8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineeering (ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

DEVELOPMENT OF A LIGHT PREPROCESSOR FOR INTERACTIVE CFD SIMULATIONS.

Manuel Julio Garcia Ruiz[1], *Juan Fernando Duque Lombana[2] and Pierre Boulanger[3]

[1] Applied Mechanics Group
EAFIT University
Carrera 49 7 Sur-50
Medellín - Colombia
mgarcia@eafit.edu.co

[2] Applied Mechanics Group
EAFIT University
Carrera 49 7 Sur - 50
Medellín - Colombia
jduquelo@eafit.edu.co

[3] AMMI Lab
University of Alberta
221 Athabasca Hall, Edmonton, Alberta - Canada
pierreb@cs.ualberta.ca

Key Words: *Computational Steering, Computational Fluid Dynamics, Interactive Design, Meshing, Fixed Grid, OpenFOAM.*

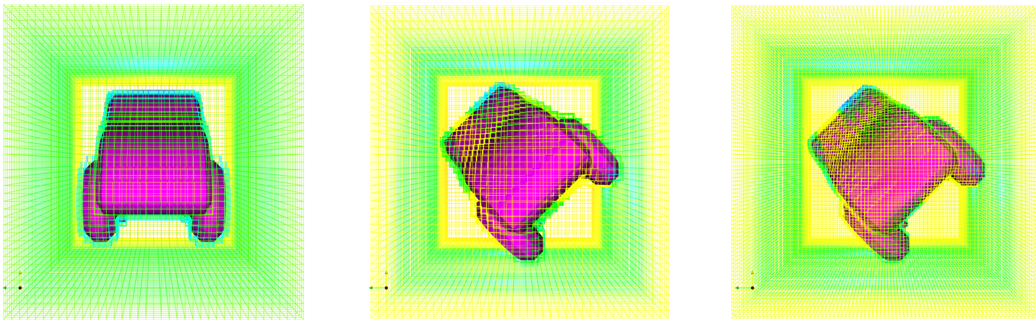
ABSTRACT

This paper introduces a fast hexahedral pre-processor for an interactive Computational Fluid Dynamics (CFD) environment which is intended for performing stable and fast CFD simulations valid during the early stages of the design process [1]. The goal behind this automatic preprocessor is to avoid the need of having to manually remesh and set boundary conditions for each case scenario present during the interactive simulation, thus allowing the designer to center his efforts during the analysis phase.

The application contains an embedded fixed grid preprocessor and a third party surface meshing algorithm adapted for geometry acquisition. The automatic CFD domain meshing is based on fixed grid meshing algorithms [3] for hexahedrons, where the elements that belong to the boundary will be fitted via a vertex projection methods [5] that will hold the correctness of the current topology and are fast enough to guarantee low computing times. A parallel implementation of the method is discussed in detail.

At present the proposed design environment solves the incompressible laminar Navier-Stokes equations using the *PISO* algorithm via the OpenFOAM Libraries [4]; the environment is inherently transient, interactions and changes to the geometry vary the initial conditions transforming the flow problem into new one (See figure 1). This information is automatically updated into the CFD module for consideration into the next time step. The results are shown to the user as they are available from the solver with the aim of soft-real-time visual feedback.

Finally, some results obtained from the computations of a possible flow case scenario in a Virtual Wind Tunnel (VWT) application are presented. This results verify the robustness and stability of the messher. This examples also show how interactive CFD simulations are a very useful tool for the early design stages where highly accurate analysis are not necessary.



(a) Original Orientation. (b) Test 1 after performing 45° rotation (c) Test 3 after performing 45° rotation

Figure 1: Example of interactive rotation operations

REFERENCES

[1] A. KELA, R. PERUCCHIO, and H. VOELCKER. “Towards an automatic finite element analysis”. *Computers in Mechanical Engineering*, pages **5571**, 1986.

[2] OH. David. “The java virtual wind tunnel -a two dimensional computational fluid dynamics simulation-”. Web, `http://raphael.mit.edu/Java` 2001.

[3] Manuel Garcia. “Fixed Grid Finite Element Analysis in Structural Design and Optimisation”. *PhD thesis, Department of Aeronautical Engineering*, The University of Sydney, March 1999.

[4] Weller, H.G.; Tabor G.; Jasak, H. and Fureby, C.: A Tensorial Approach to CFD using Object Orientated Techniques, Computers in Physics, 1998 v 12 n 6, pp 620 - 631

[5] D.-Y. Kwak and Y.-T. Im. “Hexahedral mesh generation for remeshing in three-dimensional metal forming analyses”.*Journal of Materials Processing Technology*, No. 138 (2003) 531537.