APPENDIX D

The Finite Element Personal Computer Program

With permission of the estate of Dr. Charles E. Knight, the Finite Element Personal Computer (FEPC) program is available to users of this text via the website www.mhhe.com/hutton<www.mhhe.com/hutton>. FEPC is a finite element software package supporting bar, beam, plane solid, and axisymmetric solid elements and hence is limited to two-dimensional structural applications. Dr. Knight's *A Finite Element Method Primer for Mechanical Design* is available via the website and includes basic concepts as well as an appendix delineating FEPC capabilities and limitations. The following material presents a general description of the programs' capabilities and limitations. A complete users guide is available on the website.

FEPC is actually a set of three programs that perform the operations of preprocessing (model development), model solution, and postprocessing (results analysis). FEPCIP is the input processor used to input and check a model and prepare data files for the solution program FEPC. The output processor is FEPCOP and this program reads solution output files and produces graphic displays.

All the programs are menu driven, with automatic branching to submenus when appropriate. The Files menu of the input processor is used to recall a previously stored model or store a new model. Models are stored as *filename*.MOD where filename is user specified and can contain a maximum of 20 characters. The analysis file, which becomes the actual input to the FEPC solution phase, is *filename*.ANA.

D.1 PREPROCESSING

Model definition is activated by the Model Data menu. Selection of Model Data leads to a submenu used to define element type, material properties, nodes, elements, restraints (displacement constraints), and loads. Element type is limited to bar (truss), beam, plane stress, plane strain, or axisymmetric. Only a single element type can be used in a model. Up to 10 material property sets can be used in a model and should be defined in numerical order. Nodes can be defined by direct input of node number and the *X*, *Y* coordinates of the node. Nodes for truss and beam elements are always defined in this manner. Nodes for plane and axisymmetric elements can also be defined by direct input but an automeshing capability for two-dimensional (2-D) areas is included and discussed subsequently. Similarly, elements are defined by specification of the nodes and material property number. For truss and beam elements, the order of node specification is of no consequence. However, for plane or axisymmetric elements, the nodes must be specified in a counterclockwise order around the element area. Displacement constraints are applied by setting the values and selecting the node to which the values apply. Loads are applied as nodal forces or element edge pressures for 2-D solid elements.

The 2-D Automesh Generation section of the program is used for area mesh generation of two-dimensional plane stress, plane strain, or axisymmetric models. The approach used is a coordinate transformation mapping of a grid of square elements in an integer area into a grid of elements in the geometric area. The geometric area is defined using points that are subsequently used to define lines and arcs that enclose the area. The integer grid is bounded by lines that correspond to the lines and arcs of the geometric area. Grid points bound the square elements in the integer area map to element nodes in the geometric area. The mapping process is iterative and consists of distorting the integer area grid to fit into the geometric area. Thus the user has some control over element size and shape via the size of the integer grid.

D.2 SOLUTION

When a model has been defined and saved as *filename*.ANA, the solution is generated by the FEPC.EXE program. During execution, several other files are generated and stored. Also, screen messages are issued to report on progress of the solution phase. A listing of all printed output is stored in *filename*.LST. This file contains the input data, all numerical output, and any error messages issued during execution. Two additional files are created for use by the output processor. The node and element data is placed in *filename*.MSH and nodal displacement and element stress data is stored in *filename*.NVL.

D.3 POSTPROCESSING

The output processor FEPCOP.EXE is used to display the solution results in graphic form. (The printed form of the numerical output data is in the .LST file.) Displacement results can be displayed as a plot of the deformed element mesh superimposed over a plot of the undeformed model. Displacements are scaled such that the deformed shape is exaggerated for clarity.

For two-dimensional solid models, stress components can be displayed as contour plots. The stress components available from the solution are the normal, shear, and von Mises stress components for plane stress and plane strain, and the radial, axial, shear, and hoop stress components for axisymmetric models. For models using truss or beam elements, stress components are plotted as bar charts.