# EMA 605   PROJECT #1

Use the finite element method to compute the elastic modulus and Poisson ratio of the poly-crystalline material shown. Each crystal has bar finite elements connecting its centroid to the centroids of contacting neighbors. Use EA = 23.5 kN for all bars. Nodal coordinate and element connectivity data is located in a file entitled "ProjectData" on the course web page.

FE model with 25 nodes and 42 bar elements
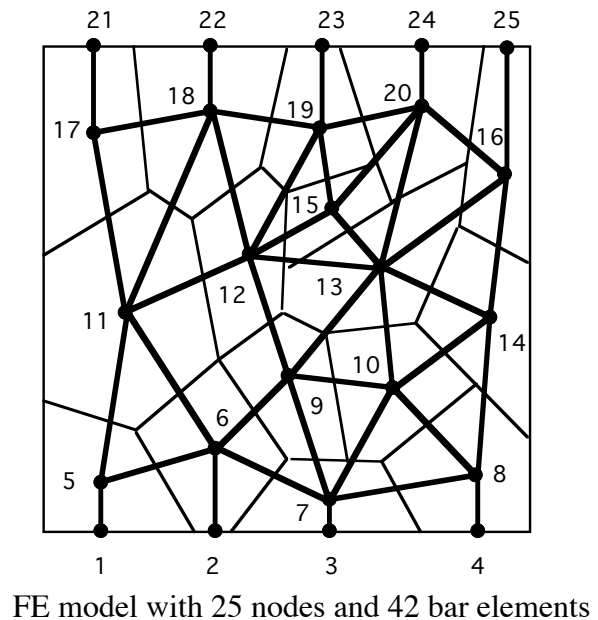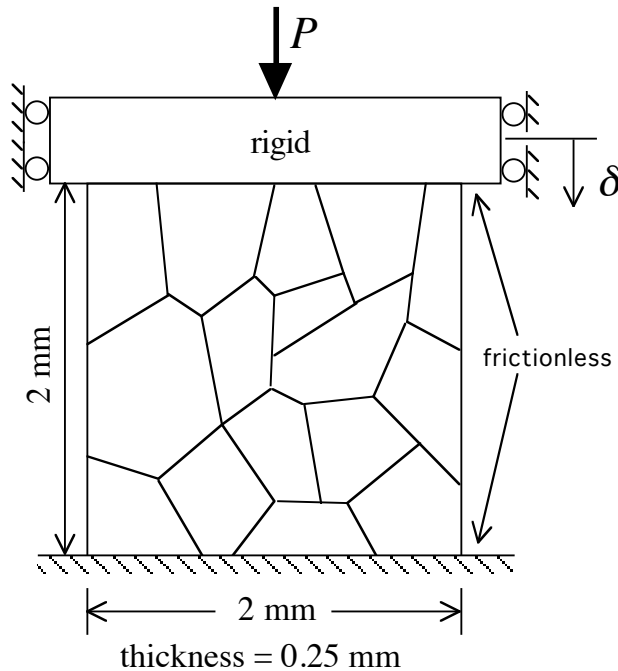
You should write an easy-to-read report (a few pages should be satisfactory) that has a brief discussion of how you modeled the problem (e.g., BCs and loading) and how you determined the modulus of elasticity and Poisson ratio. Your report should include your finite element input and output in an Appendix (if you choose Option #2, then you should also include a listing of your program in an Appendix). Your report should be well-written and it should convince the reader that your FEA results are accurate. It is recommended that you also include the results of a simple test problem (e.g., the 2-bar problem analyzed in class - the data for this is given in the file "TwoBarData" on the course web page).

Grading is as follows

50% Does your code work and do you get correct answers?
25% Did you accurately model the problem (BCs and loading)?
25% Is your report well-written.

Option No. 1 (This is the easier option, and is worth 15% of your course grade)

Compile and execute FEMCOD for the problem given above. Program FEMCOD, the user guide and other useful files are located on the course web page listed below. If you need help using the UNIX Fortran compiler, the consultants at CAE should be able to get you started. Once FEMCOD is operational, run a simple test problem. Then model the problem described above.

Option No. 2 (This is the more challenging option, and is worth 20% of your course grade)

Using the language of your choice, write your own FE code for bars in 2-D (it is fine to discuss the project with your peers, but you are expected to do your programming independently). Your code should have the capability of treating up to 50 nodes, 100 elements, and 5 material sets. It does not need to allow for non-zero prescribed displacements.   In your code, store the entire stiffness matrix. For element generation, you can use subroutine BARSTF (located within FEMCOD), and for element assembly use coding similar to Fig. 2.7-4 of CMP 3e. For equation solving, use the simple Gaussian elimination subroutine in Fig. B.2-2 of CMP 3e (but make sure the row dimension matches that in the main program). Before analyzing the materials problem shown above, you should run your program on a simple test problem. For your analysis of the materials problem, you may replace the prescribed displacement loading conditions shown above by a uniform compressive stress.

NOTE: For Option No. 1, you are free to use the machine of your choice, but you must have access to a Fortran compiler. CAE supports Fortran compilers for Linux machines only. For Option No. 2, you are free to use the language and machine of your choice.

Assigned: Tuesday, September 22, 2009
Due: Tuesday, October 27, 2009
-10% for up to one week late

---

course web page ....

http://www.engr.wisc.edu/ep/ema/courses/ema605.html
click on the link for Fall09