Actuator line method applied to grid turbulence generation for large-eddy simulations

Félix Houtin-Mongrolle, Laurent Bricteux, Pierre Benard, Ghislain Lartigue, Vincent Moureau, Julien Reveillon

To cite this version:
Félix Houtin-Mongrolle, Laurent Bricteux, Pierre Benard, Ghislain Lartigue, Vincent Moureau, et al.. Actuator line method applied to grid turbulence generation for large-eddy simulations. ERCOFTAC WORKSHOP DIRECT AND LARGE EDDY SIMULATION 12 (DLES12), Jun 2019, Madrid, Spain. hal-02149266

HAL Id: hal-02149266
https://hal.archives-ouvertes.fr/hal-02149266
Submitted on 11 Feb 2021
ACTUATOR LINE METHOD APPLIED TO GRID TURBULENCE GENERATION FOR LARGE-EDDY SIMULATIONS

F. Houtin–Mongrolle¹, L. Bricteux², P. Benard⁴, G. Lartigue¹, V. Moureau¹, J. Reveillon¹
¹ CORIA, CNRS UMR6614, Normandie Université, INSA and University of Rouen, 76801 Saint-Etienne-du-Rouvray, France
felix.houtin-mongrolle@coria.fr
² Université de Mons (UMONS), Polytechnic Faculty, Belgium

INTRODUCTION

The constant growth of computational resources allows nowadays to perform Large-Eddy Simulation (LES) on realistic flow configurations. In this framework, it is important to properly model the turbulence inflow at a low computational cost. This study addresses wind tunnel applications where the object under investigation is downstream of a turbulence grid. This grid aims to generate a highly turbulent flow and a sheared velocity profile for further use in wind turbines applications. In this work, an original strategy based on the Actuator Line Method (ALM) is proposed to emulate the effect of wind turbines blades on a flow. Indeed, this method allows to model the influence of an object on a flow and a sheared velocity profile for further use in wind tunnels applications. The proposed method is compared to usual turbulence injection strategies as well as a geometrical resolution of the turbulence generation grid [1].

ACTUATOR LINE METHOD

Definition

In the domain of wind energy, the ALM is commonly used to model the effect of wind turbines blades on a flow. Indeed, this method allows to model the influence of an object on a flow by imposing body forces distributed along the geometry. These forces are computed from the incident wind and the lift and drag aerodynamics coefficients. The overall strategy avoids handling complex and/or moving geometry. Following the formalism used in [2], the local lift and drag aerodynamics coefficients. The overall strategy avoids handling complex and/or moving geometry. Following the formalism used in [2], the local lift and drag forces experienced by each element of the grid are computed as:

\[ L = \frac{1}{2} \rho v_{rel}^2 l w C_L(\alpha), \quad (1) \]

\[ D = \frac{1}{2} \rho v_{rel}^2 l w C_D(\alpha), \quad (2) \]

where \( C_L \) and \( C_D \) are the lift and drag coefficients, \( \alpha \) is the local angle of attack, \( v_{rel} \) is the local velocity relative to the geometry, \( l \) is a local length, i.e. the chord of the emulated profile, \( w \) is the actuator element width, i.e. the distance between two actuator elements.

The body force is then projected on the Eulerian grid using a mollifying function \( \eta_e \) (see e.g. [2, 3]), defined as:

\[ \eta_e(d) = \frac{1}{\varepsilon^3 \pi^{3/2}} \exp \left[ -\left( \frac{d}{\varepsilon} \right)^2 \right], \quad (3) \]

with \( d \) being the distance between a grid node and the element position and \( \varepsilon \) a mollifier width parameter set from an estimation of the local cell size \( \Delta x \) such as \( \varepsilon = 2 \Delta x \). Thus, it provides a smooth force \( f \) distributed on the grid obtained from the forces concentrated on a given element. Finally, the body force source term \( f \) in the momentum equation can be expressed as:

\[ f(x, y, z, t) = \sum_{e=1}^{N} \left( L(t) e_x + D(t) e_z \right) \eta_e(d). \quad (4) \]

This methodology is implemented into the YALES2 flow solver [7]. YALES2 is a low Mach-number massively-parallel finite-volume LES solver using 4th-order numerical schemes on unstructured meshes. In this study, the subgrid Reynolds stress tensor is modeled using the \( \sigma \)-model [4].

Parametrization for wind tunnel turbulence grid

This study focuses on turbulence generation based on a square cylinder grid. Different experimental configurations (see e.g. [5] and [6]) showed that the aerodynamic forces of a static cylinder depends on the rod-to-rod spacing, on the rods-to-wall distance and fluctuates at a given frequency \( f \) directly linked to the Strouhal number \( St \) as \( f = St v_{rel}/l \), where \( v_{rel} \) is the relative velocity and \( l \) the local length. Therefore, a fluctuation on the Drag and Lift coefficient is added to the previous formalism as pictured in Fig. 1 and varying in time \( t \) as (equation only given for lift force coefficient):

\[ C_L(t) = \langle C_L \rangle + C_{L \mathrm{rms}} \sqrt{2} \sin \left( 2 \pi f t + \varphi \right), \quad (5) \]

where \( \langle C_L \rangle \) is the mean lift coefficient, \( C_{L \mathrm{rms}} \) its fluctuating...
amplitude and $\varphi$ is a random phase different on each rod. The aim of this random phase is to de-synchronize the vortex shedding of the rods and create a fully turbulent flow.

**APPLICATION TO GRID TURBULENCE GENERATION IN A WIND TUNNEL**

The method proposed here is applied to the turbulence generation system of the “NTNU Blind Test 5” [8] wind tunnel. Its dimensions are (see Fig. 2): $L_x \times L_y \times L_z = 14 \times 2.71 \times 1.801 \text{ m}^3$. The turbulence grid is composed of $47 \times 47 \text{ mm}^2$ square cylinder rods as described in [9]. The grid presents a constant spacing in the horizontal direction 0.24 m and the horizontal rods are arranged with an increasing spacing from the floor to the roof (see [9] for the detailed values).

This aims to generate a sheared flow.

The aim of this work is to assess the ALM (case B) and compare it to other strategies: a resolved flow around the rods (case A) and a Homogeneous Isotropic Turbulence (HIT) injection with sheared velocity injection (case C). Each of these methods requires different mesh sizes: for case A, the mesh size around the rods need to be fine enough to capture the boundary layer. This significantly increases the number of elements in the grid with a total of around $26 \times 10^6$ tetrahedra for case A. The two other methods do not require such a fine meshing with approximately $20 \times 10^6$ elements.

The preliminary results are displayed in Fig. 3, representing the vorticity field in mid-$xz$ plane. First, flow dynamics between the grid and $x_1$-plane measurement seem different between the three cases: the coherent structures destabilization length is longer for case B compared to case A. After the $x_1$-plane, the ALM case results appear in accordance with the rods-resolved case with similar turbulence dynamics. The case with HIT injection provides modest performances.

This analysis is confirmed thanks to Fig. 4 plotting the time-averaged axial velocity and the turbulence intensity at $x_1=0$ position. As expected on velocity, case C perfectly retrieve its imposed shear velocity profile while cases A and B differs up to 10% compared to the experiment. For each approach, a turbulence intensity flat profile is retrieved with a satisfying agreement on the amplitude. All these results are still under investigation with a specific attention on computing performances.

**CONCLUSIONS AND PERSPECTIVES**

An original method to emulate grid turbulence generation has been proposed and assessed using the “NTNU blind test 5” wind tunnel experiment. The preliminary results presented in this extended abstract are promising: the method allows to produce turbulence that fairly reproduces the characteristics of the turbulent flow. ALM and THI parameters still need to be adjusted and a further analysis will be conducted on different mesh refinement levels.

**REFERENCES**


