Table of Contents

Part 1 – Getting Started 1

Introducing *Examine*^{2D} 3

Layout of *Examine*^{2D} Manual 4
Acknowledgements 5
Disclaimer 5
Hardware Support 6
- Printer Support 6
- Super VGA Support 6

Hard Disk Installation Procedure 7
- Protected Mode Version 7

Installing *Examine*^{2D} to run from Windows 8
Running *Examine*^{2D} 9
- Command Line Options 9

Examine^{2D} Utilities 10
Screen Layout 10
- Upper Status Box 10
- Menu Title Box 11
- Menu Box 11
- Plot Display Box 11
- Prompt Line 11

Using the Menu System 12
- Moving the Cursor 12
- Popup Menus 13
- Special Keys 13
  - Hot Keys 13

Main Menu 14
- MODEL 14
- COMPUTE 14
- INTERPRET 14
- Read File 15
- Write File 15
- Reset All 16
- Printer Configuration 16
- EXIT 18

Hints about the Manual 18

Part 2 – Reference Manual 19

Creating a MODEL 21

Note on Units 21
Coordinate Entry 22
Stress Convention 22
SETUP 23
Analysis Options 23
Continuum Type 24
Element Type 24
Matrix Solution 25
Plane Strain 25
Consider Out-of-Plane Stress? 25
Symmetry 26
Set Limits 27
BOUNDARIES 28
ADD EXCAVATION 28
Using the Snap and Grid Options 29
Closing Excavation Boundaries 30
Correcting Mistakes 30
Drawing Arc Segments and Circles 30
Entering Geometry With Symmetry On 31
DISCRETIZATION 32
DEFAULT 32
Chge Deflt 33
SUM LENGTH 33
One Line 34
Reset 34
BOUNDARY CONDITIONS 35
DEFAULT 36
Chge Deflt 36
One Line 37
Reset 37
Show BndCd 38
Add (Free) Surface 38
Half Space 38
Discretized 39
A Note on Free Surfaces and Symmetry 39
Add Joint 40
Normal and Shear Joint Stiffness 41
Joint Aperture 41
Allow Slip 41
Slip Criterion 42
Number of Load Steps 42
Initial Joint Deformation 42
Editing Joint Properties 43

Restrictions on Use of Joints....43
Editing Boundaries....44
Del Excv....44
Move Excv....44
Copy Excv....45
Strch Excv....45
Scale Excv....46
Rot Excv....46
Split Line....46
Del Surf....47
Strch Surf....47
Del Joint....47
Strch Joint....47
Geometry Import / Export via DXF Files....48
DXF Bndry In....48
DXF Bndry Out....48
STRESS GRID....49
ADD GRID....49
ADD LINE....50
Del Grid....51
Del Lines....51
Points On....51
Points Off....51
A Note on Stress Grids and Symmetry....51
FIELD STRESS....52
CONSTANT....52
CONSTANT + Plane Strain....52
CONSTANT + Complete Plane Strain....53
GRAVITY....54
GRAVITY + Plane Strain....54
GRAVITY + Complete Plane Strain....55
USER-DEFINED Stress Field....55
Move Blk....56
Delete Blk....57
Failure Criteria....57
Hoek-Brown Failure Criterion....58
Modified Hoek-Brown Failure Criterion....58
Mohr-Coulomb Failure Criterion....59
Further Reading – Failure Criteria....59
Elastic Properties....61
Isotropic Continuum....61
Transversely Isotropic Continuum....62
Job Title....63
Toolbox....64
Autoscale....64
Zoom Win....64
Zoom Out....64
Zoom In....65

Pan 65
Snap Set 65
Grid Set 65
Input Data 66
F-Key Help 66

COMPUTE **67**

Running COMPUTE 67
Output File 68

Interpreting the Analysis **69**

SIGMA 1 70
SIGMA 2 71
SIGMA 3 71
Strength Factor 72
Displacements 73
NETDISP Utility 73
USER DATA 74
USER DATA Configuration File – e2.cfg 74
User Defined Functions 75
Secondary INTERPRET Options 77
Autoscale 77
Zoom Win & Zoom Out 77
Input Data 77
Alter Range 78
Plotting Trajectories 79
Stress Trajectories 79
Displacement Trajectories 80
Failure Trajectories 81
Overlay Traj 81
Traject Plot 81
Input Data 82
Scale Traj 82
DXF Out 82
Failure Criterion 83
Ubiquitous Joints 83
Barton-Bandis Ubiquitous Joints 84
Mohr-Coulomb Ubiquitous Joints 84
Ubiquitous Joints OFF 85
Query Data 86
Point Value 86
Line Values 87
Points File 88
Add Series 89
Graph Series 90
Graph Setup 91
Reset Series 92
Redraw 92

Display Tools 93
Display Configuration 93
Elements ON/OFF 93
Grids ON/OFF 93
Stress Block ON/OFF 94
Axes Labels 94
Contour Type 94
Color 94
Stipple 94
Contour 94
PhotoE 94
Symmetry ON/OFF 95
Display Joint Slip 95
Near Joint Contours 95
Measure 95
Label Contr 96
Delete Labs 96
Alter Color 97
Insert Text 97
Delete Text 98
Job Title 98
Move StrBlk 98
DXF File Out 99
Full Screen 100
Redraw 100

NETDISP Utility 101

Bibliography 103

Part 3 – Tutorial Manual 105

Introduction 107

Hints about the Tutorial Manual 107

A Simple Tutorial 109

MODEL 109
Set Limits 110
Entering Geometry 110
Toolbox 111
Discretization 111
Boundary Conditions 111
Stress Grid 112
Field Stress 112
Failure Criterion 113
Elastic Properties 113

Job Title....113
COMPUTE....114
INTERPRET....115

Fault Tutorial - Step 1 117

MODEL....117
Set Limits....117
Entering Geometry....118
Discretization....118
Boundary Conditions....119
Stress Grid....119
Field Stress....120
Elastic Properties....120
Job Title....120
COMPUTE....121
INTERPRET....122

Fault Tutorial - Step 2 123

MODEL....123
Add Joint....124
Re-Discretization....126
Job Title....126
COMPUTE....128
INTERPRET....128
References for Tutorial 2....129

Near Surface Tutorial - Step 1 131

MODEL....131
Set Limits....132
Entering Geometry....132
Defining the Ground Surface....132
Discretization....133
Boundary Conditions....133
Stress Grid....133
Autoscale....134
Field Stress....134
Failure Criterion....134
Elastic Properties....134
Job Title....135
COMPUTE....136
INTERPRET....136

Near Surface Tutorial - Step 2 140

MODEL....141
Redefining the Ground Surface....141
Deleting the Existing Half Space....141
Entering a Discretized Ground Surface....142
Discretization....142

Boundary Conditions 142
Surface Boundary Condition 143
Job Title 143
COMPUTE 144
INTERPRET 145

Part 1 – Getting Started

INCLUDES:

INTRODUCTION, INSTALLATION, PRINTER SUPPORT

RUNNING EXAMINE^{2D}, USING THE MENU SYSTEM

Introducing *Examine^{2D}*

EXAMINE^{2D} is a 2-dimensional plane strain indirect boundary element package for analysis of underground excavations in rock.

Before launching into an analysis with EXAMINE^{2D}, it is important to stop and consider the developmental philosophy of the program, the assumptions inherent in the analysis and the resultant limitations.

EXAMINE^{2D} was developed by the Rock Engineering Group at the University of Toronto. It is designed to be a quick and simple-to-use **parametric analysis** tool for investigating the influence of **geometry** and **in-situ stress** variability on the stress changes in rock due to excavations. The **induced stresses** in the plane of the analysis can be viewed by means of graphical screen and printer plots showing contoured stress patterns in the region surrounding the excavations. As a tool for interpreting the amount of deviatoric overstress (principal stress difference) around openings, a Strength Factor option is included which gives quantitative measures of (strength)/(induced stress) according to user defined failure criteria for the rock mass. Displacement vectors may also be plotted in this version of the program. These are meant only to illustrate general deformation trends and not to show actual magnitudes of displacements.

Some important limitations of the program which should be considered when interpreting EXAMINE^{2D} output are described below.

The assumption of **plane strain** means that the modelled excavation is of infinite length normal to the plane section of the analysis. In practice, as the out-of-plane excavation length becomes less than five times the largest cross-sectional dimension, the stress changes calculated by EXAMINE^{2D} begin to show some exaggeration since the real stress flow around the 'ends' of the excavation is not taken into account. All of the stress is 'forced' to flow around the excavation parallel to the analysis plane. This exaggeration becomes more pronounced as the out-of-plane length approaches the same magnitude as the in-plane dimensions. As long as this effect is kept in mind, the analysis may still yield useful insight into behaviourial trends in these cases.

The **elastic** boundary element analysis used in EXAMINE^{2D} dictates that the material being modelled must be assumed to be:

- ⇒ **homogenous,**
- ⇒ **isotropic** or **transversely isotropic,** and
- ⇒ **linearly elastic**.

Obviously, most of the rock masses which will be modelled possess none of these properties. The degree to which the actual rock mass being modelled deviates from these assumed properties should be kept in mind when interpreting EXAMINE2D output. Nevertheless, the induced stresses calculated and displayed by EXAMINE2D can usually prove useful, for example, when optimizing excavation geometry and/or sequencing to avoid overstress and undesirable destressing.

The displacements shown by EXAMINE2D are meant to qualitatively illustrate regional deformation trends only. The actual values of the displacements calculated by COMPUTE *include only the* ***elastic displacements*** due to the excavation. This, in reality, may constitute a very small component of the actual measured displacements in the field. In weak broken rock, the actual magnitude of displacements may be several orders of magnitude greater than the calculated elastic values. In addition, the calculated displacements depend directly on the value of the Deformation (Young's) Modulus for the rock mass, a value difficult to estimate.

The practice of performing multiple analysis runs using a range of stress and material properties to study the effect of each parameter is a prudent one in all cases.

In short, EXAMINE2D is a powerful but, nevertheless, limited tool. Like all numerical models, it should be used to enhance and supplement, but never to replace, common sense and good engineering judgement.

Layout of *Examine2D* Manual

The EXAMINE2D manual is laid out in **THREE** sections:

Part 1 – Getting Started (this section) which includes introduction, installation notes, etc.

Part 2 – Reference Manual which includes comprehensive reference information on the **Model**, **Compute** and **Interpret** modules of EXAMINE2D. The chapters are laid out according to the order of appearance in the menu tree structure.

Part 3 – Tutorial Manual which consists of three tutorials illustrating various features of EXAMINE2D. The tutorials use files which are supplied with EXAMINE2D, and which you will find in the \TUTORIAL subdirectory.

Acknowledgements

The computer program EXAMINE^{2D} was developed by the Rock Engineering Group of the Department of Civil Engineering, University of Toronto and was funded jointly by the Mining Technology Division, Centre de Technologie (Noranda), the University Research Incentive Fund (Ontario Government), and the Rock Engineering Group. The participation of the following people is acknowledged:

Project Leader	John Curran
Overall conceptual design	John Curran and Brent Corkum
Programming of MODEL and INTERPRET	Brent Corkum
with revisions by	Mark Diederichs, James McLeod
Programming of COMPUTE	Joe Carvalho and Sinnathamby Sathiamoorty
with revisions by	Sanjiv Shah, Brent Corkum, Mark Diederichs and Kathir Mailvaganam
Verification	Dr. Lei Xiao-yan
User's Guide	Brent Corkum
with revisions by	Mark Diederichs, Joe Carvalho, Igor Pashutinski

The interface for EXAMINE^{2D} is based on the FEINT user interface package (ver 3.0) developed by Brent Corkum, Rock Engineering Group, University of Toronto with funding from the Mining Research Directorate.

Special thanks to Dr. Evert Hoek for his interest and enthusiastic support of this project. The applicability of EXAMINE^{2D} to practical mining and civil engineering problems has been greatly enhanced by his involvement.

Disclaimer

The authors disclaim any responsibility for the correctness of the data generated by the EXAMINE^{2D} package, or for the consequences resulting from the use thereof. Any use or misuse of this package is the sole responsibility of the user.

Hardware Support

EXAMINE^{2D} supports the following hardware:

- IBM PC/XT/286/386/486/Pentium or compatible
- Graphics Display Adapters:

Video Graphics Array (VGA)	640x480
Enhanced Graphics Adapter (EGA)	640x350
Colour Graphics Adapter (CGA)	640x200
Multi-Color Graphics Array (MCGA)	640x480
Hercules Monochrome Display Adapter	720x348
ATT 400 Series, Compaq Plasma	640x400
VESA Super VGA	800x600

- Microsoft or Logitech Mouse and compatibles (not required)
- 80x87 math coprocessor (not required)
- 386/486 (or later) protected mode using extended memory

Suggested minimum configuration:

80386 / VGA / 80387 / Mouse

Printer Support

EXAMINE^{2D} currently supports the following printers:

- HP Laserjet, Laserjet III, Laserjet IV
- HP Paintjet, Paintjet XL, Paintjet XL300
- HP Deskjet, Deskjet 1200C
- Epson 9 and 24 Pin

See the section on Printer Configuration later in Part 1 – Getting Started, for information on the various print options available.

In addition, EXAMINE^{2D} contains DXF file creation features for use with **AutoCAD**™. All of the **INTERPRET** graphical screen output can be saved to a DXF file for printing using **AutoCAD**'s comprehensive set of printer drivers. *See Part 2 – Reference Manual for more information on DXF file output.*

Super VGA Support

The VESA Super VGA mode 800x600, 16 colors is supported. To run the program in this mode, you may use the /1 command line parameter (i.e., *examine* /1), or set the environment variable SVGA to a value of 1 by placing the following line in your autoexec.bat file:

```
SET SVGA=1
```

If the program hangs in this mode, you might require a VESA driver for your display adapter to be loaded first.

Hard Disk Installation Procedure

EXAMINE^{2D} is distributed with a simple installation program. To install EXAMINE^{2D}, insert the EXAMINE^{2D} disk into the A: or B: drive. Switch to that drive and type `INSTALL`.

Example Installation Process (installed from the A: disk drive)

```
C:\> a:
A:\> install
```

After responding to the installation prompts (naming your installation directory, protected mode installation etc.), various files and subdirectories will be contained in your EXAMINE^{2D} directory, including:

examine.exe	EXAMINE^{2D} program, performs all pre- and post- processing
compute.exe	EXAMINE^{2D} boundary element analysis engine
exam2D.uid	User identification file, contains registration information
exam2d.ico	EXAMINE^{2D} icon for use in Microsoft Windows©
example.exa	An example data file
TUTORIAL	A directory containing more EXAMINE^{2D} example files – see Part 3 (Tutorial Manual)
readme.bat	A DOS batch file for displaying readme.exm
readme.exm	Document file containing up to date information on your current version of EXAMINE^{2D}. Please read it.

Protected Mode Version

During installation, you are given a choice of installing the protected mode version of the program, which allows use of up to a maximum of 16 MB of extended memory on a 386/486 or later PC. You must have at least 2 MB of extended memory to install the protected mode version of EXAMINE^{2D}.

Installing *Examine^{2D}* to run from Windows

Once you have installed EXAMINE^{2D} on your computer, and if you have the Microsoft Windows program, you may install EXAMINE^{2D} so that you can run it from Windows. This is a useful option if you will be making a lot of screen captures of the program output for reports or other documents.

To add EXAMINE^{2D} to your Windows library, follow these steps:

- Enter the Windows program.
- Enter the Program Group that you want to add EXAMINE^{2D} to.
- Select the **File** option from the Windows Program Manager.
- Indicate that you will be installing a **New** file.
- The New Program Object window will now appear, in which you should select the **Program Item**.
- A Program Item Properties screen will appear next, in which you will enter:

Description	EXAMINE2D
Command line	The location of the EXAMINE^{2D} program, eg. D:\EXAM2D\examine.exe. It is advisable to **Browse** for the program name, to ensure that the path is correct.
Working directory	You can leave this entry blank, or if you will always be working with EXAMINE^{2D} files in a particular directory, enter the appropriate directory name.
Change Icon	You will receive a message that "No icons exist". Use **Browse** to go to the EXAM2D directory, and select exam2d.ico.

The advantage of having EXAMINE^{2D} in the Windows library is that screen captures can be taken directly from the program to the clipboard, and from there directly into your Windows based drawing or word processing package. To initiate the capture of an EXAMINE^{2D} screen, press the **Alt** and **Print Screen** keys simultaneously. Next press the **Alt** and **Tab** keys together, to send the screen capture to the clipboard. The screen can then be imported into your word processing package directly from the clipboard.

Running *Examine^{2D}*

To run the program, type *examine* at the DOS prompt. You should see the start up screen shown in Figure 1.1.

If you wish to run EXAMINE^{2D} from a directory other than the installation directory, you must place the EXAMINE^{2D} directory in your path (eg):

PATH=C:\DOS;C:\EXAM2D;

Command Line Options

EXAMINE^{2D} will automatically detect the graphics card you are using with your computer and use the mode with the highest resolution on your video card. However, a variety of command line options are available if the user wants to force the program to operate in a specific graphics mode. The syntax and command line options for EXAMINE^{2D} are:

Syntax: EXAMINE [options] [filename]

Options:

Run EXAMINE^{2D} with the /P command line option for black and white screen captures.

/C	CGA mode (640 x 200) on EGA/VGA display adapters
/E	EGA mode (640 x 350) colour on VGA display adapter
/H	HERCULES (728 x 348) monochrome adapter
/M	MCGA mode (640 x 480) two colour
/P	Monochrome emulation for all display adapters
/I	Reverse Monochrome emulation for all display adapters
/T	ATT card (640 x 400) high resolution monochrome (may not be available)
/1	Super VGA (800 x 600) colour display
/V	VGA mode (640 x 480) colour display

The /P option is particularly useful when you want a black and white screen capture or wish to work in a monochrome environment on a colour or grayscale display. The /I option should be used when you want to work in reverse black and white mode (for Windows screen captures).

You may also choose to include a filename at the DOS prompt when running EXAMINE^{2D}. This filename *may include a path* and must follow a *space* after *examine*/[options]. If no filename is given then use **Read File** in the EXAMINE^{2D} main menu to load a file.

To display information about the syntax and command line options, type *examine* /? at the DOS prompt.

Examine^{2D} Utilities

A utility program is supplied with EXAMINE^{2D} :

- NETDISP — a utility for calculating net displacements with respect to a datum state such as a previous excavation stage, or after input of a free surface.

See the end of Part 2 – Reference Manual for more information.

Screen Layout

When the EXAMINE^{2D} program is first started, the screen shown in Figure 1.1 will appear. The screen is comprised of five independent sections, as described below.

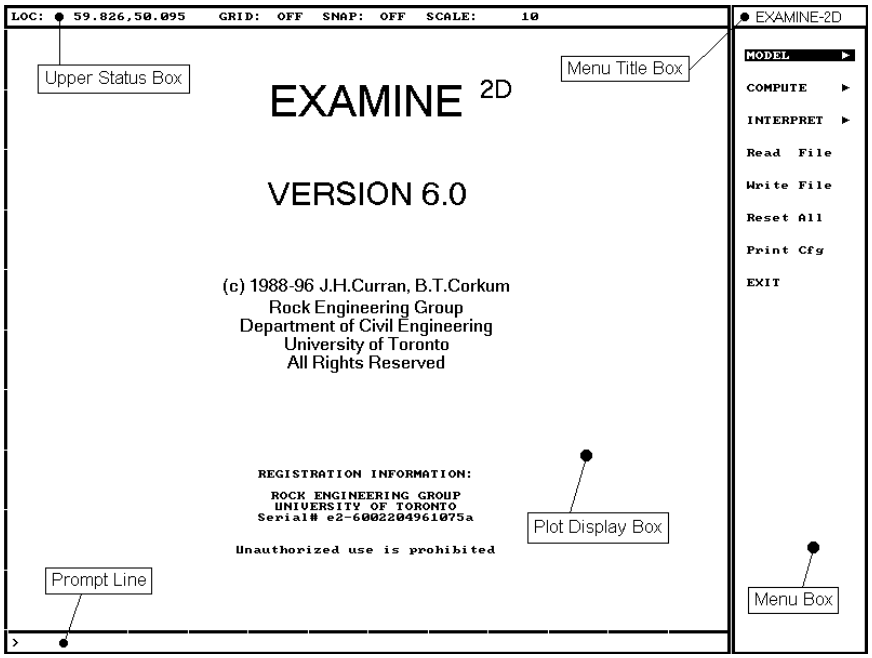


Figure 1.1: EXAMINE^{2D} start up screen.

Upper Status Box

The Upper Status Box contains several pieces of information. To the left, the LOCATOR prompt indicates the (x,y) position of the cursor on the screen and becomes active whenever the cursor is used to enter information. The GRID and SNAP flags indicate the status of these features, which can be toggled ON and OFF using the F7 and F9 keys respectively. SCALE indicates the scale of the drawings when graphics screens are displayed. Further information about these features is provided in the **Toolbox** section of the **MODEL** chapter (see Part 2 – Reference Manual).

Menu Title Box

The Menu Title Box contains the TITLE of the current menu in which you are working – in this case the EXAMINE^{2D} main menu.

Menu Box

The Menu Box contains the options for each menu – the main menu is shown in Figure 1.1. Selecting any of the options by moving the cursor to the appropriate button and pressing *Enter* or the *left mouse button*, will activate the option or take you to the sub-menu that it represents. Buttons which lead to sub-menus are identified with a triangular arrow symbol >.

Also, certain menu options will display a star (✶) beside the name of the option to indicate when a feature has been toggled ON (eg. the Stress Trajectories option), or to indicate which option is active (eg. the Field Stress or Failure Criterion types).

Plot Display Box

The Plot Display Box is used for:

- all graphical presentation of model data
- cursor selection of input and output files
- pop-up windows for input and editing of data.

Interactive graphical manipulation through the use of the cursor occurs in this area. Throughout this manual, the Plot Display Box may also be referred to as the *viewing region*, *display region*, *drawing region*, etc. depending on the context.

Prompt Line

The Prompt Line is used for various text and numerical input required throughout the program and also contains important instructions and helpful hints as you move through the options in EXAMINE^{2D}. When in doubt about an option, *read the prompt line*.

NOTE: when using **Read File** and **Write File**, you may enter a filename which includes a lengthy path — *the prompt line will scroll to allow up to 80 characters to be entered*.

Using the Menu System

EXAMINE^{2D} uses a sidebar tree structured menuing system for user interaction. Select an option by moving the highlighted menu bar with the keyboard arrow keys or by sliding the mouse up and down until you are highlighting the option you wish to select. Now either press the *Enter* key or the *left mouse button* to select the option.

*Use the **Home** and **End** keys, or the **right mouse button** to quickly toggle between the first and last options in a menu.*

- The *Home* and *End* keys will take the menu bar to the top and bottom of a menu respectively. The *right mouse button* will also take you immediately to either top or bottom of a menu. In addition, each sidebar menu is a closed loop, so that the first item in a menu can be directly accessed from the last item, and vice versa.

By alternately clicking the right mouse button *and then the* left mouse button, *the user can rapidly RETURN back through multiple sub-menu levels.*

- Menu options with an arrow symbol (>) in the right-most position of the button label indicate options which lead to sub-menus.

- Selecting the **RETURN** option in each sub-menu will return you to the previous menu. Hint: by alternately clicking the *right mouse button* (to highlight RETURN at the end of each menu) and then the *left mouse button* (to select RETURN), the user can rapidly RETURN back through multiple sub-menu levels.

- The ***<Alt X>*** keyboard combination can be used as a short cut to EXIT the program (without saving) at any stage.

- The menu options in **CAPITAL LETTERS** are the most commonly used, or mandatory options. Those in **Lower Case** are less commonly used, or non-mandatory.

- A star (✶) beside a menu option is used in the **Field Stress** and **Failure Criteria** menus, to indicate the current active field stress and failure criteria types. It is also used to indicate when **Trajectory** display has been toggled ON, and when a **Half space** is being used.

Moving the Cursor

At times you will be required to select or enter points within the drawing region using a crosshair cursor, or selection box (a small box around the cursor location). Move the cursor with the mouse if you have one, and use the *left mouse button* to enter points, or select options.

You may also use the arrow keys to move the cursor. Hold down the *left shift* key while using the arrow keys for larger cursor jumps. Hold down the *right shift* key to move the cursor pixel by pixel. Usually the *Enter* key is used to 'select' an option and the *Esc* key is used to 'escape' from the option in use.

Popup Menus

Some data is entered in EXAMINE^{2D} via popup input menus. Move about these menus with the arrow keys until the cell holding the parameter you wish to change is highlighted. To enter data, press *Enter* when you are on the cell you want to change, type the new information, and press *Enter* again (or one of the arrow keys) when you are done. You can also select an area to enter data in by clicking the mouse in a cell. To *save* newly entered values, press **<***Alt S***>** or click the mouse on the Save [ALT-S] button. To leave the popup menu without accepting the values you entered, press **<***Alt A***>** or click the mouse on the Abort [ALT-A] button.

Special Keys

In the **MODEL** portion of EXAMINE^{2D}, the keyboard ***function keys***, F1, F2 etc., can be used to quickly carry out certain functions. The following is a list of the active function keys and the operations they perform. These can be pressed at any time that you are *not* entering geometry or data in the Plot Display Box, the Prompt Line or the popup windows:

F1	Redraw	Redraws all entities (ie. excavations, grids etc.)
F2	Autoscale	Autoscales all entities to the size of the current viewport.
F3	Unused mem.	Displays the amount of memory available.
F4	Zoom Out	Scales apparent size of the model down by 75%
F5	Zoom In	Scales apparent size of the model up by 125%
F7	Grid On/Off	Toggles the Grid on and off
F8	Curs On/Off	Toggles the Cursor on and off
F9	Snap On/Off	Toggles the Snap on and off

Hot Keys

Two hot key combinations are available in EXAMINE^{2D}:

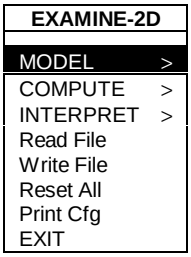
Hot key
Use the ***<Alt X>*** *keyboard combination to exit the program immediately from any menu.*

<Alt X>	**EXIT** EXAMINE^{2D} immediately, from any menu, without saving.
<Ctrl P> *or* <Ctrl O>	Dump the screen to a printer, or PCX file. User specifies either ***<Ctrl P>*** or ***<Ctrl O>*** as the active hot key combination, in **Print Cfg** menu.

Main Menu

The first screen presented by EXAMINE^{2D} is the title screen shown in Figure 1.1. The eight menu items available from this screen constitute the *main menu* – **MODEL**, **COMPUTE**, **INTERPRET**, **Read File**, **Write File**, **Reset All**, **Print Cfg**, and **EXIT**. These are described below. See Part 2 – Reference Manual, for detailed information on MODEL, COMPUTE and INTERPRET.

MODEL



This is the pre-processor module of the EXAMINE^{2D} program. All input data is entered using **MODEL**, including excavation geometry, material properties, boundary conditions, field stress, etc. **MODEL** is used to generate a new model from scratch, and also to edit an existing model.

See the *first* chapter of Part 2 – Reference Manual, for complete and detailed information on the **MODEL** option.

COMPUTE

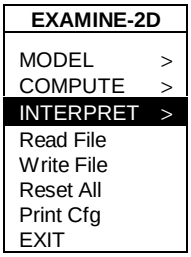
Selection of this option will prompt the user:



Answering *Yes* will initiate the EXAMINE^{2D} compute engine, and present a new screen allowing the user to see the progress of the analysis. When the analysis is complete, control is passed back to EXAMINE^{2D} for interpretation of results.

See the *second* chapter of Part 2 – Reference Manual, for more information on **COMPUTE**.

INTERPRET



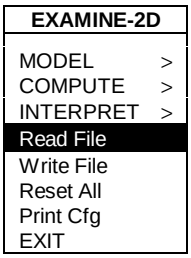
This is the post-processor (results visualization) module. It allows the user to display the results of an analysis for interpretation purposes and report preparation.

See the *third* chapter of Part 2 – Reference Manual, for complete and detailed information on the **INTERPRET** option.

Read File

This option retrieves an EXAMINE^{2D} file. Selecting **Read File** will list all EXAMINE^{2D} files in the plot display area, in alphabetical order, and the prompt line will show:

```
> Input filename or select w/ mouse [ ]:
```



To load a file you may either:

- type the name of the file (without the ***.exa*** filename extension), on the prompt line and press *Enter*, or
- position the cursor over the desired filename on the screen and select it with the *left mouse button*.

Note that if you are using the prompt line to enter a filename which includes a lengthy path, *the prompt line will scroll to allow up to 80 characters to be entered*.

If there is already a file loaded when you select **Read File**, you will see:

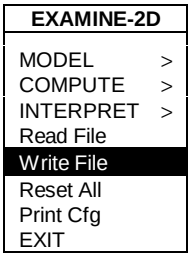


Selecting *Yes* will clear the screen of the existing model, allowing a new file to be read in as described above.

Write File

This option saves the current model as an EXAMINE^{2D} file, with an ***.exa*** filename extension. On selecting this option a list of existing EXAMINE^{2D} files will be displayed in the plot display area in alphabetical order, and the prompt line will show:

```
> Output filename or select w/ mouse [current file.exa]:
```



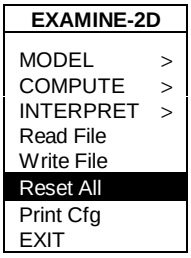
When working with a new model (or to rename an existing one), type a new filename in the prompt line, ***without*** the ***.exa*** filename extension.

When working with an existing file, the current filename will appear in the prompt line (see above) — *the user can select this by simply pressing Enter*. Alternatively, position the cursor over the desired filename on the screen and select it with the *left mouse button*. Before a file is overwritten, a pop-up prompt will appear:



Selecting *Yes* will overwrite the file with the new model.

Reset All



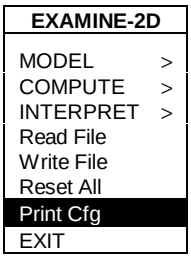
If the user wants to clear an existing model from the screen, to begin entering a new model, select the **Reset All** option. You will first be prompted:



Selecting *Yes* will clear the screen, allowing the user to start a new model from scratch.

Printer Configuration

EXAMINE^{2D} provides an option which allows for capturing the screen to various printers, or to PCX format files. To capture a screen you must first configure the program for your printer, using the **Print Cfg** option in the main menu. When this option is selected, a popup menu is displayed which allows the user to select various printer types, as well as print options.



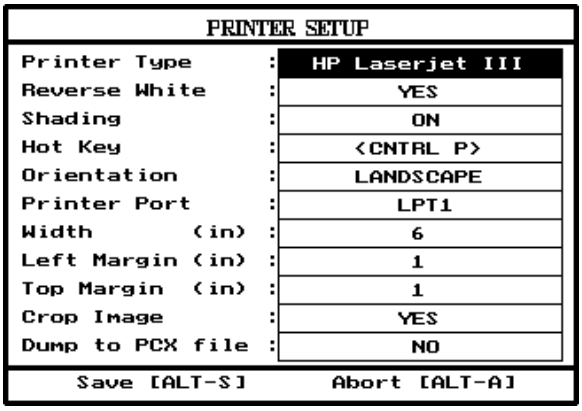


Figure 1.2: Print Configuration popup menu.

Screen captures used in EXAMINE^{2D} have been tested in CGA, EGA, VGA, and Super VGA 800x600 graphics modes. The following options can be toggled through their various selections and are available to the user when the **Print Cfg** option is selected:

Printer Type – This option allows the user to select the type of printer they will be using. The printers supported by EXAMINE^{2D} are:

HP Laserjet	HP Paintjet	HP Deskjet
HP Laserjet III	HP Paintjet XL	HP Deskjet 1200C
HP Laserjet IV	HP Paintjet XL 300	Epson 9 and 24 Pin Printers

Beeps

When you press the hot key combination to print the screen, you will hear two "beeps" — to indicate the beginning and the end of the printing process.

When sending the image to a printer, the second "beep" may take some time, as the information is transferred from the computer to the printer.

When capturing an image to a PCX file, the second beep should immediately follow the first.

Reverse White – This option allows the user to select whether *white text* is printed as *white* or *black*. (All *images* are printed with a white background.)

Shading – This option allows users with black and white printers to print colours as shaded patterns.

Hot Key – This option allows the user to toggle between two ***hot key*** combinations, <*Ctrl P*> and <*Ctrl O*>. Pressing the ***hot key*** combination initiates the printing process to the designated printer or PCX file. When you press the hot key combination to print the screen, you will hear two "beeps" to indicate the beginning and the end of the printing process.

Orientation – This option allows you to toggle between two orientations, landscape and portrait. The portrait option prints the screen in a traditional vertical page format. The landscape option prints the screen aligned with the long axis of the page (ie. at 90° to the portrait option).

Printer Port – This option allow you to toggle between three printer ports: LPT1, LPT2, or LPT3, in order to select which port the image is to be sent to.

Width, Left Margin and Top Margin – These three options allow you to vary the width of the image and the spacing of the margins of the screen to be printed. Note that the width *always* refers to the width aligned to the short direction of the paper, in both the landscape and portrait orientations.

Crop Image – This useful option allows cropping and printing of any portion of the screen, by activating a window at the time the printing process is started (ie. after pressing the hot key combination). Use the mouse or the keyboard arrow keys to locate the *upper left* and *lower right* corners of the window. Enter a window corner and toggle to the opposite corner by clicking the *left mouse button* or *Enter* on the keyboard – this can be repeated any number of times if the location is not satisfactory. When the location of the box has been determined, hit *Esc* or the *right mouse button* to start printing the cropped image. **HINT:** The box location can be adjusted very precisely by holding down the *right shift key*, and using the *keyboard arrow keys* – this will move the edges of the box in very small (single pixel) increments.

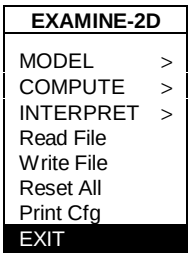
HINT: *The box location can be adjusted very precisely by holding down the* right shift key, *and using the* keyboard arrow keys, *to move the edges of the box in single pixel increments.*

*To capture an image to a PCX file, press the **hot key** combination as you would to send the image to a printer.*

*Use the **REVERSE** option to capture the PCX image with a white background.*

Dump to PCX File – This option allows you to save the screen image to a file in PCX format instead of sending it to a printer. This is useful for importing graphics images into word processor documents or drawing programs. To save an image to a file, toggle the Dump to PCX File option ON. Use the REVERSE option to capture the PCX image with a white background. To capture the image to a PCX file, press the hot key combination as you would to send the image to a printer. The file will be saved with the following preset filename format : screen?.pcx. The ? will be a number that starts at 1 and increments by 1 depending on how many PCX files already exist in the current directory – i.e. screen1.pcx, screen2.pcx, screen3.pcx etc. In some cases, programs reading in PCX files dumped with a white background, may display or print the background as grey. *To ensure a white background*, read the PCX file into a package such as Pizazz Plus for Windows, and **Save As** PCX Black & White.

EXIT



When finished working with EXAMINE^{2D}, select **EXIT** from the main menu to quit the program. If you are not in the main menu, select RETURN at the bottom of each sub-menu, until you are back to the main menu. Before exiting EXAMINE^{2D}, a pop-up prompt will ask:



Selecting *Yes* with the mouse, or typing 'y' will quit the program. Recall that you can exit the program at any stage by pressing the hot key combination *<Alt X>*.

Hints about the Manual

- Text in **BOLD** (eg. **BOUNDARIES** or **Elast Prop**) in most cases indicates a sidebar menu item.
- Text enclosed by a box eg.:

> ENTER MINIMUM X,Y COORDINATES <0,0>:

indicates a prompt line message, found at the bottom of the EXAMINE^{2D} screen.

Part 2 – Reference Manual

INCLUDES:

MODEL

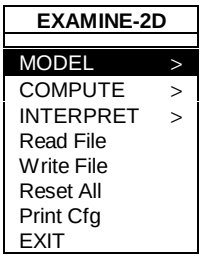
COMPUTE

INTERPRET

Creating a MODEL

Path: MODEL

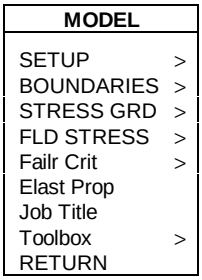
MODEL is the pre-processing module used for entering and editing the excavation geometry, material properties, in-situ stresses and boundary conditions.



The **MODEL** menu (shown at left) includes:

⇒ a **SETUP** option, for setting important model and analysis parameters (eg. Continuum Type, Element Type, Symmetry Axes, etc.)

⇒ the **BOUNDARIES** menu, for entering and editing all excavation geometry, including Boundary Conditions, Joints, Free Surfaces, Discretization, etc.



⇒ the **STRESS GRID** option, for specifying at which points stresses are to be calculated. The user specifies one or more uniform rectangular grids in which stresses are calculated at each grid point

⇒ the **FIELD STRESS** menu, for setting up the appropriate field stress conditions (eg. Constant, Gravity, User Defined)

⇒ a **Failure Criteria** option, for entering Hoek-Brown, Modified Hoek-Brown, or Mohr-Coulomb failure criterion parameters

⇒ an **Elastic Properties** option, for entering elastic properties for Isotropic or Transversely Isotropic material

Note on Units

EXAMINE^{2D} may be used with any consistent set of units (see table below), although you will find that **MODEL** will prompt you for data with:

- meganewtons (**MN**) as the force unit,
- megapascals (**MPa**) as the stress unit and
- meters (**m**) for the length unit.

Stresses generated by the boundary element analysis using these units are in MPa. We recommend that you enter all your data in the above units. All angle measurements MUST be in degrees.

LENGTH	UNIT WT	FORCE	STRESS
m	MN/m^3	MN	MPa
ft	lbf/ft^3	lbf	lbf/ft^2

Table 2.1: Consistent units (metric and imperial)

Coordinate Entry

When entering model geometry, you may use the keyboard, or the mouse, or any combination of the two, to enter coordinates:

- Use the keyboard to type coordinates in the Prompt Line at the bottom of the screen and press *Enter*. Note that the use of a **comma** to separate the x,y coordinates is actually optional, and you may simply leave a **space** between the x and y values in the Prompt Line. (This also applies whenever two (or more) numbers are being entered in the Prompt Line, for instance, the normal and shear stress boundary conditions).

- Use the mouse (or the keyboard arrow keys), to select a point by moving the cursor to that position, and press *Enter* or the *left mouse button*. Recall that the cursor position is always indicated by the LOC: (x,y location) at the left side of the Upper Status Box. The coordinate system uses the convention: *x is horizontal and increasing to the right and y is vertical and increasing upwards*.

Stress Convention

EXAMINE[2D] uses the convention:

- COMPRESSIVE stresses are **POSITIVE**

- TENSILE stresses are **NEGATIVE**.

SETUP

Path: MODEL / SETUP

MODEL
SETUP >
BOUNDARIES >
STRESS GRD >
FLD STRESS >
Failr Crit >
Elast Prop
Job Title
Toolbox >
RETURN

The **SETUP** menu includes

⇒ **Analysis Options**, for selecting important model and analysis parameters, and

⇒ **Set Limits**, for setting the initial limits on the model geometry.

These are described below.

Analysis Options

Path: MODEL / SETUP / Anlsys Opt

The **Analysis Options** in **SETUP** give you control over various analysis parameters. These options should be chosen *before* you create the model. The **Analysis Options** popup menu and the options which may be toggled in it, are shown below:

SETUP
Anlsys Opt
Set Limits
RETURN

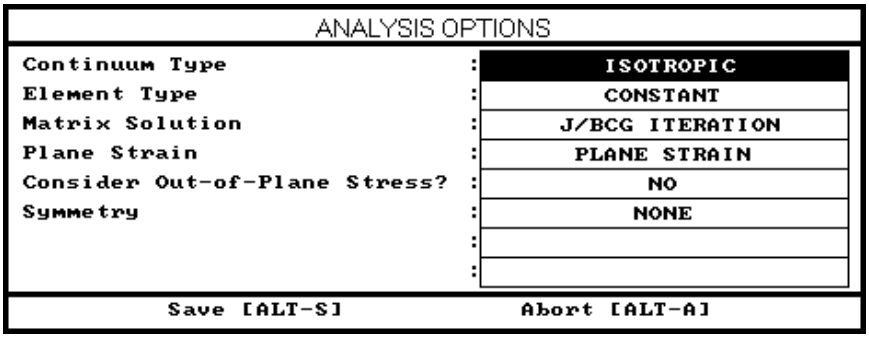


Figure 2.1: Analysis options menu.

Continuum Type	Isotropic		Transversely Isotropic	
Element Type	Constant	Linear	Quadratic	
Matrix Solution	J/BCG Iteration		Gaussian Elimination	
Plane Strain	Plane Strain		Complete Plane Strain	
Consider Out-of-Plane Stress?	Yes		No	
Symmetry	None	About X	About Y	Both

Table 2.2: Choice of analysis options.

The individual **Analysis Options** are discussed in the following sections.

Continuum Type

In EXAMINE^{2D} **Continuum Type** may be either:

⇒ **ISOTROPIC** (having the same properties in every direction) or

⇒ **TRANSVERSELY ISOTROPIC** (having different properties along two perpendicular axes).

Isotropic

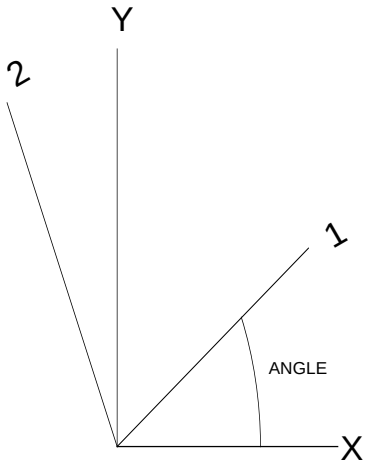
For isotropic materials, the only Elastic Properties required are:

⇒ Young's Modulus

⇒ Poisson's Ratio.

Transversely Isotropic

For transversely isotropic materials, several Elastic Properties are required (see the margin figure):



⇒ Young's Modulus in each of the two perpendicular directions,

⇒ the Shear Modulus for the two directions,

⇒ two Poisson's Ratios, one between directions 1 and 2 (see margin figure) and one between direction 1 and the perpendicular to the screen (normal to the page),

⇒ the angle between the first direction and the positive X axis.

See the Elastic Properties section later in this chapter for more information about this input.

Note:

1) you **cannot** use the **Half Space** option (see BOUNDARIES->Add Surf->Half Space) with a transversely isotropic continuum,

1) you **cannot** use non-constant elements (ie. linear or quadratic) with a transversely isotropic continuum.

Element Type

Three types of boundary elements can be used in EXAMINE^{2D}:

⇒ **CONSTANT**

⇒ **LINEAR**

⇒ **QUADRATIC**

Note:

1) Higher order elements (ie. linear or quadratic) will increase the solution time due to the increase in degrees of freedom.

1) If you are using the **Half Space** option you must use CONSTANT elements.

1) If you are using the **Transversely Isotropic Continuum** option you must use CONSTANT elements.

Matrix Solution

This option toggles between

⇒ **J/BCG ITERATION**

⇒ **GAUSSIAN ELIMINATION**

When **COMPUTE** solves the matrix representing the system of equations defined by your model, it normally attempts to use **J/BCG** (Jacobi/Bi-Conjugate Gradient) **ITERATION**, and if the system doesn't converge on a solution, Gaussian Elimination is used. If you wish to force **COMPUTE** to use GAUSSIAN ELIMINATION without attempting the iterative method, toggle the **Matrix Solution** option to GAUSSIAN ELIMINATION.

Plane Strain

This option allows you to toggle between two options:

⇒ **PLANE STRAIN**

⇒ **COMPLETE PLANE STRAIN**

In the **PLANE STRAIN** condition, one principal axis is aligned with the axis of the tunnel. Out-of-plane (axial) principal stresses are considered or ignored, depending on the setting (YES or NO) of the **Consider Out-of-Plane Stress?** option (see next section).

For the **COMPLETE PLANE STRAIN** condition, the stress field may be oriented arbitrarily in three-dimensional space. Out-of-plane stress must always be considered. The analysis generates a three-dimensional stress tensor at each grid point.

Consider Out-of-Plane Stress?

This option toggles between:

⇒ **YES**

⇒ **NO**

For PLANE STRAIN (as opposed to COMPLETE PLANE STRAIN) conditions, you can choose whether to consider out-of-plane stresses or not. Conventional two dimensional plane strain analyses ignore the out-of-plane stress component. In the analysis, however, changes in the in-plane stresses due to an excavation generate changes in the out-of-plane stress ($\Delta\sigma_z = \{ \Delta\sigma_1 + \Delta\sigma_3 \} \times \nu$) through the Poisson's effect which could have an effect on the overstress behaviour and failure mode of the material. If you wish to consider out-of-plane stresses, toggle the **Consider Out-of-Plane Stress?** option to YES**.**

If you are using the COMPLETE PLANE STRAIN option, you cannot disregard out-of-plane stresses – you must toggle the Consider Out-of-Plane Stress option to YES.

If you answer YES**,** COMPUTE **will calculate out of plane stress and** INTERPRET **will plot** σ_2 as well as σ_1 and σ_3. Further, the stress trajectories will reflect the three dimensional nature of the $\sigma_{1\text{-}3}$ plane, and the STRENGTH FACTOR calculations will use a three dimensional failure criterion. Note that σ_1 and σ_3 as plotted in **INTERPRET** will be the **real** major and minor principal stresses regardless of their in-plane or out-of-plane orientation. View the STRESS TRAJECTORIES for clarification.

If you do not wish to consider out-of-plane stress in the analysis, leave this value as NO – EXAMINE^{2D} **will consider only in-plane** σ_1 and σ_3 in computation and presentation. When using symmetry, the in-plane orientation of the horizontal (σ_H) and vertical (σ_V) stresses is set by default.

Symmetry

You can also choose some form of symmetry to decrease model solution time and to make entering the model geometry easier. The four possibilities are:

By using Symmetry, only one half or one quarter of the problem geometry needs to be defined.

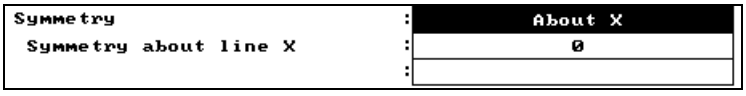
⇒ **NONE**

⇒ **ABOUT X**

⇒ **ABOUT Y**

⇒ **BOTH**

For the last three cases, enter the axis of symmetry in the cells beside Symmetry About Line X or Symmetry About Line Y, as appropriate (see the figures below). For information on **Entering Geometry with Symmetry On** see the **BOUNDARIES** section of this chapter.



Symmetry About a Line Parallel to the X-Axis

Symmetry	:	About Y
Symmetry about line Y	:	0
	:	

Symmetry About a Line Parallel to the Y-Axis

Symmetry	:	About X & Y
Symmetry about line X	:	0
Symmetry about line Y	:	0

Symmetry About Lines Parallel to Both Axes

Set Limits

Path: MODEL / SETUP / Set Limits

Set Limits allows the user to set limits on the x-y coordinate system they will be using. Select MODEL->SETUP->Set Limits and you will see the prompt:

SETUP
Anlsys Opt
Set Limits
RETURN

> ENTER MINIMUM X,Y COORDINATES <0,0>:

Enter coordinates for the **lower left** corner of your drawing region, and press *Enter* to get the prompt:

> ENTER MAXIMUM X,Y COORDINATES <119.82,100>:

To accept the current or default values shown in the prompt lines, simply press Enter *at the prompts, without typing in new coordinates.*

Enter coordinates for the **upper right** corner, and press *Enter*. The screen view will then be updated to your input dimensions. You may notice that the values shown for the Maximum X and Y coordinates are not always exactly the ones you entered. This is because they are automatically scaled to ensure that the view is not distorted and that a line has the same length horizontally and vertically.

Set Limits is intended mainly for use when initially entering the model geometry. After a model has been created, several options in the **Toolbox** can be used to adjust the viewing region with respect to the model. See the section on the **Toolbox** at the end of this chapter.

BOUNDARIES

Path: MODEL / BOUNDARIES

MODEL
SETUP >
BOUNDARIES >
STRESS GRD >
FLD STRESS >
Failr Crit >
Elast Prop
Job Title
Toolbox >
RETURN

The **BOUNDARIES** menu includes

⇒ **ADD EXCAVATION** for defining the excavation geometry

⇒ **DISCRETIZE** for creating the boundary element discretization

⇒ **BOUNDARY CONDITION** for setting up the boundary conditions

⇒ **Add Surface** for adding free surfaces to the model

⇒ **Add Joint** for adding joints to the model

BOUNDARIES
ADD EXCAV
DISCRETIZE >
BNDRY COND >
Add Surf >
Add Joint
Edit Bndry >
DXF Bndry In
DXF Bndr Out
Toolbox >
RETURN

⇒ **Edit Boundary** for modifying the excavation, joint or surface geometry

⇒ **DXF Boundary In** and **Out** for DXF file import/export

These are described below.

ADD EXCAVATION

Path: MODEL / BOUNDARIES / ADD EXCAV

*An **excavation** is defined by a series of straight line segments, which form a continuous closed polyline.*

To input the geometry of the excavation(s) that you wish to solve, use the **ADD EXCAV** option in the **BOUNDARIES** menu. The geometry of the excavations is modelled by a series of straight line segments defined by (x,y) coordinates. The lines must be drawn **counter-clockwise** and then **closed** to form a continuous closed polyline representing the geometry of an excavation – see Figure 2.2 for illustration.

The geometry can be entered either by:

⇒ entering (x,y) coordinates in the prompt line, or

⇒ interactive graphical entry with the mouse and/or arrow keys.

Follow the instructions in the prompt lines:

```
> Enter Excavation Counter-Clockwise, Start Point :
```

Enter the first vertex by typing an (x,y) coordinate pair in the prompt line and pressing *Enter*, or by moving the cursor with the mouse or keyboard arrow keys, and clicking the *left mouse button*.

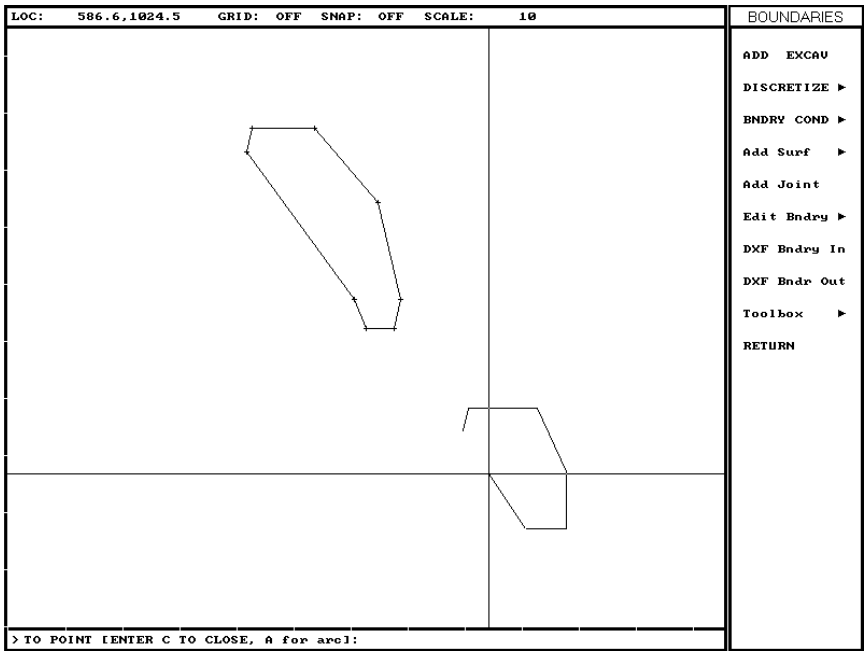


Figure 2.2: Entering excavation geometry using the cursor.

After the first point is entered, the prompt line for all subsequent points will read:

> To Point [c=close, a=arc, u=undo, Esc=done] :

Keep in mind that if you wish some parts of an excavation to have different boundary conditions from other parts, **you should start a new line segment at each change in boundary condition**, even if the regions are all on a straight line (see the BOUNDARY CONDITIONS section later in this chapter).

Using the Snap and Grid Options

The SNAP and GRID options can be useful for entering excavation geometry.

Remember that the (x,y) LOCATION of the cursor is always displayed in the Upper Status Box.

To aid in interactively drawing the excavation(s) you may wish to use the **SNAP** and **GRID** functions.

⇒ To toggle SNAP on or off, press F9

⇒ To toggle GRID on or off press F7

You will notice that while the SNAP is ON, the cursor moves in discrete increments, which helps you to place the polyline vertices which define the excavation. For example, if all excavation dimensions are multiples of some number, say 5, then set the GRID and SNAP dimensions to 5 (see the **Toolbox** section for more information) – the excavation geometry can then be rapidly and accurately entered with the mouse by snapping to grid points. Remember that the (x,y) location of the cursor is always visible at the left side of the Upper Status Box.

Closing Excavation Boundaries

*When all vertices have been entered, type **'c'** to close the excavation.*

To complete an excavation (after all vertices have been entered), type ***'c'*** (for **c**lose) and then *Enter* to connect the last vertex to the first vertex and thus close the boundary. You will automatically be returned to the **BOUNDARIES** menu.

Correcting Mistakes

*To correct mistakes while entering an excavation, type **u** to **undo** the latest action.*

If you have already closed an excavation, you will have to use Edit Boundary to edit the geometry.

If at any time you make a mistake in entering a boundary vertex you may "undo" the previous entry by typing **u** (for **u**ndo) followed by *Enter*. This will remove the most recent boundary vertex from the excavation polyline. This procedure can be repeated any number of times, to remove successive vertices from an excavation you are drawing – experiment with this option to see how it works.

Note: If you have already closed the excavation, you must use the Edit Boundary option of BOUNDARIES to edit the excavation. The **u** command is only operative while you are initially entering an excavation.

Drawing Arc Segments and Circles

*To include **arcs** or **circles** in an excavation, type the **a** command in the prompt line and follow the instructions.*

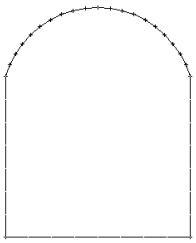
To include **arc segments** or **circles** in excavation geometry, type '**a**' (for **a**rc) and *Enter* in the prompt line, while entering an excavation. You will see the prompt:

> Number of segments in arc <20>

Press *Enter* to accept 20, or type in a new number. Enter the second (intermediate) and third (end) points on the arc at the next prompts:

> Enter second arc point:

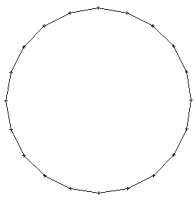
> Enter third arc point:



Experiment with this option to see how it works. Note that:

⇒ **the arc is actually made up of straight line segments.**

⇒ If you press **c** and *Enter* before selecting the *third* point, the arc will close on the start point of the current polygon.



⇒ To draw a **circle**, put the mouse on a point on the circle. Press **a** and *Enter*. Input the number of segments. Move the cursor to the point on the circle diametrically opposite the first point (note that as you move the mouse, a semicircle follows it around). Now press **c** and *Enter*, and you will have drawn a circle with a diameter between the two points.

Entering Geometry With Symmetry On

When **Symmetry** is toggled (see **SETUP**->**Analysis Opt**) only *one half* (or *one quarter* if the symmetry condition is BOTH) of the problem needs to be defined. When using symmetry, notice that part of the screen is shaded. The lines dividing the shaded and unshaded portion of the screen represent the axes of symmetry. Any point selected in the shaded portion will automatically be relocated vertically or horizontally to the nearest axis of symmetry.

To model a symmetrical excavation, input the start point on an axis of symmetry (or click the mouse with the cursor just inside the shaded region near the desired location on the axis). Move counterclockwise now in the *unshaded* region inputting desired vertices. When you come to entering the second point on the axis of symmetry, input this point from the keyboard or click the mouse with the cursor just inside the shaded region near the desired point on the axis. Now press **c** and *Enter*. The excavation will close along the axis (axes) of symmetry but mirror images of any excavations will be drawn in the shaded region.

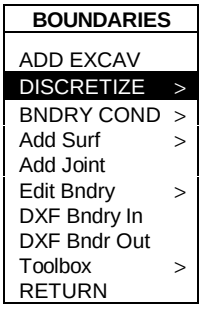
There may still be a boundary line visible along the axis (axes) of symmetry but it is ignored by the program. If a closing line is drawn from the start point to the end point and is *not* on a symmetry axis, then one of these two points was incorrectly input.

See also the sub-section ***A Note on Free Surfaces and Symmetry*** *later in this chapter.*

Discretize the problem normally (see the next section) and you will notice that if the problem has been generated correctly no elements will appear on the mirrored boundaries or on the lines of symmetry. *If this is not the case, input the boundaries again.*

DISCRETIZATION

Path: MODEL / BOUNDARIES / DISCRETIZE



The next step after defining the excavation geometry is to **DISCRETIZE** the lines making up the boundaries, including:

⇒ **excavations**,

⇒ discretized free **surfaces**

⇒ **joints**

into **boundary elements**. This process subdivides line segments into a number of elements (sub-segments), with the *total* number of elements for each boundary type specified by the user.

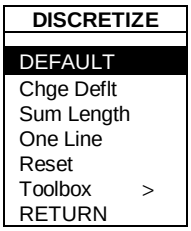
Excavations, ***surfaces*** *and* ***joints*** *can all have their own independent default discretization values.*

The default number of elements when you start up the program is 50 in total, for *each* boundary type (excavation, surface, joint). The program will subdivide the boundaries in such a way that there are **approximately** 50 elements per boundary type.

Note that **excavations**, **surfaces** and **joints** all have their own independent default discretization values, which may be independently changed using the **Chge Deflt** option (see below).

DEFAULT

Path: MODEL / BOUNDARIES / DISCRETIZE / DEFAULT



The quickest and simplest way to discretize your excavation geometry is to choose the **DEFAULT** discretization option. When you choose this option the program will automatically divide the excavations, surfaces (if present) and joints (if present) into the default number of elements. The resulting number of elements will be **approximately** equal to the default settings and will be displayed in the prompt line as follows (eg.):

> No. Excav, Surf, Joint Elem = 50,50,50

Note: DEFAULT discretization is ***not*** *automatically applied by EXAMINE²ᴰ.*

You must select DEFAULT to ***apply*** *the default discretization, otherwise EXAMINE²ᴰ will not write the file when you select COMPUTE.*

The default setting is 50 elements for excavations, 50 elements for surfaces and 50 elements for joints. These defaults can be altered independently through the **Chge Deflt** menu option described below. Selecting the **DEFAULT** option removes any previous discretization including any set with the **One Line** option.

You may rediscretize using the **DEFAULT** option at any time, using different values of the default discretization changed through the **Chge Deflt** option. However, changes in the default value(s) are reflected in your problem only *after* you select **DEFAULT** to rediscretize the model.

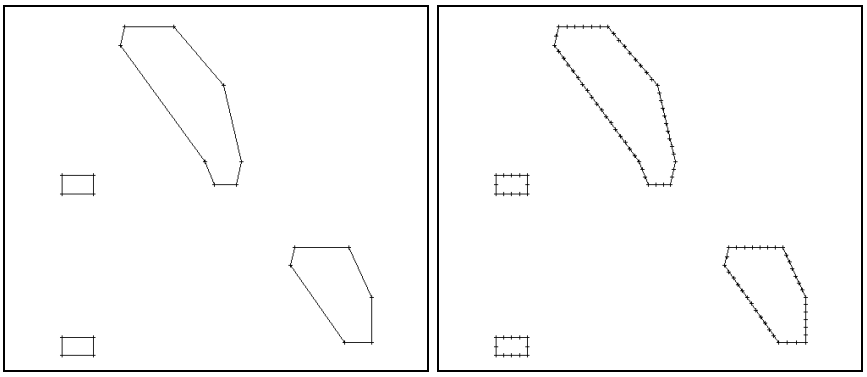


Figure 2.3: Excavations, before and after discretization.

Chge Deflt

Path: MODEL / BOUNDARIES / DISCRETIZE / Chge Deflt

DISCRETIZE
DEFAULT
Chge Deflt
Sum Length
One Line
Reset
Toolbox >
RETURN

You may change the DEFAULT number of elements that EXAMINE^{2D} uses when discretizing excavations, surfaces or joints with the **DEFAULT** option, by selecting **Chge Deflt**. The program will prompt you for new default values. For instance, if your model includes excavations, surfaces and joints, you will see:

> Excavation Default Number of Elements < 50 > :

> Surface Default Number of Elements < 50 > :

> Joint Default Number of Elements < 50 > :

*You must re-select **DEFAULT** after using **Chge Deflt**, to apply the new default discretization values.*

When you then re-select the **DEFAULT** option, the resulting number of elements will be approximately equal to the new value(s) you have entered at the above prompts.

For the standard DOS real mode version, approximately 150 constant elements can be solved directly in memory. Any more than this will cause hard disk swapping and a noticeable degradation in solution speed.

SUM LENGTH

Path: MODEL / BOUNDARIES / DISCRETIZE / Sum Length

DISCRETIZE
DEFAULT
Chge Deflt
Sum Length
One Line
Reset
Toolbox >
RETURN

There may be times when the **number** of elements is not as important as the actual **length** of each element. If you want to define each element to be of a given length, use the **SUM LENGTH** option. This option will first prompt you for an element length:

> Enter Approximate Element Length:

After entering a length, **SUM LENGTH** *will calculate the number of elements that would result if every element were that length*, and display the results in a prompt (eg.)

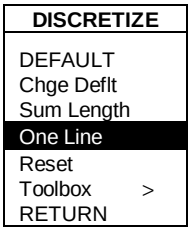
> Approx. No. Excav, Surf, Joint Elem = ## , ## , ##

NOTE: SUM LENGTH does not rediscretize for you, it is simply a function for returning the numbers at the above prompt. After using SUM LENGTH:

- *Enter the number(s) this function returns as the new default discretization values using the* ***Chge Deflt*** *option.*
- *Then select the* ***DEFAULT*** *option to discretize your geometry into elements of approximately the desired length.*

One Line

Path: MODEL / BOUNDARIES / DISCRETIZE / One Line



For maximum customization, the user can control the discretization of individual line segments using the **One Line** option. This option lets you select an already discretized line segment and change the number of elements on that line. You are first asked to select lines by moving the selection box, using the mouse or the keyboard arrow keys, until a line segment passes through it.

> Select Line(s) [u=undo, w=window, Esc=done]:

With the ***One Line*** *option, the user can customize the discretization of individual line segments.*

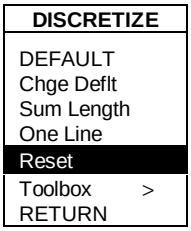
Press *Enter* or the *left mouse button* to select lines. (If you enter 'w' in the prompt line, you can create a window to select multiple line segments simultaneously. If you enter 'u', you can 'undo' the selection of a line segment.) Next, you are prompted for the number of elements that you wish to discretize the line segment(s) into.

> Enter #Elements for Line Segment(s) [##]:

When you have typed in the number of elements and pressed *Enter* the selected line(s) will be subdivided into the number of elements that you have just specified.

Reset

Path: MODEL / BOUNDARIES / DISCRETIZE / Reset



The **Reset** option allows you to erase the current discretization and restart the discretization process.

Note that **Reset** *does not reset* the default discretization *values* – these can only be changed through the **Chge Deflt** option.

BOUNDARY CONDITIONS

Path: MODEL / BOUNDARIES / BNDRY COND

BOUNDARIES
ADD EXCAV
DISCRETIZE >
BNDRY COND >
Add Surf >
Add Joint
Edit Bndry >
DXF Bndry In
DXF Bndr Out
Toolbox >
RETURN

BNDRY COND
DEFAULT
Chge Deflt
One Line
Reset
Show BndCd
Toolbox >
RETURN

The next step after you have discretized the excavations is to set the boundary conditions. The procedure for applying boundary conditions is similar to the discretization process. The **DEFAULT** boundary conditions are **stress-free**, i.e.

⇒ zero **normal** stress

⇒ zero **shear** stress

along the boundary, but either of these default conditions may be changed through the **Chge Deflt** option. As with the discretization process you may also apply customized boundary conditions to individual line segments through the **One Line** option.

NOTE: *Surface segments automatically receive a default condition equal to a stress free boundary* – the **One Line** option must be used to input a loading on a surface segment.

If you wish to input non-zero boundary conditions, observe the sign conventions defined below for normal and shear stress applied to a boundary element (line segment).

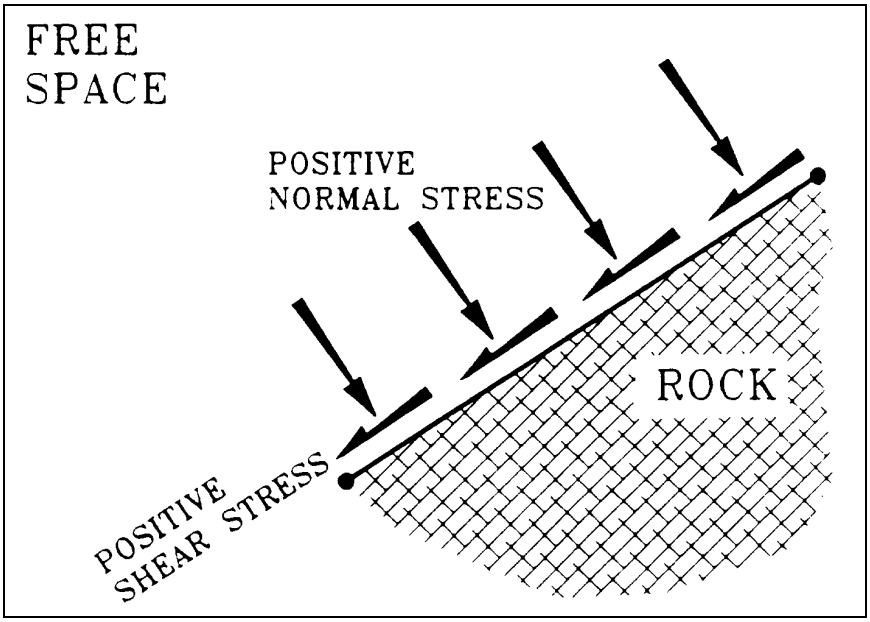
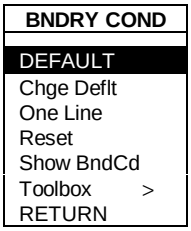


Figure 2.4: Sign Convention for Non-Default Boundary Conditions

DEFAULT

Path: MODEL / BOUNDARIES / BNDRY COND / DEFAULT



In most cases you will want to apply the **DEFAULT** stress free boundary conditions to your problem to simulate the condition in which all material is completely removed from within the boundary and zero internal pressure is applied. *You must select* this option to apply the DEFAULT boundary conditions, otherwise EXAMINE^{2D} will not write the file when you select COMPUTE.

When you select the DEFAULT boundary condition option you will see the prompt:

> Boundary Conditions Set to (0 , 0) for all Excavations

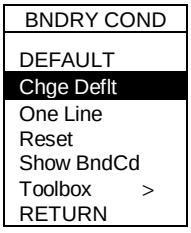
If you have changed the default boundary conditions, using the **Chge Deflt** option (see below), then selecting **DEFAULT** will erase the previous boundary condition setting for all line segments making up the excavation(s), and the above prompt will display the new default values which you have chosen, rather than (0 , 0).

*Note: Default Boundary Conditions are **not** automatically applied to excavations by EXAMINE^{2D}.*

*You must select DEFAULT to **apply** the default boundary conditions, otherwise EXAMINE^{2D} will not write the file when you select COMPUTE.*

Chge Deflt

Path: MODEL / BOUNDARIES / BNDRY COND / Chge Deflt



For cases where internal pressure is to be applied to the inside of the excavation, the **DEFAULT** boundary conditions can be changed using the **Chge Deflt** option. Select this option and input your new default normal and shear stress boundary conditions at the prompt line:

> Default Excav. Normal, Shear Stress Bndry Cond. (0,0):

(**NOTE**: be careful to observe the sign convention illustrated in Figure 2.4, for non-zero boundary conditions).

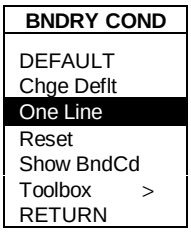
Then you must re-select **DEFAULT** (see above) to *apply* your new default boundary conditions to the excavation(s). Every element on every line segment will have the boundary conditions that you have specified through the **Chge Deflt** option. If you wish to apply non-default boundary conditions to *individual line segments*, then use the **One Line** option explained in the next section.

*After using **Chge Deflt**, you must always reselect **DEFAULT** in order to apply the new default boundary conditions to your model.*

NOTE: the **DEFAULT** and **Chge Deflt** options cannot be used to apply boundary conditions (loadings) to discretized free surfaces. Use the **One Line** option to apply loadings to free surfaces – otherwise it is assumed that the surface boundary conditions are stress free.

One Line

Path: MODEL / BOUNDARIES / BNDRY COND / One Line



You have the ability to customize boundary conditions for any individual line segment using the **One Line** option. You may select line segments from:

⇒ excavations

⇒ discretized free surfaces

⇒ joints

*With the **One Line** option, the user can customize boundary conditions on individual line segments.*

When you choose this option you will first be prompted to select the line segment or segments for which you would like to enter the non-default boundary conditions:

> Select Line(s) [u=undo, w=window, Esc = done]:

Move the cursor selection box, using the mouse or the keyboard arrow keys, and click on the desired line segment(s). If you select either **excavation** or discretized free **surface** line segments, you will see the prompt:

> Default Normal, Shear Stress Bndry Cond. (0,0):

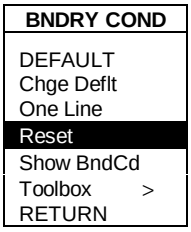
at which you may enter the custom normal and shear boundary conditions that you want to apply. These values will apply to **all** line segments which you have selected. If you select a **joint** line segment, the Joint Properties popup window will appear (rather than the above prompt line), which will allow you to modify the properties of the joint.

*Do NOT select DEFAULT **after** using the One Line option, or your custom boundary conditions will be reset to the default values.*

Note: Any custom boundary conditions only apply until you choose the **Reset** or **DEFAULT** options. This means that you should *first* select the DEFAULT option with the boundary conditions that apply to **most** of the line segments, and then select the **One Line** option to alter any segments with different boundary conditions. Also, note that boundary conditions can only be applied to *line segments* and *not to individual boundary elements*. As a result you should input your geometry in such a way as to ensure that zones with different boundary conditions are represented as individual line segments.

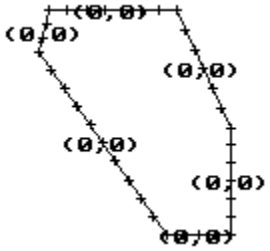
Reset

Path: MODEL / BOUNDARIES / BNDRY COND / Reset



The **Reset** option allows you to delete previously entered conditions for *all* boundary line segments. Use this option if you want to start the process of applying boundary conditions over again.

Show BndCd



BOUNDARY CONDITIONS DISPLAYED (NORMAL, SHEAR STRESS) = (0 , 0) FOR ALL SEGMENTS.

Path: MODEL / BOUNDARIES / BNDRY COND / Show BndCd

This option toggles on the display of the boundary conditions for all excavation and surface line segments, as shown in the margin figure. The **Show BndCd** option is particularly useful if you have been using the **One Line** option to customize the boundary conditions on particular line segments, and would like to examine the current boundary conditions which are in effect.

Add (Free) Surface

Path: MODEL / BOUNDARIES / Add Surf

There are two options for incorporating a **free surface** into an EXAMINE^{2D} problem, available through the **Add Surf** option of the **BOUNDARIES** menu:

BOUNDARIES
ADD EXCAV
DISCRETIZE >
BNDRY COND >
Add Surf >
Add Joint
Edit Bndry >
DXF Bndry In
DXF Bndr Out
Toolbox >
RETURN

⇒ **Half Space** (incorporate an implicit horizontal surface)

⇒ **Discretized** (user defined discretized surface, may be non-horizontal, multi-segmented)

Half Space

Path: MODEL / BOUNDARIES / Add Surf / Half Space

This method allows a horizontal surface to be included implicitly in the mathematical solution. *This means that no extra elements are required to define the surface*. To include a half space, select **Half Space** and input the elevation in the prompt line (see below), or enter it graphically with the mouse – a horizontal line will appear on the screen for this purpose.

> Enter elevation of surface [0] :

SURFACE TYPE
Half Space
Discretized
RETURN

When the Half Space option is being used, you will notice a star (✶) beside the Half Space option, to indicate that you are using this option. Note: check your field stress input to ensure that it is consistent with the presence and position of the halfspace surface line.

Although EXAMINE^{2D} will allow a halfspace to cross excavations or be placed below excavations, this will obviously yield meaningless results.

To delete or reset an implicit halfspace, simply reselect this option and answer YES to the popup prompt:

Halfspace already exists, delete? : **YES** NO

Discretized

Path: MODEL / BOUNDARIES / Add Surf / Discretized

SURFACE TYPE
Half Space
Discretized
RETURN

To input a **non-horizontal** free surface you must use the **Discretized** option of the **Add Surf** menu. Select **Discretized** and follow the prompt line instructions:

> Enter Surface (material to right), Start Point:

> To Point [a=arc, u=undo, Esc = done]:

Input the surface from left to right (i.e., keep the material to the right as you move along the surface). When you are finished with the surface line input, press *Esc* to add the surface to your model.

Discretized *surface elements may be used with the* ***Half Space*** *to model footings or other loadings.*

Use the ***One Line*** *option in the* ***BNDRY COND*** *menu to assign loading to a surface segment.*

NOTE:

⇒ This option gives you great flexibility in defining free surfaces, for instance, surface excavations can be modelled (see last tutorial in Part 3 – Tutorial Manual).

⇒ Note that when excavation boundaries are present, the first and last segments in the surface may be automatically extended a certain distance from the excavations. This is to prevent adverse effects from the end elements on the surface.

⇒ Surface line segments need to be discretized in the same way that excavations are, but they are automatically assigned zero stress boundary conditions. You must use the **One Line** option in the **BNDRY COND** menu to assign loading to a discretized surface line segment – the DEFAULT and Chge Deflt options do not have any effect on surface segments.

⇒ Discretized surface elements may be used with the **Half Space** to model footings or other loadings. However the Half Space must be defined *before* any discretized surface(s) – if the Half Space is present, the discretized surface segments will *not* be extended.

A Note on Free Surfaces and Symmetry

⇒ Free surfaces **cannot** be used with X-Axis symmetry.

⇒ Discretized surfaces **may be used** with Y-AXIS symmetry however they are not automatically extended, so you must make sure that (1) the left side of your input surface contacts the axis of symmetry, and (2) the right side extends *at least three excavation diameters* to the right of any excavations.

Add Joint

Path: MODEL / BOUNDARIES / Add Joint

As of version 5.1, EXAMINE^{2D} allows the modelling of joints or faults using the *displacement discontinuity method*.

BOUNDARIES
ADD EXCAV
DISCRETIZE >
BNDRY COND >
Add Surf >
Add Joint
Edit Bndry >
DXF Bndry In
DXF Bndr Out
Toolbox >
RETURN

To add a joint or other discontinuity to a model, select the **Add Joint** option from the **BOUNDARIES** menu. You will be prompted:

> Enter Joint, Start Point [v=vrtx snap]:

> To Point [v=vrtx snap, u=undo, Esc=done]:

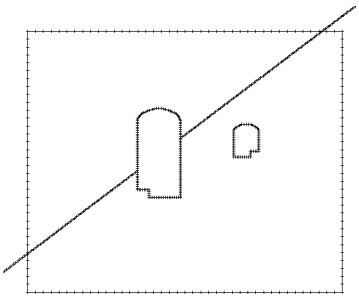
Use the mouse to graphically enter the joint coordinates, or key in (x,y) coordinates at the prompt lines, if preferred. Note that a joint may be made up of more than one line segment. When done, press *Esc* to enter the joint geometry, and the following popup window will appear, for entering all of the joint properties:

JOINT PROPERTIES

Normal Stiffness	(MPa/m)	:	250000
Shear Stiffness	(MPa/m)	:	100000
Joint Aperture	(m)	:	0.1
Allow Slip		:	NO
Slip Criterion		:	MOHR-COULOMB
Tensile Strength	(+MPa)	:	0
Cohesion	(+MPa)	:	0
Friction Angle	(deg)	:	35
Number of Load Steps		:	10
Initial Joint Deformation		:	YES

Save [ALT-S] Abort [ALT-A]

Figure 2.5: Popup window for entering Joint Properties.



If you decide to **not** add the joint to the model *at this point*, then select the *Abort* button on the popup window, or use the <*Alt A*> keyboard combination, and the joint will be removed from the model. Otherwise, enter the desired joint properties, and select *Save* or <*Alt S*> to add the joint to the model. The Joint Properties options are discussed in the following sections.

Note: it is best to add all **excavations** to your model before adding **joints** since EXAMINE^{2D} will then automatically "clip" the joint with the excavation boundaries, and remove any sections of the joint which are internal to the excavations. (Try adding a joint passing through an excavation, *before* and *after* adding the excavation, and see the results for yourself).

Normal and Shear Joint Stiffness

The joints are modelled using constant, non-linear, displacement discontinuities, as formulated in "Boundary Element Methods in Solid Mechanics" by Crouch and Starfield, 1983. Each displacement discontinuity has a **normal** and a **shear** spring **stiffness** which relate the normal and shear stress on the discontinuity to the normal and shear displacement:

$$\sigma_n = -K_n D_n$$

$$\sigma_s = -K_s D_s$$

where:

σ_n , σ_s = Normal, Shear Stress

K_n , K_s = Normal, Shear Spring Stiffness

D_n , D_s = Normal, Shear Displacement Discontinuity

The spring stiffness of the joint can be related to the thickness and Young's modulus of the infilling material by the equations:

$$K_n = -E_0 / h$$

$$K_s = -G_0 / h$$

where:

E_0 = Young's modulus of the infilling material

G_0 = Shear modulus of the infilling material

h = Joint thickness

Joint Aperture

The joint **thickness** or **aperture** is required (see above). During the analysis, the program checks that closure of the joint does not exceed the value of the aperture of the joint.

Allow Slip

Non-linear deformation behaviour can be toggled on through the **Allow Slip** option. This option allows local joint slip if the shear stress on a displacement discontinuity exceeds the shear strength as defined by the **Slip Criterion** (see below). If the Allow Slip option is NO (the default setting), then the joints will behave elastically, and there is no slip.

Slip Criterion

Currently there are two strength (slip) criteria for use with joints in EXAMINE^{2D}:

MOHR–COULOMB, where the required properties are:

⇒ tensile strength

⇒ cohesion

⇒ friction angle

BARTON–BANDIS, where the required properties are:

⇒ joint compressive strength

⇒ joint roughness coefficient

⇒ residual friction angle

Allow Slip *must be toggled to* ***YES*** *before you can enter* ***Slip Criterion*** *properties.*

If the BARTON-BANDIS Slip Criterion has been toggled, the popup window for joint properties (Figure 2.5) will allow you to enter the BARTON-BANDIS parameters instead of the MOHR-COULOMB parameters. Note that the **Allow Slip** option must be toggled to **YES** before you can enter Slip Criterion properties – otherwise you will receive a warning prompt telling you to do so:

```
> Error, First Toggle ALLOW SLIP to YES !
```

Number of Load Steps

If the joints are allowed to slip, then the relaxation of the boundary stresses on each excavation, from the initial in-situ state, to the final stress free state, must occur in a series of steps. *This is to ensure that the non-linear behaviour of the joints is properly modelled*. The number of steps, or **Number of Load Steps**, can be modified by the user.

Initial Joint Deformation

Initial Joint Deformation refers to how the joint behaves in regard to the far field stresses. If the Initial Joint Deformation is YES, then the joint will deform based on *both* the far field stresses *and* the induced stresses due to any excavations. As a result, the stress field in the vicinity of the joint will be altered from the initial far field distribution. This will be evident if you run an analysis with a *joint only* (ie. no excavations).

If Initial Joint Deformation is set to NO, then a single joint, with no excavation, has no effect at all (try running this). With no initial joint deformation, only the induced stresses from the presence of an excavation can cause joint deformation.

Editing Joint Properties

To edit the Properties of a joint *after* it has been added to a model (ie. after you have Saved the Joint Properties which you have entered), you must do this through the BOUNDARY CONDITIONS option, via the path:

MODEL->BOUNDARIES->BNDRY COND->One Line option

(see earlier in this chapter for details). The Joint Properties popup menu will appear, allowing you to modify the properties of the selected joint.

Note: if only one joint has been defined, then the properties can be viewed through the Toolbox->Input Data option. If more than one joint exists, only the number of joints will be displayed.

Restrictions on Use of Joints

If you include JOINTS in a model, you are limited to the following options:

YES!

⇒ **ISOTROPIC** continuum

⇒ **CONSTANT** elements

⇒ **PLANE STRAIN**, with **NO** OUT-OF-PLANE stress

⇒ **J/BCG** iteration

JOINTS **CANNOT** be used with the following options of EXAMINE^{2D}:

NO!

⇒ **TRANSVERSELY ISOTROPIC** continuum

⇒ **LINEAR** or **QUADRATIC** elements

⇒ **OUT-OF-PLANE** stress

⇒ **COMPLETE PLANE STRAIN**

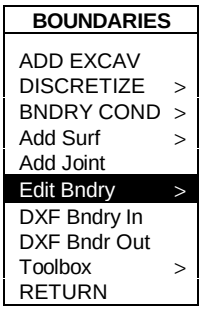
⇒ **GAUSSIAN ELIMINATION** matrix solution technique

⇒ **HALF SPACES**

If you attempt to define a Joint in conjunction with these options, you will receive a warning in the prompt line indicating the problem.

Editing Boundaries

Path: MODEL / BOUNDARIES / Edit Bndry

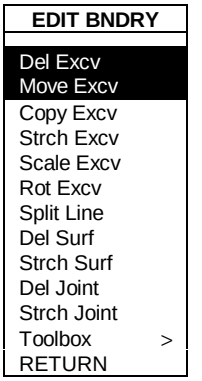


This option lets you interactively modify the model geometry. The options in the **Edit Bndry** menu allow the user to easily modify the shape, size, location etc. of any existing **excavations**, discretized **surfaces** or **joints**.

Del Excv

Path: MODEL / BOUNDARIES / Edit Bndry / Del Excv

This option allows you to *delete* an **excavation**.



> Select Excavation(s) to Erase [u=undo, w=window, Esc = done]:

1. Select an excavation by moving the selection box (cursor) using the keyboard arrow keys or mouse until the box contains part of the excavation you wish to delete – then press *Enter* or the *left mouse button*.
1. Select any number of excavations by repeating this procedure. (You may also use the 'w' command in the prompt line for selecting multiple entities – see margin note).
1. Press *Esc* and all selected excavations will be deleted and removed from the display.

***HINT**: If you enter '**w**' in the prompt line when asked to select, you may select multiple entities for editing (eg. deleting, moving or copying), by drawing a window around the desired objects – follow the instructions which you will see in the prompt line.*

Move Excv

Path: MODEL / BOUNDARIES / Edit Bndry / Move Excv

This option allows you to *move* an **excavation** to another location.

> Select Excavation(s) to Move [u=undo, w=window, Esc = done]:

1. Repeat steps 1 and 2 as in **Del Excv**, and press *Esc*.
1. The *move* is defined by giving two points, a base point and a final point, which define a distance and direction.

> Select Base Point [v=vrtx snap]:

> To Point [v=vrtx snap]:

Use the mouse (preferred) or keyboard to enter the two points, and all *selected* excavations will be translated from their original positions according to the vector defined by the two points.

Note: the base point and final point may be defined anywhere in the display region, only their relative positions are important.

Copy Excv

Path: MODEL / BOUNDARIES / Edit Bndry / Copy Excv

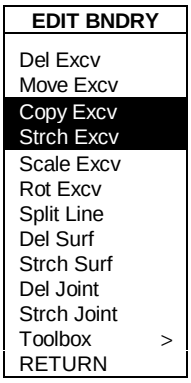
This option allows you to make a *copy* of an **excavation** and place any number of copies in new locations on the screen.

> Select Excavation(s) to Copy [u=undo, w=window, Esc = done]:

You can easily make multiple copies of an excavation by continuing to enter points at the last prompt.

This option is similar to the **Move Excv** option, except the original excavation does not get moved. *You can make several copies of an excavation at once* by continuing to give more final points when prompted. A copy will be placed relative to each point from the base point. When you have finished making copies press *Esc* to exit this option.

Strch Excv



Path: MODEL / BOUNDARIES / Edit Bndry / Strch Excv

With this option the user can modify the *shape* of an existing excavation by **moving one or more excavation vertices**. An excavation vertex is the point between two line segments making up part of an excavation. When you select this option a selection box (cursor) appears in the drawing region.

> Pick vertices to stretch [w=window, Esc=done]:

Select individual vertices by placing the selection box over a vertex and pressing *Enter* or the *left mouse button*. When you are done selecting vertices, press *Esc*. As with the **Move Excv** and **Copy Excv** options, you will be prompted to enter two points defining the stretch (use the mouse or key in coordinates at the prompt lines, if you wish– remember that the vertex snap option is always available):

> Select Base Point [v=vrtx snap]:

> To Point [v=vrtx snap]:

Note that you can select multiple vertices simultaneously, by entering '**w**' (for **w**indow) when asked to select a vertex (at the *first* prompt above). After entering **w**, create a window around the vertices you wish to move (you may graphically enter these points using the mouse):

> PICK FIRST WINDOW CORNER:

> PICK OPPOSITE WINDOW CORNER:

Then enter two points defining the stretch (as before), and all vertices within the window will be stretched to their new positions.

Scale Excv

Path: MODEL / BOUNDARIES / Edit Bndry / Scale Excv

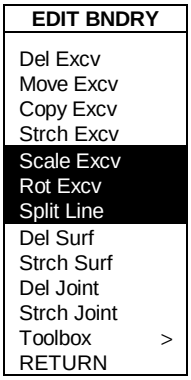
This option allows you to *scale the size* of an **excavation**, by responding to the prompts:

> Select Excavation(s) to Scale [u=undo, w=window, Esc=done]:

> Select Base Point [v=vrtx snap]:

> Enter Scaling [1 , 1] :

To increase or decrease the size of the excavation, enter numbers greater than 1 or less than 1, respectively, at the third prompt. You may independently scale the x or y axis dimensions, by entering different numbers at the third prompt eg. [1.5 , 1] to increase the x-axis dimensions by 50% and leave the y-axis dimensions the same.



Rot Excv

Path: MODEL / BOUNDARIES / Edit Bndry / Rot Excv

This option allows you to *rotate* an **excavation**, by responding to the prompts:

> Select Excavation(s) to Rotate [u=undo, w=window, Esc=done]:

> Select Base Point:

> Enter CCW Rotation (deg):

Enter a **counter-clockwise** rotation, in degrees, at the third prompt, and your selected excavation will be rotated about the base point entered at the second prompt. No scaling or stretching will take place, only a rotation is applied. To rotate an excavation about its center, pick a base point as close to the centroid of the excavation as possible.

Split Line

Path: MODEL / BOUNDARIES / Edit Bndry / Split Line

This option allows the user to *split* any of the line segments making up an **excavation**, into two segments:

> Select Excavation Line(s) [u=undo, w=window, Esc=done]:

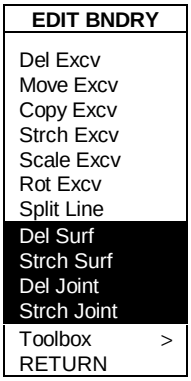
Select line segment(s) with the cursor (you may use the 'w' option) and press *Esc*, and they will be subdivided into two equal segments, as you will see by the new vertex which will appear. If desired, the user can then use the **Strch Excv** option to relocate the new vertex or vertices.

Del Surf

Path: MODEL / BOUNDARIES / Edit Bndry / Del Surf

This option allows you to delete a free surface which has been entered through the BOUNDARIES->Add Surf->Discretized option.

> Select Surface(s) to Erase [u=undo, w=window, Esc = done]:



Select the surface(s) with the cursor, press *Esc*, and the selected surface(s) will be deleted and removed from the display – see the **Del Excv** option for more info.

Note: A Half Space free surface cannot be deleted with this option, you must re-select Add Surf->Half Space to delete a halfspace surface.

Strch Surf

Path: MODEL / BOUNDARIES / Edit Bndry / Strch Surf

This option operates in the same way as the **Strch Excv** option but only discretized free surfaces are modified – see above for details.

Del Joint

Path: MODEL / BOUNDARIES / Edit Bndry / Del Joint

This option operates in the same way as the **Del Excv** and **Del Surf** options, but only joints are deleted – see above for details.

Strch Joint

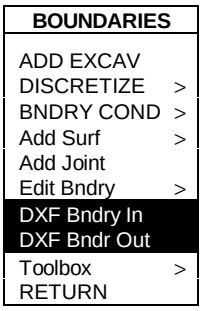
Path: MODEL / BOUNDARIES / Edit Bndry / Strch Joint

This option operates in the same way as the **Strch Excv** and **Strch Surf** options but only joints are modified – see above for details.

Geometry Import / Export via DXF Files

Path: MODEL / BOUNDARIES / DXF Bndry In **or** DXF Bndry Out

DXF Bndry In



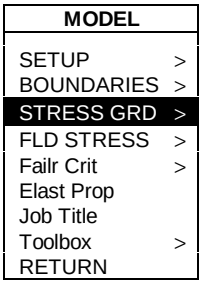
This option allows you to bring in an **AutoCAD** DXF file containing the excavation geometry. Geometries can be stored in DXF format using AutoCAD itself, or using the **DXF Bndry Out** option (see below), and later recalled using this option. The excavation geometry may be in the form of **closed polylines** or **closed lines** entered using the AutoCAD PLINE or LINE commands respectively. The polylines or lines must be entered counter-clockwise with each **closed** polyline or line representing an excavation. Any **open** line segments will be formed into surfaces. Any polylines must be composed of line segments only. This option allows you to use AutoCAD to digitize any drawings that you might have. Release 9 and 10 DXF files are supported, previous versions have not been tested. The layer on which the polylines or lines exist does not matter.

DXF Bndry Out

If you wish to save **geometry** as a DXF file for future use by EXAMINE^{2D}, then use this option. *Do not use the **DXF Out** features in **INTERPRET** for this purpose as you will save the entire screen image in addition to the geometry.*

STRESS GRID

Path: MODEL / STRESS GRD



The results generated from the boundary element analysis (**COMPUTE**) are in the form of stresses at discrete points within the rock mass. The user specifies these points by using the **STRESS GRD** option to draw one (or more) regular rectangular grids in which stresses are calculated at the grid points. These grid point stresses are later used by the post-processor, **INTERPRET**, for data visualization purposes, including plots of:

⇒ Sigma 1

⇒ Sigma 3

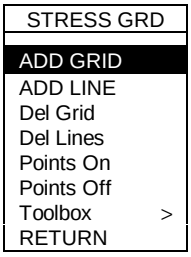
⇒ Strength Factor

⇒ Displacement

COMPUTE calculates the Strength Factor and Displacements at each grid point based on the Failure Criterion and Elastic Properties which are in effect.

ADD GRID

Path: MODEL / STRESS GRD / ADD GRID



To create a grid, select **ADD GRID** and follow the prompts:

> Select Grid Corner #1:

> Select Grid Corner #2:

> Enter Number of Intervals in X direction (##):

> Enter Number of Intervals in Y direction (##):

Draw the grid using the mouse or the keyboard arrow keys to move the cursor (or alternatively you may key in coordinates at the first two prompts), and use *Enter* or the *left mouse button* to enter a grid corner.

The default values for the number of intervals (see the third and fourth prompts above) will define square subregions on the grid with approximately 500 grid points. To accept the default values shown in the prompt lines, simply hit *Enter*, or enter your own values at the prompts, to create your own grid spacing.

When you are finished, the edges of the grid will be marked with the intervals you have chosen, and you will be returned to the **STRESS GRD** menu.

Note that stresses are calculated along the edge of the grid as well as at each internal point. There is no practical limit to the number of grids you can use, although the basic **COMPUTE** and **INTERPRET** modules are limited to approximately 15000 grid points. You may view the actual grid points where stresses are calculated by using the **Points On** option described on the following page.

Note: grid points which fall **inside** an excavation are ignored by **COMPUTE** and **INTERPRET**.

ADD LINE

Path: MODEL / STRESS GRD / ADD LINE

STRESS GRD
ADD GRID
ADD LINE
Del Grid
Del Lines
Points On
Points Off
Toolbox >
RETURN

In some situations, you may wish to calculate stresses along a **line** as opposed to a **grid,** for instance if you are checking extensometer data along a borehole. To define a line along which to calculate stresses, select **ADD LINE** from the **STRESS GRD** menu and follow the prompts:

> Start Point [v=vrtx snap, Esc=abort]:

> End Point [v=vrtx snap, Esc=abort]:

> Enter Number of Divisions <##>:

The procedure is similar to entering a **grid** – see the previous section for more info.

The results calculated along a LINE cannot be viewed within EXAMINE^{2D}. For viewing purposes, **COMPUTE** creates an **.aux** file with the same name as the data file, which can be imported by most spreadsheet packages (**Quattro**, **Lotus 123**, **Excel** etc.) for plotting purposes. However, see the note below–

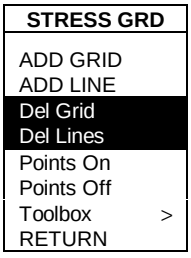
!!! **NOTE** !!!

Since the **Query Data** option has been added to the **INTERPRET** menu (as of version 6.0), stresses can now be viewed and plotted along arbitrary polylines. The values of stress (or strength factor or displacement) are interpolated from the gridpoint values, AFTER the actual boundary element analysis has been performed using a Stress Grid. This is a very useful and flexible new option, and largely replaces the purpose of the **Add Line** option. Only use the **Add Line** option if you wish to **calculate** stresses at particular points along a line, as part of the actual boundary element analysis.

Del Grid

Path: MODEL / STRESS GRD / Del Grid

This option allows you to delete an existing stress grid. When you select this option, you will be prompted:



> Place Cursor Inside Grid and hit the ENTER key

Move the cursor using the mouse or arrow keys until the crosshairs lie entirely within the grid you wish to delete. Press *Enter* or the *left mouse button*, and the grid will be removed from the display. For overlapping grids the program will delete *all* grids that the crosshair lies within so if two grids overlap and you move the crosshair into the region occupied by both grids and select, both grids will be eliminated.

Del Lines

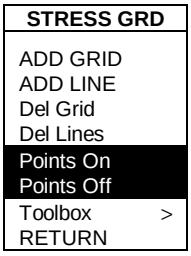
Path: MODEL / STRESS GRD / Del Lines

This option removes all stress lines which were created with the **ADD LINES** option. Note that **all** lines are immediately removed – no selections are necessary and no prompts are displayed.

Points On

Path: MODEL / STRESS GRD / Points On

This option allows you to display the actual grid of points where stresses are calculated. As with the **Del Grid** option, you will first be prompted to select a grid. Move the crosshairs inside the grid using the mouse or keyboard arrow keys and press the *left mouse button* or *Enter*, and the stress points will be displayed for the selected grid.



Points Off

Path: MODEL / STRESS GRD / Points Off

This option removes the display of grid points (for *all* grids) toggled using **Points On**.

A Note on Stress Grids and Symmetry

To align a field grid with an axis of symmetry, select either corner of the grid *just inside* the shaded region of symmetry. The grid will 'snap' to the line of symmetry.

FIELD STRESS

Path: MODEL / FLD STRESS

MODEL
SETUP >
BOUNDARIES >
STRESS GRD >
FLD STRESS >
Failr Crit >
Elast Prop
Job Title
Toolbox >
RETURN

Another parameter required for the analysis is the magnitude and orientation of the in-situ **FIELD STRESS** prior to excavation. In EXAMINE^{2D} three methods are available for defining the field stress:

⇒ **CONSTANT** for defining a constant stress field in which the principal stresses may be orientated in any direction,

⇒ **GRAVITY** for defining a gravitational stress field, and

⇒ **USER DEFINED** for stress fields which vary with depth according to user-defined criteria.

For each option, there are different parameters to be entered depending on the **Analysis Options** you have selected (eg. plane strain with or without out-of-plane stresses, or complete plane strain etc). When you have entered the field stresses, a small stress block will appear on the display indicating the relative magnitude and direction of the principal stresses – see margin figures, next page. This block may be moved or removed from the display using the **Move Blk** and **Delete Blk** options. Note that the convention for stress is POSITIVE COMPRESSION, *so in most cases you will be entering positive values for stresses*.

*Note that the convention for stress is POSITIVE COMPRESSION, so in most cases you will be entering **positive** values for stresses*

NOTE: a star (✶) beside a Field Stress option in the Menu Box indicates the current Field Stress type which is in effect.

CONSTANT

Path: MODEL / FLD STRESS / CONSTANT

The **CONSTANT** Field Stress option is used when the field stresses prior to excavation do not vary with position.

CONSTANT + Plane Strain

FLD STRESS
CONSTANT ✶
GRAVITY
USER DEF.
Move Blk
Delete Blk
Toolbox >
RETURN

If you use CONSTANT field stress with Plane Strain conditions and NO out-of-plane stresses considered, you will be required to enter:

CONSTANT STRESS FIELD	
In-plane Sigma 1' (MPa, Compression +) :	0
In-plane Sigma 3' (MPa, Compression +) :	0
Angle (deg from horiz. of sig1') :	0
Save [ALT-S]	Abort [ALT-A]

Figure 2.6a: Popup window for entering CONSTANT field stress (NO out of plane stress)

⇒ SIGMA 1' is the **major** in-plane principal stress,

⇒ SIGMA 3' is the **minor** in-plane principal stress, and

⇒ the ANGLE is measured in degrees **counter-clockwise** from the horizontal axis to the direction of the major in-plane principal stress.

If you ARE considering out of plane stresses then the popup window for entering CONSTANT field stress will have one additional line:



STRESS BLOCK



STRESS BLOCK WITH DOT REPRESENTING OUT OF PLANE STRESS VECTOR.

CONSTANT STRESS FIELD	
In-plane Sigma 1' (MPa, Compression +) :	0
In-plane Sigma 3' (MPa, Compression +) :	0
Angle (deg from horiz. of sig1') :	0
Out-of-Plane Stress :	0
Save [ALT-S]	Abort [ALT-A]

Figure 2.6b: Popup window for entering CONSTANT field stress with out of plane stress considered.

After entering the values, a stress block will appear on the display, representative of the values you have just entered. If the stress block looks wrong then try entering the data again, keeping in mind the definition of ANGLE. If out-of-plane stress is being considered, a dot will appear in the middle of the stress block representing the out of plane stress vector. Note that if SIGMA 3' = 0.0, no stress block will be drawn. This does *not* indicate an input error.

CONSTANT + Complete Plane Strain

When the Complete Plain Strain condition has been selected, the stress field may be oriented arbitrarily in three-dimensional space. Each principal stress is defined as a vector, with a direction specified as a triplet of direction cosines relative to the (X, Y, Z) axes.

CONSTANT STRESS FIELD FOR COMPLETE PLANE STRAIN			
-Stress field may be oriented arbitrarily in three-dimensional space.			
-The three principal stresses must have a direction vector specified as an X,Y,Z triplet.			
-The X axis is in the section plane, horizontal and to the right.			
-The Y axis is in the section plane vertical and up.			
-The Z axis is normal to the section plane and pointing towards you.			
-stresses should be in MPA with compression +ve			
Principal Stress Sigma A :	0		
Vector A (cosines) :	1	0	0
Principal Stress Sigma B :	0		
Vector B (cosines) :	0	1	0
Principal Stress Sigma C :	0		
Vector C (cosines) :	0	0	1
Save [ALT-S]	Abort [ALT-A]		

Figure 2.6c: Popup window for entering CONSTANT field stress with Complete Plane Strain conditions.

As indicated in the popup window:

⇒ The X-axis is in the section plane, horizontal, and to the right.

⇒ The Y-axis is in the section plane, vertical, and up.

⇒ The Z-axis is normal to the section plane and pointing towards you (out of the screen).

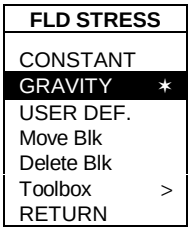
GRAVITY

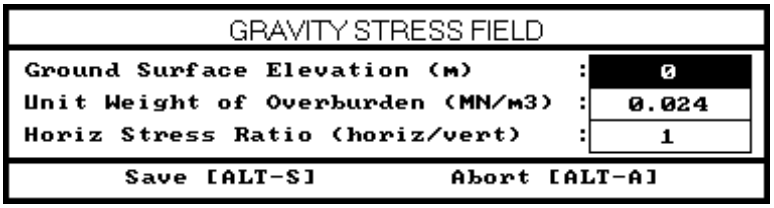
Path: MODEL / FLD STRESS / GRAVITY

The **GRAVITY** Field Stress option is used to simulate gravity field stress conditions which vary *linearly with depth from the surface.*

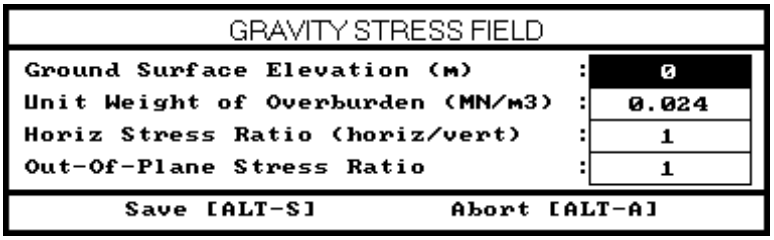
GRAVITY + Plane Strain

The popup windows for entering GRAVITY field stress parameters under Plane Strain conditions, with and without out-of-plane stresses, are shown below:





(i)



(ii)

Figure 2.6d: Popup windows for entering GRAVITY field stress, (Plane Strain conditions), **without** (i) and **with** (ii) out-of-plane stresses considered.

Note:

⇒ the Ground Surface Elevation is the vertical coordinate of the ground surface **with respect to your coordinate system**. It is **not** the depth below ground surface of your excavations!

⇒ Unit Weight of the overburden (note units: MN / m^3)

⇒ the Horizontal Stress Ratio is the in-plane horizontal stress divided by the vertical stress at any location within your model. (The vertical stress is calculated at any point by multiplying the distance from the point to the ground surface by the unit weight. The in-plane horizontal stress is then obtained by multiplying the vertical stress by the in-plane horizontal stress ratio).

GRAVITY + Complete Plane Strain

If you are using GRAVITY field stress with Complete Plane Strain, you will be required to input **two** horizontal stress ratios, K_1 and K_2, and the angle of K_1 from the Z-X plane (Z is normal to the screen, X is the horizontal right-hand axis) as indicated below:

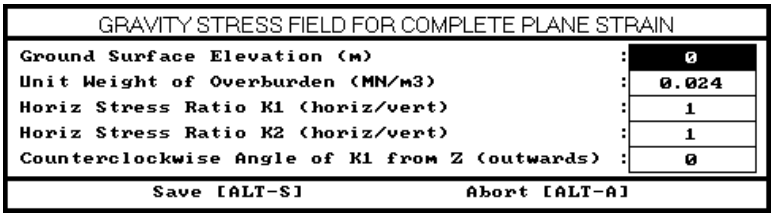
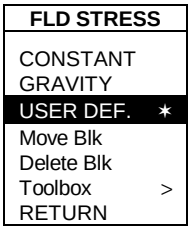


Figure 2.6e: Popup window for entering GRAVITY field stress parameters with Complete Plane Strain conditions.

USER-DEFINED Stress Field

Path: MODEL / FLD STRESS / USER DEF.



You can define a stress field which varies with depth, according to user defined stresses at two or more elevations. **You must enter data for at least two elevations**. You will first be prompted to enter an elevation, which you may key in at the prompt line (see below) or enter graphically with the mouse – a horizontal line will appear on the screen for this purpose.

> ENTER ELEVATION #1 (meters) (ESC to quit) =

After entering an elevation (50 has been entered in this example), you must input a horizontal stress and a vertical stress in the following popup window. (If you are considering out-of-plane stress, this window will have an additional line for entering this value – see the previous popup windows for similar examples).

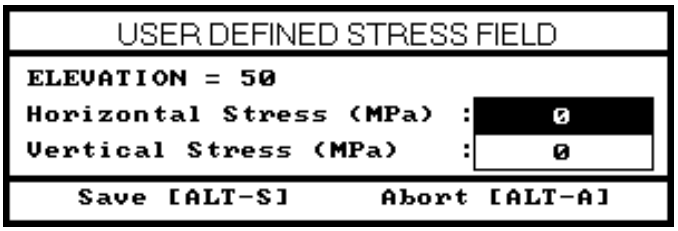


Figure 2.6f: Popup window for entering USER DEFINED stress field.

After entering data for at least two elevations, press *Esc*, and the values you have input will be displayed on the screen at the corresponding elevations.

To modify, reset, or check the values of the user defined stress field, re-select the **USER DEF.** option, and you will see the popup window:

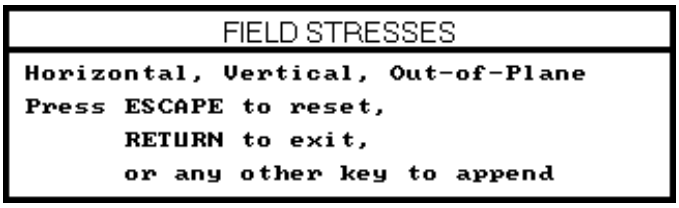


Figure 2.6g: Popup information window for resetting or modifying the user defined stress field.

⇒ *Esc* will reset all of the user defined stress field values

⇒ *Return* allows you to examine the values which you have input, without resetting or modifying them

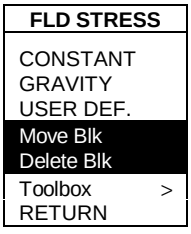
⇒ *Append* will allow you to enter new elevations and stresses to redefine the stress field

Move Blk

Path: MODEL / FLD STRESS / Move Blk



When you have finished defining the field stresses for the problem (CONSTANT or GRAVITY field stress options only, see note below), a small stress block will appear on the screen (see margin figure) indicating the relative magnitude and orientation of the field stresses.



The program places this block in the upper right hand corner of the display, paying no attention to already existing entities. As a result the block might be hard to see, in which case you can use this option to move the block to a less cluttered area of the screen. When you select this option, you will be prompted to select the point where you want the block placed. Simply move the cursor to the point and click the *left mouse button*, and the block will appear in the new location.

Note: the stress block only appears for CONSTANT or GRAVITY stress fields (not USER DEFINED), and also, does NOT appear if you are using Complete Plane Strain.

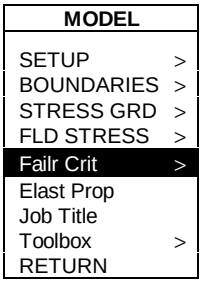
Delete Blk

Path: MODEL / FLD STRESS / Delete Blk

By selecting this option the stress block will be removed from the display.

Failure Criteria

Path: MODEL / Failr Crit



EXAMINE^{2D} can work with three failure criteria, for the purposes of strength factor determination:

⇒ **Hoek-Brown**

⇒ **Modified Hoek-Brown**

⇒ **Mohr-Coulomb**

Note that the parameters used for these criteria should reflect the strength of the **rock mass** and not necessarily the strength of an intact sample.

Since the failure criteria can be altered in **INTERPRET** without reexecuting **COMPUTE**, *it is not critical to define these values when you create the model*. Furthermore, default values will be used for strength factor calculations in **INTERPRET**, if no user values are entered through the **MODEL** option.

In two dimensions, the strength factor is calculated by dividing the rock strength in terms of:

the maximum internal shear $\tau_{max} = \frac{\sigma_{1'} - \sigma_{3'}}{2}$ 1 at failure

for a given confining pressure $\frac{\sigma_{1'} + \sigma_{3'}}{2}$ 2 at a point,

by the maximum induced internal shear at that point due to the excavation.

For background on the derivations and applicability of these failure criteria, see the references at the end of this section.

If out-of-plane stress is being considered, then an analogous three dimensional calculation of strength factor is used, incorporating σ_2 into the values for confining pressure and internal shear.

NOTE: a star (✶) beside a failure criterion option in the Menu Box indicates the current failure criterion which is in effect.

Hoek-Brown Failure Criterion

Path: MODEL / Failr Crit / Hoek-Brown

FAIL CRIT
Hoek-Brown ✶
Modif. H-B
Mohr-Coul
Toolbox >
RETURN

The **Hoek-Brown** failure criterion is an empirical relation between the principal stresses associated with the rock at failure. The relationship requires *three* parameters:

⇒ the **uniaxial compressive strength** of the **intact** rock material

⇒ the constants **m** and **s** which depend upon the properties of the rock mass and upon the extent to which it has been broken before being subjected to the excavation induced stresses.

When you choose this option, enter the parameters in the popup window shown below (default values will initially be displayed).

HOEK-BROWN FAILURE CRITERIA		
Hoek-Brown m value	:	5
Hoek-Brown s value	:	0.001
Intact Rock Compressive Strength	:	80
Save [ALT-S]		Abort [ALT-A]

Figure 2.7a: Popup window for entering Hoek-Brown failure criterion parameters.

Modified Hoek-Brown Failure Criterion

Path: MODEL / Failr Crit / Modif. H-B

FAIL CRIT
Hoek-Brown
Modif. H-B ✶
Mohr-Coul
Toolbox >
RETURN

The Modified Hoek-Brown failure criterion is used in cases where the rock mass is considered to be heavily jointed and the behaviour is not dominated by any one joint set. One of the most significant differences between the original and modified criterion is the reformulation *which eliminates rock mass tensile strength*.

The required parameters are entered in the popup window below, which will initially display default values.

MODIFIED HOEK-BROWN FAILURE CRITERIA		
Hoek-Brown mb value	:	5
Hoek-Brown a value	:	0.5
Intact Rock Compressive Strength	:	80
Save [ALT-S]		Abort [ALT-A]

Figure 2.7b: Popup window for entering Modified Hoek-Brown failure criterion parameters.

Mohr-Coulomb Failure Criterion

FAIL CRIT
Hoek-Brown
Modif. H-B
Mohr-Coul ✶
Toolbox >
RETURN

Path: MODEL / Failr Crit / Mohr-Coul

The other failure criterion which you can use is the Mohr-Coulomb failure criterion. Use this option if your strength data are in terms of the **Mohr-Coulomb** strength parameters:

⇒ **tensile strength (σ_T)** of the **rock mass**,

⇒ **rock mass cohesion intercept (c_m)** value, and

⇒ **rock mass friction angle (ϕ)**.

Enter the parameters in the popup window shown below, which appears when you select this option.

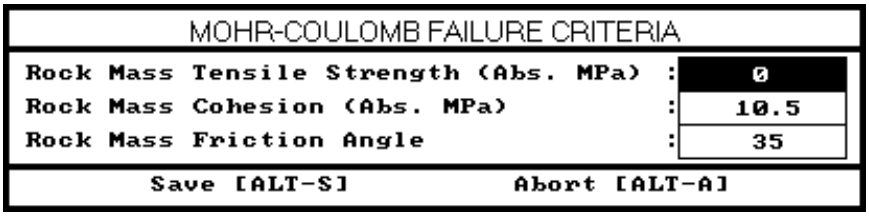


Figure 2.7c: Popup window for entering Mohr-Coulomb failure criterion parameters.

NOTE: For the Mohr-Coulomb criterion, the rock mass cohesion intercept (c_m) is used in versions 3.0+ instead of uniaxial compressive strength (UCS) for the rock mass as in previous versions. Cohesion appears in EXAMINE input and output. UCS is still the value written to the data file to ensure readability of older data files. Also note that the tensile strength can be input as a positive or negative number. It always refers, however, to a negative stress cutoff.

Further Reading – Failure Criteria

Consult the following references for detailed information:

Hoek, E. and Brown, E.T. (1988), *The Hoek-Brown failure criterion – a 1988 update*, Proc. 15th Canadian rock mechanics symposium (ed. Curran), Toronto: Dept. Civ. Engineering, University of Toronto, 31-38.

Hoek, E. (1990), *Estimating Mohr-Coulomb friction and cohesion values from the Hoek-Brown failure criterion*, Intl. J. Rock Mech. & Mining Sci. & Geomechanics Abstracts, 27(3), 227-229.

Hoek, E., Wood, D. and Shah, S. (1992) *A modified Hoek-Brown criterion for jointed rock masses*. Proc. Rock Characterization, Symp. Int. Soc. Rock Mech., Eurock '92, (ed. Hudson). London: Brit. Geol. Soc., 209-214.

Hoek, E. (1994), *Strength of rock and rock masses*, ISRM News Journal, 2(2), 4-16.

Elastic Properties

Path: MODEL / Elast Prop

MODEL
SETUP >
BOUNDARIES >
STRESS GRD >
FLD STRESS >
Failr Crit >
Elast Prop
Job Title
Toolbox >
RETURN

Before you run **COMPUTE**, you should enter **Elastic Properties** for your model – if you do not, EXAMINE^{2D} will run the analysis using default values.

Depending on whether you are dealing with a:

⇒ Isotropic Continuum *or*

⇒ Transversely Isotropic Continuum

(see **Analysis Options**) you will be required to enter different elastic constants, as described below.

*If you do not enter **Elastic Properties** for your model, default values will automatically be used in the analysis.*

Isotropic Continuum

If you are working with the Isotropic Continuum option (see MODEL->SETUP->Analysis Options), only **two** elastic constants are required, as shown in the popup window below:

⇒ **Young's Modulus**

⇒ **Poisson's Ratio** (must be between 0 and 0.5)

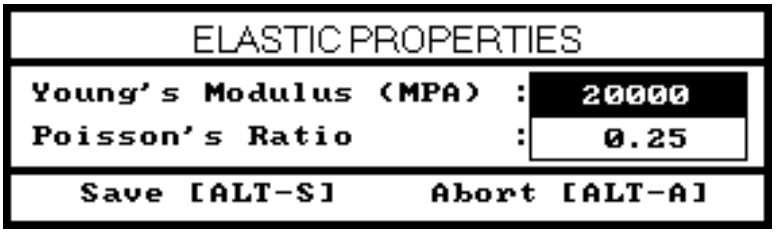


Figure 2.8a: Popup window for entering Elastic Properties for an Isotropic Continuum.

It is important to note that **Young's Modulus** is used only to calculate displacements. *Since elastic displacements normally have little meaning in a complex rock mass, except to give a relative sense of deformation, it is rarely necessary to know this value exactly.* EXAMINE^{2D} will save and compute a file without this parameter being explicitly entered – the default of 20000 will be used.

Poisson's Ratio has a moderate effect on stress calculations. The default value of .25 is adequate for most hard rocks. If you choose to consider out-of-plane stresses in your analysis (see the section on FIELD STRESS earlier in this chapter), you must however select a reasonable value for Poisson's Ratio since the out-of-plane stress change is directly related to this value.

Transversely Isotropic Continuum

If you are working with the Transversely Isotropic Continuum option (see MODEL->SETUP->Analysis Options), a number of elastic constants are required, as indicated in the popup window below. If your model has different elastic properties in two "in-plane" directions, you should use this option.

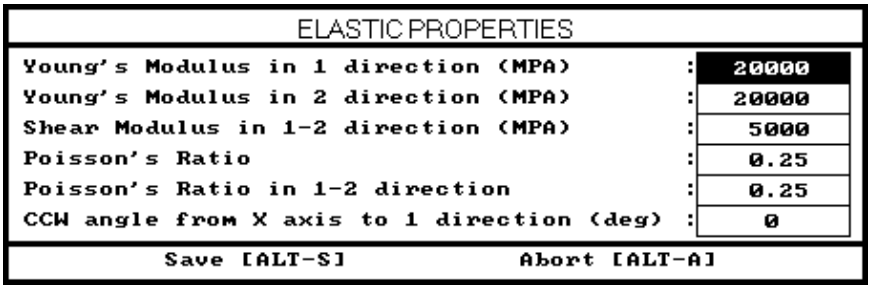


Figure 2.8b: Popup window for entering Elastic Properties for a Transversely Isotropic Continuum.

You may enter the following parameters (refer to the figure below for illustration of the various directions):

⇒ **Young's Modulus** in each of the 1- and 2- directions

⇒ **Shear Modulus** between the two directions

⇒ **Poisson's Ratio** giving the effect of an applied stress in the 1-direction on the 3-direction (the 3-direction is orthogonal to the screen and pointing towards you), and a **Poisson's ratio** giving the effect of an applied stress in the 1-direction on the 2-direction (Note: Poisson's Ratio must be between 0 and 0.5)

⇒ an **Angle** between the 1- direction and the positive X-axis.

⇒ Note: the out-of-plane elastic modulus is equal to the modulus in the 1-direction.

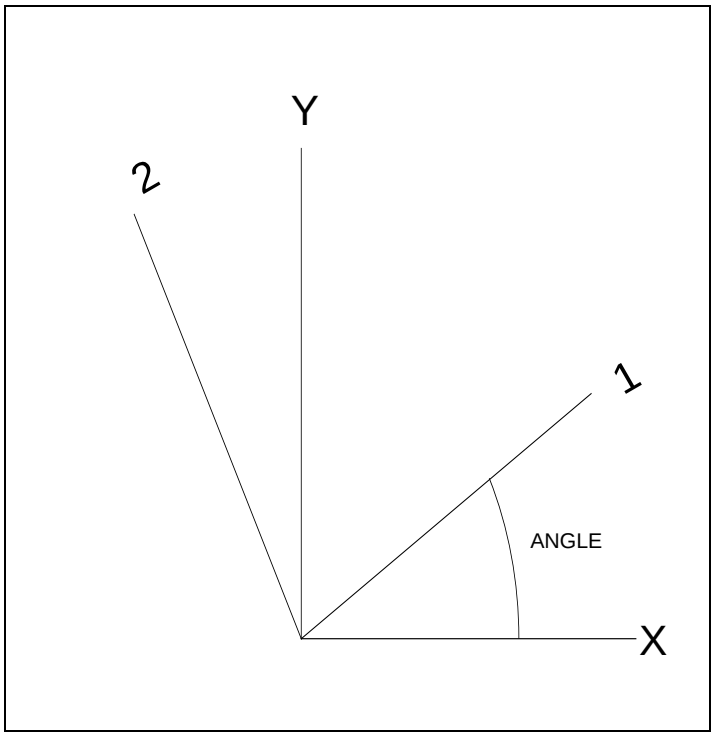


Figure 2.8c: Directions for Transversely Isotropic properties.

Job Title

Path: MODEL / Job Title

MODEL	
SETUP	>
BOUNDARIES	>
STRESS GRD	>
FLD STRESS	>
Failr Crit	>
Elast Prop	
Job Title	
Toolbox	>
RETURN	

To assign a Job Title to a model, select **Job Title** from the **MODEL** menu, type in a job description at the prompt line, and press *Enter*.

> ENTER JOB DESCRIPTION:

After the model has been analyzed and the user enters the **INTERPRET** menu, this job title will appear in the Upper Status Box at the top of the screen (when it is not being used to display cursor information, etc), and will also be written to the output file. A job title is useful whenever the screen is being captured for a printout.

Note that the Job Title may also be changed within the **INTERPRET** menu, using the **Job Title** option within **Display Tools**.

Toolbox

Path: MODEL / ... / Toolbox

MODEL
SETUP >
BOUNDARIES >
STRESS GRD >
FLD STRESS >
Failr Crit >
Elast Prop
Job Title
Toolbox >
RETURN

The **Toolbox** is present in all sub-menus under **MODEL**, including MODEL itself. This option contains a variety of useful commands for altering the display. These are described below.

⇒ Note: NONE OF THE OPTIONS DESCRIBED BELOW AFFECT THE MODEL COORDINATES, ONLY THE VIEWING REGION GETS SHIFTED.

Autoscale

TOOLBOX
Autoscale
Zoom Win
Zoom Out
Zoom In
Pan
Snap Set
Grid Set
Input Data
F-Key Help
RETURN

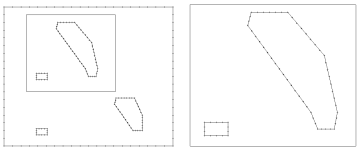
This option resets the viewport so that every entity which has been drawn will be scaled to fit on the display. In other words, all boundaries and grids will be scaled such that you will see them in the drawing region. **Autoscale** is useful for cancelling the effects of the **Zoom** and **Pan** options to quickly restore the screen appearance.

Zoom Win

This option allows you to select and zoom in on a rectangular subregion of the display using a windowing system. You will be prompted to select a window corner:

```
> ENTER WINDOW CORNER 1:
```

Move the cursor using the keyboard arrow keys or the mouse and select a corner of the subregion that you wish to zoom in on; now move the cursor and you will notice a window opening up. Select the opposite corner of the subregion.

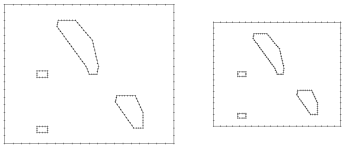


```
> TO WINDOW CORNER 2:
```

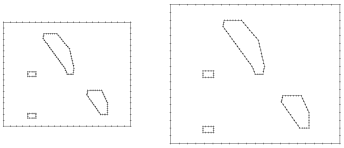
The display will now be updated, scaling the subregion that you have just selected to fit in the drawing region. Note that none of the entities, i.e., boundaries, have moved from their original location.

Zoom Out

Select **Zoom Out** and the apparent size of the model will be scaled down by 75%. Zoom Out can be selected as many times as desired to "shrink" the model.



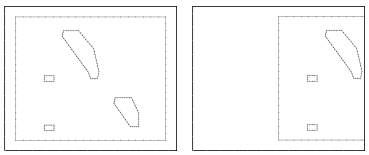
Zoom In



This is the inverse of the Zoom Out option. Each **Zoom In** will scale the apparent size of the model up by 125%. Zoom In can be selected as many times as desired to magnify the model.

To magnify a specific area of interest, use the **Zoom Win** option rather than **Zoom In** (see above).

Pan



This option allows you to pan the display around the drawing, i.e., shift the current viewport with respect to the entities. Select **Pan** and you will be first prompted to enter a base point – ie. a point from which to base the translation.

```
> ENTER BASE POINT:
```

Select a point on the display, then move the cursor to a second location representing the distance and direction of the pan, and select.

```
> TO POINT:
```

The display will be shifted such that the base point is moved to the second point you have just selected. Note that this option does not move any of the boundaries or grids from their original coordinates but rather moves the display region relative to these entities.

TOOLBOX

Autoscale
Zoom Win
Zoom Out
Zoom In
Pan
Snap Set
Grid Set
Input Data
F-Key Help
RETURN

Snap Set

This option allows you to change the current snap setting. The snap setting is the increment that the cursor will move across the screen if the snap is toggled ON using the F9 function key. The prompt will read:

```
> INPUT NEW SNAP DIMENSION <5>:
```

Enter a new number, or hit *Enter* to accept the default value.

Grid Set

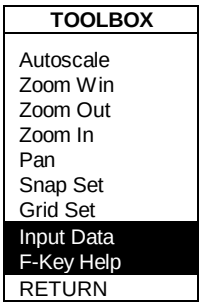
This option allows you to change the current grid setting. The grid setting is the distance between grid points when you turn the grid ON using the F7 keyboard function key.

```
> INPUT NEW GRID DIMENSION <5>:
```

Enter a new number, or hit *Enter* to accept the default value.

*See the BOUNDARIES-> ADD EXCAVATION section for more info on use of the **Grid** and **Snap** options.*

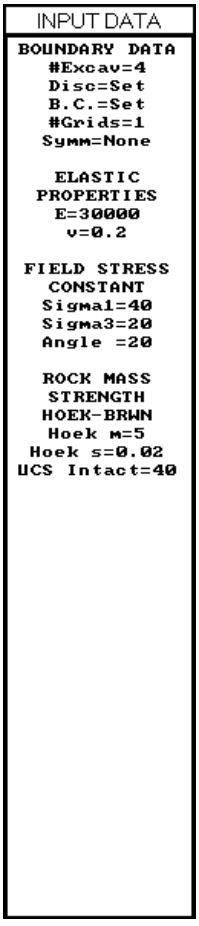
Input Data



To obtain a summary of important model input parameters (eg. boundary, strength, stress and elastic parameters), select **Input Data** from the **Toolbox**. The Menu Box area at the right of the screen will be replaced with the Input Data information – see the margin figure for an example.

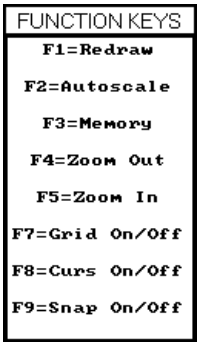
This option is useful for quickly checking program settings, and also to display for printout purposes.

To return to the **Toolbox** menu, *hit any key*. If the information fills more than one Menu Box, you will see -- **more** -- at the bottom of the display – *hit any key* to page down through the listing.



F-Key Help

This option will list the active Function Keys in the Menu Box area at the right of the screen, as illustrated at right.



The F-Keys are active any time you are *not* entering geometry, or data in the prompt line or popup windows.

For more information on the use of the eight active function keys in the **MODEL** portion of EXAMINE^{2D}, see the Introductory chapter in Part 1 – Getting Started.

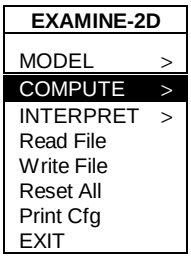
To return to the **Toolbox** menu, *hit any key*.

COMPUTE

Path: COMPUTE

COMPUTE is the 2-D boundary element analysis engine. It reads a data file generated by **MODEL** and produces a data file which can be displayed by the data interpretation module **INTERPRET**.

Running COMPUTE



COMPUTE requires an input file in a specific format compatible with the data file written by EXAMINE^{2D}'s **Write File** option. This is an ASCII file which can be edited with an ASCII DOS text editor. If you have not already done so, take a look at the example data file "example.exa" supplied with the EXAMINE^{2D} package.

To run COMPUTE select the **COMPUTE** button in the EXAMINE^{2D} main menu. If you have not already saved the current file, you will first be prompted to save the file, in the same manner as when using the **Write File** option. When the file is successfully written, you will see the prompt:



The screen will go blank for a few seconds and the COMPUTE screen will appear – see Figure 3.1. Do not touch your keyboard while COMPUTE is operating – when the problem is solved, the screen will automatically return to the EXAMINE^{2D} main menu.

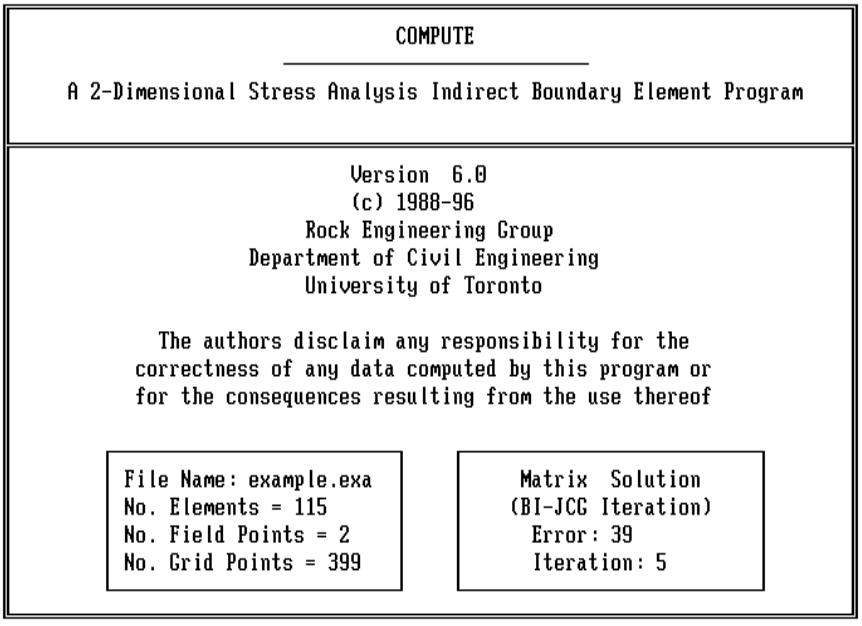


Figure 3.1: Running COMPUTE using “example” data.

To run COMPUTE on its own in batch mode, type *compute* at the DOS prompt. The COMPUTE screen will appear (Figure 3.1). If you have not entered a filename at the command line (ie. *compute filename*), you will be prompted to enter a filename. If the file has an .**exa** extension, you do not have to type the extension. If you have used **MODEL** to generate the data file just enter the filename that you entered using the **Write File** option. The program will then proceed with the analysis.

When running COMPUTE on its own (not from within EXAMINE^{2D}) it will return to the DOS prompt and not to EXAMINE^{2D} when complete.

It is important that you have all of your EXAMINE^{2D} files including **examine.exe** and **compute.exe** in the same directory. If this is not the case, the relevant directories must be included in your "PATH". See your DOS manual for instructions on how to change the DOS environment PATH variable.

Output File

Output from COMPUTE is not stored in a separate output file – **the analysis results are appended to the original data file**.

*Output from **COMPUTE** is not stored in a separate output file – **the analysis results are appended to the original data file**.*

The output is in the form of stresses and factors of safety (overstress ratio) at each of the specified grid points. Since the original input data is preserved you may run COMPUTE with a data file that has already been solved. This makes editing and re-running an analysis very easy.

If you make changes within **MODEL**, and then go directly to **INTERPRET** without first re-running **COMPUTE**, you will receive a warning prompt:

```
> Cannot Interpret Current Data File
```

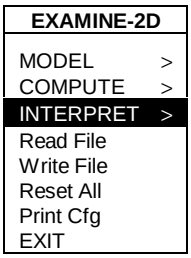
indicating that you must re-run COMPUTE. This ensures that you cannot confuse the results of an old analysis with new input parameters.

You must be careful, however, if you edit a previously analysed data file, directly with a text editor. In this case, EXAMINE^{2D} has no way of checking that your input and output data are consistent. If you do not re-run the analysis then the edited changes will show up in **INTERPRET** but the stresses and factors of safety will be a result of the old analysis. In other words **INTERPRET** will display results inconsistent with the input data. Therefore, if you are carrying out a parametric analysis, it is always best to edit your input data through the **MODEL** option, rather than editing the file directly.

Interpreting the Analysis

Path: INTERPRET

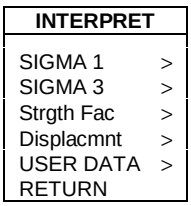
INTERPRET is the post-processing module used for data visualization and interpretation. *You must first perform a boundary element analysis on your model using **COMPUTE** before you can enter the **INTERPRET** menu.*



When you select INTERPRET from the EXAMINE^{2D} main menu, the prompt line will display:

> Reading Data – Please Wait...

If your computed data file was successfully read, the **INTERPRET** menu will appear, which will allow you to create plots of:



⇒ **SIGMA 1** (major principal stress)

⇒ **SIGMA 2** (intermediate principal stress – this option will only appear if you are considering OUT-OF-PLANE stresses)

⇒ **SIGMA 3** (minor principal stress)

⇒ **STRENGTH FACTOR** (based on your failure criterion)

⇒ **DISPLACEMENT** (this will NOT be available if you are using the Complete Plane Strain option, as COMPUTE does not calculate displacements under Complete Plane Strain)

⇒ **USER DEFINED** functions of stress, displacement. (this is also optional, and will only appear if it has been toggled ON in your e2.cfg file – see later in this chapter for more information).

If you see the following message in the prompt line when you select **INTERPRET**:

> Cannot Interpret Current Data File

it is because:

⇒ no file has been loaded, *or*

⇒ no analysis has been performed, *or*

⇒ you have made changes to your input data using MODEL, without first re-running COMPUTE (see previous page), *or*

⇒ the results (output) portion of your file has been damaged, or is invalid for some reason.

SIGMA 1

Path: INTERPRET / SIGMA 1

INTERPRET	
SIGMA 1	>
SIGMA 3	>
Strgth Fac	>
Displacmnt	>
USER DATA	>
RETURN	

Choosing **SIGMA 1** will plot contours of the **major principal stress** around the excavation(s). If out-of-plane stress is **not** being considered, this will be the in-plane major principal stress.

If out-of-plane stress is being considered, however, the **SIGMA 1** plot will display the value of σ_1 at every point *regardless of its orientation in 3 dimensions.*

SIGMA 1	
Autoscale	
Zoom Win	
Zoom Out	
Input Data	
Alter Rnge	
Stres Traj	
Query Data	>
Disp Tools	>
Print Cfg	
RETURN	

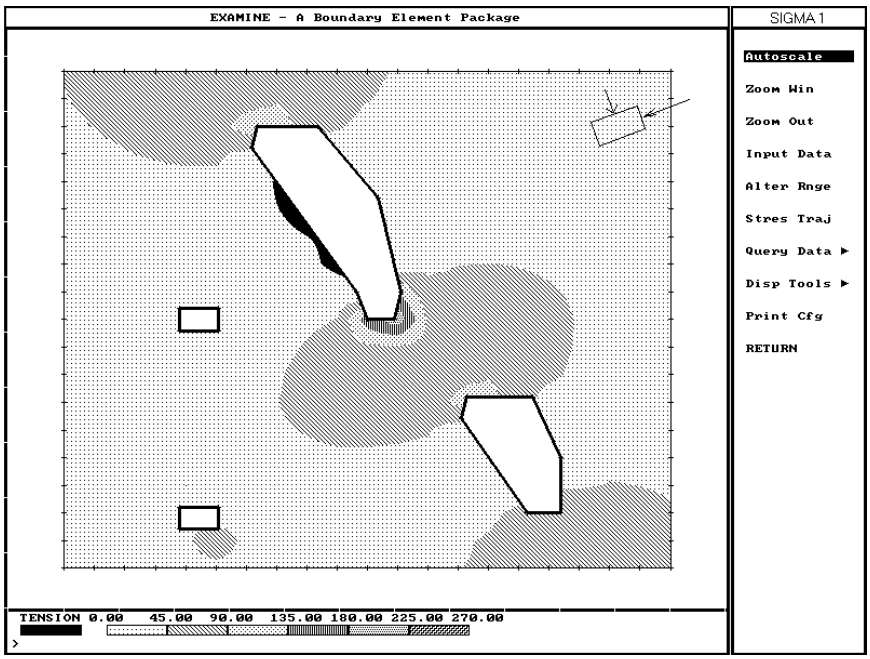


Figure 4.1: **SIGMA 1** – NOTICE the lack of detail immediately surrounding the two small haulage drifts. The grid is too coarse for these boundaries. If more local detail is needed, additional fine grids can be overlain around these openings

SIGMA 2

Path: INTERPRET / SIGMA 2

*The **SIGMA 2** stress option will only appear in the **INTERPRET** menu if you are considering OUT-OF-PLANE stress.*

If the **SIGMA 2** option is available, selecting it will plot contours of the **intermediate principal stress** around the excavation(s). If out-of-plane stress is not being considered, **this option will not be available**. If out-of-plane stress is being considered, however, this plot will display the value of σ_2 at every point *regardless of its orientation in 3-dimensional space*. Since EXAMINE^{2D} performs a plane strain analysis, and since any in-situ out-of-plane principal stress is aligned with the longitudinal axis of the excavation, any principal stress which does not fall in the plane of the analysis must naturally be orientated perpendicular to it.

SIGMA 3

Path: INTERPRET / SIGMA 3

INTERPRET	
SIGMA 1	>
SIGMA 3	>
Strgth Fac	>
Displacmnt	>
USER DATA	>
RETURN	

Choosing **SIGMA 3** will plot contours of the **minor principal stress** around the excavation(s). If out-of-plane stress is not being considered, this will be the in-plane minor principal stress. If out-of-plane stress is being considered, however, this plot will display the value of σ_3 at every point regardless of its orientation in 3-dimensional space.

SIGMA 3	
Autoscale	
Zoom Win	
Zoom Out	
Input Data	
Alter Rnge	
Stres Traj	
Query Data	>
Disp Tools	>
Print Cfg	
RETURN	

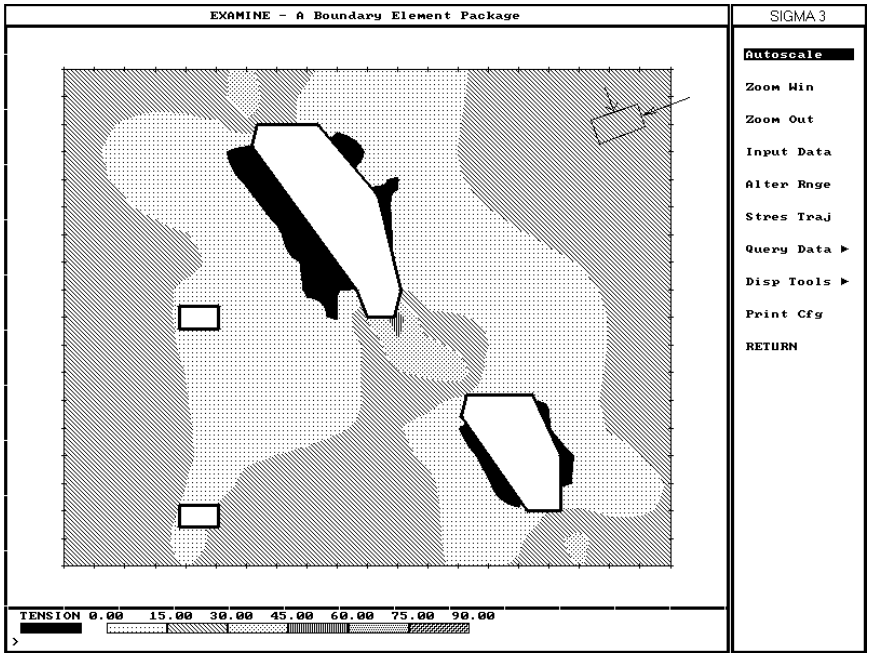


Figure 4.2: **SIGMA 3** – NOTICE that any point with SIGMA 3 (and possibly SIGMA 1) less than zero is shown as "TENSION"

Strength Factor

Path: INTERPRET / Strgth Fac

INTERPRET
SIGMA 1 >
SIGMA 3 >
Strgth Fac >
Displacmnt >
USER DATA >
RETURN

The **Strngth Factr** option will plot contours of strength factor around the excavation(s). In two dimensions the strength factor is calculated by dividing the rock strength in terms of the maximum internal shear $\frac{\sigma_{1'} - \sigma_{3'}}{2}$ 3 at failure for a given confining pressure $\frac{\sigma_{1'} + \sigma_{3'}}{2}$ 4 at a point, by the maximum induced internal shear at that point due to the excavation.

STRGTH FACT
Autoscale
Zoom Win
Zoom Out
Input Data
Alter Rnge
Stres Traj
Query Data >
Disp Tools >
Failr Crit >
Failr Traj >
Print Cfg
RETURN

Please note that this definition of **strength factor** is different from the definition of **factor of safety** used in versions prior to 3.1. Previously the factor of safety was a ratio of major principal stress, σ_1, at failure (strength) and actual induced σ_1 for a given σ_3. The new definition gives a more logical picture of overstress around an excavation. You will notice, however, that the factor of safety = 1 and the (2-D) strength factor = 1 contour lines are identical. If out-of-plane stress is being considered, the strength factors may differ from the purely 2-D case since σ_2 now has an influence on the calculation of strength factor, and σ_1 or σ_3 may actually be perpendicular to the analysis plane.

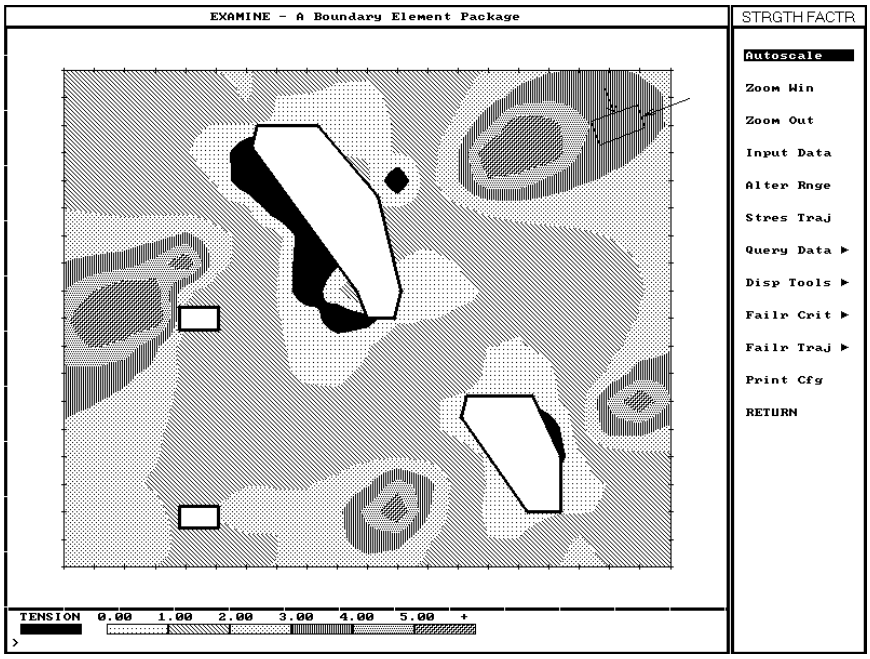


Figure 4.3: **STRENGTH FACTOR** – NOTICE that in this plot, **S.F. < 1** indicates overstress ("shear failure"), while "**TENSION**" means that SIGMA 3 is less than the calculated negative stress (tension) cutoff for the failure criterion in use.

Displacements

Path: INTERPRET / Displacmnt

INTERPRET
SIGMA 1 >
SIGMA 3 >
Strgth Fac >
Displacmnt >
USER DATA >
RETURN

This option allows you to view **vectors** or **contours** illustrating absolute magnitudes of elastic displacement. You can *contour* the displacement in:

⇒ the **X-direction**

⇒ the **Y-direction**

⇒ the **TOTAL** Absolute Displacement ($\sqrt{X^2 + Y^2}$ 5).

DISPLCMNTS
X – Abs Disp
Y – Abs Disp
Tot Abs Disp
RETURN

The same submenu appears for each option. To view displacement vectors, select **Displ Traj** from any of the three **Displacement** menus, to toggle the display of vectors ON.

It is important to realize that the displacements depend on the Young's Modulus of the rock mass which is rarely known with any certainty. The assumption of elastic displacements is also very limiting. The relative vectors, however, give a useful picture of basic displacement trends – *see Figure 4.6 for an illustration*.

X DISPLACE
Autoscale
Zoom Win
Zoom Out
Input Data
Alter Rnge
Displ Traj
Query Data >
Disp Tools >
Print Cfg
RETURN

NOTE: Since **COMPUTE** does not calculate displacements under the Complete Plane Strain condition (see the MODEL chapter for details), you will not see the **Displacements** option in the INTERPRET menu, if you work with Complete Plane Strain.

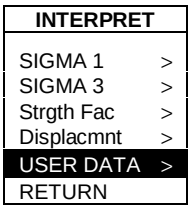
NETDISP Utility

In a boundary element model, displacements are cumulative for the entire problem. This means that if your problem includes a surface or an old excavation, the displacements measured may include tectonic uplift or old displacements respectively.

A utility called **NETDISP** is included with this release to "zero" the displacements with respect to a data file. See the end of Part 2 – Reference Manual for details.

USER DATA

Path: INTERPRET / USER DATA

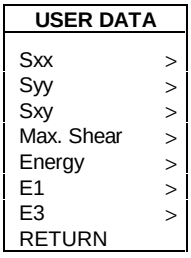


With the **USER DATA** option, the user is no longer limited to viewing principal stresses and displacements only. With this option, the user can create contour plots of:

⇒ **any stress tensor component**

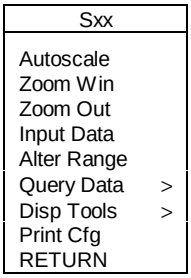
⇒ **any displacement component**

⇒ **write a mathematical expression containing these components, and view the results of this function**



The **USER DATA** option is customized through the EXAMINE^{2D} configuration file **e2.cfg**, which is supplied for this purpose.

You will find the e2.cfg file in your installation directory. Take a moment to view this file using any standard ASCII text editor. You will find that it already contains information which will create a customized user menu, as shown at left. Read on for further information.



USER DATA Configuration File – e2.cfg

Note: the information in this section is also available as comments in the e2.cfg file itself

The format of the e2.cfg file is simple. If there is an asterisk (*) in the first column of a line, then the line is a **comment**. Otherwise the line contains configuration **information**.

The syntax of a configuration **information** line is of the format:

```
<config_option> = <value>
```

It is recommended that you look at the

e2.cfg

file to examine the file format first hand.

The first non-comment line in the e2.cfg file turns the **USER DATA** option ON or OFF (when you install EXAMINE^{2D}, the default is OFF). To activate the USER DATA option, edit this line to read:

```
user_data_menu=ON
```

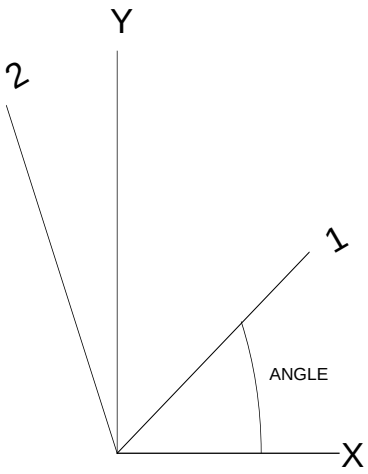
The second non-comment line can be used to change the *title* of the USER DATA menu (ie. it does not have to be called USER DATA).

```
user_data_menu_title="USER DATA"
```

User Defined Functions

The following is a list of **variables** available to the user:

s1 = major principal stress
s2 = intermediate principal stress
s3 = minor principal stress
sxx = stress in x direction (horizontal)
syy = stress in y direction (vertical)
sxy = shear stress in xy direction
dx = displacement in x direction
dy = displacement in y direction
ym = Young's modulus
g = shear modulus = ym / (2*(1+pr))
pr = Poisson ratio
sf = strength factor
pi = 3.1415926



If you are using the transversely isotropic material model, the following variables are also defined (see the margin figure for definition of the transversely isotropic directions):

e1 = elastic modulus in 1 direction & out of plane
e2 = elastic modulus in 2 direction
g = shear modulus
pr = Poisson ratio
pr12 = Poisson ratio in 1-2 direction
oang = angle between x direction and 1 direction, measured counter-clockwise

The following is a list of the basic mathematical **operators** supported:

+ = addition
- = subtraction
/ = division
*** = multiplication**
^ = to the power of (ie. 2^3 = 8)

The following **intrinsic functions** are available for the calculation of trigonometric and other quantities:

acos(x) = arc cosine of expression x
asin(x) = arc sine of expression x
atan(x) = arc tangent of expression x
cosh(x) = hyperbolic cosine of expression x
sinh(x) = hyperbolic sine of expression x
tanh(x) = hyperbolic tangent of expression x
cos(x) = cosine of expression x
sin(x) = sine of expression x
tan(x) = tangent of expression x

log10(x) = log base 10 of expression x
exp(x) = exponential e to the x
log(x) = natural logarithm of expression x
sqrt(x) = square root of expression x
abs(x) = integer absolute of expression x
fabs(x) = floating point absolute of expression x

Note: all angles must be expressed in radians

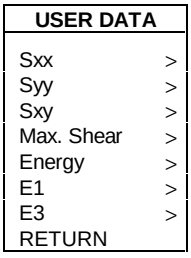
The user is allowed to specify a maximum of 12 functions, using the above **variables**, **operators** and **intrinsic functions**. Each function will be represented by a "button" in the USER DATA menu – 7 functions have been defined in the e2.cfg file supplied with EXAMINE^{2D}.

To define a function, simply define the "button label" and then define the function, in the following format:

```
option_?_title = "<button label>"

option_? = <mathematical expression>
```

where the ? = the number of the button from 1–12.



For example, the USER DATA functions supplied in your e2.cfg file when you installed EXAMINE^{2D} , were created by including the following lines in the e2.cfg file:

```
option_1_title = "Sxx"
option_1 = sxx
option_2_title = "Syy"
option_2 = syy
option_3_title = "Sxy"
option_3 = sxy
* maximum shear stress
option_4_title = "Max. Shear"
option_4 = (s1-s3)/2.0
option_5_title = "Energy"
option_5 = (s1^2 + s2^2 + s3^2 - 2.0*pr*(s1*s2 +
           s2*s3 + s1*s3))/(2.0*ym)
*major principal strain
option_6_title = "E1"
option_6 = (((1.0-pr^2)*s1)-(pr*(1.0+pr)*s3))/ym
*minor principal strain
option_7_title = "E3"
option_7 = (((1.0-pr^2)*s3)-(pr*(1.0+pr)*s1))/ym
```

NOTE:

⇒ the lines preceded by an **asterisk** (*) are comment lines

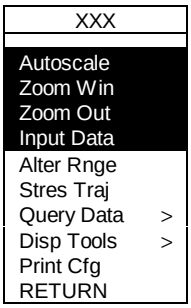
⇒ the "**button labels**" must be enclosed by quotation marks, as shown

⇒ the **functions** are NOT enclosed by quotation marks

Secondary INTERPRET Options

Most of the data interpretation options in EXAMINE[2D] are common to a number of **INTERPRET** sub-menus. These are described in the following sections without necessarily referring to specific menus.

Autoscale



Path: INTERPRET / ... / Autoscale

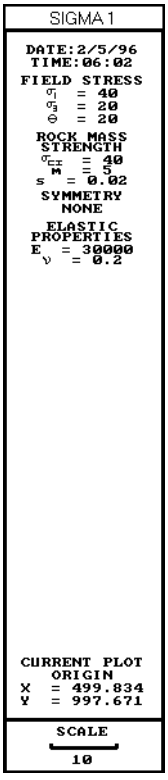
This is the same **Autoscale** option available in the MODEL->Toolbox. See that section for details.

Zoom Win & Zoom Out

Path: INTERPRET / ... / Zoom Win **and** Zoom Out

These options allow you to zoom in and out on any region of the display. These are the same options available in the MODEL->Toolbox. Read that section for details.

Input Data

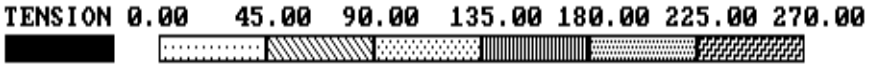


Path: INTERPRET / ... / Input Data

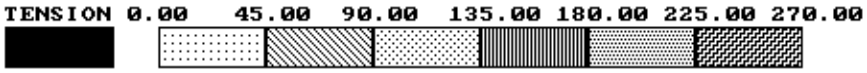
This displays a summary of input data relevant to the analysis (eg. field stress, strength and elastic parameters, etc) in the Menu Box area at the right of the screen, as shown in the margin.

Note:

- selecting Input Data from the INTERPRET menu also widens the bar legend at the bottom of the screen (see below), which is useful for display and printout purposes.



BEFORE SELECTING INPUT DATA



AFTER SELECTING INPUT DATA

Hit any key, to return from the Input Data display mode to the menu from which it was selected.

Alter Range

Path: INTERPRET / ... / Alter Rnge

XXX
Autoscale
Zoom Win
Zoom Out
Input Data
Alter Rnge
Stres Traj
Query Data >
Disp Tools >
Print Cfg
RETURN

This option lets you change the range over which **stresses** (SIGMA menus), **overstress ratios** (STRENGTH FACTOR menu) or **displacements** (DISPLACEMENT menus) are interpolated and contoured. This option is very useful, and allows the user to focus on any stress, strength factor or displacement range of interest.

Select this option and you will be prompted to enter a minimum and maximum range for the display in question (the current values will be displayed in the prompt line):

> Enter Minimum, Maximum Range Values < ## , ## > :

Type in lower and upper limits of a range of interest and press *Enter*, and the contour mapping will be regenerated with the new range which you have input. (Pressing *Enter* without entering new values, or pressing *Esc*, will return you to the menu without altering the range).

The **colours** of the contour intervals can also be changed, but this is done through the **Alter Color** option of **Disp Tools** – see the end of this chapter for details.

Note that changing the range of a contour plot can completely change the appearance of the plot, and this should always be kept in mind. The user should always examine the plot carefully, and compare with the bar legend at the bottom of the screen. The extent of 'critical' regions could appear very different, if the **range** (or **colours**) have been changed from their default settings).

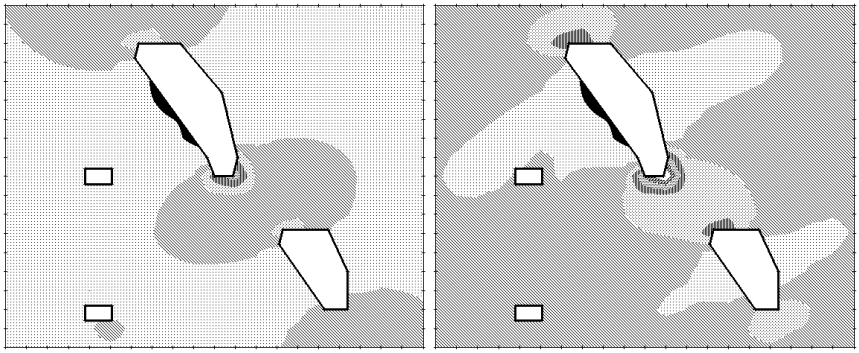


Figure 4.4: Altering the range of a contour plot can completely change its appearance – this should always be kept in mind when interpreting results. Shown here are SIGMA 1 plots for the **example** data file. At left, range = < 0 , 270 > . At right, range = < 0 , 180 >.

Plotting Trajectories

EXAMINE[2D] allows you to visualize:

⇒ **stress** trajectories (SIGMA and STRENGTH FACTOR menus)

⇒ **displacement** trajectories (DISPLACEMENT menus)

⇒ **failure** trajectories (STRENGTH FACTOR menu)

along with the contour plots. *However, note that if you are using the Complete Plane Strain option (see the MODEL chapter), 2D trajectory plotting will not be available.*

With Complete Plane Strain, trajectory plotting is not available.

Stress Trajectories

Path: INTERPRET / ... / Stres Traj

XXX
Autoscale
Zoom Win
Zoom Out
Input Data
Alter Rnge
Stres Traj
Query Data >
Disp Tools >
Print Cfg
RETURN

The **Stres Traj** option is available in the SIGMA menus as well as the STRENGTH FACTOR menu, and displays the principal **stress trajectories**, in the form of a *long line* for σ_1 and a *short line* for σ_3. The trajectories are placed and centred at every grid point, and overlie the current plot, as illustrated below.

If OUT-OF-PLANE is being considered and either the σ_1 or σ_3 line does not appear at a particular grid point, then the corresponding principal stress is perpendicular to the analysis plane at that point.

Select:

Stres Traj

in the
SIGMA
or
STRENGTH FACTOR
menus to display STRESS trajectories.

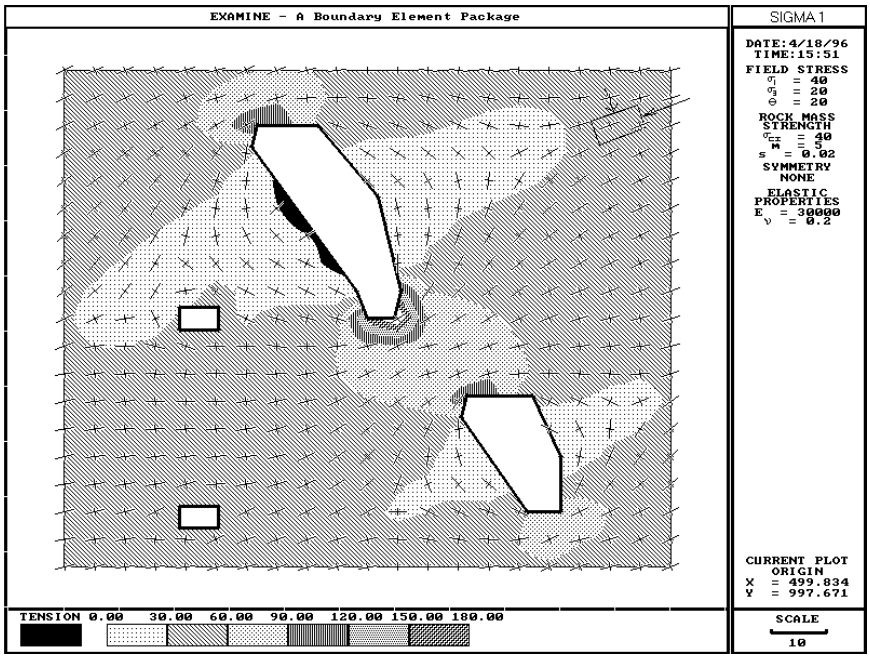


Figure 4.5: Screen display with both **Input Data** and STRESS TRAJECTORIES displayed. Notice that the *long* and *short* trajectory lines represent the directions of σ_1 and σ_3 respectively.

Displacement Trajectories

Path: INTERPRET / ... / Displ Traj

If you select **Displ Traj** from the **Displacement** menus, you will be able to view the displacement vectors, indicating the general trends in the absolute magnitude of elastic displacements.

XXX
Autoscale
Zoom Win
Zoom Out
Input Data
Alter Rnge
Displ Traj
Query Data >
Disp Tools >
Print Cfg
RETURN

A prompt will first appear, allowing you to scale the relative size (on the screen) of the displacement vectors:

> Enter scale factor as % of grid spacing [##] :

Simply press *Enter* at this prompt to accept the default value which is indicated, or type in a new value.

It is important to realize that:

⇒ the displacements depend on the Young's Modulus of the rock mass which is rarely known with any certainty.

⇒ the assumption of **elastic** displacements is also very limiting – local **inelastic** displacements may, in reality, be orders of magnitude greater, particularly in broken rock.

The relative vectors, however, give a useful picture of basic displacement trends, as shown in the figure below.

Select:

Displ Traj

in the ***DISPLACEMENT*** *menus to plot DISPLACEMENT trajectories.*

Note: when either Stress or Displacement Trajectories are toggled ON, a star (✶) will appear beside the option in the Menu Box. This can be seen in Figure 4.6.

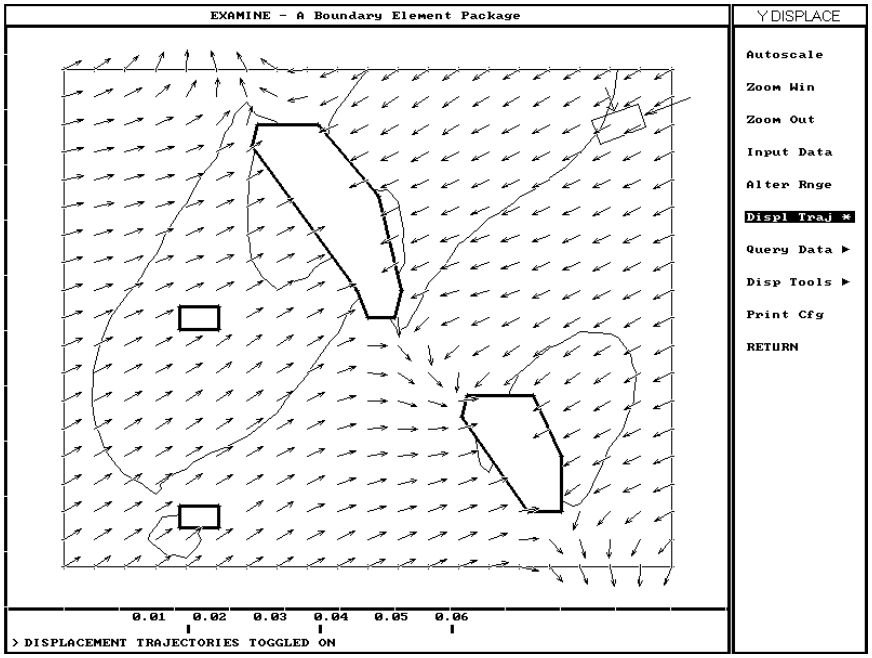


Figure 4.6: DISPLACEMENT VECTORS – NOTE that elastic displacements give only a regional sense of movement. Local inelastic displacements may, in reality, be orders of magnitude greater, particularly in broken rock.

Failure Trajectories

Path: INTERPRET / Strgth Fac / Failr Traj

STRGTH FACT
Autoscale
Zoom Win
Zoom Out
Input Data
Alter Rnge
Stres Traj
Query Data >
Disp Tools >
Failr Crit >
Failr Traj >
Print Cfg
RETURN

This option is only available in the **Strength Factor** menu of INTERPRET, and allows you to plot failure trajectories, both

⇒ **tension** and

⇒ **shear**

within the rock mass. You may overlay the trajectories on the strength factor contour plot, or plot them on their own, as described below.

Overlay Traj

Path: INTERPRET / Strgth Fac / Failr Traj / Overlay Traj

FAILURE TRAJ
Overlay Traj
Traject Plot
Input Data
Scale Traj
DXF Out
RETURN

This option overlays the failure trajectories on the strength factor contour plot. Different markers are used for whether the rock fails in **shear** or in **tension**.

Traject Plot

Path: INTERPRET / Strgth Fac / Failr Traj / Traject Plot

This option displays the failure trajectories only, without the strength factor contours – this is illustrated in the figure below.

Select:

Failr Traj

in the
STRENGTH FACTOR
menu to display
FAILURE trajectories.

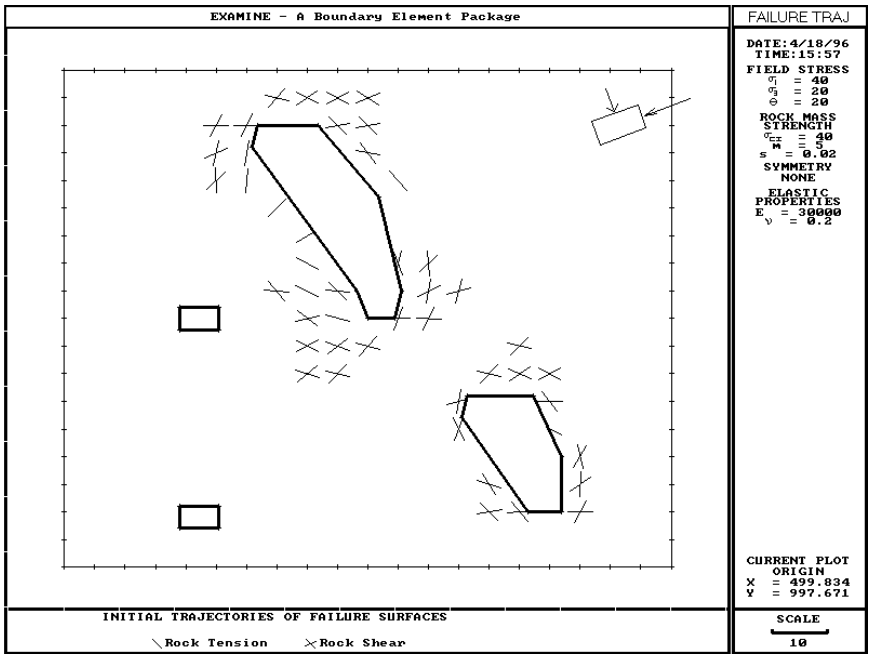


Figure 4.7: Failure trajectories plotted using **Traject Plot** option. Note the different symbols for Tension and Shear Trajectories. **Input Data** is also displayed.

Input Data

Path: INTERPRET / Strgth Fac / Failr Traj / Input Data

This option displays a summary of the input data in the right hand Menu Box area – see the previous figure for illustration. *Hit any key* to return to the **Failr Traj** menu.

Scale Traj

Path: INTERPRET / Strgth Fac / Failr Traj / Scale Traj

FAILURE TRAJ
Overlay Traj
Traject Plot
Input Data
Scale Traj
DXF Out
RETURN

This option allows you to scale the size of the failure trajectory vectors on the screen, if you wish to change the default display. Select this option and you will be prompted:

```
> Enter scale factor as % of grid spacing [ ## ] :
```

The default scaling value is indicated in the prompt line. Type in a new number if you want to change the scaling, or simply press *Enter* to accept the default value.

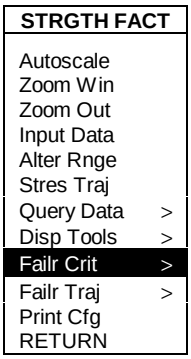
DXF Out

Path: INTERPRET / Strgth Fac / Failr Traj / DXF Out

This option allows you to save the display to a DXF (Drawing Exchange Format) file which can be read into programs such as AutoCAD. See the description in the **Disp Tools** section at the end of this chapter, for details.

Failure Criterion

Path: INTERPRET / Strgth Fac / Failr Crit



You may change the **failure criterion** of the rock *without re-running the analysis* by using the **Failr Crit** option within the INTERPRET->Strgth Factor menu.

As in the MODEL menu, you may choose between:

⇒ **Hoek-Brown**

⇒ **Modified Hoek-Brown**

⇒ **Mohr-Coulomb**

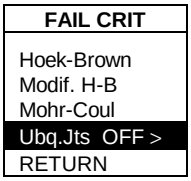
For information on these failure criteria, see the **MODEL** chapter. In addition, you may model:

⇒ **Ubiquitous Joints**

in the Strength Factor calculations. Only this option is described below, as it is only available within **INTERPRET**.

Ubiquitous Joints

Path: INTERPRET / Strgth Fac / Failr Crit / Ubq.Jts



Jointing in the surrounding rock can be accounted for by using the **Ubiq. Jts.** option. There are three possible sub-options:

⇒ **Barton-Bandis** (Ubiq. Jts. BB)

⇒ **Mohr-Coulomb** (Ubiq. Jts. MC)

⇒ **NO** ubiquitous joints (Ubiq. Jts. OFF)

In EXAMINE^{2D} ubiquitous joint modelling of joint behaviour:

Ubiquitous Joints DO NOT AFFECT STRESSES or DISPLACEMENTS!!!

⇒ up to *three* joint sets are assumed at each grid point. Each may have different properties (strength, angle with the X-axis, etc).

⇒ the strength factor is calculated for *each* joint set using the relevant equation (Mohr-Coulomb or Barton Bandis, see below), *as well as for the rock mass* (using the current rock mass failure criterion), and

⇒ the **least** strength factor is contoured.

When the contouring is complete, *note the prompt line message*:

> Ubiq. Joints Do Not Affect Stresses Or Displacements

Ubiquitous joint modelling is a post-analysis feature, *using the stresses calculated from the boundary element analysis* – the strength factor calculations have no effect on the stresses or displacements, which have already been calculated.

Barton-Bandis Ubiquitous Joints

Path: INTERPRET / Strgth Fac / Failr Crit / Ubq.Jts / Barton-Band

UBIQ. JTS
Barton-Band
Mohr-Coul
No Ubiq. Jts
RETURN

With this option, Ubiquitous Joints are modelled with the **Barton-Bandis** equation: $\tau = \sigma_n \tan[\mathrm{JRC}\ \log(\mathrm{JCS}/\sigma_n) + \phi_2]$.6 When you select this option, you will first be prompted:

> Number of Joint Sets (MAX 3) <1>:

Press *Enter* to select the default value of 1, or define multiple joint sets by entering 2 or 3 at the prompt — 3 is the maximum. Then enter the Barton-Bandis joint properties in the popup window(s) which will appear, as shown below:

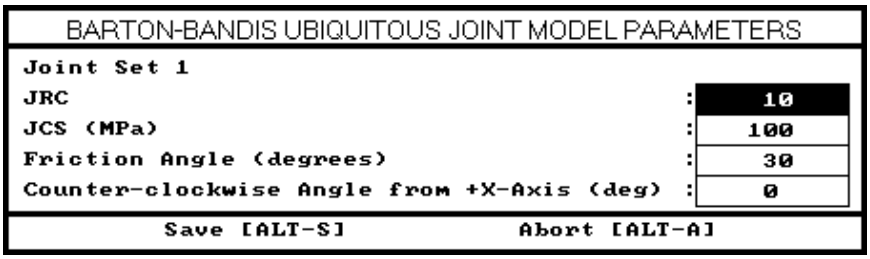


Figure 4.8: Popup window for entering Barton-Bandis Ubiquitous Joint parameters.

You will notice that the Ubiquitous Joints option on the **Failr Crit** menu changes to "Ubq.Jts BB>" when you choose this option.

Mohr-Coulomb Ubiquitous Joints

Path: INTERPRET / Strgth Fac / Failr Crit / Ubq.Jts / Mohr-Coul

UBIQ. JTS
Barton-Band
Mohr-Coul
No Ubiq. Jts
RETURN

When you select this option, Ubiquitous Joints are modelled with the Mohr-Coulomb equation for slip along a plane of weakness: $\tau = c + \sigma_3 \tan\phi$ 7, where c is the cohesion, and ϕ is the friction angle.

As described above, you will first be prompted:

> Number of Joint Sets (MAX 3) <1>:

Enter the number of joint sets, and then enter the joint properties in the popup window(s) which will appear:

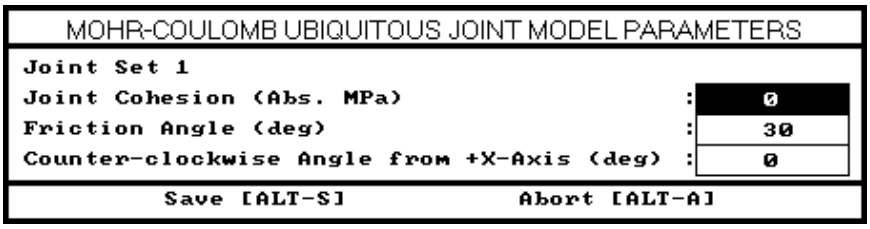


Figure 4.9: Popup window for entering Mohr-Coulomb Ubiquitous Joint parameters.

You will notice that the Ubiquitous Joints option on the **Failr Crit** menu changes to "Ubq.Jts MC>" when you choose this option.

Note that the ubiquitous joint slip planes are indicated in the stress block and in the Input Data information. This can be seen in the following figure, where 3 ubiquitous joint sets at angles of 0, 30 and 60 degrees have been defined.

When ubiquitous joints are defined, note the symbolic representation of slip planes within the stress block, and also in the Input Data display.

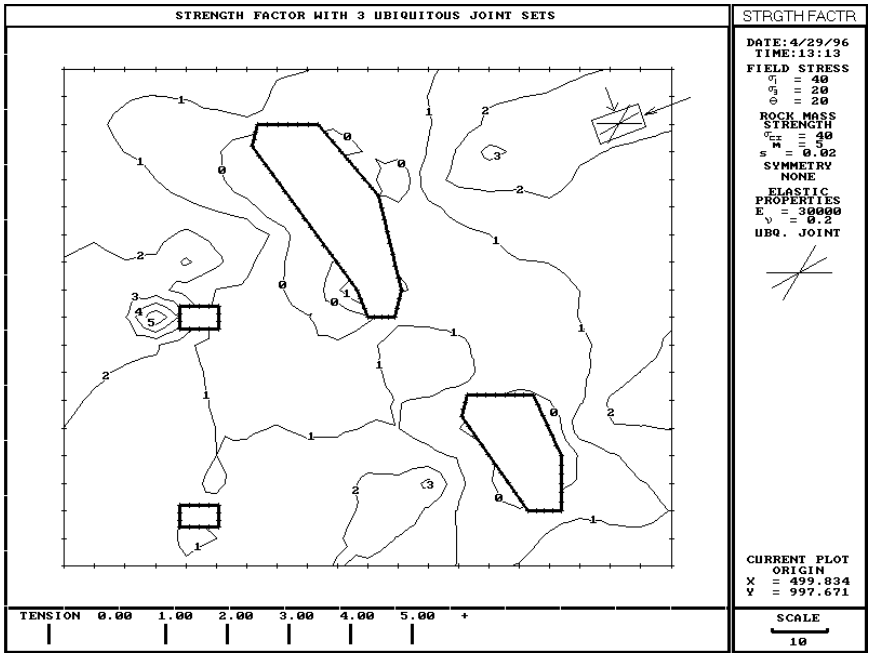


Figure 4.10: Three ubiquitous joint sets have been defined, at angles of 0, 30 and 60 degrees to the X-axis. Note that contour labels and a job title have been added in this figure.

Ubiquitous Joints OFF

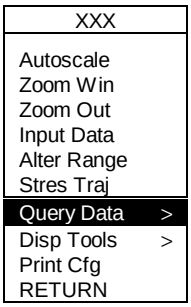
UBIQ. JTS
Barton-Band
Mohr-Coul
No Ubiq. Jts
RETURN

Path: INTERPRET / Strgth Fac / Failr Crit / Ubq.Jts / No Ubiq. Jts

This option turns OFF the Ubiquitous Joint modelling. You will notice that the Ubiquitous Joints option on the **Fail Crit** menu changes to "Ubq.Jts OFF>" when you choose this option.

Query Data

Path: INTERPRET / ... / Query Data



The **Query Data** menu provides the user with several options for viewing and plotting values at user defined points or along lines within the stress grid(s):

⇒ **display** numerical values at discrete points, or along lines

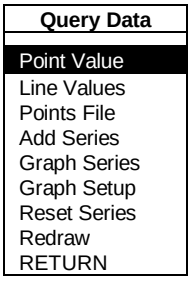
⇒ **plot** values along any arbitrary polylines

All values supplied by **Query Data** are interpolated from the existing stress grid(s), after COMPUTE has been executed – ie. the values are obtained from the existing database, after an analysis has been performed. It should be emphasized that the values are true interpolated values, and are NOT simply the nearest gridpoint value(s).

Values of stress, strength factor, or displacement, can all be obtained by using **Query Data** in the appropriate **INTERPRET** sub-menu.

Point Value

Path: INTERPRET / ... / Query Data / Point Value

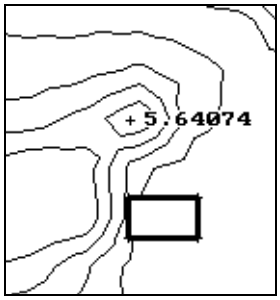


The **Point Value** option allows the user to obtain value(s) at discrete points within the stress grid(s). When you select this option you will be prompted:

```
> Enter Location [ESC = done]:
```

Use the mouse to select a point *within a stress grid*, or alternatively, key in x,y coordinates at the prompt. The interpolated value will immediately appear on the screen, at the point you have selected, as well as in the prompt which follows:

```
> Data Value at ( ## , ## ) = ## , Next Pt:
```



POINT VALUE ADDED TO STRENGTH FACTOR CONTOURS

As the prompt indicates, you may continue to enter points to obtain more values, or press *Esc* to return to the **Query Data** menu.

If you enter a point which is either:

⇒ *inside* an excavation

⇒ *outside* the stress grid(s)

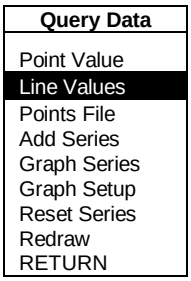
you will see the prompt:

```
> Data Not Available at this Location, Next Pt:
```

Line Values

Path: INTERPRET / ... / Query Data / Line Values

The **Line Values** option allows you to enter an arbitrary polyline, to obtain values at user defined intervals along the polyline. You will first be prompted:



> Enter Line Segment(s), Start Point:

Enter a starting point – graphically with the mouse, or key in coordinates at the prompt if you wish. Then enter further points at the prompt:

> To Point [u = undo, v = snap, c = close, a = arc]:

There are no restrictions on the polyline which you can enter. It can consist of any number of segments, follow any path (it may even cross over itself), and vertices *can be* inside excavations or outside stress grids. The only case which will be rejected is if your polyline falls *wholly* outside the stress grid(s) or *wholly* inside an excavation.

*To obtain **plots** of Values vs. Distance, you must use **Add Series** and then **Graph Series**, rather than **Line Values**.*

Note the options available at the prompt line:

⇒ enter **u** to undo mistakes (you can enter **u** any number of times to undo succesive vertex entries)

⇒ enter **v** to activate the vertex snap (this allows you to precisely snap to existing vertices of your model, eg. excavation boundary vertices)

⇒ enter **c** to close the polyline (ie. connect the first and last points, for instance if you are defining a closed polyline around an excavation

⇒ enter **a** to define segmented arcs (you may even define a circle, by entering **a** at the first prompt – see the MODEL->BOUNDARIES section for more information)

When you are finished entering your line (it may of course consist of only one segment!), press *Esc* to get the next prompt:

> Enter #Intervals for EACH Line Segment [1]:

You may subdivide the line segments into sub-intervals, by entering a number at the above prompt. Or to accept the default value of 1 (ie. no sub-intervals), simply press *Enter*. The next prompt is:

> Save Values to File (n)?

You may save the values to a file, in the format (x,y,value) – press *Enter* to accept the default of NO, or type 'y' and press *Enter* if you wish to save the values to a file. If you save to a file, you must respond to two final prompts:

> Enter Filename:

Enter a filename, *with any extension you wish* (no automatic extension is applied), and finally:

> Display Results as they are Saved (n)?

Press *Enter* if you *do not* wish to display the results on the screen (ie. you only want to save to a file), or type 'y' and *Enter* to display the results on the screen as well as saving to a file. *Values will be interpolated and displayed according to the number of intervals into which you have subdivided your polyline segment(s).*

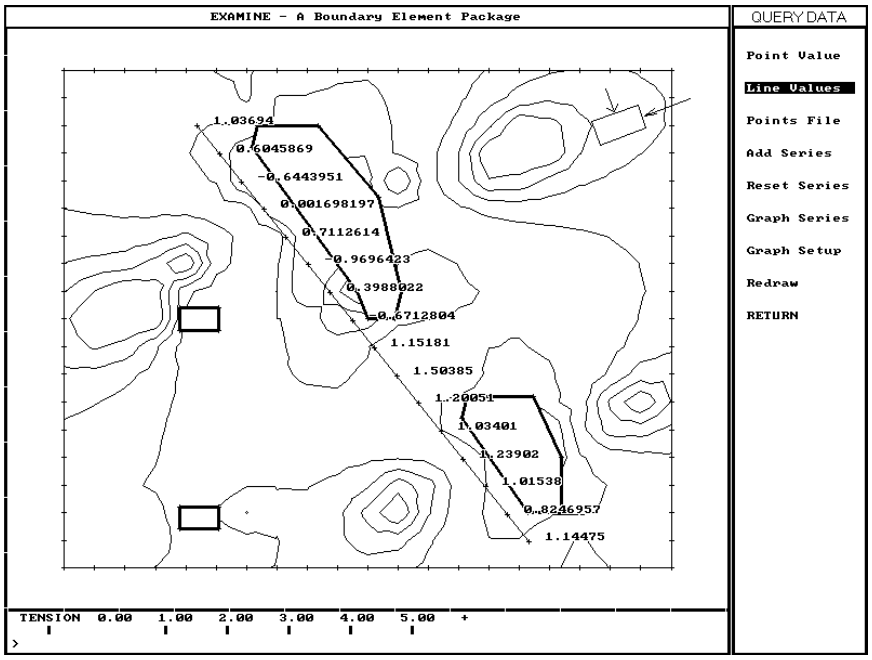


Figure 4.10: Using the Query Data->Line Values option to display Strength Factors along a user defined line (a single line segment has been entered, and divided into 15 intervals).

Points File

Query Data
Point Value
Line Values
Points File
Add Series
Graph Series
Graph Setup
Reset Series
Redraw
RETURN

Path: INTERPRET / ... / Query Data / Points File

With the **Points File** option, you may *read in* a file of (x , y) coordinates (where each line of the file is simply an (x,y) coordinate pair), for the purpose of obtaining values at the locations. You will first be prompted:

> Enter Filename Containing Points:

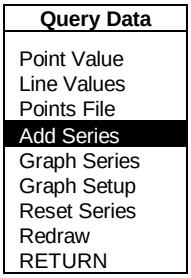
After entering a filename of your Points File, the next prompts are identical to the last three prompts of the **Line Values** option (see above), which allow you to re-save the Points File with the interpolated **values** appended to each line of the file (ie. x, y, value), and also to display the values on the screen if desired.

NOTE: you can read in a Points File which you have previously saved with the values appended – as long as the *first two numbers* on each line of the file are still the x,y coordinate locations.

Add Series

Path: INTERPRET / ... / Query Data / Add Series

Add Series allows you to enter polylines in exactly the same manner as the **Line Values** option – see above for details of this procedure. The differences are summarized below:



PLOTTING	**Add Series** is intended for use in conjunction with **Graph Series**, *to obtain plots of Values vs. Distance*, whereas **Line Values** is used for displaying numerical results on the screen – *you cannot plot the Line Values results*.
SAVING TO FILE	**Line Values** allows you to save results to a file. You cannot do this with **Add Series**.
INTERVALS	**Add Series** *does not* prompt for a number of intervals for subdividing line segments – the subdivision is automatic, and is averaged over the entire polyline, (rather than an equal interval for each line segment, as with Line Values). However the default value of the Add Series subdivision, may be changed through the **Graph Setup** option (see below).

Graph Series

Path: INTERPRET / ... / Query Data / Graph Series

After entering data with **Add Series**, select **Graph Series**, and *all* of your series will immediately be plotted, as illustrated below. You may plot any number of series on the same graph.

Query Data
Point Value
Line Values
Points File
Add Series
Graph Series
Graph Setup
Reset Series
Redraw
RETURN

Note that the number of intervals used to discretize your series, can be controlled through the #Data Points/Series option in **Graph Setup** (see below) – however, this should be set *before* you use **Add Series** (if you want to use a value other than the default). Changing this value *after* a series has been entered will have no effect on the existing plots, only series entered after the change will be affected.

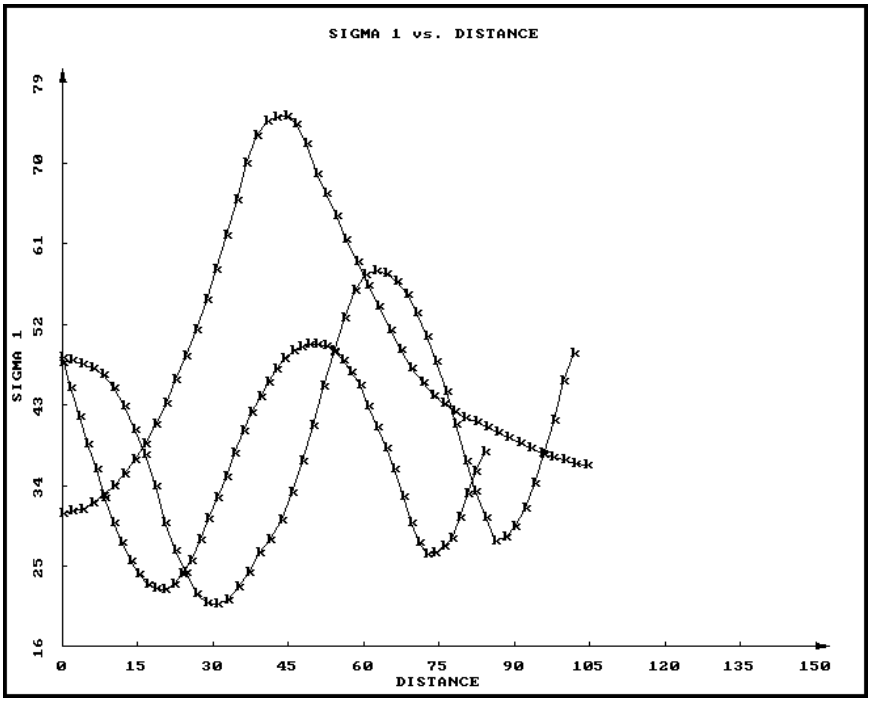


Figure 4.11: Example of plot created by **Query Data->Graph Series** (SIGMA 1 vs. DISTANCE).

*NOTE: The DISTANCE axis of the **Graph Series** plot corresponds to the distance along the line you have entered – ie. the distance is entirely relative to your path, and does not correspond to any absolute coordinate system. "Zero" distance corresponds to the first point of each series.*

Graph Setup

Path: INTERPRET / ... / Query Data / Graph Setup

You may change some of the configuration settings of the **Graph Series** plots by means of the **Graph Setup** option.

The options in the **Graph Setup** popup menu (shown below) should be self explanatory. However, note the following:

Query Data
Point Value
Line Values
Points File
Add Series
Graph Series
Graph Setup
Reset Series
Redraw
RETURN

⇒ If you wish to use a #Data Points/Series other than the default value of 50, enter a new value in the Graph Setup menu (and select *Save* at the bottom of the window), BEFORE you use **Add Series**, since changing the value of #Data Points/ Series has no effect on existing series, only on those entered subsequent to the change.

⇒ If you toggle Axes Extents to USER, and enter your own axes extents, you may find that you no longer have "round" numbers on your x and/or y axes. To remedy this, adjust your #Intervals and/or your Axes Extents to appropriate values so that you obtain the numbers you wish to see on your axes.

⇒ Changes to the Graph Setup menu are only saved while you are in a given plot menu of INTERPRET (ie. SIGMA 1, STRENGTH FACTOR, etc.) When you switch to another plot menu (or exit INTERPRET altogether), all Graph Setup options will return to their defaults.

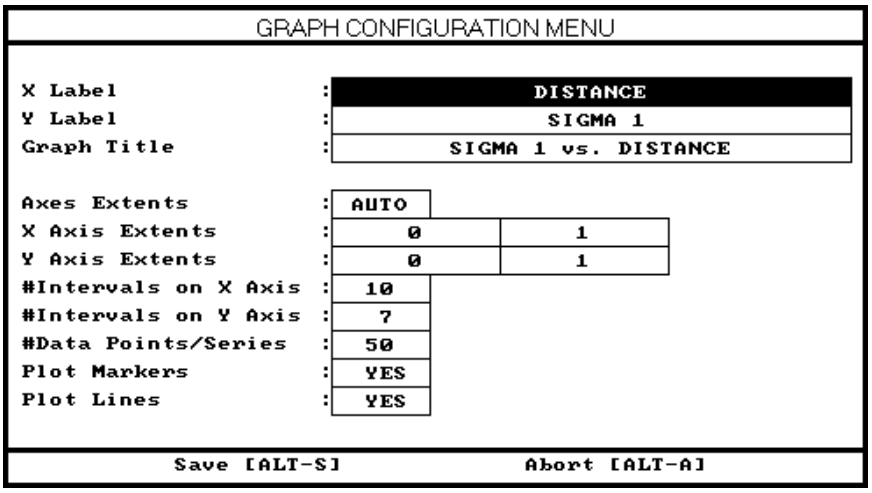
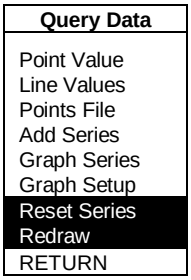


Figure 4.12: Graph Setup Menu, for changing the default settings used in the Graph Series plots.

Reset Series

Path: INTERPRET / ... / Query Data / Reset Series



Use **Reset Series** to delete all data created by the **Add Series** option – no prompts will appear, all series will be automatically deleted. Note: series are automatically deleted when you leave the plotting menu (ie. SIGMA 1, STRENGTH FACTOR, etc.) in which you created the series.

Redraw

Path: INTERPRET / ... / Query Data / Redraw

The **Redraw** option in the **Query Data** menu will:

⇒ erase all points and/or lines entered using the **Point Value** or **Line Values** options.

It *will not* delete any **Add Series** data – use **Reset Series** for this purpose.

Display Tools

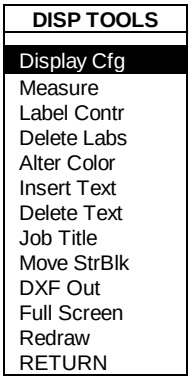
Path: INTERPRET / ... / Disp Tools



The **Disp Tools** menu is present in all (applicable) sub-menus under INTERPRET. This menu contains a variety of useful commands for customizing the display. These are described below.

Display Configuration

Path: INTERPRET / ... / Disp Tools / Display Cfg



The **Display Configuration** popup menu allows you to toggle ON or OFF various display settings in the INTERPRET menu.

New settings are saved only while currently working in the INTERPRET menu. Once you leave INTERPRET, all settings will return to their default values, with the exception of the Contour Type, which will be preserved. When you exit the program, ALL settings will return to their default values, including the Contour Type.

When you leave the INTERPRET menu, Display Configuration settings return to their default values.

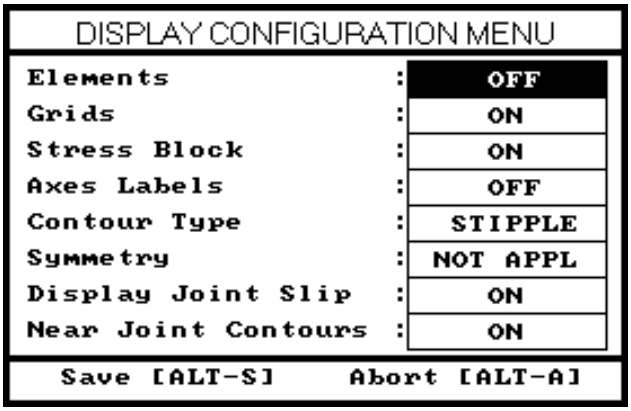


Figure 4.13: Display Configuration popup menu.

Elements ON/OFF

This option toggles the display of the boundary element discretization ON or OFF. These represent the boundary elements created when you discretize your excavation geometry.

Default is OFF.

Grids ON/OFF

This option toggles the display of stress grids ON or OFF. These are the grids defined using MODEL->STRESS GRD, which define the grid of points for stress, strength factor and displacement calculation.

Default is ON.

Stress Block ON/OFF

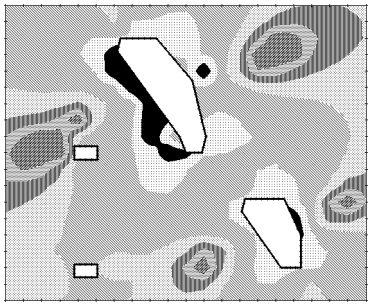
This option toggles the display of the stress block ON or OFF. Default is ON.

Axes Labels

This option will toggle the display of xy axes labels (representing distance) ON or OFF. Default is OFF.

Contour Type

The FOUR contour display modes are:



STIPPLE

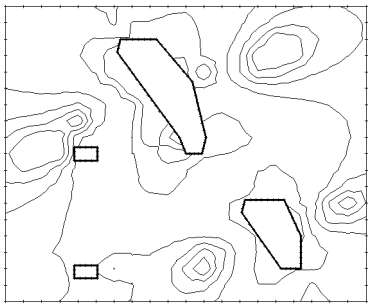
⇒ **Color**

⇒ **Stipple**

⇒ **Contour**

⇒ **Photo E**

Toggle the Contour Type by clicking with the *left mouse button*, or the *Enter* key. Note that changing the contour type does not change the range of the contour mapping or the locations of the contour lines. It is a graphical style option only. The four contour types are described below.



CONTOUR

Color

The contours are displayed as coloured zones, each zone representing a contour interval. This is the default mode for color monitors and graphics cards.

Stipple

The contours are displayed as zones with different stipple or hatch patterns, each zone representing a contour interval. This is the default mode for monochrome monitors and graphics cards. Several of the screen captures in this manual show this mode of contouring.



PHOTO E

Contour

The contours are displayed as lines representing contour range boundaries. These contours can be *labelled* using **Label Contr** in **Disp Tools**.

PhotoE

The contours are displayed as alternating BLACK and WHITE zones. This is fairly primitive in nature and the most difficult to interpret. It is however the best monochrome mode for faxing or photocopying results on lower quality equipment. It is best to use this in conjunction with the **Label Contr** option of **Disp Tools**, since the legend alone is not sufficient to identify the values of the contour intervals.

Symmetry ON/OFF

If symmetry is being used the mirrored regions of the problem will also be contoured by default. This can be turned ON or OFF with this option.

Display Joint Slip

This option only applies to JOINTS, and only applies if the Allow Slip option in the Joint Properties popup window is toggled to YES. If this is the case, then the type of deformation along the joint (ie. **elastic**, **slip** (shear), or **tension** (normal)) can be graphically represented when this option is toggled ON.

If you select **Input Data** from the menu in which you are working, the legend will indicate the type of joint deformation, as shown at right.

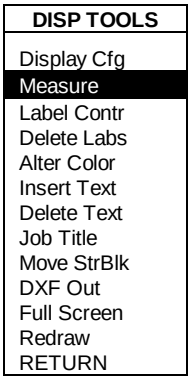


Near Joint Contours

When creating the contour plots, quite a bit of interpolation and extrapolation goes on near joints in order to create contours that are "nicely discontinous". In some instances you may want to view only the data generated directly by the compute engine — to do this, turn Near Joint Contours OFF.

Measure

Path: INTERPRET / ... / Disp Tools / Measure



You can easily measure distances anywhere on the model, using the **Measure** option of **Disp Tools**. Select **Measure** and enter two points at the prompts:

```
> Enter / Select Point 1 [v = vrtx snap]:
```

```
> Enter / Select Point 2 [v = vrtx snap]:
```

You may enter points graphically with the mouse (remember the vertex snap option is always available), or you may key in x,y coordinates at the prompt lines if you prefer. After the second point is entered, the prompt line will display the results:

```
> Distance = ##       Angle = ##
```

Note the definition of angle is positive counter-clockwise from the right hand x-axis. *The calculated angle will change by 180° depending on the order in which you input the two points.*

To delete the lines which are left on the screen after using **Measure**, use the **Redraw** option of **Disp Tools**.

Label Contr

Path: INTERPRET / ... / Disp Tools / Label Contr

A contour line within a contoured stress grid represents *a line of equal stress* (or *strength factor* or *displacement*, depending on which plot you are viewing). The **Label Contr** option allows you to place **contour labels** on the contour lines, representing the value of the contour line. Select this option and you will be prompted:

DISP TOOLS

Display Cfg
Measure
Label Contr
Delete Labs
Alter Color
Insert Text
Delete Text
Job Title
Move StrBlk
DXF Out
Full Screen
Redraw
RETURN

> Select contour boundary to label using cursor:

Move the cursor to a location **on a contour line** where you wish to place a label. Press *Enter* or the *left mouse button* and the appropriate numerical label will be placed at the cursor location. *Note that the cursor must be within one or two screen pixels of a contour line for a label to be placed – if not, no label will appear.*

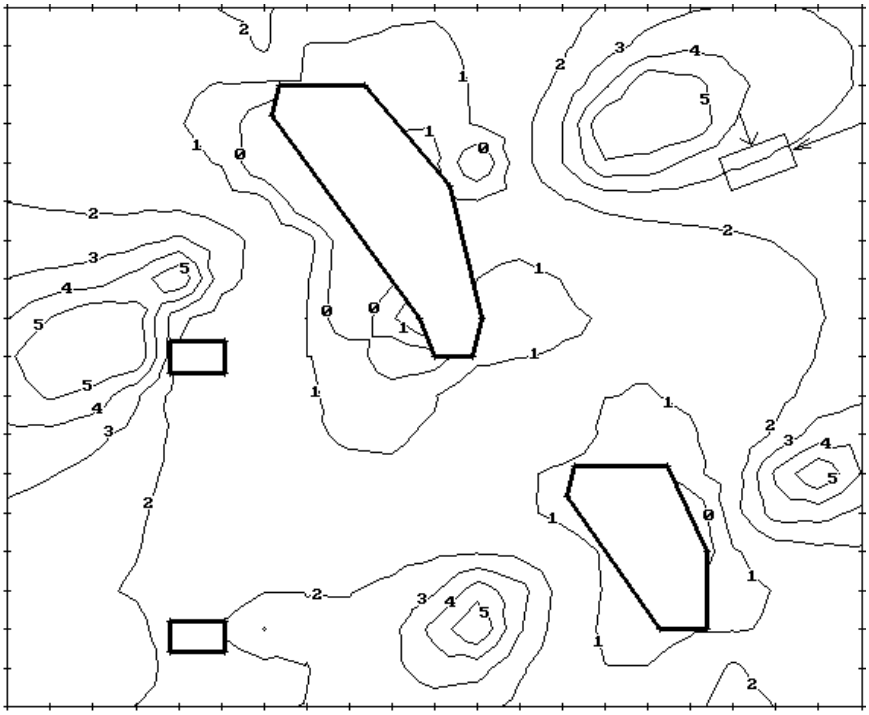


Figure 4.14: Strength Factor contours labelled using **Label Contr** (note that the LINE Contour Type is toggled).

Delete Labs

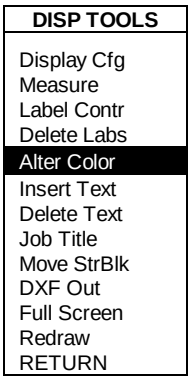
Path: INTERPRET / ... / Disp Tools / Delete Labs

To *delete* contour labels, select **Delete Labs** from the **Disp Tools** menu, and the labels will be deleted. Then select **Redraw** to redraw the screen, since the labels leave an 'outline' on the contour plot after they are deleted. (Note that **Delete Text** *will not* delete contour labels.)

Alter Color

Path: INTERPRET / ... / Disp Tools / Alter Color

The colours of contour intervals can be changed from their default settings if desired, by selecting **Alter Color** from the **Disp Tools** menu – the following information box will be displayed in the viewing region, describing how to toggle through the colour selection using the *keyboard arrow keys*:




```
Select color with LEFT and RIGHT arrow keys
Change color with UP and DOWN arrow keys
Press ESC to exit or d to restore default settings
```

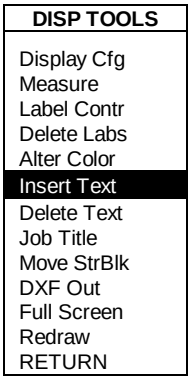
These instructions *refer to the bar legend at the bottom of the screen*, which will display a small marker – a round dot – over the contour interval to be changed. Experiment with this to see how it works.

For Contour Types other than COLOR (ie. STIPPLE, CONTOUR and PHOTO E), the same procedure is followed. With STIPPLE, **Alter Color** will scroll through the various stipple patterns, with CONTOUR, **Alter Color** will change the colours of the contour lines, and with PHOTO E, **Alter Color** will merely toggle each interval between black and white.

Insert Text

Path: INTERPRET / ... / Disp Tools / Insert Text

Text may be inserted anywhere in the drawing region, to enhance the appearance of printouts, etc. Select this option and follow the instructions in the prompt line:




```
> Text to insert:
```

```
> Select top left corner of text block:
```

```
> Press 'D' to Delete. Any other key to Continue.
```

⇒ Pressing 'any key' at the *last prompt* will add the text to the screen, and return you to the *first* prompt, if you wish to add more text. Note: if there is not enough room to fit your text at the selected point, you will get a warning prompt.

⇒ Pressing 'd' at the *last prompt*, will return you to the *second* prompt. This is useful for repositioning your text, if you are not happy with its location.

⇒ To leave the text insertion mode, press *Esc* as many times as necessary (once or twice, depending on which prompt you are at), to return to the **Disp Tools** menu.

Delete Text

Path: INTERPRET / ... / Disp Tools / Delete Text

Use **Delete Text** to delete any text added with **Insert Text**. The deleted text may leave an "outline" of its position on the screen if it was overlying any existing entitities. Select **Redraw** after using **Delete Text**, to regenerate the screen and redraw all existing entitities.

DISP TOOLS
Display Cfg
Measure
Label Contr
Delete Labs
Alter Color
Insert Text
Delete Text
Job Title
Move StrBlk
DXF Out
Full Screen
Redraw
RETURN

Job Title

Path: INTERPRET / ... / Disp Tools / Job Title

You can change the **Job Title** while within the INTERPRET menu. See the MODEL->Job Title section for info.

Move StrBlk

Path: INTERPRET / ... / Disp Tools / Move StrBlk

You will notice in the upper right hand corner of the display, the small stress block indicating the relative magnitude and orientation of the field stresses. In its default position, this block may be hard to see, or might hide information in the contours. To move the block to a less cluttered area of the screen, select this option. You will be prompted:



> Move Stressblock to Point:

Simply move the cursor to the point where you want the block placed and press Enter, and the stress block will move to the new position.

Depending on its original position, you may notice the *outline* of the stressblock in its former location, after moving it. To eliminate this, select the **Redraw** option from **Disp Tools**, to redraw the screen.

Note: this option is also found in MODEL->FLD STRESS.

DXF File Out

Path: INTERPRET / ... / Disp Tools / DXF Out

DISP TOOLS
Display Cfg
Measure
Label Contr
Delete Labs
Alter Color
Insert Text
Delete Text
Job Title
Move StrBlk
DXF Out
Full Screen
Redraw
RETURN

This option allows you to save the *entire display* to a DXF (Drawing Exchange Format) file which can be read into programs such as AutoCAD. Such a drawing file can then be used to generate report quality output using one of AutoCAD's many printer and plotter drivers.

After preparing the screen with the desired zoom limits and contour range, select **DXF Out**. You will be prompted for a file name.

```
> Enter DXF file name:
```

The .DXF extension will automatically be added, you do not need to type it in. Next input the coordinates of a PIVOT point.

```
> Select pivot point ( x , y ) :
```

This is a point which will be used to position the drawing in the AutoCAD environment. Next input the mapping coordinates for this point.

```
> Input DXF point < # , # , 0 > :
```

This second point is the point in the DXF file to which the first pivot point will be mapped. The rest of the file will be translated by the same relative distance between these two points. To preserve the original boundary coordinates simply pick any point for the first pivot point, and use the same coordinates for the second. (Hitting *Enter* for each of the two prompts will have this effect.) The final prompt is for a scale factor. This is used to expand or shrink the output. Usually a value of 1 (one) should be used.

```
> Input scaling factor < 1 > :
```

DXF OPTIONS
Input Data
Label Contr
Insert Text
RETURN

After the file is written, a new menu will appear (see the margin). This will include options allowing you to add Contour Labels, Text, or the Input Data information to your DXF plot (these features are described earlier in this chapter).

Select **RETURN** in this menu when you are finished adding features to your DXF file. This will close the file and return you to the **Disp Tools** menu. You will see the prompt:

```
> DXF file complete...
```

The output from the **DXF Out** option can be viewed by running AutoCAD (trademark of Autodesk). Use the DXFIN command to retrieve the DXF file. You may then print the output using any printer or plotter supported by AutoCAD.

NOTE that you may have to ZOOM the display within AutoCAD after importing this DXF file since the DXF file does not contain any viewport settings. If the AutoCAD screen appears blank after executing DXFIN, try to PAN or ZOOM to the coordinate range occupied by the EXAMINE^{2D} plot.

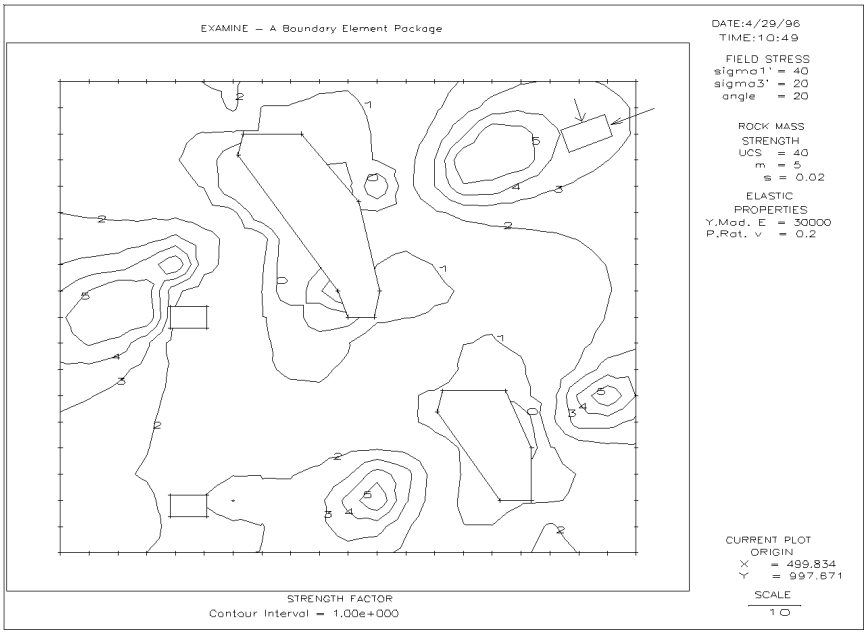


Figure 4.15: LaserJet output using **DXF Out** option and AutoCAD.

Full Screen

Path: INTERPRET / ... / Disp Tools / Full Screen

*Press **any key** to return from Full Screen mode back to the Disp Tools menu.*

This option allows you to generate a screen plot which fills the whole screen with the Plot Display Box and legend information. The Upper Status Box, the Prompt Line and the Menu Box areas disappear. You may prefer to use this option for screen captures.

Note: pressing *any key* will return you to the **Disp Tools** menu.

DISP TOOLS

Display Cfg
Measure
Label Contr
Delete Labs
Alter Color
Insert Text
Delete Text
Job Title
Move StrBlk
DXF Out
Full Screen
Redraw
RETURN

Redraw

Path: INTERPRET / ... / Disp Tools / Redraw

Use the **Redraw** option to regenerate the screen and get rid of some alterations created by use of the **Disp Tools** options. For instance, **Redraw** will erase lines on the screen created by use of the **Measure** option, and erase "outlines" which may be left on the model after deleting contour labels (**Delete Labs**), deleting text (**Delete Text**), or moving the stress block (**Move StrBlk**).

Note: **Redraw** will NOT reset the viewing region if you have been using the Zoom options. For this use **Autoscale**.

NETDISP Utility

When solving problems involving preexisting surfaces or excavations, you will probably be interested in the behaviour of the rock mass only after the creation of a new opening. Elastic stresses can be meaningful at any stage of the excavation. Displacements which are cumulative and path independent (the same regardless of excavation sequence) will not have much meaning, particularly when a surface is involved.

To calculate displacements with respect to a datum state such as a previous excavation stage or after inputting a surface, you must use the **NETDISP** utility included with the EXAMINE^{2D} package.

First you must create and compute two data files:

⇒ The *datum* file will be the first stage. This will include all previous excavations before excavating the opening of interest and/or will include a discretized ground surface.

⇒ The *appended* data file will be identical to the *datum* file but will include any new excavations.

NOTE:

⇒ Do not delete any boundaries when creating the *appended* file.

⇒ Surfaces should be stretched **into the rock mass** ("excavated") and not "bulged" into the free space.

Either of these errors will cause unpredictable results. **Do Not** change any of the problem parameters. Add new excavations only or "excavate" a surface outcrop.

After executing **COMPUTE** for both data files, exit EXAMINE^{2D}.

Type *netdisp* **at the DOS prompt.**

When prompted, input the *datum* filename, the *appended* filename, and a new *net* filename.

The program will create a new *net* file containing stresses from the *appended* file and displacements which reflect new displacements only after excavating the new openings.

NOTE: Displacements in the data files will not be meaningful. These values contain rigid body displacement components which are filtered out by the EXAMINE^{2D} interface.

Bibliography

Boundary Elements

Banerjee, P.K. and R. Butterfield: *Boundary Element Methods in Engineering Science*, McGraw-Hill Book Company, Ltd., London

Brebbia, C.A and Dominguez, J.: *Boundary Elements: An Introductory Course*, Computational Mechanics Publications, Southhampton, 1983

Crouch, S.L. and Starfield, A.M.: *Boundary Element Methods in Solid Mechanics*, George Allen and Unwin Ltd., London, 1989

Pande, G.N., Beer G. and Williams, J.R.: *Numerical Methods in Rock Mechanics*, John Wiley and Sons, West Sussex, 1990

Elasticity

Chang, Tien Sun and Frederick, Daniel: *Continuum Mechanics*, Allyn and Bacon, Inc., 1965

Fung, Y.C.: *Foundations of Solid Mechanics*, Prentice-Hall, Inc., New Jersey, 1965

Timoshenko, S. and Goodier, N.: *Theory of Elasticity*, McGraw-Hill Book Company, Ltd., New York, 1970

Rock Mechanics

Brady, B.H.G. and Brown, E.T.: *Rock Mechanics for Underground Mining*, George Allen and Unwin, London, 1985

Goodman, Richard E.: *Introduction to Rock Mechanics*, John Wiley and Sons, New York, 1980

Hoek, Evert and Brown, Edwin T.: *Underground Excavations in Rock*, The Institute of Mining and Metallurgy, London, 1981

Hoek, E., Kaiser, P.K. and Bawden, W.F.: *Support of Underground Excavations in Hard Rock*, A.A.Balkema, Rotterdam, Brookfield, 1995

Jaeger, J.C. and Cook, N.G.W.: *Fundamentals of Rock Mechanics*, Chapman and Hall, London, 1969

Part 3 – Tutorial Manual

INCLUDES TUTORIALS:

SIMPLE TUTORIAL

FAULT TUTORIAL

NEAR SURFACE TUTORIAL

Introduction

THREE tutorials are included in the following pages. The user has the option of going through the tutorials step by step, and creating the model themselves (recommended), or they may skip the modelling steps and examine the products of the finished tutorials, which can be found in the indicated files in the TUTORIAL sub-directory of your installation directory.

FILES	DESCRIPTION
simp.exa	A simple, "quick start" type of tutorial, which will get the user familiar with the basic features of EXAMINE[2D] (entering geometry, basic data interpretation features, etc).
flt1.exa flt2.exa	A tutorial featuring a "fault" – this tutorial is presented in two steps – *first* the excavations are entered and analysed *without* the fault, and then the fault is added to the model, and a new analysis performed.
ns1.exa ns2.exa	A tutorial featuring a circular tunnel, and a halfspace free surface, also presented in two steps. In the second step of the tutorial, the ground surface is changed to a discretized free surface with an excavated "ditch", and also a surface load is added above the circular tunnel.

Hints about the Tutorial Manual

- Follow instructions such as:

  Select: MODEL → SETUP → Set Limits

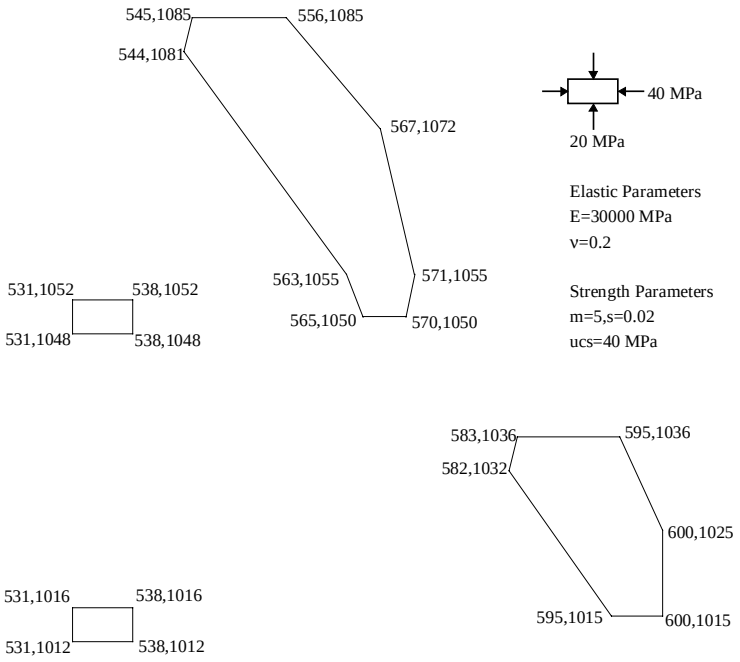
  to navigate the sidebar menu selections.

- Text in "courier" font, enclosed by a box (eg.):

```
Enter Excavation CCW,Start Point: 570,1050
To Point [c=close,a=arc,u=undo,Esc=done]: 571,1055
To Point [c=close,a=arc,u=undo,Esc=done]: 567,1072
```

  indicates the prompt line instructions (or a series of prompt line instructions), found at the bottom of the EXAMINE[2D] screen. The ***bold italic*** text at the end of a prompt line indicates the user input for the prompt line.

A Simple Tutorial



This tutorial can be done using either the educational or commercial versions of EXAMINE[2D].

If you wish to skip the model process, the finished product of this tutorial can be found in the "simp.exa" data file located in the "tutorial" directory in your installation directory.

MODEL

In this example we will model a simple set of excavations. The first step is to define the geometry and parameters required for the analysis. If you have not already done so, run the program by typing *examine* at the DOS prompt. Select the first option in the main menu, the **MODEL** option, by highlighting the **MODEL** button and pressing the *left mouse button* or the keyboard *Enter* key. This will place you in the modeler.

Set Limits

The first step in any computer-aided design process is setting the drawing limits of the region so that the limits encompass the excavation geometry.

Select: MODEL

Select: MODEL → SETUP

Select: MODEL → SETUP → Set Limits

```
ENTER MINIMUM X,Y COORDINATES <0,0>: 500,1000
ENTER MAXIMUM X,Y COORDINATES <116.092,100>: 630,1100
```

Select: MODEL → SETUP → RETURN

Entering Geometry

Now define the excavation geometry. EXAMINE^{2D} defines excavations by allowing the user to interactively draw the boundaries using the mouse, or by typing in coordinates.

Proceed to the excavation geometry menu. Enter the four excavation boundaries using the following sequence.

Select: MODEL → BOUNDARIES

Select: MODEL → BOUNDARIES → ADD EXCAV

```
Enter Excavation CCW,Start Point: 570,1050
To Point [c=close,a=arc,u=undo,Esc=done]: 571,1055
To Point [c=close,a=arc,u=undo,Esc=done]: 567,1072
To Point [c=close,a=arc,u=undo,Esc=done]: 556,1085
To Point [c=close,a=arc,u=undo,Esc=done]: 545,1085
To Point [c=close,a=arc,u=undo,Esc=done]: 544,1081
To Point [c=close,a=arc,u=undo,Esc=done]: 563,1055
To Point [c=close,a=arc,u=undo,Esc=done]: 565,1050
To Point [c=close,a=arc,u=undo,Esc=done]: c
```

Select: MODEL → BOUNDARIES → ADD EXCAV

```
Enter Excavation CCW,Start Point: 600,1015
To Point [c=close,a=arc,u=undo,Esc=done]: 600,1025
To Point [c=close,a=arc,u=undo,Esc=done]: 595,1036
To Point [c=close,a=arc,u=undo,Esc=done]: 583,1036
To Point [c=close,a=arc,u=undo,Esc=done]: 582,1032
To Point [c=close,a=arc,u=undo,Esc=done]: 595,1015
To Point [c=close,a=arc,u=undo,Esc=done]: c
```

Select: MODEL → BOUNDARIES → ADD EXCAV

```
Enter Excavation CCW,Start Point: 531,1048
To Point [c=close,a=arc,u=undo,Esc=done]: 538,1048
To Point [c=close,a=arc,u=undo,Esc=done]: 538,1052
To Point [c=close,a=arc,u=undo,Esc=done]: 531,1052
To Point [c=close,a=arc,u=undo,Esc=done]: c
```

Select: MODEL → BOUNDARIES → ADD EXCAV

```
Enter Excavation CCW,Start Point: 531,1012
To Point [c=close,a=arc,u=undo,Esc=done]: 538,1012
To Point [c=close,a=arc,u=undo,Esc=done]: 538,1016
To Point [c=close,a=arc,u=undo,Esc=done]: 531,1016
To Point [c=close,a=arc,u=undo,Esc=done]: c
```

Toolbox

At the bottom of most menus there is a **Toolbox**. This menu contains options for manipulating the drawing viewport. You can Zoom In and Out, Pan, and Autoscale the drawing. Some of the keyboard function keys also give you these abilities. The following is a list of the keyboard function keys and their use.

F1=redraw
F2=autoscale
F3=memory available
F4=zoom out
F5=zoom in
F7=grid toggle
F8=cursor toggle
F9=snap toggle

Discretization

The next step is to discretize the line segments making up the openings, into boundary elements. To do this:

Select: MODEL → BOUNDARIES → DISCRETIZE

Select: MODEL → BOUNDARIES → DISCRETIZE → DEFAULT

This process will create approximately 50 boundary elements (total) around the perimeter of the four excavations.

Boundary Conditions

After you have finished discretizing the excavation geometry, return to the **BOUNDARIES** menu to set the boundary conditions on the excavation surface.

Select: MODEL → BOUNDARIES → DISCRETIZE → RETURN

Select: MODEL → BOUNDARIES → BNDRY COND

Select: MODEL → BOUNDARIES → BNDRY COND → DEFAULT

You have now applied the default boundary conditions for excavation elements – zero normal and shear stress on the surface of the elements. (Remember that the default boundary conditions which are applied by the **DEFAULT** option can always be changed through the **Chge Deflt** option).

Stress Grid

You have now finished defining the excavation geometry. The next step is to define the locations within the rock mass where stresses and displacements are to be calculated. In EXAMINE^{2D}, stresses can either be calculated within a user defined rectangular region at constant grid spacing (**ADD GRID** option) or at constant intervals along a line segment (**ADD LINE** option). In this example we will define a rectangular grid of stress points which encompass the entire geometry.

Select: MODEL → BOUNDARIES → BNDRY COND → RETURN

Select: MODEL → BOUNDARIES → RETURN

Select: MODEL → STRESS GRD

Select: MODEL → STRESS GRD → ADD GRID

```
Select Grid Corner #1:  move cursor to approximately
                        510,1005 and select
Select Grid Corner #2:  move cursor to approximately
                        620,1095 and select
Enter Number of Intervals in X direction <23>: Enter
Enter Number of Intervals in Y direction <19>: Enter
```

By selecting the **Points On** option you can see the actual locations (the grid points) within the rock-mass where stresses are calculated in the boundary element analysis.

Field Stress

Now proceed to enter the far field stress state by returning to the **MODEL** menu and selecting the **FLD STRESS** option. In EXAMINE^{2D} there are three possible far field stress distributions: **CONSTANT**, **GRAVITY** and **USER-DEFINED**, that can be used to define the initial in situ stress field. For this tutorial we will be using a **CONSTANT** far field stress field.

Select: MODEL → STRESS GRD → RETURN

Select: MODEL → FLD STRESS

Select: MODEL → FLD STRESS → CONSTANT

In the popup menu, use the mouse or keyboard arrow keys to select the box containing the value for Sigma 1. Type in 40 for Sigma 1. Using the same process define Sigma 3 as being 20. Keep the angle at 0 degrees, this specifies that the major principal stress is horizontal since the angle is measured counter-clockwise from the positive horizontal axis. When finished, select the Save button to save the entered values and exit the popup window.

Failure Criterion

Return to the **MODEL** menu and select the **Failr Crit** option. In EXAMINE^{2D}, the definition of a failure criterion is not required since the analysis is elastic. The use of the failure criterion can be used in determing the zones of overstress based on the stress results from the elastic analysis. For this example, we will define a set of strength parameters using the Hoek-Brown strength criterion.

Select: MODEL → FLD STRESS → RETURN

Select: MODEL → Failr Crit

Select: MODEL → Failr Crit → Hoek-Brown

In the popup menu, define m to be 5.0, s to be 0.02 and the intact uniaxial compressive strength to be 40 MPa. When finished, select the Save button to save the entered values and exit the popup window.

Elastic Properties

Now define the elastic properties of the rock mass.

Select: MODEL → Failr Crit → RETURN

Select: MODEL → Elast Prop

In the popup menu define the Young's modulus as being 30000 MPa and the poisson ratio equal to 0.2. When finished, select the Save button to save the entered values and exit the popup window.

Job Title

Although it is not required, enter a job description for this example.

Select: MODEL → Job Title

```
Enter Job Description: This is a simple tutorial
```

You have now completed the modeling phase of the analysis. Return to the main menu to save the model, then run the analysis.

Select: MODEL → RETURN

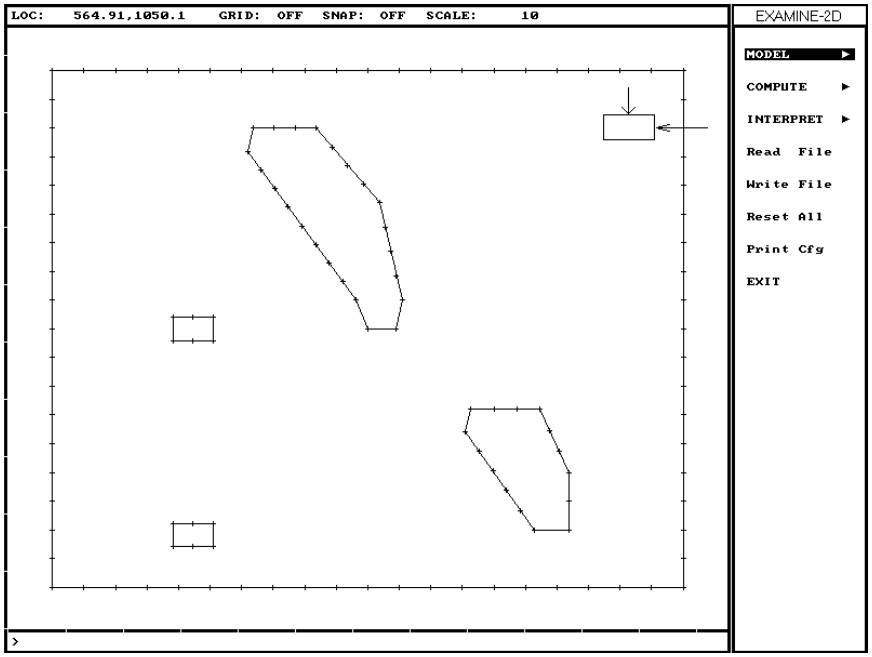


Figure T1-1: Finished model – simple EXAMINE^{2D} tutorial

COMPUTE

Before you analyze your model, save the file into a file called simple.exa. By default, all EXAMINE^{2D} model files have a .exa extension. As a result, when prompted for a filename, you do not have to, and shouldn't, type the .exa extension.

Select: Write File

```
Output filename or select w/ mouse[]: simple
```

You are now ready to run the analysis, select the **COMPUTE** option.

Select: COMPUTE



Select 'Yes' with the mouse, or type 'y', and the program will proceed in running the analysis. During the processing, the screen is updated with the progress of the analysis. When completed, the program returns you to the main EXAMINE^{2D} menu and you are ready to view the results.

INTERPRET

This phase of the analysis is the most important. Using the analysis results and the data interpretation facilities in EXAMINE2D, you will proceed with the process of determining the stress distribution within the rock mass. As well, the regions of overstress based on the elastic stress analysis can be determined. This can provide an important insight into the overall stability of the underground openings. As a result, a lot of effort has gone into the presentation of the data. There are a variety of methods for looking at the data, we will proceed in exploring some of these options.

Select: INTERPRET

You will immediately notice a wide range of datasets that can be viewed. The datasets include the principal stresses, strength factor, displacements, user defined functions, and global stress tensor components. We will first view the distribution of the major principal stress by selecting the **SIGMA 1** option.

Select: INTERPRET → SIGMA 1

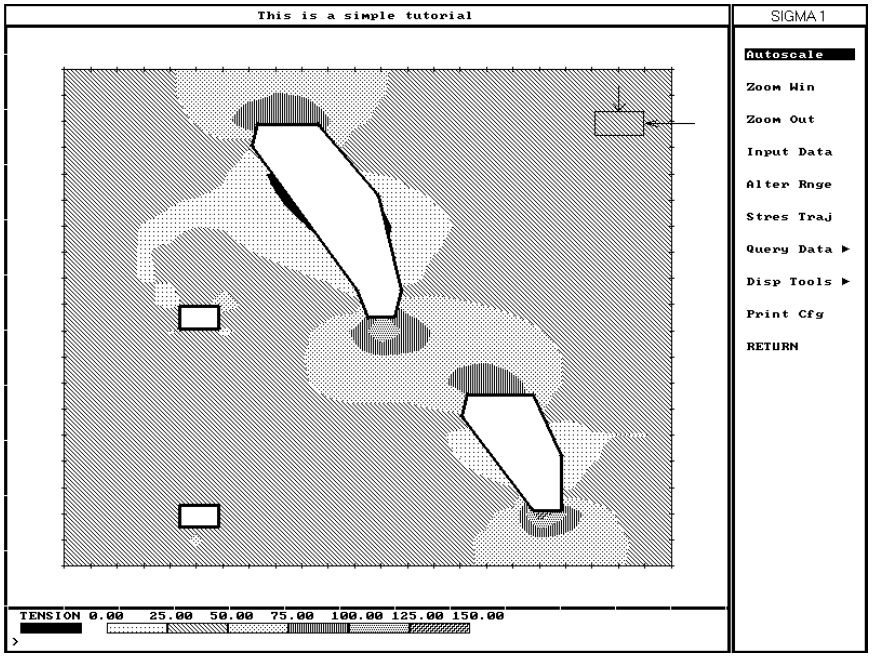


Figure T1-2: Contours of Major Principal Stress

Using the values calculated at the grid points, solid color contours are created within the user-defined stress grid. Note: the plot you obtain may not be *exactly* as shown in the above figure, if you used the mouse to enter the two points defining the stress grid. Now let's view the direction of the principal stresses at each grid point by toggling on the stress trajectories with the **Stres Traj** option.

Select: INTERPRET → SIGMA 1 → Stres Traj

The stress trajectories are shown as small cross icons where the long axis of the cross is oriented in the direction of the major principal stress while the short axis is the direction of the minor principal stress. Now turn off the trajectories by re-selecting the **Stres Traj** option, and display the model **Input Data**.

Select: INTERPRET → SIGMA 1 → Stres Traj

Select: INTERPRET → SIGMA 1 → Input Data

The **Input Data** option is useful when you wish to make a hard copy of the results for presentation purposes. (Using the **Print Cfg** option, you can select a printer, or specify that you wish to capture the screen to a PCX image file for import into your favorite word processor.) Press *any key* to exit the **Input Data** display. Let us now proceed in altering the range of intervals of the color contours.

Select: INTERPRET → SIGMA 1 → Alter Range

```
Enter Minimum, Maximum Range Values <0,150>: 0,120
```

This changes the range *interval* to 20 MPa and the maximum extent to 120 MPa.

Select: INTERPRET → SIGMA 1 → Disp Tools

Select: INTERPRET → SIGMA 1 → Disp Tools → Display Cfg

In the Display Configuration popup menu, toggle the "Contour Type" to "CONTOUR", and select Save. Note how the contour type changes from solid to line contours. Repeat this procedure, toggle the "Contour Type" back to "COLOR", and return to the main INTERPRET menu.

Select: INTERPRET → SIGMA 1 → Disp Tools → RETURN

Select: INTERPRET → SIGMA 1 → RETURN

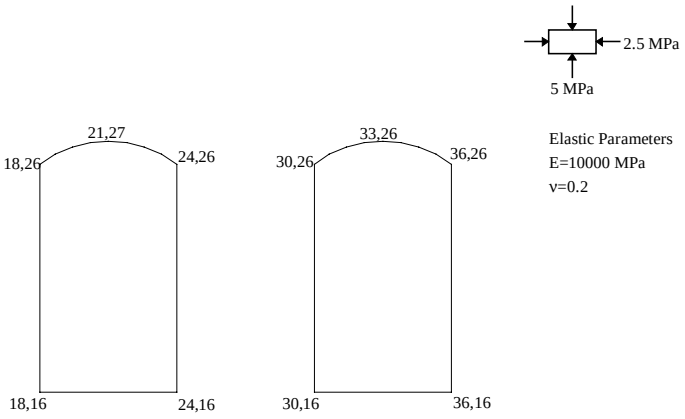
Quickly view the Strength factor results.

Select: INTERPRET → Strgth Fac

Select: INTERPRET → Strgth Fac → RETURN

Finally, notice the **USER DATA** menu option. This option is used for displaying user defined functions of stress and displacement. The actual functions are defined in a file called e2.cfg which can be found in your EXAMINE^{2D} installation directory. To add or edit a function, use a text editor to view this file and follow the directions outlined in the file. You have now finished this tutorial, exit the program by returning to the main menu and selecting **EXIT**, or by using the <*Alt X*> hotkey combination.

Fault Tutorial - Step 1



Step 1 of the Fault Tutorial can be done using either the educational or commercial versions of EXAMINE^{2D}.

If you wish to skip the model process, the finished product of this tutorial can be found in the "flt1.exa" data file located in the "tutorial" directory in your installation directory.

MODEL

The first step is to define the geometry and parameters required for the analysis. If you have not already done so, run the program by typing *examine* at the DOS prompt. Select the first option in the main menu, the **MODEL** option, by highlighting the **MODEL** button and pressing the *left mouse button* or the keyboard *Enter* key. This will place you in the modeler.

Set Limits

The first step in any computer-aided design process is setting the drawing limits of the region so that the limits encompass the excavation geometry.

Select: MODEL

Select: MODEL → SETUP

Select: MODEL → SETUP → Set Limits

```
ENTER MINIMUM X,Y COORDINATES <0,0>: 4,2
ENTER MAXIMUM X,Y COORDINATES <116.092,100>: 50,40
```

Select: MODEL → SETUP → RETURN

Entering Geometry

Now define the excavation geometry. EXAMINE^{2D} defines excavations by allowing the user to interactively draw the boundaries using the mouse or by typing in coordinates.

Proceed to the excavation geometry menu. Enter the two excavation boundaries using the following sequence.

Select: MODEL → BOUNDARIES

Select: MODEL → BOUNDARIES → ADD EXCAV

```
Enter Excavation CCW,Start Point: 18,16
To Point [c=close,a=arc,u=undo,Esc=done]: 24,16
To Point [c=close,a=arc,u=undo,Esc=done]: 24,26
To Point [c=close,a=arc,u=undo,Esc=done]: a
Number of Segments in arc <20> 8
Enter second arc point: 21,27
Enter third arc point: 18,26
To Point [c=close,a=arc,u=undo,Esc=done]: c
```

Select: MODEL → BOUNDARIES → ADD EXCAV

```
Enter Excavation CCW,Start Point: 30,26
To Point [c=close,a=arc,u=undo,Esc=done]: 30,16
To Point [c=close,a=arc,u=undo,Esc=done]: 36,16
To Point [c=close,a=arc,u=undo,Esc=done]: 36,26
To Point [c=close,a=arc,u=undo,Esc=done]: a
Number of Segments in arc <8> Enter
Enter second arc point: 33,27
Enter third arc point: c
```

Discretization

The next step is to discretize the line segments making up the openings, into boundary elements. To do this:

Select: MODEL → BOUNDARIES → DISCRETIZE

Select: MODEL → BOUNDARIES → DISCRETIZE → Chge Deflt

```
Excavation Default Number of Elements <50>: 68
```

After using **Chge Deflt**, you must still select the **DEFAULT** option to *apply* the new default discretization. EXAMINE[2D] will now discretize the geometry into (approximately) 68 boundary elements, instead of the former default value of 50.

Select: MODEL → BOUNDARIES → DISCRETIZE → DEFAULT

This process will create 68 boundary elements around the perimeter of the two excavations.

Boundary Conditions

After you have finished discretizing the excavation geometry, return to the **BOUNDARIES** menu to set the boundary conditions on the excavation surface.

Select: MODEL → BOUNDARIES → DISCRETIZE → RETURN

Select: MODEL → BOUNDARIES → BNDRY COND

Select: MODEL → BOUNDARIES → BNDRY COND → DEFAULT

You have now applied the default boundary conditions for excavation elements – zero normal and shear stress on the surface of the elements. (Remember that the default boundary conditions which are applied by the **DEFAULT** option can always be changed through the **Chge Deflt** option).

Stress Grid

You have now finished defining the excavation geometry. The next step is to define the locations within the rock mass where stresses and displacements are to be calculated. In EXAMINE[2D], stresses can either be calculated within a user defined rectangular region at constant grid spacing or at constant intervals along a line segment. In this example we will define a rectangular grid of stress points which encompass the entire geometry.

Select: MODEL → BOUNDARIES → BNDRY COND → RETURN

Select: MODEL → BOUNDARIES → RETURN

Select: MODEL → STRESS GRD

Select: MODEL → STRESS GRD → ADD GRID

```
Select Grid Corner #1: 8,6
Select Grid Corner #2: 46,36
Enter Number of Intervals in X direction <24>: 38
Enter Number of Intervals in Y direction <30>: Enter
```

Field Stress

Now proceed to enter the far field stress state by returning to the **MODEL** menu and selecting the **FLD STRESS** option. For this tutorial we will be using a CONSTANT far field stress field.

Select: MODEL → STRESS GRD → RETURN

Select: MODEL → FLD STRESS

Select: MODEL → FLD STRESS → CONSTANT

In the popup menu, use the mouse or keyboard arrow keys to select the box containing the value for Sigma 1. Type in 5 for Sigma 1. Using the same process define Sigma 3 as being 2.5. Define the angle as being 90 degrees by typing 90 in the angle box. When finished, select the Save button to save the entered values and exit the popup window.

Elastic Properties

Now define the elastic properties of the rock mass.

Select: MODEL → FLD STRESS → RETURN

Select: MODEL → Elast Prop

In the popup menu define the Young's modulus as being 10000 MPa and the poisson ratio equal to 0.2. When finished, select the Save button to save the entered values and exit the popup window.

Job Title

Although it is not required, enter a job description for this example.

Select: MODEL → Job Title

```
Enter Job Description: Fault Tutorial - Step 1
```

You have now completed the modeling phase of the analysis, return to the main menu to save the model, then run the analysis.

Select: MODEL → RETURN

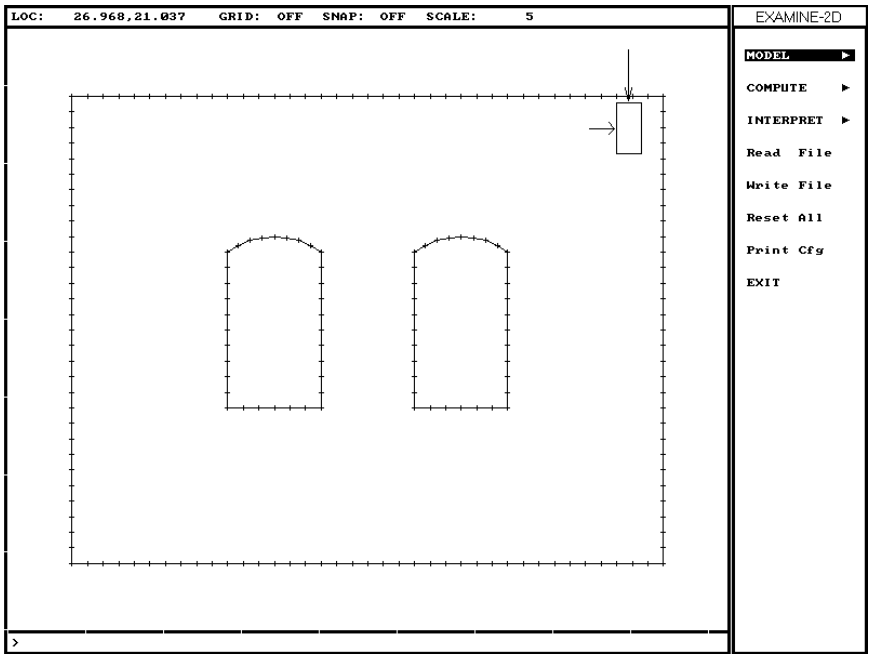


Figure T2-1: Finished model – EXAMINE^{2D} Fault Tutorial – Step 1

COMPUTE

Run compute.

Select: COMPUTE



Select 'Yes' with the mouse, or type 'y' — if you have not saved the file you will be asked for a filename and the program will proceed with saving the file for you.

```
Output filename or select w/ mouse[]: fault1
```

The program will now proceed in running the analysis. During the processing, the screen is updated with the progress of the analysis. When completed, the program returns you to the main EXAMINE^{2D} menu and you are ready to view the results.

INTERPRET

Select: INTERPRET

Select: INTERPRET → SIGMA 3

Alter the range of this plot to <0 , 6>, and then toggle on the display of Input Data by selecting the **Input Data** option from the **SIGMA 3** menu. (Note: pressing *any key* will return you from the Input Data display back to the SIGMA 3 menu.)

Select: INTERPRET → SIGMA 3 → Alter Rnge

```
Enter Minimum, Maximum Range Values <0,3.6>: 0,6
```

Select: INTERPRET → SIGMA 3 → Input Data

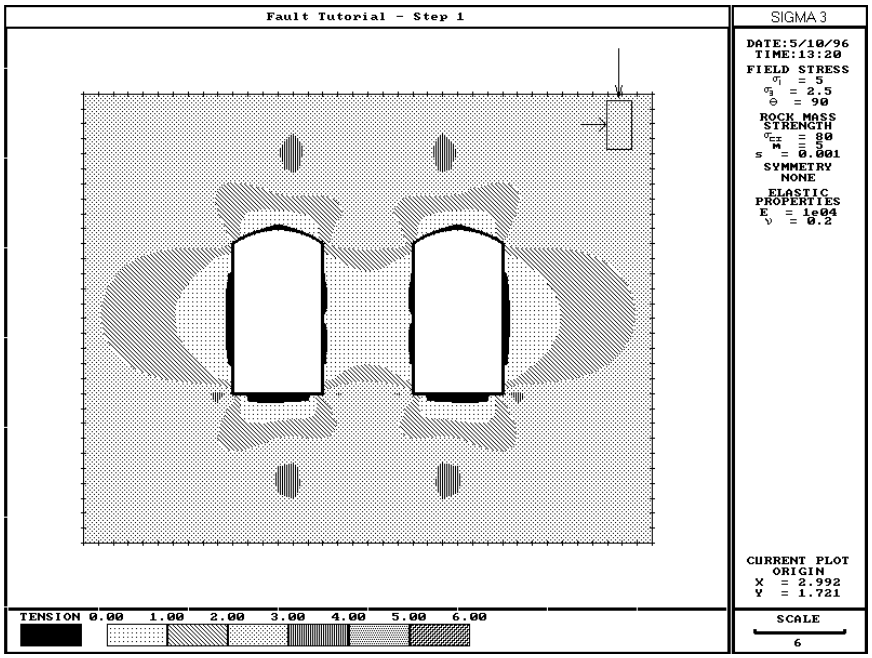
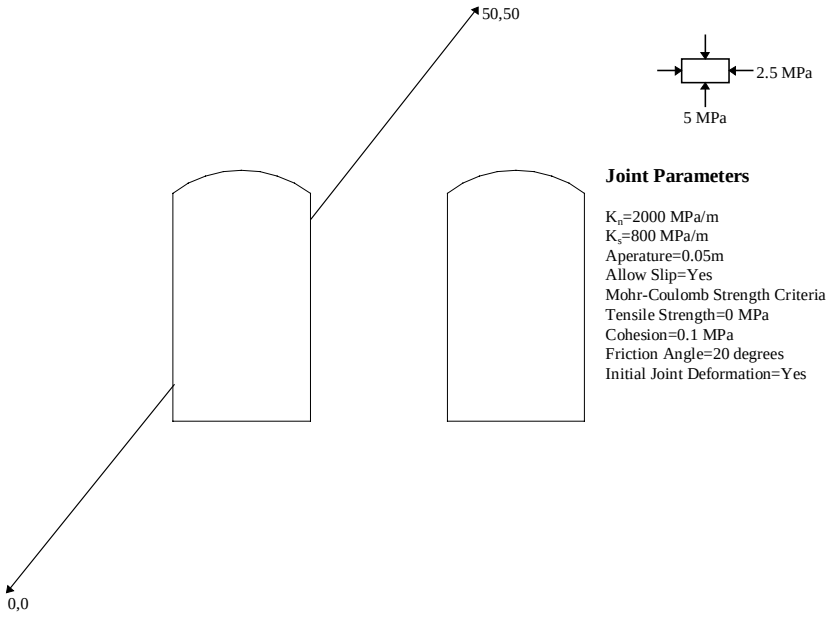


Figure T2-2: Minor Principal Stress Contours, Fault Tutorial – Step 1

Follow the instructions in the previous tutorial for using the Interpreter. In particular, view the principal stress results and total displacements. Notice how the stress contours are symmetric (see above SIGMA 3 results).

Within the **TOTAL** displacement menu try toggling on the displacement trajectories (**Displ Traj** option) to see the displacement pattern around the excavations. When you have finished looking at the results, you have completed the first step in this tutorial. Exit the program by returning to the main menu and selecting **EXIT**, or by using the <*Alt X*> hotkey combination.

Fault Tutorial - Step 2



Step 2 of the Fault Tutorial can only be done with the commercial version of EXAMINE^{2D}.

If you wish to skip the model process, the finished product of this tutorial can be found in the "flt2.exa" data file located in the "tutorial" directory in your installation directory.

If you have not completed Step 1 of this tutorial you must either do so, or copy the flt1.exa data file from the "tutorial" directory to your current working directory, and rename it fault1.exa.

MODEL

In this example we will model a fault intersecting one of the excavations from Step 1 of this tutorial. The first step is to start EXAMINE^{2D} and read in the data file from Step 1 of this tutorial. This can be done in two ways. The first method is to give the name of the file you wish to read as a command line parameter when you run EXAMINE^{2D} from the DOS prompt – i.e. type:

examine fault1

at the DOS prompt. The second method uses the **Read File** option in the main menu to allow the user to interactively select the file they wish to read. In this tutorial we will use the second method.

Run the EXAMINE^{2D} program by typing *examine* at the DOS prompt. From within the main menu read the "fault1" data file created in Step 1 of this tutorial.

Select: Read File

```
Input filename or select w/ mouse []: fault1
```

You may either type the name of the file in the prompt line, or use the mouse to interactively select the file. To use the mouse, move the mouse pointer to the file you wish to read. Notice that all the EXAMINE^{2D} files in your current directory are listed in the drawing window. Now select the file by pressing the left mouse button.

Add Joint

Now that EXAMINE^{2D} has finished reading the data file from Step 1, proceed into the modeler where you will define the fault geometry and fault contact parameters.

Select: MODEL

Select: MODEL → BOUNDARIES

Select: MODEL → BOUNDARIES → Add Joint

```
Enter Joint, Start Point [v=vrtx snap]: 0,0
To Point [v=vrtx snap,u=undo,Esc=done]: 50,50
To Point [v=vrtx snap,u=undo,Esc=done]: Esc
```

Now set the fault contact parameters by editing the values in the popup window. Set the normal stiffness of the fault to be 2000 MPa/m and the shear stiffness of the fault to be 800 MPa/m. Set the fault aperture (thickness) to be 0.05 m, and toggle the Allow Slip parameter to Yes. Keep the strength criteria as Mohr-Coulomb, keep the tensile strength at 0 and set the cohesion to be 0.1 MPa and the friction angle to be 20 degrees. Keep the Initial Joint Deformation parameter at Yes. Now select the Save button when finished.

In this example, the fault has an infilling of clay gouge with a Young's modulus of 0.1 Gpa. Using the relationships found in Crouch and Starfield (1983),

$K_n = E/h$
$K_s = G/h$

where,

E=Young's modulus of the infilling material
G=Shear modulus of the infilling material=E/(2(1+ν))
ν=poisson ratio of infilling material
h=fault aperture

the values of the fault normal and shear stiffness can be determined from the Young's modulus of the infilling material. Relationships found in Hoek (1988), Hoek (1994), and Hoek et.al (1995) can also be used to determine the Young's modulus of the infilling material based on RMR or GSI. The aperture or thickness of the fault is used to define the maximum amount of closure that can exist between the two faces of the discontinuity. When the Allow slip parameter is NO, the joint or fault will only deform elastically (no slip). Setting the parameter to YES will allow plastic deformation to occur (slip). The strength criteria are used only when the allow slip parameter is set to YES. The strength characteristics, either Mohr-Coulomb (Hoek 1990) or Barton-Bandis (Barton and Bandis 1990), of the joint can also be determined from field or laboratory measurements. The initial joint deformation parameter specifies whether the fault deforms due to the influence of the far field stresses. If this parameter is set to NO, the fault deformation is only a result of the induced stresses caused by the excavation of the two openings. Refer to Crouch and Starfield (1983) for a more in depth discussion on the definition of the various fault properties and the numerical implementation of faults and joints using displacement discontinuities.

After saving the fault contact properties you will notice that the fault was "clipped" to the excavation boundaries, thus removing the segment of fault lying within the opening – EXAMINE2D automatically clips any fault that intersects an excavation boundary.

(Note: You may later change the properties along any segment of the fault, by using the "BNDRY COND ⩾ One Line" option to select the fault line segment(s) that you wish to change. After you have selected the line segments, a popup menu will allow you to change the above fault contact parameters. The line segments will be highlighted to indicate that they have non-default contact properties.)

Re-Discretization

Notice that when the fault was added, the current discretization of the excavations was deleted. (Notice that the boundary conditions on the surface of the excavations are NOT deleted.) As a result, you need only re-discretize the geometry to complete the addition of the fault.

Select: MODEL → BOUNDARIES → DISCRETIZE

Select: MODEL → BOUNDARIES → DISCRETIZE → Chge Deflt

```
Excavation Default Number of Elements <68>: Enter
Joint Default Number of Elements <50>: 62
```

Select: MODEL → BOUNDARIES → DISCRETIZE → DEFAULT

This discretizes the fault with elements of approximately the same length as the excavation elements (~1m). Notice as well, that the stress grid interval is also about the same size. Although not strictly required, following this rule of thumb will help in producing a good set of results.

Job Title

Now let's return to the main model menu, change the job description, and run the analysis.

Select: MODEL → BOUNDARIES → DISCRETIZE → RETURN

Select: MODEL → BOUNDARIES → RETURN

Select: MODEL → Job Title

```
Enter Job Description: Fault Tutorial - Step 2
```

Select: MODEL → RETURN

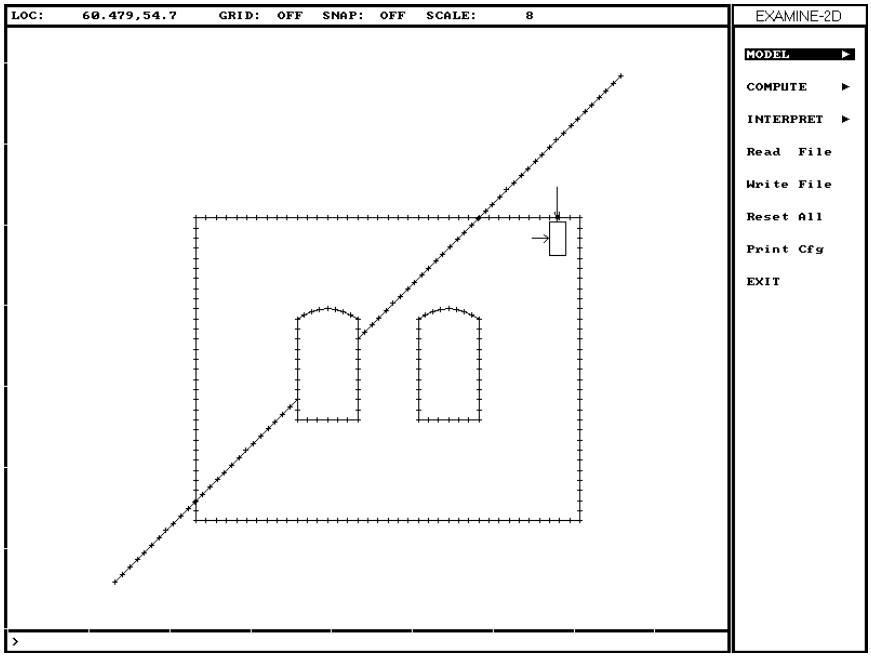


Figure T2-3: Finished model – EXAMINE^{2D} Fault Tutorial – Step 2

COMPUTE

Select: COMPUTE



Select 'Yes' with the mouse, or type 'y' — if you have not saved the file you will be asked for a filename and the program will proceed with saving the file for you.

```
Output filename or select w/ mouse[]: fault2
```

The program will now proceed in running the analysis. When completed, the program returns you to the main EXAMINE^{2D} menu and you are ready to view the results.

INTERPRET

Select: INTERPRET

Select: INTERPRET → SIGMA 3

As in Step 1 of this tutorial, alter the range of this plot to <0 , 6>, so that you can more easily compare plots.

Select: INTERPRET → SIGMA 3 → Alter Rnge

```
Enter Minimum, Maximum Range Values <0,3.3>: 0,6
```

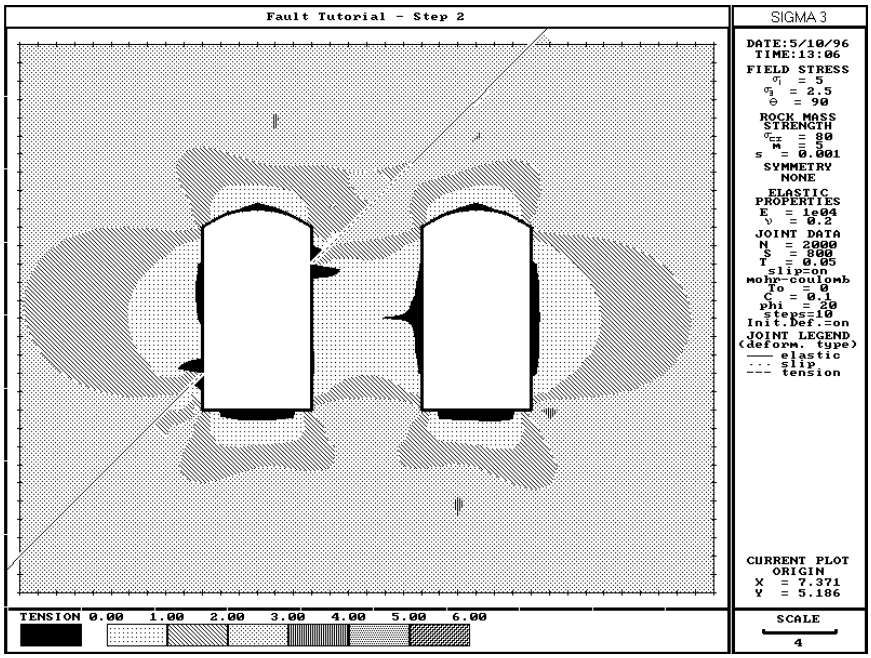


Figure T2-4: Minor Principal Stress Contours, Fault Tutorial - Step 2

Follow the instructions in the simple tutorial for using the Interpreter. In particular, view the principal stress results and total displacements and compare them to the results in Step 1. Notice how the stress contours are no longer symmetric (see above SIGMA 3 results).

Also notice the regions of fault slip, in immediate proximity to the fault-excavation contact. This is indicated by the *dotted* portion of the fault line in Figure T2-4 – see the **Input Data** legend. Note that on a colour monitor, the extent of fault slip will be shown as a dark gray, rather than dotted, line.

Within the **TOTAL** displacement menu try toggling on the displacement trajectories (**Disp Traj** option) to see the displacement pattern around the excavations. When you have finished looking at the results, you have completed the second step in this tutorial. Exit the program by returning to the main menu and selecting **EXIT**, or by using the <*Alt X*> hotkey combination.

References for Tutorial 2

Barton, N. and Bandis, S. (1990), *Review of predictive capabilities of JRC-JCS model in engineering practice*, Rock Joints (ed. Barton and Stephansson), Balkema, Rotterdam, 603-610.

Crouch, S. and Starfield, A. (1983), *Boundary element methods in solid mechanics*, GeorgeAllen & Unwin, London.

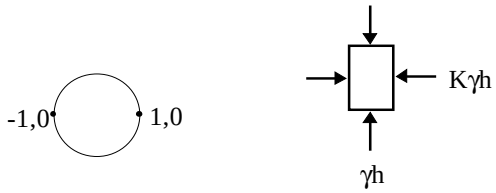
Hoek, E. and Brown, E.T. (1988), *The Hoek-Brown failure criterion - a 1988 update*, Proc. 15th Canadian rock mechanics symposium (ed. Curran), Toronto: Dept. Civ. Engineering, University of Toronto, 31-38.

Hoek, E. (1990), *Estimating Mohr-Coulomb friction and cohesion values from the Hoek-Brown failure criterion*, Intl. J. Rock Mech. & Mining Sci. & Geomechanics Abstracts, 12(3), 227-229.

Hoek, E. (1994), *Strength of rock and rock masses*, ISRM News Journal, 2(2), 4-16.

Hoek, E., Kaiser, P., Bawden, W. (1995), *Support of underground excavations in hard rock*, Balkema, Rotterdam.

Near Surface Tutorial - Step 1



ELASTIC PROPERTIES
E=50 MPa
ν=0.25

STRENGTH PROPERTIES
Tensile Strength=0 MPa
cohesion=0.04 MPa
friction angle=38°

FAR-FIELD STRESS
Gravitational Stress Field
Ground Surface Elevation=4m
Unit Weight=γ=0.02 MPa/m
Lateral Pressure Coefficient=K=0.5

This tutorial step can only be done with the commercial version of EXAMINE[2D].

If you wish to skip the model process, the finished product of this tutorial can be found in the "ns1.exa" data file located in the "tutorial" directory in your installation directory.

MODEL

In this tutorial we will model a circular tunnel, 2 meters in diameter, with it's center 4 meters below a horizontal ground surface. The first step is to define the geometry and parameters required for the analysis. If you have not already done so, run the program by typing *examine* at the DOS prompt. Select the first option in the main menu, the **MODEL** option, by highlighting the **MODEL** button and pressing the *left mouse button* or the keyboard *Enter* key. This will place you in the modeler.

The first step in any computer-aided design process is setting the drawing limits of the region so that the limits encompass the excavation geometry.

Set Limits

Select: MODEL

Select: MODEL → SETUP

Select: MODEL → SETUP → Set Limits

```
ENTER MINIMUM X,Y COORDINATES <0,0>: -10,-10
ENTER MAXIMUM X,Y COORDINATES <116.092,100>: 10,10
```

Select: MODEL → SETUP → RETURN

Entering Geometry

Now define the excavation geometry. EXAMINE^{2D} defines excavations by allowing the user to interactively draw the boundaries using the mouse or by typing in coordinates.

Proceed to the excavation geometry menu. Enter the circular excavation boundary using the following sequence. Observe how the circle is being formed as you enter the values in the prompt lines.

Select: MODEL → BOUNDARIES

Select: MODEL → BOUNDARIES → ADD EXCAV

```
Enter Excavation CCW,Start Point: -1,0
To Point [c=close,a=arc,u=undo,Esc=done]: a
Number of Segments in arc <20> 60
Enter second arc point: 1,0
Enter third arc point: c
```

Defining the Ground Surface

The next step is to define the ground surface. In EXAMINE^{2D} there are two different methods for defining a ground surface. The first method uses a numerical solution for a half space to model the ground surface. The advantage of this method is that the ground surface is not explicitly modeled using boundary elements, thus reducing the computational requirements of the analysis. The disadvantage is that only horizontal surfaces can be modeled using this method. If the ground surface is not horizontal you must use the second method, the discretized surface. This method uses boundary elements, with zero stress boundary conditions, to model the surface. In Step 1 of this tutorial the ground surface is horizontal so we will be using the half space method. Step 2 of this tutorial will explore the use of the discretized surface.

Select: MODEL → BOUNDARIES → Add Surf

Select: MODEL → BOUNDARIES → Add Surf → Half Space

```
Enter elevation of surface [0]: 4
```

Select: MODEL → BOUNDARIES → Add Surf → RETURN

Discretization

The next step is to discretize the line segments making up the circular opening, into boundary elements. To do this:

Select: MODEL → BOUNDARIES → DISCRETIZE

Select: MODEL → BOUNDARIES → DISCRETIZE → DEFAULT

This process will create 60 boundary elements around the perimeter of the opening. (Since the circular excavation consists of 60 line segments, each line segment defines a single boundary element).

Boundary Conditions

After you have finished discretizing the excavation geometry, return to the **BOUNDARIES** menu to set the boundary conditions for the excavation.

Select: MODEL → BOUNDARIES → DISCRETIZE → RETURN

Select: MODEL → BOUNDARIES → BNDRY COND

Select: MODEL → BOUNDARIES → BNDRY COND → DEFAULT

The default boundary condition for excavation elements is zero normal and shear stress on the surface of the elements.

Stress Grid

You have now finished defining the excavation geometry. The next step is to define the locations within the rock mass where stresses and displacements are to be calculated. In EXAMINE^{2D}, stresses can either be calculated within a user defined rectangular region at constant grid spacing or at constant intervals along a line segment. In this example we will define a rectangular grid of stress points which encompass the entire geometry.

Select: MODEL → BOUNDARIES → BNDRY COND → RETURN

Select: MODEL → BOUNDARIES → RETURN

Select: MODEL → STRESS GRD

Select: MODEL → STRESS GRD → ADD GRID

```
Select Grid Corner #1: -4,-4
Select Grid Corner #2: 4,3
Enter Number of Intervals in X direction <22>: 50
Enter Number of Intervals in Y direction <43>: Enter
```

Autoscale

To automatically change the viewport of the drawing window so that the entire model is visible and fits nicely, press the F2 function key. This is equivalent to entering the **Toolbox** and selecting the **Autoscale** option. As seen in Tutorial 1, the function keys act as shortcuts to many of the **Toolbox** options.

Field Stress

Now proceed to enter the far field stress state by returning to the **MODEL** menu and selecting the **FLD STRESS** option. For this tutorial, since we are in close proximity to the ground surface, we will use a gravitational far field stress.

Select: MODEL → STRESS GRD → RETURN

Select: MODEL → FLD STRESS

Select: MODEL → FLD STRESS → GRAVITY

In the popup menu, use the mouse or keyboard arrow keys to select the box containing the value for Unit Weight. Type in 0.02 for the unit weight. Using the same process define the horizontal stress ratio as being 0.5. Notice that the ground surface elevation has been automatically set to the elevation you specified for the half space. When finished, select the Save button to save the entered values and exit the popup window.

Failure Criterion

Let us now define a set of strength properties for the material being modeled.

Select: MODEL → FLD STRESS → RETURN

Select: MODEL → Failr Crit → Mohr-Coul

In the popup menu define the tensile strength as being 0.0, the cohesion as 0.04 MPa and the friction angle equal to 38 degrees. When finished, select the Save button to save the entered values and exit the popup window.

Elastic Properties

Now define the elastic properties of the rock mass.

Select: MODEL → Failr Crit → RETURN

Select: MODEL → Elast Prop

In the popup menu define the Young's modulus as being 50 MPa and keep the poisson ratio equal to 0.25. When finished, select the Save button to save the entered values and exit the popup window.

Job Title

Although it is not required, enter a job description for this example.

Select: MODEL → Job Title

```
Enter Job Description: Near Surface Tutorial - Step 1
```

You have now completed the modeling phase of the analysis. Return to the main menu to save the model, then run the analysis.

Select: MODEL → RETURN

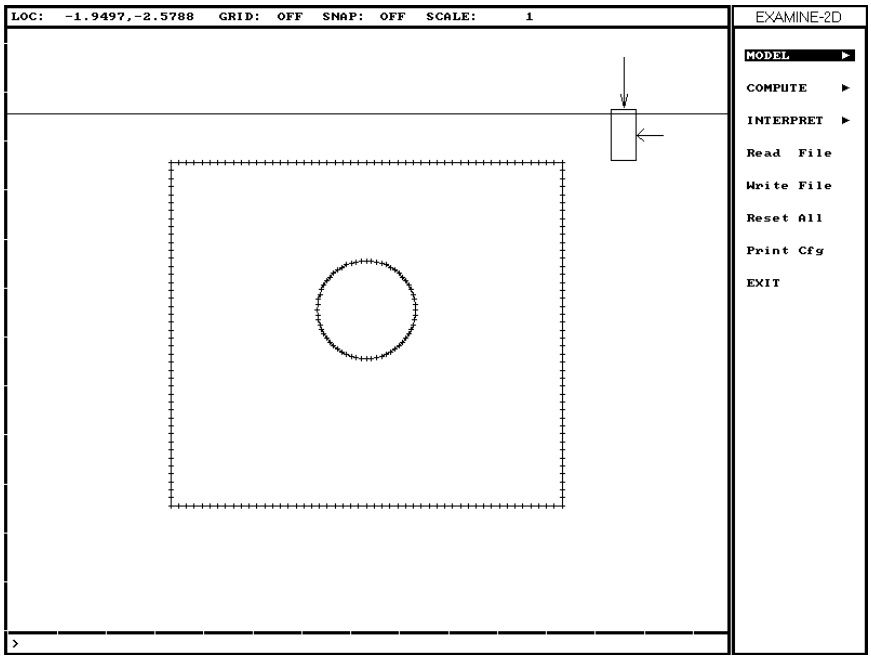


Figure T3-1: Finished model - EXAMINE^{2D} Near Surface Tutorial - Step 1

COMPUTE

Run compute.

Select: COMPUTE

```
Compute May Take Several Minutes. Are You Sure? : YES NO
```

Select 'Yes' with the mouse, or type 'y' — if you have not saved the file you will be asked for a filename and the program will proceed with saving the file for you.

```
Output filename or select w/ mouse[]: near1
```

The program will now proceed in running the analysis. During the processing, the screen is updated with the progress of the analysis. When completed, the program returns you to the main EXAMINE²ᴰ menu and you are ready to view the results.

INTERPRET

Select: INTERPRET

Select: INTERPRET → Strgth Fac

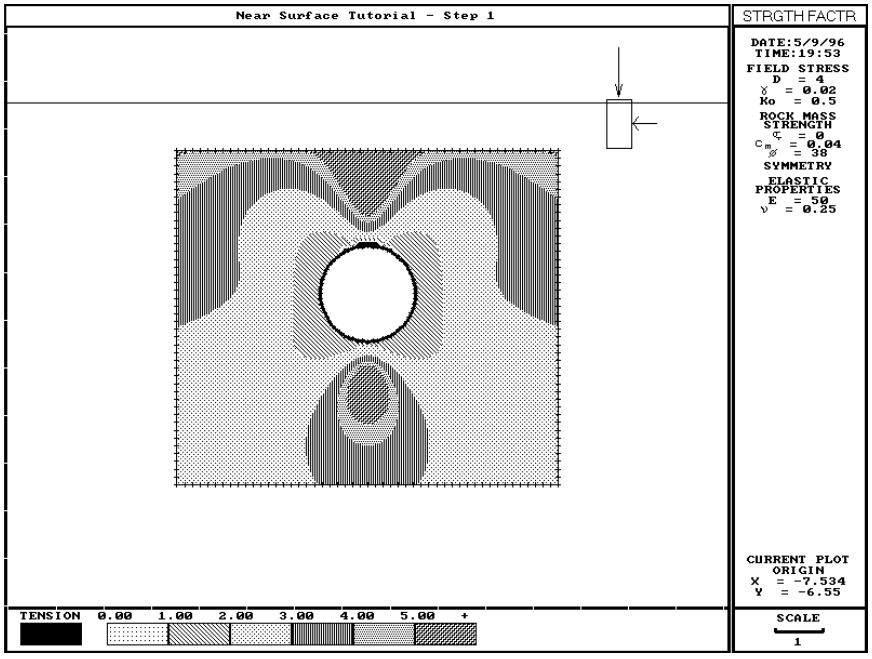


Figure T3-2: Strength Factor Contours, Near Surface Tutorial - Step 1

Notice that the regions of failure, i.e. zones with a strength factor less than one, are limited to a very small area in the crown of the tunnel.

In EXAMINE^{2D}, materials always behave elastically, with the calculation of the strength factor based entirely on this elastic stress distribution. Yielding of the material due to over-stress, and the resulting redistribution of stress is **not** accounted for. As a result, we can easily modify the strength parameters within the interpreter without having to re-run the analysis.

Let us see what happens if we reduce the cohesion of the material to a quarter of it's present value.

Select: INTERPRET → Strgth Fac → Failr Crit

Select: INTERPRET → Strgth Fac → Failr Crit → Mohr-Coul

In the popup menu keep the tensile strength as being 0.0, set the cohesion as 0.01 MPa and keep the friction angle equal to 38 degrees. When finished, select the Save button to save the entered values and exit the popup window.

Now notice that the region of failure has increased substantially. Using this capability of easily changing the strength parameters, a quick parametric analysis can be performed to study the effect of different strength properties on excavation stability.

Select: INTERPRET → Strgth Fac → Failr Crit → RETURN

Other useful data visualization features in EXAMINE^{2D} include the querying of specific data values at user defined locations, and the graphing of data along a user supplied path.

Select: INTERPRET → Strgth Fac → Query Data

Select: INTERPRET → Strgth Fac → Query Data → Point Value

Use the mouse or keyboard arrow keys to position the crosshairs at some point within the stress grid, but not within the tunnel. Press the left mouse button or the keyboard Enter key. The safety factor at this location is immediately written to the screen. You may also type in coordinates at the prompt line. You can continue to enter points and display more values. When you are done, press Escape to return to the Query Data menu.

These values are calculated by using bilinear interpolation within the grid cell where the requested data point resides. Using the **Line Values** option, values along a line or polyline can be displayed and written to an ASCII text file. This text file can then be read into a spreadsheet for further processing. Graphing of data along a user supplied path, within EXAMINE^{2D}, is also possible, using the **Add Series** option, as illustrated below.

Select: INTERPRET → Strgth Fac → Query Data → Add Series

```
Enter Line Segment(s),Start Point: -4,0
To Point [u=undo,v=snap,c=close,a=arc,Esc=done]: 4,0
To Point [u=undo,v=snap,c=close,a=arc,Esc=done]: Press
   Esc key
```

Select: INTERPRET → Strgth Fac → Query Data → Graph Series

When finished looking at the graph, press any key to return to the sidebar menu. Note: parameters controlling the layout of the graph can be changed using the Graph Setup option.

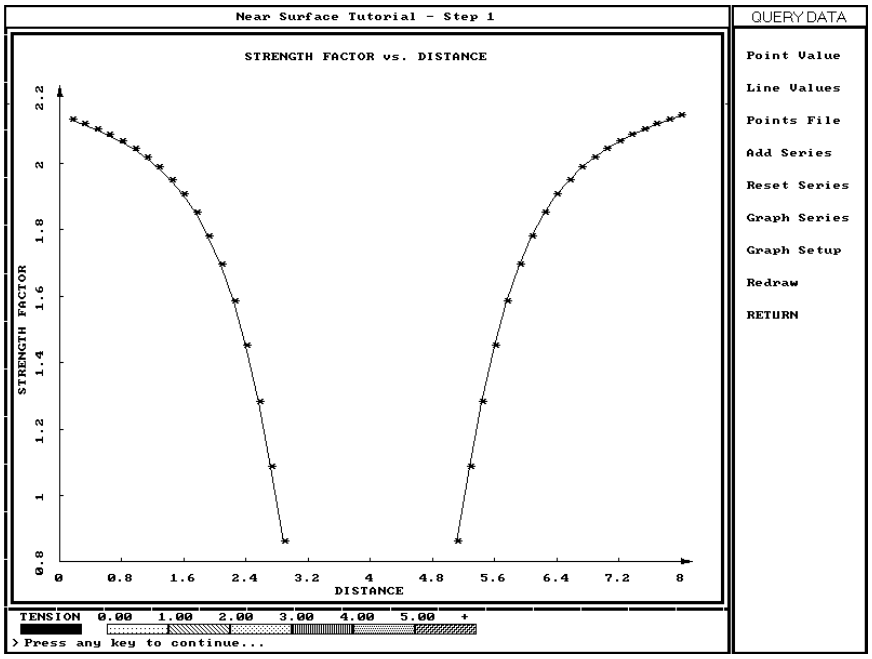
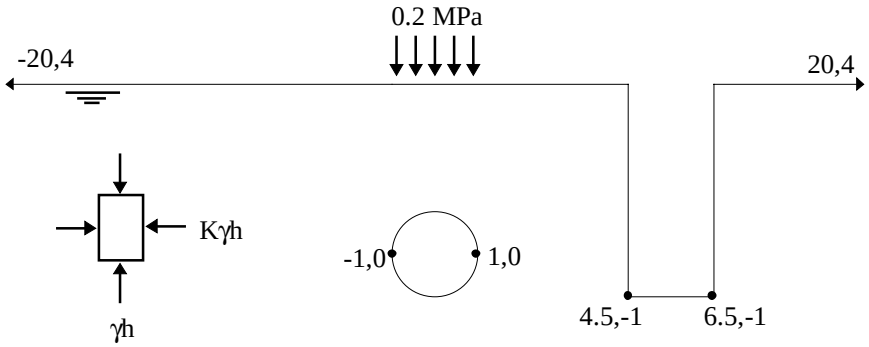


Figure T3-3: Plot of Strength Factor along a line entered with Query Data->Add Series

When you have finished looking at the results, you have completed the first step in this tutorial. Exit the program by returning to the main menu and selecting **EXIT**, or by using the <*Alt X*> hotkey combination.

Near Surface Tutorial - Step 2



ELASTIC PROPERTIES
E=50 MPa
ν=0.25

STRENGTH PROPERTIES
Tensile Strength=0 MPa
cohesion=0.04 MPa
friction angle=38°

FAR-FIELD STRESS
Gravitational Stress Field
Ground Surface Elevation=4m
Unit Weight=γ=0.02 MPa/m
Lateral Pressure Coefficient=K=0.5

This tutorial step can only be done with the commercial version of EXAMINE^{2D}.

If you wish to skip the model process, the finished product of this tutorial can be found in the “ns2.exa” data file located in the “tutorial” directory in your installation directory.

If you have not completed Step 1 of this tutorial you must either do so or copy the ns1.exa data file from the “tutorial” directory to your current working directory, and rename it near1.exa.

MODEL

In this example we will take the model created in Step 1 of this tutorial and redefine the ground surface to include a 5 meter deep trench and a uniform surface pressure directly over the circular tunnel. The first step is to start EXAMINE^{2D} and read in the data file from Step 1 of this tutorial. This can be done in two ways. The first method is to give the name of the file you wish to read as a command line parameter when you run the examine program from the DOS prompt (i.e. type "*examine near1*" at the DOS prompt). The second method uses the **Read File** option in the main menu to allow the user to interactively select the file they wish to read (see the second step of the Fault Tutorial). In this tutorial we will use the first method.

Run the EXAMINE^{2D} program and read in the data file by typing

examine near1

at the DOS prompt.

Redefining the Ground Surface

Now that EXAMINE^{2D} has finished reading the data file from Step 1, proceed into the modeler where you will first delete the current half space, then re-define the surface geometry and add a surface pressure for this tutorial.

Deleting the Existing Half Space

Select: MODEL

Select: MODEL → BOUNDARIES

Select: MODEL → BOUNDARIES → Add Surf

Select: MODEL → BOUNDARIES → Add Surf → Half Space

A popup prompt will be displayed asking whether you wish to delete the currently defined half space. Select Yes.

As mentioned in the previous tutorial, a ground surface must be entirely horizontal to model it with a half space. In this tutorial, to include the effects of the trench, we need to use the discretized surface method of modeling a free surface.

Entering a Discretized Ground Surface

Zoom Out on the drawing region by pressing the F4 keyboard function key, three or four times. This can also be done by using the **Zoom Out** option in the **Toolbox**. Now proceed to enter the new surface geometry.

Select: MODEL → BOUNDARIES → Add Surf → Discretized

```
Enter Surface (material to right), Start Point: -20,4
TO POINT [a=arc,u=undo,Esc=done]: -1,4
TO POINT [a=arc,u=undo,Esc=done]: 1,4
TO POINT [a=arc,u=undo,Esc=done]: 4.5,4
TO POINT [a=arc,u=undo,Esc=done]: 4.5,-1
TO POINT [a=arc,u=undo,Esc=done]: 6.5,-1
TO POINT [a=arc,u=undo,Esc=done]: 6.5,4
TO POINT [a=arc,u=undo,Esc=done]: 20,4
TO POINT [a=arc,u=undo,Esc=done]: Esc
```

(Note: to abort the entry of the surface, press the keyboard key combination <*Ctrl C*>. <*Ctrl C*> may also be used to abort any of the data entry prompts within the program.)

Note that as part of the ground surface polyline, a line segment which defines the position and extents of the applied surface pressure is defined. This is important since boundary pressures are applied in terms of the line entities making up the surface and/or excavation geometry, and *not* on an individual element basis.

Discretization

The next step is to discretize the line segments making up the surface into boundary elements. To do this:

Select: MODEL → BOUNDARIES → Add Surf → RETURN

Select: MODEL → BOUNDARIES → DISCRETIZE

Select: MODEL → BOUNDARIES → DISCRETIZE → Chge Deflt

```
Excavation Default Number of Elements <60>: Enter
Surface Default Number of Elements <50>: 80
```

Select: MODEL → BOUNDARIES → DISCRETIZE → DEFAULT

Boundary Conditions

The next step is to define the boundary conditions.

Select: MODEL → BOUNDARIES → DISCRETIZE → RETURN

Select: MODEL → BOUNDARIES → BNDRY COND

The default boundary condition for excavation elements is zero normal and shear stress on the surface of the elements. It is necessary to first *apply* the default boundary condition for *all* elements, prior to definition of non-default boundary conditions on individual line segments.

Select: MODEL → BOUNDARIES → BNDRY COND → DEFAULT

Surface Boundary Condition

Let us now apply a load (boundary condition) to the free surface line segment above the circular tunnel.

Select: MODEL → BOUNDARIES → BNDRY COND → One Line

```
Select Line(s) [u=undo,w=window,Esc=done]:
```

Using the mouse or keyboard arrow keys, position the selection box so that it overlays the surface line segment immediately over the circular tunnel. This is the line segment on the surface between the points -1,4 and 1,4. Now select it by pressing the left mouse button or by pressing the keyboard Enter key. You are now prompted to select another line segment which might have the same boundary conditions. Since there are none, press Esc to complete the selection process.

```
Select Line(s) [u=undo,w=window,Esc=done]: Esc
Enter Normal, Shear Stress Bndry Cond. (0,0): 0.2,0
```

You have now applied a normal surface load of 0.2 Mpa to the selected line segment. Note that this line segment is now coloured blue to distinguish it from the line segments with the default boundary condition (red colour).

Job Title

Now let's return to the main model menu, change the job description, and run the analysis.

Select: MODEL → BOUNDARIES → BNDRY COND → RETURN

Select: MODEL → BOUNDARIES → RETURN

Select: MODEL → Job Title

```
Enter Job Description: Near Surface Tutorial - Step 2
```

Select: MODEL → RETURN

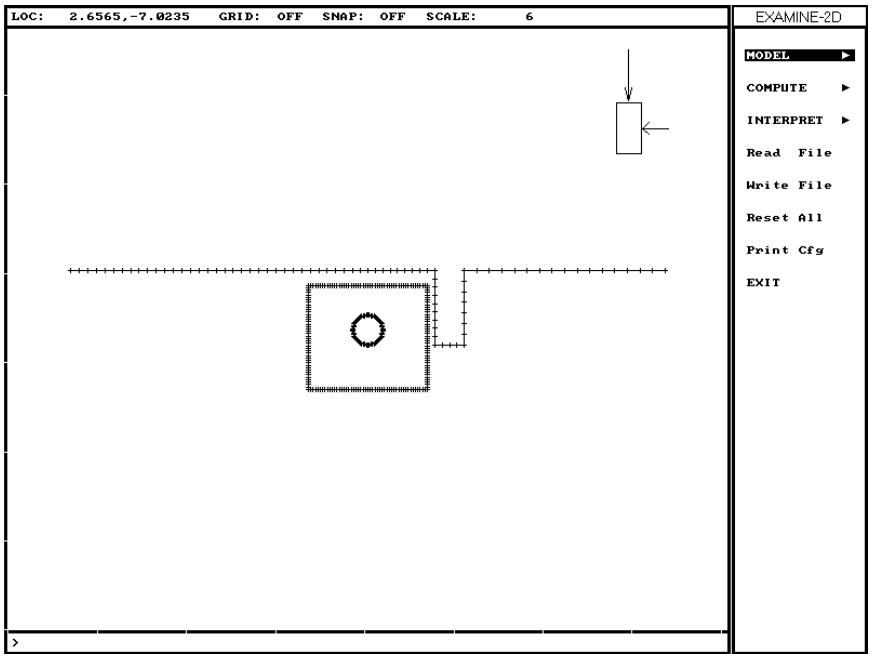


Figure T3-4: Finished model - EXAMINE^{2D} Near Surface Tutorial - Step 2

COMPUTE

Run compute.

Select: COMPUTE



Select 'Yes' with the mouse, or type 'y' — if you have not saved the file you will be asked for a filename and the program will proceed with saving the file for you.

```
Output filename or select w/ mouse[]: near2
```

The program will now proceed in running the analysis. During the processing, the screen is updated with the progress of the analysis. When completed, the program returns you to the main EXAMINE^{2D} menu and you are ready to view the results.

INTERPRET

Select: INTERPRET

Select: INTERPRET → Strgth Fac

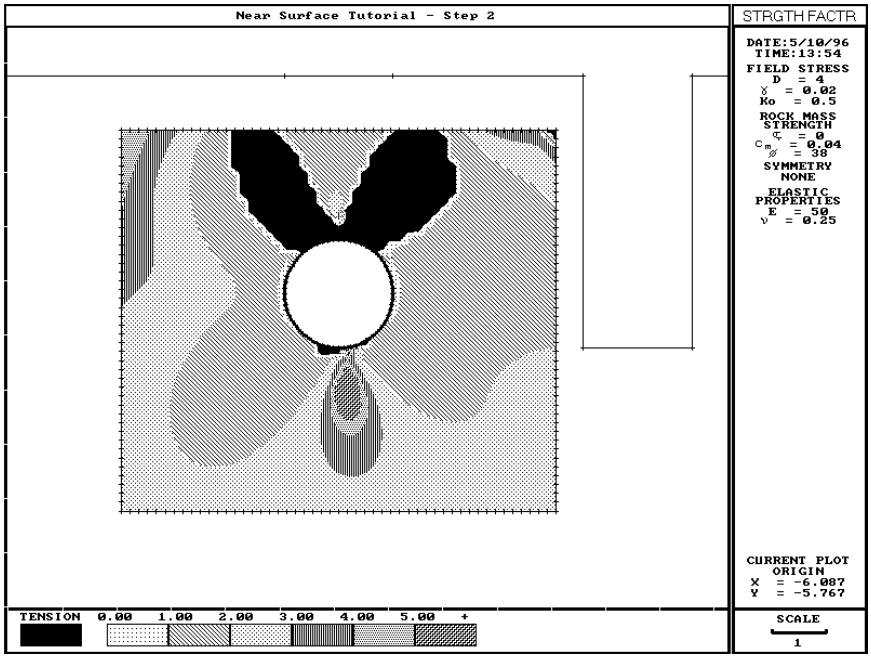


Figure T3-5: Strength Factor Contours, Near Surface Tutorial - Step 2

Zoom in on the region containing the results.

Select: INTERPRET → Strgth Fac → Zoom Win

Use the mouse or the keyboard arrow keys to define a window which encompasses the contoured stress grid of data. This window defines the new extents of the drawing region.

Notice that the regions of failure, i.e. zones with a strength factor less than one, and zones of tension, have increased substantially due to the presence of the surface load. As well, notice the influence of the trench on the distribution of strength factor between the tunnel and the trench.

When you have finished looking at the results, you have completed the second step in this tutorial. Exit the program by returning to the main menu and selecting **EXIT**, or by using the <*Alt X*> hotkey combination.