11th. World Congress on Computational Mechanics (WCCM XI)
5th. European Conference on Computational Mechanics (ECCM V)
6th. European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

ON THE MODELLING OF TURBULENT FLUID-STRUCTURE INTERACTION. APPLICATION TO A CHANNEL FLOW AROUND A CANTILEVER PLATE ATTACHED BEHIND A CIRCULAR CYLINDER

Olga Estruch[1], Oriol Lehmkuhl[1,2], Joaquim Rigola[1], Carles D. Pérez-Segarra[1] and Assensi Oliva[1]

[1] Heat and Mass Transfer Technological Center, Polytechnical University of Catalonia, ETSEIAT, Colom 11, 08222 Terrassa, Barcelona, Spain, cttc@cttc.upc.edu, http://www.cttc.upc.edu
[2] Termo Fluids S.L., Magí i Colet 8, 08204 Sabadell, Barcelona, Spain

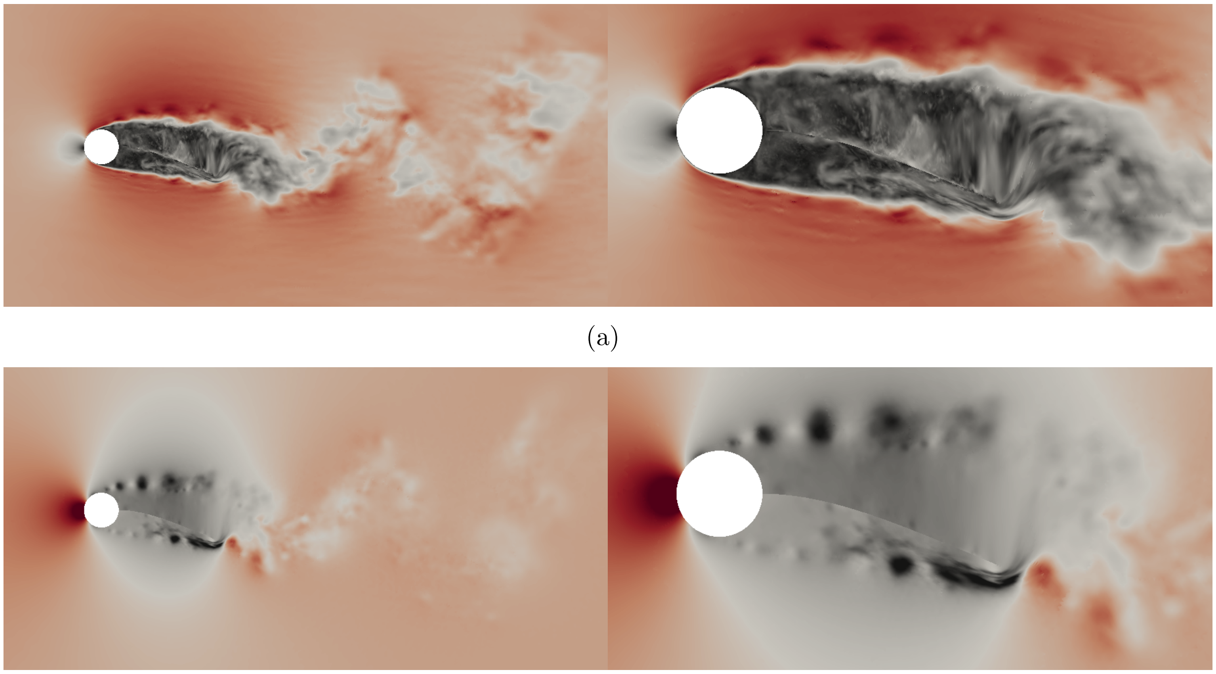
Key words: *Fluid-Structure Interaction, Dynamic Mesh, Radial Basis Function, Large-Eddy Simulation, Modal Analysis.*

Fluid-structure interactions (FSI) arise when the motion of a structure is instigated by a fluid flow and vice-versa, the flow field behavior is affected by the deformation of the structure. This type of coupling can be encountered in a great amount of industrial applications; hence, the understanding of the fluid flow behavior or the awareness of the dynamic action of the structure can be essential to improve engineering designs and optimisations. Therefore, accompanied by a great enhance in computational resources, in the last years the numerical simulation has become an important tool to predict FSI applications.

In order to contribute to the FSI field, the present work attempts the validation of the newly in-house implemented partitioned coupled algorithm focused on the study of the turbulent flow phenomena interacting with a flexible reed type structure. The benchmark case selected consists of the turbulent channel flow around an elastic thin structure clamped behind a fixed rigid non-rotational cylinder proposed in [1]. Several works can be found in the literature which deal with the numerical simulation of this type of application involving laminar flows, but for turbulent flows the sources are still scarce.

The in-house CFD&HT solver [2] is based on a three-dimensional explicit finite volume fractional-step algorithm formulated in a second-order, conservative and collocated unstructured grid arrangement. Large-eddy simulation is used to model the turbulence flow by means of the subgrid scale WALE model [3]. The dynamic action of the structure has been modelled by a specific law according modal analysis of reed theory [4], which provides a fast transient prediction of the solid deformation. Referring to the dynamic mesh procedure, a radial basis function (RBF) interpolation strategy is used [5].

In the final paper the detailed results will be presented and contrasted with the benchmark reference. For instance, in Figure 1 the numerical instantaneous velocity and pressure fields obtained are shown.



(a)

(b)

Figure 1: Instantaneous (a) velocity and (b) pressure field.

REFERENCES

[1] G. De Nayer, A. Kalmbach, M. Breuer, S. Sicklinger, R. Wüchner. A fluid-structure interaction benchmark in turbulent flow (FSI-PfS-1)–a complementary numerical/experimental investigation. *J. of Fluids and Struct.*, submitted for publication.

[2] O. Lehmkuhl, C.D. Pérez Segarra, R. Borrell, M. Sòria, A. Oliva. TERMOFLUIDS: A new parallel unstructured CFD code for the simulation of turbulent industrial problems on low cost PC cluster. In Proc. of the *Parallel CFD 2007 Conf.*, 1–8, 2007.

[3] F. Nicoud and F. Ducros. Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion*, Vol. 62, 198–200, 1999.

[4] O. Estruch, O. Lehmkuhl, J. Rigola, A. Oliva, C.D. Pérez-Segarra. Transient and dynamic numerical simulation of the fluid flow through valves based on large eddy simulation models. In Proc. of the *8th International Conf. on Compressors and their Systems*, London, 2013.

[5] O. Estruch, O. Lehmkuhl, R. Borrell, C.D. Pérez-Segarra, A. Oliva. A parallel radial basis function interpolation method for unstructured dynamics meshes. *Comp.&Fluids*, Vol. **80**, 44–54, 2013.