Term Project

A basic finite element program (FEB.FOR) was developed to demonstrate application of the finite element method in engineering. The program consists of modules and each module can be modified for specific purposes by the user. The program uses separator for data file preparation.
The program FEB.FOR is for linear static analysis of the structures. It uses 4-node isoparametric element which is suitable for 2-D analysis in plane elasticity such as concrete gravity dams. The program can take into account effect of hydrostatic pressure for inclusion of dam reservoir pressure.
Manual of NSAG-DRI was prepared for the advanced version of the program which is intended for nonlinear dynamic analysis of concrete gravity dams. The manual can be used for FEB.For program knowing that only linear part of the program is functioning.
You are asked to introduce 8-node isoparametric element (developed in the class note) into the FEB.FOR. The program should be able to use both 4-node and 8-node elements simultaneously. This can be done by assigning a code to each element.
To verify that program, you can use Pine Flat dam for the purposes of the static analysis.

1) Make two different finite element meshes of the pine flat dam.
2) Perform static analysis of the dam including hydrostatic and self weight loadings for 8-node elements using the two finite element meshes.
3) Compare the results of the two meshes and choose an appropriate mesh out of results of part 2.
4) Perform static analysis of dam including hydrostatic and self weight loading for 4-node elements using the best mesh obtained in part (3) and compare the results with those of 8-node element.
5) How would you verify your finite element program of 8-node element if you did not have the results of the analysis of the 4-node element?

For results you need to show only a) dam crest displacement b) stress at each element centre c) stress contour of the dam.

Configuration of the tallest monolith of the Pine Flat concrete gravity dam is shown. The modulus of elasticity, unit weight and Poisson's ratio of the concrete were taken as 27,580 Mpa, 2400 kg/m^3 and 0.2. The foundation rock is assumed to be rigid and therefore the nodes of finite elements are fixed at the contact points.

Make a finite element model of the dam structure and analyse the model considering hydrostatic pressure and self weight loads.