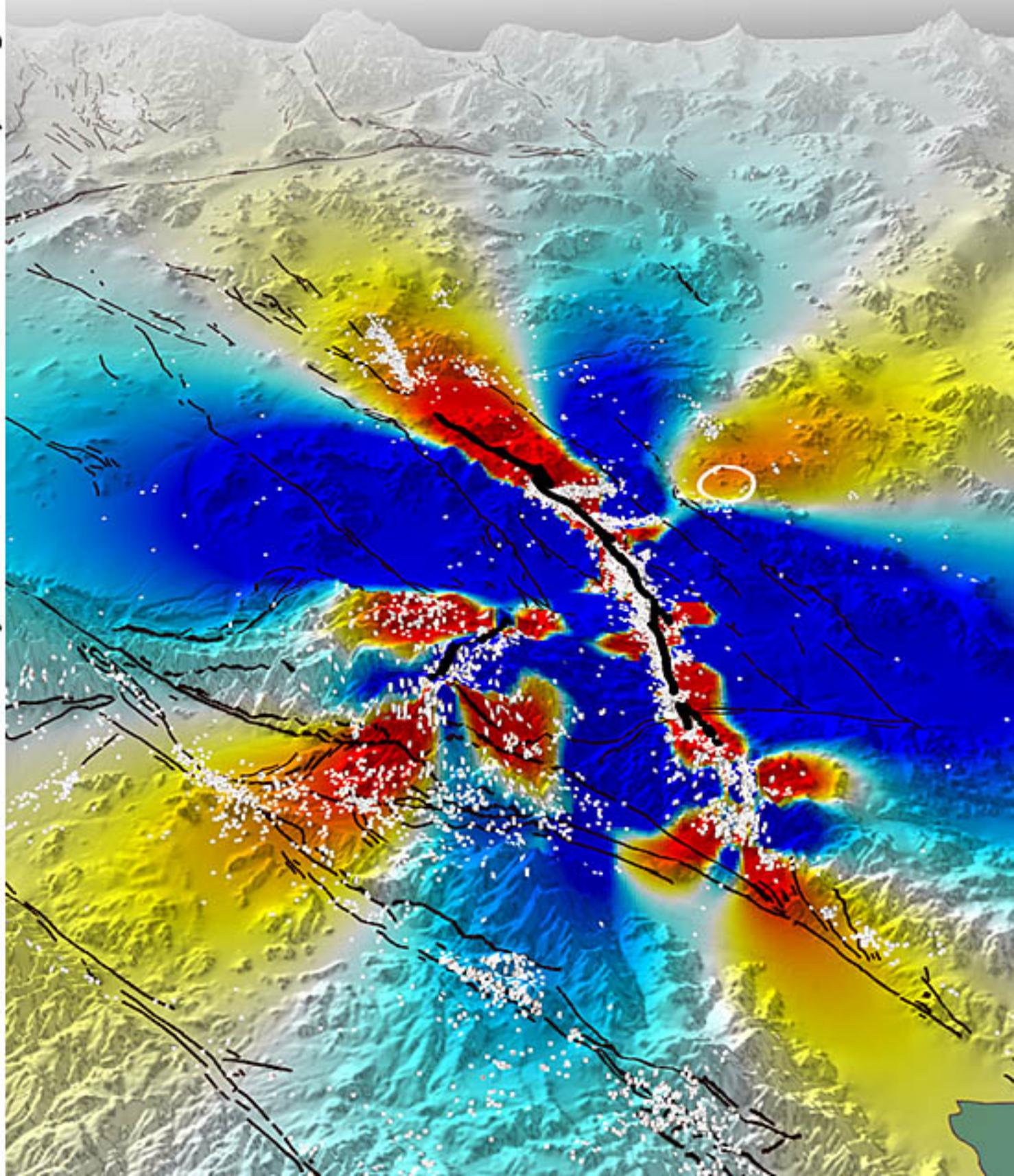


Coulomb 2.5

User
Guide

GRAPHIC-RICH DISLOCATION SOFTWARE

Shinji Toda, Ross Stein and Geoffrey King



User Guide

Coulomb 2.5 Graphic rich dislocation software

Fifth revised edition, February 27, 2003

We encourage you to use Coulomb for teaching and research. If you use it in a publication, please cite:

Toda, S., and R. S. Stein, Response of the San Andreas fault to the 1983 Coalinga-Nuñez Earthquakes: An application of interaction-based probabilities for Parkfield, *J. Geophys. Res.*, 107, 10.1029/2001JB000172, 2002.

Shinji Toda

Geological Survey of Japan, Higashi 1-1-3, Tsukuba, 305-8567, Japan

s-toda@aist.go.jp

Ross Stein

U. S. Geological Survey, 345 Middlefield Rd MS977, Menlo Park, CA94025, USA

Rstein@usgs.gov

Geoffrey King

Institut de Physique du Globe, 4, place Jussieu, 75252 Paris, France

King@ipgp.jussieu.fr

New features of Coulomb 2.5

- We now let users graphically display stresses resolved on any receiver plane. Previously, this was only available as a numerical file.
- We include an ancillary program, *Splitter*, that lets one subdivide a planar surface into any number of rectangular patches. This is very useful for inputting variable-slip source models.
- For graphic display of dilatational strains, the numerical scale automatically adjusts to the color range.
- We provide new numerical output files for some stress/strain/receiver fault combinations (all output files now identified by an '.cou' extension)

Bug fixes from former versions

- Dilatational strain exponent error in Coulomb 2.0-2.2.0 is rectified.
- For cross-section options, 'x-start' no longer has to be less than 'x-finish.'
- Supplementary information is now accepted with 'y'/'n' responses.
- Program now avoids a mathematical singularity when the calculation depth coincides with one of the fault edges (this formerly affected the graphical display only).
- Numerous typos, ambiguities, and inconsistencies in the Users Guide have –been remedied

Table of contents

<i>Table of contents</i>	<i>i</i>
Chapter 1 INTRODUCTION	1
Why Coulomb? The rationale and philosophy of this program	1
The Coulomb 2.5 Bundle contents	2
Hardware/Software requirements	2
Recommended extras	2
Two notes about this User's Guide	3
Cover images	3
Credits	3
Chapter 2 A QUICK TOUR OF COULOMB 2.5	5
Chapter 3 CONCEPTS, DIMENSIONS AND SIGN CONVENTIONS	11
To launch Coulomb 2.5	11
To quit Coulomb 2.5	11
Calculation procedure	11
To change parameters interactively	12
To save and print files	13
Prohibited procedures and actions to avoid freezes and crashes	14
Coordinates, units, and sign conventions	14
Chapter 4 INPUTMAKER AND INPUT FILES	16
Making input files with InputMaker	16
Modifying an existing file in a text editor	23
Structure and format for input files	23
Creating a new input using the <i>Data</i> > New function	24
Modifying an input file interactively from within Coulomb	25
Input file used to calculate stress on focal mechanisms	25

Chapter 5 TUTORIAL FOR DEFORMATION CALCULATIONS	27
To display the distorted grid in map view	27
To display the distorted grid in cross section	30
To display the dilatation map view	31
To display the dilatation in cross section	33
To display the horizontal displacement map view	35
To include a reference point in a horizontal vector plot	37
To display the vertical displacement in map view	38
To display the displacement cross section	38
To plot observed and predicted vectors at a specified series of points	39
To save numerical output of the strain and displacement calculations	41
A tip that will save your life (courtesy of Geof King)	42
Chapter 6 TUTORIAL FOR STRESS CALCULATIONS	43
To display the Coulomb stress change on specified planes in map view	45
To display Coulomb stress changes on the optimally oriented strike-slip, thrust, and normal faults (with the fault dip fixed) in map view	48
To save the stress calculation results as text column file	49
To plot the slip-lines (the fault planes) for the optimally-oriented strike-slip, thrust, and normal faults	51
To plot and produce a numerical output file of the principal stress axes for optimally-oriented faults	52
To plot Coulomb Stress change in Cross-section	53
To make a 3D image of the Coulomb stress change	54
Chapter 7 TUTORIAL FOR ADVANCED FEATURES	57
Resolved stress change on mapped faults	57
Graphically displaying stress changes resolved onto receiver faults	59
Freely slipping boundary elements under plane stress (Kode 105 faults)	61
Crack opening or closing (Tensile Kode 200 sources)	64

Point source of deflation or inflation (Kode 500)	65
Earthquake point source (Kode 400)	67
Resolved stresses on nodal planes of aftershocks	68
Chapter 8 OUTPUT FILES	71
Pict-formatted files of graphic images	71
Text files	71
Chapter 9 GRAPHIC TOOLS	75
Chapter 10 TAPER AND SPLITTER	77
Operation of Taper	77
Operation of Splitter	80

Chapter 1

INTRODUCTION

Why Coulomb? The rationale and philosophy of this program

Coulomb is intended both for publication-directed research and for classroom instruction. It is designed to let one calculate static displacements, strains, and stresses caused by fault slip, point sources of inflation/deflation, and dike expansion/contraction. The displacements, strains, and stresses can be on calculated at any surface at any depth.

Not surprisingly, Coulomb is optimized to investigate Coulomb stress changes on mapped faults and in aftershock files. All calculations are made in a halfspace with uniform isotropic elastic properties. The program implements elastic dislocation formulae of *Okada* [1992] and boundary element formulae of *Crouch & Starfield* [1983]. Problems such as how an earthquake promotes or inhibits failure on nearby faults, or will compress a nearby magma chamber, are germane to Coulomb. Another use would be long-term crustal deformation associated with strike-slip faults, normal faults or fault-bend folds. Finite deformation is readily visualized with the 'distorted grid' options, whereas infinitesimal deformation is mapped with displacement vectors or color gradient plots.

We believe that one learns the most when one can see the most, and when one can explore alternatives quickly and easily. So the principal feature of the program is ease of input, rapid interactive modification, and intuitive visualization of the results. The program has menus, sub-menus, check-items, and dialogue boxes to ease operation. The internal graphics are suitable for publication, and these can be easily imported into illustration and visualization programs for higher-quality figures.

The program, manual, and tutorial files are freely available from <http://quake.usgs.gov/~ross>. If you find any problems with the program or manual, please let us know. Just be patient with our replies; we get a lot of Coulomb traffic.

We encourage you to use Coulomb for your research. If you submit a scientific paper in which Coulomb was used, please cite: *S. Toda and R. S. Stein* [2002]. We would also be

grateful if you would cite *King et al.* [1994], as Coulomb is an evolution of the 1995 program, GEN, written by Geoffrey King at the Institut de Physique du Globe de Paris.

The Coulomb 2.5 Bundle contents

The Coulomb 2.5 Bundle is available either on CD-ROM or on this web site:

<http://quake.usgs.gov/research/deformation/modeling/coulomb/>

The bundle includes:

- Coulomb 2.5 standalone application (assign it ~10 MB for best results)
- InputMaker (to make input files automatically or graphically)
- Taper 1.0 (to make input files with smoothly tapering slip)
- EditII (a simple shareware text-editor to make or modify input files manually)
- UTM+ (converts between lat/lon and Cartesian coordinates; we include this program, written by James Leutgert at the USGS, with appreciation)

Hardware/Software requirements:

- A Macintosh computer with a PowerPC processor (a G3 or G4 Mac is recommended), running MacOS 7.0 or higher (MacOS 9.2 is better).
- At least 4.7 MB of RAM available to allocate to Coulomb; 10-12 MB is better.
- A color monitor of at least 600 x 400 pixel resolution. A PowerBook screen (1024 x 768 pixels) is fine.
- Microsoft Excel, to read tab-delimited text files (some output files are created as Excel documents).

Recommended extras:

- A good vector illustration program, such as Adobe Illustrator, is a must.
- A color raster-imaging application, such as Noseys (www.rsinc.com/noesys), which includes Transform for 2D color gradient, contour, or vector plots; and T3D for 3D visualization. Transform makes beautiful figures that are a fraction of the size (in Kb) of Coulomb's graphic pict files, and permits useful manipulation of the data matrices. We highly recommend Transform.
- BBEdit (BBEdit Lite is free) is better than EditII for editing input files, because you can distinguish spaces from tabs. Regardless of the editor, always use a non-kerning (uniform-spacing) font, such as Monaco, so that numbers stay properly aligned.

Two notes about this User's Guide

1. Any particularly important words of wisdom in the manual are printed in red.
2. The images used in the manual to show you what you will see on the screen were captured at low resolution (72 dpi). *But don't worry:* As you'll learn, its easy to zoom in and save much higher-resolution images from the program.

Cover images

1. The front cover image shows the Coulomb stress change caused by 1992 M=7.3 Landers, M=6.5 Big Bear and M=6.1 Joshua Tree earthquakes. See *King et al.* [1994]. The stress changes, faults and earthquakes were draped over a digital elevation model (DEM) in ArcGIS.

The inner-cover gives a taste of the possible graphical output of Coulomb 2.5:

2. The *top* image (Dilatation cross-section option) was generated in Coulomb. The fault arrows were added in Adobe Illustrator.
3. The *lower left* image was created by importing the grid and fault file into Adobe Illustrator, a drawing program. The numerical Coulomb file was imported into Transform and plotted as a color gradient image; then it was imported in the drawing program. The seismicity was plotted in QPLOT and imported into Adobe Illustrator.
4. The *lower right* image was created in Coulomb (Displacement cross-section option) and imported into Illustrator, where the arrowheads were added automatically.

Credits

The cover page and PDF layout of this Coulomb 2.5 user guide has been designed by Serkan Bozkurt (USGS, Menlo Park). We thank many users for testing the functions, finding errors and suggesting improvements.

Key Papers Related to Coulomb

Crouch, S.L., and A.M. Starfield, *Boundary Element Methods in Solid Mechanics*, 322 pp., Allen Unwin, London, 1983.

King, G.C.P., R.S. Stein, and J. Lin, Static stress changes and the triggering of earthquakes, *Bull. Seismol. Soc. Amer.*, 84 (3), 935-953, 1994.

(<http://quake.wr.usgs.gov/research/deformation/modeling/papers/landers.html>)

Okada, Y., Internal deformation due to shear and tensile faults in a half-space, *Bull. Seismol. Soc. Amer.*, 82 (2), 1018-1040, 1992.

Toda, S., R.S. Stein, P.A. Reasenber, and J.H. Dieterich, Stress transferred by the $M_w=6.5$ Kobe, Japan, shock: Effect on aftershocks and future earthquake probabilities, *J. Geophys. Res.*, 103, 24,543-24,565, 1998.

(<http://quake.usgs.gov/research/deformation/modeling/papers/kobe.html>)

Toda, S., and R.S. Stein, Response of the San Andreas fault to the 1983 Coalinga-Nuñez Earthquakes: An application of interaction-based probabilities for Parkfield, *J. Geophys. Res.*, 107, 10.1029/2001JB000172, 2002.

(<http://quake.wr.usgs.gov/research/deformation/modeling/papers/parkfield/parkfield.html>)

Toda, S., R.S. Stein and T. Sagiya, Evidence from the 2000 Izu Islands swarm that seismicity is governed by stressing rate, *Nature*, 419, 58-61, 2002.

(<http://sicarius.wr.usgs.gov>)

Stein, R.S., Earthquake conversations, *Scientific American*, v. 288 (1), 72-79, January 2003.

(e-mail rstein@usgs.gov for a PDF or reprint)

Chapter 2

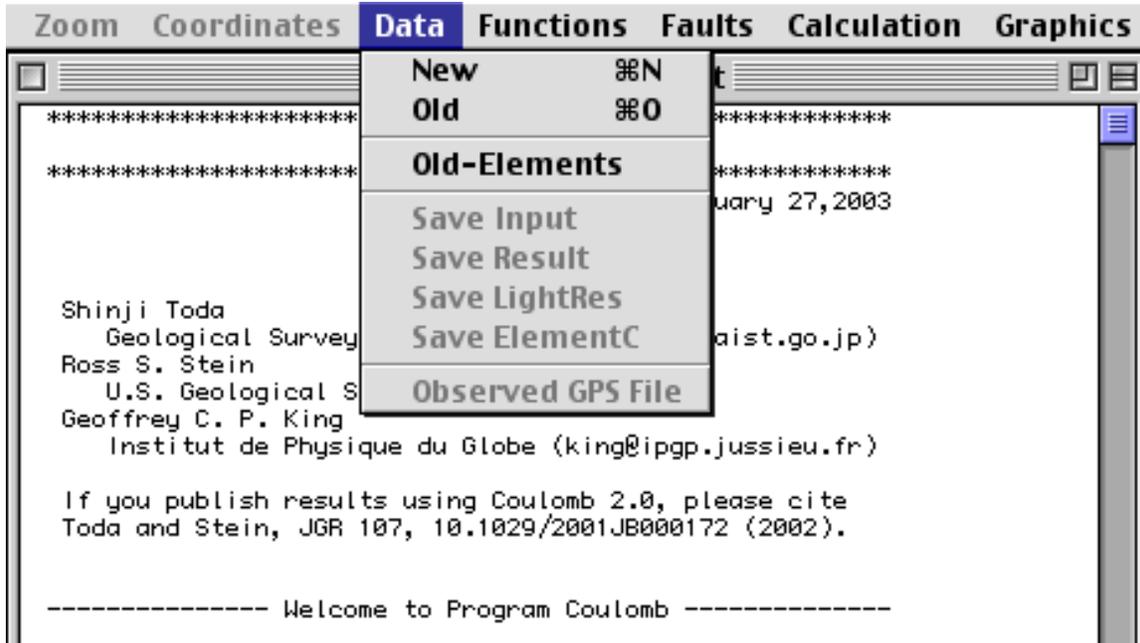
A QUICK TOUR OF COULOMB 2.5

To introduce this software, we walk you through a simple example. Here we just want you to learn the procedure to display and output graphics, and to save the results. Since this program is not a completely Mac graphical-user-interface ‘event-loop’ environment, unexpected input can cause Coulomb to crash. Very rarely, a Coulomb crash will require you to restart the Mac. So we recommend that you save all your current working documents before you start.

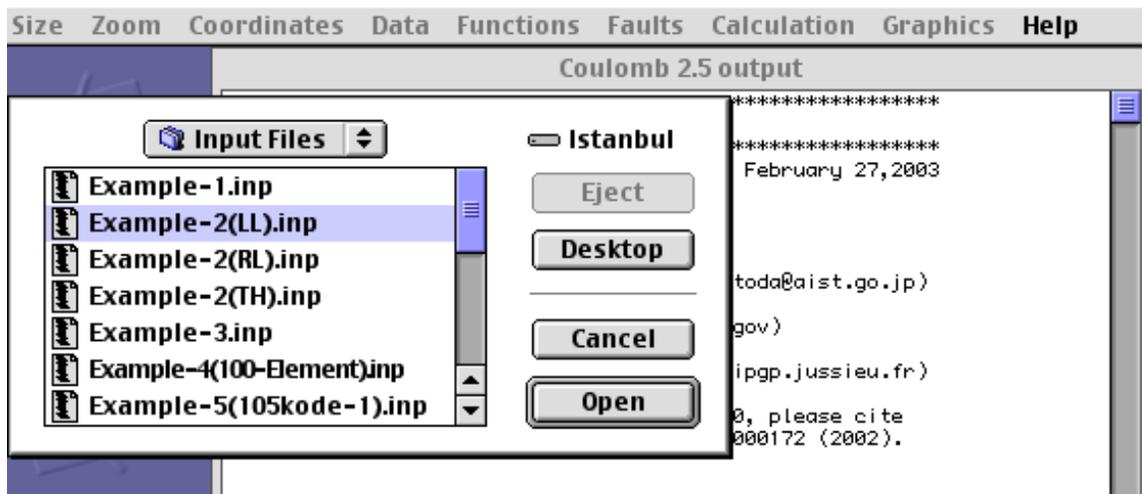
Menus and their pull-down items turn from gray to black as they become possible options. So, for example, *Functions* are gray until one selects an input file. *Faults* remain gray until one selects a *Function* that requires an additional *Fault* selection. Selections in *Functions* or *Faults* that require additional input produce pop-up *Extra Input* boxes that disappear when the data is provided. These features simplify life considerably.

To get started this software, you just need Coulomb and the ‘Input Files’ folder. You’ll soon learn how to make you own input files.

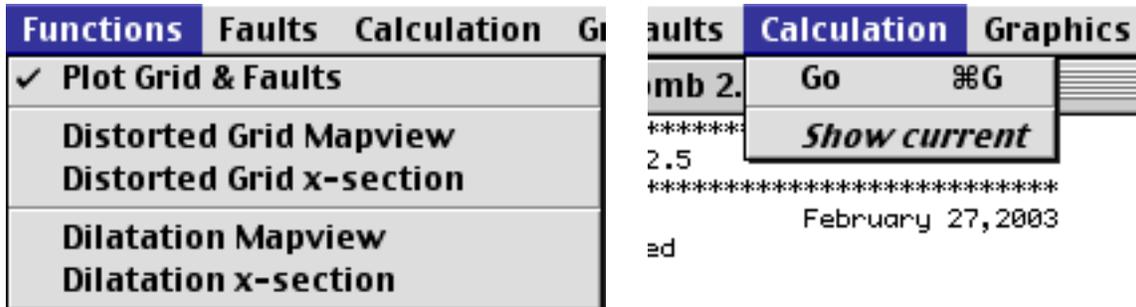
1. To launch Coulomb, double-click the Coulomb icon. A menu bar and “Coulomb 2.5 output” window appear.



2. Choose *Data* > Old.
3. Choose the input file "Example-2(LL).inp" from the dialog box, then <rtm>.

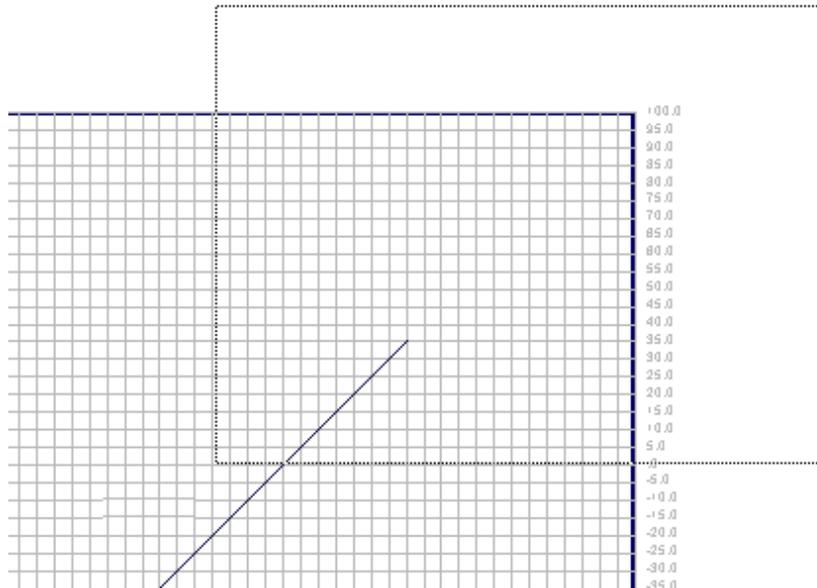
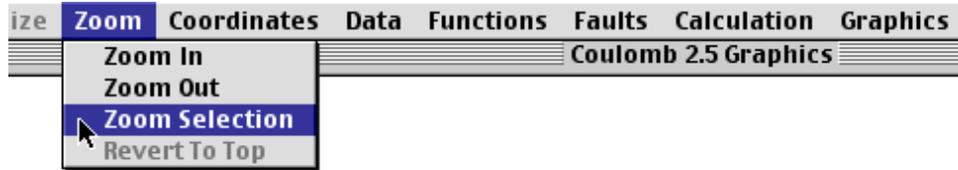


4. Choose *Functions* > **Plot Grid & Faults**. This is a check-mark item. Make sure that "Grid" is checked. Then, choose *Calculation* > **Go**.

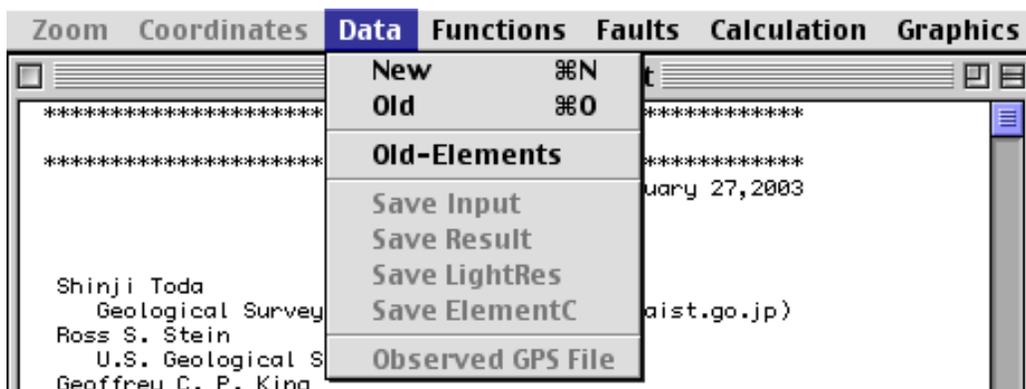


5. The calculation is performed and the program now displays the result in the “Coulomb 2.5 Graphics” window on your monitor. It’s a simple grid with a NE-striking fault in the middle of the grid.

6. If you want to save this image as pict-formatted file, choose *File* > **Save Graphics window**. You can name it as you like in the dialog box, open and modify it in Adobe Illustrator or Canvas, etc. To see more detail of the graphic object, there are two methods to do so. One is to choose *Zoom* > **Zoom In**. You can repeatedly zoom in until it beeps. The other method is to hold down the mouse button on the object, and drag until you box the area of interest (*see next screen shot*). Once an area is selected, choose *Zoom* > **Zoom Selection**. When the maximum enlargement occurs, a beep will sound. If after zooming, you want to revert the image to its original size, choose *Zoom* > **Revert To Top**.



7. You can see the (x, y) position of the cursor by choosing *Coordinates* > **Show Coordinates**. If you set "Plot size = 1.0," in the *Graphics* menu under *Size*, the precise position of the cursor is indicated in the window. You can use this function to confirm the fault position, and its handy for identifying the endpoints of a cross-section line.
8. When any the graphic is displayed, if you want to do anything except "Save file," "Coordinates," or "Zoom," you might first hit <rtm>.



9. After returning to the main window, one of the functions and an input file calculated in the last execution are indicated. To make a text-based output file, check *Data > Save Input, Save Result*, etc. (these are check-items). The content of these text output files depends on the *Function* you select. Coulomb lets you calculate using combinations of *Functions* and *Faults*. **You can repeat different kinds of calculations for an input file until the memory is full (five to six executions on a pre-G3 Mac, perhaps a dozen or more times on a later Mac).** A *Graphics* menu is used for changing the size and color of objects. The *Graphics* menu is used when the main window (text-based) is active. Why don't you now quit and relaunch Coulomb.

Output Files (basic scheme)

Menu Selections

(items in menus are grey until they become possible options)

Input/Output options

Data	
New	⌘N
Old	⌘O
Old-Elements	
Save Input	
Save Result	
Save LightRes	
Save ElementC	
Observed GPS File	

One or some of these can be checked.

Distortion/Displacement/Stress

Functions	
<input checked="" type="checkbox"/>	Plot Grid & Faults
	Distorted Grid Mapview
	Distorted Grid x-section
	Dilatation Mapview
	Dilatation x-section
	Horiz. Displacement Mapview
	Vert. Displacement Mapview
	Displacement map + x-section
	Coulomb Mapview
	Coulomb x-section
	Coulomb 3D
	Slip-line Mapview
	Slip-line x-section
	Show Regional Stress field
	Change friction ⌘F
	Change target depth ⌘D

Stress Resolved on:

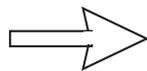
Faults	
<input type="checkbox"/>	Opt Fault
<input checked="" type="checkbox"/>	Opt Strike-Slip
	Opt Thrust [dip fixed]
	Opt Thrust
	Opt Normal [dip fixed]
	Opt Normal
	Specified Fault
	Focal Mech. File
	Primary Axes

Combinations of three menu selections make the kinds of output files.



Execution ("Run")

Calculation	
Go	⌘G
Show current	



File Output



Files saved with default names in the same folder the input file is in; they will overwrite unless renamed.

Chapter 3

CONCEPTS, DIMENSIONS AND SIGN CONVENTIONS

This chapter briefly explains what's needed to use this program. These are things you must understand to use the software correctly, and to avoid unexpected program crashes, or misleading output.

To launch Coulomb 2.5

There are two ways to launch the program as the normal Mac programs follows. Double-click the icon or highlight the icon and command-O. The "Coulomb 2.5 Output" window appears in front.

To quit Coulomb 2.5

There are four ways to quit this program.

1. Choose *File* > **Quit**.
2. Short-cut "Cmd-Q"
3. Just click the closed box on the left-top corner of the "Coulomb 2.5 Output" window.
4. Hit return key twice. You can ignore some message in the window. This almost always works.
5. If Coulomb freezes or will not quit, perform a 'forced quit' by holding down the **Apple + Option + Esc** keys.

Calculation procedure

Here is the procedure to calculate from your input file.

1. Launch Coulomb 2.5
2. *Data* > **Old** (or **Old-Elements**) to read your input file
3. If the main window "Coulomb 1.0 Output" is active, choose whichever functions you want to try from the *Functions* menu. Some of these selections trigger requests for additional input, or let you modify the parameters in new temporary windows.
4. When you calculate Coulomb stress changes (**Coulomb** Mapview, x-section, or 3D) or slip-lines (**Slip-line** Mapview or x-section), the results will also depend on the friction coefficient and the selected option in the *Faults* menu. Some of the *Fault* options (**Specified fault** or **Focal Mech. File**) trigger requests for additional input.

5. Choose *Calculation* > **Go**. All calculations and graphic plots are performed by this action.
6. You can see the graphic objects in “Coulomb 2.5 Graphics” window. When the graphics window is active, the menus *Zoom* and *Coordinates* become active (they were formerly greyed out). While the graphic plot is open, you can print, save, and zoom in on this plot.
7. To perform the next calculation, you must hit <rtrn> to return to the main window (“Coulomb 2.5 Output”).
8. The program now reverts to the main window. The foregoing process is just one cycle for the one calculation. If you want to continue different types of calculations, you can change the *Functions* and *Faults* settings. Then choose *Calculation* > **Go** as you did before. If you want to use another input file, choose *Data* > **Old** to read it in to Coulomb.
9. After you have done all calculations, quit Coulomb 2.5.

To change parameters interactively

Most of the settings and parameters have to be changed by modifying an input file using a text editor. To be flexible with the parameters, however, the software lets you change some of the input parameters interactively.

1. Fault or dike positions and slip, the regional stress field, and elastic moduli cannot be changed except by modifying the input file.
2. The output grid, the depth at which displacement, strain or stress is sampled, the friction coefficient, and all aspects of the appearance of the color plots, can be changed interactively. These changes are transient and will not permanently modify the input file. **But if you select *Data* > **Save Input**, all interactive changes and selected options will be reflected in ‘Final-Input.cou’. Coulomb overwrites these files each time you hit *Calculation* > **Go**, so rename the file if you want it preserved.**

Most such interactive changes are carried out by typing in temporary small windows whose values are generally accepted when you type ‘y’ (or something else when y is an possible input value), after which the window disappears. The following submenus give you extra pop-up windows.

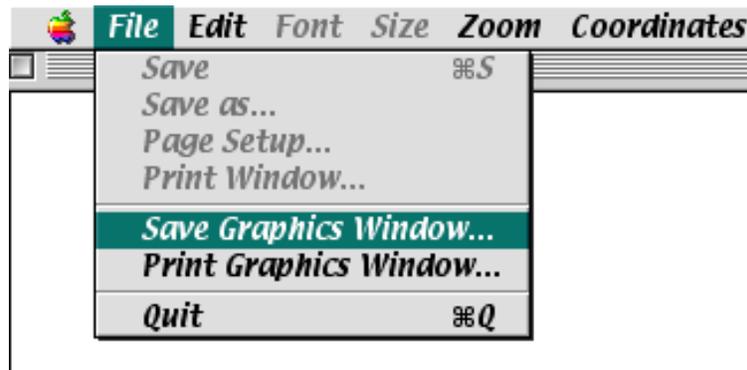
<i>File</i>	Observed GPS File (a dialog box asks you to select the file)
<i>Functions</i>	All x-section options, Coulomb 3D, Change Friction, Change Target Depth

Faults **Specified Fault** (a dialog box prompts you for the strike, dip, and rake),
Focal Mech. File (a dialog box asks you to select the file)

To save and print files

A pict-formatted graphic object is saved when you choose *File* > Save Graphics Window... name it as you like the dialog box, and put it in any folder. (If you do not rename it, Coulomb will overwrite it with the next saved file.) After saving it, you can open it by a SimpleText or a more sophisticated graphic applications such as Adobe Illustrator to modify it.

To get a high resolution (300 dpi) pict file, zoom once before saving. The size (in Kb) of the saved file gets larger and more precise with each successive zoom. When you open the pict file in, for example, Illustrator, you'll need to remove a mask that covers the graphic outside the original window. Once removed, you will see that the entire graphic is present.



Text files are saved when you check one or more of the check-items in the *Data* menu (**Save...**). **Save Input** saves the input file with all current settings that you modified interactively. **Save Result** and **Save LightRes** save tab-delimited text column files that can be viewed in Excel or displayed graphically by Transform or T3D. **Save ElementC** saves the 'element-condition matrix,' stress-changes resolved on 'receiver' planes specified in the input file. These text-based files will be saved in the same folder as the input file, regardless of where the application or its alias is located.

Coulomb 2.5 will also let you print the graphic object in the "Coulomb 2.5 Graphics" window, or text in the active window. If you want to print the graphics, just choose *File*

> **Print Graphics Window...** If you want to print the window contents, choose *File* > **Print Window**. You can change the Page Setup choosing *File* > **Page Setup**.

Prohibited procedures and actions to avoid freezes and crashes

1. With the Graphics window active and in front, you may use *File*, *Zoom*, and *Coordinates* menus. But hit <rtrn> and return to the “Coulomb 2.5 Output” window before anything else.
2. You must not do anything while the actual calculation (**Go**) is continuing. Some calculations may take a long time so that you have to wait before the plot is completed. When “*Calculation has been done. Hit return key for the next calc*” comes up at the right corner, you can continue.
3. Make sure that the “Coulomb 2.5 Output” window is in front when you choose *Calculation* > **Go** for the run. If the “ELEMENTS” window is open, it must be behind the main window.
4. Even if you allocate a lot of memory for Coulomb 2.5 using *command-I*, do not perform more than 10-20 calculations before re-loading the input. Coulomb does not have any warning message about the memory-full condition and it can suddenly crash (though it is not fatal). This appears to be less of a problem for G3 and G4 machines. It just takes a moment to quit and re-launch.

Coordinates, units, and sign conventions

Coulomb 2.5 calculates in the (x, y, z) Cartesian coordinate system. In the mapview projection on the monitor, x is positive from left to right, y is positive from bottom to top, z is positive upwards for displacement. DEPTH is positive downwards from the Earth’s surface. We use the standard *Aki & Richards* (1980) sign conventions for fault geometry and slip. The units for the input parameters are the following (see Chapters on Input and Output files for more detail):

Input

PR1 Poisson’s ratio: [dimensionless, 0-1] 0.25 is typically used

E1 Young’s modulus: [bars] 8×10^5 bars is recommended

FRIC Friction coefficient: [dimensionless; 0-1] 0.4 is a good starting value

Directions, angle, and dip: [degrees]

Grid and fault positions (x, y) : [km]

Depth: [km] downward is positive

Displacements: [m]

Faults: Right-lateral and reverse slip is positive [m]

Dikes: Opening displacement is positive [m]

Mogi point source: Deflation is positive [m³]

S1IN, S2IN, S3IN: Regional stress, positive in compression [bars]

The regional stresses are used only when either the Functions “Coulomb...” or “Slip-line...” are selected and when optimally-oriented planes (Opt...) or Focal Mech. File are selected in *Faults*. Otherwise they are ignored.

Shear modulus, G . Young's modulus [E1] and Poisson's Ratio [PR1] are input in Coulomb. To calculate the shear modulus, note that $G = E1/[2(1+PR1)]$. So for PR1=0.25 and E1=8x10⁵ bars, $G=3.2 \times 10^5$ bars or 3.2×10^{11} dyne-cm.

Output

Displacement: [m] North, East, and Up are positive

Shear Strain: [dimensionless] Right-lateral is positive

Principal Strain: [dimensionless] Extension is positive (*tensor notation*)

Dilatational Strain: [dimensionless] Dilatation is positive

Stress: [bars] Right-lateral and unclamping are positive

Chapter 4

INPUTMAKER AND INPUT FILES

All calculations in Coulomb require input files; some require successive inputs for which you will be prompted. If you already have an input file prepared, after launching the program the file is read by choosing *Data > Old*, or *Data > Old-Elements*. If an input file is inappropriate or improperly-formatted (e.g., it has tabs, or the number of faults is inconsistent with the '#fixed=', a warning dialog will appear. You must then change the file or quit the program and modify it using text editor. If you do not have an input file you can create one interactively by with InputMaker, or by choosing *Data > New* and replying to the prompts, or by modifying an existing file in EditII. Here is an input file:

```

This is a test file for the Coulomb 2.0
This is the second comment line
#reg1= 0 #reg2= 0 #fixed= 1 sym= 1
PR1= .250 PR2= .250 DEPTH= 10.0
E1= 0.800000E+06 E2= 0.800000E+06
XSYM= .000 YSYM= .000
FRIC= .800
S1DR= 24.0001 S1DP= 0.0001 S1IN= 100.000 S1GD= .000000
S3DR= 114.0001 S3DP= 0.0001 S3IN= 30.000 S3GD= .000000
S2DR= 89.9999 S2DP= -89.999 S2IN= 0.000 S2GD= .000000

# X-start Y-start X-fin Y-fin Kode rt.lat reverse dip angle top bot
xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx
1 -10.0 25.00 25.00 -5.00 100 0.0000 2.0000 30.00 5.0 15.00 WN-1

Grid Parameters
1 ----- Start-x = -50.00000
2 ----- Start-y = -50.00000
3 ----- Finish-x = 50.10000
4 ----- Finish-y = 50.10000
5 ----- x-increment = 2.000000
6 ----- y-increment = 2.000000

Size Parameters
1 ----- Plot size = 2.000000
2 ----- Shade/Color increment = 1.000000
3 ----- Exaggeration for disp.& dist. = 10000.00

Cross section default
1 ----- Start-x = -16.00000
2 ----- Start-y = -16.00000
3 ----- Finish-x = 18.00000
4 ----- Finish-y = 26.00000
5 ----- Distant-increment = 1.000000
6 ----- Z-depth = 30.00000
7 ----- Z-increment = 1.000000
  
```

Number of sources, elastic parameters, friction, and stress field, some of which can be changed interactively

Source or receiver fault parameters, none of which can be changed interactively

Plotting parameters, all of which can be changed interactively

Notice that all parameters in the input file (except for the line starting "#reg1=" and the first three characters of the source parameters marked '#') need the decimal point after each number. All format statements are shown at the end of this chapter.

Making input files with InputMaker

InputMaker 1.0, a standalone utility that comes in the Coulomb 2.5 bundle, lets you build sources interactively. You can do this by entering fault (x, y) coordinates, or by

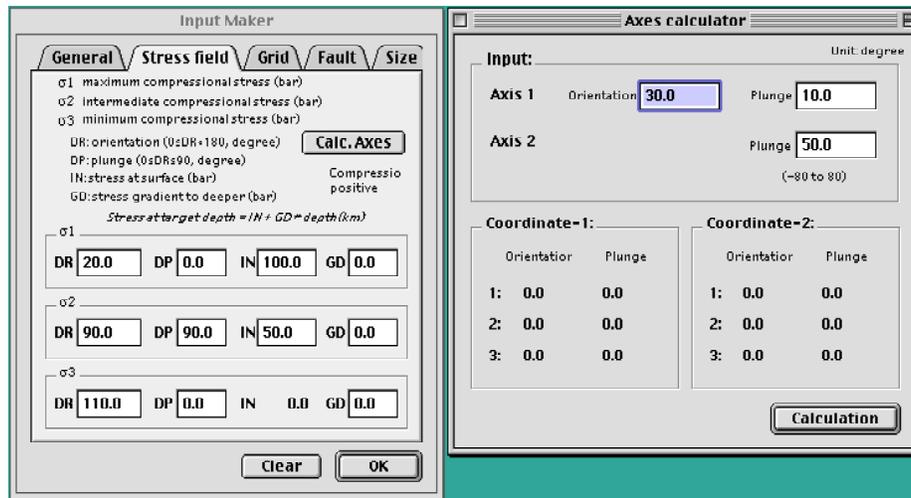
drawing the faults on the grid map. And what's more, you can drop a fault map, geological map, or satellite image on to the grid as a guide, register it to the grid, and then trace the faults to automatically enter them in the input file. InputMaker also makes certain that everything goes into the right place, and lets you view the file as you build or modify it.

The screenshot shows the 'Input Maker' window with the 'General' tab selected. The window contains several input fields and buttons. On the right side, arrows point from labels to specific input fields:

- 'Label in input file:' points to the 'Two lines of short note' section.
- 'PR1' points to the 'Poisson's ratio' field (value: 0.25).
- 'E1' points to the 'Young modulus' field (value: 8.0E+05 bar).
- 'DEPTH' points to the 'Calculation deptl' field (value: 7.5 km).
- 'FRIC' points to the 'Coefficient of friction' field (value: 0.4).

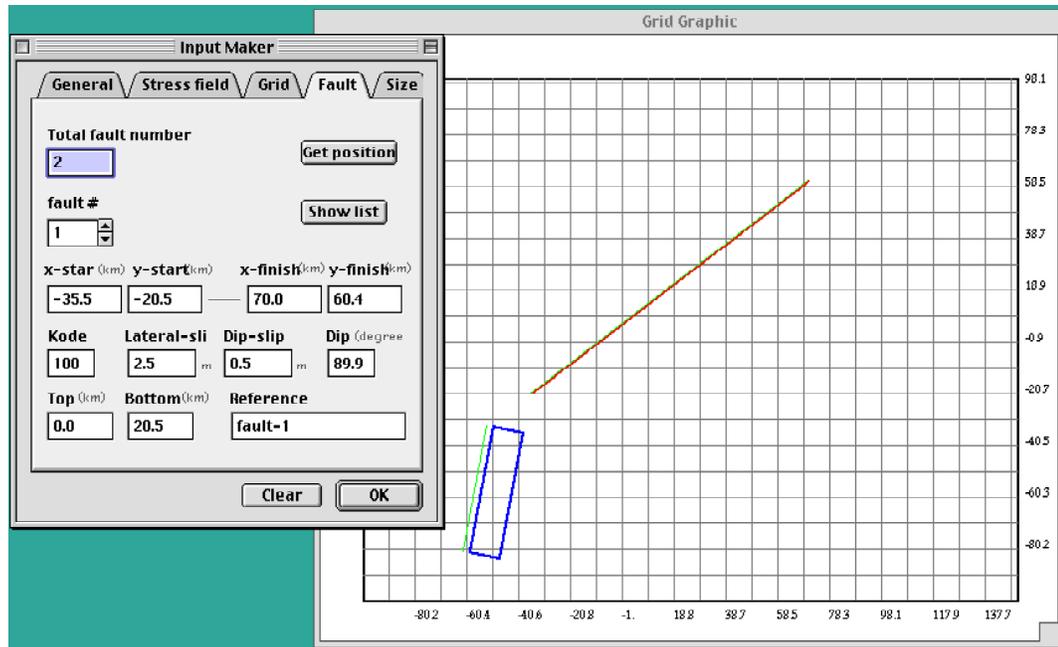
At the bottom of the window are 'Clear' and 'OK' buttons. A note at the bottom of the 'Half space parameters' section states: 'Input file uses Poisson's ratio and Young's modulus. (1 MPa = 10 bar)'.

1. Launch InputMaker. You will see several tabs along the top. The idea is to just fill in the blanks, moving from one box to the next, and then from one tab to the next. You can input either Young's modulus or the shear modulus. **After you have finished filling in the boxes in a tab (such as *General* or *Stress field*), hit the 'OK' button at the bottom. When you first do so, the "Grid graphics" window will appear; just pull it to the side. It is best to fill in the *Faults* tab last.** Remember that the *Stress* tab, shown below, will only be read if you select an optimally-oriented Coulomb stress change or slip-line option. If you hit the 'Calc. Axes' button, a calculator will pop up (see below) to assist you in defining the 3D stress field, since the three axes must be mutually perpendicular. Close the Axes calculator when done. **Hitting 'Clear' clears numbers from all boxes.**



- When you get to the *Fault* tab you have several options. One is to indicate the total number of planar (fault or dike) and point (inflation or shear) sources, and then enter each source one at a time based on (x, y) endpoints you have previously written down. After each fault source is entered, hit 'OK'. You can hit "Show file" to see the input file with these faults. If you are going to add another fault, close the input file manually (using the close box in the upper left-hand corner of the input file), then increment the "fault#", enter the new fault parameters, and hit the "OK" button. **Fault# is not the number of sub-elements for each fault (the first 3 spaces of the fault input line), which InputMaker automatically sets to 1.** Then hit "Show file" to see the additional fault in the input file. If the button reads, "Hide file", just hit it twice and it will revert. Now select *File* > **Save**, name the new input file, and open it in EditII. (Remember the right-hand rule convention described earlier so that your faults dip in the intended direction. Its useful to put a fault identifier in the 'Reference' box. The reference will appear in the input file but will not be read by Coulomb; rather this is for your own book-keeping.)
- Your other option is to select fault locations graphically. After choosing the number of faults, click "OK". The 'Grid graphics' window will appear; click anywhere on the window to make it active. To add a fault or other source, just click on a point in the grid, drag the mouse to the other end of the fault, and release. The fault will then appear as a green line. Now click back on the InputMaker window to make it active, and click "Get position". The (x, y) values for the start and endpoints of the fault will be entered. Now enter the slip, depth, etc., and click "OK" at the bottom, and this source will be added to your input file, or "Clear" if you don't like it. When you hit

“OK” the fault on the grid will turn from green to blue. You can repeat this process as many times as you like, incrementing the “fault #” each time until you have reached the total number of sources.



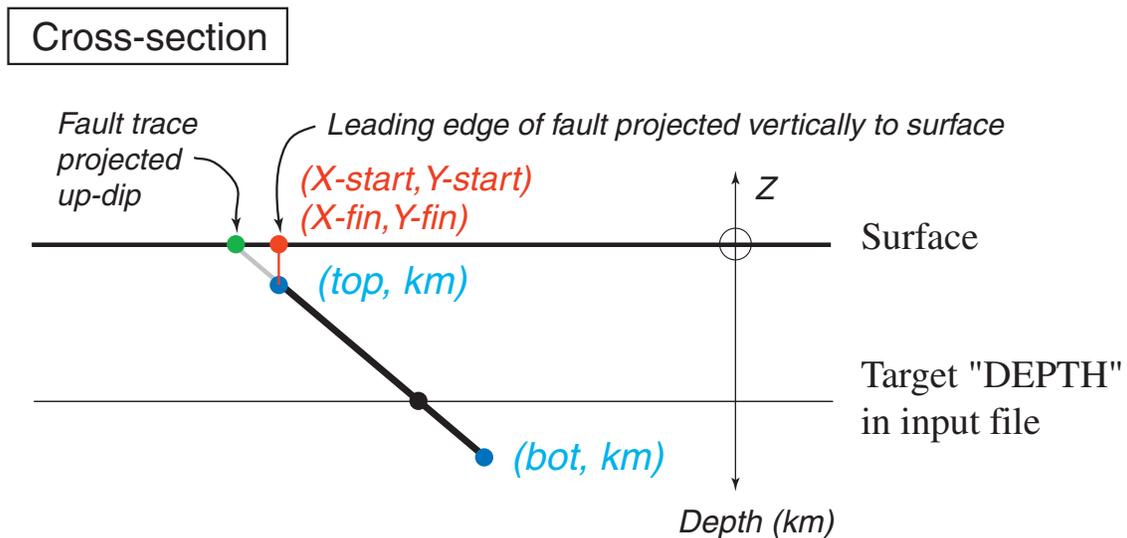
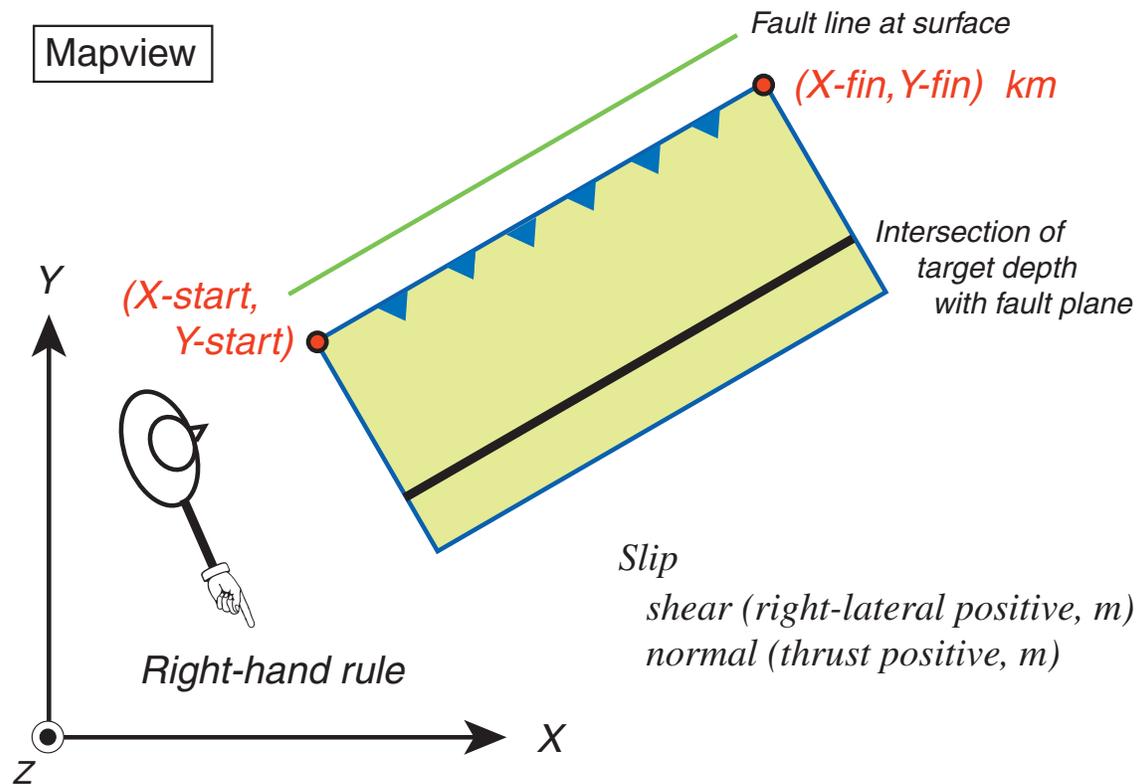
- You can put any jpeg file beneath the grid, and then trace faults over it. Just *click once* on the jpeg in the Input folder and drag it on to the ‘Grid graphic’ window. (Or, **File** > *Import* and dbl-clk on ‘IM-map.jpg’ in the input folder.) The jpeg will open *beneath* the grid, letting you trace faults to add to your input file. The jpeg will open with its upper-left corner in the upper-left corner of the grid graphic window. To register the map to the grid, notice that a line of buttons now appears on the bottom of the InputMaker screen. You can adjust the map scale (from the default 100%) and the x, and y offsets (from the 0-pixel default). If your jpeg has a km-scale bar, you can check to make sure the jpeg and grid scales are the same.

Note: In the ‘fault’ tab,

‘Lateral-slip’: vt-lat. is positive

‘Dip-slip’: reverse slip is positive

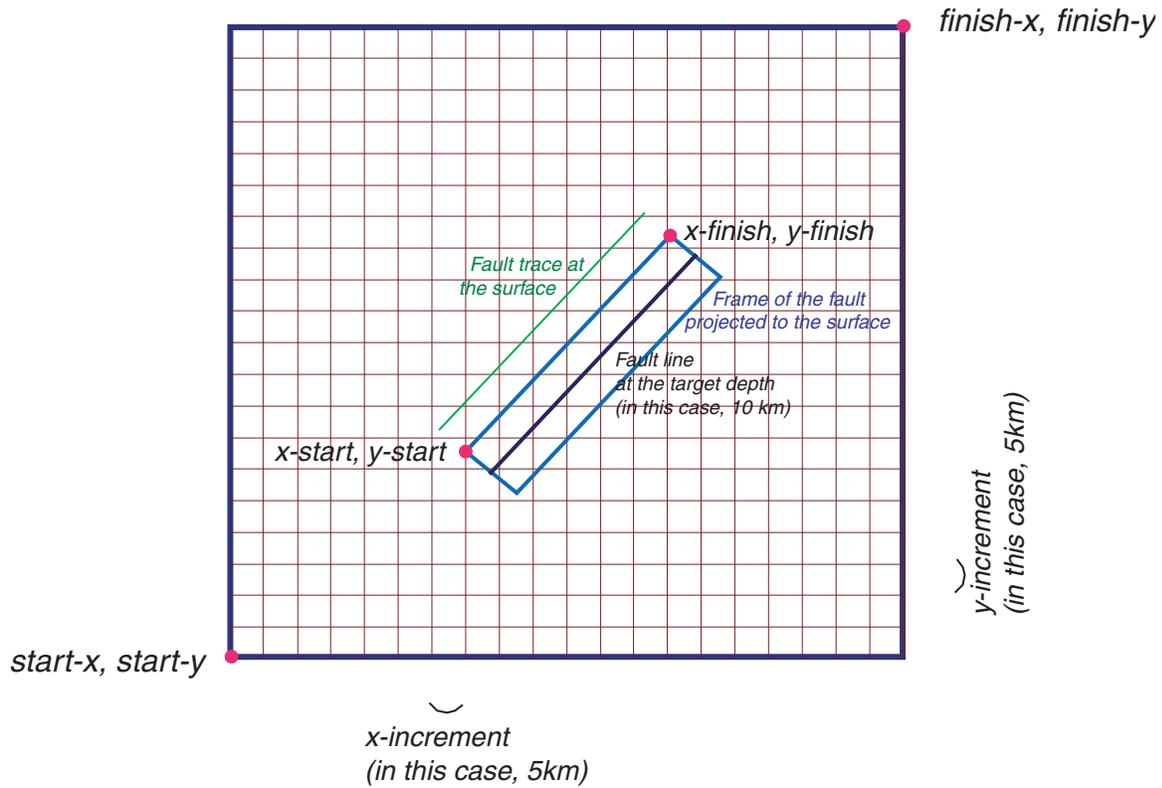
Positioning a fault in the Input file



Explanation for grid and fault position

Output graphic (for Data>Old>Example_3.inp)

Map view (Functions > Grid)
Surface projection



Input File

This is **not** a sequence number, but is used in Chapter 6 for receiver faults. It should be '1' for source faults.

#	X-start	Y-start	X-fin	Y-fin	Kode	rt.lat	reverse	dip angle	top	bot
1	-15.1	-17.00	15.20	17.20	100	0.0000	-2.0000	45.0	5.0	15.00

f-1

Grid Parameters	
1	Start-x = -50.00000
2	Start-y = -50.00000
3	Finish-x = 50.10000
4	Finish-y = 50.10000
5	x-increment = 5.000000
6	y-increment = 5.000000

Modifying an existing file in a text editor

1. Click one of the example files we prepared for tutorials.
2. To duplicate it, type 'd' holding command key (command + D), or drag the icon holding option key.
3. Rename it.
4. Launch an editor program such as QUED/M, EditII, or SimpleText, and open the file using File > Open (command + O).
5. Use a non-kerning (uniform-spaced) font, such as Monaco or Courier. Change the numbers you want, checking the units and format. If you want to add source or receiver faults or dikes, you can add a new row. Confirm that the number of the rows (faults) and the number of the fixed faults (#fixed=) in the third row in the input file are the same. Some of the editors have an option to 'show invisibles'. If your editor has the function, make sure that your modified numbers are filled in the properly formatted areas and there are no tabs. A tab will crash the program.
6. Save the file.

Taper and Splitter

We have two ancillary programs to help you transform simple rectangular sources into tapered-slip input files (Taper), or into a set of regularly distributed patches (Splitter). These are presented in Chapter 10 (page 77). Taper allows you to make more realistic schematic slip distributions without stress singularities at the fault ends. Splitter permits you to subdivide a plane for a 'variable-slip source model' from a geodetic or seismic source inversion, such as those in David Wald's repository, http://pasadena.wr.usgs.gov/office/wald/slip_models.html.

Structure and format for input files

An input file is composed of parameters for the halfspace, regional stress field, source fault positions and slip, receiver fault positions, grid information, graphical representation, and cross section information. See the figure, 'Structure of Input file', which annotates the entries. Coulomb is written in the Fortran 77 language, so you have to follow the Fortran format to avoid incorrect input. This is also shown in the figure. The numbers should be filled within the designated areas with appropriate values. **When making or modifying input files, use a non-kerning (uniform-spacing) font, such as Monaco, so that all numbers align under the xxxxxxxx headers.**

Structure and formats of the Input file

Example-1.inp

61/1657 Macintosh HD:Desktop Folder:Coulomb:Input_Files:Example-1.inp

This is an example
You can use first two lines (rows) for some comments or notes. Two lines of comments and notes.

#reg1= 0 #reg2= 0 #fixed= 3 sym= 1 #reg1 and #reg2: Number of fault region1, and region2, #fixed: number of fixed fault (should be same as the rows of listed below), sym: it should be always 1 (it is obsolete param)

PR1= .250 PR2= .250 DEPTH= 8.00 PR1,PR2: Poisson's ratio for region 1 and 2 DEPTH: Target depth [km] E1,E2: Young's modulus [bar]

E1= 0.800000E+06 E2= 0.800000E+06

XSVH= .000 YSVH= .000 Set both always zero (these are obsolete parameters)

FRIC= .400 Apparent friction of coefficient

S1DR= 89.9000 S1DP= .000000 S1IN= 100.000 S1GD= .000000 Parameters for regional stress field (for explanation and graphic plot, see Conventions figure in Chapter 2)

S2DR= 89.9000 S2DP= 90.0000 S2IN= 20.0000 S2GD= .000000

S3DR= 179.900 S3DP= .000000 S3IN= .000000 S3GD= .000000

#	X-start	Y-start	X-fin	Y-fin	Kode	shear(m)	reverse(m)	dip angle	top(km)	bot(km)	
1	-35.00	-35.00	35.00	35.01	100	2.00	0.00	90.00	0.00	12.00	source
1	-40.00	-5.00	-20.01	25.00	100	0.00	0.00	75.00	5.00	10.00	receiver-1
1	60.00	50.50	70.10	75.05	100	0.00	0.00	45.00	5.00	10.00	receiver-2

Grid Parameters

1 Start-x = -100.0000

2 Start-y = -100.0000

3 Finish-x = 100.0100

4 Finish-y = 100.0100

5 x-increment = 5.000000

6 y-increment = 5.000000

Size Parameters

1 Plot size = 1.0000

2 Shade/Color increment = 0.200000

3 Exaggeration for disp. & dist. = 10000.00

Cross section default

1 Start-x = -16.00000

2 Start-y = -16.00000

3 Finish-x = 18.00000

4 Finish-y = 26.00000

5 Distant-increment = 1.000000

6 Z-depth = 30.00000

7 Z-increment = 1.000000

Input file (Explanation)

Example-1.inp

0/2227 Macintosh HD:Desktop Folder:Coulomb:Input_Files:Example-1.inp

This is an example
You can use first two lines (rows) for some comments or notes. #a150, #a150

#reg1=0 #reg2=0 #fixed=3 sym=1 I3, I3, I3, I3

PR1= .250 PR2= .250 DEPTH= 8.00 G15.3, G15.3, G15.3

E1= 0.800000E+06 E2= 0.800000E+06 E15.3, E15.3

XSVH= .000 YSVH= .000 G15.3, G15.3

FRIC= .400 G15.3

S1DR= 89.9000 S1DP= .000000 S1IN= 100.000 S1GD= .000000 E15.6, E15.6, E15.6, E15.6

S2DR= 89.9000 S2DP= 90.0000 S2IN= 20.0000 S2GD= .000000 E15.6, E15.6, E15.6, E15.6

S3DR= 179.9000 S3DP= .000000 S3IN= .000000 S3GD= .000000 E15.6, E15.6, E15.6, E15.6

#	X-start	Y-start	X-fin	Y-fin	Kode	shear(m)	reverse(m)	dip angle	top(km)	bot(km)	
1	-35.00	-35.00	35.00	35.01	100	2.00	0.00	90.00	0.00	12.00	source
1	-40.00	-5.00	-20.01	25.00	100	0.00	0.00	75.00	5.00	10.00	receiver-1
1	60.00	50.50	70.10	75.05	100	0.00	0.00	45.00	5.00	10.00	receiver-2

Grid Parameters

1 Start-x = -100.0000 G16.7

2 Start-y = -100.0000 G16.7

3 Finish-x = 100.0100 G16.7

4 Finish-y = 100.0100 G16.7

5 x-increment = 5.000000 G16.7

6 y-increment = 5.000000 G16.7

Size Parameters

1 Plot size = 1.0000 G16.7

2 Shade/Color increment = 0.200000 G16.7

3 Exaggeration for disp. & dist. = 10000.00 G16.7

Cross section default

1 Start-x = -16.00000 G16.7

2 Start-y = -16.00000 G16.7

3 Finish-x = 18.00000 G16.7

4 Finish-y = 26.00000 G16.7

5 Distant-increment = 1.000000 G16.7

6 Z-depth = 30.00000 G16.7

7 Z-increment = 1.000000 G16.7

Input file with invisible mark (Format for Fortran 77)

Creating a new input using the Data > New function

1. Launch the Coulomb.
2. Choose Data > New. The "New File" window opens.
3. You can enter the numbers following the prompts.

4. In the final stage of the process, a dialog box appears and you can save and name it as you like.
5. The choose *Data* > **Old**, and select the new input file you just created.

Modifying an input file interactively from within Coulomb

If you choose *Data* > **Save Input** while the program is running, the parameters you modified interactively are automatically saved in “Final-Input.cou” in the folder where the original input file is located. This process does not allow you to modify the elastic constants, regional stress or fault sources. *With **Data** > **Save Input** checked, if you are running a Coulomb calculation for Specified Faults, the strike/dip/rake will be written to the input file and will automatically load when you use the input file (rename ‘Final-Input.cou’) again.*

Input file used to calculate stress on focal mechanisms

This input file is only used for the calculation for the resolved stress on the nodal planes of earthquake focal mechanisms. For this you choose *Functions* > **Coulomb Mapview**, and then *Faults* > **Focal Mech. File**. To make and modify the files, you have to use a text editor. The file format is shown below. See ‘FocalMech.inp’ in the tutorial files for more details. You can create such a file in Excel, by adjusting the column margins to be 11 characters long, and saving it as a ‘Formatted text (space-delimited)’ file. Then, open it in a text editor to check the formats and add the 3 comment lines. N = the total number of rows (nodal planes), *not* the number of focal mechanisms. The ID number in the first column will appear on the plots so you can locate the earthquakes. Note that like Coulomb, this file uses Cartesian coordinates, rather than lat/long. You can use the utility bundled with Coulomb, UTM+, to convert between lat/long and (x, y) values.

Focal Mechanism Input File

Intact input file & explanation

The screenshot shows a text editor window titled "FocalMech.inp" containing the following text:

```

Focal Mech input file
Example-1
N= 4
***
# X Y Z (depth) strike dip rake note
***
1 0.00 0.00 10.00 85.000 60.000 40.000 EQ-1
2 1.00 1.00 10.00 115.000 35.000 100.000 EQ-2
3 12.50 7.50 12.00 87.000 90.000 0.000 EQ-3
4 9.20 8.10 9.00 137.000 82.000 5.000 EQ-4
  
```

Annotations and explanations:

- Equal number:** An arrow points to the value '4' in the 'N=' line, indicating it must match the number of rows below.
- Two lines for comments and notes:** An arrow points to the 'Focal Mech input file' and 'Example-1' lines.
- Total number of earthquake (this should be the same number as listed rows below):** An arrow points to the 'N=' line.
- Earthquake number to be identified:** An arrow points to the '#' column header.
- Location of earthquake in the Cartesian coordinate (km):** An arrow points to the 'X', 'Y', and 'Z' columns.
- They should follow the Aki & Richards' convention (degree):** An arrow points to the 'strike', 'dip', and 'rake' columns.
- Short note to be referred:** An arrow points to the 'note' column header.

Input file with invisible mark & format for Fortran 77

The screenshot shows the same text editor window with the input file formatted for Fortran 77. The text is as follows:

```

FocalMechinputfile
Example-1
N= 4 I3
*****X*****Y*****Z(depth)*****strike*****dip*****rake*****note
*****
1 0.00 0.00 10.00 85.000 60.000 40.000 EQ-1
2 1.00 1.00 10.00 115.000 35.000 100.000 EQ-2
3 12.50 7.50 12.00 87.000 90.000 0.000 EQ-3
4 9.20 8.10 9.00 137.000 82.000 5.000 EQ-4
  
```

Annotations and explanations:

- I3:** A red box highlights the number '4' in the 'N=' line, with the label 'I3' below it.
- *****:** Red boxes highlight the asterisks used as column separators.
- Column Headers:** Red boxes highlight the column headers: X, Y, Z, strike, dip, rake, and note.
- Column Formats:** Red boxes highlight the data values in each row, with labels below: G10.4, G10.4, G10.4, G10.4, G10.4, G10.4, and a30.
- Line Lengths:** Red boxes highlight the end of each line, with labels 'a150' and 'a150' on the right side.

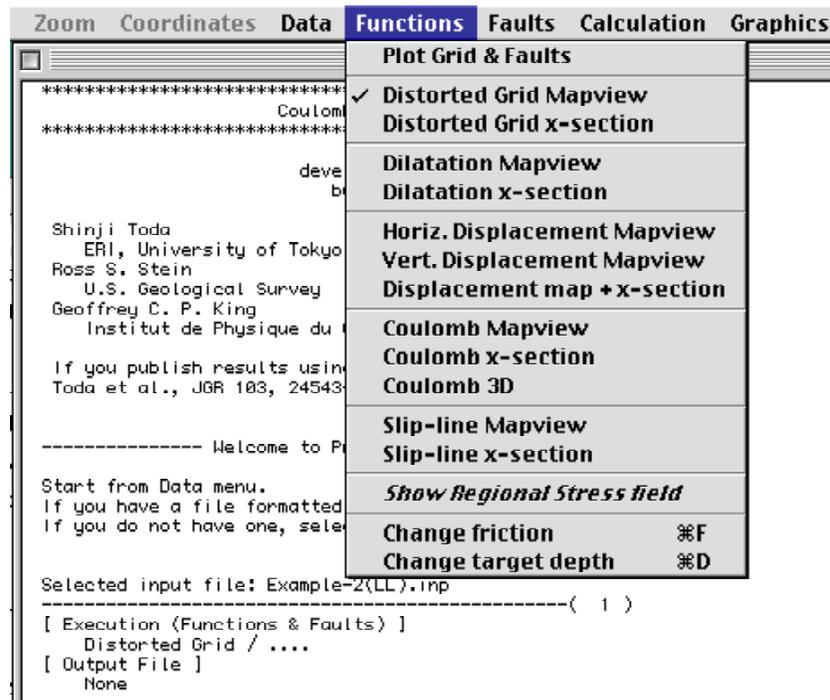
Chapter 5

TUTORIAL FOR DEFORMATION CALCULATIONS

Here we explore three options, “Distorted grid,” “Dilatation,” and “Displacement,” for map-view and cross-sections, using color-gradient and vector plots, to display the deformation field caused by earthquake faulting or volcanic intrusion.

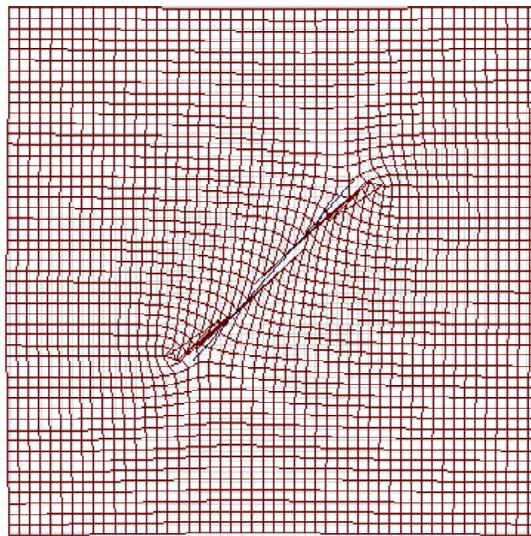
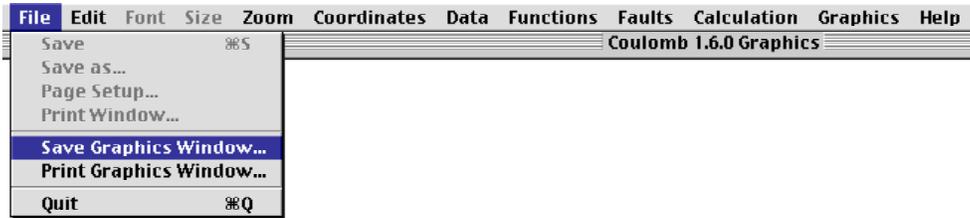
To display the distorted grid in map view

The program Coulomb can produce an image the original rectangle grid distorted by faulting in an elastic half space.

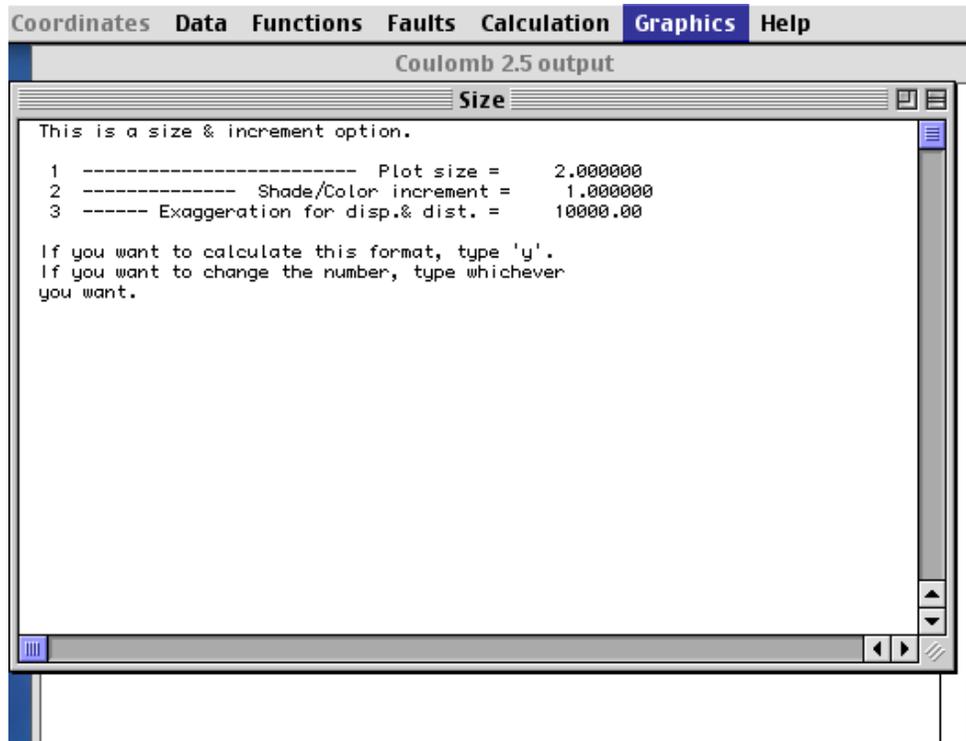


1. Start the program by double-click its icon.
2. Choose *Data* > **O****i****d**. And then, select “Example-1.inp” in the dialog and open it (keyboard short-cut: ‘cmd-O’).
3. Choose *Functions* > **D****i****s****t****o****r****t****e****d** **G****r****i****d** **M****a****p****v****i****e****w**. This is a check-item. Make sure that this option is checked.
4. Ignore the *Fault* menu, because neither Coulomb stresses nor Slip-lines are calculated.
5. Choose *Calculation* > **G****o** (keyboard short-cut ‘cmd-G’).

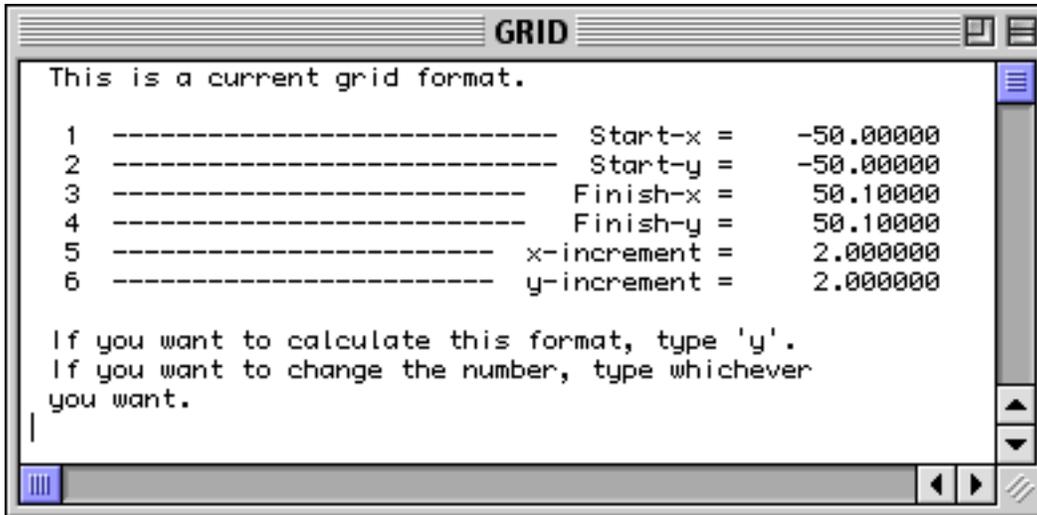
6. The following graphic appears in the “Coulomb 2.5 Graphic” window. You can see the undistorted grid in light grey behind the dark grey distorted grid.



7. You can check the fault position relative to the deformed grid. Choose *Coordinates* > **Show Coordinates**. Now the (x, y) position of the cursor appears in the lower left-hand corner of the graphics window. To tune this feature off, *Coordinates* > **Hide Coordinates**. If you want to save this image, choose *File* > **Save Graphics Window**, as you learned in the ‘Getting Started’ section.
8. If you want to change now much the deformation is exaggerated by the grid distortion, first hit <rtrn> to go back to the main window. And then choose *Graphics* > **Size**. You can see the new “Size” window. The text window also indicates the input/output files and the selected *Function*.



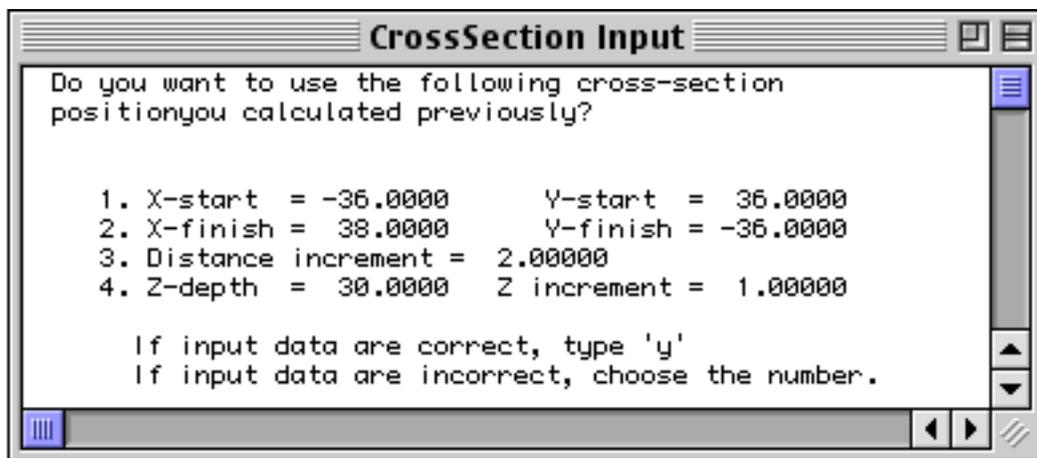
9. In the "Size" window, type '3' and input new amount of exaggeration. The default number that was included in the input file was 10000. This is a multiplier on the input fault slip. (Eg., if the input fault slip were 1 m, then the slip seen in the distorted grid would be 10,000 m or 10 km). If you want less distortion, input a value less than 10000. If you want a more exaggerated distortion, input a large number. As soon as you confirm the new number and type 0, and the "Size" window will disappear.
10. Choose *Calculation* > **Go** to get the new image of the distorted grid.
11. You can also change the grid size and *x*- and *y*-increments of the grid by choosing *Graphics* > **Grid**. When you choose this, a new "Grid" window comes up. You can type whatever you want to change. After finishing the modification for the grid, type 'y' and hit <rtrn>. The window will disappear and you will be back to the main window. Now *Calculation* > **Go** again.



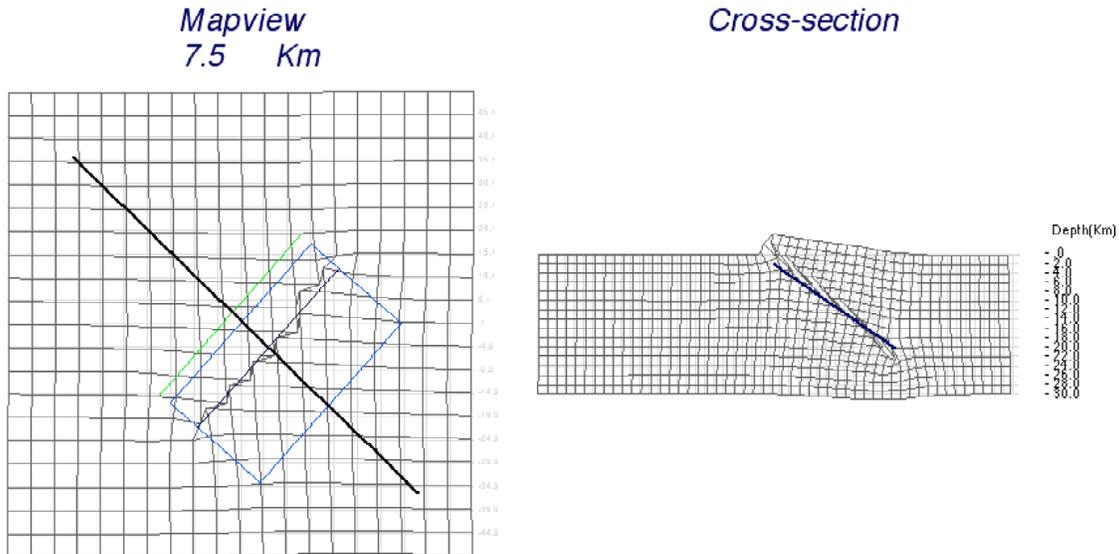
12. In this tutorial, keep the program activate for the next step and make sure that you hit <rtrn> and go back to the main window.

To display the distorted grid in cross section

1. Dbl-click 'Example_3.inp' in the 'Input Files' folder. You will see 7 lines of parameters beneath "Cross section default" at the bottom of the file. If the input file does not contain such a cross-section default parameter set, Coulomb will prompt you for them interactively whenever you select a ...x-section option in *Functions*. You can now close 'Example_3.inp' again.



2. Choose *Data* > **Old** and select 'Example_3.inp'. Then choose *Functions* > **Distorted Grid x-section**. This check-item opens the new window to verify or modify the cross section coordinates. Accept these values by typing 'y', and hit <rtrn>.
3. Choose *Calculation* > **Go** to show the distorted cross section grid image.



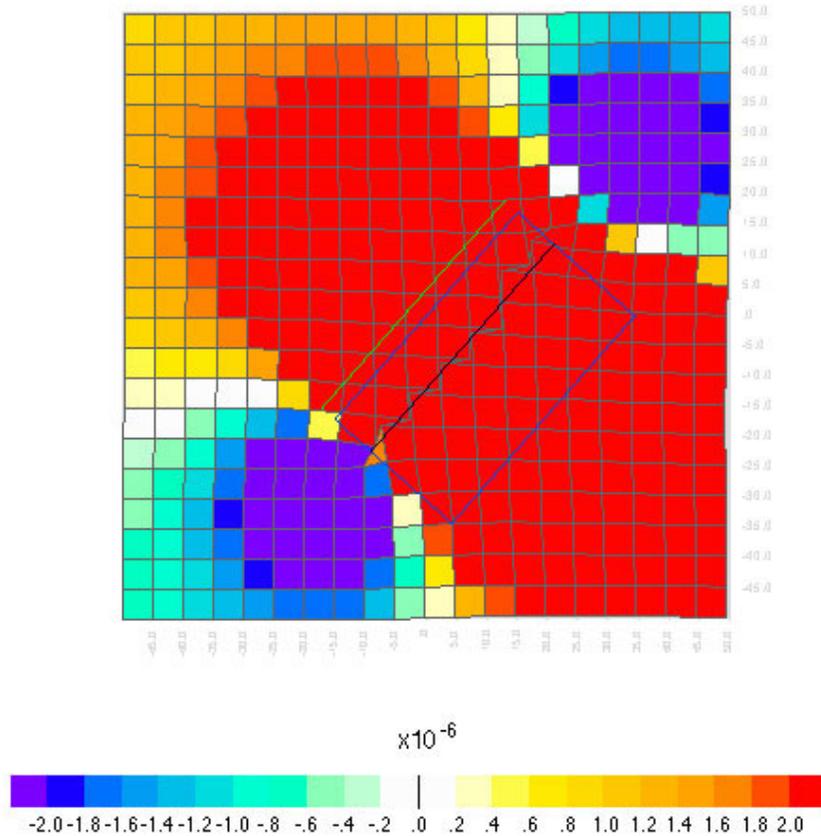
4. Experiment by zooming in on a portion of the cross-section. Drag your mouse to 'box' the cross-section (this will be identified by a light bounding box on the screen), and then choose *Zoom* > **Zoom Selection**. After viewing the zoomed image, choose *Zoom* > **Revert to Top** to return to the original size. Note that the cross-section line is shown on the mapview.
5. You can save the image (zoomed or reverted) by choosing *File* > **Save Graphics Window**.
6. To change the cross-section or mapview grid features (e.g. a deeper or coarser cross-section), go back to the main window, and repeat the procedure explained above.
7. *To create and save a default cross-section file. If you are using an input file without a cross section default, you can edit the input file in EditII manually to add one, or you can do the following: First plot the Mapview grid, select *Coordinates* > **Show Coordinates** and choose the endpoints of the cross-section, writing these down. Then choose *Data* > **Save Input**. Now select a ...x-section Function, and enter the values interactively that you wrote down, and choose *Calculation* > **Go**. Open and rename the saved input file; it now contains your default cross-section parameters for future use.*

To display the dilatation map view

'Dilatation' refers to volumetric dilatation ($e_{xx}+e_{yy}+e_{zz}$) caused by earthquake faulting or magmatic processes. This program lets you calculate the distribution of the dilatation in map view at the target depth, or in cross-section. The procedure is similar to the

Distorted Grid Mapview option. You can proceed with the following without quitting the program. Choose *Functions* > **Dilatation Mapview**. This is a check-item. Make sure it is checked.

1. Choose *Calculation* > **Go** to show the graphic image.



2. If you want to change the dilatation increment in the plot to use the full range of the

```

Size
This is a size & increment option.

1 ----- Plot size = 1.800000
2 ----- Shade/Color increment = 0.200000
3 ----- Exaggeration for disp.& dist. = 5000.000

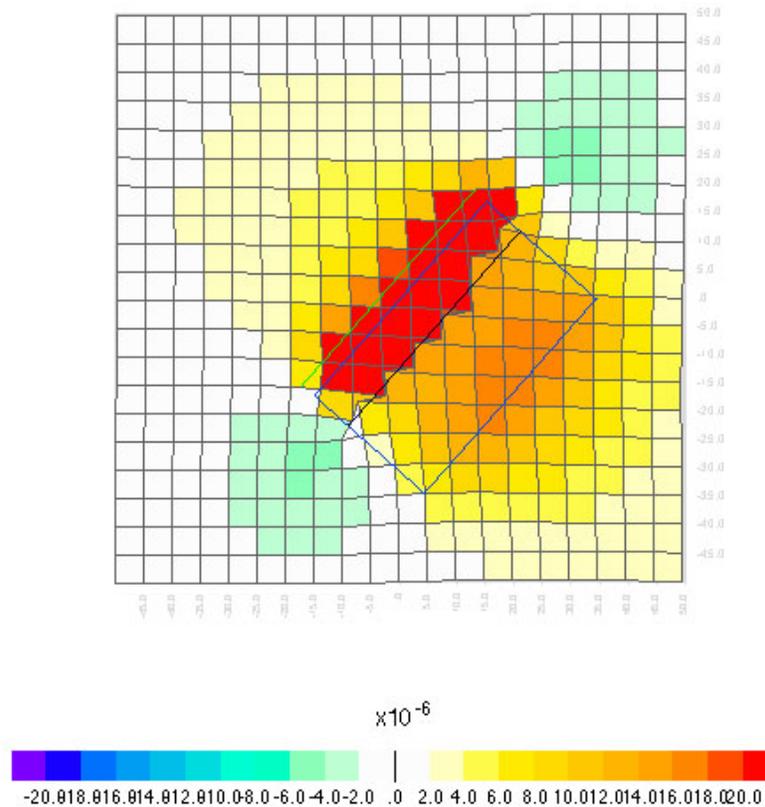
If you want to calculate this format, type 'y'.
If you want to change the number, type whichever
you want.
2
2

Do you want to change shade/color increment?
If so, type the value you want. If not, type 0.
2.0
2.0
1 ----- Plot size = 1.800000
2 ----- Shade/Color increment = 2.000000
3 ----- Exaggeration for disp.& dist. = 5000.000

If you want to calculate this format, type 'y'.
If you want to change the number, type whichever
you want.
    
```

color bar, go back to the main window after hitting the return key, and choose *Graphics* > **Size**. In the “Size” window, type 2 to change “Shade/Color increment.” Type a larger increment than the default value (for this example, choose 2.0; then type ‘y’ to close the window.

3. Choose *Calculation* > **Go** to get the new image for the dilatation. Notice that the dilatation scale has changed to reflect the new 2×10^{-6} strain increment.

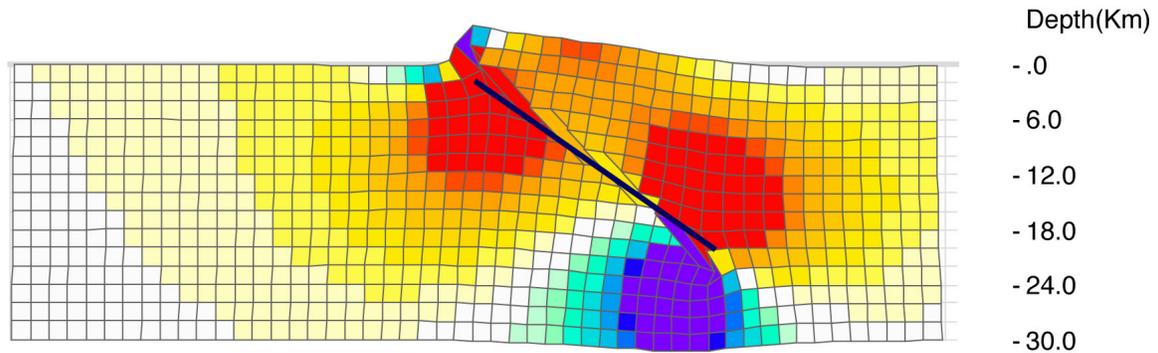


4. If you want to get a smaller range of colors in the scale, close the current graphic by hitting <rtrn>, and then choose *Graphics* > **Image**. The “Image” window opens. Type ‘3’ for the color scale settings, and then choose 2 for the detailed-color scale (for the dilatation option, B/W is not implemented). After confirming the new input, close this window by typing ‘y’. Choose *Calculation* > **Go**.

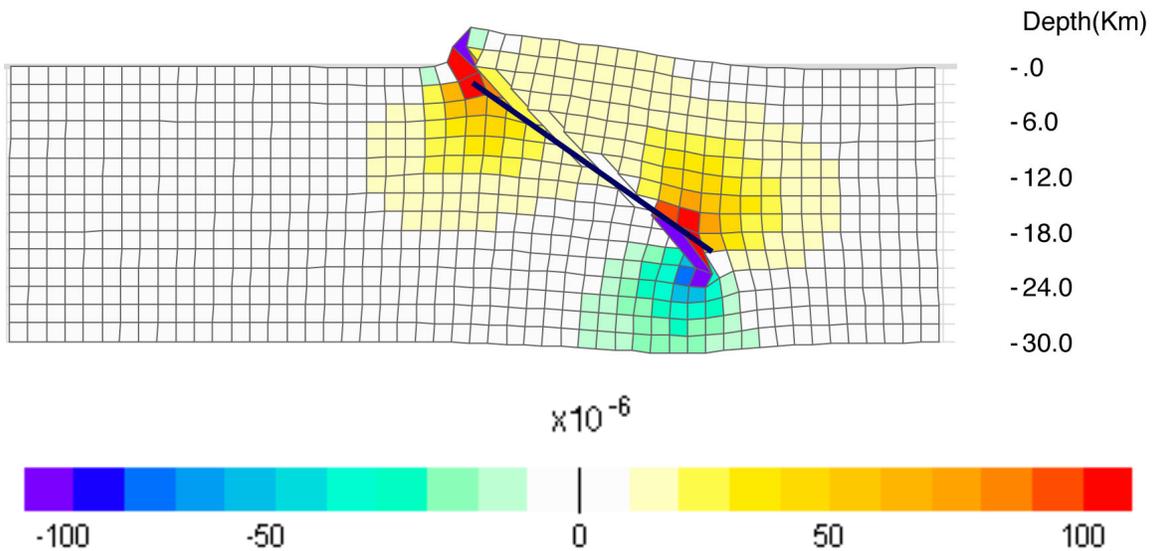
To display the dilatation in cross section

To make a plot of the cross-section of dilatation, recall how to make a cross section of the distorted grid previously.

1. Choose *Functions* > **Dilatation x-section**. When the new window appears, you can follow the same process as for the distorted grid cross section.
2. After closing the “CrossSection Input” window, choose *Calculation* > **Go**.
- 3.



Dilatation increment is 2×10^{-6} above, and 1×10^{-5} below, after changing it in *Graphics* > **Size**. Fiddling with the color scale is a valuable way to see things that would otherwise be missed. **Zoom once or twice before choosing File > Save Graphics window to get higher resolution files, as shown here**



4. In these examples you also see the distorted grid (*e.g.*, the anticlinal fold over the blind thrust). If you do not want to see such distortions, just choose *Graphics* > **Size**, select item 3, the ‘Exaggeration for disp.& dist.’ and change it to ‘1.0’ (*e.g.*, no

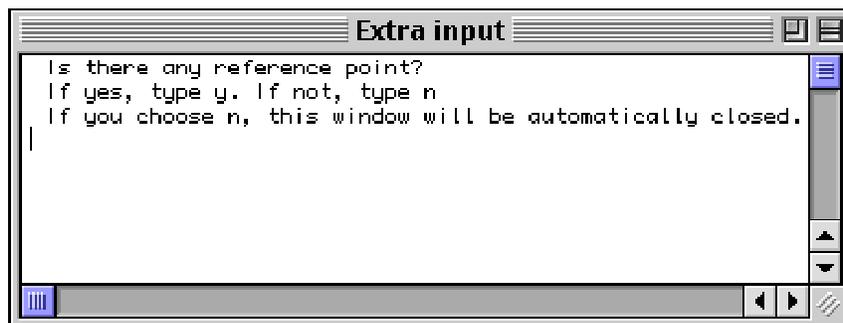
exaggeration). After accepting the revised values by typing 'y', *Calculate* > **Go** again. In the examples above, the uplift had been exaggerated by a factor of 5,000. This is equivalent to 5000 times the input 3 meters of slip, or 1.5 km, as would occur over geological time. But if you are interested in pressure solution and mineralization, or fold growth and fault rotation with accumulated slip, these tools may be useful.

5. All **...x-section** selections also plot the Mapview at the same scale side-by-side. This makes the x-section options a great place to start modeling, since you get both views at once.

To display the horizontal displacement map view

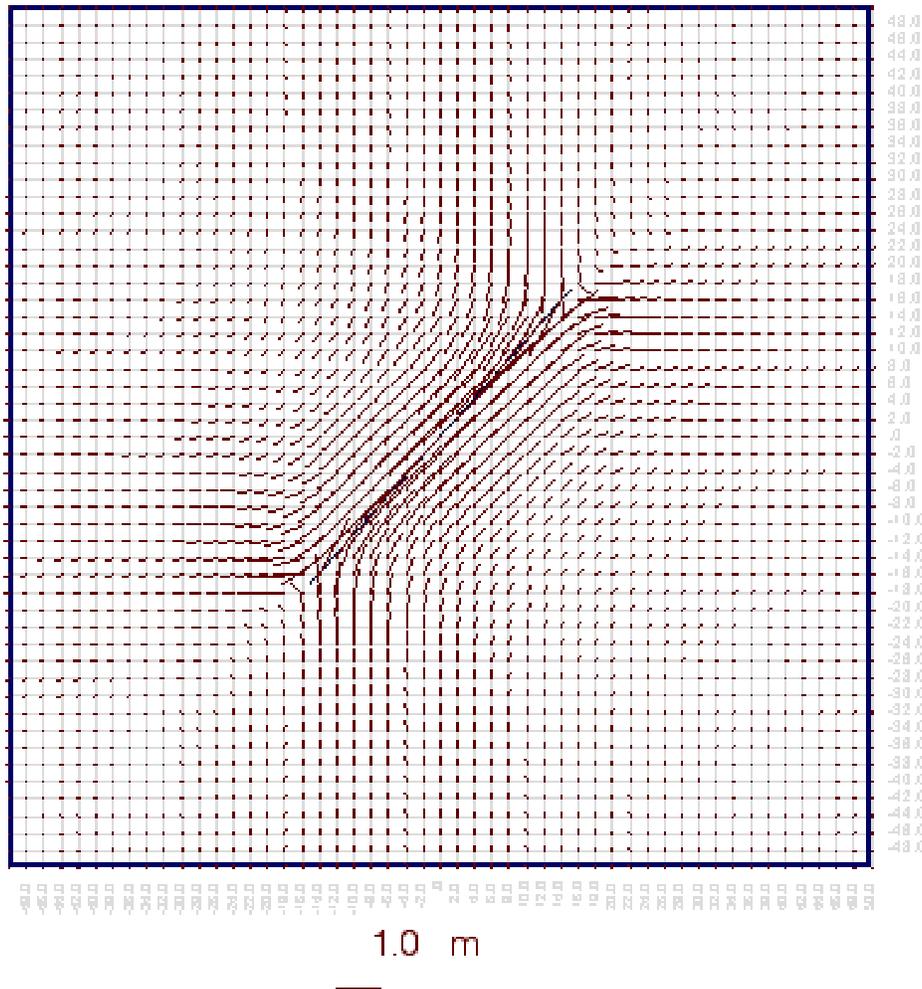
You can also make vector plots of the displacement field. Although the graphic output does not include arrowheads, the vectors are underlain by the undistorted grid, so the root of each vector is clear. Also, if you save the pict file of the graphic, you open it in Adobe Illustrator and put arrows on the ends of the lines automatically (In Illustrator 9.0:). To show the displacement in mapview, there are two options. One is an absolute reference frame (the fixed point is very far from the source fault). The other is the relative displacement from a selected point, since it is often convenient option to compare geodetic data such as the GPS or levelling observations from a reference station.

1. After launching the program and reading the input file using *Data* > **Old**, select 'Example-2(LL).inp' Then choose *Functions* > **Horiz. Displacement Mapview**. You will get an 'Extra Input' box asking if you want to hold a point fixed (in other words, the predicted displacement at the point will be subtracted from all other points). In this case, say no by typing 'n'.

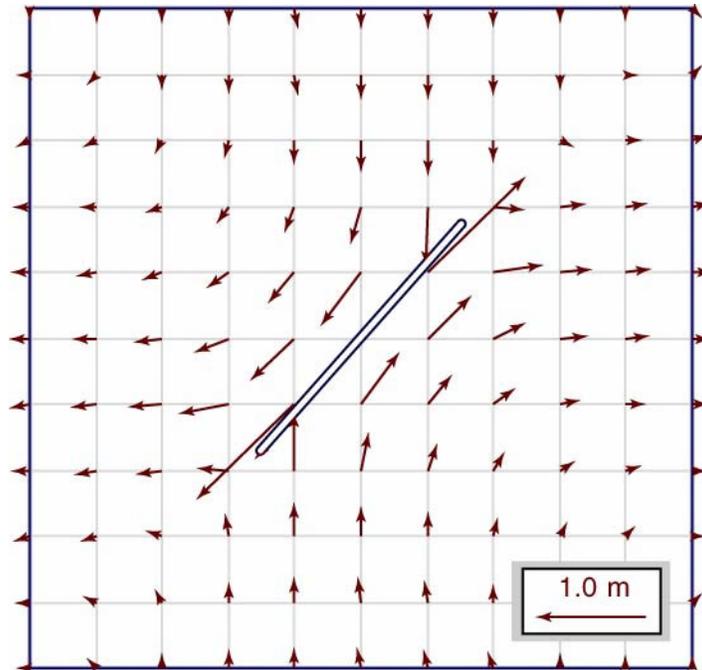


2. Choose *Calculation* > **Go**. You can see the graphic output in the graphic window. Note that a one-meter unit displacement scale bar is shown. The length of the plotted

vectors can be controlled by the Graphics option. The image you plotted here is based on the default value included in the input file.

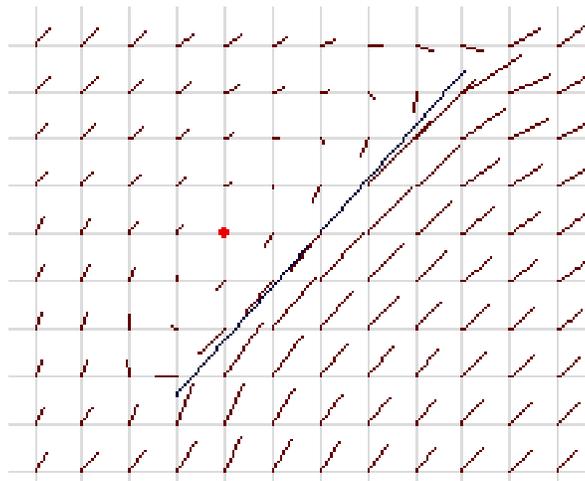


3. To recreate our modified version of this below, and coarsen the grid to 10 x 10 km (*Graphics* > **Grid**, change items 5 and 6), and change 'Exaggeration for disp.& dist.' to 30,000 (*Graphics* > **Size**, change item 3). Now your plot will look like ours without the arrowheads. Zoom once to increase the plot resolution, and select *File* > **Save graphic window**. Name it whatever.
4. Here's what you do in Illustrator 9.0 to show the arrowheads: get rid of the various bounding boxes with the Direct Selection Tool (white-cursor arrow). Click on one vector and then choose *Edit* > *Select* > *Select Fill & Stroke* (all displacement vectors—but nothing else—are now selected). Then *Filter* > *Stylize* > *Add Arrowheads* at end with *Scale* of 70%. We also spruced up the fault and moved the vector scale into the plot.



To include a reference point in a horizontal vector plot

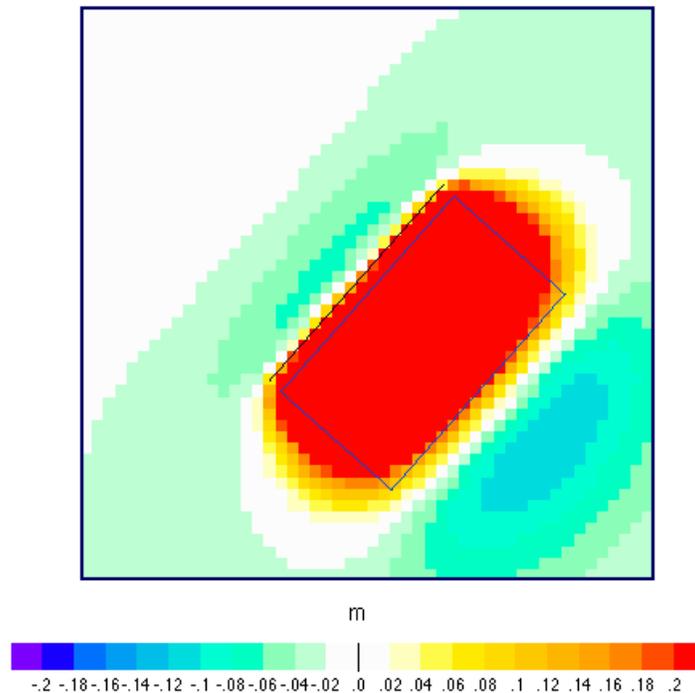
1. After launching the program and reading the input file using *Data* > **Old**, choose *Functions* > **Displacement Mapview**. This time, reply 'y' to the fixed point, and hold (0, 10) fixed to see some interesting results.
2. Choose *Calculation* > **Go**. The reference point is now shown as a large red dot with no displacement (a portion is shown below).



To display the vertical displacement in map view

You can make color contour plots just as you did for dilatation map view

1. *Data* > Old, select 'Example3.inp.' Then Choose *Functions* > Vert. Displacement Mapview. Its likely that you would want the surface displacements, so choose *Functions* > Change target depth, and set it to 0.0 km.
2. Now go to *Graphics* > **Grid**, and change the *x*, *y* increments to 5 km (items# 5 and 6). Then go *Graphics* > **Size** and change the 'Shade/Color increment' to '0.02'. Now each color change corresponds to 0.02 m of vertical displacement.
3. Now *Calculation* > **Go**. Here is the plot.



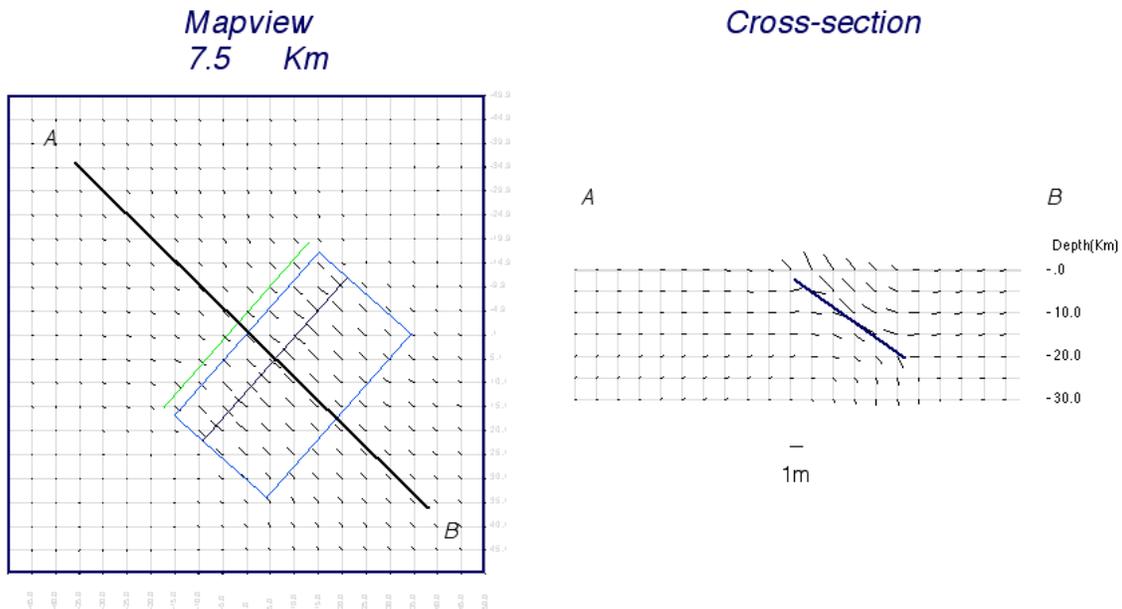
To display the displacement cross section

You can also plot displacement vectors in the cross-section. The vectors are projected onto the cross-section azimuth, with lengths scaled accordingly, and the fault shown where it is bisected by the section (if the section line cuts a corner of the fault, the fault image is truncated).

1. After launching the program and reading the input file using *Data* > **Old**, select 'Example-3.inp'. Choose *Functions* > **Displacement map + x-section**. When that sub-menu is selected, a new window appears. For practice, change the distance and depth increments to 5 km (for item #3, input '5'; for item #4, input '30, 5'; then accept

the changes by inputting 'y'. If you make a mistake, just choose *Functions* > **Displacement x-section** again).

- Now choose *Calculation* > **Go**.



- In the graphic window, you can see the Mapview in the left (not shown above), and the cross section *viewed from south* in the right side. In the map view, the section line is shown as black solid line, so you may verify the relationship between displacements in map view and cross section.
- You can save and modify the plots (e.g., add arrowheads) as described previously. You can also change the location of the cross-section. For example, its instructive to see what the displacements look like just off the end of the fault.

To plot observed and predicted vectors at a specified series of points

Often you want to compare GPS vectors (or levelling elevation changes) with model predictions, and plot the residual vectors. Here's how to do it in Coulomb.

- Data* > **Old**, choose 'Izu-1.inp'. Then *Functions* > **Horiz. Displacement Mapview**. Answer no ('n') to the 'Extra Input' query. Now select *Data* > **Observed GPS File**, and choose 'Izu-1.GPS'. **The first station listed in this file is automatically set as the reference station in the plot. All observed and modeled vectors are relative to this site, so its observed and predicted displacements are plotted as zero.** The Observed GPS input file is not tab-delimited, so keep the numbers beneath the stars (as explained in loving detail in Chapter 7). Here's what an 'Observed GPS file' looks like:

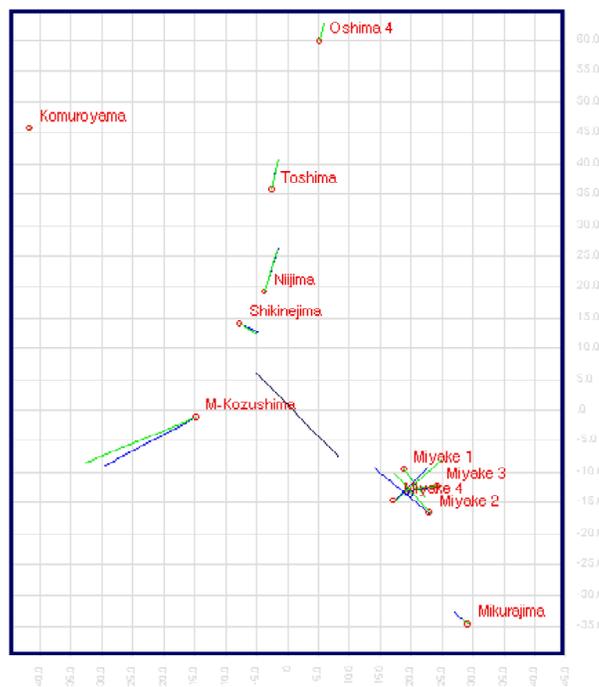
Izu-1.GPS

52/320 Macintosh HD:Coulomb f:Input Files:Izu-1.GPS

GPS input file
 Example-1
 N= 11

#	X	Y	Z (depth)	UX (m)	UY (m)	UZ (m)	Station
1	-41.6	45.7	0.0	0.000	0.000	0.000	Komuroyama
3	-2.5	35.8	0.0	0.006	0.026	0.002	Toshima
4	-3.7	19.2	0.0	0.015	0.047	0.011	Nijima
6	-7.7	14.0	0.0	0.019	-0.008	0.002	Shikinejima
9	-14.7	-1.2	0.0	-0.098	-0.051	0.038	M-Kozushima
10	18.9	-9.6	0.0	0.012	-0.016	-0.009	Miyake 1
11	24.3	-12.2	0.0	-0.031	-0.006	-0.019	Miyake 3
12	17.2	-14.7	0.0	0.036	0.035	-0.047	Miyake 4
13	23.0	-16.6	0.0	-0.059	0.049	-0.051	Miyake 2
14	29.2	-34.6	0.0	-0.014	0.013	0.000	Mikurajima
16	5.2	59.9	0.0	0.004	0.019	-0.006	Oshima 4

- To get a numerical file of the results, also select *Data* > **Save Result**. Now select *Calculation* > **Go**. The source dike is shown in black, and a point source of inflation is shown by a small blue circle (near Miyake 4). The GPS stations are the red labelled circles, and observed and predicted displacement vectors are given different colors.



1.0 m

Observation
 Calculation

- You can move the labels around and add arrowheads in Illustrator. You could also put this above a layer showing a grid of predicted vectors, or a color-gradient plot of the vertical or horizontal displacements.
- In 'Izu-1.GPS', the first station displacements happen already to be zero. If you select *Data* > **Observed GPS File** and choose 'Izu-2.GPS', Miyake 1 is listed first, and you

will see that all plotted vectors are relative to Miyake 1; it is zeroed out. The output file is called 'GPS_Disp.cou', appears below. Note that the observed and modeled dx , dy , and dz displacements are shown both with and without the reference-station displacements removed.

GPS_Disp.cou																	
A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	
#	X	Y	Z	UX(m)obs	UY(m)obs	UZ(m)obs	UX(m)obs-ref	UY(m)obs-ref	UZ(m)obs-ref	UX(m)calc	UY(m)calc	UZ(m)calc	UX(m)calc-ref	UY(m)calc-ref	UZ(m)calc-ref	Station	
1	-42	46	0	0.000	0.000	0.000	0.000	0.000	0.000	0.001	-0.001	0.004	0.001	-0.001	0.004	Komuroyama	
3	-3	36	0	0.006	0.026	0.002	0.006	0.026	0.002	0.009	0.031	0.011	0.009	0.031	0.011	Toshima	
4	-4	19	0	0.015	0.047	0.011	0.015	0.047	0.011	0.015	0.044	0.034	0.015	0.044	0.034	Nijijima	
6	-8	14	0	0.019	-0.008	0.002	0.019	-0.008	0.002	0.020	-0.011	0.014	0.020	-0.011	0.014	Shikinejima	
9	-15	-1	0	-0.098	-0.051	0.038	-0.098	-0.051	0.038	-0.118	-0.050	0.079	-0.118	-0.050	0.079	M-Kozushima	
10	19	-10	0	0.012	-0.016	-0.009	0.012	-0.016	-0.009	0.024	-0.031	0.002	0.024	-0.031	0.002	Miyake 1	
11	24	-12	0	-0.031	-0.006	-0.019	-0.031	-0.006	-0.019	-0.025	-0.005	-0.005	-0.025	-0.005	-0.005	Miyake 3	
12	17	-15	0	0.036	0.035	-0.047	0.036	0.035	-0.047	0.053	0.043	-0.041	0.053	0.043	-0.041	Miyake 4	
13	23	-17	0	-0.059	0.049	-0.051	-0.059	0.049	-0.051	-0.037	0.041	-0.009	-0.037	0.041	-0.009	Miyake 2	
14	29	-35	0	-0.014	0.013	0.008	-0.014	0.013	0.008	-0.004	0.004	0.008	-0.004	0.004	0.008	Mikurajima	
16	5.2	60	0	0.004	0.019	-0.006	0.004	0.019	-0.006	0.006	0.017	0.002	0.006	0.017	0.002	Oshima 4	

Note that the first observation point is fixed for -ref columns

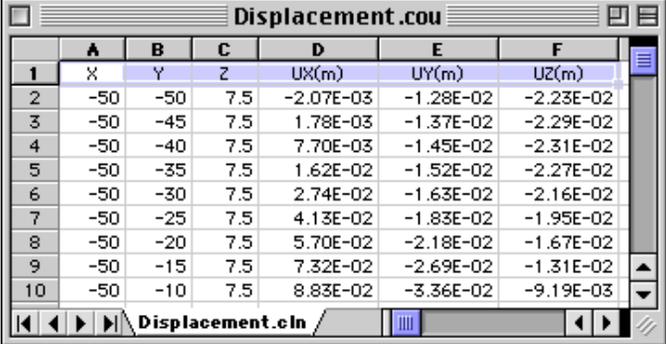
- You can easily compute and plot the residual vectors. First, in Excel, subtract the calculated ('calc-ref') displacements from the observed ('obs-ref') displacements. Now put the residual displacements into an 'Observed GPS File'. Remember that unlike the 'GPS_Disp.cou' file, the input file is not tab-delimited, so a bit of reformatting will be needed. You can now plot the residuals in Coulomb by, and giving a single source fault little or no slip. The 'calculated' vectors will disappear.

To save numerical output of the strain and displacement calculations

After reading in the input file, check **Save Result** in the *Data* menu; then select **Distorted Grid** or **Dilatation** (there are Mapview and Section versions of each) from the *Functions* menu; then choose **Go**. This will produce the following file, which can be opened in EditII but is easier to work with in Excel. If you keep **Save Result** checked, rename the file so it will not be overwritten.

Strain.cou										
	A	B	C	D	E	F	G	H	I	
1	X	Y	Z	EXX	EYY	EZZ	EXY	EYZ	EXZ	
2	-50	-50	7.5	-7.21E-07	-2.03E-07	2.78E-07	1.03E-07	-9.52E-08	-2.14E-07	
3	-50	-45	7.5	-7.92E-07	-1.57E-07	2.78E-07	2.47E-07	-8.20E-08	-2.62E-07	
4	-50	-40	7.5	-7.83E-07	-1.35E-07	2.58E-07	4.56E-07	-5.32E-08	-3.03E-07	
5	-50	-35	7.5	-6.41E-07	-1.70E-07	2.13E-07	7.20E-07	-6.51E-09	-3.26E-07	
6	-50	-30	7.5	-3.13E-07	-2.96E-07	1.38E-07	9.98E-07	5.52E-08	-3.17E-07	
7	-50	-25	7.5	2.22E-07	-5.31E-07	3.58E-08	1.22E-06	1.22E-07	-2.63E-07	
8	-50	-20	7.5	9.27E-07	-8.53E-07	-8.86E-08	1.29E-06	1.76E-07	-1.60E-07	
9	-50	-15	7.5	1.70E-06	-1.19E-06	-2.23E-07	1.16E-06	2.02E-07	-1.93E-08	
10	-50	-10	7.5	2.40E-06	-1.46E-06	-3.53E-07	8.21E-07	1.88E-07	1.34E-07	

Now close the file if you have not renamed it. With the **Save Result** option checked in *Data*, you can also select **Displacement Mapview** from the *Functions* menu; then choose **Go**. You can open the file below in Excel as well. Rename this file.



	A	B	C	D	E	F
1	X	Y	Z	UX(m)	UY(m)	UZ(m)
2	-50	-50	7.5	-2.07E-03	-1.28E-02	-2.23E-02
3	-50	-45	7.5	1.78E-03	-1.37E-02	-2.29E-02
4	-50	-40	7.5	7.70E-03	-1.45E-02	-2.31E-02
5	-50	-35	7.5	1.62E-02	-1.52E-02	-2.27E-02
6	-50	-30	7.5	2.74E-02	-1.63E-02	-2.16E-02
7	-50	-25	7.5	4.13E-02	-1.83E-02	-1.95E-02
8	-50	-20	7.5	5.70E-02	-2.18E-02	-1.67E-02
9	-50	-15	7.5	7.32E-02	-2.69E-02	-1.31E-02
10	-50	-10	7.5	8.83E-02	-3.36E-02	-9.19E-03

A tip that will save your life (courtesy of Geof King)

1. Keep **Save Input** in *Data* checked all the time. When you get a result you like and bring the graphic into Illustrator, open the associated 'Final-input.cou' from EditII, change lines 1 or 2 (the comment lines) to give the version number of Coulomb you used and the run date, save it with a new name, and copy/paste the entire file into Illustrator. You would do this even if you used Coulomb's numerical output plotted in Transform, rather than Coulomb's internal graphic files.
2. Now select the text in Illustrator and change the font size to 1 pt. You'll see an unreadable microdot of text, which you can move to the lower left corner of the graphic. You can cover it up with a white rectangle, *but do not ever remove it*; rather, keep the microdot there all the way to publication. If its covered, it will not detract from the figure, but when you 'Select all' in Illustrator, you will see it.
3. If you ever need to figure out *exactly* what parameters and version of Coulomb were used to generate that figure (as happens when reviewers demand changes or explanations), or you need to re-run the file, just copy/paste the input file back into EditII (it will automatically revert to 10 pt) and rerun Coulomb.
4. You will soon lose faith in the work of anyone who does not follow this practice.

Chapter 6

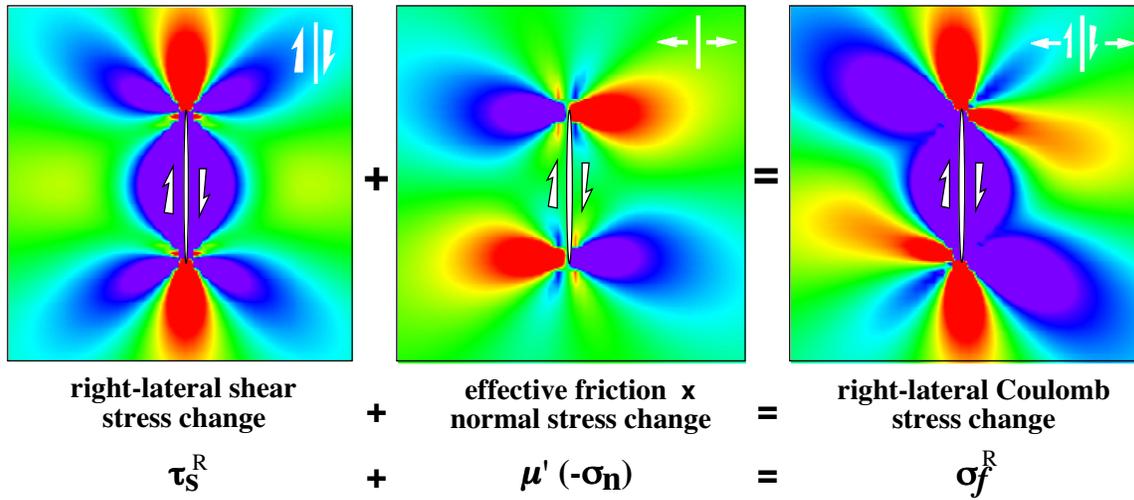
TUTORIAL FOR STRESS CALCULATIONS

Here we calculate static stress changes caused by the displacement of a fault, dike or point source (we refer to these as 'source faults'), derived from the strain field shown in previous sections. We resolve the shear and normal components of the stress change on 'receiver' fault planes. The shear stress change is dependent on the position, geometry, and slip of the source fault, and on the position and geometry of the receiver fault geometry (including its rake). The normal stress change ('clamping or unclamping') is independent only of the receiver fault rake.

We use the Coulomb failure criterion, in which failure is promoted when the Coulomb stress change is positive. We'll explore three kinds of receiver faults: (1) receiver faults listed in the input file, (2) focal mechanism files, and (3) faults optimally-oriented for failure, which are a function of the regional stress, the stress change associated with the source fault, and the assumed friction coefficient.

On the following page are Figures 2 and 3 from *King et al.* [1994], which graphically presents the Coulomb stress change resolved on vertical strike-slip faults parallel to the master fault (Fig. 2a); and resolved on optimally oriented planes (Fig. 2b) for a given regional uniaxial compression and friction coefficient. In Figure 3, the influence of the regional stress magnitude is seen on the orientation of the optimal planes, and on the stress change resolved on these planes. **These figures were made by importing the numerical stress-change output files ('Save LightRes') into Transform, and overlaying the Transform color gradients on the 'Plot Grid & Faults' and 'Slip-line Mapview' pict output files.**

A. Coulomb stress change for *right-lateral faults parallel to master fault* Stress ■ Rise ■ Drop



B. Coulomb stress change for *faults optimally oriented for failure*
N27°E regional compression (σ^r) of 100 bars; $\mu' = 0.75$ Optimum Slip Planes ■ left-lateral ■ right-lateral

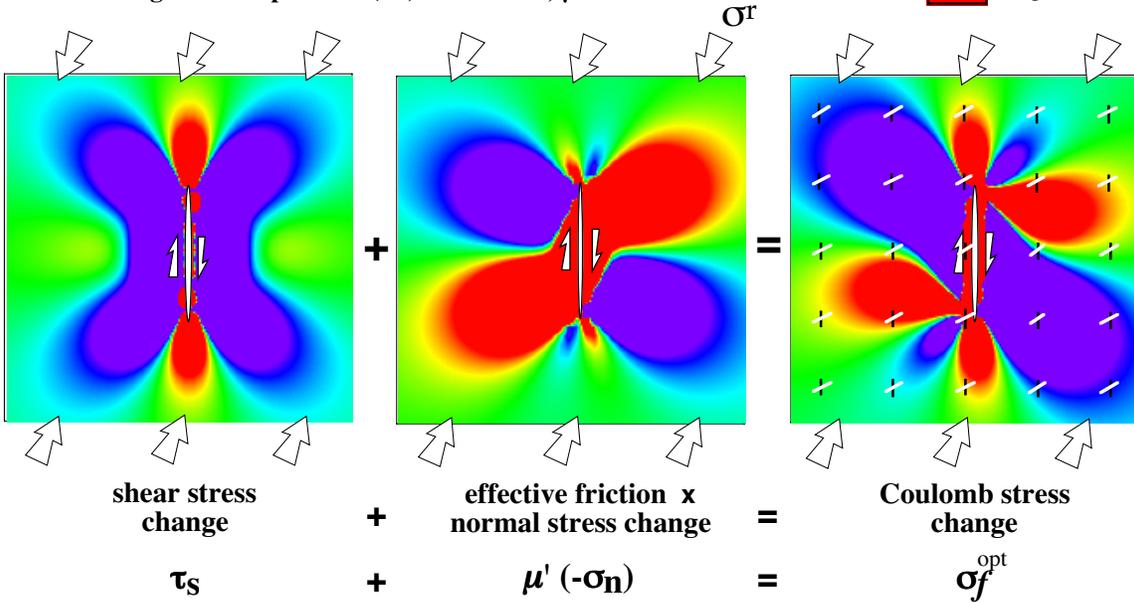


Fig. 2 from King et al (1994). Illustration of the Coulomb stress change. The panels show a map view of a vertical strike-slip fault embedded in an elastic halfspace, with imposed slip that tapers toward the fault ends. Stress changes are depicted by graded colors; green represents no change in stress. (A) Graphical presentation of equation 8 of King et al (1994), a “specified fault” calculation. (B) Graphical presentation of equation 13 of King et al (1994), for optimally-oriented strike-slip “opt strike-slip” faults.

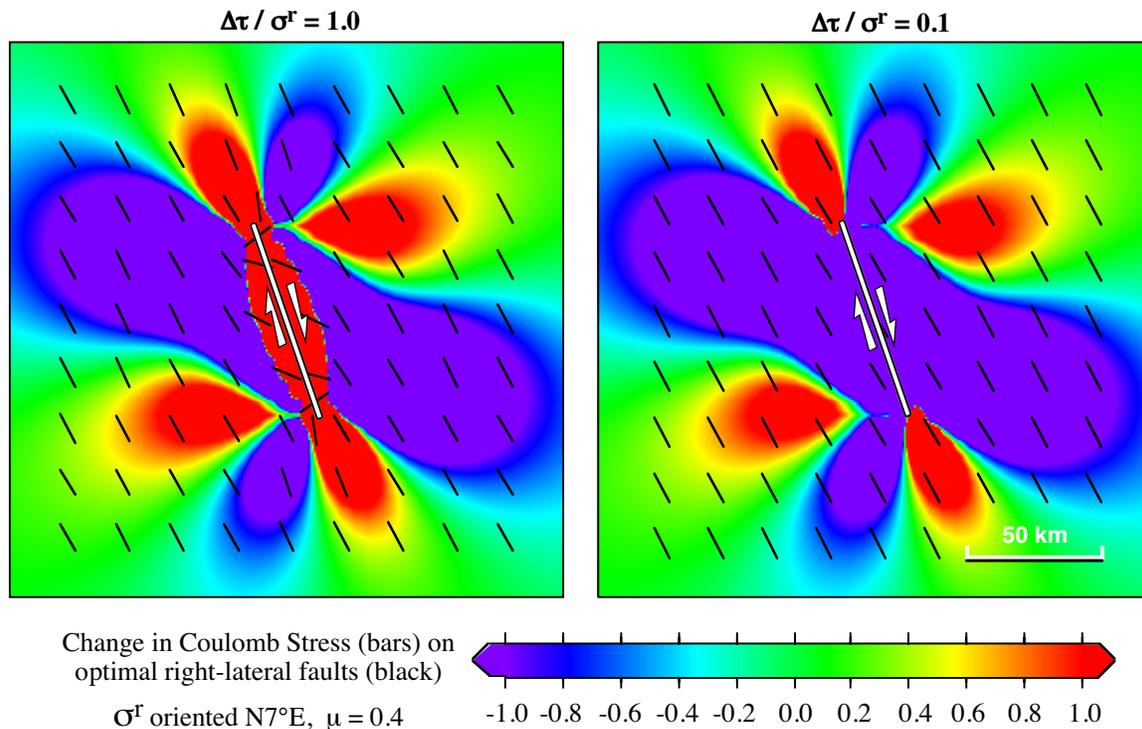


Fig. 3 from *King et al.* (1994). Dependence of the Coulomb stress change on the regional stress magnitude (σ^r), for a given earthquake stress drop ($\Delta\tau$). If the earthquake relieves all of the regional stress (*left panel*), resulting optimum slip planes rotate near the fault. If the regional deviatoric stress is much larger than the earthquake stress drop (*right panel*), the orientations of the optimum slip planes are more limited, and regions of increased Coulomb stress diminish in size and become more isolated from the master fault. In this and subsequent plots, the maximum and minimum stress changes exceed the plotted color bar range (in other words, the scale is saturated).

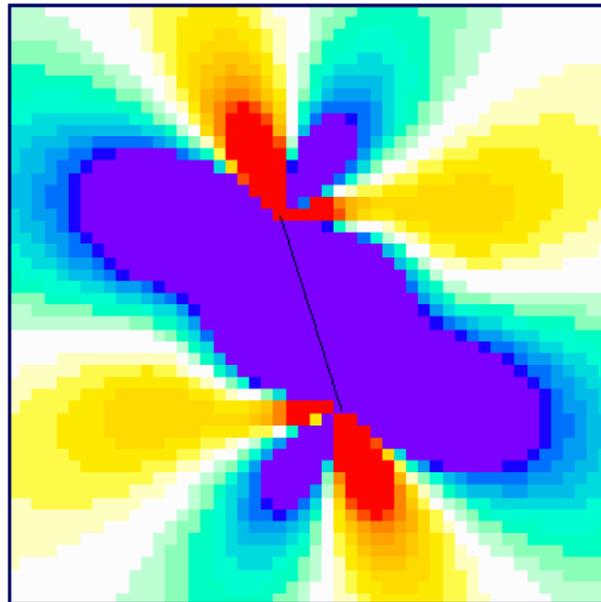
To display the Coulomb stress change on specified planes in map view

This is the simplest calculation, and is used by most researchers. To resolve the stress, you need to specify the fault strike, dip, and rake of the receiver planes following the *Aki & Richards* (Quantitative Seismology, W.H. Freeman & Co., N.Y., 1980, p. 106), and shown graphically in Chapter 3 of this manual.

1. Start the program by double-click its icon.
2. Choose *Data* > **Old**. And then, select "Example-2(RL).inp". Choose *Data* > **Save Input**.
3. Choose *Functions* > **Coulomb Mapview**. Make sure that this option is checked.
4. Choose *Faults* > **Specified**. This action opens an "Extra Input" window. You are prompted for the strike, dip, and rake of the receiver faults. To keep things simple, lets use the same values are for the source fault (strike 162°, dip 90°, rake 180°). Once you input the rake, the window automatically closes. **These values will be saved if**

you had checked *Data > Save Input* before hitting **Go**. Just rename 'Final Input.cou' and use it as the input file. Then all previous settings will be automatically followed.

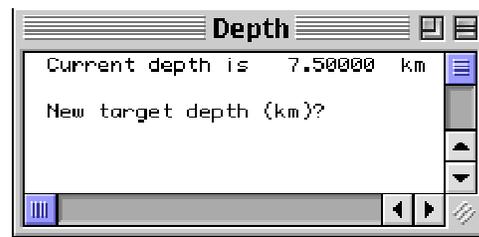
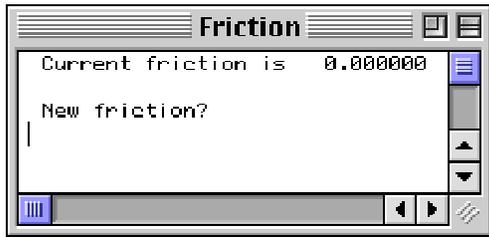
5. Choose *Calculation > Go*. The Coulomb stress changes shown as rainbow colors are now in the graphic window. You can also see the legend at the bottom of the window. Each color box in the grid is calculated at the center of the box, at the target depth specified in the input file. Calculated values of stress change may exceed the range shown in the scale bar.
6. If you want to change the color increments and thus saturation values, close the graphic window by hitting the return key and choose *Graphics > Size*. In the "Size" window, type '2' and input '0.2' for the 'Shade/color increment,' and then type 'y' to close the window. Choose *Calculation > Go* and you will get the image below. If you want to make a figure with a B/W scale (seven shades of grey are used), return to the main active window, and choose *Graphics > Image*, type '3' and select option 1. And then repeat the calculation (*Calculation > Go*).



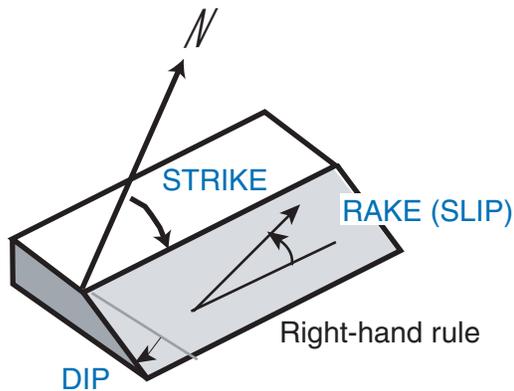
Calcula

7. Try changing the apparent coefficient of friction using *Functions > Change friction*. Change it to 0.0 and you will see that the pattern is now symmetrical since the normal stress change is not used. You can also change the target depth for the map

view using *Functions* > **Change target depth**. Note that these actions should be performed with the main window active.



CONVENTIONS

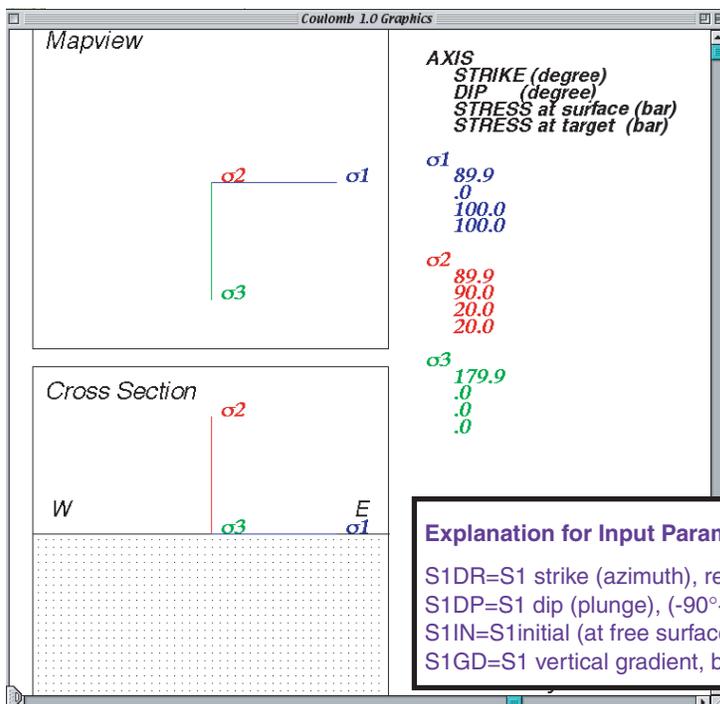
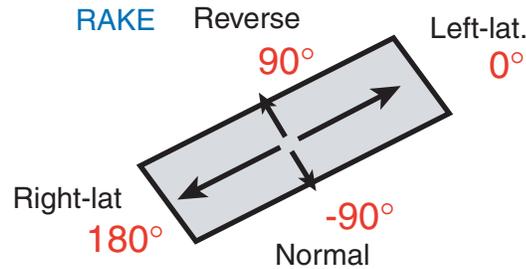


Aki & Richards (1980)

$$0 \leq \text{STRIKE} < 360^\circ \text{ (2PI)}$$

$$0 < \text{DIP} \leq 90^\circ \text{ (PI/2)}$$

$$-180^\circ \text{ (-PI)} \leq \text{RAKE} \leq 180^\circ \text{ (PI)}$$



Regional Stress Axes used for optimally-oriented Coulomb stress changes

To produce this plot:

- Data>Old>
- Example-1.inp
- Functions>
- Show Regional Stress Field
- Calculation>Go

(Note: vector lengths are not proportional to stress so that you can verify that axes are mutually orthogonal)

Explanation for Input Parameters (Lines 8-10 in input files)

- S1DR=S1 strike (azimuth), reckoned CW from North ($0^\circ \leq S1DR < 180^\circ$)
- S1DP=S1 dip (plunge), ($-90^\circ < S1DP \leq 90^\circ$)
- S1IN=S1 initial (at free surface), bars (compression is positive)
- S1GD=S1 vertical gradient, bars/km (positive for increase with depth)

To display Coulomb stress changes on the optimally oriented strike-slip, thrust, and normal faults (with the fault dip fixed) in map view

“Optimally-oriented” planes are those oriented such that the Coulomb stress change resolved on them is most positive (in our plots, most red). Here, for example, one assumes that aftershocks have a range of orientations, and will be most abundant at the depth, location and with the geometry that most promotes failure. So at every point on our grid, we find the ‘receiver’ plane along which the resolved Coulomb stress change is greatest. If the regional stress were zero, there would be a positive Coulomb stress change on some plane everywhere. But at large distances from a source fault, the earthquake stress change will be much smaller than the regional stress, so the planes rotate very little.

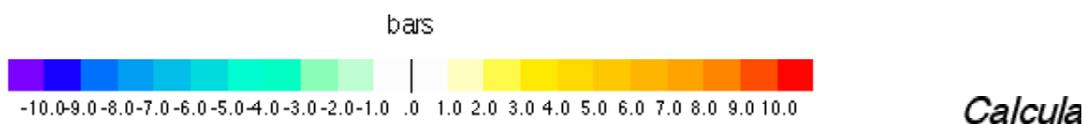
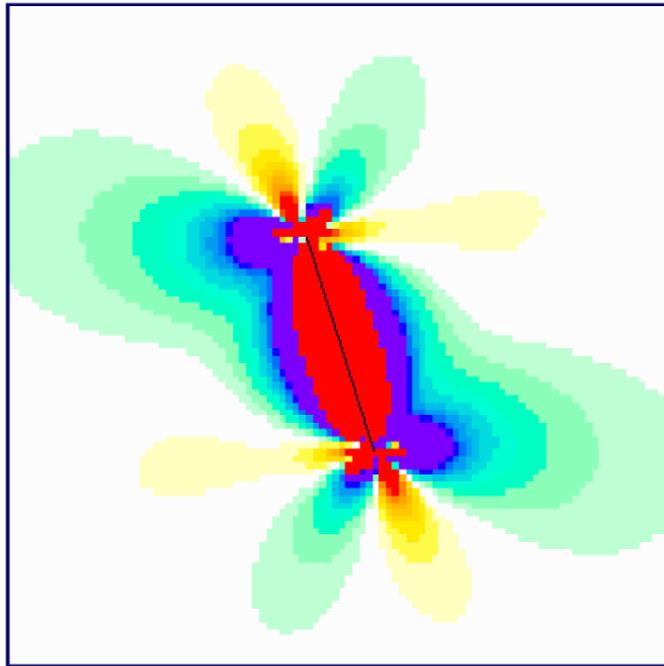
To perform this calculation, one must specify the regional stress field in the input file. “Example-1.inp” contains a sample 3D regional stress field. If you use your own input file however, make sure it is properly set up. None of this was needed when you specified the geometry of the receiver planes (**Specify Fault**), but there you implicitly assumed that such faults would be active in the regional stress field.

1. Launch Coulomb. Choose *Data* > **Old**. Select “Example-2(RL).inp”.
2. First, view the regional stress field by choosing *Functions* > **Show regional stress field**; then, *Calculation* > **Go**. This shows a mapview and x-section plot of the three principal stress axes. **Note that the vector lengths are not proportional to stress magnitude (otherwise they could be hard to see if the stress were small), but their magnitudes are listed to the right. Check that the three principal axes are mutually perpendicular, which is obligatory.** This plot is annotated, and the regional stress parameters are described in Chapter 3, on the *Conventions* page.
3. Choose *Functions* > **Coulomb Mapview**. This option should be checked.
4. Choose *Faults* > *Opt Strike-Slip*. Alternatives are *Opt Thrust [dip fixed]*, or *Opt Normal [dip fixed]*. These check-items are alternatives to *Specified Fault* you just tried. You do not have to input anything in this case.
5. For *Opt Strike-Slip* the faults dip vertically and their strike is a function of $S1$ (σ_1) and the friction coefficient μ (FRIC in Coulomb). For the thrust and normal faults, their dip is a function of the friction coefficient, and their strike is perpendicular to $S1$ (σ_1) and $S3$ (σ_3), respectively. The relationship of the angle, β , between $S1$ and the dip (for strike-slip faults) or strike (for dip-slip faults) is

$$\tan 2\beta = 1/\text{FRIC}$$

See King et al. [1994], which can be accessed online from the Coulomb web site at (<http://quake.wr.usgs.gov/research/deformation/modeling/papers/landers.html>)

6. Choose *Graphics* > **Grid**, and change the x , y increments to 1 km for higher spatial resolution. Then choose *Calculation* > **Go**. Here's the plot.



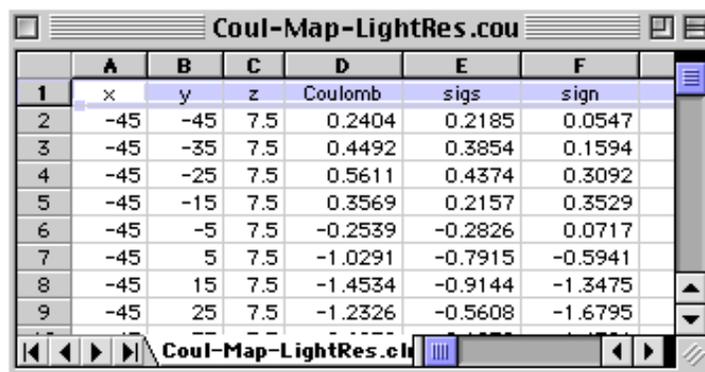
7. The Coulomb stress changes shown as rainbow colors are now in the graphic window. Try making further adjustments, such as *Functions* > **Change Friction** or **Change Target Depth**; or *Graphics* > **Size** to change the stress increment so the color scale bar will not saturate.

[To save the stress calculation results as text column file](#)

Although the program lets you save the calculation results as pict files, you may want to examine the numeric values or make higher quality figures. Coulomb color files can be large when the increments are small, so bringing numerical files into raster imaging

software can both increase the image quality and decrease the file size. Here's how to save the text column to be read by Excel (open and hit 'finished') Noesys (Transform, T3D; open them as 'text-column' and follow prompts).

1. Check *Data* > **Save LightRes** before you execute *Calculation* > **Go** in the above procedure. The output file is created in the same folder as the program Coulomb. Check to see that the new file is in the folder after the calculation is completed. It is automatically named depending on the calculation type. **Subsequent runs will overwrite the original data file unless you rename it before making new runs.** See Chapter 8 for detailed descriptions of the files and formats.



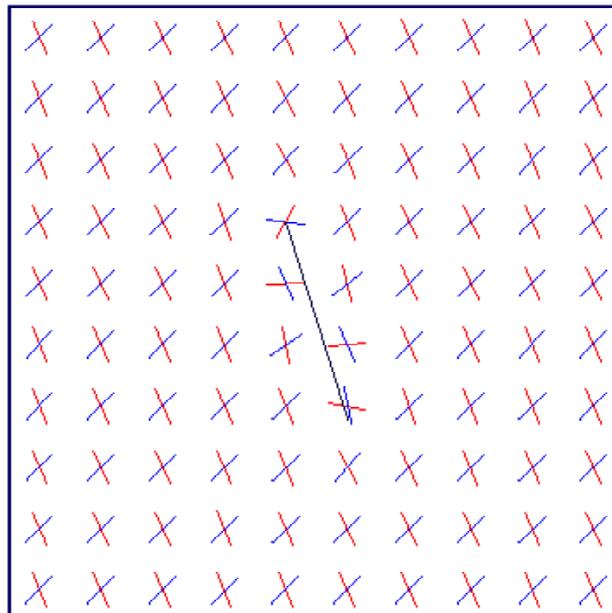
	A	B	C	D	E	F
1	x	y	z	Coulomb	sigs	sign
2	-45	-45	7.5	0.2404	0.2185	0.0547
3	-45	-35	7.5	0.4492	0.3854	0.1594
4	-45	-25	7.5	0.5611	0.4374	0.3092
5	-45	-15	7.5	0.3569	0.2157	0.3529
6	-45	-5	7.5	-0.2539	-0.2826	0.0717
7	-45	5	7.5	-1.0291	-0.7915	-0.5941
8	-45	15	7.5	-1.4534	-0.9144	-1.3475
9	-45	25	7.5	-1.2326	-0.5608	-1.6795

2. From this file you can plot the normal-stress changes in Transform (which Coulomb will not do), or re-evaluate the effect of the friction coefficient, since both components are listed. It would be wise to import the Input file into the Excel file so you can re-generate the output. Rename the Excel file if you plan more runs.
3. You could also have checked **Save Result**, which returns an additional six stress components rotated into the fault (f) coordinate system (sigfx, sigfxy, etc). They are likely more than you want; we use them principally for debugging.

To plot the slip-lines (the fault planes) for the optimally-oriented strike-slip, thrust, and normal faults

The program Coulomb lets you plot the planes for the optimally-oriented Coulomb stress changes in map view and cross-section. We recommend you look at both the stress changes and the optimal planes; if the regional stress is low relative to the earthquake stress change, the planes rotate near the source fault. Here's how to do it.

1. Quit and relaunch Coulomb unless you allocated a lot of RAM to Coulomb.
2. Choose *Data* > **Old**. Select "Example-2(RL).inp".
3. Choose *Functions* > **Slip-line Mapview**. Make sure its checked. One typically uses a coarser grid so adjacent slip-lines do not overprint, so set it to 10 x 10 km in *Graphics* > **Grid**. The slip-line length is controlled by *Graphics* > **Size** ('Exaggeration for disp. & dist. '); set this to 10,000.
4. Choose *Faults* > **Opt Strike-Slip**, also a check-item. Once again Coulomb will read the regional stress field and the friction coefficient from the input file to draw the slip-lines. Choose *Calculation* > **Go**.



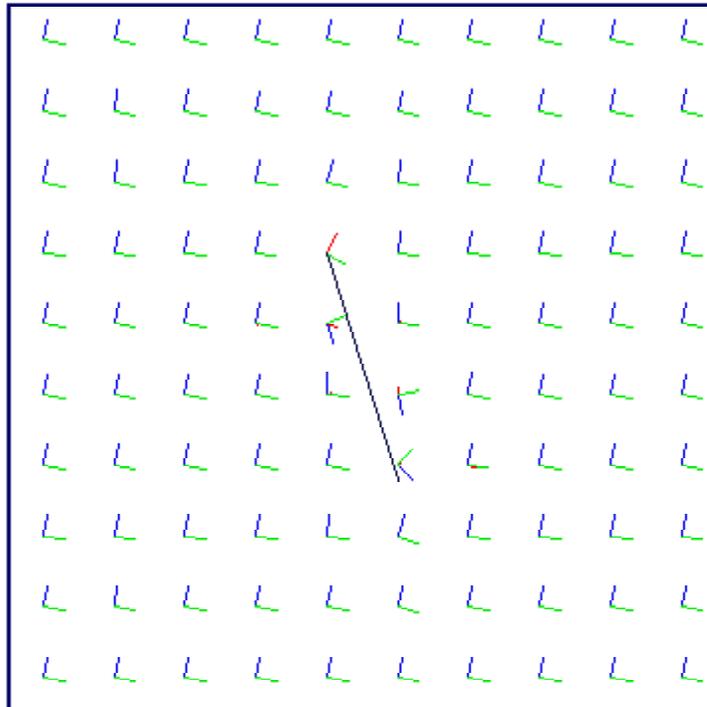
Red : right-lateral fault
Blue: left-lateral fault

Calcula
Lit retu

5. For the **Opt Strike-Slip** option, the right-lateral and left-lateral optimal planes are red and blue, respectively. You can overlay these on the stress change plots. When we do so, we often remove the planes over the negative Coulomb sites (the blue regions), because although the stress change is most positive along these planes, it is nevertheless negative, and so we would not expect aftershocks there.
6. **The Coulomb stress change on each pair of red and blue planes is identical at the point where they intersect.** For a friction coefficient equal to zero, the planes would be orthogonal, and we could compare them to aftershock focal mechanisms unambiguously. But the planes are acute for high values of friction, as explained by *King et al.* [1994]. Try checking this by selecting *Function* > **Change Friction**. For normal and thrust planes, the strike for the two planes is the same and their dip is a function of the friction coefficient.

To plot and produce a numerical output file of the principal stress axes for optimally-oriented faults

1. Choose *Data* > **Old**. Select "Example-2(RL).inp".
2. Choose *Functions* > **Slip-line Mapview**. Make sure its checked. One typically uses a coarser grid so adjacent slip-lines do not overprint, so set it to 10 x 10 km in *Graphics* > **Grid**. The slip-line length is controlled by *Graphics* > **Size** ('Exaggeration for disp. & dist. '); set this to 10,000.
3. Choose *Faults* > **Primary Axes**, also a check-item. Coulomb will read the regional stress field from the input file to draw the slip-lines. Choose *Calculation* > **Go**. Here's what you will see. Notice that the axes rotate and σ_3 deviates from being vertical close to the source, as a result of the stress changes imparted by the earthquake.



$\sigma-1$
 $\sigma-2$
 $\sigma-3$

- Now to get the numerical file, choose *Function* > **Coulomb Mapview** and then *Faults* > **Opt fault**. This will produce a 'Coul.Map-opt.cou' file, which you can open in Excel. Only part of the file is shown here.

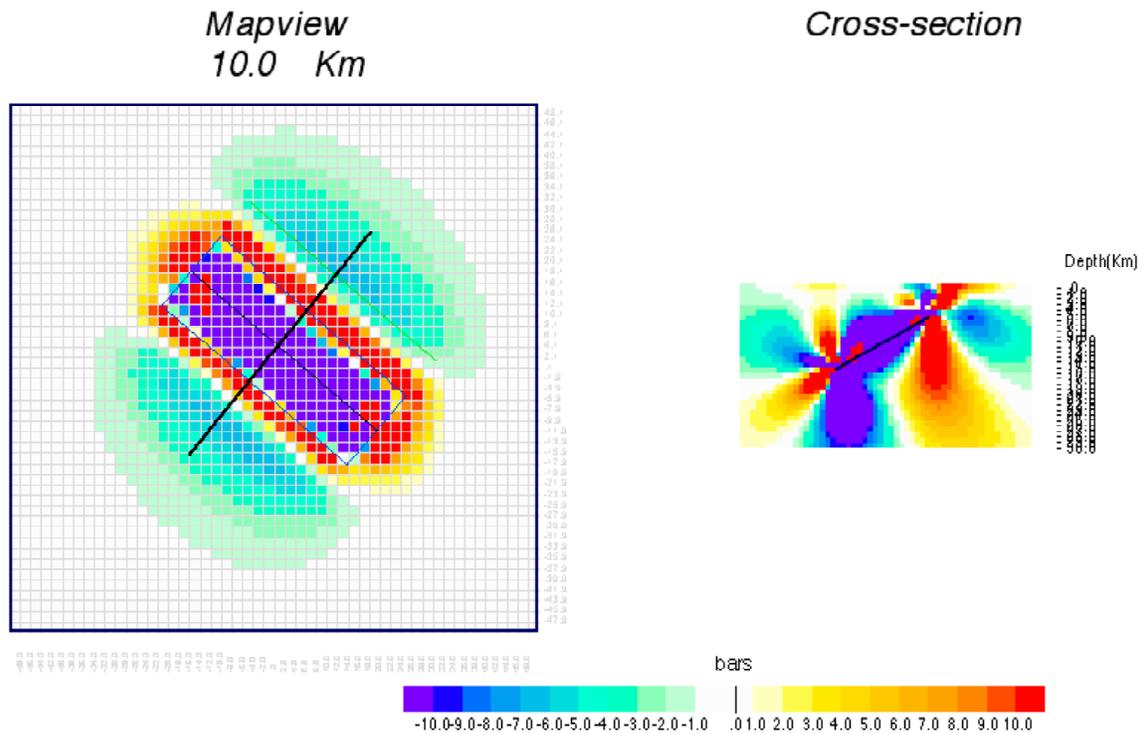
Coul-Map-Opt.cou															
A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
Selected input file: Example-2(RL).inp															
x	y	z	Coulomb	sig3	sign	opt-strike-1	opt-dip-1	opt-rake-1	opt-strike-2	opt-dip-2	opt-rake-2	prin1	p1-strike	p1-dip	prin2
-45	-45	8	0.25	0.22	0.06	204.34	89.89	179.84	136.15	89.81	0.04	-99.64	9.76	-0.07	-29.99
-45	-35	8	0.45	0.39	0.16	204.32	89.85	179.84	136.12	89.80	0.07	-99.85	9.78	-0.05	-29.99
-45	-25	8	0.56	0.44	0.31	204.18	89.86	179.89	135.98	89.85	0.09	-100.14	9.92	-0.01	-29.99
-45	-15	8	0.36	0.22	0.35	203.95	89.95	-179.99	135.75	89.99	0.05	-100.35	10.15	0.04	-30.00
-45	-5	8	-0.24	-0.27	0.08	23.77	89.86	179.82	315.57	89.79	0.06	-100.36	10.33	0.07	-30.01
-45	5	8	-1.02	-0.79	-0.58	23.81	89.63	179.69	315.61	89.57	0.23	-100.17	10.29	0.05	-30.01
-45	15	8	-1.45	-0.91	-1.35	24.08	89.48	179.66	315.89	89.49	0.36	-100.04	10.01	-0.01	-30.02
-45	25	8	-1.21	-0.54	-1.69	24.38	89.50	179.74	316.18	89.58	0.37	-100.16	9.72	-0.07	-30.02
-45	35	8	-0.66	-0.08	-1.45	24.47	89.64	179.86	316.27	89.74	0.29	-100.38	9.63	-0.09	-30.02
-45	45	8	-0.23	0.17	-0.99	24.40	89.78	179.95	316.20	89.87	0.19	-100.46	9.70	-0.08	-30.01
-35	-45	8	0.13	0.12	0.03	204.46	89.88	179.76	136.27	89.73	0.02	-99.21	9.64	-0.13	-29.98
-35	-35	8	0.49	0.43	0.14	204.51	89.81	179.72	136.32	89.67	0.07	-99.43	9.59	-0.13	-29.99
-35	-25	8	0.82	0.67	0.38	204.38	89.79	179.76	136.18	89.70	0.11	-99.90	9.72	-0.07	-29.99
-35	-15	8	0.67	0.47	0.51	204.03	89.90	179.94	135.83	89.90	0.07	-100.37	10.07	0.00	-30.00

To plot Coulomb Stress change in Cross-section

Here we use **Opt Thrust**, where the fault dip is not fixed. Rather, the dip maintains a fixed angle to the total (earthquake + regional) σ_1 , and σ_1 need not be horizontal close to the earthquake source, where the earthquake stress change can be large. Thus **Opt thrust** and **Opt thrust [dip fixed]** should differ only near the source, where **Opt thrust** will be

more positive (redder). For **Opt Fault**, not only the dip but the rake is also free to change to optimize the positive Coulomb stress change. Therefore Coulomb stress changes for **Opt Fault** will generally be the most positive of all (although we have not fully tested **Opt Fault**, so beware).

For this example, choose *Data > Old*; select 'Example-2(TH).inp'. Then *Functions > Coulomb x-section, Faults > Opt Thrust*. Then *Calculate > Go*.

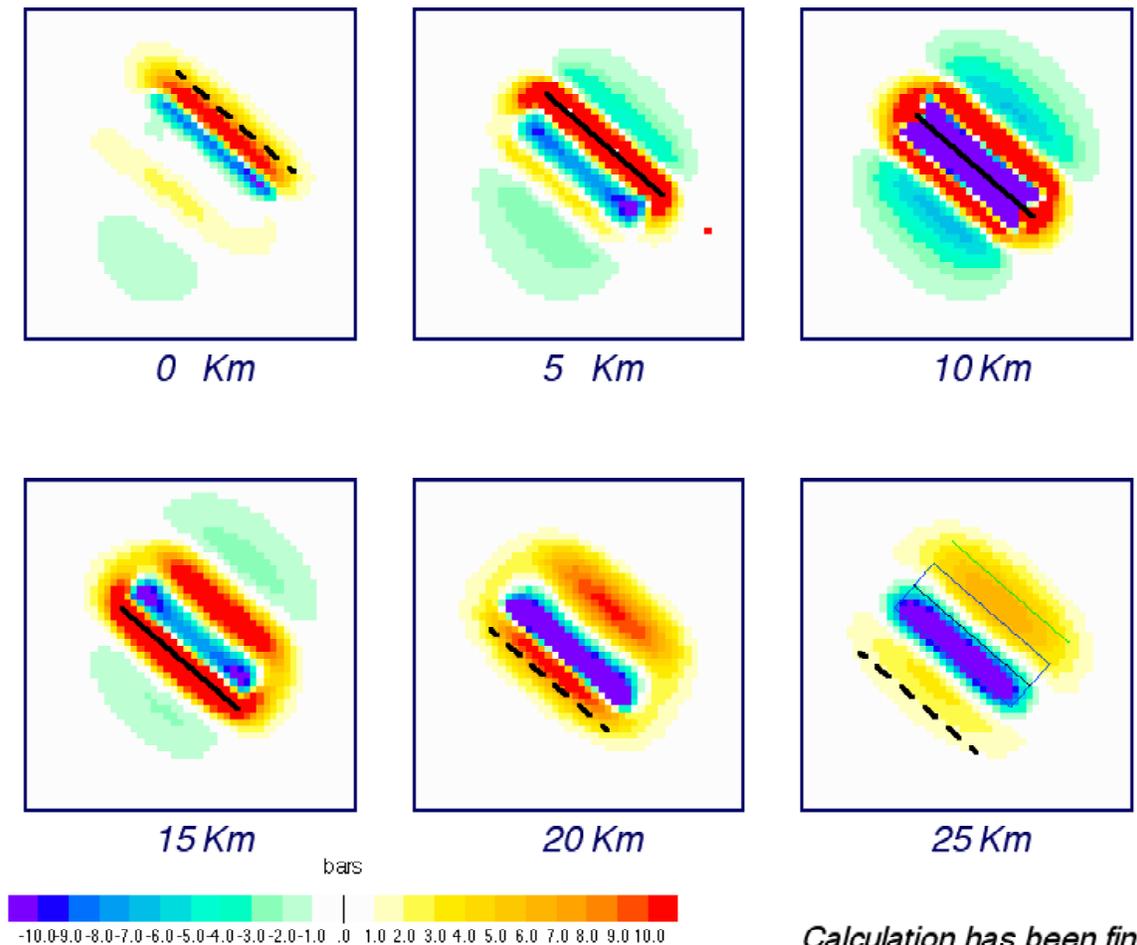


Coulomb cross-sections do not have to be vertical. An invaluable use of dipping cross-sections is explained on p. 59, in the section, 'Graphically displaying stress changes resolved onto receiver faults.'

To make a 3D image of the Coulomb stress change

In addition to the map view plot of the calculated stress result, you can make pseudo-3D images constructed from successive mapview planes. These are especially useful for studying stress changes associated with dip-slip faults, because unlike strike-slip faults, the stress changes are very depth-dependent. Numerical values can also be saved for export into scientific image rendering software, such as T3D in Noesys.

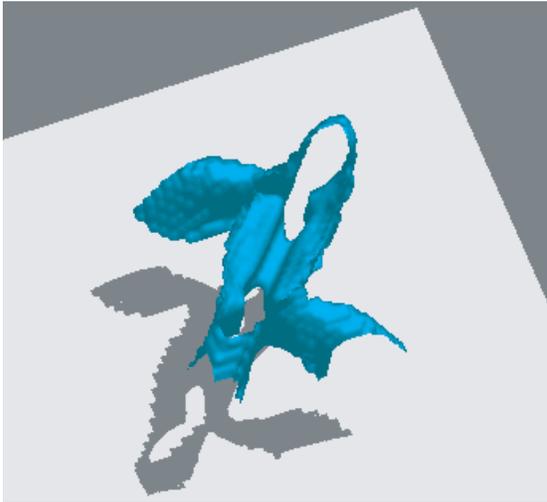
1. Select *Data* > **Old**, and use 'Example-2(TH).inp'. When you choose *Functions* > Coulomb 3D, the new window opens and you can input the depth range of the 3D image and the sampling increment in depth (the z-direction). Try, 0, 25, 5 for the upper depth, lower depth, and depth increment. Then select *Faults* > **Opt Thrust**, and *Calculation* > **Go**.
2. To get images quickly onto the screen, it would be better if the number of slices were no more than six, because one can only view 6 small map views in the *Graphics* window. You can then re-run with a finer depth increment, and open the file, checked as *Data* > **Save LightRes**, into T3D. Note that the bold black line changes position with depth, because this line is drawn where the fault plane (if solid), or its up- or down-dip projection (if dashed) intersects the horizontal plane at the stress-sampling depth.



3. To save the 3-D text column file, *Data* > **Save LightRes** (a check-item) before the calculation. See the figure on the following page, rendered with T3D in Noesys. For a good

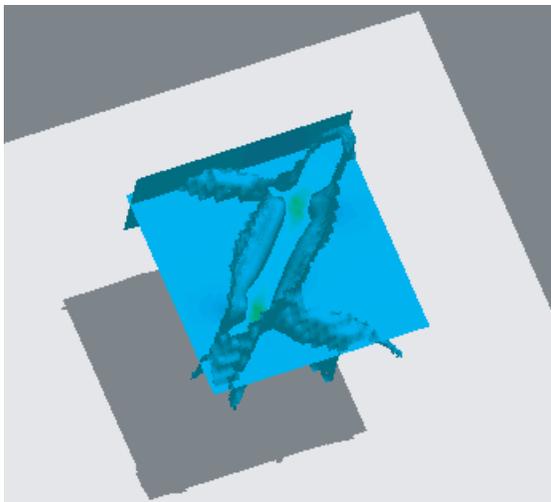
3D image, we recommend setting the x and y increments to be equal. Keep in mind that 3-D text column files tend to become huge, but T3D has no problem opening them.

3 D images made by Coul-3D-LightRes.cln with T3D (Noesys)

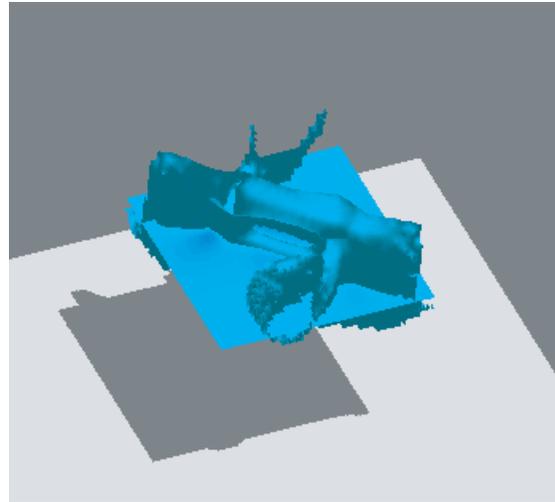


Isosurface rendering (e.g., 0-bar surface)

Examples



Slice rendering + Isosurface rendering



T3D can be purchased from www.rsinc.com/noesys

Chapter 7

TUTORIAL FOR ADVANCED FEATURES

Resolved stress change on mapped faults

To study the stress changes on active ‘receiver’ faults surrounding a ‘source’ fault (with slip or opening), we must resolve the stress change on the receiver faults, which depends on their location, geometry and inferred slip vector. It is difficult to obtain such values from the gridded calculations you have thus far learned to make. Here we place the receiver faults in the input file, with no slip (the two columns for the slip components in the input file are both set to zero). For simplicity, we always list these receiver faults after the source fault(s). We use “Example-1.inp” for this calculation. Open the input file using text-editor software such as EditII.

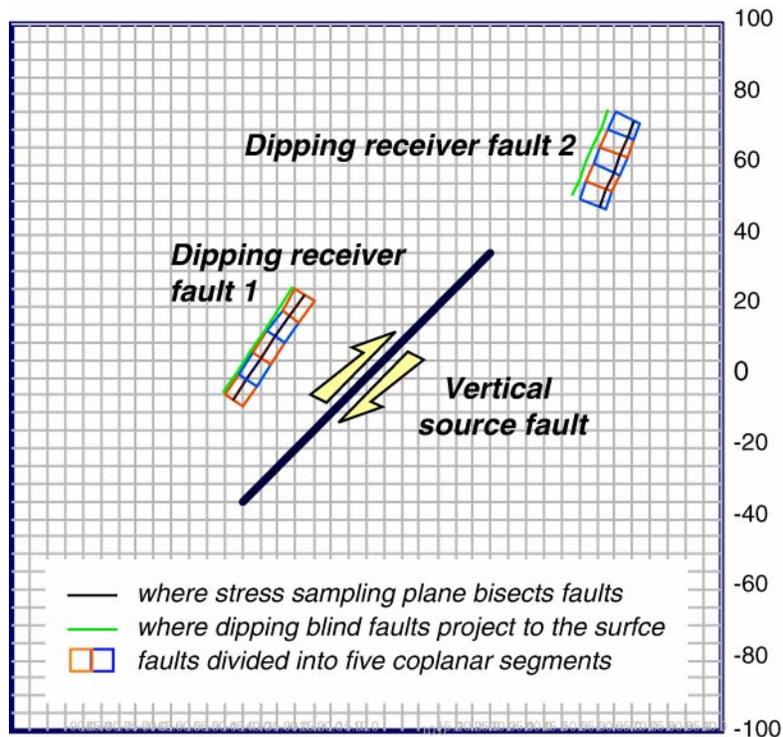
```

# X-start Y-start X-fin Y-fin Kode shear(m) reverse(m) dip angle top(km) bot(km)
xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx
1 -35.00 -35.00 35.00 35.01 100 2.00 0.00 90.00 0.00 12.00 source
5 -40.00 -5.00 -20.01 25.00 100 0.00 0.00 65.00 2.00 15.00 receiver-1
5 60.00 50.50 70.10 75.05 100 0.00 0.00 45.00 2.00 10.00 receiver-2

```

You can see three faults listed (three rows) in ‘Example-1.inp’ (above), but the two have no slip. These are receiver faults. In addition, you can see the first column in the fault list (labelled #, for number of segments), we have divided each receiver fault into five elements. This enables you to sample the stress along the receiver fault plane without the cumbersome task of inputting many faults. Now launch Coulomb. The following procedure is used for the element calculation.

1. Choose *Data* > **Old-Elements**.
2. Now, you can see that two receiver faults produce 10 lines of ‘Element Displacements.’ Put the main window in front by clicking on it anywhere. Now choose *Functions* > **Grid & Faults**; then *Calculation* > **G o**. We have annotated below what you will see on the screen.



3. Now choose *Data* > **Save ElementC**. This is a check-item, which produces the "ElementCondition.cou" file automatically in the program folder.
4. Choose *Functions* > **Coulomb Mapview** (or Coulomb x-section or 3D). For the purpose of examining the stress condition on elements, it does not matter which you choose. An 'ElementCondition.cou' file will be created as long as few as two points of a grid are calculated, so to save computation time, increase the *x*, *y* increments so the grid is very sparse. (Or, you can simultaneously save the numerical Coulomb values at the grid points, as you have done before, and the Element Condition file, by selecting *Data* > **Save LightRes**. Both items should then be checked in *Data*.)
5. Choose *Calculation* > **Go**. The graphic window appears. You can ignore the coarse stress result calculated on grid in this case. Hit <rtrn> and the main window indicates that a file, "ElementCondition.cou", has been saved.
6. Quit Coulomb and launch Excel.
7. In the Excel, choose *File* > **Open** and open "ElementCondition.cou". Because the file is tab-delimited, just hit **Finish** and all columns will be labelled.

Selected input file: Example-1.inp

Fault #	Kode #	X-center (km)	Y-center (km)	Z-Calcd. depth (km)	length (km)	strike (degree)	dip (deg)	lateral slip (m)	dip slip (m)	sig-right (bar)	sig-up (bar)	sig-max (bar)	rake-max (degree)	Source Fault
1	100	-2.5E-07	5.0E-03	8.0	99.00	44.996	90	2.0	0.0	-31.33	0.00	31.33	0.00	Receiver Fault 1
2	100	-35.67	-3.55	8.0	7.21	213.68	65	0.0	0.0	-2.45	-0.16	2.45	-3.80	
3	100	-31.68	2.45	8.0	7.21	213.68	65	0.0	0.0	-2.58	-0.18	2.59	-3.93	
4	100	-27.68	8.45	8.0	7.21	213.68	65	0.0	0.0	-2.50	-0.19	2.51	-4.39	Receiver Fault 2
5	100	-23.68	14.45	8.0	7.21	213.68	65	0.0	0.0	-2.31	-0.20	2.32	-4.85	
6	100	-19.68	20.45	8.0	7.21	213.68	65	0.0	0.0	-2.04	-0.19	2.05	-5.26	
7	100	66.56	50.67	8.0	5.31	202.36	45	0.0	0.0	-0.04	0.12	0.13	72.62	Receiver Fault 2
8	100	68.58	55.58	8.0	5.31	202.36	45	0.0	0.0	0.07	0.15	0.16	114.73	
9	100	70.60	60.49	8.0	5.31	202.36	45	0.0	0.0	0.12	0.14	0.19	129.58	
10	100	72.62	65.40	8.0	5.31	202.36	45	0.0	0.0	0.13	0.12	0.18	137.17	
11	100	74.64	70.31	8.0	5.31	202.36	45	0.0	0.0	0.13	0.11	0.17	141.95	

Lateral slip column of Kode 105 faults shows free slip results of receiver faults

8. In the Excel spreadsheet (*only the left half of the file is shown above*), each source and receiver fault element is in its own row, along with the (x, y) coordinates of the center of each element at the target depth. In addition, the segment geometry, slip (non-zero for at least one component for sources, and zero for receivers), shear, normal, and Coulomb stress; and the stress tensor components. We generally delete the rows for the source(s), particularly if there are hundreds, as is often the case for a variable-slip earthquake models. The EC files show the horizontal (right- or left-lateral), dip-slip, and maximum resolved shear stress and its associated rake direction, the normal stress change, Coulomb stress, and a myriad of other components. [See page 73 for an invaluable annotated 'Element Condition' file.](#)

Graphically displaying stress changes resolved onto receiver faults

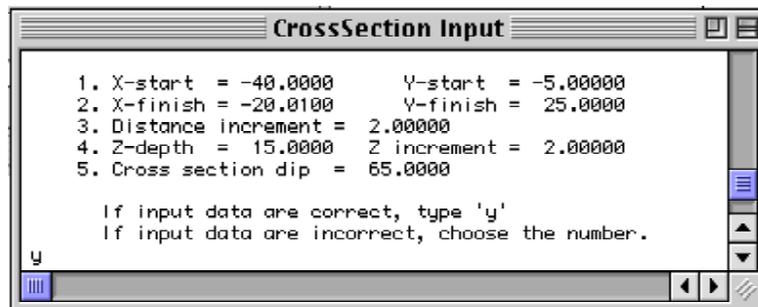
Although the stress changes on the receiver faults are tabulated in the 'ElementCondition.cou' file, often one would like to visualize these stress changes. **One can do this selecting the 'Coulomb x-section' option, and choosing a cross-section co-located with a receiver fault plane.** Such a cross-section can have any orientation, and need not be vertical. Here is an example:

Here are the three faults in Example-1.inp:

#	X-start	Y-start	X-fin	Y-fin	Kode	shear(m)	reverse(m)	dip angle	top(km)	bot(km)	
1	-35.00	-35.00	35.00	35.01	100	2.00	0.00	90.00	0.00	12.00	source
5	-40.00	-5.00	-20.01	25.00	100	0.00	0.00	65.00	2.00	15.00	receiver-1
5	60.00	50.50	70.10	75.05	100	0.00	0.00	45.00	2.00	10.00	receiver-2

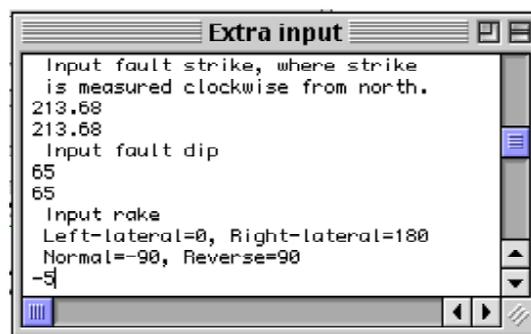
Choose *Data > Old*, and select 'Example-1.inp'. Then choose *Functions > Coulomb x-section*. Now change the x-, y-start, x-, y-finish, and the dip to be the same as for Receiver fault 1 in the input file. Next, change the Z-depth to correspond to 'bot(km)' in

the input file. Finally, let's make both the along-strike ('Distance increment') and vertical ('Z increment') increments to be 2 km. After making these changes, the final 'CrossSection Input' dialogue box should look like this:

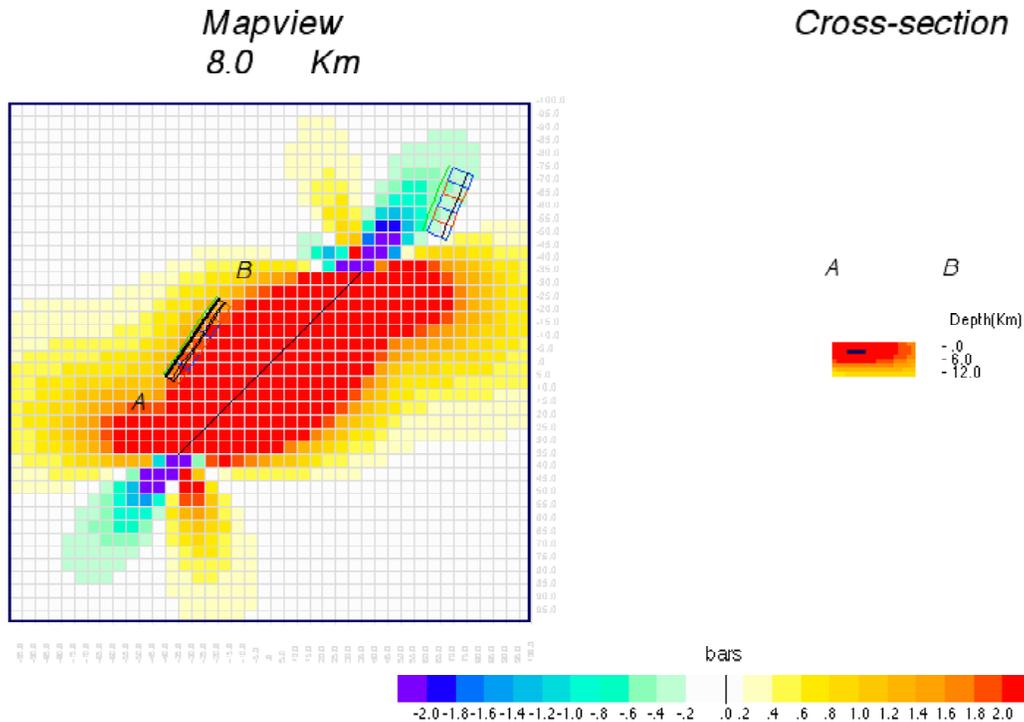


Now choose *Faults* > **Specified Fault**. Here you want to resolve the stresses onto the plane of the cross-section plane, which is the same as the Receiver fault 1 plane. So you input strike = 213.68°, dip = 65°. These values are shown in the 'ElementCondition.cou' file on page 59 for Receiver fault 1. (If you hadn't saved this file, you could re-enter 'Example-1.inp' as *Data* > **Old-Elements** and read the strike and dip of the receiver fault 1 sources from the screen).

Now choose the rake of interest to you. You could, for example, select the rake that maximizes the shear stress change (identified as *sig-max* in 'ElementCondition.cou'), called *rake-max* in 'ElementCondition.cou' above. So set rake = -5°. The 'Extra input' box should then look like this:



Now choose *Calculation* > **Go**. You can also see the A, B marking the endpoints of the cross-section. Of course you can save and edit this graphic file by hitting the *File* > **Save Graphics Window**.



You can output the numerical file of these stresses by checking *Save LightRes* and hitting **Go**; then open the resulting file, 'Coul-Sec-Spec.cou,' in Excel. Notice that you are calculating the left-lateral Coulomb stress on planes roughly parallel to the right-lateral source fault, so the stress shadow looks red. If you had chosen a rake = 180° (Coulomb stress changes on right-lateral faults), the stress shadow, as well as the stresses resolved on Receiver fault 1 in the cross-section, would be blue. You might try this.

Freely slipping boundary elements under plane stress (Kode 105 faults)

If you are studying strike-slip faulting, Coulomb allows you to calculate the amount of slip a fault would undergo to release the stress imposed by a source fault. The receiver fault is treated as a freely slipping boundary element (a fault with zero internal friction) when the Kode is set to 105, rather than 100 as used for elastic dislocations. This is based on the two-dimensional boundary element method developed by *Crouch & Starfield* (1983). Kode 105 faults must be vertical, but their top and bottom depth can be set as desired.

Here is an example of how one can use Kode 105 faults. One calculates the shear stress resolved on the San Andreas imposed by the 1992 Landers earthquake. In places it is

right-lateral; elsewhere it is left-lateral. How much would the San Andreas need to slip to shed (or relieve) this added stress? The answer obtained [King *et al.*, 1994] would be different if you let the freely slipping boundary elements lie in the upper 10 km, or if the elements extend from great depth to a minimum depth of 10 km. Further, the longer and wider a fault, the more it will slip, given the same applied stress. Another example of Kode 105 fault use is how the San Andreas in the creeping and Parkfield section would need to slip to shed the stress imposed by the 1983 Coalinga thrust earthquake 25 km away [Toda & Stein, submitted to JGR, 2001].

```

Example_5 (boundary element calculation in plane stress condition)
Fault with kode 105 should be the first row(#reg1=1). See # is 5. The fault will split into 5 elements.
#reg1= 1 #reg2= 0 #fixed= 4 sum= 1
PR1= .250 PR2= .250 DEPTH= 7.5
E1= 0.800000E+06 E2= 0.800000E+06
XSYM= .000 VSYM= .000
FRIC= .400
S1DR= 16.9900 S1DP= 0.100000E-01 S1IN= 100.000 S1GD= .000000
S2DR= 89.9900 S2DP= 89.9900 S2IN= 50.0000 S2GD= .000000
S3DR= 106.990 S3DP= -0.100000E-01 S3IN= .000000 S3GD= .000000

# X-start Y-start X-fin Y-fin Kode shear normal dip angle top bot
xxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx
5 -6.40 -26.90 5.70 -13.10 105 0.000 0.000 90.00 0.000 15.00 BB-free-slip
1 4.40 36.00 24.00 13.60 100 2.500 0.000 90.00 0.000 15.00 Landers-1
1 21.20 15.70 32.30 -6.60 100 3.500 0.000 90.00 0.000 13.00 Landers-2
1 30.30 -3.20 32.50 -27.80 100 2.000 0.000 90.00 0.000 15.00 Landers-3
1 41.20 -41.10 41.90 -49.10 100 0.500 0.000 90.00 2.000 12.50 Joshua Tree

```

Setting up the input file with Kode 105 boundary elements [see ‘Example-5-(105kode-1).inp’ above, a simplified 1992 M=7.3 Landers rupture, as a guide]:

1. You must list the Kode 105 faults *before* the elastic dislocations (kode 100 faults) in the input file. The regional stress field is ignored for this calculation.
2. Set ‘#reg 1=’ in the third line of the input file to the number of rows of Kode 105 faults. If you have one freely slipping boundary element divided into 10 patches, you still put ‘#reg1=1’, and you designate the 10 patches or elements by putting ‘10’ beneath the ### in the first 3 characters of the input line.
3. Make sure that Young’s modulus (E1) and Poisson’s ratio (PR1) for region 1 have reasonable values in the input file.
4. Ensure that the source faults listed after the receiver faults are set as Kode 100. Once you read the input file using *Data > Old-Elements*, you can immediately see the released slip on the receiver faults in the “ELEMENTS” window (*see below*). Remember the sign convention for slip: positive for right-lateral and negative for left-lateral. Thus, in this simplified example of the 1992 Landers earthquake, the Big Bear fault would need to slip 3-4 cm left-laterally to shed the stress imposed by Landers.
5. For a text output file, choose *Data > Save ElementC*. Then, *Calculate > Go*. Then open ‘Element Condition.cou’ in Excel.

Sources B.E. receivers

ELEMENTS							
Boundary element data.							
Element Kode	X(Surface) (km)	Y(Surface) (km)	X (Depth) (km)	Y (Depth) (km)	Length (km)	Strike (degree)	
1	105	-5.1900	-25.5200	-5.1900	-25.5200	3.67	41.2
2	105	-2.7700	-22.7600	-2.7700	-22.7600	3.67	41.2
3	105	-0.3500	-20.0000	-0.3500	-20.0000	3.67	41.2
4	105	2.0700	-17.2400	2.0700	-17.2400	3.67	41.2
5	105	4.4900	-14.4800	4.4900	-14.4800	3.67	41.2
6	100	14.2000	24.8000	14.2000	24.8000	29.76	138.8
7	100	26.7500	4.5500	26.7500	4.5500	24.91	153.5
8	100	31.4000	-15.5000	31.4000	-15.5000	24.70	174.9
9	100	41.5500	-45.1000	41.5500	-45.1000	8.03	175.0

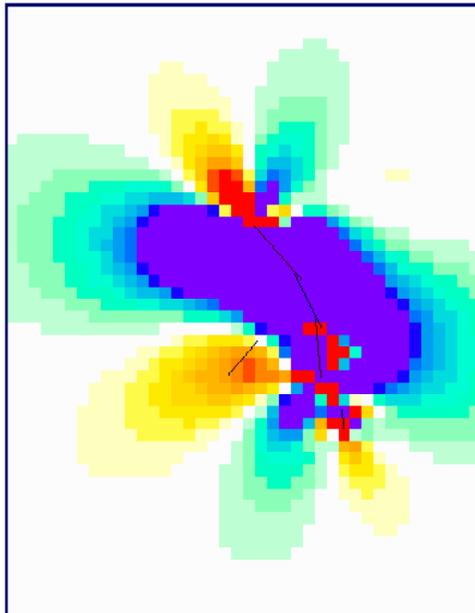
boundary element slip needed
to shed stress imposed
by source faults

Number of matrix lines = 10

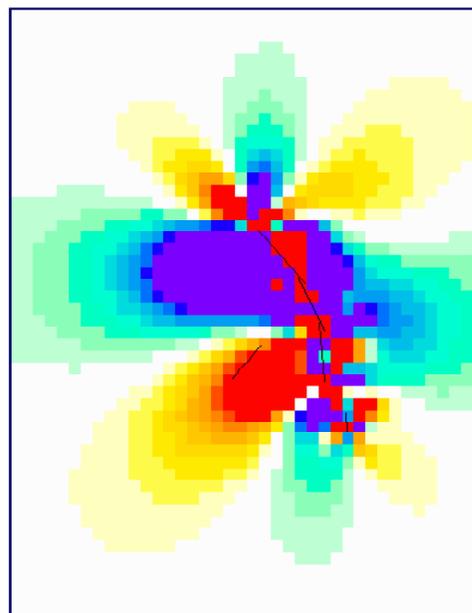
Element displacements:
(check the calculation kodes)

1) Kode = 105	Lateral Slip (m) = -0.034	Dip Slip (m) = 0.000
2) Kode = 105	Lateral Slip (m) = -0.044	Dip Slip (m) = 0.000
3) Kode = 105	Lateral Slip (m) = -0.043	Dip Slip (m) = 0.000
4) Kode = 105	Lateral Slip (m) = -0.034	Dip Slip (m) = 0.000
5) Kode = 105	Lateral Slip (m) = -0.019	Dip Slip (m) = 0.000
6) Kode = 100	Lateral Slip (m) = 2.500	Dip Slip (m) = 0.000
7) Kode = 100	Lateral Slip (m) = 3.500	Dip Slip (m) = 0.000
8) Kode = 100	Lateral Slip (m) = 2.000	Dip Slip (m) = 0.000
9) Kode = 100	Lateral Slip (m) = 0.500	Dip Slip (m) = 0.000

'Specified' fault with Big Bear parameters



'Opt strike-slip' Fault



The figure on the *left* is the Coulomb stress change ($\mu=0.4$) on **specified** left-lateral planes parallel to the Big Bear rupture (one finds a 1-bar increase at Big Bear). The figure on the *right* is the Coulomb stress change ($\mu=0.4$) on **optimally-oriented** strike-slip planes, given the N7°E regional compression axes (one sees a 2-bar increase at Big Bear). Pure red is 2.0 bars. The boundary element experiment suggests that because the fault is short, it could only slip 3-4 cm to relieve this stress increase. The Big Bear fault ruptured

in a $M=6.5$ shock 2 hr after the Landers mainshock, slipping about 50 cm in the processes.

6. Make the main 'Coulomb Output' window active again before selecting *Data > Save ElementC*. Then *Calculate > Go*. You obtain the slip values in the "ElementCondition.cou" file after you finish the Coulomb calculation. When viewing 'ElementCondition.cou' in the Excel spreadsheet, the column for "lateral slip" of the Kode 105 faults now gives the slip caused by the source fault. **Note that the program cannot calculate the dip-slip component of a Kode 105 fault, so it is always shown as zero. We have yet to implement Kode 105 for faulting in dip-slip environments.**

Crack opening or closing (Tensile Kode 200 sources)

Many magmatic intrusion problems require a dike or sill source undergoing expansion or contraction. Here we introduce the deformation and stress change caused by a tensile component of displacement, such as the opening or closing of a tabular or crack-like body, using the equations of *Okada* [1992]. For simplicity, the input file format for the Coulomb has only two slip components, labelled 'right.lat' and 'reverse'. So, when you change the Kode of the source to 200 or 300, the first of these slip components takes on a tensile-displacement interpretation by the program, as described below.

1. **Kode 200** permits tensile slip and strike-slip to be input. For Kode 200, put lateral slip in the 'rt-lat.' column and tensile slip in the 'reverse' column (*dike opening is positive; unit = meters*).
2. **Kode 300** permits tensile slip and dip-slip to be input. For Kode 300, put tensile slip in the 'rt-lat.' column (again, dike opening is positive), and put reverse slip in the 'reverse' column.
3. If the element has three components of slip (strike-slip, dip-slip, and tensile-slip), you can accomplish this by using a combination of Kode 200 and 300 elements as shown in the file below, 'Example-7(200-300kode).inp'.
4. Define the dike geometry (endpoints, dip, and top/bottom depth) as you do for faults. Unfortunately, the way we define faults/dikes does not permit one to put in a strictly horizontal plane, since the top and bottom depths would be identical, but you can put a tiny dip ($\sim 1^\circ$), and it will work fine.

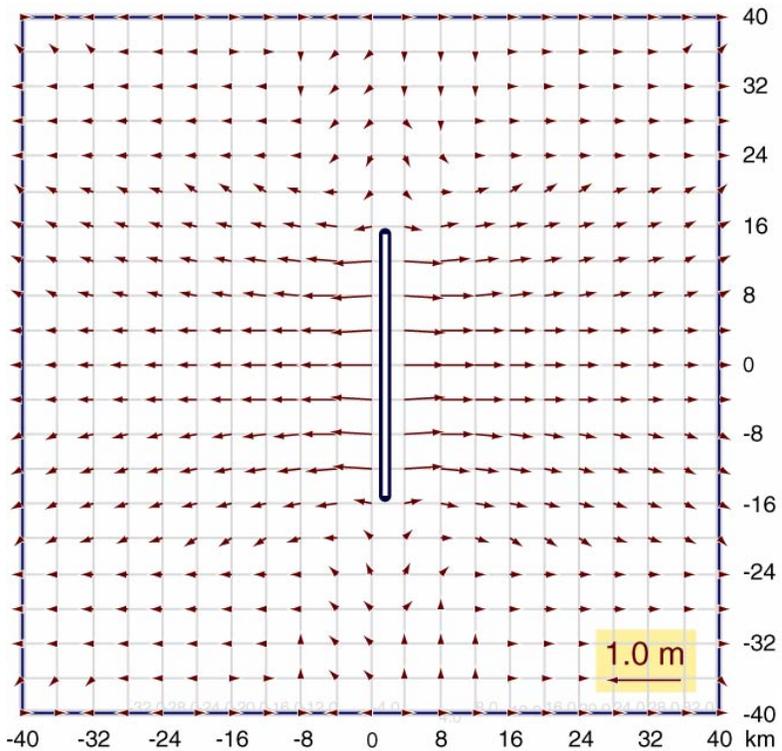
- Uniform opening or closing of a dike is likely to be unrealistic. Just as you can for faults, you can taper the displacement toward the edges by running the program, Taper, which is explained in Chapter 10.

```

Example-7(200-300kode).inp
This is a test file for the Coulomb 2.0
These two co-located tensile/shear-slip sources enable strike-slp, dip-slp, and opening to be imposed simultaneously
*reg1= 0 *reg2= 0 *fixed= 2 sym= 1
PR1= .250 PR2= .250 DEPTH= 15.
E1= 0.800000E+06 E2= 0.800000E+06
XSVM= .000 VSVM= .000
FRIC= .000
SIDR= 19.0001 SIDP= -0.0001 S1IN= 100.000 SIGD= .000000
S3DR= 89.9999 S3DP= 89.999 S3IN= 30.000 S3GD= .000000
S2DR= 109.0001 S2DP= -0.0001 S2IN= 0.000 S2GD= .000000

# X-start Y-start X-fin Y-fin Kode rt.lat reverse dip angle top bot
xxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx
1 -15.1 -17.00 15.20 17.20 200 2.0000 1.0000 60.00 5.0 25.00 rt-lat + open
1 -15.1 -17.00 15.20 17.20 300 1.0000 2.0000 60.00 5.0 25.00 open + reverse

```



Displacement due to 1 m of dike opening

Input:
Example-6(200kode).inp

Function:
Horiz. Displ. Mapview
(no reference point)

Arrowheads:
added in Illustrator

Distance scale:
edited in Illustrator

Note:
Zoom twice before saving graphic file for higher resolution.

[Point source of deflation or inflation \(Kode 500\)](#)

We have implemented the formula from Okada (1992) for a buried point source of expansion or contraction. Of course the calculated stress changes very close to such a point source are infinite and are thus unreliable, but at larger distances, this source is valuable for magmatic intrusion problems, such as studying how an intrusion event may bring other faults or dikes closer to failure. An example of this is *Nostro et al.*, Two-

way coupling between Vesuvius eruptions and southern Apennine earthquakes by elastic stress transfer, *JGR*, 103, 24,487-24,504, 1998, which is on our web site at <http://quake.usgs.gov/research/deformation/modeling/papers/nostro.html>. As with the dike/sill source, we have created two kodes for mixed volumetric and shear displacements.

```

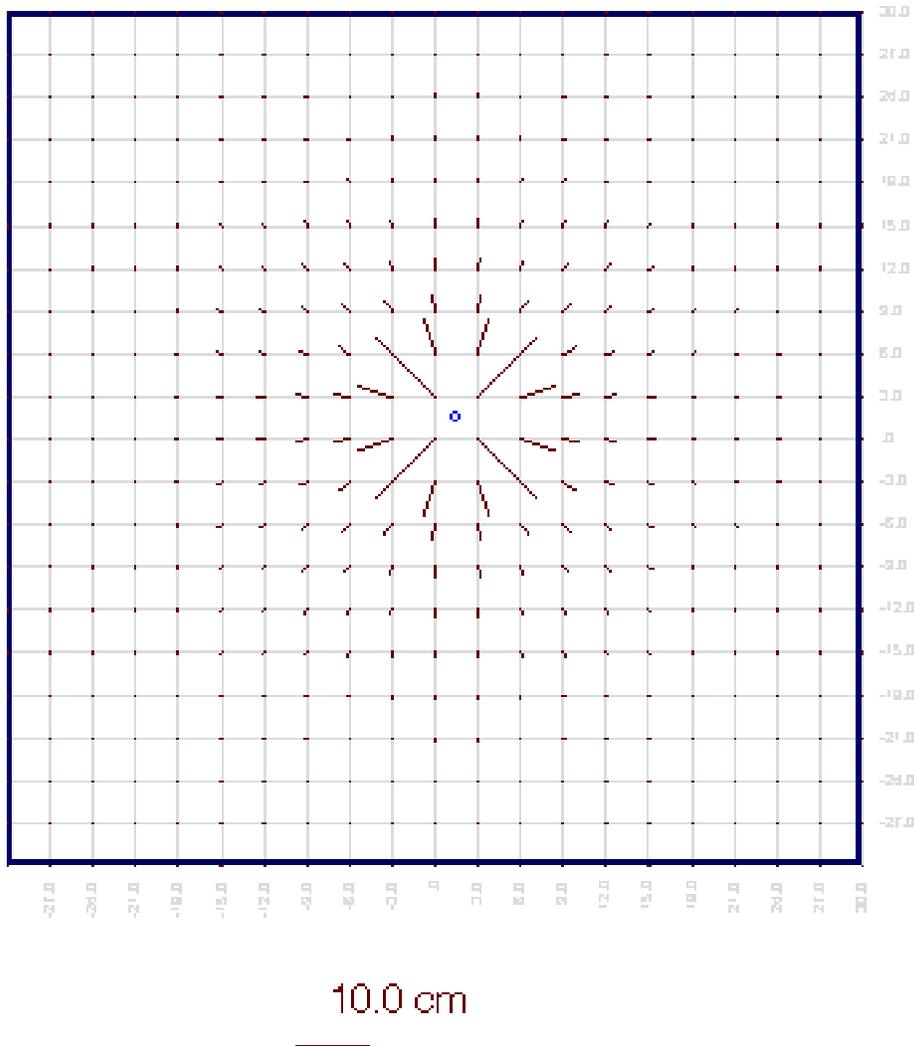
Example-9(Kode500).inp
57/1248 Macintosh HD:Coulomb f:Input Files:Example-9(Kode500).inp
This is a test file for the Coulomb 2.0
A single point source of expansion (note deflation is positive, units in m**3)
#reg1= 0 #reg2= 0 #fixed= 1 sym= 1
PR1= .250 PR2= .250 DEPTH= 0.0
E1= 0.800000E+06 E2= 0.800000E+06
XSYT= .000 YSYT= .000
FRIC= .000
S1DR= 19.0001 S1DF= -0.0001 S1IN= 100.000 S1GD= .000000
S3DR= 89.9999 S3DF= 89.999 S3IN= 30.000 S3GD= .000000
S2DR= 109.0001 S2DF= -0.0001 S2IN= 0.000 S2GD= .000000

# X-start Y-start X-fin Y-fin Kode rt.lat reverse dip angle top bot
xxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx xxxxxxxxxxxx
1 0.0 0.00 3.0 3.00 500 0.0 -2.00E+7 90.00 0.0 5.00

```

1. The unit for deflation or inflation is potency (m^3), the volumetric change associated with the evacuation or expansion of a spherical magma chamber. Contraction is positive. Potency, also called the 'geometric moment,' is equivalent to the seismic moment/G, where G is the shear modulus. As discussed previously (see boxed equation in Chapter 3), in Coulomb one inputs Young's modulus (E1) and Poisson's ratio (PR1), from which G can be computed. The units for the strike or dip slip is meters, as usual. One designates the source as Kode 500.
2. Kode 500 permits both a dike and point source of expansion to be input. Put the tensile opening in the first ('rt-lat.') column (units are m, opening is positive), and put the point source in the 'reverse' column (units are m^3 , contraction is positive).
3. The location of the point source is defined by a plane in the input file. The point is located at the midpoint of the start and end positions, and at the midpoint of the top and bottom positions. The plane defined by this geometry is used for the dike opening. So if you only calculate the point-source expansion, the length and width of the plane are unimportant; if you also want to consider opening or closing of the planar dike, then these dimensions are important.
4. The point source is indicated graphic output files as a small blue circle. Note that a dike expansion and a point-source of inflation were used in the observed and predicted GPS displacement example plotted in Chapter 4.

The following figure was made using 'Example-9(Kode500).inp' for with the *Function Horiz. Displacement Mapview*, with no reference point chosen.



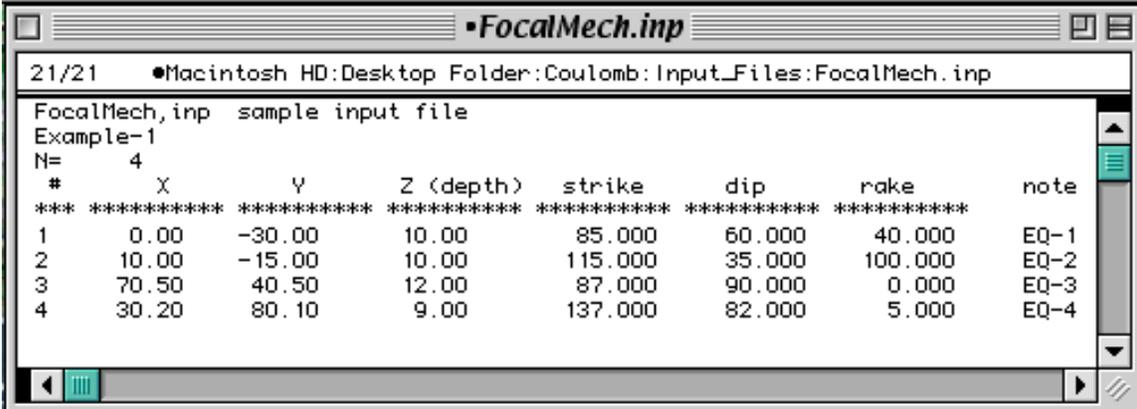
Earthquake point source (Kode 400)

To conveniently model the stress changes associated with small earthquakes or aftershocks, it may be useful to use an earthquake point-source. Such an approach is invalid close to the source, where the stress changes are infinite. Nevertheless, often one knows only the focal mechanism and the seismic moment of a shock, which is proportional to the slip times the area. A grid of point sources could also be used for a variable slip model, although this is probably easier to do by defining many adjacent rectangular patches. Here is how Kode 400 sources are defined in the input files:

1. The unit for the source is (m^3), the slip \times length \times width. Right-lateral and reverse slip are positive, as usual. The unit is equivalent to the Moment/G, where G is the shear modulus.
2. The location of the point source is defined by a plane in the input file. The point is located at the midpoint of the start and end positions, and its depth lies at the midpoint of the top and bottom positions. The plane defined by this geometry gives the strike, and dip is input as usual.
3. Point sources are indicated in graphic output files as a small blue circles. To verify the strike of the point sources, choose *Data* > **Old-Elements** and look at the 'ELEMENTS' window, or choose *Data* > **Save ElementC**, then *Calculation* > **Go**, and open the resulting 'ElementCondition.cou' file in Excel.

Resolved stresses on nodal planes of aftershocks

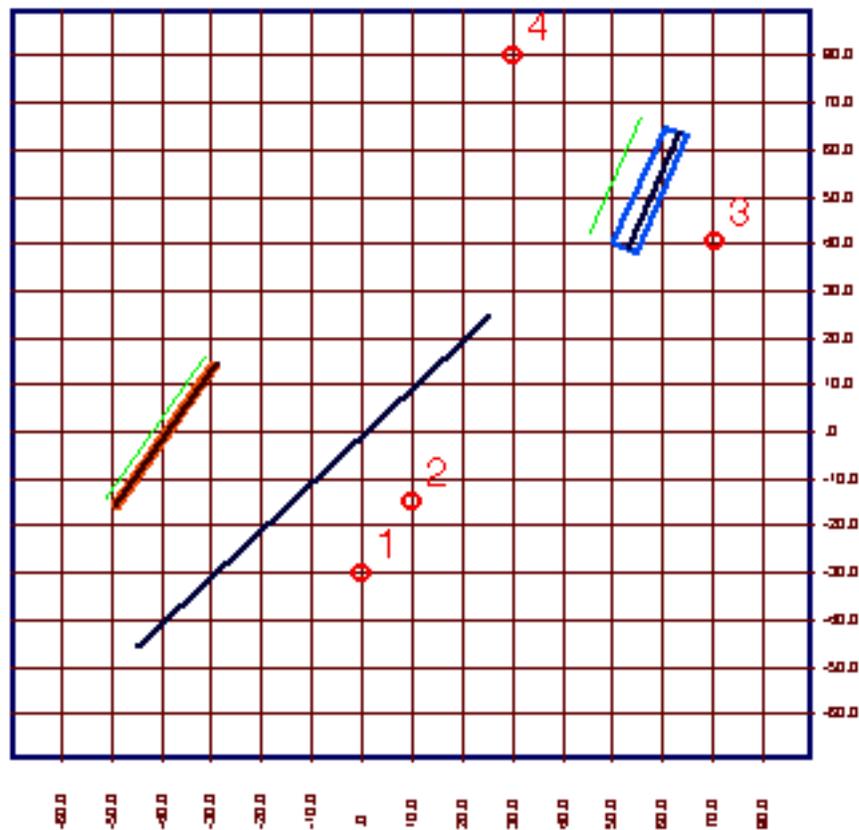
One powerful approach to Coulomb analysis is to examine stress changes on the nodal planes of seismicity or aftershocks, to answer the question, are the aftershocks brought closer to Coulomb failure by their main shock? Aftershocks are a rich source of such nodal-plane information. To examine if such earthquakes occurred on favourably oriented fault planes, the program can read another type of seismic-catalogue file (formatted for the nodal planes), and it calculates stress on the two nodal planes of each mechanism (or only one if you can eliminate one of the two alternatives). The input file is described in detail in Chapter 7. Each nodal plane is a separate line, so a typical aftershock occupies two lines, one for each nodal plane. Here is the procedure.



```
21/21 •Macintosh HD:Desktop Folder:Coulomb:Input_Files:FocalMech.inp
FocalMech,inp sample input file
Example-1
N= 4
# X Y Z (depth) strike dip rake note
*** ***** ***** ***** ***** ***** *****
1 0.00 -30.00 10.00 85.000 60.000 40.000 EQ-1
2 10.00 -15.00 10.00 115.000 35.000 100.000 EQ-2
3 70.50 40.50 12.00 87.000 90.000 0.000 EQ-3
4 30.20 80.10 9.00 137.000 82.000 5.000 EQ-4
```

1. Launch Coulomb; *Data* > **Old**, choose 'Example-1.inp' (any input file would suffice).

- To plot the numbered earthquakes on the grid, choose *Functions* > **Coulomb Mapview**. (The epicentres will not be plotted by the normal 'Plot Grid & Faults' Function).
- Choose *Faults* > **Focal Mech. File**, and then a dialog box appears.
- In the dialog box, select an input file formatted for the focal mechanism calculation (we prepared "FocalMech.inp" as an example; its in the 'input files' folder).
- Choose *Calculation* > **Go**. The grid, faults, and numbered earthquakes appear in the graphic window; the Coulomb stresses are not drawn. You can check the aftershock locations and also save this image as a figure. The calculated results are automatically saved as "FocalMech.out" in the folder where Coulomb is located.



- After quitting the program, open the "FocalMech.out" file in Excel (only the left-hand side of the file is shown below). You can examine not only the stress changes on the nodal planes given the observed rake, but also many other components (such as the maximum Coulomb stress change for the optimum rake). See chapter 8 for the detailed explanation of the components listed in "FocalMech.out".

FocalMech.out															
A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
Selected input file: Example-1.inp															
Eart #	X (km)	Y (km)	Z (km)	strike (deg)	dip (deg)	rake (deg)	sig-obs (bar)	sig-right (bar)	sig-thrust (bar)	sig-max (bar)	max-dir (degree)	sig-max(+r) (bar)	max-dir(+r) (degree)	sig-nor (bar)	coul-obs (bar)
1	0	-30	10	85	60	40	1.57	-0.37	2.01	2.05	79.73	-1.59	-139.20	-1.29	1.06
2	10	-15	10	115	35	100	0.63	2.47	0.20	2.48	175.33	-2.48	-2.55	1.92	1.40
3	70.5	40.5	12	87	90	0	-0.80	0.80	0.25	0.84	162.49	0.81	177.66	0.41	-0.64
4	30.2	80.1	9	137	82	5	-0.74	0.74	-0.13	0.75	-169.95	-0.74	5.24	-1.23	-1.24



the input
fault plane



on the
input fault
plane



on the
optimum
fault plane



with regional stress
included on the
optimal plane

Note: That you can use the 'Focal Mech. File' to calculate stress changes on receiver faults specified by strike / dip / rake / x / y / z, rather than in the standard input format of start x,y / finish x,y / rt-lat / reverse / top / bot.

This is often useful when working from standard earthquake - formatted data

Chapter 8

OUTPUT FILES

Coulomb allows you to save graphic images (pict files) and text (ascii files).

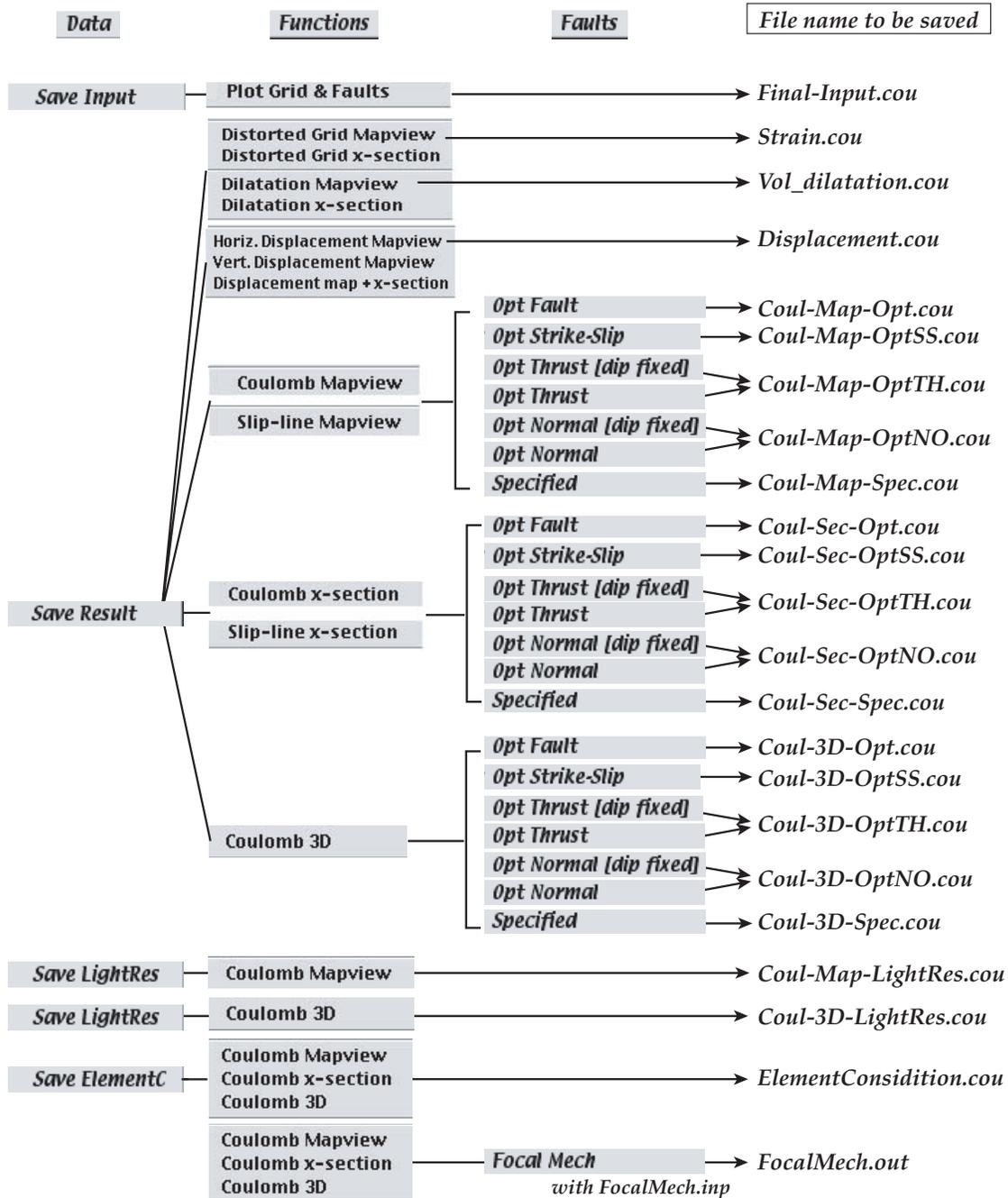
Pict-formatted files of graphic images

All graphic images are saved by choosing *File* > **Save Graphics Window...** while the graphic window ("Coulomb 2.5 Graphics") is active. You can save it to whatever folder you would like and name it whatever you wish. The saved pict-formatted file is composed of objects, such as grid lines, fault lines, color squares, etc., and a bounding frame. If you double-click the saved pict file, SimpleText automatically opens the image. To edit the file, open the file from *File* > **Open** command from Adobe Illustrator (or any graphic application that can handle pict file), and you can easily modify and group them. It is very useful, for example, to place the undistorted grid file in one layer, and the Coulomb color gradients in another layer.

Text files

The text output from Coulomb 2.5 is automatically saved in the same folder with the input file, with default names as shown in the figure "Output files (combinations & file names)." There are many kinds of column files that depend on the combination of *Data*, *Functions*, and *Faults*. Most of these are tab-delimited column files, which can be easily opened in Excel. They are also easily changed into matrix files using Transform, and the Data Utility in Noesys (or other matrix-display applications). This permits more sophisticated color gradient graphs and color rendering images, and further manipulations of the numeric files. "Final-Input.cou" should be opened by a text editor, such as EditII, rather than Excel, as it is not tab-delimited. Annotated screen shots of the various output files opened in Excel follow in the next several pages.

Output Files (combinations & file names)



Element Condition File

Kode number

The program can read four types of kode numbers.
 100: none-open fault (element) condition in a half space
 105: two dimensional lateral freely slip fault (element)
 200: crack type fault (element) condition in a half space [lateral shear + tensile]
 300: crack type fault (element) condition in a half space [tensile+ normal shear]

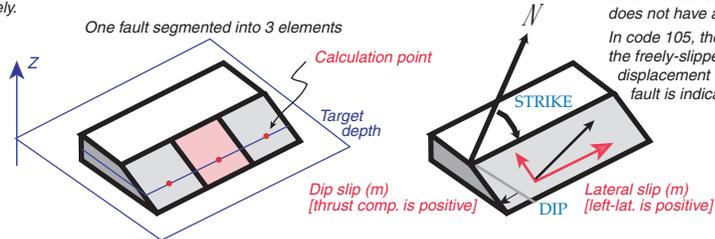
	A	B	C	D	E	F	G	H	I	J
1	Fault #	Kode #	X-center (km)	Y-center (km)	Z-Calc. depth (km)	length (km)	strike (degree)	dip (degree)	lateral slip (m)	dip slip (m)
3	1	100	2.55	0.60	7.5	38.6	40.9	90.0	-3.0	0.0
4	2	100	-23.16	-22.09	7.5	17.9	416.7	75.5	0.0	0.0
5	3	100	27.44	7.12	7.5	11.1	115.0	35.0	0.0	0.0
6										
7	Lateral slip column of Kode 105 faults shows free slip results of receiver faults									

Fault number

It numbers ascending order to the bottom.
 If a fault has multiple segmented elements, they are listed in here separately.

Coordinate (x,y,z) of the calculation point
 Depth is controlled by the input target depth.
 X, and Y indicate the center of the fault (element)

In code 100, 200, and 300, source fault has slip indicating below. Receiver fault however does not have any slip.
 In code 105, the amount of the freely-slipped lateral displacement of the receiver fault is indicated.



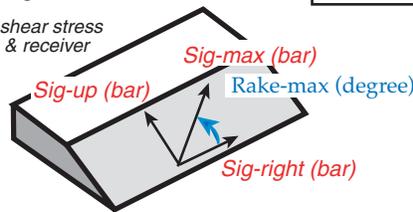
Coulomb stress change in the strike-slip direction Coulomb stress change in the dip-slip direction

K	L	M	N	O	P	Q	R	S	T
sig-right (bar)	sig-up (bar)	sig-max (bar)	rake-max (degree)	sig-max(+r) (bar)	rake-max(+r) (degree)	sig-nor (bar)	coul-s (bar)	coul-d (bar)	coul-max (bar)
80.82	0.00	80.82	180.0	-64.32	180.0	0.00	80.82	0.00	80.82
-2.87	-0.05	2.87	-1.0	1.84	18.1	-0.36	-3.02	-0.20	2.73
-0.50	2.36	2.42	78.1	1.54	94.9	0.00	-0.50	2.36	2.42

Maximum Coulomb stress change (rake indicated by 'rake-max' column)

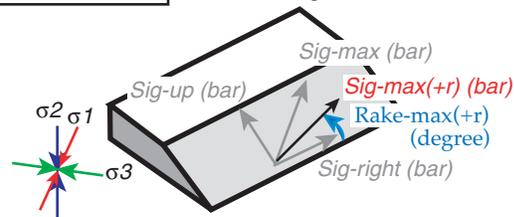
Without Regional Stress

Resolved shear stress on source & receiver faults



Common value, because of fixed fault plane

With Regional Stress



T	U	V	W	X	Y	Z	AA
coul-max (bar)	coul-max(+r) (bar)	sigfx (bar)	sigfy (bar)	sigfz (bar)	sigfxy (bar)	sigfyz (bar)	sigfyz (bar)
80.82	-64.32	-79.99	79.99	0.00	-11.51	-0.01	-0.01
2.73	1.69	0.32	-3.85	0.52	-2.53	0.02	0.35
2.42	1.54	-10.10	-2.15	1.97	1.09	-0.74	0.84

Six components of stress tensor at the calculation point

$$\text{Coul-max} = \text{Sig-max} + \mu' \times \text{Sig-nor}$$

$$\text{Coul-max (+r)} = \text{Sig-max (+r)} + \mu' \times \text{Sig-nor}$$

FocalMech.out File

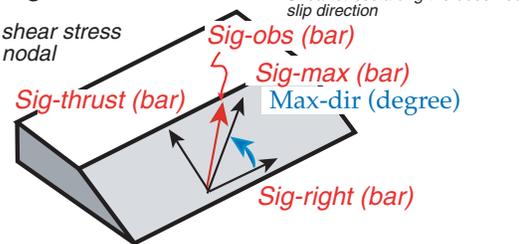
	A	B	C	D	E	F	G	H	I	J	K
1	Earthquake	X	Y	Z	strike	dip	rake	sig-obs	sig-right	sig-thrust	sig-max
2	#	(km)	(km)	(km)	(degree)	(degree)	(degree)	(bar)	(bar)	(bar)	(bar)
3	1	0	0	10	85	60	40	46.40	-34.09	31.56	46.46
4	2	1	23	10	115	35	100	2.59	0.42	2.56	2.59
5	3	12.5	7.5	12	87	90	0	47.95	-47.95	-12.68	49.60
6	4	9.2	8.1	9	137	82	5	-61.52	62.87	12.77	64.15

Cartesian coordinates (x,y,z) of the hypocenter

One of nodal planes (Aki and Richards convention)

Without Regional Stress

Resolved shear stress on one of nodal planes

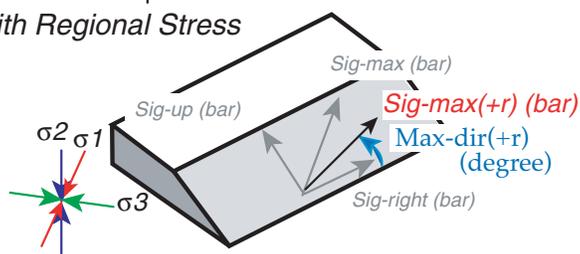


Normal stress to the nodal plane

Short notes

L	M	N	O	P	Q	R	S	T	U
max-dir (degree)	sig-max(+r) (bar)	max-dir(+r) (degree)	sig-nor (bar)	coul-obs (bar)	coul-right (bar)	coul-thrust (bar)	coul-max (bar)	coul-max(+r) (bar)	reference
42.8	43.94	23.84	-55.55	24.18	-56.31	9.34	24.24	21.72	EQ-1
99.3	0.03	9.91	-2.35	1.65	-0.52	1.62	1.65	-0.91	EQ-2
-14.8	49.52	-11.61	-101.03	7.54	-88.36	-53.09	9.19	9.11	EQ-3
168.5	53.31	134.73	-16.80	-68.24	56.15	6.04	57.43	46.59	EQ-4

With Regional Stress



$$\begin{aligned} \text{Coul-obs} &= \text{Sig-obs} + \mu' \times \text{Sig-nor} \\ \text{Coul-right} &= \text{Sig-right} + \mu' \times \text{Sig-nor} \\ \text{Coul-thrust} &= \text{Sig-thrust} + \mu' \times \text{Sig-nor} \\ \text{Coul-max} &= \text{Sig-max} + \mu' \times \text{Sig-nor} \\ \text{Coul-max (+r)} &= \text{Sig-max (+r)} + \mu' \times \text{Sig-nor} \end{aligned}$$

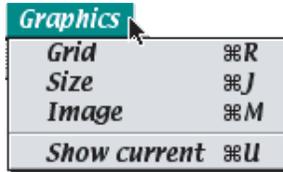
Chapter 9

GRAPHIC TOOLS

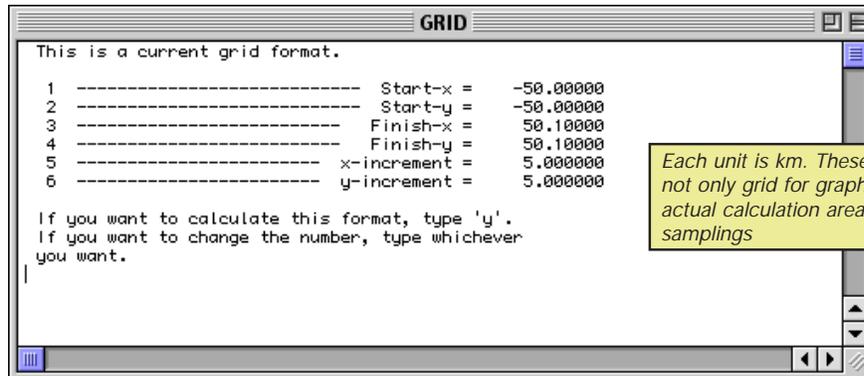
Most graphic settings are specified in the input file, such as the grid boundaries and increments, and the plot size and exaggeration for displacements and vectors. However, these can be interactively changed from the Graphics menu. In addition, certain graphic features are only accessed through the Graphics menu (see next figure). The default color bar scale, for example can be changed to a more detailed color scale, and for some plots, a gray scale. The color of the source and receiver faults can also be implemented through the graphics window. A common use of the Graphics menu is to change the 'Shade/Color increment' of a Coulomb plot so that the range of colors is fully exploited in the plot, or to saturate the colors in certain regions.

When you are in the Graphics Window with a plot on the screen, note the features you would like to change to alter or enhance your plot. Then hit return in the graphics window to close the plot. Now go to the Graphics menu to make these changes. When you are finished, re-make the plot by selecting *Calculation > Go*.

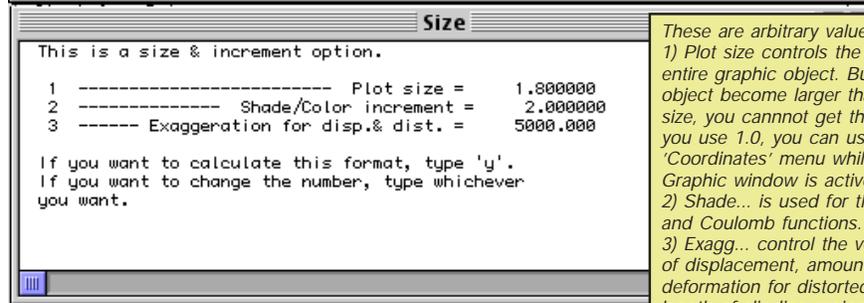
To control graphic objects in the 'Coulomb 2.5 Graphics' window



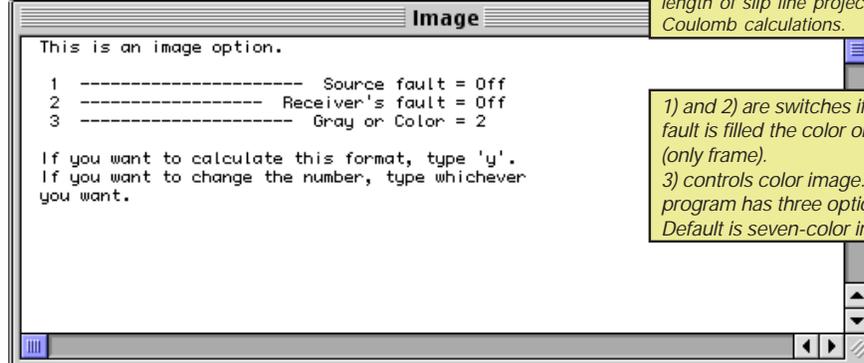
You must return to the main window to choose a submenu in Graphics menu.
As soon as you choose one of them, a small window appears. You can change the parameters in the window. And then, you have to type 0 and hit return key to close the window, before the calculation



Each unit is km. These changes not only grid for graphics but actual calculation area and samplings



These are arbitrary values.
1) Plot size controls the size of entire graphic object. But, if the object become larger than window size, you cannot get the image. If you use 1.0, you can use 'Coordinates' menu while the Graphic window is active.
2) Shade... is used for the dilatation and Coulomb functions.
3) Exagg... control the vector length of displacement, amount of deformation for distorted grid, length of slip line project for Coulomb calculations.



1) and 2) are switches if the fault is filled the color or not (only frame).
3) controls color image. The program has three options. Default is seven-color image.

Show current (command U) also opens a window, in which you can check the current settings of all graphic control parameters.

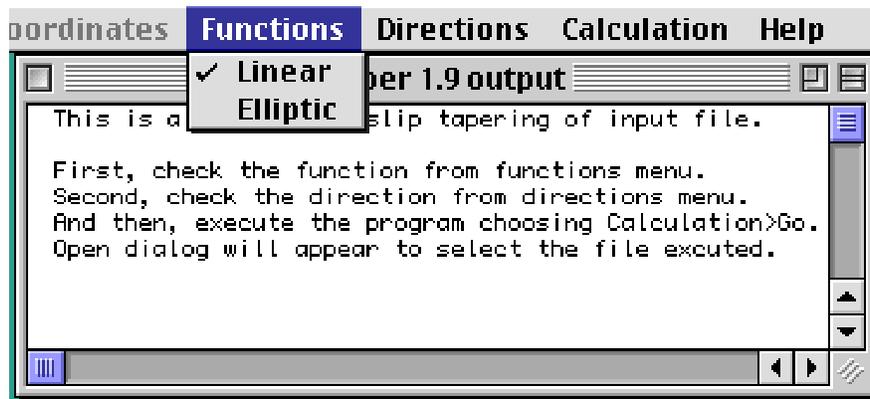
Chapter 10

Taper and Splitter

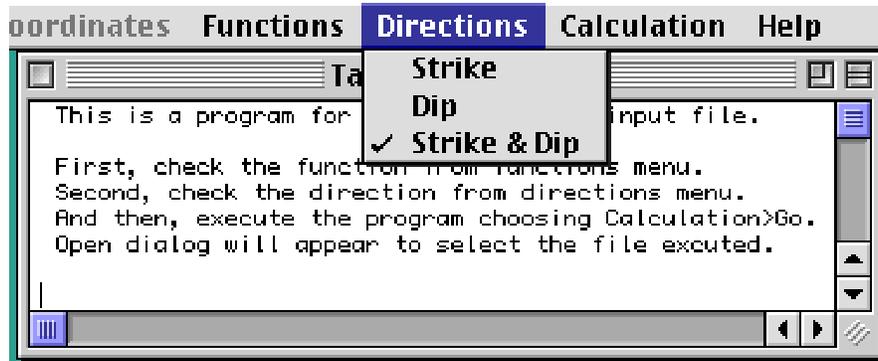
Taper is a useful stand-alone utility that allows you to create input files for planar faults or dikes with slip that varies in a regular manner along strike or down dip. Uniform slip models create stress concentrations near the fault ends that are unrealistic; taper enables you to avoid this. One reads in an input file with a standard rectangular fault or faults. One then interactively selects how slip is to diminish or taper toward the ends. **Taper also calculates the seismic moment of each fault in an input file, which is useful even if you do not want to create a tapered input file.** Taper controls the slip so that the seismic moment of the tapered-slip fault is identical to the starting uniform-slip fault. A new input file ('Tapered.in') is created in the folder in which you placed Taper. You should rename Tapered.in, because Taper will overwrite successive 'Tapered.in' files. This input file can then be read into Coulomb by *Data* > **Old**.

Operation of Taper

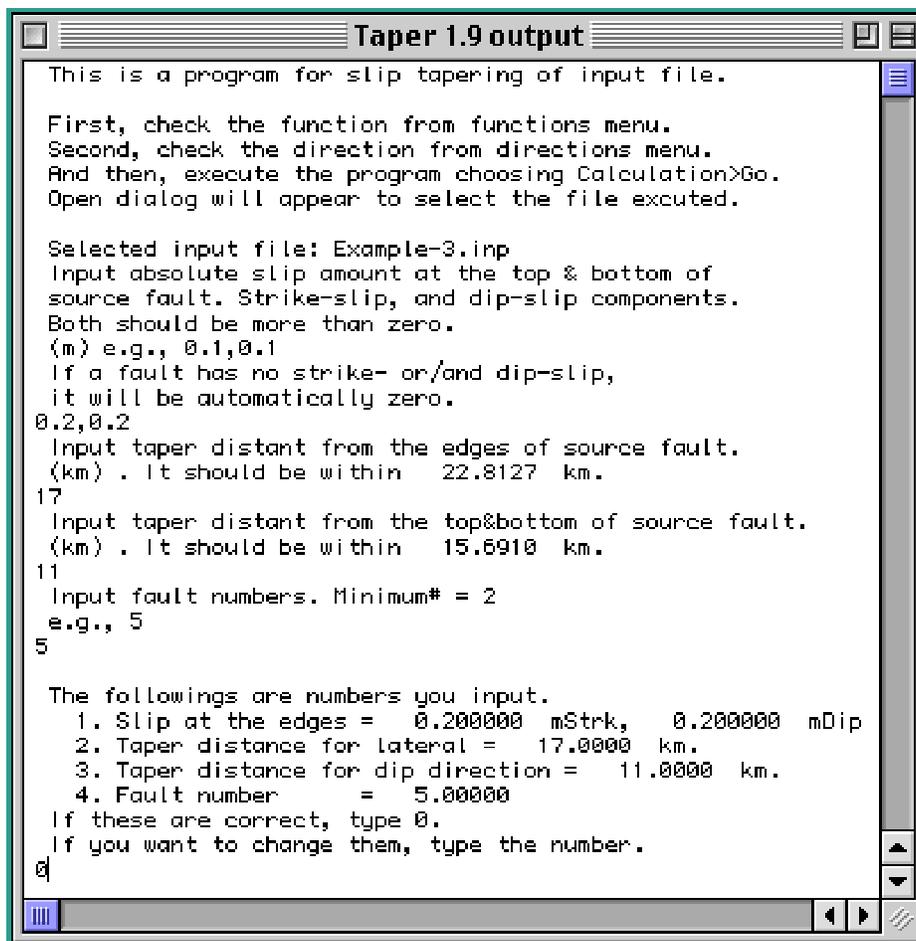
Launch Taper, and check the taper type (elliptical or linear), and direction (along-strike, down-dip, or both). A 'linear' taper can be a triangle or a trapezoid.



Then choose *Calculation* > **Go**, and you will be asked for an input file. In this example, we have selected 'Example-3.inp' which contains one 35°-dipping blind thrust fault.



Answer the screen queries, which include the minimum slip at the edges, the distance inward from the fault edges that you want the slip to taper, and the number of nested fault patches or elements (the more you use, the smoother the slip, but the longer Coulomb needs to run).



The hanging-indent is a reply in the screen shot above. Choose a along-strike or down-dip taper value at least a few km less than the maximum given in the query. The program will calculate the seismic moment and output this on the screen (**write the M_0 down and put it in one of the comment lines in the input file; this is not written to the input file it creates**). To exit Taper, click once or choose Quit from the File menu.

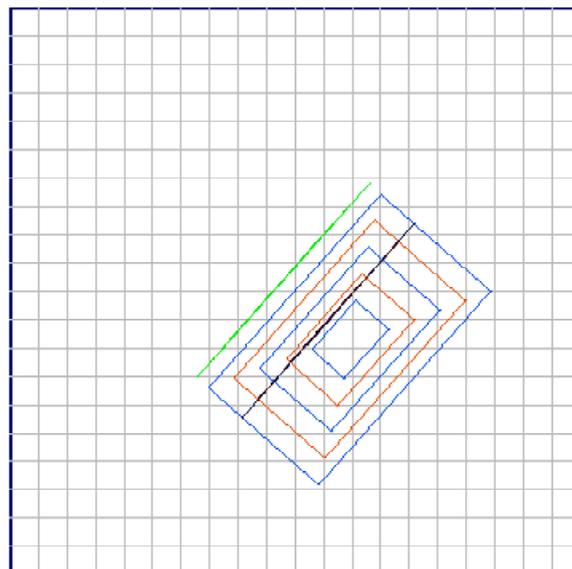
Here is the uniform-slip fault source from the input file read by Taper ('Example-3.inp'):

```
# X-start Y-start X-fin Y-fin Kode rt.lat reverse dip angle top bot
xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx
1 -15.0 -17.00 15.20 17.20 100 0.0000 3.0000 35.00 2.0 20.00
```

Below are the 5 fault sources from the output file, after we requested a linear taper along strike and down-dip.

```
# X-start Y-start X-fin Y-fin Kode shear normal dip angle top bot
xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx
1 -15.00 -17.00 15.20 17.20 100 0.0000 0.5196 35.00 2.000 20.00
1 -10.50 -15.31 14.08 12.52 100 0.0000 1.819 35.00 3.577 18.42
1 -5.997 -13.61 12.95 7.846 100 0.0000 1.819 35.00 5.155 16.85
1 -1.495 -11.92 11.83 3.170 100 0.0000 1.819 35.00 6.732 15.27
1 3.007 -10.22 10.70 -1.507 100 0.0000 1.819 35.00 8.309 13.69
```

Notice in the listing above and in the grid plot below that Taper builds the file by superimposing nested sources, each with a fraction of the total slip. Also notice that the maximum slip in the center of the fault (what you'd get by adding up the slip for all five sources) is larger than the uniform slip we started with, because the mean slip must be the same in both cases.



This file (again, *renaming is advised*) can now be read by Coulomb. Shown above is a plot of the 'grid & faults' in Coulomb. The nested patches permit the tapered slip. As usual, the black line is where the fault plane intersects the target depth, and the green line is the up-dip projection of the fault to the surface.

The calculation time for a Coulomb run is roughly proportional to the number of sources x the number of grid points; more time is required when one checks **Save Result**. So a 10-segment tapered slip fault will increase the run time by an order of magnitude. However, most runs go quickly, and so this will not be a problem. We often run variable-slip models with 100 source faults and things still go lickity-split on a G3 PowerBook, which is not a particularly fast Mac.

Splitter lets you split or subdivide a planar fault into any number of rectangular patches. These can then be assigned slip to create a 'variable slip' input model, or they can be used to resolve stress as in an 'Element Conditions' file.

Operation of Splitter

1. Open an input file (use 'Example-3.inp'). Change the number of patches (or segments) along strike in the first ### column from 1 to 10, and save it with a new name (e.g., 'Splitter-3.inp').

```

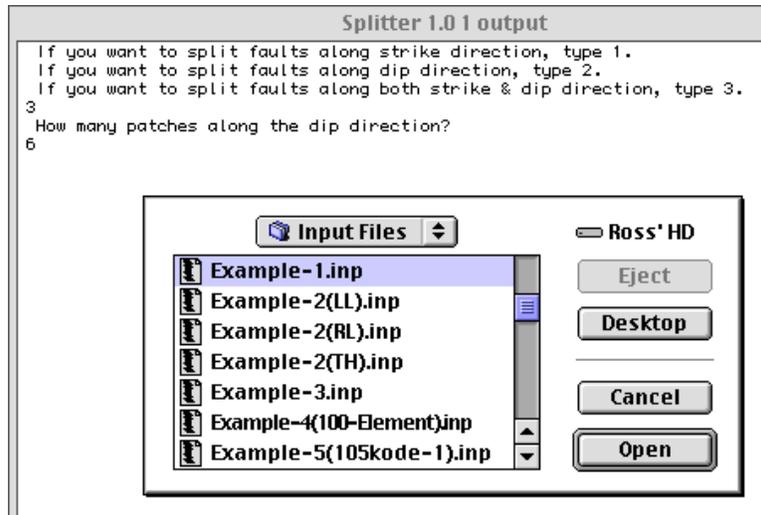
This is a test file for the Coulomb 2.5
This file is prepared to check mainly thrust faulting calculation
#reg1= 0 #reg2= 0 #fixed= 1 sum= 1
PR1= .250 PR2= .250 DEPTH= 7.5
E1= 0.800000E+06 E2= 0.800000E+06
XSVM= .000 VSVM= .000
FRIC= .400
S1DR= 19.0001 S1DP= -0.0001 S1IN= 100.000 S1GD= .000000
S3DR= 89.9999 S3DP= 89.999 S3IN= 30.000 S3GD= .000000
S2DR= 109.0001 S2DP= -0.0001 S2IN= 0.000 S2GD= .000000

# X-start Y-start X-fin Y-fin Kode rt.lat reverse dip angle top bot
xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx xxxxxxxxxxx
10 -15.0 -17.00 15.20 17.20 100 0.0000 3.0000 35.00 2.0 20.00

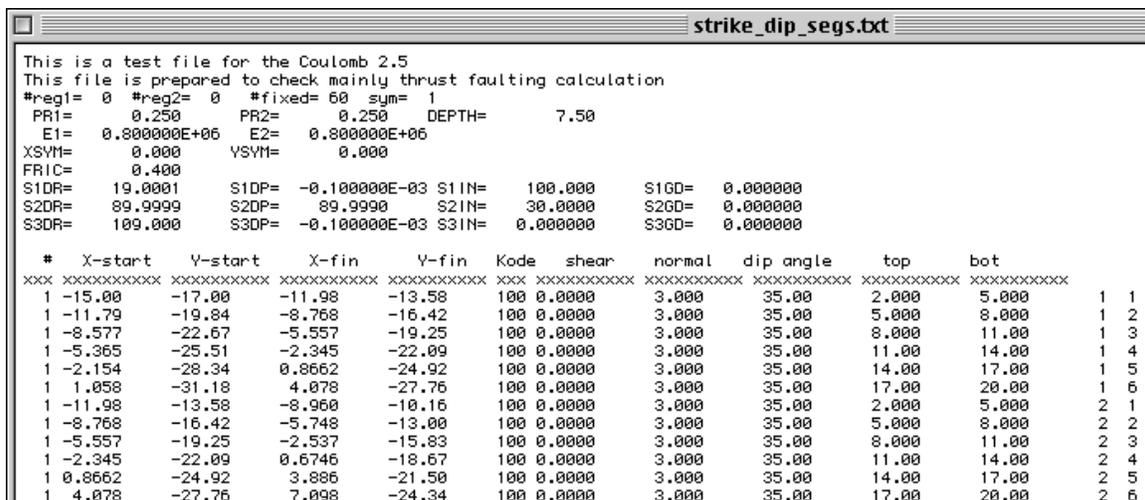
```

2. Launch Splitter. Choose option 3 for along-strike and down-dip patches (there are two simpler options). Choose 6 down-dip patches. Then select 'Splitter-3.inp' in the dialogue box and hit return. Splitter will quit and two new files will

appear in the folder where Splitter is located (not necessarily in the folder where 'Splitter-3.inp' resides), 'strike_segs.txt' and 'strike_dip_segs.txt'.



- Open EditII or any text editor, and from within the editor, open 'strike_dip_segs.txt'. Now the 30 x 18 km fault is split into sixty 3 x 3 km patches. Notice that there is a logical numbering of the patches in the comment lines at the far right of each patch. You could now assign slip values based on a variable-slip model, rename the file, and use it as an input file.



- Launch Coulomb, choose *Data* > **Old** and select 'strike_dip_segs.txt' (or whatever you've renamed it) and hit *Calculation* > **Go**. You will see the 60 evenly-

distributed rectangles in the grid. In the screen shot below, we changed the depth to 2 km so the black depth-slice line would not obscure the patches.

