Proposal 2016-2017

Topic #3 - Investigation of unsteady aerodynamic loading of a bluff-body: CFD simulations

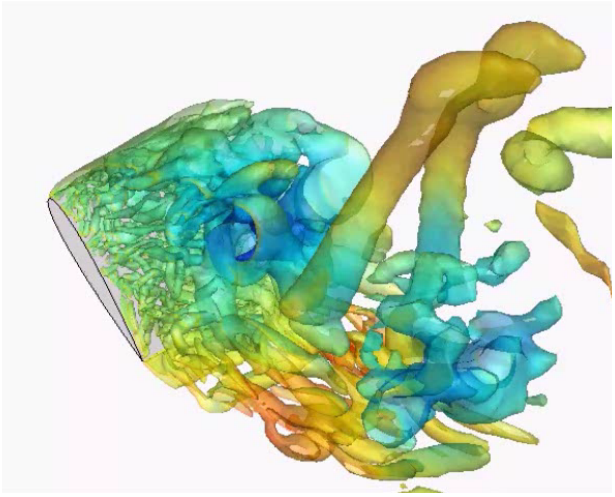
Contacts

ULg / MTFC Vincent Terrapon (vincent.terrapon@ulg.ac.be)

ULg / WTL Thomas Andrianne (t.andrianne@ulg.ac.be)

Context

Computational Fluid Dynamic (CFD) simulations represent an invaluable tool to study aerodynamic phenomena. But because the Reynolds number of typical aerodynamic applications is large, the flow is generally turbulent. Solving all the details of a turbulent flow is usually computationally too costly. An alternative approach is to directly compute the mean flow by solving the Reynolds Averaged Navier-Stokes equations (RANS). Although RANS simulations provide relatively accurate results for attached flows, they are not reliable for separated flows that are observed behind bluff bodies. A better approach is to simulate not only the mean flow, but also the largest dynamically significant scales of the turbulence (see Figure), for instance through Detached Eddy Simulations (DES). This method provides more accurate results for detached flows, but is at the same time more expensive than RANS.



Both RANS and DES simulations require a turbulence model, which introduces uncertainties. Therefore, it is critical to compare numerical results with experimental measurements. The Wind Tunnel Laboratory is equipped with a dynamic pressure measurement system that is well adapted to measure unsteady pressure distributions on the surface of a wind tunnel model. This spatio-temporal pressure field can be compared with unsteady CFD results. Decomposition techniques can then be used to retain the main part of the experimental and numerical results and to compare them adequately.

Objectives

The objective of this project is to first perform unsteady RANS (URANS) and DES simulations of the flow around rectangular bluff-bodies in a uniform flow, and then to compare the two numerical approaches with experimental data.

Once validated, the CFD models will be used to investigate the effect of the aspect ratio of the rectangular cylinders and of the free stream turbulence level on flow separation and reattachment.

Wind tunnel measurements are not part of the present work. They will be provided from previous wind tunnel test campaigns and from another master thesis project carried out in parallel. For this reason, a strong interaction between the two projects is expected to optimally use the complementarity of the two approaches.

The project consists in:

- Literature review about bluff-body aerodynamics and CFD models
- Development of a parametric 3D CFD model capable to perform incompressible unsteady RANS and DES simulations in OpenFOAM
- Validation of the CFD results through comparisons with unsteady pressure measurements
- Analysis of the effect of the aspect ratio of the bluff-body on the flow and aerodynamic loading
- Analysis of the effect of the level of the incoming free stream turbulence on the flow and aerodynamic loading

In addition to the Master thesis report, it is expected that the results be summarized in a publication that could be submitted to a conference or to an international journal, such as Journal of Fluids and Structures[1].

[1] http://www.journals.elsevier.com/journal-of-fluids-and-structures/