Advancing Layer Surface Mesh Generation

by

Jasmeet Singh

B. Tech, Indian Institute of Technology (BHU), Varanasi, 2015

A THESIS SUBMITTED IN PARTIAL FULFILLMENT
OF THE REQUIREMENTS FOR THE DEGREE OF

Masters in Applied Science

in

THE FACULTY OF GRADUATE AND POSTDOCTORAL STUDIES

(Mechanical Engineering)

The University of British Columbia

(Vancouver)

December 2019

The following individuals certify that they have read, and recommend to the Faculty of Graduate and Postdoctoral Studies for acceptance, the thesis entitled:

Advancing Layer Surface Mesh Generation

submitted by **Jasmeet Singh** in partial fulfillment of the requirements for the degree of **Masters in Applied Science** in **Mechanical Engineering**.

Examining Committee:

Carl Ollivier-Gooch, Mechanical Engineering
Supervisor

XYZ, Mechanincal Engineering
Supervisory Committee Member

PQR, LMN Department
Supervisory Committee Member

Abstract

Use of unstructured meshes in the simulation of a computational field to solve for a real world problem is ubiquitous. Specially, solving fluid flow over bodies like an airplane or a turbine computationally requires a well discretized domain, or a mesh around the surfaces of these bodies. In Computational Fluid Dynamic (CFD) simulations over these surfaces, the flow at the viscous-boundary layer of the surface is very important as the gradients in the normal direction of the flow are sharp and are orders of magnitude higher than the gradients in the tangential direction of the flow. Hence, resolving the flow field in the boundary layer is vital for accurate simulation results.

A plethora of 3D boundary layer mesh generation techniques start off from a discretization of the surface. A majority of these techniques either use surface inflation or iterative point placement normal to the surface to generate the advancing layer 3D mesh. Generating boundary layer meshes in 3D depends on the quality of the underlying surface discretization. We introduce a technique to generate advancing layer surface meshes which would improve the mesh generation pipeline for 3D mesh generation. The technique takes an input triangulation of the surface, which is fairly easy to get, even for complex geometries. Surface segments are identified and these segments are meshed independently using a advancing-layer methodology. For each surface segment, a mesh is generated by advancing layers from the identified boundaries to the surface interior while deforming the existing triangulation. As the mesh-generation technique introduced here produces a closed-mesh, we get a valid mesh at each iteration of layer advancement.

The method introduced to generate advancing layer meshes produces semi-structured quad-dominant meshes with the ability to have local control over the aspect ratio of mesh elements at the boundary curves of the surface. Semi-structured 2D anisotropic meshes in the boundary layer regions have been shown to have superior fluid flow simulation results. However, the discretization of the surfaces poses challenges in replicating the same for volume meshes. Point placement in layers, local reconnection, front recovery, front collision handling and smoothing techniques used in the study help produce a valid surface mesh at each step of mesh generation. We demonstrate the ability of the meshing algorithm to tackle fairly complex geometries and coarse initial surface discretization.

Lay Summary

Discretization of geometries using a non-regular arrangement of mesh elements, called unstructured mesh generation is used widely for simulating flow over various objects in industry and government. The region near the surfaces of objects is particularly important during the simulation process because of the extreme non-linearity in the flow characteristics near the boundaries of objects. Hence, generating a well-discretized boundary layer mesh is key to superior flow simulation results. 3D mesh generation methodologies use a surface mesh as the starting point. Hence, the surface mesh plays an important role in the overall fluid flow simulation process.

A method to generate an advancing layer surface mesh is introduced in this paper. This method could be used to generate advancing-layer quad-dominant surface meshes with the required aspect ratio at sharp corners of the surface. 3D mesh generation procedures could use this mesh to produce advancing layer mesh or any other mesh. Example meshes are generated and shown to handle complex geometries.

Preface

All the work presented in this thesis is an intellectual product of a close working relationship between Jasmeet Singh and Dr. Carl Ollivier-Gooch. The implementation of the methods, the data analysis, and the manuscript preparations were done by Jasmeet Singh with invaluable guidance from Carl Ollivier-Gooch throughout the process.

Table of Contents

List of Tables

List of Figures

Glossary

This glossary uses the handy `acroynym` package to automatically maintain the glossary. It uses the package's `printonlyused` option to include only those acronyms explicitly referenced in the LaTeX source. To change how the acronyms are rendered, change the `\acsfont` definition in `diss.tex`.

Acknowledgments

I would like to acknowledge and thank all the people who were a part of my graduate degree at UBC. These include the professors who taught me in lectures, my classmates, my labmates, people in the broader graduate community with whom I met at various academic and social events, my friends and family.

I would like to thank my supervisor, Dr. Carl Ollivier-Gooch.

Thesis Outline

1. Introduction

   - Introduction to Meshing

   - Introduction to unstructured meshing and its importance

   - Boundary Layer Phenomenon and its importance

   - Meshes that deal with such scenarios.

   - 2D previous works

   - 3D previous works

   - Surface Mesh generation methods

     - Parametric Mapping

     - Direct 3D methods

2.

Chapter 1

Introduction

If I have seen farther it is by standing on the shoulders of Giants. — Sir Isaac Newton (1855)

Computational Fluid Simulations (CFD) is a field of study where scientists and engineers architect new ways to numerically solve fluid flow equations. Before the advent of computers, numerical solutions of differential equations was done by hand. This lead to a great deal of work in the direction of creating faster algorithms to solve differential equations. An example is the development of the Fast Fourier Transform (FFT) by Cornelius Lancoz to increase the computation speed of Discrete Fourier Transform (DFT). However, since the development and advancement of computers, engineers had a significant amount of compute power to work with. This lead to the development of highly accurate methods (as compared to before) to simulate flow over various objects. These simulations have since gotten bigger and better, typically including millions of degrees of freedom, even starting to touch a billion in regular industry use.

The equations which govern the conservation of mass, momentum and energy of a moving fluid also called Navier-Stokes equations are solved in the given domain to simulate fluid flow in that domain. So as to numerically solve these equations, we need a discretization of the given domain. This discrete basis required to solve the Navier-Stokes equations is called a mesh. Simply put, a mesh is a collection of points, lines and cells that together construct the space around a body in a fluid flow.

Appendix A

Supporting Materials