```
**********************************************************
*  FORTRAN 77 Program for 3D Finite Element Analysis of  *
*  the Visco-elastic Earth (forward/inverse modelling)   *
*  Developed by S. Jason Zhao, 1999-2002.                *
**********************************************************
```

Minor restrictions:

1. The program was developed for the purpose of scientific research only, and should not be used for any commercial purposes;
2. The program should not be distributed to others without permission of the author;
3. The program can be freely modified but it is preferable to quote the author of the original program if possible;
4. Please report any bugs and suggestions to:

Email: zhao@jamstec.go.jp
subduction99@netscape.net

---

0. A list of files in this package

-readme.txt this file
-viskx.for source file
-viskx.dat data file
-start1.dat data file
-viskx1.dat data file
-viskx2.dat data file
-regox.bat batch file for running under DOS

1. Introduction

VISKX is a FORTRAN program for analyzing visco-elastic response of the 3D earth (or any other objects) to internal and external forces.

There is only one source file for the program: viskx.for.

2. Main features

(1) The program has been relatively well documented, and the details about the structures and functions for the program (and its subroutines) can be found

in the source file (viskx.FOR).

(2) Three types of elements (8-node-brick, 6-node-prism, and infinite elements) were used in the program, which enables a user (1) to
use mixed elements to investigate a problem and (2) to approximate
any geometric shapes with high accuracy.

(3) Geological/seismic faults can be simulated in the program by input of the fault points.

(4) Stress, strain rates and displacements can be evaluated for any specified points. This is extremely useful when one wants to compare the results with measurements on some observational points (such as geodetic points).
These observational points are often inside an element.

(5) The types of loads in the program include:
- -gravity load
- -hydro-static pressure
- -uniform pressure to some element faces (8-node-bricks only)
- -concentrated forces normal to each element face (8-node-bricks only)
- -nodal forces
- -traction on specified faces (<=100 faces, changeable)
- -fault/crack points

(6) types of material
- -different materials with different viscosity constants (type<=120, changeable).

3. Source, input and output files

Source file: viskx.FOR.

Input and output files:

viskx.dat --- a data-input file for the main program;
(must be prepared by users).
start1.dat -- a data-input file giving information about the inversion.
(must be prepared by users).
viskx1.dat --- a data-input file for the 8-node-brick element;
(must be prepared by users).
viskx2.dat --- a data-input file for the 6-node-prism elements;
(must be prepared by users).

Users have to supply their own mesh/grid.

Output files after running the program

-viskx.out --- an output file for the program, created automatically.
-stress01.dat -- output, stress rates
-viskx1.out -- output, deformation rates

The details for preparing data have been included in each data file.

4. Compile/link the program under DOS/Windows

Need a FORTRAN 77/90 compiler to compile/link the source file viskx.FOR, and produce an executable file: viskx.exe.

5. Run the program

(1) Five data files (viskx.dat, start1.dat,viskx1.dat, viskx2.dat) are required to run the program.

Note: the output file viskx.out and those temporary files will be automatically created after running the program. Therefore, before the next run, the output file and the temporary files must be deleted.

For those working under DOS, the batch file regox.bat in the package has included such a clean function. Under DOS to run the program, one can type:

regox.bat

(2) For those working with UNIX/LINUX, a short-shell program similar to regox.bat in function may be written (e.g., change del to rm).

The program has been tested under both DOS and UNIX systems.

(3) Details for preparing data for each type of element have been given in end of the data files.

6. Current settings in the source file
==========

(1). The parameter sentence

The maximum number of nodes (NSN)
The maximum number of material type (NMP)
The maximum dimension (NSK) of the array SK

The parameters in the sentence shown
below should be changed according to a practical application:

```
    PARAMETER(NSN=900,NMP=10,NSK=2203000)
```

where NSK~=3*NSN*(3*NSN+1)/2, -AS INITIAL NUMBER

(2) Dimensions of several arrays
The maximum number of traction loads
The maximum number of given (computational) points
The maximum number of fault points

Change the dimensions of the arrays to appropriate values
for a practical application.

Note: an array with large dimensions may cause some troubles for your
computer system, such as resulting in a hard disk failure.

7. Program structures
==========

The program is developed based on a 3D code from the FEM book:

Krishnamoorthy, C.S., Finite Element Analysis: Theory and Programming, Tata
McGraw-Hill Publishing Com. Limited, New Delhi, 551pp, 1987.

The program structures and functions of most subroutines used here have been described
in the book, though some new routines have been added.
It is recommended to go through this book to fully understand
the code structure.

8. Acknowledgments and references

This program was developed based on the following researchers' works or programs:

Chandrupatla, T.R. and A.D. Belegundu, Introduction to Finite Elements in Engineering, Prentice Hall, Englewood Cliffs, New Jersey, 414pp, 1991.

Cook, R.D., Concepts and Applications of Finite Element Analysis, John Wiley & Son, New York, 2nd ed. 537pp, 1981.

Krishnamoorthy, C.S., Finite Element Analysis: Theory and Programming, Tata McGraw-Hill Publishing Com. Limited, New Delhi, 551pp, 1987.

Melosh H.J. and A. Raefsky, The dynamical origin of subduction zone topography.

Melosh H.J. and Raefsky, A simple and efficient method for introducing faults into finite element computations, Bull. Seism. Soc. Am., 71, 1391-1400, 1981.

Smith I.M. and D.V. Griffiths, Programming the Finite Element Method, John Wiley & Sons, Brisbane, 3rd ed., 534pp, 1997.

Zienkiewicz, O.C. and R.L. Taylor, The Finite Element Method, Vol.1, Basic Formulation and Linear Problems, McGraw-Hill Book Com., London, 4th ed., 648pp, 1988.

Zienkiewicz O.C., C. Emson, and P. Bettess, A novel boundary infinite element, Int. J. Num. Meth. Engng., 19, 393-404, 1983.

9. Warranty Disclaimer
