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), Kazuro 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. (E-mail 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 University of Queensland (E-mail place@quakes.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@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, but such a simulations require for the analysis of complex fault dynamics. GeoFEM (Iizuka *et al.*, 1999) is a parallel finite element analysis system intended for solid earth field phenomena problems. This paper describes recent development in the GeoFEM project for the 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 materials.

Simulation of the generation and cycle of earthquakes is particularly important in the nonlinear analysis of solid earth phenomena, but such simulations require for the analysis of complex fault dynamics in a three dimensional heterogeneous medium. The Finite Element Method (FEM) is widely used for complex geometric and heterogeneous medium problems. The simulations require a much greater computing capacity than what is currently available, because complex phenomena such as multi-phases and a complex fault dynamics etc must be addressed. This study shows an effective method for analysis of large-scale parallel fault dynamics as a kinematic earthquake cycle by dislocation of the fault surface and as a contact problem, with a iterative solver and the augmented Lagrange method (Landers *et al.*, 1985; Landers *et al.*, 1986; Heegaard *et al.*, 1993) using GeoFEM. GeoFEM is the parallel finite element analysis system designed to handle the large-scale simulation of solid earth phenomena. Many different models need to be parallelized, coupled and integrated on advanced parallel computers in order to simulate a solid earth system. However, such a task is a difficult process that requires for a detailed knowledge of computational science. Therefor the GeoFEM parallel platform, which enables solid earth models to be parallelized and coupled, is under development to assist the developers of solid earth simulations. Parallel coupling is most important issue for multi-physics/multi-scale solid earth simulations. Details of the parallel coupling platform are presented and coupling analysis system between the LSMearth which is a particle-based model simulation system for solid earth and the GeoFEM are described.

GeoFEM Fault Analysis modules

GeoFEM employs two methods for fault analysis. One(the Contact Analysis type) uses the contact analysis technique, which uses an iterative solver with the augmented Lagrange method and is suited to the analysis of complex geometry fault dynamics via friction law. The other(the Kinmatic Earthquake Cycle Analysis type) is based on the kinematic split model, which uses dislocation stresses and is suited to the analysis of crust deformation history by earthquake cycle based on earthquake data. Both models are used for Earthquake Generation Cycle analysis. The Contact Analysis type is also used for Fault Dynamic Rupture analysis. Earthquake generation and seismic wave propagation are analysed via coupling analysis. The Contact Analysis type and Kinmatic Earthquake Cycle Analysis type methods are coupled in Earthquake Generation Cycle analysis.

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 a 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 outlines the formulation of a viscoelastic model, which is a Standard Linear Solid model (3 element model). Equation(1) shows the constitutive equation of the viscoelastic model in GeoFEM. The $\bar{\nu}, \mu, \lambda, \bar{\mu}, \bar{\lambda}$ terms in equation (1) are defined using the 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, $\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz}$. The 5 parameters are expressed using basic elastic constants (poisson ratio:ν, rigidity:μ, viscocity:η) A 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), the virtual work is as follows:

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

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

Analysis system

Dislocation calculation flow The dislocation calculation is handled in the 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 as GeoFEM mesh type data. Subduction and earthquake data is therefor handled as specific data type for dislocation analysis and earthquake data should also be read as parallel data. The fault analysis module can handle the subduction and earthquake data as parallel data using an extended utility subroutine for partitioning and reading. Figure 1 shows the viscoelastic analysis system with a kinematic earthquake cycle.

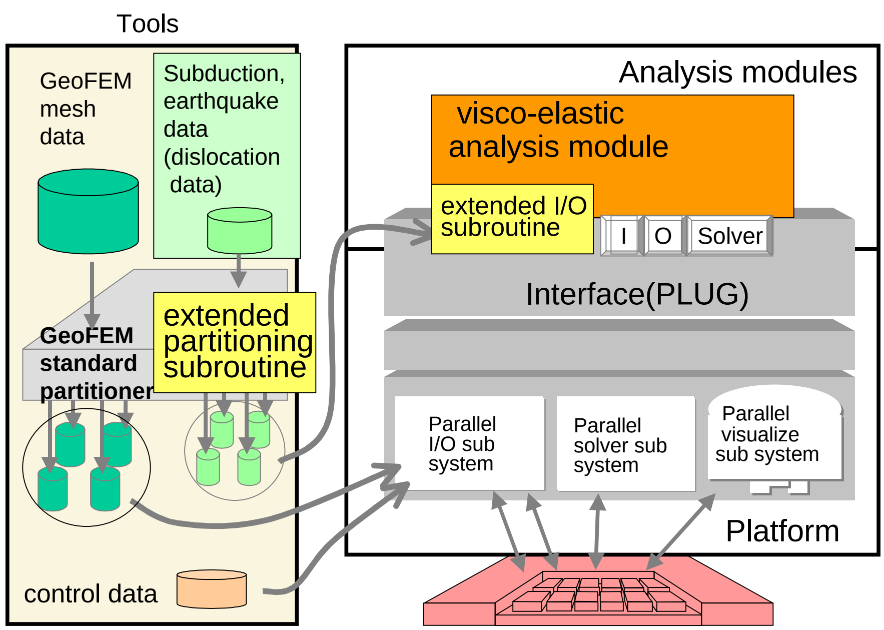

Figure 1: Viscoelastic analysis system with kinematic earthquake cycle

Results

A 1.37 MDOFs viscoelastic analysis of the Southwest Japan model (Figure 2) has been completed by parallel computation on SR2201 at the University of Tokyo. In this case, the computatinal resources for a 1 step analysis were follows, total elapsed time; 601 sec, solver elapsed time; 411 sec, number of solver iterations; 531, file volume; 75.2 MB(1 region), 91 MB(32 regions), memory; 79.1 MB (PE). Figure 3 shows the resources for the simulation. The elapsed time results (sec/step) shows that the speed increased with the number of

PEs(processor elements).

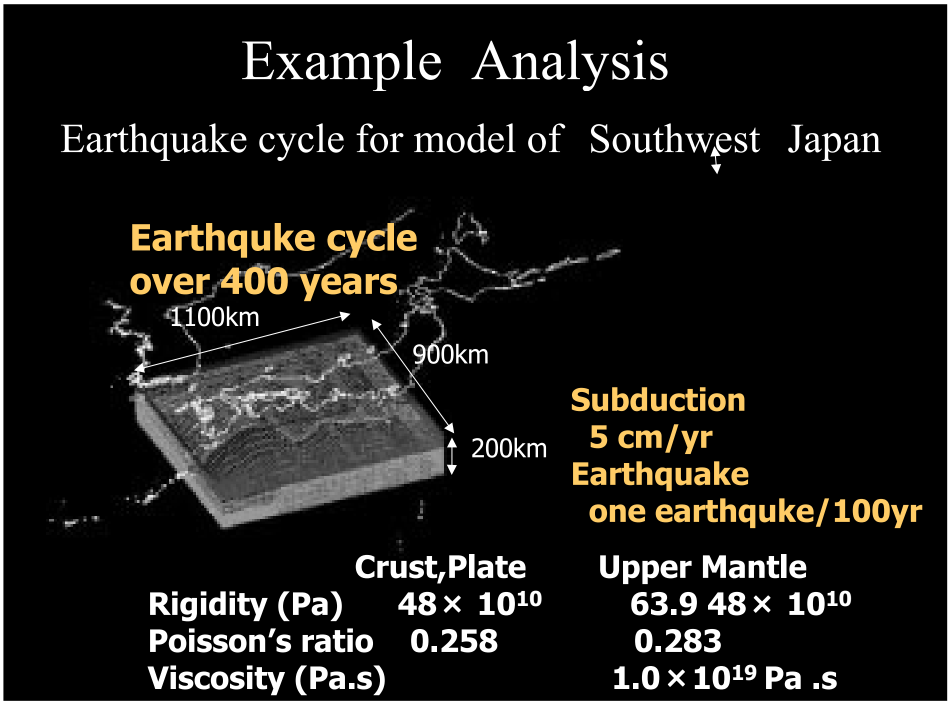

Figure 2: Visco-elastic analysis of Southwest Japan model

Figure 4 shows the results of the simulation of the Southwest Japan model.

Contact Analysis for Large-scale Parallel Fault Dynamics

GeoFEM uses an iterative solver, which is considered to be the most suitable technique for solving symmetric definite matrices 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 a simulation of the generation and cycle of an earthquaks via friction law, 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 large memory capacity and significant 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)

	PEs	Memory (MB/PE)	Elapsed Time (sec/step)	File(MB/all) input	File(MB/all) output
178,200 DOFs model	16	25.8	79.7 (1.0)	13	21
	32	13.8	44.6(1.8)	14.1	22.8
	64				
1,369,095 DOFs model	16	173.5	1193(1.0)	88	143.5
	32	90.5	593(2.0)	91	149
	64	48.2	299(3.99)	97	159

(): increase in speed

Figure 3: Computational Resource.

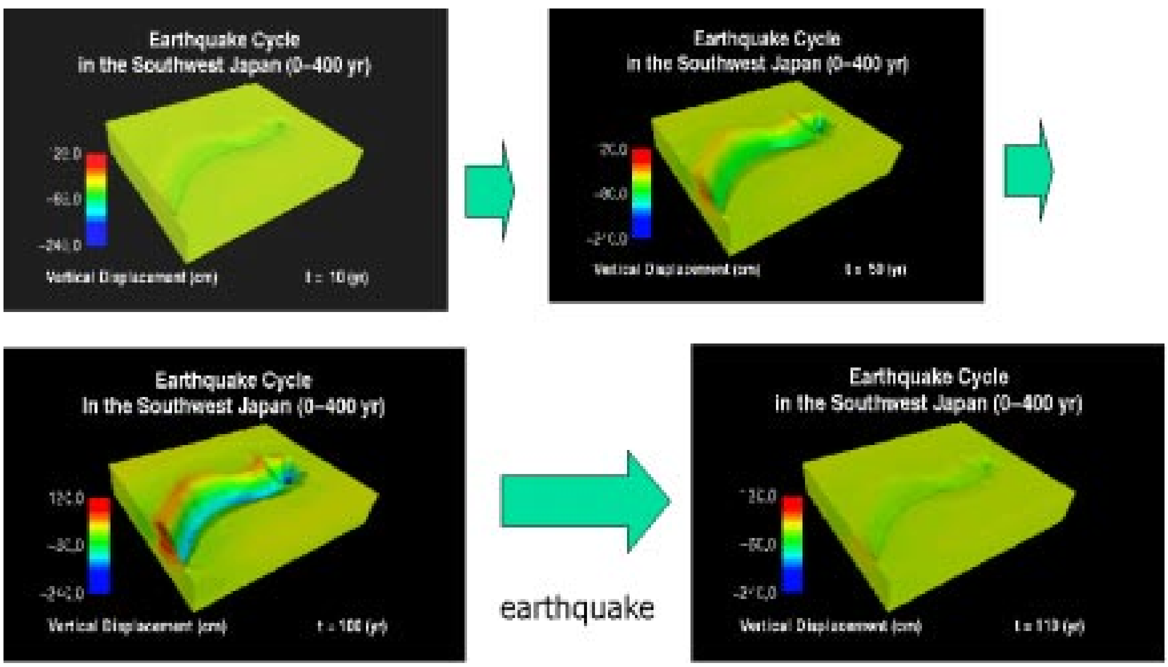

Figure 4: Result of simulation of Southwest Japan model

and Lagrange multiplier methods (Bathe *et al.*, 1985) are applied usually with the direct solver because the matrix is ill-conditioned and the iterative solver is not applicable.

This study shows an effective method for analysis of a 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, Γ_σ, $\Gamma_{\sigma c}$ and p are the domain, domain force boundary, contact body boundary and domain number respectively. In the contact problem, several domains Ω are in contact at the boundaries $\Gamma_{\sigma c}$. 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) is a kinematic contact constraint as an added condition. A kinematic contact constraint means that domains in contact along a contact surface have no penetration. The symbol kl represents a pair of contact boundaries, k and l, whereas $\{\varepsilon\}, \{u\}, \{\sigma\}, \{f_o\}, \{r_o\}, \{f_{oc}\}, \{g\}$ and $\{\bar{\Delta} u\}$ are the 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 volumetric force. In the formulation of a structural analysis problem without a contact surface, the right hand side becames zero and Eq. (9) is 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 when solving a contact problem.

Formulation of the augmented Lagrange method If penalty method is used, the penalty parameter needs a large value, which worsens the matrix condition and causes difficulties in using the iterative solver. We therefore used the augmented Lagrange method as shown below to improve the matrix condition for large-scale parallel contact analysis.

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 $()_{ALM}$.

$$
\begin{aligned}
{}^{(n+1,q+1)}\{\Delta f_{oc}\} &= {}^{(n+1,q)}\{\Delta f_{oc}\} \\
&\quad + {}^{(n+1,q+1)}\{d(\Delta f^n_{oc})\}_{PNL} + {}^{(n+1,q+1)}\{d(\Delta f^n_{oc})\}_{ALM}
\end{aligned} \tag{10}
$$

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

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

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

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

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

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 values. The gap g of the contact boundaries converges rapidly when the penalty value is large.

Parallel computation method for contact problems in GeoFEM The parallel computation method described here has been developed for parallel computers with distributed memories. To optimize the method for use on this type of computer, the entire model is divided into smaller regions, each of which is then allocated to a PE. There are two methods used to perform this domain decomposition. One approach involves the use of an iterative solver to deal with the overall degrees of freedom and the other uses an iterative method for solving the degrees of freedom condensed at the partitioned domain boundary by eliminating the inner degree of freedom in each domain. The former is used for GeoFEM because of the stability of the solver and the flexibility of its application to various problems (several iterative solvers that have been developed can only be used according to the type of problem). The iterative method (ICCG method) is used for the solver (Nakajima and Okuda, 1998).

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 after considering the flexibility and quality of the domain decomposition. To ease the contact point search, we also use 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 is explained below.

1. Figure 5 (a) shows that contact boundaries are set for the master body and the 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 Figure 5 (b). When this domain decomposes along with the 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 Figure 5 (c), domain decomposition for parallel computation is achieved by edge cutting inside the continuous domains to determine regions and overlapping areas. The external and boundary points of the division data of the continuous domains are set from this information about the overlapping areas.

4. Next, when dividing the domain by edge cutting (Fig. 5 (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, 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 is performed. If contact problem analysis proceeds, CPEP and CPBP are added to the nodes for inter-region communication (Figure 6).

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.

Example Faults in the Japanese islands were simulated for large-scale parallel contact analysis to demonstrate the validity of the proposed analytical method in large-scale computations. Figure 7 shows how fault surfaces were obtained by simulating the colliding surfaces of the Eurasia, Philippine and Pacific Sea plates. The analysis area measured 1020 km $\times$840 km $\times$600 km and the boundary conditions were as follows: boundaries running north-south, on the west side, and on the bottom were slip boundaries, and the east side was

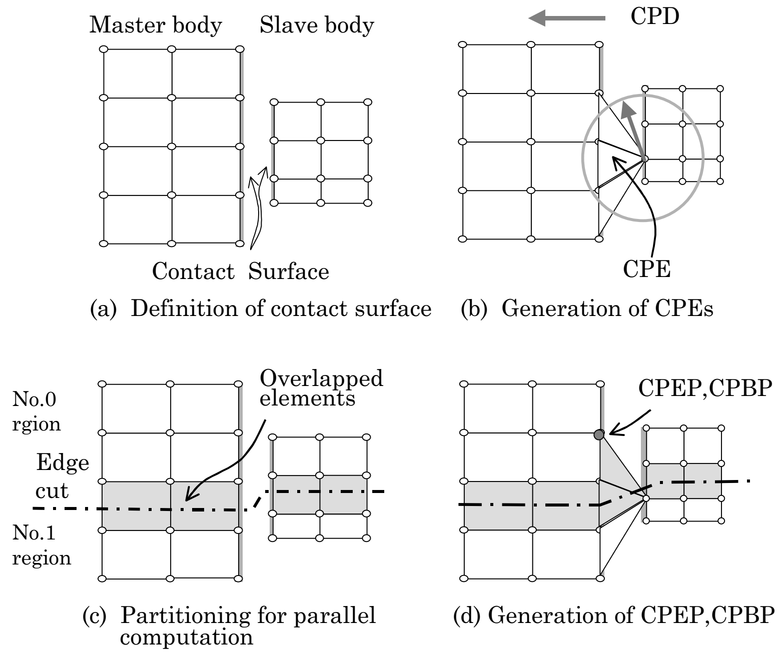

Figure 5: Partitioning of contact problem.

a 10 m constrained displacement boundary. The load was given in one step, and the non-linear equation solved using the Newton-Raphson method. We analyzed small-scale (21, 660 DOFs, 7, 220 nodes, 5, 832 elements), medium-scale (156, 066 DOFs, 52, 022 nodes, 46, 656 elements), and large-scale models (1, 183, 038 DOFs, 394, 346 nodes, 373, 248 elements). The small-scale mode was divided into 16 regions, the medium-scale model into 16 and 32 regions, and the large-scale model into 32 regions. Young's modulus was 50GPa and Poisson's ratio was 0.3 in each model.

Figure 9 shows the relationship between scale and analysis time. The penalty parameter was 10^{10}. The SR2201 computer installed at the University of Tokyo completed the computations within approximately 2.5 hours for the 1.18×10^6 DOFs model, which was the model with the largest scale. These results suggest that large-scale parallel contact analysis using the iterative solver with the augmented Lagrange method is possible.

Figure 8 shows the normal contact force contours on the fault surface. As this study focused on 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

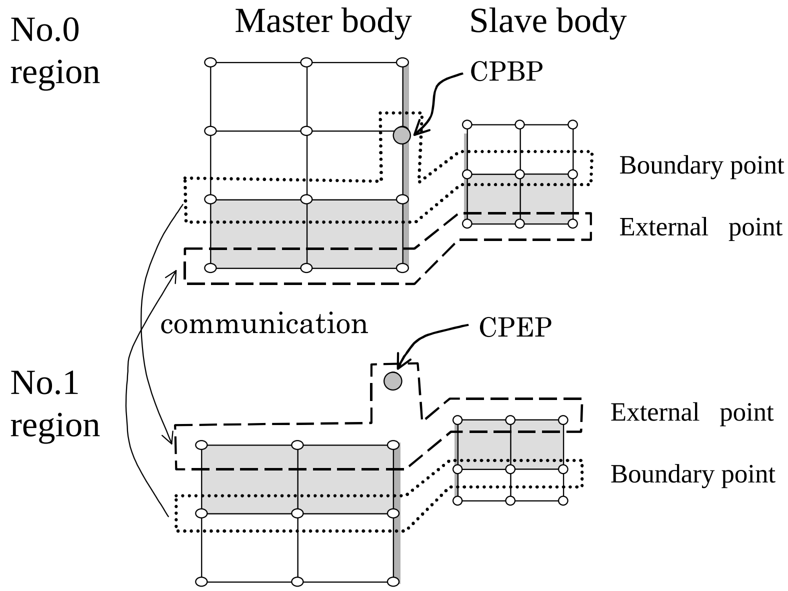

Figure 6: Communication of contact problem.

surface therefore does not have any geophysical meaning, but the distribution of contours for the small-scale and large-scale models is similar, demonstrating that parallel contact problems can be accurately analyzed on a large scale.

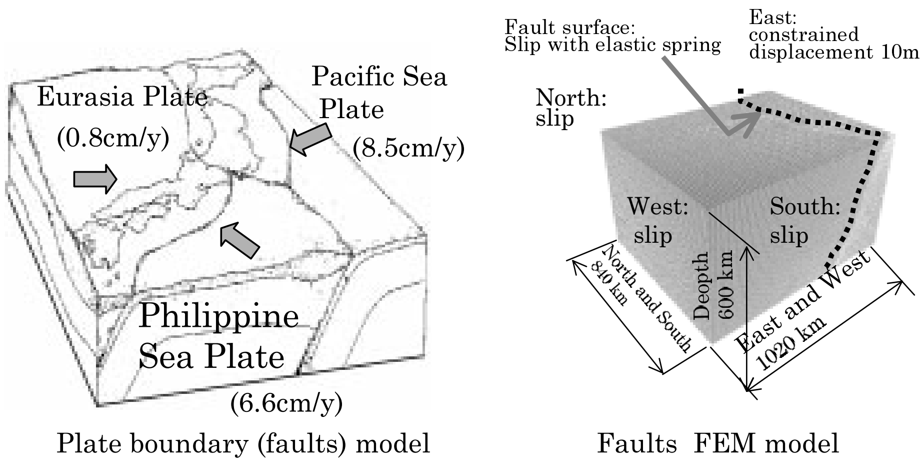

Figure 7: Fault analysis around Japan islands.

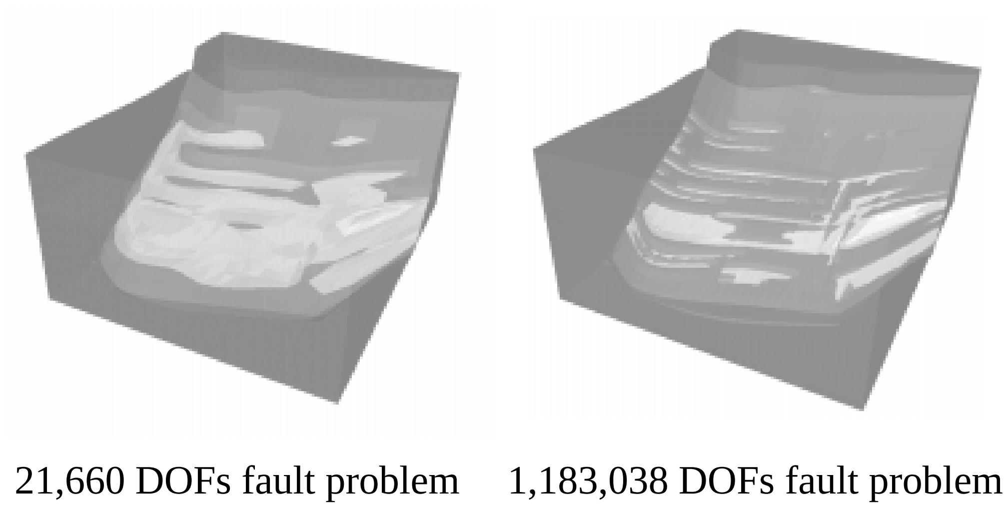

Figure 8: Normal contact force contours at fault surface.

Parallel coupling analysis platform

The GeoFEM coupler supports the communication between the modules. The coupler was developed to support a parallel/distributed environment, which is assumed by the GeoFEM programs. To use the coupler, the two GeoFEM modules transmit a "send the results data to another module" and "receive the data from another module" message to the coupler and the coupler determines what data send/receive 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 platform functions.

Assumptions

The following assumptions are made in the design of the coupler program, (Figure 10).

- The analysis spaces are partitioned into multiple regions for calculation in the data parallel manner on the distributed memory environment.
- The mesh data that 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.

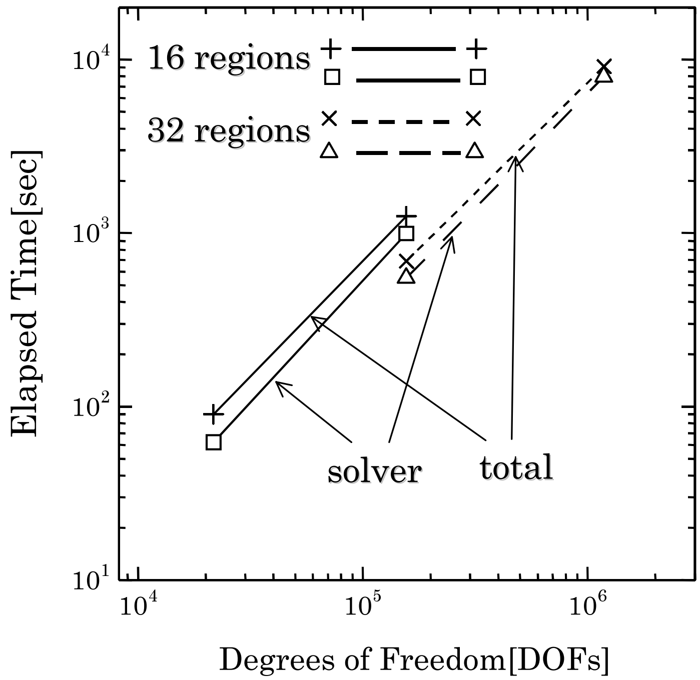

Figure 9: Relation between degrees of freedom and elapsed time.

Supported features

The features that are supported by the GeoFEM coupler program are as follows:

Analysis phase: The coupler system analyzes the multiple mesh sets and identifies the spatial relation "a node in one mesh set is involved in which element of the other mesh set".

Send/receive phase: According to the previous relations, the coupler determines which node data must be sent to the other PE and received from the the other PE. After receiving the node data, the coupler implicitly interpolates to calculate node data from nearby nodes from other PEs.

Note that the analysis phase is only needed when the spatial relation between meshes is changed, even though the send/receive phase is needed on multiple occations. The analyisi phase is a time consuming process, because the nodes and elements which are contained in both mesh sets are large, and the "spatial search" must be performed many

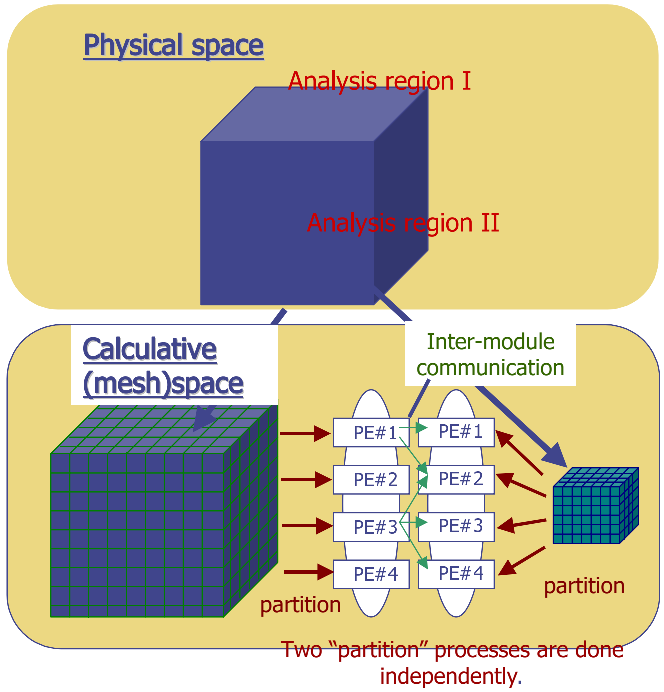

Figure 10: GeoFEM coupler.

times to examine the spatial relationship between two node sets. Therefore the analysis and the send/receive phases can be performed separately. Furthermore, considering the most simple (but ordinary) cases, nodes do not move and the spatial relation between nodes and elements dose not change. For such a simple case, it is advantageous for the analysis phase to be implemented in the pre-process manner (i.e. mesh generator/mesh partitioner like), and hence the user can omit the time consuming phase for the same mesh set.

Implementation

According to the considerations described in the previous section, the GeoFEM coupler system is implemented as follows:

Executional program(Figure 11): The analysis phase is implemented using the Xmesh (cross-mesh) independent executional program. The program reads the multiple (currently two) sets of the already partitioned mesh files and analyze the spatial relations between them. It then generates the separate files for each PEs, that contain the following information:

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

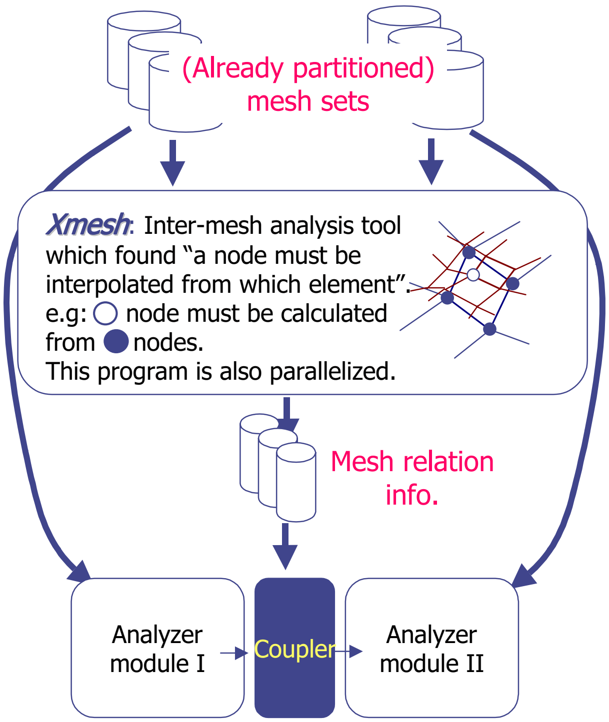

Figure 11: Implementation of GeoFEM coupler.

This xmesh program is a parallel program, because the mesh data size of GeoFEM must be assumed very huge for one PE's memory and the searching process to identify which node correspond to which element is very time consuming in one PE. Therefore parallel implementation is highly suited for the implementation process.

Communication library: The communication library is prepared to implement the send/receive phase. Two APIs are used to realize this function, i.e. "put data to the other module" and "get data from the other module". When calling the "get" subroutine, the calling program also passes the interpolation function using the "passing procedure" feature in Fortran 90. Therefore, the module programmer can design and implement appropriate interpolation method.

Example of Coupling Analysis by GeoFEM coupler

The parallel coupling system is currently under development but initial result of the static and zooming analysis has been observed. The problem was HTTR Carbon Block (CB) analysis(Figure 12) which is an engineering problem. GeoFEM is designed for the analysis of the solid earth phenomena but engineering problems provide a convenient method for the verification of the coupling analysis.

Figure 13 and Figure 14 shows the example of parallel coupling system for CB static and zooming analysis and the interface of GeoFEM coupling system. Figure 12 shows the result of a parallel coupling CB static and zooming analysis. The result shows that 1 PE result is identical to a 2 PEs result.

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 involve complex phenomena and are defined by the dynamic properties of the fault. Observations made during laboratory experiments can be extrapolated using numerical simulations to fault behaviour. Hence, numerical simulations provide an indication of the scalability of the laboratory results and provide a technique for improving understanding of how micro-scale processes in a gouge layer affect the macroscopic behavior of a fault zone. The interface between the software system developed at QUAKES (LSMearth), and the GeoFEM finite-element based software system will enable the 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

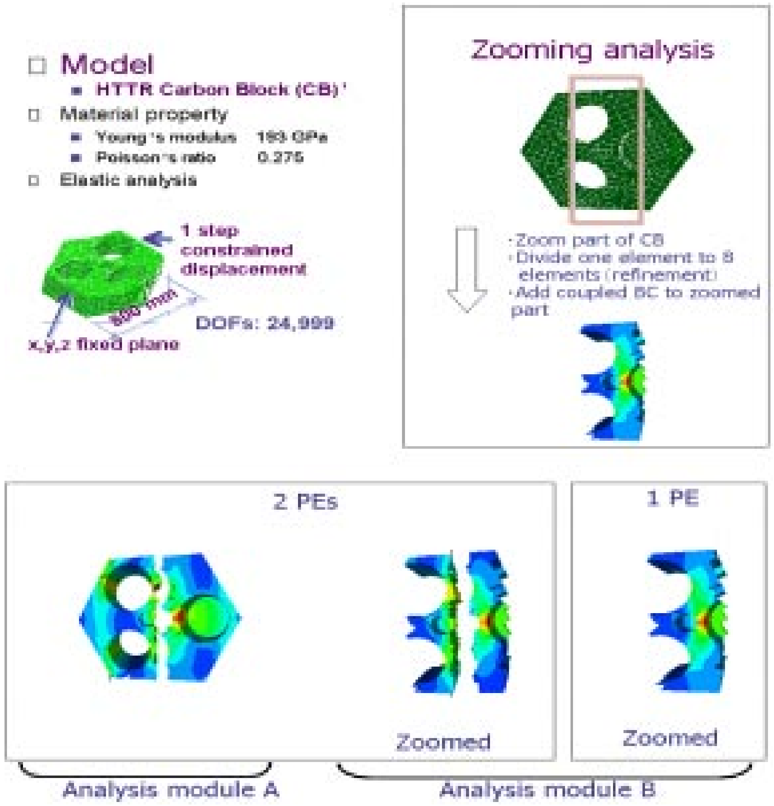

Figure 12: Example of GeoFEM parallel coupling with zooming method.

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 for a 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

Physical interface The exchange of physical values between the two models is performed 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.

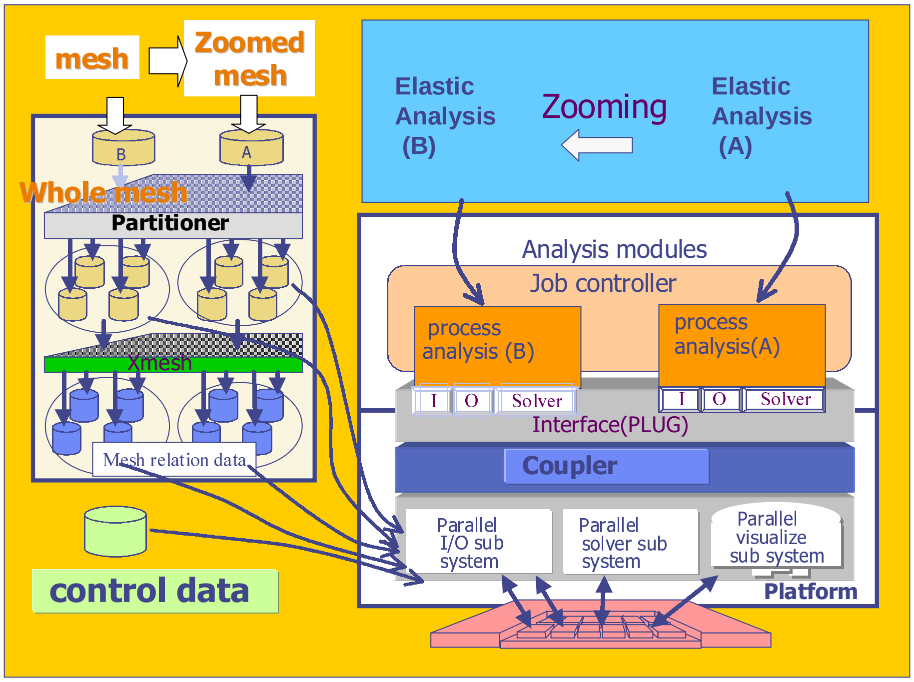

Figure 13: Implementation of GeoFEM coupling system.

Scaling An identical scale was used for both models during preliminary experiments. Hence, each node along the fault zone boundaries correspond to a particle in LSMearth. Exchange of data is performed between these nodes and particles. However, interpolation will be required between the particles and the nodes along the fault zone boundaries when using different scales are used. This will allow to use a much smaller scale for the LSMearth model than for the GeoFEM mesh. Therefor, simulation of the micro-physical process occuring at the rock grain scale will be enabled with LSMearth while macroscopic phenomena (such as elastic deformation and wave propagation) will be simulated using GeoFEM.

Transfer of forces and deformations Displacements occurring in the LSMearth model are input at the nodes of the GeoFEM mesh (Figure 15) when using different scale for GeoFEM and LSMearth to transfer forces and deformation between the models. From the displacements, deformations occur in the GeoFEM mesh and the traction forces can then be input in the LSMearth model by applying the force to the corresponding particle.

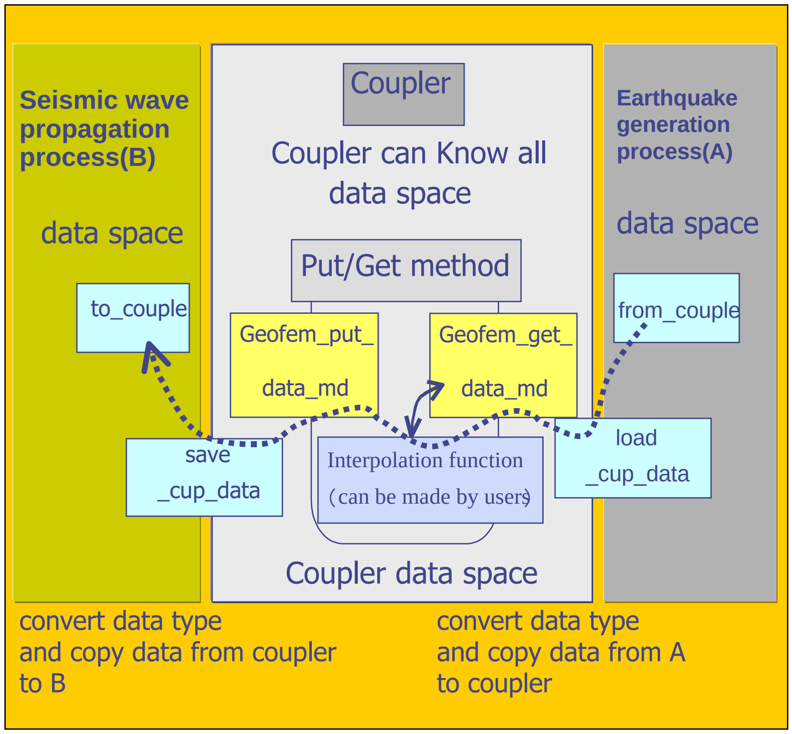

Figure 14: Interface of GeoFEM coupling system.

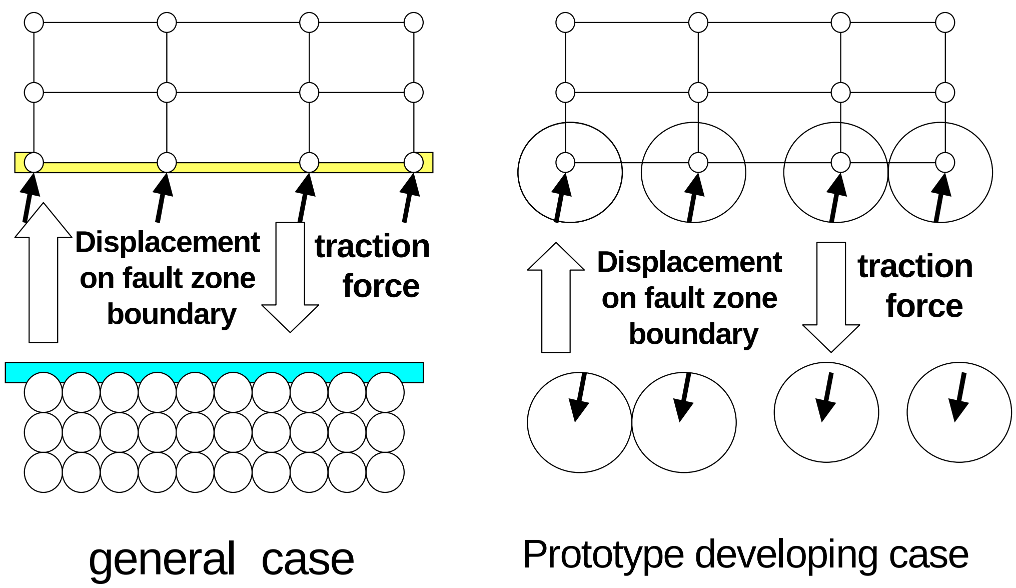

Figure 15: Physical interface

Implementation

The implementation of the hybrid model involves developing a GeoFEM main program(job controller) that controls the time evolution and calls for the two models. Exchange of data is controlled by the job controller and is performed using the 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.

Job controller The time evolution of the hybrid model is controlled by the job controller, which was written in Fortran90. The function of the job controller is (1) to initialize the models and coupler and (2) to perform the GeoFEM and LSMearth time loop. 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
```

Coupler The coupler (Figure 16) is the only module that has access to both the GeoFEM and LSMearth data spaces. To combine the two data spaces, a copy of the model data is placed in the coupler using only the save and load subroutines (saveGtoC, loadCtoG). These subroutines only have access to the coupler data space. Model data can be transfered and interpolateed from one model to the other using the get and put subroutines. These subroutines only have access to the coupler data space.

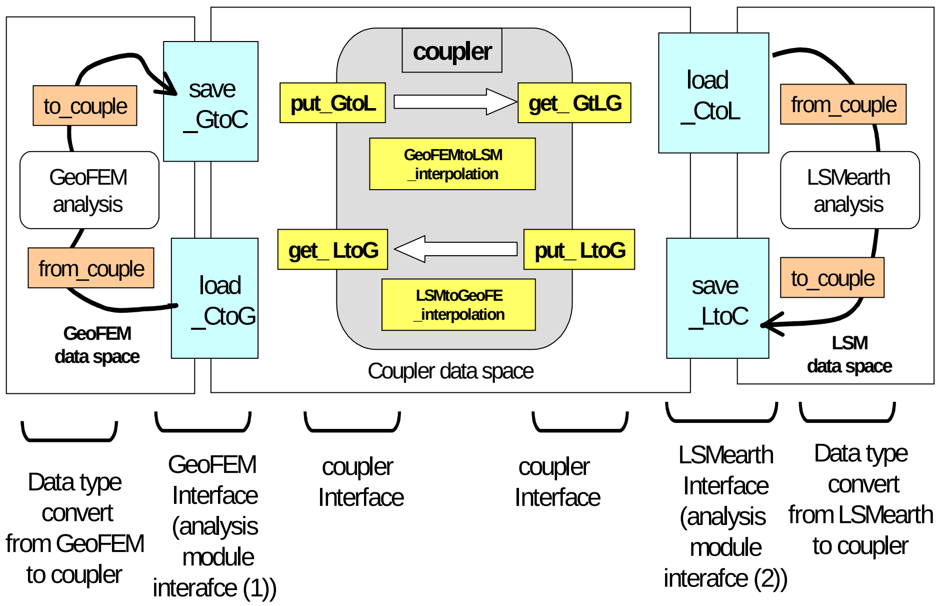

Figure 16: Coupler system configuration

GeoFEM and the LSMearth interfaces Because the job controller is based on the fault analysis module and LSMearth is written in C++, the LSMearth data and subroutines cannot be accessed directly. Hence, a C-interface is required to export LSMearth data and subroutines. Furthermore, to maintain the modularity of LSMearth, a module in LSMearth, termed the GeoFEM data exchange module, is created from which the two subroutines called loadCtoL and saveLtoC (Figure 16) can access the LSMearth data or subroutines.

LSMearth C-interface

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

Conclusion of this section

The implementation of the interface involved the development of a job controller, coupler and GeoFEM data exchange module in LSMearth. The coupler interface will ultimately operate through a message-passing interface, thereby allowing for the use of a different supercomputer for each model. This work is now under development. The interface between GeoFEM and LSMearth will allow for multi-scale simulations of large-scale fault systems and earthquakes dynamics in future(Figure 17).

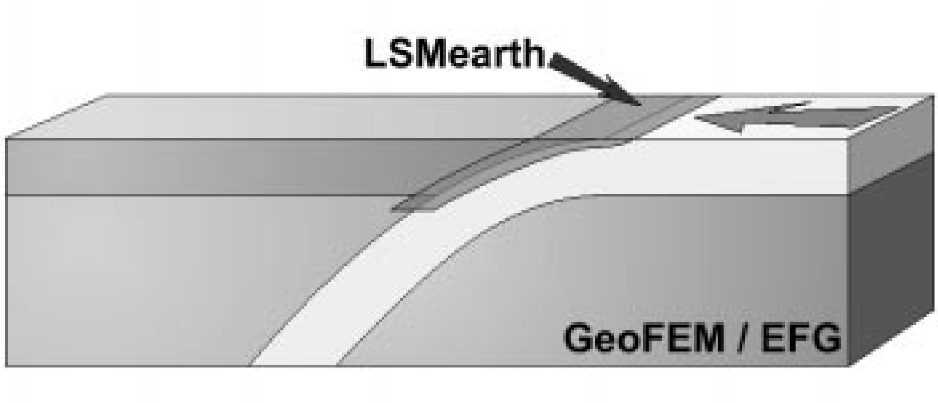

Figure 17: This figure shows the long-term aim of this project. An earthquake simulation in a subduction zone where LSMearth would be used to simulate phenomena occurring in fault gouges at the interface between the plate boundary and GeoFEM would be used to simulate wave propagations, elastic deformations and stress transfer.

CONCLUSIONS

This paper has presented some recent developments in GeoFEM for the simulation of earthquake generation and cycles. The main researches are as follows:

(1) Kinematic Earthquake Cycle Analysis for Large-scale Parallel Fault Dynamics

(2) Contact Analysis for Large-scale Parallel Fault Dynamics

(3) Parallel coupling analysis platform

(4) LSMearth and GeoFEM Coupling Analysis

The simulation of large-scale earthquakes is currently under development and the above-mentioned analysis modules will be integrated into the work.

Acknowledgments

The authors would like to thank Professor G.Yagwa from University of Tokyo, who is the supervisor of the "GeoFEM" project and all the members of the GeoFEM team for their helpful discussion and advice.

References

[1] Iizuka, M. , Garatani, K., Nakajima, K., Nakamura, H., Okuda, H. , and Yagawa, G. (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] Rundle, J.B., Henyey, T., Minster, J., and Fox, G. (1999), *General earthquake models* , 1-st ACES Workshop Proceedings, ed Mora, P., (The APEC Cooperation for Earthquake Simulation, Brisbane, Australia), 281-287.

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

[5] Sivathasan.K., Paulino, G.G., Li, X.S. and Arulanandan, K. (1998), *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).

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

[13] Nakajima, K., Okuda, H.(1998),

Parallel iterative solvers with localized ILU preconditioning for unstructured grids on workstation Cluster,

4th Japan-US Symposium on FEM in Large-Scale Computational Fluid Dynamics Proceedings, 25-30.

[14] 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.

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