Advances in Engineering Software xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

Survey

Automatic generation of structured multiblock boundary layer mesh for aircrafts

Fengshun Lu*,[a], Yufei Pang[a], Xiong Jiang[a], Junfeng Sun[a], Yong Huang[a], Ziwei Wang[a], Jiali Ju[b,c]

[a] *China Aerodynamics Research and Development Center, Mianyang, Sichuan 621000, China*
[b] *State Grid Yishui Power Supply Company, Linyi, Shandong 276400, China*
[c] *ROMTEK Innovation Limited Company, Beijing 100102, China*

ARTICLE INFO

Keywords:
Automatic mesh generation
Boundary layer
Structured grid
Multiblock
Transfinite interpolation

ABSTRACT

Structured meshes are widely utilized in the aircraft industry, despite their requirement for significant interaction between the engineers and the commercial software used in the generation process. In this research, we introduce part of our achievements while investigating the automatic generation techniques for structured meshes of arbitrary aircrafts. First, we present a data structure that eliminates the many-to-many relationship between various physical entities representing the discrete mesh. Second, we propose a novel method that can automatically generate a structured multiblock boundary layer mesh using an input surface mesh. Finally, we verify the proposed method with three cases (i.e., convex/concave steps, a four-rudder missile and the F6 wing-body half model) by the metrics Jacobian, included angles and aspect ratios. These experiments show that our proposed method can automatically generate structured multiblock boundary layer mesh for aircrafts.

1. Introduction

A computational mesh is usually needed when simulating aerodynamic flows with the computational fluid dynamics (CFD) technique. It is well acknowledged that generating meshes for complex geometries is extremely difficult and time consuming. Consequently, the automatic mesh generation is rather important to the efficiency and throughput of simulations in the aerospace and other industries [1].

The investigation of mesh generation techniques has lasted for many decades since Winslow [2] numerically constructed a triangle mesh by solving the Laplace's equation in 1966 [3–6]. There are plenty of academic journals and conferences that include topics about mesh generation, such as the annual International Meshing Roundtable [7] since 1992. Herein the computational meshes are divided into the structured and unstructured so as to review the recent progress in automatic mesh generation. Of course, this partition of mesh types is not entirely appropriate, since there are hybrid meshes that consist of both structured and unstructured parts. Despite the expense in computational speed and memory, unstructured meshes have much flexibility in placing grid points, which offers the greatest possibility for completely automatic mesh generation [8]. They have witnessed significant progress in the automatic generation techniques [9–16]. As an example of recent progress, researchers have addressed the challenges confronted while generating unstructured meshes in parallel [17,18].

Compared with unstructured meshes, the structured counterparts demand much more human intervention and the corresponding generation time is usually measured with weeks rather than hours for complex geometries. Nevertheless, the structured meshes are still widely utilized in the aircraft industry because they can efficiently achieve accurate solutions, especially for viscous flows of complex configurations. There have been many improvements in the automatic generation of structured meshes [19,20]. Allen [21] presented an automatic multiblock generation scheme for the helicopter simulation on condition that the default block structures are predefined. Benoit and Peron [22] proposed a method of automatically generating structured near-body grid around two-dimensional bodies described by polylines, which was verified with a high-lift profile. Pandya et al.[23] introduced a scripting framework to automate the surface and volume mesh generation for rocket geometries, whose input was a computer aided design (CAD) solid geometry model. ElSheikh et al.[24] presented a semi-formal design of mesh generation systems, which strictly followed the software engineering principles. In a recent survey of the development practices for mesh generation and mesh processing software, Smith et al.[25] reported that five of the evaluated 27 projects could deal with three-dimensional meshes, namely Discretizer [26], Mefisto [27], MMesh3D [28], OpenFlipper [29] and Overture [30].

From the aforementioned publications, we obtain two observations about the achievements in automatic generation of structured meshes.

---

* Corresponding author.
E-mail addresses: lufengshun@cardc.cn, lufengshun@nudt.edu.cn (F. Lu).

http://dx.doi.org/10.1016/j.advengsoft.2017.10.003
Received 11 January 2017; Received in revised form 23 August 2017; Accepted 8 October 2017

Please cite this article as: Lu, F., Advances in Engineering Software (2017), http://dx.doi.org/10.1016/j.advengsoft.2017.10.003

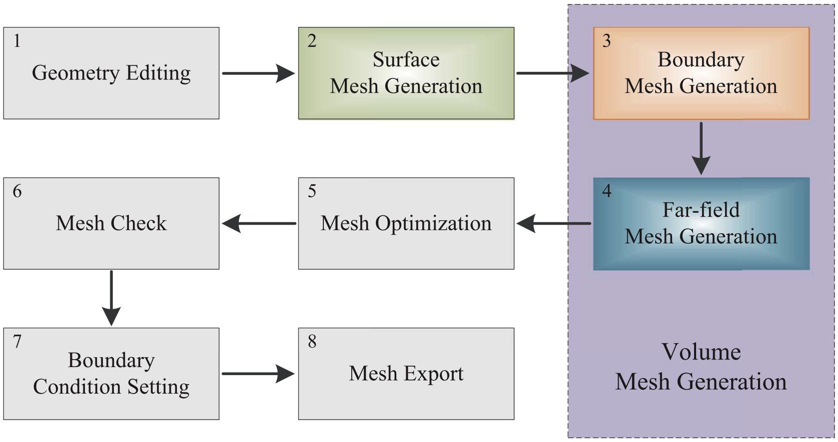

Fig. 1. Mesh generation workflow in *Spider*.

On the one hand, the application scenarios are often targeted for certain kind of geometries. For instance, Allen's work [21] mainly dealt with the mesh generation issue in helicopter simulations, while Pandya et al. [23] addressed the automatic mesh generation for rocket geometries. On the other hand, few projects address the automatic generation of structured multiblock meshes. For example, MMesh3D [28] is a semi-structured mesh generator for domains with orography and bathometry, and Overture [30] provides a module to generate overlapping grid. Instead, we put more emphasis on generality and endeavor to investigate the automatic structured mesh generation techniques for arbitrary aircrafts.

Nowadays most of the structured meshes for complex geometries are produced by lots of interactions between the engineers and commercial software like GridPro [31] and Pointwise [32]. Our team is now developing one piece of general meshing software that is called *Spider*. It has a friendly graphical user interface (GUI) and can facilitate the mesh generation for industrial applications. Interested readers can refer to Appendix A, Appendix B and Appendix C for the system architecture, software design and software testing of *Spider*, respectively.

As depicted in Fig. 1, *Spider* generates meshes in eight steps, all of which can be performed by a moderate amount of interactions between engineers and the software GUI. In this research, we concentrate on the third step and have developed an automatic procedure for generating the boundary layer mesh with an input surface mesh.

We herein present part of the work conducted in the undergoing *Spider* project and our contributions include:

- A data structure that eliminates the many-to-many relationship between various physical entities representing the discrete mesh.
- A method that automatically generates the structured multiblock boundary layer mesh.
- Detailed verification of mesh quality with the metrics Jacobian, included angles and aspect ratios.

The remainder of the paper is organized as follows. Section 2 introduces the related work on the boundary-layer mesh generation. Section 3 details our proposed method and demonstrates the three steps needed to automatically generate the boundary layer mesh. The proposed method is verified with three cases by several mesh quality metrics in Section 4. Finally, Section 5 concludes our paper and outlooks our future work.

2. Related work

The problems in physics and fluid mechanics [33,34] usually show relatively strong gradients near the immediate vicinity of a bounding surface. Therefore, extremely fine meshes have to be generated for the boundary layer so as to capture the inherent features of the investigated problems. Many researchers have addressed the issues on generating boundary layer meshes and proposed many useful methods [34–41], of which the most widely used one is the advancing-layer method [3,42] or its variants [43,44].

Lohner [33] proposed a mesh generation technique for the Navier–Stokes calculations involving complex geometries. He combined the semi-structured and unstructured meshing techniques to generate one single unstructured mesh, which accommodated the strengths of these two techniques and automatically avoided the problems usually encountered when meshing surfaces with high curvature. In another research study [35], he implemented a procedure for the generation of highly stretched grids suitable for Reynolds-Averaged Navier–Stokes (RANS) calculations. His procedure first generated an isotropic mesh and then successively enriched it with points to achieve highly stretched elements, among which the reconnection was carried out using a constrained Delaunay approach. Aubry and Lohner [41] extended the work in literature [33] and generated high aspect ratio volume grids on surfaces with ridges and corners.

Hassan et al.[34] proposed a method of generating general tetrahedral meshes suitable for use in viscous flow simulations, which could be accomplished in two steps. First, a number of unstructured layers of highly stretched elements was generated in the vicinity of solid walls. Then, the remainder of the domain was discretized by a standard advancing front procedure [45]. The proposed method was applied to a number of three-dimensional aerodynamic configurations and demonstrated the validity of the generated mesh through a three-dimensional laminar viscous flow analysis.

Garimella and Shephard [42] presented a generalized advancing layers method for meshing complex non-manifold geometric domains with anisotropic elements near the surface. It allowed multiple growth curves to emanate from each surface node, which eliminated the restriction that boundary layer prisms sharing a surface mesh edge or vertex must be joined along their sides. Note that the gaps between prisms caused by the multiple growth curves should be filled, otherwise highly anisotropic faces would be exposed to the following isotropic unstructured mesh generator.

Ito and Nakahashi [46] introduced an automated approach to generate unstructured hybrid grids comprised of tetrahedra, prisms and pyramids for high Reynolds number flow computations. With respect to the boundary layer mesh, an initial surface normal vector was calculated at each node and performed as a marching direction for the prismatic layer generation. Specially, two marching directions were given to the nodes whose folding angles at each edge on no-slip walls were more than 150°. In the following research study [47], Ito et al. proposed a hybrid mesh generation approach using an advancing layer method to generate the boundary layer mesh and an advancing front method to fill the rest of the domain. Note that multiple normals were extruded from a single surface node.