NUMERICAL SOLUTION OF THREE-DIMENSIONAL ALL SPEED FLOWS; FEATURES OF THE MODEL

by

CLOVIS RAIMUNDO MALISKA[1]

ABSTRACT -- This paper reports the features of a three-dimensional numerical model for the solution of all Mach number flows. The model employs general curvilinear coordinate systems and adopts a special linearization of the mass flux which renders to the model the capability of solving low as well as high Mach number flows. It is also incorporated in the model the use of co-located variables with to aim of producing more compact and less memory storaging computer codes. The possibility of using multiblock techniques for very complex geometries is also embodied in the numerical model.

INTRODUCTION

The solution of three-dimensional flows over arbitrary bodies is a formidable task due to the complexity of the equation system to be solved. Additionally, if all Mach numbers flows are of interest new challenging aspects are introduced due to the changing of the physics of the flow when the Mach number reaches one. This paper reports the effort realized by the SINMEC - Numerical Simulation Group - of the Mechanical Engineering Department/UFSC, in the development of a computer model for the solution of all speed flows. Such a model would permit the analysis of subsonic flows with applications in the automotive industry, civil engineering, etc, transonic aerodynamics, with its major application in the design of the large commercial aircrafts and supersonic aerodynamics for the simulation of the flows in aerospace applications.

The existing numerical methods for the solution of fluid flow problems can be cast into two main categories; the ones designed for the solution of fluid flow problems where density is constant or a function of temperature only, classified here as incompressible, and the ones where the density changes primarily with pressure, classified as compressible. Here compressibility is understood as a dependence of density with pressure. This has a direct consequence in the choice of the evolution equation for each variable of the problem.

It is well know that the development of the models in the former class above mentioned occured among the analysts involved with the solution of convection heat transfer problems, while the models in the latter class belong to the field of aerodynamics. The possibilty of having robust methods for the solution of all speed flows motivated our research group to undertake a long

1 Professor: Departamento de Engenharia Mecânica - UFSC
SINMEC - Grupo de Simulação Numérica em Mecânica dos Fluidos e Transferência do Calor
Caixa Postal 476 - Campus Universitário
88049 - Florianópolis - SC- Brazil

term research project with this main objective. This paper reports the major decisions that had to be made up to now towards this goal.

MATHEMATICAL MODEL/DOMAIN DISCRETIZATION

The equation system which governs the three-dimensional flows of interest is formed by the following equations:

- mass conservation
- momentum conservation in the x-direction
- momentum conservation in the y-direction
- momentum conservation in the z-direction
- energy conservation

The unknows are; density, u, v and w, the cartesian velocity components, temperature (energy) and pressure. An state equation of the form $P = P(\rho,T)$ closes the problem.

The equation system is written in the generalized coordinate system whose coordinates lines fits the boundary of the domain, as can be seen in FIGURE 1, where a three-dimensional grid discretizes the domain over the front part of the VLS (brazilian satellite launcher vehicle). The use of boundary-fitted grids allows more flexibility and generality of the computer code. The equation system written in the (x,y,z) coordinate system is transformed to the (ξ,η,Γ) keeping the same dependent variables in the new coordinate system. The equation system for the 3D viscous flows can be found in Maliska, Silva and Marchi (1991). Several important features of the model are now briefly described.

THE ALL SPEED FLOW MODEL

When solving fluid flow problems it is crucial the type of linearization employed in the mass conservation equation. This linearization will, in fact, define in which class the method will be cast in.

Consider the term $\partial(\rho u)/\partial x$ of the mass conservation equation. The product ρu can be linearized in different ways. If the velocity is kept constant, becoming part of the coefficients, the mass conservation equation will be understood as an equation for density determination. That is, mass conservation will be forced to be satisfied through changes in density. If one is dealing with high Mach numbers flows this is the correct strategy. However, if low Mach number flows are to be solved, this procedure is inconsistent since density is constant for these flows. Using the mass conservation equation as an equation to determine density and then using the state equation to find pressure is an approach largely used by the numerical analysts involved with the solution of high speed flows.

In the other hand, if density is kept constant in the product ρu, u must change in order to satisfy mass conservation. It is clear that this procedure is appropriate only for incompressible flows. It is also evident that keeping ρ constant the mass conservation equation can no longer be used as an equation to find density. Rather, it is used to determine pressure trough an equation derived using the so-called pressure-velocity coupling methods.

The key difference between the two class of the methods can be also viewed as the way in which the pressure-velocity/density coupling is handled. For compressible flows the strong coupling is between pressure and density, while for incompressible flows it is between pressure and velocity. Therefore, it seems natural that if one wants to solve all Mach number flows the full coupling must be taken into account, that is, the dependence of density and velocity on pressure must be considered. This is equivalent of saying that in the term ρu both density and velocity must be kept active. This require a special linearization procedure. An equation for pressure can be then obtained from the mass conservation equation replacing ρ and velocity as a function of P. The resulting equation will be an equation for pressure similar to that obtained for incompressible flows.

This idea was first put forth by Harlow and Amsdem (1971) and by Patankar (1971). The idea was also applied by Van Doormaal (1985) using the cartesian coordinate system for the solution of incompressible and compressible flows over an obstacle. The same idea was used in the model discussed in this paper in the context of curvilinear coordinate systems. Initially two dimensional and axisymmetric flows were solved, as in Silva and Maliska (1988), Maliska and Silva (1989) and Silva (1991). Recently, 3D Euler flows were computed, as reported in Marchi, Maliska and Silva (1990) and Maliska, Silva, Marchi and Valerim (1991). Independently Karki and Patankar (1989) developed a similar model as the one here described.

The special linearization applied in the mass flow is now briefly described. In the curvilinear coordinate system the mass flux is given by

$$\dot{m} = \rho U \qquad (1)$$

where U is the contravariant velocity component. Since ρ and U must be kept as variable in equation (1) the following linearization fulfills the needs

$$\dot{m} = (\rho^{*} U + \rho U^{*} - \rho^{*} U^{*}) \qquad (2)$$

where the star means evaluation at previous iteration level. The above equation recovers both Mach number limits, allowing the use of the same numerical model even for transonic calculations. The mass fluxes calculated according to equation (2) are introduced into the discretized mass conservation equation to obtain an equation for pressure after replacing ρ and U as function of pressure. The solution procedure follows then the usual approach for incompressible flows.

STAGGERED x CO-LOCATED VARIABLES

When finite-difference techniques were extensively used to solve fluid flow problems all dependent variables used to be calculated at the same point on the grid. It is reported that this arrangement introduces a weak coupling between pressure and velocity. The solution to this problem came with the paper by Harlow and Welch (1965) where they proposed the well known staggered arrangement, where the velocity control volumes are shifted related to the continuity (pressure) control volumes. The arrangement, in fact, introduces a strong coupling between pressure and velocity allowing to obtain solutions free of pressure wiggles. Because of the tremendous popularity of Patankar's book, were the co-located arrangement is not recommended, the staggered arrangement remained as the choice for the potential user of finite volume methods for incompressible flows. Because of the strong dissemination of the use of the staggered grid almost no effort was done for about 15 years in order to develop co-located models which guarantee the strong coupling between pressure and velocity. The need of solving 3D flows and because of the cumbersome implementation of three-dimensional codes using staggered grids

efforts were recently devoted to obtaining robust methods based on co-located variables.

The pioneering work of Hsu (1981), Rhie (1981) and Peric, Kessler and Scheureur(1987), dealing with incompressible flows recovered the interest in the co-located arrangement. Developments realized by Marchi, Maliska and Bortoli (1989) and Bortoli (1990) for two-dimensional compressible flows and by Marchi, Maliska and Silva (1990) for three-dimensional compressible flows demonstrated that numerical schemes using co-located variables introduce simplicity in the code implementation, less computer memory and guarantee a strong coupling between pressure and velocity/density. It seems that the use of the staggered arrangement will be restricted to the old codes since for modern three-dimensional codes it is imperative the use of co-located variables.

MULTIBLOCK TECHNIQUE

The use of a multiblock technique is the most recent feature introduced in the model. When using a structured grid to discretize the domain, as shown in FIGURE 1, it becames difficult in certain 3D problems to create a 3D grid which covers the full domain and maps onto a single parallelepiped. In these cases two alternatives can be employed; the use of nonstructured grids or the multiblock technique. The multiblock procedure divides the solution domain in several simple 3D regions. In each of these regions it is possible to create a 3D grid which maps onto a parallelepiped. The strategy now is to solve for each sub-domain and then iterate among the sub-domain to take into account the transfer of the information between blocks. The key question in designing a multiblock procedure is exactly how to transfer the information from one block to the neighbouring block such that conservation of the quantities involved are preserved. This was succefully modelled and details can found in Maliska, Silva, Marchi and Carpes Jr (1991). Up to now 2D axisymetric solutions were obtained with this technique, as can be seen in FIGURE 3, where the Cp for the supersonic flow at Mach equal to 3.75 over the VLS is plotted. The circles are experimental results, the solid lines the solution using a single block and the dashed lines using multiblocks subdivided according FIGURE 2.

SOME NUMERICAL RESULTS

The development of the numerical model was accompanied by the solution of several test problems in each decisive steps. Firstly tests were performed with the objective os assessing the ability of the model in solving very low Mach number flows. This was done using cartesian as well as generalized coordinates for flow with Mach number as low as 0.05. Following, the all speed flow model using staggered grids was implemented and several two dimensional and axisymetric flows were solved. The numerical results were compared with available results in the literature. The solution of the supersonic flow over the VLS for zero angle of attach was also solved and compared with experimental results (ONERA (1988)) obtained in wind tunel facilities.

Before going to 3D calculations the co-located arrangement was implemented in the 2D model and a full assesment of the model for 2D problems was realized. Due to the good results with the co-located arrangement for 2D flows, the extension to 3D flows were implemented. FIGURES 4, 5 and 6 depict the Cp for the supersonic/subsonic 3D flow over the VLS for Mach number of

0.497, 0.889 and 2.999. It can be seen that the results for a 60x47x19 grid agrees well with the experimental results reported in ONERA (1988). The results for the 0.899 Mach number required a more refined grid to obtain reasonable solutions. This was expected since the transonic flow offers additional difficulties for its solutions due its complex physics. Details of the validation of the 3D Euler model where results for several flow conditions and several angles of attach for the flow over the VLS can be found in Maliska, Silva, Marchi and Valerim (1991).

CONCLUDING REMARKS

The numerical model as developed up to now is suitable for solving 3D all speed flows using Euler equations. The 3D N-S formulation is now being implemented and preliminary results of 3D viscous calculation at all speed flows are predicted for the end of the current year. Two dimensional viscous calculation with turbulence and viscous dissipation using the model are reported in Men (1991). The turbulence model involved is due to Baldwin and Lomax (1978) which is largely employed in aerodynamics calculations. The turbulence model and viscous dissipation effects will also be included in the 3D viscous model.

The 2D and 3D results obtained to date demonstrated that the all speed flow model in generalized coordinates incorporating co-located variables is a robust and efficient model for aerodynamics calculations. Since it is a new approach it needs deeper investigations to asses its full potentialities and capabilities.

REFERENCES

Baldwin, B. and Lomax, H. (1978), "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows", AIAA Paper, 78-257.

Bortoli, A. L. (1990), O Uso de Variáveis Co-localizadas na Solução de Escoamentos Supersônicos sobre Corpos de Geometrias Arbitrárias, Dissertação de Mestrado, UFSC.

Harlow, F. H. and Amsden, A. A. (1971), "A Numerical Fluid Dynamics Calculation Method for All Flow Speeds", Journal of Computational Physics, Volume 8, pages 197-213 .

Harlow, F.H. and Welch J.E. (1965), "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface", Physics of Fluids, Volume 8, pages 2182-2189.

Hsu, C. (1981), A Curvilinear Coordinate Method for Momentum, Heat and Mass Transfer in Domain of Irregular Geometry, Ph.D. Thesis, University of Minnesota, USA,.

Karki, K. C. and Patankar, S. V. (1989), "Pressure Based Calculation Procedure for Viscous Flow at All Speed in Arbitrary Configurations", AIAA Journal, Volume 27, N$^{\underline{o}}$ 9, September.

Maliska, C. R. and Silva, A. F. C. (1988), Desenvolvimento de Códigos Computacionais para Solução de Problemas de Escoamentos de Alta Velocidade, Relatório preparado para o IAE/CTA, Parte III.

Maliska, C. R. and Silva, A. F. C. (1989), "A Boundary-Fitted Finite Volumes Method for the Solution of Compressible and/or Incompressible Fluid Flow Using both Velocity and Density Corrections, in Finite Element Analysis in Fluids, T. J. Chung and G. R. Karn, Eds, University of Alabama in Huntsville Press.

Maliska, C. R., Silva, A. F. C. and Marchi, C. H. (1991), Formulação Euler e Navier-Stokes Tri-Dimensional em Variáveis Co-localizadas para Escoamentos a Qualquer Velocidade, Relatório preparado para o IAE/CTA, Parte VII.

Maliska, C. R., Silva, A. F. C. and Marchi, C. H. and Carpes Jr., W. (1991), Técnica de Solução Multibloco para Problemas Axissimétricos, Relatório preparado para o IAE/CTA, Parte VIII.

Maliska, C. R., Silva, A. F. C. and Marchi, C. H. and Valerim Jr., J. (1991), Validação dos Modelos Numéricos Bi e Tri-Dimensionais - Equações de Euler, Relatório preparado para IAE/CTA, Parte IX.

Marchi, C. H., Maliska, C. R. and Bortoli, A. L. (1989), "The Use of Co-located Variables in the Solution of Supersonic Flows", 10th Brazilian Congress of Mechanical Engineering - Rio de Janeiro.

Marchi, C. H., Maliska, C. R. and Silva, A. F. C. (1990), "A Boundary-Fitted Numerical Method for the Solution of Three-Dimensional All Speed Flows Using Co-located Variables", 3$^{\underline{o}}$ ENCIT - Itapema - SC.

Men, W. S. (1991), Solução Numérica de Escoamentos Turbulentos Supersônicos com Dissipação Viscosa, Dissertação de Mestrado, Departamento de Engenharia Mecânica, UFSC.

Patankar, S. V. (1971), Calculation of Unsteady Compressible Flows Involving Shocks, Mechanical Engineering Department Report, Imperial College, London, UF/TN/A/4.

Patankar, S. V. (1981), Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Co..

Rhie, C. M. (1981), A Numerical Study of the Flow Past an Isolated Airfoil with Separation, Ph.D. Thesis, University of Illinois, Urbana-Champaign, USA.

Silva, A. F. C. (1991), Um Procedimento em Volumes Finitos para a Solução de Escoamentos de Qualquer Velocidade, Tese de Doutorado, Departamento de Engenharia Mecânica, UFSC.

Silva, A. F. C. and Maliska, C. R. (1988), "Uma Formulação Segregada em Volumes Finitos para Escoamentos Compressíveis e/ou Incompressíveis em Coordenadas generalizadas", 2$^{\underline{o}}$ ENCIT - Águas de Lindóia - SP.

Van Doormaal, J. P. (1985), Numerical Methods for the Solution at Compressible and Incompressible Flows, Ph.D. Thesis, University of Waterloo, Canada,.

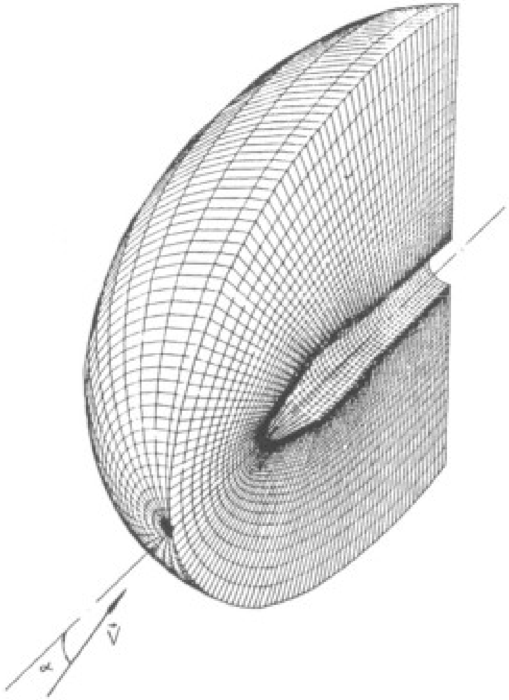

Fig. 1 - 3D boundary-fitted grid over the VLS

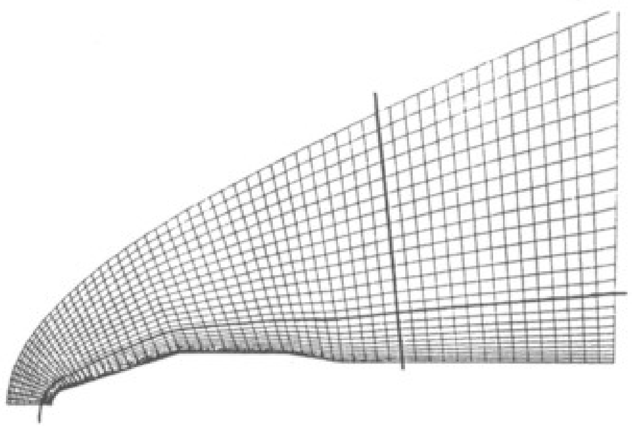

Fig. 2 - Domain sub-division for the multiblock technique

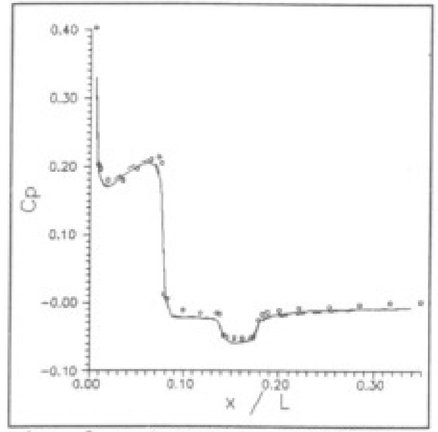

Fig. 3 - Pressure coefficient using the multiblock thechnique

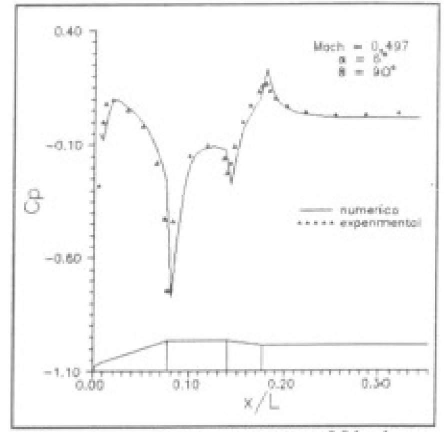

Fig. 4 - Pressure coefficient for the 3D flow, Ma=0.497

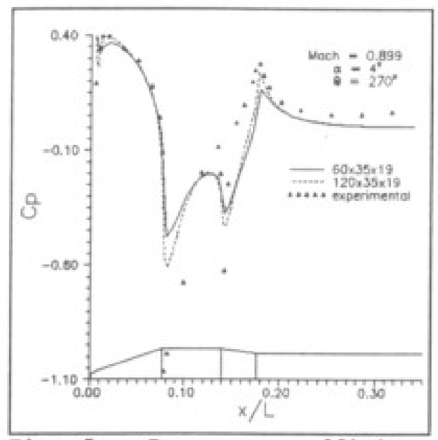

Fig. 5 - Pressure coefficient for the 3D flow, Ma=0.899

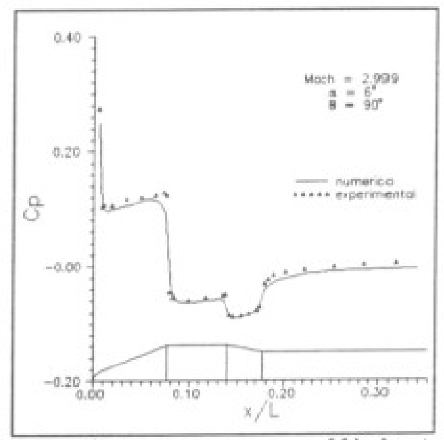

Fig. 6 - Pressure coefficient for the 3D flow, Ma=2.999