A penalisation approach to simulate root compressors

A penalisation approach to simulate root compressors

Andrea Ferrero, Francesco Larocca and
Emanuele Antonio Napoli

17TH INTERNATIONAL CONFERENCE OF NUMERICAL ANALYSIS AND
APPLIED MATHEMATICS, 23-28 SEPTEMBER 2019, RHODES, GREECE

A penalisation approach to simulate root compressors

Andrea Ferrero[1,a)], Francesco Larocca[1,b)] and Emanuele Antonio Napoli[2,c)]

[1]*Dipartimento di Ingegneria Meccanica e Aerospaziale, Politecnico di Torino, Corso Duca degli Abruzzi 24, 10129 Turin, Italy*

[2]*Dipartimento Energia, Politecnico di Torino, Corso Duca degli Abruzzi 24, 10129 Turin, Italy*

[a)]Corresponding author: andrea_ferrero@polito.it

[b)]francesco.larocca@polito.it

[c)]emanueleantonio.napoli@studenti.polito.it

Abstract. The simulation of the unsteady flow field in root compressors requires to deal with several challenges related to the presence of moving parts and complex geometries. In the present work a penalisation approach applied to the compressible Navier-Stokes equations is investigated as a possible technique to perform this kind of simulations. In particular, the compressible Navier-Stokes equations are augmented by source terms which represent the effects of the body on the fluid and then they are integrated in both the fluid and solid domains. The presence of moving bodies is taken into account by a level set function. A validation based on a grid convergence study is performed on the flow around an impulsively started cylinder. A preliminary simulation of the flow field inside a root compressor is performed in order to predict the unsteady mass flow rate through the machine.

INTRODUCTION

The improvement of energy efficiency in air compressing systems has been indicated as a key target by the energy policy of the European Union [1]. Several research efforts are presently dedicated to this goal since there is still room for significant improvements [2]. In this framework, the possibility to predict the flow field inside screw compressors and root compressors becomes a critical points to design new machines with better performances. However, this kind of simulations is challenging because it requires to deal with moving bodies in complex geometries. If a classical body fitted approach would be used then it would be necessary to manage deforming meshes which take into account the evolution of the fluid domain. A possible alternative to the body fitted approach is represented by immersed boundary techniques in which the governing equations are integrated both in the fluid region and in the solid region. In particular, the effects of the body are introducing in the fluid equations as source terms. In this way, the mesh generation and management are significantly simplified since it is no more necessary to follow the deformation of the fluid region with the grid. Among the several immersed boundary techniques which are available, the penalisation approach is one of the simplest. In this work, a penalisation approach applied to the compressible Navier-Stokes equations is applied following [3]. The accuracy of the method is investigated by simulating the flow field around an impulsively started cylinder and comparing the results of the body fitted and penalisation approaches. Finally, a preliminary simulation on a root compressor is performed in order to predict the unsteady flow field across the machine.

NUMERICAL FRAMEWORK

The penalisation approach

The compressible Navier-Stokes equations augmented by the penalisation terms in the momentum and energy equations are reported in the following, according to [3]:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \boldsymbol{u} = 0 \tag{1}$$

$$\frac{\partial \rho \boldsymbol{u}}{\partial t} + \nabla \cdot (\rho \boldsymbol{u} \otimes \boldsymbol{u}) + \nabla p = \nabla \cdot \boldsymbol{\tau} + \frac{1}{\eta} f(-\Phi(\boldsymbol{x}))(\rho \boldsymbol{u} - \rho \boldsymbol{u_s}) \tag{2}$$

$$\frac{\partial \rho e}{\partial t} + \nabla \cdot (\boldsymbol{u}(p + \rho e)) = \nabla \cdot (\boldsymbol{\tau u} + \boldsymbol{q}) + \frac{1}{\eta} f(-\Phi(\boldsymbol{x}))(\rho \boldsymbol{u} - \rho \boldsymbol{u_s}) \cdot \boldsymbol{u} \tag{3}$$

where ρ, $\boldsymbol{u}$, p, t, e and τ represent density, velocity, pressure, time, total energy per unit mass and viscous stress tensor, respectively. The function $\Phi(\boldsymbol{x})$ is the level set defined by the signed distance function from the closest wall boundary (positive in the fluid and negative inside the solid). The function f is the Heaviside function which is zero when the argument is negative and one when the argument is positive or null. The penalisation term η is a small constant: the smaller the value the better the penalisation. Usually, the term η can be related to the mesh size: the finer the mesh, the smaller the value of η. In this work η is chosen as the smallest value which gives stable computations.
The previous equations can be seen as a model for the flow in porous media. The idea is that when a grid point is inside the fluid the original Navier-Stokes equations are recovered since the source terms are automatically set to zero. In contrast, the source terms become the dominating terms of the equations for the grid points inside the body because of the large value assumed by the constant $1/\eta$: in this way the momentum and the energy inside the body are directly imposed by the source terms and the Navier-Stokes terms become negligible. The penalisation term applied to the energy equation used in this work is suitable for problems with adiabatic solid walls: the extension to isothermal solid walls is reported in [3].

Space and time discretisation

The Equation 1-3 are discretised in space by means of a discontinuous Galerkin approach in which a modal hierarchical basis is defined in each element following the approach of [4]. A second order accurate reconstruction (DG1) is adopted for all the following simulations. Convective fluxes are evaluated by means of an approximate Riemann problem solver [5] while diffusive fluxes are computed by a recovery-based approach [6]. This kind of discretisation makes it possible to easily implement adaptive schemes in which both the element size [7, 8] or the element basis functions can be automatically adapted [9] to the local flow features.
Time integration can be performed by means of both explicit and implicit schemes. The discretisation is implemented in a Fortran 90 code which has been tested on several compressible flows, with both inviscid [10] and viscous flows [6, 11, 7].

FLOW AROUND AN IMPULSIVELY STARTED CYLINDER

The flow around an impulsively started cylinder at Reynolds number $Re = 100$ and Mach number $M = 0.2$ is computed by means of the penalisation approach. The problem is studied in the frame of reference in which the air is at rest. A sequence of unstructured meshes with different resolutions is generated by Gmsh [12]. The meshes are characterised by a uniform element size distribution in the region where the cylinder moves. The characteristic length of the elements in this region is set to $l_c/D = \frac{3}{80}, \frac{2}{80}, \frac{1}{80}$ for the three meshes, respectively. Here D represents the diameter of the cylinder. Time integration is performed by a first order implicit scheme with a Courant-Friedrichs-Lewy number set to CFL=20. The force acting on the cylinder is computed by the volume integral of the source term which appears in the momentum equation. The time evolution of the drag coefficient (c_d) is reported in Figure 1 as a function of the adimensional time (which is normalised with respect to the convective time based on the cylinder diameter and the far field speed). The plot shows the results for different values of the characteristic length: as the mesh is refined the penalisation approach gives a solution which converges towards the solution obtained by the body fitted approach on the finest mesh.

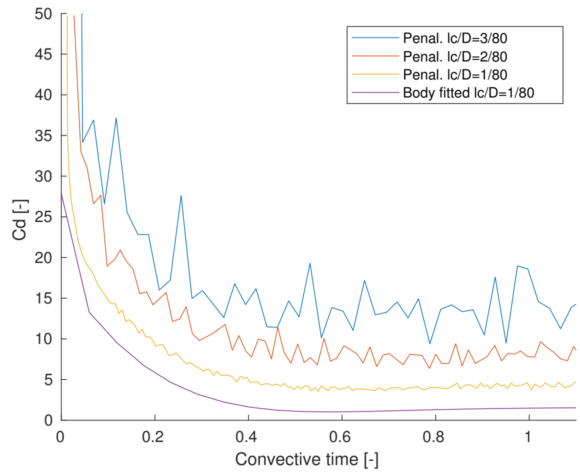

FIGURE 1. Drag coefficient for impulsively started cylinder: comparison between penalisation and body fitted approaches

ROOT COMPRESSOR: PRELIMINARY STUDY AND FUTURE PERSPECTIVES

The flow in a root compressor with two lobes is simulated by the penalisation approach. The diameter of the rotors is 90 mm and the rotors rotate at 8600 rpm. The computational domain includes two large tanks with air (T^0= 300 K, P^0=1 bar) connected by the region where the rotors move. The rotors displace air from the left tank to the right tank. This choice of the computational domain is related to the initial transient which leads to strongly oscillating flows at the beginning of the simulation: a set of preliminary simulation showed that the use of a smaller computational domain, limited to the region covered by the rotors, would lead to flow inversion at the inlet/outlet boundaries of such a reduced domain which would complicate the definition of boundary conditions. The volume of the tanks is sufficiently high to achieve pseudo-periodic conditions in the rotor region before a significant alteration of the average tank pressure takes place. The computational grid contains approximately 50000 elements (which lead to 150000 degrees of freedom per equation with the DG1 scheme) and is refined in the region where the rotors move. This mesh is suitable for a preliminary simulation but future work will be devoted to adaptive mesh algorithm which can improve the resolution in the boundary layer regions. The penalisation factor is set to $\eta = 5 \cdot 10^{-4}$ and a second order accurate explicit time integration scheme with $CFL = 0.3$ is chosen. The level set is evaluated at each time step by computing the distance between the grid points and a set of 1000 points which is used to represent the rotors.
Three instantaneous snapshots of the pressure field at the beginning of the simulation are reported in Figure 2: the plot shows clearly the propagation of the acoustic waves generated by the rotating bodies.
In order to evaluate the inlet and outlet mass flow rate which passes through the machine, two vertical control lines are defined in correspondence of the connections with the tanks. In Figure 3 the time history of the inlet and outlet mass flow rate across these control stations is reported. The mass flow rate is normalised with respect to a reference mass flow defined by the stagnation density in the left tank, the rotor diameter and a reference velocity based on the tank stagnation temperature and gas constant R ($u_{ref} = \sqrt{RT^0}$). The time reported in the plot is normalised with respet to a reference time obtained by the rotor diameter and the reference velocity. A time average is performed on the inlet and outlet mass flow rates by neglecting the initial transient: the averaged inlet mass flow rate is 5% larger with respect to the averaged outlet mass flow rate. This discrepancy can be explained by two effects. The first one is related to the fact that the solution is not exactly periodic and so a larger time window should be used for the average. The second effect could be related to the penalisation approach: an analysis of the density field shows that the density inside the rotors grows during the simulation. This error can be reduced by choosing stronger values of the penalisation factor which can be applied only by refining the mesh.

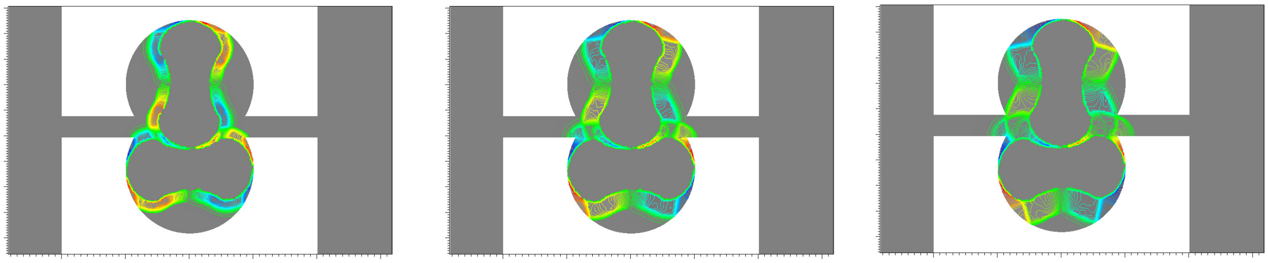

FIGURE 2. Unsteady pressure field generated by the moving rotors at the beginning of the simulation

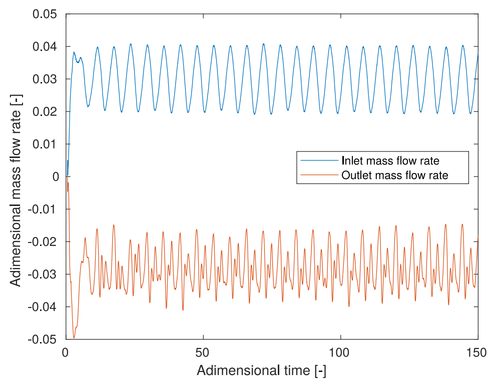

FIGURE 3. Mass flow rate across the root compressor

ACKNOWLEDGMENTS

Computational resources were provided by HPC@POLITO (http://www.hpc.polito.it).

REFERENCES

[1] P. Radgen and E. Blaustein, Stuttgart: LOG_X (2001).
[2] J. dos Santos Mascarenhas, H. Chowdhury, M. Thirugnanasambandam, T. Chowdhury, and R. Saidur, Journal of Cleaner Production (2019).
[3] R. Abgrall, H. Beaugendre, and C. Dobrzynski, Journal of Computational Physics **257**, 83–101 (2014).
[4] F. Bassi, L. Botti, A. Colombo, D. A. Di Pietro, and P. Tesini, Journal of Computational Physics **231**, 45–65 (2012).
[5] M. Pandolfi, AIAA journal **22**, 602–610 (1984).
[6] A. Ferrero, F. Larocca, and G. Puppo, International Journal for Numerical Methods in Fluids **77**, 63–91 (2015).
[7] A. Ferrero, F. Larocca, and V. Bernaschek, Advances in aircraft and spacecraft science **4**, 555–571 (2017).
[8] A. Ferrero and F. Larocca, “Adaptive cfd schemes for aerospace propulsion,” in *Journal of Physics: Conference Series*, Vol. 841 (IOP Publishing, 2017) p. 012017.
[9] A. Ferrero, A. Iollo, and F. Larocca, International Journal for Numerical Methods in Engineering **116**, 332–357 (2018).
[10] A. Ferrero and F. Larocca, Progress in Computational Fluid Dynamics, an International Journal **16**, 14–25 (2016).
[11] E. Ampellio, F. Bertini, A. Ferrero, F. Larocca, and L. Vassio, Advances in aircraft and spacecraft science **3**, 149–170 (2016).
[12] C. Geuzaine and J.-F. Remacle, International journal for numerical methods in engineering **79**, 1309–1331 (2009).