Fully Implicit algorithms for simulations in turbomachinery applications

Marwan Darwish, American University of Beirut P.O.Box 11-0236 (Department) Riad El-Solh Beirut 1107 2020 Lebanon

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

Wolfgang Sanz, Institute for Thermal Turbomachinery and Machine Dynamics Graz University of Technology 8010 Graz, Inffeldgasse 25/A

Long Abstract

Introduction

Computational Fluid Dynamics (CFD) has confirmed its fundamental use as high fidelity tool for aerodynamic design and analysis in aeronautics as well as turbo engine applications. Together with the availability of more and more powerful computing resources, current trends pursue the adoption of such high-fidelity tools and state-of-the-art technology even in the preliminary design phases. The necessity to perform many computations in a reasonable simulation time promoted the use of parallel computing to be the standard approach in the last decades, i.e. software runs on CPU clusters where the workload is distributed among different processors. Nevertheless the large amount of computational resources and time required is still an open issue to work on, mainly due to the classical semi-implicit or explicit segregated solution technologies which result in a convergence rate slow-down for large meshes.

In the present work two possible solutions to improve the computational efficiency of current technology are compared, allowing significant reduction of CFD process turn-around times at relatively low costs, namely:a novel implementation of fully implicit pressure-based coupled algorithms for All-Mach flows for unstructured grids against the well known linearized coupled density-based algorithm implemented in multi-block structured grid solver.

In what follows the developed coupling procedure is presented along with some implementation details. The resulting algorithms are then assessed by solving a number of test-case turbomachinery-related applications. Comparison between experimental and computation results are given. Transonic and supersonic flow results are also reported in order to show the capabilities of the shock-capturing techniques together with the coupled acceleration.

1. Methods

A pressure based compressible All-Mach RANS solver is developed and implementation details are given. The resulting discretization is now coupling the pressure and velocity variables. Equation (1) shows the resulting block coefficient filling.

$$\begin{bmatrix} a_C^{uu} & a_C^{uv} & a_C^{uw} & a_C^{up} \\ a_C^{vu} & a_C^{vv} & a_C^{vw} & a_C^{vp} \\ a_C^{wu} & a_C^{wv} & a_C^{ww} & a_C^{wp} \\ a_C^{pu} & a_C^{pv} & a_C^{pw} & a_C^{pp} \end{bmatrix} \cdot \begin{bmatrix} u_C \\ v_C \\ w_C \\ p_C \end{bmatrix} + \sum_{faces} \begin{bmatrix} a_{NB}^{uu} & a_{NB}^{uv} & a_{NB}^{uw} & a_{NB}^{up} \\ a_{NB}^{vu} & a_{NB}^{vv} & a_{NB}^{vw} & a_{NB}^{vp} \\ a_{NB}^{wu} & a_{NB}^{wv} & a_{NB}^{ww} & a_{NB}^{wp} \\ a_{NB}^{pu} & a_{NB}^{pv} & a_{NB}^{pw} & a_{NB}^{pp} \end{bmatrix} \cdot \begin{bmatrix} u_{NB} \\ v_{NB} \\ w_{NB} \\ p_{NB} \end{bmatrix} = \begin{bmatrix} b_C^u \\ b_C^v \\ b_C^w \\ b_C^p \end{bmatrix} \quad (1)$$

Treating both variables in an implicit manner is in essence the aim of any coupled algorithm. This is achieved here by coupling the momentum and the pressure-form of the continuity equation through a set of coefficients that represent the mutual influence of continuity and momentum on the pressure and the velocity fields.

The density-based solver is a code that was developed by at the Institute for Thermal Turbomachinery and Machine Dynamics in TU-Graz. code is written in the object orientated programming language C++ and hence its easy to maintain and extend by adding new models, boundary conditions or turbulence models.

The code solves the compressible Reynolds-averaged Navier-Stokes (RANS) equations in conservative form by means of a fully-implicit time-marching finite-volume method (Equation (3).

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_V \mathbf{U}\,\mathrm{d}V + \oint_S \left[\mathbf{f}(\mathbf{U}) - \mathbf{g}(\mathbf{U}) - \mathbf{U}\,\mathbf{w}\right]\cdot\mathbf{n}\,\mathrm{d}S = \int_V \mathbf{h}(\mathbf{U})\,\mathrm{d}V \tag{2}$$

$$\mathbf{U} = \left\{\begin{array}{c} \rho \\ \rho\,\mathbf{u} \\ \rho\,E^t \end{array}\right\} \quad \mathbf{f} = \left\{\begin{array}{c} \rho\,\mathbf{u} \\ \rho\,\mathbf{u}\,\mathbf{u}^{\mathrm{T}} + p\,\mathbf{I} \\ \rho\,E^t\,\mathbf{u} + p\,\mathbf{u} \end{array}\right\} \quad \mathbf{g} = \left\{\begin{array}{c} 0 \\ \tau \\ \tau\cdot\mathbf{u} + \mathbf{q} \end{array}\right\} \tag{3}$$

The inviscid fluxes are discretized with the upwind flux- difference splitting method of Roe. High order discretization is used fro both spatial (TVD-scheme) and transient accuracy. The viscid flux vector at the cell interfaces is constructed in a central-differencing manner, using Greenís theorem. The code uses multi block structured grids and implements a full nonlinear multigrid algorithm to accelerate the solution process.

2. Results

Experimental results are compared against CFD simulations for axial turbines geometries representative of an aero-engine gas turbine. Computational performances, local profiles and integral quantities are than evaluated for both codes and validated. Structured and unstructured based grids are also investigated in order to measure the capabilities of the unstructured solver.