6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11-15 June 2018, Glasgow, UK

Fluid Structure Interaction Problems with CIRA Structured CFD solver

D. Cinquegrana[1], P.L. Vitagliano[2]

[1] CFD Lab, Cira SCpA, via Maiorise snc 80143 Capua (ITA), d.cinquegrana@cira.it
[2] CFD Lab, Head - Cira SCpA, via Maiorise snc 80143 Capua (ITA), p.vitagliano@cira.it

Keywords: *CFD, Multiphysics Problems, FSI, Computing Methods*

A multi-block structured flow solver[3] for unsteady RANS equations has been coupled with a structural solver within the software environment managed by the open source library preCICE[1], in order to perform fluid-structure interaction simulations.

Motivation of the work is the simulation of unsteady aerodynamic problems strongly dependent from structural behavior, like flexible aircraft, rotor-craft, counter-rotating rotors, etc.

Our CFD group has recently developed a system for flow simulation of unsteady compressible RANS equations based upon structured multi-block meshes, which allows the non-conformal block to block coupling (i.e. sliding mesh) and dynamic mesh on block base (i.e. some specific blocks in the flow field can be deformed and updated at each time step). Mesh is updated outside the flow solver, which makes possible to iterate with other systems to compute, in a segregated approach, structural deformation, body dynamics and possibly other physical phenomena.

Iterations can be performed in implicit mode, by repeating each time step with corrected meshes, in a loop controlled by convergence check outside the solver.

The flow simulation system communicates by delivering local forces on specific mesh surfaces, as specified in a set-up file, and it is capable to re-mesh the flow domain starting from a set of updated geometric entities like surfaces, curves and vertices, by following specific directives. Geometric entities can be specified or modified by control points.

Structural solver, already interfaced within the preCICE framework, is the open source FEM code Calculix[2].

Details about the implementation work and some test cases will be presented.

REFERENCES

[1] Bungartz, H.J., Lindner, F., Gatzhammer, B., Mehl, M., Scheufele, K., Shukaev, A., Uekermann, B., preCICE – A Fully Parallel Library for Multi-Physics Surface Coupling, Computers and Fluids, Vol. **141** (2016), pp 250–258

[2] G. Dhondt, The Finite Element Method for Three-Dimensional Thermomechanical Applications, Wiley (2004)

[3] Marongiu C, Catalano P, Amato M, et al. U-ZEN: a computational tool solving U-RANS equations for industrial unsteady applications. In: 34th AIAA fluid dynamics conference, Portland OR, 28 June-1 July 2004. AIAA Paper 2004-2345. https://doi.org/10.2514/6.2004-2345.