Finite element analyses can generate a great deal of output, much of which is often not of interest. The **OUTPUT** commands collectively can be used to greatly restrict the amount of output to be printed. The user must, however, be aware that output which is not printed is lost and can only be regenerated by repeating the analysis. Thus if there is doubt as to whether some quantity will be of interest, it should be printed.

The purpose of the **OUTPUT ELEMENTS PRINT** commands is to eliminate unnecessary output generated by APES. More precisely, if the time history of stresses, strains, pore pressure, heat fluxes, etc. is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. In addition to the **OUTPUT ELEMENTS PRINT** commands, the user can control the type of element quantities printed (e.g., principal strains and/or stresses, average strains and/or stresses, etc.). This is done by specifying suitable keywords in conjunction with an **ELEMENT** command.

If the complete time history of nodal values of a primary dependent variable is desired only at a select few points, use of the **OUTPUT NODES PRINT** commands greatly speeds program output and facilitates inspection of the results by the user.