ANALYSIS CLINIC

What can go wrong in FEA?

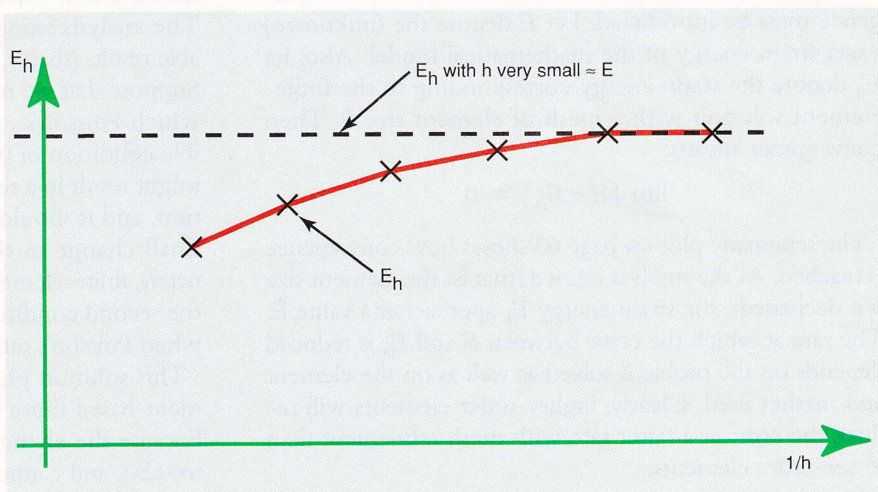
As finite-element analysis spreads to designers who may lack formal training in numerical procedures, practitioners must ask whether the most appropriate techniques are being used—and whether they are producing accurate results. **By Klaus-Jürgen Bathe**

FINITE-ELEMENT METHODS are in abundant use in today's engineering practice through various general-purpose commercial computer programs and many special-purpose programs written for specific applications. These techniques are, to an increasing extent, being used to help identify good new designs and improve designs with respect to performance and cost.

Considering the important role that finite-element methods now play in various areas of engineering, practitioners need to ask themselves whether their procedures are the most appropriate techniques available and whether the methods will lead to accurate results. These questions are particularly important because more and more design engineers who have not necessarily been trained in numerical procedures are applying finite-element techniques in their work.

As the use of these methods expands to a larger and more diverse group, users must address the important question of what can go wrong in finite-element analysis.

Klaus-Jürgen Bathe is professor of mechanical engineering at the Massachusetts Institute of Technology in Cambridge. He is also the founder and a director of ADINA R&D Inc. in Watertown, Mass.



A schematic plot of strain energies E_h of finite-element solutions is given when the element size h is decreased. The plot shows convergence as the mesh is refined.

This article is not intended to resolve the question in the broadest sense; rather, we shall focus on some aspects of the reliability of finite-element methods and their accurate use. To illustrate, we will consider linear elastic solutions and assume that the algebraic finite-element equations are solved exactly. For a more complex analysis, the same considerations hold, but additional requirements need to be addressed as well.

Mathematical Models

First of all, engineers should recall that the finite-element method is used to solve a mathematical model, which is the result of an idealization of the actual physical problem considered. The mathematical model is based on assumptions made regarding the geometry, material conditions, loading, and displacement boundary conditions. The governing equations of the mathematical model are in general partial differential equations subjected to boundary conditions. These equations cannot be solved in closed analytical form. Therefore, engineers resort to the finite-element method to obtain a numerical solution.

Consider, for example, an analysis of a valve housing of axisymmetric geometry and axisymmetric loading. In such a case, it is reasonable to assume axisymmetric conditions for analysis. The complete mathematical model and thus the analysis problem is obtained by specifying the geometry and dimensions, support conditions, material constants, and loading.

Finite-element formulations still in use could yield grossly erroneous solutions.

While engineers cannot, in general, obtain analytically the exact solution for the posed mathematical model in closed form, the exact solution of the mathematical model does exist, the solution is unique, and an approximation of this exact solution can be obtained with very high accuracy using finite-element methods.

To quantify these observations, the notion of convergence must be introduced. Let E denote the (unknown) exact strain energy of the mathematical model. Also, let E_h denote the strain energy corresponding to the finite-element solution with a mesh of element size h. Then convergence means:

$$\lim_{h \to 0} |E - E_h| = 0$$

The schematic plot on page 63 shows how convergence is reached. As the mesh is refined (that is, the element size h is decreased), the strain energy E_h approaches a value E. The rate at which the error between E and E_h is reduced depends on the problem solved as well as on the element and meshes used. Clearly, higher-order elements will reduce the error at a faster rate with mesh refinement than lower-order elements.

The Question of Reliability

Reliability in finite-element methods means that in the solution of a well-posed mathematical model, the finite-element procedures will have two attributes: The finite-element solutions will converge to the exact solution of the mathematical model as h approaches 0 for any (physically realistic) material data, displacement boundary conditions, and loading applied; and for a reasonable finite-element mesh, a reasonable finite-element solution will be obtained. Furthermore, the quality of the finite-element solution does not change drastically when the material data (or thickness of a shell) are changed.

These conditions are of crucial importance. If the first condition (convergence) is violated, then with mesh refinement a solution is approached that is not the exact solution of the mathematical model. Such erroneous solutions could lead to wrong design decisions and disastrous consequences. Of course, finite-element methods that violate the first condition should not be used.

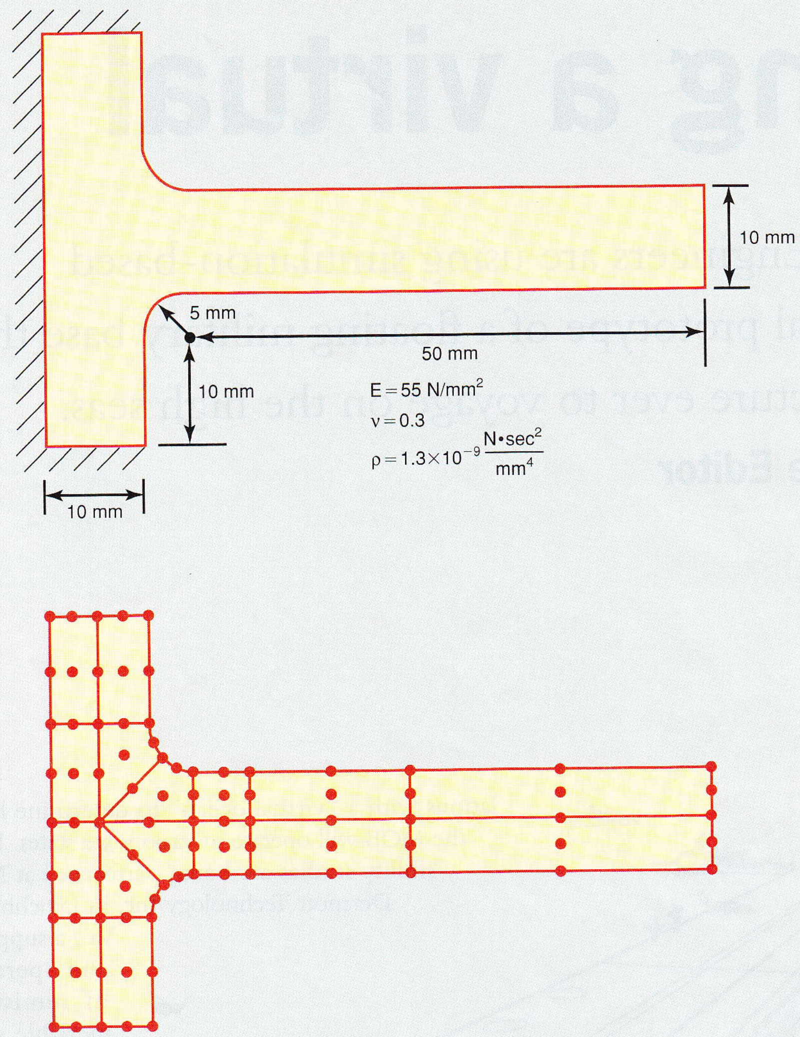
To consider the second condition—obtaining a reasonable solution using a reasonable mesh—assume, for example, that the valve housing is made of steel (Young's modulus is 200,000 megapascals; Poisson's ratio, ν, is 0.30). The analysis using a reasonable mesh has given acceptable results (that is, the error $|E - E_h|$ is acceptably small). Suppose that we now change the material to a plastic, for which Poisson's ratio is 0.49—close to the incompressible condition of 0.50. This change in material condition might result in a relatively small change in the exact solution, and it should then result only in the corresponding small change in the finite-element solution. Unfortunately, finite-element formulations are still used that violate the second condition and yield a solution grossly in error when Poisson's ratio is changed to 0.49.

This solution phenomenon is observed with displacement-based finite elements. The large errors are present because the elements are much too stiff when ν is close to 0.50, and cannot be used when $\nu = 0.50$. The underlying mechanical reason is that, considering the stresses, $p = Ke_v$, where p is pressure, K is bulk modulus, and e_v is volumetric strain. As ν approaches 0.50, K becomes very large and is infinite when $\nu = 0.50$. Also, as ν approaches 0.50, in the exact solution, e_v becomes very small and is zero when $\nu = 0.50$. Therefore, in an almost incompressible analysis, the pressure is given by a very large number (the bulk modulus) multiplied by a very small number (the volumetric strain), and must be accurately cal-

	16-element model		16×64 element model
Mode	3×3 Gauss integration	2×2 Gauss integration	3×3 Gauss integration
1	112.4	110.5	110.6
2	634.5	617.8	606.4
3	906.9	905.5	905.2
4	1,548	958.4[a]	1,441
5	2,654	1,528	2,345
6	2,691	2,602	2,664

[a]Spurious, i.e., phantom mode.

For the six smallest frequencies (in hertz) of the bracket presented on page 65, the consistent mass matrix was used. The results using a fine mesh (with 64 elements replacing each nine-node element of the 16-element mesh) are included for comparison purposes.



The geometry and material data of a bracket used in an analysis are shown on top; the 16-element mesh of nine-node elements used in the analysis is given on the bottom.

culated to balance the applied surface tractions. While the bulk modulus becomes very large, the pressure is always a finite number and usually does not change much as ν approaches 0.50.

As a remedy in displacement-based finite-element methods, "reduced integration" is employed. This means that in the numerical integration of the element stiffness matrices, the exact matrices are not evaluated. The method is simple to program and requires less computation time to establish the matrices, and with experience acceptable results are frequently obtained. However, the technique can also lead to very large errors.

Consider the frequency analysis of the bracket detailed in the table on page 64 and the figure on page 65. In this case, using nine-node elements, 3×3 Gauss integration corresponds to full numerical integration and 2×2 Gauss integration is "reduced integration." Since no analytical closed-form solution exists for the frequencies of the bracket, a very fine mesh (the 16×64 element model) was used to obtain an accurate solution. Of interest are the solutions obtained with the coarse 16-element mesh. As predicted by theory, using full integration, the frequencies of the coarse 16-element mesh are larger than the accurate solution frequencies. When reduced integration is used, some frequencies are better approximated than when using full integration, but among those few listed is a phantom frequency. Phantom frequencies do not physically exist but are only introduced by the reduced-integration scheme. If a dynamic step-by-step solution is performed, such phantom frequencies are not noticed and absorb energy, introducing large errors into the solution. Error measures would detect that the errors are large, but they are frequently not available in dynamic analysis. Thus, reduced integration is unreliable and should be avoided.

Instead, only reliable mixed finite-element formulations, which do not require reduced integration, should be used. Such formulations satisfy the first and second conditions, are currently available, and can be employed confidently for the solution of mathematical models with any material properties, loading, and boundary conditions. A displacement/pressure-based formulation is particularly attractive when incompressible materials are considered and was used to analyze a plastic bracket with Poisson's ratio equal to 0.499 (see Analysis Clinic, March). The formulation is also used effectively in nonlinear analysis, where almost incompressible conditions are widely encountered. Inelastic conditions of plasticity and creep, and rubberlike conditions, give rise to (almost) incompressible behavior.

Similar difficulties, as described for almost-incompressible analysis, are also encountered in the analysis of plate and shell structures. Here, too, some finite-element technology still uses reduced integration, and the results can be very much in error. Again, reliable formulations, which are now available, should be used instead of such reduced-integration schemes.

To sum up, finite-element methods can now be employed with great confidence, but only the methods considered reliable should be used. Earlier technology based on reduced integration should not be used, or should at any rate be employed with great care. By proceeding in this way, practitioners can have confidence that a finite-element analysis will be effective and will not go wrong. ▪

The material presented in this article is treated at further length in Finite Element Procedures *by Klaus-Jürgen Bathe (Prentice Hall, 1996).*