Proceedings of the ASME 2011 International Mechanical Engineering Congress & Exposition

IMECE2011

November 11-17, 2011, Denver, Colorado, USA

AN INTERACTIVE BOUNDARY LAYER MODELING METHODOLOGY FOR AERODYNAMIC FLOWS

Lelanie Smith, University of Pretoria, Pretoria, Gauteng, South Africa

Josua P Meyer, University of Pretoria, Pretoria, Gauteng, South Africa

Oliver F Oxtoby, Council for Scientific and Industrial Research, Pretoria, Gauteng, South Africa

Arnuad G Malan, Council for Scientific and Industrial Research, Pretoria, Gauteng, South Africa

ABSTRACT

Computational Fluid Dynamics (CFD) simulation is a computational tool for exploring flow applications in science and technology. Of central importance in many flow scenarios is the accurate modeling of the boundary layer phenomenon. This is particularly true in the aerospace industry, where it is central to the prediction of drag. Modern CFD codes as applied to modeling aerodynamic flows have to be fast and efficient in order to model complex realistic geometries. When considering viscous flows the boundary layer typically requires the largest part of computational resources. To simulate boundary layer flow with most current CFD codes requires extremely fine mesh spacing normal to the wall and is consequently computationally very expensive. Boundary layer modeling approaches have by contrast received relatively little attention, while having the potential of offering considerable computational cost savings. One boundary layer method which has proven to be very accurate is the two-integral method of Drela (1986). Coupling the boundary layer solution to inviscid external flow is, however, a challenge due to the Goldstein singularity, which occurs as separation is approached. We propose to develop a new method to couple Drela's twointegral equations with a generic outer flow solver in an iterative fashion. We introduce an auxiliary equation which is solved along with the displacement thickness to overcome the Goldstein singularity without the need to solve the entire flow domain simultaneously. In this work the incompressible Navier-Stokes equations will be used for the outer flow. In the majority of previous studies the boundary layer thickness is simulated using a wall transpiration boundary condition at the interface between viscous and inviscid flows. This boundary condition is inherently non-physical since it adds extra mass into the system to simulate the effects of the boundary layer. Here, we circumvent this drawback by the use of a mesh movement algorithm to shift the surface of the body outward without regridding the entire mesh. This replaces the transpiration boundary condition. The results obtained show that accurate modeling is possible for laminar incompressible flow and that the solutions obtained compare well to similarity solutions in the cases of flat and inclined plates and to the results of a NACA 0012 airfoil produced by the validated XFOIL code (Drela and Youngren, 2001).