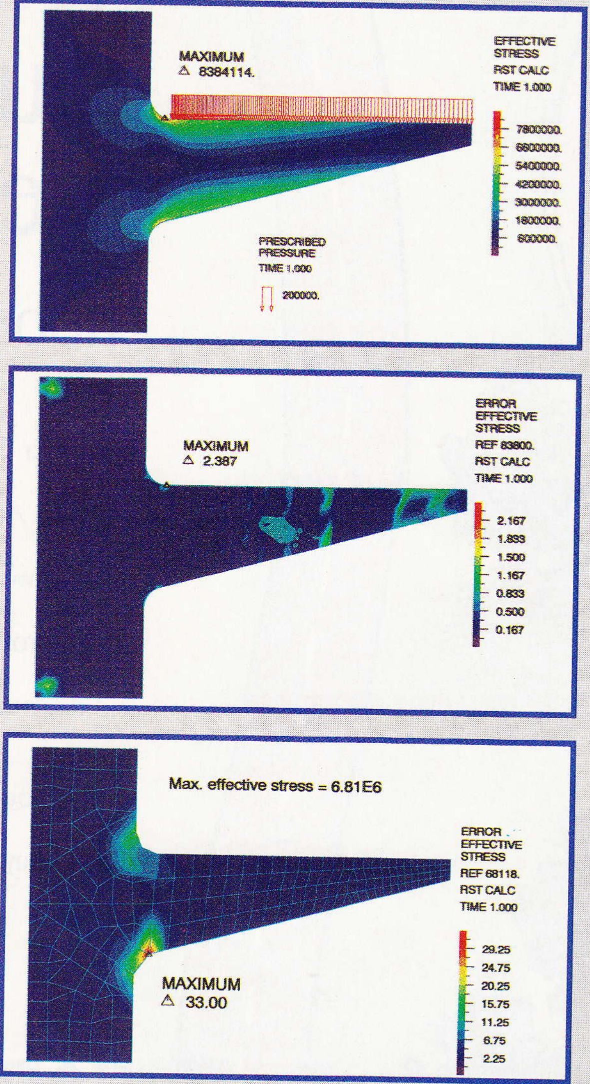
analysis CLINIC

How Automatic Is Automatic Meshing?

Analysis Clinic, which premieres with this issue, will offer tips and tools to help practicing engineers use computer-aided-engineering tools more effectively. Among the topics to be covered are finite-element analysis, computational fluid dynamics, plastic-injection-molding simulations, and preprocessing and postprocessing.

The most time-consuming step of finite-element analysis is creating a finite-element model. For this reason, FEA programs generally include software tools that help the analyst establish the finite-element model as effectively as possible. Graphical user interfaces are a common part of this process, but the most difficult and lengthy aspect of data preparation is usually generating the finite-element mesh.

Naturally, engineers prefer to use software tools that enable them to automatically create such meshes with little effort on their part. Such tools are available but they are not foolproof: Certain aspects of mesh generation must be given particular attention to avoid pitfalls and solution difficulties.



In this analysis of a steel bracket (with a Poisson's ratio of 0.30) with the ADINA system, a fine-mesh solution produces a maximum effective stress of 8.38 megapascals with approximate maximum error of 2 percent (as shown in the top two figures). A coarse-mesh solution yields a maximum effective stress of 6.81 megapascals with an approximate maximum error of 33 percent (bottom). This coarse mesh was constructed using a minimum size element for representing the geometry, which resulted in small elements at the tip but not in the fillet, where the high stresses are encountered.

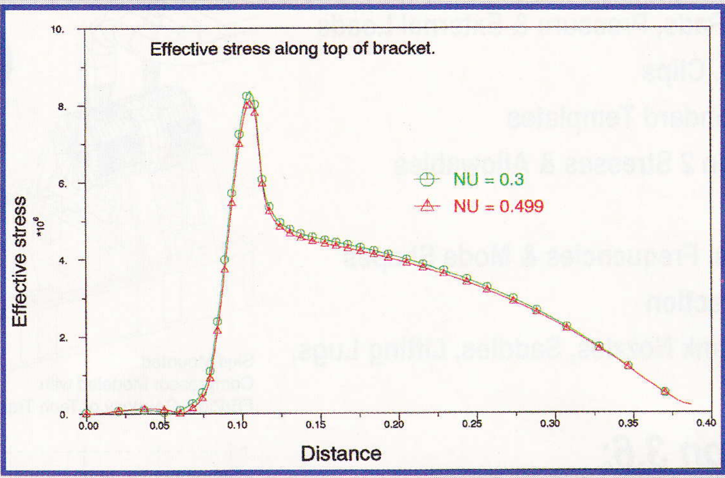
When considering the analysis process, it's useful to remember that a finite-element code solves a mathematical model, which is an idealization of a structure or system in the real world. The mathematical model includes assumptions about the geometry and displacement conditions considered; the boundary conditions (which represent the rest of the universe), including the loading; the stress-strain material conditions; the possibility that large deformations can take place; and so on. Of course, this model is established according to the analysis questions that are asked by the analyst; approximately predicting the displacements at a point, for example, may only require a beam model, whereas accurately predicting the stresses in critical areas usually requires that a more complex model be solved.

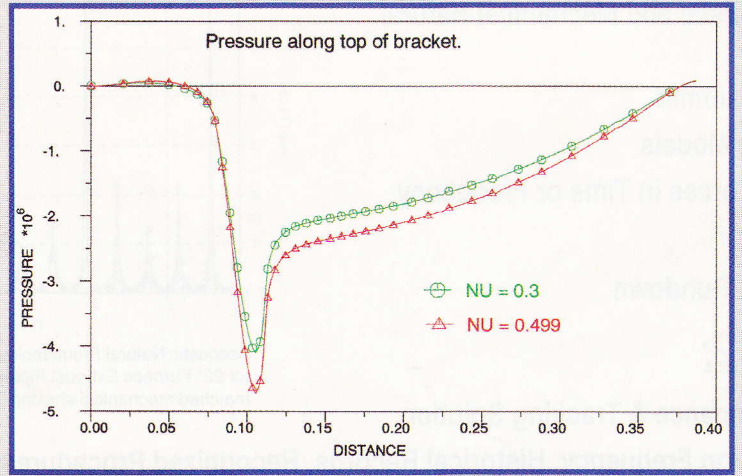
In all cases, however, the finite-element model can only predict what is contained in the mathematical model, and the finite-element analysis should solve this mathematical model to a sufficient degree of accuracy. The use of an adequate finite-element mesh is crucial to this task, but the mathematical model must also have been chosen without creating artificial analysis difficulties.

In practice, the most attractive way to analyze and thus mesh a part is to use the computer-aided-design geometry created with a solid modeler such as Unigraphics, I-DEAS, SolidWorks, and CATIA, but this geometry is frequently much too complex for analysis. For a reasonably effective analysis, the geometry must be simplified by deleting features (such as holes away from the area of interest) that will not significantly affect the desired results. This "defeaturing" process should result in a geometric entity that, when subjected to the loading and boundary conditions, will define a relatively easy-to-solve mathematical model. While this process of constructing the mathematical model appears to be a straightforward task in principle, the actual effort can be demanding, and good engineering judgment in arriving at an appropriate model is required.

Klaus-Jürgen Bathe is a professor of mechanical engineering at the Massachusetts Institute of Technology in Cambridge, where he teaches courses on mechanics and finite-element methods. He is also the founder and a director of ADINA R&D Inc. in Watertown, Mass.

Once the mathematical model is available, the next step in the analysis process is to create the mesh. Various considerations are important at this stage. An automatic finite-element mesher—one for which no element sizes and grading of element sizes are prescribed—generally will only work on the geometry and fill up the complete geometry with elements. The element sizes will be automatically determined from the geometry features, and will not be affected by the loading and boundary conditions. Such a mesh can result in quite poor analysis results because element sizes may be much too large in areas of stress concentrations. The actual error of analysis depends on the mathematical model to be solved as well as the elements (low- or high-order elements) and mesh used.





For the plane-strain analysis of this bracket using the ADINA system and a fine mesh, changing the bracket's Poisson ratio from 0.3 to 0.499 also produces an accurate solution.

A more rational way to proceed is for the engineer to prescribe reasonable element sizes for the mesh generator in crucial areas of the analysis so that the mesh is determined not only by the features of the geometry but also by the physical phenomena to be predicted—in other words, the mechanical behavior of the structure.

In this regard, sharp corners and concentrated loads are idealizations that result in infinite stresses when considering fully three-dimensional situations. Thus, mesh refinements in these areas only make sense when they are modeled to approximate quite closely the actual physical situations—the rounded "corners" and the small areas over which a distributed pressure is applied.

To address these considerations, the analyst needs to have available in the program a means to assess the error in the stress solution. Various error measures are available and are in use. In essence, they all evaluate the degree to which the governing differential equations of equilibrium are violated (the exact solution satisfies these equations exactly). If the error of the finite-element solution is too large, as assessed by the engineer, a refined mesh needs to be constructed, by using more elements (the h method), increasing the polynomial orders in the elements (the p method), or combining both methods; then, the analysis is repeated using this mesh.

At this point, there is yet another important point to be taken into account: When using a given mesh with certain material properties for a problem, analysts should use reliable and effective finite elements that uniformly perform well independent of the material properties used. Then, when considering a steel bracket and obtaining a good stress prediction with a given mesh, the same mesh will also result in a good stress prediction when the bracket is made of a plastic with a Poisson's ratio close to 0.50 (an almost incompressible material).

Only such (reliable) finite elements should be employed; otherwise, the analyst has to unduly experiment with meshes depending on the material properties used. Indeed, in the case of an almost incompressible material, certain widely used finite elements perform very poorly compared with the use of reliable and effective mixed elements. (A future feature article will address this aspect of analysis in more detail.)

Therefore, to use an automatic mesh generator successfully, engineers should keep at least three things in mind. First, analysts should not simply trust an automatic mesher to mesh an analysis part, since the program might only use the geometry features to span a mesh. Second, only reliable and effective elements should be used; moreover, their predictive capabilities should be independent of the material properties, loading, and boundary conditions. Finally, the analyst should obtain from the program that he or she uses a measure of the error of the finite-element solution as compared to the exact solution of the mathematical model.

For more information on avoiding the pitfalls and solution difficulties when using automatic meshers, see sections 1.2, 1.3, 4.3, and 4.4 of Finite Element Procedures *by Klaus-Jürgen Bathe (Prentice Hall, 1996).*