Application of smoothing techniques for aligning meshes

José M. Escobar
University of Las Palmas de Gran Canaria
Parque Científico-Tecnológico de Tafira
Las Palmas de Gran Canaria 35017
Spain
jescobar@dsc.ulpgc.es

Gustavo Montero
University of Las Palmas de Gran Canaria
Parque Científico-Tecnológico de Tafira
Las Palmas de Gran Canaria LP 35017
Spain
gustavo@dma.ulpgc.es

Rafael Montenegro
University of Las Palmas de Gran Canaria
Parque Científico-Tecnológico de Tafira
Las Palmas de Gran Canaria LP 35017
Spain
rafa@dma.ulpgc.es

Eduardo Rodríguez
University of Las Palmas de Gran Canaria
Parque Científico-Tecnológico de Tafira
Las Palmas de Gran Canaria LP 35017
Spain
barrera@dma.ulpgc.es

In this work we develop a method that deforms a given triangulation to obtain its alignment with interior curves. These curves, defined by splines, can represent internal interfaces between different materials, internal boundaries, etc. An important feature of this method is the possibility to adapt a reference mesh to curves that change their shape in the course of an evolutive process. The method is an extension of the surface mesh smoothing proposed in [2] in which the quality improvement of the mesh is obtained by an iterative process in which each node of the mesh is moved to a new position that minimizes a certain objective function. The objective function is derived from the algebraic quality measure [1] mean ratio extended to the local submesh, that is, set of triangles connected to the free node. Suppose that C is a curve sited on a surface mesh S, the basic idea consists on projecting on C some nodes of S until getting an approximate representation (interpolation) of C by linked edges of S. A node is considered projectable if there exists a position on C for which its local submesh does not result tangled. This projection implies a displacement of some nodes from its original positions and, in general, has a negative effect on the quality of the triangles close to C. To avoid this decrease in quality, the remaining nodes are also displaced to new positions following the smoothing procedure of [2]. Both the projection of nodes on C and the smoothing procedure are carried on a two-dimensional parametric space that, in our case, is a plane in which S can be projected performing a valid mesh, that is, without inverted elements. Working on the parametric space is a crucial aspect of our algorithm because the presence of barriers in the objective function avoids overlapped meshes to appear. The task to determine if a node can be projected on C and, in affirmative case, which is its optimal position, is undertaken by an objective function that incorporates the modifications introduced in [3]. All these questions will be conveniently supported by examples. In particular, we present an application of a topographic mesh which is modified in order to outline of the coastal shores.

[1] Knupp PM. Algebraic mesh quality metrics. SIAM J Sci Comp 2001;23:193 218.
[2] Escobar JM, Montero G, Montenegro R, Rodríguez E. An Algebraic Method for Smoothing Surface Triangulations on a Local Parametric Space. Int. J. Numer. Meth. Engrg. 2006;66:740-760.
[3] Escobar JM, Rodríguez E, Montenegro R, Montero G, González-Yuste JM. Simultaneous untangling and smoothing of tetrahedral meshes. Comp Meth Appl Mech Eng 2003;192:2775-2787.

Geometrical modeling and meshing of granular domains

Azeddine BENABBOU
INRIA Rocquencourt
BP 105
Le Chesnay cedex 78153
France
azeddine.benabbou@inria.fr

Patrick LAUG
INRIA Rocquencourt
BP 105
Le Chesnay cedex 78153
France
patrick.laug@inria.fr

Houman BOROUCHAKI
University of Technology of Troyes
BP 2060
Troyes cedex 10010
France
houman.borouchaki@utt.fr

Jian LU
Hong Kong Polytechnic University
Hung Hom
Kowloon
Hong Kong
jian.lu@inet.polyu.edu.hk

We consider both the problem of filling up a two dimensional domain with disks whose radii follow a user-defined distribution and the problem of generating quality meshes of such a domain. The disks must neither overlap each other nor intersect the domain boundary. The filling problem is often referred to as "random close packing". In this paper, an efficient constructive algorithm based on an advancing-front approach is proposed. Our algorithm is compared to existent constructive and dynamic methods [1,2]. The gains in CPU time and the improvements in density are reported and demonstrate the efficiency of the new method. The generated aggregates may find applications in many fields such as metallurgy, ceramics, soil science, biology, physics, chemistry and engineering. In our concern, this algorithm is used to model the nanostructure geometry where grains are approximated by disks [3,4]. The advancing-front approach provides empty areas where fronts are in collision. A point relocation algorithm using weighted Delaunay triangulation is also introduced to balance equally these areas. To generate quality meshes of the resulting granular domain, the meshing is governed by a size map depending on the thickness of the "grain boundaries" [5,6]. Furthermore, using Laguerre diagrams, we can transform the disks into polygonal cells which are very similar to the grain shapes observed in many material structures. The same meshing method is then applied to construct quality meshes for this new model.

References

[1] K. Bagi. An algorithm to generate random dense arrangements for discrete element simulations of granular assemblies. Granular Matter (2005) 7: 31-43.
[2] B.D. Lubachevsky and F.H. Stillinger. Geometric properties of random disk packings. Journal of Statistical Physics, Volume 60, Numbers 5-6, Pages: 561 - 583, September 1990.
[3] K. Lu and J. Lu. Nanostructured surface layer on metallic materials induced by surface mechanical attrition treatment. Materials Science and Engineering A, Volumes 375-377, 15 July 2004, Pages 38-45.
[4] K.S. Kumar, H. Van Swygenhoven and S. Suresh. Mechanical behavior of nanocrystalline metals and alloys. Acta Materialia, Volume 51, Issue 19, 25 November 2003, Pages 5743-5774.
[5] H. Borouchaki, P.L. George, F. Hecht, P. Laug, E. Saltel. Delaunay Mesh Generation Governed by Metric Specifications - Part I: Finite Elements in Analysis and Design (FEAD), Elsevier, vol. 25, no. 1-2, pp. 61-83, March 1997.
[6] P. Laug, H. Borouchaki. The BL2D mesh generator - Beginner's guide, user's and Programmer's manual, INRIA Technical Report RT-0194, July 1996.

Gmsh: a three-dimensional FE mesh generator with built-in pre- and post-processing facilities

Jean-Francois Remacle
Professor
Universite catholique de Louvain
Place du Levant, 1, Batiment Vinci
Louvain-La-Neuve BE 1348
Belgium
remacle@gce.ucl.ac.be

Christophe Geuzaine
Professor
Université de Liège
Institut Montéfiore
Liège 4000
Belgium
cgeuzaine@ulg.ac.be

The development of Gmsh has been under way for a decade now. Recently, Gmsh has encountered a major evolution. In this talk, we will present Gmsh 2.0 and detail some of its innovative features : -) Adaptive mesh generation algorithms applied to hybrid 3D models (STEP-IGES-Gmsh Native...), -) Mesh generation applied to ocean modelling, -) Efficient visualization of high order finite element solutions We will also share our experience in the development of an open source mesh generator and detail the perspectives for the future of Gmsh.

Triangulation of Microstructure using Recursive Subdivision and Advancing Front Technique

Daniel Rypl
CTU in Prague, Faculty of Civil Engineering
Thakurova 7
Prague 16629
Czech Republic
drypl@fsv.cvut.cz

The present paper deals with the triangulation of microstructure initially represented by a digital image obtained from Computer Tomography or any other similar scanning device. Firstly, the gray scale digital representation is thresholded into voxels of appropriate discrete values of gray corresponding to individual phases of the processed microstructure. In the next phase, the boundary voxels (and their boundary sides) of individual phases are identified. A triangulated boundary representation (of the same resolution as the initial voxel representation) is then obtained from the boundary voxel representation by replacing the boundary sides of boundary voxels by semi-regular triangulation with nodes at the centres of those boundary sides. In this triangulated boundary representation, the individual surfaces (and their boundary curves) bounding individual phases of the microstructure are identified and then subjected to recursive interpolating subdivision yielding a C1 continuous surface. In the final phase, the individual smooth boundary surfaces are triangulated using the Advancing Front Technique. Since there is available no global mapping of the recovered surfaces, the discretization is performed directly in 3D space on the surface. Note that the resolution of the final triangulation is independent of the resolution of the initial digital representations and is driven mainly by the user specification and properties (curvature) of the recovered smooth representation. The performance of the proposed approach is shown on an example. The approach for the microstructure reconstruction presented in this paper is beneficial in the sense that it is capable to handle complex topologies and its final resolution is independent of the resolution of the digital representation. Although only the surface representation has been treated in this work, the reconstructed surface triangulation can be easily employed for the solid triangulation of the microstructure, again with variable resolution. However, while the shape of the microstructure is captured quite precisely, the volume fractions of individual phases exhibits some discrepancies when compared to the initial digital representation. This also implies that statistical distribution of individual phases is not reconstructed optimally. From this point of view, the future research is to be focused on further enhancement of the quality of the reconstruction, especially in the quantitative sense.

Direct CAD Access for Design Through Analysis

Robert Haimes
MIT Aero/Astro
37-467
77 Massachusetts Ave
Cambridge MA 02139
United States
haimes@mit.edu

This talk presents a comprehensive approach for CAD based geometry handling in support of single and multi-disciplinary analysis and design. This is done through a CAD vendor-neutral Application Programming Interface (i.e. middleware). By using the Master-Model concepts found in modern CAD systems, the same CAD components can used throughout the design process from conceptual through final design (including manufacturing). This approach allows for the design intent to be encapsulated within the CAD part. Proper layout and definition of the CAD model facilitates design changes and use across all stages of design. Unlike traditional schemes, the software model presented here allows for "hands-off" automated meshing -- a requirement for higher fidelity design studies. This is accomplished by providing a dual-view of the geometry: the normal Boundary Representation (BRep) of a Solid and an associated discrete watertight tessellation (which represents the geometry with minimal counts). The discipline-based grid generator (using the API) can use either or both to satisfy geometric and the fidelity requirements of the solver. Access to the analytic entities of the BRep can be performed by evaluations (both forward and inverse) as well as conversions to NURBS. The vertices of the tessellation contain geometric parameter values (t for curves, u,v for surfaces) to facilitate augmentation through evaluation. Multidisciplinary analysis is handled through solid modeling constructs, using the geometry as a transfer media. For design applications, the engineer/designer specifies the design space directly by (a subset of) the parameters found within the CAD component. These key defining values in the model, specified during part synthesis, are exposed to carry out design studies and optimization. Defeaturing of the Master-Model can be used to make the component appropriate for the design phase and the fidelity of the analysis at hand. A turbine blade example suitable for detailed aero/structural/thermal analysis is used to illustrate these concepts.

Automatic Meshing of 3-D Respiratory Geometries

Andrew Kuprat
Senior Scientist
Pacific Northwest National Lab
902 Battelle Boulevard
P.O. Box 999; MSIN P7-58
Richland WA 99352
United States
andrew.kuprat@pnl.gov

Daniel Einstein
Scientist
Pacific Northwest National Lab
1100 Dexter Ave. N, Rm 472
Seattle WA 98109
United States
daniel.einstein@pnl.gov

We perform multiscale computation of toxicological effects of particles on the respiratory system in a 3-D CFD model of airflow with particle tracking. The correct computation of airflow and particle dose in the nose, trachea, and lungs is dependent on rapid but efficient meshing of many individual geometries. Since it is not always clear what kind of morphological characteristics will make an individual more susceptible to a certain kind of particulate, it is important that the full range of individual variation is reflected in the geometry database. Because of great individual variation, it usually takes a considerable time to create a computational mesh for each geometry. In fact, mesh generation usually dominates the amount of time required to perform a CFD computation. We have developed an automated meshing tool that automatically generates anisotropic meshes on arbitrary biological geometries. Anisotropic mesh generation is essential to limit the sizes of the mesh to under ~10 million elements. At mesh sizes larger than this it is in practice difficult to visually inspect the computation for anomalies---and visual inspection of computed airflow fields is essential for validation. Arbitrary biological geometries present unique problems in mesh generation. Airways exhibit a range of scales and a feasible approach to this problem must involve an assessment of scale-length for generation of appropriately sized elements in each region of the 3-D geometry. Our mesh generator automatically determines a scale length for each region by casting rays normal to the geometry from each vertex on the bounding surface mesh. In turn, the spacing of the vertices on the surface mesh is automatically adjusted to be consistent with the detected scale length. Problematic regions, detected either through visual inspection of computed flow fields or determined automatically through flagging by an a posteriori error estimator, are easily meshed more finely by feeding back this information into the automatic grid generator. Since our meshes our scale-invariant (i.e. tend to have the same number of layers across each airway, independent of the size of the airway), CFD computations will tend to have relative error equilibrated over the whole mesh. This means the meshes are maximally efficient, and produce the best results for the computational resources available.

Funded by NHLBI RO1 HL073598-01.

CONTINUOUS SIGHT OF INTERPOLATION ERROR FOR 3D ANISOTROPIC UNSTRUCTURED MESH ADAPTATION

Adrien LOSEILLE
Phd Student
INRIA Gamma Project
INRIA Rocquencourt
domaine de Voluceau
Le Chesnay 78153
France
adrien.loseille@inria.fr

Frederic Alauzet
INRIA Gamma Project
INRIA Rocquencourt
domaine de Voluceau
Le Chesnay fr 78153
France
Frederic.Alauzet@inria.fr

This talk is an introduction to a continuous sight of interpolation error and the deduced error model designed for anisotropic unstructured mesh adaptation. Contrary to the classical discrete approach where the interpolation error is studied on an existing discrete mesh, here the mesh is continuously modelled by a metric, i.e., a continuous prescription of elements' sizes and orientation. A continuous interpolation error expression is then proposed. The main advantage of this approach is that a global variational calculus can be performed. Thus the best mesh which minimises Lp norm of interpolation error on the whole domain is deduced. In addition, a theoretical global asymptotic second order mesh convergence is proved for the solution (even with discontinuities) whereas an 1/p order of convergence in Lp norm is usually reached in non smooth aera. Indeed, convergence of adapted solutions usually remain observed in practice only as a theoretical convergence study is tedious to carry out on non-isotopological meshes. From the practical point of view, the mapping between continuous and discrete mesh is analysed as well as the adaptive algorithm. Numerical simulation have been performed on 3D complex geometry in order to assess this continuous approach. Numerical mesh convergence order are compared to the theoretical one. Smooth flows at low Mach number as well as flows with strong shocks have been computed. We will also focus on each stage of this fully automatic adaptive scheme: (i) metric construction, (ii) 3D adaptive local remeshing, (iii) solution interpolation and (iv) flow solver.

Mesh Improvement for Quadrilateral Element using Coarsening and Refinement Techniques

Takuji Hayashi
Tokyo University of Science
2641 Yamazaki
Noda, Chiba 278-8510
Japan
takuji@me.noda.tus.ac.jp

Yoshitaka Wada
Tokyo University of Science, Suwa
5000-1 Toyohira
Chino, Nagano JP 391-0292
Japan
wada@rs.suwa.tus.ac.jp

Masanori Kikuchi
Tokyo University of Science
2641 Yamazaki
Noda, Chiba JP 278-8510
Japan
kik@me.noda.tus.ac.jp

FE analysis is one of the most powerful tools for simulation of physical phenomena. Efficient and reliable FE analysis techniques, which are local mesh refinement and multigrid solver, have been developed by the authors. Increment of multigrid iteration is often caused by existence of many distorted elements which are generated by local mesh refinement. It is very important for less number of the iterations to keep initial shape of elements as possible as good. This paper presents local mesh coarsening and refinement techniques in order to improve quadrilateral mesh quality in 2D. Coarsening is removal of the existing distorted elements to modify a mesh. Refinement is applying the refinement mesh patterns to the mesh. Coarsening techniques are loop elements coarsening. Refinement techniques are boundary elements refinement, geometric relaxation refinement and aspect ratio refinement. Loop element coarsening aims to modify complex mesh connectivity to relaxed one geometrically. This coarsening technique is useful for not only removing distorted elements but also preprocess refinement techniques. Aspect ratio refinement is applied to elements of large aspect ratio. The mesh patterns are simple subdivided elements. One element is simply divided into several aligned elements. For example, if an element is divided into 3 elements, new 3 elements is generated in the original elements, and then adjacent connected elements are recursively divided to keep connectivity of elements. Geometric relaxation refinement is applying refinement patterns and smoothing mesh. To begin with refinement mesh patterns of geometric relaxation are applied to distorted elements. Connectivity relation between distorted element and the surrounded is appropriately modified to keep element connectivity good. After geometric relaxation mesh is smoothed by laplacian smoothing. As a result of geometric relaxation refinement, geometric constraint of distorted element is relaxed and shape of elements is improved. After connectivity modification, shape of distorted elements is smoothed. In this paper how techniques improve mesh quality and several practical examples are presented. Effectiveness of the techniques is presented and discussed.

Local Mesh Modifications to Correct Curvilinear Meshes for 3D Curved Domains

Xiaojuan Luo
Research Associate
Rensselaer Polytechnic Institute
CII 7011, RPI
110, 8th street
TROY NY 12180
United States of America
xluo@scorec.rpi.edu

Mark Shephard
Professor
Rensselaer Polytechnic Institute
CII 7011, RPI
110, 8th street
TROY NY 12180
United States
shephard@scorec.rpi.edu

Lie-Quan Lee
Stanford Linear Accelerator Center
Menlo Park
Menlo Park ca 94025
United States
liequan@slac.stanford.edu

High-order finite element methods applied to curved domains requires the use of properly curved elements. The common approach to the construction of such meshes is to apply straight-sided mesh generation procedures and to then curve the mesh edges and faces on curved domain boundaries to the proper order. This approach takes advantage of the conventional unstructured mesh generators to deal with the complexity of model geometry. However, the resulting mesh may become invalid because the curving of the mesh entities to model boundaries can lead to negative determinant of Jacobian in the closures of some elements. Effective and efficiently correction of those invalid elements is critical in curvilinear mesh construction and usage. A procedure has been developed to apply curved local mesh modification operations to incrementally correct the invalid elements created during the mesh curving process. The curved local mesh modifications are built upon a set of operations including collapse, split, swap, and shape modifications etc. Those operations are applied in an order that effectively make the resulting curved elements valid [1]. The procedure is able to deal with any curved meshes with/without model information. In case that the relation between the model and mesh is given, any new mesh entity uses the model shape information to define element shapes of the proper order. In case the model information is unavailable, the new shapes are approximately computed using the existing curved surface meshes. This procedure was applied to create valid quadratic meshes for the Standford Linear Accelerator Center's (SLAC) use in performing the large-scale simulations for the superconducting cavity designs [2]. For time-domain solver T3P, which applied high-order finite element method, for computing wakefields, the results demonstrated that 40% efficiency could be achieved for using the curvilinear meshes comparing to the straight-sided ones.

[1] X.J. Luo, M.S. Shephard, R.M. O'Bara, R. Nastasia and M.W. Beall,, "Automatic p-version mesh generation for curved domains", {\it Engng. with Compters }, v20, 273-285., 2004.
[2] A.E. Candel, A.C. Kabel, K. Ko, L. Lee, Z. Li, C.K. Ng, E.E. Prudencio, G.L. Shussman and R. Uplenchwar, ``Parallel high-order finite element method for accurate field computations in wakefield and PIC simulations". 9th international computational accelerator physics conference. Oct. 2-6, Chmonix MOnt-Blanc, France, 2006.

A Rapid Meshing Technique for Studying Near-Surface Phenomena

Amanda Hines
Computer Scientist
US Army Engineer Research and Development Center, Information Technology Laboratory
3909 Halls Ferry Road
Vicksburg MS 39180
United States of America
Amanda.M.Hines@erdc.usace.army.mil

Owen Eslinger
US Army Engineer Research and Development Center, Information Technology Laboratory
3909 Halls Ferry Road
Vicksburg MS 39180
United States of America
Owen.J.Eslinger@erdc.usace.army.mil

As part of an ongoing effort to improve automated target recognition algorithms for remote sensing technologies, a suite of closely coupled numerical simulators has been developed. This platform includes high resolution thermal and moisture transport finite element models, coupled with solar and vegetation models. It is well suited for the testing of specific scenes with controlled environmental conditions, in particular meteorological and time-of-day conditions, which otherwise might be difficult and time consuming to reproduce in the field. However, this suite serves as a compliment to, rather than a replacement for, field and laboratory testing of remote sensors. The moisture and thermal codes in this numerical suite require the rapid and robust production of finite element meshes of the shallow subsurface. Each of these meshes must have realistic topography as well as the ability to include natural and manmade objects to achieve relevancy, e.g. rocks and unexploded ordinance. The mesh generation process has been achieved by taking advantage of open source (black-box) mesh generation software, and a post-process, mesh smoothing technique to ensure quality elements in the final tetrahedral mesh. In addition, the process allows for the inserted objects to be either buried, flush with or protruding from the ground surface. A simple mesh repair operation around the objects is utilized to avoid poorly shaped tetrahedra in these regions. When desired, sub-surface soil regions can be assigned as a post-process step. This ability extends to statistically generated soil distributions using site-specific information obtained from soil samples. Traditionally, finite element mesh generation for large domains can be a time consuming and highly specialized process. The meshing requirements for simulations when testing remote sensors could be burdensome as object placement becomes paramount and there exist a multitude of possible scenarios to examine. Thus, rapid mesh generation is a necessity. The approach taken in this work was to remove the entire meshing procedure away from a user utilizing a graphical user interface and mouse. Indeed, the actual process of mesh generation, mesh repair around objects, and mesh smoothing is completely hidden from the user in an automatic fashion and can be executed on a variety of operating systems. The software developed uses raw LIDAR data as an input. The data is down-sampled to create a triangulated representation of the ground surface, which is used to create the full three dimensional domain. Additionally, individual meshes of objects and their locations are supplied by the user and inserted automatically. A typical scene might be a (10m x 10m x 1m) domain with 1.5M tetrahedral elements, 300k nodes, and contain 20 or more objects. The time needed to produce this mesh will be less than 30 minutes, with the majority of time taken in the mesh smoothing algorithm. By reducing the time it takes to produce a single mesh, the number of test scenarios increases significantly. Also, the training time required for personnel to use this software is less than an hour, which allows the less experienced team members to handle meshing operations. The work presented will focus on the process of moving from LIDAR data and object meshes to the final tetrahedral mesh, with attention given to the smoothing algorithm and the treatment of objects inserted into the domain.

Techniques for Robust Target-to-Source Projection for Many-to-Many Sweeping

Michael Borden
Sandia National Labs
PO Box 5800
Mail Stop 0376
Albuquerque NM 87111
United States of America
mborden@sandia.gov

Michael Brewer
Sandia National Labs
PO Box 5800
Mail Stop 0376
Albuquerque NM 87111
United States of America
mbrewer@sandia.gov

Sweeping is a hexahedral mesh generation technique that requires mapping a quadrilateral mesh from a set of source surfaces onto a set of target surfaces. Robust algorithms exist for the special case of a single target surface (often referred to as a many-to-one sweep); however, existing algorithms have not solved the general, multiple target case (often referred to as many-to-many sweeps). In current practice, the general problem is typically manually reduced to a set of single-target problems through geometry decompositions. This manual process is time-consuming and significantly impacts the total mesh generation time. Because of the high cost of manual decomposition, significant research has focused on automatic decomposition to solve the general sweeping problem. Although previous research efforts have advanced the state of the art, they have fallen short of providing a robust solution for production use. In this work, we identify two of the major remaining issues inhibiting this automation and propose solutions to them. The two issues are 1) determining onto which source surfaces a target surface should be projected and 2) determining the appropriate projection for the target surface onto those source surfaces. As a solution to 1), we propose using "pseudo" surfaces to represent the source and target surface geometry. Pseudo source surfaces and pseudo target surfaces contain information about the real, underlying surfaces as well as additional information from the three-dimensional geometry. This combination of information provides enough data to solve 1). As a solution to 2), we propose a ray tracing technique that projects one surface onto another. We conclude with several examples which demonstrate the applicability of our solutions.

*Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockhead Martin Company, for the United States Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

Constructing A Geometric Boundary Representation From Multi-Material Voxel Data

Tathagata Ray
Post Doctorate
Rensselaer Polytechnic Institute
SCOREC, 110 8th Street
Troy NY 12180
United States
rayt@scorec.rpi.edu

Mark Shephard
Professor
Rensselaer Polytechnic Institute
CII 7011, RPI
110, 8th street
TROY NY 12180
United States
shephard@scorec.rpi.edu

Ken Greiner
Coventor Inc.
625 Mount Auburn St.
Cambridge MA 02138
United States
ken@coventor.com

Computer simulation of processes such as semiconductor fabrication are based on domains containing more than one material regions. One of the ways of representing such a multi-material domain is using volumetric representation. In this representation each voxel stores the type of material it has and some degree of information on volume fractions. In our current application, we have a constraint that each voxel can store at the most two material types for multi-material voxels and volume fractions. The problem is to extract a geometric boundary representation, denoted as G, from this volume representation such that G is valid and consistent with the volumetric data. Since G can be non-manifold, we need to define the topological entities [1] of regions, shells, faces, loops, edges and vertices of G. The motivation behind constructing G is to utilize it for input to mesh generation. There has been considerable amount of work on surface mesh extraction from volumetric data though most of it is based on scalar field data [2]. These algorithms give us a collection of polygons with connectivity information approximating the overall geometry but they do not provide us a definition for the topological entities defining the boundary representation G. In our proposed method, we construct a surface mesh M from the input data in such a way that every mesh entity, i.e. vertices, edges and faces of the surface mesh M is classified [3] on the topological entities of the geometric boundary representation G of lowest dimension. Every material region in the domain is discerned by the shell around it. Our strategy is to build M in a systematic way such that first we will approximate the shells around each region. An initial approximation to some of the faces of G is done by detecting the common portion of two shells of adjacent material regions. An initial approximation to multi-material loops and edges are done by detecting the interface of two adjacent faces. The problem has the complexity of the constraint of representing at the most two material types present in a voxel. Thus for the voxels where three or more material regions meet, there is a loss of information leading to ambiguous conditions that must be resolved in a consistent manner. Once an initial approximation of G is obtained, it can be further enhanced by detecting feature edges and corners in the surface mesh M approximating it. Our proposed method is robust enough to take care of these ambiguities.

[1] Weiler K. J.,The radial-edge structure: a topological representation for non-manifold geometric boundary representation, In J. L. Encarnacao, M.J. Wozny, H.W. McLaughlin, eds, Geometric Modeling for CAD applications, p. 3-36, Elseviers Science Publisher B. V. (North-holland), Amsterdam, 1988.
[2] Ju T., Losasso F., Schaefer S., and Warren J., Dual contouring of Hermite data, ACM Transaction on Graphics(Proc. SIGGRAPH'02), v. 21(3), p. 339-346, 2002.
[3] Beall M. W. and Shephard M.S., A general topology based mesh data structure, International Journal for Numerical Methods in Engineering, v. 40, p. 1573-1596, 1997.

Adaptive Tetrahedral Mesh Generation for Intelligent Forging Simulation

Mincheol Lee
Post-doctoral Researcher
Gyeongsang National University
900 Gazwa-Dong
Jiju 660-701
Korea, Republic of
mclee@gnu.ac.kr

ManSoo Joun
Professor
Gyeongsang National University
900 Gazwa-Dong
Jinju 660-701
Korea, Republic of
msjoun@gnu.ac.kr

In this paper, we present an adaptive tetrahedral element generation technique for intelligent metal forming simulation based on a Delaunay method. Various mesh quality and density control schemes are used. A systematic and general approach to optimal triangular mesh generation on the surface is used for mesh quality control. The local transformation processes are used to enhance mesh quality as well as to generate the improved mesh from the input mesh. After application of the nodal smoothing scheme as a whole, the optimal nodal smoothing scheme is used for optimizing local mesh quality. The presented technique is applied to automatic forging simulation. In the forging simulation, curvature of the surface and die-material interference as well as the state variables including effective strain and effective strain-rate are considered in determining the mesh density distribution to reduce the geometrical error. The capability of the presented approach is tested through its application to intelligent forging simulation.

On Combining Mesh Redistribution with h-Adaptivity

Derek Gaston
Sandia National Labs
P.O. Box 5800
MS 0382
Albuquerque NM 87185-0382
United States
drgasto@sandia.gov

Larisa Branets
ExxonMobil URC
3319 Mercer St, NW509A
Houston TX 77027
United States
larisa.v.branets@exxonmobil.com

Graham Carey
University of Texas at Austin
201 East 24th Street
1 University Station, C0200
Austin TX 78712
United States
carey@cfdlab.ae.utexas.edu

Benjamin Kirk
Aerospace Engineer
NASA Lyndon B. Johnson Space Center
Mail Code EG3
2101 NASA Parkway
Houston TX 77058
United States
benjamin.kirk@nasa.gov

John Peterson
Research Asst.
Univ. of Texas at Austin
201 E. 24th St., C0200
Austin TX 78712
United States
peterson@cfdlab.ae.utexas.edu

Roy Stogner
University of Texas at Austin
1 University Station, C0200
Austin TX 78712
United States
roystgnr@ices.utexas.edu

In this study the use of mesh redistribution strategies for mesh smoothing or clustering in combination with adaptive mesh refinement (AMR) is investigated. These two strategies for optimal mesh design are usually viewed independently and there have been many studies of their individual application. Here we focus on a 2-stage algorithm in which: (1) redistribution is first applied on coarser meshes, and then (2) AMR is applied to further adapt the mesh to fine scale features such as interior or boundary layers. The concept is implemented to function within the 'libMesh' C++ finite element framework developed by the CFDLab at the University of Texas in Austin. Originally libMesh was designed as an extensible platform for AMR capable of supporting adaptive h with p-type refinement for Galerkin, Petrov Galerekin (SUPG) and Discontinuous Galerkin finite element formulations [1]. By taking advantage of OOP practices we have defined a redistribution ‘object’ to incorporate redistribution. The mesh redistribution methodology and software is based on the mesh optimization formulation in [2,3,4]. Error indicators computed for adaptivity in libMesh are incorporated into the optimization functional for cell quality in the mesh redistribution scheme via a weighting function for the variational optimization process. The optimization problem is solved using a damped Newton method in an algorithm that also untangles invalid ‘folded’ meshes. This functionality enables automatic mesh redistribution. In the “Texas Two Step” algorithm we first apply redistribution on an initial coarse mesh to keep the nonlinear optimization problem small and then apply adaptive mesh refinement to the resulting graded mesh. In this way the redistribution process automatically clusters and aligns the mesh with layers and local solution features while preserving cell quality. Then the AMR scheme takes over and further refines the mesh through local subdivision. This process can be used with both isotropic AMR and anisotropic AMR. However, the use of the prior redistribution step alleviates the need for an anisotropic AMR scheme to some extent. Furthermore, it provides more efficient meshes than AMR alone and avoids solving large nonlinear optimization problems with redistribution alone. This strategy is applied to several representative finite element application problems including compressible inviscid applications for flows with shocks and problems with corner singularities or features due to surface tension effects modeled by the Laplace-Young equation.

References:
[1] B. Kirk, J. W. Peterson, R. H. Stogner, and G. F. Carey. libMesh: A C++ Library for Parallel Adaptive Mesh Refinement/Coarsening Simulations. 'Engineering with Computers Special Infrastructure Issue', in press 2006, DOI:10.1007/s00366-006-0049-3
[2] L. Branets. A Variational Grid Optimization Method Based on a Local Cell Quality Metric. PhD thesis, University of Texas at Austin, 2005.
[3] L. Branets and G. F. Carey. Smoothing and adaptive redistribution for grids with irregular valence and hanging nodes. In 'Proceedings of the 13th International Meshing Roundtable', pages 333-344, Williamsburg, VA, 2004.
[4] L. Branets and G.F. Carey. A local cell quality metric and variational grid smoothing algorithm. 'Eng. with Comp.', 21:19-28, 2005.

A log(L/s)-competitive Algorithm for No-Large-Angle Triangulation

Don Sheehy
Carnegie Mellon University
5130 Wean Hall
5000 Forbes Avenue
Pittsburgh PA 15213
United States of America
dsheehy@cs.cmu.edu

Todd Phillips
Carnegie Mellon University
5000 Forbes Avenue
Pittsburgh PA 15213
United States of America
tp517@cs.cmu.edu

Gary Miller
Carnegie Mellon University
5000 Forbes Avenue
Pittsburgh PA 15213
United States of America
glmiller@cs.cmu.edu

We explore the problem of producing a conforming triangulation of an arbitrary planar straight-line graph (PSLG) in which all angles of the resulting triangulation are bounded by a constant. We present the Overlay Stitch Meshing (OSM) algorithm, a simple to implement algorithm that gives the first log(L/s)-competitive size guarantee for no-large-angle triangulation. Here, L/s, also known as the spread of the input, is the ratio of the diameter of the input to the smallest distance between disjoint input features. Previously, only worst case bounds were known. The bound on the largest output angle is 170 degrees. The algorithm has three phases. 1. The Overlay Phase: A delaunay refinement algorithm produces a triangulation of the input vertices, ignoring input edges for now. 2. The Stitching Phase: The input edges are stitched into the overlay mesh, adding vertices at intersection points. In this phase some of the overlay edges are discarded if they only contribute intersection points forming large angles. 3. The Completion Phase: After the stitching phase, some of the faces are not triangles, but all have at most 6 sides. These faces must be triangulated to complete the triangulation. Although the output is not a Delaunay triangulation, OSM leverages Delaunay refinement technology without the usual caveats regarding input angles or O(n L/s) output size. Despite its simplicity, the algorithm admits strong guarantees. The competitive size analysis uses new tools for analyzing no-large-angle triangulations.

Isosurface Stuffing: Fast Tetrahedral Meshes with Good Dihedral Angles

Jonathan Shewchuk
Associate Professor
University of California at Berkeley
625 Soda Hall
CS Division, UC Berkeley
Berkeley CA 94720-1776
United States of America
jrs@cs.berkeley.edu

Francois Labelle
University of California at Berkeley
535 Soda Hall
CS Division, UC Berkeley
Berkeley CA 94720-1776
United States of America
flab@cs.berkeley.edu

The isosurface stuffing algorithm fills an isosurface with a uniformly sized tetrahedral mesh whose dihedral angles are bounded between 10.7 degrees and 165 degrees. All vertices on the boundary of the mesh lie on the isosurface. The algorithm is whip fast, numerically robust, and easy to implement because, like Marching Cubes, it generates tetrahedra from a small set of precomputed stencils. A variant of the algorithm creates a mesh with internal grading: on the boundary, where high resolution is generally desired, the elements are fine and uniformly sized, and in the interior they may be coarser and vary in size. Isosurface stuffing is the first algorithm that simultaneously copes with boundaries of complex shape and rigorously guarantees the suitability of its tetrahedra for finite element methods. Our angle bounds are guaranteed by a computer-assisted proof. We illustrate the use of isosurface stuffing for dynamic remeshing in a fluid simulation with moving liquid surfaces.

Formulation of Delaunay Field for Hexahedral Meshing

Damrong Guoy
Research Scientist
University of Illinois at Urbana-Champaign
2253 Digital Computer Lab
1304 W. Springfield Avenue
Urbana IL 61801
United States
guoy@uiuc.edu

Jeff Erickson
Associate Professor
University of Illinois at Urbana-Champaign
3304 Siebel Center for Computer Science
201 N. Goodwin Avenue
Urbana IL 61801
United States
jeffe@cs.uiuc.edu

We present a new concept motivated by Delaunay triangulation that is relevant to hexahedral meshing. Given an n-dimensional domain, we apply empty-sphere criterion in appropriate functional form and construct a scalar field, called Delaunay field, on the domain. Delaunay field has zero set coincides with domain boundary, and its gradient vector field is continuous almost everywhere. The mesh elements can be generated in a promising advancing-front fashion by advancing the level set according to the gradient flow. Some mathematical properties of Delaunay field will be presented. A simple numerical procedure to construct Delaunay field will be described. We will also discuss outstanding issues related to critical points and topological noise.

Geometry Based Parallel Mesh Generation

Joe Walsh
VP Business Development
Simmetrix Inc.
10 Halfmoon Executive Park
Clifton Park NY 12065
United States of America
jwalsh@simmetrix.com

There has been a growing need in the analysis community to solve lerger and larger problems, with some problems approaching or surpassing billions of elements. This has resulted in an advent of a wide range of parallel solvers, however, parallel mesh generation has generally not kept up with the solver technology. This presentation will discuss some of the issues associated with providing parallel mesh generation over a range of problems. These issues will include the selection of SMP or DMP architectures, the need for mesh partitioning independent of the parts (i.e. large flow volumes, large EM field volumes) and the need to drive both surface and volume meshing in parallel from the geometry. Example results of geometry based parallel mesh generation using Simmetrix Parallel MeshSim technology will also be presented.

3D Surface Mesh Regeneration Considering Curvatures

Antonio Miranda
Researcher
Tecgraf-PUC/Rio
Av. Prof. Fausto Moreira, 306, Ap 1407
Bairro: Barra da Tijuca
Rio de Janeiro RJ 22793-340
Brazil
amiranda@tecgraf.puc-rio.br

Anthony Ingraffea
Cornell Fracture Group
643 Rhodes Hall
Ithaca NY
United States of America
ari1@cornell.edu

Luiz Fernando Martha
Tecgraf/PUC-Rio
Rua Marquês de São Vicente, 225
Rio de Janeiro RJ 22453-900
Brazil
lfm@tecgraf.puc-rio.br

Paul Wawrzynek
Cornell Fracture Group
641 Rhodes Hall
Ithaca NY
United States of America
wash@fac.cfg.cornell.edu

This work describes an automatic algorithm for generating unstructured triangulations for arbitrarily shaped three-dimensional surfaces. This generic surface can be: a triangulated mesh, a set of points or an analytical surface (such as a collection of NURBS patches). To be generic, the algorithm requests three support functions as input. The first, given the position of the point, returns the desired size of element. The second, given the current edge in the boundary contraction algorithm, receives the ideal apex point that forms a triangle. The third, given a point in space, returns the closest point on the geometrical support surface. The algorithm incorporates aspects of well-known meshing procedures of the advancing front technique and includes some original steps. The advancing front technique starts with a boundary that is represented by a set of oriented edges. Triangular elements are “extracted” or “pared” from the region one at a time. As each element is extracted, the boundary is updated and the process is repeated. The procedure terminates when the entire region is meshed. In this algorithm, the advancing front process is divided into two phases to ensure generation of valid triangulations. In the first phase, a geometry-based element generation is pursued to generate elements of optimal shape. After this ideal phase is exhausted, and no more optimal elements can be generated, a topology-based element generation takes place, creating valid, but not necessarily well shaped, elements in the remaining region. To make the procedure reliable for complex geometries, some additional tests are required. In the case where the algorithm is used to re-mesh triangulated surfaces, two auxiliary data structures, an octree and Alternating Digital Tree (ADT), are used to create efficient implementations of the three support functions. The octree is used for two purposes. First, to provide local policies used to define the discretization of the surface mesh. The second is to define the sizes of triangular elements to be generated during the advancing-front procedure. The ADT structure is used to quickly determine which triangles in the original triangulation are "close" to a point and to identify high curvatures in the original surface. In these locations, a smoothing transition of elements is required. Finally, examples of re-meshing are presented in order to verify the algorithm proposed in this work.

An Immersive Topology Environment for Generalized Hex and Tet Meshing of CAD Models

Steven Owen
Sandia National Laboratories
PO Box 5800 M.S. 0376
Albuquerque NM 87185
United States of America
sjowen@sandia.gov

Brett Clark
Sandia National Laboratory
PO Box 5800, MS 0376
Albuquerque NM
United States of America
bwclark@sandia.gov

Intermittent users of mesh generation technology often require significant time and overhead to build a mesh suitable for simulation. For a given CAD model, the user may be required to perform a number of complex decisions and operations to successfully prepare the model for analysis. Although current software tools may provide the needed capabilities, without a thorough understanding of their use and limitations, this process can become frustrating and time consuming, requiring a major investment of time to become proficient. The Immersive Topology Environment for Meshing (ITEM) is a wizard-like environment, built on top of the CUBIT Geometry and Meshing Toolkit, intended to help guide the intermittent user through the process of preparing a CAD model for simulation. ITEM is focused on three main objectives: 1) guiding the user through the simulation model preparation workflow; 2) providing the user with intelligent options based upon the current state of the model; and 3) where appropriate, automating as much of the process as possible. To accomplish this, a diagnostic+solution approach is taken. Based upon diagnostics of the current state of the model, specific solutions for a variety of common tasks are provided to the user. Some of these tasks include geometry simplification, small feature suppression, resolution of misaligned assembly parts, decomposition for hex meshing, and source and target selection for sweeping. The user may scroll through a list of intelligent solutions for a specific diagnostic and entity, view a graphical preview of each solution and quickly perform the solution to resolve the problem. In many cases, automatic solutions for these tasks can be generated and executed if the user chooses. This talk will discuss the various diagnostics and geometric reasoning algorithms and approaches taken by ITEM to determine solutions for preparing an analysis model. Several real-world examples will be used to demonstrate the effectiveness of the environment.

Thistle mesh generator

Juan Almara
SciSolver.com
San Juan 82
Parana ER E3100AFB
Argentina
almara@usa.com

This paper describes a tetrahedra mesh generator, with uniform or variable density, for general domains in three dimensions, supporting multiple sub domains with holes [1]. The input data is specified as space polygons with boundary point spacings. It provides high automation in the mesh generation process, as minimum user intervention is required. Borrowing key concepts from other mesh generators [2][3], more than one internal point is included at a time using a heuristic method, testing proximity constraints before insertion. This operation is repeated until no points can be added. A pseudorandom, reproducible variation, can be incorporated to each nodal point, to blur geometrical patterns. Then, the mesh is generated using a Delaunay tetrahedralization [4][5][6][7], and is partially optimized. The algorithm can produce regular high quality meshes, and is able to discretize complex domains, particularly when locally graded meshes are needed, and is capable of handling very large scale variations in node density, for fluid mechanics applications. The implementation is based on dynamic data structures. The output depends on the final use: may be a list of tetrahedrons, including geometric properties, with a sub list of related neighbors, for standard finite elements processing, or a list of field nodes and related neighbors in the support domain, for meshless methods. Several examples are included that demonstrate the capabilities of the technique, with timings and quality metrics.

[1] Siqueira, Marcelo, "An Introduction to Algorithms for Constructing Conforming Delaunay Tetrahedrizations", http://www.cis.upenn.edu/grad/documents/siquera.pdf
[2] Kornberger, Bernhard, "Replacing polygons by circles and solids by balls", http://www.ig.jku.at/strobl/2006-06bkornberger.pdf
[3] Shimada, Kenji, Gossard, David, "Bubble Mesh: Automated Triangular Meshing of Non-Manifold Geometry by Sphere Packing", http://www.andrew.cmu.edu/user/shimada/papers/95-acm-sm-shimada.pdf
[4] Crespin, B., "Dynamic Triangulation of Variational Implicit Surfaces using Incremental Delaunay Tetrahedralization", http://www.msi.unilim.fr/~crespin/lyon_recherche.html
[5] Pav, Steven, Walkington, Noel, "Robust Three Dimensional Delaunay Refinement", http://www.imr.sandia.gov/papers/imr13/pav.pdf
[6] Cheng, Siu-Wing, Poon, Sheung-Hung, "Three-Dimensional Delaunay Mesh Generation", http://www.cs.ust.hk/~scheng/homepage/pub-area.htm
[7] Cohen-Steiner, David, Colin de Verdiere, Eric, Yvinec, Mariette, "Conforming Delaunay Triangulations in 3D", http://www.di.ens.fr/~colin/textes/00conf.pdf