

NUMERICAL STUDY BETWEEN STRUCTURED AND UNSTRUCTURED MESHES FOR EULER AND NAVIER-STOKES EQUATIONS

Gustavo Bono^a and Armando M. Awruch^b

^a*Mechanical Engineering Posgraduate Program, Federal University of Rio Grande do Sul, Av.
Sarmiento Leite 425, 90050-170 Porto Alegre, Brazil, gbono@mecanica.ufrgs.br,
<http://www.mecanica.ufrgs.br/promec>*

^b*Applied and Computational Mechanical Center, Federal University of Rio Grande do Sul, Av.
Osvaldo Aranha 99, 90035-190 Porto Alegre, Brazil, amawruch@ufrgs.br,
<http://www.ppgec.ufrgs.br/cemacom>*

Keywords: Finite element method, Compressible flows, Structured mesh, Unstructured mesh.

Abstract. The mesh influence for the solution of compressible flows with two explicit schemes is presented. The investigations measured the order of accuracy, absolute error and the computational cost with structured and unstructured meshes. Two explicit Taylor-Galerkin schemes (one-step and two-step) with a linear tetrahedral and tri-linear hexahedral elements are employed to analyze the compressible Euler and Navier-Stokes equations. Finally, some numerical analyses in inviscid and viscous flows are discussed. This article includes a reflection of an oblique shock and a viscous flow over an airfoil.

1 INTRODUCTION

The rapid growth in the power and availability of computers in recent years has led to the development of many schemes for solving the Euler and Navier-Stokes equations successfully, and as a result numerical simulations are beginning to complement or even replace experimental measurement. They allow simulations of internal and external fluid flow at different boundary conditions. In the aerospace industry today Computational Fluid Dynamics (CFD) plays an increasingly important role as a tool for design and analysis.

Computational methods are continuously being required to deliver more accurate solutions for more complex realistic configurations at lower computational cost. Traditionally, structured mesh (quadrilateral or hexahedral elements) approach have been employed to discretize the computational domain. The natural ordering of quadrilateral/hexahedral elements enables the construction of very efficient numerical algorithms for solving the flow equations. While amenable to algorithmic efficiency, structured mesh are inherently difficult for discretizing complex geometries. In the case the complex configurations, the physical domains has to be decomposed into different subdomains (blocks), and the mesh has to generated separately for individual blocks.

Unstructured mesh methods originally emerged as a viable alternative to the structured mesh techniques for discretizing complex geometries. In unstructured meshes triangular elements in two dimensions and tetrahedral elements in three-dimensions are used. This not only provides greater flexibility for discretizing complex domains but also enables straightforward implementation of adaptive techniques where node may be added or deleted, while mesh connectivity is updated locally, in order to enhance solution accuracy. Although unstructured mesh methods provide flexibility for discretizing complex geometries, then have be the drawback of requiring larger in-core memory and more CPU effort than their structured counter-parts.

The choice of the type of mesh element to be employed in the aerospace and aeronautical problems depends on the delivered accuracy, efficiency, and flexibility of the numerical solver. A two dimensional study by Aftosmis et al. (1994) observed little difference in accuracy between equivalent meshes of quadrilateral and triangular elements. Hexahedral meshes have a better accuracy than the tetrahedral meshes in three dimensional experiments (Baker, 2005).

The time integration, for instance, can be performed in one of the two classical approaches, explicit or implicit techniques. Implicit methods are computationally more expensive, but have less stringent stability bounds. Explicit methods are relatively simple to code and implemented, and are easily cast in a form suitable for efficient parallelization. They require less memory than implicit methods, since only one flow field solution needs to be stored at a time.

In the current work, two explicit Taylor-Galerkin schemes for solving the compressible Euler and Navier-Stokes equations in the context of structured and unstructured meshes are investigated. The present comparison intends to emphasize important features of these numerical schemes and elements. The tests include the reflection of an oblique shock for which there exists a closed-form solution to the compressible Euler equations, as well as the laminar transonic flow past an airfoil.

2 THE GOVERNING EQUATIONS

Let $\Omega \subset R^{n_{sd}}$ and $(0,T)$ be the spatial and temporal domains, respectively, where $n_{sd} = 3$ is the number of space dimensions, and let Γ denote the boundary of Ω . The spatial and

temporal coordinates are denoted by \mathbf{x} and t . We consider the conservation law form of the Navier-Stokes equations governing unsteady compressible flows with no source terms, written here in their dimensionless form

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}_i}{\partial x_i} + \frac{\partial \mathbf{G}_i}{\partial x_i} = 0 \quad (1)$$

where \mathbf{U} is the unknown vector of the conservation variables, \mathbf{F}_i and \mathbf{G}_i are, respectively, the convective and diffusive flux vectors given by

$$\mathbf{U} = \begin{Bmatrix} \rho \\ \rho v_i \\ \rho e \end{Bmatrix}, \quad \mathbf{F}_i = \begin{Bmatrix} \rho v_j \\ \rho v_i v_j + p \delta_{ij} \\ v_j (\rho e + p) \end{Bmatrix}, \quad \mathbf{G}_i = \begin{Bmatrix} 0 \\ -\tau_{ij} \\ -\tau_{ji} v_i - q_j \end{Bmatrix} \quad (2)$$

with $i, j = 1, 2, 3$. Here v_i is the velocity component in the direction of the coordinate x_i , ρ is the specific mass, p is the thermodynamic pressure, τ_{ij} are the components of the viscous stress tensor, q_j is the heat flux vector, e is the total specific energy and δ_{ij} is the Kronecker delta function.

For a calorically perfect gas, the equation of state and internal energy i are given by the following equations

$$p = (\gamma - 1) \rho i, \quad i = c_v T = e - \frac{1}{2} v_i v_i \quad (3)$$

where T is the temperature and $\gamma = c_p / c_v$ with c_p and c_v being the specific heat coefficients at constant pressure and constant volume, respectively. The viscous stress tensor τ_{ij} and the heat flux vector q_j are defined as

$$\tau_{ij} = \lambda v_{k,k} \delta_{ij} + \mu (v_{i,j} + v_{j,i}), \quad q_j = -k_{ij} T_{,k} \quad (4)$$

where $k = 1, 2, 3$. Here k_{ij} represents the components of the conductivity tensor, λ and μ are the volumetric and dynamic viscosity coefficients, respectively. The dynamic viscosity and coefficient of thermal conductivity depend on temperature and therefore are modeled using Sutherland's law (White, 1974)

$$\mu = \left(\frac{T}{T_{ref}} \right)^{3/2} \frac{T_{ref} + S_\mu}{T + S_\mu}, \quad k = \left(\frac{T}{T_{ref}} \right)^{3/2} \frac{T_{ref} + S_k}{T + S_k} \quad (5)$$

where T_{ref} is the temperature reference and $S_\mu = 110$ K and $S_k = 194$ K for atmospheric air.

Initial and boundary conditions must be added to equations (1)-(2) in order to define uniquely the problem.

3 A TAYLOR-GALERKIN FORMULATION

The numerical scheme is obtained expanding in Taylor series the governing equation and applying after the space discretization process, using the Finite Element Method (FEM) in the context of the classical Bubnov-Galerkin scheme. These approach can be interpreted as the finite element version of the Lax-Wendroff scheme used in finite differences. Two schemes for explicit time integration (one-step and two-step methods) are investigated for solving the compressible inviscid/viscous flow problems.

This temporal integration provides second-order accuracy for time derivative. The formulation exclusively employs tetrahedral and hexahedral finite elements which provides second-order spatial accuracy. Linear unstructured finite elements were chosen because they can be easily generated for complex geometries and exactly integrated without numerical quadrature. To obtain important savings in CPU time and computer memory, an analytical evaluation of the eight node hexahedral element matrices was performed.

3.1 Time discretization: The one-step scheme

The one-step scheme is similar to that presented by Donea (1984). Expanding the conservation variables \mathbf{U} at $t = t^{n+1}$ in Taylor series including the first and second derivatives, resulting in

$$\Delta\mathbf{U}^{n+1} = \Delta t \left(\frac{\partial\mathbf{U}}{\partial t} \right)^{n+s_1} + \frac{\Delta t^2}{2!} \left(\frac{\partial^2\mathbf{U}}{\partial t^2} \right)^{n+s_2} + O(\Delta t^3) \quad (6)$$

with $\Delta\mathbf{U}^{n+1} = \mathbf{U}^{n+1} - \mathbf{U}^n$, being s_1 and s_2 the implicitness parameters defined such that

$$\begin{aligned} \frac{\partial\mathbf{U}^{n+s_1}}{\partial t} &= \frac{\partial\mathbf{U}^n}{\partial t} + s_1 \frac{\partial\Delta\mathbf{U}^{n+1}}{\partial t} & 0 \leq s_1 \leq 1 \\ \frac{\partial^2\mathbf{U}^{n+s_2}}{\partial t^2} &= \frac{\partial^2\mathbf{U}^n}{\partial t^2} + s_2 \frac{\partial^2\Delta\mathbf{U}^{n+1}}{\partial t^2} & 0 \leq s_2 \leq 1 \end{aligned} \quad (7)$$

Substituting equation (7) into equation (6), and adopting $s_1 = s_2 = 1/2$, the following expression is obtained

$$\Delta\mathbf{U}^{n+1} = \Delta t \left(\frac{\partial\mathbf{U}^n}{\partial t} + \frac{1}{2} \frac{\partial\Delta\mathbf{U}^{n+1}}{\partial t} \right) + \frac{\Delta t^2}{2} \left(\frac{\partial^2\mathbf{U}^n}{\partial t^2} + \frac{1}{2} \frac{\partial^2\Delta\mathbf{U}^{n+1}}{\partial t^2} \right) \quad (8)$$

Substituting equation (1) and its second derivative into equation (8), and neglecting high-order terms, we obtain

$$\begin{aligned} \Delta\mathbf{U}_{I+1}^{n+1} &= \Delta t \left[-\frac{\partial\mathbf{F}_i^n}{\partial x_i} - \frac{\partial\mathbf{G}_i^n}{\partial x_i} + \frac{\Delta t}{2} \frac{\partial}{\partial x_k} \left(\mathbf{A}_k^n \frac{\partial\mathbf{F}_i^n}{\partial x_i} \right) \right] + \\ &+ \frac{\Delta t}{2} \left[-\frac{\partial\Delta\mathbf{F}_{iI}^{n+1}}{\partial x_i} - \frac{\partial\Delta\mathbf{G}_{iI}^{n+1}}{\partial x_i} + \frac{\Delta t}{2} \frac{\partial}{\partial x_k} \left(\mathbf{A}_k^n \frac{\partial\Delta\mathbf{F}_{iI}^{n+1}}{\partial x_i} \right) \right] \end{aligned} \quad (9)$$

where I is an iteration counter, $\Delta\mathbf{F}_i^{n+1} = \mathbf{F}_i^{n+1} - \mathbf{F}_i^n$, $\Delta\mathbf{G}_i^{n+1} = \mathbf{G}_i^{n+1} - \mathbf{G}_i^n$ and \mathbf{A}_i is the convection Jacobian defined as $\mathbf{A}_i = \partial\mathbf{F}_i/\partial\mathbf{U}$ (Hughes and Tezduyar, 1984).

In expression (9), the variables at time level $n+I$ are involved in the left and right sides of the equation, therefore it is necessary to use an iterative scheme.

3.2 Time discretization: The two-step scheme

The two-step scheme is similar to that presented by Kawahara and Hirano (1983). In the first step, corresponding to the time interval $[t^n, t^{n+1/2}]$, the unknown vector \mathbf{U} at $t = t^{n+1/2}$ is expanded in Taylor series, resulting in

$$\Delta \mathbf{U}^{n+1/2} = \left(\frac{\Delta t}{2} \right) \frac{\partial \mathbf{U}^n}{\partial t} + \frac{\left(\frac{\Delta t}{2} \right)^2}{2!} \frac{\partial^2 \mathbf{U}^n}{\partial t^2} + O(\Delta t^3) \quad (10)$$

with $\Delta \mathbf{U}^{n+1/2} = \mathbf{U}^{n+1/2} - \mathbf{U}^n$. Substituting equation (1) and its second derivative into equation (10), and neglecting high-order terms, gives

$$\Delta \mathbf{U}^{n+1/2} = \frac{\Delta t}{2} \left[-\frac{\partial \mathbf{F}_i^n}{\partial x_i} - \frac{\partial \mathbf{G}_i^n}{\partial x_i} + \frac{\Delta t}{4} \frac{\partial}{\partial x_i} \left(\mathbf{A}_i^n \frac{\partial \mathbf{F}_j^n}{\partial x_j} \right) \right] \quad (11)$$

In the second step, \mathbf{U} at time t^{n+1} is determined by expanding equation (1) in Taylor series, obtaining the following expression

$$\Delta \mathbf{U}^{n+1} = \Delta t \left(\frac{\partial \mathbf{U}}{\partial t} \right)^n + \frac{\Delta t^2}{2!} \left(\frac{\partial^2 \mathbf{U}}{\partial t^2} \right)^n + O(\Delta t^3) \quad (12)$$

with $\Delta \mathbf{U}^{n+1} = \mathbf{U}^{n+1} - \mathbf{U}^n$. Substituting equation (1) and its second derivative into equation (12), and neglecting high-order terms, the following expression is obtained

$$\Delta \mathbf{U}^{n+1} = \Delta t \left[-\frac{\partial \mathbf{F}_i}{\partial x_i} - \frac{\partial \mathbf{G}_i}{\partial x_i} + \frac{\Delta t}{2} \frac{\partial}{\partial x_i} \left(\mathbf{A}_i \frac{\partial \mathbf{F}_j}{\partial x_j} \right) \right]^{n+1/2} \quad (13)$$

where the convection Jacobian \mathbf{A}_i is defined as $\mathbf{A}_i = \partial \mathbf{F}_i / \partial \mathbf{U}$ (Hughes and Tezduyar, 1984).

3.3 Spatial discretization

Applying the classical Bubnov-Galerkin weighted residual method in the context of the FEM to equation (9), for the one-step scheme, and equations (11) and (13), for the two-step scheme, the spatial discretization are obtained. The computational domain was divided into a finite number of tri-linear hexahedral elements (structured mesh) or linear tetrahedral elements (unstructured mesh). The consistent mass matrix is substituted by the lumped mass matrix and then these equations are solved with an explicit scheme.

The proposed schemes are conditionally stable, and the local stability condition for element E is given by

$$\Delta t_E = \delta \frac{L_E}{a + (\mathbf{v}_i \mathbf{v}_i)^{1/2}} \quad (14)$$

where L_E is a characteristic dimension of the element, a is the sound speed and δ is a safety coefficient (in this work the coefficient adopted were $\delta = 0.1$ or 0.3).

In order to stabilize the solution numerically, specially to control oscillations, and overshoots in the vicinity of the steep gradients such as of shock waves, discontinuities, and vortex sheets, where dissipation effects or shear stress take place in very thin layers of the flow, it is necessary to add numerical damping to the flow solver. An artificial viscosity model, as proposed by Argyris et al. (1990), is used due to its simplicity and efficiency in terms of CPU time. An artificial viscosity is added explicitly to the non-smoothed solution as follows

$$\mathbf{U}_s^{n+1} = \mathbf{U}^{n+1} + \mathbf{M}_L^{-1} \mathbf{D} \quad (15)$$

where \mathbf{M}_L is the assembled lumped mass matrix, \mathbf{U}_s^{n+1} and \mathbf{U}^{n+1} are the smoothed and non-smoothed solutions at $t + \Delta t$, respectively. The vector \mathbf{D} is given by

$$\mathbf{D} = \sum_{ele} CFL \text{ CAF } S_{ele} [\mathbf{M} - \mathbf{M}_L]_{ele} \mathbf{U}_{ele}^n \quad (16)$$

where ele is an index referred to a specific element, $CFL = \Delta t / \Delta t_E$ is the local Courant-Friedrichs-Lewy number, CAF is an artificial damping coefficient given by the user, S_{ele} is a pressure sensor at element level obtained as an average of nodal values S_i . Values of S_i are components of the following assembled global vector

$$S_i = \sum_{ele} \frac{|(\mathbf{M} - \mathbf{M}_L)_{ele} \mathbf{p}|_i}{|[\mathbf{M} - \mathbf{M}_L]_{ele} \mathbf{p}|_i} \quad (17)$$

where \mathbf{p} is the pressure vector of a specific element, the bars indicate that absolute values of the corresponding terms must be taken and, finally, \mathbf{M} is the consistent mass matrix at element level.

The constant CAF must be specified with care in order to avoid interferences of artificial and physical viscosities. In this work $0.8 \leq CAF \leq 1.0$ were adopted.

4 NUMERICAL EXAMPLES

In this section two examples are presented in order to evaluate the accuracy, the capability and the performance of the two explicit Taylor-Galerkin schemes in the context of structured / unstructured meshes applied to the solution of compressible inviscid and viscous problems.

The test problems have been geometrically modeled as 3-D problems, the boundary conditions are chosen so that the 2-D flow field is simulated. Finally, it is assumed that the fluid has a specific heat ratio and a constant Prandtl number equal to 1.4 and 0.72, respectively.

4.1 Reflection of an oblique shock in a inviscid flow

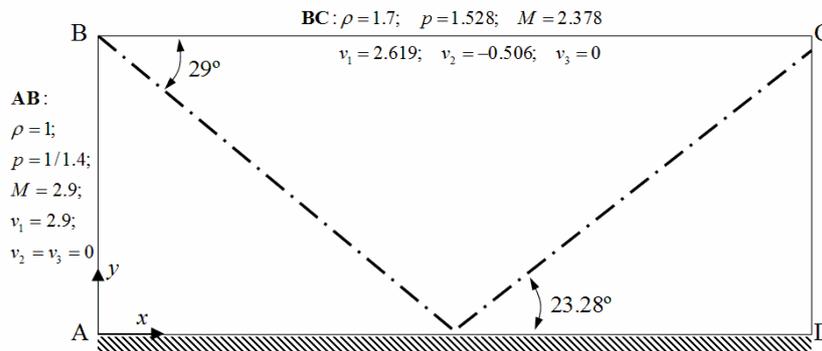


Figure 1: Reflection of an oblique shock and boundary conditions

The rectangular domain $\{0 \leq x \leq 4.10, 0 \leq y \leq 1.0\}$ and boundary conditions of this problem are shown in Figure 1. Along the inflow ABC all variables are fixed; zero normal velocity is imposed at the wall (AD) and along CD all variables are left free. The exact solution is formed by an incident shock with angle 29° and a reflected shock with angle

23.28°. The solution after the second shock is: $\rho = 2.687$; $p = 2.934$; $M = 1.942$; $v_1 = 2.401$ and $v_2 = v_3 = 0$.

Four uniform meshes M1, M2, M3 and M4 consisting of $21 \times 11 \times 2$, $41 \times 21 \times 2$, $61 \times 31 \times 2$ and $81 \times 41 \times 2$ nodes, respectively were considered. The meshes with hexahedral elements contains 200, 800, 1800 and 3200 elements and the meshes with tetrahedral 1000, 4000, 9000 and 16000 elements, respectively. The mesh with tetrahedral is obtained by subdividing each hexahedral element into five tetrahedral elements.

In Figure 2 the meshes with hexahedral and tetrahedral elements, respectively, are shown

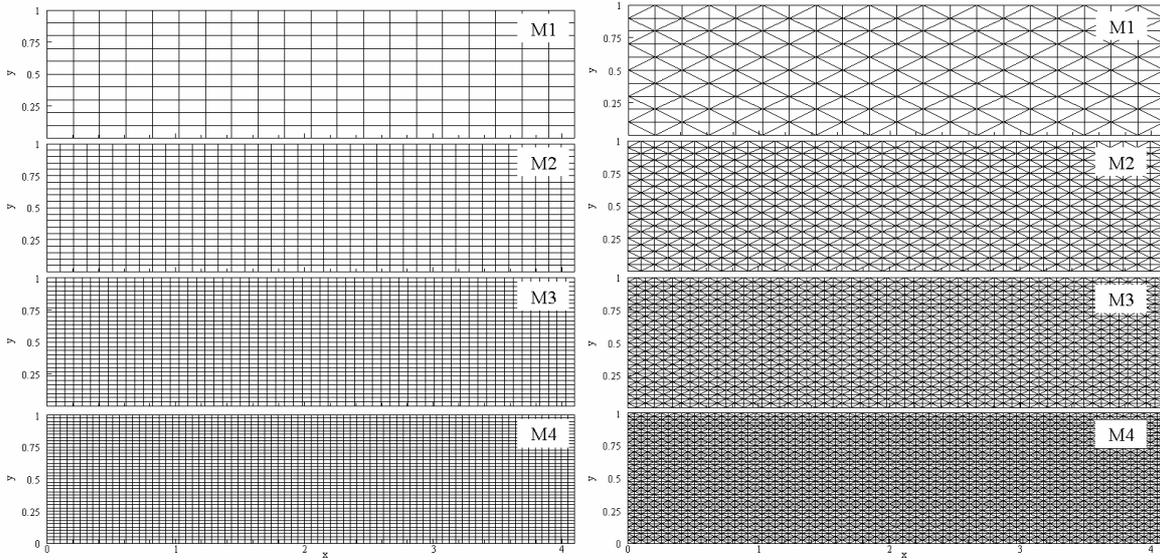


Figure 2: Meshes with hexahedral and tetrahedral elements

Figures 3 and 4 show the specific mass at $y = 0.25$ obtained with the four meshes of hexahedral (H) / tetrahedral (T) elements with the one-step and with the two-step (P2) schemes. The artificial damping coefficient used in this example, defined in equation (16), was $CAF = 1.0$.

The results for hexahedral meshes exhibit a good agreement with exact solutions and numerical results reported in Shakib et al. (1991). A small oscillation in the vicinity of the second shock is observed when tetrahedral meshes are used. Also, these tetrahedral meshes fail to give an exact specific mass behind the second shock. In the mesh M4 only the tetrahedral mesh with one-step scheme overestimate the specific mass.

The accuracy of the numerical solution for the problem is measured by the average error defined in the L_1 norm and the L_2 norm. For specific mass, the two norms are given by

$$L_1 = \frac{1}{nno} \sum_{i=1}^{nno} \left| \frac{\rho_i - \rho_i^e}{\rho_i^e} \right| \quad \text{and} \quad L_2 = \sqrt{\frac{1}{nno} \sum_{i=1}^{nno} \left(\frac{\rho_i - \rho_i^e}{\rho_i^e} \right)^2} \quad (18)$$

where ρ^e represents the exact solution and nno is the number of nodes.

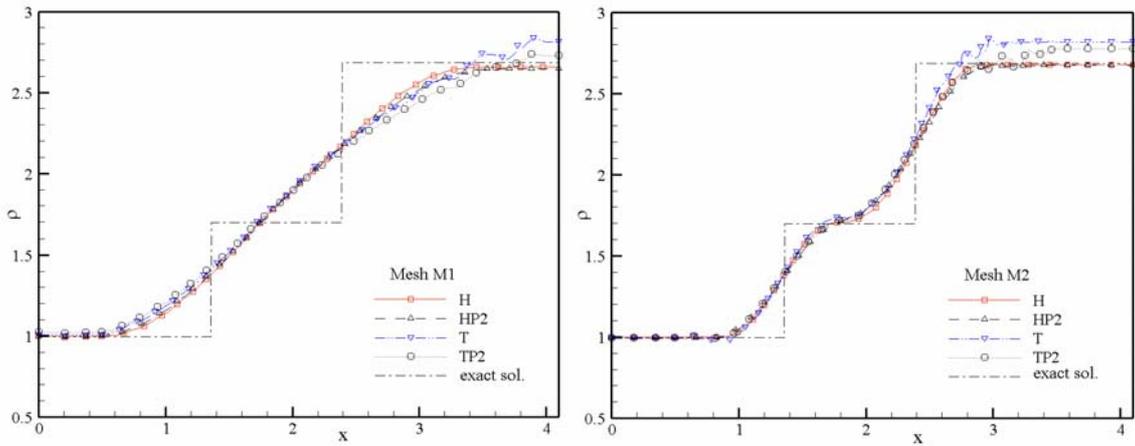


Figure 3: Comparison between exact solution and numerical results of the specific mass along the line $y = 0.25$ for mesh M1 and M2

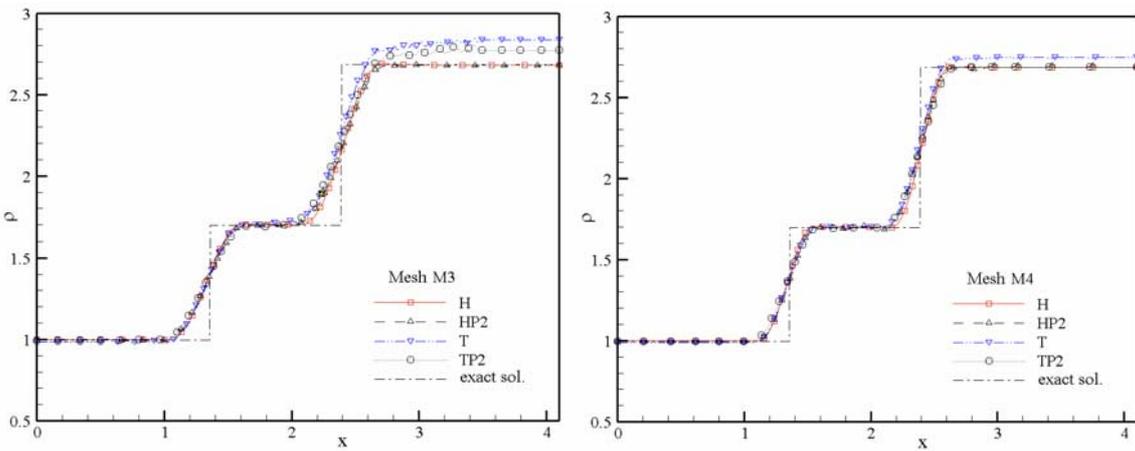


Figure 4: Comparison between exact solution and numerical results of the specific mass along the line $y = 0.25$ for mesh M3 and M4

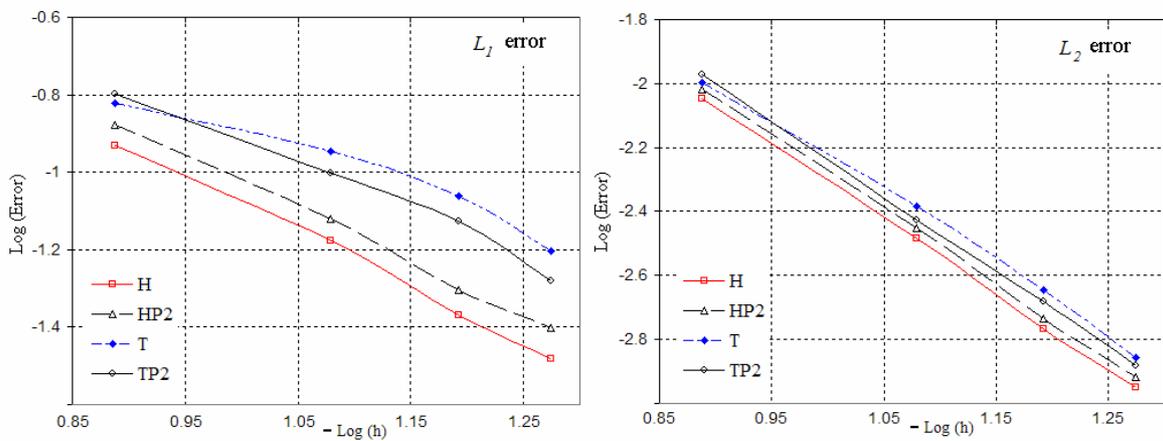


Figure 5: Convergence study for the reflection of an oblique shock problem

In the error convergence study, we presents plots of $\log L_1(\Omega)$ and $\log L_2(\Omega)$ versus

$\log(h)$. The results for the specific mass are presented in Figure 5 for hexahedral / tetrahedral elements with the one-step and the two-step schemes. Analyzing Figure 5 it is observed that the error decreases as the grid size decreases. The average mesh density h for the hexahedral and tetrahedral meshes is defined as $h = (1/nno)^{1/3}$.

In structured meshes a two-step scheme is slightly less accurate than the one-step scheme, while in unstructured meshes a one-step scheme is slightly less accurate than the two-step scheme. Finally, we point out that hexahedral elements have better accuracy than tetrahedral elements. The error is lower for hexahedral elements, independent of the finite element mesh.

The asymptotic convergence rate of errors, given in Table 1, which is measured over the two finest meshes, is just slightly less than 2.0 for $L_1(\Omega)$ norm. Note, however, that the order of convergence is slightly over 2.0 for $L_2(\Omega)$ norm. This is indeed consistent with the second-order accuracy of the spatial discretization that was used.

element / scheme	L_1	L_2
H	1.378	2.257
HP2	1.213	2.241
T	1.715	2.588
TP2	1.907	2.470

Table 1: Order of convergence for the example of the reflection of an oblique shock problem for meshes M1, M2, M3 and M4.

Finally, tests were performed on three meshes as shown in Figure 6. The first mesh M5 is unstructured with almost uniform tetrahedral elements. The mesh M6 with tetrahedral element was initially refined in the shock waves regions and the last mesh M7 has distorted hexahedral elements. The mesh M2 is similar to mesh M7 without distorted elements. The meshes M5, M6 and M7 consisting of 3828, 6356 and $41 \times 21 \times 2$ nodes, respectively. The three meshes contains 14580, 27595 and 800 elements, respectively.

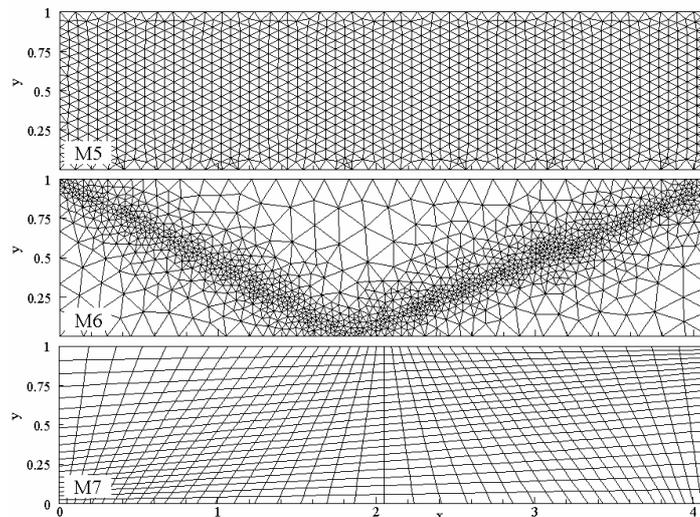


Figure 6: Meshes M5, M6 and M7 with hexahedral and tetrahedral elements

Table 2 compares the mass specific error in both $L_1(\Omega)$ and $L_2(\Omega)$ norms for hexahedral / tetrahedral elements with the one-step and the two-step schemes. In structured meshes (M7) a two-step scheme is lightly less accurate than the one-step scheme, while in unstructured meshes (M5 and M6) a one-step is slightly less accurate than the two-step scheme. The same behavior is observed with meshes M1, M2, M3 and M4.

The mesh with distorted hexahedral elements (M7) is less accurate than the mesh with regular hexahedral elements (M2). It is evident that the distorted element has a strong influence in the accuracy.

meshe s	norm L_1		norm L_2	
	one-step	two-step	one-step	two-step
M2H	0.0665	0.0758	0.00327	0.00352
M5	0.221	0.218	0.00664	0.00658
M6	0.299	0.288	0.00524	0.00501
M7	0.0771	0.0879	0.00358	0.00386

Table 2: Mass specific error for the example of the reflection of an oblique shock problem with meshes M5, M6 and M7.

4.2 Transonic viscous flow around an airfoil

To investigate these schemes in viscous flows and the influence of different type of elements, the flow around a NACA 0012 airfoil is computed. The flow is specified by $M_\infty = 0.80$, the viscosity by $Re = 500$ and an angle of attack is equal to 10 deg. With these conditions, a separation bubble extends over more than an half of the upper surface. The airfoil wall is assumed to be adiabatic, and non-slip condition is specified for the velocity on the surface of the airfoil.

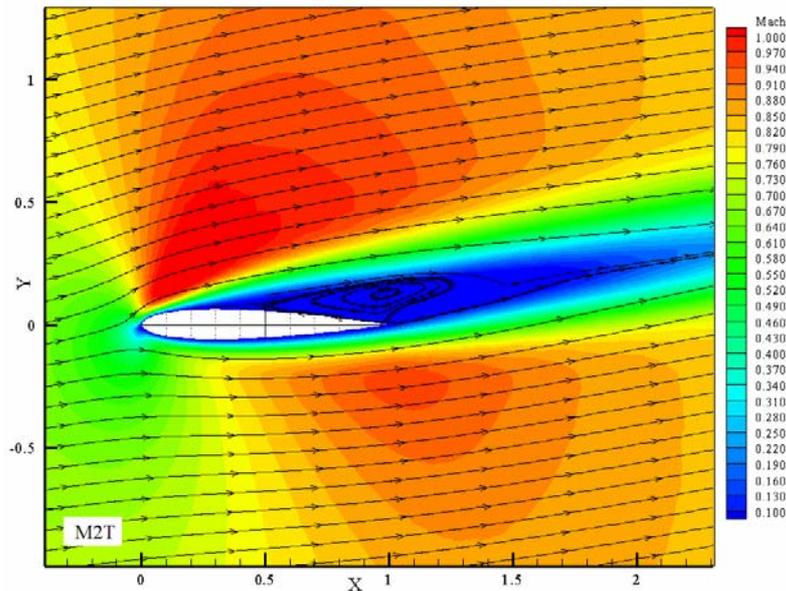


Figure 7: Mach number contours on the mesh M2 with tetrahedral elements

Solutions were obtained for three meshes consisting of 19680, 27860 and 40450 nodes,

respectively. The meshes with hexahedral elements have 9600, 13600 and 19800 elements meanwhile the meshes with tetrahedral have 48000, 68000 and 99000 elements, respectively. The mesh with tetrahedrals is obtained by subdividing each hexahedral into five tetrahedrals.

A sample of Mach number contours and the streamlines at the mesh with 27860 nodes and 68000 tetrahedrals using the one-step scheme is shown in Figure 7.

Contours of pressure for the mesh with 19680 nodes and 9600 hexahedrals using the one-step and two-step schemes are shown in Figures 8-*a* and 8-*b*. Both solutions are very similar and agree with those reported in Tang and Hafez (2001). A comparison of the friction coefficient distributions and the pressure coefficient distributions obtained from the finest meshes with hexahedral and tetrahedral elements using the one-step scheme are plotted in Figure 9. The present results are also in good agreement with those reporters by Hafez and Wahba (2007). In the leading ant trailing edges the difference between the present solutions and Hafez and Wahba’s solution for the friction coefficient distributions may be due to different formulations used.

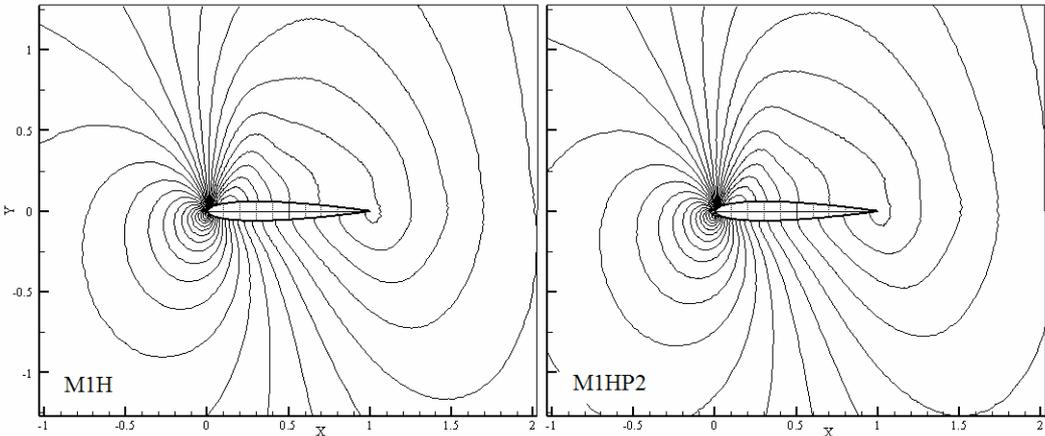


Figure 8: Contours of pressure on the mesh with hexahedral elements for the NACA 0012 airfoil using the one-step and two-step schemes (*a* and *b*, respectively)

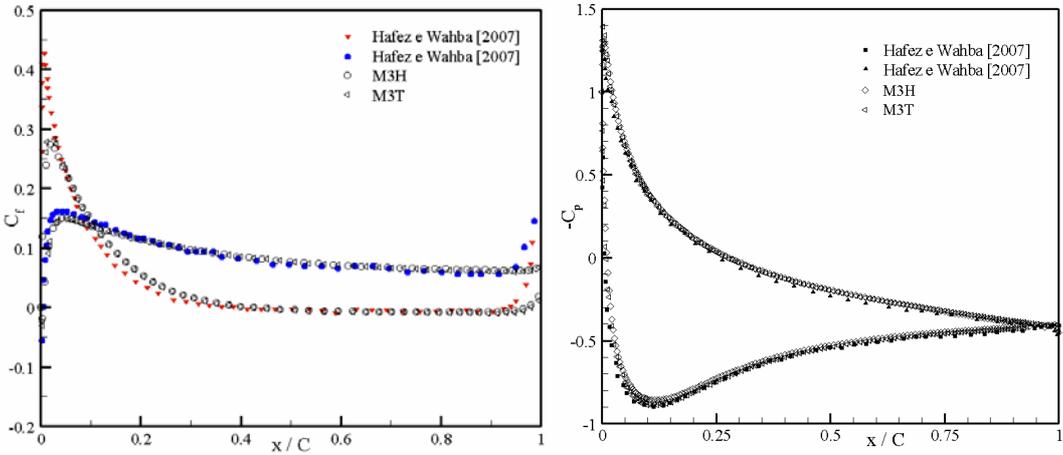


Figure 9: Friction coefficient and pressure coefficient distributions over a NACA 0012 airfoil, compared with result obtained by Hafez and Wahba (2007)

The behavior of the drag and lift coefficients with grid refinement on the hexahedral (H) and tetrahedral (T) meshes and the one-step / two-step (P2) schemes are plotted in Figure 10,

in order to further quantify the accuracy level of the computed pressure distribution. The artificial damping coefficient (CAF) used with the one-step / two-step schemes were 0.9 and 0.8, respectively.

It is interesting to note the monotone behavior of the drag coefficient with mesh refinement. The small difference between the lift coefficient on the M2 and M3 meshes indicate that the solutions on those meshes may be regarded as nearly mesh independent, however, the difference for the drag coefficient on those meshes is not quite grid independent yet. The drag and lift coefficients obtained by Forsyth and Jiang (1997) ranged between $0.2430 \leq C_d \leq 0.2868$ and $0.4145 \leq C_l \leq 0.5170$, respectively. The mesh with tetrahedral elements using the two-step scheme underpredicts the drag and lift coefficients compared with the same meshes using the one-step scheme.

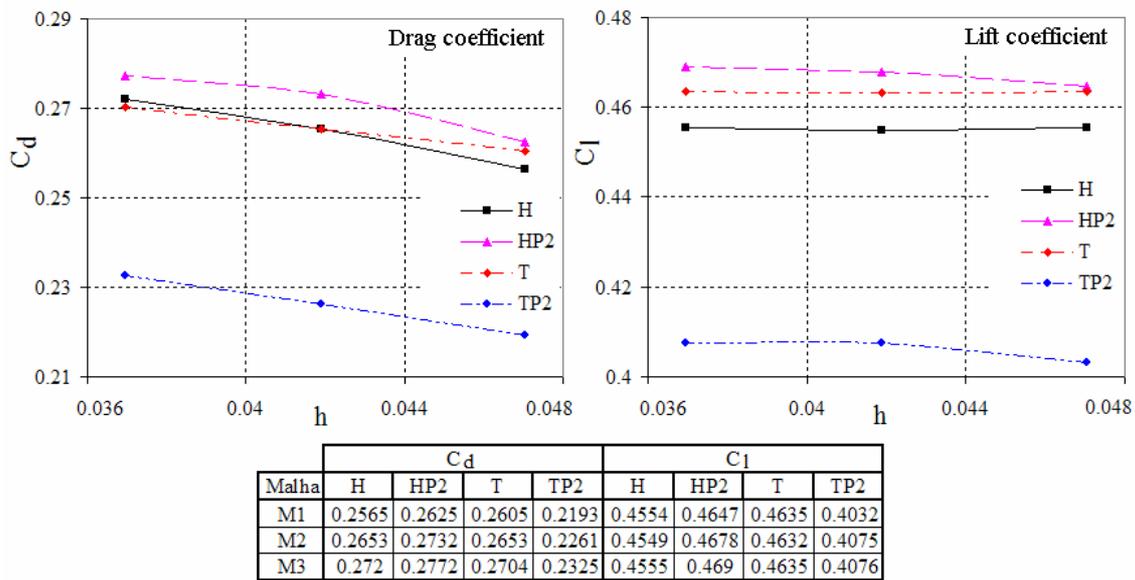


Figure 10: Variations of the drag and lift coefficients for the transonic viscous flow around a NACA 0012 airfoil

5 CONCLUSION

This paper emphasizes the accuracy and the efficiency of two explicit Taylor-Galerkin schemes in the context of meshes with hexahedral and tetrahedral elements. A comparison among the one-step and two-step scheme using hexahedral and tetrahedral elements leads to the following conclusions: (a) Simulations with meshes employing hexahedral elements, using either the one-step or the two-step scheme, are more accurate and robust in both example studied here, with respect to results employing meshes of tetrahedral elements; (b) In structured meshes a two-step scheme is slightly less accurate than the one-step scheme, while in unstructured meshes a one-step scheme is slightly less accurate than the two-step scheme, for the non-viscous flow problem; (c) Simulations with meshes employing tetrahedral elements using the two-step scheme is the most inaccurate in the case of the transonic viscous flow. Clearly, the artificial viscosity plays a fundamental role in the accuracy. The one-step scheme is less diffusive than the two-step scheme allowing to take a smaller value of the coefficient CAF in Equation (16).

As in the one-step scheme some vectors and matrices are stored to avoid their calculation during the iterative process, the two-step scheme requires less memory (approximately 75 %

with respect to the one-step scheme) but it demands more processing time.

In spite of the conclusions given previously, it is necessary to take into account that meshes with tetrahedral elements are more suitable for complex geometries. Another important subject is mesh adaptation, which is an essential tool to obtain accurate results in problems involving, for example, strong shock waves, as in the first application (the inviscid flow). Adaptive meshes will be considered in future works.

6 ACKNOWLEDGEMENTS

The authors would like to thank CNPq, which supported the project under the Edital Universal 2004 No. 472053/2004-3.

REFERENCES

- M. Aftosmis, D. Gaitonde and T.S. Tavares. Behavior of linear reconstruction techniques on unstructured meshes. *AIAA Journal*, 33:2038–2049, 1995.
- J. Argyris, I.S. Doltsinis and H. Friz. Study on computational reentry aerodynamics. *Computer Methods in Applied Mechanics and Engineering*, 81:257–289, 1990.
- T.J. Baker. Mesh generation: art or science. *Progress in Aerospace Science*, 41:29–63, 2005.
- J. Donea. A Taylor-Galerkin for convective transport problems. *International Journal for Numerical Methods in Engineering*, 20:101–119, 1984.
- P.A. Forsyth and H. Jiang. Nonlinear iteration methods for high speed laminar compressible Navier-Stokes equations. *Computer & Fluids*, 26:249–268, 1997.
- M. Hafez and E. Wahba. Simulations of viscous transonic flows over lifting airfoils and wings. *Computer & Fluids*, 36:39–52, 2007.
- T.J.R. Hughes and T.E. Tezduyar. Finite element methods for first-order hyperbolic systems with particular emphasis on the compressible Euler equations. *Computer Methods in Applied Mechanics and Engineering*, 45:217–284, 1984.
- M. Kawahara and H. Hirano. A finite element method for high Reynolds number viscous fluid flow using two step explicit scheme. *International Journal for Numerical Methods in Fluids*, 3:137–163, 1983.
- F. Shakib, T.J.R. Hughes and Z. Johan. A new finite element formulation for compressible fluid dynamics: X. The compressible Euler and Navier-Stokes equations. *Computer Methods in Applied Mechanics and Engineering*, 89:141–219, 1991.
- C. Tang and M. Hafez. Numerical simulation of steady compressible flows using a zonal formulation. Part II: viscous flows. *Computer & Fluids*, 30:1003–1016, 2001.
- F.W. White. *Viscous fluid flow*. McGraw Hill, 1974.