CFD Analysis of Turbine Cascade Unsteady Aerodynamics Using a Hybrid POD Technique

Vladyslav Skilskyy, Viola Rossano, and Giuliano De Stefano(✉)

Engineering Department, University of Campania Luigi Vanvitelli, 81031 Aversa, Italy
giuliano.destefano@unicampania.it

Abstract. This study presents a computational investigation into the unsteady aerodynamics of a low-pressure turbine cascade, utilizing computational fluid dynamics (CFD) with a primary focus on enhancing efficiency. The proposed approach combines a classical proper orthogonal decomposition with a modern machine learning technique. This hybrid methodology demonstrates its effectiveness by accurately predicting the unsteady flow over the turbine blade. Crucially, the solution retains the essential features of the original physics-based computational model. This study represents a potential significant advancement in improving the efficiency of CFD solutions, enabling future resource-conscious scale-resolving simulations of complex aerodynamic flows without sacrificing solution accuracy.

Keywords: Computational Fluid Dynamics · Compressible Turbulent Flow · Turbine Blade Aerodynamics · Proper Orthogonal Decomposition

1 Introduction

Low-pressure turbine (LPT) blades are intricately designed to enhance aerodynamic efficiency at low-pressure ratios, prioritizing the recovery of exhaust energy, while minimizing back pressure [1]. The airflow around one blade naturally becomes turbulent due to the wake produced by upstream blades, resulting in unsteady airflow that interacts with blade movements, creating a complex nonlinear scenario. Accurately predicting this fluid-structure interaction (FSI) phenomenon is crucial, particularly concerning potential flutter instabilities, especially with extended blades. Despite advancements in understanding LPT flow dynamics under both laminar and turbulent conditions, addressing the transitional region in the flow remains challenging, especially considering the effects induced by the passage of wake flow from upstream blade rows [1]. Over the past decade, considerable efforts have been devoted to developing effective numerical approaches for aero-mechanical analysis, primarily tailored for industrial design

© The Author(s), under exclusive license to Springer Nature Switzerland AG 2024
O. Gervasi et al. (Eds.): ICCSA 2024, LNCS 14814, pp. 355–365, 2024.
https://doi.org/10.1007/978-3-031-64608-9_23

considerations [2]. However, these approaches might prove insufficient when simulating flow patterns around the LPT blades, particularly for separated flow conditions. Therefore, there is a need to develop more accurate and detailed numerical models, especially to fully comprehend the complex airflow dynamics in modern LPT experiencing vibrations. For instance, a nonlinear frequency-domain technique was employed in [3] to predict pressure coefficients and aero-damping for multistage turbine blades, achieving good agreement compared to time-domain methods.

In the industrial context, low-fidelity methods remain desirable due to their particular emphasis on aero-damping analysis. However, this emphasis often comes at the cost of neglecting the intricate physics underlying FSI. This pragmatic approach introduces what is commonly referred to as a "black-box effect," wherein a deep understanding of fundamental physical processes is sacrificed. This becomes particularly critical in the design of modern aeronautical LPT blades featuring slender airfoils, where flutter may emerge as a significant constraint. In the current era of extensive data utilization, the pivotal role of data-driven models in industrial research cannot be overstated. In contrast to complex numerical approaches to turbulent flow modeling like large-eddy simulation (LES), which requires highly detailed grids to capture shocks and boundary layers, data-driven models emerge as a highly efficient alternative. Surrogate models have clearly proven their effectiveness in dealing with challenges related to optimizing shapes [4], positioning data-driven models as a very promising approach, surpassing the limitations of hybrid approaches based on Reynolds-averaged Navier-Stokes (RANS) and LES methodologies [5], and conventional wind tunnel experiments [6].

While commonly employed, there is a lack of comprehensive research explaining the reasons for preferring neural network (NN) or other artificial intelligence tools over alternative methods like proper orthogonal decomposition (POD) for the reconstruction of transonic flows. This becomes particularly crucial when considering sample efficiency, a factor frequently overlooked in such evaluations. Conventionally, the belief is that deep learning demands an extensive dataset compared to more traditional tools, but this viewpoint is questioned in our current research. This study aims at improving the accuracy of the POD approach by using a deep learning algorithm to estimate the unsteady compressible aerodynamics behaviour in a low-pressure turbine cascade.

This represents a significant advancement over previous works [7,8], which primarily focused on only one of these methods. The integration of POD and machine learning techniques brings about several benefits. Firstly, it significantly reduces computational complexity, making it more efficient than using a single method. Secondly, it allows for the extraction of essential flow features, which may not be possible with a single method, leading to a better understanding of the dynamic behavior of the turbine cascade. Finally, it preserves the essential features of the original physics-based computational model, ensuring that the solution's accuracy is not compromised.

The manuscript is organized as follows: Section 2 provides a comprehensive overview of the CFD model. The proposed methodology is introduced in Sect. 3, where the POD technique is briefly reviewed, and the hybrid POD-DNN method is presented. The preliminary results are presented in Sect. 4, accompanied by a thorough analysis of the main sources of error. Finally, Sect. 5 offers some concluding remarks.

2 CFD Model

In this study, the T106A LPT blade profile acts as a suitable benchmark for the proposed data-driven model, representing the mid-section of the PW2037 LPT rotor passage blade. Widely examined in previous studies, this profile offers valuable insights into complex phenomena like flow separation and transition in engine regions characterized by high-pressure gradients, along with moderate Mach and Reynolds numbers [9].

The computational domain, which is depicted in Fig. 1, outlines the geometric characteristics of the turbine blade, featuring a chord length of 1.0 m, an axial chord of 0.8663 m, a pitch of 0.9557 m, and a domain breadth of 2.287 m in the homogeneous spanwise direction. The domain's origin is situated at mid-span along the leading edge of the turbine blade, with the inflow plane positioned 0.92 m upstream from the leading edge, and the outflow plane situated 1.86 m downstream. In this configuration, the Reynolds number based on the chord of the turbine blade is set to 2×10^5, while the Mach number is 0.601. The air is assumed to behave as an ideal gas with constant viscosity and its dynamics is governed by the compressible Navier-Stokes equations. To highlight a nuanced aspect that has received limited attention in the existing literature, a cylindrical wake generator was intentionally positioned ahead of the blade, following the methodology proposed in [9].

The mesh generation process utilized the Pointwise software V18.6R2. Initially, a two-dimensional triangle-based unstructured mesh was created on the blade surface. To take into account the boundary layers near the various solid surfaces (of both blade and cylinder),the T-Rex method was employed to generate a boundary layer mesh consisting of 30 layers of quadrilaterals beneath the triangles. The minimum wall spacing was set at 3×10^{-5} m, with a growth rate of 1.2 from one layer to another.

Numerical simulations were conducted using the CFD code Ansys Fluent v23.R1 [10], employing a pressure-based solver, due to the relatively low Mach number. The freestream turbulence intensity was imposed to be 2%, and the blade surface was subjected to isothermal no-slip boundary condition. Using the finite volume (FV) method, the governing equations were discretized applying the conservation principles directly to each computational cell [11]. The SIMPLEC procedure managed the coupling between pressure and velocity. A second-order upwind scheme was utilized for pressure discretization, energy, and turbulence variables, while a central differencing scheme was employed for momentum.

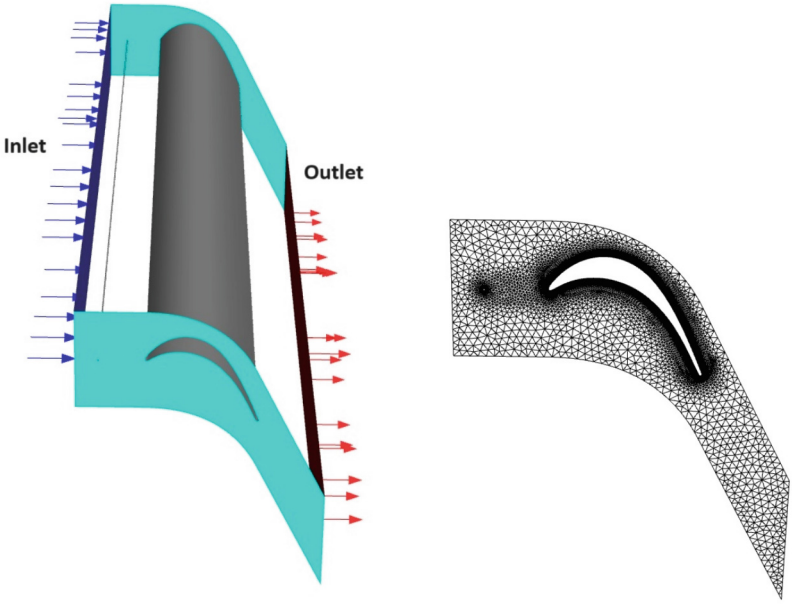

Fig. 1. Computational domain and spatial mesh.

Table 1. Flow specification

Parameter	Value
Total temperature at inlet (T_{in})	350 K
Static pressure at inlet (P_{in})	7,340 Pa
Total pressure at inlet ($P_{\text{total,in}}$)	7,770 Pa
Static pressure at exit (P_{exit})	6,950 Pa

In the context of applied scientific research, our recent findings, e.g. [12, 13], suggest that employing unsteady RANS models offers a feasible and cost-effective alternative for predicting complex aerodynamic flows with engineering accuracy, even for rough walls [14], which is particularly important for turbine blades. Therefore, a hybrid RANS-LES model utilizing the Stress-Blended Eddy Simulation (SBES) technique was employed for turbulence modeling [15]. This approach dynamically switches between RANS and LES turbulence models using a shielding function dependent on the local mesh resolution [16].

Concerning boundary conditions, velocity inlet and pressure outlet conditions were specified in the streamwise direction. Periodic boundary conditions were applied to the upper, lower, and lateral walls. The inlet and outlet thermo-fluid dynamic conditions were set according to the specifications detailed in Table 1.

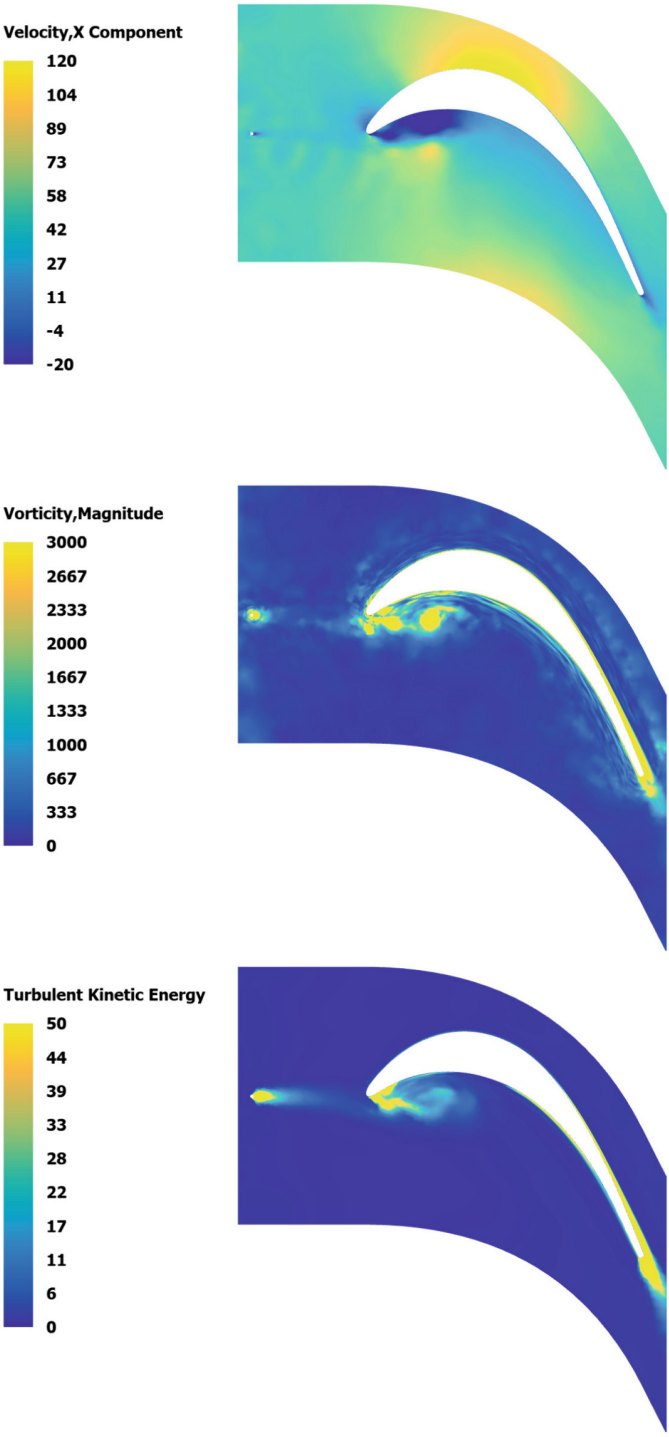

Fig. 2. Flow field variables from CFD analysis at $t = 0.01\,\mathrm{s}$.

3 Methodology

3.1 POD Technique

By now, POD can be considered a strong and widely used technique in the field of applied CFD. Originating from Lumley's work [17], POD has been refined by Sirovich, particularly through the method of snapshots [18]. The method is similar to Fourier analysis but stands out by often needing fewer modes to create a highly efficient system representation. This distinct feature emphasizes the importance of POD as a valuable option compared to Fourier-type analysis when dealing with the original partial differential equations describing fluid flow [19] (Fig. 2).

The POD expansion allows the representation of Fourier coefficients for a scalar variable $u(x,t)$ (representing a generic fluid dynamic variable), defined in the time interval $0 < t < T$ [20]. Mathematically, this variable is expressed as:

$$u(x,t) = \sum_{n=1}^{\infty} c_n(x)\phi_n(t), \tag{1}$$

where $\phi_n(t)$ denotes a set of orthonormal basis functions. The functions $c_n(x)$ are random variables, particularly relevant when dealing with random processes like fluid turbulence, and are given by:

$$c_n(x) = \int_0^T x(t)\phi_n^*(t)dt \ . \tag{2}$$

where $\phi_n^*(t)$ denotes the complex conjugate of the orthonormal basis function $\phi_n(t)$. The above integral captures the interaction between data and basis functions, yielding coefficients c_n that encapsulate the contribution of each mode to the overall system. The POD method minimizes the mean square error of projections, expressed as:

$$error = \langle \|u(x,t) - \hat{u}(x,t)\|^2 \rangle. \tag{3}$$

In this expression, where u represents the original data, $\hat{u}(x,t)$ signifies the reconstructed data using a reduced set of modes. The aim is to optimize this projection, ensuring that the reconstructed data closely approximates the original dataset. This minimization process allows for a more compact representation of the system's dynamics, emphasizing the most significant features while reducing computational complexity. To obtain POD modes, the following eigenvalue problem needs to be solved:

$$\int_0^T R(t_1,t_2)\phi_n(t_2)\,dt_2 = \lambda\phi_n(t_1). \tag{4}$$

Here, $R(t_1,t_2)$ is the auto-covariance of the fluid variable $u(x,t)$ evaluated in different time instants (t_1,t_2). The eigenvalues of each POD mode represent the

energy contained in that mode, with magnitudes arranged in decreasing order ($\lambda_n > \lambda_{n+1}$).

In this approach, CFD data at a specific time interval are treated as a snapshot. The method of snapshots aims at reducing the computational cost of solving the eigenvalue problem in Eq. (4), which involves an $n \times n$ matrix. The auto-covariance matrix R is approximated as a sum of snapshots:

$$R_{ij} = \frac{1}{M} \sum_{n=1}^{M} x_i^n x_j^n. \tag{5}$$

Here, the snapshots are assumed to be sufficiently separated in time or space, beyond the correlation time or distance. While commonly employed, doubts arise about the applicability of this method in some kind of flows, due to its assumption of uncorrelated snapshots, as mentioned in [20].

3.2 Hybrid POD-DNN Method

In this study, we investigate the application of the back-propagation neural network (BPNN) to predict flow fields based on POD principles, following the method of Xie et al. [21]. Renowned for its well-established theory, exceptional performance, and broad applicability, BPNN proves to be a robust tool for flow field predictions.The BPNN minimizes the mean square error between predicted ($\hat{y}_i$) and actual values (y_i). The neuron, which represents the fundamental unit of a neural network, combines weighted inputs and generates an output using an activation function. The network structure, enhanced by multiple hidden layers in a Deep Neural Network (DNN), influences the overall functionality.

The common activation function ReLU is preferred, owing to its near-linear response, simplifying the application of gradient-based optimization algorithms [22]. The network is trained to minimize output errors using a quadratic function. The training process aims at enhancing predictions by minimizing the disparity between actual and predicted values across all sample points, measured by the mean squared error (MSE) loss function:

$$J_{\text{MSE}} = \frac{1}{N} \sum_{i=1}^{N} (y_i - \hat{y}_i)^2, \tag{6}$$

where N represents the total number of training examples. The optimization process, driven by the gradient descent algorithm, enhances the network's performance:

$$g_t(\theta) = \frac{\partial \mathcal{J}}{\partial \theta} \tag{7}$$

In summary, the POD methodology allows the identification of primary modes and their associated modal coefficients, providing insights into the main characteristics of the resolved flow field. Essentially, POD discerns fundamental patterns and their respective strengths within the CFD dataset. Subsequently,

the obtained modal coefficients are utilized to train a particular DNN, referred to as AI_Airfoil_CFD, which is available at https://github.com/Jameshin/AI_Airfoil_CFD. This DNN is specifically designed to predict the modal coefficients under unknown conditions, such as situations not previously observed.

In conclusion, once trained, the DNN can efficiently reconstruct the local flow field with a low computational cost based on the POD orthogonal basis. This implies that deep learning framework can predict the flow distribution without the need to solve higher-order non-linear partial differential equations. In practical terms, this methodology facilitates an efficient prediction of the flow field, proving applicable in scenarios where the direct numerical solution of flow equations would be computationally highly demanding.

4 Preliminary Results

In this showcase study, the ability of the POD-DNN method to reconstruct the velocity field is assessed by comparing velocity data to ground-truth values obtained through CFD, at four different time instants. The preliminary results, which are shown in Fig. 3 in terms of velocity contours, demonstrate the accurate modeling of separation zones by using the POD-DNN technique. In fact, the method provides results that are fully consistent with the expected physical scenario. Despite turbulent flow in the turbines, the hybrid POD-DNN algorithm accurately estimated fluid separation near the trailing edge, proving its effectiveness in resolving complex turbulence with high accuracy. The deviation between the values predicted by POD-DNN and CFD results was assessed through the computation of relative mean squared errors, as outlined in Table 2.

The POD-DNN approach offers a significant advantage not only in terms of accuracy but also in its ability to rapidly generate results compared to conventional numerical methods. To assess the difference in computational efficiency between CFD and POD-DNN, we conducted a comparative analysis, focusing on the computational time required. This involved running identical simulations using both approaches and recording the time taken to reach the solution. The speedup is evaluted as the ratio of computational time between CFD and POD-DNN methods, providing a quantitative measure of the efficiency of POD-DNN compared to traditional CFD simulations. The individual computational times for a single simulation, utilizing a dedicated GPU node are detailed in Table 3.

Table 2. Error estimation

Time (s)	RMSE
7.6×10^{-5}	1.835415×10^{-13}
0.0014	2.320662×10^{-13}
0.007	8.444527×10^{-13}
0.013	7.502180×10^{-13}

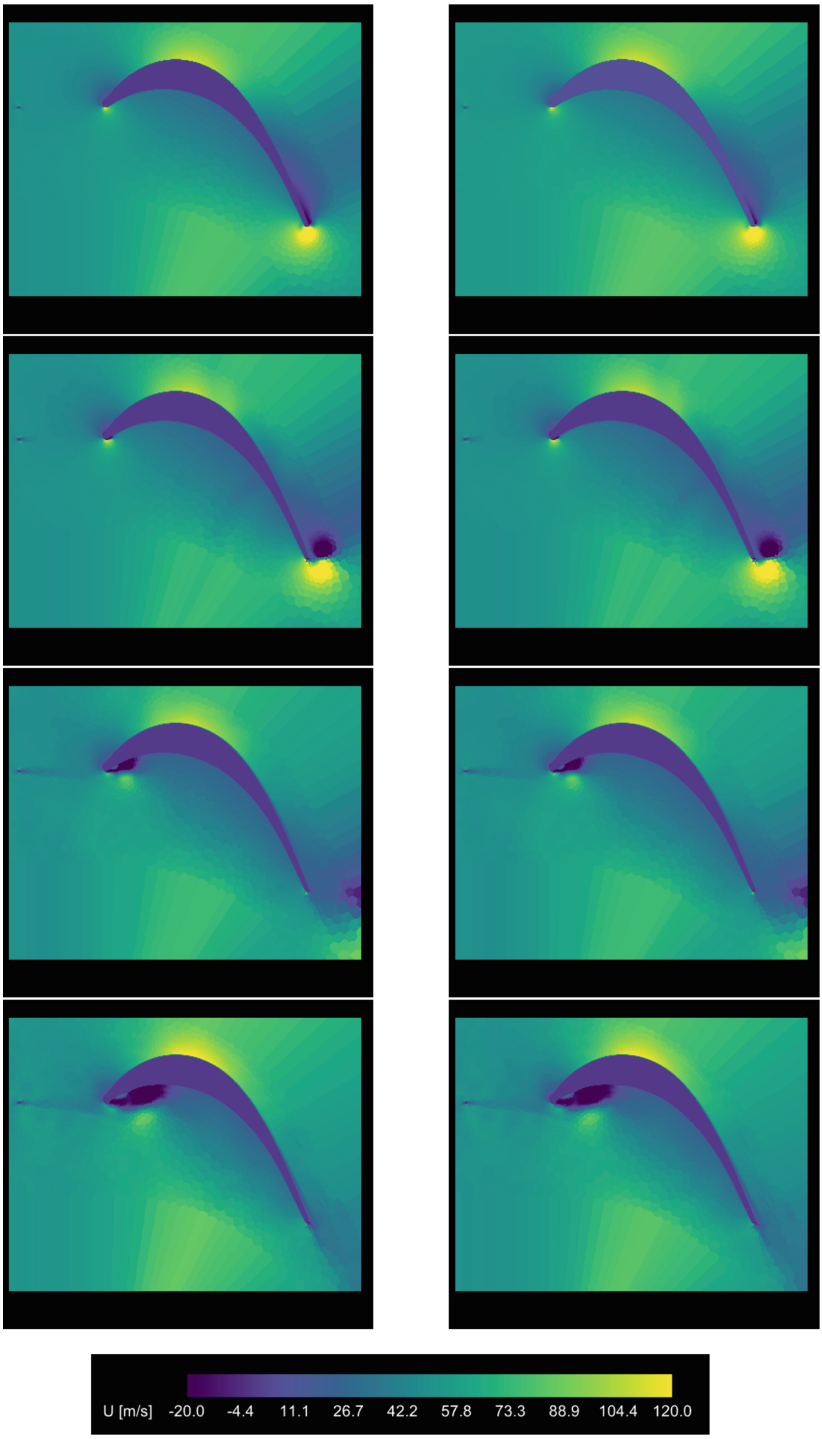

Fig. 3. Axial velocity field at the (x, y) plane for various time instants corresponding to $t = 7.6e-05$ s, 0.0014 s, 0.007 s, 0.013 s, organized from top to bottom rows for CFD and POD-DNN, displayed from left to right columns.

Table 3. Computational time

Method	Time (s)	Speedup
CFD	864000	–
POD-DNN	11520	75

5 Conclusion

This study explores the feasibility of rapidly obtaining solutions for flow fields in turbomachinery applications. The objective is to test a novel approach that integrates POD with deep learning to estimate unsteady aerodynamic flows. To generate data for unsteady flows, hybrid RANS-LES simulations are performed, and the acquired dataset is employed to train the method, while validating the effectiveness of the model. The present POD-DNN framework demonstrated the ability of accurately capturing the features of the flow field. In addition, it is demonstrated the potential for conducting simulations with low computational cost, while addressing complexities in the resolved flow field. Finally, it is important to acknowledge the limitations of the POD-DNN method, particularly when extrapolating fields in strongly nonlinear conditions outside the parameter space for which the data are available, where the relationship between inputs and outputs can become unpredictable. Additionally, there exist several potential ways for improving the proposed method. The exploration of integrating alternative machine learning techniques, such as reinforcement learning or genetic algorithms, could prove beneficial in augmenting the performance of the methodology. Furthermore, the incorporation of experimental data holds the potential to enhance the model's accuracy by providing real-world insights that are not captured by computational simulations.

References

1. Besem, F.M., Kielb, R.E.: Influence of the tip clearance on a compressor blade aerodynamic damping. J. Propuls. Power **33**, 227–233 (2017)
2. Rahmati, M.T., He, L., Wells, R.G.: Interface treatment for harmonic solution in multi-row aeromechanic analysis. In: Turbo Expo: Power for Land, Sea, and Air, vol. 4, pp. 1253–1261 (2010)
3. Rahmati, M.T., He, L., Wang, D.X., Li, Y.S., Wells, R.G., Krishnababu, S.K.: Nonlinear time and frequency domain methods for multirow aeromechanical analysis. J. Turbomach. Trans. ASME **136**, 041010 (2014)
4. Liu, J., Song, W.-P., Han, Z.-H., Zhang, Y.: Efficient aerodynamic shape optimization of transonic wings using a parallel infilling strategy and surrogate models. Struct. Multidiscip. Optim. **55**, 925–943 (2017)
5. Rossano, V., De Stefano, G.: Testing a generalized two-equation turbulence model for computational aerodynamics of a mid-range aircraft. Appl. Sci. **13**, 11243 (2023)

6. Zhu, J., Liu, L., Liu, T., Shi, Y., Su, W., Wu, J.: Lift and drag in two-dimensional steady viscous and compressible flow: I. far-field formulae analysis and numerical confirmation. In: 45th AIAA Fluid Dynamics Conference, vol. 2305 (2015)
7. Pan, Y., An, X., Lei, Y., Ji, C.: An improved neural network for modeling airfoil's unsteady aerodynamics in transonic flow. Phys. Fluids **36**(1), (2024)
8. Fonzi, N., Brunton, S.L., Fasel, U.: Data-driven modeling for transonic aeroelastic analysis. J. Aircraft **61**(2), 625–637 (2024)
9. Iyer, A.S., et al.: High-order accurate direct numerical simulation of flow over a MTU-T161 low pressure turbine blade. Comput. Fluids **226**, 104989 (2021)
10. ANSYS Inc., ANSYS Fluent (Version 23R1)
11. De Stefano, G., Denaro. F.M., Riccardi, G.: High-order filtering for control volume flow simulation. Int. J. Numer. Meth. Fluids **37**, 797–835 (2001)
12. Rossano, V., Cittadini, A., De Stefano, G.: Computational evaluation of shock wave interaction with a liquid droplet. Appl. Sci. **12**, 1349 (2022)
13. Rossano, V., De Stefano, G.: Hybrid VOF-Lagrangian CFD modeling of droplet aerobreakup. Appl. Sci. **12**, 8302 (2022)
14. Salomone, T., Piomelli, U., De Stefano, G.: Wall-modeled and hybrid large-eddy simulations of the flow over roughness strips fluids **8**, 10 (2023)
15. Mendez, M.A., Ianiro, A., Noack, B.R., Brunton, S.L.: Data-Driven Fluid Mechanics: Combining First Principles and Machine Learning. Cambridge University Press (2023)
16. Rossano, V., De Stefano, G.: Scale-resolving simulation of shock-induced aerobreakup of water droplet. Computation **12**, 71 (2024)
17. Berkooz, G., Holmes, P., Lumley, J.L.: The proper orthogonal decomposition in the analysis of turbulent flows. Annu. Rev. Fluid Mech. **25**, 539–575 (1993)
18. Sirovich, L.: Turbulence and the dynamics of coherent structure. Part I, II, III. Quat. Appl. Math. **3**, 583 (1987)
19. Duggleby, A., Paul, M.R.: Computing the Karhunen-Loève dimension of an extensively chaotic flow field given a finite amount of data. Comput. Fluids **39**(9), 1704–1710 (2010)
20. Gorder, R.: Use of proper orthogonal decomposition in the analysis of turbulent flows. Report, Fluid Turbulence Course, University of Washington (2010)
21. Xie, C., Yuan, Z., Wang, J.: Artificial neural network-based nonlinear algebraic models for large eddy simulation of turbulence. Phys. Fluids **32**, 116610 (2020)
22. Jarrett, K., Kavukcuoglu, K., Ranzato, M., LeCun, Y.: What is the best multi-stage architecture for object recognition? In: 2009 IEEE 12th International Conference on Computer Vision, pp. 2146 (2009)