The mesh generation

Describe general methods (structured, unstructured, hybrid, adaptive, etc.) and discuss their key features and applications

A key step of the finite element method for numerical computation is mesh generation. One is given a domain (such as a polygon or polyhedron; more realistic versions of the problem allow curved domain boundaries) and must partition it into simple " elements" meeting in well-defined ways. There should be few elements, but some portions of the domain may need small elements so that the computation is more accurate there. All elements should be " well shaped" (which means different things in different situations, but generally involves bounds on the angles or aspect ratio of the elements). One distinguishes " structured" and " unstructured" meshes by the way the elements meet; a structured mesh is one in which the elements have the topology of a regular grid. Structured meshes are typically easier to compute with (saving a constant factor in runtime) but may require more elements or worse-shaped elements. Unstructured meshes are often computed using quadtrees, or by Delaunay triangulation of point sets; however there are quite varied approaches for selecting the points to be triangulated

The simplest algorithms directly compute nodal placement from some given function. These algorithms are referred to as algebraic algorithms. Many of the algorithms for the generation of structured meshes are descendents of " numerical grid generation" algorithms, in which a differential equation is solved to determine the nodal placement of the grid. In many cases, the system solved is an elliptic system, so these methods are often referred to as elliptic methods.

It is difficult make general statements about unstructured mesh generation algorithms because the most prominent methods are very different in nature. The most popular family of algorithms is those based upon Delaunay triangulation, but other methods, such as quadtree/octree approaches are also used.

Delaunay Methods

Many of the commonly used unstructured mesh generation techniques are based upon the properties of the Delaunay triangulation and its dual, the Voronoi diagram. Given a set of points in a plane, a Delaunay triangulation of these points is the set of triangles such that no point is inside the circumcircle of a triangle. The triangulation is unique if no three points are on the same line and no four points are on the same circle. A similar definition holds for higher dimensions, with tetrahedral replacing triangles in 3D.

Quadtree/Octree Methods

Mesh adaptation, often referred to as Adaptive Mesh Refinement (AMR), refers to the modification of an existing mesh so as to accurately capture flow features. Generally, the goal of these modifications is to improve resolution of flow features without excessive increase in computational effort. We shall discuss in brief on some of the concepts important in mesh adaptation.

Mesh adaptation strategies can usually be classified as one of three general types: r-refinement, h-refinement, or p-refinement. Combinations of these are also possible, for example hp-refinement and hr-refinement. We summarise these types of refinement below.

r-refinement is the modification of mesh resolution without changing the number of nodes or cells present in a mesh or the connectivity of a mesh. The increase in resolution is made by moving the grid points into regions of activity, which results in a greater clustering of points in those regions. The movement of the nodes can be controlled in various ways. On common technique is to treat the mesh as if it is an elastic solid and solve a system equations (suject to some forcing) that deforms the original mesh. Care must be taken, however, that no problems due to excessive grid skewness arise.

h-refinement is the modification of mesh resolution by changing the mesh connectivity. Depending upon the technique used, this may not result in a change in the overall number of grid cells or grid points. The simplest strategy for this type of refinement subdivides cells, while more complex procedures may insert or remove nodes (or cells) to change the overall mesh topology.

In the subdivision case, every “ parent cell” is divided into “ child cells”. The choice of which cells are to be divided is addressed below. For every parent cell, a new point is added on each face. For 2-D quadrilaterals, a new point is added at the cell centroid also. On joining these points, we get 4 new “ child cells”. Thus, every quad parent gives rise to four new offsprings. The advantage of such a procedure is that the overall mesh topology remains the same (with the child cells taking the place of the parent cell in the connectivity arrangement). The subdivision process is similar for a triangular parent cell, as shown below. It is easy to see that the subdivision process increases both the number of points and the number of cells

A very popular tool in Finite Element Modelling (FEM) rather than in Finite Volume Modelling (FVM), it achieves increased resolution by increasing the order of accuracy of the polynomial in each element (or cell).

In AMR, the selction of “ parent cells” to be divided is made on the basis of regions where there is appreciable flow activity. It is well known that in compressible flows, the major features would include Shocks, Boundary Layers and Shear Layers, Vortex flows, Mach Stem , Expansion fans and the like. It can also be seen that each feature has some “ physical signature” that can be numerically exploited. For eg. shocks always involve a density/pressure jump and can be detected by their gradients, whereas boundary layers are always associated with rotationality and hence can be dtected using curl of velocity. In compressible flows, the velocity divergence, which is a measure of compressiblity is also a good choice for shocks and expansions. These sensing paramters which can indicate regions of flow where there are activity are referred to as ERROR INDICATORS and are very popular in AMR for CFD.

Just as refinement is possible by ERROR INDICATORS as mentioned above, certain other issues also assume relevance. Error Indicators do detect regions for refinement, they do not actually tell if the resolution is good enough at any given time. In fact the issue is very severe for shocks, the smaller the cell, the higher the gradient and the indicator would keep on picking the region, unless a threshold value is provided. Further, many users make use of conservative values while refining a domain and generally end up in refining more than the essential portion of the grid, though not the complete domain. These refined regions are unneccesary and are in strictest sense, contribute to unneccesary computational effort. It is at this juncture, that reliable and resonable measure of cell error become necessary to do the process of “ coarsening”, which would reduce the above-said unnecessary refinement, with a view towards generatin an “ optimal mesh”. The measures are given by sensors referred to as ERROR ESTIMATORS, literature on which is in abandunce in FEM, though these are very rare in FVM.

Control of the refinement and/or coarsening via the error indicators is often undertaken by using either the ‘ solution gradient’ or ‘ soultion curvature’. Hence the refinement variable coupled with the refinement method and its limits all need to be considered when applying mesh adaptation

A hybrid model contains two or more subsurface layers of hexahedral elements. Tetrahedral elements fill the interior. The transition between subsurface hexahedral and interior tetrahedral elements is made using degenerate hexahedral (pyramid) elements.

High quality stress results demand high quality elements, i. e., aspect ratios and internal angles as close to 1: 1 and 90°, respectively, as possible. High quality elements are particularly important at the surface. To accommodate features within a component, the quality of elements at the surface of a hexahedral model generally suffers, e. g., they are skewed. Mating components, when node-to-node contact is desired, can also adversely affect the models element quality. Even more difficult is producing a tetrahedral model that contains high quality subsurface elements. In a hybrid model, the hexahedral elements are only affected by the surface mesh, so creating high quality elements is easy.

Minimal effort is required to convert CAD data into surface grids using the automated processes of pro-surf. These surface grids are read by pro-am. The surface grid is used to extrude the subsurface hexahedral elements. The thickness of each extruded element is controlled so that high quality elements are generated. The interior is filled automatically with tetrahedral elements. The pyramid elements that make the transition are also generated automatically.

A hybrid model will generally contain many more elements than an all-hexahedral model thus increasing analysis run-time. However, the time saved in the model construction phase – the more labor intensive phase – more than makes up for the increased run-time. Overall project time is reduced considerably. Also, as computing power increases, this “ disadvantage” will eventually disappear.

Hexahedral Meshing

ANSYS Meshing provides multiple methods to generate a pure hex or hex dominant mesh. Depending on the model complexity, desired mesh quality and type, and how much time a user is able to spend meshing, a user has a scalable solution to generate a quick automatic hex or hex dominant mesh, or a highly controlled hex mesh for optimal solution efficiency and accuracy.

Mesh Methods:

Automated Sweep meshing

- Sweepable bodies are automatically detected and meshed with hex mesh when possible

- Edge increment assignment and side matching/mapping is done automatically
- Sweep paths found automatically for all regions/bodies in a multibody part
- Defined inflation is swept through connected swept bodies
- User can add sizing controls, mapped controls , and select source faces to modify and take control over the automated sweeping
- Adding/Modifying geometry slices/decomposition to the model also greatly aids in the automation of getting a pure hex mesh.

Thin Solid Sweep meshing

- This mesh method quickly generates a hex mesh for thin solid parts that have multiple faces as source and target.
- Can be used in conjunction with other mesh methods
- User can add sizing controls, mapped controls, and select source faces to modify and take control over the automated sweeping
- MultiZone Sweep meshing

- This advanced sweeping approach uses automated topology decomposition behind the scenes to attempt to automatically create a pure hex or mostly hex mesh on complicated geometries
- Decomposed topology is meshed with a mapped mesh or a swept mesh if possible. A user has the option to allow for free mesh in sub-topologies that can't be mapped or swept.
- Supports multiple source/target selection
- Defined inflation is swept through connected swept bodies

- User can add sizing controls, mapped controls and select source faces to modify and take control over the automated meshing
- Hex-dominant meshing
- This mesh method uses an unstructured meshing approach to generate a quad dominant surface mesh and then fill it with a hex dominant mesh
- This approach generally gives nice hex elements on the boundary of a chunky part with a hybrid hex, prism, pyramid, test mesh internally

Tetrahedral Meshing

The combination of robust and automated surface, inflation and tet meshing using default physics controls to ensure a high-quality mesh suitable for the defined simulation allows for push-button meshing. Local control for sizing, matching, mapping, virtual topology, pinch and other controls provide additional flexibility, if needed.

Mesh Methods:

Patch conforming mesh method:

- Bottom-up approach (creates surface mesh, then volume mesh)
- Multiple triangular surface meshing algorithms are employed behind the scenes to ensure a high quality surface mesh is generated, the first time
- From that inflation layers can be grown using several techniques
- The remaining volume is meshed with a Delaunay-Advancing Front approach which combines the speed of a Delaunay approach with the smooth-transitioned mesh of an advancing front approach

- Throughout this meshing process are advanced size functions that maintain control over the refinement, smoothness and quality of the mesh
- Patch independent mesh method:
- Top-down approach (creates volume mesh and extracts surface mesh from boundaries)
- Many common problems with meshing occur from bad geometry, if the bad geometry is used as the basis to create the surface mesh, the mesh will often be bad (bad quality, connectivity, etc.)
- The patch independent method uses the geometry only to associate the boundary faces of the mesh to the regions of interest thereby ignoring gaps, overlaps and other issues that give other meshing tools countless problems.
- Inflation is done as a post step into the volume mesh. Since the volume mesh already exists, collisions and other common problems for inflation are known ahead of time.

Note: For volume meshing, a tetrahedral mesh generally provides a more automatic solution with the ability to add mesh controls to improve the accuracy in critical regions. On the contrary, a hexahedral mesh generally provides a more accurate solution, but is more difficult to generate.

Shell and Beam Meshing

For 2-D planar (axisymmetric), shell and beam models, ANSYS Meshing provides efficient tools for quickly generating a high quality mesh to accurately simplify the physics.

Mesh Methods for shell models:

Default surface meshing

- Multiple surface meshing engines are used behind the scenes to provide a robust, automated surface mesh consisting of all quad, quad dominant or all tri surface mesh.
- User can add sizing controls, and mapped controls to modify and take control over the automated meshing
- Uniform surface meshing
- Orthogonal, uniform meshing algorithm that attempts to force an all quad or quad dominant surface mesh that ignores small features to provide optimum control over the edge length

Describe key features of ALL existing meshing options in Ansys Mesh module and discuss their applications

The meshing tools in ANSYS Workbench were designed to follow some guiding principles:

- Parametric: Parameters drive system
- Persistent: Model updates passed through system
- Highly-automated: Baseline simulation w/limited input
- Flexible: Able to add additional control w/out complicating the workflow
- Physics aware: Key off physics to automate modelling and simulation throughout system
- Adaptive architecture: Open system that can be tied to a customer's process

CAD neutral, meshing neutral, solver neutral, etc.

By integrating best in class meshing technology into a simulation driven workflow, ANSYS Meshing provides a next generation meshing solution.