The University of Iowa
College of Engineering
53:243/58:255 Computational Inelasticity
Fall Semester 2005
Instructor: C.C. Swan

Assignment #4: Due: 28 October 2005

The objective of this assignment is to implement the Backward Euler integration algorithm and consistent tangent operator for the elastoplasticity model treated in Assignment #3. Your numerical implementation will be written in subroutine which you will compile and link together with a research finite element program FENDAC. Details on compiling and linking are provided below and a manual for the program FENDAC can be found at
`engineering.uiowa.edu/~swan/software/fendac.pdf`

For those who can possibly do so, it will be easiest if you write your material model subroutine in Fortran. (For those who program only in C or some other language, please see me to make special arrangements.)

a. As the starting point, you are given a Fortran material model subroutine mod16d2.f. When you are finished modifying this routine, it will compute updated stresses and the appropriate material tangent operator. At present, this model does only linear elasticity, so you will need to modify it quite extensively to implement your elastoplasticity model.

b. Please implement both a continuum tangent operator for your model, and a consistent tangent operator. When the logical variable passed into your module `ltan=.true.` and when `itan=0`, you should compute and return the continuum tangent operator. Alternatively, when `ltan = .true.` and `itan=1` compute and return the consistent tangent operator. This will allow you to easily compare the performance characteristics of the two tangent operators.

c. There are a number of test problems which you will find in the class directory on the ECSS system `(/usr/ui/class/examples/cee5330/hw4)`. Three are simple material test type computations, and the fourth is a more complicated boundary value problem. When you run the first three, make sure that the computed answers obtained with the program FENDAC are correct.

d. In addition to correctness of solutions, for all problems verify that your consistent tangent operator gives asymptotically quadratic rates of convergence in the nonlinear global iterations of the FEM code. Also, compare the rates of convergence achieved with the continuum and consistent tangent operators.

Supporting details:

The relevant class directory for this assignment is on the ECSS HP-UX system and is: /usr/ui/class/examples/cee5330/hw4 and hereafter, this directory is simply referred to as the hw4 directory.

In this directory you will find:

- A file named mod16d2.f. This is the file in which you will need to implement your material model.
- A file named makefile. Once you've written a version of mod16d2.f that you think should work, type makefile and the fortran compiler will try to compile your subroutine(s) and then link them together with the remainder of the FEM program FENDAC.
- Three data files named: uni.data, shear.data and bcap.data. You will use these FENDAC data files as test problems which your material model should solve.

Additional Information for Assignment #4

1.) From the ``hw4'' directory, copy over the file setup53243 and execute it by typing source setup53243 <Enter>. Each time you start a new login session to continue work on this assignment, you will need to retype this command. By executing this setup script, you allow yourself to use a number of executables and libraries in the hw4 directory without needing to make copies.
2.) To compile your subroutines in the file mod16d2.f and link them with FENDAC, you'll need to copy the file makefile from the hw4 directory. Each time you want or need to recompile your routines simply type make <Enter> and your routines in mod16d2.f will be recompiled and linked with FENDAC, creating a new version of the executable FENDAC.HP.
3.) Before you can test your subroutine(s), you must have successful compilation, with no fatal errors.
4.) To actually run FENDAC.HP, type fhp-m <Enter> .When prompted, enter a value of 1 to continue execution. When prompted for the name of the data set, type for example uni <Enter> to make FENDAC.HP run the problem described in the data file uni.data. When FENDAC.HP is finished executing, there will be a file uni.results and a number of TAPE**.uni files in the directory you are working in. These files enable you to:
    i. view the undeformed and deformed FEM meshes for the problem you're analyzing;
    ii. view element time histories of various stress and strain components;
    iii. view time histories of various nodal displacements.
5.) To view the FEM mesh for a specific problem that you've already runs successfully, type, for example, ropostg uni <Enter> to view the mesh associated with the uni data set. The display program will then ask you a few questions about your display preferences and then show the mesh. [For the problems of this assignment, always answer 0 or No to the question on display of volume fractions.] To advance the display to the next view, click on the right mousebutton. After the last available display, you'll exit from the ropostg program.
6.) For the uni and shear data sets, which are really simulations of material property tests, you will want to display the element time histories saved in TAPE86.***. To create plots after running the uni data set, type ethplt < TAPE86.uni <Enter>. The program ethplt

will then create two files `graph1.ps` and `graph2.ps` in your working directory. It might also send these to your default printer as well. To view these plots on the screen of your terminal, use the utility `ghostview` or any other suitable postscript viewer.

7.) For the `bcap` data set, you are computing the bearing capacity of an elasto-plastic half-space. Displacements beneath the applied load are saved in `TAPE88.bcap`. Loading is applied to the half-space at a rate that increases uniformly in time. To view the displacements beneath the load, type the command `nnthplt < TAPE88.bcap <Enter>` and this will create a new `graph1.ps` file in your directory. This file can also be viewed using `ghostview`.