Journal of Physics: Conference Series

Towards the development of an advanced wind turbine rotor design tool integrating full CFD and FEM

To cite this article: Lorenzo Cozzi *et al* 2022 *J. Phys.: Conf. Ser.* **2265** 042050

The Science of Making Torque from Wind (TORQUE 2022) IOP Publishing
Journal of Physics: Conference Series **2265** (2022) 042050 doi:10.1088/1742-6596/2265/4/042050

Towards the development of an advanced wind turbine rotor design tool integrating full CFD and FEM

Lorenzo Cozzi[1], Juri Bellucci[1], Matteo Giovannini[1], Francesco Papi[2], Alessandro Bianchini[2]

[1] MORFO Design Srl, via Madonna del Piano 6, 50019, Sesto Fiorentino, Italy
[2] Department of Industrial Engineering, University of Florence, via di Santa Marta 3, 50139 Firenze, Italy

Author contact email: Lorenzo.Cozzi@morfodesign.it

Abstract. Large, highly flexible wind turbines of the new generation will make designers face unprecedent challenges, mainly connected to their huge dimensions. To tackle these challenges, it is commonly acknowledged that design tools must evolve in the direction of both improving their accuracy and turning into holistic, multiphysics tools. Furthermore, the wind turbine industry is reaching a high level of maturity, and ever more accurate and reliable design tools are required to further optimise these machines. Within this framework, the study shows the development of an integrated platform for blade design integrating 3D CFD flow simulations and 3D-FEM structural analysis. Artificial intelligence techniques are applied to develop an optimization procedure based on the proposed tool. The potential of the new platform has been tested on the well-known test case of the MEXICO rotor, for which an optimization of the blade design has been carried out. Exploring a design space sampled with 2000 CFD and FEM computations, increases in blade torque have been obtained at each of the three tip-speed ratios (TSR) investigated, ranging from 6% at the nominal TSR to 14% at the lowest one, while stresses on the blade are kept almost unaltered.

1. Introduction and objectives

The development of the new generation of wind turbines poses new challenges to designers [1]. In particular, the continuous upscaling of modern rotors implies not only the presence of significant aeroelastic effects while in operation (in turn, leading to aeroelastically-tailored design techniques for passive load alleviation), but also the exploitation of winds at height exceeding the atmospheric boundary layer [2]. In these regions, unprecedent flow features are to be expected, particularly regarding the influence of turbulence. These factors, together with many others discussed in [3], are showing that the design and simulation methods in use up to now by industry and academia, and particularly the ubiquitous Blade Element Momentum Theory (still used, and also predictive in the majority of the applications), are reaching their limits in terms of accuracy and are about to be replaced by higher-fidelity methods, like Computational Fluid Dynamics (CFD). For this latter to be extensively applied, however, high computing resources need to be further developed. Wind energy applications indeed suffer from the so-called “scale mismatch”, i.e., the need of accounting in a single simulation of length scales ranging from those of atmospheric flows (order of 10^2 m) to those of the boundary layer on the blades (order of 10^{-3}m), thus making the computational cost increase significantly [4]. A further key trend noticeable in current wind turbine research is the urgent need for making the design process of wind turbines as holistic as possible [3]. The complex interactions between the aero-servo-elastic components (also including hydrodynamics in case of floating offshore turbines) are showing that a design process based on iterations between separate groups of experts could not bring to the optimum or converges too slowly in comparison to the needs of a fast-growing industry like that of wind energy. To this end, design tools should not only be more accurate, but also able to account simultaneously for as many "sciences" as possible, to more efficiently explore the design space and produce accurate predictions of wind turbine performance (e.g., [5]). If some examples do exist already in the literature (e.g., [6] [7]), they are still very often based on engineering methods that make use of conventional aerodynamics based on airfoil polars. Moreover, wind turbine blades are now reaching an unprecedented level of refinement and are being optimized to meet structural and aerodynamic targets to reduce the levelized cost of energy (LCOE). To further improve on these designs, numerical tools with an ever-increasing level of fidelity and accuracy are required. Within this context, the present study describes the first steps towards the development of an integrated software platform for the aero-mechanical design of wind turbine blades based on blade-resolved 3D CFD flow simulations and 3D-FEM for the structural analysis. The two environments are not only connected, but also integrated within an optimization framework. While still not at the level of a complete design tool, this study proves that the proposed approach is extremely promising for a future implementation at larger scale. In particular, the potential of the new platform has been tested on the well-known test case of the MEXICO rotor [4].

2. Methodology

The significant novelty of the proposed approach relies on the adaptation to the wind energy sector of a design approach originally developed for aerospace applications. The approach is thought to be used in the early design phase, so that effective designs can be identified with confidence since the beginning of the process, strongly reducing the number of iterations towards the optimal solution. The conventional blade design procedure consists of obtaining tabulated aerodynamic coefficients for 2D airfoils. These coefficients are then used in low to mid fidelity engineering tools to simulate the wind turbine rotor. In contrast to these approaches, the proposed tool uses blade-resolved CFD to model rotor aerodynamics, eliminating the need for aerodynamic tables. This way, complex radial flow interactions can be solved, as opposed to them being included through empirical models, greatly improving the reliability of the methods. Moreover, as detailed in the following sections, the procedure developed in this paper will allow designers to discard the traditional approach based on "families" of airfoils and start considering way bigger design spaces, potentially largely improving design choices. As discussed previously, computational cost has somewhat hampered the use of 3D CFD/FEM in wind energy. Using a response-surface approach however, as already demonstrated in the aeronautical field [8], allows for high-fidelity computations to be leveraged while containing computational costs. More in detail, the proposed methodology adopts CFD and FEM results for a first coarse grain sampling, then it exploits advanced AI tools (mainly based on Artificial Neural Networks) to build a meta-model able to generalize these first results accurately. Finally, optimization techniques are used to explore the response surface, looking for multi-objective, constrained optima. The complete automation of the whole procedure, and its ability to exploit current HPC (High-Performance-Computing) resources are also key features of the new methodology. Moreover, while the procedure is tested in this study only for a turbine blade, its extension to other components like the nacelle or the tower is straightforward. Furthermore, the modularity of the approach allows the implementation of different targets, or different tools, thus allowing the effective tailoring to each specific design activity requested by a company or a design team.

2.1. Optimization/design procedure

The workflow of the new design environment is presented in Figure 1. As already mentioned, it relies on a high level of automation, where the designer is involved only in design choices, while all other phases of pre-processing, calculation, and evaluations are carried out autonomously. As the focus is on the design, and not only on the analysis of a given geometry, the first important aspect concerns the ability of the system to "generate" geometries. For this reason, the first step of the design process is the definition of the geometry in a parametric form. To this end, an in-house parametrization tool was developed, where the

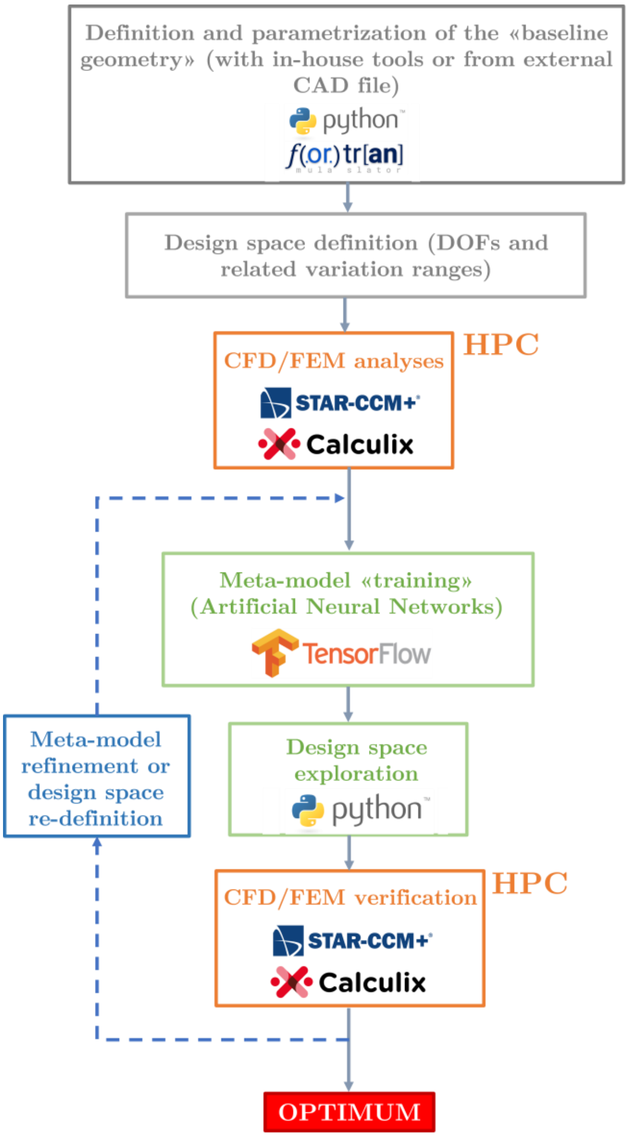

Figure 1: Scheme of the optimization chain.

rotor is defined by means of an arbitrary number of blade profiles stacked along the rotor span. Each of these profiles is described parametrically by means of B-splines curves; then, the whole 3D shape of the blade is handled by the spanwise distribution of the parameters' values. The tool can generate a CAD model of a blade starting from few parameters, and can be also adapted to fit a geometry, as, for example, one coming from a previous design. The parameterization software was mainly developed in Fortran, while the open-source OpenCascade Libraries (OCC) [9] were used for the CAD generation.

The next step of the procedure concerns the definition of the design space, hence the choice of the degrees of freedom (DOFs) to be considered for the design and their range of variation. The design space is then sampled adopting quasi-random/low discrepancy sequences to select the geometries to be evaluated by means of CFD and FEM analyses. These CAE results are adopted as a first database for building the response surface. In the first implementation of the procedure used for this study, the commercial code STAR-CCM+ was adopted for CFD calculations, coupled with the opensource FEM solver "Calculix" [10] used for structural analyses. In the new design system, the pre-processing phase (CAD import, meshing, simulation setup), the calculations and post-processing phase have been automated by means of in-house software (mainly using Python and Java scripts). Once the design space has been sampled (i.e., CFD and FEM calculations have been carried out for all selected geometries), a meta-model based on artificial neural networks (ANNs) is trained on the CFD/FEM results. The meta-model, that follows a connectionism approach, is developed leveraging an inhouse software based on the open-source Tensor Flow framework [11]. In particular, the ANN adopted is a feed forward network with two hidden levels, each level consisting of eight neurons [12] [13]. According to Hecht-Nielsen theorem, this structure of the network can compute any function f(x) with a certain accuracy. The type of training is supervised and adopt a gradient-based back-propagation algorithm. The activation function is the ReLu and the optimizer for training is the Adam algorithm. The inputs of the meta-model are the DOFs of the design space, while the outputs are the objective functions of the considered problem (e.g., the power output, the maximum stress, the maximum displacement, etc.). ANNs are used as they have proven to be effective in aero-mechanical optimization, as they allow to reach a proper generalization accuracy even for complex problems while keeping a reduced number of training examples. The in-house AI tool allows the selection of the proper model, the tuning of hyper-parameters, and an estimation of the reliability of the response surface. This latter is then explored, using in-house developed search tools taken from data analysis. These tools allow to select proper objective functions (OFs) and to set the constraints required for the optimal solutions (e.g., limits on the mechanical stress or deformation). The optimal configurations are then verified through CFD and FEM analyses to confirm the accuracy of the ANN prediction, and to increase the confidence in the performance of the new designs. Some discrepancies, indeed, may arise when comparing meta-model predictions and CFD/FEM data for the optimal solutions. As a matter of fact, indeed, it is easy for this to happen when one starts by considering a broad design space, thus leading to an initial sampling that may not be adequate to describe the overall response surface. In this case, however, the new solutions can be exploited to retrain the ANNs, and then start again the research for optima, thus adopting an iterative approach.

The whole procedure ends when a satisfactory solution has been reached. As a further advantage of the approach, it is important to point out that the whole database, or better, the whole knowledge acquired during the process can be further exploited for additional designs/arrangements. It is well known that it is more a practice than an exception that a design is subject to successive revisions. If a new constraint or a slightly different target is asked for the design, the trained ANNs can provide almost in real-time a new solution matching the new requirements.

3. Numerical tools and case study

The main numerical tools adopted in the new design system are discussed in the present section. As already noted, however, these tools can be customized or replaced quite easily to match specific requirements or customized company standards. The last sub-section also briefly presents the MEXICO rotor used as a case study for the present work, and the tuning of the approach on this example case.

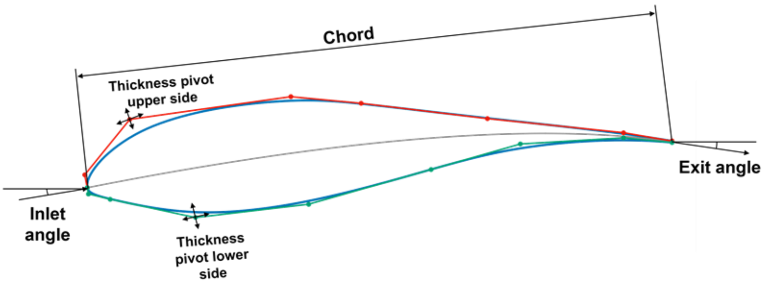

Figure 2: Parametrization parameters

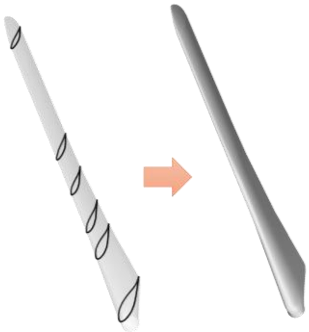

Figure 3: Blade CAD geometry

3.1. Parametrization

The geometry parametrization is a crucial phase of the whole procedure. Adopting an effective parametrization strategy, it is possible to express the complex geometry of a turbine blade airfoil through a limited set of design parameters. In the in-house parametrization tool, developed in the framework of the SUITE project [14], the 3D blade geometry is obtained by stacking a set of 2D sections along the spanwise direction. For each 2D section the blade airfoil is defined starting from a building curve called pseudo-camberline, on which the upper and lower thickness distributions are mounted. Figure 2 shows the layout of both the pseudo-camberline (grey-coloured curve) and the two sides of the airfoil (blue-coloured in the image) defined in terms of thickness distributions. The pseudo-camberline is defined by means of a B-spline curve where the number of control points can vary from 3 to 5, and these points can be expressed in terms of a set of design parameters such as inlet/exit angle, twist, chord, etc. The thickness distributions consist of B-spline curves with an arbitrary number of control points, expressed in terms of non-dimensional curvilinear abscissa and distance from the pseudo-camberline.

The 3D shape of the blade is therefore obtained by defining several "control" sections along the span or directly defining the spanwise distributions of the parameters (through B-spline curves). The CAD model is obtained by a loft among all the control sections. Special treatments are then devoted to parts such as the tip cover of the blade and the root portion connecting the first control section to the nacelle, that are usually generated through morphing procedures to match the specific shape adopted in these regions.

The CAD generation tool builds both a 3D blade geometry (see Figure 3) as well as models best suited for CAE analysis. In these latter ones, other components can be added (e.g., nacelle) and the fluid domain boundaries can be included. The process of generating the 3D CAD model from a prescribed set of parameters takes just a few seconds, when run on a single core.

3.2. CFD

The software used to generate the meshes and perform the aerodynamic simulation of the wind turbine geometries, is the commercial code STAR-CCM+. The mesh details, as well as the numerical setup to be considered for all the calculations must be defined at the beginning of the activity, by creating a “default” model. The current procedure was developed adopting a steady-state (i.e., RANS) approach, neglecting the influence of the tower on the aerodynamic performance of the rotor. The introduction of more complex models is straightforward but would clearly impact on the overall computational cost. RANS approach allows to limit the fluid domain to a single sector of the turbine, including only one blade and assuming cyclic symmetry conditions on the periodicity interfaces of the sector, thus reducing the computational cost of each simulation. Time-saving is essential when building an environment based on thousands of CFD runs as the one needed within the current approach.

3.3. FEM

The structural analysis of the blade geometries is carried out by means of the open-source FEM solver CalculiX, widely employed in many different applications also including wind turbines [15]. The discretization of the 3D blade has been performed using Gmsh [16], an open-source software able to build surface and volume meshes and export them in formats compatible with the most used opensource and commercial FEM solvers.

FEM analysis with CalculiX can focus on both the static and the dynamic behaviour of the blade, simply selecting the proper model during the computation setup. Furthermore, several kinds of loadings and constrains can be applied, allowing to simulate the blade structural behaviour as close as possible to its actual configuration. Finally, different types of materials can be modelled, ranging from traditional metallic alloys with an isotropic linear-elastic behaviour to more complex multi-layer composite materials with anisotropic properties.

3.4. MEXICO rotor and validation of the approach

An extensive validation of the CFD setup has been performed using the Mexnext Phase 3 experimental campaign data [17] on the MEXICO turbine. This validation is not necessary for the procedure itself but is reported here to underline the quality of the results that can be obtained with a full 3D approach, thus pointing out the potential benefit of adopting such method for blade design.

As discussed, calculations were carried out considering steady-state conditions and neglecting the influence of the tower. In the MEXICO turbine, indeed, the axial distance between blade tip and pylon is higher than the blade span, so that the tower is expected to have a limited impact on the blade performance.

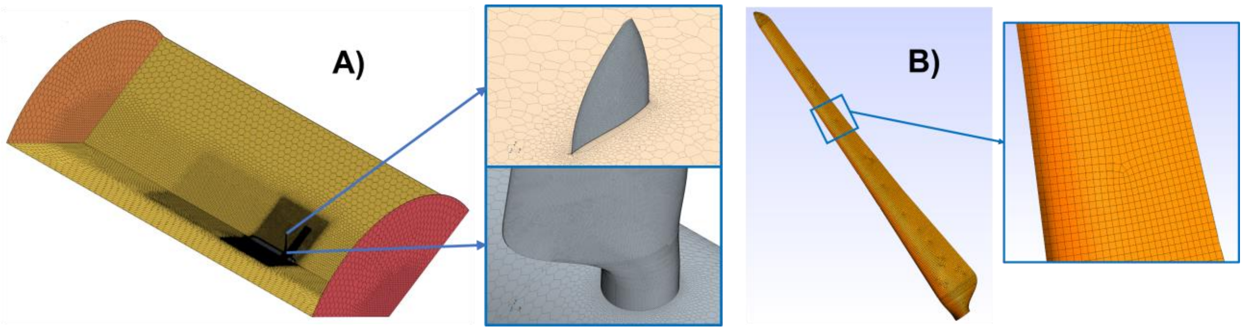

Figure 4: A) CFD computational domain and mesh; B) Structural mesh layout

The layout of the computational domain adopted for CFD simulation is displayed in Figure 4A. As shown in the image, the domain inlet and outlet interfaces are far upstream and downstream of the turbine blade, to ensure a proper simulation independence from the position of the boundaries. In particular, the inlet interface is located 4 blade spans upstream of the turbine, the outlet is located 8 blade spans downstream and the top interface is 4 blade spans away from the axis.
Polyhedral elements have been adopted to discretize the fluid domain (see Figure 4A), with an overall mesh dimension of around 6M cells, chosen as a trade-off between accuracy and computational time through a dedicated sensitivity analysis, not reported here for brevity reasons. The meshes feature a tailored refinement close to the blade, that guarantees a proper resolution in the fluid domain region where the highest gradients are registered. Some details of the adopted meshes on the blade are shown in the zoom boxes of Figure 4A.

As far as the numerical scheme is concerned, a segregated solver has been used with second order discretization for both the convective fluxes and energy equation. The k-ω SST model by Menter [18] has been adopted for the turbulence closure. A fully turbulent approach has been considered for the validation phase because the MEXICO turbine was experimentally investigated with turbulators forcing boundary layer transition immediately downstream of the leading edge. Coherently, fully turbulent conditions have been also considered for the optimization campaign. Different schemes, e.g., with transitional models, are at hand in case the application would require them.

The experimental data include both integral parameters (e.g., axial trust, torque) and distributions (e.g., blade loadings, velocity trend in the turbine axis direction, etc.) for 3 different operating conditions at the same rotational speed, allowing to have a wide evaluation of the CFD setup quality. Figure 5 reports a comparison between predicted and measured turbine torque outputs, for different values of turbine tip speed ratio (TSR). CFD simulations lead to a generally good prediction of the measured data in terms of both values and trend. The reference torque adopted for the plot is the CFD value obtained at the turbine design point (DP).

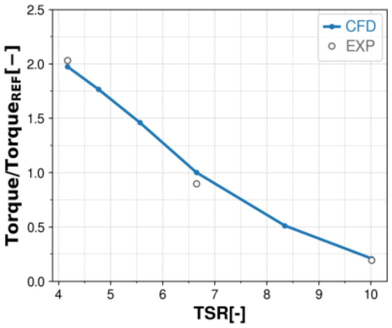

Figure 5: Torque output trend

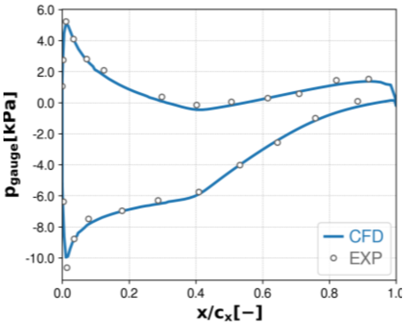

Figure 6: Blade tip loading at DP

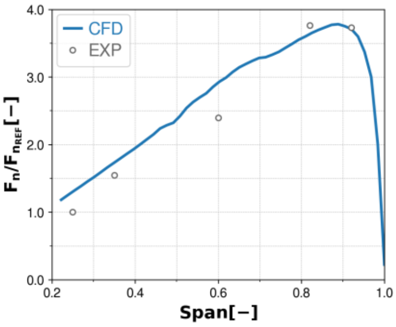

Figure 7: Normal force at DP

A comparison between CFD and measured blade loading, for a section close to the tip at the turbine design condition is presented in Figure 6. Numerical results show good agreement with experiments on both the suction side and the pressure side. Furthermore, the spanwise distributions of aerodynamic normal force is reported in Figure 7. The distribution tends to show a better agreement between CFD curve and experimental data getting close to blade tip section, while higher discrepancies can be seen in the lower part of the blade span, with the numerical prediction tending to slightly overestimate the forces exchanged between the fluid and the blade. As already underlined also by other researchers [17], these differences are probably connected to higher secondary flows in the test case with respect to the simplified CFD model adopted, due to non-stationary interactions. However, these results show an overall good agreement with experiments, thus proving the reliability of the CFD setup, in line with previous results on the same test case [17].

As far as blade structural analysis is concerned, a CalculiX setup fitted for wind turbine applications has been adopted. As wind turbine blades have a hollow structure and are extremely slender, the problem can be tackled using SHELL elements [19]. Therefore, in the pre-processing phase, a surface mesh consisting of second order quad elements, as the one shown in Figure 4B, is generated using Gmsh. The conversion of the surface elements of this mesh in shell elements is then performed directly by CalculiX, once the thickness of the layers is provided. The adopted surface meshes have around 60K elements. Again, the grid size was selected as a trade-off between accuracy and computational time after a sensitivity analysis, which is not reported here for brevity. A cantilever blade configuration with fixed root section has been used to perform the static analyses. Applied loads include a) pressure distribution on the blade coming from CFD analysis; b) centrifugal force; c) gravity, considering the worst-case configuration in which it is aligned with the tangential direction. The material used has been duralumin, an aluminium–copper alloy with enhanced mechanical properties.

4. Optimization campaign

As discussed, the new optimization/design environment has been tested on the case study of the MEXICO wind turbine, which represented the baseline (i.e., BSL configuration) for the optimization campaign.

The first step of the campaign consisted in the ‘fitting’ of the MEXICO geometry, performed using the parametrization tool presented in Section 3.1. The tool indeed features also a “fitting-mode” in which the parametrization parameters are automatically optimized to find the geometrical configuration that fits better a datum geometry. Amongst all parametrization parameters, only a limited amount has been picked to set the degrees of freedom (DOFs) used to define the design space of the optimization campaign. The campaign has been carried out using 6 control sections along the blade span (corresponding to the positions of the master airfoils of the MEXICO turbine), with 3 master sections characterized by 8 DOFs and 3 sections with just 2 DOFs. The chosen DOFs for the master sections are the inlet/outlet blade angle, twist angle, chord and position along the profile curvilinear abscissa and in the normal-like direction of the pivot thickness control points on the airfoil upper and lower side. An outline of the profile geometry with reference to all the DOFs of a master section, excluding the twist angle, is shown in Figure 2. The use of a pivot strategy to modify the thickness distributions allows to test very different configurations without the need for introducing too many DOFs. Once the pivot is moved, the other thickness control points of the distribution are in fact moved coherently with a scaling procedure. The other 3 control sections have only chord and twist as DOFs, while the remaining 6 parameters are derived from their master section. The use of these slave sections was chosen to reduce the overall amount of DOFs of the campaign, in a trade-off between computational cost and design space extension. From the above, the number of geometrical DOFs for the present optimization was 30. At the same time, the wind velocity was varied among different calculations, thus allowing the prediction of the turbine performance at different operating conditions. The range of variation in the wind velocity has been selected around the MEXICO turbine design condition.

As the optimization was more dedicated to proving the potentiality of the new approach than at a specific application, the target of the optimization was selected considering typical design targets that can be encountered for wind turbines. More in detail, the optimization was focused on the improvement of the turbine power output at design condition and at other TSR values in the range investigated in the experimental campaign of the MEXICO project, while keeping acceptable values of increase in maximum stress and tip displacement.

The range of variation of the DOFs has been chosen, based on previous experiences in similar applications, around the BSL configuration.

4.1. Clouds

In the presented optimization campaign, the design space has been sampled with 2000 CFD and FEM computations. Leveraging on the developed automatic procedure, each simulation required less than 30 minutes when run on 64 cores (AMD Epyc 7452 “Rome”), from the geometry generation to the post-processing of CFD and FEM results. Once all calculations were carried out, the database of numerical solutions has been used to train the surface-response meta-model based on artificial neural networks already introduced in section 2.1. The surface has been explored with data-analysis search-tools to look for optimal solutions using the following constraints: a) maximization of the turbine torque output; b) minimization of maximum stress and displacement on the blade; c) TSR close to the turbine design point value (i.e., 6.7). Setting these constrains in the search algorithm, a multi-dimensional pareto front was built. The predicted performance of these optimal geometries, found by the search algorithm, has then been verified through CFD and FEM analysis. The optimal solution has been picked between this set of geometries proposed by the meta-model, with a choice based on their verified aero-mechanical performance.

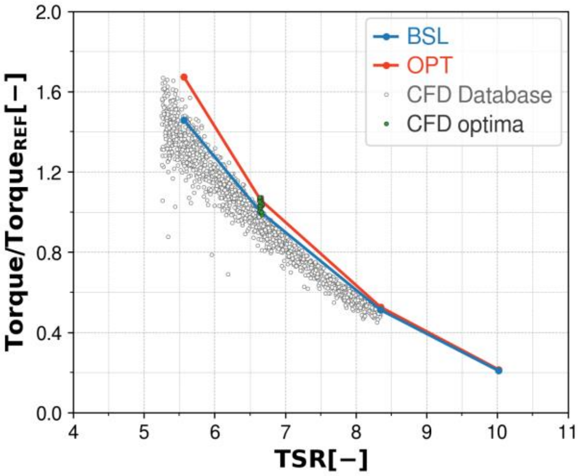

Figure 8: Torque trend at varying TSR

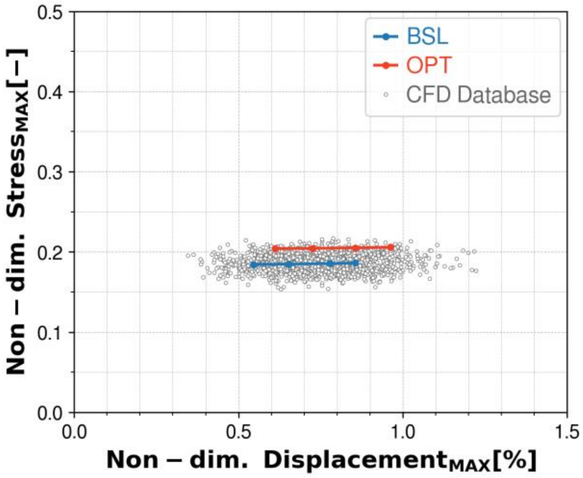

Figure 9: Max stress vs. max displacement cloud

The optimization clouds in terms of turbine torque output at varying TSR are reported in Figure 8. In the image, the geometries used to build the database are represented by the grey dots, while the geometries suggested by the meta-model and then verified through CFD/FEM analysis are reported as green dots. Finally, the curves of BSL and optimum (OPT) geometries are coloured blue and red, respectively. The reference value used to scale the plot y axis data is the output torque of the BSL geometry at the design condition, predicted by CFD analysis. As clearly visible from the abovementioned plot, the optimum curve at varying TSR presents values and slopes always higher than the ones of the original geometry. Therefore, the aerodynamic performance of the new geometry is significantly improved within all the operating range considered.

Regarding the structural behaviour, a plot reporting maximum equivalent stress on the blade against maximum displacement is shown in Figure 9. The plot includes the values registered for all the database geometries, and the curves for BSL and OPT geometries. The values of maximum displacement are scaled with the axial blade-tower clearance. The values of maximum equivalent stress are scaled with the yield stress of the material adopted for the FEM analysis. Both the values of stress and displacement of the OPT geometry are slightly higher than the ones of the BSL configuration, but the maximum stress is largely below the yield stress of the material and the maximum displacement increase is less than 3 millimetres.

4.2. *Optimum analysis*

A more detailed comparison between BSL and OPT results is presented in this section.

The increase in torque output registered for the OPT geometry with respect to the BSL one at varying TSR is reported in Figure 10A. An increase of approximately 6% has been reached for the turbine design condition, but aerodynamic performance improvements are registered for all the investigated operating conditions, with benefits increasing moving towards lower values of TSR (where an increase of more than 14% is obtained). It is important to underline that no specific attention was devoted to the operating curve of the current turbine (for which scarce data were available), as the optimization simply looked for an increase in torque output. The proposed approach, however, allows the introduction of more complex design targets in the case other conditions should be important for the design. As an example, it could be possible to introduce constraints or objectives considering a control strategy (e.g., stall control) as well as overall performance that integrate over a given wind distribution.

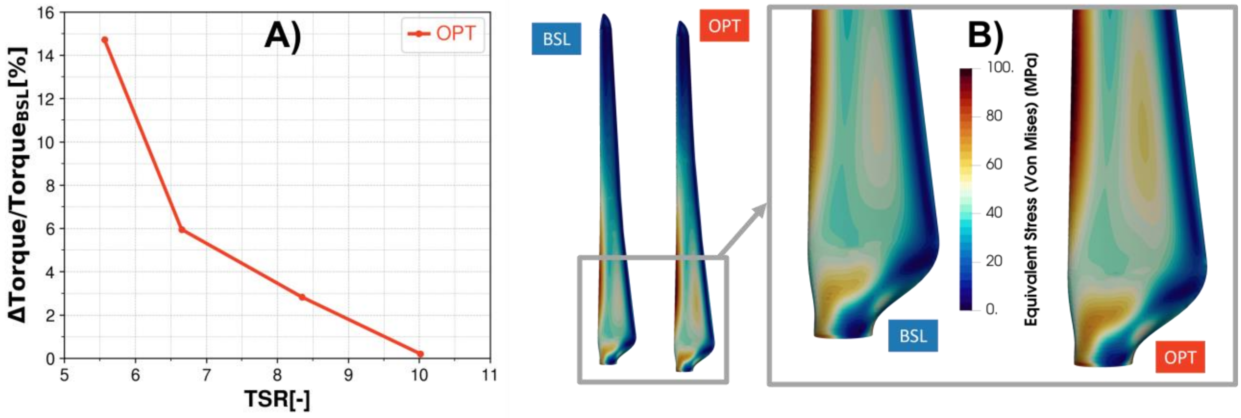

Figure 10: A) Torque output OPT vs. BSL; B) Equivalent stress maps (BSL vs. OPT)

A comparison of the blade loadings at different spanwise positions for the two configurations is reported in Figure 11. The OPT geometry tends to have a lower incidence towards hub and tip sections. Moreover, a slightly higher loading is observed after 40% axial chord for the distributions at a span over 50%.

FEM analyses then proved that such results are obtained with a small impact on the structural behaviour of the blade. An equivalent stress contour surface map, allowing a comparison between the structural behaviour of BSL and OPT blades is presented in Figure 10B. As a general comment, the two configurations share a very similar stress pattern. When considering the zoomed portion of the image, one can appreciate the actual differences, with the OPT map showing a slightly wider radial extension of the high-stress area at the leading edge and more extended intermediate stress areas at mid-chord and close to the blade root. After all, as reported in Figure 9, the maximum equivalent stress of the OPT blade is just few percent higher than the BSL one, and surely well far from any critical values (~20% vs 18% of the yield stress of the blade material). Analogously, the maximum displacement of the tip section increases of few millimetres, with practically no impact on the blade-tower clearance.

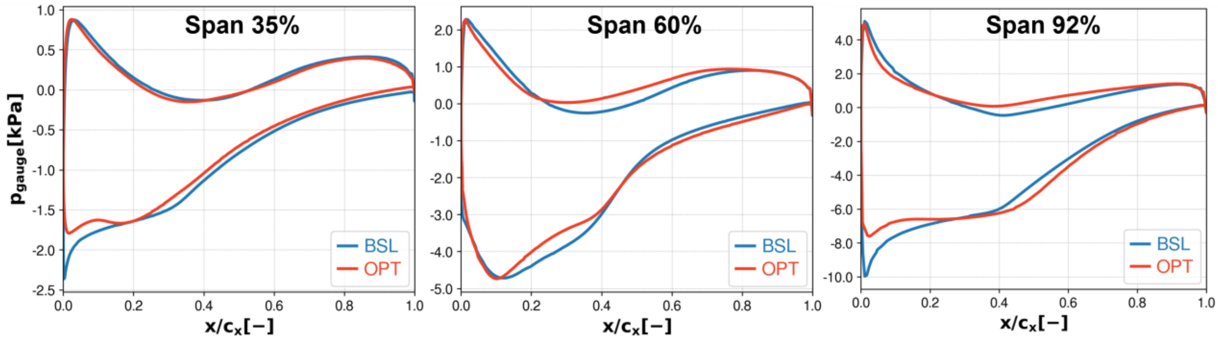

Figure 11: Blade loadings comparison between BSL and OPT geometries

5. Conclusions

The present study describes the realization of a new design and analysis platform for wind turbines. Currently, blade design only is included, but an extension to the entire turbine is at hand. The platform is modular and can include solvers for the aerodynamic and structural parts of different fidelity; these are connected through in-house pieces of software able to generate design spaces, analyse them with AI techniques and produce a response surface that can be then interrogated to find the optimum. In the current implementation, aerodynamics is solved with 3D RANS CFD while the structural analysis is addressed with 3D FEM. The new platform has been tested on the well-known MEXICO rotor to showcase its potential. After a validation of the methods against experiments, a blade optimization procedure has been carried out. Using only 2000 CFD and FEM computations, increases in blade torque have been obtained at each of the three tip-speed ratios (TSR) investigated, ranging from 6% at the nominal TSR to 14% at the lowest one, while stresses on the blade are kept almost unaltered. This has been done with HPC resources, adopting around 250 cores for less than 10 days. While still relevant, this computational cost is close to industrial feasibility, yet provides an overall accuracy that is much higher than conventional engineering methods. Moreover, once built, the design system can be further exploited as it is able to sample the design space practically in real-time, when searching for a new design or when different requirements are set for the blade during the final design phase. In turn, this proves the great potential of the proposed design platform, which will be further developed in the next future to make it become an accessible tool for wind turbine industry.

Acknowledgments

The data for the use-case have been supplied by the consortium which carried out the EU FP5 project Mexico: 'Model rotor EXperiments In COntrolled conditions'. The consortium received additional support to perform the New Mexico measurements from the EU projects ESWIRP and INNWIND.EU.
The research leading to the present paper has received funding from the Italian government (Regione Toscana) with the SUITE project (POR FESR Toscana 2014-2020, ASSE I – AZIONE 1.1.5 sub-azione A1).

References

[1] A. Bianchini, "Trends, Prospects, and R&D Directions in Wind Turbine Technology☆," in *Reference Module in Earth Systems and Environmental Sciences*, Elsevier, 2019.

[2] P. Veers, K. Dykes, E. Lantz, S. Barth, C. L. Bottasso, O. Carlson and al., "Grand challenges in the science of wind energy," *Science,* vol. 366, p. eaau2027, 2019.

[3] G. A. M. Kuik, J. Peinke, R. Nijssen, D. Lekou, J. Mann and al., "Long-term research challenges in wind energy -- a research agenda by the European Academy of Wind Energy," *Wind Energy Science,* vol. 1, pp. 1-39, 2016.

[4] S. Hewitt, L. Margetts and A. Revell, "Building a Digital Wind Farm," *Archives of Computational Methods in Engineering,* vol. 25, pp. 879-899, 11 2018.

[5] A. Dal Monte, S. De Betta, M. Raciti Castelli and E. Bernini, "Proposal for a coupled aerodynamic-structural wind turbine optimization," *Composite Structures,* no. 159, pp. 144-156, 2017.

[6] H. Wang, X. Jiang, Y. Chao and al., "Numerical optimization of horizontal-axis wind turbine blades with surrogate model," *IMechE Part A: J Power and Energy,* vol. 235, no. 5, pp. 1173-1186, 2020.

[7] P. Bortolotti, K. Dixon, E. Gaertner, M. Rotondo and G. Barter, "An efficient approach to explore the solution space of a wind turbine rotor design process," *Journal of Phisics,* p. 12, 2020.

[8] M. Giovannini, F. Rubechini, M. Marconcini, A. Arnone and F. Bertini, "Analysis of a LPT Rotor Blade for a Geared Engine: Part I — Aero-Mechanical Design and Validation," 2016.

[9] "Open Cascade," [Online]. Available: https://www.opencascade.com/.

[10] "CalculiX: A Free Software Three-Dimensional Structural Finite Element Program," [Online]. Available: http://www.calculix.de.

[11] "TensorFlow: An end-to-end open source machine learning platform," [Online]. Available: https://www.tensorflow.org.

[12] C. M. Bishop, Pattern Recognition and Machine Learning (Information Science and Statistics), Berlin, Heidelberg: Springer-Verlag, 2006.

[13] I. Goodfellow, Y. Bengio and A. Courville, Deep Learning, MIT Press, 2016.

[14] "SUITE: SvilUppo di un ambiente Innovativo per l'analisi e l'ottimizzazione di turbine eoliche basato su intelligenza arTificiale e tecniche per high pErformance computing," [Online]. Available: http://www.morfodesign.it/wp-content/uploads/2021/04/Poster_SUITE_v3.pdf.

[15] W. Finnegan, Y. Jiang, N. Dumergue, P. Davies and J. Goggins, "Investigation and Validation of Numerical Models for Composite Wind Turbine Blades," *Journal of Marine Science and Engineering,* vol. 9, 2021.

[16] "Gmsh: A three-dimensional finite element mesh generator with built-in pre- and post-processing facilities," [Online]. Available: https://gmsh.info.

[17] J. Schepers, T. Lutz, K. Boorsma, S. Gomez-Iradi, I. Herraez, L. Oggiano, H. Rahimi, P. Schaffarczyk, G. R. Pirrung, H. A. Madsen, W. Z. Shen and P. Weihing, "Final Report of IEA Wind Task 29 Mexnext (Phase 3)," 2018.

[18] F. R. Menter, "Two-Equations Eddy Viscosity Turbulence Models for Engineering Applications," *AIAA J.,* vol. 32, pp. 1598-1605, 8 1994.

[19] M. Bucalem and K. Bathe, "Finite element analysis of shell structures," *ARCO,* vol. 4, pp. 3-61, 1997.