



HAL
open science

Adaptive anisotropic meshing for incompressible navier stokes using a VMS solver with boundary layer

Thierry Coupez, Elie Hachem

► **To cite this version:**

Thierry Coupez, Elie Hachem. Adaptive anisotropic meshing for incompressible navier stokes using a VMS solver with boundary layer. 5th International Conference on Adaptive Modeling and Simulation, ADMOS 2011, Jun 2011, Paris, France. pp.125-133 - ISBN: 978-849396403-0. hal-00724517

HAL Id: hal-00724517

<https://minesparis-psl.hal.science/hal-00724517v1>

Submitted on 21 Aug 2012

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

ADAPTIVE ANISOTROPIC MESHING FOR INCOMPRESSIBLE NAVIER STOKES USING A VMS SOLVER WITH BOUNDARY LAYER

T. COUPEZ* AND E. HACHEM†

MINES ParisTech, Center for Materials Forming (CEMEF)
UMR CNRS 7635, BP 207, 06904 Sophia-Antipolis, France

*e-mail: thierry.coupez@mines-paristech.fr

†e-mail: elie.hachem@mines-paristech.fr

Key words: Adaptive Modeling, Simulation

Abstract. In this work, we propose to show that adaptive anisotropic meshing based on a posteriori estimation can be addressed for incompressible Navier Stokes at High Reynolds number. The proposed a posteriori estimate is based on the length distribution tensor approach and the associated edge based error analysis. It will be shown that boundary layer can be produced automatically on an unstructured mesh basis. The Finite Element flow solver is based on a Variational MultiScale (VMS) method, which consists in here of decomposition for both the velocity and the pressure fields into coarse/resolved scales and fine/unresolved scales. This choice of decomposition is shown to be favourable for simulating flows at high Reynolds number. The stabilization parameters are determined rigorously taking into account the anisotropy of the mesh using a directional element diameter.

1 INTRODUCTION

Anisotropic mesh adaptation enables to capture scale heterogeneities that can appear in numerous physical or mechanical applications including those having boundary layers, shock waves, edge singularities and moving interfaces ([1], [2], [3], [4]). In these cases, discontinuities or gradient of the solution are highly directional and can be captured with a good accuracy by using anisotropic meshing at a low extra number of elements. Several approaches to build easily unstructured anisotropic adaptive meshes are then often based on local modifications ([5], [6], [7]...) of an existing mesh. Indeed, it mainly requires extending the way to measure lengths following the space directions and can be done by using a metric field to redefine the geometric distances. In parallel, theories of anisotropic a posteriori error estimation (*i.e.* [8]) have been well developed, leading to some standardization of the adaptation process; production of metrics from the error

analysis of the discretization error and steering of remeshing by these metrics. It follows that most adaptive anisotropic meshing techniques take a metric map as input. For practical reasons, meshing tools requires often a nodal metric map. In fact, during the remeshing operations, the elements are much more volatile than the mesh nodes and therefore defining fields on a continuous basis ease their reconstruction, interpolation or extrapolation. By following the lines in [1], we recall that a different route for the metric construction was described and it is done directly at the node of the mesh without any direct information from the element, neither considering any underlying interpolation. It is performed by introducing a statistical concept: the length distribution function. A second order tensor was introduced to approximate the distribution of lengths defined by gathering the edges at the node. Therefore, we can compute the error along each edge and in the direction of each edge.

In this work, we propose an extension of this theory to take into account Multicomponent fields (tensors, vectors, scalars). Rather than taking several metric intersection and calculation, we propose in here an easy way to account for several fields in the a posteriori analysis while producing a single metric field. Therefore, for the Navier Stokes equations, this approach is able to account for all components of the velocity field and to follow the interpolation error estimation for the velocity (and not the velocity norm). By using an adequate scaling, the boundary layers with highly stretched element are produced automatically. The proposed analysis is implemented in the context of adaptive meshing under the constraint of a fixed number of nodes, with the advantage to avoid evaluating precisely the constant arising in the error analysis and also to provide a useful tool for practical applications.

The paper is structured as follows. In section 2 we introduce the node based metric framework and describes the anisotropic mesh adaptation procedure governed by the length distribution tensor. In section 3, the interpolation edge error for multicomponent fields is described. Section 4 presents the developed VMS Navier-Stokes solver. Section 5 provides some numerical results and examples.

2 Length distribution tensor approach

The first step consists in building a continuous natural metric field of a given mesh. In another words, the desired metric field is for which all the edges have a unit length ([1]). This unit mesh metric is constructed directly at the nodes by collecting informations from the edges vectors as follows:

Let \mathbf{X}^{ij} be the edge vector and \mathbb{A}^{ij} be a transformation changing it into a unit length vector:

$$|\mathbb{A}^{ij} \mathbf{X}^{ij}| = 1 \text{ then for } \mathbb{M} = t\mathbb{A}^{ij}\mathbb{A}^{ij}, \text{ we get } (\mathbb{M} \mathbf{X}^{ij}, \mathbf{X}^{ij}) = 1.$$

\mathbb{M} is a positive symmetric tensor but not definite yet. In order to build \mathbb{M}^i such as the edge length of node, i , is almost equal to 1, the immediate solution is to sum up the

previous relation:

$$\sum_{j \in \Gamma(i)} (\mathbb{M}^i \mathbf{X}^{ij}, \mathbf{X}^{ij}) = \sum_{j \in \Gamma(i)} 1 \implies \mathbb{M}^i : \left(\sum_{j \in \Gamma(i)} \mathbf{X}^{ij} \otimes \mathbf{X}^{ij} \right) = |\Gamma(i)| \quad (1)$$

where $|\Gamma(i)|$ denotes the cardinality of subset $|\Gamma(i)|$. Consequently, the length distribution tensor at node i and denoted as \mathbb{X}^i is defined by:

$$\mathbb{X}^i = \frac{1}{|\Gamma(i)|} \sum_{j \in \Gamma(i)} \mathbf{X}^{ij} \otimes \mathbf{X}^{ij} \quad (2)$$

Note that, \mathbb{X}^i is a positive symmetric matrix when there exist at least d non aligned edge vectors.

3 Interpolation edge error for multicomponent fields

In what follows, we assume that the approximation error is proportional to the interpolation error, allowing using directly the proposed estimate. We begin by recalling that:

$$\exists c > 0, \|u - u_h\| < c \|u - \pi_h u\| \quad \text{with} \quad \pi_h u(\mathbf{X}^i) = u(\mathbf{X}^i), \quad \forall i \in \mathcal{N} \quad (3)$$

The basic idea is then to compute the error along the edge and in the direction of each edge. We start by introducing u as a multicomponent fields ($u_k; k = 1, \dots, n$). It can be seen as the assembly of multiple scalars, vectors, tensors fields, or the combination of any of these. Following the lines in [1], we compute the multiple gradient recovery operator defined by:

$$\mathbf{G}_k^i \cdot \sum_{j \in \Gamma(i)} ((\mathbf{X}^{ij} \otimes \mathbf{X}^{ij})) = \sum_{j \in \Gamma(i)} U_k^{ij} \mathbf{X}^{ij} \quad (4)$$

Then, by taking $\mathbf{U}^i = \sum_{j \in \Gamma(i)} U^{ij} \mathbf{X}^{ij}$, we get for each $k = 1, \dots, n$:

$$\mathbf{G}_k^i = (\mathbb{X}^i)^{-1} \mathbf{U}_k^i \quad (5)$$

In order to obtain a new metric depending on the error analysis, one has to recalculate a new length for each edge and then to use it for rebuilding the length distribution tensor.

Let s_{ij}^k , $i, j \in \mathcal{N}$ be the set of edges scaling (stretching) factors defined by:

$$\begin{aligned} \tilde{e}_{ij} &= (s_{ij}^k)^2 e_{ij} \\ |\tilde{\mathbf{X}}^{ij}| &= s_{ij}^k |\mathbf{X}^{ij}| \end{aligned} \quad (6)$$

where \tilde{e} and $|\tilde{\mathbf{X}}|$ denote respectively the target error and associated edge length.

The continuous metric field defined at the mesh nodes is obtained by considering:

- i) A be a given number of edges of the mesh,
- ii) $(e_{ij} = |\mathbf{G}^{ij} \cdot \mathbf{X}^{ij}|)$ be the calculated error along and in the direction of the edges.
- iii) $p \in [1, d]$ be an exponent to be defined.

We obtain:

$$\mathbb{M}^i = \left(\frac{1}{d} \sum_{j \in \Gamma(i)} (s_{ij}^k)^2 \mathbf{X}^{ij} \otimes \mathbf{X}^{ij} \right)^{-1} \quad (7)$$

where

$$s_{ij}^k = \left(\frac{\lambda}{e_{ij}^k} \right)^{1/p} \quad (8)$$

and

$$\lambda = \left(\frac{\sum_i \sum_{j \in \Gamma(i)} e_{ij}^{\frac{p}{p+2}}}{A} \right)^{\frac{p+2}{p}} \quad (9)$$

minimizes the error for a fixed number of edges. Moreover, when $s_{ij} = 1, \forall i, \forall j \in \Gamma(i)$ then the mesh is optimal. For the Navier-Stokes equations, we compute the error estimation based on the velocity vector and its norm. Therefore, u can be seen in this case as the combination of the normalized velocity components and the velocity norm. This will enable from one hand to adapt along the change of different directions whereas the velocity norm gathers informations only on the velocity magnitude.

4 VMS: incompressible Navier-Stokes solver

In this section the general equations of time-dependent Navier-Stokes equation are solved. The stabilizing schemes from a variational multiscale point of view are described and presented. Both the velocity and the pressure spaces are enriched which cures the spurious oscillations in the convection-dominated regime as well as the pressure instability.

Following the lines in [9], we consider an overlapping sum decomposition of the velocity and the pressure fields into resolvable coarse-scale and unresolved fine-scale $\vec{u} = \vec{u}_h + \vec{u}'$ and $p = p_h + p'$. Likewise, we regard the same decomposition for the weighting functions $\vec{w} = \vec{w}_h + \vec{w}'$ and $q = q_h + q'$. The unresolved fine-scales are usually modelled using residual based terms that are derived consistently. The static condensation consists in substituting the fine-scale solution into the large-scale problem providing additional terms, tuned by a local time-dependent stabilizing parameter, that enhance the stability and accuracy of the

standard Galerkin formulation for the transient non-linear Navier-Stokes equations. Thus, the mixed-finite element approximation of the incompressible Navier-Stokes problem can read:

$$\begin{aligned} & \text{Find a pair } (\vec{u}, p) \in \mathcal{V} \times \mathcal{Q}, \text{ such that: } \forall (\vec{w}, q) \in V_0 \times Q_0 \\ & \left\{ \begin{array}{l} \rho (\partial_t(\vec{u}_h + \vec{u}'), (\vec{w}_h + \vec{w}'))_{\Omega} \\ \quad + \rho ((\vec{u}_h + \vec{u}') \cdot \nabla(\vec{u}_h + \vec{u}'), (\vec{w}_h + \vec{w}'))_{\Omega} \\ \quad + (2\eta \underline{\dot{\epsilon}}(\vec{u}_h + \vec{u}') : \underline{\dot{\epsilon}}(\vec{w}_h + \vec{w}'))_{\Omega} \\ \quad - ((p_h + p'), \nabla \cdot (\vec{w}_h + \vec{w}'))_{\Omega} = \left(\vec{f}, (\vec{w}_h + \vec{w}'))_{\Omega} \\ (\nabla \cdot (\vec{u}_h + \vec{u}'), (q_h + q'))_{\Omega} = 0 \end{array} \right. \end{array} \quad (10)$$

To derive the stabilized formulation, we first solve the fine scale problem, defined on the sum of element interiors and written in terms of the time-dependant large-scale variables. Then we substitute the fine-scale solution back into the coarse problem, thereby *eliminating the explicit appearance of the fine-scale while still modelling their effects*. At this stage, three important remarks have to be made:

- i) when using linear interpolation functions, the second derivatives vanish as well as all terms involving integrals over the element interior boundaries;
- ii) the subscales will not be tracked in time, therefore, quasi-static subscales are considered here;
- iii) the convective velocity of the nonlinear term may be approximated using only the large-scale part;

Consequently, applying integration by parts and then substituting the expressions of both the fine-scale pressure and the fine-scale velocity into the large-scale equations, we get

$$\left\{ \begin{array}{l} \rho (\partial_t \vec{u}_h, \vec{w}_h)_{\Omega} + (\rho \vec{u}_h \cdot \nabla \vec{u}_h, \vec{w}_h)_{\Omega} \\ \quad - \sum_{K \in \Omega_h} (\tau_K \mathcal{R}_M, \rho \vec{u}_h \nabla \vec{w}_h)_K + (2\eta \underline{\dot{\epsilon}}(\vec{u}_h) : \underline{\dot{\epsilon}}(\vec{w}_h))_{\Omega} \\ \quad - (p_h, \nabla \cdot \vec{w}_h)_{\Omega} + \sum_{K \in \Omega_h} (\tau_C \mathcal{R}_C, \nabla \cdot \vec{w}_h)_K \\ \quad = \left(\vec{f}, \vec{w}_h \right)_{\Omega} \quad \forall \vec{w}_h \in V_{h,0} \\ (\nabla \cdot \vec{u}_h, q_h)_{\Omega} - \sum_{K \in \Omega_h} (\tau_K \mathcal{R}_M, \nabla q_h)_K = 0 \quad \forall q_h \in Q_{h,0} \end{array} \right. \quad (11)$$

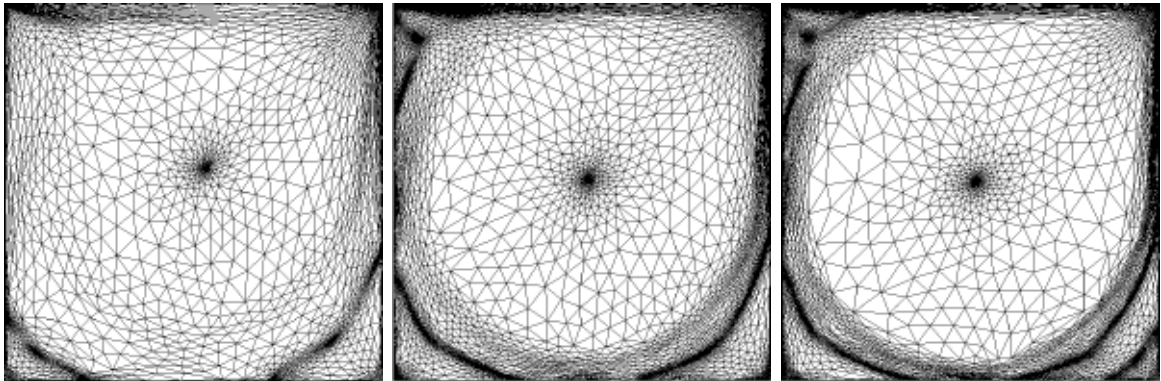


Figure 1: Anisotropic meshes for Reynolds 1000, 5000 and 10,000

When compared with the Galerkin method (10), the proposed stable formulation involves additional integrals that are evaluated element wise. These additional terms, obtained by replacing the approximated \vec{u}' and p' into the large-scale equation, represent the effects of the sub-grid scales and they are introduced in a consistent way to the Galerkin formulation (see [9] for details). All of these terms, controlled by some stabilizing parameters, enable to overcome the instability of the classical formulation arising in convection dominated flows and to satisfy the inf-sup condition for the velocity and pressure interpolations.

5 Numerical examples

The performance of the method will be assessed by benchmarking the driven cavity and by comparing to very accurate reference solutions. In such simulations we show that boundary layers as well as vortices can be well captured at high Reynolds numbers. Results are compared with the literature and show that the flow solvers based on stabilized finite element method is able to exhibit good stability and accuracy properties on such anisotropic meshes.

5.1 Driven flow cavity problem (2-D)

We begin to numerically solve the classical lid-driven flow problem. This test has been widely used as a benchmark for numerical methods and has been analyzed by a number of authors [9, 10, 11]. Dirichlet boundary conditions prescribe on the upper boundary at $y = 1$, and zero elsewhere on Γ . The source term is identical to zero. The viscosity is adjusted in order to obtain Reynolds number of 1,000, 5,000, 10,000, 20,000, and 100,000.

All numerical experiments are done with a fixed number of nodes. As shown in figure 1, we obtained for Reynolds number less than 10,000 a converged meshes (10,000 nodes). All the boundary layers as well as the vortices are well captured and highlighted. As shown in figure 2, the results are in very good agreement with the reference having a 601x601 mesh [10].

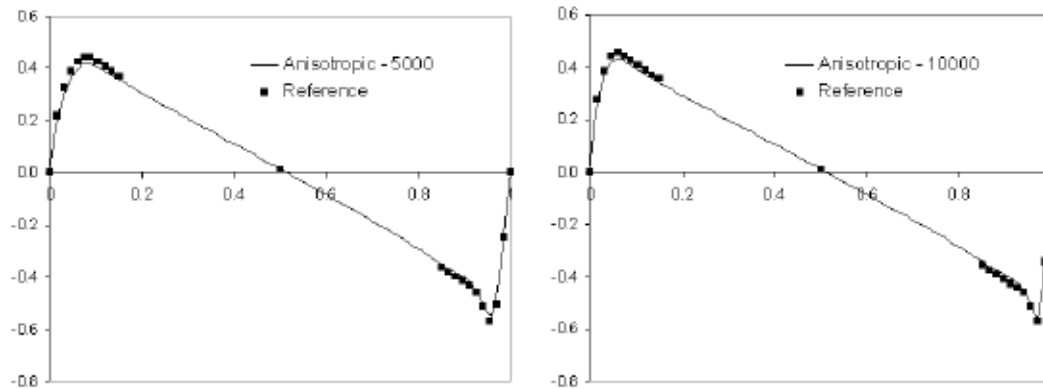


Figure 2: Velocity profiles for u_y along $y = 0.5$

Note that for Reynolds numbers 20,000 and 100,000, the number of nodes is fixed to 20,000 and the solutions are unsteady. We plot in figure 3 only snapshots at a certain time step. Again, the developed incompressible Navier-Stokes VMS solver shows to be very efficient and robust especially when using highly stretched elements.

6 CONCLUSIONS

We proposed in this paper a posteriori estimate based on the length distribution tensor approach and the associated edge based error analysis. It is addressed for incompressible Navier Stokes at High Reynolds number. We show that boundary layer can be produced automatically on an unstructured mesh basis. All the meshes are obtained by solving an optimization problem under the constraint of a fixed number of edges in the mesh. The Variational MultiScale (VMS) method is shown to be favourable for simulating flows at high Reynolds number. The numerical results show that the flow solvers based on stabilized finite element method is able to exhibit good stability and accuracy properties on such anisotropic meshes.

REFERENCES

- [1] T. Coupez. Metric construction by length distribution tensor and edge based error for anisotropic adaptive meshing. *Journal of Computational Physics* (2011) **230**:2391–2405.
- [2] F. Alauset, *Adaptation de maillage anisotrope en trois dimension. Applications aux simulations instationnaires en Mecanique des Fluides*. Ph.d. thesis, Université de Montpellier II, 2003.

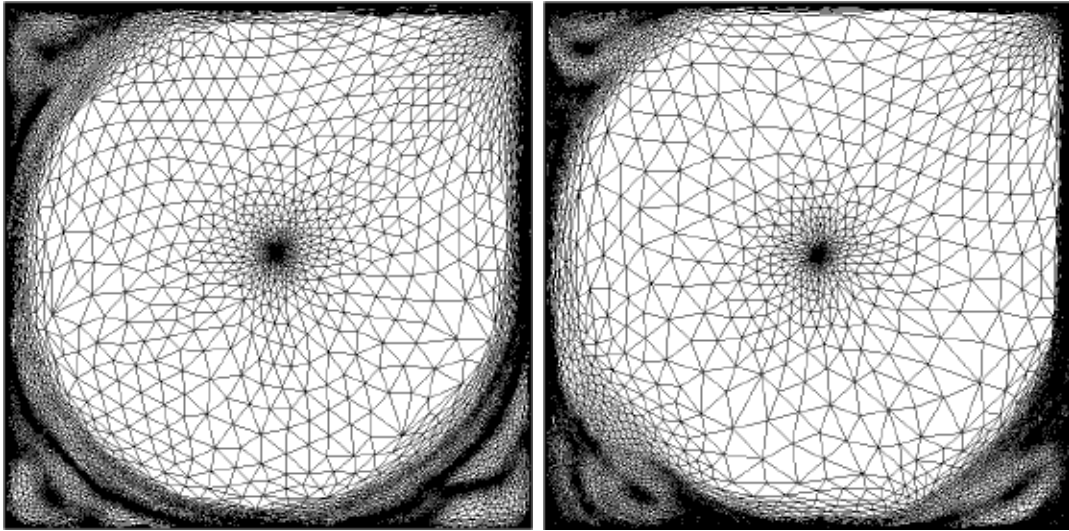


Figure 3: Snapshots of the anisotropic meshes for Reynolds 20,00 and 100,000

- [3] P.J. Frey, F. Alauzet. Anisotropic mesh adaptation for CFD computations. *Computer Methods in Applied Mechanics and Engineering* (2005) **194**:5068–5082.
- [4] C. Gruau, T. Coupez. 3D tetrahedral, unstructured and anisotropic mesh generation with adaptation to natural and multidomain metric. *Computer Methods in Applied Mechanics and Engineering* (2005) **194**:4951–4976.
- [5] T. Coupez. A mesh improvement method for 3D automatic remeshing. *Numerical Grid Generation in Computational Fluid Dynamics and Related Fields*, Pineridge Press (1994) **194**:615–626.
- [6] X. Li, M. Shephard, M. Beall. 3D anisotropic mesh adaptation by mesh modification. *Computer Methods in Applied Mechanics and Engineering* (2005) **194**:4915–4950.
- [7] J.-F. Remacle, X.M. Shephard, J. Flaherty. Anisotropic adaptive simulation of transient flows. *International Journal for Numerical Methods in Engineering* (2005) **62**:899–923.
- [8] L. Formaggia, S. Perotto. Anisotropic error estimates for elliptic problems. *Numerical Mathematics* (2003) **94**:67–92.
- [9] E. Hachem, B. Rivaux, T. Kloczko, H. Dignonnet, T. Coupez. Stabilized finite element method for incompressible flows with high Reynolds number. *Journal of Computational Physics* (2010) **229**:8643–8665.

- [10] U. Ghia, K.N. Ghia, C.T. Shin. High-Re solutions for incompressible flow using the NavierStokes equations and a multigrid method. *Journal of Computational Physics* (1982) **48**:387–411.
- [11] M. Sahin, R.G. Owens. A novel fully-implicit finite volume method applied to the lid-driven cavity problem. Part I: high Reynolds number flow calculations. *International Journal for Numerical Methods in Fluids* (2003) **42**:57–77.