Panel Sessions at COMPUMAG 'Rio

The material collected in the following is a synthesis of the presentations of a panel session held at Compumag Rio last November. In this Conference two panel sessions were organized, and both were followed attentively by conference participants, were praised by attendees as timely and interesting and generated very lively scientific discussions for the whole duration of the Conference. It has consequently been felt that the whole community of the ICS could benefit from the inclusion in the Newsletter of a summary of the presentations of both sessions (the second one will appear in the next issue). The synthesis of the Mesh Generation Panel Session presented here, kindly prepared by the authors under the coordination of their Session Panel Chair, is also supplemented by a significant set of references that might be appreciated also by the members of the Society which attended Compumag Rio.

Mesh generation is certainly a key issue in continuum physics simulations, and particularly so in three dimensional electromagnetics, which presents several specific features and difficulties (multiregion, multiply-connected domains, dynamic range of significant dimensions, open boundaries, etc.) that set it apart from other disciplines in this context. In spite of these difficulties, automatic meshing is essential to simplify preprocessing and make electromagnetic analysis simpler, faster and suitable for insertion in automatic loops for design parameterization, sensitivity analysis and design optimization, so that the pressure to develop reliable automatic meshing procedures is certainly very strong. Of course, automatic meshing is not enough in itself, since a quality of the meshing sufficient to obtain an acceptable accuracy should be also guaranteed. This goal could be pursued in a variety of ways, but in three dimensional electromagnetic analysis is not exactly easy: if you want to know more, read on ...

What follows is a synthesis of the presentations held in the Panel Session organized and chaired by Prof. Jean Louis Coulomb, of the Laboratoire d'Electrotechnique de Grenoble, who also collected the contributions of the panelists. Contributions are given in the order of presentation followed at Compumag Rio and are intended as a sequence of individual opinions expressed by the panelists on important specific topics rather than an integrated summary of the Session.

Panel session "Mesh Generation"

COMPUMAG'97, Rio de Janeiro, Brazil

Automatic finite element generation using dynamic bubble system

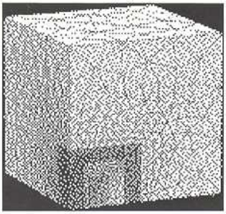
Hideo Yamashita
Electric Machinery Laboratory, Faculty of Engineering, Hiroshima University, Japan
yama@eml.hiroshima-u.ac.jp

One possible method for the automatic mesh generation for finite element analysis (FEA) uses a dynamic bubble system [1]. The method features two separate routines; one for the automatic generation of a set of nodes in accordance with the desired mesh density and the dynamic bubble system, and one for the automatic generation of finite elements according to the Delaunay algorithm and using the set of nodes generated from the previous routine. The generation of bubbles is performed frontally - starting from vertices and edges - and ending up with surfaces and volumes of the analysis region after which a dynamic movement of generated bubbles is performed. The dynamic bubble system consists of a set of bubbles which are defined by their radii, masses and positions in space according to their central coordinates. Each bubble obeys Newton's Second Law of Dynamics, where the acting forces are the van der Waals' forces between them. Input data for the method are just the outline of the analysis domain and the radii of some or all vertices that define in the model.

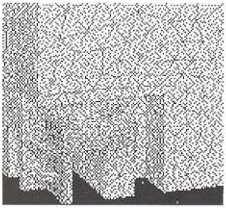
First, the vertex bubbles are set by user or automatically approximated using some approximation technique. Next, the generation of edge bubbles is performed and movement according to the existing dynamic forces is performed, followed by the generation and movement in the same manner as above, this time, however, for the set of facet bubbles. At the end, according to the previous procedures, volume bubbles are generated inside the entire analysis domain. When the dynamic stability of the entire generated dynamic system of bubbles is achieved, the movement stops and each center of a bubble becomes a possible node in the finite element mesh. Then, a tetrahedral finite element mesh is generated utilizing the Delaunay algorithm and the above generated set of nodes. In order to avoid problems due to the existence of convex regions, initially the entire analysis domain is divided into a very coarse, so called pre-tetrahedral mesh, again utilizing the Delaunay triangulation method and only the nodes that outline the entire analysis domain.

The method enables easy mesh density control using simple exponential functions with a very small amount of input data and very short computation time. The algorithm is applicable to the automatic mesh generation of complex shapes and structures such as those usually encountered in various electromagnetic devices (rotating machines and transformers). Due to its modest requirement for input data and its ability to generate meshes with graded mesh densities, the method could also be very useful for adaptive mesh refinements in FEA [2].

To verify its usefulness, the method was applied for automatic mesh generation of an inductor test model. The generated finite element meshes for the entire domain and for the air area are shown in Fig.1. From Fig.1, it is apparent that the method produces a high quality mesh division map with graded mesh density. The average value of the quality coefficient defined as a ratio between three times radius of the inscribed sphere and the radius of circumscribed sphere, for this mesh was 0.88, which is very close to the ideal value of 1.0 obtained for the equilateral tetrahedron.



(a)



(b)

Figure 1. Application model: (a) Final division map (b) Air region

Mixing hexahedra, prisms and tetrahedra

Jean-Louis Coulomb
Laboratoire d'Electrotechnique de Grenoble, INPG/UJF-CNRS, France
Jean-Louis.Coulomb@leg.ensieg.inpg.fr

Another well known problem in mesh generation, which is common for example in electromagnetics and acoustics is the presence of regions of free space which embed solid objects. Different methods show interesting features but lack generality (Delauney, advancing front, extrusion, transfinite mapping, etc.). How to keep the positive aspects of these generation methods while minimizing the drawbacks? By mixing them, and taking into account the problems of conformity of the mesh. In the toolbox we present, three kinds of mesh generators are made available: an automatic mesh generator, a mapped mesh generator and a mesh technique based on extrusion. The purpose of our toolbox is to allow the user to mix these mesh generators in order to increase the accuracy of the solution and to reduce computational cost. We have also built a pyramid element [5] that can satisfy the continuity requirements between rectangular and triangular faces. The following table summarizes the different popular mesh generators proposed by the toolbox. We can divide them into two main families of methods: automated and assisted generators.

Table 1. Mesh Generators

Methods	Finite Elements	Algorithms
Automated generators	Triangle/Tetrahedron	Delaunay triangulation
Assisted mesh generators	Quadrangle, Triangle Prism, Brick Tetrahedron	Extrusion Transfinite mapping

The example in Fig. 2 is taken from the car industry. It is a heating device, used to modify physical properties of a car engine. The frequency of the current in the inductor is 200 kHz and it involves eddy currents in the metal part. This example, solved with an A-V magneto dynamic formulation, is rather expensive because of the four complex unknowns per node. Furthermore, the accuracy of the solution is very sensitive to the discretisation of the skin depth, where induced currents are present. Fig. 2 shows the mesh of the conductor, the air and the coil. The conductor is meshed with prisms, the air with tetrahedra and the inductor with bricks. A strong continuity is ensured at the interface between free and mapped meshes by the pyramid elements (Fig. 3). Indeed, the mesh has to be very thin in the small width (0.5 mm) where eddy currents occur, that's why prisms are very useful in this case. A tetrahedral mesh of the whole structure would have generated too many nodes, and would have increased significantly the computational cost of the simulation.

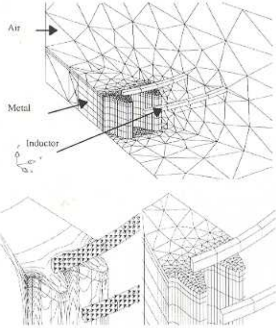


Figure 2. Mixing bricks (conductor), prisms (metal) and tetrahedra (air) and linking with pyramids.

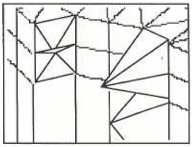


Figure 3. Pyramids on the interface metal/air.

Error Estimation and Adaptive Mesh Generation

Lutz Jaenicke
Algemeine Electrotechnik BTU Cottbus, Germany
Lutz.Jaenicke@aet.TU-Cottbus.DE

Error Estimation

Since the real solution of the field problem is unknown, the error distribution has to be computed using an error estimator. Several schemes have been proposed in recent years [6,7,8,9,10,11,12,13], all of them looking for a compromise between speed and reliability. An error estimator should have two properties: it should give a local error distribution that is the basis for the adaptive mesh refinement, and it should give some absolute measure that can be used for judging on convergence of the solution process.

Convergence

The error of the finite element solution converges as $|||e||| = O(h^{\nu})$, $\nu=\min(p,\lambda)$ with $|||e|||$ being the energy norm of the error, h the element size, p the polynomial order of the elements and λ the intensity of the singularities [14,15]. As the element size h is difficult to define, the number of degrees of freedom (*NDF*) can be introduced. As h decreases with $NDF^{-1/2}$ in the 2D case and $NDF^{-1/3}$ in the 3D case, the convergence rate can be calculated from $|||e||| = O(NDF^{-\nu/2})$ or $|||e||| = O(NDF^{-\nu/3})$.

If an error estimator delivering the energy norm like the one proposed in [6] is used, the estimated error $|||\bullet|||$ can be compared with the total energy $|||u|||$ stored in the field problem itself, leading to the definition of a relative error $|||\bullet|||/|||u|||$ which shows the same convergence behaviour as $|||\bullet|||$, since the computed stored total energy remains nearly constant. Figures 4 and 5 show the convergence of the finite element solutions for a 2D and a 3D example problem, respectively [16].

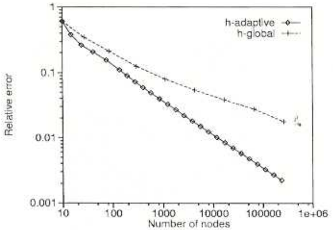


Figure 4. Relative error vs. number of nodes for a 2D problem.

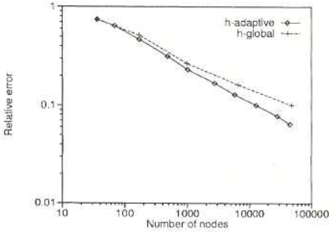


Figure 5. Relative error vs. number of nodes for a 3D problem.

The convergence behaviour clearly shows that the number of degrees of freedom necessary to achieve the required accuracy rises fast when going to 3D, making an adaptive (and hence well fitted) mesh very desirable. Due to the geometric problems in the 3D case (see the later remarks regarding robustness) adaptive mesh generation is seldom found in commercial and scientific codes.

Adaptive Mesh Generation

Beside the possibility of reducing the required memory (often at the cost of rising computer time), the adaptive mesh generation offers another great benefit. Starting with a very simple mesh, that can be automatically generated, the software will take over the time consuming task of adjusting the mesh to the problem. To a certain degree the computation of field problems can become a "fire and forget" task which does not require human intervention and delivers good results. Experience shows, however, that the user should understand both the field problem and its physics, as well as the algorithms, methods, and limitations behind the software. If the physical problem is not modelled properly, wrong results that are not recognized by the user can appear due to the "everything is automatic" attitude.

Automatic Adaptive Mesh Generation

Z. Cendes
Ansoft Corporation, Pittsburgh, PA 15219, USA
zol@ansoft.com

There are many advantages to automatic adaptive mesh generation. It eliminates the sometimes tedious task of manual mesh generation by unloading this work to the computer. It produces an optimal mesh that reduces computational costs by minimizing mesh size. It provides accuracy indicators by which one can determine the convergence of the solution. And it is more robust than manual mesh generation, avoiding the many pitfalls encountered in a user directed process.

While the above algorithm is straightforward, it does contain two complex steps. One is the process by which the elements are made; the other is the procedure used to compute the errors. For the first of these complex steps, it is necessary to create a mesh with arbitrary geometries and to refine it locally. This is possible today only with triangles in two-dimensions and tetrahedra in three dimensions. The essential algorithm for this process is called Delaunay tessellation and was applied in finite element computation for example in [9,17]. For the second step, it is necessary to determine the error in each finite element. This process can be accomplished for example by the error energy residual algorithm in [18].

A key component of Delaunay tessellation applied to finite element mesh generation is the preservation of object surfaces. The Delaunay algorithm in a "pure" sense creates a triangular or tetrahedral mesh for an arbitrary set of points without considering object surfaces. In finite element computation, there is an added requirement to maintain the integrity of the surfaces of the objects in the model. Until recently, the leading approach to object surface preservation was the "add point" algorithm. In this algorithm, points are added to the object surface whenever an element face cuts through that surface. In theory, if enough points are added on a surface, eventually the surface will be tessellated entirely by element faces. In practice, this approach is expensive and error prone: complex models often generate endless loops and excessive numbers of surface points.

Fortunately, a procedure that guarantees the integrity of the surface has been developed recently. In this approach, the surface mesh is created first, before attempting to form the volume mesh. Points are added on the surface to meet strict mathematical criteria. The volume mesh is created only after these strict criteria have been met. It can be proved mathematically that a valid volume mesh that matches the previously generated surface mesh always exists provided that the surface mesh satisfies the proper criteria.

The other major component of automatic adaptive mesh generation is the error energy residual algorithm. This algorithm is described in detail in [19]. It consists of two essential steps: (1) computing the residual in each finite element, and (2) computing the local energy error from the element residuals. From interpolation theory we know that the optimal rate of convergence of a 3D finite element solution is $N^{[-(2p+2)/3]}$ where N is the number of points and p is the polynomial order. Tests show that error energy algorithm provides this optimal convergence rate.

Robustness and generality issues in the generation of tetrahedral meshes for computational electromagnetics

Piergiorgio Alotto
University of Genova, Italy
alotto@die.unige.it

The current state of the art in tetrahedral mesh generation shows a well-understood, stable and computationally efficient [21] set of techniques to improve, from the point of view of accuracy, existing, boundary-conforming meshes. The need for such strategies arises because of the badly shaped elements which can be generated by the first stages of most mesh generation algorithms (Fig. 6).

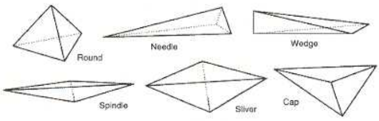


Figure 6. Optimal and various types of badly shaped elements.

The two main families of mesh improvement techniques operate, after selecting proper quality criteria [20, 23], either on nodes or on edges/faces: the first through addition, removal or movement, the latter through swaps [22]. The best results in terms of quality improvement are achieved by carefully combined methods [24]. Recently, the use of adaptive techniques [26], with proper node addition strategies [25], is becoming more widespread also due to the sinking cost of sufficiently powerful hardware (Fig. 7).

Figure 7. Initial and adapted meshing of a C-shaped magnet.

While mesh improvement techniques, which operate on given boundary-conforming meshes, are today routinely used, the subject of three-dimensional mesh generation itself is still open to research. A number of different issues contributes to the lack of sufficient robustness and generality of current meshing algorithms and packages, such as: choice of algorithm, implementation of a given algorithm, finite precision arithmetic, interaction, or lack thereof, with the modelling environment. In computational electromagnetics and other areas of research which make heavy use of gridding techniques a number of different mesh generation strategies are currently used to generate 3D tetrahedral meshes: spatial decomposition [34,33], advancing front [29], Delaunay/point insertion [32], convex decomposition [27, 28]. All of these algorithms have their own strengths and weaknesses and no one appears to be clearly superior to all others in general. Depending on the chosen technique, thousands of elements per second can be generated on rather inexpensive equipment in structural analysis and CFD (1000 elements per second on Sun Sparc 2 [35], 15000 elements per second on HP735 [36], 3000 elements per second on HP735 [31]) and multi-million element grids are also a reality (20 Million elements [30]). Although such results may seem, and indeed are, impressive there are no similar achievements in the computational electromagnetics community. This very brief overview of the current state-of-the-art shows that more research in this key technology is necessary to further strengthen and simplify the use of finite element codes.

References

[1] V.Cingoski, R.Murakawa, K.Kaneda, H.Yamashita, "Automatic mesh generation in finite element analysis using dynamic bubble system", *Journal Applied Physics*, Vol. 81, No.8, Part2, 15, pp.4085-4087, 1997.

[2] R.Murakawa, V.Cingoski, K.Kaneda, H.Yamashita, "Adaptive Finite Element Analysis Using Dynamic Bubble System", *11th Conference on the Computation of Magnetic Fields COMPUMAG'97*, PC2-1, pp. 255-256, 1997.

[3] O.C. Zienkiewicz, *The Finite Element Method in Engineering Science*, McGraw-Hill, London, 1971.

[4] Ramsey A., "Comparison of HEX and TET meshes", *Benchmark*, June 1995, 11-13, 1984.

[5] Coulomb J.L., Zgainski F.X., Mar_chal Y., "A Pyramidal Element to link Hexahedral, Prismatic and Tetrahedral Edge finite Elements", *IEEE Transactions on MAGNETICS*, Vol. 33, No. 2, March 1997, pp 1362-1365.

[6] R. E. Bank and A. Weiser, "Some a posteriori error estimators for elliptic partial differential equations", *Mathematics of Computation*, vol. 44, no. 170, pp. 283—301, April 1985.

[7] J. Penman and M. D. Grieve, "An approach to self adaptive mesh generation", *IEEE Transactions on Magnetics*, vol. 21, no. 5, pp. 2567—2570, September 1985.

[8] R. L. Ferrari and A. R. Pinchuk, "Complementary variational finite-element solution of eddy current problems using field variables", *IEEE Transactions on Magnetics*, vol. 21, no. 5, pp. 2242—2245, September 1985.

[9] Z. J. Cendes, D. Shenton, and H. Shahnasser, "Magnetic field computation using Delaunay triangulation and complementary finite element methods", *IEEE Transactions on Magnetics*, vol. 19, no. 6, pp. 2551—2554, November 1983.

[10] J. Penman and J. R. Fraser, "Dual and complementary energy methods in electromagnetism", *IEEE Transactions on Magnetics*, vol. 19, no. 6, pp. 2311—2316, November 1983.

[11] P. Fernandes, P. Girdinio, P. Molfino, and M. Repetto, "Local error estimates for adaptive mesh refinement", *IEEE Transactions on Magnetics*, vol. 24, no. 1, pp. 299—302, January 1988.

[12] P. Girdinio, P. Molfino, G. Molinari, L. Puglisi, and A. Viviani, "Finite difference and finite element grid optimization by the grid iteration method", *IEEE Transactions on Magnetics*, vol. 19, no. 6, pp. 2543—2546, November 1983.

[13] C. S. Biddlecombe, J. Simkin, and C. W. Trowbridge, "Error analysis in finite element models of electromagnetic fields", *IEEE Transactions on Magnetics*, vol. 22, no. 5, pp. 811—813, September 1986.

[14] O. C. Zienkiewicz and R. L. Taylor, *The Finite Element Method - Basic Formulation and Linear Problems*, McGraw-Hill, London, 4 edition, 1989.

[15] O. C. Zienkiewicz and J. Z. Zhu, "Adaptivity and mesh generation", *International Journal for Numerical Methods in Engineering*, vol. 32, pp. 783—810, 1991.

[16] L. JŠnicke and A. Kost, "Error estimation and adaptive mesh generation in the 2d and 3d finite element method", *IEEE Transactions on Magnetics*, vol. 32, no. 3, pp. 1334—1337, May 1996.

[17] D. N. Shenton and Z. J. Cendes, "Three-Dimensional Finite Element Mesh Generation Using Delaunay Tessellation", *IEEE Trans.*, Vol. MAG-21, pp. 2535-2538, 1985.

[18] D. N. Shenton and Z. J. Cendes, "MAX - An Expert System for Automatic Adaptive Magnetics Modeling", *IEEE Trans.*, Vol. MAG-22, pp. 805-807, 1986.

[19] C. W. Steele, *Numerical Computation of Electric and Magnetic Fields*, 2nd ed., pp. 239-249, Chapman and Hall, New York, 1997.

[20] A. Liu, B. Joe, "Realtionship between tetrahedron shape measures", *available on Internet.*

[21] N.A.Golias, T.D. Tsibukis, "An approach to refining three-dimensional tetrahedral meshes based on Delaunay transformations", *IJNME*, vol. 37, pp.793-812, 1994.

[22] L. Kettunen, K. Forsman, "Tetrahedral mesh generation in convex primitives", *IJNME*, vol. 38, pp. 99-117, 1995.

[23] V.N. Parthasarathy, C.M. Graichen, A.F. Hathaway, "A comparison of tetrahedron quality measures", *Finite Elements in Analysis and Design*, vol. 15, pp. 255-261, 1993.

[24] L.A. Freitag, C. Ollivier-Gooch, "A comparison of tetrahedral mesh improvement techniques", *available on Internet.*

[25] A. Plaza, G.F. Carey, "About local refinement of tetrahedral grids based on bisection", *available on Internet.*

[26] P. Alotto, A. Castagnini, P. Girdinio, P. Fernandes, "Error estimation and adaptive meshing in 3D electrostatic and magnetostatic problems", *proc. 11th Compumag Rio.*

[27] B. Chazelle, "Convex partitions of polyhedra: a lower bound and worst-case optimal algorithm", *SIAM J. Comput*, vol. 13, no.3, pp.489-507, 1984.

[28] B. Joe, "Tetrahedral mesh generation in polyhedral regions based on convex polyhedron decompositions", *IJNME*, vol. 37, pp. 693-713, 1994.

[29] R. Loehner, P. Parikh, "Generation of three-dimensional unstructured grids by the advancing front method", *IJNM Fluids*, vol. 8, pp.1135-1149, 1988.

[30] J.D. Baum, H. Luo, R. Loehner, "Numerical simulation of blast in the World Trade Center", AIAA-95-0085, 1995.

[31] P. J. Frey, H. Borouchaki, P.L. George, "Delaunay tetrahedralization using and advancing front technique", *available on Internet.*

[32] P.L. George, F. Hecht, E. Saltel, "Automatic mesh generator specified boundary", *Computer Methods in Appl.Mech and Eng.*, vol. 92, pp. 269-288, 1991.

[33] E.K. Buratynski, "A fully automatic three dimensional mesh generator for complex geometries", *IJNME*, vol. 30, pp. 931-952, 1990.

[34] S. A. Mitchell, S.A. Vavasis, "Quality mesh generation in three dimensions", *Proc. 8th Annual Conference on Computational Geometry*, Berlin Germany, Aug. 1992.

[35] N.P. Weatherhill, O. Hassan, "Efficient three dimensional Delaunay triangulation with automatic point creation and imposed boundary constraints", *IJNME*, vol. 37, pp. 2005-2039, 1994.

[36] H. Borouchaki, S.H. Lo,"Fast Delaunay triangulation in three dimensions", *Comput. Meth. Appl. Mech. Eng.*, vol. 128, pp. 153-167, 1995.