Available online at www.sciencedirect.com

ScienceDirect

Procedia Structural Integrity 8 (2018) 422–432

Structural Integrity Procedia

www.elsevier.com/locate/procedia

AIAS 2017 International Conference on Stress Analysis, AIAS 2017, 6–9 September 2017, Pisa, Italy

Fluid structure interaction analysis: vortex shedding induced vibrations

N. Di Domenico[a], C. Groth[a], A. Wade[b], T. Berg[c], M.E. Biancolini[a,*]

[a] Department of Enterprise Engineering - University of Rome Tor Vergata, Via del Politecnico, 1, 00133, Rome, Italy
[b] ANSYS UK Ltd, Sheffield Business Park, 6 Europa View, Sheffield, S9 1XH
[c] ANSYS Sweden AB, Vestagatan 2B, 416 64 Göteborg, Sweden

Abstract

Fluid Structure Interaction (FSI) numerical modelling requires an efficient workflow to properly capture the physics involved. Computational Structural Mechanics (CSM) and Computation Fluid Dynamics (CFD) have to be coupled and at the moment there is a lack of monolithic solvers capable to tackle industrial applications that involves high fidelity models which mesh can be comprised of hundred millions of cells. This paper shows an efficient approach based on standard commercial tools. The FEM solver ANSYS® Mechanical™ is used to extract a given number of eigenmodes. Then the modal shapes are imported in the CFD solver Fluent® using the Add On RBF Morph™. Updating the modal coordinates it is possible to adapt the shape of the model by taking into account the elasticity of the CFD model. Transient analysis is faced using a time marching solution by updating the shape of the mesh at each time step (weak coupling, evaluated as single DOF systems and integrating modal forces over the CFD grid). Numerical performances and solution accuracy of this approach are analyzed on a practical application (NACA0009 Hydrofoil) for which experimental data are available. A comparison between proposed method and experiment is provided. Transient coupled solver is used for the computation of eigenvalues in water by post processing the free vibration response in calm fluid.

Peer-review under responsibility of the Scientific Committee of AIAS 2017 International Conference on Stress Analysis

Keywords: Radial Basis Functions; RBF; Fluid Structure Interaction; FSI; Modal Superposition; Vortex shedding; vibrations

1. Introduction

Fluid structure interaction (FSI) is a very interesting topic, subject of ongoing investigations in the scientific community. Given its intrinsic complexity, this multiphysics phenomenon introduces an high level of complication in the analysis methods especially when using an high fidelity approach; for this reason, when possible, the structural analyst often underestimates the pressure variations inside or outside structural domains, substituting the action of the evolving flow with a constant wall loading. In a similar manner the CFD analyst, if acceptable, doesn't takes into

* Corresponding author. Tel.: +39 0672597124.
E-mail address: biancolini@ing.uniroma2.it

Peer-review under responsibility of the Scientific Committee of AIAS 2017 International Conference on Stress Analysis
10.1016/j.prostr.2017.12.042

consideration structural elasticity, employing as boundaries rigid walls. There are however applications in which these simplifying assumptions can't be made and in which both the physics ruling the problem must be faced. The system behavior is in these cases governed by the interaction between fluid and structure.

The interaction can be the working principle of the component itself (reed valves action, parachute canopy unfolding, movement of a sheet of paper within a printing device and more); can be due to the lightweight design of the structure (aircraft design); or can be exploited to finely tune the design taking advantage of interaction (Formula 1 wings). The ability to provide the analyst with advanced and accurate tools able to correctly reproduce this multidisciplinary phenomenon allows to accurately predict the behavior of existing systems but also to design advanced and higher performance products. For this reason many efforts have been made in the last decades on this topic and several approaches are available in literature to tackle the problem.

The most common high fidelity approach to solve FSI is the 2-way partitioned method, that foresees a mutual interaction between the structural finite element method (FEM) and the computational fluid dynamics (CFD) codes. This coupling is complex from both the numerical and the operative points of view, being solver-dependent and requiring several data exchanges between them, making this method computationally heavy and hard to be set-up. CFD loads are first extracted from the fluid dynamics numerical grid and transferred to the FEM code where the structural response is evaluated; deformations are then synchronized back to the CFD code in order to measure the introduced flow variation and the iteration carried out again until a certain convergence criteria (i.e: displacement or flow variation) is met. While well validated in literature this workflow poses however several problems. Generally speaking structural and fluid dynamical meshes are indeed not matching, requiring different grid discretizations to correctly catch the different nuances given by the physics involved; in order to transfer the loads between the different grids a mathematically complex mapping algorithm is required, employed also to transfer back structural displacements. Several actions are moreover required by the workflow including I/O, results checking and format conversion. Last but not least an appropriate algorithm must be used to deform the CFD grid, propagating the displacements known at boundaries into the volume. This task is challenging and crucial, requiring an algorithm that is robust, efficient and accurate.

In the 2-way partitioned coupling data transfers can be a bottleneck, since mapping and mesh deformation are required at each iteration. This difficulty, encountered in steady state problems, is felt much more when dealing with transient simulations, in which the data transfer and the mesh update must be accomplished continuously, typically at each time step when employing a weak coupling or at each inner iteration for strong coupling (Benra et al. (2011)).

At the moment there is a lack of monolithic solvers capable to tackle industrial applications involving high fidelity models comprised of hundred millions of cells.

In this paper an efficient approach based on the modal theory, able to tackle transient FSI simulations with industrial meshes and employing standard commercial tools, is shown. Imagining the structural deformation as a linear superimposition of its modal shapes, an eigenvalue analysis is carried in ANSYS® Mechanical™ in order to extract a given number of eigenmodes. Data exchange, following this approach, is required only once at initialization. Modal shapes are imported in the CFD solver ANSYS® Fluent® using the Add On RBF Morph™, a commercial mesh morphing tool based on Radial Basis Functions (RBF), making the numerical grid implicitly elastic and deforming the numerical grid at each time step. By embedding the modes in the CFD solver no data exchange is required and mesh update can be efficiently accomplished for each time step amplifying linearly the modal shapes stored in memory.

RBF proved to be a powerful mesh morphing tool (De Boer et al. (2007)) and have been successfully applied in several fields of research also for geometrical modeling (Kojekine et al. (2003), Reuter et al. (2003)), shape optimization (Cella et al. (2017), Biancolini et al. (2014)) and adjoint data filtering (Groth (2015)) among others.

For FSI applications RBF have been applied using Reduced Order Methods (Castronovo et al. (2017)). For static analyses was also employed the 2-way (Cella and Biancolini (2012), Keye (2009)) and the modal superposition approach (Biancolini et al. (2016), Andrejašič et al. (2016)). A notable example of modal embedding for transient FSI analyses using RBF is demonstrated in literature by Van Zuijlen et al. (2007) on the AGARD 445.6 wing.

In the present work the mesh update task is accomplished using the commercial morpher RBF-Morph (RBF-Morph (2016)). It was first developed as an on-demand module for a Formula 1 top team and then placed on the market as an add-on for the CFD solver ANSYS Fluent (Biancolini (2010)). The code proved its efficiency in several aerospace (Biancolini and Cella (2010); Biancolini and Groth (2014)) and non aerospace applications (Biancolini et al. (2014)). A comprehensive description of the theory behind the tools and their applications is given in (Biancolini (2012)).