XII CONGRESSO BRASILEIRO DE ENGENHARIA MECÂNICA - BRASÍLIA - BRASIL (DEZEMBRO, 1993)
12th BRAZILIAN CONGRESS OF MECHANICAL ENGINEERING - BRASÍLIA - BRAZIL (DECEMBER, 1993)

NUMERICAL COMPUTATIONS OF THE 3D FLOW OVER THE VLS - BRAZILIAN LAUNCH VEHICLE

C. H. Marchi & C. R. Maliska

Laboratório de Simulação Numérica em Mecânica dos Fluidos e Transferência de Calor
SINMEC, Depto. Eng. Mecânica, UFSC
Caixa postal 476, CEP 88040-900, Florianópolis, SC, Brazil

SUMMARY

The numerical solution of the three dimensional flow over the Brazilian Satellite Launch Vehicle (VLS) is realized in this work. The solution is carried out for the inviscid flow with Mach number of 0.50, 0.90 and 3.0, covering the subsonic, transonic and supersonic regime. A numerical model which uses co-located variables and is suitable for the solution of all speed flows is employed. The numerical results are compared with the available experimental ones, and good agreement is observed.

INTRODUCTION

Due to the fast development of high speed computers and of the numerical techniques for the solution of partial differential equations, the use of computational codes for the aerodynamic design of aerospace vehicles has increased considerably. The association of selected experiments in wind tunnels with numerical experiments in computers, permits a better design with much lower costs. With both techniques the aim is to obtain the pressure center, drag and normal coefficients, and the pressure coefficient, used in the prediction of the vehicle trajectory, its performance and its structural design.

In the present work the numerical model of Marchi et al (1990) is employed for the solution of the three dimensional inviscid flow over the VLS, whose geometry is depicted in Fig. 1. The main features of the model are the use of co-located variables in a boundary-fitted framework and the versatility of solving all speed flows. The solution is obtained for Mach number of 0.50, 0.90 and 3.0 with an angle of attack of six degrees. The VLS is under development at the Instituto de Aeronáutica e Espaço (IAE) and it is supposed to launch artificial satellites which are being developed by the Instituto Nacional de Pesquisas Espaciais (INPE).

It is demonstrated that the model employed here is a useful tool in the design of aerospace vehicles and, due to its generality, it can be also employed in the aerodynamic design of automotive vehicles and in the determination of the forces acting upon building structures due to the wind action.

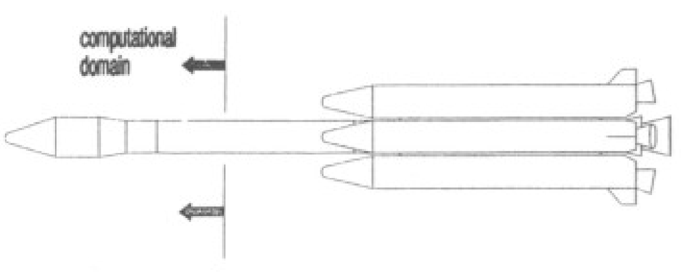

Figure 1. Full configuration of the VLS vehicle.

THE MATHEMATICAL AND THE NUMERICAL METHOD

The mathematical model employed considers the inviscid flow of a perfect gas. The governing equations written in a natural coordinate system (ξ,η,γ) are given by

$$\frac{1}{J}\frac{\partial}{\partial t}(\rho\phi) + \frac{\partial}{\partial \xi}(\rho U\phi) + \frac{\partial}{\partial \eta}(\rho V\phi) + \frac{\partial}{\partial \gamma}(\rho W\phi) + \hat{P}^{\phi} = 0 \quad (1)$$

where J, t and ρ, are the jacobian of the transformation, time and density, (U,V,W), the contravariant velocity components, and $\hat{P}^{\phi}$ an appropriate source term. When the scalar ϕ is set equal to 1, u, v, w and T, Eq.(1) recovers the mass conservation equation, the three components of the momentum equations (Euler equations in this case), and the energy equation, where u, v and w are the cartesian velocity components and T the temperature. The closure of the mathematical model is achieved using the constitutive relation given by the state equation as

$$p = \rho RT \quad (2)$$

where p is the thermodynamic pressure and R the gas constant.

The numerical methodology used employs the finite volume approach (Patankar, 1980) based in a boundary-fitted framework (Thompson et al., 1974), with a co-located scheme for the dependent variables, as described in Marchi et al. (1989) and the all speed methodology developed by Silva and Maliska (1988). The use of co-located variables in three-dimensional numerical schemes using boundary-fitted grids is extremely convenient since it simplifies the cumbersome procedure needed when staggered grids are employed.

The equations recovered by Eq.(1) are solved implicitly in a segregated way, with the Euler equations used for calculating u, v and w, the mass conservation relation for finding pressure and the energy equation for determining temperature. Density is obtained through the state equation. The SIMPLEC method of Van Doormaal and Raithby (1984) is employed for treating the pressure-velocity coupling. The same idea embodied in the SIMPLEC procedure was employed earlier by Rushmore and Taulbee (1978). The five systems of algebraic equations are solved by an ADI procedure as described in Silva et al. (1991).

PROBLEM DEFINITION

Computational Domain. The flow domain under investigation covers only the forepart the VLS, about 25% of its full length L, as shown in Fig. 1, because beyond this region the flow is affected by the rocket boosters. To solve the booster region it requires a very large computational grid, which is, at the moment, beyond our computational capabilities. Fig. 2 shows the type of discretization used for the numerical solution. The grid was generated algebraicaly using an nonuniform spacing normal to the rocket surface with ratio 1,20. The full computational domain has, therefore, dimensions of 4L in the upstream direction and 8L normal to the surface, where L is the full length of the vehicle. Such a large domain, in the upstream and normal directions, is necessary due to the elliptic nature of the subsonic and the transonic calculations. Additionaly, as seen in Fig. 2, the computational domain covers only 180^0 in the azimuthal direction because the yaw angle of the flow is zero and, therefore, the three dimensionality of the flow is admited to result only from the angle of attack of the vehicle.

It is employed 12 volumes in the azimuthal and 70 in the normal directions, with 96 volumes in the streamwise direction for the supersonic calculatins and 192 for the remaining cases. This represent grids wich ranges from 80,000 to 160,000 elemental volumes.

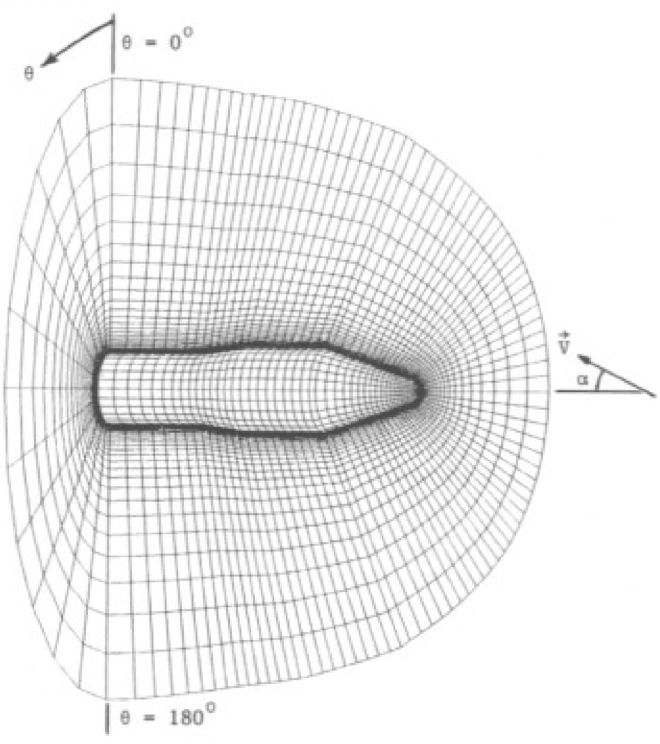

Figure 2. Grid close to the VLS vehicle.

Boundary Conditions. At the downstream region, that is, the plane normal to the vehicle axis shown in Fig. 2, locally parabolic flow is assumed. In the azimuthal planes of 0^0 and 180^0 symmetry boundary conditions are employed, while at the body surface slip flow with zero normal velocity component is used for the Euler equations and adiabatic conditions for the energy equation. Finally, the free-stream conditions precribed are shown in Tab. 1.

Table 1. Free-stream conditions.

M_∞	0.50	0.90	3.0
α [degrees]	6.0	6.0	6.1
p_∞ [kPa]	210.9	106.4	10.90
T_∞ [K]	277.1	260.3	144.8

NUMERICAL RESULTS

In Figs. 3 to 5, the pressure coefficients are reported for Mach numbers of 0.50, 0.90 and 3.0. Solid and dotted lines denote the numerical results obtained in the present work and the symbols represents the exprimental results of Moraes Jr. (1991). The angle θ is defined in Fig. 2 and α is the vehicle angle of attack. Recall that L is the full length of the VLS. The pressure coefficient plotted in the above mentioned figures is defined by

$$Cp = \frac{(p_w - p_\infty)}{\frac{1}{2}\rho_\infty |\vec{V}_\infty|^2} \tag{3}$$

where p_w is the pressure at the vehicle surface. Additional informations with respect to the experimental data used in Figs. 3 to 5 can be found in Moraes Jr. and Neto (1990).

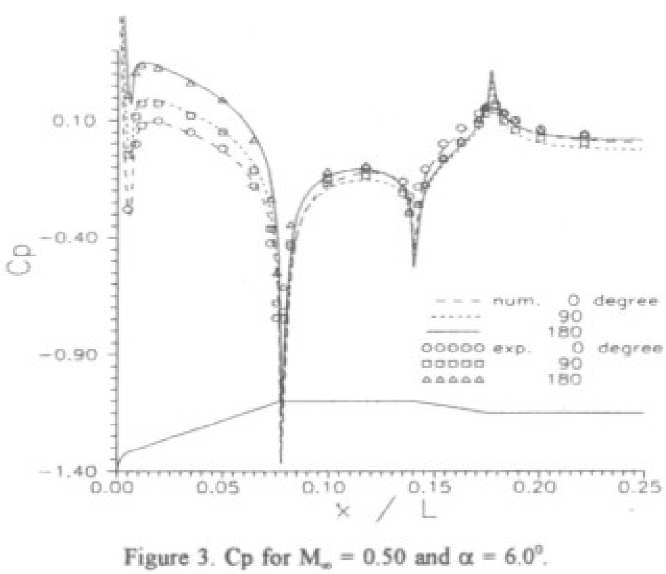

Figure 3. Cp for $M_\infty = 0.50$ and $\alpha = 6.0^0$.

As can be seen in Fig. 3 and 5, where the results for the subsonic and supersonic cases are shown, the agreement between the numerical and experimental results are very good. The agreement observed for the different θ values, which corresponds to very distinct physical situations, demonstrated that the three-dimensional flow is being correctly solved.

For the transonic case, where elliptic and hyperbolic effects take place simultaneously, it is well known that the capture of the flow details requires high grid resolution. In the present work the grid was refined up to the computational capabilities available. Even though, the numerical results do not fit the experimental ones in two regions. These regions are the expansions occuring when the flow leaves the cone and when the diameter of the launcher starts to decrease. In the first expansion the pressures calculated numerically are higher than the experimental ones. The filling is that a even more refined grid can improve the quality of the results. In the second region the experimental results do not show the existence of a expansion for $\theta = 0^0$. For other θ values the experimental results do show an expansion with the numerical ones agreeing reasonably well. It seems that a best fitting can be achieved between numerical and experimental results in the first expansion if higher grid resolution is employed, since this was the trend observed in Maliska et al. (1991), when the grid resolution study was carried out. In the second region one can see that near the kink of the vehicle there is no enough experimental points for $\theta = 0^0$. It may be possible that if a intermediate pressure point were used, the expansion could have been captured. One can also speculate that the secondary flow caused by the angle of attack may influence the flow expansion in the leeside of the vehicle, that is at $\theta = 0^0$. The numerical results, although not fitting exactly the numerical ones, clearly demonstrate that the flow charactristics are well captured. Additional numerical computations of transonic flows over the VLS can be found in Maliska et al. (1991).

A global view of the results can be seen in Figs. 6 to 8, where the Mach number contours are shown for the region close to the vehicle. Fig. 6 and 7 show the ellyptic character of the flow with the isolines demonstrating the influence of the launcher upstream. The non-symmetry flow due to the angle of attack is also clearly pictured in these figures. By its turn, Fig. 8 depicts a typical hyperbolic flow with a attached shock close to the launcher nose.

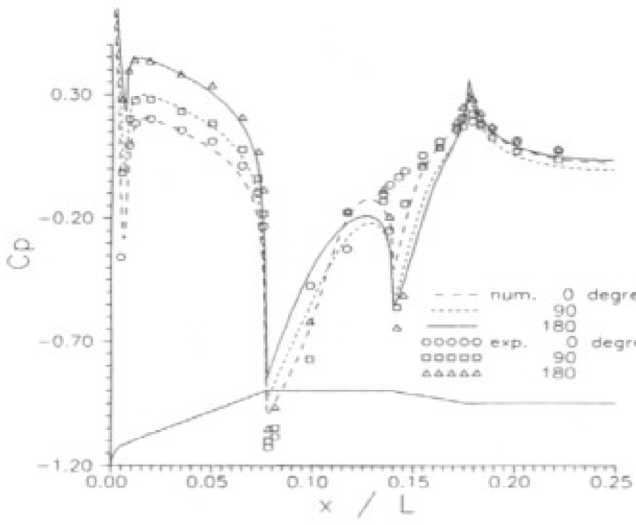

Figure 4. Cp for $M_\infty = 0.90$ and $\alpha = 6.0^0$.

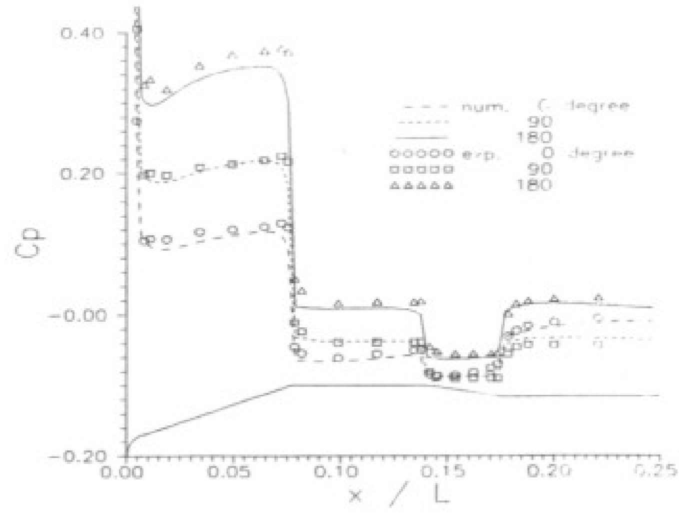

Figure 5. Cp for $M_\infty = 3.0$ and $\alpha = 6.1^0$.

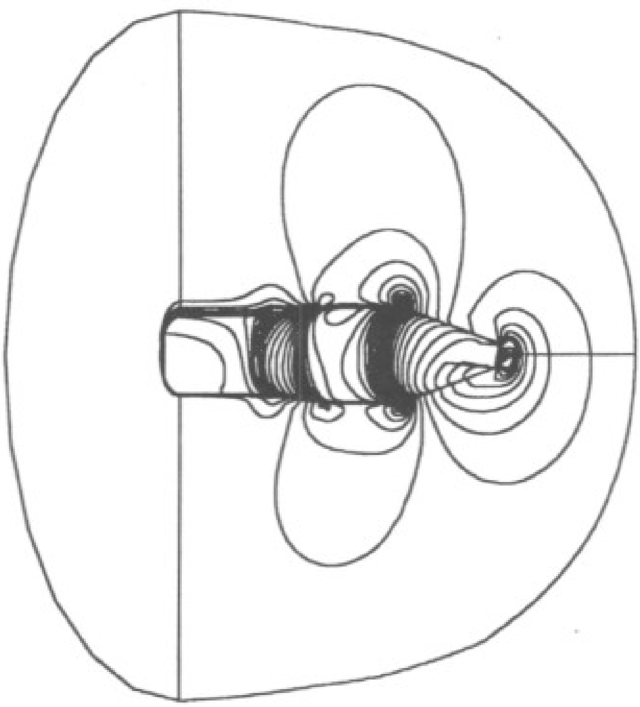

Figure 6. Mach number contours for $M_\infty = 0.50$.

CONCLUDING REMARKS

The computation of subsonic, transonic and supersonic flows over the VLS was realized with great versatility using the all speed flow methodology. The present results, in conjuction with several others obtained by the authors, render to the method confidence for the solution of general aerodynamic problems. It is already implemented in the model the multiblock facility which will permit the solution of the flow over the complete geometry of the VLS including the four boosters. This is of fundamental importance for predicting the stages separation in order to avoid collision between them. In a companion activity the viscous terms and a turbulence model are being included in the model, such that viscous heating problems can also be considered.

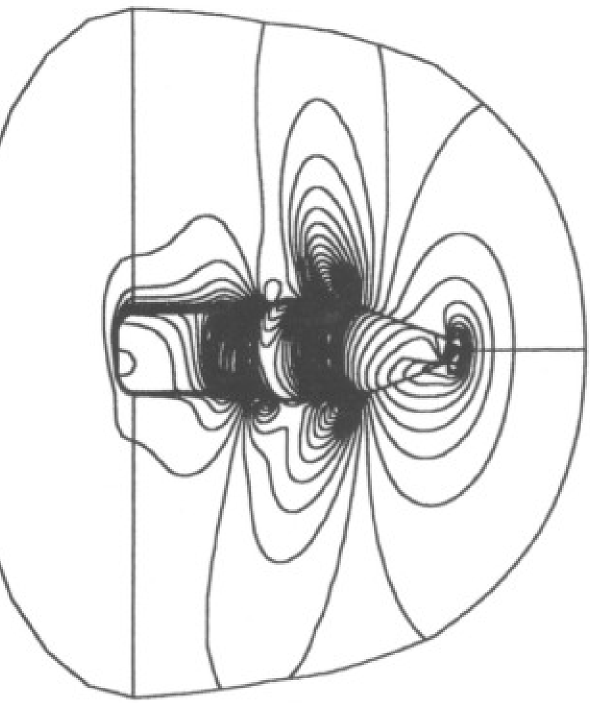

Figure 7. Mach number contours for $M_\infty = 0.90$.

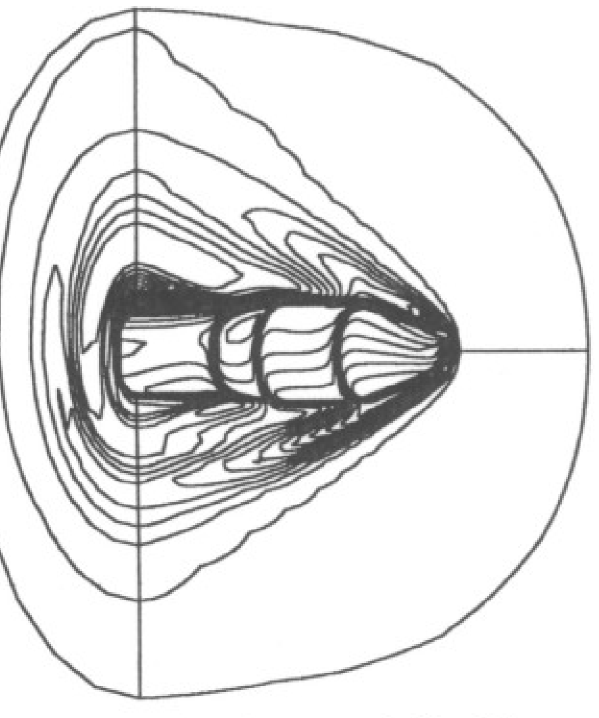

Figure 8. Mach number contours for $M_\infty = 3.0$.

ACKNOWLEDGMENTS

The authors wants to acknowledge the partial financial support provided by IAE and to Dr. Paulo Moraes Jr. for making available the experimental results.

REFERENCES

#Maliska, C.R., Silva, A.F.C., Marchi, C.H. and Valerim Jr., J., "Validação dos Modelos Numéricos Bi e Tridimensionais - Equações de Euler; Relatório ao IAE/CTA; Parte IX", SINMEC/EMC/UFSC, Florianópolis, 1991.

#Marchi, C.H., Maliska, C.R. and Bortoli, A.L.", "The Use of Co-Located Variables in the Solution of Supersonic Flows", Proceedings of the 10[th] Brazilian Congress of Mechanical Engineering, pp. 157-160, Rio de Janeiro, Brazil, 1989.

#Marchi, C.H., Maliska, C.R. and Silva, A.F.C., "A Boundary-Fitted Numerical Methods for the Solution of Three Dimensional All Speed Flows Using Co-Located Variables", Proceedings of the 3[rd] Brazilian Thermal Science Meeting, pp. 351-356, Itapema, Brazil, 1990.

#Moraes Jr., P., "Private Communication", São José dos Campos, 1991.

#Moraes Jr., P. & Neto, A.A., "Aerodynamic Experimental Investigation of the Brazilian Satellite Launch Vehicle (VLS)", Proceedings of the 3[rd] Brazilian Thermal Science Meeting, pp. 2 11-215, Itapema, Brazil, 1990.

#Patankar, S.V., "Numerical Heat Transfer and Fluid Flow", McGraw-Hill, New York, 1980.

#Rushmore, W.L. & Taulbee, D.B., "Numerical Investigation of Developing Pipe Flows of Arbitrary Curvature", Computers and Fluids, Vol. 6, pp. 125-140, 1978.

#Silva, A.F.C. & Maliska, C.R., "Uma Formulação Segregada em Volumes Finitos para Escoamentos Compressíveis e/ou Incompressíveis em Coordenadas Generalizadas", Proceedings of the 2[nd] Brazilian Thermal Science Meeting, pp. 11-14, Águas de Lindóia, Brazil, 1988.

#Silva, A.F.C., Marchi, C.H., Livramento, M.A. and Azevedo, J.L.F., "On the Effects of Vectorization for Efficient Computation of Three Dimensional Segregated Finite Volume Solutions", Proceedings of the 11[th] Brazilian Congress of Mechanical Engineering, pp. 109-112, São Paulo, Brazil, 1991.

#Thompson, J.F., Thames, F.C. and Mastin, C.W., "Automatic Numerical Generation of Body-Fitted Curvilinear Coordinate System for Field Containing Any Number of Arbitrary Two-Dimensional Bodies", J. Comp. Phys., Vol. 15, pp. 299-319, 1974.

#Van Doormaal, J.P. & Raithby, G.D., "Enhancements of the Simple Method for Predicting Incompressible Fluid Flows", Numerical Heat Transfer, Vol. 7, pp. 147-163, 1984.