A conservative sliding mesh coupling procedure for U-RANS flow simulations

Abstract

Purpose – Simulate unsteady flows with surfaces in relative motion using a multi-block structured flow solver.

Design/methodology/approach – A procedure for simulating unsteady flows with surfaces in relative motion was developed, based upon a structured multi-block U-RANS flow solver1. Meshes produced in zones of the flow field with different rotation speed are connected by sliding boundaries. The procedure developed guarantees that the flux conservation properties of the original scheme are maintained across the sliding boundaries during the rotation at every time step.

Findings – The solver turns out to be very efficient, allowing computation in scalar mode with single core processors as well as in parallel. It was tested by simulating the unsteady flow on a prop fan configuration with two counter-rotating rotors. The comparison of results and performances with respect to an existing commercial flow solver (unstructured) are reported.

Originality/value – This paper fulfils an identified need to allow for efficient unsteady flow computations (structured solver) with different bodies in relative motion.

Keywords CFD, Dual time stepping, Finite volumes, Multi-block structured flow solver, Polygon clipping, Open Rotor, Sliding mesh, URANS.

Paper type *Research paper*.

Introduction

Motivation for the present work is the development of a method for flow simulation around configurations with parts in relative motion, like aircraft with prop-fan, propellers, multi-stages compressor and turbines. Unsteady periodic flow can be successfully simulated by U-RANS solvers. Currently available techniques are quite expensive and demanding in terms of computational time and memory size, which is a limit to the affordable mesh dimensions and, in turn, to the accuracy of the results.