XII Congresso 22 a 24 de setembro de 2004
Interno de Iniciação Científica da UNICAMP
Ginásio Multidisciplinar da UNICAMP

CNPq

UNICAMP

T769

AUTOMATIC MESH REFINEMENT FOR THE STUDY OF NAVIER-STOKES FLUID FLOWS

Luís Guilherme Mello Décourt (Bolsista UNIEMP/EMBRAER) e Prof. Dr. Philippe Remy Bernard Devloo (Orientador), Faculdade de Engenharia Civil, Arquitetura e Urbanismo – FEC , UNICAMP

Mesh generation is a fundamental step of the finite element method. The mesh generation procedure consists in dividing a computational domain, according to some established patterns, into smaller and better known geometric forms (ex: triangles or quadrilaterals); in many occasions it is desirable to divide elements in given regions to increase the precision of simulation of a property that we wish to study (considering that each element of a mesh consists in a source of information for the calculations). This sub-division of elements is called mesh refinement, and this work consists in using mesh refinement techniques to generate automatically refined longitudinal meshes in boundary layers (i.e. directional refinement); the resulting meshes are used to improve the accuracy of Navier-Stokes fluid simulations of the boundary layer of an aircarft wing. Previously we developed a software that generated images of the refinement patterns; visualization is useful to analyze the quality of the refined meshes. Then, based in that analysis, we could start the refinement procedure. All the software development of this work uses the object oriented programming philosophy (C++).
CFD - Mesh Refinement - C++ Programming