Mesh adaptation for compressible flows

Bernadette Palmerio

UNITÉ DE RECHERCHE
INRIA-SOPHIA ANTIPOLIS

Rapports de Recherche

N°1683

Programme 6
Calcul scientifique, Modelisation et logiciels numeriques

MESH ADAPTION FOR COMPRESSIBLE FLOWS

Bernadette PALMERIO

Mai 1992

Institut National
de Recherche
en Informatique
et en Automatique

Sophia Antipolis
B.P. 109
06561 Valbonne Cedex
France
Tél.: 93 65 77 77

MESH ADAPTION FOR COMPRESSIBLE FLOWS

Bernadette PALMERIO

Laboratory of Mathematics, CNRS URA 168
University of Nice, Parc Valrose, 06000 NICE, FRANCE
and INRIA, 2004 route des lucioles, 06560 VALBONNE, FRANCE

Invited paper at **ICFD Conference on Numerical Methods for Fluid Dynamics**, april 7-10 1992, Reading (UK).

ABSTRACT : We describe here some recent developments in mesh adaption for compressible viscous inviscid flow simulation. One of the most challenging task is to capture efficiently thin layers such as boundary layers, mixing layers. Two important issues, the coupling between mesh and flow, and the use of flat elements for stretching are considered in the case of deformation mesh systems. The paper is illustrated by results obtained by INRIA-Rocquencourt, INRIA-Sophia-Antipolis and Dassault-Saint Cloud teams **[4,5,28,29]**, **[12,16]**, **[18,22]**.

METHODES DE MAILLAGES ADAPTATIFS POUR DES ECOULEMENTS COMPRESSIBLES

RESUME : On présente une synthèse sur les différents méthodes et ingrédients concernant l'adaption de maillage pour des écoulements compressibles visqueux ou non. Le but principal est en général de capturer des structures minces telles que couches limites ou couches de mélanges. Ce contexte fait apparaître deux points importants, le couplage maillage-écoulement et l'usage d'éléments plats, que l'on utilise dans le cas de déformations de maillage. Cette présentation est illustrée par des travaux réalisés à l'INRIA-Rocquencourt, à l'INRIA-Sophia-Antipolis et par une équipe de Dassault-Saint Cloud **[4,5,28,29]**, **[12,16]**, **[18,22]**.

Introduction

One of the principles of numerical analysis is to compute the solution of approximate problems, instead of the solution of continuous ones. The accordance between the discrete solution and the continuous one results from various, often antagonist, options such as choosing a fine mesh or saving computer cost. In most cases, the discrete solution is constructed in two phases:
Firstly a mesh is generated from the point of view of taking into account geometrical data, and **then** the solution is computed on the mesh, from the physical data, by solving a differential system.
This two-phase point of view is not the most reasonable one. Indeed it generally a posteriori appears that the first mesh is not conveniently adapted to the continuous solution that was looked for ; this can be observed locally in the computational domain; in some regions, the mesh is non necessarily refined; in some other ones, important details are not satisfactorily captured.
In fact, the real problem is to find a couple mesh+solution, that allows obtaining with a cost as low as possible an approximation to a given accuracy of the continuous problem. Let us remark that, especially in CFD, a level of accuracy is generally what is required since it is generally decided by the mathematical/physical model; for example, in many industrial cases, the full potential transonic model is not to be computed in more than one percent of accuracy, since the model itself does not represent the real flow that accurately. A second remark is that we generally need to know the flow variables only on a few locations, such as the body; thus accuracy may have not to be uniform.

Finally a complete adaptive CFD analysis is a method in which:

- *m-method*: the model has to be chosen, may be adaptively in the future (e.g. applying local turbulence models).

- *p-method*: the local accuracy has to be chosen.

- *h-method*: the mesh has to be chosen.

Note that the local efficiency could also be taken into account (through an *ad hoc* combination of structured and non structured methods for example).
In the sequel we shall not consider these types of adaptivity and we restrict to a fixed order method; in this condition, both local density and local connectivity have to be handled, this refers to *h-r methods*, in which these two features are handled with differents ways. For example, in case of element division and node movement this type of combination leads still to rather tricky problems and no theoretically well based strategy exists (see **[17]** for effort in this direction).

1 Spatial convergence and error estimate

Before reviewing all the difficulties existing in 2-D and 3-D, and the many methods to handle them, we shall discuss some definitions (ideas, notions) in the 1-D case

which allows not to consider the topology difficulties.
The problem is the following: find the solution of a differential problem on a given interval to a given accuracy, for the smallest cost. The standard theory for approximation assumes that more or less explicitly the mesh is uniform, since the "cost" of the mesh is only measured by the mesh size (finite differences, finite volume) or the maximal mesh size (finite element).
The order of approximation of the method is therefore a kind of rule to estimate the price to pay for reaching a given accuracy with a series of uniform meshes. This point of view calls two remarks.
Shock sharp capturing: The calculation of a solution having a discontinuity on the interval is generally done in this context (uniform mesh) by shock capturing. Unfortunatly, the accuracy is limited to first-order by both, *(firstly)*, the difficulty to represent the shock by an interpolation of discrete values,*(secondly)*, the mechanism for stabilizing the approximation near the discontinuity.
Therefore the global efficiency is only first order, that is, accuracy is a linear function of the inverse of the CPU cost, assuming that the CPU cost is proportional to the number of nodes.
Shock/transition capturing: Let us consider now that the solution to be seeked presents, instead of a discontinuity a sharp gradient in one point, while being a regular function; think for example to a quasi-shock in the weakly viscous Burgers' equation solution. In this case, the standard theory claims that asymptotically, when Δx tends to zero, second-order accuracy is obtained. Unfortunately, this behaviour does appear only if the sharp gradient is conveniently **captured**, that is, if a sufficient number of nodes is devoted to its representation.
Conversely, if the mesh is too coarse with respect to the width of this sharp transition, then the approximation is not able to distinguish it from a discontinuity and will behave as a first-order scheme.
Now one important point in CFD is that generally the computation of the solution is costly and its meshes are just fine enough to capture difficult details, so that mesh convergence (convergence of the discrete solution to continuous solution) may essentially appear as first-order.
As sketched below, this point of view does not take into account an accurate measure of the cost. Let us consider now that meshes are not uniform and that (for the sake of simplicity) the computational cost is proportional to number of nodes. The two above examples are now considered with adaptive grids, and in terms of number of nodes. When adding nodes, it is possible to devote a convenient number of nodes to capturing the shock; for example, with a second-order method it is not expensive to divide by four the local space increment, Δx in the neighbouring of the discontinuity at each new mesh, in which the mesh size is divided by two elsewhere; in the regular intervals, since the mesh is divided by two, the local error is divided by four (second-order accuracy); near the discontinuities, since the mesh is divided by four, so is the local error. It can be therefore expected that, with essentially twice as many nodes, we obtain an error four times smaller, so the second-order accuracy is recovered.
For the case of a **sharp transition**, this approach will behave in a similar manner

and second-order accuracy can be obtained much sooner (in terms of number of nodes) than with an uniform mesh.
This point of view can be applied to evaluate the efficiency of an adaptative method with respect to uniform mesh and to other adaptative methods.

2 Criterion

It is a quantity that allows to give in a local manner informations about either the quality of the current mesh with respect to the flow and/or some features of a family of more desirable meshes. There are many families of criteria:
(1)*criteria based on* **truncation error**, (2) *criteria based on* **approximation error**, (3) *criteria based on* **convergence measures**, (4) *criteria based on* **variational principles**, (5) *criteria based on* **physical quantities**

2.1 Truncation error related criteria

2.1.1 Criteria using the general context of discrete approximation

This criteria family measures how the equation to be solved is satisfied by the discrete solution. This can be done by computing on each node a finite difference truncation error and by approximating further derivatives by divided differences. The approximation error ${e^*}_h$ between the discrete solution U_h of $A_hU_h = f_h$ and the continuous solution U of $AU = f$ is obtained from the truncation error $e_h = A_hU - f_h$ by using the inverse of the discrete equation:

$$\begin{cases} AU = f \quad , \quad A_hU_h = f_h \\ e_h = U - U_h \quad , \quad t_h = A_he_h \quad , \quad e_h{}^* = A_h^{-1}t_h \end{cases}$$

The truncation error is the most local part of an approximation; the truncation error is a really local measure of the approximation error and therefore it is very interesting to use it for local refinement. It is possible to evaluate the error with only taking into account $AU_h - f$, in which, either A is approximated with the first terms of the equivalent or modified equation (these terms are then by finite differences), or A is replaced by another approximation method (this point of view is Gear's one for ordinary differential equations [**8**]). Generally, the second scheme is (or, is assumed to be) more accurate, but it can also be an approximation on a dual set of points (for example, one elementwise least-square approximation can be used to control a Galerkin approximation).

2.1.2 Criteria relying on particular properties of the scheme

A recent idea is to introduce "control approximation schemes" that present less numerical viscosity; for example, the error of TVD calculations [**21**] can be evaluating from central difference residual.

2.1.3 Criteria using mathematical properties

A last idea [18], [4] consists of evaluating error through a differential equation, that is not discretized in the system to solve, but that is essentially satisfied by the solution of the continuous system; we refer as an example for the conservation of entropy S along streamlines for steady cases, the criterion is chosen equal to $U.\nabla S$.

2.2 Criteria based on approximation error

The approximation error is still more difficult to measure. In the elliptic case a theory has been developed by Babuska [1] and co-workers. This has been extensively used by several authors [17], [7], [24].

2.3 Criteria based on convergence measures

More direct evaluations of the mesh can be obtained when nested meshes are used. Indeed the difference $U_{2h} - U_h$ between the solution obtained on nested meshes is a measure of the mesh efficiency, taking in particular into account the capture phenomenon described in Section 1.
If nested meshes are not applied, one idea is to extend the well known mesh convergence measure of the spectral methods. Indeed when we apply a Chebyshev approximation to a given problem, the coefficient of the polynomials decreases with the polynomial number; when the n-th coefficient is at zero machine, it is said that a maximum accuracy is attained and that no extra polynomial is needed. In the case of a one-dimensional uniform mesh, it is possible to apply this kind of strategy: we can construct a sequence of finite-element basis functions with decreasing mesh size and with Gram-Schmidt orthogonality; then mesh convergence can be measured from the scalar products between these basis functions and the obtained finite-element solution. Extension to non-structured meshes is likely difficult; simplified versions can be derived by building only one level coarser than the current mesh.

Rather popular principles that are also used, are relying on the error finite element analysis through interpolation error. By Cea's lemma

$$|U - U_h| \leq C\,|U - P_h U| \tag{1}$$

The interpolation error $|U - P_h U|$ is a majorant of the approximation error; it can be splitted in three orthogonal directions; this allows directional refinements or stretchings. We refer to [22] and to [19] for the implementation of the directional stretching.

2.4 Criteria based on variational principles

A last class of mesh adaption criteria rely on variational principles. The idea is to consider the extra variables corresponding to the node coordinates in a similar way as flow variables at these nodes. In the Moving Finite Element method [**3,14,15**], the Galerkin variational principle is applied. Another option is to consider least square minimization. In both cases, the problem is non-linear with respect to the node location variables; it is then more complicated that the corresponding usual non adaptive problems.

2.5 Criteria based on physical quantities

A last option is to try to capture a family of specified structures in the solution by choosing a criterion that will be a sensor for in the term of gradient or further order derivatives, that are assumed to be found in these structures. We refer to [**5**] for examples.

3 Constructing the new mesh

3.1 Basic remarks

3.1.1 Relation between flow solver and mesh adaptor

An important notion in the mesh adaption theory is whether or not there is some coupling between mesh and solver. We shall call a **static adaptator** any process that will compute a mesh from a given flow solution obtained on an initial mesh without computing any extra flow features. This software can a priori be combined with different flow solvers. **Dynamic adaption** solves a coupling between mesh regeneration and flow solution. It can be performed, for example, by the alternation of flow solution and static adaption, either for time evolutive calulations or for accurate steady solution.

3.1.2 Metric and topology

The two important features of a mesh are the **metric** and the **topology**. We call **topology** the set data described by integers; for example, in structured case topology is reduced to the number of nodes in each direction; in block-structured one, topology is defined as these two integers for each block, together with integers defining the connectivity; in a non-structured finite element context, it can be defined as element to element neighbouring topology; for example, the number of elements type of each element (triangles, quadrilaterals), the type of interpolation and local connectivity (boundary and internal elements) can be the basic data of the topology. The **metric** is the set of all the real numbers describing the mesh; typically it is the set of coordinates.

3.1.3 Main options

From a very theoretical point of view, constructing the mesh at the same time the flow solution is computed should essentially multiply the complexity of the task

by a factor around two, in relation with the number of variables.
In fact, this analysis applies only when the number of nodes and the topology are known; in practical cases the topology has to be adapted, which represents a very heterogeneous set of informations, that is not easily mapped in a finite dimensional vector space. This is a reason why many methods are static ones, the main test for classifying static mesh adaption is considering whether the topology is changed; we therefore distinguish variable-topology and constant-topology mesh methods. when the topology is not changed, we shall call it mesh deformation.

3.2 Variable topology methods

3.2.1 Mesh enrichment methods

They rely on the distinction between regions of the mesh that are kept as they are, and other regions that are supposed to become finer. Generally, nodes are added and connectivity with the unchanged part of the mesh has to be organized; this connectivity can be either conformal or not:
Conformal reconnection can be done by applying a local mesh regenerator such as a Voronoi generator **[2], [13]** or an advancing front generator **[23]**. Another method for dividing is to apply some division patterns **[18], [25]**. Several of these methods tend to improve the mesh quality **[22]**, **[2]**, **[13]**, **[24]**.

Figure 1: Unsteady flow generated by hot jet, adapted mesh, Mach number contours, species contours, from **[12]**.

Non conformal reconnection relies on a numerical approximation method that extends to non conformal meshes; this approach is generally used for structured meshes, through hierarchical methods, that are also used in finite elements [**6**].
In **Figure 1** (see [**12**]) is shown a typical example of dynamical enrichment-unrefinement computation; in the corner of a vessel a supersonic jet is incoming; the proximity of a second duct will produce a dissymmetrical instability in the flow; the regions in which the flow is not trivial are very different between two time level of the computation, since the jet is continuously in expansion.

3.2.2 Reconstruction methods

When a fast mesh generator method is avalaible, it can be extended to mesh adaption; at each mesh modification, the mesh is globally re-built. We refer to (1) the advancing front method [**23**], (2) Voronoi mesh re-generators [**5,28,29**], [**2**], [**13**].

Figure 2: Incompressible Navier-Stokes flow past a NACA 001 2 with a Reynolds number equal to 1000; successive meshes (800, 1356, 576, 802 nodes) and corresponding iso-velocity contours, from [**28**].

In the reconstruction proposed by [**28**], the mesh is rebuilt by applying a

Voronoi mesh generator, in which the local two by two matrix is introduced and taken into account during the two phases of node introduction and diagonal swapping. An example of calculation is presented in **Figure 2**.

3.2.3 Method for adapting the metrics: deformation

Due to the constant topology, mathematical developments for mesh deformation methods are much more easy to derive. But huge difficulties remain in mesh admissibility (no overlapping). The deformation can be explicitly defined; this easier in the one dimensional case, we refer to **[16]** and even

The deformation can be computed from a discrete mesh system that can be a discretization of a continuous system or not (analytic, algebraic definitions). The definition of mesh systems began early with conformal mapping, and then with elliptic and hyperbolic mesh generators (see for example **[26]**, **[27]**). The main difficulty is to have a very general system that will fulfill the following conditions:
- being a mapping (no overlapping) from the reference geometry to the real one,
- to produce a mesh that satisfies some quality conditions (orthogonality, no obtuse angles, no unneeded stretching or element flattening),
- to be able to adapt to solutions that can have singularities: layers, shocks...

3.2.4 Mesh systems

Mesh systems are well developed in the case of structured meshes. The following effects are generally modelled: step space length, cell area, stretching, orthogonality.
In the case of non-structured meshes and in particular for triangles or tetrahedra, some equivalent notions exist: edge length, element volume/area, obtuse angles.
We obtain generally a discrete scheme, that depends only on immediate direct neighbours; in term of differential equations, this correspond to first and second derivatives.

3.2.5 Terms against overlapping, deformation case

The research of non-overlapped meshes leads to the choice between methods that strictly impose:
i) that the mesh does not overlap, even during the solution process,
ii) that it tends to produce, as the converged solution, a mesh that does not overlap.
In the first case, it is more easy to reach a mesh that is admissible/not overlapping; but generally the equation is not defined if the mesh is not admissible. One example has been proposed in **[20]** for the case of non-structured triangulations. Let us consider a subregion of a mesh that is made of one node and its neighbours. One method to impose that each neighboring triangle has a positive area is to model forces, the strength of which are proportional to the inverse of each triangle:

$$||f|| = \frac{1}{area(T)}$$

Therefore a force will have the right sense and a finite length when the area is strictly positive. When the area of T tends to zero, then the length tends to infinity; when the triangle is of zero area, the length is not defined; when a triangle has a negative area it is difficult to have in this way a force that will push toward an admissible mesh configuration. To put into a mesh system this model, one method consists of introducing in the mesh system a penalizing term proportional to the gradient of the inverse of element area.

$$\nabla\pi \ = \ \nabla\frac{1}{area(T)}$$

One disadvantage of this approach is that, because of nonlinearity, iterations must be applied for solution and these iterations must stay handling non-overlapping meshes, in order to keep the penalizing term well defined. One way to nonlinearly iterate is to apply a Gauss-Seidel iteration with a nodewise block (two by two) iteration with nonlinear Newton solution on each node, see **[20]** for details. A second option is to replace the above terms by differences that have still some meaning when the mesh is overlapped. For example, we can try to have uniform repartition of the areas; extra terms can be defined as follows:

$$\nabla\pi \ = \ \nabla(area(T))$$

A third alternative can be to minimize the difference between the largest and the smallest area of T:

$$\mathbf{Min}_{(}\mathbf{x_i},\mathbf{y_i}) \ = \ [\ \mathbf{Max}_{T neighbour\ of\ i}(area(T)) \ - \ \mathbf{Min}_{T neighbour\ of\ i}(area(T))]^2$$

One disadvantage of this approach is that it does not ensure that no area will be negative or zero at convergence of the iterative mesh system solver. Note that to obtain the advantage of both approaches we can use the first formula when the mesh is not admissible and the second one if yes.
A last remark is that these models can be considered either as penalizing terms or as terms contributing to adaptation, for example, by introducing a coefficient K depending on of flow that will monitor the size of each neighboring areas:

$$\nabla\pi \ = \ \nabla[K(flow)\ /\ area(T)]$$

3.2.6 Model for non-obtuse angles, deformation case

We do not know much work about it, we think that this point should be better investigated.

3.2.7 Terms for adaption to the flow, deformation case

The information that is available from the flow should ideally contain a direction of stretching, the strength of stretching and the local mesh size.
This should allow (see **[20]**) to apply stretching in the good direction. But this can be in conflict with existing topology and lead to flaten elements; in **[9]**, **[19]** the

stretching is operating through a spring model: a spring corresponds to each edge; the spring produces an attraction between the nodes at each extremity of the edge. The strength of the spring is monitored by the flow. This has allowed to obtain meshes rather stretched in arbitrary direction as illustred in the flow around a NACA0012 airfoil, with a farfield Mach number at infinity equal to 8, see **Figure 3**. The combination of repulsion terms with adaptive attractive ones seems a useful provisional approach. Indeed, for many adaptive models, elements with zero surface are perfect, since the truncation error is zero. One main disadvantage of this attraction-repulsion combination is that the local ellipticity of the system can change its sign and degenerate; this can lead to numerical difficulties.

Figure 3: Euler flow past a NACA0012 with a Mach number equal to 8 and an angle of attack equal to 20 degrees, initial and final mesh (600 nodes), from **[20]**.

4 Coupling the mesh and the flow

4.1 Nonstationary case

Mesh adaption for unsteady case is a rather old subject since Lagrangian methods and ALE **[11]** ones can be interpreted as mesh adapted unsteady methods. An illustration of the efficiency of the ALE approach in combination with an implicit scheme is showed with a calculation of a tulip flame instability from **[16]**, sketched in **Figure 4**, in which the mesh is adapted in one direction. We want to emphasize that this quality of adaption is, as far as we know, not yet obtained for sharp layers that are not aligned with the mesh. At least in the case of Euler flows, mesh enrichment/coarsening algorithm can be sucessfully applied to problems of which the details of solution are very non uniformly distributed: see **[12]**. We want also to point out that, in the case of unsteady calculations, the efficiency of the adaption will strongly depend on the ability of the criterion to guess the areas of interest in the one or several coming time-step, we refer as example to **[12]**, **[31]**.

Figure 4: Capture of a tulip flame by unidirectional adaption ; adapted mesh and reaction rate contours corresponding to t=31.62ms (duct dimensions $L_x = 150$ mm, $L_y = 38.1$ mm), from **[16]**.

4.2 Stationary case

For steady cases, two factors generally lead to decouple the flow and the mesh modification/regeneration:
i) Handling modifications of the mesh topology in a coupled way with flow solution is a complex task (but it is done, for example with an unsteady enrichiment/coarsening method). The evaluation of convergence to steady-state is difficult.
ii) Mesh adaption is generally thought as an extra device to combine with one or several existing flow codes.
Because of these two factors, a static approach is generally chosen ; the mesh adaption phase is more or less applied interactively between two flow calculations.

We have recently (see **[22]**) investigated in which conditions this kind of approach is a convergence process toward a global (coupled) mesh+flow solution.
In the deformation model introduced in Section 3.2, a mesh **M** and a flow **W** are converged together if the mesh system $\mathcal{M}$ is solved with spring forces depending on the flow solution and if the flow solution **W** is computed on this mesh from the

system $\mathcal{F}$:
Flow equation: $\mathcal{F}(\mathbf{M}, \mathbf{W}) = 0$.
Mesh equation: $\mathcal{M}(\mathbf{M}, \mathbf{W}) = 0$.
In the spring model the strengths of springs are computed both from mesh quantities and flow ones. In the static approach two options have been applied:
Option 1: $\mathbf{W_i}$ is a value at node i on the old mesh,
Option 2: an interpolation algorithm is applied from the operator $\mathbf{I_{M^n -> M^{n+1}}}$ in order to define the value of $\mathbf{W}$ at the new location $\mathbf{X_i}$, $\mathbf{Y_i}$ in the new mesh. Therefore the following systems are solved:
Option 1: $\mathcal{M}(\mathbf{M^{n+1}}, \mathbf{W^n}) = 0$.
Option 2: $\mathcal{M}(\mathbf{M^{n+1}}, \mathbf{I_{M^n -> M^{n+1}}} \mathbf{W^n}) = 0$.

We first remark that Option 1 will give wrong results if only one step of static adaption is applied. Indeed the mesh will be concentrated, not on the location where sharp gradients are detected but in accordance to the new location of nodes in which these sharp gradients where located in the old mesh (Option 1).

Figure 5: Steady 1-D diffusion-convection problem; mesh convergence for option 1 and option 2, adapted solution, from **[22]**.

We want further emphasize that **Option 1 is also a bad option inside an interactive loop**; to illustrate this point we have compared the two options on a one-dimensional example, the advection-diffusion equation:

$$W_x - \epsilon W_{xx} = 0 \quad W(0) = 0 \quad W(1) = 0 \tag{2}$$

A TVD scheme combined with central difference for the diffusion term is applied to discretize the above differential system; a spring model with second derivatives of W as spring strength is applied for mesh system.

Figure 6: 2-D test-case, Supersonic flow over a flat plate, right: adapted mesh, left solution's Mach contours (Mach=3., min=0.00, max=2.80, ΔM=.5).

At each phase of the adaption the flow is exactly solved and the mesh system is fully converged. The convergence to an adapted solution is evaluated from the mean square of difference between two successive meshes. In **Figure 5** it is illustrated that Option 1 does not yield convergence, while Option 2 is fast and gives in practice a solution in five to ten static phases. Turning to 2-D experiments, we present in **Figure 6** what can be obtained by applying this approach to a flow around a flat plate .

An important issue is the behavior of the numerical scheme when the mesh is not fine enough; indeed the real thickness of the layer will be fitted only with adaption the final adaptive mesh and not with the initial one; with a coarse mesh the scheme can be either oscillatory or artificially viscous; in the second case, many layers will have their thickness over predicted and most part of the adaptive effort will be dedicaced to get progressively closer to exact thickness.
The method is also applied to a ramp flow chosen among the test cases of the Workshop [30] (Test case III.2). The farfield Mach number is 10. and the Reynolds number is 143800/m; the ramp angle is 20 degrees. The second derivative of the Mach number is used as an adaption criterion.

Figure 7a: Supersonic flow past a ramp, initial mesh.

An initial non adapted mesh is first used: it is already manually stretched; the stretching is obtain by using a geometrical progression and the thickness of the resulting mesh boundary layer is choosen in a good accordance with the viscous boundary to be captured; this mesh is presented in **Figure 7a**. The result of a first computation using this mesh is sketched in **Figure 7b**; we observe a very frequent problem with the manual stretching: at the separation location, the viscous layer is out of the mesh concentration. Also the thickness of the layer at its beginning is polluted by a coarse capture of the first stagnation point (edge of the plate).

Figure 7b: Supersonic flow past a ramp, final mesh.

We present in **Figure 7c** the result of one phase of mesh adaption, i.e. one mesh change followed by one new calculation of the flow ; main features are well captured, although the adaption criterion seem to take too much nodes out from the recirculation zone.

Figure 7c: Supersonic flow past a ramp, blow-up near the stagnation point in the final results, mesh and corresponding Mach contours.

5 Concluding remarks

In this talk, some important issues have been pointed out. For many applications, variable topology method (enrichment and coarsening, regeneration) proved to be very efficient, in a static point of view or in a unstationary dynamical one.
However, for viscous flows involving different type of layers, not only a static point of view is not sufficient since several (or even ten) static phases may be necessary to properly capture the existing layers, but also the cost constraints lead to apply very flat elements. We have shown that flat elements are easily handled if a mesh deformation is applied in one direction of the mesh topology (structured mesh); conversely, in variable topology methods, very flat elements are difficult or costly to obtain. Between these two stand-points, the capture of layers in arbitrary directions and on non structured meshes stays challenging. Finding methods that, like MFE, compute in a coupled manner the couple flow+mesh stays one of the main issues for the next years.

6 Aknowledgements

We thank Doctor M.J. BAINES for his invitation at ICFD Conference, we thank also Doctor A. DERVIEUX for kindly aiding us to prepare this paper and Doctors M.-O. BRISTEAU, H. GUILLARD, B. LARROUTUROU, N. MAMAN, B. MANTEL, B. N'KONGA, J. PERIAUX, M. G. VALLET for allowing us to reproduce figures from their works.

7 References

[1] I. BABUSKA, O.C. ZIENKIEWICZ, J. GAGO, E.R. de A. OLIVEIRA, Accuracy estimates and adaptive refinements in finite element computations, J. Wiley and son (1986).

[2] T.J. BAKER, Generation of tetrahedral meshes around complete aircraft, Numerical Grid Generation in Computational Fluid Dynamics, 425-436, J. Haüser, C. Taylor eds, Pineridge Press Swansea (1986).

[3] M. J. BAINES, An analysis of the moving finite-element procedure, Siam J. Numer. Anal., 28 No5 1323-1349 (1991). of Compressible Flow, University of Reading Report 4/85, (1985).

[4] M. O. BRISTEAU, J. PERIAUX, Finite element methods for the calculation of compressible viscous flows using self adaptive mesh refinements, Lecture Notes in CFD, Von Karman Institute for Fluid Dynamics, Belgium, March 1986.

[5] M. O. BRISTEAU, B. MANTEL, M. G. VALLET, Adaptive stretched Finite Elements for viscous incompressible and compressibles flows, in IMACS'91, july 22-26, Trinity College Dublin, Ireland , 413-414, vol.1 (1991).

[6] R. E. BANK, A. H. SHERMAN, Refinement algorithms and data structures for regular local mesh refinement, R. Stepleman et al. eds., IMACS/North-Holland

Publishing Company (1983).

[7] R. E. BANK, B. D. WELFERT, A posteriori error estimates for the Stokes problem, SIAM Journal Numer. Anal., 28 No3 591-623 (1991).

[8] C. W. GEAR, Numerical initial value problems in ordinary differential equations, Prentice Hall series in Automatic Computation (1971).

[9] P. A. GNOFFO, Finite Volume adaptive grid algorithm applied to planetary entry flowfields, AIAA J., 19 No 9, 1249- 1254 (1983).

[10] H. GUILLARD, R. PEYRET, on the use of spectral methods for numerical solution of stiff problems, Computational Methods on Appl. Mech. Eng., 66 17-43 (1988)

[11] C.W. HIRT, A. A. AMSDEN, J. L. COOK, An arbitrary Lagrangian-Eulerian computing method for all flow speeds, J. Comp. Phys. 14 (1974), 227-253.

[12] N. MAMAN., B. LARROUTUROU, Dynamical mesh adaption for two-dimensional reactive flow simulation, 3rd International Conference on Numerical Grid Generation and Related Fields, S. Arcilla, J. Hauser, J.F. Thompson eds, Elsevier 1991.

[13] D. J. MAVRIPLIS, Adaptive mesh generation for viscous flows using Delaunay triangulation, Journal of computational Physics 90, 271-291 (1990).

[14] K. M. MILLER, R. N. MILLER, Moving finite elements I, SIAM Journal Numerical Analysis, Vol 18 No 6, 1019-1032 (1981).

[15] K. M. MILLER, R. N. MILLER, Moving finite elements II, SIAM journal Numerical Analysis, Vol 18 No 6, 1033-1057 (1981).

[16] B. N'KONGA, G. FERNANDEZ, H. GUILLARD, B. LARROUTUROU, Numerical investigations of the tulip flame, comparisons with experimental results, to appear in Combustion Science and Technology.

[17] J.T. ODEN, Adaptive methods in Computational Fluid Dynamics , in Finite Element Analysis In Fluids 2-10, Huntsville, University of Alabama, U.S.A, Chung T.J., Karr G.R. eds, UAH Press, 1989.

[18] B. PALMERIO, V. BILLEY, J. PERIAUX, A. DERVIEUX, A Self adaptive refinement and Finite elements for solving the Euler equations, Numerical Methods For Fluid Dynamics II, 369-387, K.W. Morton, M.J. Baines eds, Clarendon Press , Oxford (1986).

[19] B. PALMERIO, A two dimensional F.E.M. adaptive moving node method for steady Euler flow simulation, Computer Methods in Applied Mechanics and Engineering 71 (1988), 315-340, North-Holland.

[20] B. PALMERIO, A. DERVIEUX, 2-D and 3-D unstructured mesh adaption relying on physical analogy, Numerical grid Generation in Computational Fluid Mechanics 88 653-663, Pineridge Press.

[21] B. PALMERIO, C. OLIVIER, A. DERVIEUX, On the relation between TVD and Mesh Adaption and application to Navier-Stokes Calculations, Notes on Numerical Fluid Mechanics, No. 29, Vieweg, Braunschweig (1990).

[22] B. PALMERIO, A. DERVIEUX, On weak and strong coupling between mesh adaptors and flow solvers, 12th International Conference on numerical methods in

Fluid Dynamics, University of Oxford, jul. 9-13 1990, Lecture Notes in Physics 371, 540-544 (1990), K.W. Morton ed., Springer-Verlag.

[23] K. MORGAN, J. PERAIRE, Finite-element method for compressible flows, Computational fluid dynamics, Lecture series 1987-04, von Karman Institute for fluid dynamics (1987).

[24] M. RIVARA, Adaptive finite element refinement and fully irregular and conforming triangulations, Accuracy Estimates and Adaptive Refinements in Finite Element Computations, 359-370, Edited By I. Babushka, O. C. Zienkiewicz, J. Gago, E. R. de A. Oliveira (1986), John Wiley & Sons Ltd.

[25] M. RIVARA, A grid generator based on 4-triangles conforming mesh-refinement algorithms, International Journal For Numerical Methods In Engineering, vol. 24, 1343-1354 (1997).

[26] J. L. STEGER, An introduction to grid generation using partial differential equations, the third Joint Europe/US Short Course in Hypersonics, october 4 (1990), RWTH Aachen, University of Technology, D-5100,Aachen, FRG.

[27] J.F. THOMPSON, Z. U. A. WARSI., C. W. MASTIN, Numerical grid generation, Foundations and applications, North-Holland (1985).

[28] M. G. VALLET, Génération de maillages anisotropes adaptés , Application à la capture de couches limites, INRIA Report 1360, dec. 1990.

[29] M. G. VALLET, F. HECHT, B. MANTEL, Anisotropic control of mesh generation based upon a Voronoi type method, 3rd International Conference on Numerical Grid Generation and Related Fields, 93-105, S. Arcilla, J. Hauser, J. F. Thompson eds, Elsevier 1991.

[30] Workshop “Hypersonic flow for reentry problems”, J. A. Desideri, R. Glowinski, J. Periaux eds, No.1 (1991), Springer Verlag.

[31] S. T. ZALESAK, R. LOHNER, Minimizing numerical dissipation in modern shock-capturing schemes, in Finite Element Analysis In Fluids, Huntsville, University of Alabama, U.S.A, 1205-1217, Chung T.J., Karr G.R. eds, UAH Press, 1989.