

A steady pseudo-compressibility approach based on unstructured hybrid finite volume techniques applied to turbulent premixed flame propagation

Wladimir M. C. Dourado

ASA-P – IAE – CTA

São José dos Campos - 11228-904 – SP – Brazil

wladimyrmd@terra.com.br

Pascal Bruel

Centre National de la Recherche Scientifique

Laboratoire de Combustion et de Détonique

BP 40109 - 86961 Futuroscope Chasseneuil Cedex FRANCE

bruel@lcd.ensma.fr

João L. F. Azevedo

ASE-N – IAE – CTA

São José dos Campos - 11228-904 – SP – Brazil

azevedo@iae.cta.br

Abstract. A pseudo-compressibility method for turbulent reactive flows with heat release is adapted to an unstructured finite volume hybrid grid scheme. Premixed reactants are considered and a flamelet approach for combustion modeling is adopted, using a continuous quenched mean reaction rate. The spatial discretization is an overlapped cell vertex approach. Additional artificial dissipation is added to stabilize the final discretized governing equations. To integrate the final form of discretized governing equations, a three-stage hybrid time-stepping scheme is used. An infinite planar flame propagating freely is considered as test case, and the calculated solution using the proposed method coincides with the results available in the literature.

Keywords: turbulent combustion, quenched flame, finite volume method, unstructured hybrid grid, pseudo-compressibility method.

1. Introduction

Low Mach number steady and unsteady reactive flows in the sense of Majda and Sethian, 1985, are often encountered in many systems of practical interest such as furnaces or jet engines. In such cases, the pressure can be considered as being thermodynamically constant e.g. the density variations are uniquely related to the temperature changes due to the heat released by the combustion process. When compressible flow solvers are employed in such situations, their convergence rate is becoming so slow that specific procedures such as preconditioning techniques have to be employed to improve them, but in the limit $Ma \rightarrow 0$ even the recourse to such techniques is not sufficient to ensure a satisfactory convergence rate (Dourado and Azevedo, 1996, Dourado and Azevedo, 1999, Dourado et al., 2000). Accordingly, specific numerical strategies have to be employed such as the Simple, the PISO or the pseudo-compressibility technique (Chorin, 1967). The latter technique presents the immense advantage over the two others that its introduction in a compressible flow solver is easy and leaves the structure of the computer program largely unmodified while allowing the calculations of both inert and reacting flows (Corvellec, 1998, Corvellec et al., 1999).

On the other hand, during the last two decades, the finite volume unstructured grid technique has shown its ability to cope with a wide range of complex compressible flow geometries (Jameson et al., 1986, Barth and Jespersen, 1989) including the use of mesh refinement techniques (Batina, 1989, Trépanier et al., 1991, and Nomura and Hughes, 1992) that are appealing for properly resolving the high spatial gradients that are potentially present in the flows under investigation.

Thus, the temptation is strong to combine the advantages of both the pseudo-compressibility method and the finite volume unstructured grid technique to develop a computer code able to deal with low Mach number inert or reacting flows.

Along these lines, the present work is devoted to the development and the test of such a combination that results in the development of a 2-D computer code that will be tested on the geometry of an 1-D freely

propagating turbulent premixed flame whose velocity is known, thanks to the formulation retained for the combustion modeling.

2. Physical Modeling

2.1. Combustion Model

Since the present study is mainly concerned with the numerical method of solution of the governing equations rather than the modeling of the combustion process itself, only a brief description of the modeling is given here. The formulation retained is similar to that of Corvellec et al., 1999. This choice is motivated by the fact that such a modeling yields a turbulent premixed flame that propagates at a velocity that can be directly derived from the so-called KPP analysis (Kolmogorov et al., 1937, Corvellec et al., 1999). Consequently, such a test case permits an unambiguous evaluation of the capability of the code to predict the correct turbulent flame speed which is intrinsic to the set of equations considered. So, the recourse to the comparison with experimental data is not required at this stage as long as it is not the physical modeling itself which is to be tested.

A simplified Bray-Moss-Libby like formulation (Bray and Moss, 1977, Bray et al., 1984, Bray et al., 1988, thereafter referred to as BML) with a quenched mean reaction rate is used. Accordingly, the isenthalpic combustion process takes place in infinitely thin-laminar like reactive interfaces (flamelets) whose thickness is less than the Kolmogorov scale and whose laminar flame velocity is negligible compared to the velocity fluctuations associated with turbulence. An observer recording the temperature signal at a given point within the flow would obtain basically two values of the temperature signal e.g. $T = T_b$ in the fully burnt products and $T = T_r$ in the unburned mixture. The combustion process can then be represented through the evolution of a single bi-valued progress variable c ($c = 0$ in unburned mixture and to $c = 1$ in the fully burnt products) which plays the role of a reduced temperature via the following relation:

$$c = \frac{T - T_r}{T_b - T_r} . \quad (1)$$

The zero Mach number hypothesis leads to the following equation of state:

$$\rho T = \rho_r T_r = \rho_b T_b . \quad (2)$$

or alternatively :

$$\rho = \frac{\rho_r}{1 + \chi c} , \quad (3)$$

where the heat release parameter χ is defined by:

$$\chi = \frac{(T_b - T_r)}{T_r} . \quad (4)$$

With use of the classical Favre average (Favre, 1965), the governing equation for the mean progress variable $\tilde{c} = \frac{\overline{\rho c}}{\bar{\rho}}$ can be written as:

$$L(\tilde{\rho} \tilde{c}) = \bar{w} , \quad (5)$$

where the L operator regroups the convective and diffusive operators.

To ensure the existence of a unique propagation velocity for the turbulent flame brush, a continuous quenched form for the mean reaction rate \bar{w} is chosen (Corvellec, 1998, Corvellec et al., 1999), namely:

$$\bar{w} = \begin{cases} 0 & \text{if } \tilde{c} \leq \chi c^* \\ C_w (1 + \chi) \frac{(\tilde{c} - \chi c^*)(1 - \tilde{c})}{(1 + \chi c^*)^2} & \text{otherwise. } otherwise \end{cases} \quad (6)$$

As explained in the forthcoming sections, the governing equations are discretized by a cell vertex method using a centered spatial discretization scheme. Therefore, the source term is treated differently in comparison with the approach presented by Corvellec et al., 1999, and it is not given in split form.

2.2. Pseudo-Compressibility Technique

The pseudo or artificial compressibility technique was proposed initially by Chorin, 1967, to deal with inert incompressible flows. It has been extended to steady reactive flows by Bruel et al., 1996, and to unsteady reactive flows by Corvellec et al., 1999. The basic idea behind this technique consists in modifying the continuity

equation to introduce a finite sound speed during the course of the convergence process toward the solution. Thus, the continuity equation permits *de facto* the calculation of the static pressure and reads as:

$$\frac{1}{\beta} \frac{\partial \bar{p}}{\partial \tau} + \frac{\partial \bar{\rho} \tilde{u}_i}{\partial x_i} = 0, \quad (7)$$

where β is the pseudo-compressibility factor expressed in $m^2 s^{-2}$ and τ is the pseudo-time. This equation is physically meaningful when convergence is reached in τ , e.g. when $\frac{\partial \bar{p}}{\partial \tau}$ vanishes. The pseudo-sound speed is given by:

$$a = \sqrt{u^2 + \beta}, \quad (8)$$

and the local corresponding pseudo-Mach number M_a^* defined by $M_a^* = \frac{u}{a}$ is thus always less than one as soon as β is larger than zero. For a 1-D turbulent flame and using an implicit structured finite differences approach, Bruel et al., 1996, have shown that i) the value of the pseudo-compressibility factor β influences strongly the convergence rate of the calculations and ii) a value $\beta \simeq 5u_{ref}^2$ to $10u_{ref}^2$ where u_{ref} is the largest convective velocity expected in the flow is found to be a good compromise in terms of convergence rate.

3. Mean Governing Equations

In spite of the fact that the test case considered here is one-dimensional, the governing equations are cast into a two-dimensional form since an unstructured approach for dealing with such equations cannot be formulated otherwise. For the simplified combustion model retained here, the main mean variables that describe the turbulent reactive flow are the density $\bar{\rho}$, the Cartesian velocity components \tilde{u}_i , the static pressure \bar{p} and the progress reaction variable \tilde{c} . The steady governing equations that include the pseudo-compressibility approach are given by:

$$\frac{\partial q}{\partial \tau} + \frac{\partial E_{e_i}}{\partial x_i} - \frac{\partial E_{v_i}}{\partial x_i} = S \quad (9)$$

with $x_1 \equiv x$ and $x_2 \equiv y$, the various vectors can be expressed as:

$$q = \begin{Bmatrix} \bar{p} \\ \bar{\rho} \tilde{u} \\ \bar{\rho} \tilde{v} \\ \bar{\rho} \tilde{c} \end{Bmatrix} \quad E_{e_x} = \begin{Bmatrix} \beta \bar{\rho} \tilde{u} \\ \bar{\rho} \tilde{u}^2 + \bar{p} \\ \bar{\rho} \tilde{u} \tilde{v} \\ \bar{\rho} \tilde{u} \tilde{c} \end{Bmatrix} \quad E_{e_y} = \begin{Bmatrix} \beta \bar{\rho} \tilde{v} \\ \bar{\rho} \tilde{v} \\ \bar{\rho} \tilde{v}^2 + \bar{p} \\ \bar{\rho} \tilde{v} \tilde{c} \end{Bmatrix} \\ E_{v_x} = \begin{Bmatrix} 0 \\ \frac{\sigma_{xx}}{\rho D} - \bar{\rho} \widetilde{u''u''} \\ \frac{\sigma_{xy}}{\rho D} - \bar{\rho} \widetilde{u''v''} \\ \frac{\rho D \frac{\partial c}{\partial x} - \rho \widetilde{u''c''}}{\rho D} \end{Bmatrix} \quad E_{v_y} = \begin{Bmatrix} 0 \\ \frac{\sigma_{xy}}{\rho D} - \bar{\rho} \widetilde{u''v''} \\ \frac{\sigma_{yy}}{\rho D} - \bar{\rho} \widetilde{v''v''} \\ \frac{\rho D \frac{\partial c}{\partial y} - \rho \widetilde{v''c''}}{\rho D} \end{Bmatrix} \quad S = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ \bar{w} \end{Bmatrix}, \quad (10)$$

where \bar{w} is the mean reaction rate given by Eq. (6) and σ_{ij} is the laminar stress tensor expressed for a Newtonian fluid by:

$$\sigma_{ij} = \mu_l \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right). \quad (11)$$

In Eq. (10), $D = \frac{\nu_l}{Sc_l}$ is the laminar diffusion coefficient, ν_l the kinematic laminar viscosity and Sc_l the laminar Schmidt number. The Reynolds stresses, present in Eq. (10) are closed with a Boussinesq-like expression (Frisch, 1995), namely:

$$\bar{\rho} \widetilde{u''_i u''_j} = -\mu_t \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) + \frac{2}{3} \left(\bar{\rho} \tilde{k} + \mu_t \frac{\partial \tilde{u}_k}{\partial x_k} \right) \quad (12)$$

where $\tilde{k} = \frac{1}{2} \widetilde{u''_i u''_i}$ is the turbulence kinetic energy. The turbulent viscosity coefficient can be expressed as a function of \tilde{k} and of its dissipation rate $\tilde{\epsilon}$ by (Launder and Spalding, 1974):

$$\mu_t = \bar{\rho} C_\mu \frac{\tilde{k}^2}{\tilde{\epsilon}}, \quad (13)$$

with C_μ being equal to 0.09.

In the present work, the turbulence field is supposed to be frozen at the level that prevails in the fresh reactants so that $\tilde{k} = \tilde{k}_r$ and $\tilde{\epsilon} = \tilde{\epsilon}_r$ with:

$$\tilde{k}_r = \frac{3}{2} u_r'^2, \quad \tilde{\epsilon}_r = C_\mu^{0.75} \frac{\tilde{k}_r^{3/2}}{l_i}, \quad (14)$$

where $u_r' = \sqrt{u_r'^2}$ and l_i , to be prescribed, stand for the rms velocity fluctuations and the turbulence integral length that prevail in the fresh reactants, respectively.

The turbulent mass flux $\widetilde{\rho u'' c''}$ present in Eq. (10) is also expressed through a gradient approximation, namely (Bray et al., 1984):

$$\widetilde{\rho u'' c''} = -\bar{\rho} D_t \frac{\partial \tilde{c}}{\partial x_i}, \quad (15)$$

where $D_t = \frac{\nu_t}{Sc_t}$ is the turbulent diffusion coefficient, $\nu_t = \mu_t/\bar{\rho}$, the kinematic turbulent viscosity and Sc_t the turbulent Schmidt number.

The closed form of Eqs. (9) is now written in an integral form using the Green's theorem:

$$V \frac{\partial}{\partial \tau} \int_V q dV + \oint_S \mathbf{E}_e \cdot \mathbf{\bar{n}} dS = \oint_S \mathbf{E}_v \cdot \mathbf{\bar{n}} dS + \int_V \mathbf{S} dV, \quad (16)$$

with $\mathbf{\bar{n}}$ being the outgoing normal unitary vector to surface S of volume V . In two dimensional Cartesian notation, the $\mathbf{\bar{n}} dS$ product is equal to $\Delta y \vec{i} - \Delta x \vec{j}$ where \vec{i} and \vec{j} are the basis vectors. \mathbf{E}_e and \mathbf{E}_v are the convective and diffusion flux vectors whose components are given in Eq. (10).

4. Spatial Discretization

As the control volume can be constructed *a priori* with different type of elements (triangles and/or quadrangles) on hybrid grid topologies, the elementary cell of the grid considered here is simply called an element. Then, the control volume V is defined by the external boundary surface of all elements sharing a common node, as sketched in Fig. (1).

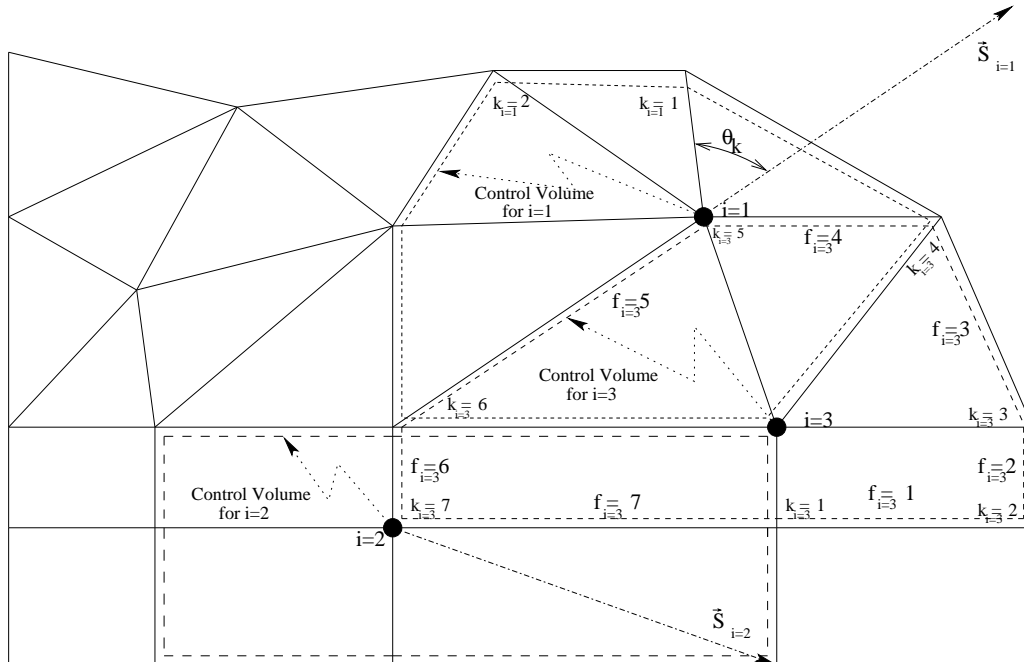


Figure 1: Domain of influence of nodes 1, 2 and 3 and its local stretching for hybrid unstructured grid.

The spatial discretization adopted here is based on a cell center finite volume method adapted for cell vertex hybrid elements (Jameson et al., 1986). This scheme proved to be efficient for dealing with steady and unsteady inert flows as reported by Dourado and Azevedo, 1996, Dourado and Azevedo, 1999, Dourado et al., 2000, Mavriplis et al., 1989, Mavriplis and Venkatakrishnan, 1995, and Mavriplis, 1998.

The discretized form of Eq. (16) is written as:

$$V \frac{\partial \bar{q}_i}{\partial \tau} + \sum_{f=1}^{nf} (\bar{E}_{e,f} \Delta y - \bar{E}_{e,y} \Delta x) = \sum_{f=1}^{nf} (\bar{E}_{v,f} \Delta y - \bar{E}_{v,y} \Delta x) + V \bar{S} , \quad (17)$$

where

$$\bar{q} = \frac{1}{V} \int_V \mathbf{q} dV \quad (18)$$

is the average main variables vector, and

$$\bar{S} = \frac{1}{V} \int_V \mathbf{S} dV \quad (19)$$

is the average source term vector. \bar{E} is the average flux vector at face f calculated as the arithmetic average of the two fluxes located on nodes which delimit this face. Further details on triangular and hybrid elements in unstructured grids are presented in the studies that have been referred to above.

5. Artificial Dissipation

For high Reynolds number flows, it is frequently required to stabilize the calculations by adding explicitly some artificial dissipation to damp out numerical oscillations. These additional dissipative terms must be kept at a level that does not endanger the accuracy of the numerical solution. Accordingly, and as suggested by Dourado et al., 2000, the dissipation terms introduced in the systems of equations are written as:

$$D_a(q_i) = \sum_{k=1}^{nn} [\Delta_k - \Delta_i] \left[\frac{A_k + A_i}{2} \right] , \quad (20)$$

where

$$\Delta(q) = -k^{(4)} \nabla^2 q , \quad (21)$$

and nn is the number of nodes located on the boundary of the control volume i , as it is sketched in Fig. (1).

The constant $k^{(4)}$ which defines the level of the fourth order dissipation term is taken equal to $1/256$ for all the calculations presented in this work and the coefficient A is adjusted according to the grid stretching by:

$$A_k = \alpha_{1k} \cos^2 \theta_k + \alpha_{2k} \sin^2 \theta_k , \quad (22)$$

where i and k subscript refer to the variable evaluation at nodes i and k , respectively. At each mesh point, the coefficients α_1 and α_2 of Eq. (22) are calculated with the maximum eigenvalue λ associated with the convective terms by:

$$\alpha_1 = \phi(s) \frac{1}{s+1} \lambda , \quad \alpha_2 = \phi(s^{-1}) \frac{s}{s+1} \lambda ,$$

where s represents the magnitude of the stretching vector \vec{S} (see Fig. (1)) and $\phi(r) = 1 + r^{2/3}$. For an unstructured mesh, the maximum eigenvalue λ at each mesh point is evaluated by the following discrete approximation:

$$\lambda = \sum_{f=1}^{nf} |\bar{u}_f \Delta y_f - \bar{v}_f \Delta x_f| + \bar{a}_f \sqrt{\Delta x^2 + \Delta y^2} , \quad (23)$$

with \bar{u} , \bar{v} and \bar{a} being the average values of the velocity components and of the sound speed calculated from the two nodes which delimit the face f and cumulated over all the boundary faces of the considered control volume. Finally, since the code is edge based database organized, the artificial dissipation term formulation was enhanced in order to be usable in the context of hybrid grids where triangular and quadrilateral elements coexist.

One should observe that, although quadrilateral meshes were used in the present simulations, the code treats all grids fully unstructured. The reader should be aware that the definition of an unstructured grid is not necessary related to the shape of the control volumes but to the form in which these control volumes are addressed in the code.

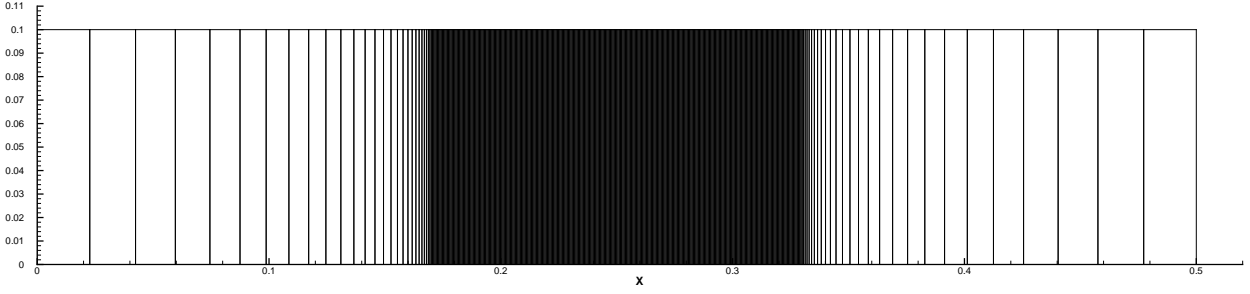


Figure 2: Computational mesh with 710 nodes and 354 quadrilateral elements named here as *mesh 1*.

6. Time Marching

As exposed before, when one has recourse to the pseudo-compressibility method to solve the flow governing equations, the solution is physically meaningful only when a steady state in pseudo-time is reached, so, even to seek steady state solutions, a time-stepping procedure has to be used.

Thus, the discretized equations system (17) is first written as:

$$V_i \frac{dq_i}{d\tau} + [C(q_i) - Dv(q_i) - D_a(q_i)] = S(q_i) \quad i = 1 \dots n. \quad (24)$$

where the residuals of the convective, diffusive and stabilizing terms are denoted by $C(q)$, $D_v(q)$ and D_a , respectively, and the source term is represented by $S(q)$.

Then, a Runge-Kutta explicit three-stage hybrid time-stepping scheme is adopted due to its satisfactory performance, simplicity and low computational cost. The scheme proposed by Manzari et al., 1998, to advance the solution in pseudo-time is implemented here under the following form:

$$q_i^{(0)} = q_i^n \quad (25)$$

$$q_i^\kappa = q_i^{(0)} - \alpha_\kappa \frac{\Delta\tau}{V_i} [C(q_i^{(\kappa-1)}) - D_v(q_i^{(\kappa-1)}) - S(q_i^{(\kappa-1)}) - D_a(q_i^{(\kappa-1)})] \quad \text{for } \kappa = 1, 2, 3 \quad (26)$$

$$q_i^{n+1} = q_i^{(3)}. \quad (27)$$

The operators C , D_v and S are created using the properties vector $q_i^{(\kappa-1)}$ from stage $\kappa - 1$, while the added artificial dissipation term is calculated only in the first stage and held constant throughout the next stages. The coefficients appearing in Eq. (27) are given the values $\alpha_1 = 0.6$, $\alpha_2 = 0.6$ and $\alpha_3 = 1.0$.

7. Results

The 1-D turbulent premixed flame referenced by Corvellec, 1998, as *Case I* is considered here to test the present numerical approach. The parameters defining such a slow propagating flame are given in Table 1.

Table 1: Parameters defining the 1-D turbulent premixed flame considered as test case.

χ	χc^*	S_t (m/s)	\bar{u}_r (m/s)	u_r' (m/s)	l_i (m)	Sc_t	C_w ($kg\ m^{-3}\ s^{-1}$)	ρ_r ($kg\ m^{-3}$)
5	0.03958	0.5	0.5	1.0	0.001	0.75	200	1.1886

The boundary condition imposed at the reactants side e.g. $\bar{u}_r = S_t$, $\tilde{c} = \bar{c} = 0$, has been chosen so that the turbulent flame brush should be steady in the computational domain which consists of a strip of $0.5\ m$ long and $0.1\ m$ wide with only one division in the latter direction. An example of grid, called *mesh 1*, with 355 nodes in the longitudinal direction with 354 quadrilateral elements is presented in Fig. (2). In order to properly resolve the strong gradients that are to be expected near the quenching point, the mesh is highly refined in the region located between $0.17\ m \leq x \leq 0.33\ m$ with a minimum mesh spacing Δx_{min} equal to $5.333 \times 10^{-4}\ m$.

Due to the high Reynolds number flow hypothesis, the laminar diffusive transports will be neglected in front of their turbulent counterparts. To ensure 1-D like conditions on such a 2-D mesh, a symmetry condition has been imposed at the lower and upper boundary lines parallel to the flow and are applied to the corresponding nodes and faces. This condition corresponds to the imposition of zero transverse velocity and gradients. At the burned products side, the pressure is imposed and all other variables are extrapolated.

The initial condition for the progress variable \tilde{c} is given by a ramp form function that goes from $\tilde{c} = 0$ at $x = 0.27\ m$ to $\tilde{c} = 1$ at $x = 0.32\ m$. The Courant-Frederich-Lewis (CFL) number used in these calculations was

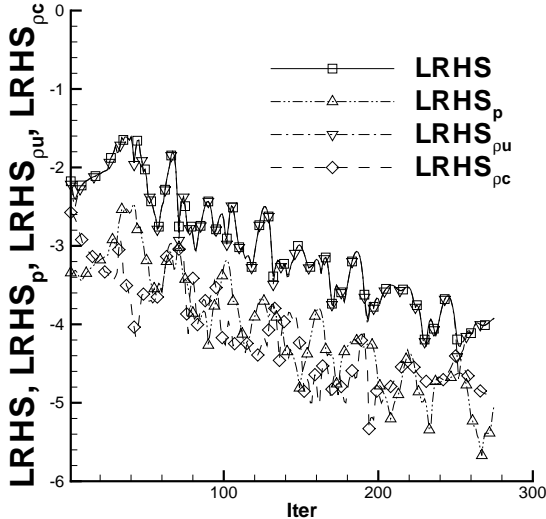


Figure 3: Convergence history of residue using the *mesh 1* with $\beta = 75 \text{ (m}^2/\text{s}^2)$.

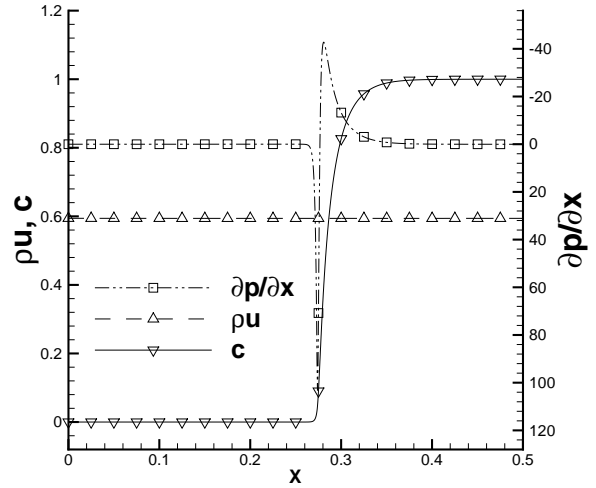


Figure 4: Profiles of mean reaction progress variable, continuity equation (ρu) and pressure gradient along x direction for *mesh 1* with $\beta = 75 \text{ (m}^2/\text{s}^2)$.

equal to 0.94 and is constant throughout the domain. Accordingly, the pseudo time step $\Delta\tau$ is variable in the computational domain with a minimum value used in the region of greatest refinement. Such a CFL number is given by:

$$CFL_{\tau} = \lambda_b \frac{\Delta\tau}{\Delta x_{min}} \quad , \quad (28)$$

where $\lambda_b = u_b + a_b$ is the largest convective eigenvalue based on the pseudo-sound speed given by Eq. (8). As a consequence, the minimum pseudo-time step for *mesh 1* is equal to $2 \times 10^{-5} \text{ s}$. The convergence criterion set to 10^{-4} was met after 27500 iterations through a convergence history presented in Fig. (3). The corresponding profiles of pressure gradient, mean progress variable and mass flux obtained on *mesh 1* are presented in Fig. (4).

The curvature at the foot of the flame brush is extremely strong and this illustrates why such a highly stretched mesh has to be used to capture properly the progress variable and pressure gradients in that zone. The profile of pressure gradient plotted against the progress reaction variable \tilde{c} is displayed in Fig. (5).

A good agreement between the present results and those of Corvellec, 1998, is obtained with a difference in the maximum pressure gradient of less than 6%. Consequently, the criterion convergence chosen appears to be correctly adapted to get accurate converged results.

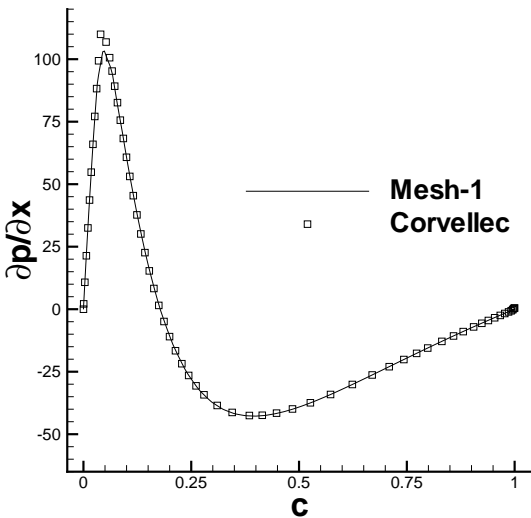


Figure 5: Profile of pressure gradient versus mean reaction progress variable for the *mesh 1* with $\beta = 75 \text{ (m}^2/\text{s}^2)$.

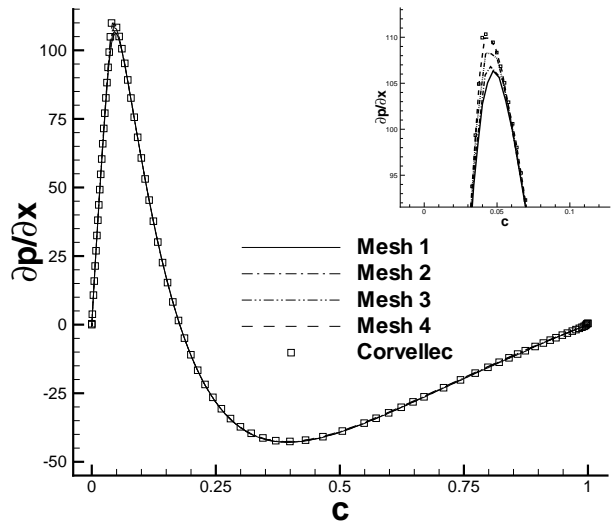


Figure 6: Profile of pressure gradient versus mean reaction progress variable for the *mesh 1, 2, 3 and 4*.

To investigate the sensitivity of our results to the mesh refinement, three other meshes have been considered that differ from *mesh 1* by their refinement: i) *mesh 2* possesses 50 % more divisions in the x direction in

the interval $0.17m \leq x \leq 0.33m$ with $\Delta x_{min} = 3.56 \times 10^{-4} m$; ii) *mesh 3* has 100% more divisions in the x direction on the same interval, which gives $\Delta x_{min} = 2.67 \times 10^{-4} m$ and iii) *mesh 4* has the same number of nodes as *mesh 3* but the interval of refinement is reduced to $0.22m \leq x \leq 0.30m$, which gives a minimum mesh discretization $\Delta x_{min} = 1.33 \times 10^{-4} m$. It should be remembered that in her study, Corvellec, 1998, was using an auto-adaptive mesh refinement with a Δx_{min} equal to $7.1 \times 10^{-5} m$. The results obtained with the different grids are presented in Fig. (6) where the profiles of pressure gradients are plotted versus the mean reaction progress variable. It can be clearly seen that as soon as the mesh is sufficiently refined, no substantial difference can be observed with the results by Corvellec, 1998, obtained, it should be stressed with a completely different numerical formulation.

A short analysis of the influence of the choice of the pseudo-compressibility factor $\hat{\beta}$ on the convergence rate was carried out here and Fig. (7) presents the convergence history of the residue of x -direction momentum equation observed for several $\hat{\beta}$. The dependence of the number of iterations required to meet the convergence criterion $LRHS_{\rho u} \leq 10^{-4}$ on the value of $\hat{\beta}$ is presented in Fig. (8).

The optimum value of $\hat{\beta}$ around 10 obtained here confirmed the results reported by Bruel et al., 1996, and show that such an optimum is poorly dependent on the numerical approach retained and thus can be considered as specific of the pseudo-compressibility approach itself.

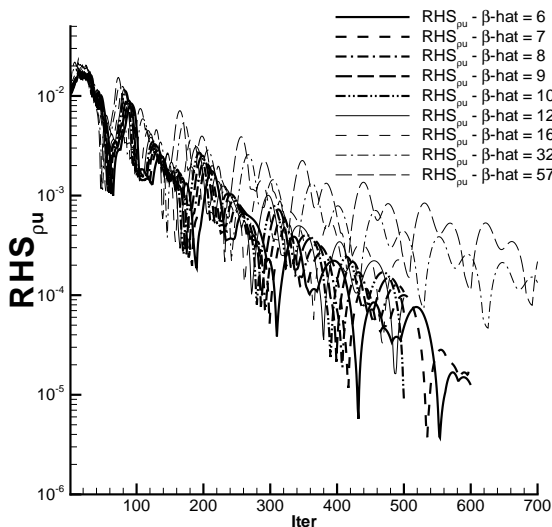


Figure 7: Error history for x -direction momentum equation for several reduced pseudo-compressibility factor $\hat{\beta}$ using *mesh 1*.

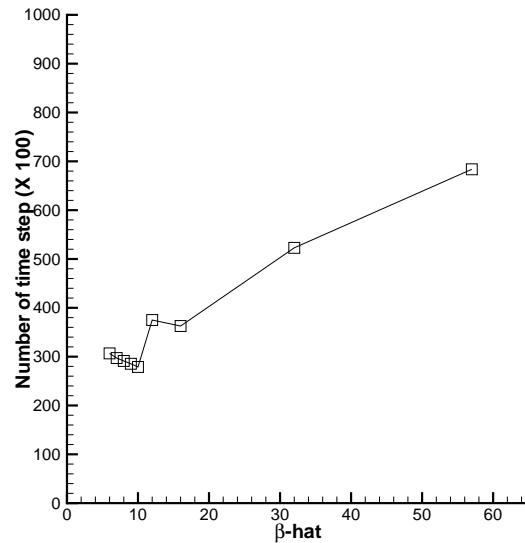


Figure 8: Number of time step necessary to achieve converged solution.

8. Concluding Remarks

The proposed combination of the pseudo-compressibility method with an unstructured formulation to solve the governing equations of a turbulent premixed flame proves to work properly and to produce results that compare well with those obtained in the literature on the very same flow geometry. Due to the centered type of spatial discretization used here, a fourth-order artificial numerical dissipation has been added to stabilize the calculations but the results show that the accuracy of the final solution has been preserved. The use of an explicit three-stage hybrid time stepping scheme shows that it is powerful enough to calculate the steady solution in the pseudo-time of the discretized governing equations (24). Nevertheless, the use of such an explicit time stepping scheme has the drawback of limiting the maximum possible CFL-number value resulting in an increased total number of pseudo-time steps required to achieve convergence. If needed, an implicit formulation could be considered to speed-up the convergence process. Finally, an auto-adaptive Lagrangian-Eulerian approach on unstructured grid (see Trépanier et al., 1991, and Nomura and Hughes, 1992) could greatly help in reducing the total number of grid elements required to resolve properly the high gradients present in the flow while preserving the overall accuracy of the calculations. Future work will focus on these directions.

9. Acknowledgments

The authors would like to acknowledge the financial support from Conselho Nacional de Desenvolvimento Científico e Tecnológico, CNPq, and Coordenação de Aperfeiçoamento de Pessoal de Nível Superior, CAPES. A large part of this work has been carried out during the stay of the first author at the Laboratoire de Combustion et de Détonique (Poitiers, France) that belongs to the Centre National de la Recherche Scientifique (CNRS)

and in the framework of a "co-tutelle" thesis prepared under the co-supervision of the University of Poitiers and ITA. The authors further acknowledge the partial support of CNPq under the Integrated Project Research Grant No. 522.413/96-0.

10. References

- Barth, T. J. and Jespersen, D. C., 1989, The Design and Application of Upwind Schemes on Unstructured Meshes, "Proceedings of 27th AIAA Aerospace Sciences Meeting", number AIAA-89-0366 in AIAA Aerospace Sciences Meeting. AIAA.
- Batina, J. T., 1989, Vortex-Dominated Conical-Flow Computations Using Unstructured Adaptively-Refined Meshes, "Proceedings of AIAA 20th Fluid Dynamics, Plasma Dynamics and Lasers Conference", number AIAA 89-1816 in AIAA Fluid Dynamics, Plasma Dynamics and Lasers Conference. AIAA.
- Bray, K. N. C., Champion, M., and Libby, P. A., 1988, Mean Reaction Rates in Premixed Turbulent Flames, "22th Symposium (International) on Combustion", pp. 763–769. The Combustion Institute.
- Bray, K. N. C., Libby, P. A., and Moss, J. B., 1984, Flamelet Crossing Frequencies and Mean Reaction Rates in Premixed Turbulent Combustion, "Combustion Science and Technology", Vol. 41, pp. 143–172.
- Bray, K. N. C. and Moss, J. B., 1977, A Unified Statistical Model of the Premixed Turbulent Flame, "Acta Astronautica", Vol. 4, pp. 291–319.
- Bruel, P., Karmed, D., and Champion, M., 1996, A Pseudo-Compressibility Method for Reactive Flows at zero Mach Number, "International Journal of Computational Fluid Dynamics", Vol. 7, pp. 291–310.
- Chorin, A. J., 1967, A Numerical Method for Solving Incompressible Viscous Flow Problems, "Journal of Computational Physics", Vol. 2, pp. 12–26.
- Corvellec, C., 1998, "Numerical and Analytical Study of a Combustion Zone Propagating Characteristics Developing on Premixed Turbulent Flow Modeled by a Flamelet Approach Type", PhD Thesis, Université de Poitiers - Ecole Nationale Supérieure de Mécanique et d'Aérotechnique (ENSMA), (in French, original title is "Étude numérique et analytique des caractéristiques propagatives d'une zone de combustion se développant au sein d'un écoulement turbulent prémélangé modélisée par une approche de type *Flammelette*").
- Corvellec, C., Bruel, P., and Sabelnikov, V., 1999, A Time-Accurate Scheme for the Calculations of Unsteady Reactive Flows at Low Mach Number, "International Journal for Numerical Methods in Fluids", Vol. 29, pp. 207–227.
- Dourado, W. M. C. and Azevedo, J. L. F., 1996, Flow Simulation with Unstructured Meshes over Basic Automotive Configurations with All Speed Method, ABCM, editor, "Proceedings of the 6th Brazilian Congress of Engineering and Thermal Sciences and 6th Latin American of Mass and Heat Transfer, ENCIT/LATCYM 96", Vol. I, pp. 553–558, Florianópolis, SC. ABCM, (in Portuguese, original title is "Simulação de escoamentos com malhas não-estruturadas sobre configurações automotivas básicas com um método para toda a faixa de velocidade").
- Dourado, W. M. C. and Azevedo, J. L. F., 1999, Analysis of an All Speed Method in Laminar Flows Using Unstructured Meshes, ABCM, editor, "15th Brazilian Congress of Mechanical Engineering," Vol. CD-ROM, Águas de Lindóia, SP - Brazil. ABCM.
- Dourado, W. M. C., Azevedo, J. L. F., and Bruel, P., 2000, Application of the All-Speed Method to Steady and Unsteady Flows on Unstructured Grids, ABCM, editor, "8th Brazilian Congress of Thermal Engineering and Sciences", Vol. CD-ROM, Porto Alegre, RS, Brazil.
- Favre, A., 1965, Compressible Turbulent Gas Equations, "Journals de Mécanique", Vol. 4, pp. 361–391, (in French, original title is "Equations des Gaz Turbulents Compressibles").
- Frisch, U., 1995, "Turbulence", Cambridge University Press.
- Jameson, A., Baker, T. J., and Weatherill, N. P., 1986, Calculation of Inviscid Transonic Flow over a Complete Aircraft, AIAA, editor, "Proceedings of the 24th AIAA Aerospace Sciences Meeting", number AIAA 86-0103 in AIAA Aerospace Sciences Meeting, Reno, Nevada.
- Kolmogorov, A., Petrovsky, I., and Piscounov, N., 1937, Study of the Diffusion Equation with Growth of the Quantity of Matter and its Application to a Biology Problem, "Bulletin of The Moscow State University, International Serie", Vol. 1, Section A, No. 6.

- Launder, B. E. and Spalding, D. B., 1974, The Numerical Computational of Turbulent Flows, "Computer Methods in Applied Mechanics and Engineering", Vol. 3, pp. 269–289.
- Majda, A. J. and Sethian, J., 1985, The Derivation and Numerical Solution of the Equations for Zero Mach Number Combustion, "Combustion Science and Technology", Vol. 42, pp. 185–205.
- Manzari, M. T., Hassan, O., Morgan, K., and Weatherill, N., 1998, Turbulent Flow Computation on 3D Unstructured Grids, "Finite Elements in Analysis and Design", Vol. 30, pp. 353–363.
- Mavriplis, D. J., 1998, Multigrid Strategies for Viscous Flow Solvers on Anisotropic Unstructured Meshes, ICASE Report 98-6, ICASE, NASA Langley Research Center, Hampton, Virginia 23681-2199.
- Mavriplis, D. J., Jameson, A., and Martinelli, L., 1989, Multigrid Solution of the Navier-Stokes Equations on Triangular Meshes, NASA CR- 89-11, Institute for Computer Applications in Science and Engineering - ICASE, NASA Langley Research Center Hampton, Virginia 23665-5225.
- Mavriplis, D. J. and Venkatakrishnan, V., 1995, A Unified Multigrid Solver for the Navier-Stokes Equations on Mixed Element Meshes, ICASE Report 95-53, ICASE - Institute for Computer Applications in Science and Engineering, Mail Stop 132C, NASA Langley Research Center Hampton, VA 23681-0001.
- Nomura, T. and Hughes, T. J. R., 1992, An Arbitrary Lagrangian-Eulerian Finite Element Method for Interaction of Fluid and Rigid Body, "Computer Methods in Applied Mechanics and Engineering", Vol. 95, pp. 115–138.
- Trépanier, J. Y., Reggio, M., Zhang, H., and Camarero, R., 1991, A Finite-Volume Method for the Euler Equations on Arbitrary Lagrangian-Eulerian Grids, "Computers and Fluids", Vol. 20, No. 4, pp. 399–409.