CFD FOR TURBOMACHINERY: METHODS AND APPLICATIONS

HRVOJE JASAK[1,2], GREGOR CVIJETIĆ[1], TESSA UROIĆ[1], LUKA ČULIĆ[1]
[1]*University of Zagreb, gregor.cvijetic@fsb.hr, tessa.uroic@fsb.hr, lukaculic1@gmail.com, hrvoje.jasak@fsb.hr*
[2]*Wikki Ltd., h.jasak@wikki.co.uk*

Keywords: *Turbomachinery, compressible flow, incompressible flow, multiple reference frames, block–coupled solver, characteristic boundary conditions*

Abstract

Turbomachinery CFD has been extensively used for decades and variety of tools have been developed for dealing with specific phenomena appearing in these machines. The tools enable significant computational savings, or provide simplified models for preliminary evaluation, which resulted in establishing CFD as a common, efficient R&D technique in industry. In this paper we present the methods for modeling complex flows inside various types of turbomachinery. A general overview of tools for turbomachinery simulations available in `foam-extend` is given, with the application demonstrated for both incompressible and compressible flow problems. Appropriate methods should be chosen depending on the physics and geometry involved:

- steady-state or transient approach,
- single blade passage or full annulus configuration,
- type of rotor-stator interaction interface,
- compressible or incompressible problem,
- implicitly block–coupled or segregated pressure-velocity system, etc.

All options will be addressed for specific test cases presented in the paper. Furthermore, the mathematical and numerical model applied in the cases will be presented.
In the paper, three incompressible test cases will be considered: a Francis turbine, a centrifugal pump and a ship propeller. A comparison of steady-state multiple reference frames (MRF) approach [1], transient approach and Harmonic Balance method [2] (quasi–steady state) will be given and results will be validated against experimental data, where available. Compressible solvers described in [3, 4] will be evaluated for three test cases as well, each comprising of different flow regime: low compressibility Onera M6 wing, high compressibility Aachen turbine and transonic NASA Rotor 67 test case. Methods for treating the rotor–stator interface, mixing plane and general grid interface (GGI) [5], will be presented and tested in the Aachen turbine test case. Coupling of the pressure and velocity for both the compressible and incompressible flow equations will be presented in the segregated and implicitly coupled formulation.

Mathematical Model

Transient equations provide a solution with a fully resolved flow in time, compared to steady-state or Harmonic Balance methods, which introduce temporal simplifications or approximations to reduce the computational CPU time. The high cost of computational time make the transient approach less attractive for everyday use. To alleviate this problem, simplified methods are often used: multiple reference frame (MRF) which is a steady state approach and the Harmonic Balance (HB) method.
In MRF the rotation effects are modelled by solving the conventional steady–state equation set with terms accounting for rotation: the Coriolis force and centrifugal force, which are included in the momentum equation, either as a source term (segregated approach) or implicitly in the matrix (implicit approach).
Equations 1-6 represent the governing equations for transient, MRF and HB approach, respectively. As previously noted, the MRF equations do not include the temporal derivation term and have the additional term $-\omega \times (\rho \mathbf{u})$.

Transient equation set:

$$\frac{\partial \rho}{\partial t} + \nabla \bullet \rho \mathbf{u} = 0\,, \tag{1}$$

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \bullet (\rho \mathbf{u}\mathbf{u}) - \nabla \bullet (\mu \nabla \mathbf{u}) = -\nabla p\,. \tag{2}$$

Steady-state equation set with MRF:

$$\nabla \bullet \rho \mathbf{u} = 0\,, \tag{3}$$

$$\nabla \bullet (\rho \mathbf{u}_{rel} \mathbf{u}) - \nabla \bullet (\mu \nabla \mathbf{u}) = -\nabla p - \omega \times (\rho \mathbf{u})\,. \tag{4}$$

Harmonic Balance equation set:

$$\nabla \bullet \rho_{t_j} \mathbf{u}_{t_j} = -\frac{2\omega}{2n+1} \left(\sum_{i=1}^{2n} P_{i-j} \rho_{t_i} \right)\,, \tag{5}$$

$$\nabla \bullet (\rho_{t_j} \mathbf{u}_{t_j} \mathbf{u}_{t_j}) - \nabla \bullet (\mu \nabla \mathbf{u}_{t_j}) = -\nabla p_{t_j} \; - \frac{2\omega}{2n+1} \left(\sum_{i=1}^{2n} P_{i-j} \rho_{t_i} \mathbf{u}_{t_i} \right)\,, \tag{6}$$

$$\text{for } j = 1 \ldots 2n+1.$$

Index rel denotes the relative velocity while index t_j denotes the time instant for which the solution is sought. HB source term accounts for the temporal coupling between the time instants - the use of different indices t_i and t_j on the LH and RH side should be noticed.
HB is considered to be a quasi steady–state method as all the equations are steady-state, but mutually coupled by the right hand side term, obtained using the Fourier decomposition on a transient set of equations, which necessarily restricts the application of HB to periodic problems only. For details on the derivation, the reader is referred to [6, 7]. HB simulations can be computed with an arbitrary number of harmonics n, which affects the accuracy of the method. The number of equations scales with $2n + 1$, meaning that for $n = 1$ harmonic, three sets of velocity and pressure equations will be solved. Therefore, by increasing the number of harmonics, the CPU time is also increased. Depending on the problem, the optimal number of harmonics can change, as some transient flow features are more complex than others. The major benefit of HB compared to MRF is that although $2n + 1$ time instants are solved for, the solution can be reconstructed at any point in time, thus the solution for the whole period is available.

Numerical Model

In the first subsection of the numerical model, the segregated and block–coupled implicit approach for solving the Navier–Stokes equations will be analysed. In the second section, turbomachinery–specific boundary conditions will be presented.

Pressure–Velocity Coupling

The derivation of the pressure equation as a Schur complement from the continuity and momentum equations can be found in [4]. The final form of the incompressible, steady–state system can be written as:

$$\begin{bmatrix} [A_{\mathbf{u}}] & [\nabla(.)] \\ [\nabla \bullet (.)] & [\nabla \bullet (D_{\mathbf{u}}^{-1} \nabla(.))] \end{bmatrix} \begin{bmatrix} \mathbf{u} \\ p \end{bmatrix} = \begin{bmatrix} 0 \\ \nabla \bullet (D_{\mathbf{u}}^{-1} \overline{\nabla(p^o)}) \end{bmatrix}\,, \tag{7}$$

where $[A_{\mathbf{u}}]$ is the momentum matrix (convection and diffusion operators), $D_{\mathbf{u}}^{-1}$ is the inverse of the diagonal of the momentum matrix, p^o is the previously available pressure solution and is only used in the Rhie-Chow correction, [8]. Overline indicates face-interpolated cell-centred pressure gradient. The convection term is linearised by using the old available values of flux. The formulation is valid for both the segregated and implicit block–coupled approach, as the off–diagonal terms are treated explicitly in the segregated approach. As stated previously, the additional terms coming from the MRF method are treated as a source term in the segregated approach, while they are added into the matrix in the implicit approach. In implicitly coupled system, there is no need for underrelaxation (or very little) which leads to faster time–to–solution, and also more stability and robustness in converging the solution. The drawback is a high memory demand caused by the 4 times larger matrix dimension of the linear system and, due to the properties of the matrix, more complex linear solvers have to be used, such as the selective algebraic multigrid method, [9].

Boundary Conditions

Usually, the geometry of turbines, pumps and other machines with rotating parts is symmetric and it is beneficial to exploit the fact by applying special boundary conditions which will enable significant reduction in mesh cell count. Some will be presented in this work: periodic boundary conditions for single blade passage simulations, mixing plane [10] and general grid interface [11] for inter–domain interpolation. In order to use the sliding mesh technique in a transient simulation with multiple domains (rotor and stator) which are not connected, a general grid interface (GGI) is used. GGI handles the interpolation between non–conformal boundaries (non–matching faces) of two domains. Common use of GGI is presented in [1]. If a single blade passage with both rotor and stator configuration is simulated, the two domains usually have a partial overlap which should be specially handled. This is done with the aid of `overlapGgi` in `foam-extend`, which mathematically creates the full annulus of the paired interfaces and than performs interpolation. In order to reduce the mesh size, spatially periodic flow can be assumed in turbomachinery, meaning that the same flow pattern appears in each blade passage. This allows the reduction of the simulation domain to a single blade passage with the aid of suitable periodic boundary condition on meridional boundaries. In `foam-extend` such boundary condition is called `cyclicGgi`, and it handles the transformation and interpolation from one meridional patch to its matching pair and vice versa. For more details, the reader is referred to [10].

Results

Several cases will be utilized for demonstration of turbomachinery capabilities, for both the incompressible and compressible flow.

Incompressible flow

For problems with low compressibility, the standard incompressible solver can be used, thus reducing the number of coupled governing equations solved and complexity of the problem.
Francis 99 is a water turbine test case available from the Francis–99 Workshop [12], with the experimental data available. The turbine consists of 28 stator and 30 rotor blades, followed by a draft tube outlet, Figure 1. The rotational velocity is 335 rpm and best efficiency is achieved at 0.2 m^3/s flow rate. We compared the two steady–state methods: MRF and HB, for the best efficiency point. In the final paper, results for the high load and part load operating points will be presented as well. Integral values are compared in Table 1.

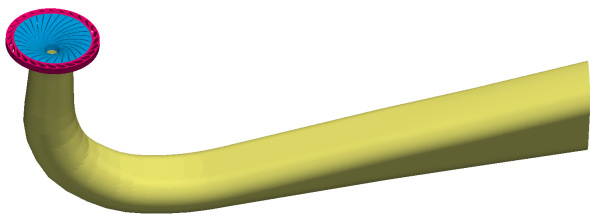

Figure 1: Stator (red), rotor (blue) and draft tube (yellow).

Table 1: Comparison of integral quantities.

	P [W]	H [m]	η [%]
Experiment	21 617	11.94	92.39
HB simulation	22 457	11.53	94.40
Error	3.74%	3.43%	2.13%

The second test case is a centrifugal pump, for which the geometry is shown in Figure 2. The flow will be simulated using the transient solver with sliding mesh rotor–stator interface and also using the steady–state approach. For steady–state, we will use a MRF and compare the results and convergence of the segregated and coupled approach.

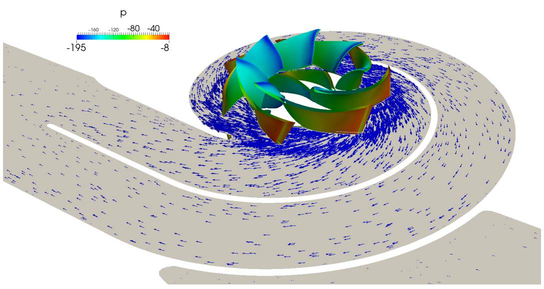

Figure 2: Centrifugal pump geometry with the pressure field on the rotor and velocity vectors.

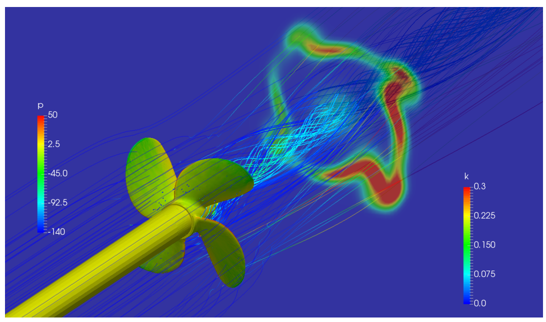

Figure 3: A ship propeller simulation results.

The third test case is a ship propeller, similar to the one shown in Figure 3, for which the transient and steady state approach will be compared (MRF with segregated and coupled solver, and HB solver).

Compressible flow

A low compressibility test case which will be presented is the flow around the Onera M6 wing at Mach number equal to 0.7, for which experimental data is available, [13]. A comparison of the segregated and implicitly coupled approach will be given. Aachen test case, Figure 5, will be presented to demonstrate the use of rotor-stator interface treatment since the configuration of the test case is stator-rotor-stator, coupled with the MRF solver will be used. The final test case is a high compressibility transonic NASA rotor 67. The geometry is shown in Figure 4, for which MRF segregated and coupled compressible solver will be benchmarked against experimental data.

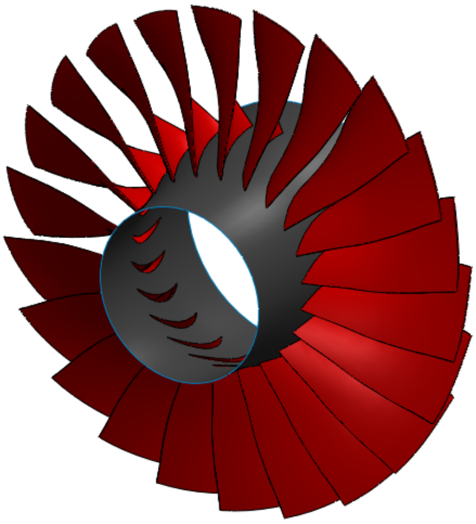

Figure 4: NASA rotor 67 geometry, full annulus.

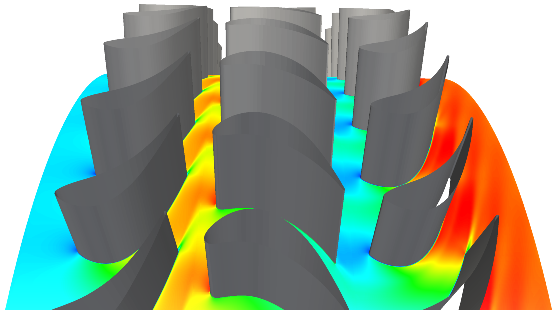

Figure 5: Aachen test case geometry and velocity field.

References

[1] H. Jasak and M. Beaudoin, "OpenFOAM turbo tools: From general purpose cfd to turbomachinery simulations," in *Proceedings of ASME-JSME-KSME Joint Fluids Engineering Conference (AJK2011-FED)*, 2011.

[2] H. Jasak and G. Cvijetić, "Implementation and validation of the harmonic balance method for temporally periodic non–linear flows," in *ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*. American Society of Mechanical Engineers, 2016, pp. V02CT39A009–V02CT39A009.

[3] I. De Dominicis, G. Cvijetić, M. Willetts, and H. Jasak, "Enhanced turbomachinery capabilities for foam-extend: Development and validation," in *11th OpenFOAM Workshop*, 2016.

[4] T. Uroic and H. Jasak, "Implicitly coupled pressure-velocity solver," in *OpenFOAM Selected papers of the 11th Workshop*, 2016.

[5] M. Beaudoin and H. Jasak, "Development of a generalized grid interface for turbomachinery simulations with OpenFOAM," in *Open source CFD International conference*, vol. 2. Berlin, 2008.

[6] G. Cvijetic, H. Jasak, and V. Vukcevic, "Finite volume implementation of the harmonic balance method for periodic non–linear flows," in *54th AIAA Aerospace Sciences Meeting*, 2016, p. 0070.

[7] G. Cvijetic, I. Gatin, V. Vukcevic, and H. Jasak, "Harmonic Balance developments in OpenFOAM," *Computers & Fluids*, 2018. [Online]. Available: http://www.sciencedirect.com/science/article/pii/S0045793018300793

[8] C. Rhie and W. Chow, "A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation," in *AIAA Journal*, vol. 21, 1983, pp. 1525–1532.

[9] T. Uroic and H. Jasak, "Block-selective algebraic multigrid for implicitly coupled pressure-velocity system," vol. 167, pp. 100–110, 2018.

[10] M. Beaudoin, H. Nilsson, M. Page, R. Magnan, and H. Jasak, "Evaluation of an improved mixing plane interface for OpenFOAM," in *IOP Conference Series: Earth and Environmental Science*, vol. 22, no. 2. IOP Publishing, 2014, p. 022004.

[11] M. Beaudoin and H. Jasak, "Development of a General Grid Interface for Turbomachinery simulations with OpenFOAM," in *Open Source CFD International Conference*. American Society of Mechanical Engineers, 2008.

[12] C. Trivedi, M. J. Cervantes, and O. G. Dahlhaug, "Experimental and numerical studies of a high-head francis turbine: A review of the francis-99 test case," *Energies*, vol. 9, no. 2, p. 74, 2016.

[13] "ONERA M6 Wing." [Online]. Available: https://www.grc.nasa.gov/www/wind/valid/m6wing/m6wing.html