FEAP - - A Finite Element Analysis Program

Version 8.5 Multiscale User Manual

Robert L. Taylor & Sanjay Govindjee
Department of Civil and Environmental Engineering
University of California at Berkeley
Berkeley, California 94720-1710

October 2017

Contents

Chapter 1

INTRODUCTION

This manual serves as a supplement to the *FEAP* User manual[1] available at the web site `www.ce.berkeley.edu/feap`.

The multiscale FE^2 module for *FEAP* requires the final build to be completed using access to MPI send and receive commands. The program will be executed using multiple processors (or cores). To use the multiscale program with *openmpi* the final compilation must be performed using the OpenMPI libraries. Alternatively, any MPI build may be used. This build is performed after first compiling a serial version of *FEAP*. There is a `makefile` in directory `ver85` and in the subdirectory `fe2` to assist in the compilation.

Multi-scale analysis is implemented as a two-scale method in which the main problem is described according to the descriptions in the *FEAP* User Manual. The only exception is for material behavior to be described by a finer scale analysis using a *representative volume* (RVE) formulation. For these materials the material data input consists of

```
MATErial ma
  SOLId
    RVE,<filename>
```

for finite deformation problems or

```
MATErial ma
  SOLId
    SRVE,<filename>
```

for small deformation problems. Multiple RVE types may be described by specifying different `filename` parameters to the `RVE/SRVE` record.

N.B. Problem solutions tested to date include: Plane strain; Axisymmetric; Three dimensional solids.

The main problem input file must start with `I` or `i` followed by a name. For example, the main file `Iblk3` describes the main problem to be solved. For each processor used a file `Iblk3.01`, `Iblk3.02`, etc. must be prepared. Each of these files contains the simple structure:

```
NOCOunt
UFEAP <* * optional title>

INCLUde solve_mpi

STOP
```

The RVE file (name given to `filename`) is also a standard *FEAP* input file except that the `feap` start record is omitted and all values on the control record must be specified (*i.e.*, values for nodes, elements, material sets, mesh dimension, dofs/node, and nodes/element). Thus the start of the file becomes:

```
numnp numel nummat ndm ndf nen

MATErial 1
  ect.

END ! mesh
ELINk
   ...
```

in which numerical values are given for `numnp`, etc. Periodic boundary conditions may be given by linking the edges in all directions of the RVE. Note that nodes must match on linked edges. Alternatively, the boundaries of the RVE may be fully restrained using `EBOUndary` commands. In this case it is not necessary for the boundary nodes to match on opposite faces. Finally, a full Taylor condition in which all nodes of the RVE are fixed may be specified as

```
TBOUndary
```

Once execution begins *FEAP* will compute how many calls to each of the different RVE's will be required and proportion the different processors to achieve a balance. At this time it is assumed each RVE will require about the same compute time to converge each solution step.

The INCLude solve_mpi in each RVE describes the solution steps that can be performed. At this time it is recommended that the solution for an RVE only be given as

```
batch
  mpi  start sw its ! Start RVE solution (sw = if parameter)
  loop,infinite     ! Permits infinite solution iterations
    mpi  get        ! Get F from macro solution: Input U, H
    if sw-12        ! Test for time step (isw = 12)
      noprint       ! Do nothing statement needed
    else sw-3       ! For all 3 < isw < 12
      loop          ! Iterate local solution (no number)
        tang,,1
      next          ! End local unit problem
    endif
    mpi  send       ! Send stresses to macro solution: Out U,H
  next              ! Next macro solution
end             ! End of batch execution file
```

The parameter its specifies the maximum number of iterations for the TANG command. At the start of each time step only one iteration is performed, thus, any inelastic material should return an elastic predictor in the first iteration. In subsequent iterations up to its iterations are performed for each RVE.

The solution steps for the main problem are given in a standard manner (batch execution is recommended) except that the first command must always be

```
RVE <POINt num>
```

to allocate the files to each processor. The option POINt specifies a quadrature point in the main problem for which specified deformation gradients (or strains) along with returned RVE stress and moduli are written to files (currently unit 90, 91, 92, respectively). These may be used to run a serial *FEAP* analysis using the PERIodic mesh command along with the HILL solutions commands. The results for the local stress and displacement contours may then be displayed for this point.

Bibliography

[1] R.L. Taylor. *FEAP - A Finite Element Analysis Program, User Manual*. University of California, Berkeley. `projects.ce.berkeley.edu/feap`.