A GPU-accelerated compressible RANS solver for Fluid-Structure Interaction simulations in turbomachinery

Andrea Gadda, Department of Aerospace Sciences and Technologies, Politecnico Di Milano, Milano, Italy

Luca Mangani, Department of Fluid Mechanics and Hydraulic Machines, Lucerne University of Applied Sciences and Arts, Luzern, Switzerland

Paolo Mantegazza, Department of Aerospace Sciences and Technologies, Politecnico Di Milano, Milano, Italy

Ernesto Casartelli, Department of Fluid Mechanics and Hydraulic Machines, Lucerne University of Applied Sciences and Arts, Luzern, Switzerland

SYMPOSIA ON ROTATING MACHINERY
ISROMAC
Since 1985

Long Abstract

Introduction

Computational Fluid Dynamics (CFD) is a fundamental tool for industrial applications. The usual approach relies on the negligible structural deformation due to aerodynamic loads. In some cases, however, where power efficiency is the ultimate goal (e.g. propulsion), an accurate prediction of the structure-flow interaction is mandatory. This is particularly true for trim and flutter analysis of aircrafts, helicopter and turbomachinery blades. Turbomachinery trim and flutter predictions, in particular, still represent a challenge due to phenomena like rotor-stator interactions, separations and shock waves. The usual time-linearized, frequency-domain strategies can be inadequate when this kind of strong non-linear phenomena occur in the flow, making necessary time-domain simulations [1, 2] or the harmonic balance technique [3]. Besides flutter analysis, another important aspect, not yet adequately investigated, is the trim formulations, a fundamental aspect for an accurate steady analysis that aims to consider static blade elasticity for the performance evaluation of turbomachineries. In this paper a pioneer work describing the impact of static deformation on the performances will be provided. Alongside with accurate results, computational efficiency is indispensable. The purpose of this article is to show the architecture of a GPU-accelerated Fluid-Structure Interaction (FSI) solver for compressible viscous flows. The proposed approach is validated with typical industrial cases. The effects of trimmed solutions on the most important integral quantities (i.e. massflow, characteristic curves, mass-averaged outflow profiles) are investigated and a comparison with pure aerodynamic results is provided.

1. Methods

A compressible RANS solver is implemented and parallelized on GPUs to provide accurate results for the analysis and optimization of aerodynamic designs for internal and external flows. For what concern the aerodynamic side of the problem, an explicit time-stepping formulation is chosen to fully exploit GPU architectures [4] in hybrid unstructured meshes. Different convergence acceleration techniques and formulations aimed to reduce the domain size are combined together to further reduce the computational time [5, 6].

Aerodynamics and structural properties are combined together using a Fluid-Structure Interaction (FSI) formulation. In order to obtain an accurate and computationally efficient formulation, the structural FEM model is reduced to a modal representation [7]. This way, structural properties are recovered knowing for each of the lower frequency chosen modes, the shape, the mass, the damping and the stiffness. Obviously modes have to be computed using a FEM solver considering the influence of the rotor angular velocity, however this is done only once before the FSI simulation. Structural and aerodynamic meshes are free to be completely different in term of number and position of computational nodes. In order to achieve consistence between aerodynamic loads and structural displacements, a fluid-structure interface is implemented by mean of Radial Basis Functions (RBF) [7]. Basically the two sets of nodes are related through a matrix that is computed only one time, is then stored and finally used for all the successive trim or unsteady simulations. An Inverse Distance Weighting (IDW) [8, 7] formulation is the basis of the algorithm used to move internal domain aerodynamic nodes knowing boundary displacements, and is opportunely tuned to load the thousands of GPU cores. Finally Arbitrary Lagrangian Eulerian (ALE) and Multiple Reference of Frame (MRF) formulations are combined together to handle rotor cases with mesh deformation. In particular, the ALE framework can be used to solve steady rotor cases without actually rotating the mesh, thanks to MRF, avoiding the computational effort of updating metrics. However, when trim or unsteady analysis with mesh deformation are required for rotor cases, it is possible to deform the mesh with the modal shape only, leaving the MRF formulation the task of recovering the effects of domain rotation.

The solver is parallelized using OpenCL thanks to its capability to handle GPUs and CPUs of different vendors and architectures, enlarging the solver compatibility. Furthermore the code uses the powerful OpenFOAM API to provide pre-processing and post-processing features.

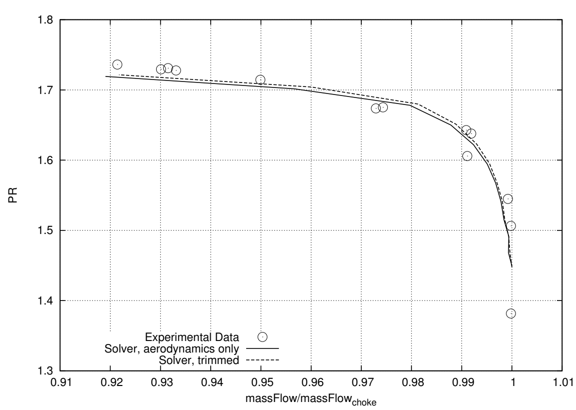
2. Results

Rotor 67 trim

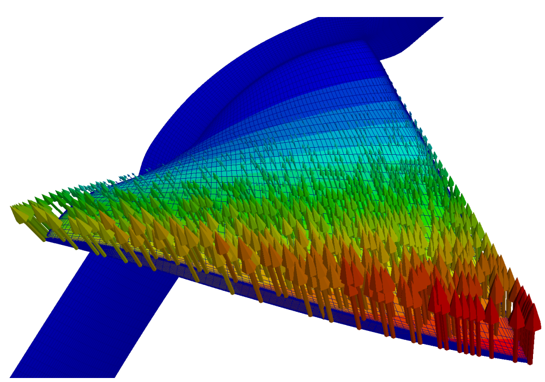
The trim analysis of the NASA rotor 67 [10] transonic fan is provided as representative of turbo propulsion applications. Preliminary tests are made on a 1’100’000 elements mesh. The structural model alongside with the first 3 modes of the rotor are computed with Code Aster and provided as input to the developed solver.

Figure 1(a) shows the differences between the pure aerodynamic solution and the trimmed solution for what concerns the total pressure characteristic curve. Of course the differences between the curves are relatively small since usually turbomachinery blades deformations are very small, unlike e.g. plane wings or helicopter blades where deformations are more appreciable. However it is possible to notice the more accurate solutions provided by the trimmed formulation. Obviously experimental data are always "trimmed" since in reality fluid-structure interaction cannot be avoid.

Figure 1(b) shows the mesh displacements of the blade attached to the hub for the last analyzed point, just before the stall region. It is possible to see that the bending displacements dominate blade deformation.



(a) Total pressure ratio.



(b) Trim displacements.

GPU acceleration results

The advantages provided by the GPU in term of computational time reduction [6, 5] are analyzed using the time per iteration per cell as the performance metrics. Thanks to the wide OpenCL hardware compatibility, the same code is executed on one CPU ore, multiple CPU cores and on GPU in order to show the scalability capabilities of the solver. The authors available hardware is represented by an Intel i7 3930K CPU (6-core, ~ 170 $GFLOPS$ single precision, ~ 500 USD) and an AMD 290X (~ 5600 $GFLOPS$ single precision, ~ 400 USD). Simulations are carried out using single precision in order to take advantage of the hardware peak performance.

References

[1] Yun Zheng and Yang Hui. Coupled fluid-structure flutter analysis of a transonic fan. *Chinese Journal of Aeronautics*, 24(3):258–264, 2011.

[2] Volker Carstens, Ralf Kemme, and Stefan Schmitt. Coupled simulation of flow-structure interaction in turbomachinery. *Aerospace Science and Technology*, 7(4):298–306, 2003.

[3] Earl H Dowell and Kenneth C Hall. Modeling of fluid-structure interaction. *Annual Review of Fluid Mechanics*, 33(1):445–490, 2001.

[4] M. Favale and A. Gadda. A gpu parallelized two fields full potential formulation for real gases. Master's thesis, Politecnico di Milano, 2013.

[5] G. Romanelli, L. Mangani, E. Casartelli, A. Gadda, and M. Favale. Implementation of an explicit density-based unstructured cfd solver for turbomachinery applications on graphical processing units. In *ASME Paper*, 2015.

[6] L. Mangani, G. Romanelli, A. Gadda, and Casartelli E. Comparison of acceleration techniques on cfd open-source software for aerospace applications. In *AIAA Paper*, 2015.

[7] Giulio Romanelli, Michele Castellani, Paolo Mantegazza, and Sergio Ricci. Coupled csd/cfd non-linear aeroelastic trim of free-flying flexible aircraft. In _ *53rd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference* _, 2012.

[8] E Casartelli, L Mangani, G Romanelli, and T Staubli. Transient simulation of speed-no load conditions with an open-source based c++ code. In *IOP Conference Series: Earth and Environmental Science*, volume 22, page 032029. IOP Publishing, 2014.

[9] Lucian Hanimann, Luca Mangani, Ernesto Casartelli, Thomas Mokulys, and Sebastiano Mauri. Development of a novel mixing plane interface using a fully implicit averaging for stage analysis. *Journal of Turbomachinery*, 136(8), 2014.

[10] Andrea Arnone. Viscous analysis of three-dimensional rotor flow using a multigrid method. *Journal of turbomachinery*, 116(3):435–445, 1994.