Lire la première partie de la thèse
Part II

Mesh adaptation methods for LES of swirling flows
Chapter 5

Introduction to solution based mesh adaptation

Since Part I demonstrated that one of the most important parameters controlling the quality of LES in a swirler is the mesh, the natural following question is to know how to ensure that the best mesh is used. Of course, using extremely fine meshes everywhere could be a solution [76] but it is not always possible, practical or economical especially for optimization where multiple configurations must be tested. A possible strategy is to use mesh refinement/coarsening to relocate points only where they are needed, keeping constant the overall numerical cost of the simulation (if it is not otherwise desired to further increase the simulation accuracy).

In general, as already mentioned in the introduction of this work (see section 1.2 and which is reported here), a second issue concerning meshing in the LES context arises. At present time there is no standard procedure to generate a LES-suited grid. The mesh, which is the most important LES parameter since it determines the filter size in implicit filtering LES, see section 3.2.2, and influences modeling in a non-trivial way\(^1\), is chosen more or less arbitrarily.

Because of the importance of the numerical grid in LES and because of the lack of standard meshing procedures, the second part of this thesis focuses on mesh adaptation, that is the ability of manipulating a grid based on a set of criteria. This methodology, which has been applied to a very limited number of LES of canonical flows [37, 38, 39], it has never been applied to complex industrial configurations.

This chapter provides: a description of mesh adaptation and the main definitions used, the mesh adaptation methods (here referred as INRIA [25, 3, 2, 63]) and the numerical tool (MMG3D) which performs the so-called metric based adaptation. Then, a review of the error analysis in the LES context and the mutual interaction/compensation of the various error sources (see [31, 71, 53]) are presented.

The presented tools and the error analysis are then re-arranged in Chapter 6 in order to build a set of mesh adaptation criteria suitable for LES. The mesh adaptation strategy and the constraints used to build the adapted mesh are discussed there.

\(^1\)All models used in the LES solvers employed here, rely on the Boussinesq hypothesis, or eddy viscosity hypothesis, which links the kinetic energy dissipation rate to the resolved velocity field strain rate (as for molecular diffusion in a Newtonian fluid, see also section 3.2.2). The resolved flow field is the input for modeling and modeling modifies the resolved flow field in a circle.
Finally, in Chapters 7-9, mesh adaptation is tested on low Reynolds number turbulent jets in order to study the efficiency of the developed sensors and the validity of mesh adaptation strategy. Chapter 7 is dedicated to the Dellenback experiment [17] in the case of a purely axial jet, Chapter 8 is dedicated to the same experiment [17] but in the case of a swirled jet, Chapter 9 is dedicated to the LOTAR experiment shown in the first part of this thesis.

5.1 Mesh adaptation: basic definitions.

The aim of mesh adaptation is to improve "the accuracy of the numerical solution of a given problem, by modifying the domain discretization according to size and directional constraints" [3]. Such constraints are determined by the adaptation criterion used (for instance in [25, 3, 2, 63] the criterion is based on the linear interpolation error). Criteria can be casted in two different families depending on the fact that they rely:

- on the geometrical properties of the mesh elements (for instance a measure of the element stretching such as equivolume skewness), and which therefore requires only the grid to be evaluated, or,

- on the numerical solution of the problem (and therefore on the discretized equations).

The first family of methods relies on the fact that the shape of the elements of the mesh will affect the numerical resolution of the problem. As reported in [69], "as the element’s dihedral angles become too large, the discretization error in the finite element solution increases [45]; as angles become too small the condition of the element stiffness matrix increases [26]". These algebraic constraints need to be satisfied to avoid errors generated by the numerical scheme. Such methods can be named \textit{a priori} since they do not rely on the computed numerical solution but on the mesh only.

The second family of methods aims to adapt the mesh based on the numerical solution, as there is a correlation between the error on the numerical solution and the local mesh size [25], as for instance happens to be the case for LES. Independently of the error estimate chosen (which is clearly dependent on the problem to be solved and which may not always be derived analytically), such methods can be defined \textit{as a posteriori} since they rely on the computed numerical solution.

The techniques employed to manipulate the mesh, relocate nodes, inserting etc., are the same for the two families. They are [37]:

- the $h$ technique, that consists in node insertion or deletion
- the $r$ technique, that consists in relocating grid points
- edge swapping, local reconnection of grid points modifying element faces

Another technique, not discussed here, is the $p$ technique which consists in increasing the order of the basis function (for Finite Element methods, which implies a local increase of accuracy). Many mesh adaptation programs exist, for instance [62]: Yams,
5.2. METRIC-BASED MESH ADAPTATION METHODS

Forge3d, Fun3d, Gamic, MeshAdap, Mmg3d, Mom3d, Tango, and even a direct CAD mesher as CENTAUR has a mesh adaptation module (which however does not allow mesh coarsening).

5.2 Metric-based mesh adaptation methods

As stated in [3], the aim of mesh adaptation is to improve the accuracy of a numerical solution by modifying the grid based on a given criterion. This criterion can be derived from the numerical solution of the problem which is used to generate an estimate of the approximation error (difference between the exact and the numerical solution of the problem).

In the work of Frey and Alauzet [25], Alauzet et al. [3], Dobrzynski and Frey [2], Loseille et al. [63], the approximation error is estimated via the linear interpolation error as for elliptic problems the approximation error is bounded by the linear interpolation error (according to the Cea’s lemma [14]). The interpolation error is the error arising from approximating a continuous solution with a discrete one using a finite number of degrees of freedom. For the variational formulation of a set of partial differential equations (finite volumes or finite elements), the exact and the numerical solution (that is the one to be really interpolated) do not necessarily coincide anywhere in the domain [4]. Because of this limitation, the linear interpolation error in the context of finite elements/volumes can be only estimated as the departure of a quadratic interpolation of the real solution from a linear one [33, 21].

This implies the departure of the solution obtained with a linear basis function from a quadratic basis function [33] (that will give a better or at least equal, representation of the real solution). In a 1D case the point-wise linear interpolation error is the difference between the two curves shown in Fig. 5.1, the parabola and the piecewise linear function used to interpolate it.

In 1D this error is bounded by the second order derivative of the real solution [33, 21] as by interpolating a linear function with linear basis function, the interpolation error is
zero (obvious). In a 2D or 3D case the second order partial derivatives of the solution are casted in the Hessian tensor \( (H) \). The linear interpolation error can be expressed analytically as a function of the element size and orientation with respect to the Hessian matrix (Loseille and Alauzet [62]) as,

\[
|u - \Pi_h u|_{L^1(K)} \leq \frac{K}{40} \sum_{i=1}^{6} \bar{e}_i^T H \bar{e}_i,
\]

(5.1)

(the inequality becomes an equality when \( u \) is elliptic or parabolic) and where:

- \( u \) is a quadratic function (as the interpolation error is estimated using a quadratic interpolation of the function),
- \( \Pi_h u \) is the linear interpolation of \( u \),
- \( L^1(K) \) is the \( L^1 \) norm evaluated on a tetrahedra \( K \),
- \( K \) is the volume of the tetrahedra,
- \( \bar{e}_i \) are the set of edges of the tetrahedron \( K \) taken in a particular order,
- \( H \) is the Hessian matrix of the solution.

The linear interpolation error can be evaluated exactly using Eq.(5.1) or it can be estimated by substituting the Hessian matrix with a "bounding" metric \( (M_b) \), in the sense that

\[
\bar{e}_i^T H \bar{e}_i \leq \bar{e}_i^T M_b \bar{e}_i \quad \forall \bar{e}_i \in \mathbb{R}^3,
\]

(5.2)

where \( H \) denotes the positive symmetric matrix deduced from \( H \) by taking the absolute values of its eigenvalues [62] (i.e. the eigenvalues of \( H \) are equal to the absolute value of the eigenvalues of \( H \), \( \lambda^i_H = |\lambda^i_H| \) with \( i = 1, 2, 3 \)).

The interpolation error of Eq.(5.1) will be bounded by the quantity obtained by replacing \( H \) with \( M_b \) in Eq.(5.1). A metric is a symmetric definite-positive form expressed as a matrix \( M \). Metrics are used to define the dot product in Euclidean metric spaces (therefore the notion of distance \( d_M \) between points):

\[
(\bar{u}, \bar{v})_M = \bar{u}^T M \bar{v},
\]

(5.3)

where \( (\cdot,\cdot)_M \) is the dot product defined for the Euclidean space with a metric \( M \). In the 2D-3D space of classical mechanics, metrics are the identity matrices \( I_2 \) and \( I_3 \).

A metric is easily visualized by transformation of geometric entities (Fig. 5.2). The unit ball in the metric space defined by the identity matrix (which is the locus of points whose distance from the ball center is lower-equal than \( r = \lambda^{1/2} = 1 \), where \( \lambda = 1 \) is the eigenvalue of the identity matrix) is transformed into an ellipsoid with its axis aligned with the principal direction of \( M \) [62]. The metric defines distance \( d_M \) between points and therefore "deforms" the locus of points (such as the unit ball) which requires the notion of distance to be defined.

If a metric is defined continuously, so that, at each point of the space \( M = M(x) \), a **Riemannian space** is defined: the unit ball becomes a different ellipsoid at each location
5.2. METRIC-BASED MESH ADAPTATION METHODS

Figure 5.2: Isolines of $l_M = \text{const.}$ (evaluated via equation 5.4) in $I_2$, left, Euclidean space with a constant metric, center, Riemannian space, right. From Loseille and Alauzet [62]

and the isolines in Fig. 5.2 become complex curves. Since Riemannian spaces are curved, the length of a vector $\overrightarrow{ab}$ in a Riemannian space is calculated using the straight line parametrization $\gamma(t) = \overrightarrow{aa} + t\overrightarrow{ab}$ with $t \in [0, 1]$:

$$l_M(\overrightarrow{ab}) = \int_0^1 \sqrt{\overrightarrow{ab}^T M(a + t\overrightarrow{ab}) \overrightarrow{ab}} \ dt,$$

(5.4)

Once the linear interpolation error is evaluated using the bounding metric $M_b$ of Eq.(5.2) coupled with Eq.(5.1), the mesh can be adapted in order to minimize it\(^3\). The minimization of the overall interpolation error is achieved adapting the mesh in order to get all the tetrahedra which compose the mesh of the same length (Eq.(5.4)).

To summarize, the steps followed by the mesh adaptation method here named as INRIA are:

1. to build a metric ($M$) based on the Hessian of the solution (Eq.(5.2)),

2. to evaluate the length of the edges of each element of the original mesh in the metric space $M$ (Eq.(5.4)),

3. to create a tetrahedra with sides of unit length in the metric space defined by $M$.

\(^2\)Note that in [25, 2, 3, 4] formulations different from Eq.(5.1) are used. Only Eq.(5.1) is reported here since it is the latest developed by Loseille and Alauzet[62].

\(^3\)An exact evaluation of the linear interpolation error is proposed by Loseille and Alauzet[62] and it will be briefly reported here. Because for each tetrahedron $K$ only one metric for which its edges are unit with respect to the metric:

**Definition 3** for $K$ only one $M : l_M(\overrightarrow{e_i}) = 1$ for $i=1..6$ ;

it is possible to define a Riemannian space using the mesh elements [62]. Based on this continuous metric definition is possible to evaluate the linear interpolation error continuously and express it as a function of the Hessian of the solution and of the local metric $M$ of definition 3. So Eq.(5.1) becomes [62]:

$$u - \Pi_h u_{L^2(K)} = \frac{K}{40} \sum_{i=1}^6 \overrightarrow{e_i}^T H \overrightarrow{e_i} = \frac{7}{240} \det M^{-\frac{1}{2}} \text{trace}(M^{-\frac{1}{2}}HM^{-\frac{1}{2}}).$$

(5.5)

where $M = R\lambda R^T$ is the spectral decomposition of the metric and $M^{-\frac{1}{2}} = R\lambda^{-\frac{1}{2}} R^T$. 
Step 1 (evaluation of the linear interpolation error) is peculiar to the INRIA mesh adaptation method [25, 2, 3, 4] while step 2-3 can be used for any mesh adaptation method since they deal with grid manipulation only and not with an error estimate. Fig. 5.3 shows an example of a mesh adapted using the linear interpolation error estimate (step 1) and the metric concept (steps 2-3).

To conclude, since it is impossible to tessellate the 3D space only with the regular tetrahedron, the unit mesh (a mesh of tetrahedra with sides of unit length, see also definition 3), can only be approached by an adaptation program [62].

5.3 MMG3D

MMG3D is a program developed by C. Dobrzynski and P. Frey to adapt the mesh based on a given metric field. It performs the second and the third steps defined at the end of section 5.2. The metric field \(M\) is only an input of the program and it is not evaluated or modified by MMG3D. The re-meshing is performed using an algorithm based on a modified Delaunay criterion by evaluating the length of the elements edges in the given metric space (Eq. 5.4). The Delaunay criterion is clarified by Fig. 5.4: the couple of triangles on the left does not respect the Delaunay criterion since the vertex \(P\) lies inside the circle circumsphere of triangle \(K\) while the couple of triangles on the right does. MMG3D simply tries to generate elements (in 3D) of the type of Fig. 5.4 right, i.e. it generates tetrahedra which respect the modified Delaunay criterion. The only difference with the classic Delaunay approach is that the circle circumsphere (or the circumsphere in 3D) is evaluated in the metric space.
given by the user as input using Eq.(5.4).
Together with the $h$ technique (node insertion/deletion) MMG3D is able to displace the grid nodes ($r$ technique) and to perform edge swapping, i.e. all techniques itemized at the end of section 5.1. A more detailed description of the algorithm used by MMG3D to manipulate the grid is given in reference [18] and it is not discussed here since the program is used as a "black box".

Fig. 5.5 gives an example of a mesh adapted with MMG3D (using a Hessian based metric). From Fig. 5.5 it is evident that the mesh, adapted to capture shocks, is as anisotropic as the Hessian of the pressure field is (pressure changes strongly across the shock). MMG3D can handle only tetrahedral elements in a 3D space, hexahedral and pentahedral elements are decomposed by the program in tetrahedra to be adapted. A 2D version of the code,
MMG2D, is available but is not used here. The number of mesh nodes can be trivially kept constant by deactivating node insertion/deletion. The quality of each tetrahedron $K$ is evaluated as:

$$Q_K = \frac{e_s^3}{|K|},$$  \hspace{1cm} (5.6)

with $e_s = \sqrt[6]{\sum_{i=1}^{6} (l_i)^2}$ and $l_i$ is the length of the edge $i$ evaluated using Eq.(5.4) and $|K|$ is the element volume. The quality index is equal to one for equilateral tetrahedra (in the metric space $M$). The quality index is dependent on the metric imposed: if a constant, isotropic metric is defined everywhere in the domain, the program will try to generate equilateral tetrahedra everywhere. The size of the elements is directly controlled via the metric as it will be explained in chapter 6.

### 5.4 Error estimate in the LES context

As mentioned in section 5.1, the *a posteriori* mesh adaptation strategy relies on a quality measure or error estimate. Such an estimate does not exist in mathematical form for LES and in general for the Navier-Stokes equations. Concerning LES, the most famous effort in such direction (a study of the main error components in LES) was made by Ghosal [31, 94] and his analysis is briefly presented here. The "overall" error in LES cannot be lower than the error arising from the approximation of a continuous solution ($u$) with a discrete one using a finite number of degrees of freedom. This error is called the projection (or interpolation) error and it is intrinsic, since it is derived by using a finite number of degrees of freedom to represent a solution with an higher number of degrees of freedom. This error is minimal when:

$$e \quad P(u) \quad u_d = 0, \hspace{1cm} (5.7)$$

where $P()$ is the best projection operator of the space of continuous solutions to that of the discrete solutions and $u_d$ is the numerical solution. Eq.(5.7) simply states that the numerical solution can only approach the best projection $P()$ of the exact solution $u$. The Navier-Stokes equations can be written in the symbolic form:

$$\partial_t u = N(u)$$  \hspace{1cm} (5.8)

where $\partial_t u$ is the time derivative of $u$ and $N()$ is the nonlinear Navier-Stokes operator. By applying the projection operator $P()$ to Eq.(5.8), we obtain:

$$\partial_t P(u) = P(N(u)).$$  \hspace{1cm} (5.9)

The discrete form of the Navier-Stokes equation can be written as:

$$\partial_t u_d = N_d(u_d).$$  \hspace{1cm} (5.10)
where \( \partial_t u_d \) is the time derivative of \( u_d \) and \( N_d() \) is the discrete Navier-Stokes operator. By subtracting Eq.(5.10) from Eq.(5.9) and adding and subtracting \( N_d(P(u)) \) to the right-hand side, we obtain:

\[
\partial_t (P(u) - u_d) = P(N(u)) - N_d(u_d) + N_d(P(u)) - N_d(P(u)),
\]

(5.11)

\[
\partial_t e = (N_d(P(u)) - N_d(u_d)) = P(N(u)) - N_d(P(u)),
\]

(5.12)

which can be written, using the composition operator " ◦ " which is not distributive, as:

\[
\partial_t e = N_d e = P(N(u)) - N_d(P(u)).
\]

(5.13)

Error \( e \) is zero at the boundary and in the initial condition (since it \( u \) and \( u_d \) satisfy the same initial and boundary conditions), therefore the necessary and sufficient condition for it to remain zero is the right-hand side source term of Eq.(5.13) to be zero. In Ghosal’s analysis, error in LES is therefore defined as this source term:

\[
P(N(u)) - N_d(P(u)),
\]

(5.14)

that is, the extent of the departure of the numerical solution from the best possible on a given grid. Eq.(5.14) includes all the error sources apart from the projection error (or interpolation error which is implicit in Ghosal’s analysis) that are:

- modeling error due to the inaccuracy of the subgrid model,
- discretization error because of the discrete differentiation operator,
- aliasing error due to the computation of non-linear terms.

Note that all SGS models used in this thesis rely on the eddy viscosity assumption which approximates the residual stress tensor as a function of the resolved velocity field (Eq.(3.5)). Turbulent viscosity (\( \nu_t \) in Eq.(3.5)), is usually taken as \( \nu_t > 0 \) to ensure a purely dissipative SGS model avoiding the backscatter of kinetic energy from the smallest to the biggest scales of motion. This assumption is found to be unrealistic [86] but avoids numerical instabilities.

Focusing on Ghosal’s [31] theory, in order to simplify the analysis, the modeling error is erased by using an ”ideal subgrid model” (obtained by imagining to filter instant by instant a DNS solution and giving this output to LES), while the solution (\( u \)) is chosen with an energy spectrum in the form proposed by Von Karman and the quasi-normality hypothesis is employed to evaluate certain non linear terms.

Under these simplifications the main result of Ghosal [31] is that ”the finite-differencing operator error remains larger than the subgrid force over most of the wavenumber range for a second order operator while the situation is improved for higher-order schemes but even for an eighth-order scheme the numerical error is smaller than the subgrid term only about half of the wavenumber range” (Fig. 5.6). The aliasing error has the opposite behavior, being higher for higher-order schemes. Indeed aliasing error was found to be of minor importance and only relevant for very high-order discretization methods [84], so that most of the numerical analysis concentrates on the discretization error [71].
The main consequence of Ghosal’s analysis is that an increase of the resolution (mesh refinement) does not improve the ratio of numerical error over subgrid term as the subgrid scale term drops faster than the numerical error. However, mesh refinement reduces the size of the corrupted eddies so that their effect on the overall flow behavior could vanish or being negligible [66] and this is the path followed in this thesis.

In general, the only option able to modify the ratio of numerical error over SGS force is to increase the SGS contribution. To obtain this for a given mesh resolution, it is required to decrease the ratio grid spacing over filter width:

$$r = \frac{h}{\Delta}. \quad (5.15)$$

Decreasing $r$ means to perform explicit filtering LES. Apart from the analytic and numerical difficulties in performing explicit filtering LES, such a strategy would imply an increase of the numerical cost without adding resolution: the mesh would not be used optimally [66, 12].

### 5.4.1 Verification of Ghosal’s analysis

The theoretical results of Ghosal were verified numerically by Chow and Moin [16] for a HIT, showing that a filter ratio of at least $\frac{1}{4}$ is required (for a second order numerical scheme) to avoid numerical corruption of the results. However, a filter ratio of $\frac{1}{4}$ would require, for the same resolution, a simulation $4^4 = 256$ times more expensive than a simulation with a filter ratio of one (under the CFL condition). Kravchenko and Moin [56] verified the effects of the aliasing error on the turbulent channel flow, showing that ”different (but analytically equivalent) formulations of the nonlinear terms give different results because of finite differences and aliasing errors” [16].

Using explicit filtering LES, Geurts and Fröhlich [30], were able to separate the numerical error contribution from the modeling error. For an HIT simulation, the a posteriori error indicator $\delta_E(\Delta, r)$ proposed in Geurts and Fröhlich [30] is:

$$\delta_E(\Delta, r) = \left| \frac{E_{LES}(\Delta, r) - E_{DNS}(\Delta, r)}{E_{DNS}(\Delta, r)} \right|; \quad (5.16)$$
where \( E \) is the volume averaged kinetic energy for LES \( (E_{LES}) \) and for a filtered DNS \( (E_{DNS}(\Delta, r)) \) evaluated at the same instant, while the norm \( f^2 = \int_{t_0}^{t_1} f(t)^2 dt / (t_1 - t_0) \) represents time averaging. Since in a decaying HIT kinetic energy can only be dissipated, the parameter chosen to be compared with the exact error of Eq.(5.16) is the subgrid activity:

\[
s = \frac{\langle \epsilon_t \rangle}{\langle \epsilon_t \rangle + \langle \epsilon_\mu \rangle};
\]

where \( \langle \rangle \) represents the ensemble average operator and \( \epsilon_t \) and \( \epsilon_\mu \) are the turbulent and molecular dissipation of kinetic energy. If \( s \to 0 \) the simulation is a DNS, if \( s \to 1 \) is a LES with an infinite Reynolds number (a poorly resolved LES). The subgrid activity parameter \( s \) is the ratio of the dissipation related to the modeled turbulence over the total dissipation rate (modeled + viscous).

In Fig. 5.7 the total error of Eq.(5.16) is plotted against the subgrid activity (Eq.(5.17)) for different mesh resolutions. For \( s > 0.5 \) and \( r > \frac{1}{2} \) (so a LES in which the SGS term is of importance, \( s > 0.5 \), and not contaminated by numerical error \( r > \frac{1}{2} \)) all simulations fall on the same curve. In such a dissipation dominated simulation, the subgrid activity is a good estimate of the total error. However a decaying HIT LES has some peculiarities which are hard to find in the flows examined in this thesis and global, "one number", estimates such as Eq.(5.17) are unlikely be accurate.

The discretization error \( (\delta_{E,d}) \) of Fig. 5.7 (b) is evaluated varying the filter size, changing in Eq.(5.16) \( E_{DNS}(\Delta, r) \) with \( E_{LES}(\Delta, r) \) with \( r \to 0 \). The magnitude of the numerical error increases with increasing value of \( r \) and dominates for \( r = 1 \), a further verification of Ghosal’s analysis.
5.4.2 Mutual error compensation

Decaying HIT simulations can be used to build the so-called error landscapes, [71]. By systematically varying the Smagorinsky parameter $C_s$ (the Smagorinsky constant), the spatial resolution $N$ and the Reynolds number, Meyers et al. [71] find the optimal refinement strategy (grid size-model constant) for which the prediction of a given flow property (such as total kinetic energy or enstrophy, see Fig. 5.8) is improved because of the mutual compensation of numerical and modeling error.

![Figure 5.8: Error landscapes, from [71]. Error similar to Eq.(5.16) is plotted against $C_s$ and the number of discretization point. (a) error on the total kinetic energy, (b) error on the total enstrophy $\varepsilon = \langle \omega_i \omega_i / 2 \rangle$ with $\omega_i = \nabla \times u \rangle$.](image)

A similar non-monotone behavior was experienced in the work of Meyers and Sagaut [72]. The classic turbulent channel was simulated in DNS (but with a LES like resolution). The resulting zero error isosurfaces of the skin friction coefficient (Fig. 5.9) and the mean velocity profile correspond. However, the zero error isoline for the skin friction corresponds to a strong error on the Reynolds stresses, showing as the error on velocity fluctuations compensates with the lack of wall resolution therefore obtaining the proper skin friction coefficient.

![Figure 5.9: Turbulent channel of Meyers and Sagaut [72]: contour plot of the skin-friction error as function of the mesh size in the streamwise and spanwise directions ($\Delta^+ x$ and $\Delta^+ z$).](image)
Note that all these studies were performed for academic turbulent flows (typically homogeneous turbulence or channel flows) and that we should not expect them to be directly applicable to complex swirled flows for which these simple and intuitive methods cannot hold because of the flow complexity.
CHAPTER 5. INTRODUCTION TO SOLUTION BASED MESH ADAPTATION

5.5 Mesh adaptation in the LES context

Very few works dealing with mesh adaptation in LES can be found in the literature and none in the context of complex, industrial configurations as the one studied in this thesis. Here are briefly reported the results of Hertel et al. [37], who uses a methodology and criteria similar to the one developed in this thesis, and some of the results of Hoffman [38, 39]. In the work of Hertel et al. [37], a self-adaptive method is presented. The adaptation technique employed is the $r$ (nodes displacement) which keeps the mesh connectivity untouched allowing the mesh adaptation process to be executed in parallel to the LES solver. Mesh movement is included in the LES equations via the ALE (Arbitrary Lagrangian Eulerian) formulation, an approach first proposed by Huang and Russel [40]. Nodes density is determined by the monitor function, $G$, that is a symmetric definite-positive matrix. The objective of such method is to achieve uniformity of nodes distribution with respect to $G$, i.e. to obtain an error distribution homogeneous in space.

This target is achieved by minimizing a functional derived from $G$ and from the nodes coordinates. If $G$ is chosen to be the Hessian, $H$, of the solution, the linear interpolation used by INRIA (see section 5.2) can be recovered [41]. The monitor functions tested in [37] are:

1. the gradient of the mean streamwise velocity,
2. turbulent viscosity ratio, $\frac{<\nu_t>}{<\nu_t>+<\nu>}$,
3. turbulent kinetic energy ratio, $TKE = \frac{<k_{sag}>}{<k_{res}>+<k_{sag}>}$,
4. shear stress ratio, $\frac{<\tau^{mod}_{12}>}{<\tau^{mod}_{12}>+<\tau^{mod}_{11}>}$, with $\tau^{mod}_{12} = \nu_t(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})$.

Adaptation is tested [37] on the flow over a periodic hill (Fig. 5.10) for which highly resolved LES data were available [27].

Figure 5.10: Geometry of Hertel et al. [37] and Fröhlich et al. [27] and flow field cut ($u$=streamwise flow speed).

Results obtained with the adapted meshes (≈135000 nodes), are compared with the highly-resolved LES of 4.7M points, in terms of mean flow velocity profiles (Fig. 5.11).
Not all sensor fields, or criteria for adapting the mesh, were able to improve comparison with the highly resolved LES (Fig. 5.11). The gradient of mean velocity was the most successful in this respect because the structured grid was refined “near the windward side near the hill crest where a strong acceleration takes place” [37], yielding to a wall resolved mesh in the wall normal direction ($\Delta y^+ \approx 6$), and at the reattachment position. Results of [37] show no dependency on the initial grid used.

A totally different approach is studied by Hoffman [38, 39]. Hoffman introduces a linearized dual problem in order to improve the prediction of a given quantity of interest (lift, drag, etc.). Based on that, he evaluates the various error sources [94] (coming from discretization and modeling) obtaining good result in drag or lift prediction in simple configuration as the one shown in Fig. 5.12. However, as shown in section 5.4 where results of Meyers and Sagaut [72] are reported, good results on a single flow quantity do not imply that the whole flow is better resolved because of the so-called mutual error compensation.

### 5.6 Summary

In section 5.1 errors in the LES context have been reviewed. In general all error components act on a LES flow whose solution ($u_d$) will differ from the exact solution of the Navier-
Stokes equations \((u)\) introducing the so-called approximation error:

\[
\varepsilon_a = u - u_d .
\]

By definition \(\varepsilon_a = 0\) in LES (since the number of degrees of freedom of LES is lower than the number of degrees of freedom of the flow). The approximation error is a function of three different components:\(^5\)

\[
\varepsilon_a = f(\varepsilon_m, \varepsilon_n, \varepsilon) ,
\]

where: \(\varepsilon_m\) is the modeling error, \(\varepsilon_n\) is the numerical error (therefore dispersion and diffusion), \(\varepsilon\) is the interpolation (or projection) error (Eq.(5.7)). From the literature it has emerged that:

1. finite differences and finite volumes operators have large numerical errors (larger than the SGS term) concentrated at the highest wave numbers (smallest structures) \([12, 31]\),

2. the combined effects of the three error components of Eq.(5.19) are unknown and could compensate each others \([16, 71, 72]\),

3. mesh refinement reduces the size of the corrupted eddies so that their effect on the overall flow behavior could vanish or being negligible \([66]\) (that is a different way of saying that LES is consistent with the Navier-Stokes equations).

The last property (localized mesh refinement) is used in this thesis to improve the accuracy of LES results (since explicit filtering LES or even higher order numerical schemes are not investigated). A possible refinement strategy is to refine everywhere the mesh (Moureau et al. \([76]\)) but such an approach is evidently expensive. The strategy used here is to use a flow property (such as the velocity gradient as in Hertel et al. \([37]\)) to increase the resolution only where needed. The mesh adaptation method used can therefore be defined as an \(a\ posteriori\) method since the mesh is adapted based on the computed flow. The manipulation of the mesh is performed using the program MMG3D, presented in section 5.3, which requires as input a metric field. The flow property chosen to adapt

\(^5\)Eq.(5.19) puts together all error sources while Ghosal's \([31]\) analysis does not consider interpolation error since it cannot be erased, \(\varepsilon_a = 0\).
the mesh is therefore transformed in a metric field to be read by the mesh manipulator. MMG3D performs the second and third steps shown in section 5.2 which are in common with the mesh adaptation method here used and the "INRIA" approach (Frey and Alauzet [25], Alauzet et al. [3], Dobrzynski and Frey [2], Loseille et al. [63]). While in the INRIA method the metric field derives directly from the Hessian of the solution, here it the metric field is constructed based on flow properties.
Chapter 6

Suitable sensors for LES

This chapter presents the criteria used for the \textit{a posteriori} mesh adaptation method developed during this thesis and that will be validated in the test cases of chapters 7-9. Since \textit{a posteriori} mesh adaptation requires a computed flow field, this chapter shows how to generate a set of criteria/sensors using a LES solution.

Generating an adaptation criterion is the intermediate step between the LES solver and the mesh adapter (Fig. 6.1). The mesh adapter (that is the program that moves, insert or delete nodes, elements, etc.) requires as input a metric field (see section 5.3). Therefore, creating an adaptation criterion consists in generating a metric field (see section 5.2) which can be read by the mesh adapter. Fig. 6.1 sketches the three mesh adaptation steps. The first step of Fig. 6.1 consists in performing a LES. The mesh and the solution of LES are the inputs of the metric generator. The metric generator analyses the flow field and generates as output a metric field depending on which criterion and constraints the user choses.

The mesh and the metric field are given to the mesh adapter which modifies the mesh based on the size and directional constraints prescribed by the metric. The LES solver used in this thesis (AVBP and YALES2) have been described in section 3.2.2 while the mesh adapter (MMG3D) has been described in section 5.3.

This chapter focuses on the metric generator that is the "decision maker" of the whole process: based on its analysis of the LES flow field it prescribes how the adapter has to ma-
nicipulate the grid. The metric generator was developed by the author of this manuscript. It is written in C and it is named ADDA (which stands for nothing). Part of the metric generator tasks consists in translating the mesh from the LES format (in hdf5) to a format which can be read by MMG3D: it basically works as a link between the LES solver and the mesh adapter.

6.1 How to build a metric

We start our description by introducing some basic concepts that will be used in this chapter. A metric is a symmetric definite-positive form expressed as a matrix \( M \). Metrics are used to define the dot product in Euclidean metric spaces (therefore the notion of distance \( d_M \) between points, Eq.(5.3)). A metric \( M \) in \( \mathbb{R}^3 \) can be diagonalized as follows:

\[
M = R \begin{pmatrix} 
\lambda_1 & 0 & 0 \\
0 & \lambda_2 & 0 \\
0 & 0 & \lambda_3 
\end{pmatrix} R^T ,
\]

(6.1)

where \( \lambda_1, \lambda_2, \lambda_3 \) are its three eigenvalues and \( R \) is an orthonormal matrix composed of the eigenvectors of \( M \) which verifies \( RR^T = R^T R = I_3 \) where \( I_3 \) is the identity matrix \[63\]. The three eigenvalues have to be strictly positive (a metric “deforms” the Euclidean space, if one of the eigenvalues is zero, it means that a dimension has disappeared) and symmetric (a metric defines distance; if the metric is not symmetric, distance would depend on orientation \( \overline{AB} \neq \overline{BA} \)).

There are two possible conditions: either the three eigenvalues are equal \( (\lambda_1 = \lambda_2 = \lambda_3) \) therefore the metric is isotropic in all directions or one of the eigenvalues is different from the others \( (\lambda_1 = \lambda_2) \) therefore the metric is anisotropic. An isotropic metric will force the mesh adapter to create isotropic elements while an anisotropic metric will result in anisotropic elements. In this thesis only the first path (isotropic metric) will be investigated since for implicit filtering LES the mesh has to be isotropic for LES to be consistent with the Navier-Stokes equations \[66\]. Also, the use of an isotropic metric reduces the number of parameters of the study performed here.

Since all metrics used here are isotropic in all directions \( (\lambda_1 = \lambda_2 = \lambda_3 = \lambda) \), they can be expressed as:

\[
M = \lambda \ I_3,
\]

(6.2)

simply. The main task of the metric generator of Fig. 6.1 is to generate a metric field, so to determine at each node \( n \) of the mesh a metric \( M(n) \). Generating a metric field corresponds to evaluate the function:

\[
M(n) = \lambda(n) \ I_3 = \begin{pmatrix} 
\lambda(n) & 0 & 0 \\
0 & \lambda(n) & 0 \\
0 & 0 & \lambda(n) 
\end{pmatrix}.
\]

(6.3)

6.1.1 Metric based on a flow property

The simplest metric fields developed are based on a flow property \( \Phi \). The time-averaged value of a flow property \( (\Phi_{mean}) \) is used to generate the metric as follows. First the mean
length \(L(n)\) of the side of the tetrahedra sharing a node \((n)\) is evaluated. Then a factor of scaling \(s(n)\) is evaluated as:

\[
s(n) = \frac{\Phi_{\text{mean}}}{\Phi(n)},
\]

(6.4)

where \(\Phi(n)\) is the flow property \(\Phi\) evaluated at the node \(n\). The metric eigenvalue at the node \(n\) based on the flow property \(\Phi\) (i.e. \(\lambda_{\Phi}(n)\)) is evaluated as:

\[
\lambda_{\Phi}(n) = \frac{1}{(L(n) \times s(n))^2},
\]

(6.5)

and used is Eq.(6.3) instead of \(\lambda\).

The resulting mesh size (\(\Delta\)) of Eq.(6.5) is \(\Delta = L(n) \times s(n)\) which is also the radius of the unit ball in the metric space defined by the metric (see section 5.2).

If \(\Phi(n)\) is higher than the mean (\(s(n) < 1\)), the size of the tetrahedron\(^1\) will be reduced according to Eq.(6.4-6.5), vice-versa the size of the tetrahedron will be increased.

\(\Phi\) can be any of the flow properties: fluctuation of the mean velocity or sub-grid dissipation, ratio of turbulent over laminar viscosity, etc. .

### 6.1.2 Metric based on the velocity gradient tensor

Building a metric using the gradient of the mean\(^2\) velocity (\(u_{\text{mean}}\)) requires a different method to be applied. Since the mean velocity gradient tensor (\(u_{\text{mean}}\)) is not necessarily symmetric or definite positive it cannot be directly used as a metric field and some algebraic manipulations are required. These algebraic manipulations are the same proposed in Nicoud et al. [80] to build the SIGMA SGS model, see section 3.2.2, and a metric field based on the singular values of the velocity gradient is built.

Singular values are defined as the square root of the eigenvalues of the matrix \(A^T A\) where \(A\) is a matrix with real entries and \(A^T\) is its transpose. Singular values are chosen because they are by definition nonnegative [80] and this simplifies their use to create a metric.

Singular values of the gradient of the mean velocity are evaluated as follows. First the matrix:

\[
G = u_{\text{mean}}^T u_{\text{mean}},
\]

(6.6)

is evaluated at each node \(n\) (the dependency on \(n\) is dropped for clarity). \(G\) is by construction symmetric and semi-definite positive. Then the eigenvalues of \(G\) \((G_1, G_2, G_3)\) are used to evaluate the singular values \((\eta_1, \eta_2, \eta_3)\) of \(u_{\text{mean}}\) as \(\eta_i = \sqrt{G_i}\) for \(i = 1, 2, 3\).

Singular values are not necessarily equal to each other, \(\eta_i = \eta_j\), and some of them can be zero. Therefore the mean singular value \(\eta_M = \sum_{i=1}^3 \eta_i / 3\) is used instead to build a metric field \(M_{u}(n)\) as:

\[
M_{u}(n) = \eta_M(n) I_3 = \begin{pmatrix}
\lambda_{u(n)} & 0 & 0 \\
0 & \lambda_{u(n)} & 0 \\
0 & 0 & \lambda_{u(n)}
\end{pmatrix}.
\]

(6.7)

\(^1\)The size of a tetrahedron depends on the scaling factor (Eq.(6.4)) of all nodes belonging to that tetrahedron.

\(^2\)We assume that the mean and the gradient operator commute so \((u_{\text{mean}}) = (u)_{\text{mean}}\).
where $\lambda u(n) = \eta_M(n)$ is by definition the eigenvalue of the metric $M_u(n)$. The peculiarity of the metric field of Eq.(6.7) is that the bigger the velocity gradient is, the smaller the element size is.

### 6.1.3 Metric intersection

Along with a metric field based on a flow property (Eqs. (6.4-6.5)) or based on the velocity gradient (Eq.(6.7)) a third method is used here. This method consists in mixing the previous two using the principle of metric intersection as explained by Alauzet and Frey [4].

For instance, if it is desired to adapt the mesh based on a criterion relying on both a flow property $\Phi$ and on the velocity gradient, two metric fields are evaluated:

- $M_\Phi(n) = \lambda_\Phi(n) I_3$, using Eqs. (6.4-6.5),
- $M_u = \lambda_u(n) I_3$, using Eq.(6.7) and the algorithm explained in section 6.1.2.

The intersection of $M_\Phi(n)$ and $M_u(n)$ is evaluated as:

$$INT(n) = M_\Phi(n) \cap M_u(n) = \lambda_\Phi(n) I_3 \cap \lambda_u(n) I_3.$$

Eq.(6.9) expresses formally that $INT(n)$ is the intersection of two concentric balls of different radii (any isotropic metric can be visualized by a circle in 2D or by a ball in 3D, see section 5.2). The ball of intersection $INT(n)$ obviously corresponds to the smallest ball between $\lambda_\Phi(n) I_3$ and $\lambda_u(n) I_3$, ergo:

$$INT(n) = M_\Phi(n) \cap M_u(n) = max[\lambda_\Phi(n), \lambda_u(n)] I_3,$$

since the square of the radius of the ball is inversely proportional to its characteristic eigenvalue $\lambda$.

### 6.2 Constraints

The mesh needs to satisfy some constraints to be used by a LES solver. Element shape, gradation (variation of the volume of neighbor elements), minimum and maximum cell size, total number of nodes are important properties which can influence the quality, the efficiency and even the possibility of doing a LES of a given flow. In this section are presented all manipulations performed on the metric field in order to match the external constraints additional to the $a posteriori$ adaptation criterion.

#### 6.2.1 Anisotropy

As mentioned in section 5.1, element shape cannot be arbitrary. Tetrahedra with an excessive equivolume skewness (or aspect ratio etc.) can cause numerical difficulties causing the simulation to crash. Equivolume skewness is defined as:

$$E_{Qv} = \frac{V_c}{V_c} \frac{V}{V},$$

(6.10)
where $V_c$ is the volume of an equilateral tetrahedron with the same circumsphere of the current tetrahedron and $V$ is the volume of the current tetrahedron. Fig. 6.2 clarifies Eq.(6.10) for a 2D tetrahedron, i.e. a triangle. Elements which are too stretched in one direction have a large equivolume skewness and cannot be used: a constraint on the element shape is imposed.

![Figure 6.2: Equivolume skewness for 2D tetrahedra (i.e. equisurface skewness for different triangles). Triangle A has a high equisurface element, triangle B has a low equisurface skewness while triangle C, which is the equilateral triangle inscribed in the same circle circumference of A & B, has obviously an equisurface skewness of zero. The same concept is applicable to tetrahedra considering the circumsphere instead of the circle circumference.](Image)

This constraint is trivially satisfied for the isotropic metrics of section 6.1 (since the mesh adapter should generate only isotropic elements if the metric is purely isotropic). However, since it is impossible to tessellate the 3D space only with the regular tetrahedron [62], and because of the difficulties arising by not deteriorating the original CAD surface during the re-meshing procedure, the mesh adapter could generate highly stretched elements. For this reason the a priori optimization based algorithm of Freitag et al. [24] (employing only nodes displacement) was used to improve the quality of the worst elements in terms of equivolume skewness in the case that LES had numerical stability problems.

### 6.2.2 Minimum & maximum cell size

The time step of a LES under the Courant-Friedrichs-Lewy (CFL) condition is determined (the 1D case is used here for simplicity) by:

$$dt = \min\left(\frac{dx}{c}\right) CFL,$$

(6.11)

where $dx$ the grid spacing, $c$ is the wave speed (which is $c = u + c_{\text{sound}}$ for compressible solvers or $c = u$ for incompressible solvers where $u$ is the absolute value of largest between the convective and the diffusive flow speeds and $c_{\text{sound}}$ is the speed of sound) and $CFL$ is the CFL number chosen. If the wave speed is approximately constant everywhere in the domain it is evident that, for a given CFL, the minimum cell size ($dx_{\text{min}}$) determines the simulation time advancement and therefore efficiency.

The maximum cell size ($dx_{\text{max}}$) has to be fixed since any criterion could lead to a metric field which has an eigenvalue close to zero locally (for instance a metric field based on the velocity gradient is zero for a parallel flow).
6.2. Constraints

Based on these constraints ($dx_{max}$ and $dx_{min}$ defined by the user) if any eigenvalue ($\lambda(n)$) of the metric field exceeds the boundaries imposed by $dx_{max}/dx_{min}$, i.e. $\lambda(n) > (1/dx_{min})^2$ or $\lambda(n) < (1/dx_{max})^2$, an artificial clipping is employed:

$$\lambda(n) = \left( \frac{1}{dx_{max}} \right)^2,$$

or,

$$\lambda(n) = \left( \frac{1}{dx_{min}} \right)^2.$$

if it exceeds the minimum/maximum bounds respectively.

### 6.2.3 Gradation

The cell size gradation is the variation of the size of the edges of neighbor elements. Gradation is here computed as the ratio of the maximum eigenvalue between all nodes of a tetrahedron to the minimum eigenvalue between all nodes of the same tetrahedron. Formally, $\lambda_{T_{min}} = \min(\lambda_{i_{min}})$ is the smallest eigenvalue of the tetrahedron, $\lambda_{T_{max}} = \max(\lambda_{i_{max}})$ is the largest eigenvalue (where $i = 1, 2, 3, 4$ are the nodes of the tetrahedron). In case that the ratio

$$g_\lambda = \sqrt{\frac{\lambda_{T_{max}}}{\lambda_{T_{min}}}},$$

(6.14)

is higher then the desired level of gradation ($\beta$) the eigenvalues $\lambda_{max}$ and $\lambda_{min}$ are modified as follows.

First the mean eigenvalue is evaluated as $\lambda_{T_{mean}} = (\lambda_{T_{max}} + \lambda_{T_{min}})/2$. Then $\lambda_{T_{max}}$ and $\lambda_{T_{min}}$ are modified accordingly to the gradation level imposed by the user ($\beta$):

$$\lambda_{T_{max}} = \beta \lambda_{T_{mean}},$$

(6.15)

$$\lambda_{T_{min}} = \frac{\lambda_{T_{mean}}}{\beta}.$$

(6.16)

The result is that their ratio $g_\lambda$ becomes:

$$g_\lambda = \sqrt{\frac{\lambda_{T_{max}}}{\lambda_{T_{min}}} = \beta}.$$

(6.17)

Every non-boundary node belongs to two tetrahedra at least. Therefore, imposing a maximum gradation $\beta$ for a tetrahedron corresponds, even if indirectly, to impose a gradation level between the size of the tetrahedron and its neighbour tetrahedra which share the same nodes.

### 6.2.4 Number of nodes

The cardinality of the grid nodes (the "number of nodes") obviously determines the computational cost of a simulation (in the best case scenario the computational cost increases

---

3The element size of the adapted mesh ($\Delta$) is inverse proportional to $\lambda^2$, $\Delta = \lambda^{-\frac{1}{2}}$.

4The square root is chosen since the mesh size is proportional to $1/\lambda^2$. 

linearly with the number of nodes). The number of nodes of the adapted mesh is determined as a function of the metric field as follows.

Let us consider a metric $M(n)$ evaluated at a point $n$ of the mesh. The volume of the equilateral tetrahedra that is of unit length (i.e. the equilateral tetrahedron whose edges are of unit length in the metric space defined by $M(n)$) with respect to the metric is:

$$V^M(n) = \det(M(n))^{\frac{1}{2}} V^{REF} = (\lambda(n)^3)^{\frac{1}{2}} V^{REF}. \quad (6.18)$$

$V^{REF}$ is the volume of an equilateral tetrahedron of unit length inscribed in a sphere of unit radius:

$$V^{REF} = \left(\frac{2}{3} \frac{\sqrt{6}}{6} \right)^3 \frac{\sqrt{2}}{12} = 0.5132 [m^3]. \quad (6.19)$$

The ratio of the nodal volume $V^{OLD}(n)$ of the original mesh and the volume estimated by Eq.(6.18) is named $\gamma(n)$:

$$\gamma(n) = \frac{V^{OLD}(n)}{V^M(n)}. \quad (6.20)$$

The resulting total number of nodes on the adapted mesh can be approximated as:

$$N^{NEW} \approx N^{OLD} \sum_{n=1}^{N^{OLD}} \gamma(n), \quad (6.21)$$

where $N^{OLD}$ is the number of nodes of the original mesh to be adapted.

Suppose that $N^{NEW}$ is different from the desired number of nodes on the adapted mesh ($N^{TARGET}$), we introduce a scale factor $s$ between these two terms:

$$s = \frac{N^{NEW}}{N^{TARGET}}. \quad (6.22)$$

By multiplying each eigenvalue $\lambda(n)$ of $M(n)$ by a parameter $\alpha$ equal to:

$$\alpha = \left(\frac{N^{NEW}}{N^{TARGET}}\right)^{\frac{2}{3}} = s^{\frac{2}{3}}, \quad (6.23)$$

we obtain that,

$$V^M(n, \alpha) = (\alpha^3 \lambda(n)^3)^{\frac{1}{2}} V^{REF} = \alpha^{\frac{3}{2}} V^M(n) = s V^M(n), \quad (6.24)$$

therefore,

$$\gamma(n, \alpha) = \frac{V^{OLD}}{V^M(n, \alpha)} = s^{\frac{1}{2}} \gamma(n), \quad (6.25)$$

giving,

$$N^{NEW}(\alpha) = \sum_{n=1}^{N^{OLD}} \gamma(n, \alpha) = s \sum_{n=1}^{N^{OLD}} \gamma(n) = s N^{NEW} = N^{TARGET}. \quad (6.26)$$

Is therefore possible to estimate the number of nodes of the adapted mesh a priori (before the mesh adapter generates it). Obviously, any gradation present in the original metric
and any anisotropy will be left untouched by the scaling procedure (\(\alpha\) of Eq. (6.23) is applied to the whole metric field). This method allows to determine the numerical size of the problem without modifying the properties of the original metric. Using the definition of Loseille and Alauzet [62], whose analysis inspired the method used here to impose the number of nodes of the mesh, the original and the ”scaled” metric field (and therefore adapted mesh) are ”embedded”, since they have the same anisotropic ratios (in this case is trivially satisfied since the metric is isotropic) and orientations and they only differ from the number of nodes.

6.3 Simulation efficiency

In the final section of this chapter is introduced the definition of the simulation efficiency that will be used later on.

**Definition 4** The simulation efficiency is defined as the ratio of the simulation physical time \(t_{\text{simu}}\) over the computational time \(t_{\text{CPU}}\), the computational time) using a given solver, machine and number of processors:

\[
\text{Eff} = \frac{t_{\text{simu}}}{t_{\text{CPU}}},
\]

(6.27)

Considering a constant \(t_{\text{simu}}\), for instance \(t_{\text{simu}} = 5\) convective flow through times, in order to reach a converged statistic for a given flow, the simulation efficiency will be inversely proportional to \(t_{\text{CPU}}\): the faster is the computation for a given solver, machine and number of processors, the more efficient is the simulation.
Chapter 7

Test case A: the Dellenback experiment, axial case

7.1 Description of the experiment

Dellenback’s experiment [17] studied a confined jet at moderate Reynolds number and produced a detailed set of measurements. Measurements were taken at different swirl ($S$, Eq.(2.2)) and Reynolds numbers (note that the measurement position differs for the axial and swirled case) and a number of LES studies have been performed for this flow [103, 104]. In this thesis two cases will be examined:

the first case corresponds to the fluid dynamics condition of $Re = 30000$ and $S = 0.0$ and it is named here “axial” case,

the second case corresponds to the condition of $Re = 30000$ and $S = 0.6$ and it is named here “swirled” case (Chapter 8).

As explained in the first part of this thesis (see Chapter 2) the introduction of a swirl velocity component highly modifies the flow topology: the axial and swirled flows examined here have few common features except the geometry. The axial case is chosen for its simplicity which allows to easily understand the effects of mesh adaptation while the swirled case (closer the the topic of this manuscript) is the logical next step before analyzing more complex geometries. This chapter focuses on the axial case while chapter 8 focuses on the swirled case.

The LES solver chosen to test adaptation techniques is YALES2 (see section 3.2.2) since it gives reliable results at low numerical cost for such low Mach number flow ($Ma = 0.025$). In Fig. 7.1 the geometry of the Dellenback case is shown: it consists in two coaxial cylinders with the flow streaming from the smaller (with a diameter of $D_1 = 0.0508[m]$) expanding in the larger one (with a diameter twice larger, $D_2 = 0.1016[m]$). The simulation domain expands along the streamwise direction for $L = 0.7112[m] = 14D_1$, allowing to analyze numerical results down to the measurement position $12D_1$ downstream of the inlet (or $10D_1$ downstream of step expansion).

Three boundary conditions are used:

an inlet where the experimental velocity profile obtained at the first measurement
plane is imposed ($2D_1$ upstream of the step expansion) as the same approach was used for this flow in [103, 104],

an outlet (a so-called convective boundary condition[85]),

solid, adherent and adiabatic walls everywhere else.

The average jet speed over the upstream section\(^1\) is $9.13 \text{[m/s]}$ which corresponds to the experimental Reynolds number chosen (considering a kinematic viscosity $\nu = 1.517 \times 10^{-5}$[m$^2$/s] and $D_1 = 0.0508$[m]). Note that no turbulence is injected at the inlet since the proper amount of synthetic turbulence would depend on the particular mesh resolution (considering the effects of dissipation) even tough the experimental conditions correspond to a fully developed turbulent flow in the upstream pipe. This investigation focuses mainly on the downstream part of the domain where the effects of such an approximation are negligible ($5D_1$ downstream of the inlet turbulence intensity reaches the experimental level) as will be shown in appendix C. Table 7.1 summarizes the B.C.s used.

\begin{table}[h]
\centering
\begin{tabular}{|c|c|c|}
\hline
BC NAME & IMPOSED PROPERTY & TARGET VALUE \\
\hline
inlet & EXP velocity profile & $U_{\text{ref}} = 8.9$[m/s] \\
outlet & pressure & 101300 [Pa] \\
walls & adherence, impermeability, adiabaticity & \\
\hline
\end{tabular}
\caption{Imposed values for boundary conditions sketched in Fig. 7.1.}
\end{table}

The SGS model chosen for all LES of this chapter is \textit{WALE} [79] whose properties near the wall are similar to the SIGMA SGS model (i.e. the turbulent viscosity generated by the model is proportional to the cube of the wall distance, see section 3.2.2). All simulations are initialized with a zero velocity flow.

\footnote{In Dellenback et al.[17] the Reynolds number is evaluated using "the average velocity in the upstream tube". However, as shown in the experimental results[17] for the "axial" case, the mean velocity profiles normalized by the centerline jet speed shows absolutely no dependence on the Reynolds number.}
CHAPTER 7. TEST CASE A

7.2 Axial case, homogeneous meshes

In a first step, the effects of a non-adaptative mesh refinement strategy are compared by using four meshes (H1 to H4). This is done for the axial case which corresponds to a confined, unswirled jet at $Re = 30000$ ($Re = U/D_1/\nu$). Results are compared with experimental data taken at 8 measurement position (Fig. 7.2). Four different meshes are used for the basic LES (basic means without adaptation) ranging from mesh H1 to H4 (where "H" stands for homogeneous and the number for the refinement level). Results obtained with the homogeneous meshes are used as reference together with experimental data. This choice is motivated by the lack of a reliable error estimate which would justify an increase or a reduction of the mesh resolution. Therefore mesh adaptation is used here to re-distribute grid nodes based on a given criterion and comparison will be performed between homogeneous and adapted meshes of the same efficiency level (Eq.(6.27)).

The set of meshes of the basic case are isotropic, homogeneous and with a cell size that smoothly increases from the inlet to the outlet by a factor of 2 (elements close to the outlet are twice as large as elements close to the inlet). Meshes are fully composed of tetrahedra with a maximum equiangle skewness limited to $E_q = 0.9$ and a stretching ratio of 1.3, both imposed as input of the commercial mesh generator used to create them (CENTAUR).

![Figure 7.2: Measurement planes names (above) and axial position with respect to the expansion for the axial case of Dellenback[17]. The white line is zero mean axial velocity of LES H3, the picture is colored by the mean axial velocity just as a reference.](image)

The set of basic LES and the characteristics of the meshes used are summarized in table 7.2 while $H1$ and $H4$ are shown in Fig. 7.3. Note that meshes H1 and H2 are finer at the inlet than inside the domain in order to improve the interpolation of the experimental velocity profile (a mesh size of 1[mm] is imposed at the inlet to get 50 nodes in the diameter). This refinement, which is necessary to match the mean velocity profile downstream at the first measurement plane (plane 2), introduces an additional cost which is zero for the fine meshes H3 and H4 (which are finer than 1[mm] at the inlet). When the inlet refinement is removed the efficiency of the simulation ($E$ defined by Eq.(6.27)) scales roughly as the fourth power of the mesh size$^2$, i.e. $E_{H4} = 0.009$ $E_{H3}(\Delta H_4/\Delta H_3)^4 = 0.013$, see table 7.2. The inlet refinement causes a decrease of the efficiency, for instance LES H2 is just 1.48 times more efficient then LES H3 while it should be 2.3.

$^2$Considering that the numerical cost scale as $\Delta^{-3}$ and cost of time advancement as $\Delta^{-1}$.
Table 7.2: table of simulations performed. The numerical efficiency (Eq.(6.27)) is normalized by the efficiency of LES H1 while the average mesh size \( \Delta \) is adimensionalized by the Kolmogorov length scale for a HIT at the same Reynolds number \( \eta_K = Re^{-\frac{3}{4}} D_1 = 2.24 \times 10^{-5}[m] \)

<table>
<thead>
<tr>
<th>mesh name</th>
<th>number of nodes</th>
<th>number of tetra</th>
<th>numerical efficiency Eq.(6.27)</th>
<th>( \Delta/\eta_K )</th>
</tr>
</thead>
<tbody>
<tr>
<td>H1</td>
<td>0.3M</td>
<td>1.6M</td>
<td>1</td>
<td>130</td>
</tr>
<tr>
<td>H2</td>
<td>1.3M</td>
<td>7.5M</td>
<td>0.31</td>
<td>82</td>
</tr>
<tr>
<td>H3</td>
<td>2.3M</td>
<td>13.4M</td>
<td>0.21</td>
<td>68</td>
</tr>
<tr>
<td>H4</td>
<td>25M</td>
<td>111M</td>
<td>0.009</td>
<td>34</td>
</tr>
</tbody>
</table>

Figure 7.3: Meshes H1 and H4. H1 has \( \approx 26-52 \) nodes for diameters \( D_1-D_2 \) respectively, while H4 has \( \approx 84-168 \) nodes for diameters \( D_1-D_2 \) respectively (from the figure it is almost impossible to distinguish elements of mesh H4 which are 4 times smaller than the one of H1).

7.2.1 Flow field of the homogeneous meshes LES

First, the accuracy of LES is tested by increasing the mesh density everywhere as shown in table 7.2. The flow field changes significantly with this homogeneous refinement. When increasing the mesh resolution the turbulent structures generated in the shear layer of the jet get smaller and stronger, a larger amount of the flow turbulence is directly resolved (as evidenced by the snapshot of the Q criterion in Fig. 7.4). The Q criterion [44] is:

\[
Q = \frac{1}{2}(\Omega_{ij}\Omega_{ij} - S_{ij}S_{ij}),
\]

where \( \Omega_{ij} \) and \( S_{ij} \) are the anti-symmetric and the symmetric part of the velocity gradient tensor. Where \( Q \) is positive (rotation dominates strain) a vortex core is present. The turbulent structures generated in the shear layer move toward the centerline of the flow while they are convected downstream by the mean flow as evident in Fig. 7.4.

Mesh resolution modifies these dynamics. With a low resolution these structures reach the centerline sooner (Fig. 7.4) causing an increase in the turbulence intensity (\( TI = u/U_{ref} \) where \( u \) is the root mean square of the velocity considered) in the axial (streamwise) direction along the centerline of the flow (Fig. 7.5). With a higher mesh resolution these structures reach the centerline later (Fig. 7.4) causing the peak of turbulence intensity to be shifted downstream (see H3 in Fig. 7.5). The distribution of axial
and radial turbulence intensities ($TI = \frac{u}{u_{\text{ref}}} = \frac{u}{8.92\, \text{m/s}}$) shown in Fig. 7.5 shows that turbulence along the flow centerline becomes more isotropic as the flow develops.

The mean axial velocity fields of Fig. 7.7 show that the jet reattachment position (at the end of the potential core of the jet where the two shear layer merges) moves downstream from H1 to H3 and that the jet penetration increases with the mesh resolution (compare the zero and the top speed isolines in Fig. 7.7).

The rms fields of Fig. 7.7 show that the shear layer instability grows faster in LES H1 and H2 than in H3 and H4 (compare the isolines of the rms of the axial velocity in Fig. 7.7). Comparison of the mean axial velocity with experimental measurements is good for all LES (Fig. 7.8) up to plane 6, then the prediction of LES H1 and H2 rapidly deteriorate while H3 and H4 provide a good agreement with experimental data. Note that H4 results has to be taken with care since statistics are collected for a shorter time.

Comparison of the rms of the axial velocity is less accurate in all planes. One observation arises from the rms velocity profiles. Turbulence intensity at plane 4 is higher (and it compares better with experimental data) for the low resolution LES than for the higher resolution (Fig. 7.8(b)) as the shear layer instability grows faster. The dependency of the instability growth rate on the mesh resolution can be explained as an effect of the numerical noise generated close to the wall in LES H1 (see Fig. 7.9). This numerical noise (which remains undamped because the SGS model produces no turbulent viscosity at the wall and laminar viscosity is not sufficient to damp it) triggers the instability upstream and helps it to grow faster. Turbulence affects the mean velocity profile with H1 giving the best prediction at plane 4 (the only exception between all measurement positions).

Appendix C shows results of LES H3 with turbulence injection at the inlet of the domain. Injected turbulence ameliorates prediction in this case, labelled H3\textsubscript{TJ}, at plane 4 without being negatively affecting the velocity profiles downstream, obtaining therefore a good comparison with experimental data in the whole domain.

The transition from a laminar (as no turbulence is injected at the inlet) to a fully turbulent jet ends approximately at plane 5 (Fig. 7.9c) where a strong vortex pairing is made evident by the peak of turbulence intensity at a radial position corresponding to the upstream radius.

Finally, Fig. 7.6 shows the jet kinetic energy: all simulations reach convergence after 0.7[s] except LES H4 where the solution is initialized from a mesh of the same resolution as H3 and then refined isotropically. A further transient period is required for LES H4 to start oscillating around a mean value that is higher than LES H1-H2-H3 (as obvious since more flow structures are explicitly resolved in H4). As evident from Fig. 7.6, H4 statistics are collected for a shorter time.
Figure 7.4: Snapshots of Q criterion for simulations of table 7.2.
CHAPTER 7. TEST CASE A

Figure 7.5: Distribution along the centerline of axial and radial (along the centerline radial and tangential TI overlaps) turbulence intensity \(TI = u'/U_{ref}\) for LES H1 and H4 of table 7.2.

Figure 7.6: Time evolution of the L2 norm of U over the whole computational domain for LES of table 7.2. Note that H4 results has to be taken with care since statistics are collected for a shorter time.
7.2. AXIAL CASE, HOMOGENEOUS MESHES

Figure 7.7: Mean flow & RMS for LES of table 7.2
(a) Mean axial velocity profile at plane 2

(b) Mean axial velocity profile at plane 4

(c) Mean axial velocity profile at plane 5

(d) Mean axial velocity profile at plane 6

(e) Mean axial velocity profile at plane 7

(f) Mean axial velocity profile at plane 8

(g) Mean axial velocity profile at plane 9

(h) Mean axial velocity profile at plane 10

Figure 7.8: Mean axial velocity profiles at the measurement planes of Fig. 7.2 for simulations of table 7.2.
7.2. AXIAL CASE, HOMOGENEOUS MESHES

Figure 7.9: TI axial profiles ($TI = \frac{u_{RMS}}{u_{ref}} = \frac{u_{RMS}}{8.92 \text{ m/s}}$) at the measurement planes of Fig. 7.2 for simulations of table 7.2.
7.2.2 Pressure drop and LES quality estimates

The prediction of pressure drop, i.e. the difference between total pressure measured at the INLET and total pressure measured at the OUTLET of the domain\(^3\), does not change significantly with the mesh resolution, see table 7.3. Note that pressure drop was not measured experimentally.

<table>
<thead>
<tr>
<th>mesh/LES name</th>
<th>Total pressure drop $y_{mean}$</th>
<th>$\frac{f_V \langle \frac{\nu}{\nu} \rangle dV}{\nu dV}$</th>
<th>$\langle max(\frac{\nu}{\nu}) \rangle$</th>
<th>Pope criterion $\text{Eq.}(4.1)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>H1</td>
<td>40.8 [Pa]</td>
<td>10.6</td>
<td>6.3</td>
<td>149.0</td>
</tr>
<tr>
<td>H2</td>
<td>39.6 [Pa]</td>
<td>8.1</td>
<td>3.0</td>
<td>87.1</td>
</tr>
<tr>
<td>H3</td>
<td>39.1 [Pa]</td>
<td>7.1</td>
<td>2.3</td>
<td>69.6</td>
</tr>
<tr>
<td>H4</td>
<td>38.8 [Pa]</td>
<td>4.4</td>
<td>0.82</td>
<td>33.3</td>
</tr>
</tbody>
</table>

Table 7.3: Pressure drop, surface averaged $y_{mean}$ and time-volume average of turbulent viscosity (normalized by laminar viscosity), time average of the max value of turbulent viscosity (normalized by laminar viscosity), time-volume average Pope criterion.

The comparison of Fig. 7.10 (which shows the variation of total pressure against the axial position) and Fig. 7.5 shows that the jump in total pressure corresponds to the position where turbulence intensity is the highest, i.e. where the turbulent structures generated in the shear layer reach the centerline of the flow, at the end of the potential core of the jet.

\(^3\)Note that $\Delta P/P$, as usually pressure drop is presented, is in the order of 0.04% ($\Delta P/P = \frac{40}{101300} = 4 \times 10^{-4}$), therefore it is preferred to present results in terms of $\Delta P$ simply.
7.2. AXIAL CASE, HOMOGENEOUS MESHES

Figure 7.11: Time evolution of the maximum ratio of laminar over turbulent viscosity.

The characteristic time scale $\tau$ of an eddy of size $l$ in the inertial subrange is:

$$\tau = \left( \frac{l^2}{\epsilon} \right)^{\frac{1}{3}} \left( \frac{l_0^2}{\epsilon_0} \right)^{\frac{1}{3}}, \quad (7.2)$$

where $l$ is the characteristic length scale of the eddy, $\epsilon$ is the dissipation rate of kinetic energy at a length scale $l$, $\epsilon_0$ is the dissipation rate of kinetic energy imposed by the largest eddies of size $l_0$ and characteristic velocity $u_0$.

$$\epsilon_0 = \frac{u_0^3}{l_0}. \quad (7.3)$$

Using Eq.(7.2) the velocity gradient can be approximated as:

$$u = \frac{u}{l} = \frac{1}{\tau} \left( \frac{2}{3} \frac{l}{\epsilon_0} \right)^{\frac{1}{3}}, \quad (7.4)$$

Using Eq.(3.6), turbulent viscosity can be expressed (assuming that the differential operator $D_m(\tilde{u})$ of Eq.(3.6) and the velocity gradient are in a linear proportion, as for the Smagorinsky model) as:

$$\nu_{SGS} \propto \Delta^2 u. \quad (7.5)$$

Coupling Eq.(7.6) and Eq.(7.4) with $l = \Delta$ we obtain:

$$\nu_{SGS} \propto \Delta^2 u \propto \Delta^{\frac{4}{3}} \epsilon_0^{\frac{1}{3}}. \quad (7.6)$$

Let us consider two different HIT at a different mesh resolution. Using Eq.(7.6) and considering a large amount of samplings, turbulent viscosity in the two HIT should be in a proportion:

$$\frac{\nu_{SGS}^a}{\nu_{SGS}^b} \left( \frac{\Delta^0}{\Delta^b} \right)^{\frac{4}{3}} = r^{\frac{4}{3}} \Delta. \quad (7.7)$$
This simple scaling factor \( r^{4/3} \Delta \) is valid for a filtered DNS. Results of table 7.3 recast in Fig. (7.12) show that Eq. (7.7) approximates very well the trend of turbulent viscosity plotted against the average cell size (where \( \nu_t \) is averaged in time and space) even though the assumptions are not entirely satisfied.

Finally, Fig. 7.13 shows the Pope criterion (Eq. (4.1)) for simulations H1 and H4 using time-averaged turbulent viscosity. From Fig. 7.13 it appears that not even the 111 million of elements mesh H4 (with an average mesh size just 34 times larger than the Kolmogorov length scale for an HIT at the same Re) is able to reach the minimum, arbitrary, requirement of 0.8 set by Pope [89] (see also table 7.3) everywhere.
7.3 Axial case, adapted meshes

The previous section has shown what the effects of a homogeneous mesh refinement method (from H1 to H4) could be. We begin now to see how an adaptive mesh strategy can improve the results and which sensor should be used for refinement. In this section mesh adaptation is tested using three different sensors:

- the first sensor is named "GRAD" and is based on the mean velocity gradient (see section 6.1.2). Using the "GRAD" sensor the local mesh size is inversely proportional to the magnitude of the velocity gradient;

- the second sensor is named "RMS" and is based on a flow property \( \Phi \) (Eq.(6.3-6.4-6.5)) where \( \Phi = RMS = (u_{ji}^2)^{1/2} \); grid nodes are equi-distributed with respect to the intensity of the resolved turbulence and this sensor is chosen because the jet dynamics are driven by turbulence as explained in section 7.2.1;

- the third sensor is named "MIX" and it combines the previous two using the concept of metric intersection (see section 6.1.3).

All metrics are isotropic (see Chapter 6) and the mesh size gradation, set to 1.3, is based on experience as this value is commonly used at CERFACS for LES, even though a lower value would have been more appropriate for the simulation of jets, as suggested by Moreau [75].

Meshes adapted using the solution of LES H2 of table 7.2 are named A2 ("A" stands for adapted and "2" since sensors are based on the flow field of LES H2) and they have approximately the same numerical efficiency (Eq.(6.27)) as LES H2. Mesh \( A_{1MIX} \) is obtained based on mesh and solution of LES H1 of table 7.2 instead. This test is set up to evaluate the effects of the flow field used to build the sensor. In order to compare different sensors/meshes, mesh \( A_{1MIX} \) is built to have the efficiency (Eq.(6.27)) of the other adapted grids (this target is obtained by keeping constant the number of nodes and the minimum cell size in the domain). The characteristics of the meshes are summarized in table 7.4 while grids are shown in Fig. 7.14.

<table>
<thead>
<tr>
<th>mesh/LES name</th>
<th>number of nodes</th>
<th>number of tetra</th>
<th>numerical efficiency</th>
<th>( \Delta / \eta_K )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( A_{2GRAD} )</td>
<td>1.26M</td>
<td>7.2M</td>
<td>0.33</td>
<td>68.75</td>
</tr>
<tr>
<td>( A_{2RMS} )</td>
<td>1.3M</td>
<td>7.48M</td>
<td>0.23</td>
<td>71.42</td>
</tr>
<tr>
<td>( A_{2MIX} )</td>
<td>1.18M</td>
<td>6.78M</td>
<td>0.30</td>
<td>77</td>
</tr>
<tr>
<td>( A_{1MIX} )</td>
<td>1.17M</td>
<td>6.9M</td>
<td>0.34</td>
<td>76.5</td>
</tr>
</tbody>
</table>

Table 7.4: Table of simulations.

Mesh \( A_{2GRAD} \) is refined at the location where the velocity gradient is higher, i.e. at the solid boundaries and in the shear layer (Fig. 7.14). Mesh \( A_{2RMS} \) is refined downstream of the step expansion, along the centerline of the flow while it is coarsened everywhere else (Fig. 7.14). Refinement/coarsening zones correspond to the locations where resolved turbulence is higher/lower respectively (see Fig. 7.7-7.9). Meshes \( A_{2MIX} \) and \( A_{1MIX} \) are a compromise solution between \( A_{2GRAD} \) and \( A_{2RMS} \). Meshes \( A_{2MIX} \) and \( A_{1MIX} \)
are refined both in the shear layer and along the centerline of the flow (Fig. 7.14). Note that $A_{2MIX}$ differs from $A_{1MIX}$ because of the different solution used to generate the metric field ($H2$ vs. $H1$). Obviously, since the number of nodes is kept constant, a mesh refinement somewhere implies a coarsening somewhere else (as shown by comparing Fig. 7.3 and Fig. 7.14).

Boundary nodes are not modified by adaptation in order to avoid deteriorating the ability of the mesh to match the CAD geometry. Note that this choice of not modifying the boundary nodes on the CAD surface is a disadvantage in certain cases, for example if the initial mesh is coarse, this would limit the capacities of adaptation to improve its quality. At the same time, this is also an advantage because it allows to use the mesh generator only once and never have to come back to the CAD data in the subsequent re-meshing phases. However, if the initial mesh is sufficiently resolved at the walls, it is a reasonable method.
Figure 7.14: Meshes of table 7.4
Flow fields of LES obtained using the meshes of table 7.4 are shown in Fig. 7.15. LES $A_{2RMS}$ is characterized by a very short jet penetration and by a high turbulence intensity a few diameters downstream of the jet expansion. On the other hand, LES $A_{2GRAD}$, $A_{2MIX}$ and $A_{1MIX}$ show a higher jet penetration and lower turbulence levels. The flows of LES $A_{2MIX}$ and $A_{1MIX}$ are barely distinguishable in Fig. 7.15.

Not all sensors improve LES prediction and only the "mixed" sensors ($A_{1MIX}$ and $A_{2MIX}$) show a good comparison with experimental data, a prediction similar to LES H3 but with an efficiency which is 50% higher. Sensors $A_{2GRAD}$ and $A_{2RMS}$ alone are not able to improve results. $A_{2GRAD}$ is too coarse downstream where the turbulent structures generated in the shear layer reach the centerline of the flow: a too coarse mesh at this location causes an underestimation of the jet top speed because of an excessive dissipation due to modeling ($\nu_t \Delta^4$). $A_{2RMS}$ is too coarse before the expansion (Fig. 7.14), causing

![Flow Fields of LES](image)
the LES prediction to deteriorate rapidly. Note that the low resolution upstream of the expansion favors the growth of the shear layer instability (see Fig. 7.9) so that LES $A2_{RMS}$ compares better with experimental data at plane 4 for both mean and turbulence intensity (a phenomenon already experienced in section 7.2.1 for LES H1). As evident from Fig. 7.14, meshes $A1_{MIX}$ and $A2_{MIX}$ slightly differ, but they give similar results (Fig. 7.16-7.17). Such a phenomenon can be explained simply by the fact that none of the two meshes is optimal but both are close to it.

### 7.3.1 Pressure drop and LES quality

Improving the prediction of pressure losses in LES remains a significant issue. This section discusses the effects of mesh adaptation on pressure losses predictions.

Table 7.5 shows the characteristics of the flow field of the LES obtained with the adapted meshes. The comparison of table 7.5 and table 7.3 shows that in none of the adapted meshes neither the volume-time average value of the turbulent viscosity neither its maximum value diminish (see also Fig. 7.18(a)) with respect to LES of section 7.2.1. However, pressure drop$^4$ does not diminish significantly in all LES compared to the homogeneous case. Pressure drop remains almost constant even though the zone where the adapted meshes are finer corresponds to regions where the mean SGS dissipation$^5$, $\epsilon = \langle \tau_{ij}S_{ij} \rangle$, is higher (Fig. 7.19(a)).

The mean value of $y^+$ slightly improves in all LES, but an increase of the wall resolution seems to have little influence on pressure drop for this flow (as shown in table 7.3, LES H4 predicts a pressure drop 3.5% lower than H1 despite a 30% reduction of the mean $y^+$ in the domain).

<table>
<thead>
<tr>
<th>mesh/LES name</th>
<th>Total pressure drop</th>
<th>$y^+_\text{mean}$</th>
<th>$f_v \frac{\langle u_+^2 \rangle}{f_v} dV$</th>
<th>$&lt; \max(\frac{\nu_t}{\nu}) &gt;$</th>
<th>Pope criterion Eq.(4.1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>H2</td>
<td>39.6 [Pa]</td>
<td>8.1</td>
<td>3.0</td>
<td>87.1</td>
<td>0.63</td>
</tr>
<tr>
<td>$A2_{GRAD}$</td>
<td>39.1 [Pa]</td>
<td>6.9</td>
<td>6.0</td>
<td>133</td>
<td>0.56</td>
</tr>
<tr>
<td>$A2_{RMS}$</td>
<td>38.5 [Pa]</td>
<td>10.6</td>
<td>3.7</td>
<td>110</td>
<td>0.59</td>
</tr>
<tr>
<td>$A2_{MIX}$</td>
<td>39.1 [Pa]</td>
<td>7.2</td>
<td>4.3</td>
<td>89</td>
<td>0.57</td>
</tr>
<tr>
<td>$A1_{MIX}$</td>
<td>38.9 [Pa]</td>
<td>6.9</td>
<td>4.2</td>
<td>101</td>
<td>0.58</td>
</tr>
<tr>
<td>H4</td>
<td>38.8 [Pa]</td>
<td>4.4</td>
<td>0.82</td>
<td>33.3</td>
<td>0.79</td>
</tr>
</tbody>
</table>

Table 7.5: Total pressure ($P_{TOT} = P + \frac{1}{2}u_+^2$) drop, surface averaged $y_+$, time-volume averaged turbulent viscosity (normalized by laminar viscosity), time averaged of the max turbulent viscosity (normalized by laminar viscosity), time-volume averaged Pope criterion.

The volume averaged Pope criterion of all adapted LES of table 7.5 gets worse compared to the homogeneous meshes. For instance LES $A2_{MIX}$ is able to give a comparison against experimental data as good as LES H3 but its Pope criterion is 13% lower. The low values of the Pope criterion (compare Figs. 7.13-7.19(b)) are consistent with the high values of turbulent viscosity in the domain.

$^4$Note that $\Delta P/P$, as usually pressure drop is presented, is in the order of 0.04% ($\Delta P/P = 40/101300 = 4 \times 10^{-4}$), therefore it is preferred to present results in terms of $\Delta P$ simply.

$^5$Note that $\epsilon$ is evaluated in real time and then averaged.
CHAPTER 7. TEST CASE A

Figure 7.16: Mean axial velocity profiles at the measurement planes of Fig. 7.2 for simulations of table 7.4.
Figure 7.17: TI axial velocity \((TI = \frac{u_{RMS}}{u_{ref}} = \frac{u_{RMS}}{8.92 \text{ m/s}})\) at the measurement planes of Fig. 7.2 for simulations of table 7.4.
CHAPTER 7. TEST CASE A

Figure 7.18: (a) Time evolution of the maximum ratio of laminar over turbulent viscosity (lines are regression curves). (b) Total pressure drop against streamwise position for LES of table 7.4.

Figure 7.19: Flow properties of LES A2\textsubscript{MIX} of table 7.4.
7.3.2 Why adaptation works

The comparison of Figs. 7.8-7.16 reveals that solutions on meshes adapted using the "MIX" sensor show a better match with experimental data than results obtained on an homogeneous mesh such as LES H2 (results are directly compared in Fig. 7.20). This improvement is obtained despite the fact that none of the common estimates used to evaluate LES quality improves after adaptation (such as the Pope criterion [89] or the ratio of turbulent over laminar viscosity, compare tables 7.3-7.5).

This phenomenon can be explained as follows. Mesh A2_{MIX} (as well as A1_{MIX} which gives similar results) are refined in the shear layer and, downstream, along the centerline of the flow (Fig. 7.14). At the same locations turbulent activity is more intense (Fig. 7.15-7.17). The zones where the grid is finer overlap with the zones where most of the eddies are located (Fig. 7.21). Most of the mesh resolution of meshes A2_{MIX} and A1_{MIX} is therefore targeted to resolve turbulence. Fig. 7.22 shows how different the resolved turbulence fields of LES A2_{MIX} and A1_{MIX} are with respect to H2. Fig. 7.22 shows a volume averaged Probability Density Function (PDF) of the Q criterion for some of the LES analyzed in this chapter. This PDF is obtained as follows. The interval \([0, Q_{MAX}]\), \(Q_{MAX} = 3 \times 10^6\) is divided in 100 sub-intervals \(Q(1), Q(2), ..., Q(100)\). For each sub-interval "j" the number of nodes having a \(Q \in [Q(j), Q(j+1)]\) is evaluated and weighted by the nodal volume. The weighting factor is motivated by the fact that homogeneous and non-homogeneous meshes are compared. Formally:

\[
PDF_j(\%) = \sum_{i \in [Q(j), Q(j+1)]} \left( \frac{V_i}{V_{TOT}} \right) \times 100, \quad (7.8)
\]

where \(PDF_j(\%)\) is the PDF of Q to be in the interval \([Q(j), Q(j+1)]\), \(V_i\) is the volume at the node "i" while,

\[
V_{TOT} = \sum_{i \in [0, Q_{MAX}]} V_i, \quad (7.9)
\]

\(^6\)Note that the Pope criterion improves in the refined regions.
Figure 7.21: Mesh H2 and A2 and Q criterion (Eq.(7.1)) of the correspondent LES. In the bottom figure, the line represents the isovolume ($V = 0.4 [mm^3] - 0.73 [mm^3]$) and it envelops the zone where the grid is finer.

is the volume of all the nodes with a Q criterion in the range $[0, Q_{MAX}]$ (the normalization condition). $PDF_j(\%)$ can be transformed easily in the portion of the whole domain volume occupied by eddies with an “intensity” $Q = (Q(j) + Q(j + 1))/2$ by substituting
V_{TOT} with the total volume of the domain in Eq.(7.8).
As evident from Fig. 7.22 the PDF (Eq.(7.8)) of LES A2_{MIX} and A1_{MIX} is the same of LES H3: turbulence is resolved similarly on these meshes and better than on meshes H2 or H1 and this is the reason why results improve after adaptation. This flow is dominated by the dynamics of the turbulent structures generated in the shear layer, convected downstream by the mean flow and diffused by their own chaotic behavior. Mesh resolution is increased (for meshes A2_{MIX} and A1_{MIX}) exactly along the primary path of main vortical activity, reducing errors due to modeling and to numerics.

Figure 7.22: PDF(%) (Eq.(7.8)) of the Q criterion (Eq.(7.1)) for LES H1,H2,H3,H4 and A1_{MIX} and A2_{MIX}.
Chapter 8

Test case B: the Dellenback experiment, swirled case

This chapter deals with the second adaptation test case. While in chapter 7 mesh adaptation was tested on the LES of the "axial" case of the Dellenback experiment [17] (a confined jet at high Reynolds number), here it is tested on the "swirled" case of the same experiment. As already mentioned in chapter 7 and as explained in the first part of this thesis (see Chapter 2), the introduction of a swirl velocity component strongly modifies the flow topology and the flow in the "axial" and "swirled" cases have little in common. The introduction of a swirl velocity component triggers a set of flow instabilities, such as the vortex breakdown or the PVC (described in detail in section 2.2.2) which are not present in the "axial" case of Chapter 7. The swirled case here tested corresponds to the condition of \( Re = 30000 \) and \( S = 0.6 \).

8.1 Numerical settings

The LES solver used here is YALES2. Boundary conditions used are those of chapter 7. Three B.C.s are used: an inlet (2\( D_1 \) upstream of the step expansion), an outlet at the end of the domain, and solid, adherent and adiabatic walls everywhere else.

Axial and tangential experimental velocity profiles are imposed at the inlet in a different manner. Both velocity profiles reproduce the experimental (normalized) curve at the first measurement plane multiplied by a characteristic speed\(^1\). The characteristic speed differs for the axial and tangential velocities. For the axial speed, it is \( U_{ref}^{ax} = 10\text{[m/s]} \) which corresponds to a Reynolds number\(^2\) of 30000. For the tangential speed, the characteristic velocity is \( U_{ref}^{tan} = 8.2\text{[m/s]} = U_{ref}^{ax} \). This choice is motivated by the fact that, by choosing the same speed to re-dimensionalize the axial and the tangential velocity profiles (i.e. \( U_{ref}^{ax} = U_{ref}^{tan} \)) the swirl number (Eq.(2.2)) was off by 16\% with respect to the experimental condition chosen\(^3\) (\( S = 0.7 \) vs. \( S = 0.6 \)). As in the axial case of chapter 7, no turbulence is

---

\(^1\)The experimental curve is normalized by the jet top speed, i.e. profiles of \( U/U_{ref} \) are shown in Dellenback et al. [17].

\(^2\)In Dellenback et al. [17] the Reynolds number is evaluated using "the average velocity in the upstream tube".

\(^3\)This discrepancy is present also in the experimental data: by integrating numerically the experimental velocity profile the swirl number results 16\% higher than the expected value of \( S = 0.6 \). This phenomenon
injected at the inlet. The effects of such an approximation are negligible since turbulence develops naturally because of the injected vorticity (the swirl velocity component) and because of the large coherent structures (such as the PVC) which trigger turbulence. Table 8.1 summarizes the B.C.s used.

<table>
<thead>
<tr>
<th>BC NAME</th>
<th>IMPOSED PROPERTY</th>
<th>TARGET VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>inlet</td>
<td>EXP velocity profile</td>
<td>$U_{ref}^{ax} = 10[m/s], U_{ref}^{tan} = 8.2[m/s]$</td>
</tr>
<tr>
<td>outlet</td>
<td>pressure</td>
<td>$101300 [Pa]$</td>
</tr>
<tr>
<td>walls</td>
<td>adherence, impermeability, adiabaticity</td>
<td></td>
</tr>
</tbody>
</table>

Table 8.1: Imposed values for boundary conditions of the geometry of Fig. 8.2.

The geometry of the swirled case differs from the geometry of the axial case. An expansion is mounted at the end of the second, downstream, cylinder. This expansion was also present in the experimental settings (Fig. 8.1, the expansion is indicated with the letter K).

This additional geometry (expansion K in Fig. 8.1 and Fig. 8.2) is required to be included in the simulation domain the flow downstream of the test section which interacts with the central recirculation zone (CRZ). As evident from Fig. 8.2 the central recirculation zone extends throughout the pipe and downstream inside the expansion K. LES prediction without the expansion K (i.e. using the same geometry of chapter 7 shown in Fig. 7.1) were bad and showed a trend opposite to what usually experienced: increasing the mesh resolution results got worse. This phenomenon is due to the convective outlet boundary is likely to be related to fact that the experimental velocity profiles where acquired by post-processing the electronic paper were they are reported. Such method/measure is evidently subject to errors.
condition\cite{85} used which is not suited to feed recirculation zones. The outlet is therefore pushed downstream and the flow is free to develop entering in the expansion "K". The geometry used for the swirled case (and the measurements planes used for the comparison LES-EXP data) is shown in Fig. 8.2.

![Image showing measurement planes and axial position with respect to the first expansion for the swirled case of Dellenback\cite{17}.

The SGS model chosen for all LES of this chapter is SIGMA \cite{79} where the turbulent viscosity is proportional to the cube of the wall distance (see section 3.2.2). The SIGMA model is preferred to WALE since it does not generate turbulent viscosity in the case of a solid-body rotation. It is therefore more suited for this flow than WALE because experimental data shows that, the flow is in solid body rotation (along the centerline of the geometry) in most of the domain J. All simulations are initialized with a zero velocity flow.

### 8.2 Swirled case, homogeneous meshes

The swirled case corresponds to a confined jet at $Re = 30\,000$ and $S = 0.6$. The set of basic LES and the characteristics of the meshes used are summarized in table 8.2. Results

<table>
<thead>
<tr>
<th>mesh LES name</th>
<th>number of nodes</th>
<th>number of tetra</th>
<th>numerical efficiency Eq.(6.27)</th>
<th>$\Delta/\eta_K$</th>
</tr>
</thead>
<tbody>
<tr>
<td>H0</td>
<td>0.3M</td>
<td>1.5M</td>
<td>1</td>
<td>134</td>
</tr>
<tr>
<td>H1</td>
<td>0.6M</td>
<td>3.4M</td>
<td>0.513</td>
<td>106</td>
</tr>
<tr>
<td>H2</td>
<td>1.7M</td>
<td>10M</td>
<td>0.016</td>
<td>62</td>
</tr>
</tbody>
</table>

Table 8.2: Table of simulations. The numerical efficiency (Eq.(6.27)) is normalized by the efficiency of LES H0 while the average mesh size $\Delta$ is non-dimensionalized by the Kolmogorov length scale for a HIT at the same Reynolds number ($\eta_K = \frac{Re - \frac{3}{4}}{D_1} = 2.24 \times 10^{-5}[m]$). The average mesh size ($\Delta$) refers to the first two cylinders where measurements were taken (J in Fig. 8.1-8.2).
are compared with experimental data taken at 8 measurement positions (Fig. 8.2). Three
different meshes are used for the basic LES (basic means without adaptation) ranging from
mesh H0 to H2 (where "H" stands for homogeneous and the number for the refinement
level). Names refer to meshes and simulations indifferently. As in chapter 7, results
obtained with the homogeneous meshes are used as reference together with experimental
data.
The flow field is highly unstable and perturbed by the presence of a Precessing Vortex
Core (PVC). Fig. 8.3 shows the PVC (made evident by a low pressure iso-surface) and the
instantaneous, velocity vectors. In swirled flows, the PVC is a small tornado that lies on
the boundary of the reverse flow zone between the zero velocity and zero streamline [78].

The flow field changes with homogeneous refinement. LES H0 and H1 show a very
small CRZ (in Fig. 8.4 it is visualized by the zero axial velocity isoline) and a secondary
recirculation zone inside the expansion K. On the contrary, the CRZ of LES H2 extends
along the whole length of the pipe and merges with the secondary recirculation zone.
However, Fig. 8.4 is misleading. The apparent continuity of the CRZ appears only in the
mean field of LES H2, while looking at one snapshot (Fig. 8.5) is evident that the CRZ is
somehow "intermittent" with the pockets of zero axial velocity (the black line along the
centerline of the flow in Fig. 8.5) which are not connected (this has been verified using 3D
isosurfaces of zero axial velocity).
Similarly, the mean tangential velocity field (Fig. 8.6) changes with homogeneous refine-
ment, and LES H2 shows a higher tangential speed throughout the pipe.

The differences in the flow fields of LES of table 8.2 are made evident in Figs. 8.4-
8.6. The velocity profiles in Figs. 8.7-8.8 show that predictions of LES H2 are better
with respect to LES H0 or H1 in all planes and LES H2 gives a good comparison with
experimental data. However, homogeneous refinement has a smaller effect in the swirled
case than in the the axial case of Chapter 7 (compare with Fig. 7.8 which shows how
homogeneous refinement can change LES results). Note that a variation of the swirl level
introduced at the inlet could modify the velocity profiles significantly. However, the aim
of these tests is not to reproduce experimental data by adjusting the swirl level, even though, as explained earlier, the amount of swirl in the flow is a source of uncertainty for this experiment.

The RMS fields of LES of table 8.2 appear very similar in Fig. 8.9, with a zone of higher RMS values after the first step expansion. These high RMS values are due to the PVC (Fig. 8.3). Similarly, the zone of higher axial velocity RMS along the centerline of the flow is linked to the "intermittency" of the "pockets" of backflow shown in Fig. 8.5. Only the zone downstream of the 1.4\([m/s]\) isoline can be considered as "pure" turbulence.
Fig. 8.10 shows the RMS profiles at plane number 3 for LES of table 8.2: clearly not injecting turbulence at the inlet has a limited effect on the flow since all LES are able to reproduce the experimental trend. This phenomenon is linked to the fact that vorticity (i.e. tangential velocity) is injected in the domain and this helps turbulence to develop. All LES converge in terms of kinetic energy after a transient period of \( 0.6[s] \) (Fig. 8.11). After \( 0.6[s] \), kinetic energy then increases in time with a small drift because of the slow convergence of the flow in the downstream expansion whose time scale is significantly larger.
Figure 8.7: Mean axial velocity profiles at the measurement planes of Fig. 8.2 for simulations of table 8.2.
8.2. SWIRLED CASE, HOMOGENEOUS MESHES

(a) Mean tangential velocity profile at plane 3
(b) Mean tangential velocity profile at plane 4
(c) Mean tangential velocity profile at plane 6
(d) Mean tangential velocity profile at plane 7
(e) Mean tangential velocity profile at plane 8
(f) Mean tangential velocity profile at plane 10
(g) Mean tangential velocity profile at plane 11
(h) Mean tangential velocity profile at plane 12

Figure 8.8: Mean tangential velocity profiles at measurement planes of Fig. 8.2 for simulations of Table 8.2.
Figure 8.9: RMS of axial velocity for LES of table 8.2

Figure 8.10: RMS axial velocity for LES of table 8.2 at plane 3 of Fig. 8.2.
8.3 Quality estimates for the homogeneous meshes

The quality of LES of table 8.2 is monitored via Pope’s criterion (Eq. 4.1) and the amount of turbulent viscosity generated by the SGS model, both time and volume averaged, the mean value of the non-dimensional wall distance ($y^+$) for wall nodes, the cumulative PDF ($\int PDF(Q)dQ$) of the Q criterion (Eq. 7.1) as discussed in section 7.3.2. All these quantities are monitored in test section J (see Fig. 8.1-8.2) and summarized in table 8.3.

As for the axial case of chapter 7, turbulent viscosity follows the theoretical curve of Eq.(7.7), see Fig. 8.12. Also, as expected, the mean value of $y^+$ diminishes as the mesh resolution is increased. Comparing table 7.3 with table 8.3 reveals that turbulent viscosity is significantly lower in the swirled case. This phenomenon can be related to the different SGS model used (WALE vs. SIGMA) and mainly to the fact that, in the swirled case, most of the energy is contained and transported by the largest organized flow structures which are almost inviscid (the tangential velocity profile does not change between plane 8 and 12, see Fig. 8.8).

\[ \text{Table 8.3: Surface time-averaged } y^+ \text{ and time-volume average of turbulent viscosity (normalized by laminar viscosity), time-volume average Pope criterion, cumulative probability of the Q criterion (as elucidated in section 7.3.2).} \]

\[
\begin{array}{|c|c|c|c|c|}
\hline
\text{mesh/LES name} & y^+_{mean} & \frac{\int_{V} \frac{<v^+_{max}>dV}{dV}}{V} & \text{Pope criterion} & Q_{cum} = \int_{10^5} PDF(Q)dQ \\
\hline
H0 & 22.4 & 5.6 & 0.60 & 13\% \\
H1 & 16.8 & 3.7 & 0.65 & 16\% \\
H2 & 15.4 & 2.0 & 0.72 & 21\% \\
\hline
\end{array}
\]

\[ \text{4Only the volume averaged turbulent viscosity is monitored since the max value of such quantity was significantly high inside the expansion K, a zone that was not isotropically refined as the test section J.} \]
Fig. 8.13 shows the $Q = 1 \times 10^6 [1/s^2]$ isosurface of the Q criterion for LES of table 8.2. The PVC in the upstream tube is distorted and destroyed soon downstream of the expansion giving birth to numerous smaller structures. The amount of small vortices increases with the mesh resolution (Fig. 8.13 and Fig. 8.14). These vortices are then convected downstream by the mean flow (Fig. 8.13 shows in transparency the Q criterion field). A measure of the different amount of small eddies is given in Fig. 8.14 which shows the PDF of the Q criterion (Eq.(7.8)) as discussed in section 7.3.2. Three informations can be extracted from Fig. 8.14. When the mesh resolution is increased:

the amount of high frequency eddy cores augments since the cumulative probability,

$$Q_{cum} = \int_{a}^{\infty} PDF(Q)dQ,$$

increases (see table 8.3), "a" = $10^5 [1/s^2]$ in Eq.(8.1) is chosen arbitrarily. Note that $Q_{cum}$ includes also the PVC;

The highest frequency present in the flow increases.

The slope of the PDF changes. This phenomenon is likely to be the effect of filtering on the spectrum of turbulent kinetic energy, see Pope [89], which affects the transfer of energy from the larger to the smaller scales (a similar trend can be observed in Fig. 7.22 also).

Finally, Pope’s criterion (Eq. 4.1) shows the expected trend, increasing with mesh refinement (see table 8.3). Pope’s criterion remains below the arbitrary value of 0.8 set by Pope [89] for all LES. The fields of Pope’s criterion for LES of table 8.2 are shown in Fig. 8.15.
8.3. QUALITY ESTIMATES FOR THE HOMOGENEOUS MESHES

Figure 8.13: Snapshots of Q criterion for simulations of table 8.2. The field of the Q criterion (black dots are zones at $Q > 1 \times 10^6$) is made transparent to make visible the 3D structure of the PVC and of the smaller eddies.

Figure 8.14: $PDF(\%)$ (Eq.(7.8)) of the Q criterion (Eq.(7.1)) for LES H1,H2,H3,H4 and A1$_{MIX}$ and A2$_{MIX}$. 
Figure 8.15: Pope criterion (Eq. 4.1) for simulations table 8.2.
8.4 Adapted meshes

Mesh adaptation is now tested using the two sensors which were able to improve LES results in chapter 7 ("GRAD" and "MIX"), plus a third sensor based on turbulent viscosity. The "GRAD" sensor is based on the mean velocity gradient tensor (see section 6.1.2). The MIX sensor (here named "MIX_{RMS}") is the intersection of the "GRAD" sensor and of a sensor based on a flow property $\Phi$ (see section 6.1.1) where $\Phi = RMS = (\bar{u}_i^2)_{ij}$. Similarly, the third sensor (here named "MIX_{\nu_T}") is obtained as the intersection of the "GRAD" sensor and a sensor based on a flow property $\Phi$, where $\Phi = \nu_T$ (see section 6.1.1).

Meshes are adapted using the solution of LES H2 of table 8.2 and are named as the sensors used to generate them, i.e. "GRAD", MIX_{RMS} and MIX_{\nu_T}. Meshes and LES are summarized in table 8.4.

<table>
<thead>
<tr>
<th>mesh/LES name</th>
<th>number of nodes</th>
<th>number of tetra</th>
<th>numerical efficiency (normalized by H0)</th>
<th>$\Delta/\eta_K$</th>
</tr>
</thead>
<tbody>
<tr>
<td>GRAD</td>
<td>0.6M</td>
<td>3.1M</td>
<td>0.33</td>
<td>100</td>
</tr>
<tr>
<td>MIX_{RMS}</td>
<td>0.6M</td>
<td>3.3M</td>
<td>0.37</td>
<td>100</td>
</tr>
<tr>
<td>MIX_{\nu_T}</td>
<td>0.6M</td>
<td>3.5M</td>
<td>0.45</td>
<td>100</td>
</tr>
</tbody>
</table>

Table 8.4: Table of simulations performed for mesh adaptation.

Figs. 8.16 and 8.17 show the adapted meshes of table 8.4 and mesh H1 of table 8.2 as a comparison. Mesh GRAD is refined:

- close to the walls,
- in the internal and external shear layer between the central recirculation zone and the corner recirculation zone,
- along the centerline of the flow because of the radial gradient of axial and tangential velocity.

Meshes MIX_{RMS} and MIX_{\nu_T} are obviously refined at the same locations of mesh "GRAD" and in the zones where the flow criterion used (the RMS of the flow speed or the modeled turbulence respectively) is higher.

Note that for this flow the zone where the RMS of the jet velocity is higher corresponds to the location of the PVC and it is not directly connected to turbulence as in chapter 7. For this reason a sensor based on turbulent viscosity (MIX_{\nu_T}) was tested.

As for the axial case of chapter 7, the minimum cell size was set to the one used in LES H1 in order to keep the numerical efficiency constant while the levels of gradation (see section 6.2.3) is set as 1.35. Even if the minimum cell size was similar for all meshes of table 8.4, it was not possible to control the simulation time step with an accuracy sufficient to get the same numerical efficiency. However LES MIX_{RMS} and MIX_{\nu_T} have approximately the same efficiency as LES H1.

---

5 Value commonly used at CERFACS and also in Chapter 7
Figure 8.16: Meshes of table 8.4 and mesh H1 of table 8.2 as reference.
Figure 8.17: Meshes of table 8.4 and mesh H1 of table 8.2 as reference.
CHAPTER 8. TEST CASE B

All adapted meshes are refined in the junction between the test section J and the expansion K (see Fig. 8.2 and Fig. 8.17) while they are all coarsened with respect to H1 in the remaining part of the expansion K. In this zone (inside the expansion K and far from the junction with the test section) turbulent viscosity was extremely high because of the large mesh size. Here, the element size for mesh \( MIX_\nu_T \) was limited by evaluating the sensor \( \Phi = \nu_t \) only inside the test section J while fixing the mesh size in the expansion K as a fraction of the cylinder diameter.

8.5 Flow field and simulations quality

The flow field changes after adaptation. The length of the central recirculation zone augments in all adapted meshes/LES compared to the LES on H1 (compare Fig. 8.19 and Fig. 8.4). The CRZ gets closer to LES H2 (which is significantly more expensive numerically). Results get closer to experimental data at all planes (Fig. 8.20-8.21). Globally, results improve with adaptation.

As shown in Chapter 7, despite a better comparison with experimental data in terms of velocity profiles, not all quality measures improve. Comparing table 8.3 with table 8.5, shows that Pope’s criterion (Eq. 4.1) and the mean value of turbulent viscosity inside the test section J get worse for all adapted meshes. On the other hand, the time and surface averaged \( y^+ \) is improved in all adapted meshes with a value even smaller than LES H2.

<table>
<thead>
<tr>
<th>mesh/LES name</th>
<th>( y^+ )</th>
<th>( \frac{\int_V \langle \nu_T &gt; dV}{\int_V \nu_T dV} )</th>
<th>Pope criterion Eq. 4.1</th>
<th>( Q_{cum} = \int_{10^5} PDF(Q)dQ ) of Fig. 8.14</th>
</tr>
</thead>
<tbody>
<tr>
<td>GRAD</td>
<td>12.5</td>
<td>6.6</td>
<td>0.57</td>
<td>12%</td>
</tr>
<tr>
<td>MIX_RMS</td>
<td>13.6</td>
<td>6.8</td>
<td>0.51</td>
<td>23%</td>
</tr>
<tr>
<td>MIX_\nu_T</td>
<td>13.6</td>
<td>6.3</td>
<td>0.52</td>
<td>22%</td>
</tr>
</tbody>
</table>

Table 8.5: Surface time-averaged \( y^+ \) and time-volume average of turbulent viscosity (normalized by laminar viscosity), time average of the max value of turbulent viscosity (normalized by laminar viscosity), time-volume average Pope criterion.

Lower \( y^+ \) values can have an effect on the axial velocity profiles: between all adapted meshes the one which shows the longer recirculation zone (at \( r = 0 \)) is also the one which predicts the best match with experimental data in the boundary layer (at \( r \approx 2D_1 \)), see Fig. 8.20 at planes 8-10-11. Indeed, LES GRAD matches experimental data perfectly at plane 8 close to the walls (Fig. 8.20(e)) while at the same plane it predicts the highest backflow speed. Similarly, at plane 10, it predicts the highest speed between all adapted meshes at \( r \approx 2D_1 \) and it reproduces the central backflow better. This observation suggests to test the effects of a wall refinement only, by increasing the resolution close to the wall in the second cylinder of the test section J. Mesh H1 was refined at the walls but keeping the overall number of nodes constant. However, with this new mesh, despite the fact that the mean \( y^+ \) was reduced to 13.9 with respect to 16.8 of LES H1 (more precisely in the second, larger cylinder went from 16.1 in H1 to 11.8 in the wall-refined H1) there was no appreciable effect on the velocity profiles.

By looking at the Q criterion (Fig. 8.22), at its PDF obtained from one snapshot (Fig. 8.23,
Eq.(7.8)) and at its cumulative probability (Eq.(8.1)), the best quality LES are $\text{MIX}^{\text{RMS}}$ and $\text{MIX}^{\epsilon}$ with a quality level similar to LES H2. On the contrary, these quantities get worse for LES GRAD.

While in the axial case of Chapter 7 it was easy to identify the reason why adaptation improved results, in the swirled case studied here it is very difficult to understand the jet dynamics and their interactions. The presence of a swirl velocity component increase the complexity of the flow significantly, which is particularly unstable because of the presence of a very large PVC. Such a condition, a large PVC which impacts the whole flow, is very far from the flow of the aeronautical swirler shown in the first part of this manuscript in which the PVC is confined into a small portion of the domain (see section 4.6.2).

The swirled case can therefore be considered as a very peculiar jet being very far from the common configurations and flows studied at CERFACS and the difficulties encountered are linked to the well-known complexity of swirl flows.
Figure 8.18: Mean axial velocity and axial velocity isolines for LES of table 8.4.

Figure 8.19: Mean axial velocity and axial velocity isolines for LES of table 8.4.
Figure 8.20: Mean axial velocity profiles at the measurement planes of Fig. 8.2 for simulations of table 8.4.
Figure 8.21: Mean tangential velocity profiles at measurement planes of Fig. 8.2 for simulations of table 8.4.
Figure 8.22: Snapshots of Q criterion for simulations of table 8.4. The field of the Q criterion (black dots are zones at $Q > 1 \times 10^6$) is made transparent to make visible the 3D structure of the PVC and of the smaller eddies.

Figure 8.23: $PDF(\%)$ (Eq.(7.8)) of the Q criterion (Eq.(7.1)) for LES of table 8.4 and H1 of table 8.2.
Lire la troisième partie de la thèse