Title:
Design Optimization of Customizable Centrifugal Industrial Blowers for Gas Turbine Power Plants

Authors:
Vincenzo Castorani, v.castorani@pm.univpm.it, Università Politecnica delle Marche
Daniele Landi, d.landi@staff.univpm.it, Università Politecnica delle Marche
Marco Mandolini, m.mandolini@staff.univpm.it, Università Politecnica delle Marche
Michele Germani, m.germani@staff.univpm.it, Università Politecnica delle Marche

Keywords:
Industrial Fan, Virtual Prototyping, Geometric Optimization, Genetic Algorithm, Response Surface Methodology

DOI: 10.14733/cadconfP.2018.174-178

Introduction:
The design process of any mechanical part requires rapid decision tools and methods to support engineering tasks. A quick and good product designing is achievable through virtual analyses, capable to investigate functional aspects and reduce the necessity to build physical prototypes and perform tests. This leads to lower cost, shorter time to market and an overall higher quality level. Virtual Prototyping (VP) tools allow to analyze, at the same time, different product features: geometry, kinematics, fluid dynamics, mechanical response, etc.

Fan is a type of machine used to move a fluid, typically a gas such as air, exploiting the kinetic energy of a rotating impeller. It consists mainly of two components: housing and rotor. Generally, it is powered by an electric motor, but also other sources can be exploited. It has been widely used in the most different industrial sectors. In particular, fans play a key role in a proper operation of turbo-gas power plants. Indeed, gas turbines need a large volume of ambient air during their running. Inadequate air supply can lead toward reduction in power output and overall gas turbine efficiency. The impeller plays a key role in a proper operation of turbo-gas power plants. Indeed, gas turbines need a large volume of ambient air during their running. Inadequate air supply can lead toward reduction in power output and overall gas turbine efficiency. The correct matching of impeller and casing is a decisive factor in design process. Volute area should not be too large or too small, as both conditions lead to pressure distortion

In the last decade, thanks to the fast development of computing power and technology, virtual prototyping - VP methods and tools have been broadly exploited. In industry, VP has become a fundamental analysis tool for engineering design [3]. Moreover, through design optimization methods, numerical simulations results can help in achieving optimal design.

Nowadays, the optimization of mechanical products has been evolved to the stage of automatic design optimization through the use of dedicated platforms, such as ModeFrontier® or ISight®. These software are able, in a single framework, to integrate and communicate with each other CAD/CAE/DfC (Design for Cost) tools with optimization algorithms and methodologies.

Optimization analysis have been playing a crucial role in developing competitive products. The optimization of an existing product can be dictated by several factors such as: regulation, market needs, customizations, company strategies, etc. However, the integration of an optimization phase inside the traditional design processes can increase delays and cost. This risk grows when complex software tools are used during product design. Generally, computational analysis tools seem to be a possible solution to reduce cost and time during project cycle. Nonetheless, a workflow methodology is required to support engineers in the decision-making process. Without a methodological approach, the use of virtual prototyping tools could be time expensive and onerous for the designers. Software like FEM (Finite

Elements Method) and FVM (Finite Volumes Method) tools require skilled and trained users: the traditional users have to achieve new technical and simulation competences in order to really improve their efficiency. Further difficulties emerge in the design of custom and complex products, where, the time to market, the cost and the high geometric dimensions make nearly impossible the creation of physical prototypes.

There is extensive literature on the design and optimization of industrial fan. The main focus is looking for the blade profile parameters with the best aerodynamic performance, minimum pressure loss and maximum working capacity under definite boundary conditions. As example, Lee and Lim [6] developed an optimized design for a centrifugal blower assessing different configurations of fan ribs and external case. They analyzed flow characteristics by using both numerical and experimental approaches. Hariharan and Govardhan [5] investigated an alternative kind of volute for an industrial centrifugal impeller. They made a detailed performance comparison between a parallel wall volute and its equivalent rectangular volute for different geometric configurations. The numerical simulation was performed through the use of Ansys® CFX®. Tang et al. [7] proposed an aerodynamic optimization of a transonic fan through CFD analysis. Their goal was to enhance the fan adiabatic efficiency. They used steepest descent optimization (SDO) and gradient-enhanced response surface model (GERSM) as optimization methodologies. Zhong et al. [8] presented a study aiming to optimize the operation of exhaust fans in a metro line. They found the best operating conditions with the aid of a one-dimensional simulation software. Baloni et al. [2] increased the performance of a centrifugal blower modifying the volute geometry. They coupled numerical simulation, carried-out with Ansys® FLUENT®, along with Taguchi method and ANOVA approach. Unfortunately, few researchers have addressed the problem of optimization of a centrifugal fan for air supply of a large-scale gas turbine. The main works cited, refer to a small or medium standardized fan, where it is possible to study a large number of parameters and perform many experimental tests.

In the gas turbine sector, for improving the overall efficiency of power plants, engineers require customized fans that finely satisfy the functional requirements. For large geometric dimensions (impellers of 2 meters of diameter and air flow of over 40000 [m^3/h]) the flow rate interval between two blower sizes could be too high for respecting the overall ventilation efficiency of the plant. For such a kind of blowers (large and customized) the physical experimentation is not feasible. Therefore, it is necessary to develop an approach for the rapid optimization of large customized centrifugal fans using virtual prototyping and optimization methods. In particular, the presented approach foresees Ansys® CFX® for the virtual prototyping and ModeFrontier® for the optimization, the latter based on the Response Surface Methodology (RSM) [1] and Genetic Algorithms (GAs) [4].

Methodological approach:

The approach presented in this paper, as shown in Fig. 1, integrates three different levels of analysis: parametric CAD model, virtual prototyping analysis and design optimization.

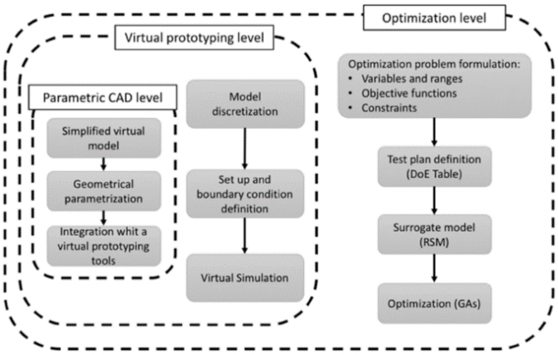


Fig. 1: Simulation and geometrical optimization approach.

The first step of the approach is the generation of the 3D CAD model of the fan. In this phase, engineers, by identifying the less important geometrical entities, reduce and simplify the geometrical complexity

Proceedings of CAD'18, Paris, France, July 9-11, 2018, 174-178

of the real product (as shown in Fig. 2). The resulting geometry is a closed volume which excludes all those elements with a low influence on mechanical behavior: through holes, threads, small fillets and chambers, electrical components, etc. At this level, the main geometrical and non-geometrical parameters have to be identified and a parametric simplified CAD geometry is created.

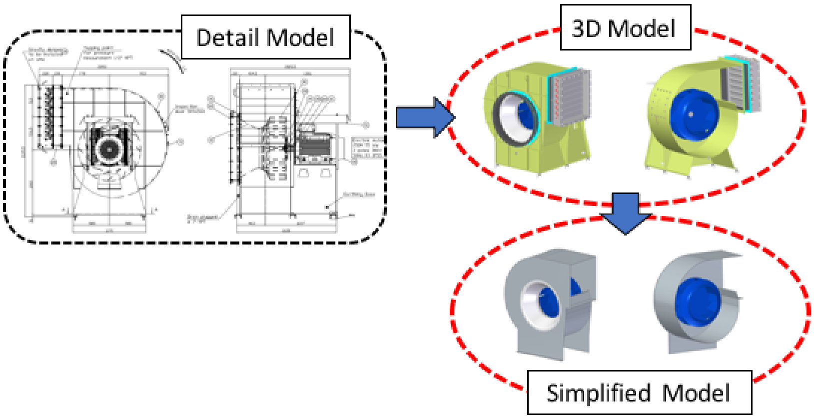


Fig. 2: Parametric CAD model of an industrial fan for gas turbine air supply.

The virtual prototyping level employs numerical simulation techniques to create a digital model of the product able to reproduce its behavior under certain working condition. Currently, Finite Element Method (FEM) is one of the most widely used method to perform virtual engineering analysis.

The simulation process is carried-out in three steps: pre-processing, processing and post-processing. During the pre-processing phase the finite element model is built: the CAD geometry is discretized and the set up and boundary conditions are defined. At the end of the numerical model processing, results are assessed (post-processing) Fig. 3. Results analysis is a necessary operation both to check simulation reliability and analyze product behavior.

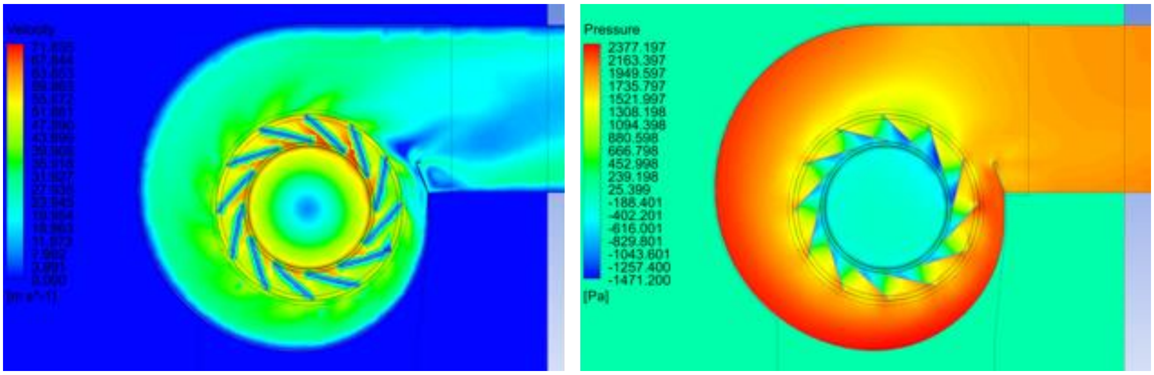


Fig. 3: Trend of velocity (left figure) and pressure (right figure).

The optimization process is based on virtual experiments and meta-modelling techniques according to the necessity of reducing time and costs during the design phase. The first step consists in the optimization problem formulation. This task is carried-out choosing the variables and the experimental domain to investigate, defining the goals to achieve and constraints to satisfy. Then, a DoE table provides the experimental plan for the virtual analyses.

Test case:
The applicative target concerns the fluid-dynamic optimization of an existing blower in order to increase ventilator efficiency (air flow rate maximization and absorbed power minimization). The optimization process, concerns the investigation of the wheel blades, while the fan volute and wheel dimensions are fixed for this application. The existing impeller has 12 backwards rectangular profile blades with an angle of 45 ° to the radial direction. The impeller is coupled with a 55 [kW] electric motor that allows an angular speed of 1480 [rpm] and a flow rate of 55000 [m^3/h].

A virtual test reproduces the behavior of a blower during the use phase applying the same boundary conditions. In this paper, ANSYS® CFX® was used to perform the CFD simulations considering air as an incompressible fluid. Each steady simulation calculates the air flow rate at a specific condition of rotational speed and pressure load. The CFD simulations were performed using k-ε equations in steady conditions and the MRF (Multiple Reference Frames) approach to describe the behavior of the rotating parts. After each simulation, a check is necessary to verify if the electric motor can produce the resultant torque related to the analyzed condition. Numerical simulations have been validated through experimental tests.

Fig. 4 shows a comparison between experimental tests and CFD simulations. The simulated curves of air flow (red line) can reproduce the real behavior of an air blower within a 5 % gap respect to real data. The same considerations can be made about the trend of the absorbed power (Fig. **4** right).

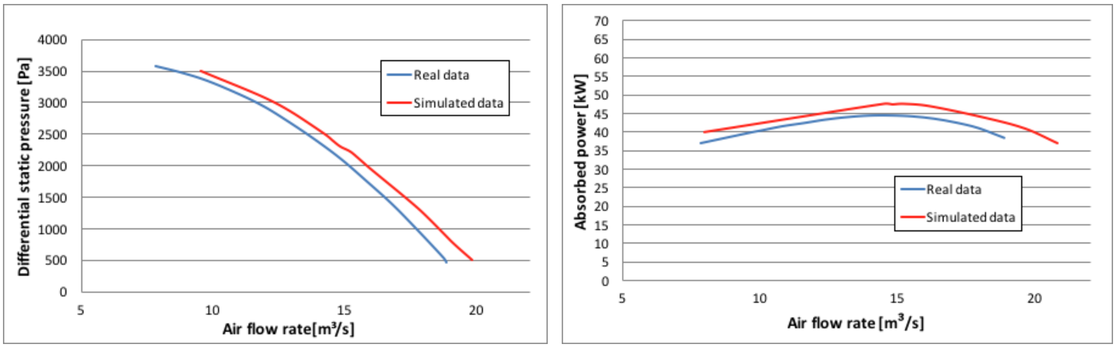
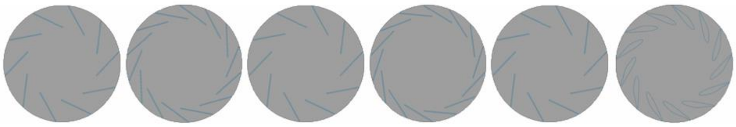


Fig. 4: Comparison between real (blue line) and simulated values (red line).

The blower design is focused on the impeller optimization. The proposed DoE optimization was based on the analysis of a L4 orthogonal array (Taguchi's method). Blower parameters are: functional, geometrical, and operational. The functional parameters are: the max air flow rate and the max static pressure. Rotational speed, torque and energy efficiency can be considered as operational parameters. Finally, the geometrical parameters concern the impeller dimensions such as: diameters, angles, thicknesses, and number of blades. This study was focused, as shown in Fig. 5, on three geometrical variables: blade angle (between 35° and 55° with 5° of step), blade number (between 10 and 14 with 1 of step) and two different blade profiles (-1 and 1). Dimensions such as diameters and thickness were considered as a constant. Tab. **1** shows the obtained results. The configuration n°10 represents the initial fan design. Three configurations with an efficiency greater than 90 % have been identified. Considering the minimum required flow was 16 [m^3/s], the optimal configuration is the number 6. This configuration, respect to the existing design, allows to 14 % enhance the air flow rate and decrease the absorbed power by a 5 %.



10 Blade *14 Blade* *35° angle* *55° angle* *-1 profile* *+1 profile*

Fig. 5: Blade geometrical variables.

Configuration	Blade angle	Blade number	Blade profile	Air flow rate [m^3/s]	Power [kW]	Efficiency [%]
1	35	10	-1	18	46,8	75,46
2	40	13	1	15,6	48,6	62,97
3	50	11	-1	14,3	35,6	78,8
4	55	14	-1	11,7	24,2	94,85
5	40	11	1	16	47,9	65,53
6	35	13	-1	19,4	41,3	92,16
7	50	10	1	13,6	34,4	77,56
8	50	14	-1	14,5	30,4	93,58
9	35	11	1	16	50,5	62,16
10	45	12	-1	16,7	43,4	75,49
11	55	10	1	11,4	27,3	81,92
12	40	10	-1	18,1	42,6	83,36
13	35	14	1	16,7	57,2	57,28
14	45	14	1	15,3	42,9	69,97

Tab. 1: DoE table.

Conclusions and future development:

This paper presented an approach for the optimization of customizable centrifugal industrial blowers for gas turbine air supply. The parametric 3D CAD model, which includes the impeller and its volute is the result of a simplification process. Using a CFD tool, the fluid dynamic performance for each planned experiment has been simulated. The optimization process is based on specific objective functions (air flow rate maximization and absorbed power minimization). The use of DoE method, RSM and Gas allows reducing the quantity of experiments, while the introduction of VP avoid the manufacturing of physical prototypes. The approach, used for a blower, enhanced the ventilator efficiency by 18 %.

References:

[1] Box, G. E. P.; Draper, N. R.: Response Surfaces, Mixtures, and Ridge Analyses, John Wiley & Sons, Inc., Hoboken, NJ, USA, 2007.

[2] Baloni, B. D.; Pathak, Y.; Channiwala, S. A.: Centrifugal Blower Volute Optimization Based on Taguchi Method, Computers & Fluids, 112, 2015, 72–78. https://doi.org/10.1016/j.compfluid.2015.02.007.

[3] Castorani, V.; Landi, D.; Germani, M.: Determination of the Optimal Configuration of Energy Recovery Ventilator through Virtual Prototyping and DoE Techniques, Procedia CIRP, 50, 2016, 52–57. https://doi.org/10.1016/j.procir.2016.05.019.

[4] Goldberg, D. E.: Genetic Algorithms in Search, Optimization and Machine Learning, 1st ed., , Addison-Wesley Longman Publishing Co., Inc., Boston, MA, USA, 1989.

[5] Hariharan, C.; Govardhan, M.: Improving Performance of an Industrial Centrifugal Blower with Parallel Wall Volutes, Applied Thermal Engineering, 109, 2016, 53–64. https://doi.org/10.1016/j.applthermaleng.2016.08.045.

[6] Lee, Y. T.; Lim, H. C.: Performance Assessment of Various Fan Ribs inside a Centrifugal Blower, Energy, 94, 2016, 609–22. https://doi.org/10.1016/j.energy.2015.11.007.

[7] Tang, X.; Luo, J.; Liu, F.: Aerodynamic Shape Optimization of a Transonic Fan by an Adjoint-Response Surface Method, Aerospace Scienze and Technology, 68, 2017, 26–36. https://doi.org/10.1016/j.ast.2017.05.005.

[8] Zhong, L.; Tong, Y.; Zhang, Z.; Du, A.; Wang, Y.: Optimization of Operation of Exhaust Fans in Nanjing Metro Line 10, Procedia Engineering, 205, 2017, 773–78. https://doi.org/10.1016/j.proeng.2017.10.009.