Second African Conference on Computational Mechanics – An International Conference – AfriCOMP11
January 5 – 8, 2011, Cape Town
A.G. Malan, P. Nithiarasu and B.D. Reddy (eds)

# EFFICIENT MODELLING OF AERODYNAMIC FLOWS IN THE BOUNDARY LAYER FOR HIGH PERFORMANCE COMPUTING

**Lelanie Smith, Oliver Oxtoby*, Arnaud G. Malan* and Josua P. Meyer****
Advanced Computational Methods, Council for Scientific and Industrial Research,
PO Box 395, Pretoria 0001, South Africa
*amalan@csir.co.za; *ooxtoby@csir.co.za
Department of Mechanical and Aeronautical Engineering, University of Pretoria,
Pretoria 0002, South Africa
**Josua.Meyer@up.ac.za

## SUMMARY

A unique technique to couple boundary-layer solutions with an inviscid solver is introduced. The boundary-layer solution is obtained using the two-integral method to solve displacement thickness with Newton's method, at a fraction of the cost of a full viscous solution. The boundary-layer solution is coupled to an existing inviscid solver. Coupling occurs by moving the wall to a streamline at the computed boundary layer thickness and treating it as a slip boundary, then solving the flow again and iterating. The proposed method obtained good results when compared to analytical solutions for flat and inclined plates presented in this paper.

**Key Words: Boundary layer, mesh movement, interactive coupling**

## 1. INTRODUCTION

Modern Computational Fluid Dynamics (CFD) codes as applied to modeling aerodynamic flows have to be fast and efficient. HPC in CFD offers software that improves the accuracy and speed of complex simulation scenarios, causing a new power of flow simulation. Current CFD packages based on Reynolds-Averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) methods either approximate the boundary layer effects or neglect viscous effects altogether on the surface of the body.

When considering viscous flows the boundary layer requires the largest part of computational resources, with RANS turbulence models being the most widely employed. Boundary layer approaches on the other hand have received relatively little attention, while having the potential of offering considerable computational cost savings.

## 2. METHOD AND RESULTS

The application of numerical techniques to solve the boundary layer equations allows treatment of more realistic geometries and the fulfillment of boundary conditions on the actual surface [1]. The various analyses and design algorithms that have been developed employ one of two distinct approaches to calculate aerodynamic flows: the full RANS approach or the interactive viscid-inviscid approach.

This study is concerned with the interactive approach which is based on coupling the solution of the viscous and inviscid flow equations with an interaction law. The viscous and inviscid flows are strongly coupled usually through a wall transpiration boundary condition on the inviscid flow. These interactive approaches are much less computationally expensive than solving the full Navier-Stokes equations and both show equal accuracy advantages.

The governing equations involve a reduced formulation of the Navier-Stokes equations known as the boundary layer equations. However, this simplification of the Navier-Stokes equations, are still non-linear which presents numerical difficulties when solved and difficulty increases as Reynolds number increases [6]. Approximation of the boundary layer problem is obtained through the momentum integral method attributed to von Karman. This approximation solution does not depend on the similarity assumption and the shape of the boundary layer velocity profile can change significantly. Consequently, this method can be extended to any flow regime with complex geometries and includes effects such as transition and separation [5].

Traditionally boundary layer methods are tied to the approximation of a thin viscous layer and the fact that the external velocity and pressure is known. However, for cases of limited separation these two parameters cannot be specified and that the boundary layer adjusts through the displacement thickness effect and therefore an interactive approach is needed [5].

Drela substituted equations for momentum and displacement thickness into the momentum integral equations to obtain the momentum integral equation in terms of momentum and displacement thickness. These equations are known as the two equation integral formulation based on dissipation closure for both laminar and turbulent flows, which eliminate the direct link between the profile shape and the pressure gradient, making it suitable for flow with strong interaction [3].

The two equations based on Drela's work is presented as:

$$\frac{d\theta}{dx_1} + \left(2 + H - M_e^2\right)\frac{\theta}{u_e}\frac{du_e}{dx_1} = \frac{C_f}{2} \qquad \text{[i]}$$

$$\theta\frac{dH^*}{dx_1} + \left[2H^{**} + H^*(1-H)\right]\frac{\theta}{u_e}\frac{du_e}{dx_1} = 2C_D - H^*\frac{C_f}{2} \qquad \text{[ii]}$$

Where θ is momentum thickness, H shape parameter, $C_D$ dissipation coefficient, $C_f$ skin friction coefficient, H* energy thickness shape parameter and H** is the density thickness. $U_e$ is the inviscid velocity.

The fundamental difficulty with these equations is that they contain more than two independent variables and hence some assumptions about the additional unknowns will have to be made to obtain a solution. Laminar closure equations empirically derived from the Falkner-Skan profile family, solves this problem accurately [2]. These closure equations are all dependent on the shape parameter and Reynolds number for the laminar case.

Backward differencing is used to discretise equations [i-ii] and they are then solved using a global Newton's method. The backwards differencing scheme is unconditionally stable and is an implicit difference method, which has the advantage that there are no limitations on the step size and a smaller number of iterations will be needed to reach convergence.

The flow solver, with which the boundary layer code is interacting, is based on the Characteristic-Based split (CBS) scheme, which is very similar to the original Chorin split and has similarities with other projection schemes widely employed in incompressible flow calculation. Viscous terms are included although they are negligible for inviscid flow and the flow solver is not required to resolve the boundary layer.

The interaction method between the solver and the boundary layer equations happen without the use of the traditional fictitious transpiration velocity. The flow solver converges to a solution where the residual is less than the specified tolerance. The solver then uses the boundary layer thickness obtained from the boundary layer solution to move the mesh to the outer edge of the boundary. The mesh is re-preprocessed and the residual is calculated again until the residual is less than the convergence tolerance. This might seem like a very crude approach but it is sufficient for small displacements in aerodynamic applications.

The results obtained show that for laminar flow the solution compare very well in the cases of flat and inclined plates. We present the results for the case of an inclined plate compared to the similarity Falkner-Skan solution [7]:

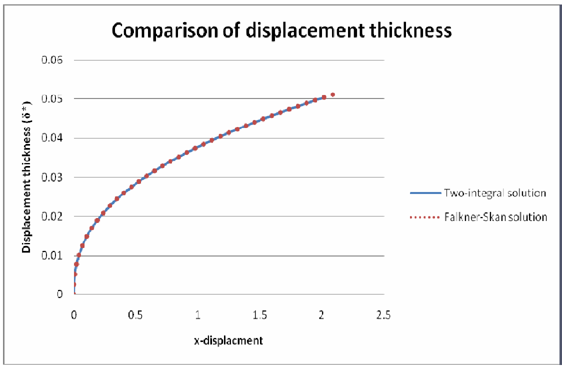

**Figure 1: Comparison between the two-integral solution and the Falkner-Skan similarity solution**

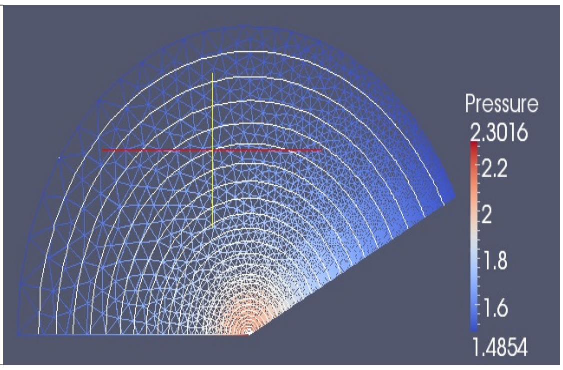

**Figure 1: Pressure contours and unstructured mesh ($\beta=0.3$)**

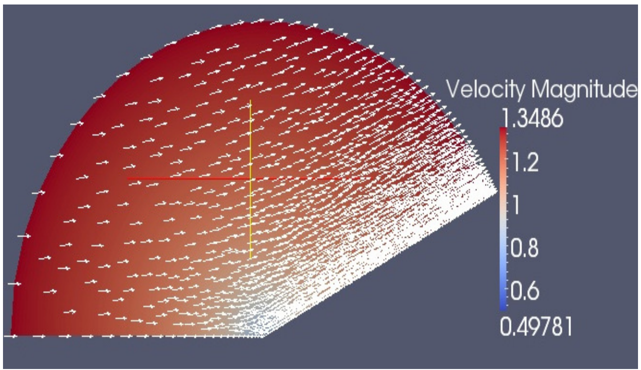

**Figure 2: Velocity distribution in m.s$^{-1}$ for the entire flow field (β=0.3)**

## 3. CONCLUSIONS

A methodology for the calculation of the boundary thickness and the interaction with the inviscid solver has been successfully developed for incompressible attached laminar flow in two dimensions.

## REFERENCES

[1] Cebeci T and Cousteix, J. 2005. *Modeling and Computation of Boundary Layer Flows.* Horizon Publishing. California. Los Angeles.

[2] Drela M. 1985. *Two-dimensional transonic aerodynamic design and analysis using the Euler equations.* PHD dissertation. Department of Aeronautics and Astronautics. Massachusettes Institute of Technology. Cambridge. Massachusetts.

[3] Drela M and Giles MB. 1987. Viscous-Inviscid analysis of transonic and low Reynolds number airfoils. *American Institute of Aeronautics and Astronautics Journal.* Vol 25, No 10, pp1347 – 1355.

[4] Hirsch C. 1995. *Numerical Comptutation of Internal and External Flows. Volume 2: Computational Methods for Inviscid and Viscous Flows*. John Wiley and Sons Ltd. Wiltshire.

[5] Katz J and Plotkin A. 2001. *Low Speed Aerodynamic. 2nd Edition*. Cambridge University Press. New York City.

[6] Schlichting H and Gersten K. 2000. *Boundary Layer Theory 8th edition*. Springer-Verlag. Berlin.

[7] White FM. 2006. *Viscous Fluid Flow 3rd Edition*. McGraw-Hill. Singapore.