ANALYSIS CLINIC

Current Directions in Meshing

The preparation of finite-element models for analysis is being increasingly automated, with new developments expected to greatly advance the field.

By Klaus-Jürgen Bathe

THE KEY TO AN EFFICIENT finite-element analysis (FEA) is frequently an effective finite-element mesh. The construction of this mesh is the first step in the analysis process. While guidance is available introducing nonspecialists to basic issues regarding the use and application of meshing tools (see Analysis Clinic, March), there is a need for further discussion of more-advanced meshing issues and for comment on the current state of meshing procedures. Such techniques are widely available and are broadly used in many solution packages for finite-element analysis of solids, structures, and fluids, as well as packages for finite-difference and control-volume analysis of fluids.

There are two broad types of mesh-generation methods—schemes for structured and unstructured meshes. The techniques for obtaining structured grids are based on rules for the grid-subdividing of geometries and mapping techniques. Solving partial differential equations is frequently an effective way to establish the grading.

Structured grids, as is implied by the name, have a clear structure. The techniques used to generate them produce quadrilateral or triangular cells, or elements, in two-dimensional analysis, and tetrahedral or hexahedral elements in three-dimensional simulations. Commercial finite-element programs offer options for structured-grid generation for solid mechanics and fluid-flow calculations, and aerospace companies use this meshing approach in their programs for the solution of the Euler and Navier-Stokes equations. Grid points can be distributed along lines with effective spacing, and well-graded grids can be constructed. The approach is very effective when relatively simple geometries are considered, like the stress analysis of a turbine blade or fluid flow over an airplane wing. However, this approach is also used in mesh generation for complex geometries using multiblock methods, in which the complete geometry is considered to be an assemblage of blocks. Each block is to be equivalent topologically to a cube and meshed. The major difficulty is to set up the connections between the blocks for complex geometries and varying grid-point spacing.

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

Techniques for unstructured grids are more novel and are extensively used in finite-element analysis, both for structures and fluids. As the name implies, a mesh obtained using an unstructured-grid-generating scheme does not show structure in the placement of the elements. The methods for unstructured grid generation have received intensive research and development attention because they can mesh any complex geometry, using triangular elements for two-dimensional analysis and tetrahedral elements for three-dimensional analysis.

Unstructured Grids

There are two major approaches to generating unstructured grids: the advancing-front method and the Delaunay triangulation. Considering three-dimensional analysis, the starting point in each scheme is a surface triangulation. In the advancing-front method, this triangulation is known as the initial front, and tetrahedral elements are created on each triangle. This is achieved by creating new points in the interior of the domain. The current front always consists of the exposed element faces in the domain. The front is advanced either by establishing new points or by using existing points to create new elements. In this way, the complete domain is filled with tetrahedra. Grid-point spacing is controlled by sources placed in the domain or by specifying a background mesh.

The Delaunay method is quite different. All boundary points of the surface triangulation are used to form what is at first a rather coarse triangulation of the complete domain. Then, additional points are inserted in the domain by a refinement strategy. The key to efficiency is the point-insertion procedure.

The major difference between the two approaches is that in the Delaunay method, a rather coarse domain triangulation is created using boundary points into which further points are inserted, while in the advancing-front procedure points are created inside an empty domain.

When using meshing schemes, it is important to be able to generate anisotropic meshes. For thin structures, elements must be used that are thin in one direction and long in the other two. In fluid flows, boundary layers need to be modeled with very thin elements aligned with the boundary.

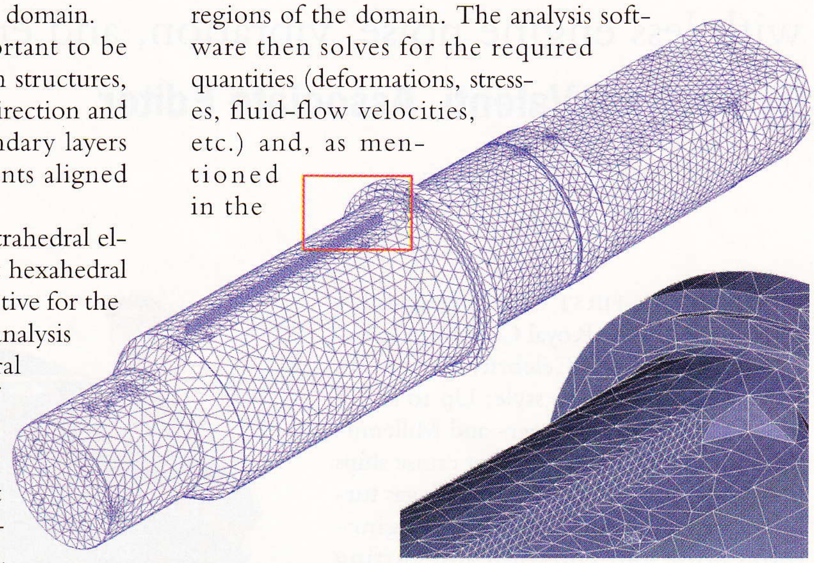
While free-form meshing schemes using tetrahedral elements are in wide use, it is well known that hexahedral elements would, in many cases, be more effective for the analysis. This holds specifically in structural analysis problems. Unfortunately, an effective general unstructured mesh-generation scheme of hexahedral elements for complex geometries does not yet exist. What can of course be achieved is to generate tetrahedra and split each into hexahedral elements. But these elements are then too distorted to be effective. Also, layers of hexahedral elements can be generated from the surface of the geometric domain, as far as possible, to finally fill up the remaining space with tetrahedra. An important consideration is that higher-order hexahedral elements are relatively insensitive to mesh distortions and can be employed with high aspect ratios to model thin structures. Therefore, general and effective schemes for unstructured hexahedral meshing that can accommodate grading would be very useful.

An increasingly common application of finite-element methods is seen in computer-aided design (CAD). The geometry is built in a CAD system, such as CATIA, Pro/ENGINEER, or Unigraphics, and the meshing scheme works directly on this geometric representation. In such cases, the geometry needs to be built only once, and the structural or fluid-flow system can be analyzed directly. For example, the figure on this page shows geometry created within Pro/E and meshed directly in ADINA with its free-form mesher, based on the advancing-front method. Such a direct connection between CAD and FEA systems is very useful in product design, because the analysis can be part of the design cycle.

While it is a most elegant concept, the direct finite-element analysis of a CAD part is not without difficulties. Small features, such as holes, chamfers, and fillets, may require very small elements in the meshing and result in a large finite-element discretization—one that could well be too large for the hardware resources available. Hence, a defeaturing of certain geometric features, not important for the analysis results sought, is often necessary. In the analysis of the solid shown in the figure, such defeaturing was not performed, and sufficiently small-size elements were used to model all small features given in the solid.

Adaptive Meshing

Powerful software for meshing is already available, the current limits are well understood, and new algorithms and programs can be expected that will continuously advance the state of meshing capabilities. Using currently available software, the analyst has great flexibility in meshing, but must prescribe element sizes and grading in the different regions of the domain. The analysis software then solves for the required quantities (deformations, stresses, fluid-flow velocities, etc.) and, as mentioned in the earlier article on meshing, calculates error indicators (as far as available) for these results. If the error is too large, the analyst needs to remesh, that is, refine the discretization in certain areas, and perform further solutions.



This tetrahedral mesh was automatically created with ADINA for Pro/E part geometry.

An area of great potential is the automatic adaptation of the mesh, without intervention by the analyst, and automatic continued solution until the required accuracy has been reached. Such mesh adaptation generally means that in certain areas of the analysis domain the size of the elements is decreased (or increased) and the order of the elements may be increased (or decreased). In concept, such adaptation is most appealing, but there are difficulties when complex practical situations are considered.

These difficulties pertain to the use of appropriate error indicators and decision-making processes, and the implementation of the remeshing schemes that adapt the mesh effectively and stop the solution cycle when acceptable results have been reached. A broader approach, when considering this automation, should also include the selection of an effective mathematical model that does not contain undue solution difficulties such as artificial singularities. To effectively automate the choice of the mathematical model and its solution, major advances in theoretical issues and software development are still required and will provide an exciting challenge for the years to come. ■