U.S. DEPARTMENT
OF THE INTERIOR
U.S. GEOLOGICAL SURVEY

3D-DEF: A USER'S MANUAL

(A three-dimensional, boundary element modeling program)

by

Joan Gomberg[1]

and

Michael Ellis[2]

Open-File Report 93-547

[1]USGS, MS 966
Box 25046
Denver Federal Center
Denver, CO 80225
gomberg@gldvxa.cr.usgs.gov

[2]Center for Earthquake Research and Information
Memphis State University
Memphis, TN 38152
ellis@ceri.memst.edu

1993

Introduction

If you have figured out elastic deformation in two-dimensions and are game for a new challenge, our three-dimensional boundary element program is for you. Note that this manual assumes that you understand the basics of elastic deformation theory and the ideas that underlie boundary element algorithms.

We have attempted to not limit the code to a run with a specific operating or graphics system (only a Fortran compiler is required). The codes are written in generic Fortran. They were developed using Macintosh Language Systems Fortran but should compile with almost any Fortran compiler. Note that no structures are used. To date we have compiled and run 3D-DEF on an Apple Computer Corp. Macintosh SE and Quadra 800, and several SUN and Sparc workstations.

This manual contains the following information:

The code does contain some internal comments for those brave enough to look into it. Please note that, although many basic tests have been performed, there are modes of operation that have not been thoroughly tested.

I. Program structure

Four files contain source code (Fortran text):

3D_MAIN.F OKADA_SUB.F   XYZ_OUTPUT.F   VECTOR_OUTPUT.F

In order to compile the source code (convert it to instructions understood by the computer) two include files are required. These are:

SIZES.INC   UNITS.INC

(Include files contain Fortran statements that are inserted into the source code automatically upon compilation. They simplify making changes to the program; for example, to change the maximum number of allowable elements (see Section II.B) one would only modify a single parameter specified in the file SIZES.INC. All appropriate array dimensions within

the four source code files would then be changed appropriately without having to edit them.)

SIZES.INC contains program dimensions that depend on the number of elements. The largest array (computer memory accessed as an indexed matrix) has dimensions of (MAX3_ELEM)*(MAX3_ELEM+1) where MAX3_ELEM=3*MAX_ELEM and MAX_ELEM is the maximum number of elements allowed. You should make this number as large as you will need for your specific problem by editing the file SIZES.INC before compiling the program. If the program does not run and the computer returns an 'insufficient memory' error, you will have to reduce this number.

UNITS.INC contains all file unit numbers ('tags' used by the computer to identify a file when reading or writing to it). You can change these as needed and re-compile the source codes to implement the changes.

The basic parameter specification (input) is done in the code contained in the file 3D_MAIN.F. All major calculations are done in routines contained in files 3D_MAIN.F and OKADA_SUB.F. Routines that control the output grid and information written to column formatted files are contained in the file XYZ_OUTPUT.F. Each row of these output files contains the X,Y,Z coordinates of each grid point and specified calculated quantities. These column formatted output files are read by routines in the file OUTPUT_VECTOR.F and the information they contain is reformatted in a vector format (as required by many plotting programs such as MATLAB, IDL, etc.). Vector reformatting is optional.

The program does not generate any graphical output - displaying the output is left up to you.

II. Definitions and conventions

A. Green's functions.
　　　　3D-DEF uses Green's functions calculated using subroutines provided by Y. Okada. The Green's functions relate the deformation field (displacements and displacement gradients) to a rectangular dislocation in a homogeneous half-space (a semi-infinite medium bounded by a planar free-surface). Descriptions of the dislocation theory underlying the routines may be found in Okada, Y., Internal deformation due to shear and tensile faults in a half-space, *Bull. Seismo. Soc. Amer.*, v. 82, 1018-1040, 1992.

B. Selected definitions.
　　　　The meaning of an element and the coordinate systems used in the computational algorithms are defined in this section. Three coordinate systems are used. A *local* coordinate system is defined for each element because the Green's functions (see Section II.A) are most efficiently calculated in the local coordinate system of the element. An *in-plane* coordinate system is also defined for each element because the boundary conditions are specified either on or orthogonal to the element surface. A *global* coordinate system is required so that the effect of the deformation field associated with one element on another element can be calculated. Because the parameters describing the deformation are calculated in local coordinates, they must be transformed into a global system that is common to all elements.

Plane (Element):　A planar elastic dislocation surface. Physically, an element may be thought of as a fault plane, a dike, or a sill having constant strike and dip.

Sub-element: Each element can be subdivided into smaller sub-elements (sub-faults or fault segments). For each element, all the sub-elements will have equal dimensions. The dimension of each sub-element in the strike direction will be equal to the (element's total length along strike)/(number of sub-elements along strike) and in the dip direction to the (element's total width along dip)/(number of sub-elements along the dip direction). The width, length and numbers of element subdivisions in each direction are specified in the input file. The terms element, sub-element, fault, plane, and dislocation will be used interchangeably throughout this manual.

Global coordinates: The coordinate system common to all elements is: +X is horizontal (parallel to the free surface) and points east, +Y is horizontal and points north, and +Z is vertical pointing up (away from the half-space) with respect to the free surface. See Figure 1.

Reference point, X_o, Y_o, Z_o: For each element, the reference point determines the position of all planes relative to one another in the global coordinate system. The intersection of the edge of an element that is parallel to the free surface and closest to the free surface (shallowest edge) defines a line. The reference point defined by the user is the end point of this line such that, looking away from the reference point, the hanging wall on the right.   The program actually uses a different reference point internally. The internal reference point is defined like the user defined point except that it lies along the deepest edge of the element. The internal reference point is calculated from the user specified reference point and the element's geometry. The user defined point is used because it is easier to visualize. See Figure 1.

Local coordinates (X_L, Y_L, Z_L): This is the coordinate system defined for each plane. It is required for the Green's functions' calculations. The X_L and Y_L axes define a horizontal plane that is parallel to the free surface. The +X_L axis is along strike, the +Y_L axis is perpendicular to the strike, and the +Z_L axis is vertical. The system is right handed such that looking in the strike direction (+X_L) the hanging-wall is on the right and the +Y_L axis points into the footwall. See Figure 1.

Planar (or In-plane) coordinates (X_P, Y_P, Z_P): This coordinate system is defined for each plane such that X_P is along the strike direction (=X_L), Y_P is along the dip direction and is positive going up-dip from the internal (program's) reference point X_o,Y_o,Z_o. +Z_P is normal to the plane with positive defined so that the system is right handed. See Figure 1.
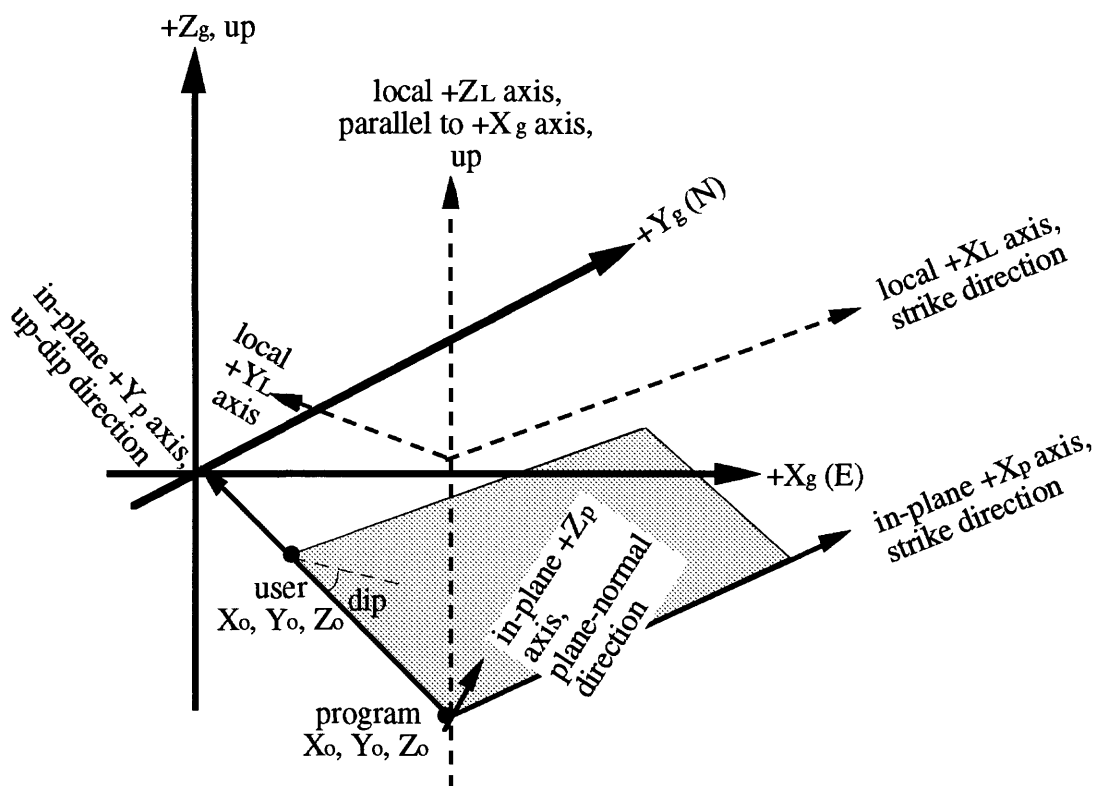
Figure 1.

C. Boundary condition codes.

The boundary condition codes define the constraints on each sub-element, one boundary condition for each directional (strike, dip, normal) component of stress or displacement. Boundary conditions may include the magnitude of the directional components of stress, of <u>absolute</u> displacements, or of <u>relative</u> displacements on a sub-element. For a single sub-element, mixed (stress and displacement) boundary conditions may be specified.

The stresses are continuous everywhere and displacements are discontinuous across elements. Understanding the difference between absolute and relative displacements is very important when specifying boundary conditions and interpreting the program output. Once you understand the difference, you are well on your way to mastery of boundary element modeling! The program calculates (solves for) <u>relative</u> displacement on all of the sub-elements in the model such that the strain energy within the half-space is a global minimum. Relative displacement is simply the net dislocation or slip of one side of the element with respect to the other side (for example, the fault offset observed after an earthquake). The relative displacements solved for by the program (that minimize the strain energy) are determined by the user specified boundary conditions on the sub-elements. The relative displacement on a particular sub-element or group of elements may also be specified by the user, rather than being treated as an unknown. <u>Absolute</u> displacement refers to the motion of the footwall side of the element with respect to the global coordinates (HOWEVER: the displacements are specified in the planar coordinates of the sub-element; that is: strike direction, dip direction, fault-normal direction). Mathematically, the relative displacement is the difference between the absolute displacements of either side of the element. For example, the normal component of relative displacement, D_n, is

$$D_n = u_n^- - u_n^+$$

where u_n^- is the magnitude of the motion of the footwall (absolute displacement) in the direction normal to the element and is u_n^+ is the absolute displacement of the hanging wall.

The user specifies one boundary condition code (a number) for each sub-element. Each code actually constrains the magnitude of three parameters (stresses or displacements) at the center of the sub-element, one in the strike direction, one in the dip direction, and one in the direction normal to the sub-element surface. The boundary condition codes allowed are:

Code#	Direction in the Plane of the Sub-element		
	strike (shear)	dip (shear)	normal (tensile)
1	absolute displacement	absolute displacement	absolute displacement
2	stress	stress	stress
3	stress	absolute displacement	stress
4	absolute displacement	stress	stress
5	stress	stress	absolute displacement
10	relative displacement	relative displacement	relative displacement
11	angle (degrees)	stress (at specified angle)	relative displacement
12	stress	stress	relative displacement
13	stress	relative displacement	relative displacement
14	relative displacement	stress	relative displacement

Note that Code #11 and Code #12 specify the same stress conditions; for Code #11 the user specifies the direction of shear as an angle measured from the strike direction, which is then resolved into shear stress conditions in the strike and dip directions.

D. Background deformation specification.

It is possible to include a uniform deformation field, referred to as a background deformation field. (For example, suppose one wanted to investigate how slip was partitioned among a network of faults driven by a uniform simple-shear strain field.) The field is specified as a stress, strain, or displacement gradient tensor. Internally these tensors are converted to a stress tensor and a rigid body rotation (if a displacement gradient tensor is specified). The background stresses are subtracted from the stress boundary conditions before solving for the relative displacements on the sub-elements. Because the background deformation is added to that associated with the sub-element displacements, the resultant stresses on the sub-elements are thus guaranteed to equal the boundary condition stress value. Use of a background stress or strain tensor will have no affect on the sub-element relative displacements if you use Codes 1 or 10 because the boundary conditions do not involve stress. If a displacement gradient tensor is specified, then the only affect will be to modify the output displacements by any rigid body rotations associated with the background displacement gradient field.

Section III.B explains how to specify the input to include (or not include) a uniform background deformation field.

E. Displacement direction, sign, and fault geometry conventions.

The orientation of each fault (element) is specified as follows:

1. When looking in the strike direction, the hanging wall is on the right.
2. The strike angle is defined in decimal degrees, increasing clockwise from north.
3. Dip is defined as the angle between the horizontal (parallel to the free surface) and the fault plane. It is measured between two lines that are orthogonal to the

strike direction, one line is parallel to the horizontal and the other to the fault surface. Dip angles are specified in degrees and range from 0° to 90°.

Fault (element) relative displacement sign conventions are:

Dislocation Component	Sign	Faulting Mode
Strike	>0 (positive)	left-lateral
	<0 (negative)	right-lateral
Dip	>0 (positive)	thrust
	<0 (negative)	normal
Tensile (normal to plane)	>0 (positive)	opening
	<0 (negative)	closing

F. Units.

Specify input element (fault) dimensions in kilometers and displacements in centimeters. Output displacements have units of centimeters, strains are dimensionless. Stresses (input and output) have the same units as the Young's modulus specified in the input file.

G. Tensor notation.

Stress tensors are written as a vector internally as:

Vector Index	Stress Component
1	Sxx
2	Sxy
3	Sxz
4	Syy
5	Syz
6	Szz

x,y,z refers to whatever coordinate system is being used at the time the values are examined (for example, directly out of the subroutines that calculate the Green's functions x,y,z refers to the local XL,YL,ZL coordinate axes). Strain tensors are also written in the same manner.

III. Input file description.

A. Introduction to the input file.

The program requires one input file and the amount of interactive input is variable depending on what options the user selects. All input parameters may be written to a file so that the only interactive step is to respond to the program prompt for the input file name. Alternatively, the user can interactively enter the type and characteristics of the information written to output files.

All lines in the input file containing numbers are free formatted so the format is not important but items cannot be omitted. Text lines that begin with an asterisk are provided only for the user's benefit and are ignored by the program. Note that while you can write anything in these lines, the number of asterisk lines and blank lines must not change.

The first part of the input file contains parameters that describe the material properties of the medium, the element geometry, and the boundary conditions. The second part of the file controls which properties of the deformation field are calculated and output and where within the half space (and in what coordinate system) these properties are calculated. This second part of the file may be omitted and the needed input entered interactively.

B. First part of the input file (material properties, element geometry, boundary conditions).

An example and explanation of the first part of an input file is now provided. The text between the dashed lines below illustrates a sample file followed by explanation of each of the lines.

```
-------------------------------------------------------------------------------------
* POISSONS RATIO   YOUNGS MOD #PLANES COEFF INTERNAL FRICTION      BACKGROUND DEFORMATION
 .25               7.E10       2       0.6                         displacement gradient
* Background deformation tensor
0 0 0 0 -.00001 0 0 0 0

* for each element or plane specify the
* REFERENCE:XO    YO    ZO     DIMEN.:STK  DIP  #SUB-ELEMS:STK DIP  GEOM:STK    DIP
            50.   59.   .0           136. 15.             3   2         48.     90.
            122.  180.  .0           54.  15.             1   2         37.     90.

* for each plane (at each fixed distance along strike, going up-dip):
* PLN,SUB-ELEM          code      BC-shear(STK)    BC2-shear(DIP)  BC3-normal
* 1    (STK,DIP)
       1  1             10        -270.            0.0             0.0
       1  2             10        -270.            0.0             0.0
       2  1             10        -270.            0.0             0.0
       2  2             10        -270.            0.0             0.0
       3  1             10        -270.            0.0             0.0
       3  2             10        -270.            0.0             0.0
* 2    (STK,DIP)
       1  1             10        -270.            0.0             0.0
       1  2             10        -270.            0.0             0.0
-------------------------------------------------------------------------------------
```

The first line of the input file must begin with an '*' and can contain any comments thereafter; here the comments describe what the line following contains. The second line contains 5 fields; the first four are numbers, and the last is a character string. The numbers are (from left to right) the Poisson's ratio (dimensionless), the Youngs modulus (units of your choice), the number of elements or planes in the model, and the coefficient of internal friction. The coefficient of internal friction is used to determine potential failure planes, assuming a Coulomb failure criterion; if you don't select the 'failure plane' output (see Section IV) this coefficient has no influence on the results. The final item in this line is a character string flag set to specify the type of background deformation field that will be specified; the character string should be 'none' for no background deformation field, 'stress' for a background stress field, 'strain' for a background strain field, or 'displacement gradient' for a background displacement gradient field. The next line is blank if the above flag is 'none' or contains the elements of the background deformation tensor. The order of stress or strain tensor components is xx, xy, xz, yy, yz, zz where x,y,z refer to the global coordinate axes. For a flag set to 'displacement' the order of displacement gradient tensor components is dUx/dx, dUx/dy, dUx/dz, dUy/dx, dUy/dy, dUy/dz, dUz/dx, dUz/dy, dUz/dz.

A blank line and two comment lines are then followed by the next group of numbers that describe each plane, one line per plane. The first three numbers are the user reference point of the plane in global coordinates (see Section 2B). The fourth and fifth numbers describe the dimensions of the plane; the length (in the strike direction) and width (dip direction). The sixth and seventh numbers describe how the plane is sub-divided into sub-elements in the strike and dip directions. Sub-elements of a single plane all have the same dimensions; their individual length equals the plane length divided by the sixth number (under #SUB-ELEMS:STK in the sample file) and their width equals the plane width divided by the seventh number (under DIP). The last two numbers are the strike and dip of

the plane in decimal degrees (see Section 2D). In this sample file two planes are specified, the first is divided into six sub-elements (three along strike and two down dip) and the second into two sub-elements.

The next group of numbers (following a blank and three comment lines) specify the boundary conditions for each sub-element, one row per sub-element. Figure 2 below illustrates how the sub-elements are identified. The first two integers are sub-element indices but are not actually used by the program; they simply help the user keep track of the sub-elements. The first sub-element must be the one at the internal reference point or equivalently, the origin of the in-plane coordinate system (looking at the plane from the hanging wall the first sub-element is in the lower (deepest) left corner). The element dip index is the second number of the pair (right index) and the values increase up-dip. The first number of the pair is the strike index and these increase with distance from the origin. For each strike index, the dip index is incremented. Diagramatically the indices of the sub-elements of the first plane in the sample file are:
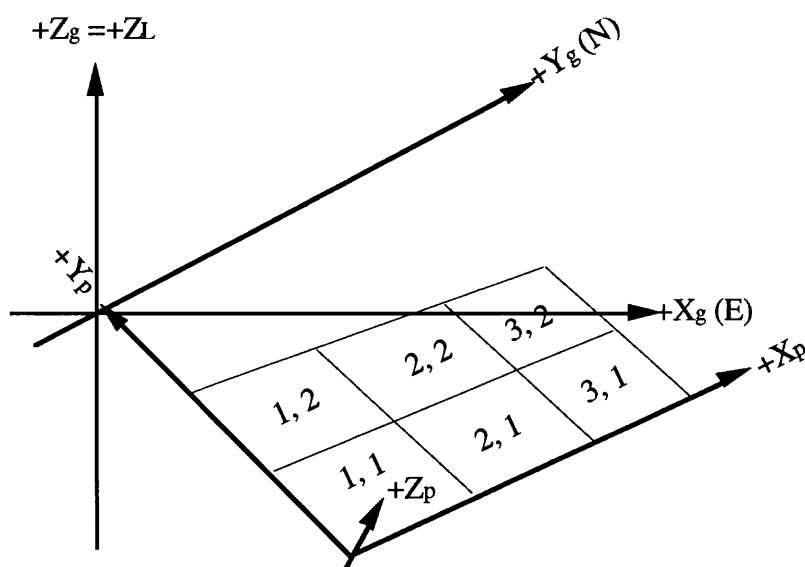

Figure 2

C. The second part of the input file (output control).

The second part of the input file may be viewed as having 5 "blocks" of input, the total number of lines in each block is determined by input in previous blocks. As a result, this part of the input file may change dramatically from model to model. The first (non-comment) line of the first block (#1) determines whether the output parameters will be specified interactively or non-interactively. If interactively, nothing else in the file matters-- the user is prompted from standard output for the parameters. If non-interactively is selected, this block contains two more lines. (NOTE: as will be explained in more detail below, greater control over the number and content of created files may be achieved via the interactive specification of output control.)

The line following the interactive/non-interactive choice determines the gridding of the solution; that is, whether the desired output parameters describing the deformation field are calculated at:

    i) points on a plane (of any orientation)

ii) points within a volume (with vertical and horizontal sides), or

iii) a list of user-specified points.

If output of parameters describing the deformation on a plane is desired, the line must start with the character string "plane". For a gridded volume, the line must start with "vol". For output at a set of user-specified points, the line starts with "user". In almost all cases, only the first letter of the line is read by the program.

The next line determines the type of output, "XYZ" or "vector". For XYZ (column) output, the output files of deformation parameters contain one line per point, with the X, Y, and Z coordinates of the point given one on each line. For vector output, the values within the gridded region (which may be either a plane or a volume) are output into a file as an array of numbers, without coordinates. The coordinates of each point are determined by their position within the array (Seem Obscure? Patience, Grasshopper! Enlightenment may be obtained by reading more about this issue below in the following examples!). NOTE: if vector output is requested, both column output files *and* vector output files are created because the column output files are always created, and the program generates the vector files from the column output files.

The next block (#2) determines the coordinate system for calculating various parameters describing the deformation field. The two choices of coordinate systems are "global" or "in-plane"; the latter is specified with anything that doesn't start with a 'g' or 'G' (in-plane, planar, etc.). This option only applies to output on a gridded plane and affects only output stresses, strains, displacements, and gradients of displacements. Global output coordinates means that stresses, strains, displacements, and gradients of displacement are coordinate stresses, strains, etc. in the same reference frame in which the faults were specified (that is, Sxx, Sxy, Sxz, etc.). In-plane (or planar) coordinates refer to the gridded plane, that is: the output stresses, strains will be referred to the strike, dip, and normal direction of the gridded plane (that is, Sss, Ssd, Ssn, etc).

The next block (#3) determines which parameters describing the deformation field to calculate. The choices are: element displacements (that is, one side relative to the other of each element), and at each grid or user specified point, absolute displacements, gradients of displacements, strains, stresses, strain invariants, strain orientations, rigid body rotations, and failure plane orientations. These are described in detail in Section IV.

The next block (#4) is a line containing the suffix that will be appended to the output files.

The final block (#5) specifies the points at which output is calculated. If a user array was specified in block #1, then this portion contains the number of points, and their coordinates (see example below). If a gridded plane was specified in block #1, then this section contains the orientation and size of the plane in one line, and the number of points on the plane at which to calculate the output on a second line (see example below). If a volume is specified, the first line of this block contains the coordinates of two of the corners of the rectangular prism of the volume, and the second line contains the number of X, Y, and Z grid points within the volume.

In the examples below, user input that effects the outcome of the model (that is, which output files are created, what they are called, where the values are calculated, etc.) is shown in *Italics*. The minimum input required to obtain the desired result is underlined. Everything else in the second part of the file is ignored by the program, but is included to make the input file more intelligible to the user.

(NOTE: lines that begin with '*' must be in the input file because they are used by the program to separate blocks of differing types of input. However, they may contain any text following the '*'.)

EXAMPLE #1

For a non-interactive run with a gridded volume, the second part of the file might look like this (ignore dashed lines):

```
-----------------------------------------------------------
-----------------Block #1 ----------------------------
* Run everything else non-interactively
Not Interactive  (alternative is 'Interactive' grid specification)
Volume grid (alternative is 'Plane grid')
XYZ Output  (alternative is Vector output - results in both output types)
-----------------Block #2 ----------------------------
* Coordinate system for output (global only for volume grids)
global
-----------------Block #3 ----------------------------
* Output information (each line results in a separate output file)
stress (in global coordinates)
strain (in global coordinates)
gradients of displacement (in global coordinates)
rigid body rotations (about the global X(E),Y(N),Z(up) axes)
orientation of principal strain axes (w.r.t the global X(E),Y(N),Z(up) coordinates)
displacements (in global coordinates)
invariants of the strain field (independent of coordinate system)
failure planes (in global coordinates)
element relative displacements (strike,dip,normal directions of each element's plane)
-----------------Block #4 ----------------------------
* Output file suffix
out1
-----------------Block #5 ----------------------------
* XYZ Coordinates of the upper NW and lower SE corners of the volume:
0,0,0, 90,-90,-10
* Grid spacing increments in X (E), Y (N),Z (up) directions:
9,9,2
-----------------------------------------------------------
```

Commentary on example #1:   For a gridded volume, all output parameters, except element relative displacements, will be in global coordinates (no matter what you write in the appropriate line). The list of files containing various characteristics of the calculated deformation field can be in any order. The word on each line sets an appropriate flag so that an output file containing the parameters described by the word is generated. The program only interprets the first letter (the first 4 letters for 'stress' and 'strain') so there is some flexibility in what words are used. For example, you must write '*gradients of displacement*' (if fact, simply 'g' will do) to generate an output file containing the components of the displacement gradient tensor; use of 'displacement gradients' would be incorrect and (because the key word starts with a 'd' instead of a 'g') result in an output file containing displacements.

EXAMPLE #2

For a non-interactive run with a gridded plane the second part of the file might look like this (ignore dashed lines):

```
-----------------------------------------------------------
-----------------Block #1 ----------------------------
* Run everything else non-interactively
Not Interactive  (alternative is 'Interactive' grid specification)
Plane grid (alternative is 'Volume grid')
Vector output (alternative is 'XYZ'--vector output yields both types of output files)
```

```
-----------------Block #2 -----------------------------
* Coordinate system for output (global only for volume grids)
planar (in-plane coordinates of gridded plane--must not begin with "g")
-----------------Block #3 -----------------------------
* Output information (each line results in a separate output file)
stress (coordinate system of the gridded plane;X=strike,Y=dip,Z=normal directions)
strain (coordinate system of the gridded plane;X=strike,Y=dip,Z=normal directions)
gradients of displacement (coord. system as for stress and strain)
rigid body rotations (about the global X(E),Y(N),Z(up) axes)
failure planes (about the global X(E),Y(N),Z(up) axes)
orientation of principal strain axes (w.r.t the global X(E),Y(N),Z(up) coordinates)
displacements (coord. system of the gridded plane;X=strike,Y=dip,Z=normal directions)
invariants of the strain field (independent of coordinate system)
element relative displacements (strike,dip,normal directions of each element's plane)
-----------------Block #4 -----------------------------
* Output file suffix
out2
-----------------Block #5 -----------------------------
* Xo,Yo,Zo (Ref. corner),strike (cw wrt N),dip (0-90),length (strike), width (dip):
0,0,0,90,90,10,5
* Number of grid points in the strike and dip directions:
5,5
--------------------------------------------------------------
```

Commentary on example #2:   For a gridded plane, if the coordinate system requested is anything but '*global*' output parameters that are still referenced to the global coordinate system include:

> *rigid body rotations*
> *orientation of principal strain axes*
> *failure plane descriptions*

output parameters that are in the coordinate system of the gridded plane (X=strike, Y=dip, Z=fault-normal directions) include:

> *stress tensor elements*
> *strain tensor elements*
> *gradients of displacement (tensor elements)*
> *displacements*

Element relative displacements correspond to the strike, dip, and fault-normal directions of *each element's plane*. The list of files containing various characteristics of the calculated deformation field can be in any order but the first character must be correct (see gridded volume description above).

EXAMPLE #3
For a non-interactive run on a user-specified array of points:

```
--------------------------------------------------------------
-----------------Block #1 -----------------------------
* Run everything else non-interactively
Not Interactive (alternative is 'Interactive' grid specification)
User array (alternative is 'Plane grid' or 'Volume grid')
XYZ Output (alternative is Vector output - results in both output types)
-----------------Block #2 -----------------------------
* Coordinate system for output (global only for volume grids)
global
-----------------Block #3 -----------------------------
* Output information (each line results in a separate output file)
stress (in global coordinates)
strain (in global coordinates)
gradients of displacement (in global coordinates)
rigid body rotations (about the global X(E),Y(N),Z(up) axes)
```

```
orientation of principal strain axes (w.r.t the global X(E),Y(N),Z(up) coordinates)
displacements (in global coordinates)
invariants of the strain field (independent of coordinate system)
failure planes (in global coordinates)
element relative displacements (strike,dip,normal directions of each element's plane)
------------------Block #4 ----------------------------
* Output file suffix
out1
------------------Block #5 ----------------------------
* Number of points followed by index,x,y,z
4
1 10 5 -5
2 12 6 -5
3 14 5 -5
4 16 6 -5
------------------------------------------------------------
```

EXAMPLE #4

For an interactive run the second part of the file might look like this (ignore dashed lines):

```
------------------------------------------------------------
*Run everything else interactively
Interactive grid specification
------------------------------------------------------------
```

Commentary on example #4: The user will be asked by the program to answer a sequence of questions that provide the program with the required input parameters.

IV. Output file descriptions

A. Column output format.

Nine output files are created with column format. Each one has a similar format. The suffix ??? in the following discussion is replaced with the suffix that you specified in the input file (or answered interactively). Each line contains an observation point x,y,z (in global coordinates) and the parameters describing the various characteristics of the deformation field at that point. Those characteristics are:

File	Contents
stressten.???	Six independent elements of the stress tensor, in the order: Sxx, Sxy, Sxz, Syy, Syz, Szz. If output on a plane is requested and global output is not requested, then these correspond to the in-plane coordinates. (For example, Sss, Ssd, Ssn, Sdd, Sdn, Snn where 's' refers to the strike direction, 'd' to the dip direction, and 'n' to the normal direction. Thus, Szz is really Snn and corresponds to the stress that is normal (tensile) to the fault plane, Sxz corresponds to Ssn and is the shear stress (sometimes called a traction) in the strike direction.) Otherwise the tensor components are referred to the global coordinate system.
strainten.???	Six independent elements of the strain tensor, Exx, Exy, Exz, Eyy, Eyz, Ezz, and the volumetric strain (Exx+Eyy+Ezz, independent of the coordinate system). If output on a plane in in-plane coordinates is requested, then these values may be output in the in-plane coordinates (as described in stressten.???).

displacements.??? Displacements, Ux, Uy, Uz, in the x,y,z directions, respectively. If output on a plane in in-plane coordinates is requested, then these values may be output in in-plane coordinates; Ux is in the strike direction, Uy in the up-dip direction, Uz normal to the plane.

dgten.??? Displacement gradient tensor, dUx/dx, dUx/dy, dUx/dz, dUy/dx, dUy/dy, dUy/dz, dUz/dx, dUz/dy, dUz/dz. As for the above, these may correspond to the in-plane coordinates if the user has selected the plane and in-plane (not global) options.

strainorient.??? Principal strains, and their plunge and trend with respect to the global axes. The values corresponding to the maximum principal strain are output first, and the minimum principal strain values are output last.

invariants.??? Strain & stress invariants including the volume change, failure stresses, octahedral shear stress, and strain energy density.

rbr.??? Rotations about the global x, y, and z axes.

failure.??? For each of the two optimal failure planes, the strike and dip of the plane and the direction cosines of the slip vector are given. The strike is clockwise from north, the dip is from the horizontal, and the direction cosines are with respect to X(E), Y(N), Z(up).

elements.??? Each line of the output file will contain the coordinates of the center of the element in global coordinates or in in-plane coordinates, and the relative displacements in the strike, dip, and tensile (normal) directions.

Programmer notes:
The first eight files are opened in a subroutine named 'open_opts', closed in a subroutine named 'close_opts', and written to in the subroutine named 'XYZ_Results'. The file "elements.???", is opened and closed in subroutine 'final_write' and written to in 'XYZ_elements'.

The first eight output files may be re-formatted in a vector format (that is, as desired by plotting programs such as MATLAB, IDL, etc.); this re-formatting is of course optional. Remember that if you change the format of these column formatted files, you must also change the routines in OUTPUT_VECTOR.f since they must read these files. The element displacements may also be written out in vector format but this is independent of the column formatted file.

B. Vector output format
If the user requests vector output, then the deformation output parameters will be re-written in vector format (in new files). Each parameter will be written to its own file. If the program is being run non-interactively, then all parameters of a particular selected type will be written out. For example, if the user has specified 'gradients of displacement' in the input file, nine output files will be generated - one for each element of the displacement gradient tensor. If running in an interactive mode, the user can select which parameters to be written out (that is, only the dUx/dz, dUy/dz, dUz/dz components of the displacement

gradient tensor--or whatever subset is desired). See the description of the column formatted files to find out what coordinate systems may be used for the output parameters. Note that for in-plane coordinates, x is the strike direction, y is the dip direction, and z is normal to the plane. For a gridded plane, each line of the output file corresponds to a row (constant depth) on the plane; the values from left to right on each line correspond to the increasing strike direction. The first row corresponds to the shallowest depth and last row to the greatest depth on the plane or in the volume. For a gridded volume, consider the volume to be a stack of horizontal planes, the first plane is the shallowest (closest to the surface). The output is the same as for the gridded plane for each of the horizontal planes. The first value of each plane corresponds to the northwestern corner.

The following describes the output file possibilities for each output type requested.

Output type	Files	Contents
element displacements	elemds#.???	strike component of displacements
	elemdd#.???	dip component of displacements
	elemdn#.???	tensile component of displacements
	(#=index number, 1... no. of planes)	
stress	Sxx.???	xx component of the stress tensor
	Sxy.???	xy component of the stress tensor
	Sxz.???	xz component of the stress tensor
	Syy.???	yy component of the stress tensor
	Syz.???	yz component of the stress tensor
	Szz.???	zz component of the stress tensor
strain	Exx.???	xx component of the strain tensor
	Exy.???	xy component of the strain tensor
	Exz.???	xz component of the strain tensor
	Eyy.???	yy component of the strain tensor
	Eyz.???	yz component of the strain tensor
	Ezz.???	zz component of the strain tensor
gradients in displacements	dUxdx.???	gradient of x-displacement in x-direction
	dUxdy.???	gradient of x-displacement in y-direction
	dUxdz.???	gradient of x-displacement in z-direction
	dUydx.???	gradient of y-displacement in x-direction
	dUydy.???	gradient of y-displacement in y-direction
	dUydz.???	gradient of y-displacement in z-direction
	dUzdx.???	gradient of z-displacement in x-direction
	dUzdy.???	gradient of z-displacement in y-direction
	dUzdz.???	gradient of z-displacement in z-direction
rigid body rotations	rbr_deg_x.???	rotation about the X (E) axis
	rbr_deg_y.???	rotation about the Y (N) axis
	rbr_deg_z.???	rotation about the Z (vertical) axis
orientation of principal axes	prince_max.???	magnitude of the max. principal strain
	plunge_max.???	plunge with respect to horizontal of the max. principal strain

	trend_max.???	trend (cw with respect to N) of the max. principal strain
	prince_int.???	magnitude of the intermediate principal strain
	plunge_int.???	plunge with respect to horizontal of the intermed. principal strain
	trend_int.???	trend (cw with respect to N) of intermediate principal strain
	prince_min.???	magnitude of the min. principal strain
	plunge_min.???	plunge with respect to horizontal of the min. principal strain
	trend_min.???	trend (cw with respect to N) of the min. principal strain
displacements	disp1.???	displacements in the x-direction.
	disp2.???	displacements in the y-direction.
	disp3.???	displacements in the z-direction.
invariants	delv.???	volumetric strain
	aftershocks.???	failure stress
	oss.???	octahedral shear stress
	work.???	strain energy density
failure planes	strike1.???	strike of first failure plane
	dip1.???	dip of first failure plane
	ce1.???	slip vector dir. cos. with respect to X(E)
	cn1.???	slip vector dir. cos. with respect to Y(N)
	cz1.???	slip vector dir. cos. with respect to Z(up)
	strike2.???	strike of second failure plane
	dip2.???	dip of second failure plane
	ce2.???	slip vector dir. cos. with respect to X(E)
	cn2.???	slip vector dir. cos. with respect to Y(N)
	cz2.???	slip vector dir. cos. with respect to Z(up)

For the element displacements output on each plane, three file types are created for each plane -

elemds#.???	strike component of displacements
elemdd#.???	dip component of displacements
elemdn#.???	tensile component of displacements

where # is the number of the plane as listed in the input file. Each contains a component of the displacement such that each line of the file corresponds to the sub-elements along strike at a fixed depth (dip index). The first line for each plane is the top (closest to the surface) of the plane and lines increment with increasing depth. A set of lines is written for each plane, and each plane is separated by a blank row.

V. Acknowledgements