



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

Vincenzo Castorani¹ , Daniele Landi² , Marco Mandolini³  and Michele Germani⁴ 

¹Università Politecnica delle Marche, v.castorani@pm.univpm.it

²Università Politecnica delle Marche, d.landi@staff.univpm.it

³Università Politecnica delle Marche, m.mandolini@staff.univpm.it

⁴Università Politecnica delle Marche, m.germani@staff.univpm.it

Corresponding author: Daniele Landi, d.landi@staff.univpm.it

ABSTRACT

Nowadays, design processes demand agile and flexible tools and methods to meet market needs. Virtual prototyping techniques are widespread in design strategies and practices, because these technologies reduce the project development lead-time and cost related to physical prototyping. The aim of this paper is the study and application of an approach for the modeling, simulation and geometrical optimization of fans for gas turbine air supply. 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. There is extensive literature on the design and optimization of industrial fan, but main works refer to a small or medium standardized fan, where it is possible to study many parameters and perform many experimental tests. The paper presents an approach for the efficiency optimization of large and customizable centrifugal industrial blowers for gas turbine air supply. The design variables investigated in this study were the blades quantity, orientation and shape. The proposed optimization method has been used for the design optimization of a blower for gas turbine power plant. The response surfaces allowed defining correlation between design variables and efficiency. The optimized design was 18 % more efficient than the original one.

Keywords: Industrial Fan, Virtual Prototyping, Computer Aided Engineering, Geometric Optimization, Genetic Algorithms, Response Surface Methodology.

DOI: <https://doi.org/10.14733/cadaps.2019.1098-1111>

1 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.

Centrifugal blowers are energy consuming machines 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. During operation, the fluid is collected either by a volute or a series of diffusing passages and leaves impeller at high velocity and pressure. The proper design of a volute efficiently fulfills primary function of flow collection and conversion of kinetic energy into pressure energy, before delivering the fluid to the exit of a turbomachine. The volute does not take part in the head generation and consequently all theoretical considerations of casing design deal with losses. The impeller is the rotating element that transfers energy to the fluid. The major working surface of the rotor is the blade. Blade forms include backward-inclined (BI), radial, and forward-curved (FC) blades. 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 [14,15,20].

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 [7]. 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) require skilled and trained engineers: 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 strict time to market, the high manufacturing cost and the overall dimensions make nearly impossible the creation of physical prototypes.

In literature several simulation and optimization methods have been applied in the design of mechanical components. Examples of impeller design have been studied in different applications such as: turbines, compressors, pumps, air-conditioning systems, household appliances and so on. 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 [4]. 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 using dedicated platforms, such as ModeFrontier® or ISight®. This software are able, in a single framework, to integrate and communicate with other CAD/CAE/DfC (Design for Cost) tools with optimization algorithms and methodologies.

There is an extensive literature on the design and optimization of industrial fans. The focus is looking for the blade profile parameters with the best aerodynamic performance, minimum pressure loss and maximum working capacity under definite boundary conditions [5]. As example, Lee and Lim [12] 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 [9] 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 using Ansys® CFX®. Kesare and Swami [10] reached the optimal configuration of a centrifugal blower for food storage by means of FEA method. They investigated the blower static and modal response considering different materials. Tang et al. [17] 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. [21] 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. [1] 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 optimization problem for centrifugal fans used in large-scale gas turbine. Literature refers to small or medium standardized fans, where it is possible to study many 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 diameter greater than 2 [m] and air flow over 40000 [m³/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 and efficient optimization of large customized centrifugal fans using virtual prototyping and optimization methods. The presented approach foresees Ansys® CFX® for the virtual prototyping and ModeFrontier® for the optimization, the latter based on the Response Surface Methodology (RSM) [2] and Genetic Algorithms (GAs) [8].

2 METHODOLOGICAL APPROACH

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

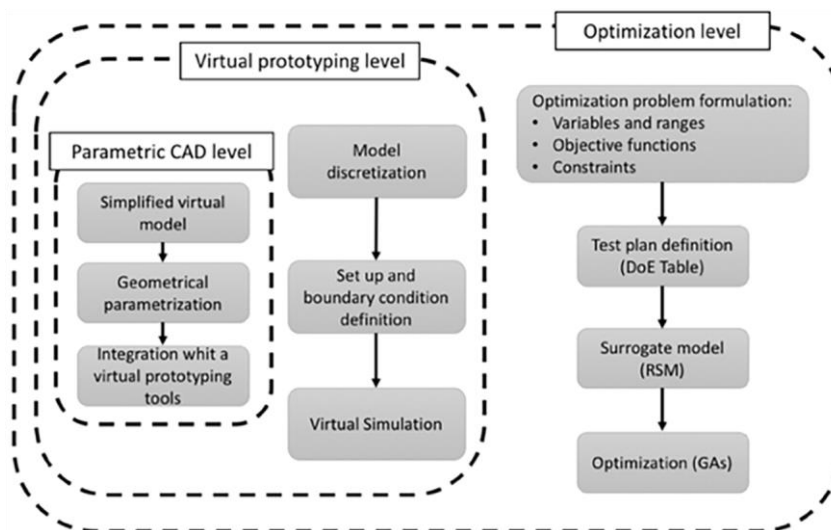


Figure 1: Modeling, simulation and geometrical optimization approach.

2.1 Parametric CAD level

The first step of the approach is the generation of the 3D CAD model of a centrifugal blower. In this phase, engineers, by identifying the less important geometrical entities, simplify the geometrical complexity of the physical product. This operation is performed using mechanical 3D CAD tools. The computational effort for performing the simulations depends on the shape complexity of the CAD model. Even the presence of a single and relatively small geometric detail (feature) can increase the size of the underlying discrete physical simulation problem by as much as 10-fold [11,18]. The computational time to perform a finite element analysis on a part with hundreds of small features, as shown in Figure 2., could be very long. Extremely large computational times may limit the usefulness of simulations during the design cycles. Complex models may often lead to ill-conditioned matrices and hence working with non-simplified complex models may produce inaccurate results [13]. The use of more and more powerful processing units is not an answer to the problem associated with highly complex models. To get accurate results in a timely manner, simulations should be performed using simplified models that retain the only important details. In general, there is no standard procedure for model simplification, but this phase is performed according to the objective of the subsequent virtual simulation. For example, structural analysis provides a different simplification compared to thermos-fluid dynamic or modal analysis [18].

In this research study, the goal of numerical models is to simulate the fluid-dynamic behavior of the fan and to measure the efficiency of the product. Therefore, the resulting geometry is a closed volume which excludes all those elements with a low influence on the fluid dynamic behavior: through holes, threads, small fillets and chambers, electrical components, etc. At this level, the main geometrical and non-geometrical parameters have been identified and a parametric simplified CAD geometry was created. The determination of the CAD design variables will be discussed in the next paragraph optimization level.

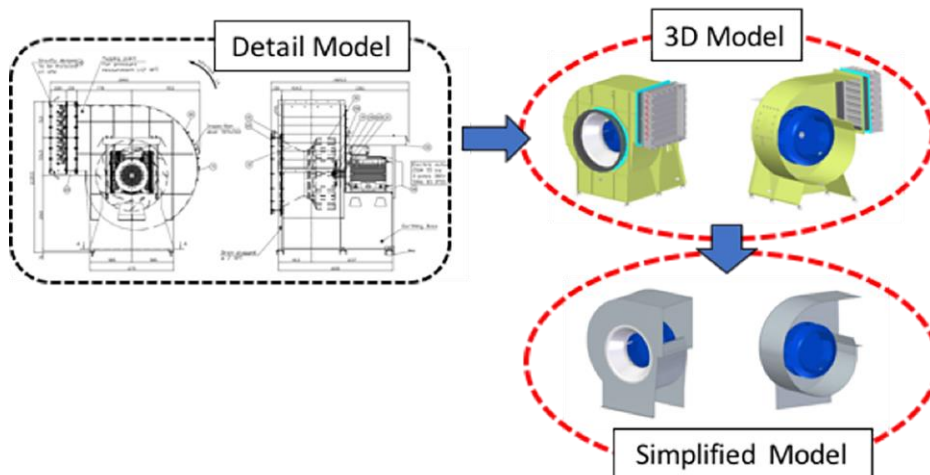


Figure 2: Parametric CAD model of an industrial fan for gas turbine air supply.

2.2 Virtual prototyping level

The virtual prototyping level employs numerical simulation techniques to create a digital model of the product able to reproduce its behavior under certain working conditions. Currently, FEM or FVM are two of the most widely used methods to perform virtual engineering analysis.

VP uses a digital model called "virtual prototype" for testing and evaluating specific characteristics of a product/manufacturing process in a computational environment. Therefore, in virtual prototyping, faults concerning fabrications, product design and production planning can be

detected in a compressed time frame before great expenditures are committed. This advantage significantly reduces the number of physical iterations and thereby the associated manufacturing overheads that leads to faster and cost-effective product development. Once the virtual prototyping is created, the model may be sent directly to physical fabrication or to customers for the approval/comments. Since digital models are mostly used in VP, the cost incurred in repeating the process to optimize prototype quality is minimal. Through VP method it is possible to speed-up the design process and reduce/avoid fabrication of physical prototypes.

Independently by the adopted software tool, the simulation process is carried-out in three steps: pre-processing, processing and post-processing. During the pre-processing phase, the numerical model is built: the CAD geometry is discretized, while set up and boundary conditions are defined. The set-up phase involves defining the type of simulation (structural, fluid-dynamic, modal etc.), the principal equations of the model, and the operational conditions. The processing phase consists in the resolution of the equation defined in the previous step. The last step of VP level is the post-processing. In this phase the results obtained from the virtual simulation are collected and processed. Each physical quantity can be displayed both as a graphical trend (plane contour, vector, streamline etc.) and as a numerical table. The virtual prototyping level includes always the CAD level. However, it is not always possible to have two distinct levels (CAD and VP) because it is possible to realize the geometries inside the VP environment. The integration with a parametric CAD tools is fundamental for the interaction between different designers.

2.3 Optimization level

Companies can experience great benefits in integrating and exploiting methods and tools for Multi Objective Optimization (MOO). The use of MOO to increase product performances while respecting design requirements and constraints is a crucial aspect for achieving a competitive product.

In this research study, 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 a proper formulation of the optimization problem.

Optimization problem formulation is carried-out following these actions:

1. definition of design variables and their bounds;
2. statement of the objective function;
3. formulation of the design constraints.

Then, a Design of Experiments - DoE table provides the experimental plan for the virtual analyses.

The DoE [3] is a statistical technique for the design and organization of experiments that enables engineers to gain as much product knowledge as possible with the least consumption of resources (i.e. with the minimum amount of experiments). The experimental plan is characterized by tests where the values of the design variables are changed simultaneously on the contrary of the traditional One Factor At a Time - OFAT approach. In literature numerous DoE strategies have been proposed; for the aim of this paper, the Taguchi method [16] is suggested.

Once performed the numerical simulations, according to the DoE plan, the Response Surface Methodology - RSM [2] is used to build the relationship between variables and objective functions. RSM consists of a group of statistical and mathematical methods able to perform thousands of design evaluations in short time accelerating the optimization process. It can use a small amount of data efficiently with a smart exploitation of the available computational resources. Typically, the approximate functional relationship between design and response variables is achieved through a second order polynomials expression given by:

$$y = f(x_1, x_2, \dots, x_n) + \epsilon \quad (2.1)$$

where y is the response, f is the response function, x_1, x_2, \dots, x_n are the design variables, n is the number of design variables and finally ϵ is the statistical error. Generally, a normal distribution with mean zero and variance is assumed for ϵ .

After the generation of the response surface, a suitable optimization algorithm has to be chosen to find the optimal solution.

In this paper, genetic algorithms - GAs [8] are used to solve the optimization problem. The genetic algorithms are very general algorithms that can be applied to all different types of problem: continuous, discrete, non-differentiable. Moreover, these methods are easy to use and to program and allow the achievement of global optimum solutions. GAs are based on Darwin's theory of natural selection: "*survival of the fittest*". The main phases of genetic algorithms can be synthesized in the following steps (as shown in Figure 3.):

1. Initial population generation: this step starts with the definition of an initial population (design points are generated using the allowable values for each design variable) and of a fitness function;
2. fitness evaluation: fitness values are evaluated and assigned to the respective design solutions;
3. reproduction: a set of design solutions are selected from the current population and passed to the next one;
4. crossover: picked elements of the new generation exchange characteristics among themselves;
5. mutation: this process avoids that the reproduction and crossover phases result in a loss of valuable genetic materials.
6. if the stopping criterion is met the optimization is concluded and the optimum is found.

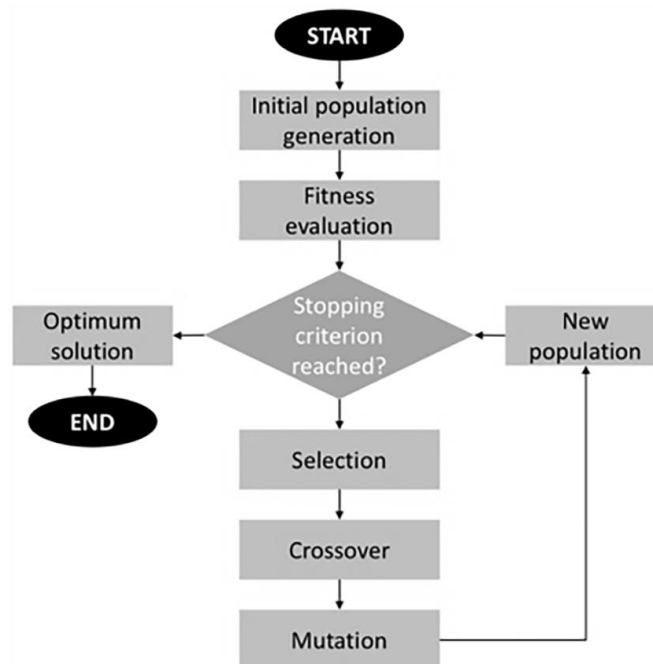


Figure 3: Genetic algorithms sequence.

In this research, NSGA-II algorithm has been used due to its high efficiency [8]. NSGA-II uses a smart multi-search elitism able to retain very good solutions without causing premature convergence into local-optimal frontiers [9].

3 CASE STUDY

This section presents a case study concerning the efficiency optimization of a large and customizable centrifugal industrial blower for air supply of a gas turbine. The fan efficiency optimization will be reached through fluid-dynamic optimization. The starting point is the 3D CAD model. In this case, Solid-Edge® by Siemens® was used to model the control volumes. The surfaces have been made as "smoother" as possible, to allow prism layers to be extruded from the surface in the next mesh phase. Where possible, sharp angles should be replaced with radius fillets of 2 [mm]. Details smaller than a millimeter have been eliminated. Holes, threads, structural reinforcements external to case have been removed.

Three control volumes and related interfaces have been created, i) the volume that contains the negative of the impeller (the volume in which rotation will be applied), ii) the volume that represents the air inside the case, iii) the volume that represents the surrounding environment. For the latter, there is no defined standard dimensions, but the volume depends by the specific case study. For each type of simulation there are best practices, deriving from company know how, that define the external environment dimensional standards. In this type of simulation, the dimensions of the external computational volume should be at least 3 blower lengths in front and behind.

The numerical simulations have been performed using commercial CFD code ANSYS® CFX® 18.2. The computational domain includes the external environment where the fan sucks the air, fills 3-D volute and impeller and the outlet duct. A structured tetrahedral mesh was generated using ANSYS mesh (as shown in Figure 4.). To reduce numerical diffusion during simulation, high quality mesh was created at all domains. A grid independency check has been performed for the fan's mesh. It was found that extending the cells count above 3,526,536 does not significantly influence final results.

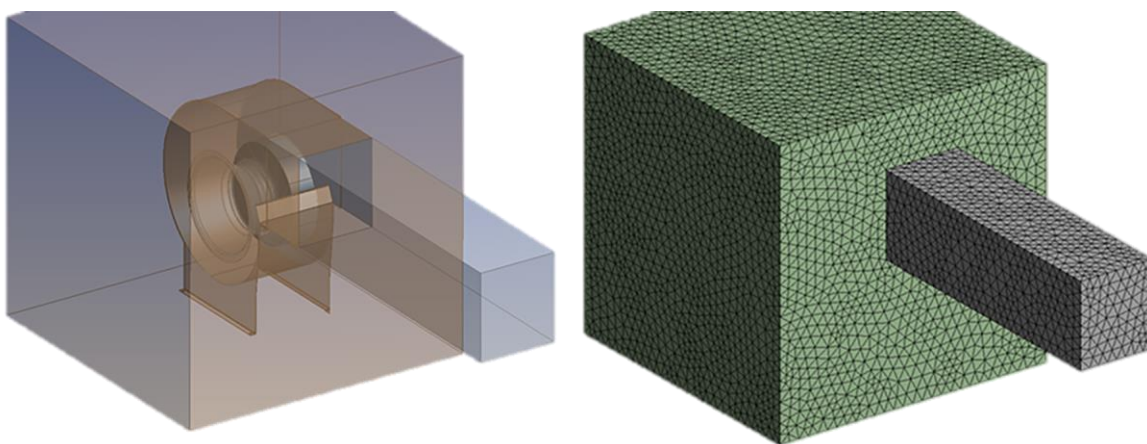


Figure 4: Geometrical model (left figure) and mesh (right figure).

Figure 5. shows two of the most important mesh quality indicators: orthogonal quality and skewness coefficient. To achieve a high-quality discretization, the elements must assume orthogonality quality values close to 1 and skewness coefficients close to zero. The average orthogonal quality, for the considered mesh, was of 0.87, while the skewness was of 0.17.

Steady state simulations were carried out for five different values of impeller rpm and outlet pressure, to simulate the different operation conditions and build the fan's characteristic curve. The continuity, energy and Navier-stokes equations were solved using a second order discretization technique. The equations for mass, momentum and energy conservation can be stated as follows in a stationary frame:

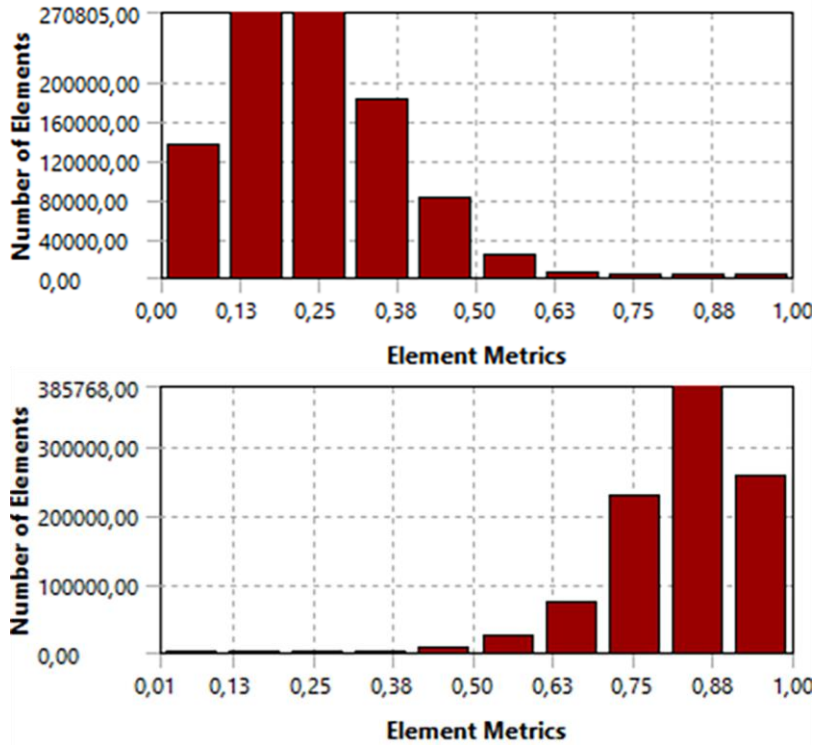


Figure 5: Mesh quality, skewness coefficient (top) and orthogonal quality (bottom).

$$\text{Continuity equation:} \quad \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad (3.1)$$

$$\text{Momentum equation:} \quad \frac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + \nabla \tau + \mathbf{S}_M \quad (3.2)$$

$$\text{Total energy equation:} \quad \frac{\partial(\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{U} h_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mathbf{U} \tau) + \mathbf{U} \cdot \mathbf{S}_M + \mathbf{S}_E \quad (3.3)$$

where ρ is the fluid density, \mathbf{U} is the vector of velocity, p is the static pressure, τ is the shear stress, \mathbf{S}_M the momentum source, h_{tot} the specific total enthalpy, λ is the thermal conductivity, T the static temperature and \mathbf{S}_E the energy source.

The high speed turbulent flow inside the volute is captured using k - ε turbulence model and the MRF (Multiple Reference Frames) approach to describe the behavior of the rotating parts. k is the turbulence kinetic energy defined as the variance of the fluctuations in velocity. ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate). Transport equations, for k - ε model, are:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho k U_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon + P_{kb} \quad (3.4)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \varepsilon U_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon + C_{\varepsilon 1} P_{\varepsilon b}) \quad (3.5)$$

where μ is the dynamic viscosity, μ_t is the turbulent viscosity, σ_k turbulence model constant for the k equation, σ_ϵ turbulence model constant for the ϵ equation, P_k the shear production of turbulence, P_{kb} and $P_{\epsilon b}$ represent the influence of the buoyancy forces and $C_{\epsilon 1}$ and $C_{\epsilon 2}$ the Reynolds stress model constants.

Due to compressible nature of the flowing air, ideal gas assumption was included. Total pressure inlet and mass flow outlet boundary condition were imposed. The interface between impeller and volute were facilitated by frozen rotor technique. All the walls were considered as adiabatic and the required y^+ value of 25 for k - ϵ turbulence model was maintained. The convergence criteria for simulation were sets at sixth order of residuals, as well as the key variables like pressure at outlet, mass flow at inlet were monitored. At the end of the numerical model processing, results were assessed (post-processing). Table 1 shows torque and power values for different outlet pressure at 1480 rpm, while Figure 6 and Figure 7 the trend of velocity and pressure for the Point3.

	Point1	Point2	Point3	Point4	Point5	Point6
Inlet pressure [Pa]	0	0	0	0	0	0
Outlet pressure [Pa]	500	800	1300	2600	2900	3500
Different static pressure [Pa]	500	800	1300	2600	2900	3500
Air flow [m ³ /s]	19,3	18,4	17,4	14	13,8	10,6
Torque [Nm]	289	296	305	303	305	305
Power [kW]	44,7	45,9	47,3	47	47,2	41,3

Table 1: Torque and power for different value of outlet pressure and 1480 rpm.

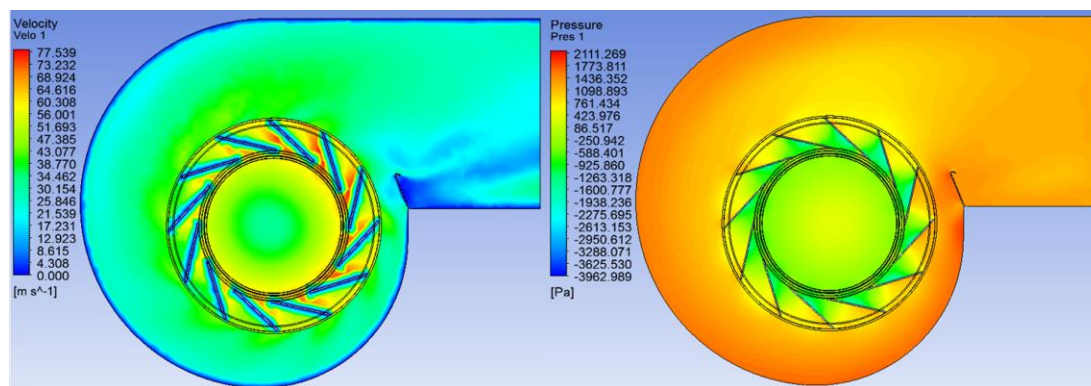


Figure 6: Trend of velocity (left figure) and pressure (right figure).

To ensure the goodness of CFD virtual simulations, the characteristic curves of the fan were compared with the experimental curves. In Figure 8. it is possible to observe a good correspondence between the real model (blue line) and the virtual model (red line). The maximum error between the two models is less than 5%.

The methodological approach was applied for the fluid dynamics optimization of an existing blower to increase ventilator efficiency. ModeFrontier® was used as tool for the optimization analysis. The considered objective function involves the fluid-dynamic efficiency maximization. The fan performance is represented by its discharge volume-flow-rate, static pressure, and static efficiency

at a rotation speed. Thus, these parameters are the primary measuring variables. The static efficiency can be determined by:

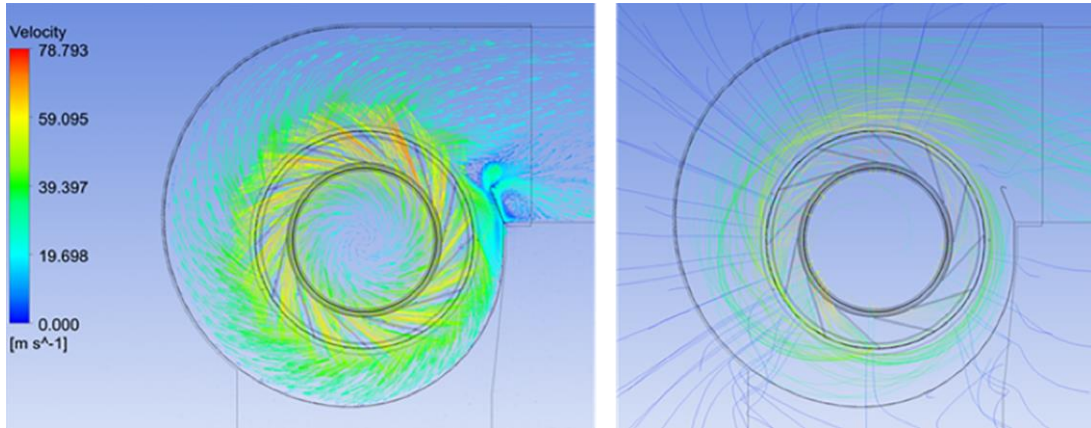


Figure 7: Trend of velocity, vector (left figure) and streamline (right figure).

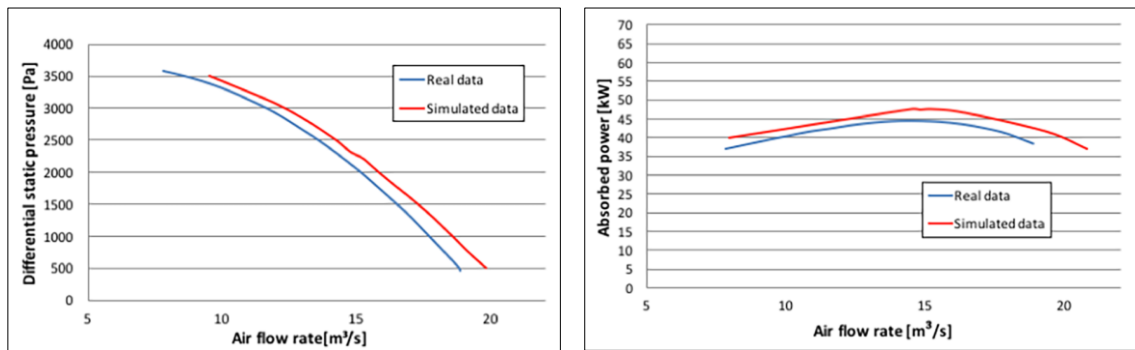


Figure 8: Comparison between real (blue line) and simulated values (red line) for the differential static pressure (on the left) and for the absorbed power (on the right).

$$\eta = \frac{P_s Q}{W} \quad (3.1)$$

where W is the fan input power, P_s is the static pressure, and Q is the air volume flow rate.

In this analysis, the rotation speed was fixed at 1480 rpm. In addition, two constraints were posed: the minimum air flow rate and the maximum absorbed power. These constraints were the inputs data necessary for the proper operation of the gas turbine. The minimum air flow required was 16 [kg/s], while according to the available electrical motors, the allowable maximum absorbed power was 45 [kW].

The optimization process, described below, concerns the investigation of the wheel blades, while the fan volute and wheel dimensions were fixed for this application. The existing impeller has 12 backwards rectangular profile blades with an angle of 45 ° to the radial direction. The impeller was coupled with a 45 [kW] electric motor that allows an angular speed of 1480 [rpm] and a flow rate of 55000 [m³/h].

In this study, the blower design was focused on the impeller optimization since the blade geometry has an important role in the performance of this unit. When a centrifugal blower works,

the kinetic energy of rotating blades increases the air pressure against the fluid-dynamic resistance. The proposed DoE optimization was based on the analysis of a L4 orthogonal array (Taguchi's method). Blower parameters can be divided in three groups: functional, geometrical, and operational. The functional parameters were: the maximum air flow rate and the maximum 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 Figure 9., on three geometrical variables: blade angle (between 35° and 55° with 5° of step), blades quantity (between 10 and 14) and two different blade profiles (A and B). Dimensions such as diameters and thickness were considered constant.

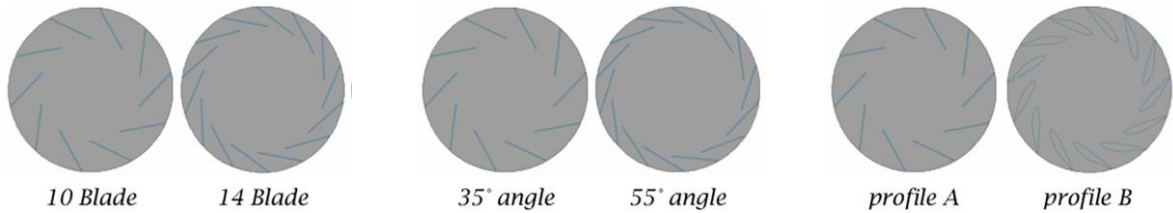


Figure 9: Blade geometrical variables.

3.1 Results

In accordance with the presented approach, Tab. 1 shows the results obtained from the numerical simulations of the DoE. The configuration n° 10 represents the initial fan design (efficiency = 75%). Tab. 1 highlights the availability of many other design solutions with a higher efficiency.

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

Table 1: DoE table.

The DoE results have been used to create the response surfaces. The polynomial SVD algorithm has been chosen to determine the correlation between design variables and objectives. Figure 10. shows

the response surface of the efficiency as function of blade angle and blades quantity (on the left for the profile A and on the right for the profile B). Efficiency increases with blade angle while decreases with blade number. The blade profile A guarantees higher efficiency than the profile B (+ 10 % considering the maximum reached value).

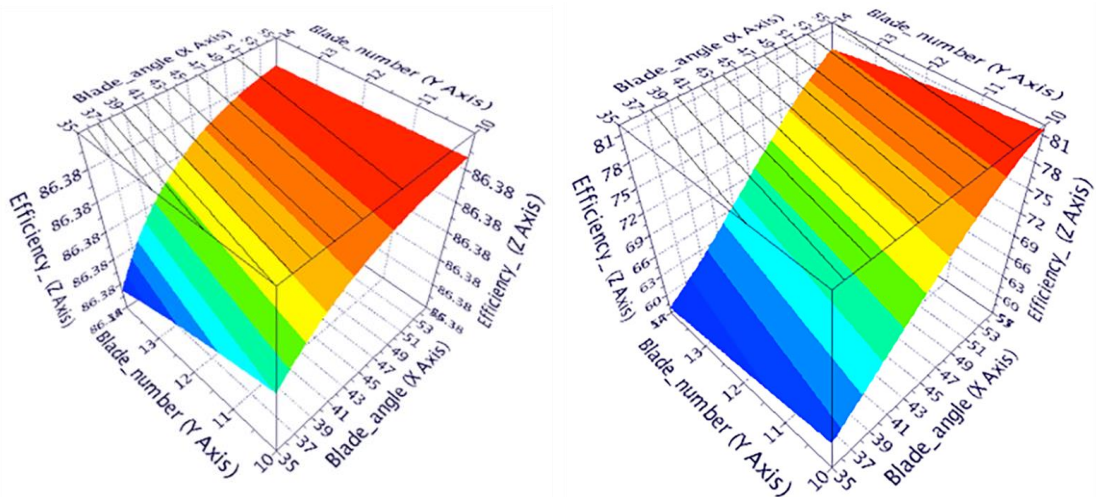


Figure 10: Trend of the efficiency as function of blade number and blade angle (on the left for the profile A and on the right for the profile B).

The optimal configuration has been researched, through NSGA-II algorithm, only among the fans with profile A, as this kind of profile allows to reach, for all the investigated conditions, higher efficiencies. Considering the posed constraints (minimum air flow and allowable power), the optimal configuration is made by 14 blades with an angle of 55° (in Figure 11. the green line highlights the optimal configuration). This configuration, with respect to the existing design, allows to enhance by 18 % the efficiency and by 14 % the air flow rate and decrease the absorbed power by a 5 %. With the same electrical motor used in the initial model it is possible to estimate a reduction in operating costs of around 10%.

In addition to increase the fluid-dynamic efficiency of the fan, which will lead to a lower operating cost, the proposed approach makes possible to simplify the design phase, reducing time and cost. Exploiting the achieved correlations (Figure 10. and Figure 11.), it is possible to evaluate, in a fast and easy way, the real performance of the fan as function of geometrical parameters. Moreover, thanks to the virtual analysis, it is possible to enhance the product knowledge avoiding/reducing the physical prototyping phase. It was estimated a time and cost reduction of about 10 %.

4 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 virtual analysis has been compared with real experiments obtaining an acceptable average error related to the calculation of the air flow rate, the resistance torque and power. The optimization process was based on one specific objective function: efficiency maximization.

The use of DoE technique, RSM and GAs allowed reducing the quantity of experiments, while the introduction of VP avoid the manufacturing of physical prototypes. The use of a DOE approach allows designer to investigate several geometry configurations with the possibility to simulate all geometry

variation in a parallel analysis. This approach gives a working methodology for designers in order to achieve a pragmatic study of operation parameters in the virtual analysis for reducing cost and time in the design phase. The approach, used for a blower, enhanced the ventilator efficiency by 18 %. In addition, it was estimated a time and cost saving during the design phase of about 10 %.

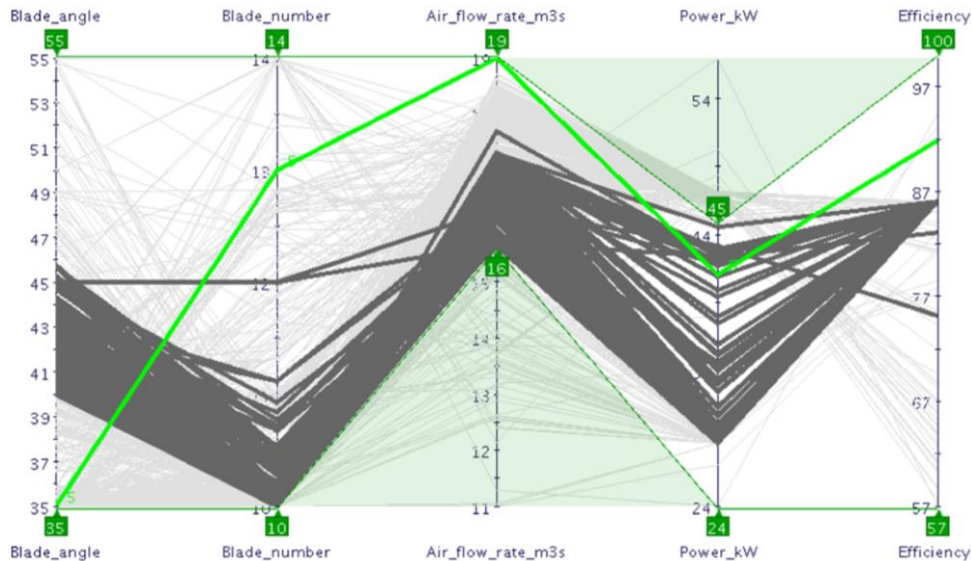


Figure 11: Optimal fan design.

Future works should focus on taking into consideration other design aspects and variables: mechanical, modal analysis, noise, weight, costs etc. The fluid-dynamic model could be refined for reducing the estimation error. Moreover, to give a general validation to the proposed approach, it is necessary to test different kind of customizable industrial centrifugal blowers than that one presented in this paper.

Vincenzo Castorani, <http://orcid.org/0000-0001-8514-5548>

Daniele Landi, <http://orcid.org/0000-0003-0809-3662>

Marco Mandolini, <http://orcid.org/0000-0003-0962-5982>

Michele Germani, <http://orcid.org/0000-0003-1988-8620>

REFERENCES:

- [1] 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>.
- [2] Box, G. E. P.; Draper, N. R.: *Response Surfaces, Mixtures, and Ridge Analyses*, John Wiley & Sons, Inc., Hoboken, NJ, USA, 2007. <https://doi.org/10.1002/0470072768>
- [3] Box, G.E.P., Hunter, S.J., Hunter, W.G.: *Statistics for Experimenters: Design, Innovation, and Discovery*, John Wiley & Sons, Inc., 2005.
- [4] 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>.
- [5] Cicconi, P.; Landi, D.; Germani, M.; Russo, A.C.: A support approach for the conceptual design of energy-efficient cooker hoods, *Applied Energy* Volume 206, 15 November 2017, Pages 222–239. <https://doi.org/10.1016/j.apenergy.2017.08.162>

- [6] Corriveau, G., Guilbault, R., Tahan, A.: Genetic algorithms and finite element coupling for mechanical optimization, *Advances in Engineering Software*, 41, 2010, 422–426. <https://doi.org/10.1016/j.advengsoft.2009.03.008>
- [7] Ertas, A., Jones, J.C.: *The engineering design process*, John Wiley & Sons, Inc., New York, 1996.
- [8] Goldberg, D. E.: *Genetic Algorithms in Search, Optimization and Machine Learning*, 1st ed., Addison-Wesley Longman Publishing Co., Inc., Boston, MA, USA, 1989.
- [9] Hariharan, C.; Govardhan, M.: Improving Performance of an Industrial Centrifugal Blower with Parallel Wall Volute, *Applied Thermal Engineering*, 109, 2016, 53–64. <https://doi.org/10.1016/j.applthermaleng.2016.08.045>.
- [10] Kesare, S. V.; Swami, M. C.: Analysis and optimization of Centrifugal Blower by using FEA, *International Journal of Engineering Development and Research*, 4, 2016.
- [11] Lee, K. Y.; Armstrong, C. G.; Price, M. A.; Lamont, J. H.: A small feature suppression/unsuppression system for preparing B-Rep models for analysis, *Proceedings of the 2005 ACM symposium on solid and physical modeling*, 2005. <https://doi.org/10.1145/1060244.1060258>
- [12] 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>.
- [13] Saad, Y.: *Iterative methods for sparse linear systems*, Society for Industrial and Applied Mathematics, 2003.
- [14] Stepanoff, A. J.: *Turbo blowers: theory, design and application of centrifugal and axial flow compressors and fans*, John Wiley & Sons, Inc., New York, 1955.
- [15] Stepanoff, A. J.: *Centrifugal and axial flow pumps: theory, design and applications*, John Wiley & Sons Inc., New York, 1957.
- [16] Taguchi, G.; Yokoyama, Y.: *Taguchi Methods: Design of Experiments*, ASI Press, 1993.
- [17] Tang, X.; Luo, J.; Liu, F.: Aerodynamic Shape Optimization of a Transonic Fan by an Adjoint-Response Surface Method, *Aerospace Science and Technology*, 68, 2017, 26–36. <https://doi.org/10.1016/j.ast.2017.05.005>.
- [18] Thakur, A.; Gopal, A.; Gupta, S.: A survey of CAD model simplification techniques for physics-based simulation applications, *Computer-Aided Design*, 41(2), 2009, 65-80. <https://doi.org/10.1016/j.cad.2008.11.009>
- [19] White, D. R., Saigal, S., Owen, S. J.: Meshing complexity of single part CAD models, *Proceedings of the 12th international meshing roundtable conference*, 2003.
- [20] Wroster, R.C.: The flow in volutes and its effect on centrifugal pump performance, *Proc Inst Mech Engg*, 177, 1963, 843-875. https://doi.org/10.1243/PIME_PROC_1963_177_061_02
- [21] 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>.