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

NUMERICAL SIMULATION OF TURBULENT FSI BENCHMARK CASES

Thorsten Reimann, Awais Ali, Dörte C. Sternel* and Michael Schäfer

Institute of Numerical Methods in Mechanical Engineering (FNB), TU Darmstadt, Germany
http://www.fnb.tu-darmstadt.de

Key words: *fluid-structure interaction, turbulence, bench mark cases*

During the last decade, great effort has been made in order to enhance coupling schemes with respect to stability and efficiency. It has become common practice to observe the progress with the help of a set of standardized FSI test cases.

We present numerical studies of turbulent FSI test cases. The coupled problem is solved with an implicit partitioned coupling approach between fluid and structural domains. The unsteady RANS equations are solved using a parallel block-structured finite-volume solver of second-order accuracy in space and time. An elliptic relaxation scheme ($\zeta - f$) is employed for model closure. To simulate the structural behaviour, a finite-element code, also of second order accuracy, is used. The non-matching grids are coupled and interpolated by the MpCCI software. Force extrapolation is used for FSI convergence acceleration. Comparison of simulation results with the benchmark experimental data show a satisfactory agreement. This work serves as the basis for FSI simulations with more advanced turbulence resolving methods (like LES or Hybrid RANS/LES), which are already in progress. Here we will discuss our findings about the suitability of the FSI simulations with present approach based on the simulated test cases.