Hybrid RANS-LES Modelling on a strongly detached turbulent flow around tandem cylinder at high Reynoldsnumber

Gual Skopek, Marc[a], Braza, Marianna[a], Hoarau, Yannick[b]

a. *IMFT: Allée du Professeur Camille Soula, 31400 Toulouse, France*
Email: Marc.Gual-Skopek@imft.fr
b. *INP(ENSEEIHT)-CNRS: 2, Rue Charles Camichel, 31071 Toulouse cedex 7*

Abstract :

The turbulent flow around a generic configuration of a landing gear ('the tandem cylinder') is simulated and analysed physically at $Re = 1.6 \cdot 10^5$, by means of hybrid RANS-LES turbulence modelling approaches, which are compared with URANS ones. Especially, the statistical part of the Delayed Detached Eddy Simulation (DDES) is reconsidered by means of the Organised Eddy Simulation, OES, to take into account non-equilibrium turbulence effects. The results are compared with experiments carried out at the NASA-Langley Research Centre, in the context of ATAAC EU-program.

Keywords : Tandem Cylinder, Hybrid RANS-LES, Detached Turbulent Flow

1 Introduction

For separated flows Unsteady Reynolds-Averaged-Navier-Stokes (URANS) equations have proven to be a poorly adapted approach, but showing a good resolution of the attached boundary layer. Also URANS-methods have been proven fast and robust. Therefore the *Advanced Turbulence Simulation for Aerodynamic Application Challenges* (ATAAC) project focusses on different hybrid RANS-LES schemes. This schemes will be tested on various test cases with highly detached turbulent flow, such as the Tandem Cylinder configuration.
This case has a strong industrial relevance, as the Tandem Cylinders are a generic configuration of a commonly used landing gear of a commercial airplane. The flow is at Reynolds number $Re = 1,6 \cdot 10^5$, based on the free stream velocity U_∞ and the cylinder's diameter D. Many experiments [1, 2] and simulations [3] were performed at the NASA-Langley research center for this test case.
This results have shown, that the flow around the tandem cylinders is characterized by a highly complex flow. The flow shows a strong turbulent detachment and a shear layer roll-up around the first cylinder. This detachment of the flow at the first cylinder creates an unsteady interaction between the first and the second cylinder and a highly unsteady detachment of the boundary layer at the second cylinder. For this reason this test case was chosen to test a number of hybrid numerical schemes. So, simulations with an advanced numerical approach, like the *Organized Eddy Simulation* (OES) and the hybrid approaches such as the DES (Detached-Eddy-Simulation) and the DDES (Delayed DES) were performed.
It is needed to be said, that all variables (time, velocity, vorticity...) are given non-dimensionalized with the inflow velocity and the cylinders diameter : U_∞/D.

2 Test Case

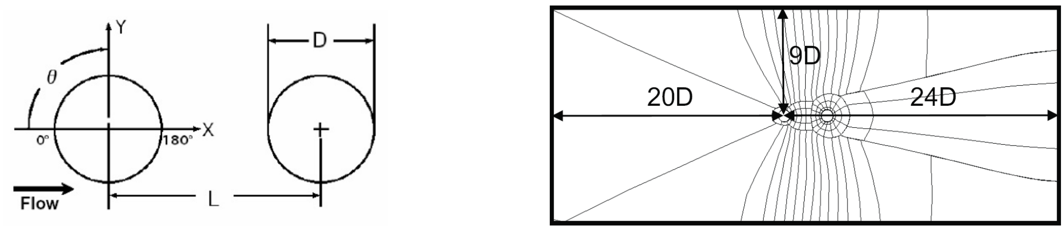
The data of the simulations are compared to the experimental data, therefore the experimental setup must be explained.

2.1 Experimental Setup and Data

The experiments were performed in the Basic Aerodynamic Research Tunnel (BART) at the NASA-Langley Research Institute. It is a subsonic, atmospheric wind tunnel used to investigate the fundamental characteristics of complex flow fields and to acquire data for CFD-validation and development. The free-stream turbulence is less than $0,10\%$. To create a fully turbulent boundary layer and assure a post-critical flow separation, transition strips were attached along the entire span of the upstream cylinder. Measurements have proven the effectiveness of the transition strips. The c_p-distributions of a cylinder with strips were compared to data of a cylinder without strips acquired at the post-critical Reynolds number of $Re = 8,6 \cdot 10^6$ [4].
Figure 1 (a) shows the configuration of the cylinders. They are placed in streamwise direction with a distance between their centers of $L = 3,7D$. The spanwise length of the cylinders is $L_z = 16D$. The field data was obtained using 2D-PIV, while the surface data was obtained via static pressure orifices and piezoelectric resistive transducers.
To be able to compare the results of the experiment with the simulations, the boundary layer is assumed to be fully turbulent on the whole surface and the wind tunnel is assumed to be periodic in spanwise direction. This enables the comparison with the high L_z/D-ratio used in the experiment, without the need of an inmense number of points in the third direction.

2.2 Numerical Setup

For the ATAAC-project it is really important to be able to compare the results between all the members, so the Mesh was provided in 2D by New Technologies and Services (NTS), St. Petersburg, Russia. To ensure the same boundary conditions as in the experiments, the inlet was set to $20D$, the outlet to $24D$ and the walls to $9D$ from the center of the first cylinder, as seen in Figure 1 (b).
The 2D-Mesh was extruded in the third dimension with the proposed $L_z = 3D$ span-size and the boundary is given a periodic condition. The definitive grid-size is about 16 million points.
The time interval for saving unsteady data should be able to allow resolutions of frequencies up to a Strouhal number $Sr = 0.2$ and should not be larger than $0.075D/U_\infty$. In this case a resolution of the frequency of $7,5$ points was chosen, so a time step of $\Delta t = 0.01$ is used for the calculations. This time step will capture the most important turbulent scales and allow a stable and fast calculation.



(a) Tandem Cylinder Configuration (b) Calculation field with blocking

Fig. 1 – Setup : Configuration and Blocking

3 Numerical Method

The Navier-Stokes-Multi-Block (NSMB) code solves the compressible Navier-Stokes equations using a finite volume formulation on Multi-Block structured grids. Various spatial discretization schemes are available like Jameson's central difference, Roe or AUSM+. The time integration is based on the full matrix implicit LU-SGS (Lower-Upper Symmetric Gauss-Seidel) method and the dual-time stepping. In the present work the artificial compressibility method is employed, as the inflow Machnumber $M_\infty = 0.12$ is lower than $M = 0.3$ and must therefore be seen as a incompressible fluid. NSMB is parallelized using the Message Passing Interface.The governing equations are the preconditioned unsteady Navier-Stokes equation written in conservative form with dual-time stepping.
The simulations were performed at the french High-Computing-Centre CINES (Centre Informatique National de l'Enseignement Supérieur) in Montpellier with the structured, compressible, finite-volume

solver NSMB (Navier-Stokes-Multiblock) on 512 processors with a random access memory of 8GB per processor.

4 Turbulence Modelling

In the context of standard URANS turbulence modellig two eddy-viscosity models are compared in this study, the k-ω-Baseline and the k-ω-SST (Shear-Stress-Tensor) models proposed by Menter [5]. In the SST model an eddy-viscosity limiter was introduced, originally derived for steady state flows, to prevent an overprediction of the ratio $\frac{-\langle u'v' \rangle}{k}$.

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, \Omega F_2)} \tag{1}$$

in order to ensure that $-\langle u'v' \rangle = 2a_1 k$ on highly sheared regions. The constants used in this work are the ones given by the Organised Eddy Simulation (see section 4.1) and/or the ones proposed by Menter [5].
In the context of capturing non-equilibrium turbulence effects in unsteady flows by distinction of the coherent scales from the chaotic turbulence, the Organized Eddy Simulation (OES) approach is introduced.

4.1 Organisd Eddy Simulation : OES

The Organised Eddy Simulation is an Advanced URANS modelling approach, that aims at capturing non-equilibrium turbulence effects especially in separated flows. This approach distinguishes the turbulent structures to be resolved from those to be modelled not by their size -like the Large Eddy Simulation (LES)-, but by their physical nature - organised or chaotic. In the present class of inhomogeneous turbulent flow there are strong non-equilibrium turbulence effects, which cause shape and slope modification in the inertial range of the turbulence energy spectrum. As production is not equal to dissipation, the organised part of the motion corresponds to distiguished peaks in the spectrum that corresponds to the resolved turbulence. The periodic nature of the stalled flow behind a cylinder allows the definition of phase-averaged quantities, so the flow is decomposed into a periodic component and a random fluctuation : $< U_i >= \overline{U_i} + \tilde{U_i}$. The phase-averaged Navier-Stokes equations are solved. It was shown by means of the DRSM modelling [6, 7] that an advanced eddy-diffusion coefficient C_μ can be derived from the second-order modelling and suggested values were of order $0,02 - 0,04$. Furthermore, an anisotropic OES approach provides a tensorial eddy-diffusion coefficient, also derived by projection of the DRSM on basis of the principal direction of the strain rate [8]. In the present study the simplified OES-approach has been used, adapting a constant C_μ-coefficient.

4.2 Detached Eddy Simulation : DES

The Detached Eddy Simulation is a three-dimensional simulation that uses the statistical turbulence modelling equations, switching the turbulence length scale between its RANS expression and the subgrid length scale as in LES [9, 10]. So a RANS model is chosen for the near-wall region and the LES approach is chosen for the regions of flow detachment. This switch between the length scales allows the use of a single set of turbulence modelling equations and does not require the definition of a spatial interface between RANS and LES regions. The switch for the length scale is defined in the following equation : $\tilde{d} = \min(d, C_{DES}\Delta)$, where C_{DES} is the DES constant, calibrated by means of the homogeneous, isotropic turbulence spectrum, Δ is defined as $\Delta = \max(\Delta x, \Delta y, \Delta z)$, which is the largest dimension of the elementary control volume cell and d is the URANS length scale. For a two-equation model, like the $k - \omega$ model, the distance is defined as $d = \sqrt{k}/\beta\omega$.

4.3 Delayed Detached Eddy Simulation : DDES

The DES definition of the length scale can bring a transition from URANS to LES within the boundary layer which would produce non-physical artefacts. Therefore a new definition of the turbulence length scale was introduced with a near-wall damping function : $\tilde{d} = d - f_d \cdot \max(0, d - C_{DES}\Delta)$, with f_d

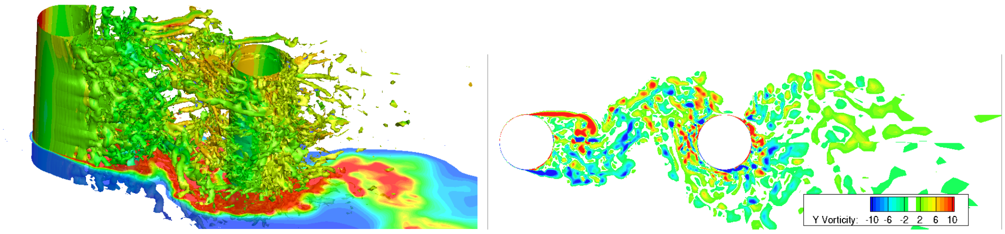
as the damping function, which is 1 away from the wall and 0 in the near-wall regions. So, if $f_d = 0$, $\tilde{d} = d$, which yields to RANS modelling and if $f_d = 1$, $\tilde{d} = \min(d, C_{DES}\Delta)$, which yields the classical DES modelling approach.

4.4 DDES-OES

In order to higher the turbulence intensity of the flow, to simulate the high-Reynolds effects created by the transition strips in the experiment a new definition for the turbulence length scale and the turbulent viscosity was introduced. The length scale used in these calculations is the OES one l_{OES} instead of the RANS one l_{RANS}. The turbulent viscosity is calculated by means of the SST and the OES definition : $\nu_t = \min(\nu_t^{SST}, \nu_t^{OES})$.

5 Discussion of Results

The flow field is divided into the two regions, which show the most important physical phenomena, such as flow separation and reattachment. The first region discussed in this paper is the gap region. It consists of the flow between the two cylinders. Figure 5 shows a good comparison between the experiment and the simulation. The high 3D effects can be seen in Figure 2 (a).

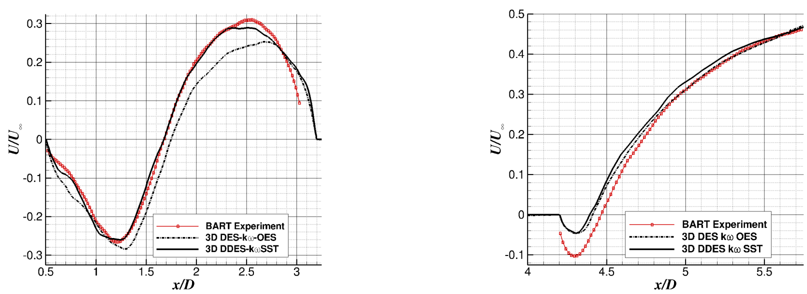


(a) Isovorticity Magnitude 12 (b) Spanwise Vorticity

Fig. 2 – 3D DDES-k-ω-SST-OES

5.1 Gap Region

The flow in the gap region is of great physical importance, as it determines the behaviour of the flow around the second cylinder. Figure 5 shows mean velocity field around the cylinders with the streamlines. The counterrotating vortices indicate the separation of the boundary layer at the first cylinder. The reattachment point of the shear layer at around $x/D = 1.6$ is at a similar location as in the experiments. This is also seen in Figure 3 (a) in the velocity profiles of the flow in the middle plane between the two cylinders. A good concordance between the simulations and the experiment, shows the good abilities of the hybrid approaches to predict difficult flows.

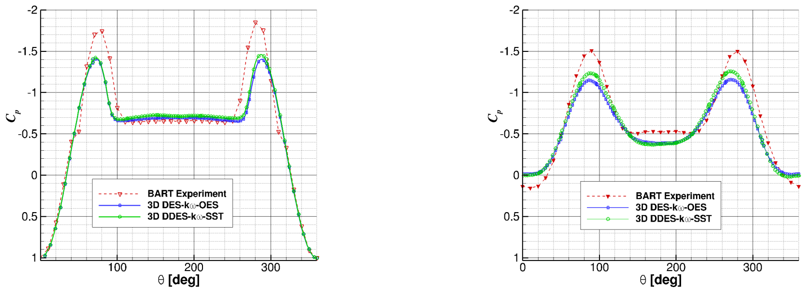


(a) Front Cylinder (b) Rear Cylinder

Fig. 3 – Mean Velocity Profiles

5.2 Near-Wake Region

The second region discussed is the near-wake region behind the second cylinder. It consists of the flow just after the second cylinder. Figure 5 illustrates that the flow around the second cylinder is governed



(a) Front Cylinder (b) Rear Cylinder

FIG. 4 – Mean Pressure on Tandem Cylinders

by the gap flow. It is seen, that the strong turbulence caused by the detachment and shear layer flow at the first cylinder highers the turbulence level in the boundary layer of the second cylinder. This is seen in the later detachment and the earlier reattachment of the shear layer compared to the first cylinder. Figure 3 (b) shows an earlier reattachment of the flow for the simulations in comparison to the experiment. This is given by the higher turbulent intensity created by the earlier detachment of the boundary layer in the first cylinder (see Figure 4 (a)) compared to the experiments. The later detachment and the higher turbulence level are also seen in Figure 4 (b) were the pressure recovery for the second cylinder is higher for the simulations than for the experiments.

5.3 Experiment - Simulation

The differences between the experiment and the simulations are probably due to the transition strips used in the experiments. The flow before the transition strip is laminar/transitional and the transition strips invoke a higher turbulence not only in the boundary layer, but also in the outer flow. This means that the flow will have supercritical-like behaviour, but the turbulence invoked by the turbulence modelling will still be less than that of the experiment. Even though the turbulence is higher and, therefore, the momentum transfer perpendicular to the mean flow direction is much higher, the boundary layer of the simulation separates earlier.

A turbulent boundary layer is known to grow faster than a laminar/transitional one and a thick boundary layer will detach earlier than a thin one, as its inertial energy will be of a lesser magnitude. In this comparison, this is due to the longer running length of the turbulent boundary layer in the simulation, as the first 50° of the experiment show a laminar/transitional flow regime. The earlier

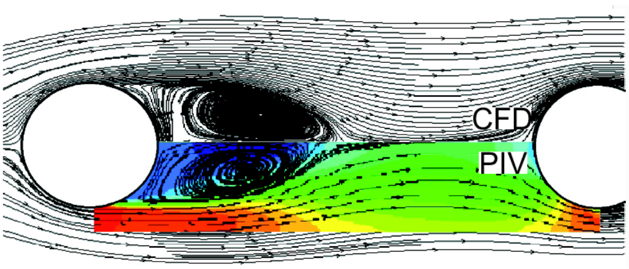


FIG. 5 – Mean Streamlines - Experiment and 3D DDES-k-ω-SST-OES

separation causes a higher turbulent intensity between the cylinders, which highers the turbulence in the boundary layer of the second cylinder and brings it to the later detachment and the earlier reattachment of the shear layer, having the same running length of the turbulent boundary layer, as both (experiment and simulation) show a strong turbulence before the second cylinder.

6 Conclusion

In the present study, it has been shown that the hybrid approach DDES-k-ω-SST-OES gives a satisfactory prediction of the thin shear layers downstream of the separation, of the Kelvin-Helmholtz vortices in the separated shear layers and of the wrap-around vortex process regarding the second cylinder. The exisiting discrepancies in the c_p-coefficient have to be investigated in a future study by ensuring an effectively high turbulence level arising in the upstream boundary layer, in order to imitate the action of the strips. Furthermore, as an outlook, computations by taking into account transition will be conducted.

For the most matching simulation, more monitoring points will be included in the field to extract the most important frequencies to be able to make a spectral analysis of the noise levels of the tandem cylinders and compare these with the experiments performed at the Quiet Flow Facility (QFF) at the NASA-Langley Research Center.

7 Acknowledgments

The authors want to strongly express their gratitude to the High-Computing facility Centre Informatique National de l'Enseignement Supérieur (CINES) in Montpellier for hosting the vast computations and to Dr. Hilde Ouvrard as a contact person for all her help. Part of this work has been carried out in the context of the national ANR research program "ECINADS".

Références

[1] L. N. Jenkins, M. R. Khorrami, M. M. Choudhari and C. B. McGinley : Characterization of unsteady flow structures around tandem cylinders for component interaction studies in airframe noise. *AIAA 2005-2812, 2005*

[2] L. N. Jenkins, D. H. Neuhart, C. B. McGinley, M. M. Choudhari and M. R. Khorrami : Measurements of Unsteady Wake Interference Between Tandem Cylinders. *AIAA-2006-3202, 2006, 36th AIAA Fluid Dynamics Conference and Exhibit, 5-6 Jun. 2006, San Francisco*

[3] M. R. Khorrami, D. P. Lockhard, M. M. Choudhari, L. N. Jenkins, D. H. Neuhart and C. B. McGinley : Simulations of bluff body flow interaction for noise source modeling. *AIAA-2006-3202, 2006*

[4] A. Roshko : Experiments on the flow past a circular cylinder at very high Reynolds number. *Journal of Fluid Mechanics, Vol. 10, Issue 3, 345-356, 1961*

[5] F. R. Menter : Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA-Journal, 32(8) :1598-1605, August 1994*

[6] Y. Hoarau : Analyse Physique par Simulation Numérique et Modélisation des Écoulements Décollés Instationnaires autour des Surfaces Portantes. *Thése Doctorale, Institut National Polytechnique de Toulouse, 2002*

[7] M. Braza, R. Perrin, Y. Hoarau : Turbulence properties in the cylinder wake at high Reynolds numbers. *Journal of Fluids and Structures 22, 757-771, 2006*

[8] R. Bourguet, M. Braza, G. Harran and R. El Akoury : Anisotropic Organised Eddy Simulation for the prediction of non-equilibrium turbulent flows around bodies. *Journal of Fluids and Structures, 24 :8, 1240-1251, 2008*

[9] A. Travin, M. Shur, M. Strelets, P. Spalart : Detached-eddy simulations past a circular cylinder. *Flow, Turbulence and Combustion 63, 293-313, 2000*

[10] C. Mockett, R. Perrin, T. Reimann, M. Braza and F. Thiele : Analysis of detached-eddy simulation for the flow around a circular cylinder with reference to PIV data. *Flow, Turbulence and Combustion, Volumen 85, Number 2, 167-180, 2009*