Unsteady Turbulent Tandem Cylinders Simulation

End of studies project

Lokman BENNANI

School supervisor: Gilles GRONDIN
Supervisor: Jean-Christophe JOUHAUD

September 8, 2010
# Contents

Abstract 3

Aknowledgements 4

Introduction 5

1 Modelling and numerical methods 7
   1.1 The Navier-Stokes equations 7
   1.2 U-RANS: Unsteady Reynolds Averaged Navier Stokes 7
   1.3 LES: Large Eddy Simulation 8
   1.4 Numerical schemes 10
   1.5 Boundary conditions 11
   1.6 2D LES limitations 12

2 The tandem cylinders case 15
   2.1 Methodology 15
   2.2 Physical analysis 16
   2.3 The experimental data 18

3 Two dimensional computations 20
   3.1 The mesh 20
   3.2 Computational parameters 22
   3.3 U-RANS 2D 22
      3.3.1 Mean field 22
      3.3.2 Unsteady field 24
   3.4 Two dimensional LES 26
      3.4.1 Mean field 26
      3.4.2 Unsteady field 27
   3.5 Conclusion 30

4 Three dimensional computations 32
   4.1 U-RANS 32
      4.1.1 Mean field 33
      4.1.2 Unsteady field 37
4.2 LES .................................................. 39
   4.2.1 Mean field .................................... 40
   4.2.2 Unsteady field ................................. 42
4.3 DES with chimera mesh ............................. 43
   4.3.1 Chimera mesh .................................. 44
   4.3.2 Results and comments ......................... 46
4.4 DES with near wall modelling ....................... 47
4.5 Conclusion ......................................... 49

Conclusion ............................................. 50

A Signal processing ................................... 52
   A.1 PSD using Welch’s averaged periodogram .......... 52
   A.2 Algorithm validation ............................... 54
   A.3 Decimation ......................................... 54
   A.4 Signal processing procedure ....................... 55

B Fjortoft’s theorem ................................... 56

C CERFACS ............................................ 58
Abstract

This project deals with the numerical simulation of the unsteady turbulent flow past two cylinders in a tandem configuration. This problem comes from aeroacoustic issues. In the view of investigating the noise radiated by a landing gear a validation case is needed to assess the suitability of the currently available numerical procedures. The goal of this project is to compute mean and unsteady flowfields associated with this problem. Several approaches will be used and compared. The numerical results will also be compared to an available experimental database furnished by NASA.

The first phase of the project will concern two dimensional simulations. These simulations will provide us with preliminary results at a reduced computational cost. Then we will proceed to three dimensional simulations. We will see how this improves the results and what solutions can be proposed to further these improvements.

This project also deals with signal processing tools used for post-treatment. A power spectral density estimator will be coded in fortran. It will be based on Welch’s averaged periodogram method.
I wish to thank Mr Jean-Christophe Jouhaud for having proposed this end of studies project.

I also wish to thank Mr Jean-François Boussuge for having proposed his ideas and new approaches to this problem. I also thank him and Mr Thierry Poinset for welcoming me in the CFD team. Many thanks to Thomas Leonard for all his help.

Finally I would like to thank the trainee students with whom I spent these six months and of course all of the AAM team.
Introduction

Nowadays, constraints on noise emitted by civil aircrafts are becoming increasingly strict. In such a context, we may often think of jet engine noise for example. Jet engine noise remains the predominant source during take off. However, during landing or approach phases, airframe noise plays a significant role (fig 1). The landing gear is indentified as one of the main noise sources during these phases.

Thus, in order to better control this noise generation, it is important to identify the physical phenomena from which it originates. This will enable the design of new shapes or noise reducing devices. One way of achieving this is by using computer based noise prediction tools. It is the goal of computational aeroacoustics (CAA). Noise generated aerodynamically occurs firstly as the result of moving bodies. The moving body and resulting unsteady flow generate pressure fluctuation that are radiated as noise. It is called impulsive noise. The second noise generator is turbulence. Thus aerodynamic noise is generally composed of broadband noise resulting from the turbulence and tonal components generated by impulsive noise [28].

Presently there are three main technics for simulating turbulent flows:

- DNS (Direct Numerical Simulation) which consists in solving all the turbulent scales (no turbulence model). Of course this calls for high computational ressources, and is at present out of reach for most industrial problems.

Figure 1: Noise contributors (source: airbus)
• The RANS (Reynolds Averaged Navier Stokes) method where all the turbulence is modelled. In this form it solves a steady flowfield however an unsteady formulation exists, namely U-RANS.

• An intermediate apporach called LES (large Eddy Simulation) in which the larger turbulent scales are computed, and the smaller ones modelled.

The description of the properties of turbulence is necessary to predict the radiated noise. Calculating noise with a RANS method requires assumptions and approximations about the statistical properties of turbulence[28]. The U-RANS method will only give access to the very large unsteady motion and thus only low frequency noise can be estimated. On the other hand LES enables a much finer description and thus radiated noise can be predicted on a much larger frequency range. LES can be used to directly calculate the large acoustic wave producing pressure fluctuations[17]. Moreover the larger turbulent scales are known to be high noise generators [28]. Thus LES appears to be an adapted method for aeroacoustics. However, for complex geometries, such as landing gears, wall resolved LES is still too computationally expensive. Moreover an academic validation case which is needed to assess the suitability of the current numerical methods. This leads us to consider a simplified geometry : the tandem cylinders. The tandem cylinders provides a prototype of the different problems encountered.

Before presenting the tandem cylinders in more details, let’s first take a look at the models and numerical methods we intend to use. We will then present the tandem cylinders and the underlying physics. Finally a presentation of the 2D and 3D computations results will be done.
Chapter 1

Modelling and numerical methods

The computations are run using ONERA’s finite volume elsA code. Although elsA is a compressible fluid dynamics solver, for the sake of simplicity we will present the models and methods in their incompressible form. Moreover the tandem cylinders is a low Mach case (M=0.128) and thus compressibility effects are low.

1.1 The Navier-Stokes equations

Let us consider a viscous Newtonian fluid and an incompressible flow. The Navier-Stokes equations which describe the motion of the fluid are expressed in the following way:

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j}
\]

Solving these equations by direct numerical simulation leads to prohibitive calculation times. Thus we use alternative approaches, namely U-RANS and LES, to solve these equations. In the following sections we present the basic ideas underlying these approaches.

1.2 U-RANS: Unsteady Reynolds Averaged Navier Stokes

This method is based on the procedure introduced by Reynolds. The idea is to express all quantities as the sum of a mean and fluctuating part: \(f_i = F_i + f'_i\). After applying the Reynolds decomposition, the U-RANS
equations are expressed as [2, 1]:

\[ \frac{\partial U_i}{\partial x_i} = 0 \]

\[ \rho \frac{\partial U_j}{\partial t} + \rho U_i \frac{\partial U_j}{\partial x_i} = \frac{\partial}{\partial x_i} (2\mu S_{ij} + \tau_{ij}) - \frac{\partial P}{\partial x_j} \]

Where:

\[ \tau_{ij} = -\rho u'_i u'_j \]

Nevertheless, this leads us to more unknowns than we have equations. The system needs to be closed. To do so, the eddy viscosity hypothesis is commonly used. Originally proposed by Boussinesq, it assumes that the turbulent fluctuations have an effect similar to that of molecular viscosity. This leads to:

\[ \tau_{ij} = 2\mu T S_{ij} - \frac{2}{3} \rho k \delta_{ij} \]

Where \( k = \frac{1}{2} \overline{u'_i u'_j} \) is the turbulent kinetic energy. We then use a turbulence model to completely close the problem. It is worth noting that the U-RANS method can only capture unsteady phenomena that are not turbulent in nature. In our case we hope to capture the basic physics of the problem which features coherent structures.

1.3 LES: Large Eddy Simulation

LES makes use of a fundamental concept in turbulence, that of the energy cascade. In his view of the energy cascade, Richardson considers that the turbulence is composed of eddies of characteristic size \( l \). An eddy being a turbulent motion localised in a region of size \( l \) (although there is no precise definition). Each eddy has a characteristic velocity \( u(l) \) and timescale \( \tau(l) \).

The idea of the energy cascade is that the large eddies transfer energy to the smaller eddies by a break up process. This process continues until viscosity becomes effective and dissipates the kinetic energy[1]. By using dimensional arguments, and in the framework of locally isotropic homogeneous turbulence, Kolmogorov and Obukhov predicted that in the inertial subrange\(^1\), the energy spectrum is:

\[ E(\kappa) = C_k \epsilon^{2/3} \kappa^{-5/3} \]

(where \( \epsilon \) is the mean rate of kinetic energy transfer per unit mass). This result is consistent with the cascade process. The idea of LES is to represent the larger eddies whereas the effects of the smaller eddies are modelled.

\(^1\)That is the range where the cascading process occurs
To be more precise, LES consists in filtering the Navier-Stokes equations using a low pass filter. Applying a filtering operator to the Navier-Stokes equations yields:

\[
\frac{\partial \bar{U}_j}{\partial t} + \frac{\partial \bar{U}_i \bar{U}_j}{\partial x_i} = \nu \frac{\partial^2 \bar{U}_j}{\partial x_i \partial x_j} - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_j}
\]

We define:

\[
\tau_{ij} = \bar{U}_i \bar{U}_j - \bar{U}_i \bar{U}_j
\]

By using the incompressibility constraint, one finds:

\[
\frac{\partial \bar{U}_j}{\partial t} + \bar{U}_i \frac{\partial \bar{U}_j}{\partial x_i} = \nu \frac{\partial^2 \bar{U}_j}{\partial x_i \partial x_j} - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_i}
\]

We then use the eddy viscosity hypothesis. We thus assume that the small eddies draw energy from the larger eddies by a process similar to that of molecular diffusion. This yields:

\[
\tau^d = \nu_{SGS} (\nabla \bar{U} + \nabla^T \bar{U})
\]

Where \( \tau^d \) is the deviator of \( \tau \):

\[
\tau_{ij}^d = \tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij}
\]

We now include the scalar \( \frac{1}{3} \tau_{kk} \) with the pressure in a modified pressure term:

\[
\Pi = \bar{p} + \frac{1}{3} \tau_{kk}
\]

Finally:

\[
\frac{\partial \bar{U}_j}{\partial t} + \bar{U}_i \frac{\partial \bar{U}_j}{\partial x_i} = \frac{\partial}{\partial x_i} \left( [\nu + \nu_{SGS} \frac{1}{\rho}] \frac{\partial \bar{U}_j}{\partial x_i} \right) - \frac{1}{\rho} \frac{\partial \Pi}{\partial x_j}
\]
The closure problem now consists in expressing $\nu_{SGS}$ as a function of $\bar{U}$. This is done by using a subgrid scale model. There are two subgrid scale models available in elsA, namely the Smagorinsky model and the WALE (Wall Adapting Local Eddy viscosity) model. They are expressed in the following way:

- **Smagorinsky** [20, 12]: $\nu_{SGS} = (C_S \bar{\Delta})^2 (2|\bar{S}|)^{1/2}$

- **WALE** [18]: $\nu_{SGS} = (C_w \bar{\Delta})^2 \frac{(S^d_{ij} S^d_{ij})^{3/2}}{(S_{ij} S_{ij})^{5/2} + (S^d_{ij} S^d_{ij})^{3/2}}$

Where:

- $\bar{S}^d_{ij} = \frac{1}{2}(\bar{g}^2_{ij} + \bar{g}^2_{ji}) - \frac{1}{3} \bar{g}^2_{kk} \delta_{ij}$
- $\bar{g}_{ij} = \frac{\partial \bar{U}_i}{\partial x_j}$
- $\bar{g}^2_{ij} = \bar{g}_{ik} \bar{g}_{kj}$
- $\bar{S}_{ij} = \frac{1}{2} \left( \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right)$

### 1.4 Numerical schemes

The choice of the numerical schemes is of great importance when performing numerical simulations. In the case of LES we are explicitly calculating a part of the turbulence, it is advised to employ schemes that have been validated on academic cases. For example in the case of freely decaying isotropic homogeneous turbulence, figure 1.2 shows that the skew-symmetric form of the Jameson flux better predicts the energy spectrum and avoids spectral aliasing problems.
Therefor we have chosen to select the skew-symmetric formulation of the Jameson flux for our computational procedure. In the framework of LES the artificial viscosity parameters, originally conceived for shock capturing and stabilisation, have been reduced to the maximum in order to avoid excessive dissipation of the turbulence.

Another issue concerns the choice of the viscous flux discretisation. When test calculations were realised on the academic case of freely decaying isotropic turbulence a five point stencil discretisation did not correctly dissipate the small scales. This led to an energy accumulation in the high wave number range. However a three point stencil discretisation, although less accurate did not exhibit this problem. Thus a corrected five point stencil discretisation was introduced, and it is precisely this viscous flux that we choose.

1.5 Boundary conditions

The choice of the boundary conditions has to be carefully made. In the first instants of the computations acoustic waves appear. When these waves get to the outer boundaries of the computational domain they must be allowed to evacuate. Their reflection (possibly caused by a unadapted boundary condition) will cause perturbations in the computed flow. In order to allow this evacuation Navier-Stokes Characteristic Boundary Conditions are used (nscbc). They are based on the concept of characteristic lines [21].
As for the cylinders an adiabatic wall boundary condition was chosen.

### 1.6 2D LES limitations

Before presenting the results of the two dimensional LES case, it seems important and interesting to present some principles of two dimensional turbulence. When performing a large eddy simulation with a two dimensional treatment, we are assuming that at least the resolved turbulence is two dimensional. It is most unlikely that this is the case for the tandem cylinders. Thus we wish to justify the use of LES for such a case but also to present the errors induced by two dimensional treatment.

Two dimensional turbulence differs from three dimensional turbulence in several aspects. One of the basic physical mechanisms of the energy cascade is vortex stretching. In the inertial subrange, that is the range where the cascading process occurs, viscosity has a minor effect. Thus according to Kelvin’s theorem, as the vorticity tubes stretch, their cross section decreases. Therefore variations of vorticity on smaller scales appears. The stretching results from the action of the velocity gradients [10, 19, 15].

Taking the curl of the momentum equation for an incompressible flow with no non-conservative forces and a barotropic fluid yields:

$$\frac{\partial \omega_i}{\partial t} + u_j \frac{\partial \omega_i}{\partial x_j} = \omega_j \frac{\partial u_i}{\partial x_j} + \nu \frac{\partial^2 \omega_i}{\partial x_j \partial x_j}$$

Where $\omega_j \frac{\partial u_i}{\partial x_j}$ is the vortex stretching/tilting term.

If we assume the flow to be two dimensional, $\frac{\partial u_i}{\partial x_j} = 0$ if $i = 3$ or $i = j = 3$ and $\omega_j = 0$ if $j = 1, 2$ and consequently $\omega_j \frac{\partial u_i}{\partial x_j} = 0$ if $i = 1, 2, 3$.

Therefor there is no vortex stretching for two dimensional flows. This implies a conservation law for another quantity, enstrophy, defined as:

$$\frac{1}{2} \omega_i \omega_i$$

Or in spectral form:

$$\int_0^\infty k^2 E(k, t) dk$$

Fjortoft’s theorem (see appendix B then shows that in two dimensions, energy is more easily transferred from the small eddies to the larger ones. Kraichnan has suggested that two dimensional turbulence exhibits two kinds of inertial ranges [7]:

- An energy transfer range from small eddies to large eddies of the form $E(\kappa) = C \epsilon^{2/3} \kappa^{-5/3}$
- An enstrophy transfer range from large eddies to small eddies of the form $E(\kappa) = C' \beta^{2/3} \kappa^{-3}$
where $\epsilon$ is the mean rate of kinetic energy transfer per unit mass and $\beta$ is the main rate of enstrophy transfer per unit mass. Figure 1.3 illustrates these processes[15].

![Double cascading spectrum of forced two-dimensional turbulence](image.png)

Thus in two dimensions, the turbulent subgrid eddy viscosity concept meets with severe limitations linked to the reverse energy transfer. To take this reverse energy transfer into account the eddy viscosity should be allowed to take negative values [8, 14]. Moreover as we saw earlier, the selected models and numerical schemes are built and validated to capture three dimensional turbulence.

Nevertheless a feature of two dimensional turbulence is the formation of Kelvin-Helmholtz type coherent vortices [15]. We know the coherent vortices are an important aspect of the physics of the tandem cylinders case. Thus we may hope to capture some of the physics on the basis of this approximation.

Another limitation of this case comes from the time step. The choice of the time step for large eddy simulation isn’t an easy matter. First it should be chosen to ensure that spatial resolution is increased by a finer grid or a higher order scheme finer structures will be captured. Thus the time step should be decreased for consistency, in order to be adapted to these finer structures. To do so, we can use a CFL number based criterion. Although the CFL number comes from stability analysis, it also has a physical meaning. Choosing a time step so as to fulfill $CFL < 1$ means that between two computed instants, the information will not propagate further than the cells boundaries, and thus will not be lost. Nevertheless as the mesh size in the near wall regions is set by the constraint $y^+ \sim 1$ choosing $CFL < 1$ implies a physical time step of the order of $\Delta t_p \sim 10^{-8}s$. It is asked that the averages be performed on at least 25 periods of the principal shedding frequency, evaluated at approximately 175 Hz. This yields 0.15s of physical time, thus 15 million iterations. Knowing that 200 seconds are necessary for 1000 iterations, we find that such a computation would take 3 million seconds, that is to say 34 days (on 32 processors). In the view of the limitations of this case as far as the physics of the flow are concerned it is surely not necessary to devote
so much time to it. Thus we chose to select an implicit time integration scheme. The time step was chosen in such a way that lowering it would not change the results significantly.

These limitations imply that the results of this case should be analysed with caution. However, along with the two dimensional U-RANS case, this will provide us with preliminary results at a computationally cheap price. It will also enable a first glimpse of the physics of the flow.

It is worth noting that the CFL criterion for an LES timestep is not unique. There also exists a criterion based on $\Delta t^+ = \frac{u_2^2 \Delta t}{\nu}$. For example $\Delta t^+ < 1$ is sometimes used for academic flows (ex: plane channel) [17].
Chapter 2

The tandem cylinders case

The case consists of two cylinders of diameter $D$ separated by a distance $L$. The inflow is uniform with streamwise velocity $U_0 = 44m/s$ and Mach number $M = 0.128$. The flow is parallel to the cylinders center-to-center line. The diameter is equal to $5.715cm$. The Reynolds number based on the diameter and the inflow velocity is $Re = 1.66 \times 10^5$. The geometry is sketched fig 2.1:

![Figure 2.1: The tandem cylinders](image)

2.1 Methodology

The tandem cylinders constitutes what is called a benchmark problem. That is, a problem for which an experimental database is available, thus enabling the validation of the numerical results and code to code comparisons. We will start with two dimensional computations. They will provide preliminary results at reduced computational cost. However much care will be taken when analysing the results as three dimensional effects are known to be of...
importance[3]. Then we will move on to three dimensional computations where several methods will be tested.

2.2 Physical analysis

The tandem cylinders configuration is unsteady and turbulent. Three main physical phenomena are of upmost importance:

- Shear layer separation: in the near wall regions the kinetic energy variations are due to pressure and viscous forces. The viscous deformation work irreversibly transforms (dissipates) kinetic energy into heat. This results in a diminishing of the kinetic energy. If the adverse pressure gradient is too high the loss of kinetic energy may be sufficient to stop the flow. It will no longer be able to stay attached to the wall. This phenomenon is called separation [11].

- Kelvin-Helmholtz instabilities: they are the result of the shear flow’s inflexional velocity profile. Let us consider the shear flow fig 2.2 and consider a fluid particle A at the center of the layer.

![Figure 2.2: Roll up process](image)

This particle has more vorticity than the particles at higher or lower y. Thus if we imagine that a perturbation moves a particle B (originally on the y = 0 axis) to a higher position by velocity induction a roll up
process will occur. We can also see this by considering a rotational zone that would be given a sinusoidal form by a perturbation (fig 2.3). Velocity induction will lead to a roll up process. The resulting vortices are called Kelvin-Helmholtz vortices [15].

\[ \text{Figure 2.3: Roll up process [15]} \]

- Von Karman street (vortex shedding): when the Reynolds number is high enough the cylinder’s wake exhibits periodic vortex shedding. One can interpret the formation of these vortices as the result of an instability caused by a doubly inflexional velocity profile. The instability is triggered by the shear layers[15].

From an experimental point of view, the tandem cylinders have already been studied extensively for subcritical Reynolds numbers ($< 10^5$). For example Zdrakovich’s study sets the path for a classification of the flowfield patterns encountered at different spacings [16, 3, 4]. Although in our study, we are interested in a higher Reynolds number, such information is interesting for it informs us on the phenomena we may be expected to observe.

- $L/D < 1.1$: The cylinders behave as a single bluff body and vortex shedding occurs only on the downstream cylinder.
- $1.1 < L/D < 1.6$: The shear layers from the upstream cylinder attach alternatively to the downstream cylinder. Vortex shedding occurs only on the downstream cylinders.
- $1.6 < L/D < 2.5$: The shear layers from the upstream cylinder attach to the downstream cylinder. Vortex shedding occurs only on the downstream cylinders.
• $2.5 < L/D < 3.2$ : Intermittent vortex shedding appears in the gap between the cylinders and on the downstream cylinder.

• $3.2 < L/D < 3.8$ : Bistable flow switching between intermittent and constant shedding.

• $L/D > 3.8$ : Vortex shedding occurs on both cylinders. They exhibit a behaviour similar to that of a single cylinder.

In our present case, as observed experimentally, the flow will accelerate on the front part of the cylinder whereas it will decelerate on the rear part. This deceleration will be associated with a recompression thus creating an adverse pressure gradient leading to the separation of the shear layer. Also the shear flow in the boundary layer will lead to the formation of Kelvin-Helmholtz vortices leading to a roll up process. Theses vortices will then induce vortex shedding. The Kelvin-Helmholtz instabilities will then in turn influence the flow separation on the second cylinder [3].

2.3 The experimental data

The experimental database which is used to validate the numerical results was provided by NASA. These experiments were realised in the Basic Aerodynamic Research Tunnel (BART) and Quiet Flow Facility (QFF) at NASA Langley research center [4].

The BART is a subsonic windtunnel. When realising the measurments the free-stream velocity was 44 m/s (Mach=0.128). The Reynolds number based on the cylinder diameter was $1.66 \times 10^5$. The free-stream turbulence level was 0.067\%. The cylinder’s span was $b = 12.4D$ which corresponds to the entire height of the BART tunnel. Moreover the upstream cylinder was tripped between azimuthal locations of $50^\circ$ and $60^\circ$ to ensure a fully turbulent process.

The QFF is specifically designed for anechoic testing and is an open jet facility. The cylinder’s span was $b = 16D$. For the same reasons as previously the upstream cylinder was tripped. The Mach number for this test was 0.1285 (43.3 m/s) with dynamic pressure 1145 Pa.
Figure 2.4: NASA Langley test facilities
Chapter 3

Two dimensional computations

3.1 The mesh

The main difference between the U-RANS mesh and the LES mesh concerns the number of cells in the spanwise direction. Thus the mesh will be common for the two dimensional cases. The mesh in the near wall region is built so as to fulfill $y^+ \sim 1$.

Moreover in order to perform parallel computations the mesh has to be split in several blocks. In this case the computations are all going to be run on 32 processors. Thus in order to optimise the parallel computation the mesh was split into 32 blocks while trying to include the same number of elements in each of them. The goal of this operation is to have a well balanced processor loading. Sometimes (as in the three dimensional cases) it is not possible and the mesh is split into more blocks (for example 35) which leads to several blocks being assigned to one processor. This increases the load balancing.

The main characteristics of the mesh are:

- Mesh type: Structured C mesh
- Number of nodes: 625344
- Number of cells: 306502
- Number of blocks: 32
- Load balancing: 0.38%
Figure 3.1: Full domain mesh

Figure 3.2: Near wall mesh
3.2 Computational parameters

The computational parameters are also the same in the two cases, although the time step could have been reduced in the U-RANS case.

- Perfect gas model ($\gamma = 1.4$)
- Jameson flux
- Implicit time integration scheme
- Time step (non dimensional) $\Delta t = 0.005$. Yielding $CFL_{x_{max}} = \frac{u_{max} + c}{\Delta x_{min}} = 1.45$ and $CFL_{y_{max}} = 90$
- Temporal scheme : Backward Euler
- Dual time-stepping algorithm : gear
- Number of subiterations : 15
- LU-SSOR scalar resolution

All the computations were made on CERFACS’s IBM IDATAPLEX machine : octopus. It is an 82 node cluster, each node being composed of two quad core intel nehalem processors.

This highly interactive and complex flow exhibits a long duration initial transient phase that can easily be mistaken for the established unsteady flow. Therefor as required for the benchmark, a flow particle must be allowed to travel a distance of 50 diameters (that is 0.065 s of simulation time) before sampling and averaging. In this case this means that 65000 preliminary iterations are needed. This corresponds to approximately 14h of computation time.

3.3 U-RANS 2D

In this section we present the results of the 2D U-RANS computation. The two equation $k - \omega$ menter turbulence model was used for it has a better behaviour in the near wall regions. The numerical results often don’t match the experimental data, therefor we present only the main results. Once the initial transient phase was passed the computation was run for 200000 iterations and took approximately 2 days (on 32 processors).

3.3.1 Mean field

The time averaged pressure coefficient $Cp$ for both cylinders are presented figure 3.3.
The computed curves display an asymmetric distribution with two suction peaks. While the location of the suction peaks is fairly well predicted, the computation overpredicts their magnitude. The computed pressure recovery on the back of the cylinders is also overpredicted and displays strong irregularities.

We now present the centerline mean velocity in the gap region and in the wake of the rear cylinder, as shown fig 3.4. These curves enable an evaluation of the accuracy of the computed recirculation zones. We can see that they are underpredicted or not captured by the computation. Once again, the magnitude is overpredicted.
3.3.2 Unsteady field

The *rms* pressure coefficients presented fig 3.5 display strong overpredictions. We also observe a non physical behaviour in the pressure recovery zone of the front cylinder. Other than that the shape of the curves seem to match.

![RMS pressure coefficients](image)

(a) Front cylinder  
(b) Rear cylinder

Figure 3.5: RMS pressure coefficients

The unsteady field was mainly investigated through power spectral densities. Unsteady pressure signals where obtained by probing the field at locations \( \theta = 135^\circ \) and \( \theta = 45^\circ \) on the front a rear cylinder respectively. Results are presented fig 3.6 :

![Power spectral densities](image)

(a) Front cylinder 135\(^\circ\)  
(b) Rear cylinder 45\(^\circ\)

Figure 3.6: Power spectral densities

The shedding frequency is fairly well predicted for the front cylinder, although the magnitude is wildly overpredicted. As for the rear cylinder,
the impingement frequency is slightly underestimated but the magnitude remains consistent with the experimental data.

Finally, to provide a better picture of the unsteady flow we present two snapshots in fig 3.7. One representing the spanwise component of vorticity, the other representing $\frac{\nabla \rho}{\rho}$.

![Figure 3.7: Unsteady flow](image)

As we can see, the computation effectively captures the vortex shedding motion, especially in the gap between the cylinders. In the wake of the
second cylinder, the structures tend to spread and fragment. This is due to a strong coarsening of the grid in that region, inducing their dissipation.

3.4 Two dimensional LES

Taking into account the known limitations presented earlier, we present the 2D LES results. The WALE subgrid scale model was used for the computation. Only the main results are presented for the same reasons as the 2D U-RANS case. The computation was run for 200000 iterations and took approximately 2 days (on 32 processors).

3.4.1 Mean field

Firstly we present in figures 3.8 the time averaged pressure coefficients:

![Mean pressure coefficients](image)

Figure 3.8: Mean pressure coefficients

The front cylinder exhibits once again overpredicted suction peak magnitudes and irregularities in the pressure recovery zone. However, the results on the rear cylinder more or less agree with the experimental data.

Figure 3.9 shows that both recirculation zones were captured. But once again their sizes are underestimated, and the mean velocity values overpredicted.
3.4.2 Unsteady field

Fig 3.10 displays the \( \text{rms} \) values of the pressure coefficients:

The \( \text{rms} \) magnitudes are again hugely overpredicted. Moreover the front cylinder exhibits non physical twin suction peaks.

We now present the power spectral densities of the pressure signals in figure 3.11:
No shedding frequency can be clearly observed. The curves are smooth due to a higher segmentation and the use of zero padding. This was necessary in order to obtain exploitable power spectral densities. Despite the excessive smoothing, what seems to be a peak (a lump) appears at approximately $210 - 230\,\text{Hz}$ for both cylinders. Thus the 2D LES overpredicts the shedding frequency.

Finally, as we did earlier, we present shots of the spanwise vorticity field and density gradient fig 3.12:
We observe, consistently with what is expected, much finer scales of motion than in the U-RANS case. In addition to vortex shedding, Kelvin-Helmholtz type instabilities are resolved. Vortex shedding is more obvious on an animated visualisation of the flow. Nevertheless, as mentioned before, we must be cautious about this case. Fig 3.13 shows a snapshot of the experimental spanwise vorticity.
As we can see, the shear layers stretch quite a bit before they roll up, thus giving birth to vortex shedding. This is not the case for the 2D LES where the shear layers seem to stay attached to the cylinder walls. This induces a possibly non physical flowfield.

### 3.5 Conclusion

Fig 3.14 shows the experimental results for spanwise pressure correlations. We observe a decay in the spanwise correlation of pressure fluctuations. This ascertains the presence of three dimensional effects.

2D treatment is not adapted to the tandem cylinders. Although it enables to capture some of the basic phenomena there are many drawbacks to these cases. We summarize the most important of them:

- Overpredicted magnitudes.
- Poorly calculated, or non captured recirculation zones.
- Non physical results.
This is most likely due to the lack of 3D effects. This causes the absence of spanwise dissipation which leads to the observed overpredictions. Moreover, wall treatment is a known difficulty for LES. Finally, the use of LES in a 2D formulation probably led to non physical phenomena.
Chapter 4

Three dimensional computations

In this section we move on to three dimensional computational procedures. As we saw earlier, two dimensional treatment isn’t sufficient. The goal here is to accurately capture, quantitatively and qualitatively, the main physics of the flow. As far as the mesh is concerned it is common to U-RANS, DES with wall modelling and LES cases. It is simply an extruded form of the two dimensional mesh presented earlier.

The computations were realised using the same computational resources as in the two dimensional case.

4.1 U-RANS

As for the 2D U-RANS simulations, the two equation \( k - \omega \) Menter turbulence model was selected for the 3D U-RANS computations. The non dimensional time step was highered to \( \Delta t = 0.05 \). To do so the \( \chi_4 \) artificial viscosity was set to a typical RANS value of 0.016.

To eliminate the initial transient 6500 iterations were needed and took approximately 4 days of computation.

Once the initial transient phase was passed the computation was run for 15000 iterations and took approximately 9 days (on 32 processors). The mesh characteristics are the following:

- Mesh type: Structured C mesh
- Number of nodes: 9 692 832
- Number of cells: 9 181 560
- Number of blocks: 32
- Load balancing: 0.38%
4.1.1 Mean field

Let us start by presenting in fig 4.1 the mean coefficient of pressure obtained:

Figure 4.1: Mean pressure coefficients

The front cylinder $C_p$ is excellent agreement with the experimental values, although the suction peak magnitudes are very slightly underestimated. No irregularities appear in the pressure recovery zone, matching the experiment perfectly. As for the rear cylinder, the numerical values don’t match as well. The suction peak magnitudes are underpredicted. However, good agreement is obtained in the pressure recovery zone with respect to the BART experiment. Overall, the $C_p$ distributions are symmetric and the suction peak locations are well predicted.

We now take a look at the centerline mean streamwise velocity distributions in fig 4.2:
Both recirculation zones are captured although overpredicted. The magnitudes are in fair agreement in the gap and overestimated in the aft. Overall the shape of the curves are in agreement with the experiments.

The centerline 2D TKE distributions are presented fig 4.3. Fairly good agreement is obtained in the gap region. In the aft the magnitudes and peak value location are overpredicted. This is consistent with what was observed on the mean velocity distributions.

The 2D TKE along $x = 1.5D$ and $x = 4.45D$ are displayed fig 4.4:
In spite of slightly over and under predicted areas, the gap results match the experiment overall. This is not the case in the aft region. The peak magnitude is a little overestimated as is it’s location. We also note the presence of a large dip in the central area.

Here we compare computed and experimental views of the 2D TKE field in the gap and the aft.

---

**Figure 4.4: 2D TKE Distributions**

(a) $X=1.5D$

(b) $X=4.45D$
The shape and localisation of concentrated computed 2D TKE are in good agreement with the experiment. Moreover the magnitudes are correctly predicted although seemingly slightly overestimated in the aft, consistently with previous observations.

To better visualise the recirculation zones the 2D streamlines are presented fig 4.6. We clearly see the formation of both recirculation zones.
4.1.2 Unsteady field

The \textit{rms} pressure coefficients are now presented, fig 4.7:

![Figure 4.6: 2D streamlines](image)

![Figure 4.7: RMS pressure coefficients](image)

On the front cylinder the suction peak \textit{rms} magnitudes are overpredicted as well as the pressure recovery zone. Irregularities appear on the front part of the cylinder ($\theta = 0^\circ$) and in the pressure recovery zone. The distribution of the rear cylinder is in better agreement with the experimental data. Nevertheless the peaks are underpredicted and the dip in the pressure recovery zone is overpredicted. The distributions tend to be symmetric.

The psd (fig 4.8) are unfortunately unfavorable to an accurate estimation of the shedding frequency. If we consider the lump of the front cylinder psd as a peak then the shedding frequency is a little underpredicted. As for the rear cylinder, clearly, no information can be deduced. The levels are also underpredicted.
In order to evaluate the presence of three dimensional effects two unsteady pressure probes were placed on each cylinder. One at midspan and one near the boundary. We can thus evaluate the correlation of these signals:

- Front cylinder: \( r_{xy}(0) = 0.8783 \)
- Rear cylinder: \( r_{xy}(0) = 0.5346 \)

As we can see the rear cylinder signals are more decorrelated. Overall a certain level of decorrelation is attained meaning that three dimensional effects have been captured. To better emphasize this fact we present fig 4.9 a slice of the Mach number field in the \( y = 0 \) plane. We can clearly see the non uniform distribution after the front cylinder.
Finally to get a better picture of the unsteady flow we present fig 4.10 the spanwise vorticity field. As we can see the field is more consistent with the experimental shot (fig 3.13).

4.2 LES

The 3D LES computations were realised by Thomas Leonard, Ph.D student at cerfacs. He kindly provided us with some of his results that we present in
the following. The WALE subgrid model was used with a non dimensional time step $\Delta t = 0.005$. The time integration is implicit and the numerical schemes remain the same as in the 2D case. Although not really suited for LES the mesh is the same as for 3D U-RANS case.

4.2.1 Mean field

As we can see fig 4.11 overall the results match quite well. However the first cylinder still exhibits secondary lumps near the suction peaks. This is probably due to the fact that the mesh isn’t adapted to LES. The spanwise refinement isn’t sufficient and thus the 3D effects are still not accurately captured. In both cases the pressure recovery zone is curved and doesn’t present the expected plateau.

In the following we observe that although they display similar shapes, the 2D TKE is this time underpredicted. This is consitent with the fact that the mesh isn’t adapted to LES. The coarsness in the spanwise direction induces dissipation.
Finally we see fig 4.13 the size of the recirculation zone in the gap is underestimated while the magnitude is overpredicted. As for the aft, the curves almost match. The recirculation zone is slightly overestimated but the magnitudes remain of the same order.
4.2.2 Unsteady field

The \textit{rms} pressure coefficients (fig 4.14 were better calculated than in the 2D case although still overpredicted. They display shapes similar to the experimental curves. We notice the disappearance of the non physical double suction peaks on the front cylinder. This means they were effectively due to the lack of three dimensional effects. The overprediction is probably also due to the lack of three dimensionality.

For this computation case 32 probes were placed in the spanwise direction on the first cylinder only. This enabled spanwise averaging of the power spectral densities. However a satisfying result still wasn’t achieved (fig 4.15). We note the presence of high frequency noise and a clear peak at the shedding
frequency is hard to define. The shedding frequency can be estimated, and would be slightly underestimated.

![Front cylinder PSD](image)

Figure 4.15: Front cylinder PSD

As in the U-RANS case, we evaluate the correlation of two pressure signals. One is probed mid span and the other at the boundary. We find $r_{xy}(0) = 0.999$ meaning that the signals are almost entirely correlated and thus that three dimensional effects have probably been poorly captured. This is consistent with the fact that once again the mesh is not adapted for LES.

### 4.3 DES with chimera mesh

As mentioned in the two earlier sections, the computation domain needs to be extended in the spanwise domain. In order to do this by maintaining a reasonable computation time, one idea is to use the chimera meshing technic [27] combined with the DES method [26].

The difficulties associated with LES in the near wall regions has led to consider hybrid models that attempt to combine the best aspects of RANS and LES. Detached Eddy Simulation is such a method. The idea is to use RANS modelling in the near wall regions and treat the rest of the domain in LES.

The idea of the chimera technic is to split the mesh into several overlapping parts. This will enable a sufficiently fine mesh in the near cylinder region and a very coarse mesh in the outer regions thus lowering the computation time. Using DES will allow us to use a higher time step than for LES. The Spallart-Allmaras one equation turbulence model[2] was used in order to reduce the computational cost.
4.3.1 Chimera mesh

The mesh is made up of five overlapping parts. These parts have different spanwise refinement. The mesh is coarser in the outer region and refined in the near cylinder region where the finer scales have to be captured.

- Two cylinder meshes with 64 cells per diameter spanwise. The front cylinder has 256 circumferential nodes whereas the rear cylinder has 360. The fact that the flow is more perturbed on the rear cylinder led to this choice. These meshes are in red in fig 4.16.

- A mesh that englobes both cylinders. It has 32 cells per diameter spanwise. The cell sizes were made to be approximately uniform in the $x-y$ plane with $\delta x = \delta y = 0.02D$. This mesh is in blue in fig 4.16.

- A wake mesh with 32 cells per diameter spanwise. This mesh is in brown in fig 4.16.

- Finally, a global mesh with 5 cells per diameter. This mesh is in black in fig 4.16.

In fig 4.16(b) we show an example of an overlap region. These overlap regions were taken into account as ‘overlap’ boundary conditions in the python topology scripts.
Figure 4.16: Chimera Mesh and overlapping region

The main characteristics of this mesh are the following:
4.3.2 Results and comments

As we can see on the following snapshot (fig 4.17), this solution doesn’t provide satisfactory results.

It seems as if the flow is highly influenced by the mesh. The flow structures are dissipated or don’t pass through the overlapping zones. It could have been a transient initial process but computations over more than 5000 iterations provide no improvement. The definition of the overlapping zones were checked and it appears they are correctly set.
In fact the problem comes from the method itself. The chimera method is non-conservative. The interpolation process between overlapping mesh zones is not accurate enough to enable the convection of the highly unsteady flow structures. Thus the chimera mesh solution was abandonned.

4.4 DES with near wall modelling

The idea is to use DES with wall modelling. The wall model will thus be a RANS wall model. This permits a coarser grid in the near wall regions and a higher time step. Once again the idea is to lower the computational cost. A coarser 2D mesh was extruded along $3D$ with 32 cells per diameter in the spanwise direction. The Spalart-Allmaras turbulence model was once again used. We hope that this computational procedure will enable a more accurate capturing of the physics.

The mesh characteristics are the following:

- Mesh type: Structured mesh
- Number of nodes: 26 949 122
- Number of cells: 26 063 232
- Number of blocks: 35
- Load balancing: 2.31%

Unfortunately, even though the computation is still running and may not be fully converged, the steady pressure coefficients displayed fig 4.18 show an overprediction of the suction peaks and an underestimation of the pressure recovery zone.
These results were obtained after 5000 iterations. For the same num-

Figure 4.18: Pressure coefficients
ber of intermediate iterations the steady pressure coefficients had already converged in the U-RANS 3D case. This means that this method may not provide any significant improvement on the 3D U-RANS case.

4.5 Conclusion

Results were significantly improved by considering the problem in its natural three dimensional form. Nevertheless the mesh was more adapted to U-RANS than LES. Therefore the LES case still requires more spanwise mesh refinement in order to provide satisfying results. The shedding frequencies were still hard to determine. This is due to the fact that the signals were too short. Longer signals are needed for better spectral analyses. We also conclude that the overpredictions observed in the two dimensional case were effectively due to the lack of spanwise dissipation and three dimensional effects.

The non-functioning of the chimera solution is most probably due to the non-conservative nature of this method and that the interpolation order is too low. Using a higher order interpolation could provide a solution to this problem.

Finally the DES with wall modelling could have provided an interesting alternative. But the preliminary results show that this case may also not provide satisfying results.
Conclusion

To conclude we see that the tandem cylinders features many physical phenomena. The experimental results show that the second cylinder is the main noise generator. The psd’s display higher levels for the second cylinder. The physical phenomenon from which this originates is the impingement of the vortical structures on the front of the rear cylinder. The other noise generating phenomenon is vortex shedding. The unsteady flow generated by the vortical structures will generate pressure fluctuations which will be radiated as noise.

Qualitatively the U-RANS and DES with wall modelling computations were able to capture the main physical phenomena. Quantitatively the best results were achieved by the 3D U-RANS case.

The unsteady and turbulent nature of this problem challenges the numerical procedures. The Banc-I workshop which took place the 10 – 11\textsuperscript{th} of June in Stockholm shows that no Navier-Stokes solver is able to correctly solve this problem in it’s entirety \[9, 13, 24, 25, 5\] although fairly satisfying results are obtained in some cases. The best results were achieved by a Lattice Boltzmann solver. This enabled a very fine mesh in the spanwise direction (128 nodes per diameter with 16D span) and simulations over long physical times. The procedure was reasonably cheap (4 days on 256 processors). This goes to show how important it is to take into account three dimensional effects. As we saw the three dimensional cases offer considerably improved results. Moreover they are much more consistent with the actual physics of the flow. For signal processing purposes (and improved convergence) it is also important to simulate the flow over long physical times. The longer the signal, the better the signal processing procedures will work.

Therefor our suggestions for improving the computational procedure are:

- Refine the mesh in the spanwise direction with at least 3D span. This will enable the capturing of three dimensional effects which are of prime importance to this case.
- Refine the mesh in the wake of the rear cylinder to avoid premature overdiffusion and possible perturbations.
• Simulate over longer physical times.

This implies that more computational resources be used for this case. Moreover considering the size of the mesh, DES methods are surely the best compromise at the time being. A Large Eddy Simulation with near wall resolution would be too costly. However the use of wall models could provide a solution for a large eddy simulation of the tandem cylinders.
Appendix A

Signal processing

A.1 PSD using Welch’s averaged periodogram

During the post treatment of the results several signal processing tools are used. The most important of them is a power spectral density algorithm that we coded in fortran. This algorithm is based on Welch’s averaged periodogram method [22] which we wish to present here. If $x(t)$ is a stationnary random process its auto-correlation function is defined as:

$$\gamma_{xx}(\tau) = E[x^*(t)x(t + \tau)]$$

Wiener-Kintchine’s theorem then yields the power spectral density:

$$\Gamma_{xx}(f) = \int_{-\infty}^{+\infty} \gamma_{xx}(\tau)e^{-2j\pi ft}d\tau$$

But in practice we deal with a single realisation of $x$ and thus we do not have access to $\gamma_{xx}$. Instead we can calculate the time averaged auto-correlation function:

$$R_{xx}(\tau) = \frac{1}{2T_0} \int_{-T_0}^{T_0} x^*(t)x(t + \tau)dt$$

Or in its discrete form:

$$r_{xx}(m) = \frac{1}{N} \sum_{n=0}^{N-m-1} x^*(n)x(n + m), m = 0, ..., N - 1$$

$$r_{xx}(m) = \frac{1}{N} \sum_{n=|m|}^{N-1} x^*(n)x(n + m), m = -1, ..., -(N - 1)$$

Where $N$ is the number of samples. The corresponding estimate of the power spectral density is:

$$P_{xx}(f) = \sum_{n=-(N-1)}^{N-1} r_{xx}(m)e^{-2j\pi fm}$$
And by substituting the expression for $r_{xx}$ we find:

$$P_{xx}(f) = \frac{1}{N} \left| \sum_{n=0}^{N-1} x(n)e^{-2j\pi fn} \right|^2 = \frac{1}{N} |X(f)|^2$$

This power spectral density estimate is called the periodogram. This estimate is asymptotically unbiased in the sense that taking its expected value in the limit $N \to +\infty$ will yield $\Gamma_{xx}(f)$. However, its variance does not go to zero as $N \to +\infty$. Therefore the periodogram is not a consistent estimate of the true power spectral density. We need to find a method for reducing the variance of this estimate. This is precisely the goal of Welch’s method. The algorithm is the following:

- Divide the data sequence into $L$ overlapping segments (typically 50% overlap):

  $$x_i(n) = x(n + iD) \quad n = 0..M-1 \text{ and } i = 0..L-1$$

- Multiply each segment by a window function $w(n)$ (Hamming for example).

- Fourier transform $x_i(n)w(n)$ resulting in the “modified” periodogram:

  $$P_{xx}^i(f) = \frac{1}{MU} \left| \sum_{n=0}^{N-1} x_i(n)w(n)e^{-2j\pi fn} \right|^2$$

  $U$ is the window power normalisation factor:

  $$U = \frac{1}{M} \sum_{n=0}^{M-1} w^2(n)$$

- The average of these modified periodograms yields the Welch power spectral density estimate:

  $$P_{xx}^W(f) = \frac{1}{L} \sum_{n=0}^{L-1} P_{xx}^i(f)$$

In the fortran program, the Fourier transforms were computed using an open source Fast Fourier Transform algorithm: FFT pack 5. It is worth noting that in the above expression the power spectral density is a continuous function of $f$. The program’s output is of course discrete. The spectral resolution is directly linked to the size of the segments: $df = \frac{1}{T_{seg}}$, where $T_{seg}$ is the duration of a segment. Moreover, high segmentation will lead to lower variance but also low resolution. On the opposite, low segmentation will yield a less consistent estimate but with a higher resolution. We thus see that a compromise has to be found when using this algorithm.
A.2 Algorithm validation

To validate the algorithm we consider signal composed of a sinusoid with a 200Hz component and noise sampled at 1000Hz on a duration of 1s. We compare the results fig A.1:

![Figure A.1: Psd estimated with fortran program and Matlab](image)

We see that both results are almost identical. Moreover the results are consistent. The peak at 200Hz is well resolved. We thus consider this as a validation of the fortran program.

A.3 Decimation

The output signals of our computations have a very high sampling rate (100-200 KHz). As we are only interested in the low frequency range (0-3500Hz) we use a Matlab resampling procedure called decimation. This routine filters the input data with a low pass eighth order Chebychev filter. Then it reduces the sampling rate to the desired lower rate. The filtering process is necessary to avoid spectral alliasing problems.
A.4 Signal processing procedure

The signal processing procedure will thus be as follows:

- Determine the sampling rate of the output signal.
- Resample the signal 7500 Hz sampling frequency using Matlab’s decimation routine.
- Use the Welch’s method algorithm to compute the power spectral density estimates.
Appendix B

Fjortoft’s theorem

We wish here to briefly present Fjortoft’s theorem \([23, 15]\) which provides the basis for the phenomenology of two dimensional turbulence. In two dimensional turbulence the kinetic energy conservation law still holds, yielding:

\[
\frac{d}{dt} \int_0^\infty E(k, t) dk = -2\nu \int_0^\infty k^2 E(k, t) dk
\]

And the absence of vortex stretching yields a conservation law for enstrophy:

\[
\frac{d}{dt} \int_0^\infty k^2 E(k, t) dk = -2\nu \int_0^\infty k^4 E(k, t) dk
\]

As in the inertial range viscosity doesn’t affect the dynamics, the right hand sides can be set to zero. If we now consider these conservation laws truncated in order to retain only three modes \(k_1, k_2 = 2k_1\) and \(k_3 = 3k_1\), we find that the energy variations \(\delta E_i\) must satisfy:

\[
\delta E_1 + \delta E_2 + \delta E_3 = 0
\]

\[
k_1^2 \delta E_1 + k_2^2 \delta E_2 + k_3^2 \delta E_3 = 0
\]

This yields:

\[
\delta E_1 = -\frac{5}{8} \delta E_2
\]

\[
\delta E_3 = -\frac{3}{8} \delta E_2
\]

\[
k_1^2 \delta E_1 = -\frac{5}{32} k_2^2 \delta E_2
\]

\[
k_3^2 \delta E_3 = -\frac{27}{32} k_2^2 \delta E_2
\]

Therefore if \(k_2\) loses kinetic energy \((\delta E_2 < 0)\) as would be the case for freely decaying turbulence, then more of it will be transferred to \(k_1\) and
more enstrophy will be transferred to \( k_3 \). Thus in two dimensional turbulence, energy is more easily transferred from the higher wave numbers to the lower wave numbers. Physically the inverse energy transfers in decaying two dimensional isotropic turbulence are due to the pairing of large energetic eddies [15].
Appendix C

CERFACS

CERFACS is the European Center for Advanced Research and Training in Scientific Computation. It’s goal is to develop advanced computational methods in order to solve large problems relevant to research as well as industry. The seven current shareholders of CERFACS are EADS, CNES, ONERA, TOTAL, Meteo-france, EDF and SAFRAN. The activities at CERFACS comprise CFD, parallel algorithms, climate modelling, aviation and environement. The CFD team’s partners include Airbus, Turbomeca and Snecma amongst others. This project took place in the AAM (Advance Aerodynamics and Multiphysics) groupe, itself a part of the CFD team along with the combustion groupe.
Bibliography


