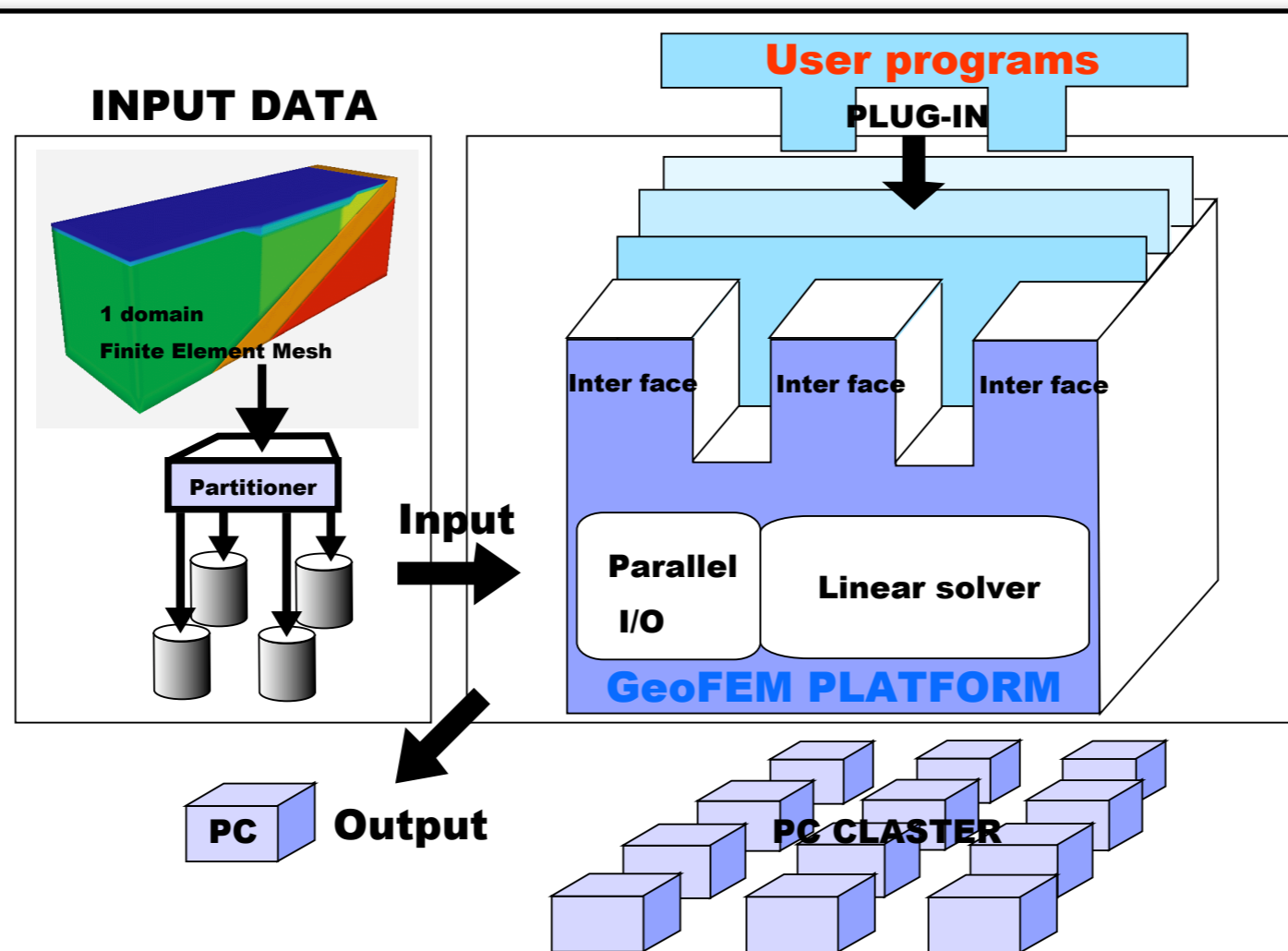


# What is GeoFEM ?

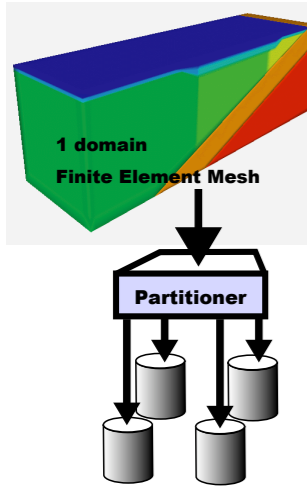
- ✓ Parallel FEM Code for Solid Earth Simulation
- ✓ Developed by **RIST ( Research Organization for Information Science and Technology )** as a product of **GeoFEM Project** ( 1998~2002 )
  - ◎ Free Software
    - Source codes and Detailed Documents are downloadable from <http://geofem.tokyo.rist.or.jp>
  - ◎ GeoFEM works on **PC Cluster(linux), SR2201(Hitachi), SR8000(Hitachi), SX4(NEC), Earth simulator(NEC)** etc.

# Overview of GeoFEM system

- ✓ Parallel calculation & Programming languages
  - **MPI( Message Passing Interface ) Library** is used for parallel computation
  - Mainly Written in **Fortran 90** (Visualization programs are written in C)

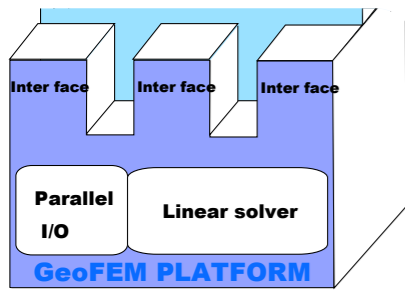


- ✿ Utility tool ( Partitioner for 1 domain FE mesh )
- ✿ **GeoFEM Platform** ( Parallel I/O, Linear equation solver )
- ✿ **User Program** (treated as **MODULE SUB-PROGRAMs of Fortran 90**)



## ✓ Utility tool:

- (Unfortunately, GeoFEM does **NOT** support generation of FE meshes.) If you prepare an original mesh of **GeoFEM FORMAT\***, it generates input data for parallel calculation from original mesh. (\* AS for **GeoFEM FORMAT**, please see GeoFEM HP.)



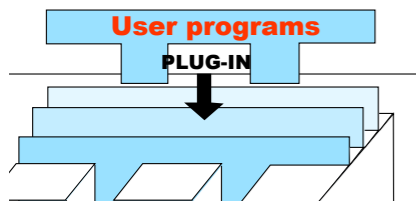
## ✓ GeoFEM Platform:

- SUBROUTINES for BASIC CALCULATION for FE ANALYSIS ( Parallel I/O, Linear equation Solver etc. )

## ✓ User Program:

- It depends on analysis of your own interest. But GeoFEM contains several basic modules as following,

- ➡ Thermal conduction analysis ([geofem-v6.0.1/src/thermal/](#))
- ➡ Wave Propagation ([geofem-v6.0.1/src/dynamic\\_linear/](#))
- ➡ Static or **Quasi-Static Deformation analysis** ([geofem-v6.0.1/src/static\\_linear/](#), [geofem-v6.0.1/src/static\\_contact/](#))



# INSTALL of GeoFEM

- ✓ Download tarball ([geofem-v6.0.1.tgz](#)) from GeoFEM Homepage
  - ➔ GeoFEM Homepage: <http://geofem.tokyo.rist.or.jp>
- ✓ Expand tarball, and Move "geofem-v6.0.1/" Directory
- ✓ Type 'make' (after several changes in Makefile of this directory)
- ✓ Many Executable Files will be generated
  - ➔ ex. Executable file for Quasi-Static Deformation module (ST) will be generated in "geofem-v6.0.1/src/static\_contact/" directory
    - ◎ Using ST, you can easily calculate **POSTSEISMIC Crustal Deformation** after earthquakes

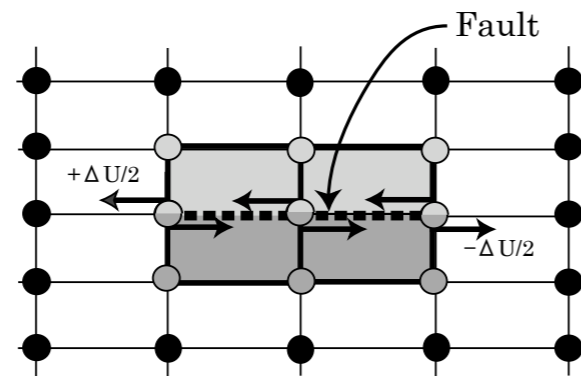
# Quasi-static Deformation Analysis Module (ST)

- ST supports **split node technique** and **linear visco-elastic materials**

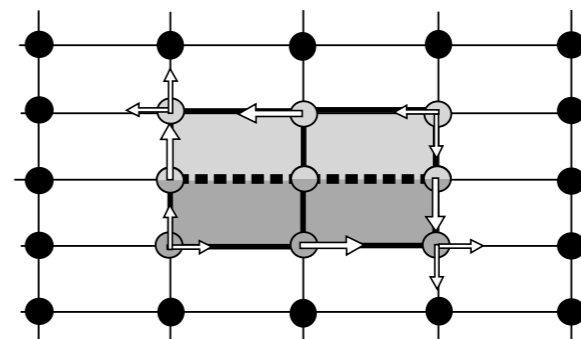
## ✓ Dislocation (Fault Slip as an Initial Condition)

### ❖ Split node (Melosh & Raephsy, 1981, BSSA)

(a) Node splitting



(b) Equivalent Nodal Force



## ✓ Material (several V.E. materials are available)

❖ elastic



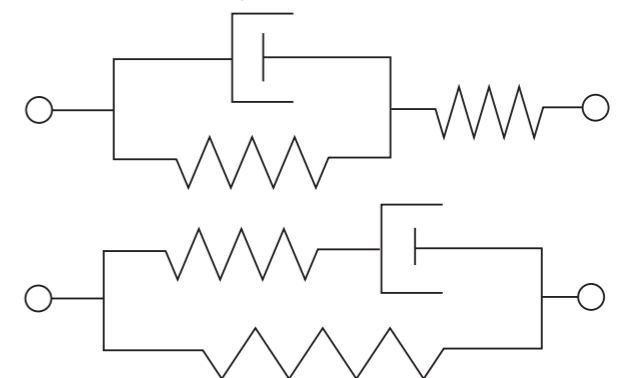
elastic

❖ Maxwell V.E.



Maxwell

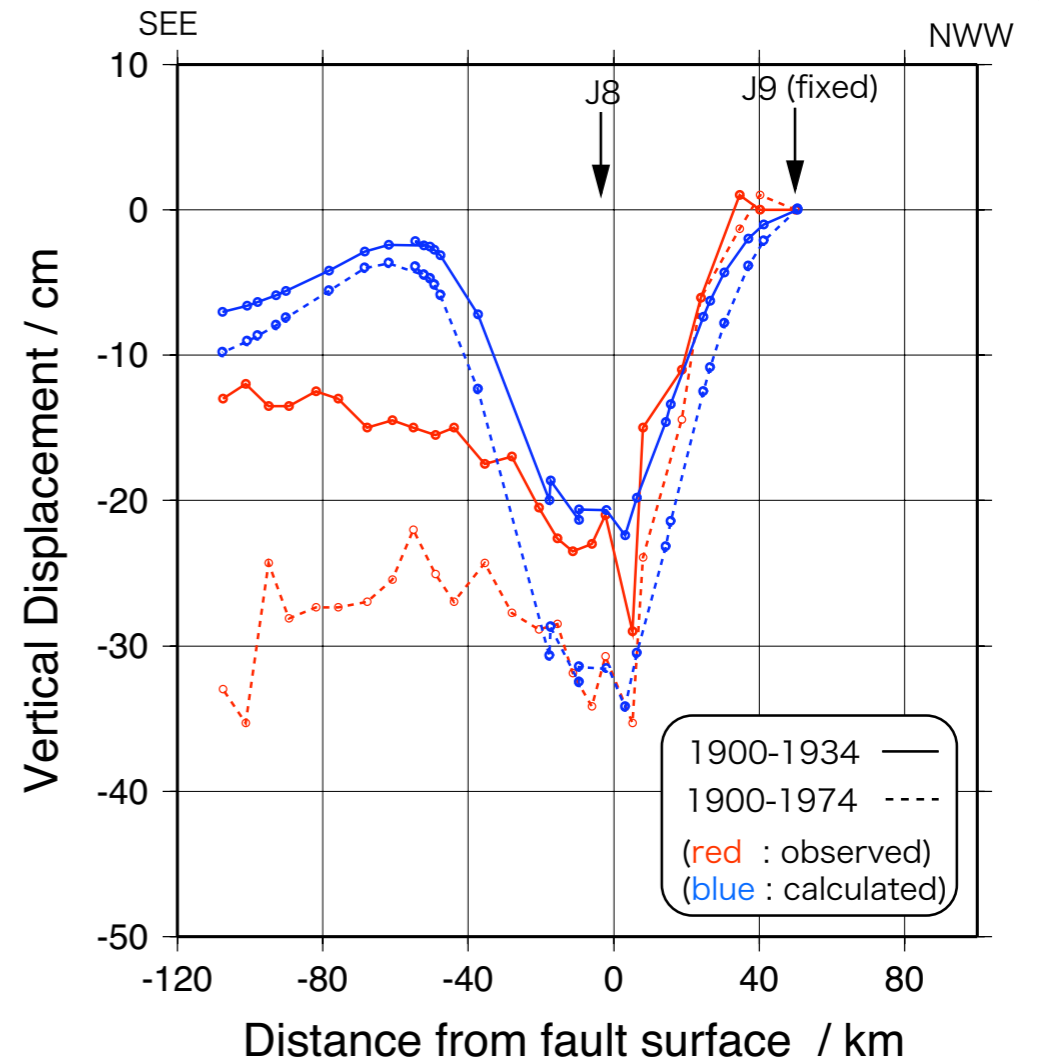
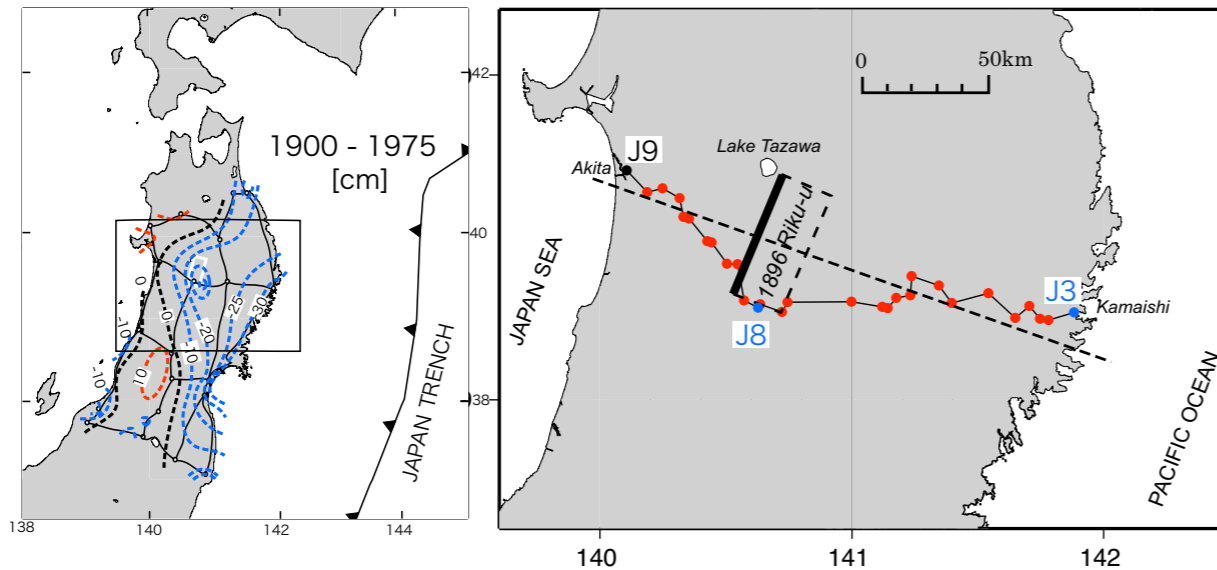
❖ Standard linear Solid



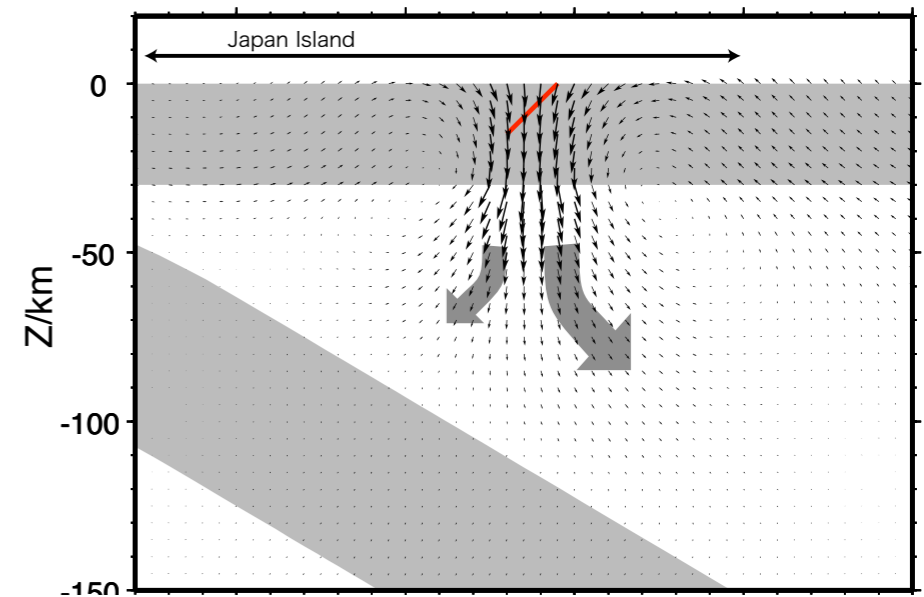
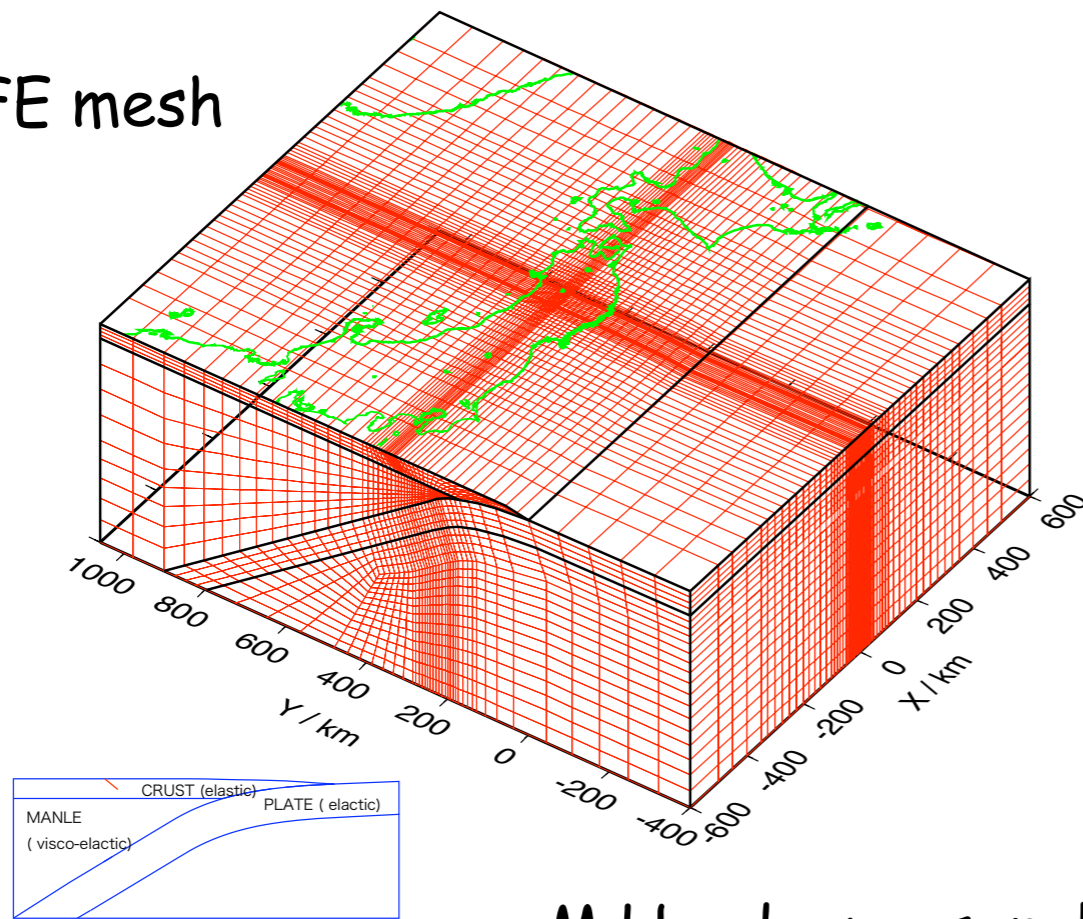
Standard Linear Solid

# Example of Postseismic Deformation calculation using GeoFEM(ST)

## Observation data & Sites



## FE mesh



M.Hyodo, personal communication, 2005

If you are interested in GeoFEM,  
PLEASE DOWNLOAD and USE it.

THANK YOU!

\* There are NO detailed documentations for use of quasi-static module(ST) in GeoFEM homepage, so if you have any question about the use of ST, please contact :  
[hyodo@jamstec.go.jp](mailto:hyodo@jamstec.go.jp)

Mamoru HYODO

Earth Simulator Center(ESC),

Japan Agency for Marine-Earth Science and Technology (JAMSTEC)