GENERAL INFORMATION

General Information

The APES\(^1\) computer program has been written to perform one-, two-, or three-dimensional quasi-static analyses of continua. For two-dimensional analyses, the solution domain can be idealized assuming conditions of plane strain, plane stress, or torsionless axisymmetry. Since the formulation used in the program takes into account the coupling between stress and flow, APES is particularly well-suited for the analysis of earth structures. It is important to note, however, that the program is written in such a manner as to avoid unnecessary computations when this coupling is not desired in an analysis. In short, the efficiency of the program is not compromised in such a case.

With the exception of lexing and parsing software developed elsewhere, the APES computer program is written in Fortran 95 \([2]\), thus making use of the advances offered by the language. In keeping with modern programming practice, the design of the program is quite modular and facilitates the incorporation of additional solution algorithms, equation solvers, element types and of constitutive models.

This documentation assumes general familiarity with the finite element method; novices in this area are referred to standard texts on the subject \(\text{e.g., [1, 7, 10, 14]}\).

Historical Note

APES grew out of the SAC-2 and SAC-3 computer programs. The original versions of these programs were written at the University of California Davis in 1983 \([3, 4, 5]\). A FORTRAN-77 version of the SAC-2 program, which improved the input and output options and which incorporated additional material models, followed \([6]\). This was followed by enhanced two- \([8]\) and three-dimensional \([9]\) versions of the program.

The present version of the APES computer program has the following features:

- The program is written in Fortran 95 \([2]\), thus taking advantage of features of the language which lead to more efficient code.

- To facilitate program input and output, a simple problem-oriented language (POL) approach \([11]\) is used in APES. The model generation phase of an APES analysis is sufficiently flexible so as to easily accept data generated by stand-alone mesh pre-processors (e.g., \([12]\)).

- An enlarged library of element types has been developed. This includes one-dimensional (bar and beam) elements, zero-thickness interface elements, and irreducible and mixed two- and three-dimensional continuum elements.

- The number of material idealizations has been increased.

- The highly efficient SKYLDU skyline equation solver \([13]\) for symmetric and/or non-symmetric systems has been implemented.

\(^1\)APES = Analysis Program for Earth Structures.
• Greater flexibility in displaying the results of analyses has also been incorporated into the program.
References


