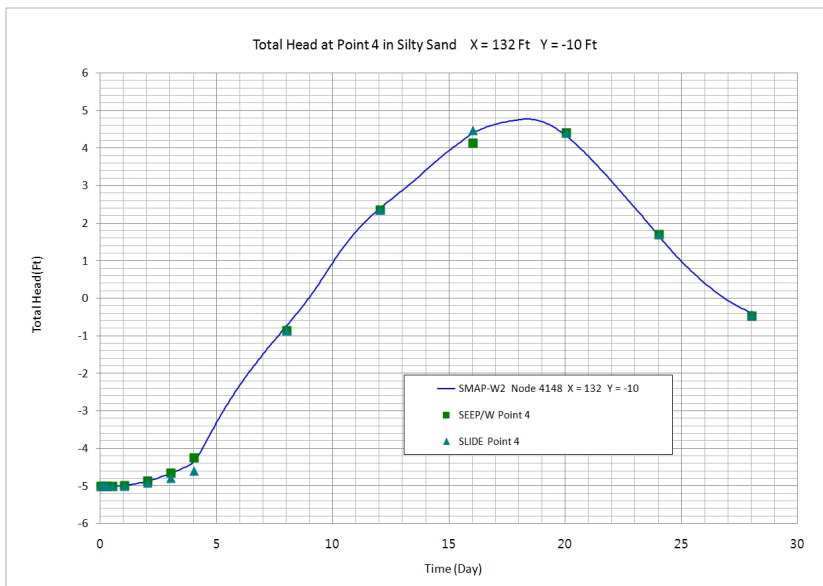


Figure 4.40 Total head variation at 10 feet below toe

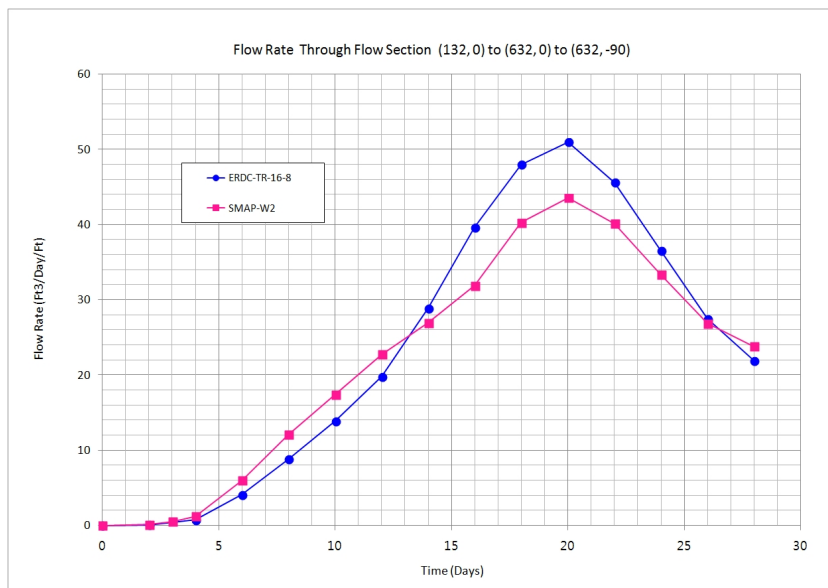


Figure 4.41 Flow rate through landside ground surface

Table 4.6 Listing of main input file VP5.Man

```

* CARD 1.1
* TITLE
  Verification Problem 5 (ERDC TR-16-8 GENERIC LEVEE)
* CARD 2.1
* IP  ILUMP  IBATCH  ISST  IMSEEP  IDFVEL  IFVOL  ISFACE
  1    1      -1      1      0        0        1        1
* CARD 3.1
* NCYCL  NSTCL  NITER  MCYCL  TOLER  IRANGE  IRNODE
  280    0      20     10     0.01    0        0
* CARD 3.2
* SIP
  0.5
* CARD 3.2.1
* IDELT  DT
  0      0.1
* CARD 4.1
* NUMNP
  5691
* CARD 4.2
* CMFAC  XREF
  1.0    0.0
* CARD 5.1
* NCONT
  5512
* CARD 5.2.1
* NTNC
  3
* CARD 5.2.2.1
* TITLE
  MATERIAL 1 (Lean Clay)
* CARD 5.2.2.2
* MATNO  MATFN  MW(ft2/lb)  GW(lb/ft3)  KH(ft/d)  KV(ft/d)  THETA
  1      11     1.0E-05     62.4        0.0028    0.0028    0.
* CARD 5.2.2.3
* VWCsat  VWCres  alpha(ft2/lb)  n      m      Uswcc  SCFac  Gwm  Pr
  0.5     0.05    0.004885        1.23  0.187  0      1.     62.4  0.
* CARD 5.2.2.1
* TITLE
  MATERIAL 2 (Silty Sand)
* CARD 5.2.2.2
* MATNO  MATFN  MW(ft2/lb)  GW(lb/ft3)  KH(ft/d)  KV(ft/d)  THETA
  2      11     1.0E-05     62.4        0.28      0.28      0.

```

4-58 SMAP-W2 Example Problem

```
* CARD 5.2.2.3
* VWCsat VWCres alpha(ft2/lb) n      m      Uswcc SCFac Gwm Pr
0.35    0.035 0.04975      2.24 0.5536 0    1.    62.4 0.
* CARD 5.2.2.1
* TITLE
MATERIAL 3 (Uniform Sand)
* CARD 5.2.2.2
* MATNO MATFN MW(ft2/lb) GW(lb/ft3) KH(ft/d) KV(ft/d) THETA
3      0      1.0E-05    62.4      28      28      0.
* CARD 9.4.1
* NTIME NTIM
2      16
* CARD 9.4.2
* TIME FN1 FN2
0.     -5.   -5.
2.      0.   -5.
4.      4.   -5.
6.      8.   -5.
8.     12.   -5.
10.    16.5  -5.
12.    17.   -5.
14.    17.5  -5.
16.    17.   -5.
18.    16.5  -5.
20.    12.   -5.
22.     8.   -5.
24.     4.   -5.
26.     2.   -5.
28.     1.   -5.
300.    1.  -5.
* CARD 9.5.1
* NWPRF NWPR IWPFI INTP
0      0      0      0
* CARD 10.1
* NTPRNT
1
* CARD 10.2.1
* NHPEL
8
* CARD 10.2.2
* NEL1
1  2  3  4  5  6  7  8
* CARD 10.3.1
* NHPMT
6
* CARD 10.3.2
* NODE1 NODE2 NODE3 NODE4 NODE5 NODE6
3020   3389   3539   3701   3818   4148
* CARD 10.4.1
* NTIME
11
```



```

* CARD 10.4.2
* TIME1  TIME2  TIME3  TIME4  TIME5  TIME6  TIME7
  6.    10.    12.    14.    16.    18.    20.
 22.    24.    26.    28.
* CARD 10.5.1
* NPHD
  4
* CARD 10.5.2
* WP1    WP2    WP3    WP4
  0.0    1000.  2000.  3000.
* CARD 10.6.1
* NTHD
  9
* CARD 10.6.2
* TH1    TH2    TH3    TH4    TH5    TH6    TH7    TH8    TH9
  0      2      4      6      8      10     12     14     16
* CARD 10.7.1
* NFLN   ISPL   DELV
  3      1      0.
* CARD 10.7.2
* NOD1   NOD2   NOD3
  373    590    993
* CARD 10.8
* NFSEC   IBFQ
  1      0
* Flux Section (132, 0) to (632, 0) to (632, -90)
* CARD 10.8.1
* NODES
  81
* Potential Seepage Face (Ground), Generated by Group Mesh
* LIST OF NODES
  4142  4173  4204  4235  4266  4297  4328  4359  4390  4421
  4452  4483  4514  4545  4576  4607  4638  4669  4700  4731
  4762  4793  4824  4855  4886  4917  4948  4979  5010  5041
  5072  5103  5134  5165  5196  5227  5258  5289  5320  5351
  5382  5413  5444  5475  5506  5537  5568  5599  5630  5661
* Total Head Boundary 2, Generated by Group Mesh
* LIST OF NODES
  5661  5662  5663  5664  5665  5666  5667  5668  5669  5670
  5671  5672  5673  5674  5675  5676  5677  5678  5679  5680
  5681  5682  5683  5684  5685  5686  5687  5688  5689  5690
  5691
* END OF DATA

```

4.6 Steady-State Seepage Through Earth Dam

The sixth verification problem concerns a steady-state seepage flow through a homogeneous isotropic earth dam shown in Figure 4.42. Figure 4.43 shows unsaturated hydraulic conductivity as a function of suction to model the unsaturated part of dam above the phreatic surface. This example is the same as the Example 9-5 in Physical and Geotechnical Properties of Soils by Bowles (1984) and also solved using two different computer programs; SEEP/W and SLIDE. SMAP-W2 analyses used the same model parameters as reported in the above reference.

Three different finite element meshes are prepared as shown in Figure 4.44. Mesh 1 is generated by the Group Mesh Generator. Refer to [VP6 Group Mesh.pdf](#). Mesh 2 and 3 are generated by the Earth Dam Mesh Generator which is the special feature in Addrgn-2D. Refer to [VP6 Earth Dam Mesh.pdf](#).

Table 4.7 shows the listing of main input file VP6.Man for Mesh 1. The other input files for Mesh 2 and 3 are similar to Mesh 1 input.

Figure 4.45 shows computed results of selected flow lines and total head contours along with the phreatic surface in red line. All three analyses with different finite element meshes show almost the same total head contours. Note that the flow path starting at the water elevation of upstream slope is slightly above the the phreatic surface for most part. This means that there are some water flow in unsaturated zone above the phreatic surface.

Table 4.8 shows steady-state flow rate on downstream seepage face. Overall, SMAP-W2 results are very close to those published results. SMAP-W2 results are not sensitive to the number of elements. They all show flow rate of about $1.39 \times 10^{-3} \text{ m}^3/(\text{min. m})$.

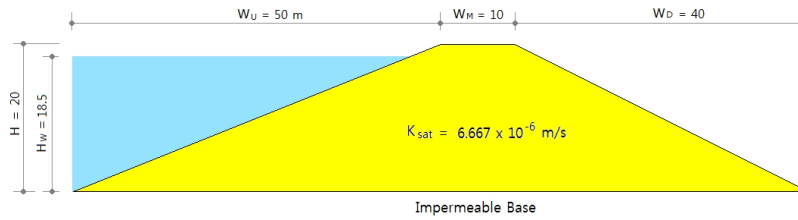


Figure 4.42 Earth dam cross section for VP6

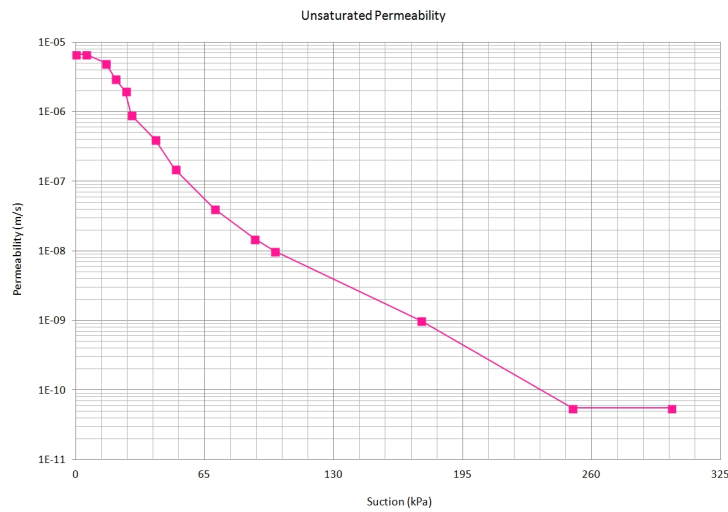
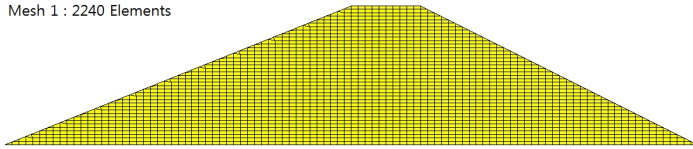
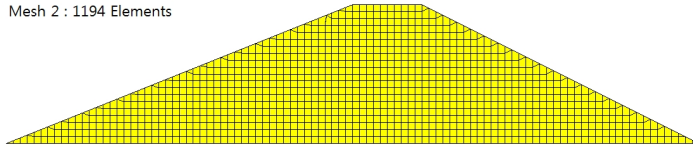


Figure 4.43 Unsaturated hydraulic conductivity

Mesh 1 : 2240 Elements



Mesh 2 : 1194 Elements



Mesh 3 : 561 Elements

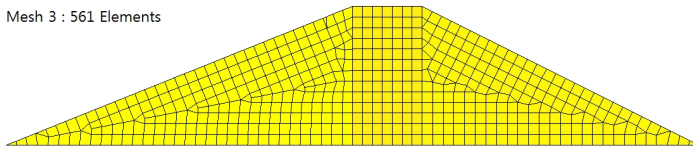
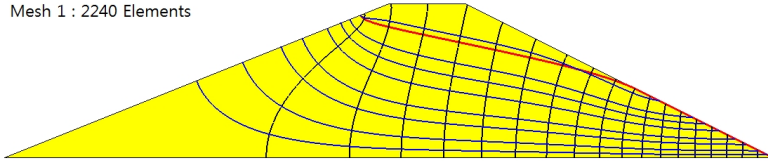
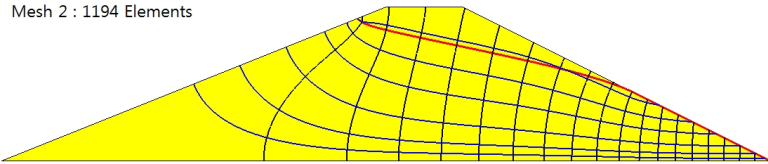


Figure 4.44 Finite element meshes

Mesh 1 : 2240 Elements



Mesh 2 : 1194 Elements



Mesh 3 : 561 Elements

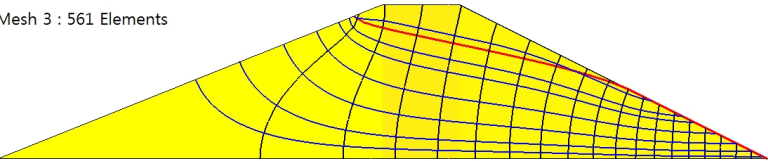


Figure 4.45 Flow lines and total head contours

Table 4.7 Listing of main input file VP6.Man

```

* CARD 1.1
* TITLE
  Verification Problem 6
* CARD 2.1
* IP  ILUMP  IBATCH  ISST  IMSEEP  IDFVEL  IFVOL  ISFACE
  1    1      -1      0      0      1      1      1
* CARD 3.1
* NCYCL  NSTCL  NITER  MCYCL  TOLER  IRANGE  IRNODE
  100    1      20     10     0.01    0      0
* CARD 3.2
* SIP
  0.5
* CARD 3.2.1
* IDELT  DT
  0      0.1
* CARD 4.1
* NUMNP
  2301
* CARD 4.2
* CMFAC  XREF
  1.0    0.0
* CARD 5.1
* NCONT
  2240
* CARD 5.2.1
* NTNC
  1
* CARD 5.2.2.1
* TITLE
  MATERIAL 1
* CARD 5.2.2.2
* MATNO  MATFN  MW(1/kPa)  GW(kN/m3)  KH(m/s)  KV(m/s)  THETA
  1      1      1.0E-05    9.81      6.67e-6  6.67e-6  0.
* CARD 5.2.2.3
* VWCsat  VWCres  alpha(1/m)  n  m  Uswcc  SCFac  Gwm  Pr
  0.45    0.15    0.1      0    0    0      1.    9.81  0.
* CARD 9.4.1
* NTIMF  NTIM
  1      2
* CARD 9.4.2
* TIME  FN1
  0.     18.5
  10000. 18.5

```

```

* CARD 9.5.1
* NWPRF      NWPR      IWPF      INTP
  1          15          0          1
* CARD 9.5.2
* WP (kPa)    K (m/s)      Mw (1/kPa)
-1000.        6.67E-6      1.0E-5
  0.          6.67E-6      1.0E-5
  5.          6.67E-6      1.0E-5
 15.          5.00E-6      1.0E-5
 20.          3.00E-6      1.0E-5
 25.          2.00E-6      1.0E-5
 28.          9.00E-7      1.0E-5
 40.          4.00E-7      1.0E-5
 50.          1.50E-7      1.0E-5
 70.          4.00E-8      1.0E-5
 90.          1.50E-8      1.0E-5
100.          1.00E-8      1.0E-5
174.          1.00E-9      1.0E-5
250.          5.50E-11     1.0E-5
1000.         5.50E-11     1.0E-5
* CARD 10.1
* NTPRNT
  1
* CARD 10.2.1
* NHPEL
  8
* CARD 10.2.2
* NEL1
  1  2  3  4  5  6  7  8
* CARD 10.3.1
* NHPMT
 11
* CARD 10.3.2
* NODE1
  1  2  3  4  5  6  7  8  9 10 11
* CARD 10.4.1
* NTIME
  3
* CARD 10.4.2
* TIME1      TIME2      TIME3
  0.1        1          10
* CARD 10.5.1
* NPHD
  4
* CARD 10.5.2
* WP1        WP2        WP3        WP4
  0          50        100        150

```

4-66 SMAP-W2 Example Problem

```
* CARD 10.6.1
* NTHD
18
* CARD 10.6.2
* TH1      TH2      TH3      TH4      TH5
1          2          3          4          5
6          7          8          9          10
11         12         13         14         15
16         17         18
* CARD 10.7.1
* NFLN      ISPL      DELV
8           1          0.0
* CARD 10.7.2
* NOD1      NOD2      NOD3      NOD4      NOD5
266         404       541       633       732
802         838       875
* CARD 10.8
* NFSEC      IBFQ
1            0
* Flux Section (Landside Seepage Surface)
* CARD 10.8.1
* NODES
41
* Potential Seepage Face (Ground), Generated by Group Mesh
* LIST OF NODES
1441 1482 1522 1561 1599 1636 1672 1707 1741 1774
1806 1837 1867 1896 1924 1951 1977 2002 2026 2049
2071 2092 2112 2131 2149 2166 2182 2197 2211 2224
2236 2247 2257 2266 2274 2281 2287 2292 2296 2299
2301
* END OF DATA
```


Table 4.8 Steady-state flow rate on seepage face

Progam	Total Elements	Flow Rate $\text{m}^3/(\text{min. m})$	Reference
Bowles	Flow nets	1.103×10^{-3}	Bowles, 1984 (Example 9.5a)
	Equation	1.280×10^{-3}	Direct computation with Eq. 9.4
RS2	391	1.378×10^{-3}	SLIDE verification manual, 2022
SEEP / W	295	1.410×10^{-3}	Chapuis R, et al. 2001
	1145	1.370×10^{-3}	Chapuis R, et al. 2001
SMAP-W2	2240	1.3897×10^{-3}	Mesh 1
	1194	1.3906×10^{-3}	Mesh 2
	561	1.3893×10^{-3}	Mesh 3

Group Mesh Example Problem

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

Two example problems are presented:

1. [Arch Tunnel](#)
Shows step-by-step procedure to create and modify group meshes.
2. [Finite Element Mesh Modification](#)
Illustrates how to modify existing finite element meshes using [Mesh Generator](#).

5.1 Arch Tunnel

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

This example has the following three parts:

Part 1 : Creating Arch Tunnel (Figure 5.1)

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

Part 2 : Adding Rock Bolts (Figure 5.2)

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

Part 3 : Adding Utility Tunnel (Figure 5.3)

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

Table 5.1 shows the construction sequence.

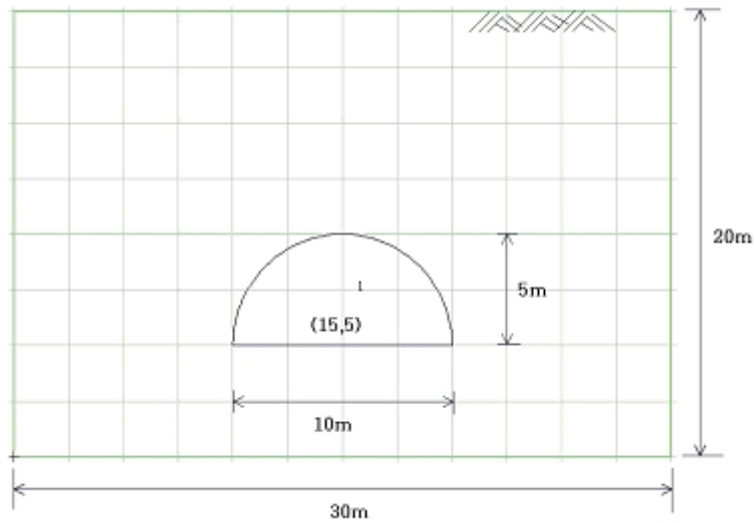


Figure 5.1 Arch tunnel (Part 1)

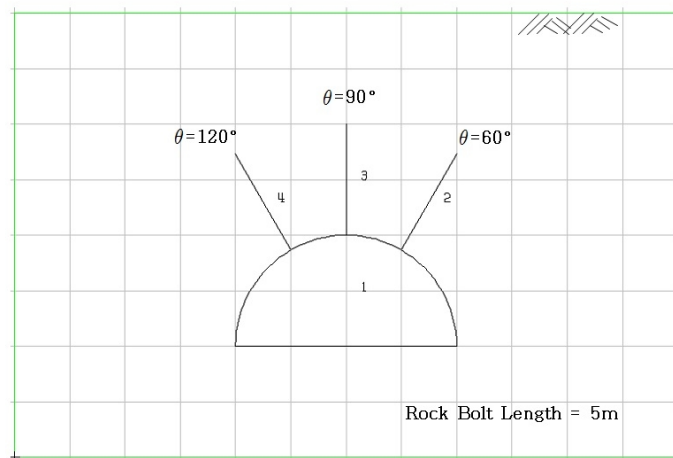


Figure 5.2 Arch tunnel with rock bolts (Part 2)

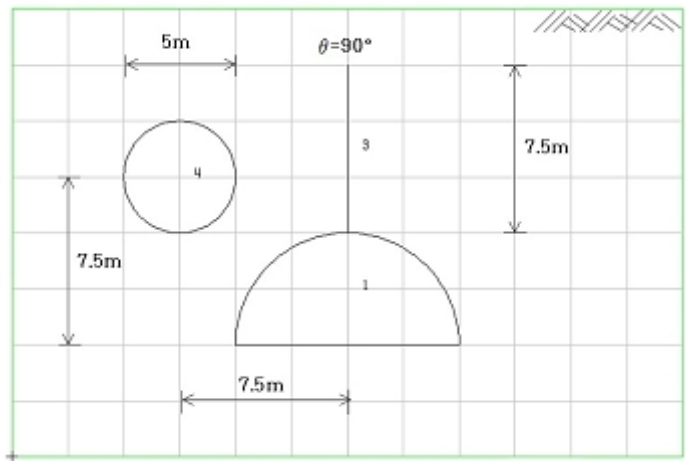


Figure 5.3 Arch tunnel with utility tunnel (Part 3)

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

Table 5.1 Construction sequence

5.1.1 Part 1: Creating Arch Tunnel

Part 1 consists of the following main actions:

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

Step 1: Group Mesh Generator (New)

Access [Group Mesh Generator](#) by selecting the following menu items in [SMAP](#) (Figure 5.4):

Run → Mesh Generator → Group Mesh → New

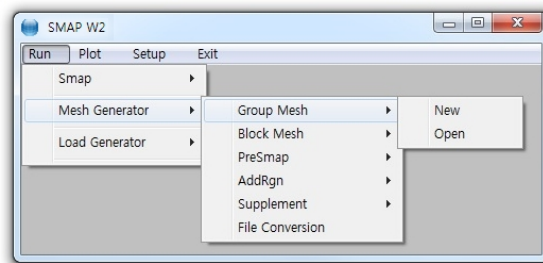


Figure 5.4 Accessing group mesh generator (New)

Step 2: Group Input (New)

Select [Built-in Base Mesh](#) in Figure 5.5. Click [OK](#).

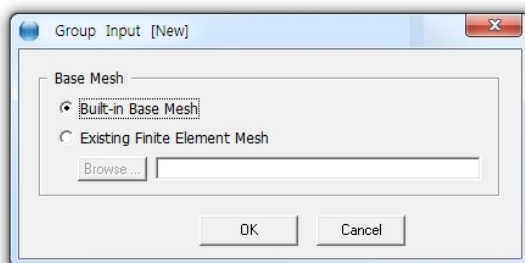


Figure 5.5 Group input (New)

5-6 Group Mesh Example

Step 3: Group Menu and Dialog

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

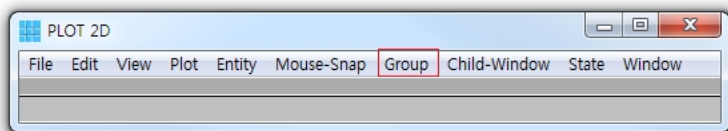


Figure 5.6 Group menu

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

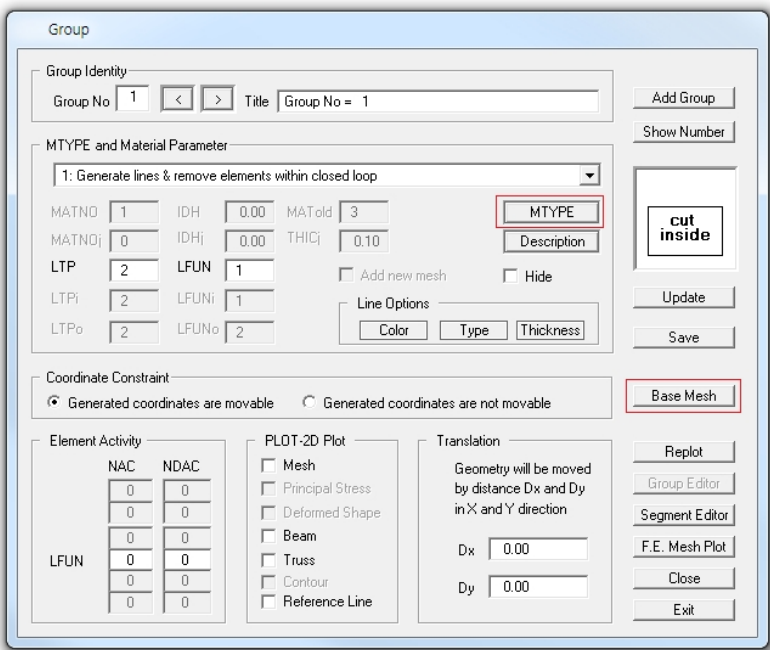


Figure 5.7 Group dialog with initial default values

Step 4: Built-in Base Mesh

Click **Base Mesh** button in **Group** dialog.

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

Figure 5.8 Built-in base mesh dialog

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

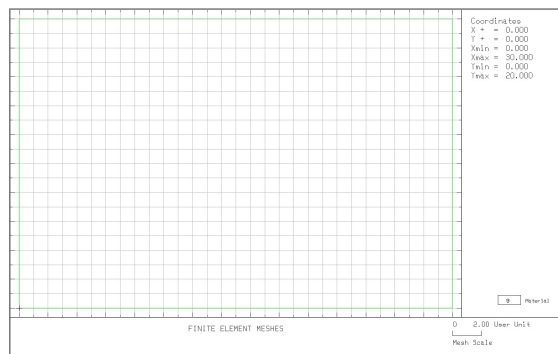


Figure 5.9 Base mesh on drawing board

5-8 Group Mesh Example

Step 5: MTYPE

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

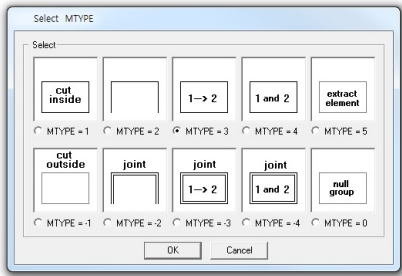


Figure 5.10 MTYPE dialog

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

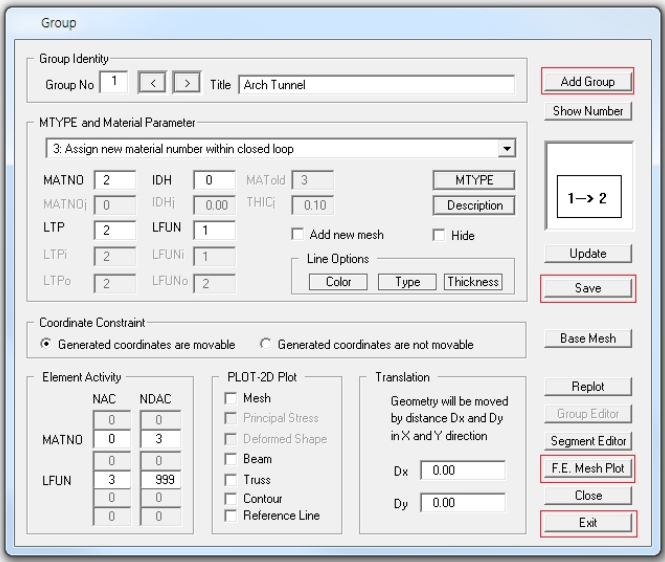


Figure 5.11 Group dialog with MTYPE = 3

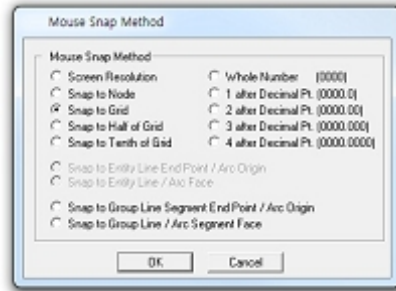
Step 6: Mouse Snap

Click **Mouse-Snap** menu in **PLOT-2D**.

Select **Snap to Grid** in Figure 5.12. Click **OK**.

Figure 5.12

Mouse snap dialog

**Step 7: Add Group**

Click **Add Group** button in **Group** dialog.

Table 5.2 summarizes group parameters used for arch tunnel.

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

Group No	Seg. No	Line Segment				Arc Segment						IEND
		Beginning Point		Ending Point		Origin		Radius and Angle				
		X	Y	X	Y	X _o	Y _o	R _x	R _y	Θ _b	Θ _e	
1	1	10	5	20	5							2
	2					15	5	5	5	0	180	2

Table 5.2 Group parameters for arch tunnel

Step 8: Line Segment

Click **Draw** button in **Line Segment** dialog in Figure 5.13.

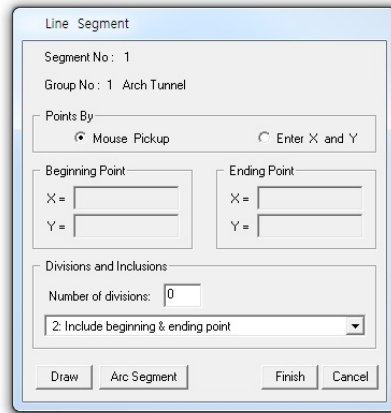


Figure 5.13 Line segment dialog

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

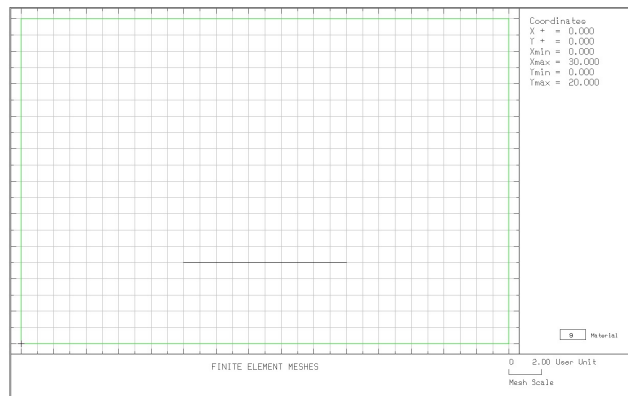


Figure 5.14 Line segment on drawing board

Step 9: Arc Segment

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

Figure 5.15 Arc segment dialog

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

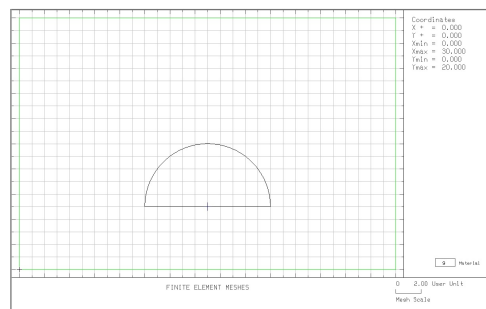


Figure 5.16 Arc segment on drawing board

Step 10: Save

Click **Save** button in **Group** dialog.

Group.Meg is saved as shown in Figure 5.17.

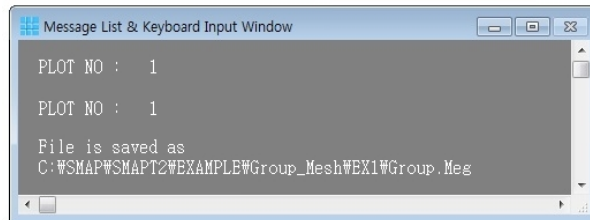


Figure 5.17 Message for file save

Step 11: Finite Element Mesh

Click **F.E. Mesh Plot** button in **Group** dialog.

Click **Yes** in Figure 5.18.

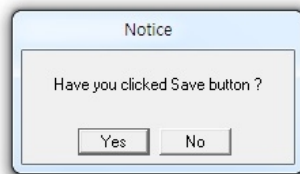


Figure 5.18 Notice for finite element mesh plot

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

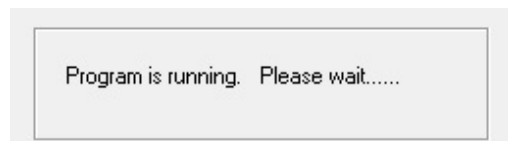


Figure 5.19 Notice while generating finite element mesh plot

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

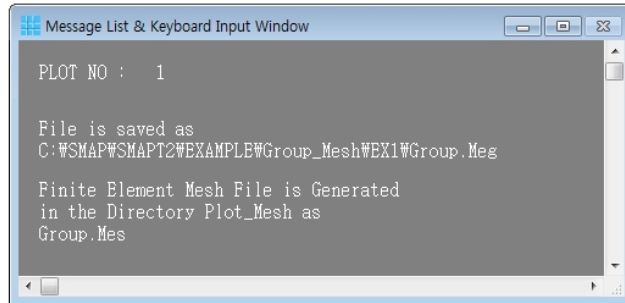


Figure 5.20 Message for finite element mesh file

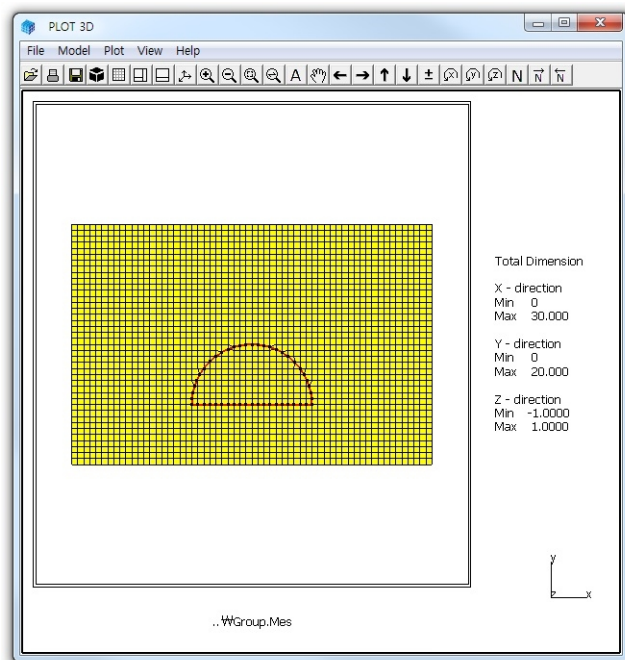


Figure 5.21 Finite element mesh plot

Step 12: Exit

Click **Exit** button in **Group** dialog.

Click **OK** in **Exit** dialog as shown in Figure 5.22.

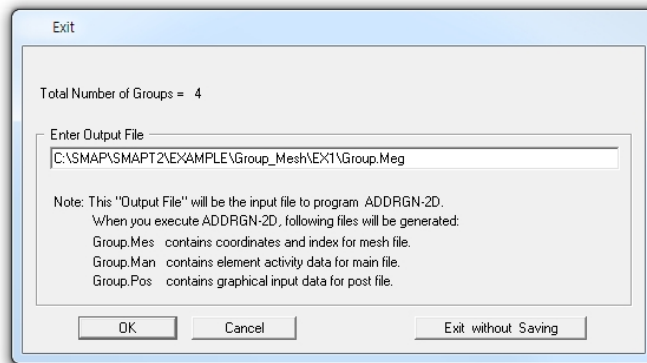


Figure 5.22 Exit dialog

5.1.2 Part 2: Adding Rock Bolts

Part 2 consists of the following main actions:

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

Step 13: Group Mesh Generator (Open)

Access [Group Mesh Generator](#) by selecting the following menu items in [SMAP](#) (Figure 5.4):

[Run](#) → [Mesh Generator](#) → [Group Mesh](#) → [Open](#)

Step 14: Group Input (Open)

File open dialog will be displayed as in Figure 5.23.

Select group mesh file [Group.Meg](#) in Part 1 and click [Open](#).

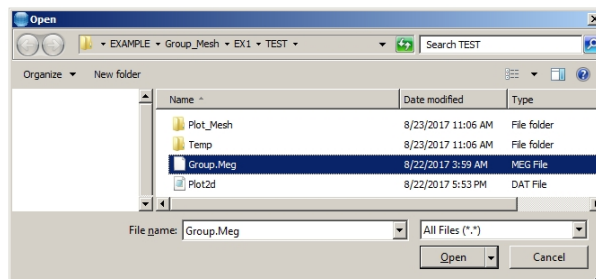


Figure 5.23 File open dialog

Step 15: Group Menu and Dialog

Click [Group](#) menu in [PLOT-2D](#) as shown in Figure 5.6.

[Group](#) dialog for Group No 2 is displayed with initial default values.

Step 16: MTYPE

Click [MTYPE](#) button in [Group](#) dialog.

Select [MTYPE=2](#) in [MTYPE](#) dialog in Figure 5.10.

Click [OK](#).

Step 17: Group No 2 for Rock Bolt 1

Table 5.3 summarizes group parameters for rock bolts.

Rock bolt is modeled by a straight radial line in [Arc Segment](#).

Group No	Bolt No	MTYPE	Elem. Type (LTP)	Mat. No (LFUN)	Element Activity		Radius and Angle				IEND
					NAC	NDAC	R _x	R _y	Θ _b	Θ _e	
2	Bolt-1	2	Truss (3)	1	4	999	5	10	60	60	-2
3	Bolt-2	2	Truss (3)	1	4	999	5	10	90	90	-2
4	Bolt-3	2	Truss (3)	1	4	999	5	10	120	120	-2

Table 5.3 Group parameters for rock bolts

Group No 2 represents [Rock Bolt 1](#) with a length of 5m at 60 degrees.

Fill in input fields for [Group](#) dialog as shown in Figure 5.24.

The screenshot shows the 'Group' dialog box with the following settings:

- Group Identity:** Group No: 2, Title: Rock Bolt 1
- MTYPE and Material Parameter:**
 - MTYPE: 2 (Generate lines)
 - MATNO: 1, IDH: 0.00, MATold: 3, THICK: 0.10
 - MATNO: 0, IDH: 0.00, THICK: 0.10
 - LTP: 3, LFUN: 1
 - LTP: 2, LFUN: 1
 - LTP: 2, LFUN: 2
 - Line Options: Color, Type, Thickness
- Coordinate Constraint:**
 - ☒ Generated coordinates are movable
 - ☐ Generated coordinates are not movable
- Element Activity:**
 - NAC: 0, NDAC: 0
 - LFUN: 4, 999
 - LFUN: 0, 0
 - LFUN: 0, 0
- PLOT:2D Plot:**
 - ☐ Mesh
 - ☐ Principal Stress
 - ☐ Deformed Shape
 - ☐ Beam
 - ☐ Truss
 - ☐ Contour
 - ☐ Reference Line
- Translation:**
 - Geometry will be moved by distance Dx and Dy in X and Y direction
 - Dx: 0.00, Dy: 0.00

Figure 5.24 Group dialog for Rock Bolt 1

Step 18: Mouse Snap

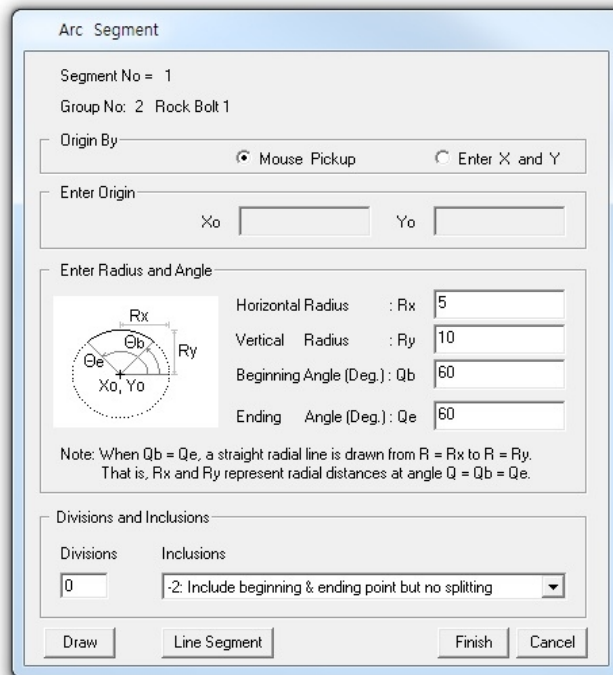
Click **Mouse-Snap** menu in **PLOT-2D**.
Select **Snap to Grid** in Figure 5.12.
Click **OK**.

Step 19: Add Group

Click **Add Group** button in **Group** dialog.

Step 20: Arc Segment

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



The image shows the 'Arc Segment' dialog box. It has a title bar 'Arc Segment'. Inside, 'Segment No = 1' and 'Group No: 2 Rock Bolt 1' are displayed. There are two radio buttons for 'Origin By': 'Mouse Pickup' (selected) and 'Enter X and Y'. Below this is a section 'Enter Origin' with input fields for 'Xo' and 'Yo'. Another section 'Enter Radius and Angle' contains a diagram of an arc with labels 'Rx', 'Ry', 'Qb', 'Qe', and 'Xo, Yo'. To the right of the diagram are input fields: 'Horizontal Radius : Rx' (5), 'Vertical Radius : Ry' (10), 'Beginning Angle (Deg.) : Qb' (60), and 'Ending Angle (Deg.) : Qe' (60). A note below states: 'Note: When Qb = Qe, a straight radial line is drawn from R = Rx to R = Ry. That is, Rx and Ry represent radial distances at angle Q = Qb = Qe.' At the bottom, there is a section 'Divisions and Inclusions' with 'Divisions' set to '0' and 'Inclusions' set to '-2: Include beginning & ending point but no splitting'. At the very bottom are buttons for 'Draw', 'Line Segment', 'Finish', and 'Cancel'.

Divisions and Inclusions	
Divisions	Inclusions
0	-2: Include beginning & ending point but no splitting

Figure 5.25 Arc segment dialog for Rock Bolt 1

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

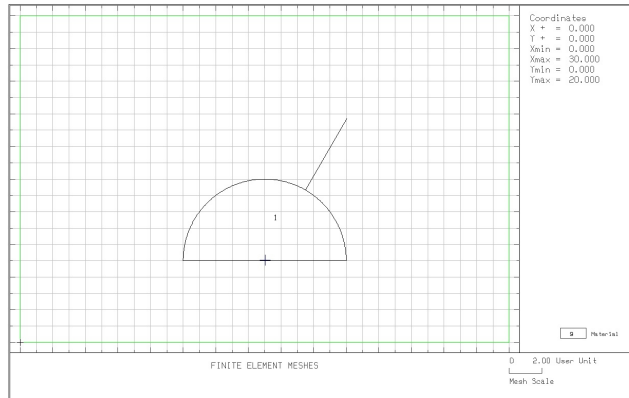


Figure 5.26 Rock Bolt 1 on drawing board

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

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

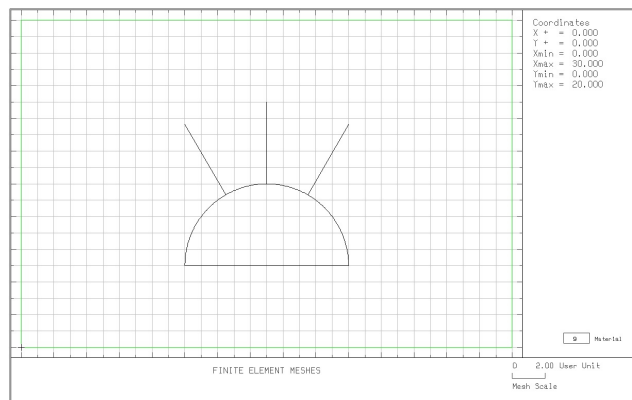


Figure 5.27 All three rock bolts on drawing board

Step 22: Finite Element Mesh

Click **F.E. Mesh Plot** button in **Group** dialog.

Follow the same procedure as in Steps 10 and 11.

Finite element meshes are shown in Figure 5.28

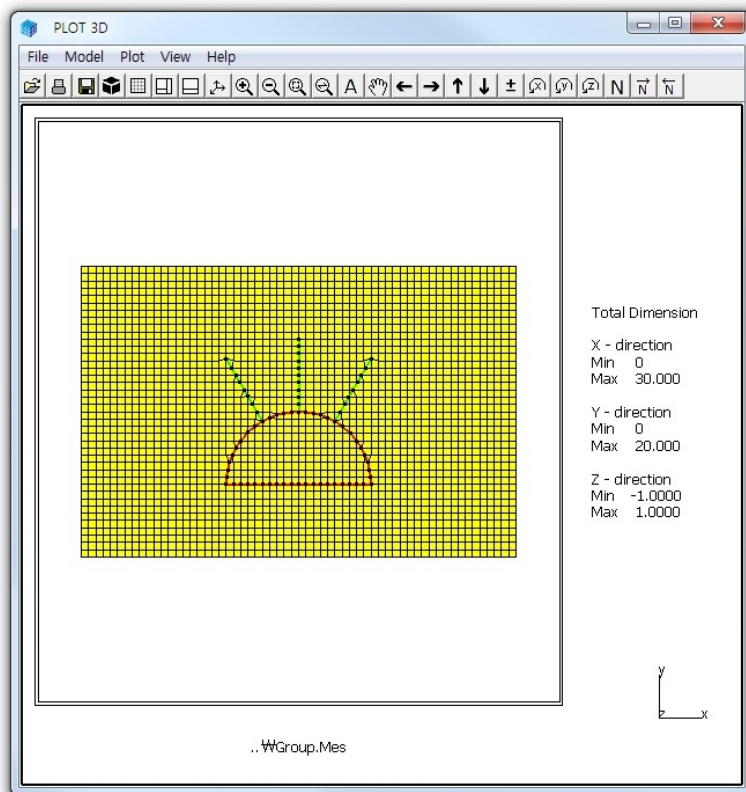


Figure 5.28 Finite element mesh plot

Step 23: Exit

Click **Exit** button in **Group** dialog.

Click **OK** in **Exit dialog** as in Figure 5.22.

5.1.3 Part 3: Adding Utility Tunnel

Part 3 consists of the following main actions:

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

Step 24: Open Group Mesh File in Part 2

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

Step 25: Remove Rock Bolt 1

Select Group No 2 in **Group** dialog.

Click **MTYPE** button in **Group** dialog.

Select **MTYPE=0** in **MTYPE** dialog in Figure 5.10.

Click **OK**.

Click **Update** and then **Replot** buttons in **Group** dialog.

A new plot with the Group No 2 missing is displayed in Figure 5.29

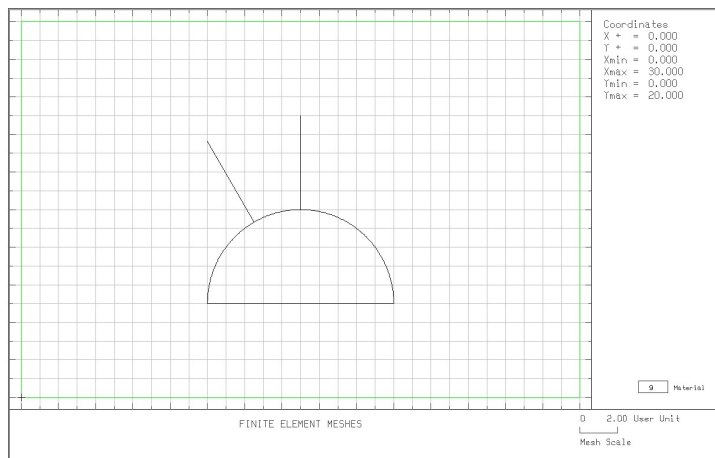


Figure 5.29 Rock Bolt 1 removed on drawing board

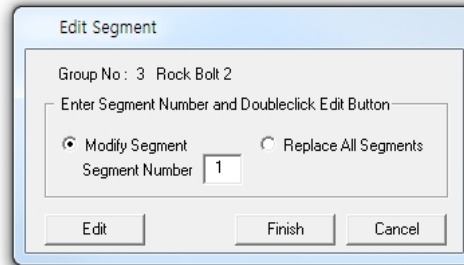
Step 26: Change Length of Rock Bolt 2

Select Group No 3 in **Group** dialog.

Click **Edit Group** button in **Group** dialog.

Click **Edit** button in **Edit Segment** dialog in Figure 5.30.

Figure 5.30
Edit segment dialog
for Group No 3



Fill in input fields for **Arc Segment** dialog as shown in Figure 5.31.

Click **Draw** and then **Finish** in **Arc Segment** dialog.

Click **Finish** in **Edit Segment** dialog.

Figure 5.31
Arc segment dialog with
rock bolt length modified

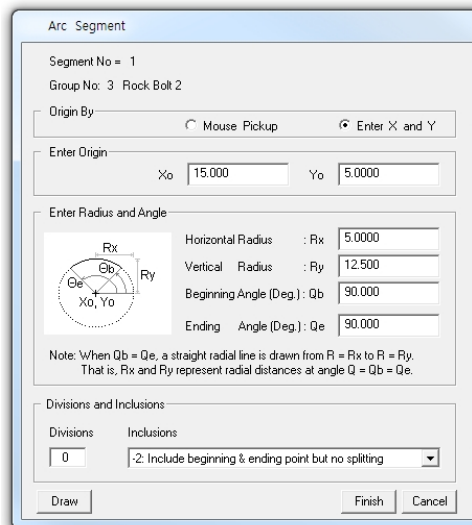


Figure 5.32 shows a new plot with longer Rock Bolt 2.

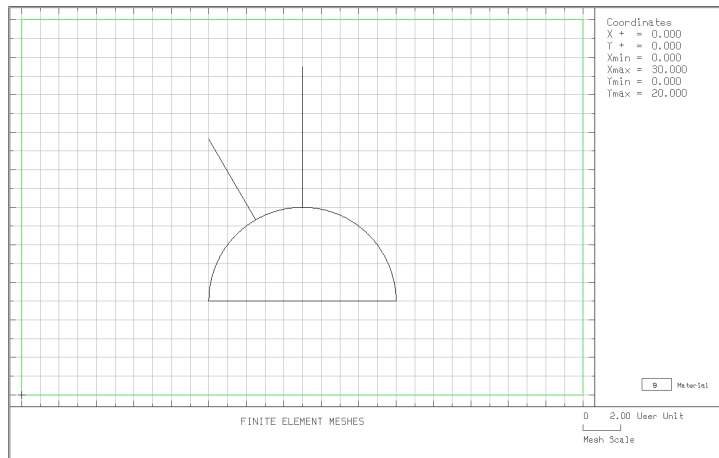


Figure 5.32 Longer Rock Bolt 2 on drawing board

Step 27: Replace Rock Bolt 3 by Utility Tunnel

Select Group No 4 in **Group** dialog.

Click **MTYPE** button in **Group** dialog.

Select **MTYPE=1** in **MTYPE** dialog in Figure 5.10.

Click **OK**.

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

Click **Edit Group**.

Group

Group Identity
 Group No: 4 | Title: Utility Tunnel

MTYPE and Material Parameter
 1: Generate lines & remove elements within closed loop
 MATNO: 1 | IDH: 0.00 | MATold: 3 | MTYPE: [button]
 MATNOj: 0 | IDHj: 0.00 | THICj: 0.10 | Description: [button]
 LTP: 2 | LFUN: 2 | Add new mesh: [checkbox] | Hide: [checkbox]
 LTPi: 2 | LFUNi: 1 | Line Options: Color [button] | Type [button] | Thickness [button]
 LTPo: 2 | LFUNo: 2

Coordinate Constraint
☒ Generated coordinates are movable
☐ Generated coordinates are not movable

Element Activity

	NAC	NDAC
LFUN	0	0
	0	0
	0	0
	5	999
	0	0

PLOT-2D Plot
☐ Mesh
☐ Principal Stress
☐ Deformed Shape
☐ Beam
☐ Truss
☐ Contour
☐ Reference Line

Translation
 Geometry will be moved by distance Dx and Dy in X and Y direction
 Dx: 0.00
 Dy: 0.00

Buttons: Edit Group, Show Number, Update, Save, Base Mesh, Replot, Group Editor, Segment Editor, F.E. Mesh Plot, Close, Exit

Figure 5.33 Group dialog for Utility Tunnel

5-24 Group Mesh Example

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

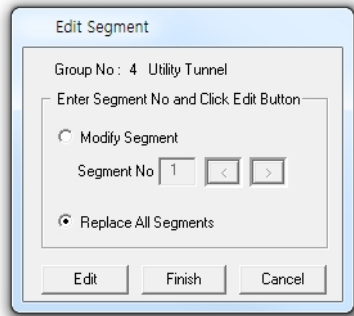


Figure 5.34 Edit segment dialog for Group No 4

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

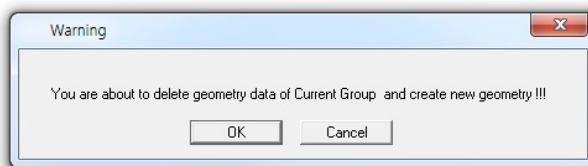
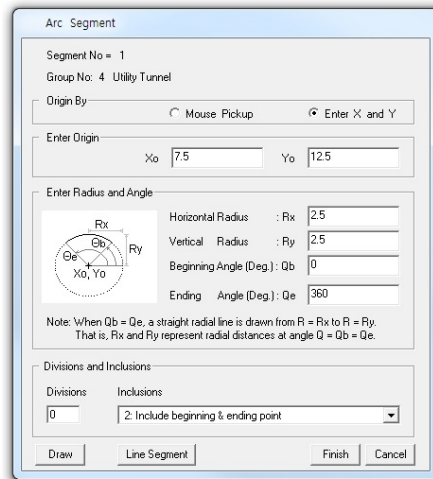


Figure 5.35 Warning message

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



The **Arc Segment** dialog box is shown with the following settings:

- Segment No: 1
- Group No: 4 Utility Tunnel
- Origin By: ☐ Mouse Pickup ☒ Enter X and Y
- Enter Origin: X_o = 7.5, Y_o = 12.5
- Enter Radius and Angle:
 - Horizontal Radius : R_x = 2.5
 - Vertical Radius : R_y = 2.5
 - Beginning Angle (Deg.): Q_b = 0
 - Ending Angle (Deg.): Q_e = 360
- Note: When Q_b = Q_e, a straight radial line is drawn from R_x = R_x to R_y = R_y. That is, R_x and R_y represent radial distances at angle Q = Q_b = Q_e.
- Divisions and Inclusions:
 - Divisions: 0
 - Inclusions: 2 Include beginning & ending point
- Buttons: Draw, Line Segment, Finish, Cancel

Figure 5.36 Arc segment dialog for Utility Tunnel

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

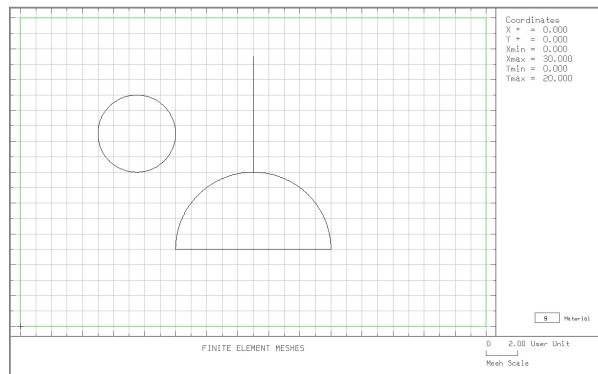


Figure 5.37 Arch and Utility Tunnels on drawing board

Step 28: Finite Element Mesh

Click **Save** and **F.E. Mesh Plot** button in **Group** dialog.

Follow the same procedure as in Steps 10 and 11.

Finite element meshes are shown in Figure 5.38

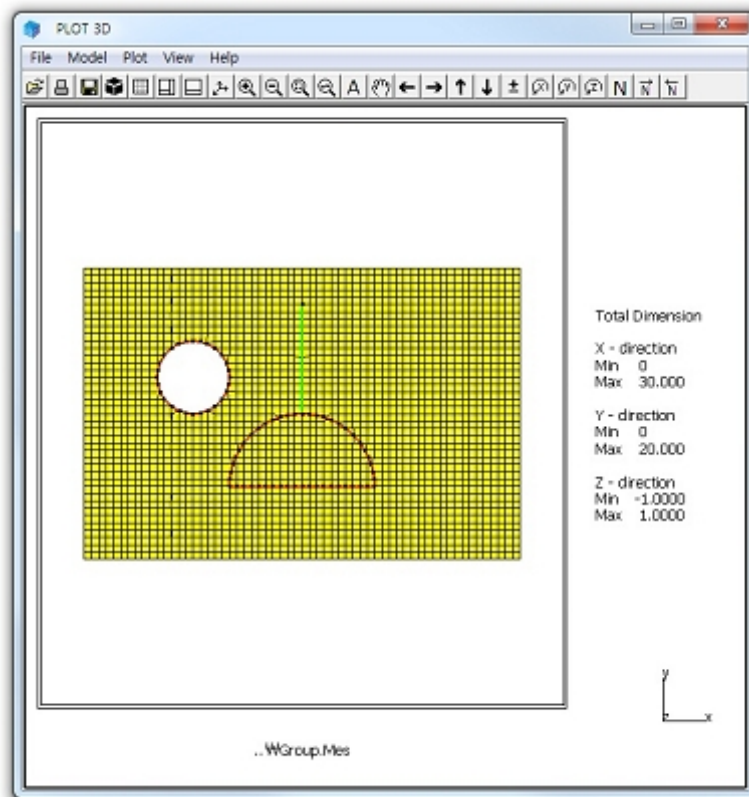


Figure 5.38 Finite element mesh plot

Step 29: Exit

Click **Exit** button in **Group** dialog.

Click **OK** in **Exit dialog** as in Figure 5.22.

5.2 Finite Element Mesh Modification

This example illustrates how to modify existing finite element meshes using [Mesh Generator](#).

5.2.1 Overview

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

Editing finite element meshes has three parts: [Nodal Boundary](#), [Nodal Coordinate](#) and [Element Material](#). These editing modes can be accessed from [Mesh](#) menu in [PLOT-2D](#) as shown in Figure 5.39.

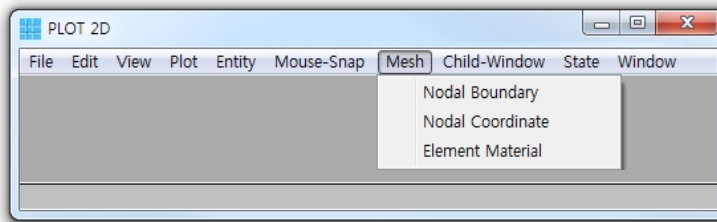


Figure 5.39 Menu for editing finite element mesh

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

Figure 5.40 shows existing finite element mesh with six layers of natural soils. The top layer of this existing mesh is to be replaced by sand embankment with reduced width as schematically shown in Figure 5.41.

This modification involves following three works:

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

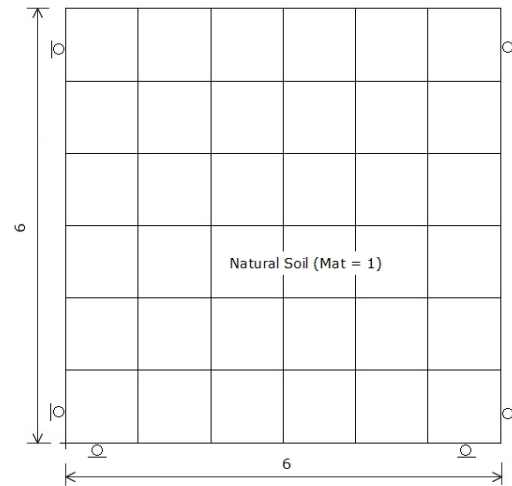


Figure 5.40 Existing finite element mesh

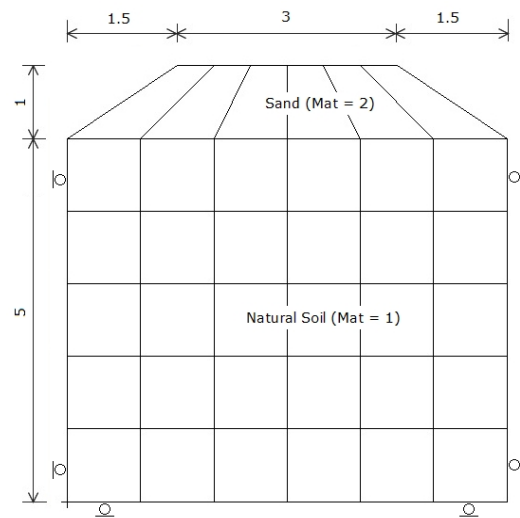


Figure 5.41 Modified finite element mesh

5.2.2 Change Top Surface Nodal Coordinates

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

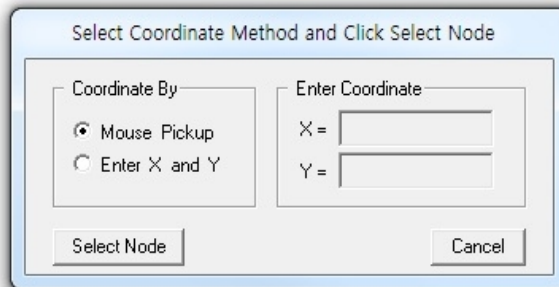


Figure 5.42 Edit coordinate dialog

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

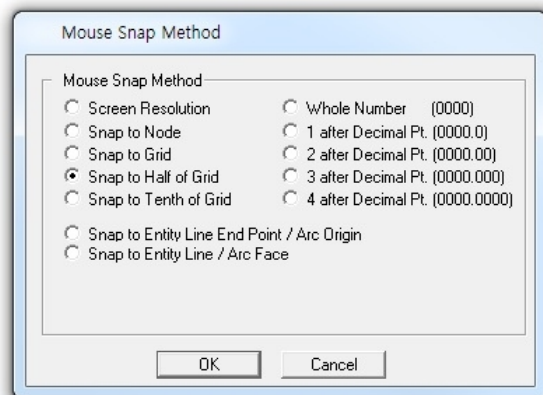


Figure 5.43 Mouse snap method

Click **Select Node** in Figure 5.42.

When you select the node by **Mouse Right Click**, the selected node is marked as an open circle on the drawing board as in Figure 5.44.

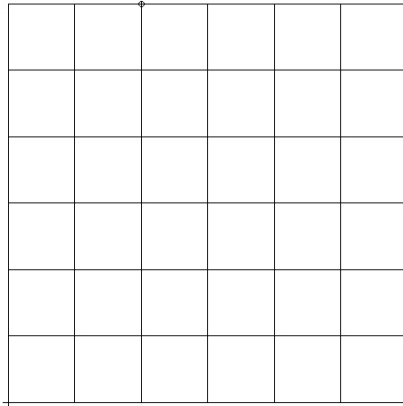


Figure 5.44 First selected node on drawing board

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

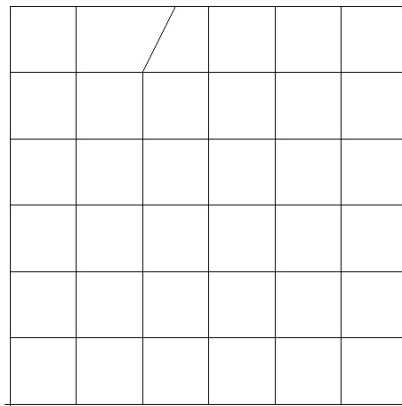


Figure 5.45 New position of first selected node

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

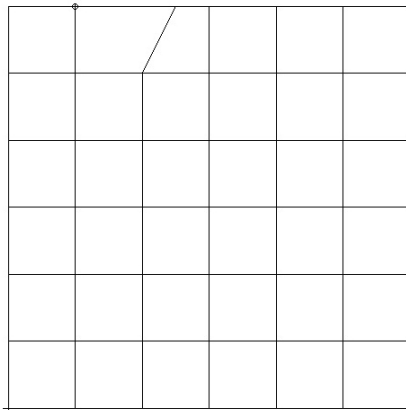


Figure 5.46 Second selected node on drawing board

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

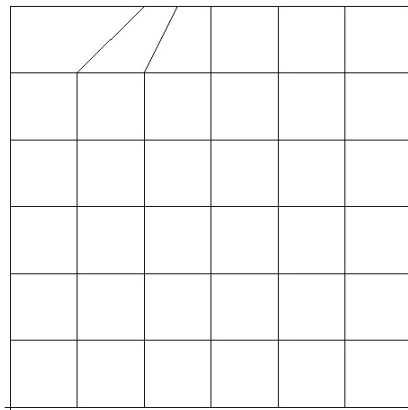


Figure 5.47 New position of second selected node

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

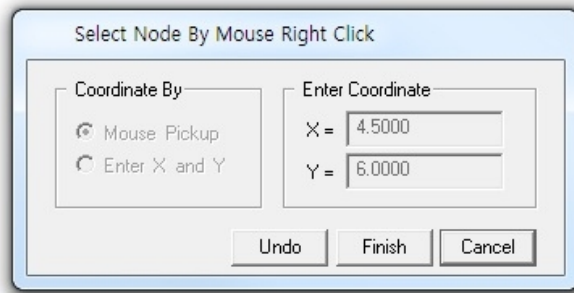


Figure 5.48 Edit coordinate dialog

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

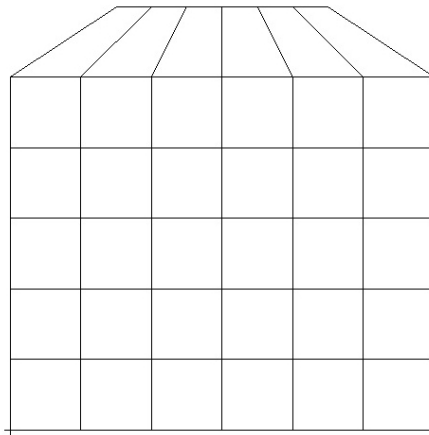
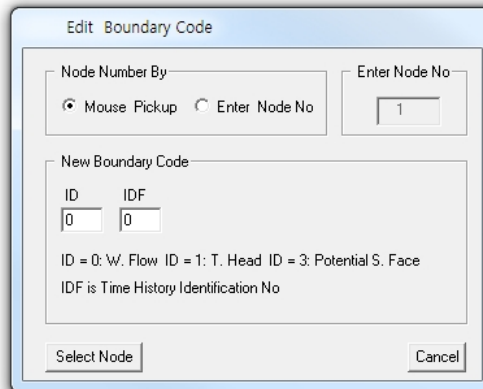


Figure 5.49 Final finite element mesh

5.2.3 Change Top Surface Nodal Boundaries

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

Figure 5.50
Edit boundary dialog

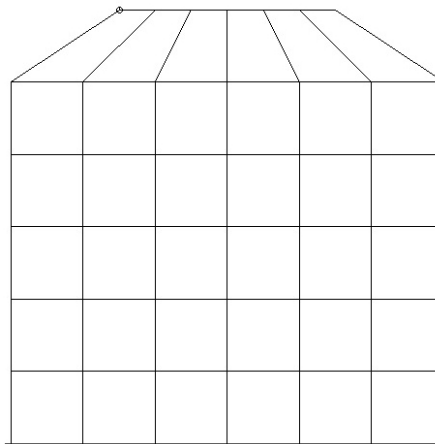


The dialog box is titled "Edit Boundary Code". It contains two main sections. The first section, "Node Number By", has two radio buttons: "Mouse Pickup" (which is selected) and "Enter Node No". To the right of these is a text box labeled "Enter Node No" containing the number "1". The second section, "New Boundary Code", contains two text boxes labeled "ID" and "IDF", both containing the number "0". Below these boxes is a legend: "ID = 0: W. Flow ID = 1: T. Head ID = 3: Potential S. Face" and "IDF is Time History Identification No". At the bottom of the dialog are two buttons: "Select Node" and "Cancel".

Click **Select Node** in Figure 5.50.

When you select the node by **Mouse Right Click**, the selected node is marked as an open circle on the drawing board as in Figure 5.51.

Figure 5.51
Selected node on drawing board



Change the boundary codes as in Figure 5.52 so that the top left node is specified by total head and then click [Apply Code](#) button.

Figure 5.52
Modified boundary code for
top left node

The dialog box is titled "Select Node By Mouse Right Click". It contains two sections. The first section, "Node Number By", has two radio buttons: "Mouse Pickup" (selected) and "Enter Node No". To the right of these is a text box labeled "Enter Node No" containing the number "1". The second section, "New Boundary Code", contains two input fields: "ID" with the value "1" and "IDF" with the value "1". Below these fields is a legend: "ID = 0: W. Flow ID = 1: T. Head ID = 3: Potential S. Face" and "IDF is Time History Identification No". At the bottom of the dialog are two buttons: "Apply Code" and "Cancel".

In the same way, select the top right node, modify boundary codes, and click [Apply Code](#). Since all boundary codes are modified, click [Finish](#) button in Figure 5.53.

Figure 5.53
Modified boundary code for
top right node

The dialog box is titled "Select Node By Mouse Right Click". It contains two sections. The first section, "Node Number By", has two radio buttons: "Mouse Pickup" (selected) and "Enter Node No". To the right of these is a text box labeled "Enter Node No" containing the number "43". The second section, "New Boundary Code", contains two input fields: "ID" with the value "1" and "IDF" with the value "1". Below these fields is a legend: "ID = 0: W. Flow ID = 1: T. Head ID = 3: Potential S. Face" and "IDF is Time History Identification No". At the bottom of the dialog are three buttons: "Apply Code", "Finish", and "Cancel".

Click **General View** from the **View** menu. Select **Flow Boundary Code** in **General View Options** dialog as shown in Figure 5.54 and then click **OK** button. Modified flow boundary codes are shown in Figure 5.55.

Figure 5.54
General view
for flow boundary code

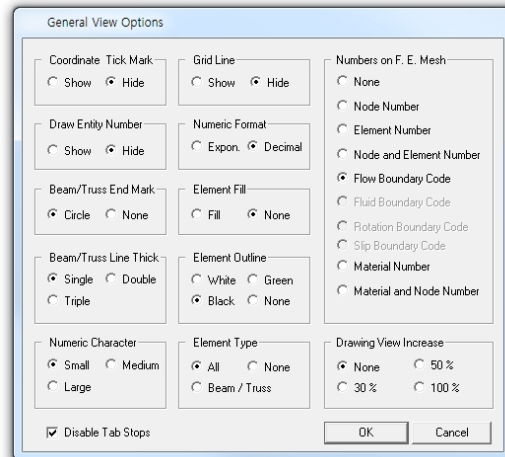
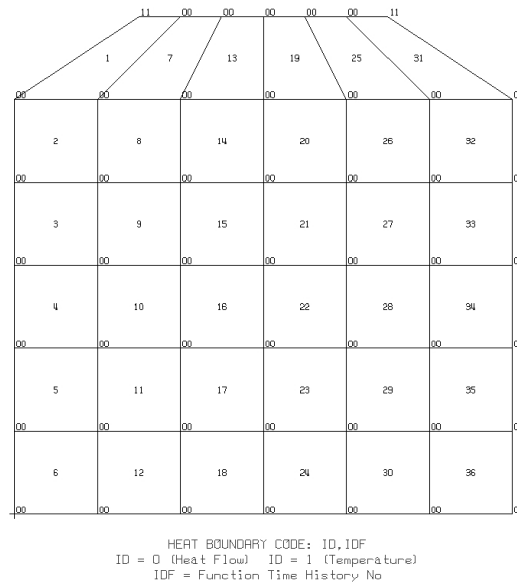


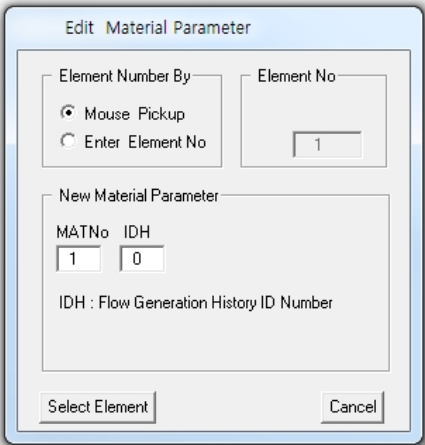
Figure 5.55
Modified flow
boundary code plot



5.2.4 Change Top Layer Element Materials

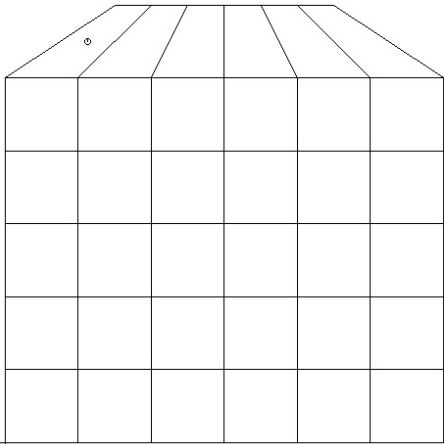
Click **Element Material** from the **Mesh** menu, then **Edit Material Parameter** dialog in Figure 5.56 is displayed.

Figure 5.56
Edit element material dialog



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

Figure 5.57
Selected element on drawing board



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

Figure 5.58
Modified material number
for element 1

The dialog box is titled "Select Element By Mouse Right Click". It contains two main sections. The first section, "Element Number By", has two radio buttons: "Mouse Pickup" (which is selected) and "Enter Element No". To the right of this section is a text box labeled "Element No" containing the value "1". The second section, "New Material Parameter", contains two input fields: "MATNo" with the value "2" and "IDH" with the value "0". Below these fields is the text "IDH : Flow Generation History ID Number". At the bottom of the dialog are two buttons: "Apply" and "Cancel".

Repeat the same procedure for the other elements on the top layer. Once finished, click **Finish** button in Figure 5.59.

Figure 5.59
Modified material number
for element 31

The dialog box is titled "Select Element By Mouse Right Click". It contains two main sections. The first section, "Element Number By", has two radio buttons: "Mouse Pickup" (which is selected) and "Enter Element No". To the right of this section is a text box labeled "Element No" containing the value "31". The second section, "New Material Parameter", contains two input fields: "MATNo" with the value "2" and "IDH" with the value "0". Below these fields is the text "IDH : Flow Generation History ID Number". At the bottom of the dialog are three buttons: "Apply", "Finish", and "Cancel".

Click **General View** from the **View** menu. Select **Material Number** in **General View Options** dialog as shown in Figure 5.60 and then click **OK** button. Modified material number is shown in Figure 5.61.

Figure 5.60
General view
for material number

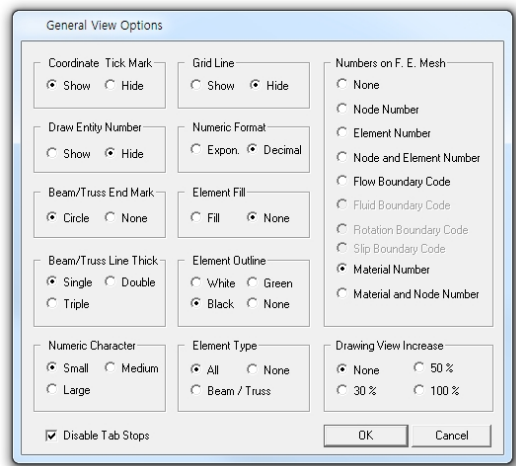
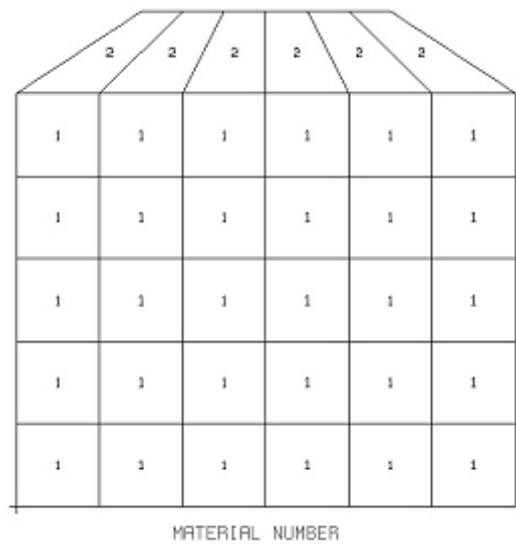


Figure 5.61
Modified material number
plot



Block Mesh Example Problem

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

Two example problems are presented:

1. [Single Element](#)
Shows step-by-step procedure to create block mesh.
2. [Square Earth Dam](#)
Builds block mesh for square earth dam.

6.1 Single Element

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

This example is to build single square element in Figure 6.1 by using block mesh generator. This single element is subjected to total head change through top surface.

This example involves following seven main steps:

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

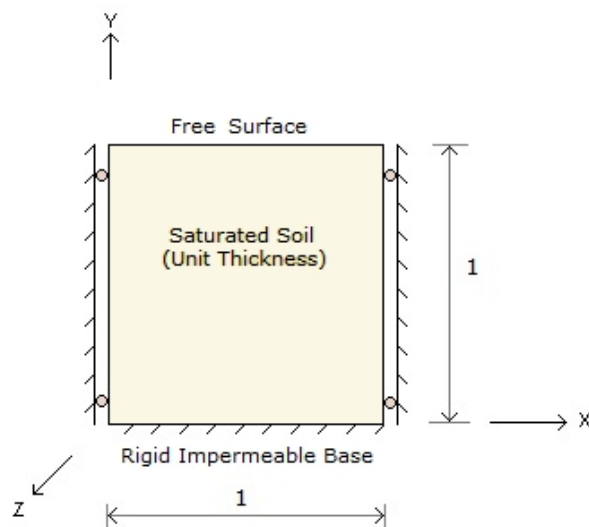


Figure 6.1 Single element

Step 1: Access Block Mesh Generator (New)

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

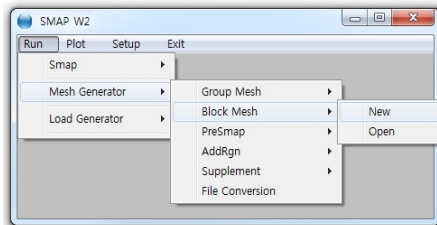


Figure 6.2 Accessing block mesh generator

Step 2: Set Work Plane

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

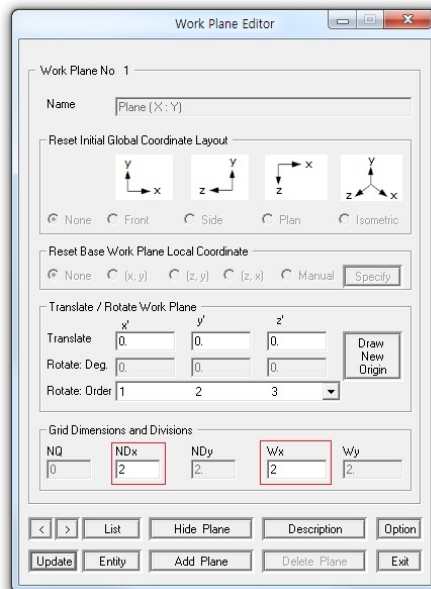


Figure 6.3 Work plane editor

Step 3: Build Cube Entity

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

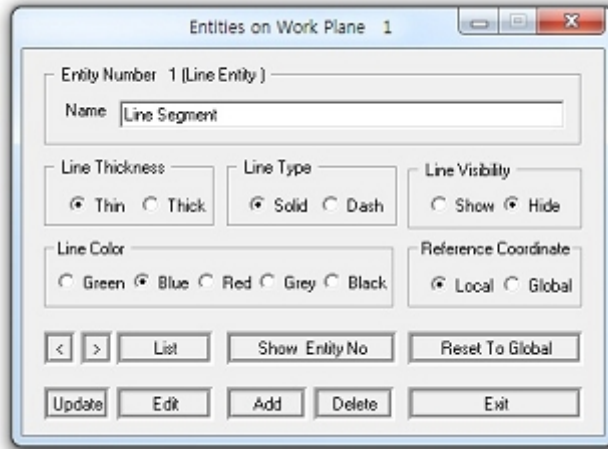


Figure 6.4 Entity editor

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

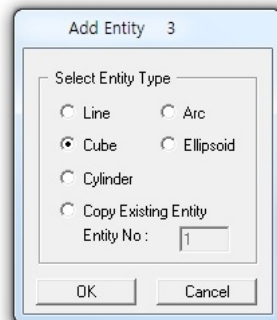


Figure 6.5 Entity type selection

5. Modify input fields of L_x , L_y , and L_z as shown in Figure 6.6.
6. Click **Draw Cube Entity** button.

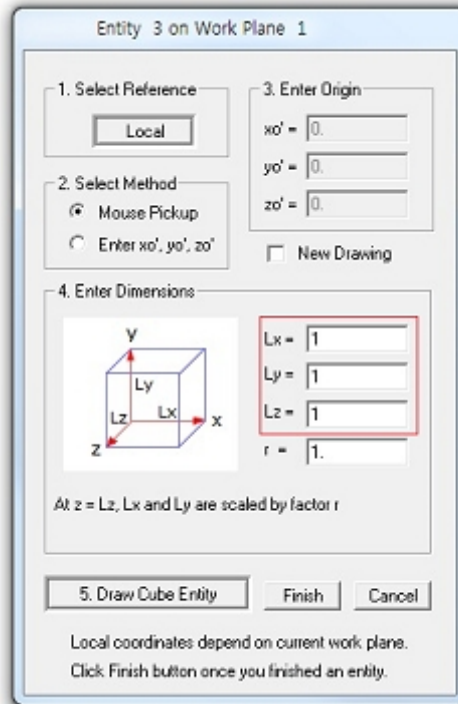


Figure 6.6 Cube entity

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

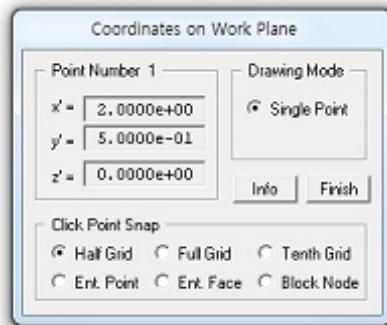


Figure 6.7 Coordinates on work plane

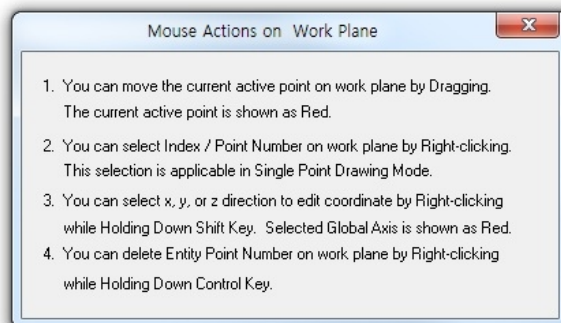


Figure 6.8 Mouse actions on work plane

9. Click **Axis** toolbar as shown in Figure 6.9.

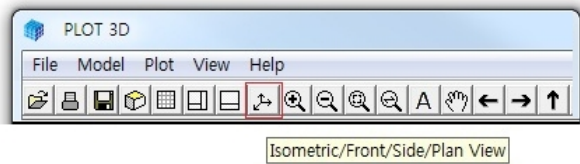


Figure 6.9 Axis toolbar

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

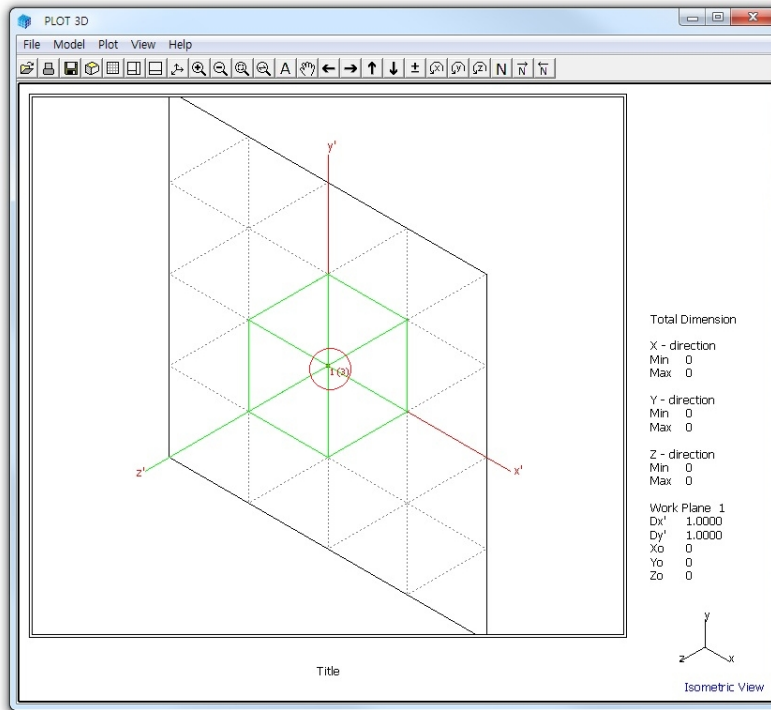


Figure 6.10 Cube entity on isometric work plane

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

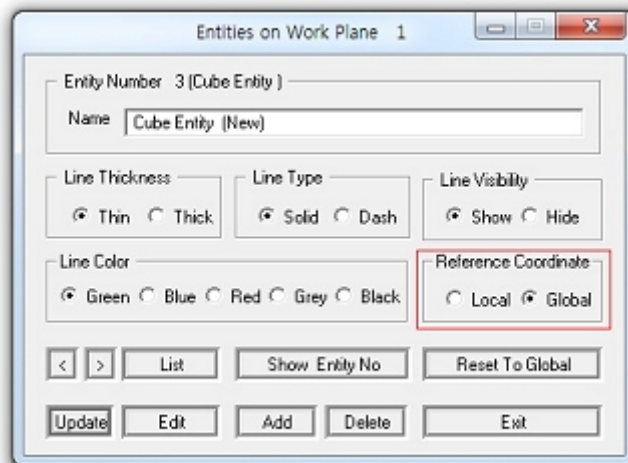


Figure 6.11 Entity editor

Step 4: Build Quad Block

1. Click **Block Editor** toolbar in Figure 6.12.

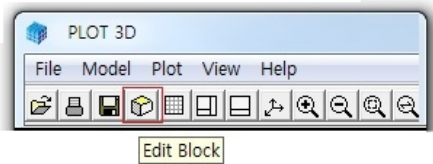


Figure 6.12 Block editor toolbar

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

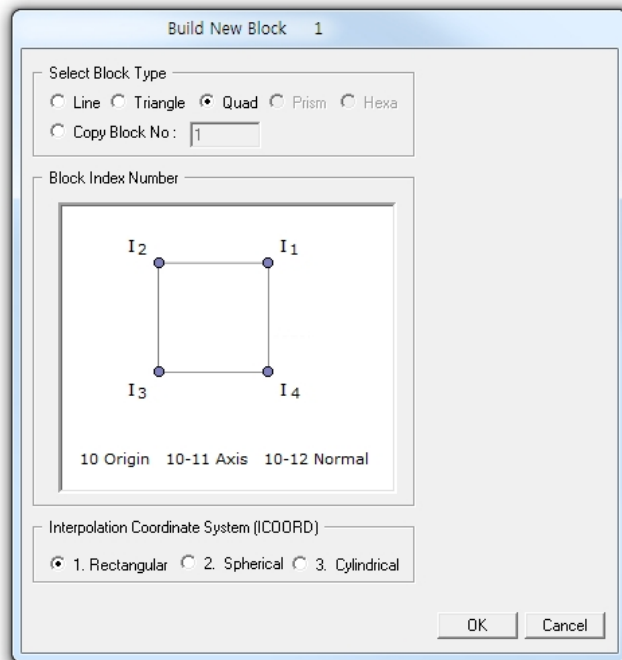


Figure 6.13 Block type selection

6-10 Block Mesh Example

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

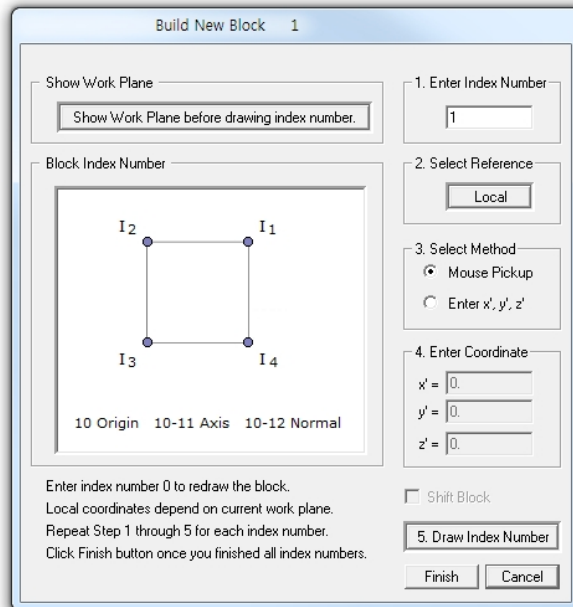


Figure 6.14 Quad block

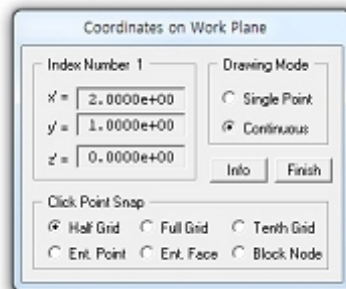


Figure 6.15 Coordinates on work plane

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

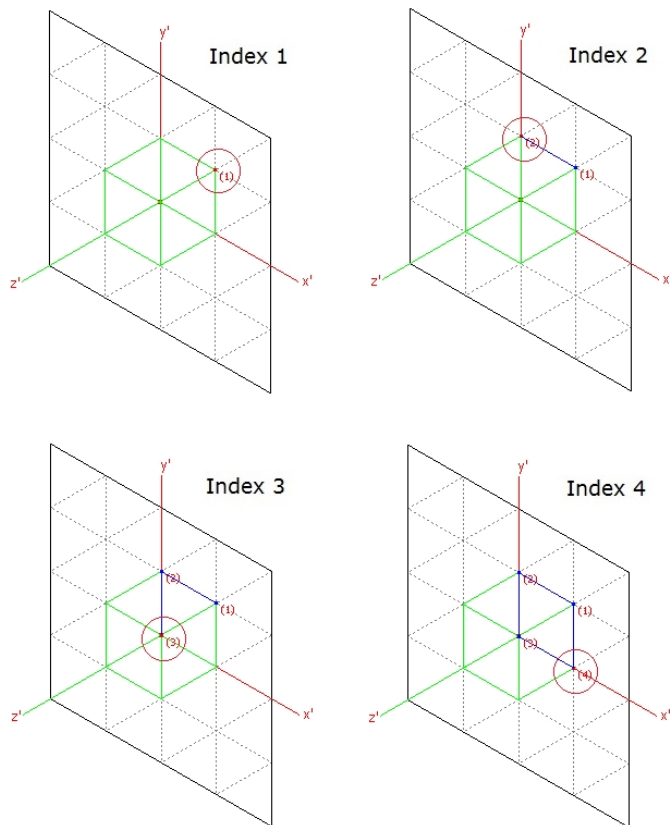


Figure 6.16 Index numbers for quad block

Now, the geometry of quad block is completed.

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

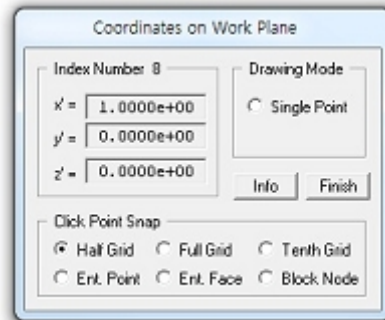


Figure 6.17 Coordinates on work plane

7. Get back to **Work Plane Editor** dialog and click **Entity**.
8. Select **Entity Number 3**, **Hide** for line visibility, click **Update**, and click **Exit** in Figure 6.18.

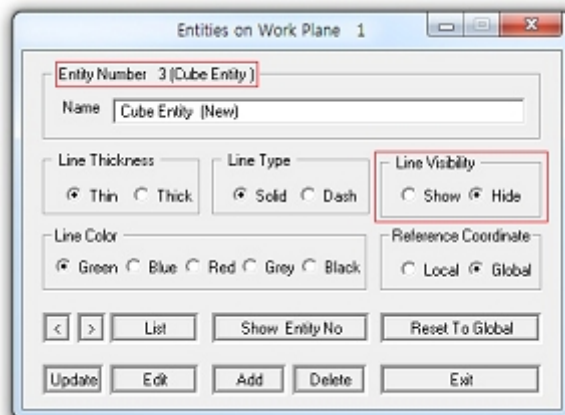


Figure 6.18 Entity editor

9. Modify **Title** and **Material & Element Generation Parameters** in **Block Editor** as shown in Figure 6.19.
10. Click **Save** and type in file name as **EX1**.

Block Editor

Title:

Block No: 1 [Quad Block]

Name:

Interpolation Coordinate System (ICoord):

☒ 1. Rectangular ☐ 2. Spherical ☐ 3. Cylindrical

Coordinate Modification (IMode):

☒ 0. Do not modify ☐ 1. Modify coordinate using node M10 as origin

Interpolation Scheme (ILAG):

☒ 0. Serendipity ☐ 1. Lagrangian ☐ 2. Surface Sector

Reference Node Numbers:

(M10) Origin. Negative value means arc shape over 180 degrees in sphere or cylinder

(M11) Defining cylinder axis M10-M11 (M12) Other cylinder axis M10-M12

Material and Element Generation Parameters:

MATNO	NDx	NDY	IDH
1	1	1	0

Mid Node: Alpha X: Alpha Y:

Figure 6.19 Block editor

Step 5: Edit Block Boundary Code

1. Click **Edit Boundary** in Figure 6.19.
2. Set the boundary codes as shown in Figure 6.20.
3. Click **IBTYPE** button to see description of boundary in Figure 6.21.
4. Click **Update** and then **OK** buttons.

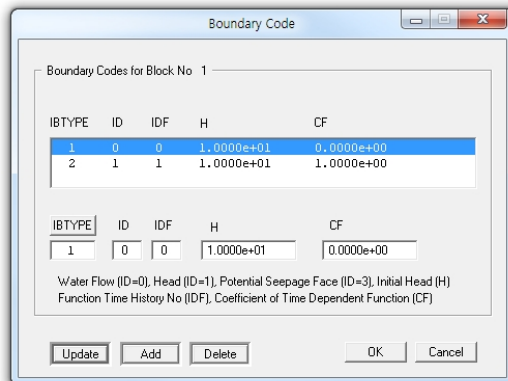


Figure 6.20 Boundary code editor

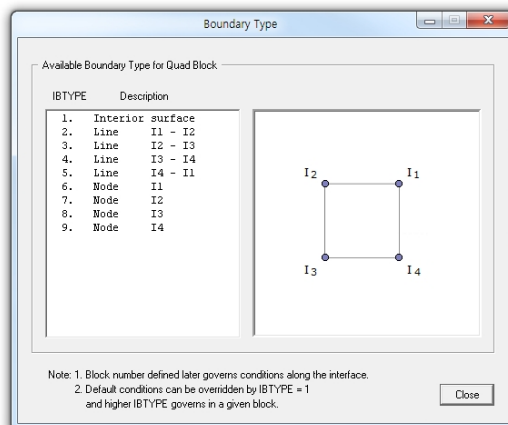


Figure 6.21 Boundary type for quad block

Step 6: View Water Flow Boundary Code

1. Select **View → General** in PLOT-3D menu.
2. Select **Water Flow Boundary Code** and click **OK** in Figure 6.22.
3. Click **Save** in Figure 6.19.

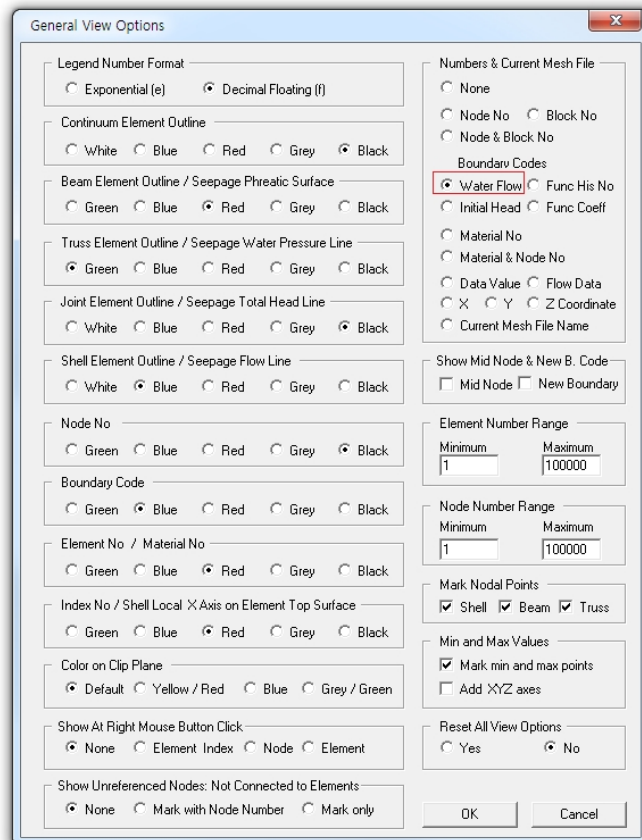


Figure 6.22 General view options

- Click **Show Numbers** toolbar as shown in Figure 6.23.



Figure 6.23 Show numbers toolbar

- Water flow boundary codes are shown in Figure 6.24.

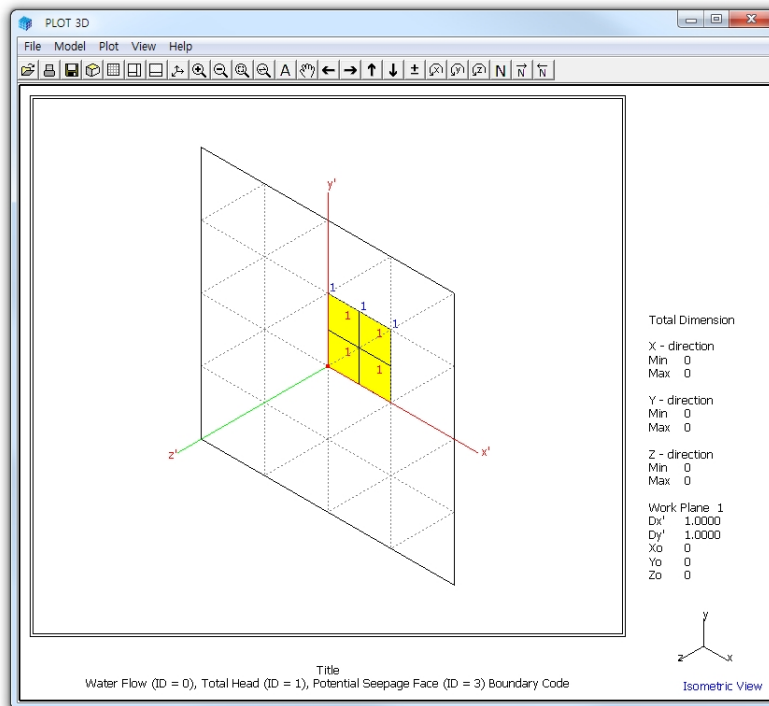


Figure 6.24 Water flow boundary codes on drawing board

Step 7: Plot Finite Element Mesh

1. Click **Show F. E. Mesh** in Figure 6.19.
2. Figure 6.25 shows the finite element mesh.

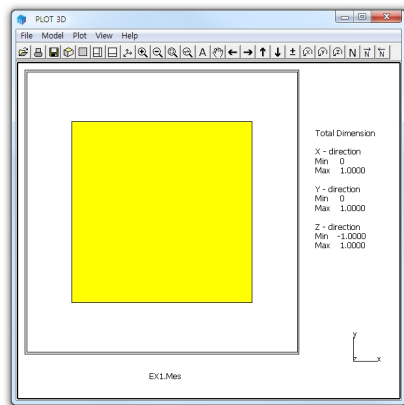


Figure 6.25 Finite element mesh

3. Follow same procedure to plot flow boundary code in Step 6.
4. Figure 6.26 shows water flow boundary code.

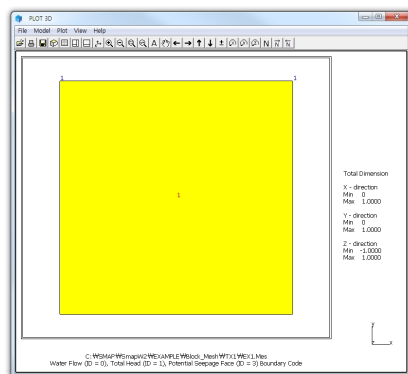


Figure 6.26 Water flow boundary codes

6.2 Square Earth Dam

This example illustrates how to build block mesh for square earth dam. Square earth dam has the dimensions of 100 x 100 units with all impermeable boundaries except total head specified on top surface.

This example has the following two parts:

Part 1: Creating Square Earth Dam(Figure 6.27)

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

Part 2: Modifying Square Earth Dam (Figure 6.28)

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

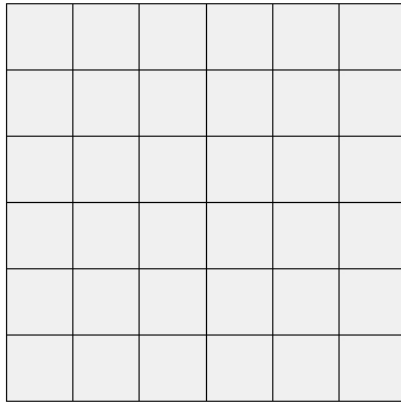


Figure 6.27 Square earth dam with constant element size

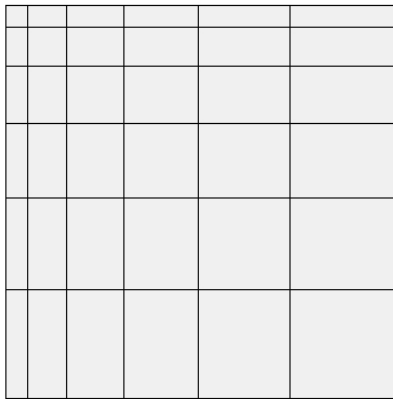


Figure 6.28 Square earth dam with variable element size

6.2.1 Part 1: Creating Square Earth Dam

Part 1 consists of the following seven main steps:

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

Step 1: Access Block Mesh Generator (New)

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

Run → Mesh Generator → Block Mesh → New

Step 2: Set Work Plane

Prebuilt [Work Plane](#) is displayed on drawing board along with [Work Plane Editor](#) dialog. Modify [NDx](#) and [Wx](#) in Figure 6.29 and click [Update](#) button.

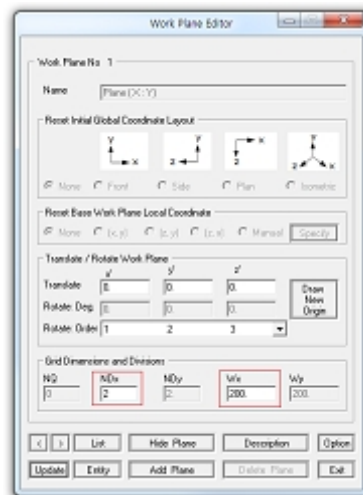


Figure 6.29 Work plane editor

Step 3: Build Quad Block

Follow the same procedure as in Step 4 in the first example.

1. Click [Axis](#) toolbar as shown in Figure 6.9.
2. Click [Block Editor](#) toolbar in Figure 6.12.
3. Select [Quad](#) for block type and click [OK](#) in Figure 6.13.
4. Click [Draw Index Number](#) in Figure 6.14.
5. [Coordinates on Work Plane](#) dialog is displayed as in Figure 6.15.

Index Numbers on Quad Block

6. Click the points for index numbers for quad block as in Fig. 6.30.

Now, the geometry of quad block is completed.

7. Click [Finish](#) in Figure 6.17.
8. Click [Finish](#) in Figure 6.14.
9. Modify [Title](#) and [Material & Element Generation Parameters](#) in [Block Editor](#) dialog as shown in Figure 6.31.

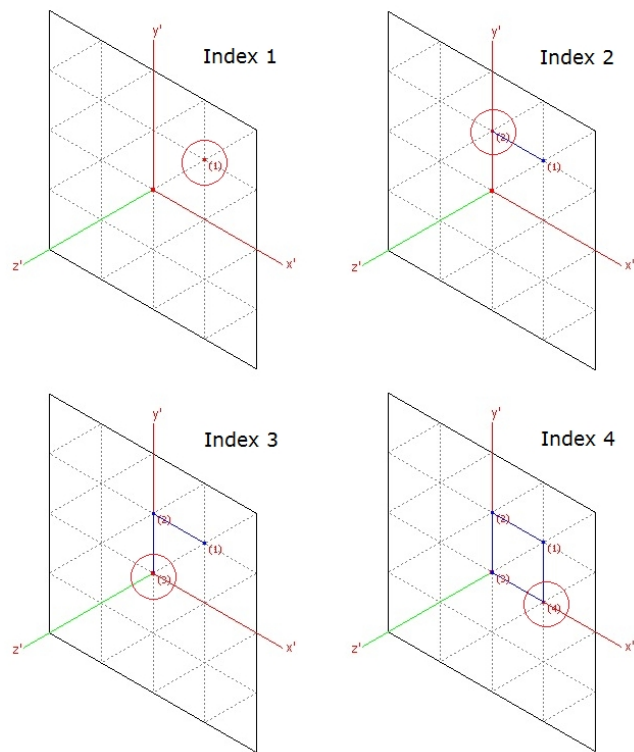


Figure 6.30 Index numbers for quad block

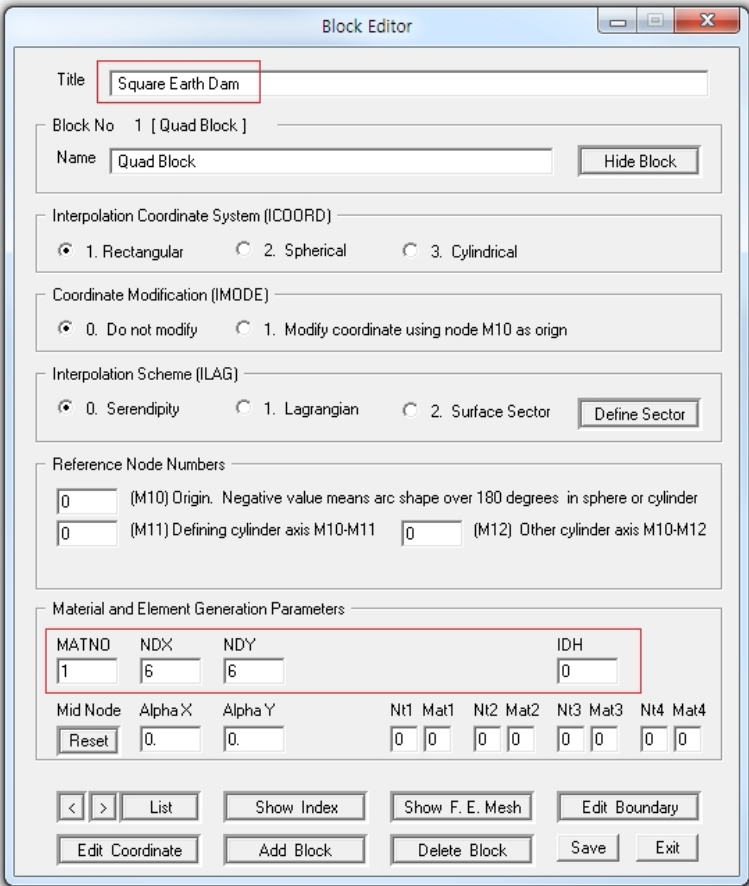


Figure 6.31 Block editor

Step 4: Edit Block Boundary Code

1. Click **Edit Boundary** in Figure 6.31.
2. Set the boundary codes as shown in Figure 6.32.
3. Click **IBTYPE** button to see description of boundary in Figure 6.33.
4. Click **Update** and then **OK** buttons in Figure 6.32.

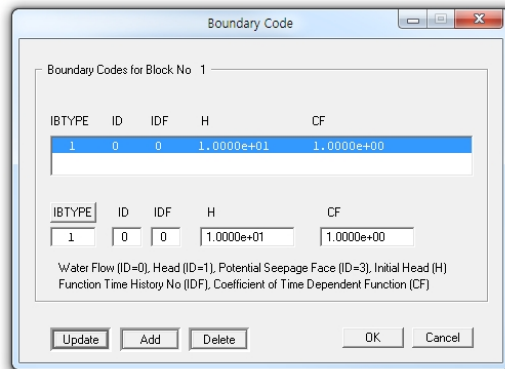


Figure 6.32 Boundary code editor

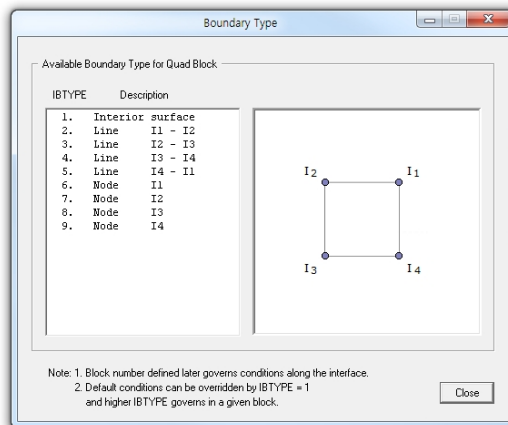


Figure 6.33 Boundary type for quad block

Step 5: Set Global Boundary Code

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

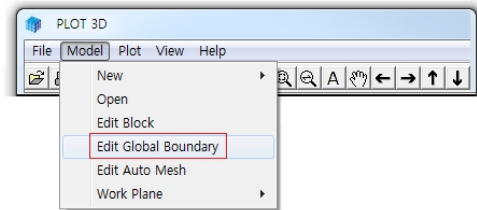


Figure 6.34 Edit global boundary menu

2. Set the boundary codes as shown in Figure 6.35.
3. Select **Yes override block boundary**.
4. Click **Save** and type in file name as **EX2**.

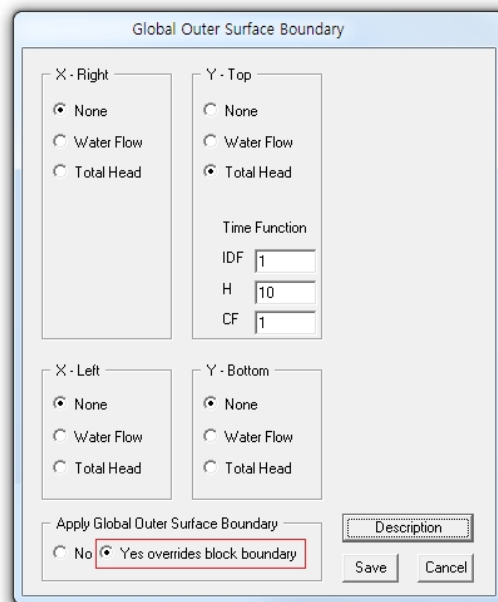


Figure 6.35 Global outer surface boundary

Step 6: View Water Flow Boundary Code

1. Select **View → General** in PLOT-3D menu.
2. Select **Water Flow Boundary Code** and click **OK** in Figure 6.22.
3. Click **Show Numbers** toolbar as shown in Figure 6.23.
4. Water flow boundary codes are shown in Figure 6.36.

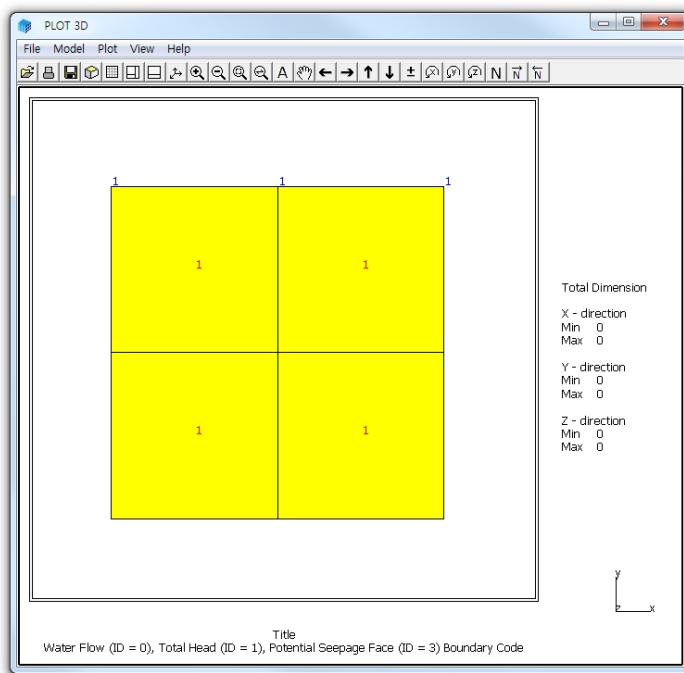


Figure 6.36 Water flow boundary codes on drawing board

Step 7: Plot Finite Element Mesh

1. Click [Show F. E. Mesh](#) in Figure 6.31.
2. Figure 6.37 shows the finite element mesh.

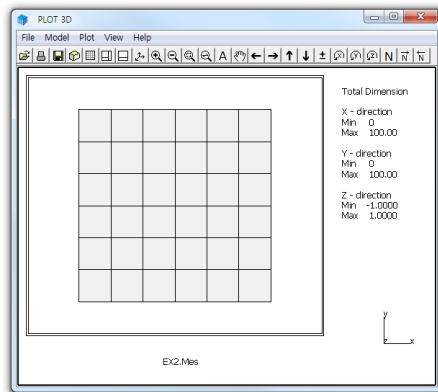


Figure 6.37 Finite element mesh

3. Follow same procedure to plot water flow boundary in Step 6.
4. Figure 6.38 shows water flow boundary code.

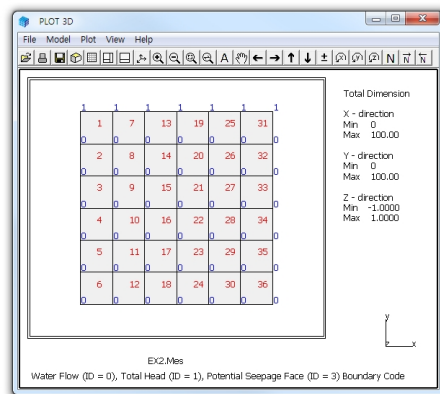


Figure 6.38 Water flow boundary codes

6.2.2 Part 2: Modifying Square Earth Dam

Part 2 consists of the following three main steps:

1. Access block mesh generator (Open)
2. Modify element generation parameters
3. Plot finite element mesh

Step 8: Access Block Mesh Generator (Open)

1. Access [Block Mesh Generator](#) by selecting the following menu items in [SMAP](#) (Figure 6.2):
[Run](#) → [Mesh Generator](#) → [Block Mesh](#) → [Open](#)
2. Click [Browse](#) button in [Open Input File](#) dialog in Figure 6.39.
3. Select the input file [EX2.Meb](#) generated in Part 1.

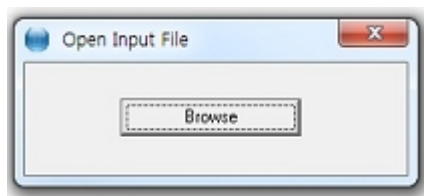


Figure 6.39 Open input file

Step 9: Modify Element Generation Parameters

1. Click **Block Editor** toolbar in Figure 6.12.
2. Modify **Alpha X**, **Alpha Y** as in Figure 6.40.
3. Click **Reset**.
4. Click **Save**.

The screenshot shows the 'Block Editor' window with the following settings:

- Title:** Square Earth Dam
- Block No:** 1 [Quad Block]
- Name:** Quad Block
- Interpolation Coordinate System (ICoord):** 1. Rectangular (selected)
- Coordinate Modification (IMode):** 0. Do not modify (selected)
- Interpolation Scheme (ILAG):** 0. Serendipity (selected)
- Reference Node Numbers:**
 - (M10) Origin: 0
 - (M11) Defining cylinder axis M10-M11: 0
 - (M12) Other cylinder axis M10-M12: 0
- Material and Element Generation Parameters:**

MATNO	NDX	NDY	IDH	N1	Mat1	N2	Mat2	N3	Mat3	N4	Mat4
1.	6	6	0	0	0	0	0	0	0	0	0

Below the table, the 'Mid Node' section is highlighted with a red box:

Mid Node	Alpha X	Alpha Y
Reset	0.3	0.3

Buttons at the bottom include: < > List, Show Index, Show F. E. Mesh, Edit Boundary, Edit Coordinate, Add Block, Delete Block, Save, and Exit.

Figure 6.40 Block editor

6-30 Block Mesh Example

5. Click **Work Plane** toolbar and then click **Show Plane** button.
6. Block mesh is shown in Figure 6.41.

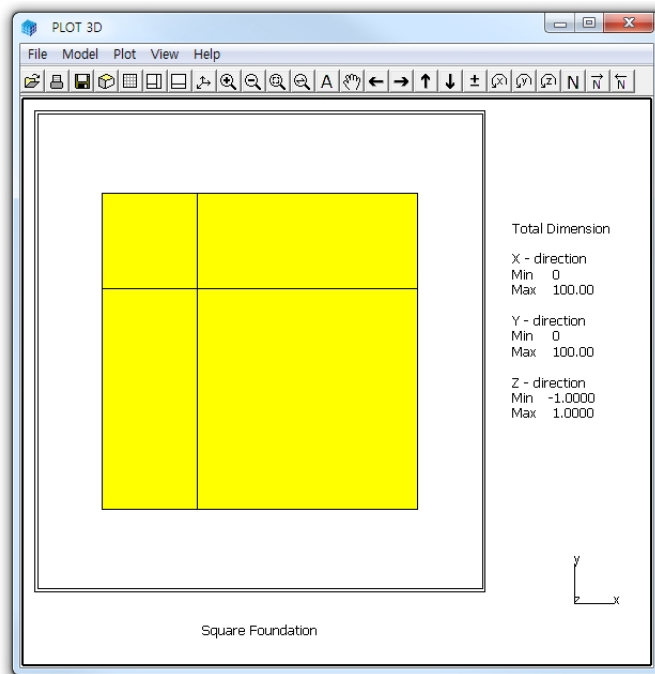


Figure 6.41 Block mesh on drawing board

Step 7: Plot Finite Element Mesh

1. Click **Show F. E. Mesh** in Figure 6.40.
2. Figure 6.42 shows the finite element mesh.

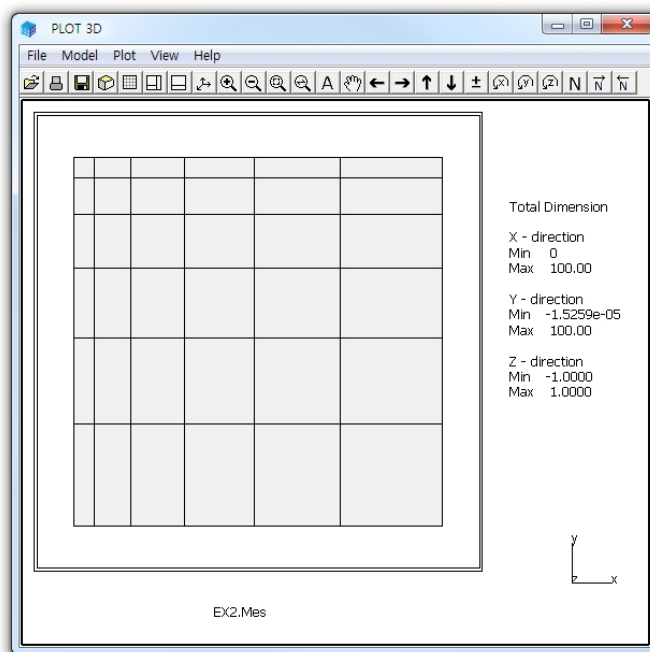


Figure 6.42 Finite element mesh

PRESMAP

Example Problem

PRESMAP menu includes six Pre-Processing programs: PRESMAP-2D, NATM-2D, CIRCLE-2D, PRESMAP-3D, CROSS-3D, GEN-3D, and PRESMAP-GP. These Pre-Processing programs are mainly used to generate Mesh File described in Section 4.3 of User's Manual. Refer to SMAP-W2 User's Manual:

- Section 5 for input parameters for PRESMAP programs.
- Section 3.2.2 for running PRESMAP programs.

7.1 PRESMAP-2D

PRESMAP-2D includes **Model 1**, **2**, **3**, and **4**. **Model 1** is basic pre-processor which can be applied to model various types of problem geometry.

Model 2 is the special pre-processor developed to model near-field around underground openings such as tunnels, culverts, etc. **Model 3** is the special pre-processor developed to model triangular and rectangular shape geometry. **Model 4** is the useful pre-processor to generate layered embankments having slope.

7.1.1 Model 1

A typical underground tunnel is chosen here to illustrate mesh generations using [PRESMAP-2D Model 1](#) and [2](#). Figure 7.1 shows geological condition around tunnel consisting of four layers: weathered soil, weathered rock, soft rock, and hard rock. Figure 7.2 shows in detail tunnel cross section including shotcrete and rock bolt dimensions.

For convenience, the tunnel problem geometry is divided into three regions as shown in Figure 7.3; Core, Near-field, and Far-field regions. By symmetry, only right half of the tunnel geometry is considered.

[Model 1](#) is used to generate Core and Far-field region meshes. And [Model 2](#) is used to generate Near-field region mesh. Near-field region mesh generation will be explained in the next section. And assembly of Core, Near-field, and Far-field regions will be explained in ADDRGN-2D Example Problems in Section 8.1.

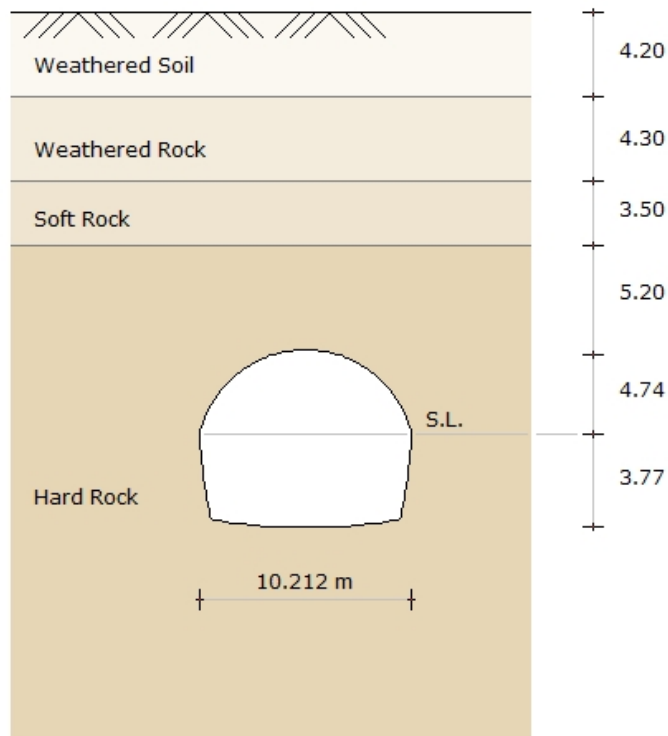
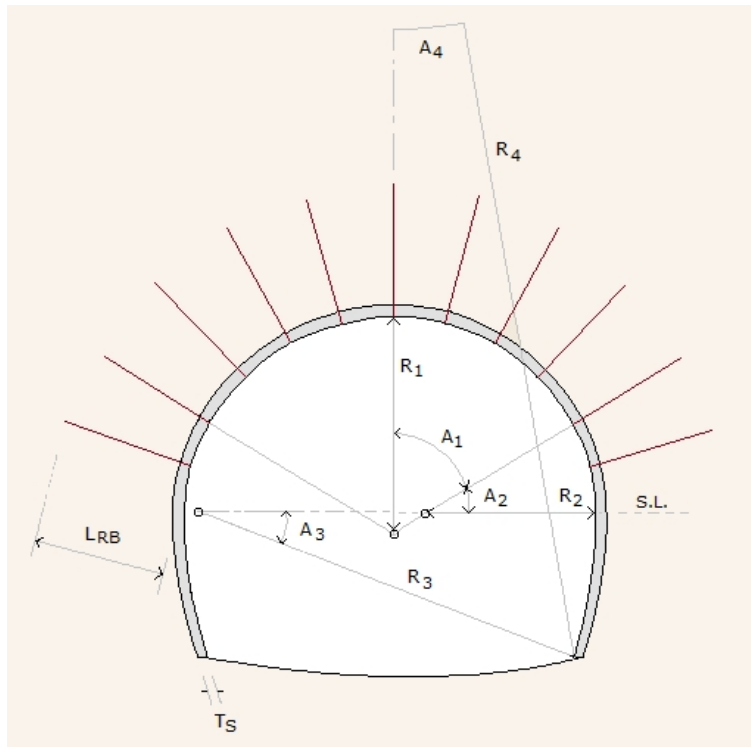


Figure 7.1 Geological condition



$$\begin{aligned} R_1 &= 5.24 \text{ m} & A_1 &= 60^\circ \\ R_2 &= 4.24 \text{ m} & A_2 &= 30^\circ \\ R_3 &= 9.86 \text{ m} & A_3 &= 19.781^\circ \\ R_4 &= 23.86 \text{ m} \end{aligned}$$

Number of Rock Bolts (NUMRB)	= 11
Length of Rock Bolts (LRB)	= 3.0 m
Spacing of Rock Bolts (TSPACING)	= 1.2 m
Thickness of Shotcrete (TS)	= 15 Cm
Thickness of Liner (TL)	= 30 Cm
Reinforcing Bar Area (ASI)	= 22 Cm ²
Reinforcing Bar Area (ASO)	= 22 Cm ²

Figure 7.2 PD-2 tunnel section detail

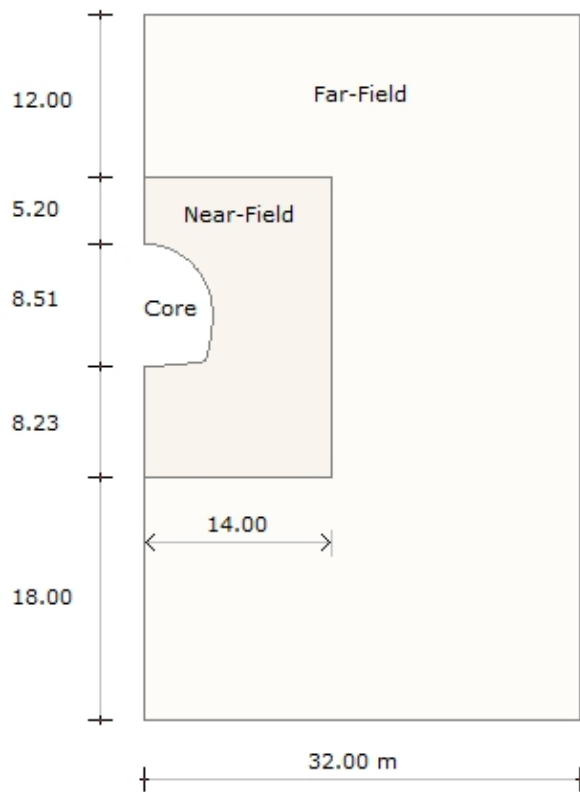


Figure 7.3 Core, Near-field, and Far-field regions

7.1.1.1 Core Region Mesh Generation

Figure 7.4 shows the block diagram for the Core region. Three blocks are used in the horizontal direction (NBX=3) and four blocks in the vertical direction (NBY=4). Block numbers should be in order from top to bottom and left to right. Top 9 blocks (Block numbers 1,2,3,5,6,7,9,10, and 11) represent upper half of tunnel core to be excavated first and bottom 3 blocks (Block numbers 4,8, and 12) represent lower half of tunnel core to be excavated later.

Each block can be consisted of 4 to 9 block nodes depending on whether you can include side and center block nodes. For those blocks facing the tunnel wall of the Core region, side block nodes are included to form the curve. Note that when the side block node is not specified, the straight line will be formed along that side.

Block index should be specified in counterclockwise. For example, the index of Block 4 can be written as $I_1=11$, $I_2=4$, $I_3=5$, $I_4=12$, $M_5=0$, $M_6=0$, $M_7=7$, $M_8=0$, $M_9=0$. Next, each block is further divided into elements. For example, Block 4 has 2 elements in the horizontal direction (NDX=2) and 6 elements in the vertical direction (NDY=6). It should be noted that to be compatible, the same number of divisions be specified along the two adjacent blocks. For example, Blocks 4, 8, and 12 have 6 elements in the vertical direction so that the generated elements can share the same nodal points along the boundaries of these blocks.

Since the tunnel is symmetry about y axis, the boundary condition along the y axis is specified as the roller which allows the displacement in the y direction and the boundary condition at all other nodes is specified to be free. And material number.4 representing hard rock is specified for all blocks since the Core region belongs to the hard rock layer as shown in Figure 7.1.

Table 7.1 shows the listing of input file, [CORE.Rgn](#), which has been prepared according to the [PRESMA-2D Model 1](#) in Section 7.2.1 of User's Manual. Note that the format of the [PRESMA-2D](#) output file is the same as that of Mesh File in SMAP-W2 User's Manual. Graphical outputs are shown in Figure 7.5.

Table 7.1 Listing of input file CORE.Rgn

```

* INPUT DATA FOR PRESMAP-2D MODEL 1
* CARD 1.1
  PD-2 CORE REGION GENERATION
* CARD 1.2
* IP
  0
* CARD 1.3
* NBLOCK  NBNODE  NSNEL  CMFAC  TEMPI
   12      30      1      1.0  10.
* CARD 1.4
* NBX  NBY  MIDX  MIDY  NF  NSNODE
   3    4    0    0    1    1
* CARD 2.1
* NODE  X      Y
   1    0.0    4.74
   2    0.0    3.16
   3    0.0    1.58
   4    0.0    0.0
   5    0.0   -3.77
   6    0.684  4.695
   7    0.76   -3.7579
   8    1.356  4.562
   9    1.488  2.819
  10    1.594  1.425
  11    1.702  0.0
  12    1.517  -3.722
  13    2.005  4.341
  14    2.273  -3.662
  15    2.62   4.038
  16    2.9204 2.4907
  17    3.157  1.273
  18    3.404  0.0
  19    3.025  -3.577
  20    3.19   3.66
  21    3.776  -3.47
  22    3.705  3.205
  23    4.157  2.69
  24    4.538  2.12
  25    4.783  1.623
  26    4.962  1.097
  27    5.07   0.5534
  28    5.106  0.0
  29    4.96   -1.693
  30    4.524  -3.337

```

7-8 PRESMAP-2D Example Problem

```
* =====
* CARD 3.1
* BLNAME
* BLOCK 1
* CARD 3.2
* IBLNO
* 1
* CARD 3.3
* I1 I2 I3 I4 M5 M6 M7 M8 M9
* 8 1 2 9 6 0 0 0 0
* CARD 3.5
* MATNO NDX NDY IDH
* 4 2 2 0
* CARD 3.6
* NFSIDE
* 0
* =====
* BLOCK 2
* 2
* 9 2 3 10 0 0 0 0 0
* 4 2 2 0
* 0
* =====
* BLOCK 3
* 3
* 10 3 4 11 0 0 0 0 0
* 4 2 2 0
* 0
* =====
* BLOCK 4
* 4 3.337
* 11 4 5 12 0 0 7 0 0
* 4 2 6 0
* 0
* =====
* BLOCK 5
* 5
* 15 8 9 16 13 0 0 0 0
* 4 2 2 0
* 0
```



```
* =====  
BLOCK 6  
6  
16 9 10 17 0 0 0 0 0  
4 2 2 0  
0  
* =====  
BLOCK 7  
7  
17 10 11 18 0 0 0 0 0  
4 2 2 0  
0  
* =====  
BLOCK 6  
6  
16 9 10 17 0 0 0 0 0  
4 2 2 0  
0  
* =====  
BLOCK 7  
7  
17 10 11 18 0 0 0 0 0  
12 12 12 12 12 12 12 12 12  
4 2 2 0  
0  
* =====  
BLOCK 8  
8  
18 11 12 19 0 0 14 0 0  
4 2 6 0  
0  
* =====  
BLOCK 9  
9  
22 15 16 24 20 0 0 23 0  
4 2 2 0  
0
```

7-10 PRESMAP-2D Example Problem

```
* =====  
BLOCK 10  
10  
24 16 17 26 0 0 0 25 0  
4 2 2 0  
0  
* =====  
BLOCK 11  
11  
26 17 18 28 0 0 0 27 0  
4 2 2 0  
0  
* =====  
BLOCK 12  
12  
28 18 19 30 0 0 21 29 0  
4 2 6 0  
0  
* =====
```

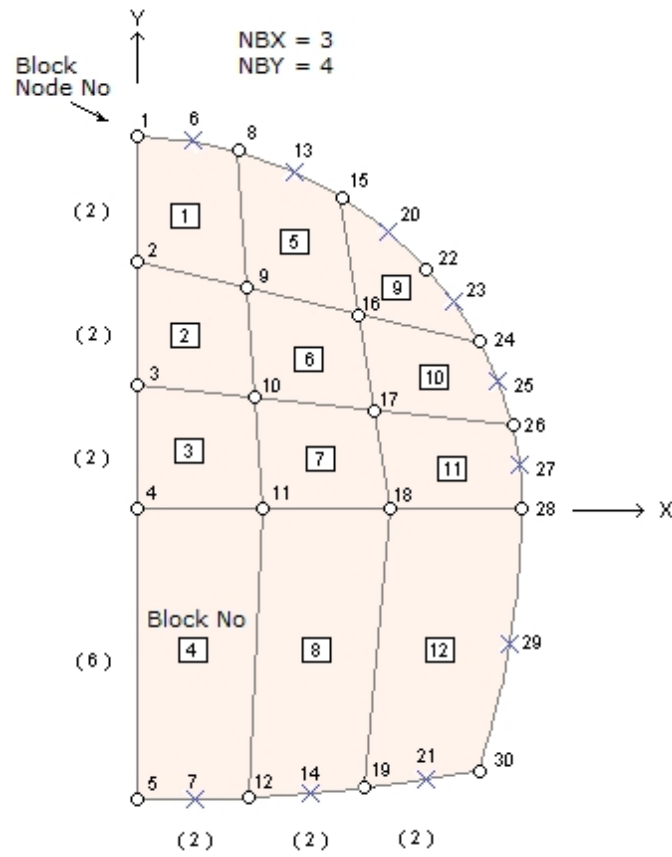


Figure 7.4 Core region block diagram

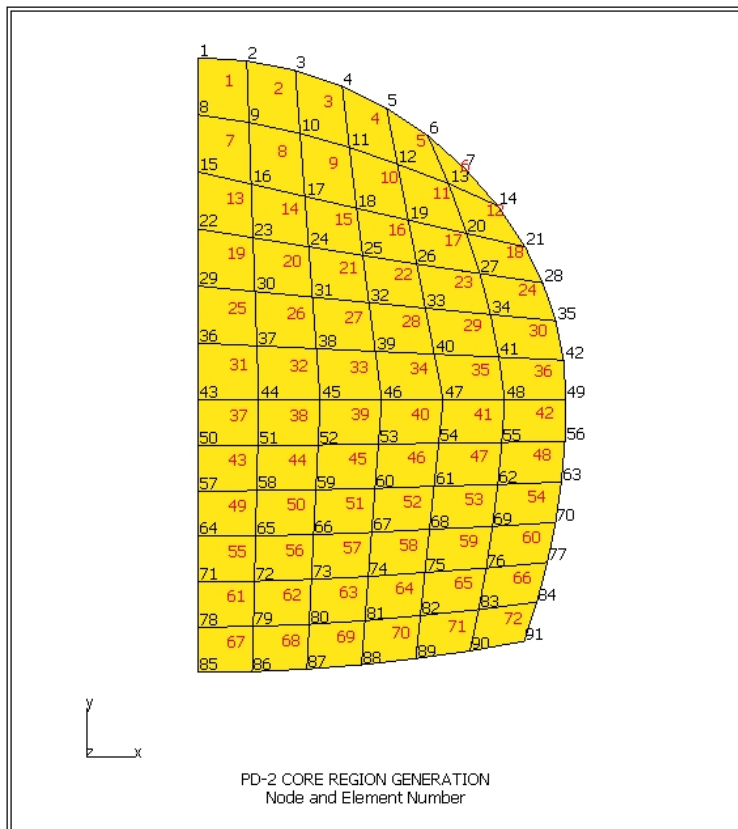


Figure 7.5 Generated element and node numbers for Core region

7.1.1.2 Far-Field Region Mesh Generation

Figure 7.6 shows the block diagram for the Far-field region. Two blocks are used in the horizontal direction ($NBX=2$) and 6 blocks in the vertical direction ($NBY=6$). Block numbers 1 and 7 represent weathered soil ($MATNO=1$). Block numbers 2 and 8 represent weathered rock ($MATNO=2$). Block numbers 3 and 9 represent soft rock ($MATNO=3$). And the rest of blocks represent hard rock ($MATNO=4$) except Block numbers 4 and 5 ($MATNO=0$). Note that Block numbers 4 and 5 are void blocks. Elements in this void blocks are not generated in Far-field region, but will be generated in Core and Near-field regions.

You can specify the index of each block as for Core region. Side block nodes are used here to make element sizes bigger as the elements are away from the tunnel core. To simulate plane strain condition at the remote boundary, boundary conditions for the left, right, and bottom are specified as the roller.

Table 7.2 shows the listing of input file, [FAR.Rgn](#), which has been prepared according to the [PRESMAP-2D Model 1](#) in Section 7.2.1 of User's Manual. Generated element and node numbers are shown in Figure 7.7. Note that the Far-field element number starts from 337, considering that there are 336 elements in Core and Near-field regions.

7-14 PRESMAP-2D Example Problem

Table 7.2 Listing of input file FAR.Rgn

```
* INPUT DATA FOR PRESMAP-2D MODEL 1
* CARD 1.1
  PD-2 FAR-FIELD REGION GENERATION
* CARD 1.2
* IP
  0
* CARD 1.3
* NBLOCK  NBNODE  NSNEL  CMFAC  TEMPI
   12      31      337    1.0   10.
* CARD 1.4
* NBX  NBY  MIDX  MIDY  NF  NSNODE
   2    6    0    0    1    1
* CARD 2.1
* NODE   X      Y
   1    0.0    21.94
   2    0.0    17.74
   3    0.0    13.44
   4    0.0     9.94
   5    0.0     0.0
   6    0.0   -12.0
   7    0.0   -19.2
   8    0.0   -30.0
   9   14.0    21.94
  10   14.0    17.74
  11   14.0    13.44
  12   14.0     9.94
  13   14.0     0.0
  14   14.0   -12.0
  15   14.0   -19.2
  16   14.0   -30.0
  17   21.2    21.94
  18   21.2    17.74
  19   21.2    13.44
  20   21.2     9.94
  21   21.2     0.0
  22   21.2   -12.0
  23   21.2   -30.0
  24   32.0    21.94
  25   32.0    17.74
  26   32.0    13.44
  27   32.0     9.94
  28   32.0     0.0
  29   32.0   -12.0
  30   32.0   -19.2
  31   32.0   -30.0
```

```
* =====
* CARD 3.1
* BLNAME
* BLOCK 1
* CARD 3.2
* IBLNO
* 1
* CARD 3.3
* I1 I2 I3 I4 M5 M6 M7 M8 M9
* 9 1 2 10 0 0 0 0 0
* CARD 3.5
* MATNO NDX NDY IDH
* 1 6 1 0
* CARD 3.6
* NFSIDE
* 0
* =====
* BLOCK 2
* 2
* 10 2 3 11 0 0 0 0 0
* 2 6 1 0
* 0
* =====
* BLOCK 3
* 3
* 11 3 4 12 0 0 0 0 0
* 3 6 2 0
* 0
* =====
* BLOCK 4
* 4
* 12 4 5 13 0 0 0 0 0
* 0 6 6 0
* 0
* =====
* BLOCK 5
* 5
* 13 5 6 14 0 0 0 0 0
* 0 6 6 0
* 0
```

7-16 PRESMAP-2D Example Problem

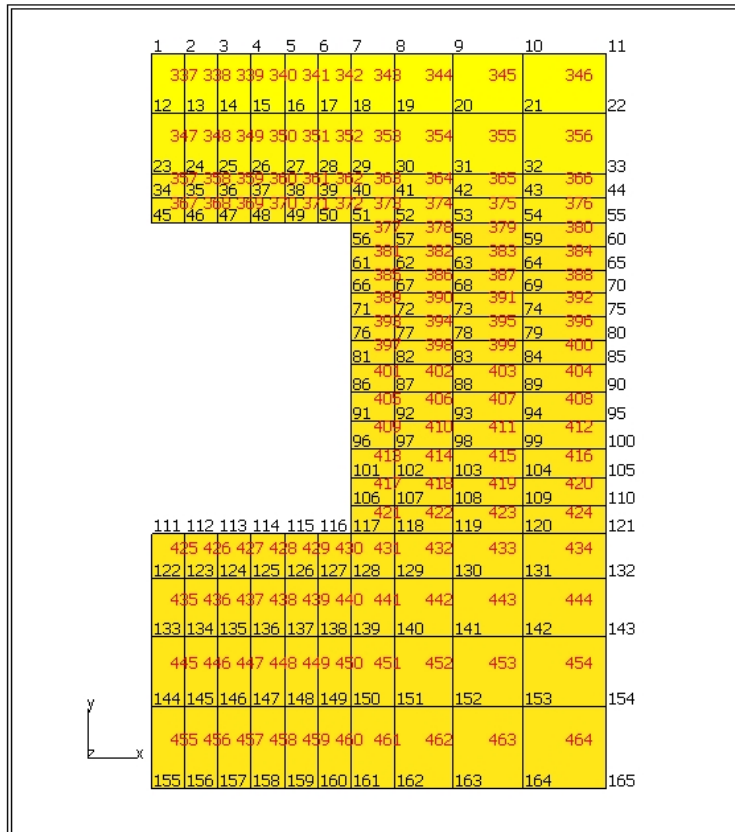
```
* =====  
BLOCK 6  
6  
14 6 8 16 0 7 0 15 0  
4 6 4 0  
0  
* =====  
BLOCK 7  
7  
24 9 10 25 17 0 18 0 0  
1 4 1 0  
0  
* =====  
BLOCK 8  
8  
25 10 11 26 18 0 19 0 0  
2 4 1 0  
0  
* =====  
BLOCK 9  
9  
26 11 12 27 19 0 20 0 0  
3 4 2 0  
0  
* =====  
BLOCK 10  
10  
27 12 13 28 20 0 21 0 0  
4 4 6 0  
0  
* =====  
BLOCK 11  
11  
28 13 14 29 21 0 22 0 0  
4 4 6 0  
0
```



```
* =====  
BLOCK 12  
12  
29 14 16 31 22 15 23 30 0  
4 4 4 0  
0  
* =====  
* END OF DATA
```



Figure 7.6 Far-field region block diagram



PD-2 FAR-FIELD REGION GENERATION
Node and Element Number

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

7.1.2 Model 2

Model 2 is the special pre-processor developed to model Near-field region around the underground openings. The Near-field region shown in Figure 7.3 is taken here as an example problem.

As shown in Figure 7.8, eight subregions are used to construct the Near-field region. And each subregion consists of three blocks. Then each block is further divided in radial and tangential directions. For example, Block number 5 in Subregion 2 has 5 elements in radial direction and 6 elements in the tangential direction. Note that element sizes in third block increase gradually in radial direction. Parameters specific to each subregion are tabulated in Table 7.3.

Table 7.4 shows the listing of input file, **NEAR.Rgn**, which has been prepared according to the **PRESMAP-2D Model 2** in Section 7.2.2 of User's Manual. Generated element mesh is shown in Figure 7.9.

Table 7.3 Parameters specific in Near-field region

NSUBR = 8 NDRF = 2 NDRS = 5 NDRT = 4
DRF = 0.15 m DRS = 2.85 m

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

Global block numbers are in order from surface

to outer edge and counterclockwise.

Local block numbers in each subregion are in order
from surface to outer edge.

Example : In Subregion 2,
First block = 4 , Second block = 5, Third block = 6

Table 7.4 Listing of input file NEAR.Rgn

```

* INPUT DATA FOR PRESMA-2D MODEL 2
* CARD 1.1
  PD-2 NEAR-FIELD MESH GENERATION
* CARD 1.2
* IP
  0
* CARD 1.3
* NSNEL  NSNODE  NF  CMFAC  TEMPI
   73      67    1   1.0   10.
* CARD 1.4
* NSURB  NDRF  NDRS  NDRT  DRF  DRS
   8      2    5    4   0.15  2.85
* =====
* CARD 2.1
* SUBNAME
  SUBREGION 1
* CARD 2.2
* ISUBNO
   1
* CARD 2.3
* ISBTYPE  LSFTYPE  NSEC
   1        1      6
* CARD 2.4.2 (LSFTYPE = 1)
* R      Xo  Yo  TA  TB
  23.86  0.0  20.09  270.  280.93
* (ISBTYPE = 1)
* CARD 2.5.3
* Xc  Yc  Xd  Yd
   0.0  -12.  14.0  -12.
* CARD 2.7
* MATNO1  IDH1
   4      0
* MATNO2  IDH2
   4      0
* MATNO3  IDH3
   4      0
* CARD
* NFSIDE
   0

```

```
* =====
SUBREGION 2
2
1 1 6
9.86 -4.754 0.0 340.22 360.
14.0 -12. 14.0 0.0
4 0
4 0
4 0
0
* =====
SUBREGION 3
3
0 1 2
4.24 0.866 0.0 0.0 15.0
1
14.0
0
14.0 3.31
4 0
4 0
4 0
0
* =====
SUBREGION 4
4
0 1 2
4.24 0.866 0.0 15.0 30.0
0
14.0 3.31
0
14.0 6.63
4 0
4 0
4 0
0
```

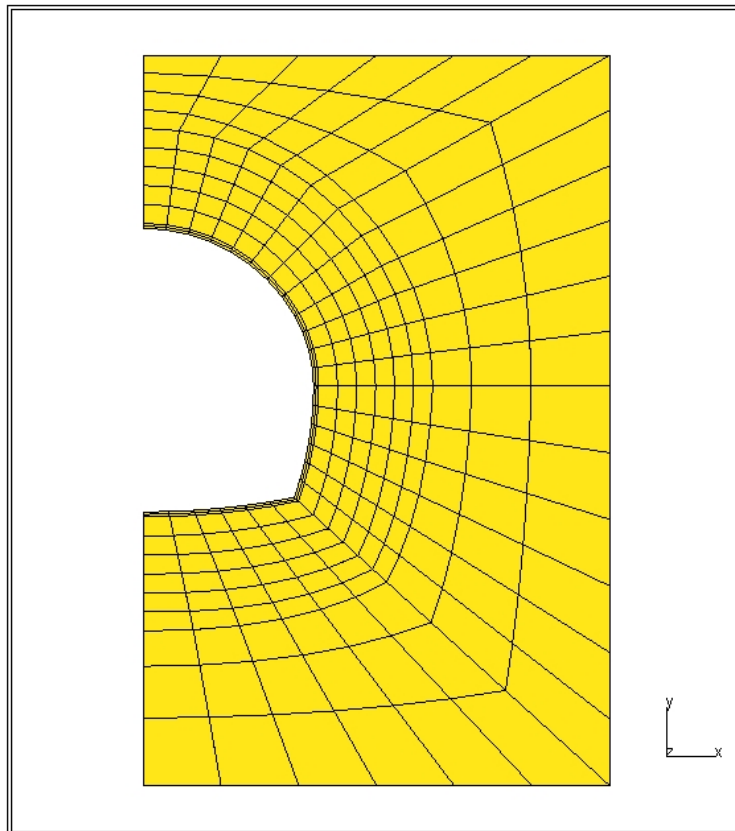
7-24 PRESMAP-2D Example Problem

```
* =====
SUBREGION 5
5
0 1 2
5.24 0.0 -0.5 30.0 45.0
0
14.0 6.63
0
14.0 9.94
4 0
4 0
4 0
0
* =====
SUBREGION 6
6
0 1 2
5.24 0.0 -0.5 45.0 60.0
0
14.0 9.94
0
9.33 9.94
4 0
4 0
4 0
0
* =====
SUBREGION 7
7
0 1 2
5.24 0.0 -0.5 60. 75.0
0
9.33 9.94
0
4.67 9.94
4 0
4 0
4 0
0
```



```
* =====  
SUBREGION 8  
8  
0 1 2  
5.24 0.0 -0.5 75.0 90.0  
0  
4.67 9.94  
0  
0.0 9.94  
4 0  
4 0  
4 0  
0  
* =====
```





PD-2 NEAR-FIELD MESH GENERATION

Figure 7.9 Generated mesh for Near-field region

7.1.3 Model 3

Model 3 is a useful pre-processor to generate triangular or rectangular meshes. It is much easier to use compared to **Models 1** and **2**. But you have to specify the boundary codes manually.

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

Table 7.5 shows the listing of input file, **GM3.Rgn**, which has been prepared according to the **PRESMAP-2D Model 3** in Section 7.2.3 of User's Manual. Generated element and node numbers are shown in Figure 7.11.

Table 7.5 Listing of input file GM3.Rgn

```

* INPUT DATA FOR PRESMAP-2D MODEL 3
* CARD 1.1
  MESH GENERATION SURROUNDING PIPE ( GM3 )
* CARD 1.2
* IP
  0
* CARD 1.3
* NBLOCK  NBNODE  NSNEL  NSNODE  CMFAC
      6      12      171      1      1.0
* CARD 2.1
* NODE      X      Y
  1  .324920E+02  .100000E+03
  2  .809020E+02  .587790E+02
  3  .100000E+03  .000000E+00
  4  .809020E+02  -.587790E+02
  5  .324920E+02  -.100000E+03
  6  100.        100.
  7  125.        50.
  8  150.        0.
  9  125.       -50.
 10  100.       -100.
 11  200.        100.
 12  175.        50.
* =====
* CARD 3.1
* =====
* IBLNO  IBLTYPE  MATNO  IDH
      1      2      2      0
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
  6  1  2  7  0  0  0  0  0  0  0  0  0  0  0  0
* =====
* IBLNO  IBLTYPE  MATNO  IDH
      2      2      2      0
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
  7  2  3  8  0  0  0  0  0  0  0  0  0  0  0  0
* =====
* IBLNO  IBLTYPE  MATNO  IDH
      3      2      2      0
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
  8  3  4  9  0  0  0  0  0  0  0  0  0  0  0  0

```

7-30 PRESMAP-2D Example Problem

```
* =====
* IBLNO  IBLTYPE  MATNO  IDH
*   4      2      2      0
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
*   9  4  5 10  0  0  0  0  0  0  0  0  0  0  0  0
* =====
* IBLNO  IBLTYPE  MATNO  IDH
*   5      2      2      0
* FOR IBLTYPE = 2
* I1 I2 I3 I4 M5 M6 M7 M8 M9 M10 M11 M12 M13 M14 M15 M16
*  11  6  7 12  0  0  0  0  0  0  0  0  0  0  0  0
* =====
* IBLNO  IBLTYPE  MATNO  IDH
*   6      4      2      0
* FOR IBLTYPE = 2
* I1 I2 I3 M4 M5 M6 M7 M8 M9 M10 M11 M12
*   7  8 12  0  0  0  0  0  0  0  0  0
* =====
```

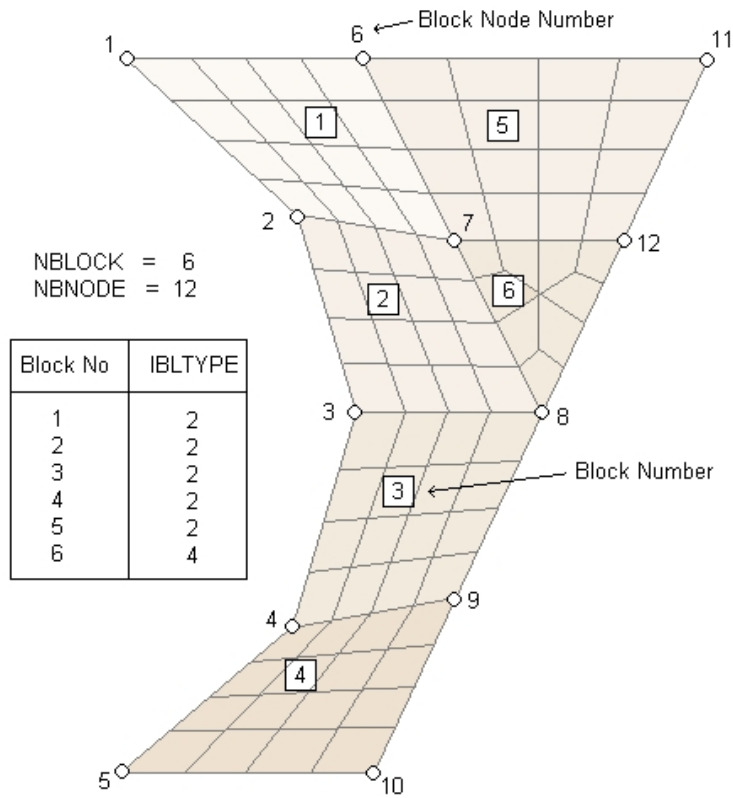
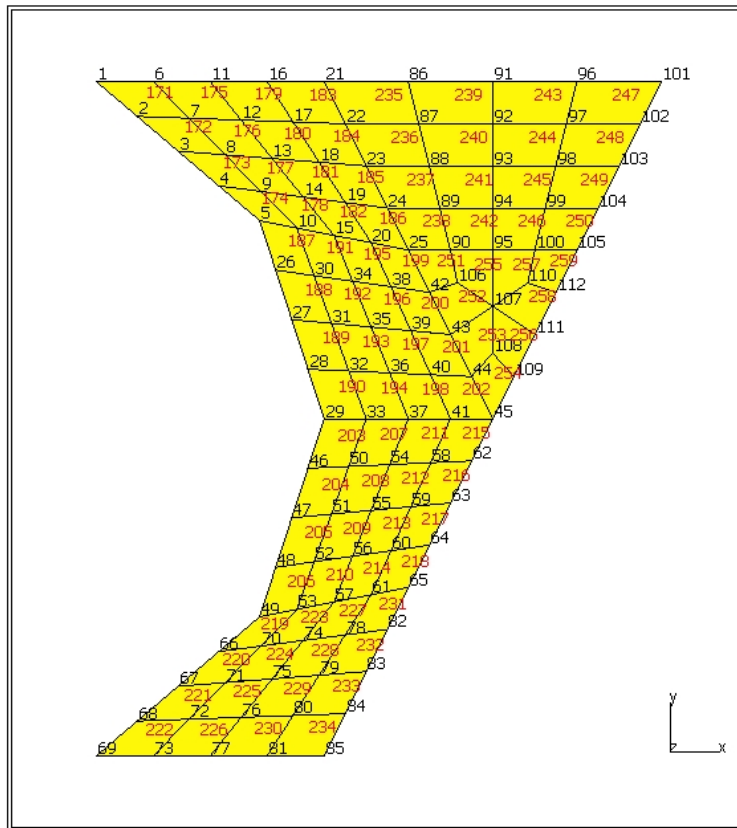


Figure 7.10 Block diagram for Model 3 example problem



MESH GENERATION SURROUNDING PIPE (GM3)
Node and Element Number

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

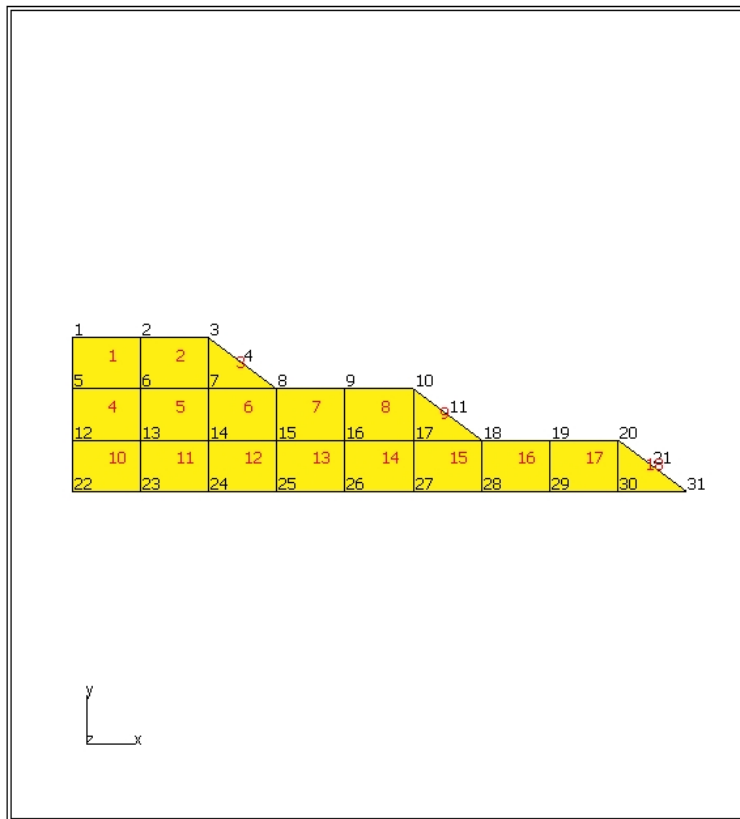
7.1.4 Model 4

Model 4 is a useful pre-processor to generate horizontally layered dams or embankments. It is easy to use but the boundary codes should be specified manually.

As **Model 4** example problem, an embankment with 3 layers is considered. Table 7.6 shows the listing of input file, **GM4.Rgn**, which has been prepared according to the **PRESMAP-2D Model 4** in Section 7.2.4 of User's Manual. Generated element and node numbers are shown in Figure 7.12.

Table 7.6 Listing of input file GM4.Rgn

```
* CARD 1.1
* TITLE
  EXAMPLE PROBLEM FOR PRESMAP-2D MODEL 4
* CARD 1.2
* NLayer  NDiv  ITRANGL
   3       3     1
* CARD 1.3
* NSNEL   NSNODE  CMFAC
   1       1      1.0
* CARD 2.1
* XB1  YB1  YB2  XB2
  0.0  3.0  0.0  12.
* CARD 3.1
* MATNO  IDH
   3      0
* END OF DATA
```



EXAMPLE PROBLEM FOR PRESMAP-2D MODEL4
Node and Element Number

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

7.2 NATM-2D

NATM-2D is the special pre-processing program to generate automatically two-dimensional finite element meshes and boundary conditions for NATM tunnels. NATM-2D has four different models:

Model 1	Single Tunnel (Half Section)
Model 2	Single Tunnel (Full Section)
Model 3	Two Tunnel (Symmetric Section)
Model 4	Two Tunnel (Unsymmetric Section)

Once you have executed NATM-2D, you will obtain following files:

<u>Output File</u>	Mesh File including all elements (Continuum, Beam, and Truss). <u>Output File</u> is the user specified name.
BEAM.Dat	Mesh File including only beam elements.
TRUSS.Dat	Mesh File including only truss elements.
AD.Dat	Card Group 8 in Main File representing default element activities for upper and lower parts of Core, Shotcrete, and Rock Bolt including Joint and Lining elements.
LINING.Dat	Mesh File for Beam-Spring Lining Analysis. This file will be generated only for ILNCOUPL=1.

A typical PD2 tunnel shape is chosen here to illustrate mesh generation using NATM-2D as shown in Figure 7.13. For each model, we will present:

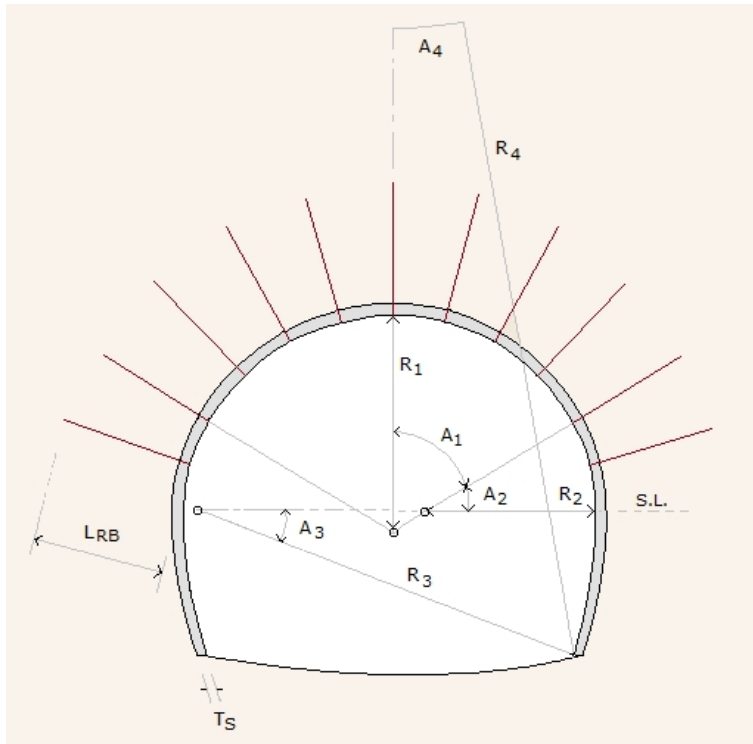
- Listing of input file
- Schematic tunnel section view
- Graphical output of finite element mesh

Table 7.7 Listing of input file PD2-1.Dat

```

* CARD 1.1
* TITLE
  NATM-2D MODEL 1 EXAMPLE PROBLEM
* CARD 1.2
* IUNIT
  2
* CARD 1.3
* MODEL  IGEN  IEXMESH  ILNCOUPL
  1      0      0          0
* CARD 2.1
* HT      HL      W      DELTAX  DELTAX  NDYMAX
  21.94  30.    20.    2.0      2.0     40
* CARD 3.1
* NLAYER
  4
* CARD 3.2
* LAYERNO  H      IDH
  1         4.2    0
  2         4.3    0
  3         3.5    0
  4        39.94   0
* CARD 4.1
* R1      A1      R2      A2  R3      A3      R4      GR      GA
  5.24   60.    4.24   30.  9.86   19.781  23.86  1.0    0.5
* CARD 4.2
* INVSHOT  TS
  0         0.3
* CARD 4.3
* NUMRB  LRB      LSPACING  TSPACING  NSRB
  11     3.0      0.8       1.2       2
* CARD 5.1
* LDTYPE  DGW  GAMAW
  0       2.0  1.0
* END OF DATA

```

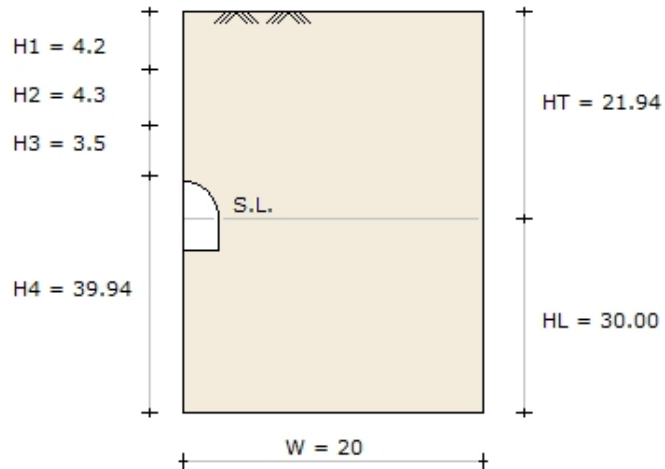


$$\begin{aligned}
 R_1 &= 5.24 \text{ M} & A_1 &= 60^\circ \\
 R_2 &= 4.24 \text{ M} & A_2 &= 30^\circ \\
 R_3 &= 9.86 \text{ M} & A_3 &= 19.781^\circ \\
 R_4 &= 23.86 \text{ M}
 \end{aligned}$$

Number of Rock Bolts (NUMRB)	= 11
Length of Rock Bolts (LRB)	= 3.0 M
Spacing of Rock Bolts (TSPACING)	= 1.2 M
Thickness of Shotcrete (TS)	= 15 Cm
Thickness of Liner (TL)	= 30 Cm
Reinforcing Bar Area (ASI)	= 22 Cm ²
Reinforcing Bar Area (ASO)	= 22 Cm ²

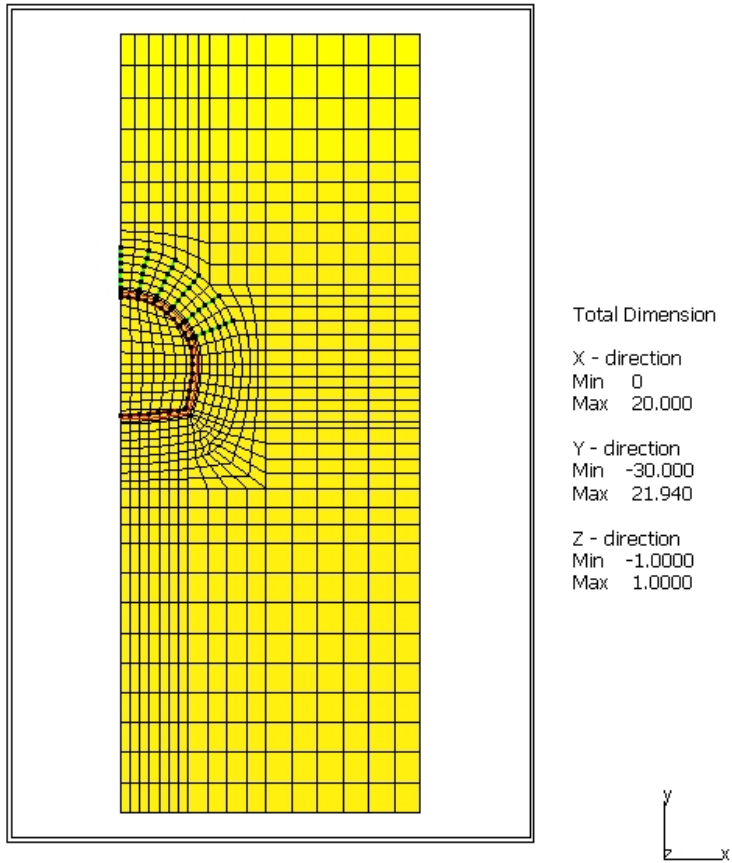
Figure 7.13 Tunnel dimensions used for example problem

MODEL=1 Single Tunnel (Half Section)



DELTA X = 2.0 DELTA Y = 2.0 NDYMAX = 40

Figure 7.14 Schematic tunnel section view for Model 1 example



NATM-2D MODEL 1 EXAMPLE PROBLEM

Figure 7.15 Generated finite element mesh for Model 1 example

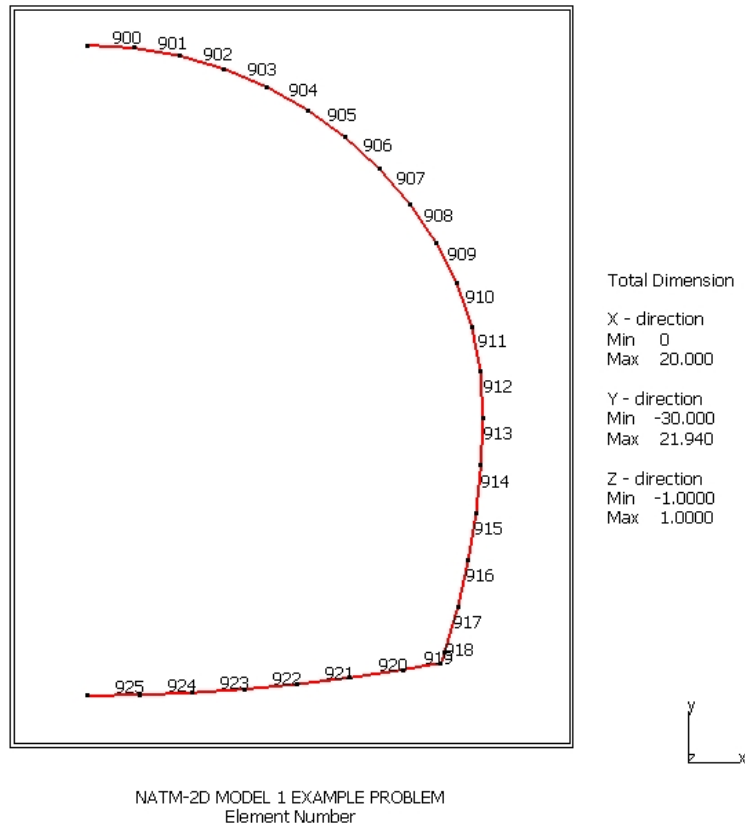


Figure 7.16 Generated beam element number for Model 1 example

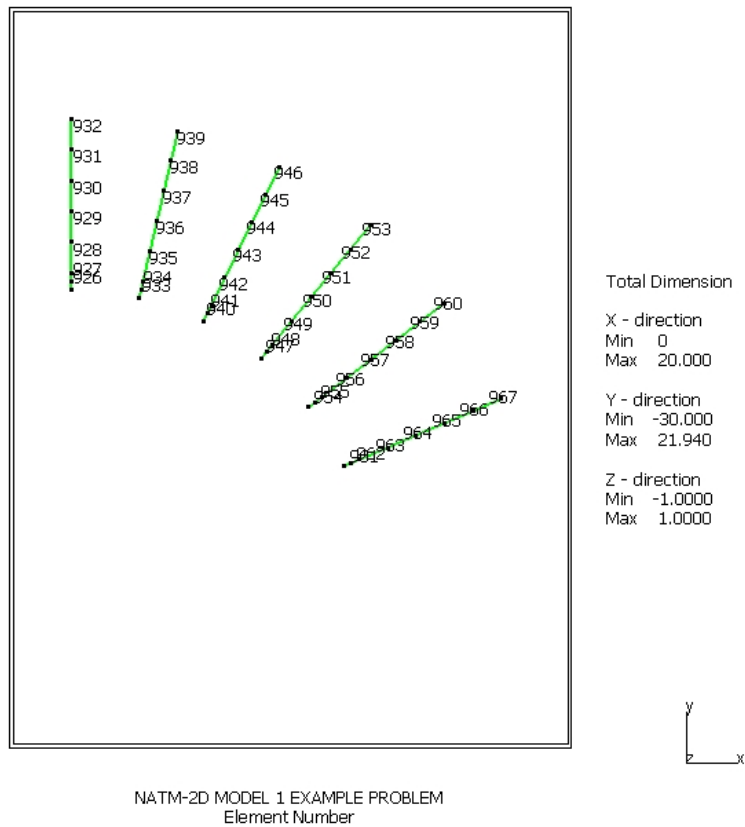


Figure 7.17 Generated truss element number for Model 1 example

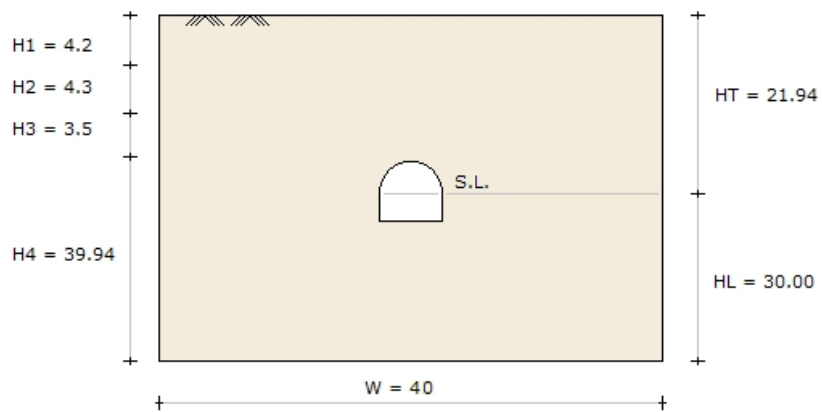
Table 7.8 Listing of input file PD2-2.Dat

```

* CARD 1.1
* TITLE
  NATM-2D MODEL 2 EXAMPLE PROBLEM
* CARD 1.2
* IUNIT
  2
* CARD 1.3
* MODEL  IGEN  IEXMESH      ILNCOUPL
  2      0      0          0
* CARD 2.1
* HT      HL      W      DELTAX  DELTAX  NDYMAX
  21.94  30.    40.    2.0      2.0      40
* CARD 3.1
* NLAYER
  4
* CARD 3.2
* LAYERNO  H      IDH
  1         4.2    0
  2         4.3    0
  3         3.5    0
  4        39.94   0
* CARD 4.1
* R1      A1      R2      A2  R3      A3      R4      GR      GA
  5.24   60.    4.24   30.  9.86   19.781  23.86  1.0    0.5
* CARD 4.2
* INVSHOT  TS
  0         0.3
* CARD 4.3
* NUMRB  LRB      LSPACING  TSPACING  NSRB
  11     3.0      0.8       1.2       2
* CARD 5.1
* LDTYPE  DGW  GAMAW
  0       2.0   1.0
* END OF DATA

```

MODEL=2 Single Tunnel (Full Section)



DELTAX = 2.0 DELTAY = 2.0 NDYMAX = 40

Figure 7.18 Schematic tunnel section view for Model 2 example

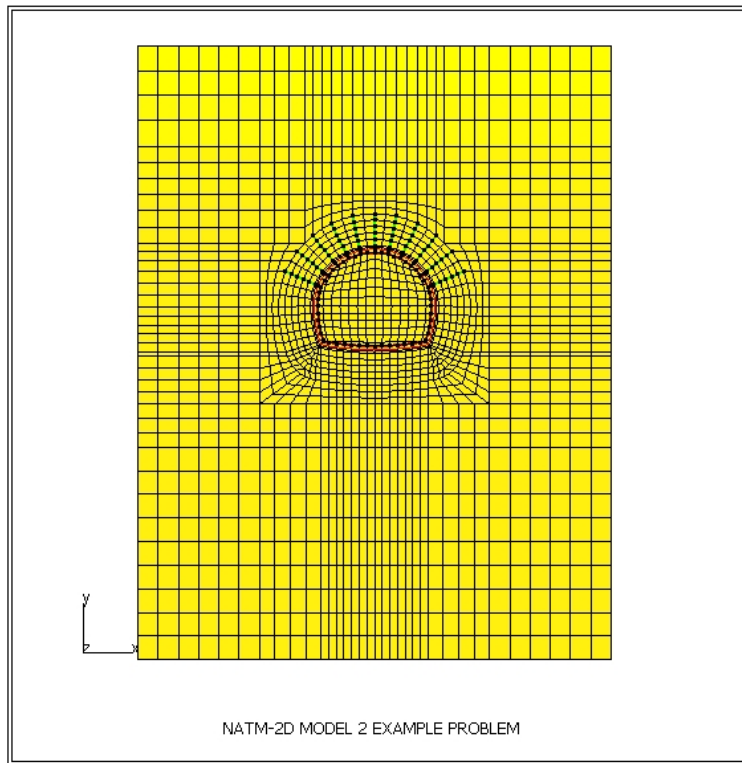


Figure 7.19 Generated finite element mesh for Model 2 example

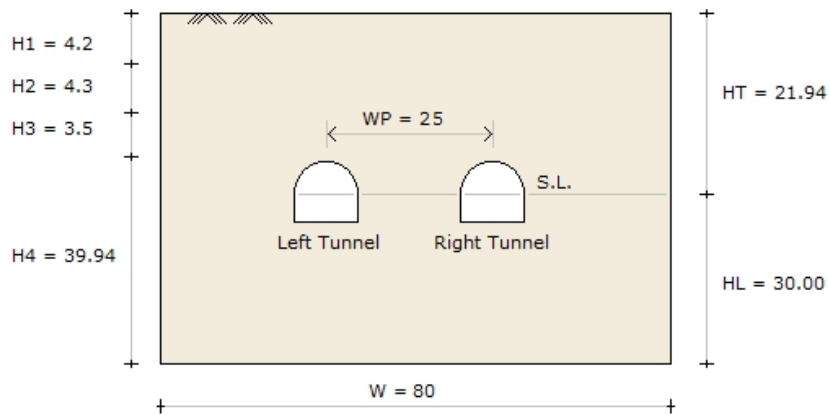
Table 7.9 Listing of input file PD2-3.Dat

```

* CARD 1.1
* TITLE
  NATM-2D MODEL 3 EXAMPLE PROBLEM
* CARD 1.2
* IUNIT
  2
* CARD 1.3
* MODEL  IGEN  IEXMESH  ILNCOUPL
  3      0      0        0
* CARD 2.1
* HT      HL      W      WP      DELTAX  DELTAY  NDYMAX
  21.94  30.    80.    25.    2.0      2.0     40
* CARD 3.1
* NLAYER
  4
* CARD 3.2
* LAYERNO  H      IDH
  1         4.2    0
  2         4.3    0
  3         3.5    0
  4        39.94  0
* CARD 4.1
* R1      A1      R2      A2  R3      A3      R4      GR      GA
  5.24   60.    4.24   30.  9.86   19.781  23.86  1.0   0.5
* CARD 4.2
* INVSHOT  TS
  0         0.3
* CARD 4.3
* NUMRB   LRB      LSPACING  TSPACING  NSRB
  11      3.0      0.8        1.2        2
* CARD 5.1
* LDTYPE   DGW      GAMAW
  0         2.0     1.0
* END OF DATA

```

MODEL=3 Two Tunnel (Symmetric Section)



$\text{DELTA}X = 2.0$ $\text{DELTA}Y = 2.0$ $\text{NDYMAX} = 40$

Figure 7.20 Schematic tunnel section view for Model 3 example

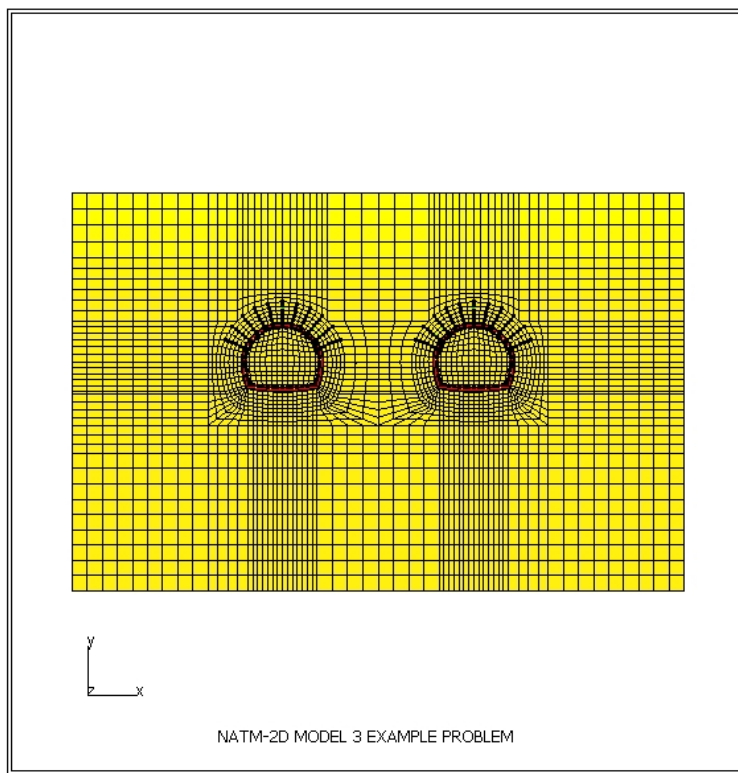


Figure 7.21 Generated finite element mesh for Model 3 example

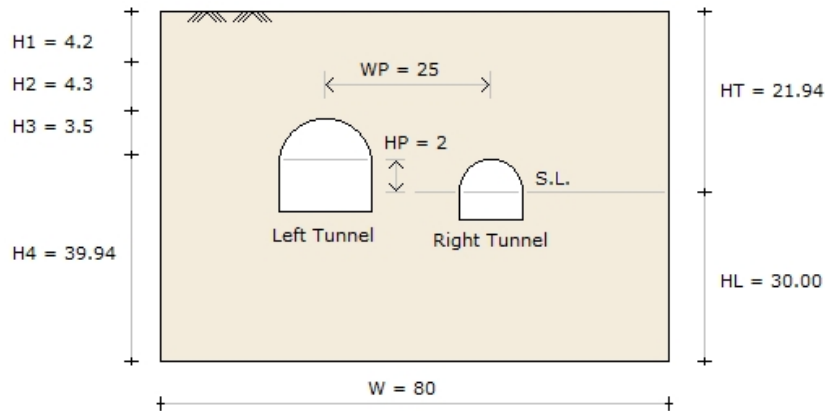
Table 7.10 Listing of input file PD2-4.Dat

```

* CARD 1.1
* TITLE
  NATM-2D MODEL 4 EXAMPLE PROBLEM
* CARD 1.2
* IUNIT
  2
* CARD 1.3
* MODEL  IGEN  IEXMESH  IILNCOUPL
  4      0      0      0
* CARD 2.1
* HT      HL      W      WP      HP      DELTAX  DELTAY  NDYMAX
  21.94  30.    80.    25.    2.0      2.0      2.0      40
* CARD 3.1
* NLAYER
  4
* CARD 3.2
* LAYERNO  H      IDH
  1         4.2    0
  2         4.3    0
  3         3.5    0
  4        39.94  0
* RIGHT TUNNEL
* CARD 4.1
* R1      A1      R2      A2  R3      A3      R4      GR      GA
  5.24   60.    4.24   30.  9.86   19.781  23.86  1.0    0.5
* CARD 4.2
* INVSHOT  TS
  0         0.3
* CARD 4.3
* NUMRB   LRB      LSPACING  TSPACING  NSRB
  11      3.0      0.8      1.2      2
* LEFT TUNNEL
* CARD 4.1
* R1      A1      R2      A2  R3      A3      R4      GR      GA
  7.24   60.    6.24   30. 11.86   21.781  25.86  1.0    0.5
* CARD 4.2
* INVSHOT  TS
  0         0.35
* CARD 4.3
* NUMRB   LRB      LSPACING  TSPACING  NSRB
  15      3.0      0.8      1.2      2
* CARD 5.1
* LDTYPE   DGW      GAMAW
  0         2.0     1.0
* END OF DATA

```

MODEL=4 Two Tunnel (Unsymmetric Section)



DELTA X = 2.0 DELTA Y = 2.0
NDYMAX = 40

Right Tunnel Tunnel dimensions are shown in Figure 7.16

Left Tunnel $R_1 = 7.24 \text{ M}$ $A_1 = 60^\circ$
 $R_2 = 6.24 \text{ M}$ $A_2 = 30^\circ$
 $R_3 = 11.86 \text{ M}$ $A_3 = 21.781^\circ$
 $R_4 = 25.86 \text{ M}$

Number of Rock Bolts (NUMRB) = 15
 Length of Rock Bolts (LRB) = 3.0 M
 Spacing of Rock Bolts (TSPACING) = 1.2 M
 Thickness of Shotcrete (TS) = 35 Cm

Figure 7.22 Schematic tunnel section view for Model 4 example

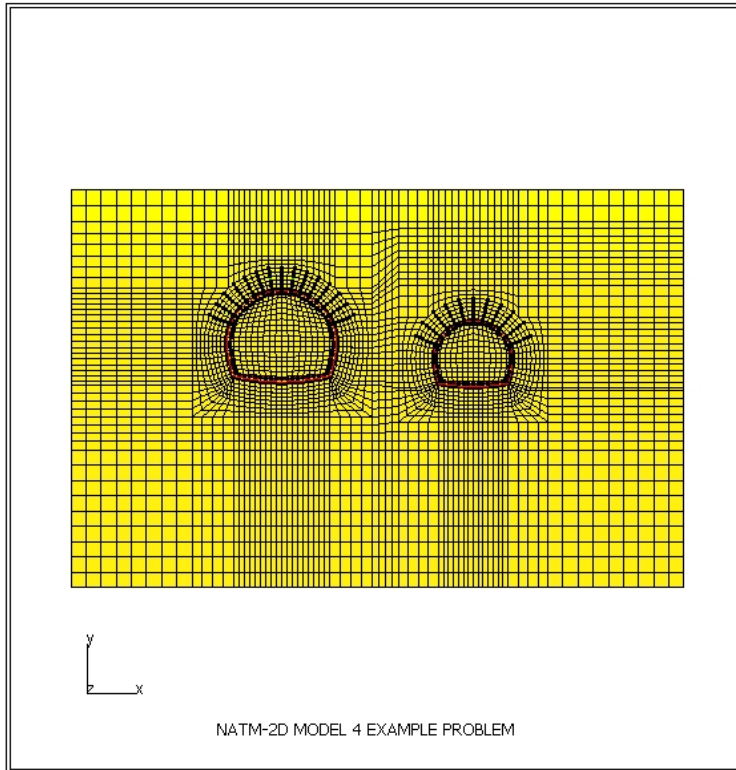


Figure 7.23 Generated finite element mesh for Model 4 example

7.3 CIRCLE-2D

CIRCLE-2D is the special pre-processing program to generate automatically two-dimensional finite element meshes and boundary conditions for circular sections. CIRCLE-2D has three different models:

Model 1	Quarter	Section
Model 2	Half	Section
Model 3	Full	Section

CIRCLE-2D is described in Section 7.4 of User's Manual and can be selected in the following order:

[Run](#) → [Mesh Generator](#) → [PreSmap](#) → [Circle 2D](#)

When you finish the execution of CIRCLE-2D, select [PLOT-3D](#) to plot the generated finite element mesh.

Three example problems are presented here to show all three types of available models. Figure 7.24 shows schematic section views which are used for example problems.

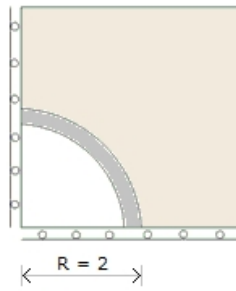
For each model, we will present:

- Listing of input file
- Graphical output of finite element mesh

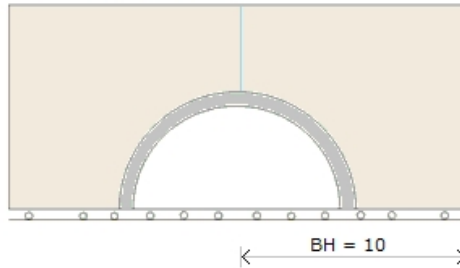
Model = 1
(Quarter Section)

COREMAT1 = 1
COREMAT2 = 2
COREMAT3 = 3

JOINTMAT = 4
NEARMAT = 5



Model = 2 (Half Section)



Model = 3 (Full Section)

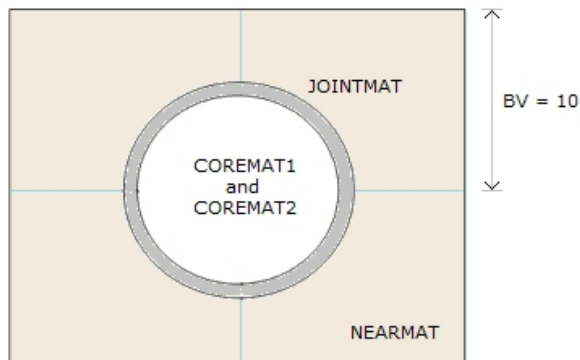


Figure 7.24 Schematic section views for CIRCLE-2D examples

Table 7.11 Listing of input file CIR1C_Q.Dat (MODEL = 1)

```
* CARD 1.1
* TITLE
  MODEL 1 (COARSE, ALL QUAD)
* CARD 1.2
* MODEL      NSNEL      NSNODE
  1          1          1
* CARD 2.1
* R          FINEMESH   NEARMESH   NDIV      BH        BV
  2.0        0          0          5         10.0      10.0
* CARD 3.1
* COREMAT1   COREMAT2   COREMAT2J   JOINTMAT   NEARMAT
  1          2          3          4          5
* END OF DATA
```

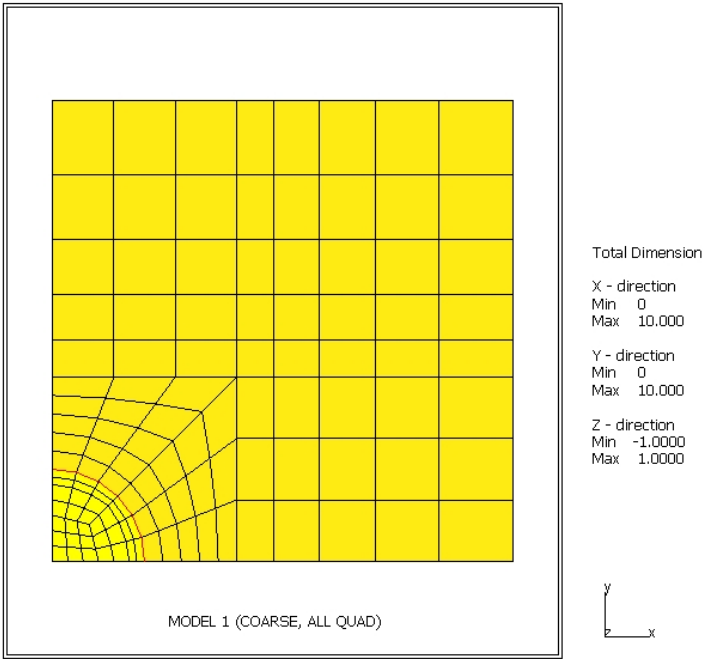


Figure 7.25 Generated finite element mesh for MODEL = 1

Table 7.12 Listing of input file CIR2C_Q.Dat (MODEL = 2)

```
* CARD 1.1
* TITLE
  MODEL 2 (COARSE, ALL QUAD)
* CARD 1.2
* MODEL      NSNEL      NSNODE
  2           1          1
* CARD 2.1
* R           FINEMESH   NEARMESH   NDIV      BH      BV
  2.0         0          0          5         10.0   10.0
* CARD 3.1
* COREMAT1   COREMAT2   COREMAT2J   JOINTMAT   NEARMAT
  1           2          3           4          5
* END OF DATA
```

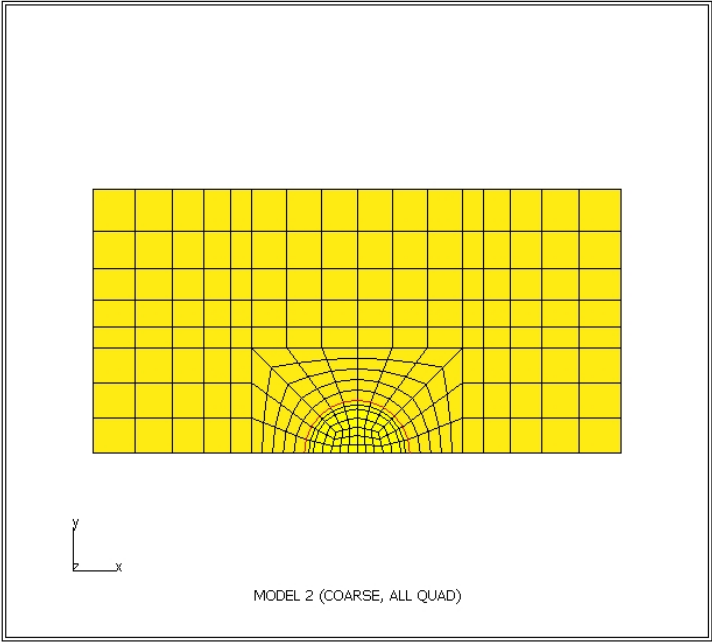


Figure 7.26 Generated finite element mesh for MODEL = 2

Table 7.13 Listing of input file CIR3C_Q.Dat (MODEL = 3)

```
* CARD 1.1
* TITLE
  MODEL 3 (COARSE, ALL QUAD)
* CARD 1.2
* MODEL      NSNEL      NSNODE
  3          1          1
* CARD 2.1
* R          FINEMESH   NEARMESH   NDIV      BH      BV
  2.0        0          0          5         10.0   10.0
* CARD 3.1
* COREMAT1   COREMAT2   COREMAT2J   JOINTMAT   NEARMAT
  1          2          3          4          5
* END OF DATA
```

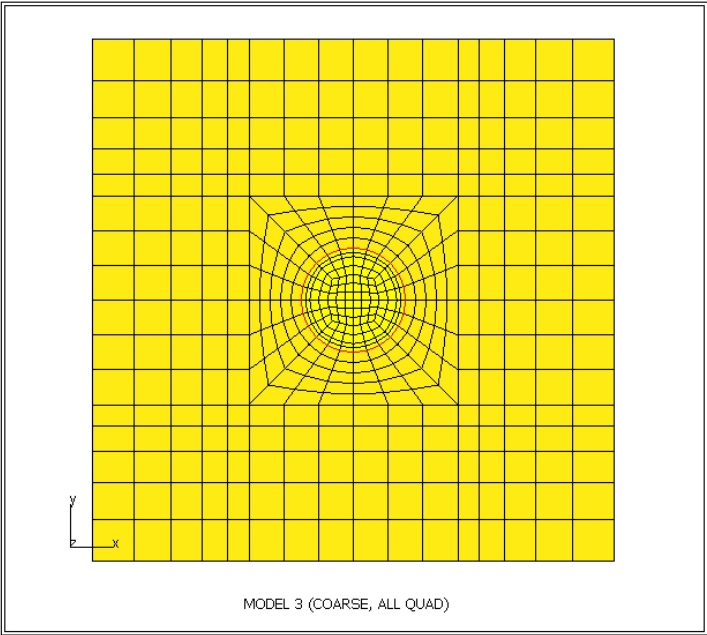


Figure 7.27 Generated finite element mesh for MODEL = 3

7.4 PRESMAP-GP

PRESMAP-GP is the general purpose pre-processor which can be used to generate coordinates, element indexes, and boundary codes of various geometries modeled by truss, beam, shell or continuum elements. Input parameters of PRESMAP-GP have been described in detail in Section 7.9 of User's Manual.

Input file for PRESMAP-GP is also called block mesh file which can be generated or modified by [Block Mesh Generator](#) described in Section 6 of User's Manual.

PRESMAP-GP can be selected in the following order.

[Run](#) → [Mesh Generator](#) → [PreSmap](#) → [Presmap GP](#)

When you finish the execution of PRESMAP-GP, select [PLOT-3D](#) to plot the generated mesh.

7.4.1 Example 1: 2-D Line/Surface Blocks

Example 1 shows you how Beam and Continuum elements are generated using various types of blocks. There are a total of 3 blocks consisting of a line block, a triangle surface block, and a quad surface block. Detailed block information is listed in Table 7.24.

Input block meshes and generated finite element meshes are presented in the following order:

[Input Block Meshes](#)

Figure 7.56 Node and block numbers

Figure 7.57 Material numbers

Figure 7.58 Water flow boundary codes

Figure 7.59 Function time history numbers

Figure 7.60 Initial total head

Figure 7.61 Time function coefficient

Generated Finite Element Meshes

- Figure 7.62 Node and element numbers
- Figure 7.63 Material numbers
- Figure 7.64 Water flow boundary codes
- Figure 7.65 Function time history numbers
- Figure 7.66 Initial total head
- Figure 7.67 Time function coefficient

Table 7.24 Listing of input file EX1.Meb

```

StartPresmap
VersionNo = 7.000
* CARD 1.1
* TITLE
  LINE/SURFACE/ ELEMENT GENERATION
* CARD 1.2


| NBLOCK | NBNODE | NSNODE | NSNEL | IGBND | ISMAP | CMFAC | ICOMP |
|--------|--------|--------|-------|-------|-------|-------|-------|
| 3      | 6      | 1      | 1     | 0     | 9     | 1.000 | 1     |


*=====
* CARD 1.3
* Global Outer Surface Boundary
* X - Right Boundary


| IHG | IDF | H             | CF            |
|-----|-----|---------------|---------------|
| 2   | 7   | 7.000000E-001 | 7.700000E+000 |


* X - Left Boundary


| IHG | IDF | H             | CF            |
|-----|-----|---------------|---------------|
| 0   | 0   | 0.000000E+000 | 0.000000E+000 |


* Y - Top Boundary


| IHG | IDF | H             | CF            |
|-----|-----|---------------|---------------|
| 0   | 0   | 0.000000E+000 | 0.000000E+000 |


* Y - Bottom Boundary


| IHG | IDF | H             | CF            |
|-----|-----|---------------|---------------|
| 0   | 0   | 0.000000E+000 | 0.000000E+000 |


* Z - Front Boundary


| IHG | IDF | H             | CF            |
|-----|-----|---------------|---------------|
| 0   | 0   | 0.000000E+000 | 0.000000E+000 |


* Z - Back Boundary


| IHG | IDF | H             | CF            |
|-----|-----|---------------|---------------|
| 0   | 0   | 0.000000E+000 | 0.000000E+000 |


*=====

```

```

* CARD 1.4
* Automatic Finite Element Generation Control Parameters
* Min Length      Max Element
  1.000          10000
=====
* CARD 2.1
* NODE   X           Y           Z
  1       4.0         6.5         0.0
  2       0.0         2.0         0.0
  3       5.9         0.8         0.0
  4       7.0         7.0         0.0
  5       7.0         1.0         0.0
  6       5.72        3.87        0.0
=====
StartBlock
* CARD 3.0
* IBETYPE
  1
* CARD 3.1
* BLNAME
  BLOCK 1
* CARD 3.2
* ICOORD IMODE  ILAG
  1       0       0
* CARD 3.3
* I1      I2
  1       3
* M3
  0
* M4
  0
* M5      M6      M7
  0       0       0
* CARD 3.4.1
* NBOUND
  1
* CARD 3.4.2
* IBTYPE ID      IDF      TH      CF
  1       1       8       8.000000E-001  8.800000E+000
* CARD 3.5
* MATNO  NDX
  1       4
EndBlock

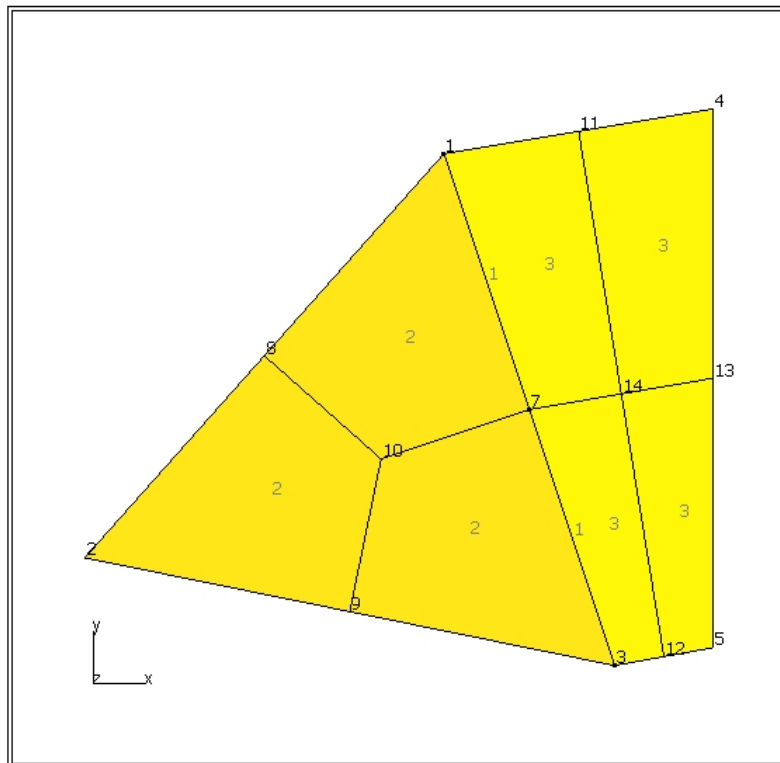
```

```

=====
StartBlock
* CARD 3.0
* IBETYPE
-2
* CARD 3.1
* BLNAME
BLOCK 2
* CARD 3.2
* ICOORD IMODE ILAG
1 0 1
* CARD 3.3
* I1 I2 I3
1 2 3
* M4 M5 M6
0 0 0
* M7
0
* M8 M9 M10
0 0 0
* CARD 3.4.1
* NBOUND
1
* CARD 3.4.2
* IBTYPE ID IDF TH CF
1 1 8 8.000000E-001 8.800000E+000
* CARD 3.5
* MATNO NDXY
4 4
* IDH
0
EndBlock
=====
StartBlock
* CARD 3.0
* IBETYPE
2
* CARD 3.1
* BLNAME
BLOCK 3
* CARD 3.2
* ICOORD IMODE ILAG
1 0 1
* CARD 3.3
* I1 I2 I3 I4
4 1 3 5

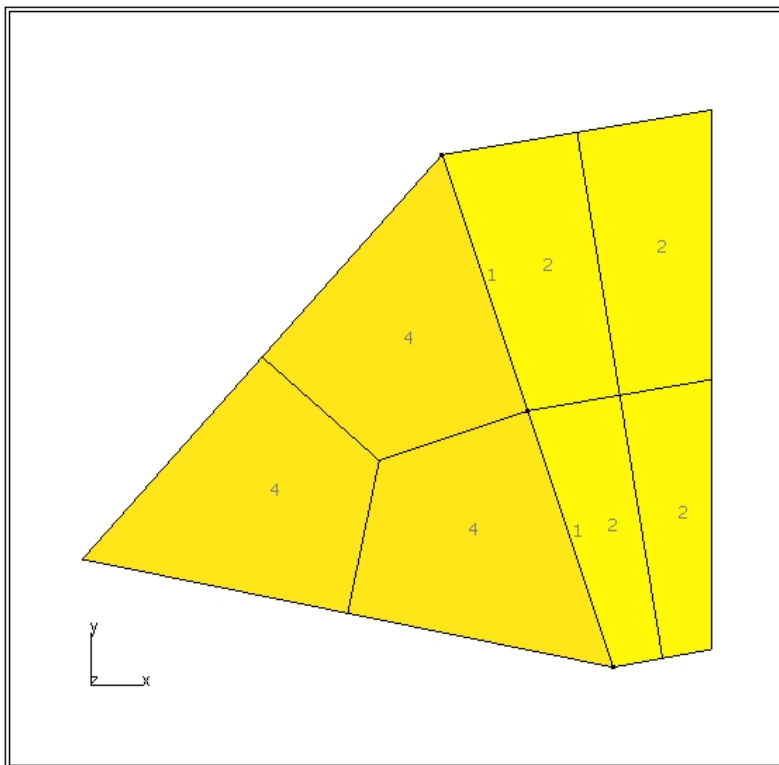
```

```
* M5      M6      M7      M8
0        0        0        0
* M9
0
* M10     M11     M12
0        0        0
* CARD 3.4.1
* NBOUND
1
* CARD 3.4.2
* IBTYPE ID      IDF      TH      CF
1        1        8      8.000000E-001  8.800000E+000
* CARD 3.5
* MATNO  NDX      NDY
2        2        4
* NT1     NT2     NT3     NT4
0        0        0        0
* MAT1     MAT2     MAT3     MAT4
0        0        0        0
* IDH
0
EndBlock
*=====
EndOfLastBlock
```



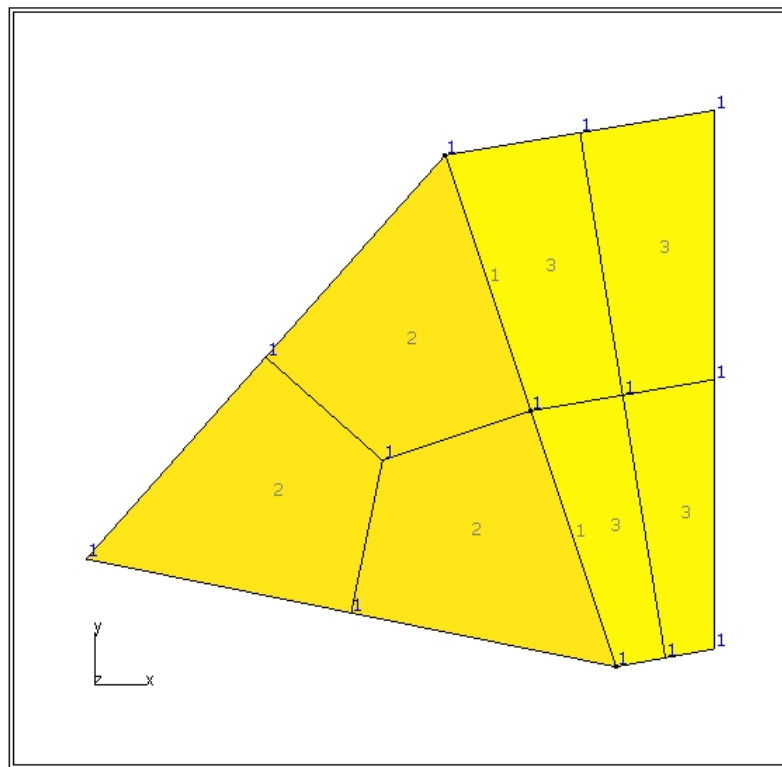
LINE/SURFACE/ ELEMENT GENERATION
Node and Block Number

Figure 7.56 Node and block numbers for Example 1



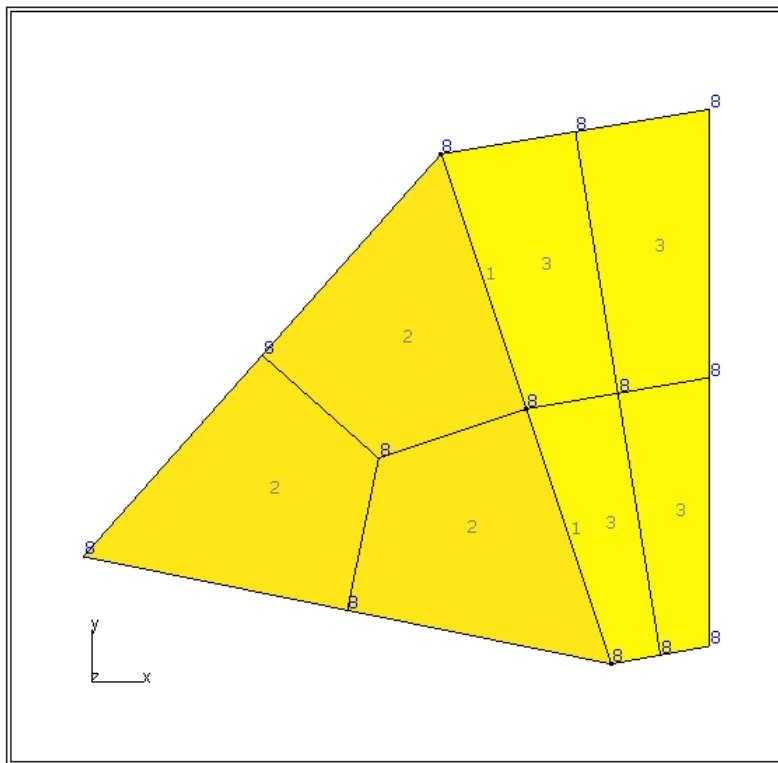
LINE/SURFACE/ELEMENT GENERATION
Material Number

Figure 7.57 Material numbers



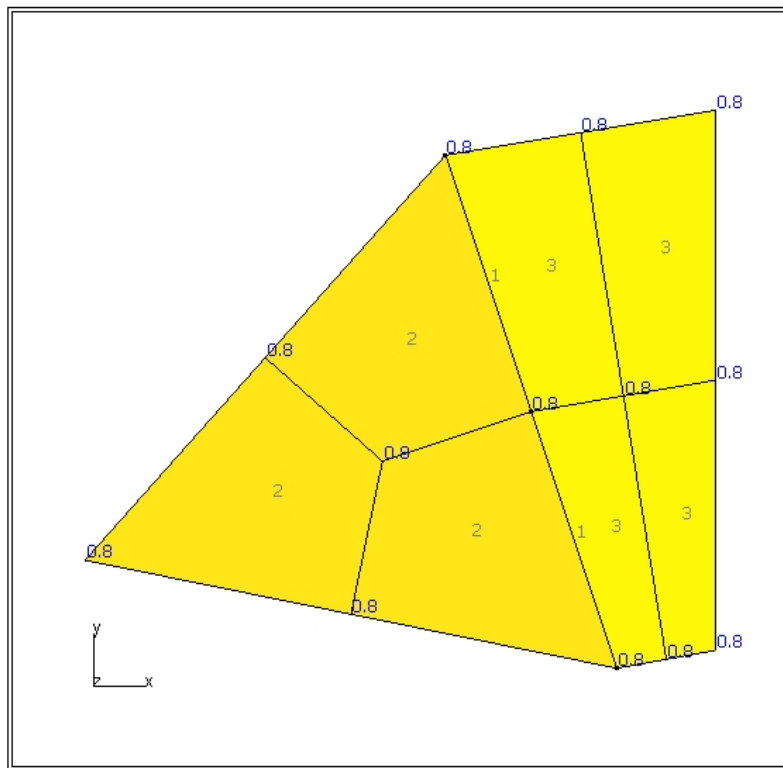
LINE/SURFACE/ ELEMENT GENERATION
Water Flow (ID = 0), Total Head (ID = 1), Potential Seepage Face (ID = 3) Boundary Code

Figure 7.58 Water flow boundary codes



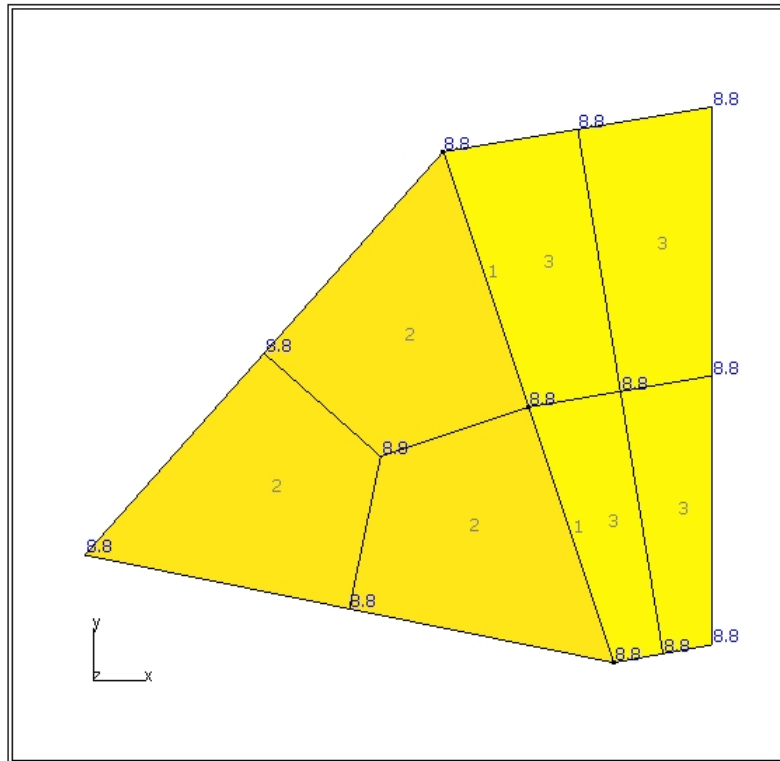
LINE/SURFACE/ ELEMENT GENERATION
Function Time History No.: IDF (Negative Value for Hydrograph)

Figure 7.59 Function time history numbers



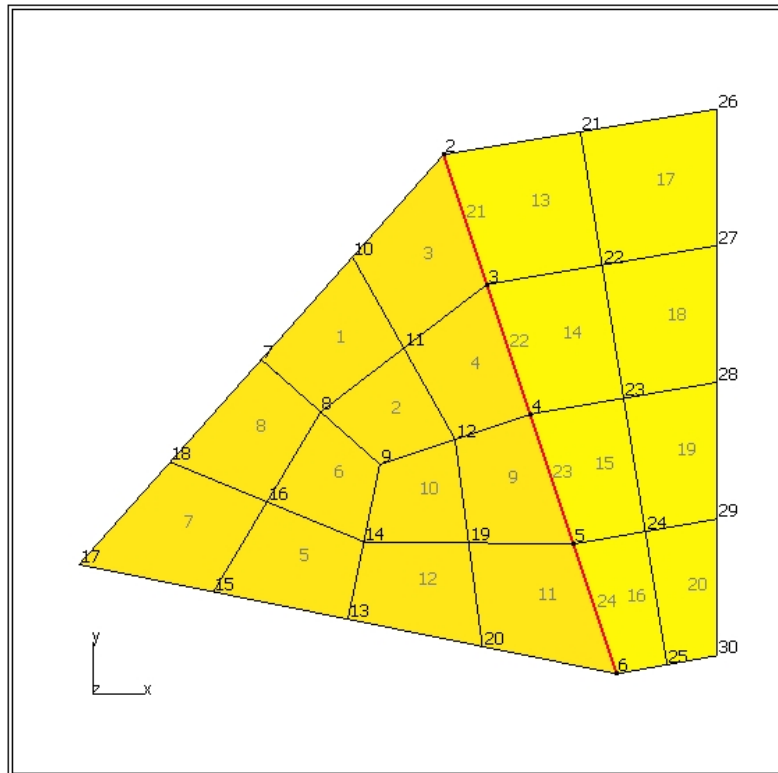
LINE/SURFACE/ ELEMENT GENERATION
Initial Total Head: H

Figure 7.60 Initial total head



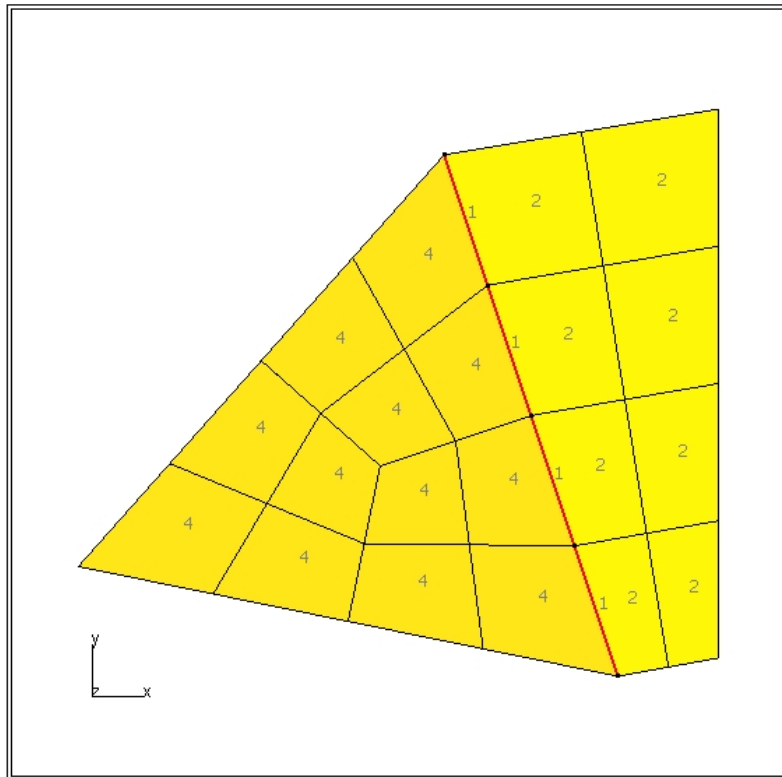
LINE/SURFACE/ ELEMENT GENERATION
Coefficient of Time Dependent Function: CF

Figure 7.61 Time function coefficient



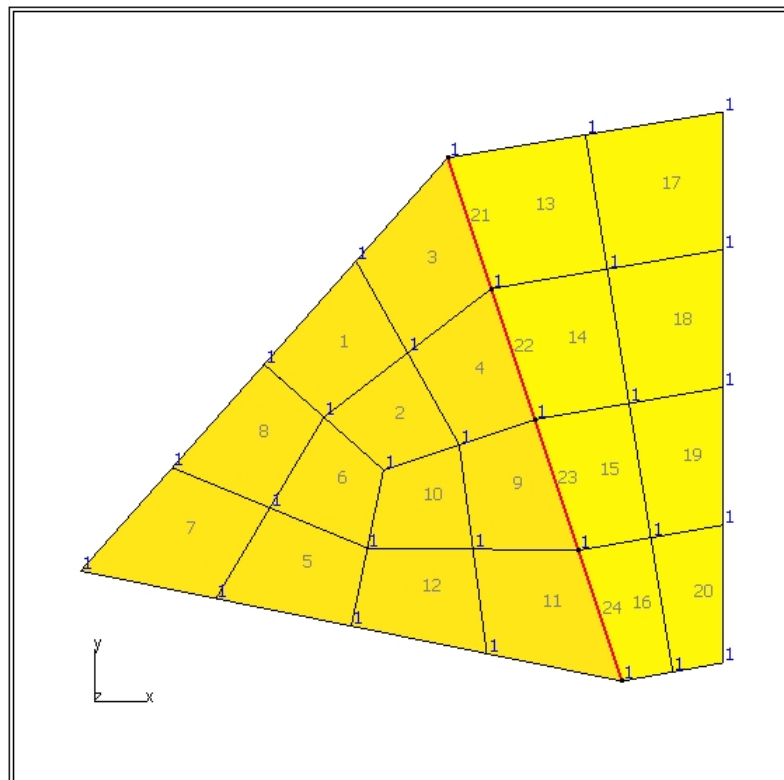
EX1.Mes
Node and Element Number

Figure 7.62 Node and element numbers



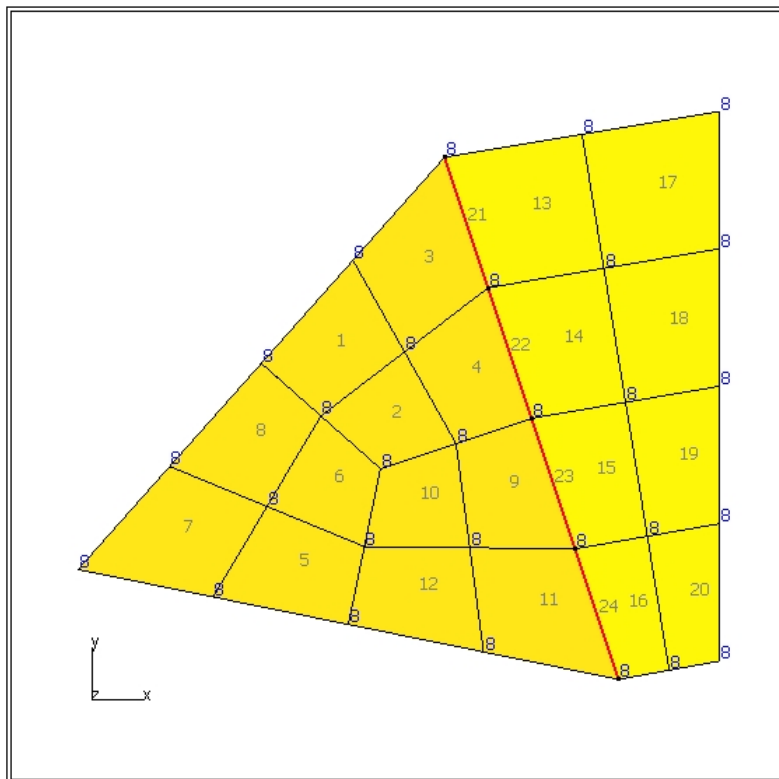
EX1.Mes
Material Number

Figure 7.63 Material numbers



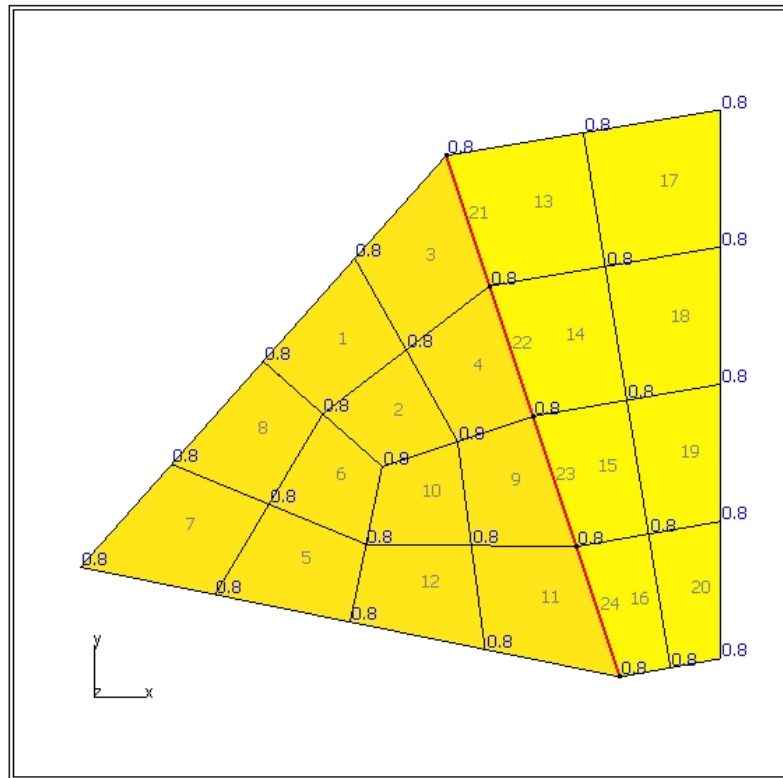
EX1.Mes
Water Flow (ID = 0), Total Head (ID = 1), Potential Seepage Face (ID = 3) Boundary Code

Figure 7.64 Water flow boundary codes



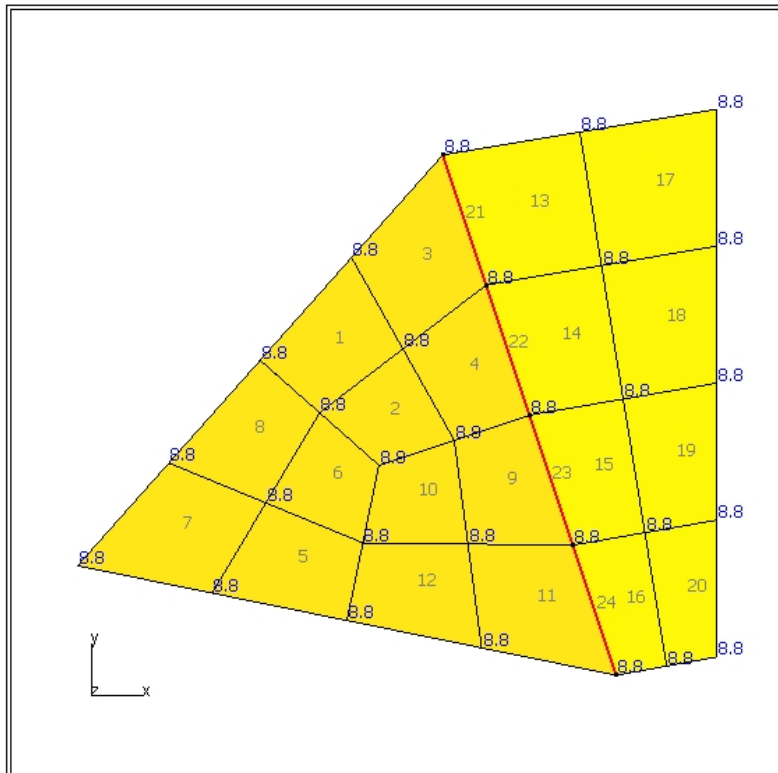
EX1.Mes
Function Time History No: IDF (Negative Value for Hydrograph)

Figure 7.65 Function time history numbers



EX1.Mes
Initial Total Head: H

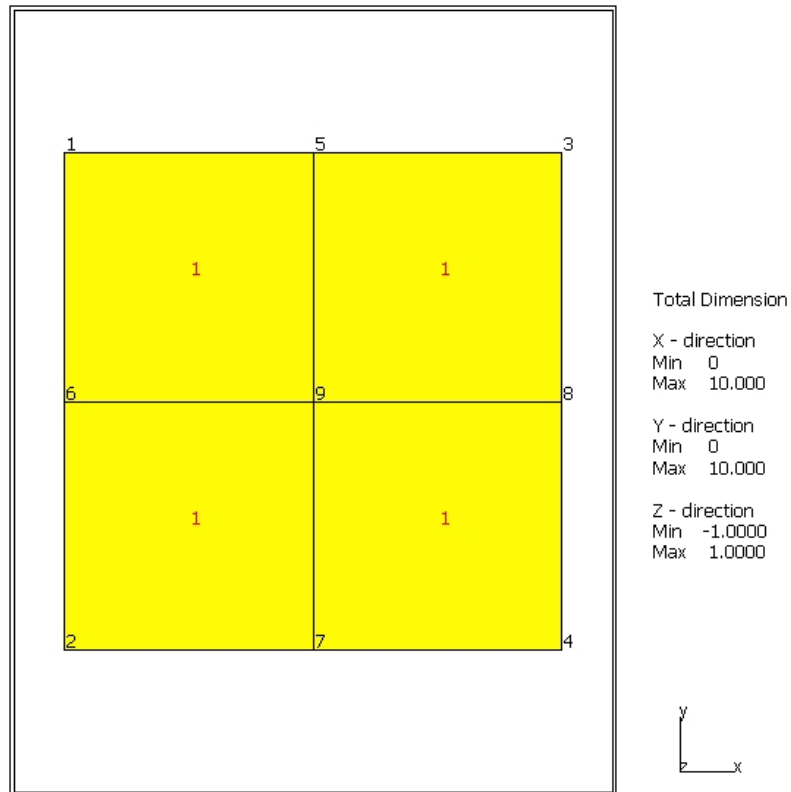
Figure 7.66 Initial total head



EX1.Mes
Coefficient of Time Dependent Function: CF

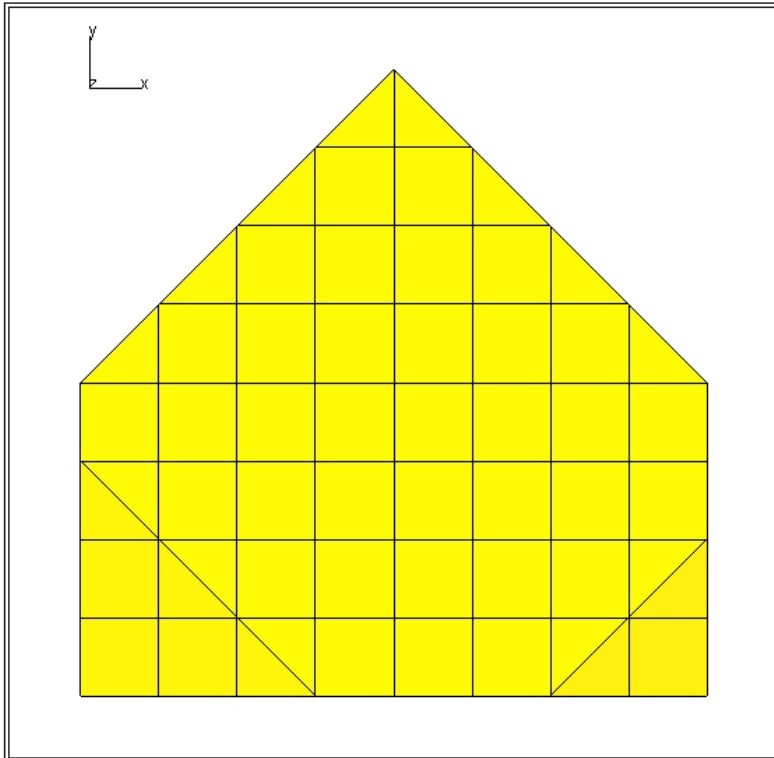
Figure 7.67 Time function coefficient

7.4.2 Example 2: Surface with Corner Triangles



SURFACE ELEMENT GENERATION WITH CORNER TRIANGLES.
Node and Block Number

Figure 7.68 Block mesh for Example 2



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX2\EX2.Mes

Figure 7.69 Finite element mesh for Example 2

7.4.3 Example 3: Circular Sector

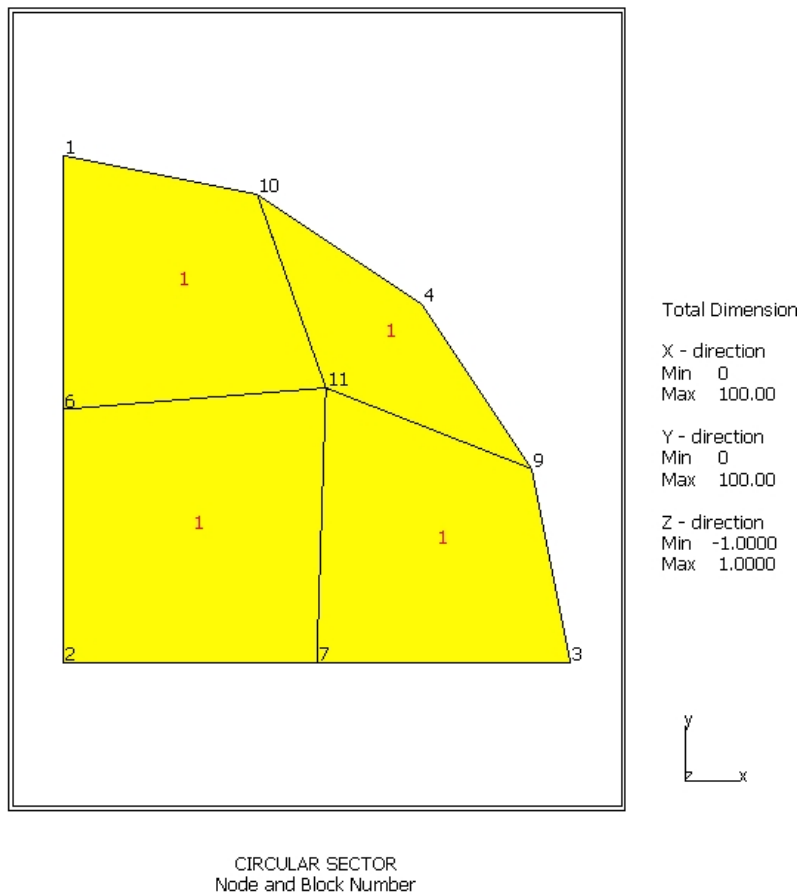
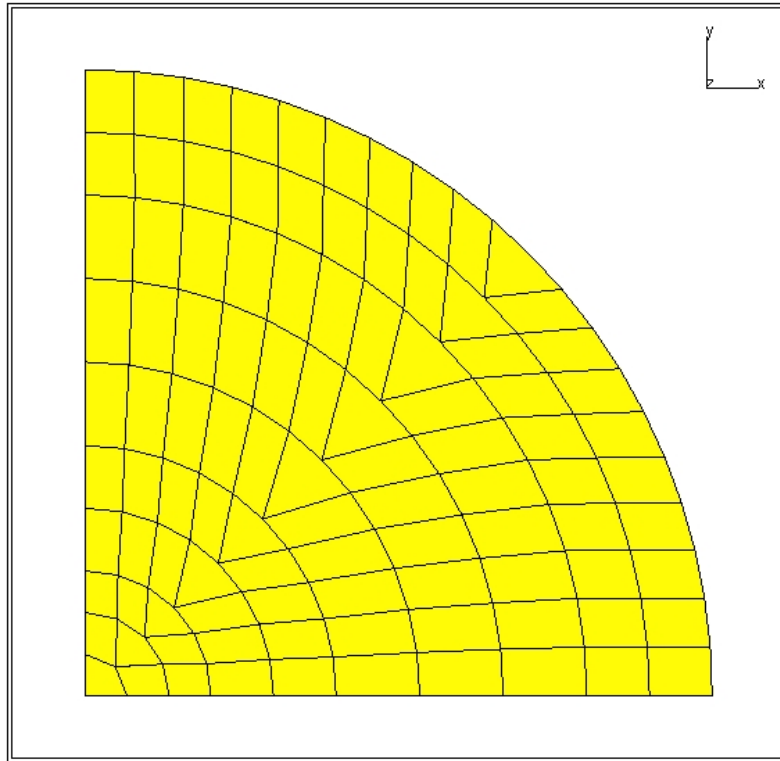


Figure 7.70 Block mesh for Example 3



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX3\EX3.Mes

Figure 7.71 Finite element mesh for Example 3

7.4.4 Example 4: Straight Line Sector

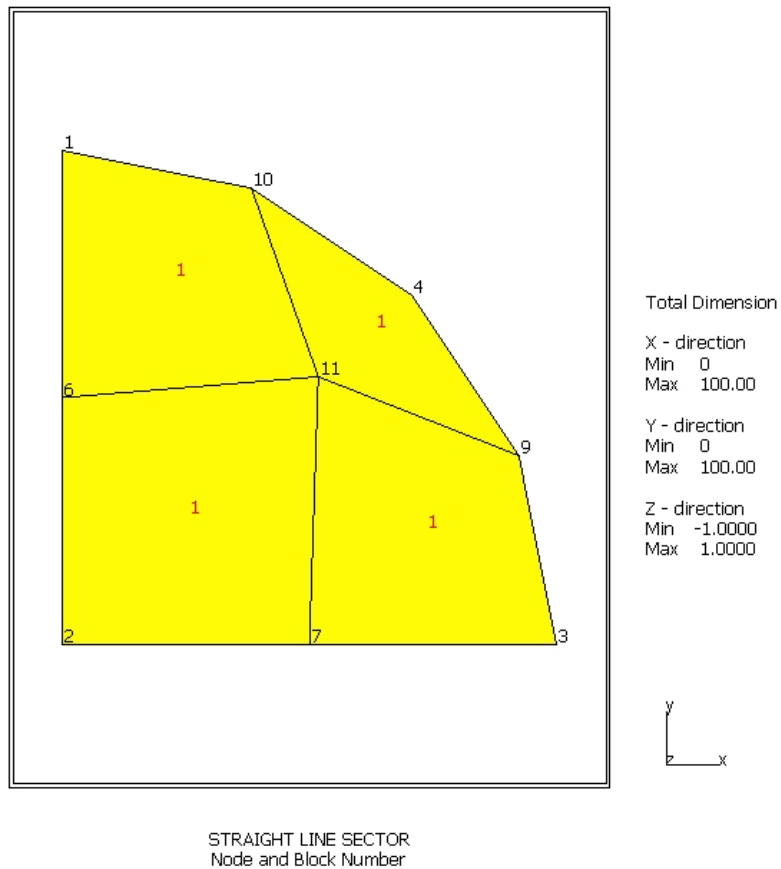
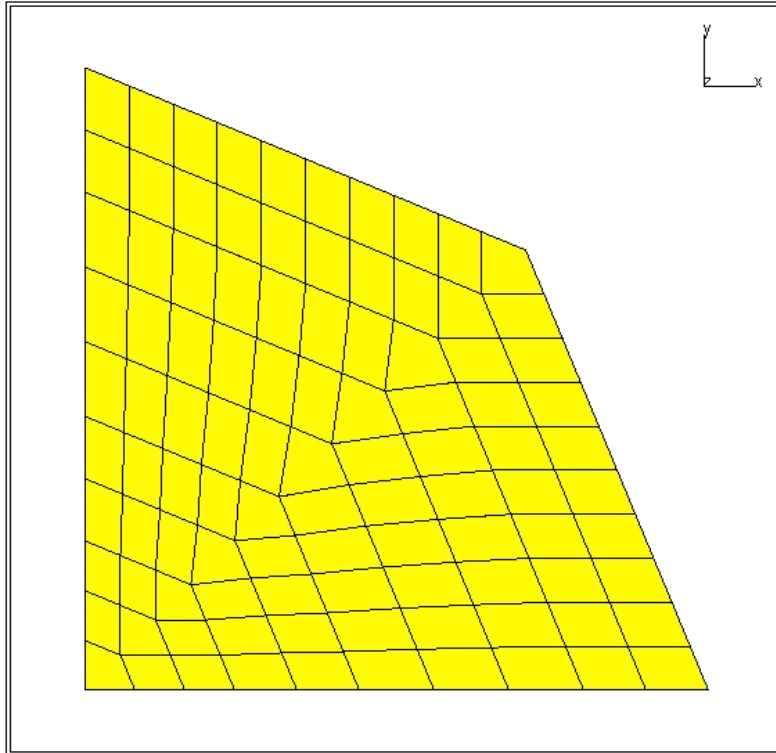


Figure 7.72 Block mesh for Example 4



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX4\EX4.Mes

Figure 7.73 Finite element mesh for Example 4

7.4.5 Example 5: Surface and Line Element (1)

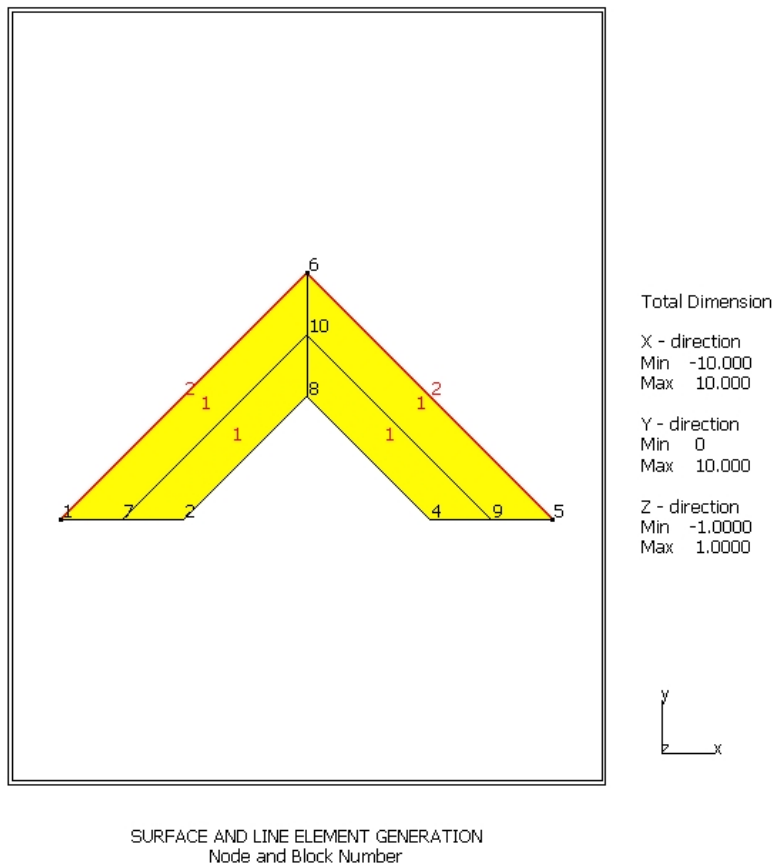
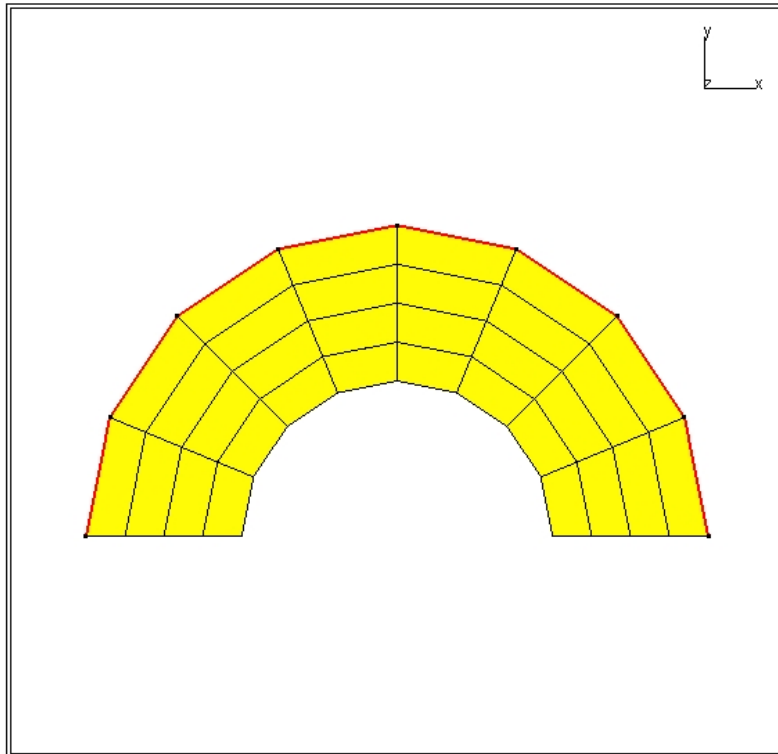


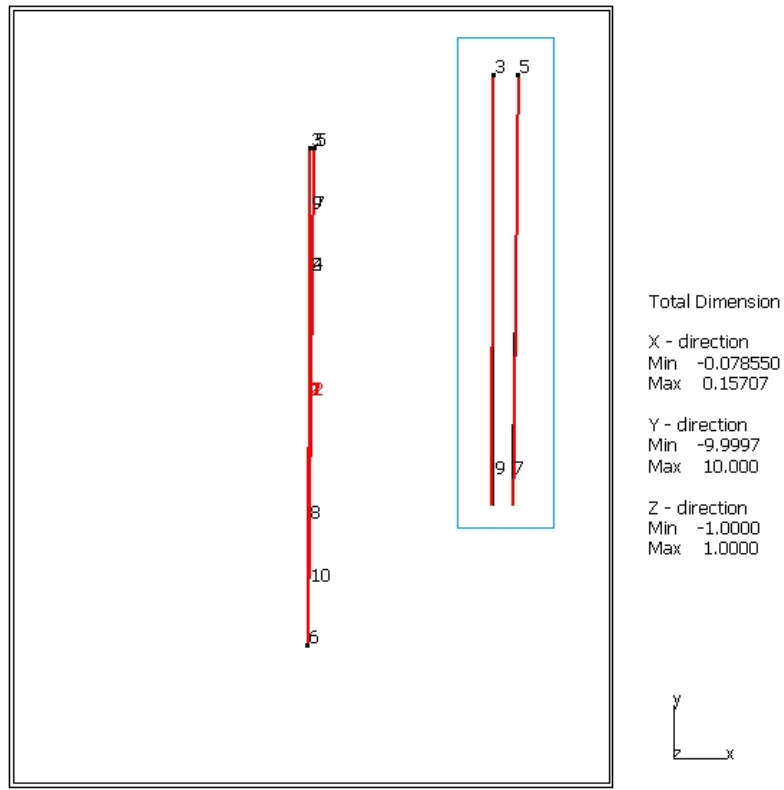
Figure 7.74 Block mesh for Example 5



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX5\EX5.Mes

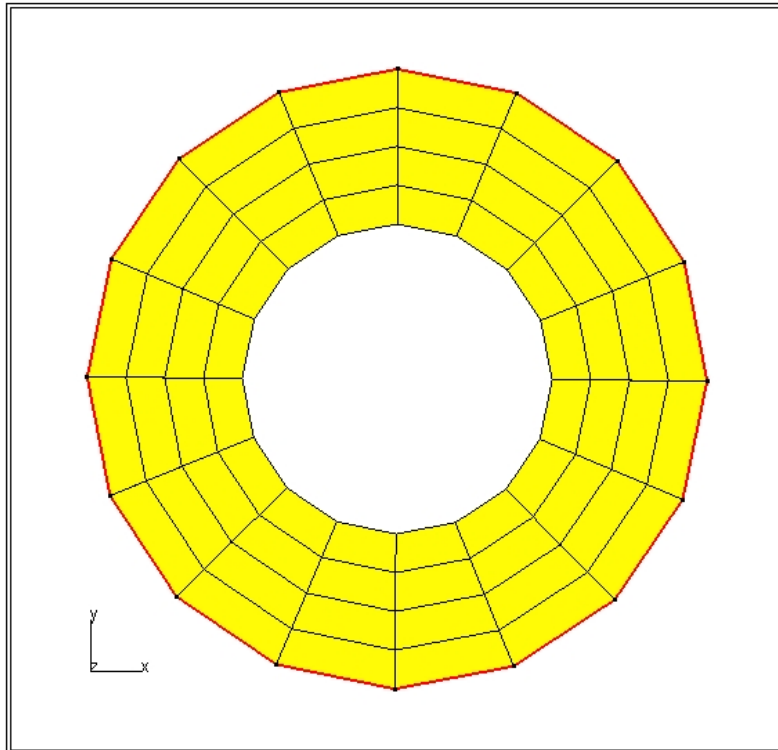
Figure 7.75 Finite element mesh for Example 5

7.4.6 Example 6: Surface and Line Element (2)



SURFACE AND LINE ELEMENT GENERATION: EX6.RGN [2D-S-1B]
Node and Block Number

Figure 7.76 Block mesh for Example 6



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX6\EX6.Mes

Figure 7.77 Finite element mesh for Example 6

7.4.7 Example 7: Surface and Line Element (3)

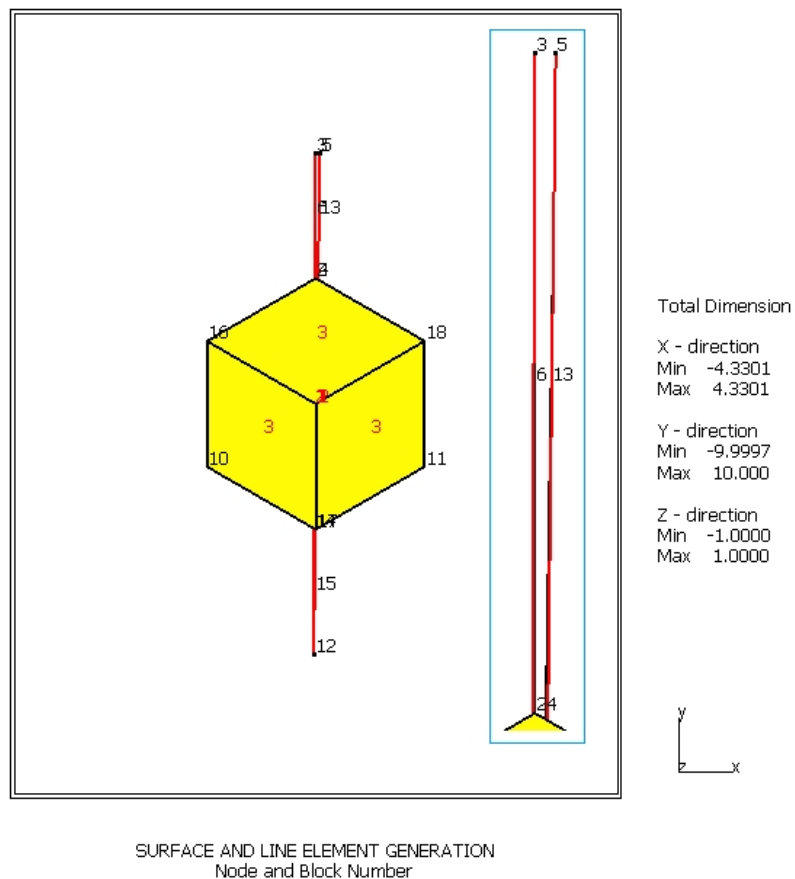
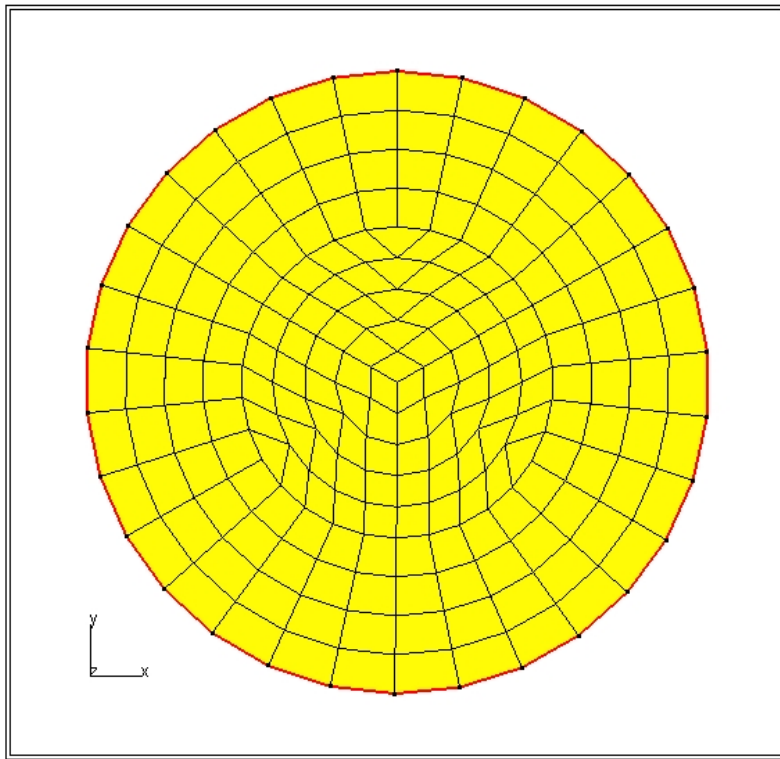


Figure 7.78 Block mesh for Example 7



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX7\EX7.Mes

Figure 7.79 Finite element mesh for Example 7

7.4.8 Example 8: Cement-Soil Road

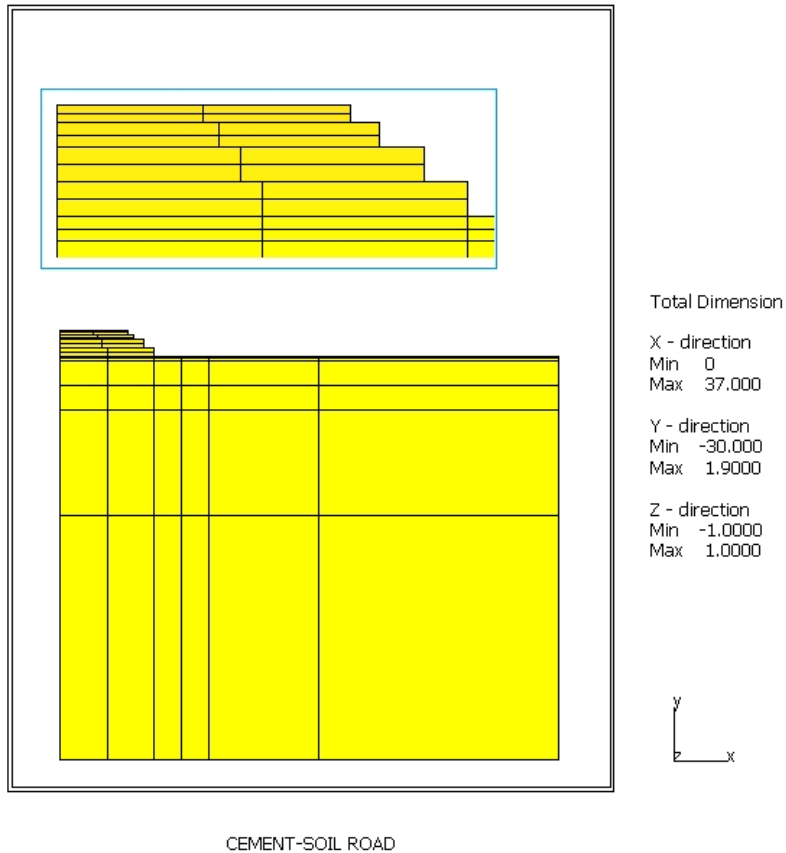
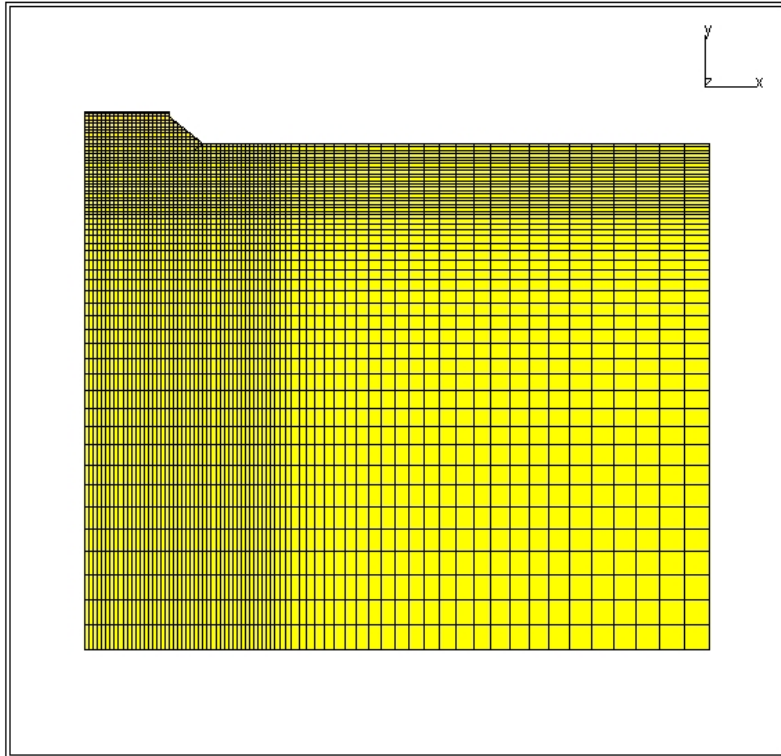


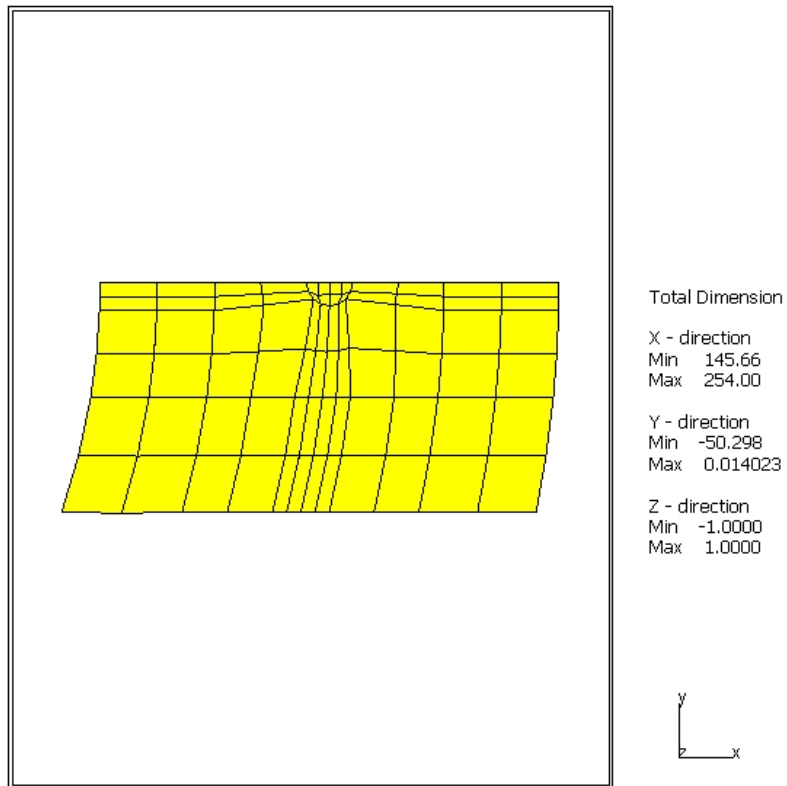
Figure 7.80 Block mesh for Example 8



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX8\EX8.Mes

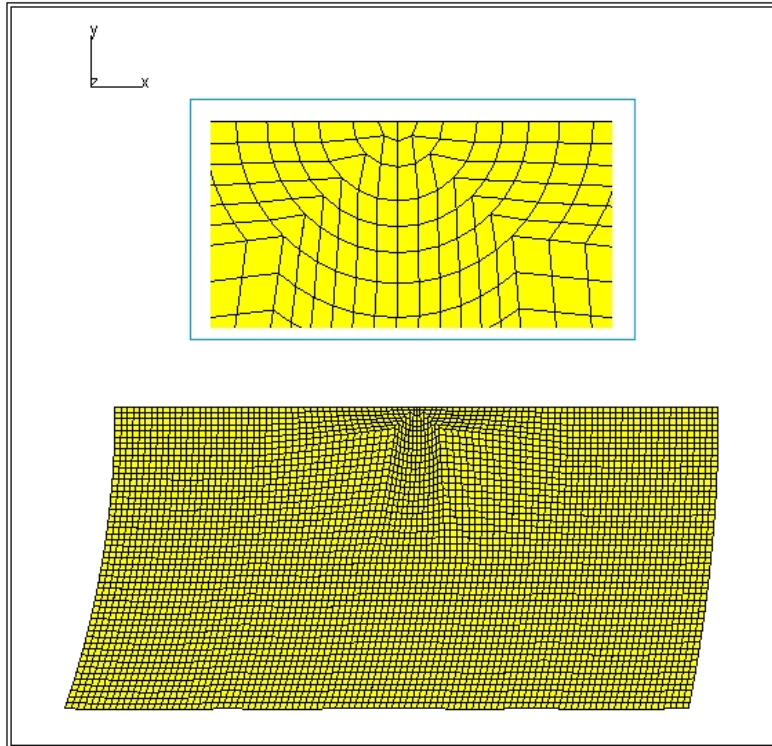
Figure 7.81 Finite element mesh for Example 8

7.4.9 Example 9: Tunnel in Spherical Geometry



Tunnel Subjected To Spherical Wave

Figure 7.82 Block mesh for Example 9



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX9\EX9.Mes

Figure 7.83 Finite element mesh for Example 9

7.4.10 Example 10: Horseshoe Tunnel

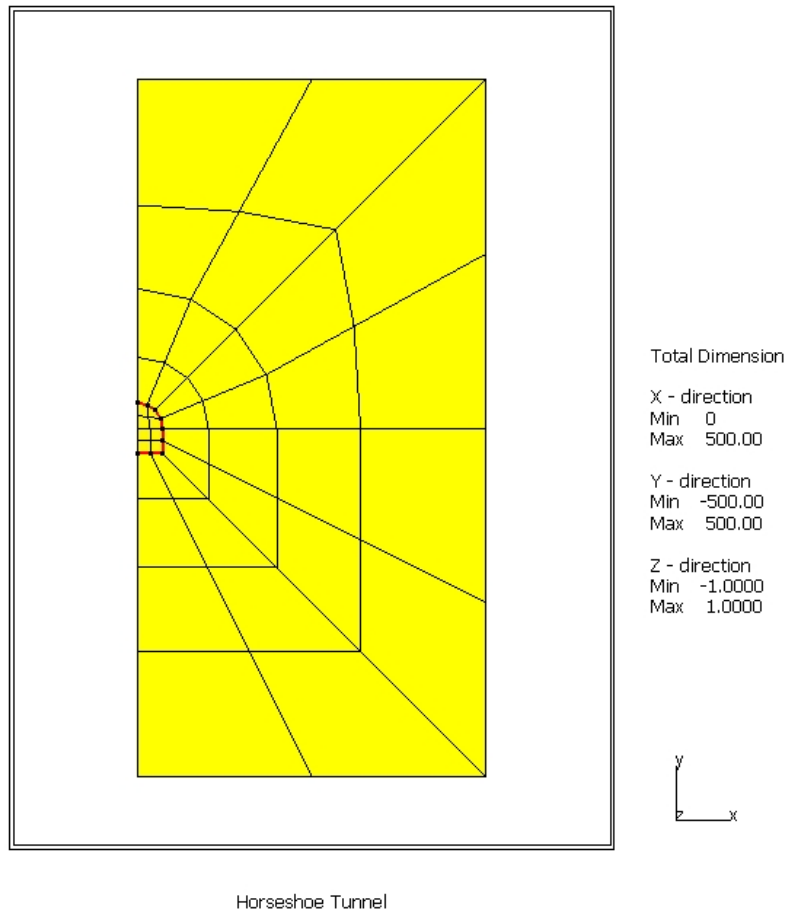
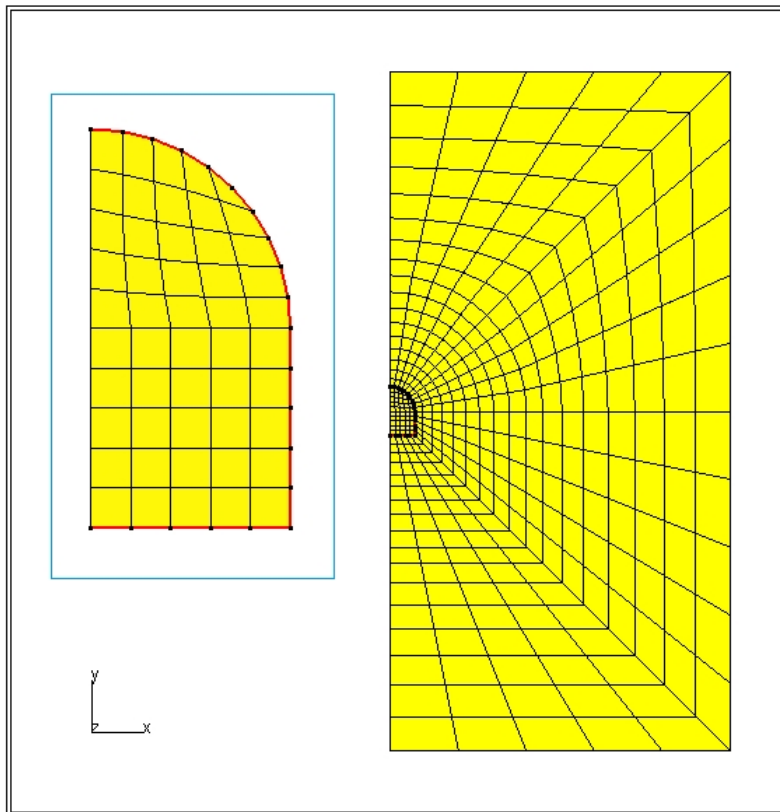


Figure 7.84 Block mesh for Example 10



C:\SMAP\SMAPW2\EXAMPLE\PRESMAP\PRESM-GP\EX10\EX10.Mes

Figure 7.85 Finite element mesh for Example 10

7.4.11 Example 11: Wedge Surface Block

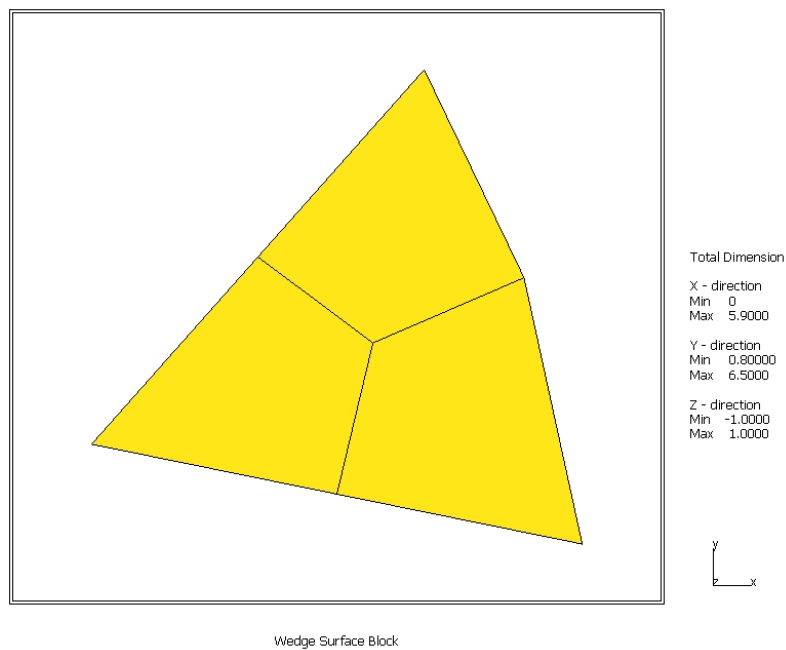


Figure 7.86 Block mesh for Example 11

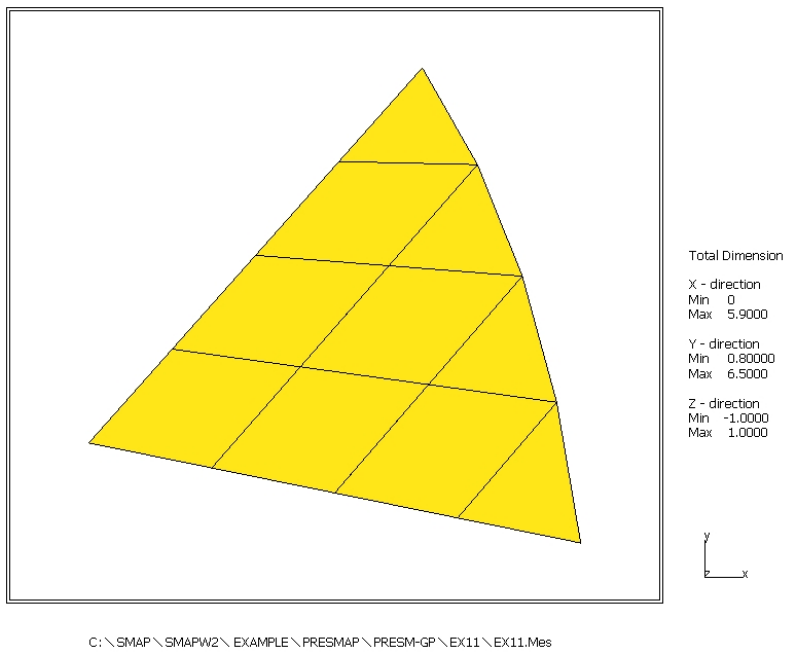


Figure 7.87 Finite element mesh for Example 11

ADDRGN

Example Problem

ADDRGN is the pre-processing program which has the following functions:

- Combine two different meshes
- Modify existing meshes
 - Change coordinates
 - Change boundary codes
 - Cut elements
 - Change material numbers
- Generate finite element meshes (ADDRGN-2D)

Refer to SMAP-W2 User's Manual:

- Section 5 for group mesh generation (ADDRGN-2D)
- Section 8 for input parameters

8.1 ADDRGN-2D

ADDRGN-2D is the two dimensional pre-processor which is used to combine, modify, or generate finite element meshes.

ADDRGN-2D can be selected in the following order:

Run → Mesh Generator → AddRgn → Addrgn 2D

When you finish the execution of ADDRGN-2D, select **PLOT-3D** to plot modified or generated mesh.

8.1.1 Combining Meshes

In the PRESMAP-2D Example Problem in Sections 7.1.1 and 7.1.2, three different regions (Core, Near-field, and Far-field) are generated using Models 1 and 2. Now, we want to combine all these different regions into one using ADDRGN-2D. Note that [CORE.Mes](#), [NEAR.Mes](#) and [FAR.Mes](#) are the output files corresponding to the input file [CORE.Rgn](#), [NEAR.Rgn](#) and [FAR.Rgn](#) respectively.

Element numbers 1 to 72 are assigned for [CORE.Mes](#), 73 to 336 for [NEAR.Mes](#) and 337 to 464 for [FAR.Mes](#). When we combine two regions, element numbers should be continuous through the regions. So, let's first add [NEAR.Mes](#) (called REGION B) to [CORE.Mes](#) (called REGION A) to make [CONE.Mes](#) (called COMBINED REGION). Next, let's add [FAR.Mes](#) (called REGION B) to [CONE.Mes](#) (called REGION A) to make the final mesh [CNF.Mes](#) (called COMBINED REGION). ADDRGN input files are listed in Tables 8.1 and 8.2.

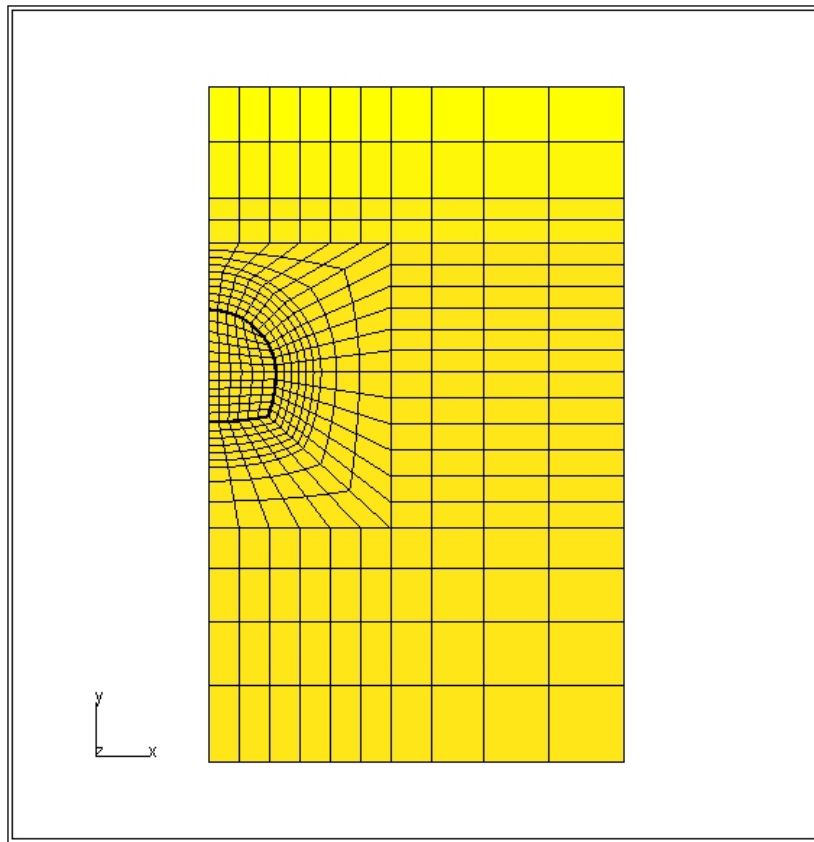
Figure 8.1 shows the element meshes of combined region representing all three regions.

Table 8.1 Listing of input file ADD2D-1.Dat

```
* ADD2D-1.Dat
* CARD 1.1
* IMOD = 0 : ADD REGION B TO REGION A
  0
* CARD 2.1
* FILEA : Input file name containing REGION A
  CORE.Mes
* FILEB : Input file name containing REGION B
  NEAR.Mes
* FILEC : Output file name to store COMBINED REGION
  CONE.Mes
* CARD 2.2
* INTERFACE
  0
* END OF DATA
```

Table 8.2 Listing of input file ADD2D-2.Dat

```
* ADD2D-2.Dat
* CARD 1.1
* IMOD = 0 : ADD REGION B TO REGION A
  0
* CARD 2.1
* FILEA : Input file name containing REGION A
  CONE.Mes
* FILEB : Input file name containing REGION B
  FAR.Mes
* FILEC : Output file name to store COMBINED REGION
  CNF.Mes
* CARD 2.2
* INTERFACE
  0
* END OF DATA
```



CNF.Mes

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

8.1.2 Modifying Mesh

In this example, we want to generate symmetric meshes using ADDRGN-2D. As the existing mesh, we take the [CORE.Mes](#) which has been generated using PRESMAP-2D Model 1 (refer to Section 7.1.1.2). Note that [CORE.Mes](#) represents the right side of the tunnel core. ADDRGN input file to generate [Left Core](#) is listed in Table 8.3. The output file [LCORE.Mes](#) contains [Left Core](#) whose graphical output is shown in Figure 8.2.

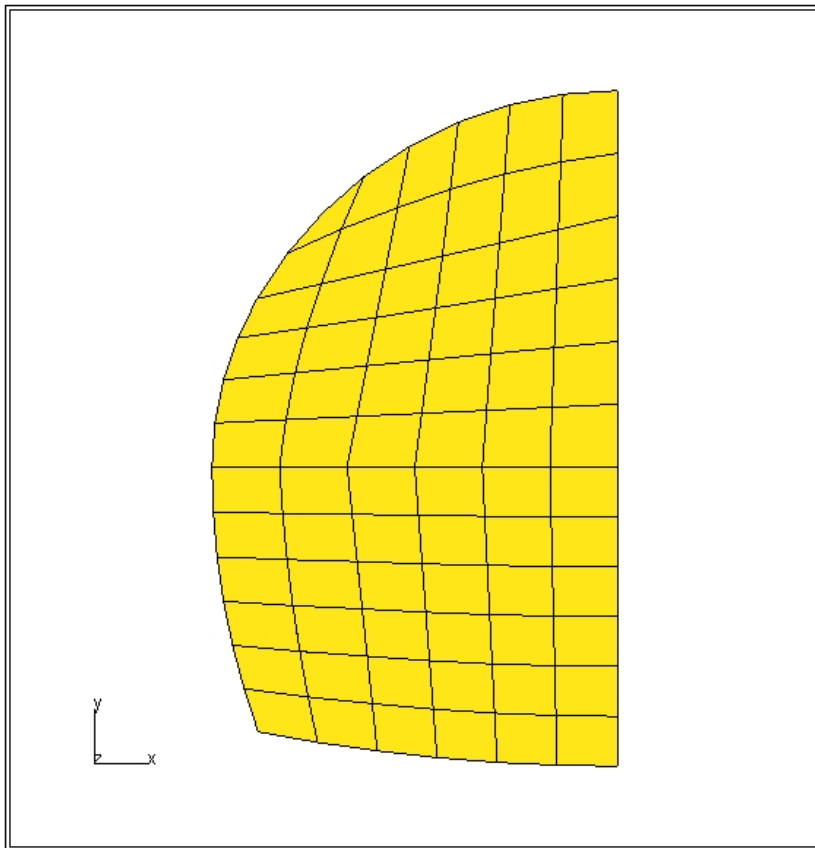
By combining both left and right core regions as instructed in Table 8.4, we can generate a whole core region, [WCORE.Mes](#). Graphical output of [WCORE.Mes](#) is shown in Figure 8.3.

Table 8.3 Listing of input file ADD2D-3.Dat

```
* ADD2D-3.Dat
* CARD 1.1
* IMOD = 1 : MODIFY EXISTING MESH
1
* CARD 3.1
* FILEA : Input file name to be modified
CORE.Mes
* FILEM : Output file name to store modified mesh
LCORE.Mes
* CARD 3.2
* NSNEL NSNODE
73 1
* CARD 3.3
* IEDIT = 0 : CHANGE COORDINATES
0
* CARD 3.3.1.1
* Xo Yo Xonew Yonew
0.0 0.0 0.0 0.0
* CARD 3.3.1.2
* Xscale Yscale
-1.0 1.0
* END OF DATA
```

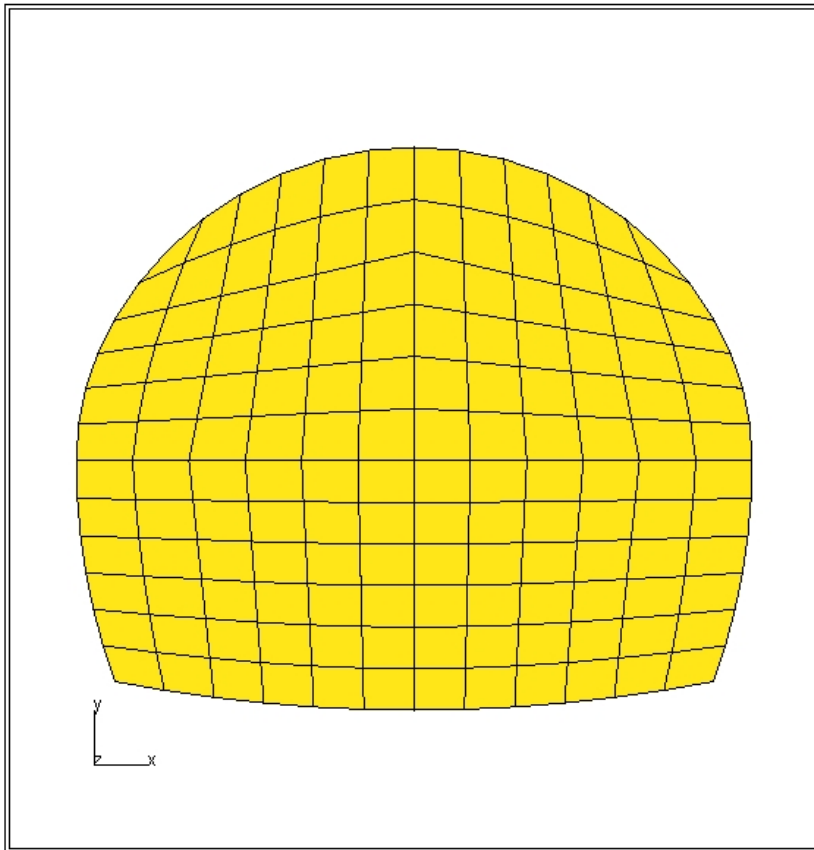
Table 8.4 Listing of input file ADD2D-4.Dat

```
* ADD2D-4.Dat
* CARD 1.1
* IMOD = 0 : ADD REGION B TO REGION A
0
* CARD 2.1
* FILEA : Input file name containing REGION A
CORE.Mes
* FILEB : Input file name containing REGION B
LCORE.Mes
* FILEC : Output file name to store COMBINED REGION
WCORE.Mes
* CARD 2.2
* INTERFACE
0
* END OF DATA
```



LCORE.Mes

Figure 8.2 Left core mesh, LCORE.Mes



WCORE.Mes

Figure 8.3 Combined whole core mesh, WCORE.Mes

8.1.3 Generating Mesh

This example is to show a powerful mesh generation feature using ADDRGN-2D. All you need to do is to specify the locations, dimensions and material numbers of structures along with few instructions for mesh generation. ADDRGN-2D will do the rest of the work to build the Mesh File.

As the first example, we take a simple problem as schematically shown in Figure 8.4. A utility tunnel with a diameter of 4 meters is located 6 meters below the ground surface. Table 8.5 shows the full listing of input file [ADD2D-5.Dat](#). The base mesh consists of 3 blocks in the horizontal direction and 1 block in the vertical direction.

The first group represents soft rock underlying soil. And the second group represents the utility tunnel. Tunnel liner is modeled by beam element and the interface between the liner and the surrounding soil is modeled by joint element which will allow the slippage and separation. Finite element meshes generated by ADDRGN-2D are shown in Figures 8.5 and 8.6. It should be noted that the joint thickness in Figure 8.6 is exaggerated to show clearly both inner and outer joint faces. The real joint thickness is specified in material property card in Main File.

Table 8.5 Listing of input file ADD2D-5.Dat

```

* ADD2D-5.Dat
* CARD 1.1
* IMOD = 2 : GENERATE BASE MESH AND THEN MODIFY
* IMOD JK
  2    9
* CARD 4.1
* NBX  NBY
  3    1
* CARD 4.2
* XO    YO
  0.0  0.0
* CARD 4.3
* W      DX    ALPAX
  14.0  0.3   -0.3
  21.0  0.3    0.5
  11.0  0.3    0.3
* CARD 4.4
* H      DY    ALPAY
  20.0  0.3    0.5
* CARD 4.5
* IGMOD
  1
* -----
* CARD 3.1
* FILEA
  BMESH.Dat
* FILEM
  ADD2D-5.Mes
* CARD 3.2
* NSNEL  NSNODE
  1      1
* CARD 3.3
* IEDIT = 4 : BUILD USER-SPECIFIED CURVES.
  4
* CARD 3.3.5.1
* NODE
  0
* CARD 3.3.5.2
* NOEL
  0
* CARD 3.3.5.3
* IBOUND
  0

```



```

* CARD 3.3.5.4
* NGROUP
  2
* XREF  YREF
 14.0  20.0
* ----- GROUP 1 -----
*
*                               SOFT ROCK
*
* CARD 3.3.5.4.1.1
* MTYPE
  3
* CARD 3.3.5.4.1.2
* MATNO IDH  LTPI  LMAT
  7    0    0    0
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO  YLO
  6      1    0    0.0  0.0
* CARD 3.3.5.4.2.2
* NP   X      Y
  1   0.0   0.0
  2  46.0   0.0
  3  46.0  13.0
  4  31.0  12.0
  5  19.0   8.0
  6   0.0   4.0
* CARD 3.3.5.4.3
* NSEGMENT
  6
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  1      1    0    3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  2      1    0    3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  3      1    0    2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  4      1    0    2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  5      1    0    2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  6      1    0    2

```

8-12 ADDRGN-2D Example Problem

```
* ----- GROUP 2 -----  
*           UTILITY TUNNEL  
* CARD 3.3.5.4.1.1  
* MTYPE  
*   -3  
* CARD 3.3.5.4.1.2  
* MATNO IDH MATNOJT IDHJT THICJT  LTPI, LMATI, LTPO, LMATO  
*   3    0    4      0    0.1    2    5    2    6  
* CARD 3.3.5.4.2.1  
* NPOINT MOVE IREF  XLO  YLO  
*   1      0    1    8.0 -6.0  
* CARD 3.3.5.4.2.2  
* NP  X      Y  
*   1  2.0   0.0  
* CARD 3.3.5.4.3  
* NSEGMENT  
*   1  
* CARD 3.3.5.4.3.1  
* SEGNO LTYPE NDIV IEND  
*   1      2    0    2  
* CARD 3.3.5.4.3.2  
* X0    Y0    RX    RY    THETA_B  THETA_E  
*  0.0   0.0   2.0   2.0   0.0      360.  
* -----  
* END OF DATA
```

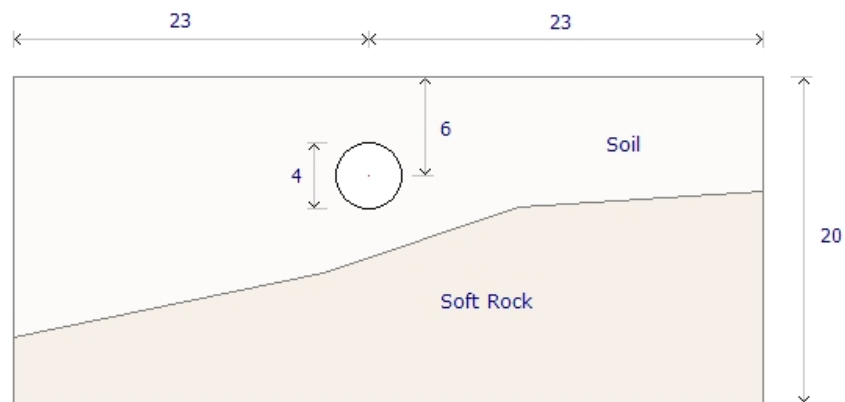
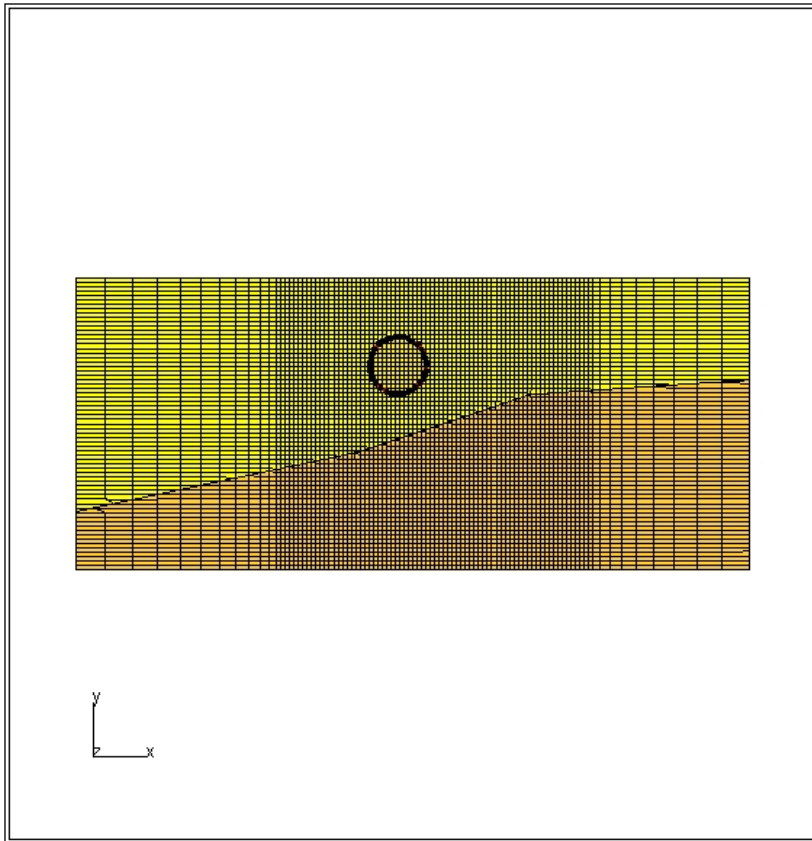
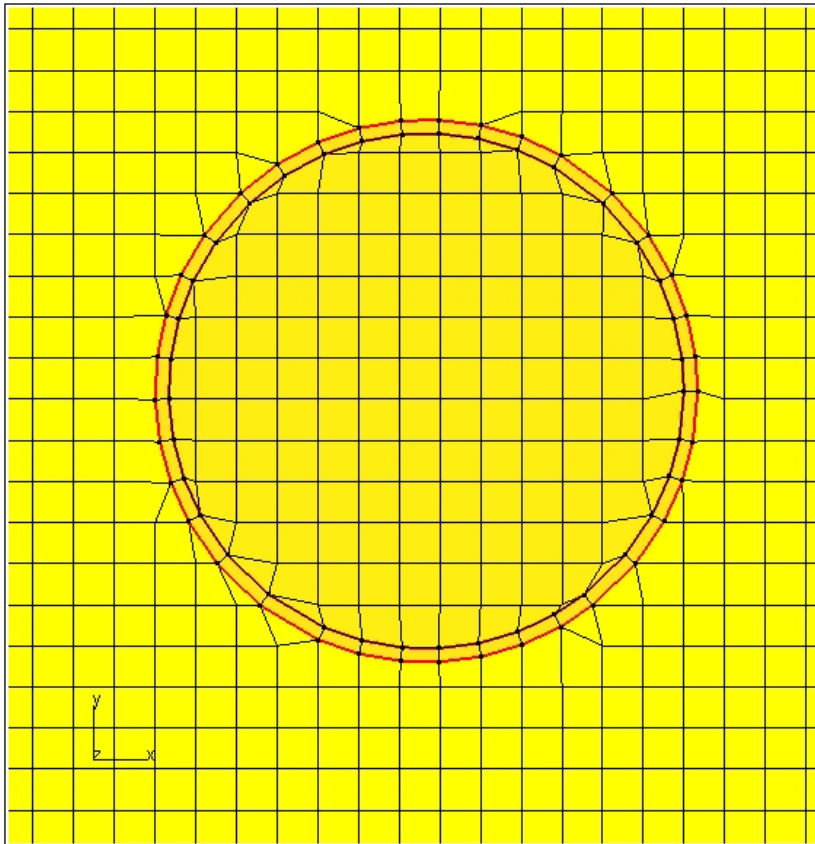


Figure 8.4 Schematic section view.



ADD2D-5.Mes

Figure 8.5 Overall finite element mesh



ADD2D-5.Mes

Figure 8.6 Finite element mesh around utility tunnel.

As the second example, we take a complex problem as schematically shown in Figure 8.7. The problem geometry includes different types of underground structures; strut, anchor bar, pile, utility tunnel, subway tunnel, rock bolt, foundation and fault zone. Table 8.6 shows the partial listing of input file **ADD2D-6.Dat**. The base mesh consists of 3 blocks in the horizontal direction and 2 blocks in the vertical direction.

For detailed description of input parameters, refer to Section 8.2 in SMAP-3D User's Manual. Joint elements are used to model the fault zone and the interfaces between surrounding medium and the structures such as pile and tunnels. Figure 8.8 shows overall finite element mesh generated by ADDRGN-2D. Detailed finite element meshes are shown in Figure 8.9 for the excavation zone and in Figure 8.10 for the tunnels and foundation. As in the previous example, the joint thickness in Figures 8.9 and 8.10 is exaggerated to show clearly both inner and outer joint faces. The real joint thickness is specified in material property card in Main File.

Table 8.6 Listing of input file **ADD2D-6.Dat** (Partial Listing)

```
* ADD2D-6.Dat
* CARD 1.1
* IMOD = 2 : GENERATE BASE MESH AND THEN MODIFY
* IMOD JK
  2    9
* CARD 4.1
* NBX  NBY
  3    2
* CARD 4.2
* XO    YO
  0.0  0.0
* CARD 4.3
* W      DX      ALPAX
  14.0  0.3    -0.3
  21.0  0.3     0.5
  11.0  0.3     0.3
* CARD 4.4
* H      DY      ALPAY
  23.0  0.3     0.5
  16.0  0.3     0.3
```

```

* CARD 4.5
* IGMOD
1
* -----
* CARD 3.1
* FILEA
  BMESH.Dat
* FILEM
  ADD2D-6.Mes
* CARD 3.2
* NSNEL  NSNODE
  1      1
* CARD 3.3
* IEDIT = 4 : BUILD USER-SPECIFIED CURVES.
  4
* CARD 3.3.5.1
* NODE
  0
* CARD 3.3.5.2
* NOEL
  0
* CARD 3.3.5.3
* IBOUND
  0
* CARD 3.3.5.4
* NGROUP
  22
* XREF  YREF
  14.0  39.0
* ----- GROUP 1 -----
*
*          MAKING GROUND SURFACE
*
* CARD 3.3.5.4.1.1
* MTYPE
  -1
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO  YLO
  8      1      0      0.0  0.0
* CARD 3.3.5.4.2.2
* NP      X      Y
  1  46.0  34.0
  2  39.0  34.0
  3  33.0  39.0
  4  18.0  39.0
  5  12.0  34.0
  6   0.0  34.0
  7   0.0   0.0
  8  46.0   0.0

```

8-18 ADDRGN-2D Example Problem

```
* CARD 3.3.5.4.3
* NSEGMENT
  8
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  1      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
  2      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  3      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
  4      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  5      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
  6      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  7      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
  8      1      0      2
* ----- GROUP 2 -----
*
*                               SOFT ROCK
*
* CARD 3.3.5.4.1.1
* MTYPE
  3
* CARD 3.3.5.4.1.2
* MATNO IDH  LTPI  LMAT
  7    0    0    0
* CARD 3.3.5.4.2.1
* NPOINT MOVE  IREF  XLO  YLO
  6      1      0    0.0  0.0
* CARD 3.3.5.4.2.2
* NP   X      Y
  1  46.0    0.0
  2  46.0   33.0
  3  31.0   32.0
  4  19.0   28.0
  5   0.0   24.0
  6   0.0    0.0
```



```

* CARD 3.3.5.4.3
* NSEGMENT
  6
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  1      1      0      3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  2      1      0      3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  3      1      0      0
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  4      1      0      0
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  5      1      0      0
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  6      1      0      3
* ----- GROUP 3 -----
*
*                               FAULT
*
* MTYPE
  -2
* CARD 3.3.5.4.1.2
* MATNOJT IDHJT THICJT  LTPI, LMATI, LTPO, LMATO
  5      0  -0.1      0      0      0      0
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO  YLO
  4      1      0    0.0  0.0
* CARD 3.3.5.4.2.2
* NP  X      Y
  1  46.0  29.0
  2  29.0  19.0
  3  16.0  14.0
  4   0.0  10.0
* CARD 3.3.5.4.3
* NSEGMENT
  3
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
  1      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEDN
  2      1      0      2

```

8-20 ADDRGN-2D Example Problem

```
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
      3      1      0      2
* ----- GROUP 4 -----
*
*                                FOUNDATION
*
* CARD 3.3.5.4.1.1
* MTYPE
      4
* CARD 3.3.5.4.1.2
* MATNO  IDH  LTPI  LMAT
      2      0      0      0
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO  YLO
      8      1      0    0.0  0.0
* CARD 3.3.5.4.2.2
* NP    X      Y
      1  34.5  29.0
      2  34.5  30.0
      3  32.5  30.5
      4  32.5  39.0
      5  31.5  39.0
      6  31.5  30.5
      7  29.5  30.0
      8  29.5  29.0
* CARD 3.3.5.4.3
* NSEGMENT
      8
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
      1      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
      2      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
      3      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
      4      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
      5      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
      6      1      0      2
* CARD 3.3.5.4.3.1
```

```

* SEGNO  LTYPE  NDIV  IEND
   7      1      0      2
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
   8      1      0      2
* ----- GROUP 5 -----
*
*                LEFT UTILITY TUNNEL
*
* CARD 3.3.5.4.1.1
* MTYPE
   -3
* CARD 3.3.5.4.1.2
* MATNO IDH MATNOJT IDHJT THICJT  LTPI, LMATI, LTPO, LMATO
   3      0      4      0    -0.1      2      5      2      6
* CARD 3.3.5.4.2.1
* NPOINT  MOVE  IREF  XLO  YLO
   1      0      1    8.0  -6.0
* CARD 3.3.5.4.2.2
* NP  X      Y
   1  2.0    0.0
* CARD 3.3.5.4.3
* NSEGMENT
   1
* CARD 3.3.5.4.3.1
* SEGNO  LTYPE  NDIV  IEND
   1      2      0      2
* CARD 3.3.5.4.3.2
* X0  Y0  RX  RY  THETA_B  THETA_E
  0.0  0.0  2.0  2.0  0.0    360.
* ----- GROUP 6 -----
*
*                RIGHT UTILITY TUNNEL
*
* CARD 3.3.5.4.1.1
* MTYPE
   -3
* CARD 3.3.5.4.1.2
* MATNO KF MATNOJT KFJT  THICJT  LTPI, LMATI, LTPO, LMATO
   3      0      4      0    -0.1      2      5      2      6
* CARD 3.3.5.4.2.1

```

8-22 ADDRGN-2D Example Problem

```
* ----- GROUP 22 -----
*
*           SUBWAY TUNNEL
*
* CARD 3.3.5.4.1.1
* MTYPE IGPOST OVERLAY GCOLOR GLTYPE GLTHIC GHIDE
*  -3    0    0    0    0    0    0
* Card 3.3.5.4.1-1
* MAT IDH MATj IDHj THICj LTi LMi LTo LMo
*   3    0    4    0 -0.100  2    5    2    6
* CARD 3.3.5.4.2.1
* NPOINT MOVE IREF XLO YLO
*   4    1    1    0.0  0.0
* CARD 3.3.5.4.2.2
* NP   X      Y
*   1  26.  24.
*   2  20.  24.
*   3  20.  20.
*   4  26.  20
* CARD 3.3.5.4.3
* NSEGMENT
*   4
* CARD 3.3.5.4.3.1
* SEGNO LTYPE NDIV IEND
*   1    2    0    2
* Card 3.3.5.4.3.1-1
* Xo     Yo     Rx     Ry     Qb     Qe
* 23.    24.    3.0    3.0    0.0   180.
* Card 3.3.5.4.3.1
* SEGNO LTYPE NDIV IEND
*   2    1    0    2
* Card 3.3.5.4.3.1
* SEGNO LTYPE NDIV IEND
*   3    1    0    2
* Card 3.3.5.4.3.1
* SEGNO LTYPE NDIV IEND
*   4    1    0    2
* -----
* END OF DATA
```

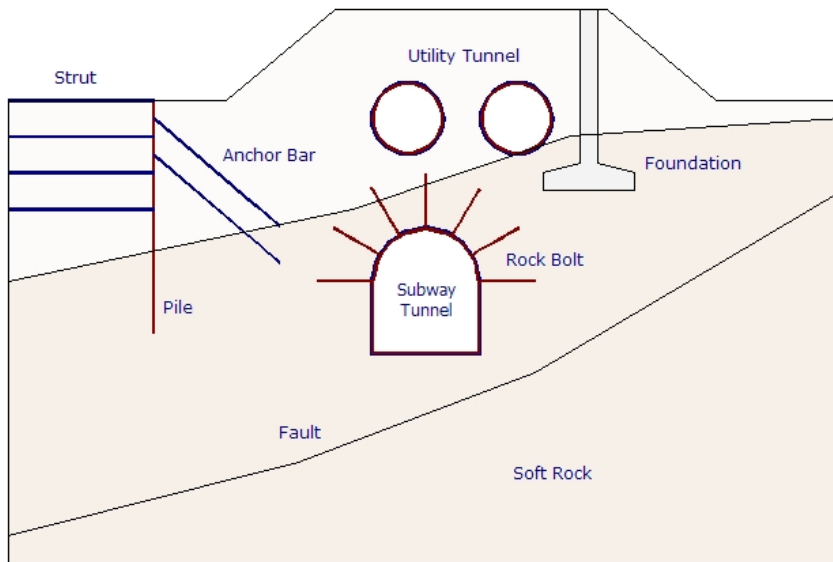
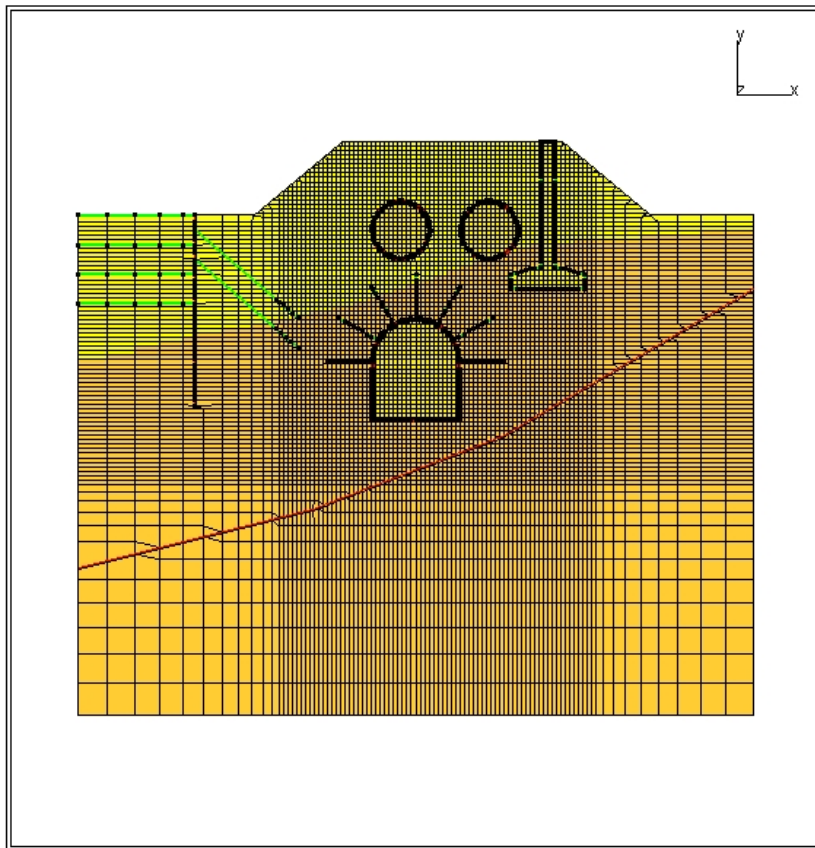
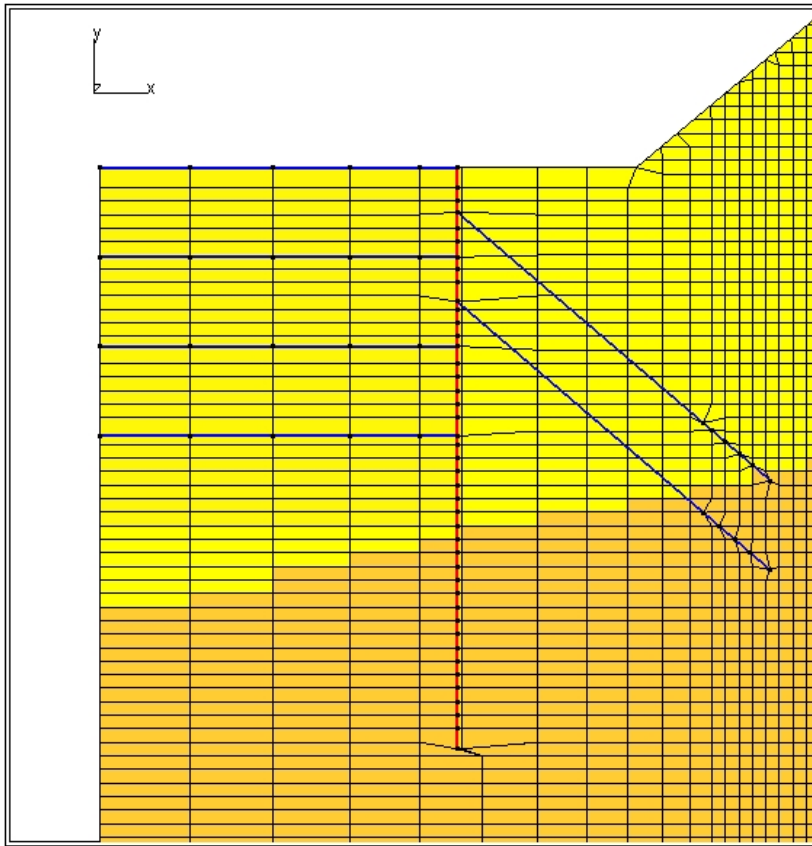


Figure 8.7 Schematic section view



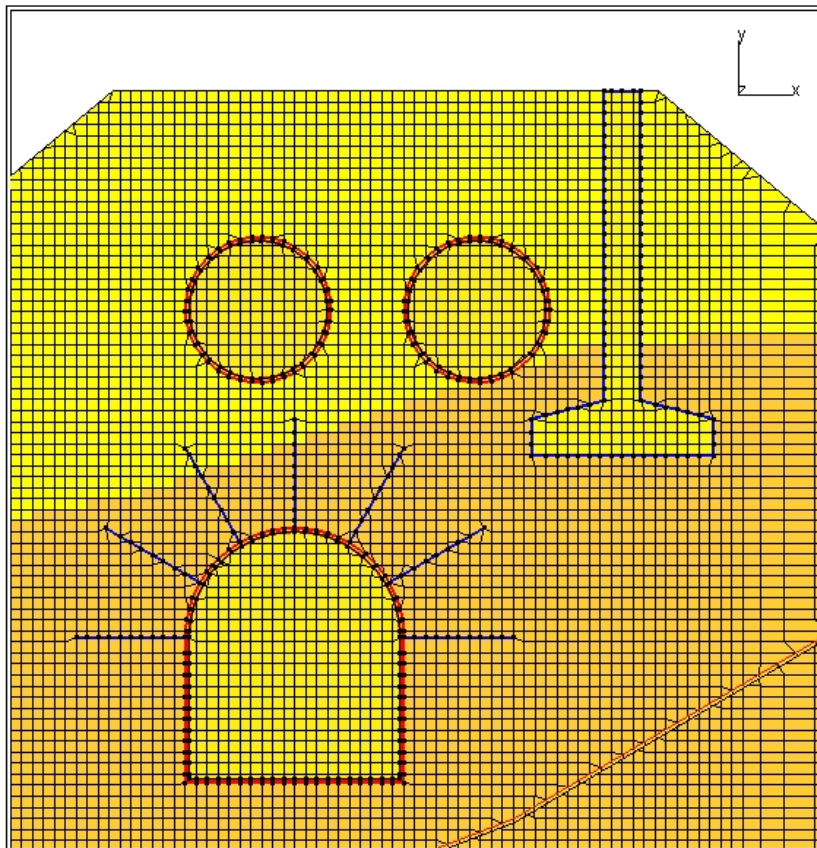
ADD2D-6.Mes

Figure 8.8 Overall finite element mesh



ADD2D-6.Mes

Figure 8.9 Finite element mesh around excavation zone



ADD2D-6.Mes

Figure 8.10 Finite element mesh around tunnels and foundation

SUPPLEMENT

Example Problem

SUPPLEMENT Menu contains supporting programs which are useful to prepare input data for pre- and main-processing programs of SMAP-W2.

Running SUPPLEMENT is described in Section 3.2.6 of User's Manual and can be selected in the following order:

Run → Mesh Generater → Supplement → Edit, XY. Cards or Shrink File

EDIT is used to run text editor.

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

CARDS generates element activity data in Card 8 in Section 4.4 Main File.

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

9.1 XY Example Problem

XY is the supporting program which computes coordinates of mid points, cross points, or normal points. Full description of XY is presented in Section 9.3 of User's Manual.

As an example, we select **NF=6** which computes coordinates of point normal to the circular arc as shown in Figure 9.1.

9-2 SUPPLEMENT Example Problem

Table 9.1 illustrates options available to the program XY and the user inputs specific to NF=6. Computed coordinates of the normal point are stored in the output file **XY.Out** and are listed in bottom part of Table 9.1.

Table 9.1 XY Example Problem

Type file name to store output: **XY.Out**

- NF = 0 End of Computation

 - 1 Compute Midpoint on Straight Line
 - 2 Compute Midpoint on circular Arc
 - 3 Compute Intersection Point of Two Straight Lines
 - 4 Compute Intersection Point of Circular Arc and Straight Line
 - 5 Compute Points Normal to Straight Line
 - 6 Compute Points Normal to Circular Arc

NF= **6**

R, Xo, Yo, TA
5.0 0.0 0.0 0.0

TAC, CD
45.0 3.0

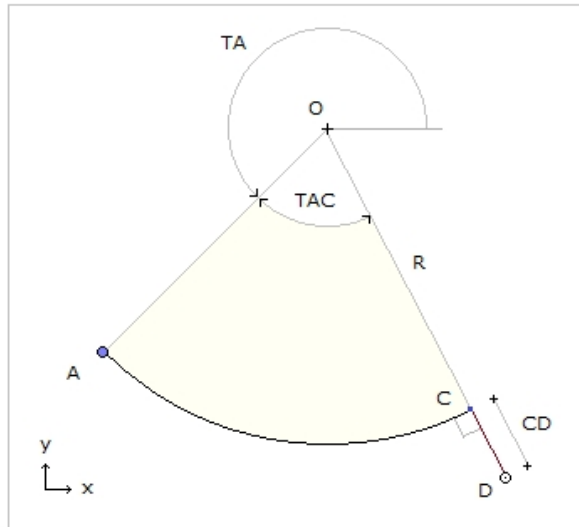
User inputs are **bold**.

Output file contains following information:

Compute Points Normal to Circular Arc

R = 5.
Xo = 0. Yo = 0.
TA = 0.
TAC = 45. CD = 3.
XC = 3.535527 YC = 3.535540
XD = 5.656844 YD = 5.656865

NF = 6 Compute Points Normal to Circular Arc



INPUT:

$R,$ $X_o,$ $Y_o,$ TA
 $TAC,$ CD

R = 5.0
 X_o, Y_o = 0.0, 0.0
 TA = 0.0
 TAC = 45.0
 CD = 3.0

Figure 9.1 XY example problem

9.2 CARDS Example Problem

CARDS is the supporting program which is written to aid the preparation of SMAP-3D input cards. Currently, there is only one routine available to generate element activity data in Card Group 8.2 of Users Manual.

Table 9.2 shows user inputs for the example problem. Generated element activity data is stored in the output file, **CARDS.Out**, which is listed in Table 9.3.

Table 9.2 User inputs for CARDS example problem

CARD NO = 0	Exit
8.2	Element Activity

CARD NO = **8.2**

Type file name to store output: **CARDS.OUT**

NF = 0	End of Generation
1	Generate Element Activity / Deactivity

NF = **1**

NEL (start), NEL (end), NAC, NDAC

101 120 0 6

NF = 0	End of Generation
1	Generate Element Activity / Deactivity

NF = **1**

NEL (start), NEL (end), NAC, NDAC

121 130 3 50

NF = 0	End of Generation
1	Generate Element Activity / Deactivity

NF = **0**

User inputs are **bold**.

Table 9.3 Listing of output file CARDS.Out

* NEL	NAC	NDAC
101	0	6
102	0	6
103	0	6
104	0	6
105	0	6
106	0	6
107	0	6
108	0	6
109	0	6
110	0	6
111	0	6
112	0	6
113	0	6
114	0	6
115	0	6
116	0	6
117	0	6
118	0	6
119	0	6
120	0	6
*		
121	3	50
122	3	50
123	3	50
124	3	50
125	3	50
126	3	50
127	3	50
128	3	50
129	3	50
130	3	50
* NFAD = 30		

10.1 LOAD-2D

LOAD-2D is the pre-processing program which can be used to generate initial head, external water flow, specified head and potential seepage face boundary. For the detailed description of input parameters, refer to section 11 of User's Manual.

LOAD-2D can be selected in the following order:

Run → Load Generator → Load 2D

When you select LOAD-2D, Load Generation Dialog will be displayed as in Figure 10.1. You need to specify input file names for Load and Mesh Data.

10.1.1 Example 1

Example 1 is to show most of load generations related to seepage problem as schematically shown in Figure 10.2. This problem is the same as SMAP-W2 Verification Problem 4 which is the axially symmetric well subjected to sudden drawdown of left side water table to the bottom of upper block.

Left side surface of upper block represents potential seepage face boundary, left side surface of lower block represents total head of 0.305m and the right side surface represents total head of 1.22m. Initial total heads are constant value of 1.22m over the whole region. Two different load time histories for left and right side water tables are considered.

Mesh Data contains information for nodal coordinates and element indexes. VP4_NB.Mes represents Mesh Data graphically shown in Figure 10.3. Load Data contains information for loads to be generated. Load_W2.dat in Table 10.1, has been prepared according to LOAD-2D User's Manual.

10-2 LOAD-2D Example Problem

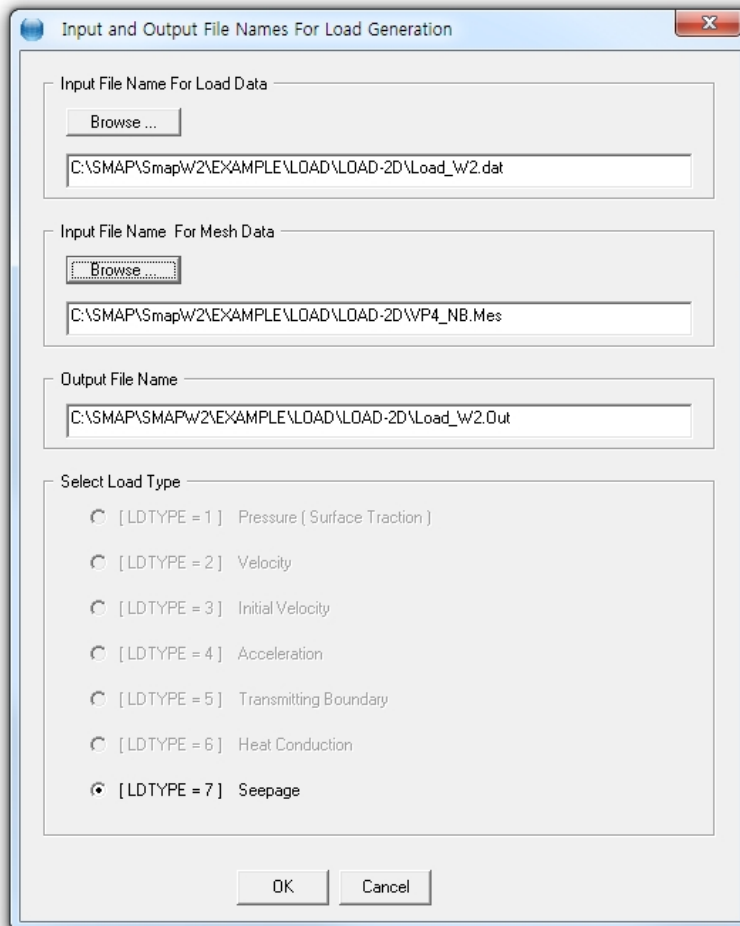


Figure 10.1 Load generation dialog

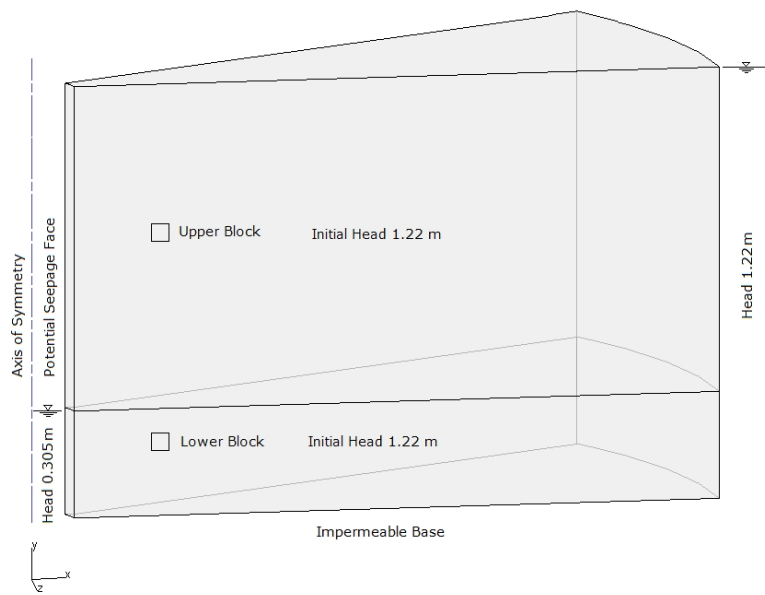


Figure 10.2 Schematic view of load for Example 1

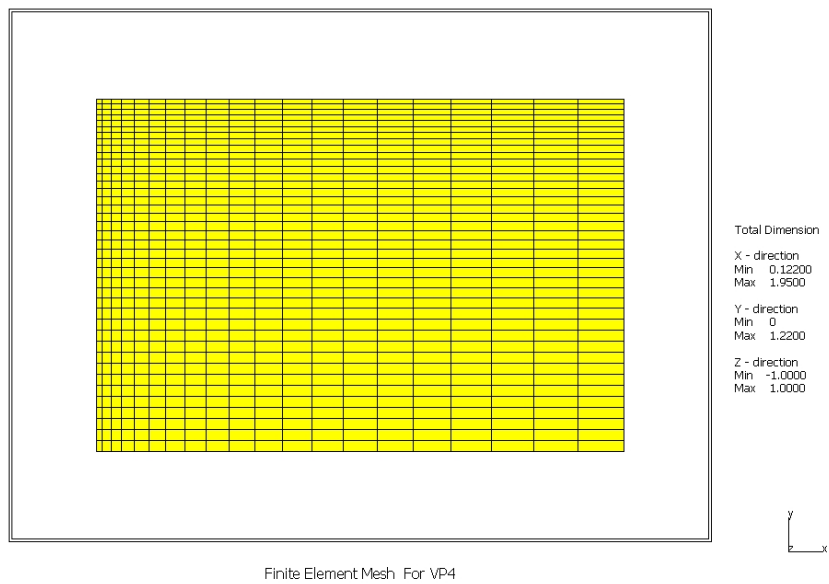


Figure 10.3 Finite element mesh for Example 1

Table 10.1 Listing of load input file Load_W2.dat for Example 1

```

* LOAD-W2  INPUT
* CARD 1.1
* TITLE
  EXAMPLE 1  LOAD-W2 [LDTYPE = 7]
* CARD 1.2
* NCTYPE
  1
* =====
* CARD 2.1
* NUMLS
  3
* -----
* Potential Seepage Face
* CARD 2.2.1
* LSNO   LSTYPE (Node Group)
  1       3
* CARD 2.2.4
* NUMNODG
  1
* Upper Block Left Surface
* CARD 2.2.5
* NSR   JCR   NJR   ICR   NIR
  1     1     33    0     1
* -----
* Total Head 1
* CARD 2.2.1
* LSNO   LSTYPE (Node Group)
  2       3
* CARD 2.2.4
* NUMNODG
  1
* Lower Block Left Surface
* CARD 2.2.5
* NSR   JCR   NJR   ICR   NIR
  694   1     8     0     1
* -----
* Total Head 2
* CARD 2.2.1
* LSNO   LSTYPE (Node Group)
  3       3

```

10-6 LOAD-2D Example Problem

```
* CARD 2.2.4
* NUMNODG
  2
* Upper Block Right Surface
* CARD 2.2.5
* NSR   JCR   NJR   ICR   NIR
  661   1     33    0     1
* Lower Block Right Surface
* CARD 2.2.5
* NSR   JCR   NJR   ICR   NIR
  854   1     8     0     1
* =====
* CARD 3.1
* NUMLF
  2
* -----
* INITIAL HEAD
* CARD 3.2.1
* LFNO
  1
* CARD 3.2.2
* A-0      A-X      A-Y
  1.22     0.0      0.0
* -----
* CF
* CARD 3.2.1
* LFNO
  2
* CARD 3.2.2
* A-0      A-X      A-Y
  1.0      0.0      0.0
* =====
* CARD 4.1
* NUMLH
  2
* -----
* CARD 4.2.1
* LHNO
  1
* CARD 4.2.2
* NUMTP
  2
```

```

* CARD 4.2.3
* T1      T2
  0.0     1000.
* CARD 4.2.4
* C1      C2
  0.305   0.305
* -----
* CARD 4.2.1
* LHNO
  2
* CARD 4.2.2
* NUMTP
  2
* CARD 4.2.3
* T1      T2
  0.0     1000.
* CARD 4.2.4
* C1      C2
  1.22    1.22
* =====
* INITIAL HEAD
* CARD 5.1
* IBTYPE
  1
* CARD 5.1-1
* LFNO_IH  LHED_IH
  1        0
* -----
* POTENTIAL SEEPAGE FACE
* CARD 5.1
* IBTYPE
  6
* CARD 5.1-6
* LSNO_PS  LFNO_PS  LHNO_PS
  1        2        0
* -----
* TOTAL HEAD 1
* CARD 5.1
* IBTYPE
  5
* CARD 5.1-6
* LSNO_HB  LFNO_HB  LHNO_HB  LHED_HB
  2        2        1        0

```

10-8 LOAD-2D Example Problem

```
* -----
* TOTAL HEAD 2
* CARD 5.1
* IBTYPE
  5
* CARD 5.1-6
* LSNO_HB  LFNO_HB  LHNO_HB  LHED_HB
  3         2       2        0
* -----
* END
* CARD 5.1
* IBTYPE
  0
* -----
* END OF INPUT DATA
```

The output file [Load_W2.out](#) contains load time histories for total heads for left and right boundary surfaces. The format of the generated load output is compatible to the format of Card Group 9.4 in SMAP-W2 main input.

The output file [LoadingSurface.out](#) contains all nodes representing flow sections where flow rates are computed. The format of the generated load output is compatible to the format of Card Group 10.8 in SMAP-W2 main input.

For IBTYPE = 1, 4, 5, or 6, specified initial heads or boundary conditions are saved in mesh file ([NewMeshFile.Mes](#)).

XY Graph Example Problem

[XY Graph](#) is a two-dimensional graph consisting of lines connecting each pair of data points, which can be plotted by [PLOT XY](#) or [EXCEL](#). [XY Graph User's Manual](#) describes all the basic functions associated with XY graph creation and modifications.

Two example problems are presented:

1. [New Graph](#)

Shows step-by-step procedure to create and modify XY graph.

Main actions:

- Access XY graph
- Edit initial Draft XY
- Modify XY graph by Edit dialog
- Open XY graph on Excel Spreadsheet

2. [SMAP Result](#)

Plots SMAP results specified in Card Group 12 in SMAP Post File.

Main actions:

- Execute SMAP-W2 example
- Access SMAP result
- Access PLOT XY in Plot menu
- Modify XY graph by Edit dialog
- Open XY graph on Excel Spreadsheet

11-2 XY Graph Example

11.1 New Graph

The main objective of this first example is to show the step-by-step procedure to create and modify XY graph.

This example consists of the following main actions:

- Access XY graph
- Edit initial Draft XY
- Modify XY graph by Edit dialog
- Open XY graph on Excel Spreadsheet

Step 1: Access XY Graph (New)

Access [XY Graph](#) by selecting following items in [SMAP](#) (Figure 11.1):
[Plot](#) → [XY](#) → [PLOT XY](#) → [New](#)

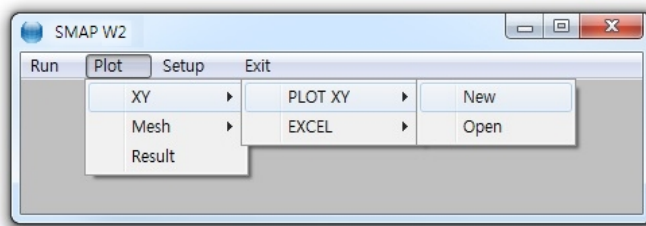


Figure 11.1 Accessing XY graph (New)

Step 2: Edit Initial Draft XY

Once selected, initial default file [XY.dat](#) will be opened by [Notepad](#) as listed in Table 11.1.

Edit the first plot in this default file as listed in Table 11.2.
And then save and exit.

Modified graph will be displayed on [PLOT XY](#) drawing board as shown in Figure 11.2.

Table 11.1 Draft XY Data (Initial Default [File XY.dat](#))

```

Plot No. 1
Sub Title 1
XLabel-1
YLabel-1
0      10
100    20
.000000E+00 .123456E+06
Curve 1
Legend
10,      20
90,      30
.000000E+00 .123456E+06
Curve 2
Legend
.000000E+00 .987654E+06
Plot No. 2
Sub Title 2
XLabel-2
YLabel-2
0      100
1000   200
.000000E+00 .123456E+06
Curve 1
Legend
100     200
900     300
.000000E+00 .123456E+06
Curve 2
Legend
.000000E+00 .987654E+06
Plot No. 3
Sub Title 3
XLabel-3
YLabel-3
0      100
1000   200
.000000E+00 .123456E+06
Curve 1
Legend
200,    200
900,    300
.000000E+00 .123456E+06
Curve 2
Legend
.000000E+00 .987654E

```

11-4 XY Graph Example

Table 11.2 Modified Draft XY Data ([File XY.dat](#))

```
Example 1
Stress History
Time (Sec)
Stress (MPa)
0      10
100    20
.000000E+00 .123456E+06
Vertical
Stress
0      20
100    30
.000000E+00 .123456E+06
Horizontal
Stress
.000000E+00 .987654E+06
Plot No. 2
Sub Title 2
XLabel-2
YLabel-2
0      100
1000   200
.000000E+00 .123456E+06
Curve 1
Legend
100    200
900    300
.000000E+00 .123456E+06
Curve 2
Legend
.000000E+00 .987654E+06
Plot No. 3
Sub Title 3
XLabel-3
YLabel-3
0      100
1000   200
.000000E+00 .123456E+06
Curve 1
Legend
200,   200
900,   300
.000000E+00 .123456E+06
Curve 2
Legend
.000000E+00 .987654E
```

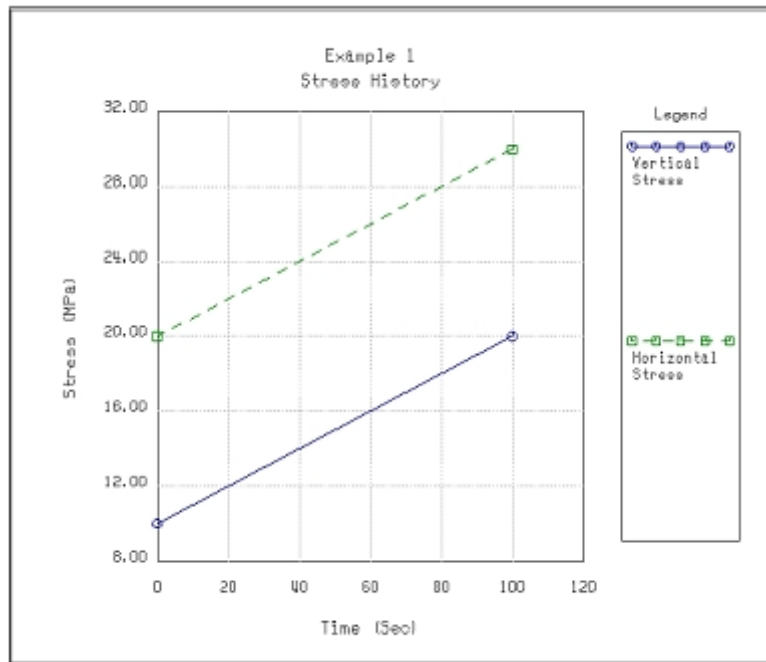


Figure 11.2 Modified graph on PLOT XY

11-6 XY Graph Example

Step 3: Modify XY Graph by Edit Dialog

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

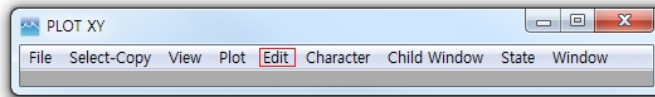


Figure 11.3 Edit menu in PLOT XY

[Edit dialog](#) will be displayed as shown in Figure 11.4.

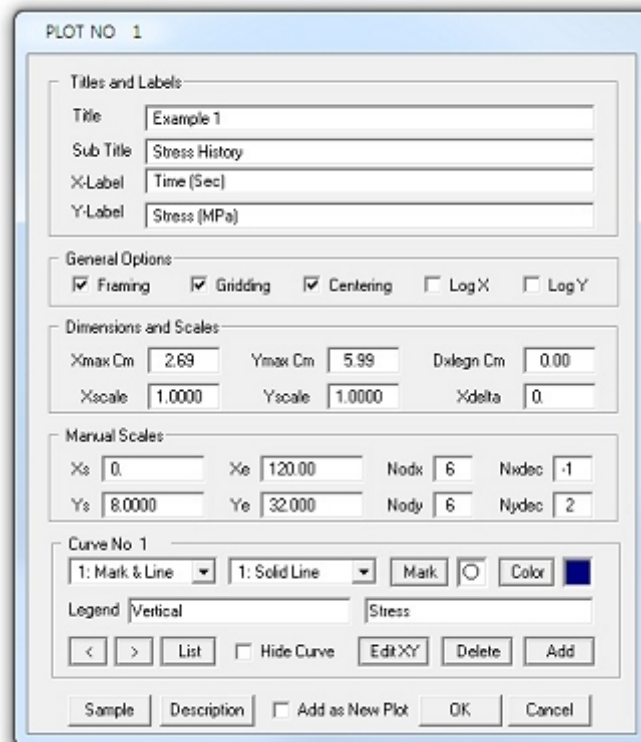


Figure 11.4 Edit dialog

There are many different options available for changing view of XY graphas described in detail in Section 12.3 in [XY Graph User's Manual](#).

Here, change the color of the first curve into Red.

Click [Color](#) button and select Red from [Color Palette dialog](#).

Then Red color will be updated for first curve as shown in Figure 11.5.

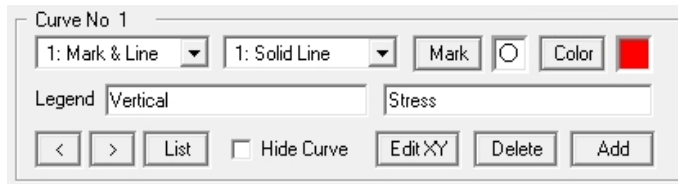


Figure 11.5 Updated red color in Edit dialog

Click [OK](#) button in [Edit dialog](#).

Then updated plot will be displayed on [PLOT XY](#) as in Figure 11.6.

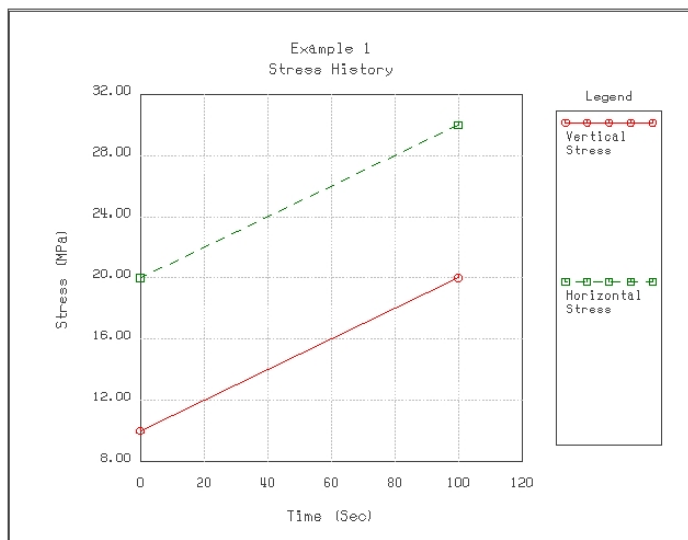


Figure 11.6 Updated first curve on PLOT XY

11-8 XY Graph Example

Step 4: Open XY Graph on Excel Spreadsheet

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

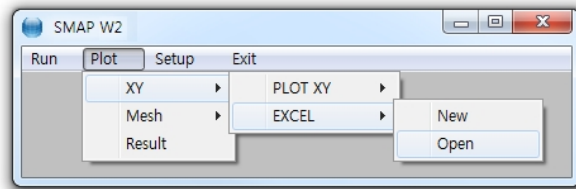


Figure 11.7 Accessing XY graph on Excel (Open)

Open **XY.dat** in the current working directory.

XY graph will be displayed on **Excel Spreadsheet** as in Figure 11.8.

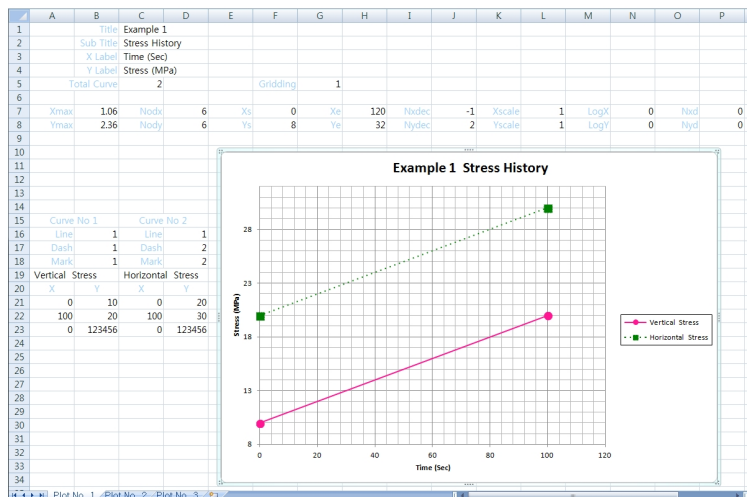


Figure 11.8 XY graph on Excel spreadsheet

Refer to more samples in the following directory:

C:\Smap\SmapW2\Example\XY_Graph\Excel XY Graph Sample.docx

11.2 SMAP Result

The main objective of this second example is to show step-by-step procedure to plot SMAP results specified in Card Group 12 in SMAP Post File. This example involves SMAP-W2 Example Problem 1 (1-D Transient Linear Consolidation).

This example consists of the following main actions:

- Execute SMAP-W2 example
- Access SMAP result
- Access PLOT XY in Plot menu
- Modify XY graph by Edit dialog
- Open XY graph on Excel Spreadsheet

Step 1: Execute SMAP-W2 Example

Execute [SMAP-W2](#) by selecting the following menu items in [SMAP](#) (Figure 11.9): [Run](#) → [Smap](#) → [Execute](#)

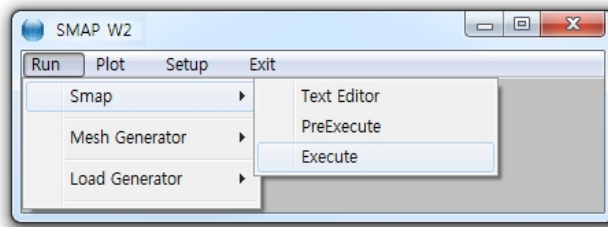


Figure 11.9 Execute SMAP-W2 example problem

Note that [SMAP-W2 Example Problem 1](#) includes XY graph specified in Card Group 12 in SMAP Post File [Vp1.Pos](#) as listed in Table 11.3

Step 2: Access SMAP Result

Access [SMAP Result](#) by selecting the following menu items in [SMAP](#) : [Plot](#) → [Result](#)

11-10 XY Graph Example

Table 11.3 SMAP-W2 post file ([File Vp1.Pos](#))

```
* PLOT-2D Information
* CARD 11.1
* NPTYPE
  0
* PLOT-XY Information
* CARD 12.1
* IPTYPE
  3
* Card 12.4.1
* IPLOT
  1
* Card 12.4.2
* NOTM
  2
* Card 12.4.3
* TLIST1 TLIST2 ...
  3.71    37.1
* Card 12.4.4
* NDPQ
  1
* Card 12.4.5
* KY2
  51
* Card 12.4.6
* ISCALD  ILTNUM  XSTART
  1        1      0.25
*
* Card 12.4.7
  2
* NSTAR   NINCR   NPONT
  1        1      20
  0
* Card 12.4.8
* TPFAC   SDFAC
  1        1
* Card 12.4.9
* TITLE / X-LABEL / Y-LABEL
  Profiles of Pore Water Pressure (SMAP-W2)
  M
  Pore Water Pressure (t/m2)
* CARD 12.1
* IPTYPE
  0
* End of Data
```


Step 3: Access PLOT XY in Plot Menu

Select **PLOT XY** in **Plot Menu** dialog in Figure 11.10.

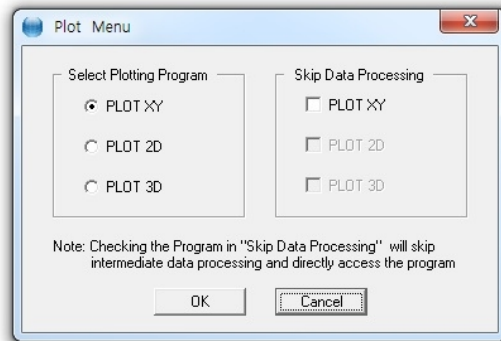


Figure 11.10 Plot menu dialog

Step 4: Modify XY Graph by Edit Dialog

Once XY graph is displayed on **PLOT XY**, access **Edit dialog** by clicking the **Edit** menu in **PLOT XY** as shown in Figure 11.11

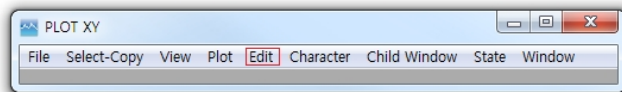


Figure 11.11 Edit menu in PLOT XY

11-12 XY Graph Example

Modify [Edit dialog](#) as shown in Figure 11.12.

The main modification is to change scales and color in XY graph.

Click [OK](#) button in [Edit dialog](#).

PLOT NO 1

Titles and Labels

Title: Profiles of Pore Water Pressure (SMAP-W2)
Sub Title: AT TIMES .371000E+01 .371000E+02
X-Label: M
Y-Label: Pore Water Pressure (t/m2)

General Options

☒ Framing ☒ Gridding ☒ Tick Mark ☐ Log X ☐ Log Y

Dimensions and Scales

Xmax Cm: 12.70 Ymax Cm: 12.70 Dxlegn Cm: 0.00
Xscale: 1.0000 Yscale: 1.0000 Xdelta: 0.

Manual Scales

Xs: 0. Xe: 10.000 Nodx: 5 Nxdec: -1
Ys: 0. Ye: 20.000 Nody: 4 Nydec: -1

Curve No 1
1: Mark & Line 1: Solid Line Mark ☐ Color ☒

Legend: T = .3710E+01

< > List ☐ Hide Modify XY Edit XY Delete Add

Sample Description ☐ Add as New Plot OK Cancel

Figure 11.12 Edit dialog

Modified XY graph is shown in figure 11.13.

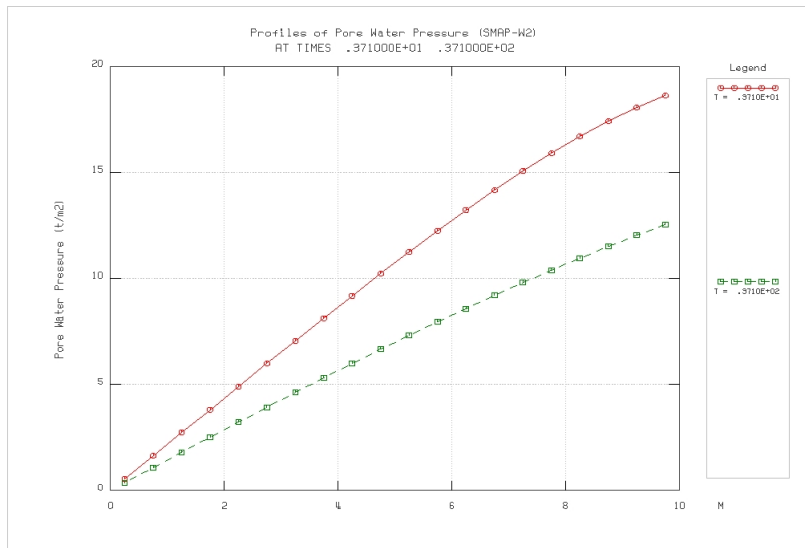


Figure 11.13 XY graph on PLOT XY

Refer to more samples in the following directory:

C:\Smap\SmapW2\Example\XY_Graph\PLOT XY Graph Sample.docx

11-14 XY Graph Example

Step 5: Open XY Graph on Excel Spreadsheet

Access **XY Graph** by selecting following items in **SMAP** (Figure 11.14):
Plot → XY → EXCEL → Open

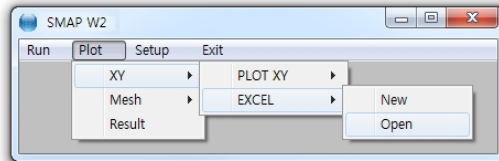


Figure 11.14 Accessing XY graph on Excel (Open)

Open [PlotXy.Lin](#) in the current working directory.
XY graph will be displayed on **Excel Spreadsheet** as in Figure 11.15.

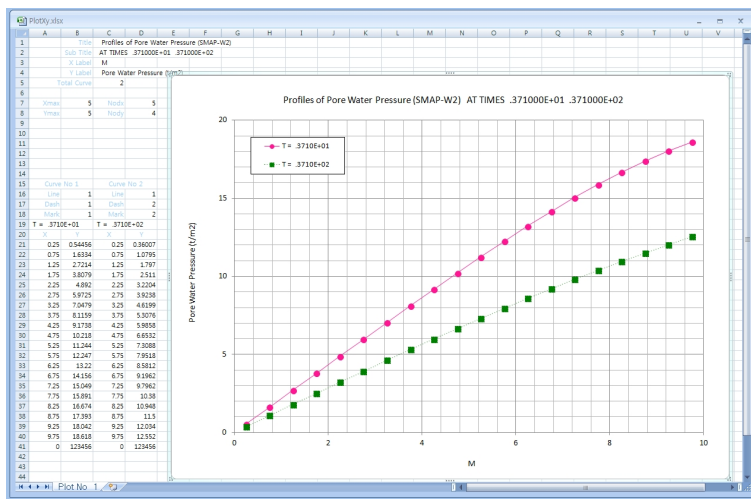


Figure 11.15 XY graph on Excel spreadsheet

Refer to more samples in the following directory:

C:\Smap \SmapW2 \Example \XY_Graph \Excel XY Graph Sample.docx

SWCC Example Problem

SWCC program supports following SWCC models:
van Genuchten (1980), Fredlund and Xing (1994)
and Brutsaert (1966) & Gardner (1958)

Graphical output file:

SWCC.Lin plots following graphs as a function of soil suction:

- Volumetric water content
- Normalized volumetric water content
- Specific moisture capacity
- Relative hydraulic conductivity
- Absolute hydraulic conductivity in log scale
- Absolute hydraulic conductivity in arithmetic scale
- Volumetric water content vs absolute hydraulic conductivity

Text output files:

PresfunctionM.dat

Permeability and specific moisture capacity

PresfunctionV.dat

Permeability and volumetric water content

PresfunctionA.dat

Permeability, volumetric water content and water capacity

Note:

Graphical file SWCC.Lin can be plotted by PLOT-XY.

Text output files can be used as input data for SMAP-F2/W2/W3.

12-2 SWCC Example

Prepare Input File

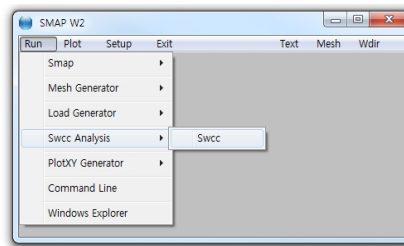
This example represents SM soil data fitted to van Genuchten Model by M.W. Gui, etc (Ref. Comparison of Two Water Storage Functions of Soil on Pore Water Pressure of Earth-Filled Dam Under Changing Environment, 28th International Symposium on Automation and Robotics in Construction, ISARC 2011 - Seoul, Korea, Republic of Duration: 29 Jun 2011 → 2 Jul 2011)

Input file in Table 12.1 is prepared according to [SWCC Users Manual](#).

Access SWCC

Access [SWCC](#) by selecting following items in [SMAP](#).
[Run](#) → [Swcc Analysis](#) → [Swcc](#)

Figure 12.1 Accessing SWCC



Select Input File

Once clicked SWCC, browse input file as shown in Figure 12.2.

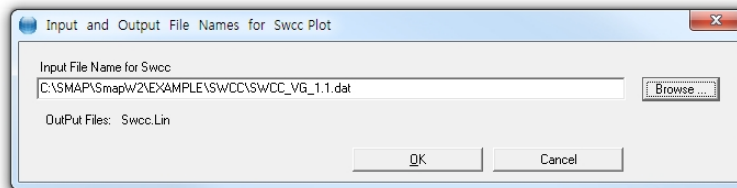


Figure 12.2 Selecting SWCC input file

Output Files

Text output file is listed in Table 12.2.

Graphical outputs are shown in Figures 12.3, 12.4 and 12.5.

Table 12.1 SWCC input file (SWCC_VG1.1.dat)

```

* CARD 1.
* TITLE
  van Genuchten Model (1980)
* SM soil from Table 1 in Ref. 6
* CARD 2
* MATFN  MW(m2/kN)  GW(kN/m3)  KSAT(m/d)  XSmin  XEmax  DLZ
  11      5.0E-05    9.81      0.6048    0.01   1e+06  0.02
* CARD 3
* VWCsat  VWCres    a          n          m          Uswcc  SCFac  Gwm
  0.35    0.0      0.06816  1.411    0.2913   0      1      9.81
* END OF DATA

```

Table 12.2 SWCC output file (PresFunctionA.dat)

```

* CARD 9.5.1
* NWPRF      NWPR      IWPF
  1          402        1
*
* CARD 9.5.2
*   N   H [L]      TH [L3/L3]    AK [L/T]      CA [1/L]
  1   .000000E+00  .350000E+00  .604800E+00  .000000E 00
  2   .101937E-02  .349997E+00  .545881E+00  .480503E-02
  3   .106741E-02  .349996E+00  .544785E+00  .489682E-02
  4   .111771E-02  .349996E+00  .543669E+00  .499037E-02
  5   .117039E-02  .349996E+00  .542532E+00  .508571E-02
  6   .122555E-02  .349995E+00  .541375E+00  .518287E-02
  7   .128331E-02  .349995E+00  .540197E+00  .528188E-02
  8   .134379E-02  .349995E+00  .538998E+00  .538278E-02
  9   .140712E-02  .349995E+00  .537778E+00  .548561E-02
  -   -           -           -           -
  396 .773259E+05  .404292E-02  .452231E-08  .214901E-07
  397 .809701E+05  .396712E-02  .451766E-08  .201380E-07
  398 .847861E+05  .389273E-02  .451285E-08  .188711E-07
  399 .887820E+05  .381974E-02  .450791E-08  .176838E-07
  400 .929661E+05  .374812E-02  .450282E-08  .165713E-07
  401 .973475E+05  .367784E-02  .449758E-08  .155287E-07
  402 .101935E+06  .360888E-02  .449221E-08  .145517E-07
* END OF DATA

```

12-4 SWCC Example

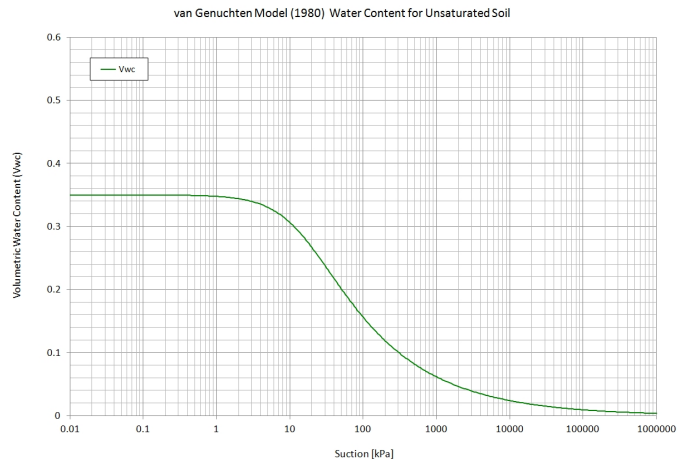


Figure 12.3 Volumetric water content

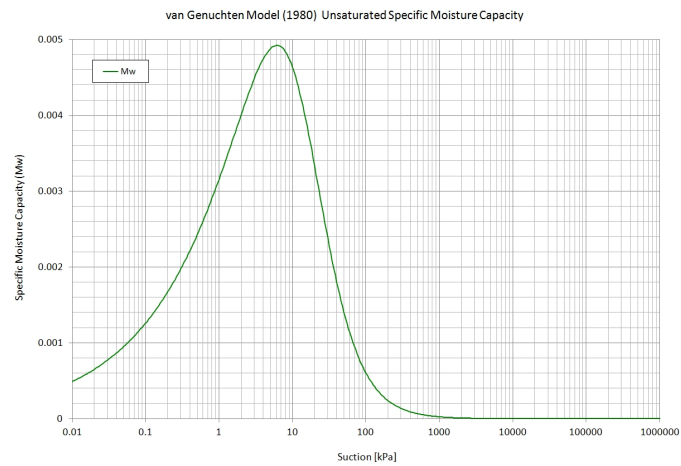


Figure 12.4 Specific moisture capacity

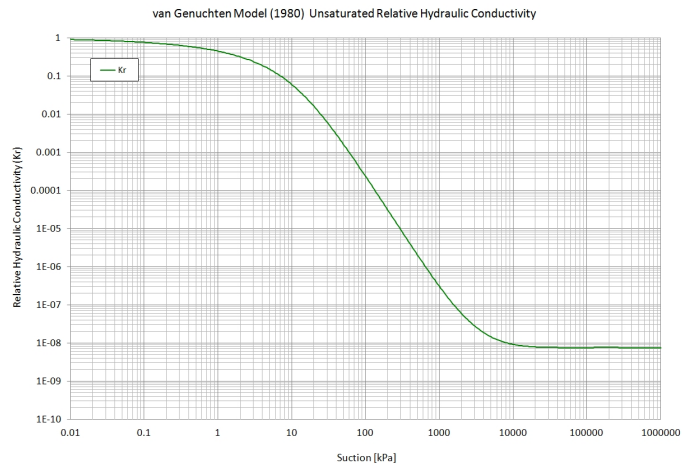


Figure 12.5 Relative hydraulic conductivity

Go to [Edit](#) > [Preferences](#) > [Page Display](#) > Uncheck [Enhance Thin Lines](#)

SMAP[®] - W2

Structure Medium Analysis Program
2-D Seepage Analysis

Theory

Contents**1. Introduction**

1.1 Introduction	1-1
----------------------------	-----

2. Finite Element Formulations

2.1 Introduction	2-1
2.2 Finite Element Formulation	2-2

3. SWCC Models

3.1 SWCC Models	3-1
3.1.1 Introduction	3-1
3.1.2 van Genuchten (1980)	3-2
3.1.3 Fredlund and Xing (1994)	3-3
3.1.4 Brutsaert(1966) and Gardner (1958)	3-4
3.1.5 Simple Gardner (1958)	3-5

4. Reference 4-1

Introduction

1.1 Introduction

This paper introduces the nonlinear finite element computer program SMAP-W2 developed by COMTEC RESEARCH. The program has specific applications for modeling nonlinear transient seepage problems in saturated and unsaturated soils including potential seepage face. Other practical applications of SMAP-W2 include designing hydraulic structures and seepage through dams, embankments, retaining walls and slopes.

Section 2 describes derivation of finite element formulation for seepage analysis in three-dimensional body. We set the mass balance in control volume, construct global mass equilibrium equation using the Galerkin residual method and then convert volume integration into surface integration using Gauss divergence theorem. Finally we construct finite element formulation by discretizing the continuous system using shape functions and numerical integration technique.

Section 3 presents SWCC models which represent Soil Water Characteristic Curve where unsaturated volumetric water content is expressed as a function of matric suction.

Finite Element Formulation

2.1 Introduction

Section 2.2 derives finite element formulation for seepage in three-dimensional body. First, we set the flow balance in control volume as shown in Figure 2.1. Then we construct global flow equilibrium equation using the Galerkin's residual method. Then we convert volume integration into surface integration using Gauss divergence theorem. Finally we construct finite element formulation by discretizing the continuous system using shape functions and numerical integration technique.

2.2 Finite Element Formulation

For a differential cubic element volume $dv = dx \, dy \, dz$

See Figure 2.1 for flow rate in x direction.

$\dot{Q}_x$: Flow rate in x direction

$\dot{Q}_y$: Flow rate in y direction

$\dot{Q}_z$: Flow rate in z direction

Flow rate by hydraulic conduction (J. B. J. Fourier, 1822)

$$\begin{aligned}\dot{Q}_x &= -k_x (dy \cdot dz) \partial H / \partial x \\ \dot{Q}_y &= -k_y (dx \cdot dz) \partial H / \partial y \\ \dot{Q}_z &= -k_z (dx \cdot dy) \partial H / \partial z\end{aligned}\tag{2.1}$$

Derivatives of Equation 2.1

$$\begin{aligned}\partial \dot{Q}_x / \partial x &= -k_x (dy \cdot dz) \partial^2 H / \partial x^2 \\ \partial \dot{Q}_y / \partial y &= -k_y (dx \cdot dz) \partial^2 H / \partial y^2 \\ \partial \dot{Q}_z / \partial z &= -k_z (dx \cdot dy) \partial^2 H / \partial z^2\end{aligned}\tag{2.2}$$

Flow balance within control volume $dx \, dy \, dz$ during dt

$$Q_I + Q_G = Q_O + Q_R\tag{2.3}$$

Flow coming into the control volume

$$Q_I = \dot{Q}_x \cdot dt + \dot{Q}_y \cdot dt + \dot{Q}_z \cdot dt\tag{2.4}$$

Flow generated within the control volume

$$Q_G = \dot{q}_G \cdot dv \cdot dt\tag{2.5}$$

Flow coming out from the control volume

$$\begin{aligned}Q_O &= (\dot{Q}_x + (\partial \dot{Q}_x / \partial x) \cdot dx) \cdot dt \\ &+ (\dot{Q}_y + (\partial \dot{Q}_y / \partial y) \cdot dy) \cdot dt \\ &+ (\dot{Q}_z + (\partial \dot{Q}_z / \partial z) \cdot dz) \cdot dt\end{aligned}\tag{2.6}$$

Flow remained within volume resulted in total head increase

$$Q_R = \lambda \cdot dV \cdot (\partial H / \partial t) \cdot dt \quad \text{where } \lambda = m_w \gamma_w \quad (2.7)$$

Substituting Equations 2.1 and 2.2 into 2.3 with 2.4 through 2.7

$$k_x (\partial^2 H / \partial x^2) + k_y (\partial^2 H / \partial y^2) + k_z (\partial^2 H / \partial z^2) + \dot{q}_G - \lambda (\partial H / \partial t) = 0 \quad (2.8)$$

Assume total head field using shape functions

$$\begin{aligned} H &= [N_i] [H_i] \\ [N_i] &= [N_1 \ N_2 \ N_3 \ N_4 \ N_5 \ N_6 \ N_7 \ N_8] \\ [H_i]^T &= [H_1 \ H_2 \ H_3 \ H_4 \ H_5 \ H_6 \ H_7 \ H_8] \end{aligned} \quad (2.9)$$

Residual by applying Equation 2.9 into 2.8

$$R = (k_x \partial [N_i, x] / \partial x + k_y \partial [N_i, y] / \partial y + k_z \partial [N_i, z] / \partial z) [H_i] + \dot{q}_G - \lambda [N_i] [\dot{H}_i] \quad (2.10)$$

Applying Garlerkin's weighted residual method

$$\int_V [N_i]^T \cdot R \, dV = 0 \quad : \quad i = 1 \text{ to } 8 \quad (2.11)$$

Divergence theorem

$$\int_V \mathbf{A} \cdot (\partial \mathbf{B} / \partial \mathbf{x}) \cdot dV = - \int_V \mathbf{B} \cdot (\partial \mathbf{A} / \partial \mathbf{x}) \cdot dV + \int_S \mathbf{A} \cdot \mathbf{B} \cdot d\mathbf{s} \quad (2.12)$$

Divergence theorem for x component

$$\int_V [N_i]^T k_x (\partial [N_r, x] / \partial x) dV = - \int_V k_x [N_r, x]^T [N_r, x] dV + \int_S k_x [N_i]^T [N_r, x] n_x d\mathbf{s} \quad (2.13)$$

Divergence theorem for y component

$$\int_V [N_i]^T k_y (\partial [N_r, y] / \partial y) dV = - \int_V k_y [N_r, y]^T [N_r, y] dV + \int_S k_y [N_i]^T [N_r, y] n_y d\mathbf{s} \quad (2.14)$$

Divergence theorem for z component

$$\int_V [N_i]^T k_z (\partial [N_r z] / \partial z) dv = - \int_V k_z [N_r z]^T [N_r z] dv + \int_S k_z [N_i]^T [N_r z] n_z ds \quad (2.15)$$

Substituting Equations 2.10 into 2.11 with 2.13 through 2.15

$$[c] \cdot [\dot{H}_i] + [k] \cdot [H_i] = [\dot{q}_h] + [\dot{q}_B] \quad (2.16)$$

Element capacity matrix

$$[c] = \int_V \lambda [N_i]^T \cdot [N_i] dv \quad (2.17)$$

Element hydraulic conductivity matrix

$$[k] = \int_V k_x [N_r x]^T [N_r x] dv + \int_V k_y [N_r y]^T [N_r y] dv + \int_V k_z [N_r z]^T [N_r z] dv \quad (2.18)$$

Element flow generation vector

$$[\dot{q}_h] = \int_V [N_i]^T \cdot \dot{q}_G dv \quad (2.19)$$

Element specified boundary flow vector

$$\begin{aligned} [\dot{q}_B] &= \int_S [N_i]^T (k_x [N_r x] [H_i] n_x + k_y [N_r y] [H_i] n_y + k_z [N_r z] [H_i] n_z) ds \\ &= \int_S [N_i]^T (k_x (\partial H / \partial x) n_x + k_y (\partial H / \partial y) n_y + k_z (\partial H / \partial z) n_z) ds \\ &= \int_S [N_i]^T \cdot f_B \cdot ds \end{aligned} \quad (2.20)$$

Where f_B is specified flux on the element surface

Interpolation of x, y and z coordinates

$$\begin{aligned} x &= x(\xi, \eta, \zeta) = N_1(\xi, \eta, \zeta) x_1 + \dots + N_8(\xi, \eta, \zeta) x_8 \\ y &= y(\xi, \eta, \zeta) = N_1(\xi, \eta, \zeta) y_1 + \dots + N_8(\xi, \eta, \zeta) y_8 \\ z &= z(\xi, \eta, \zeta) = N_1(\xi, \eta, \zeta) z_1 + \dots + N_8(\xi, \eta, \zeta) z_8 \end{aligned} \quad (2.21)$$

Derivatives of shape functions wrt ξ, η, ζ coordinates by chain rule

$$\begin{aligned} N_{1,\xi} &= (N_{1,x}) \cdot (x, \xi) + (N_{1,y}) \cdot (y, \xi) + (N_{1,z}) \cdot (z, \xi) \\ N_{1,\eta} &= (N_{1,x}) \cdot (x, \eta) + (N_{1,y}) \cdot (y, \eta) + (N_{1,z}) \cdot (z, \eta) \\ N_{1,\zeta} &= (N_{1,x}) \cdot (x, \zeta) + (N_{1,y}) \cdot (y, \zeta) + (N_{1,z}) \cdot (z, \zeta) \end{aligned} \quad (2.22)$$

$$\begin{bmatrix} N_{1,\xi} \\ N_{1,\eta} \\ N_{1,\zeta} \end{bmatrix} = \begin{bmatrix} x, \xi & y, \xi & z, \xi \\ x, \eta & y, \eta & z, \eta \\ x, \zeta & y, \zeta & z, \zeta \end{bmatrix} \begin{bmatrix} N_{1,x} \\ N_{1,y} \\ N_{1,z} \end{bmatrix} \quad (2.23)$$

Derivatives of shape functions wrt x, y and z coordinates

$$\begin{bmatrix} N_{1,x} \\ N_{1,y} \\ N_{1,z} \end{bmatrix} = \begin{bmatrix} x, \xi & y, \xi & z, \xi \\ x, \eta & y, \eta & z, \eta \\ x, \zeta & y, \zeta & z, \zeta \end{bmatrix}^{-1} \begin{bmatrix} N_{1,\xi} \\ N_{1,\eta} \\ N_{1,\zeta} \end{bmatrix} \quad (2.24)$$

$N_{1,x}, N_{1,y}$ and $N_{1,z}$ can be evaluated at integration point (ξ, η, ζ)

$$\begin{bmatrix} L_x(1) \\ L_y(1) \\ L_z(1) \end{bmatrix} = \begin{bmatrix} J \end{bmatrix}^{-1} \begin{bmatrix} P(1,1) \\ P(2,1) \\ P(3,1) \end{bmatrix} \quad (2.25)$$

Interpolation of total head field

$$H = H(\xi, \eta, \zeta) = N_1(\xi, \eta, \zeta) \cdot H_1 + \dots + N_8(\xi, \eta, \zeta) \cdot H_8 \quad (2.26)$$

Derivatives of total head field wrt x, y and z coordinates

$$\begin{aligned} H_{,x} &= N_{1,x} \cdot H_1 + \dots + N_{8,x} \cdot H_8 \\ H_{,y} &= N_{1,y} \cdot H_1 + \dots + N_{8,y} \cdot H_8 \\ H_{,z} &= N_{1,z} \cdot H_1 + \dots + N_{8,z} \cdot H_8 \end{aligned} \quad (2.27)$$

Numerical integration in ξ, η and ζ coordinates

$$\begin{aligned} \int_V F(x, y, z) dx dy dz &= \int_V F(\xi, \eta, \zeta) \cdot |J(\xi, \eta, \zeta)| d\xi d\eta d\zeta \\ &= F_1(\xi_1, \eta_1, \zeta_1) \cdot |J_1| \cdot W_1 + \dots + F_8(\xi_8, \eta_8, \zeta_8) \cdot |J_8| \cdot W_8 \end{aligned} \quad (2.28)$$

Numerical integration of capacity matrix using 2x2x2 Gauss points

$$[c] = \sum_{i=1}^2 \sum_{j=1}^2 \sum_{k=1}^2 \lambda [N(\xi, \eta, \zeta)]^T \cdot [N(\xi, \eta, \zeta)] \cdot |J(\xi, \eta, \zeta)| \cdot L_i L_j L_k \quad (2.29)$$

Numerical integration of conductivity matrix using 2x2x2 points

$$\begin{aligned} [k] &= \sum_{i=1}^2 \sum_{j=1}^2 \sum_{k=1}^2 (k_x [N, x(\xi, \eta, \zeta)]^T \cdot [N, x(\xi, \eta, \zeta)] \\ &\quad + k_y [N, y(\xi, \eta, \zeta)]^T \cdot [N, y(\xi, \eta, \zeta)] \\ &\quad + k_z [N, z(\xi, \eta, \zeta)]^T \cdot [N, z(\xi, \eta, \zeta)]) \cdot |J(\xi, \eta, \zeta)| \cdot L_i L_j L_k \end{aligned} \quad (2.30)$$

Numerical integration of flow generation vector using 2x2x2 points

$$[\dot{q}_h] = \sum_{i=1}^2 \sum_{j=1}^2 \sum_{k=1}^2 \dot{q}_G \cdot [N(\xi, \eta, \zeta)]^T \cdot |J(\xi, \eta, \zeta)| \cdot L_i L_j L_k \quad (2.31)$$

Numerical integration of specified flow vector using 2x2 points

$$\begin{aligned} [\dot{q}_B] &= \sum_{i=1}^2 \sum_{j=1}^2 f_B \cdot [N_s(r, t)]^T \cdot |J(r, t)| \cdot L_i L_j \\ [N_s] &= [N_{s1} \ N_{s2} \ N_{s3} \ N_{s4}] \end{aligned} \quad (2.32)$$

r, t : Natural coordinate on element surface
where boundary flux is specified

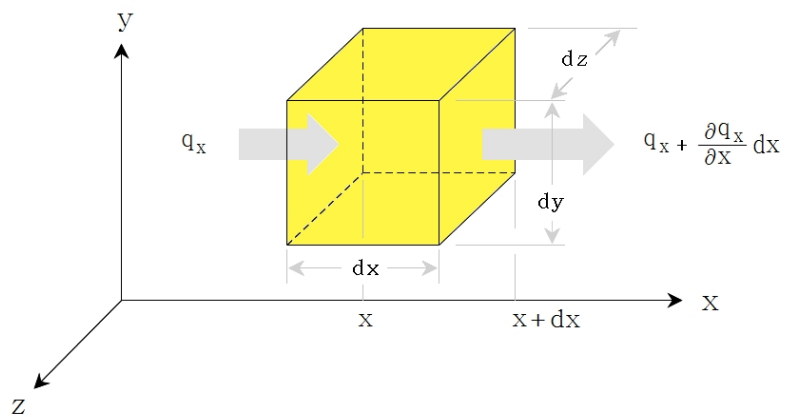


Figure 2.1 Flow in differential control volume

SWCC Models

3.1 SWCC Models

3.1.1 Introduction

SWCC represents Soil Water Characteristic Curve where unsaturated volumetric water content is expressed as a function of matric suction. Specific moisture capacity, which is required for the transient seepage analysis, is obtained by differentiating the volumetric water content with respect to matric suction.

Most empirical equations for the unsaturated hydraulic conductivity are obtained by relating to the soil water characteristic curves.

SMAP supports following four SWCC models:

- van Genuchten (1980)
- Fredlund and Xing (1994)
- Brutsaert (1966) and Gardner (1958)
- Simple Gardner (1958)

SWCC model parameters for the typical soils (sand, silt and clay) are presented in the following tables.

- Table 3.1 for van Genuchten (1980) and Leong & Rahardjo (1997)
- Table 3.2 for Fredlund and Xing (1994)
- Table 3.3 for Brutsaert (1966) and Gardner (1958)
- Table 3.4 for van Genuchten (1980) from Carsel & Parrish (1988)

3.1.2 van Genuchten (1980)

Volumetric water content for unsaturated soils is given by

$$\theta = \theta_r + \frac{(\theta_s - \theta_r)}{[1 + (a \psi)^n]^m} \quad (3.1)$$

where

ψ	Suction as water pressure or pressure head
θ	Volumetric water content
θ_s	Saturated volumetric water content
θ_r	Residual volumetric water content
a, n, m	Material parameters from fitting data
	Default $m = 1 - 1/n$

Differentiation of water content with respect to suction

$$\frac{d\theta}{d\psi} = - \frac{(\theta_s - \theta_r) m n a (a \psi)^{n-1}}{[1 + (a \psi)^n]^{m+1}} \quad (3.2)$$

Specific moisture capacity (m_w) for unsaturated soils is given by

$$\begin{aligned} m_w &= (d\theta/d\psi) (1/\gamma_w) && \text{when } \psi \text{ is water pressure head} \\ m_w &= (d\theta/d\psi) && \text{when } \psi \text{ is water pressure} \end{aligned} \quad (3.3)$$

where γ_w is unit weight of water

Permeability for unsaturated soils is given by

$$k = k_s \frac{(1 - (a \psi)^{n-1} [1 + (a \psi)^n]^{-m})^2}{[1 + (a \psi)^n]^{m/2}} \quad (3.4)$$

where k_s is saturated permeability.

3.1.3 Fredlund and Xing (1994)

Volumetric water content for unsaturated soils is given by

$$\theta = C(\psi) \frac{\theta_s}{\left(\ln \left[e + \left(\frac{\psi}{a} \right)^n \right] \right)^m} \quad (3.5)$$

where

ψ	Suction as water pressure or pressure head
ψ_r	Suction at residual water content
θ	Volumetric water content
θ_s	Saturated volumetric water content
a, n, m	Material parameters from fitting data

$$C(\psi) = 1 - \frac{\ln(1 + \psi/\psi_r)}{\ln(1 + 10^6/\psi_r)} \quad (3.6)$$

Differentiation of water content with respect to suction

$$\frac{d\theta}{d\psi} = C' \frac{\theta_s}{\left(\ln \left[e + \left(\frac{\psi}{a} \right)^n \right] \right)^m} - C \frac{\theta_s}{\left(\ln \left[e + \left(\frac{\psi}{a} \right)^n \right] \right)^{m+1}} \frac{m n (\psi/a)^{n-1}}{a \left[e + \left(\frac{\psi}{a} \right)^n \right]} \quad (3.7)$$

$$C'(\psi) = - \frac{1}{(\psi_r + \psi) \ln[1 + 10^6/\psi_r]} \quad (3.8)$$

Unsaturated relative permeability is given by

$$k_r = \frac{\int_{\ln(\psi)}^b e^{-y} [\theta(e^y) - \theta(\psi)] \theta'(e^y) dy}{\int_{\ln(\psi_{aev})}^b e^{-y} [\theta(e^y) - \theta_s] \theta'(e^y) dy} \quad (3.9)$$

ψ_{aev} = air-entry value of suction

$b = \ln(1,000,000)$

Leong and Rahardjo (1997) proposed simplified form of unsaturated hydraulic conductivity by assuming $C(\psi) = 1$.

$$k = \frac{k_s}{\left(\ln \left[e + \left(\frac{\psi}{a} \right)^n \right] \right)^m} \quad (3.10)$$

where k_s is permeability at saturation.

3.1.4 Brutsaert (1966) and Gardner (1958)

Unsaturated volumetric water content is given by Brutsaert (1966)

$$\theta = \theta_r + (\theta_s - \theta_r) \frac{A_1}{A_1 + (\psi)^{n_1}} \quad (3.11)$$

where

ψ	Suction as water pressure head
θ	Volumetric water content
θ_s	Saturated volumetric water content
θ_r	Residual volumetric water content
A_1, n_1	Material parameters from fitting data

Differentiation of water content with respect to suction

$$\frac{d\theta}{d\psi} = -(\theta_s - \theta_r) \frac{A_1 n_1 (\psi)^{n_1-1}}{\left[A_1 + (\psi)^{n_1} \right]^2} \quad (3.12)$$

Unsaturated permeability is given by Gardner (1958)

$$k = k_s \frac{A_2}{A_2 + (\psi)^{n_2}} \quad (3.13)$$

where

k_s	Permeability at saturation.
A_2, n_2	Material parameters from fitting data

3.1.5 Simple Gardner (1958)

This Simple Gardner equation is used for Verification Problem 2.

Volumetric water content for unsaturated soils is given by

$$\theta = \theta_r + (\theta_s - \theta_r) e^{-\alpha \psi} \quad \psi > 0 \quad (3.14)$$

where

ψ	Suction as water pressure head
θ	Volumetric water content
θ_s	Saturated volumetric water content
θ_r	Residual volumetric water content
α	Material parameters

Differentiation of water content with respect to suction head

$$\frac{d\theta}{d\psi} = -(\theta_s - \theta_r) \alpha e^{-\alpha \psi} \quad (3.15)$$

Specific moisture capacity (m_w) for unsaturated soils is given by

$$m_w = (d\theta / d\psi) (1 / \gamma_w) \quad (3.16)$$

where γ_w is unit weight of water

Permeability for unsaturated soils is given by

$$k = k_s e^{-\alpha \psi} \quad (3.17)$$

where k_s is saturated permeability.

And α is the same as that used in Equation 3.14 for water content.

Table 3.1 SMCC model parameters from Gui et al, 2011 (Ref 6)

Soil Type	k_s (m/d)	θ_s	van Genuchten (1980) $m = 1-1/n$		Leong & Rahardjo (1997)		
			a (kPa ⁻¹)	n	a (kPa)	n	m
CL	$1.29 \cdot 10^{-10}$	0.45	0.016	1.180	70.54	1.211	0.4620
ML	$2.45 \cdot 10^{-8}$	0.37	0.0172	1.209	104.0	1.016	0.5904
SM	$7 \cdot 10^{-6}$	0.35	0.0682	1.411	21.42	1.734	0.8811

CL: Lean Clay ML : Silt SM : Silty Sand

k_s : Saturated permeability

θ_s : Saturated volumetric water content

Table 3.2 SMCC parameters from Fredlund et al, 1994 (Ref 3)

Soil Type	$k_s \times 10^6$ (m/s)	θ_s	a	n	m	P_r (kPa)
Touchet silt loam	-----	0.43	8.34	9.90	0.44	30
Columbia sandy loam	-----	0.458	6.01	11.86	0.36	30
Yolo light clay	0.123	0.375	2.70	2.05	0.36	100
Guelph loam (Drying)	3.917	0.52	5.61	2.24	0.40	300
Guelph loam (Wetting)	0.7	0.43	3.12	4.86	0.23	100
Superstition sand	18.3	-----	2.77	11.20	0.45	300

Table 3.3 SMCC model parameters from Mavroulidou, 1999 (Ref 8)

Soil Type	k_s (m/d)	θ_s	θ_r	Brutsaert (1966)		Gardner (1958)	
				A_1 (m)	n_1	A_2 (m)	n_2
Silty Sand	8.4	0.3	0.0	0.0634	2.9	0.0003	5.0
Touchet Silt	0.432	0.469	0.127	1.505	2.849	0.3302	5.192
Jurong Clay	$6.74 \cdot 10^{-4}$	0.394	0.320	1.3705	0.5876	0.0011	2.0

Silty Sand : Vauclin et al (1976)

Touchet Silt : Brooks and Corey (1966)

Jurong Clay : Leong and Rahardjo (1995)

θ_r : Residual volumetric water content

Table 3.4 van Genuchten parameters from Carsel & Parrish, 1988 [Ref 2]

Soil Type	k_s (m/s)	θ_s	θ_r	a (kPa^{-1})	n
Sand	8.25E-05	0.43	0.045	1.477	2.68
Loamy Sand	4.05E-05	0.41	0.057	1.264	2.28
Sandy Loam	1.22E-05	0.41	0.065	0.7645	1.89
Sandy Clay Loam	3.64E-06	0.39	0.100	0.6013	1.48
Loam	2.89E-06	0.43	0.078	0.3670	1.56
Silt Loam	1.25E-06	0.45	0.067	0.2039	1.41
Clay Loam	7.22E-07	0.41	0.095	0.1937	1.31
Silt	6.94E-07	0.46	0.034	0.1631	1.37
Clay	5.55E-07	0.38	0.068	0.08155	1.09
Sandy Clay	3.33E-07	0.38	0.100	0.2753	1.23
Silty Clay Loam	1.94E-07	0.43	0.089	0.1019	1.23
Silty Clay	5.55E-08	0.36	0.070	0.051	1.09

 Note: $m = 1 - 1/n$

References

1. Brutsaert, W., "Probability Laws for Pore-Size Distributions," Soil Science, 1966; 101, pp. 85-92.
2. Carsel, F.C., and Parrish, R.S., "Developing Joint Probability of Soil Water Retention Characteristics," Water Resources Research, 1988; Volume 24, Issue 5, pp. 755-769.
<https://doi.org/10.1029/WR024i005p00755>
3. Fredlund, DG, and Xing, A., "Equations for the Soil-Water Characteristic Curve," Canadian Geotechnical Journal, 1994; 31(4): 521-32. <https://doi.org/10.1139/t94-061>
4. Gardner, W.R., "Some Steady-State Solutions of the Unsaturated Moisture Flow Equation with Application to Evaporation from a Water Table," Soil Science, 1958; 85(4), pp. 228-232.
5. Genuchten, Van, "A Closed-Form Equation for Predicting the Hydraulic Conductivity of Unsaturated Soils," Soil Science Society of America Journal, 1980; 44(5): 892-8.
6. Gui, M.W., Wu, C.H. and Lu, C., "Comparison of Two Water Storage Functions of Soil on Pore-Water Pressure of Earth-Filled Dam under Changing Environment," Conference: 28th ISARC, June 2011; pp. 538-543. DOI:10.22260/ISARC2011/0097
7. Leong, E.C., and Rahardjo, H., "Review of Soil-Water Characteristic Curve Equations," A.S.C.E. Journal of Geotechnical and Environmental Engineering, 1997; 123(12), pp. 1106-1117.
8. Mavroulidou, M., "Numerical Modeling of Seepage in the Presence of Phreatic Surfaces," Doctoral Thesis, University of Surrey, United Kingdom, 1999. 363 Pages.

9. Tracy, F.T., Brandon, L. and Corcoran, M.K., "Transient Seepage Analysis in Levee Engineering Practice," US Army Corps of Engineers, July 2016; ERDC TR-16-8.
10. France, P.W., Parekh, C.J., Peters, J.C. and Taylor, C., "Numerical Analysis of Free Surface Seepage Problems," ASCE Journal of Irrigation & Drainage Division, 1971, Vol. 97, No 1, pp. 165-179.
<https://doi.org/10.1061/JRCEA4.0000773>
11. Bathe, K.J. and Khoshgoftaar, M.R., "Finite Element Free Surface Seepage Analysis Without Mesh Iteration," Internal Journal for Numerical and Analytical Methods in Geomechanics, 1979, 3(1), pp. 13-22. <https://doi.org/10.1002/nag.1610030103>
12. Taylor, G.S. and Luthin, J.N., "Computer Methods for Transient Analysis of Water-Table Aquifers," Water Resources Center, Ohio State University, Jan. 1971.
13. Hall, H.P., "An Investigation of Steady flow Toward a Gravity Well," La Houille Blanche, 10, 1955.
14. Bowles, J.E., "Physical and Geotechnical Properties of Soils," 1984, 2nd ed., McGraw-Hill, New York.
15. Chapuis, R.P., Chenaf, D., Bussiere, B., Aubertin, M. and Crespo, R., "A Users Approach to Assess Numerical Codes for Saturated and Unsaturated Seepage Conditions," Canadian Geotechnical Journal, Vol. 38, No. 5, Oct. 2001.
<https://doi.org/10.1139/t01-055>
16. Rockscience, "Slide 2: Slide Groundwater Verification Manual, Rockscience Inc., 2022.
<https://static.rocscience.cloud/assets/verification-and-theory/Slide2/Slide-Groundwater-Verification-Manual-2022.pdf>

17. Ho, P.G., "The Prediction of Hydraulic Conductivity from Soil Moisture Suction Relationship," B.Sc Thesis, 1979, University of Saskatchewan, Saskatoon, SK, Canada.
18. Fredlund, D.G., Xing, A. and Hung, S., "Predicting the Permeability Function for Unsaturated Soils Using the Soil-Water Characteristic Curve," 1994, Department of Civil Engineering, University of Saskatchewan, Saskatoon, SK, Canada.