Parallel Simulation System for Earthquake Generation: Fault Analysis Modules and Parallel Coupling Analysis

Mikio Iizuka [(1)], Daigo Sekita [(2)], Hisashi Suito [(3)], Mamoru Hyodo [(3)], Karzuro Hirahara [(3)], David Place [(4)], Peter Mora [(4)], Osamu Hazama [(5)], and Hiroshi Okuda [(6)]

(1) Research Organization for Information Science & Technology (E-mail iizuka@tokyo.rist.or.jp) (2) Mitsubishi Reserch Institute,Inc. (sekita@mri.co.jp (3) Earth and Planetary Sciencs, Nagoya University (E-mail suito@eps.nagoya-u.ac.jp, hyodo@eps.nagoya-u.ac.jp, hirahara@eps.nagoya-u.ac.jp) (4) QUAEKS, Department of Earth Sciences, The Univercity of Queensland (E-mail place@earthsciences.uq.edu.au, mora@earth.uq.edu.au) (5) Yokohama National University (E-mail hazama@typhoon.cm.me.ynu.ac.jp) (6) Department of Quantum Engineering and Systems Science, The University of Tokyo (E-mail okuda@garlic.q.t.u-tokyo.ac.jp)

Abstract

Solid earth simulations have recently been developed to address issues such as natural disasters, global environmental destruction and the conservation of natural resources. The simulation of solid earth phenomena involves the analysis of complex structures including strata, faults, and heterogeneous material properties. Simulation of the generation and cycle of earthquakes is particularly important solid earth phenomena, but such a simulation requires analysis of a complex fault dynamics. GeoFEM (Iizuka *et al.*, **1999) is a parallel finite element analysis system intended for problems of solid earth field phenomena. This paper shows recent research of GeoFEM for simulation of earthquake generation and cycles.**

Key words: Solid earth simulations, Generation and cycle of earthquakes, GeoFEM, Parallel finite element analysis

Introduction

Solid earth simulations have recently been developed (Rundle *et al.*, 1999; Bielak *et al.*, 1999; Zienkiewicz *et al.*, 1999; Sivathasan *et al.*, 1998; Zhao *et al.*, 1998) to address issues such as natural disasters, global environmental destruction and the conservation of natural resources. The simulation of solid earth phenomena involves the analysis of complex structures including strata, faults, and heterogeneous material properties. Simulation of the generation and cycle of earthquakes is particularly important in the nonlinear analysis of solid earth phenomena, but such a simulation requires analysis of a complex fault dynamics in three dimensional heterogeneous medium. Because complex phenomena such as multi-phases and a complex fault dynamics etc must be addressed, simulations require much greater computing capacity than what is currently available. GeoFEM is the parallel finite element analysis system, which is designed to handle the large-scale simulation of solid earth phenomena. This study shows an effective way to analyze large-scale parallel fault dynamics as kinematic earthquake cycle by dislocation on plate surface and as contact problem, with iterative solver and the augmented Lagrange method (Landers *et al.*, 1985; Landers *et al.*, 1986; Heegaard *et al.* , 1993) using GeoFEM. In order to simulate solid earth system, many different models should be parallelized, coupled and integrated on the advanced parallel computers. But that is very difficult for non-specialist in computational science. Therefor the parallel platform like GeoFEM, which enables the solid earth models to be easily parallelized and coupled, should be needed. Parallel coupling is most important issue for multi-physics/multi-scale. We also explain the parallel coupling platform. Finally, the paper will show LSMearth which is particle-based model and GeoFEM coupling analysis system.

Kinematic Earthquake Cycle Analysis for Large-scale Parallel Fault Dynamics

This section shows recent research for module development of large-scale kinematic earthquake cycle with viscoelastic analysis in three dimensional heterogeneous medium. This method is based on the kinematic split model using dislocation stress.

Formulation of visco-elastic model and FEM analysis

This section outline the formulation of viscoelastic model which is Standard Linear Solid model (3 elements model). Equation(1) shows the constitutive equation of viscoelastic model in GeoFEM. The $\bar{\nu}, \mu, \lambda, \bar{\mu}, \bar{\lambda}$ in equation (1) should be setted for Maxwel, Voigt, Standard linear models.

$$\{\sigma\} + \bar{\nu}\{\dot{\sigma}\} = 2\mu\{\varepsilon\} + \lambda\{\varepsilon_v\} + 2\bar{\mu}\{\dot{\varepsilon}\} + \bar{\lambda}\{\dot{\varepsilon}_v\} \tag{1}$$

Where, $\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz}$. Above-mentioned 5 parameters is expressed by basic elastic constants (poisson ratio:ν, rigidity:μ, viscocity:η) which is used as basic constants in geological area. Time discreted constitutive equation is obtained by using the central difference method as follows,

$$\{\Delta\sigma\} = [S]\{\Delta\varepsilon\} - \frac{\Delta t}{\Delta t/2 + \bar{\nu}}\{Re\} \tag{2}$$

$$S_{ii} = \frac{1}{\Delta t/2 + \bar{\nu}}\left\{\Delta t\left(\lambda/2 + \mu\right) + \bar{\lambda} + 2\bar{\mu}\right\} \quad (i = 1 \sim 3) \tag{3}$$

$$S_{ii} = \frac{1}{\Delta t/2 + \bar{\nu}}\left(\Delta t\mu/2 + \bar{\mu}\right) \quad (i = 4 \sim 6) \tag{4}$$

$$S_{ij} = \frac{1}{\Delta t/2 + \bar{\nu}}\left(\Delta t\lambda/2 + \bar{\lambda}\right) \quad (i \neq j, i = 1 \sim 3) \tag{5}$$

$$\{Re\} = \{\sigma\}_t - 2\mu\{\varepsilon\}_t - \lambda\{\varepsilon_v\}_t \tag{6}$$

By using Equation(2), virtual work is as follows,

$$\int_\Omega \lfloor\delta\varepsilon\rfloor[S]\{\Delta\varepsilon\}dV + \int_\Omega \lfloor\delta\varepsilon\rfloor\left[{}^{(n)}\{\Delta\sigma\} - \frac{\Delta t}{\Delta t/2 + \bar{\nu}}{}^{(n)}\{Re\}\right]dV - \int_\Gamma \lfloor\delta u\rfloor^{(n+1)}\{f_o\}dS = 0 \tag{7}$$

FEM analysis is based on equation (7). Kinematic earthquake cycle is expressed by dislocation on plate surface (Suito*et al.*, 1999). The dislocation is constrained by inner force which is obtained by using equation (2) for dislocation displacement. Dislocation of subduction and earthquake can be handled in fault analysis module(static_contact) in GeoFEM.

Analysis system

Dislocation calculation flow Dislocation calculation is handled in fault analysis module as follows,

```
Program static_conatact
  read subduction and earthquake data
  do /* time integration loop  */
     calc inner dislocation force (by Equaion (2) )
     stress rcover
     make stiff
     call parallel solver
  untile end of simulation
```

parallel handling of subduction and earthquake data GeoFEM can not handle subduction and earthquake data type by GeoFEM mesh data type. Subduction and earthquake data is handled as specific data type for dislocation analysis. Subduction and earthquake data should also be red as parallel data. Fault analysis module can handle the subduction and earthquake data as parallel data using extended utility subroutine for partitioning and reading. Fig. 1 shows the viscoelastic analysis system with kinematic earthquake cycle.

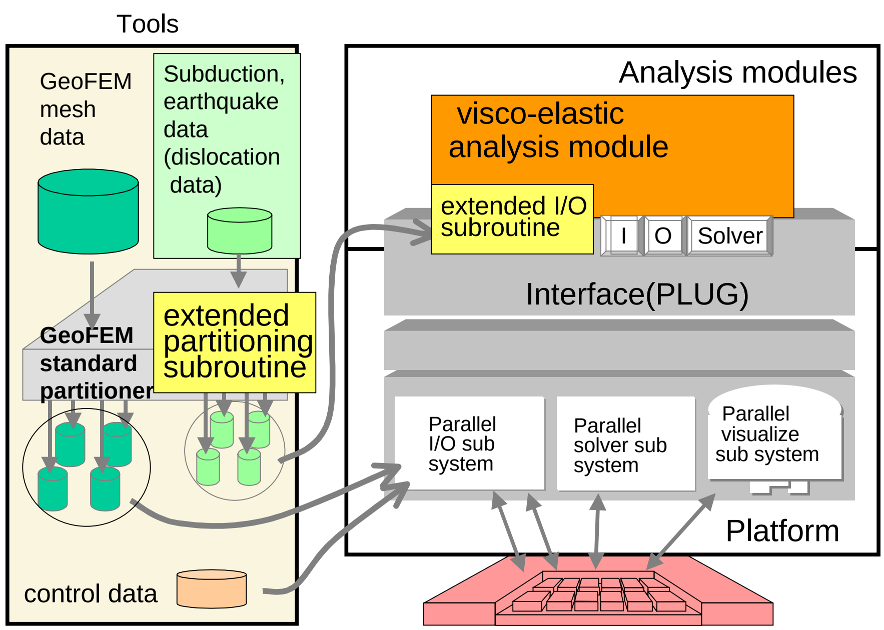


Figure 1: Viscoelastic analysis system with kinematic earthquake cycle

Results

1.37 MDOFs viscoelastic analysis of Southwest Japan model(Fig. 2) has been completed by parallel computaiton. In this case for 1 step analysis, computatinal resource is as follows, total elapsped time; 601 sec, solver elapsed time; 411 sec, number of solver iteration; 531, file volume; 75.2 MB(1 region), 91 MB(32 regions), memory; 79.1 MB (PE). Fig. 4 shows the result of simulation of Southwest Japan model.

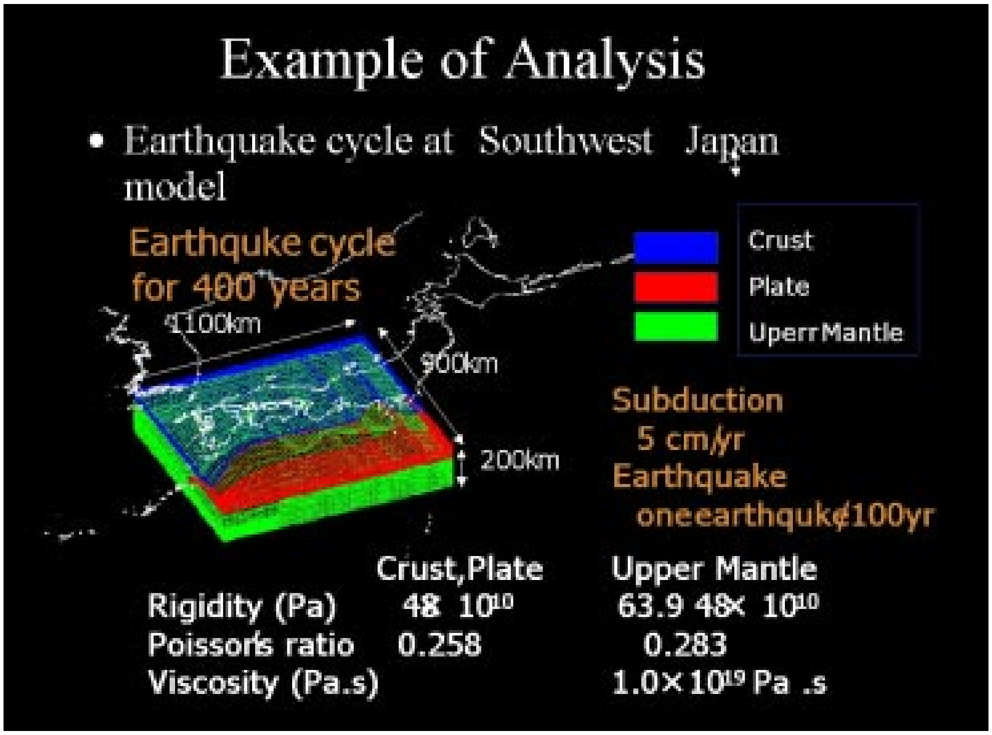


Figure 2: Visco-elastic analysis of Southwest Japan model

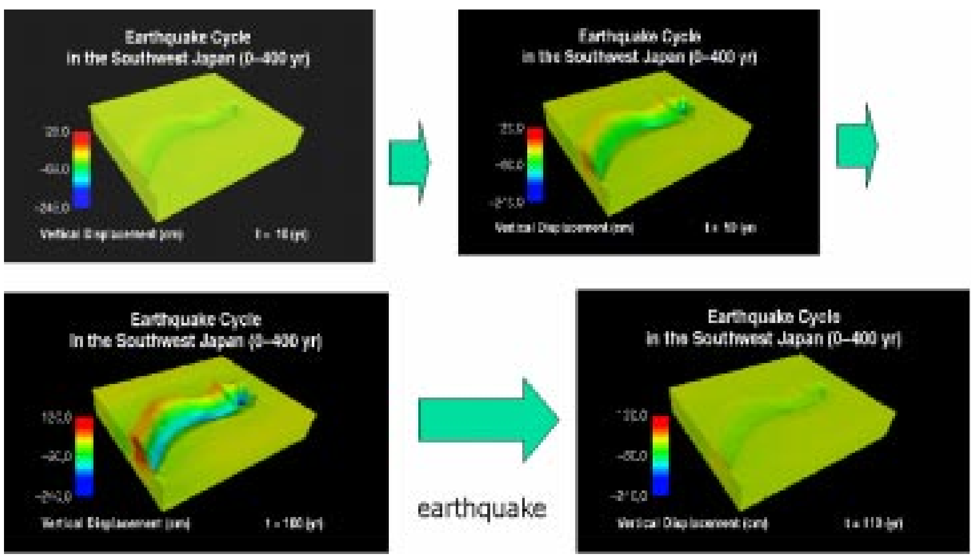


Figure 3: Result of simulation of Southwest Japan model

This research is being accomplished under the collaborative work between RIST/GeoFEM and Nagoya univ. group.

Contact Analysis Analysis for Large-scale Parallel Fault Dynamics

GeoFEM has conventionally used an iterative solvers. This is considered to be the most suitable means of solving symmetric definite matrices and produces outstanding results in the field of large-scale linear elastic analysis (Garatani *et al.*). A current challenge is to develop nonlinear analysis methods based on these results.

To perform simulation of the generation and cycle of earthquak via friction low, contact problems must be solved using the large-scale finite element method where parallel computation is essential for such large-scale finite element analysis to be practical. Using the direct solver in large-scale parallel computation is difficult because it requires a huge memory capacity and a huge amount of communication between processors.

However, iterative solvers are not yet sufficiently versatile to be used for all structural analysis problems. To deal with the contact problem by imposing contact constraints, the penalty (Belytschko *et al.*, 1991) and Lagrange multiplier methods (Bathe *et al.*, 1985) are being applied, usually with the direct solver because the matrix is ill-conditioned and the iterative solver is not applicable.

This study shows an effective way to analyze large-scale parallel contact problem using GeoFEM, with the iterative solver and the augmented Lagrange method to improve matrix conditions. We also explain the application of parallel computation.The paper will show an example of large-scale parallel contact problem analysis of simulated faults that run across the Japanese islands.

Formulation of contact problem analysis using the augmented Lagrange method

Formulation of contact problem analysis This section outlines the formulation of frictionless elastic contact problem analysis. Here, Ω,, $_{\sigma}$, , $_{\sigma c}$ and p are the domain, domain boundary of force, contact body boundary and the domain number respectively. In the contact problem, several domains Ω are in contact at the boundaries , $_{\sigma c}$. The formula is given by the following virtual work and added conditions:

$$\sum_{p}\left[\int_{\Omega_p}\lfloor\delta\varepsilon\rfloor\{\sigma\}dV-\int_{\Gamma_{\sigma p}}\lfloor\delta u\rfloor\{f_o\}dS-\int_{\Omega_p}\lfloor\delta u\rfloor\{r_o\}dV\right]$$
$$=-\sum_{kl}\left[\int_{\Gamma_{\sigma ckl}}\lfloor\delta\left(\bar{\Delta}u\right)\rfloor\{f_{oc}\}dS\right] \tag{8}$$

$$\int_{\Gamma_{\sigma ckl}}\lfloor\delta\left(\bar{f}_c^n\right)\rfloor g dS=0 \tag{9}$$

where, Eq. (8) shows the balance of force and Eq. (9) a kinematic contact constraint as an added condition. A kinematic contact constraint means that domains in contact along a contact surface have no penetration。 And the symbol kl represent a pair of contact boundaries, k and l, whereas $\{\varepsilon\}, \{u\}, \{\sigma\}, \{f_o\}, \{r_o\}, \{f_{oc}\}, \{g\}$ and $\{\bar{\Delta}u\}$ are strain, displacement, stress, external force, body force, contact force, contact boundary gap, and relative displacement, respectively.

The first term of Eq. (8) shows the internal force, the second term shows the traction force and the third term shows the volume force. In the formulation of structural analysis without a contact surface, the right hand side becames zero and Eq. (9) was not required. Therefore, it is a feature of the contact problem to have the contact force term in the formulation and to add the geometrical condition (no penetration) of the contact surface. This added condition causes considerable difficulties in solving the contact problem.

Formulation of the augmented Lagrange method The augmented Lagrange method can be formulated as follows by applying the incremental and Newton-Raphson methods and combining a modified increment with the penalty term $()_{PNL}$ and the augmented Lagrange term $()_{ALM}$.

$$^{(n+1,q+1)}\{\Delta f_{oc}\}={}^{(n+1,q)}\{\Delta f_{oc}\}$$

$$+^{(n+1,q+1)}\{d(\Delta f_{oc}^{n})\}_{PNL}+{}^{(n+1,q+1)}\{d(\Delta f_{oc}^{n})\}_{ALM} \tag{10}$$

$$\{d(\Delta f_{oc}^{n})\}_{ALM}=\{\alpha g\} \tag{11}$$

Contact problem analysis using the augmented Lagrange method is formulated as follows:

$$\begin{aligned}
&\sum_{p}\left[\int_{\Omega_p}\lfloor\delta\varepsilon\rfloor^{(n+1,q+1)}\{d(\Delta\sigma)\}dV\right]\\
&-\sum_{kl}\left[\int_{\Gamma_{\sigma ckl}}\lfloor\delta\left(\bar{\Delta}u\right)\rfloor dS^{(n+1,q+1)}\{d(\Delta f_{oc}^{n})\}_{PNL}\right]\\
&=-\left[\sum_{p}\left[\int_{\Omega_p}\lfloor\delta\varepsilon\rfloor\left({}^{(n+1,q)}\{\Delta\sigma\}+{}^{(n)}\{\sigma\}\right)dV\right.\right.\\
&\qquad\left.-\int_{\Gamma_{\sigma p}}\lfloor\delta u\rfloor^{(n+1)}\{f_o\}dS-\int_{\Omega_p}\lfloor\delta u\rfloor^{(n+1)}\{r_o\}dV\right]\\
&+\sum_{kl}\left[\int_{\Gamma_{\sigma ckl}}\lfloor\delta\left(\bar{\Delta}u\right)\rfloor dS\left({}^{(n)}\{f_{oc}\}+{}^{(n+1,q)}\{\Delta f_{oc}\}\right.\right.\\
&\qquad\left.\left.-{}^{(n+1,q)}\{d(\Delta f_{oc}^{n})\}_{ALM}\right)dS\right]
\end{aligned} \tag{12}$$

$$\{d(\Delta f_{oc}^{n})\}_{PNL}=\{\alpha d(\Delta g)\} \tag{13}$$

$$\{d(\Delta f_{oc}^{n})\}_{ALM}=\{\alpha g\} \tag{14}$$

Modification is repeated on the right side (augmented Lagrange term) until the gap g on the contact boundaries becomes zero. The penalty term on the left-hand side makes the matrix non-singular and convergence can be achieved over a wide range of penalty value. When the penalty value is large, in particular, the gap g of the contact boundaries converges rapidly.

Parallel computation method for contact problems in GeoFEM The domain decomposition method for contact problem analysis applied in the present study is introduced below. Contact problems can be solved in two ways; one gathers contact boundaries within a single region for processing, whereas the other divides and allocates the contact boundaries to each region.This study used the latter considering the flexibility and quality of domain decomposition. To ease the contact point search, we also used a method that has overlapping information about nodes with contact potential within the designated distance. The domain decomposition method for contact problems and communication during parallel computation are explained below.

1. Fig. 4 (a) shows that contact boundaries are set for the master body and slave body for which contact is expected. The node-to-segment model is used as a finite element model and therefore the study focused on contact between the contact boundary element surfaces at the contact boundaries of the master body (master segment) and the nodes at the contact boundaries of the slave body (slave node).

2. Overlapping information on nodes with contact potential inside the designated distance within the domain boundary is necessary. Before dividing the domain, the contact potential distance (CPD) is considered to select contact potential elements (CPE), as shown in Fig. 4 (b). When this domain decomposes along with these CPEs, they are shared as overlapping elements in each region, leading to sharing of the node data necessary for contact problem analysis.

3. First, as shown in Fig. 4 (c), domain decomposition for parallel computation is achieved by edge cutting inside the continuous domains to determine regions and overlapping areas. From this information about the overlapping areas, we set the external and boundary points of the division data of the continuous domains.

4. Next, when dividing the domain by edge cutting (Fig. 4 (d)), the new nodes that are not included in the overlapping elements of the continuous domain are generated as external, boundary points with contact potential. These nodes are called contact potential external points (CPEP) and contact potential boundary points (CPBP). For the domain decomposition method used here, contact boundaries are divided among the regions, the data for contact problem analysis are automatically shared at the boundaries, and inter-region communication to search for contact points is no longer needed.

5. During parallel computation, communication occurs only between the external point and the boundary point if no contact problem analysis takes place. If contact problem analysis proceeds, CPEP and CPBP are added to the nodes for inter-region communication (Fig. 5).

Because only small amount of distortion were handled in this study, CPD is sufficient by the length of one element. Even if large slips are to be handled, this method should be feasible by setting the slip-potential distance at CPD.

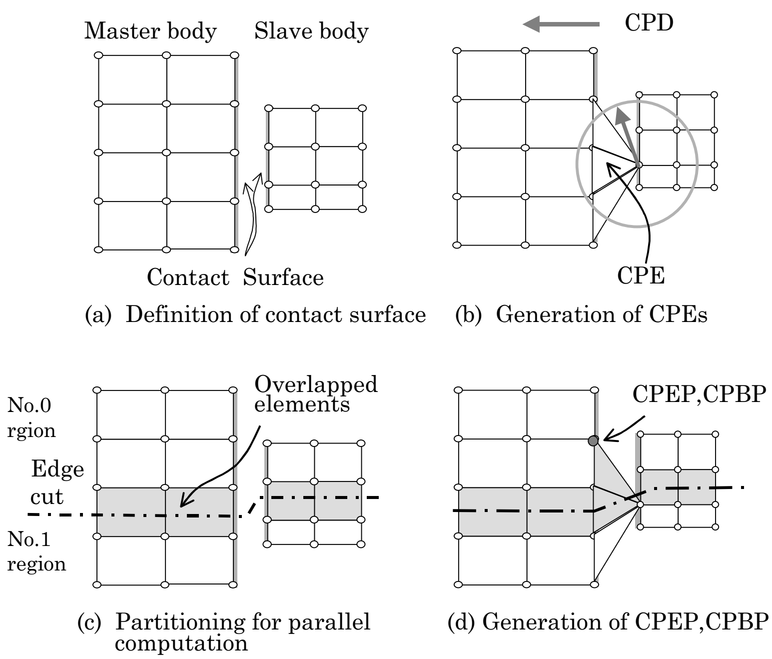


Figure 4: Partitioning of contact problem.

Analysis Example We simulated faults in the Japanese islands for large-scale parallel contact analysis, to demonstrate the validity of the proposed analytical method in large-scale computations. Fig. 6 shows now fault surfaces were obtained by simulating the colliding surfaces of the Eurasia, Philippine and Pacific Sea plates. The area for analysis measures 1020km $\times$840 km $\times$600 km and the boundary conditions were as follows : boundaries running

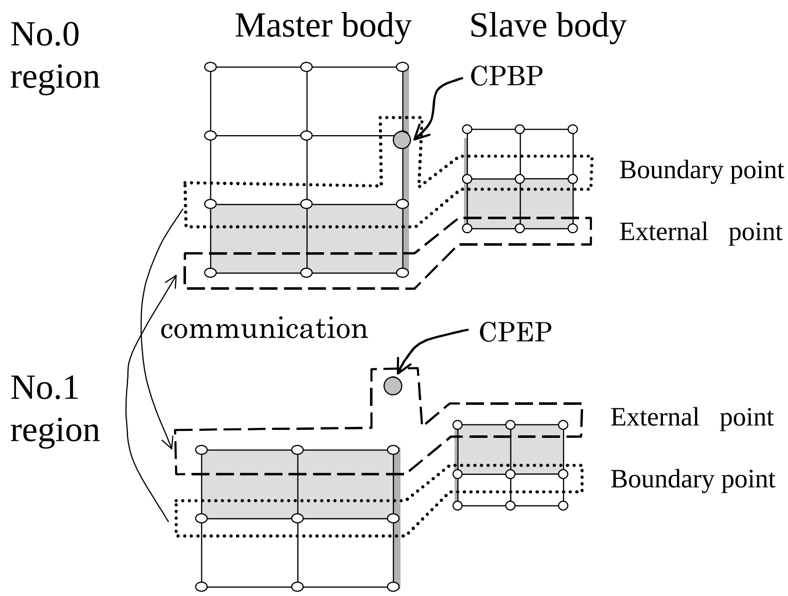


Figure 5: Communication of contact problem.

north-south, on the west side, and on the bottom are slip boundaries, and the east side is a 10 m constrained displacement boundary. The load was given in one step, and the non-linear equation was solved using the Newton-Raphson method. Young's modulus is set to 50GPa and Poisson's ratio is 0.3. We analyzed small-scale (21, 660 DOFs, 7, 220 nodes, 5, 832 elements), and large-scale models (1, 183, 038 DOFs, 394, 346 nodes, 373, 248 elements).

Fig. 7 shows the contour of normal contact force on the fault surface. As this study focused on the analyzing large-scale parallel contact problems, we used idealized boundary conditions and frictionless models of the fault surface. The normal contact force on the fault surface therefore does not have any geophysical meaning, but the contours distribution of the analysis results for the small-scale and large-scale models is similar, demonstrating that parallel contact problems can be accurately analyzed on a large scale.

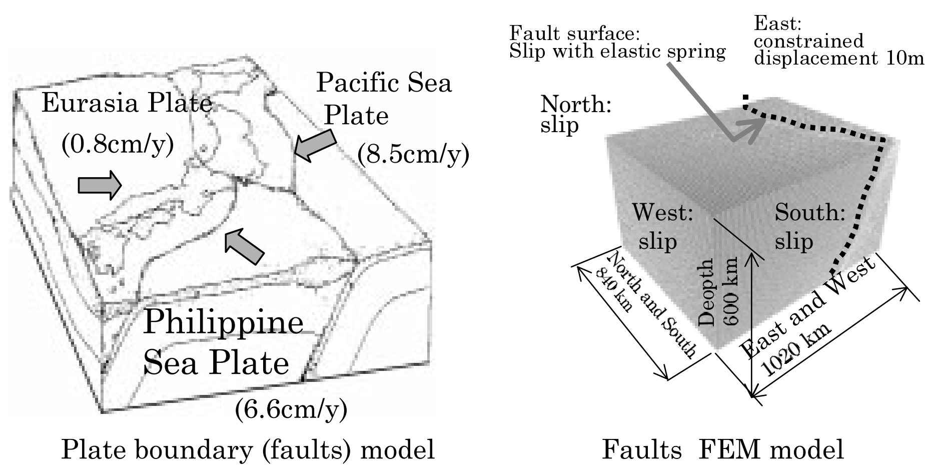
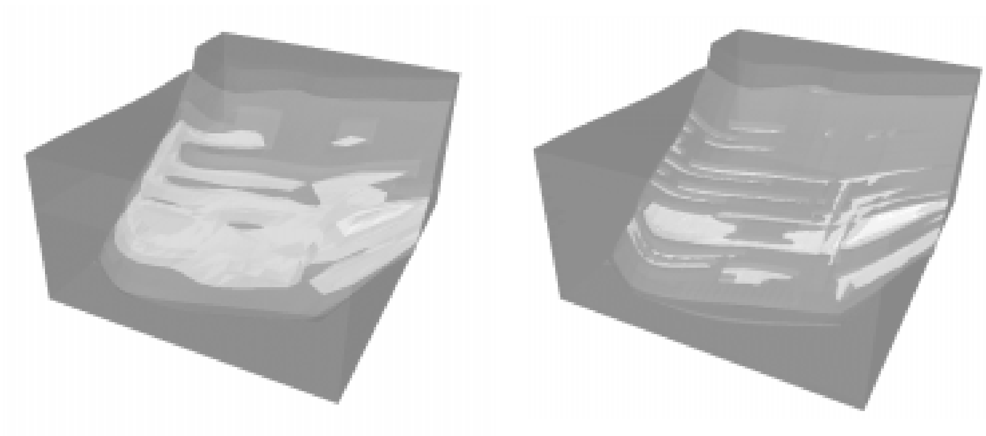


Figure 6: Fault analysis around Japan islands.



21,660 DOFs fault problem   1,183,038 DOFs fault problem

Figure 7: Contour of normal contact force at fault surface.

Parallel coupling analysis platform

The GeoFEM coupler supports the communication between the modules. Of course the coupler was made to support parallel/distributed environment which is assumed by the GeoFEM programs. To use coupler, the two GeoFEM modules just say "send the result of data to another module" and "receive the data from another module" to the coupler and the coupler decide to send/receive what data to/from which PE. Thus a module programmer need not to know the mesh information for the counterpart module. The coupler is embedded in the GeoFEM platform with the other functions of the platform.

Assumptions

To design the coupler program, the following assumptions are done(Fig. 8).

- The analysis spaces are partitioned to multiple regions to calculate in the data parallel manner on the distributed memory environment.
- The mesh data which are used by modules are partitioned independently, i.e there is any assumption for the partitioning rule ("which node belongs to which PE") among the mesh set.

Supported features

The features which are supported by the GeoFEM coupler program follow:

Analyze phase: The coupler system analyzes the multiple mesh sets and find the spatial relation "a node of one mesh set is involved by which element of another mesh set".

Send/receive phase: According to the previous relations, the coupler decides which node data must send the other PE and really sends and receives data. After receiving the node data the coupler implicitly interpolates to calculate the node data from the multiple nearby nodes data from the other PE.

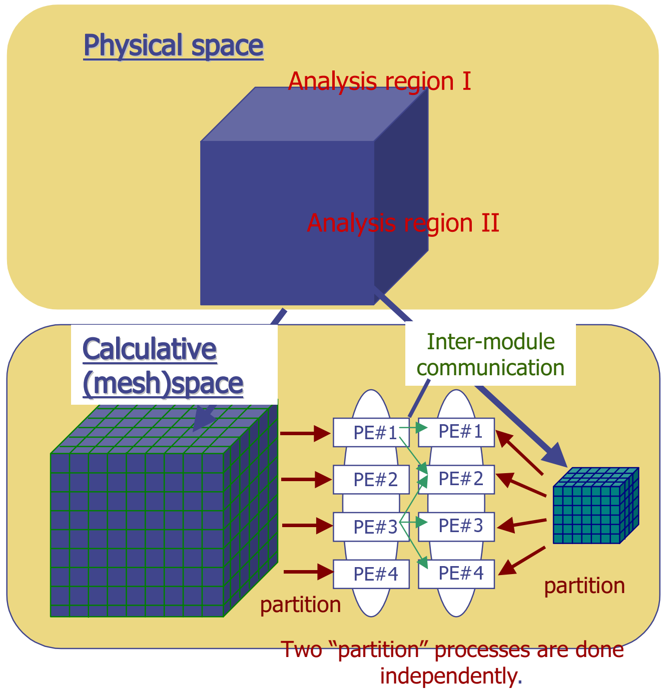


Figure 8: GeoFEM coupler.

Note that the analyze phase is just needed when the spatial relation between meshes is changed even though the send/receive phase is needed multiple times. The analyze phase is very time consuming process because the node and element which are contains in both mesh sets are huge, and to examine the spatial relationship between two node sets, the "spatial search" must done many times. Therefore the analyze phase and the send/receive phase can be done separately. Furthermore, considering the most simple (but ordinary) cases, any node don't move and the spatial relation between nodes and elements doesn't change. For such a simple case, it is very advantageous the analysis phase is implemented in the pre-process manner (i.e. mesh generator/mesh partitioner like), and the user can omit the very time consuming phase for the same mesh set.

Implementation

According to the consideration described in the previous section, the coupler system of GeoFEM is implemented as:

Executional program(Fig. 9): The analysis phase is implemented as the independent executional program named "Xmesh (cross-mesh)". This program reads the multiple (currently two) sets of the already partitioned mesh files and analyze the spatial relations between the sets and generates the files (separately for PEs) which contain the following infos:

- Which node data must be sent to which PE.
- Which node data will be received from which PE.
- Which node data form the other module can be use to calculate the node data using interpolation.

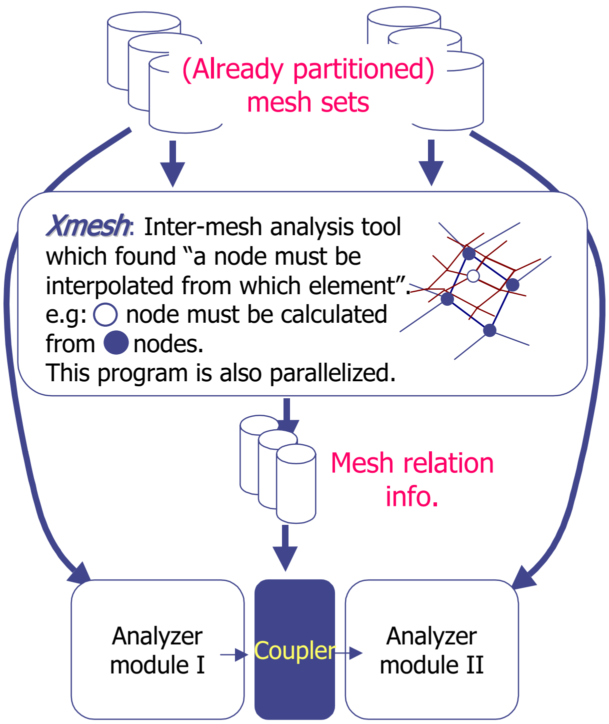


Figure 9: Implementation of GeoFEM coupler.

This xmesh program itself is a parallel program like GeoFEM because the mesh data size of GeoFEM must be assumed very huge for one PE's memory and the searching process for "which node corresponding to which element" analysis is very time consuming for one PE's power. Therefore parallel implementation is very desirable for implementing the phase.

Communication library: The communication library is prepared to implement send/receive phase. The APIs which realize this function are two, i.e. "put data to the other module" and "get data from the other module". When calling the "get" subroutine, the calling program also pass the interpolation function using "passing procedure" feature of Fortran 90. Therefore, the module programmer can easily design the most suitable interpolation method and use it.

Example of Coupling Analysis by coupler of GeoFEM

Now, parallel coupling system is under development. The elementary result, static & zooming analysis, has been got.Test problem is HTTR Carbon Block (CB) analysis(Fig. 12) which is engeneering problem. GeoFEM handles the solid earth phenomena but engeneering problem is convenient for verification of coupling analysis.

Test problem is HTTR Carbon Block (CB) analysis(Fig. 12). Fig. 10 and Fig. 11 shows the example of parallel coupling system for CB static & zooming analysis and interface of GeoFEM coupling system. Fig. 12 shows the result of parallel coupling CB static & zooming analysis. The result shows 1 PE result is same as 2 PEs result.

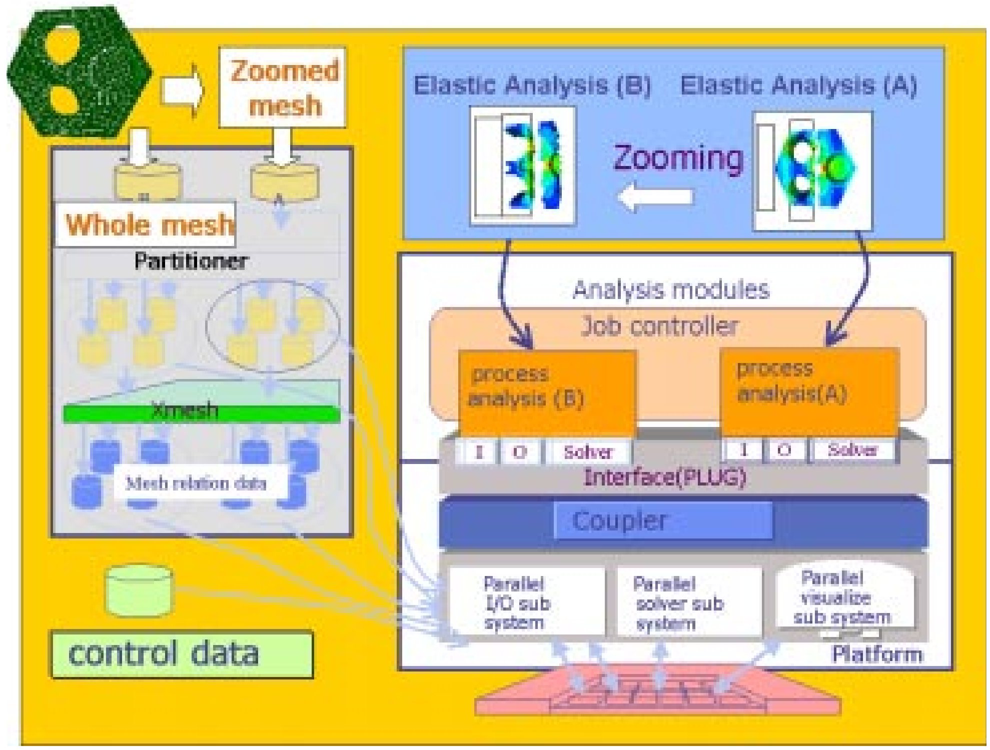


Figure 10: Implementation of GeoFEM coupling system.

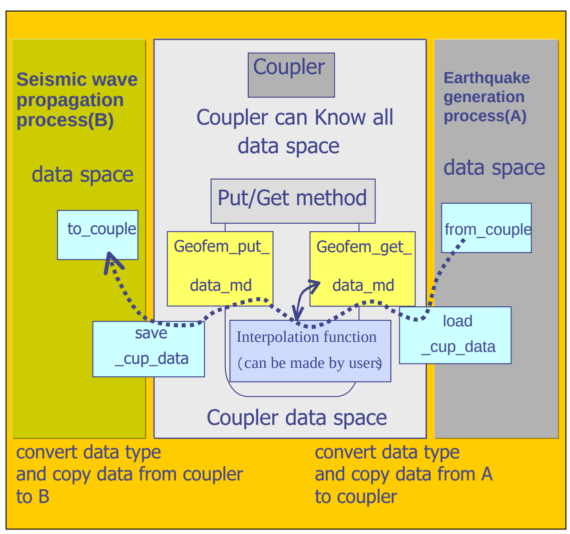


Figure 11: Interface of GeoFEM coupling system.

LSMearth and GeoFEM Coupling Analysis

This section shows another coupling analysis system for GeoFEM with LSMearth(Mora, *et al.*, 1999, Place and Mora, 2000).

Earthquake processes invole complex phenomena and depends on fault dynamics. Different complex phenomena that occur at various scale control the fault dynamics. Observations made during laboratory experiments can be extrapolated using numerical simulations to fault

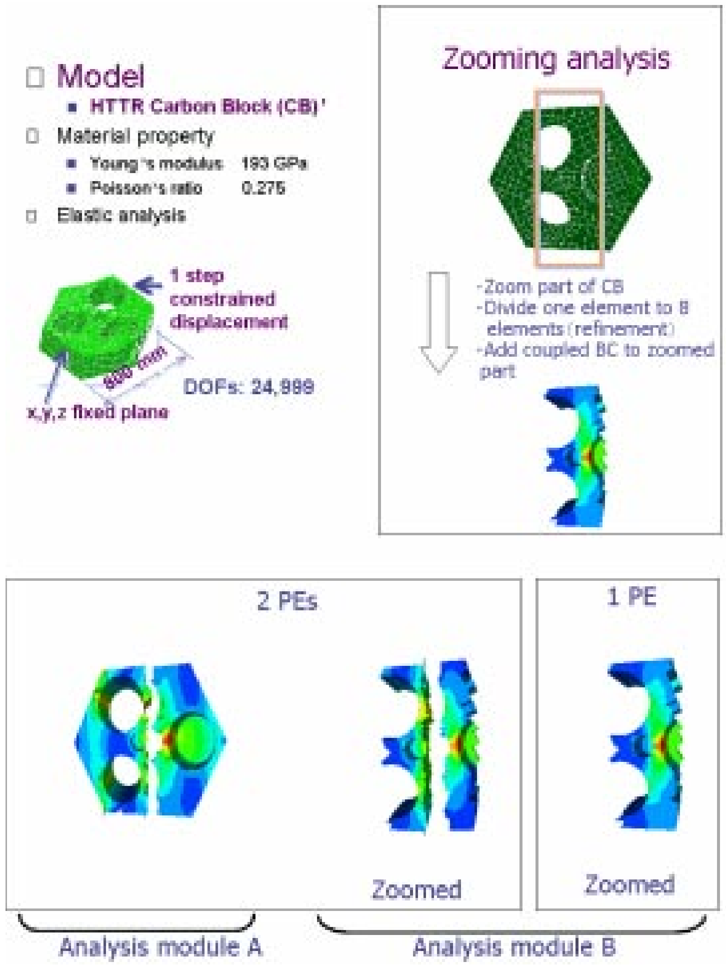


Figure 12: Example of GeoFEM parallel coupling with zooming method.

behaviour. Hence, numerical simulations provide a clue on the scalability of laboratory results and are a means to improve understanding on how such micro-scale processes in a gouge layer affects the macroscopic behavior of fault zone. The interface being developed between the software system developed at QUAKES (LSMearth), and a finite-element based software system - GeoFEM will enable simulation of processes occurring at the microscopic scale using the particle-based model (LSMearth) and simulation of processes occurring at the macroscopic scale, such as plastic deformation and wave propagation, using the finite-element method (GeoFEM). Using this approach, the effects of microscopic phenomena on the macroscopic behavior of a large-scale fault system can be studied. This hybrid method will also extend the resolution of numerical experiments of fault zone behavior by allowing more efficient simulation of those parts of models well approximated as a continuum such as elastic regions outside the gouge zone. This section shows the conceptual design for an interface between LSMearth and GeoFEM.

Conceptual design

The physical interface The exchange of physical values between the two models is done through the fault zone boundaries between the models. Forces and displacements are exchanged between particles of LSMearth and Nodes of GeoFEM along the fault zone boundaries.

Scaling During preliminary experiments, the same scale is used for both models. Hence, each node along the fault zone boundaries correspond to a particle in LSMearth. Exchange of data is performed between these nodes and particles. In the future, when using different scale for GeoFEM and LSMearth, interpolation will be required between the particles and the nodes along the fault zone boundaries. This will allow to use a much smaller scale for the LSMearth model than for the GeoFEM method. Hence, micro-physics occuring at the rock grain scale will be simulated with LSMearth while macroscopic phenomena (such as elastic deformation and wave propagation) are simulated with GeoFEM.

Transfer of forces and deformations To transfer forces and deformation between the models, displacements occurring in the LSMearth model are input at the nodes of the GeoFEM mesh (Fig. 13). From the displacements, deformations occur in the GeoFEM mesh, the traction forces can then be input in the LSMearth model by applying the force to corresponding particle.

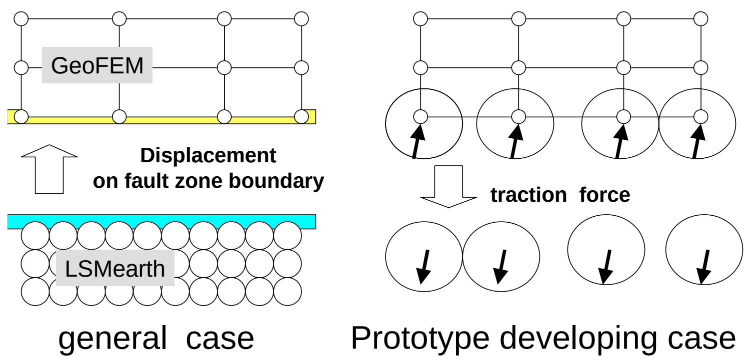


Figure 13: Physical interface

Implementation

The implementation of the hybrid model consists of developing a GeoFEM main program (termed job controller) that controls the time evolution and call for the two models. Exchange of data is controlled by the job controller and is performed using a coupler. The job controller is designed as a GeoFEM main module and based on the GeoFEM-fault analysis module, which allows the access of GeoFEM's functions.

The job controller The time evolution of the hybrid model is controlled by the job controller which is written in Fortran90. The function of the job controller is (1) to initialize the models and the coupler and (2) to perform the time loop of GeoFEM and LSMearth. During the initialization, when calling the subroutine init_coupler(), connections between nodes of the GeoFEM mesh and particles of LSMearth are specified.

```
Program HModel
  call init_lsm_geofem()
  call init_geofem()
  call init_lsm()
  call init_coupler()
```

```
 do /* Time step control for GeoFEM */
    call DoGeoFem()
    do /* Time step control for LSM */
     call DoLSM()
    until end of GeoFEM time step
  until end of simulation
end
subroutine DoGeoFEM                subroutine DoLSM
call get LtoG()                    call get GtoL()
call load CtoG()                   call load CtoL()
call dynamic contact()             call LSMearth oneStep()
call save GtoC()                   call save LtoC()
call put GtoL()                    call put LtoG()
end                                end
```

The coupler The coupler (Fig. 14)is the only module which has access to both the GeoFEM data space and the LSMearth data space. To combine the two data space of GeoFEM and LSMearth, a copy of the models data is placed in the coupler using only "save" and "load" subroutines(GtoC, CtoG). These subroutines have only access to the coupler data space. Model data can be transfer and interpolate from one model to the other using "get" and "put" subroutines. These subroutines have only access to the coupler data space.

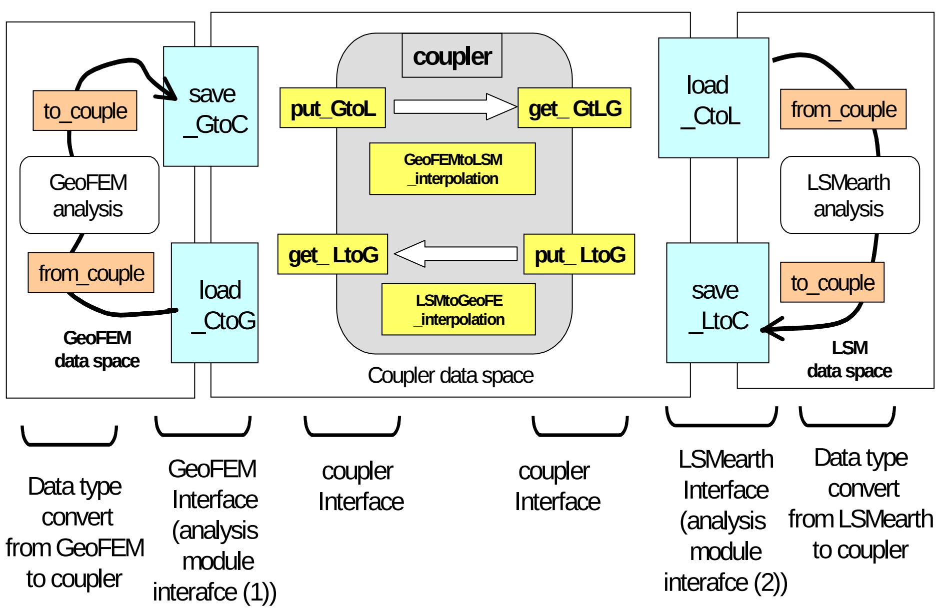


Figure 14: Coupler system configuration

The GeoFEM interface and the LSMearth interface Because the job controller is based on the fault analysis module and LSMearth is written in C++, the LSMearth data and subroutines cannot be access directly. Hence, a C-interface is required to export LSMearth data and subroutines. Furthermore, to keep the modularity of LSMearth, a module in LSMearth, termed GeoFEM data exchange module, is created from which the two subroutines loadCtoL and saveLtoC (Fig. 14) can access LSMearth data or subroutines.

The LSMearth C-interface

Because the GeoFEM is written in Fortran90 and LSMearth in C++, a C-interface is required to call C++ functions from a Fortran program.

Conclusion of this section

The implementation of the interface involves the development of a job controller, a coupler and a GeoFEM data exchange module in LSMearth. Ultimately the coupler interface can operates through a message-passing interface allowing the use of different super-computer foreach model. This work is now under development. The interface GeoFEM-LSMearth will allow multi-scale simulations of large-scale fault system and the dynamics of earthquakes to be performed in future(Fig. 15).

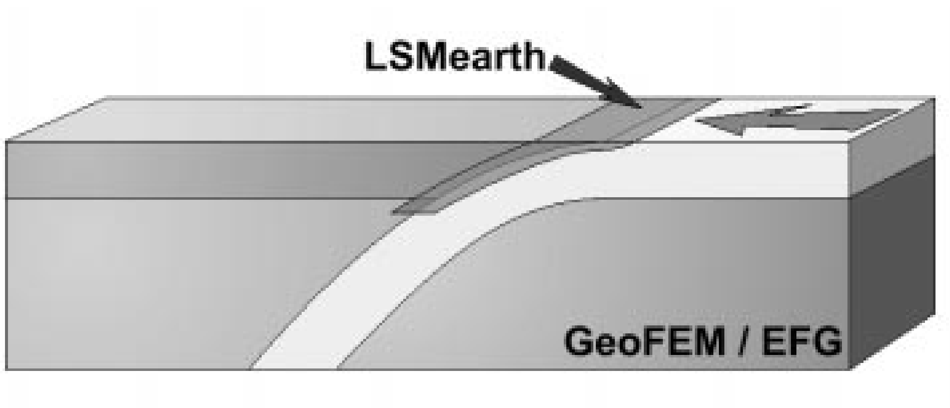


Figure 15: This figure shows the long-term goal of this project. An earthquake simulation in a subduction zone where LSMearth would be used to simulate phenomena occurring in fault gouges at the interface between the plate boundary and GeoFEM would be used to simulate wave propagations, elastic deformations and stress transfer. coupler system configuration

CONCLUSION

This paper showed recent researchs of GeoFEM for simulation of earthquake generation and cycles.The researchs are as follows:

(1) Kinematic Earthquake Cycle Analysis for Large-scale Parallel Fault Dynamics
(2) Contact Analysis Analysis for Large-scale Parallel Fault Dynamics
(3) Parallel coupling analysis platform
(4) LSMearth and GeoFEM Coupling Analysis.

Now, the reserch of analysis for the large-scale Southwest Japan Earthquake simulation is being studied. Above-mentioned analysis module is being integrated in this reserch.

Acknowledgments

Author would like to thank Professor G.Yagwa, University of Tokyo, the supervisor of the "GeoFEM" project and all the members of the GeoFEM team for their discussion and advice.

References

[1] Mikio Iizuka, Kazuteru Garatani, Kengo Nakajima, Hisashi Nakamura, Hiroshi Okuda and Genki Yagawa (1999),
GeoFEM : High-Performance Parallel FEM Geophysical Applications,
ISHPC99, Second International Symposium Proceedings, High Performance Computing, Lecure Notes in Computer Science 1615,
292-303.

[2] John B.Rundle, Tom Henyey, J.-Bernard Minster, and Geoffrey Fox (1999), *General earthquake models* , 1-st ACES Workshop Proceedings, ed Mora, P., (The APEC Cooperation for Earthquake Simulation, Brisbane, Australia), 281-287.

[3] Jacobo Bielak and Omar Ghattas (1999), *Computational challenges in seismology* , 1-st ACES Workshop Proceedings, ed Mora, P., (The APEC Cooperation for Earthquake Simulation, Brisbane, Australia), 325-328.

[4] Zienkiewicz, O.C., M. Huang and M. Pastor, *Numerical prediction for Model No 1*, In *Verification of Numerical Procedures for the Analysis of Soil Liquefaction Problems 1* (ed. Arulanandanand Scott)(Balkema, Rotterdam, 1993), pp.259-274.

[5] Sivathasan.k., Paulino, G.G., Li, X.S. and Arulanandan, K., *Validation of Site Characterization Method for the Study of Dynamic Pre Pressure Response, Geotechnial Special Publication No75, Volume one, Geotechnical Earthquake Engineering and Solid Dynamics III* (ASCE, Seattle, Washington 1998)

[6] Zhao C., Hobbs B. E. and Mühlhaus H. B. (1998), *Finite element modelling of temperature gradient driven rock alteration and mineralization in porous rock masses*, Compu. Meth. Appl. Mech. Eng. *165*, 175-187.

[7] J.A.Landers and R.L. Taylor (1985) , *An augmented Lagrangian formulation for the finite element solution of contact problems* , *Rep. No. UCB/SESM-85/09* , University of California, Berkley.

[8] J.A.Landers and R.L. Taylor (1986) , *An augmented Lagrangian formulation for the finite element solution of contact problems* , *Rep. No. AD-A166 649* , University of California, Berkley.

[9] Heegaard, J.-H.and Curnier, A. (1993), *An Augmented Lagrangian Method for Discrete Large-Slip Contact Problems*, Int.J.for Num.Meth.in Eng., *36*, 569-593.

[10] Suito, H., Hirahara, K.(1999), Simulation of Postsismic Deforamtion *Simulation of Postsismic Deformation caused by the 1896 Riku-u Earthquake, Northeast Japan: Re-evauation of the viscosity in the upper mantle* , Geophysical Reserchletters, Vol. 26, No.16, 2561-2564.

[11] Garatani, K., Nakamura, H., Okuda, H., Yagawa, G., *GeoFEM: High Performance Parallel FEM for Solid Earth*, Proceedings of 7th Hight performance Computing and Networking (HPCN Europe '99), LNCS-1593, 133-140.

[12] T.Belytschko and M.O.Neal (1991), *Contact-Impact by the Pinball Algorithm with Penalty and Lagrangian Methods*, Int.J.Numer.Methods Eng., *31*, 547-572.

[13] Mora, P., Place, D., Abe, S., Weatherley, D., and Keane, T. (1999) , *The lattice Solid Model: towards a realstic simulation model for earthquake micro-physics and the development fo a virtual laboartory for the earthquake cycle* , 1-st ACES Workshop Proceedings, ed Mora, P., (The APEC Cooperation for Earthquake Simulation, Brisbane, Australia), 121-128.

[14] Place, D. and Mora P, *A 3D Virtual Earth Simulator for Earthquake Micro-physics: LSMearth*, Second ACES Workshop, Japan (2000).