Optimization of a squirrel cage fan

Alla Eddine Benchik Le Hocine, Sébastien Poncet, Hachimi Fellouah

Department of Mechanical Engineering, Université de Sherbrooke
2500 Boul. de l’Université, Sherbrooke (QC) J1K 2R1 Canada
Email: alla.benchikh.le.hocine@USherbrooke.ca

Abstract—The restrictions related to air quality are increasing making the improvement of the air system important. The squirrel cage fan (SCF), also known as forward-curved multi-blade centrifugal fan, is widely used in vacuum systems. Most of researches so far used commercial software to study and optimize the SCF. In the present study, a complete automatic optimization process loop is developed based only on open source libraries: Dakota, Salome and OpenFOAM. Up to seven design parameters are selected. The Latin Hypercube Sampling (LHS) method is preferred to determine the design points and then the Kriging and Efficient Global optimization (EGO) metamodels are built. A 3D incompressible simple flow SOLVER is coupled to the Multiple reference frame (MRF) approach to model the flow in the SCF. An efficiency improvement of 8.46% is reached by the EGO approach. A strong vortex is observed in the cut-off region. The optimal design is finally validated against the produced prototype, with an error of 3.4% on the efficiency.

Index Terms—Squirrel cage fan, OpenFoam, Dakota, Design of Experiments, Surrogate model, Latin Hypercube Sampling

I. INTRODUCTION

The society is highly concerned and more sensible to the air quality of their outdoor and indoor environments. The COVID-19 pandemy certainly strengthens the researches about the improvement and adaptation of ventilation systems [1]. The improvement of the HVAC (heating, ventilation and air-conditioning) system component efficiency is becoming a necessity to ensure a trade-off between particle captation and power consumption. The squirrel cage fan (SCF) has been extensively used for decades in HVAC systems and other household appliances (bath room, range hood) [2]. The SCF is a special centrifugal fan and also known as forward-curved multi-blade centrifugal fan. In general, the SCF is prone to flow separation around the blades due to their two-dimensional circular arc profile [3], and is also subject to non-uniformity of the flow at the impeller inlet due to the sharp flow direction turning [4], [5]. These flow mechanisms and patterns lead to lower performances of the SCF and the increase of the power consumption [6]. Thus, an improvement of the flow condition inside the SCF is necessary.

Design of experiments (DOE) and metamodels have been used gradually in the optimization process of squirrel cage fans due to their ability to converge towards optimum designs efficiently. Kim and Seo [7] investigated numerically the improvement of the SCF efficiency by studying the effect of cutoff location and radius, and impeller width. The authors coupled numerical calculations and the response surface method (RSM) or also known as metamodels. For the numerical calculations, 3D incompressible RANS simulations were run using the commercial software CFX and the turbulence model k-ω. The flow around the blades and inside the impeller was not modeled to reduce computing time and was calculated by an impeller force model. For the response surface method, the quadratic polynomial model available in the commercial software SPSS was used. Kim and Seo [7] validated their numerical calculations against the experimental data of Kim and Kang [8] by hot-wire probe around the reference SCF design. At the inlet, an acceptable agreement was observed for the axial distribution of the radial velocity component with a maximum error of 20%. The quadratic polynomial model allowed to improve the maximum efficiency and the static pressure coefficient by 38.8% and 1.66%, respectively. Any validation of the obtained optimum design by a metamodel was accomplished by CFD or experimental tests. In a recent study by Zhou et al. [9], the Kriging surrogate model was coupled with CFD to optimize the blade profile for a centrifugal SCF. The authors used the modified Hicks-Henne function to design a simple arc blade, where three amplitude coefficients were selected as input parameters for the optimization process. In their study, only the blade profile function was optimized and the other part of the SCF remained identical. For the numerical calculations, a 3D incompressible RANS approach was used with the turbulence model k-ω SST and the commercial software ANSYS Fluent. A latin hypercube design was employed to obtain the sets of sampling points and the NSGA-II method was used to determine the optimal design from the Kriging response surface. Zhou et al. [9] observed a good agreement between the predicted values by the Kriging model and the CFD with an average relative error of 2.74% and 2.61% for the efficiency and flowrate, respectively. Furthermore, the optimal design improved the efficiency by 4.21% compared to the initial design. The authors validated also their numerical model for the optimal design against experimental data with a maximum error of 3.2% in terms of efficiency.

Most of the published studies that investigated and optimized the SCF focused on the impeller or the volute parameters separately. Also, they used commercial softwares for the CFD calculations and optimization like ANSYS Fluent or CFX [2], [10], [11]. In the current study, full 3D simulations are carried out in order to optimize the efficiency and force applied on the impeller by using only opensource softwares. Different geometrical parameters of the impeller and volute are integrated simultaneously in an optimization loop using the Latin Hypercube Sampling (LHS), CFD and metamodels. The optimization process is also governed by an opensource soft-
ware named Dakota. The first numerical results obtained by the Design Of Experiment (DOE) method are used to construct a response surface of a Surrogate Based Optimization (SBO, Metamodels). Furthermore, an advanced optimization method named Efficient Global Optimization (EGO) is employed to obtain the best configuration. The numerical performance of the optimal configuration is carefully validated against a new set of experimental data.

II. NUMERICAL MODELING

A 3D steady-state Navier-Stokes incompressible solver has been employed by using the opensource OpenFOAM 6 libraries to model and optimize the SCF performances.

A. Geometrical modelling

Figure 1 shows a 3D sketch of the SCF in Configurations 1 and 2. In both configurations, a Latin Hypercube Sampling (LHS) has been used in the Design of Experiments (DoE) technique. The main difference between both configurations remains in the design of the hub. For Configuration 1, a conical hub is constructed with a diameter and a height of 0.12 m and 0.077 m, respectively. For Configuration 2, the hub corresponds to a cylinder with a diameter and a height of 0.04 m and 0.066 m, respectively. The external diameter ($D_2$) and the height of the impeller ($H_2$) are equal to 0.1524 m and 0.0762 m, respectively. The SCF is a 51 bladed fans equipped with 4 digit NACA having a thickness of 2.2 mm.

The SCF is placed in an open environment represented herein by a half sphere with a diameter of $D_{\text{ext}} = 1.2$ m (Fig.2). The volute is connected to an exit duct with a diameter and length of $D_p = 0.10$ m and $L_p = 0.92$ m, respectively. A transition zone with a length $L_{\text{tr}} = 0.051$ m is employed to link the volute outlet and the exit duct.

B. Numerical methods

The calculations of the flow dynamics around the SCF are performed by a 3D steady-state incompressible Reynolds averaged Navier-Stokes (RANS) solver named SimpleFoam (OpenFoam 6), which was successfully applied to model the flow around fans and turbines [12]–[14].

A fully second-order scheme is used for the spatial discretization in order to minimize excessive numerical dissipation. The Laplacian and gradient terms are discretized by a bounded Gauss linear numerical scheme. A linear approach is selected for the interpolation scheme. The SIMPLE algorithm enables to overcome the pressure-velocity coupling. The generalized geometric-algebraic multi-grid (GAMG) solver with the combined Diagonal incomplete-Cholesky/Gauss Seidel (symmetric) smoother is selected to solve the pressure. The preconditioned bi-conjugate gradient (PBiCG) solver with Diagonal incomplete-LU (DILU) pre-conditioner is used to solve the rest of the discretized equations.

The turbulent flow is modeled by the $k - \omega$ SST closure developed by Menter [15]. It combines the robust formulation of the $k - \omega$ Wilcox model [16] in the near wall region and the $k - \varepsilon$ away from the wall. Very satisfactory results were obtained by multiple authors when using the $k - \omega$ SST turbulence closure in different fan configurations [17]–[19], and even better results than other two-equation models as the $k - \varepsilon$ family [20].

C. Numerical parameters

The different boundary conditions of the computational domain are shown in Figure 2. Three main regions are defined: the rotor (impeller), the surrounding environment and the volute with the exit pipe. The rotor region lies inside the volute and includes the fan blades, the rotating ring and the hub. The rotor rotating speed is set to 900 rpm. An inlet mass flowrate condition of 0.0588 kg/s (105 cfm) is imposed at the half-sphere surface with a turbulence intensity of 5%. The air parameters are assumed to be constant and evaluated at 293 K. For the impeller, a no slip wall condition is imposed at the blade surfaces, hub, rotating ring and disc plate. The same condition is also applied on the volute and the exit pipe surfaces. A pressure outlet is used at the pipe outlet surface where a static pressure of 24.91 Pa is imposed. The multiple reference frame (MRF) approach is used to model the rotation motion of the rotor region.

Figure 3 shows different views of the unstructured grid mesh generated by the open-source Salome library. The mesh is
composed of tetrahedral elements. Ten prismatic layers are generated around the blades with a stretching factor of 1.1. Different mesh refinements with factors equal to 4 and 3 are imposed in the volute and the exit pipe, respectively. The average total number of elements in the whole domain is around 11.7 million cells, with 9 and 2.2 million elements in the rotor and volute regions, respectively. The maximum value of the wall coordinate satisfies the requirement for a low-Reynolds number approach ($y^+ < 0.9$). The selected mesh parameters are obtained based on a mesh convergence study.

Fig. 3. Views of the mesh distribution: (a) front; (b) rear; (c) lateral.

Each RANS calculation took approximately 1-2 days using 32 processors (AMD Opteron 6172). During all the optimization process, around 4736 processors were used for a total time of 296 days. The convergence is achieved when: (i) the global fan efficiency deviation between two subsequent iterations gets below 0.1%; (ii) all residuals are lower than $10^{-7}$; and (iii) the mass imbalance is lower than $10^{-6}$. A relaxation factor of 0.3 was used for the velocity components, $k$ and $\omega$.

### III. OPTIMIZATION PROCESS AND METHODS

#### A. Optimization loop

The optimization procedure is displayed on Fig.4. Initially, the design space is determined by the selected input parameters and their minimum and maximum bounds. In Configurations 1 and 2, five input parameters are selected, while in Configuration 3, two more design parameters are added to expand the design space of Configuration 2. The input variables are:

1) $D_1/D_2$ is the ratio between the wheel interior diameter and the wheel external diameter;
2) $\beta_1$ is the blade leading edge angle (angle of attack);
3) $\beta_2$ is the blade trailing edge angle (outlet angle);
4) $R_c/D_2$ is the ratio between the cutoff radial position and the wheel external diameter;
5) $\theta_c$ is the cut-off angular position;
6) $D_2$ is the wheel external diameter;
7) $r_c/D_2$ is the ratio between the cutoff radius and the wheel external diameter.

The different input variables are introduced in Dakota and a DOE is conducted by using LHS. The LHS is a stratified sampling method where the uncertain variable range is divided into $N$ parts (segments) with equal probability. A random sample is selected from each of the equal probability segments. The $N$ values for each of the selected parameters are combined in a shuffling operation to construct a set of $N$ parameter vectors with a specified correlation structure. The LHS method has the advantage to require fewer samples than the traditional Monte-Carlo method for the same accuracy in statistics.

![Fig. 4. Optimization loop process for the LHS method.](image)

As the LHS technique is run by Dakota, a set of input parameters variables are defined. In the next step, Salome is loaded and the entire computational domain is created automatically based on Python scripts. As the geometry is constructed, a Python script is used to check any irregularity in the geometry. In the next step, the Salome mesh library is used and a Python script with all the mesh parameters is executed in order to generate the unstructured mesh. A CheckMesh function of OpenFOAM is used in order to ensure the mesh quality. After, all the calculations are run simultaneously on a cluster. Once accomplished, a post-processing is executed to extract the objective functions herein: the global efficiency and the forces applied on the impeller.

#### B. Metamodel approach

The optimal design obtained by the LHS method represents the best configuration over limited design points. In order to explore the entire design space and to confirm or find another optimal design, a metamodel (Surrogate-Based Optimization SBO) has to be constructed. The metamodel construction process is displayed in Fig.5. Multiple surrogate models are used in the design optimization such as radial basis functions, polynomial regression, neural networks, etc. In this study,
Optimal design exploration
Optimization algorithm used on the meta-model predictions

Best design

Validation
Optimal design validation by CFD

Update design points
Add the new design point

Meta-model construction
The construction is based on the obtained design points

Numerical calculations
High-fidelity simulations of the design points

Design space exploitation
Design points selected by LHS

Fig. 5. Metamodel (surrogate based) optimization workflow.

the Kriging metamodel is selected to its ability to predict accurately the efficiency on axial and centrifugal fans [21] and to model different function typologies. The Kriging surrogate (Gaussian process) is based on the achievement of a Gaussian stochastic process to the modeled objective functions. Mathematically, the Kriging function prediction at a point (design parameter) \( x \) is defined as:

\[
\hat{f}(x) = \hat{\mu} + \psi R^{-1}(Y - 1\hat{\mu})
\]  

where \( \hat{\mu} \) is the maximum likelihood estimator of the random field mean and \( \hat{f}(x) \) is the predicted objective (response) function for the variable \( x \). \( Y \) represents a set with a dimension \( N \) of the calculated data and is expressed as follows:

\[
Y = [f(x^{(1)})...f(x^{(N)})]
\]

\( f(x) \) represents the value obtained by CFD at the design points \( (x) \) selected by the LHS method. \( \psi \) in Eq.1 is a vector with a length \( N \) representing the basis functions. The basis functions \( \psi \) are weighted by the term \( R^{-1}(Y - 1\hat{\mu}) \), where \( R \) is the \( N \times N \) correlations matrix among the design points.

In addition to the different metamodels, the efficient global optimization (EGO) method is also applied to predict the objective functions. The EGO has been developed by Jones et al. [22] and is based on the Kriging metamodel. The EGO model contrary to the surrogate model has the ability to select the next sample point with the maximum probability that the global minimum exists in the design space. The probability is defined as an expected improvement (EI) function:

\[
E(I(x)) = \left(f_{\text{min}} - \hat{f}\right)\Phi\left(\frac{f_{\text{min}} - \hat{f}}{s}\right) + s\phi\left(\frac{f_{\text{min}} - \hat{f}}{s}\right)
\]

where \( f_{\text{min}} \) is the obtained minimum value from the CFD calculations, \( \hat{f} \) is the predicted value by the Kriging model. The parameter \( s \) is the standard error at \( x \). \( \phi(\cdot) \) and \( \Phi(\cdot) \) are the standard normal density and distribution function, respectively. \( I(x) \) is the improvement at the point \( x \) defined as:

\[
I(x) = \max\left(0, f_{\text{min}} - \hat{f}(x)\right)
\]

The iterative process will continue until a global minimum is found and validated against the CFD results. A convergence tolerance of \( 10^{-2} \) is selected for the efficiency and the force applied on the wheel. Contrary to standard metamodels, the EGO approach needs more computational resources to converge toward an optimal design.

IV. RESULTS AND DISCUSSION

In this section, the results of the optimization process by using LHS and metamodels are presented for the three configurations.

A. Performance comparison

The optimal value of the efficiency and \( F_{xy} \) are determined after completing all the optimization process for the LHS approach and constructing the surrogate models according to the charts displayed in Figures 4 and 5. The optimization objective is to minimize \( F_{xy} \) in order to avoid any mechanical damage and to maximize the SCF efficiency.

All the predicted values of the objective functions by the different methods are presented in Table II. In Configuration 1, the Kriging model predicts with accuracy the efficiency and \( F_{xy} \) values with errors of 2.16% and 8.4%, respectively. The major source of error is essentially due to its limitation in complex responses, where the flow is highly turbulent with 3D vortices interacting with the impeller blades. The application of the EGO based on the Kriging metamodel allows the improvement of the efficiency and \( F_{xy} \) by 2.35% and 5.71%, respectively, compared to the optimal LHS design. The EGO method converges after 10 iterations. As the design space is expanded by the EGO approach compared to the Kriging, the probability to find an optimal design which meets the objective function requirements is increased. In Configuration 2, by using the Kriging model and EGO, the initial optimal design obtained by the LHS is improved in a short time and few calculation resources with an improvement of 2.63% and 28.45% for the efficiency and \( F_{xy} \), respectively. As in Configuration 1, the optimal design with the EGO model improves the objective functions specifically \( F_{xy} \) by 11.49% compared to the Kriging model validation. In Configuration 3, the Kriging model provides an accurate prediction with errors of 1.13% and 8.15% for the efficiency and \( F_{xy} \), respectively. The EGO
model provides an optimal design with an improvement of the efficiency by 3.27% and 4.22% compared to the Kriging OpenFOAM (OF) validation and LHS, respectively.

**B. Mean flow field**

Figure 6 displays the 2D contours of the mean velocity magnitude $U$ (m/s) on plane 1 $(x = 0, y = 0, z = H_2/2)$ for Configurations 1, 2 and 3.

In order to investigate the effect of the observed vortex on the flow around the blades, the parietal distribution of the static pressure on the impeller is presented in Fig.8. The impeller is oriented towards the volute exit. In Configurations 1 and 2, the flow is separated for multiple blades all along the height due to the strong interaction with the 3D vortex (Fig.7). However, in Configuration 3, the flow is separated along few blades, mostly in the tip region. As the flow is more attached along the blades in Configuration 3, the produced efficiency is higher.

**C. Validation**

As the SCF in Configuration 3 produced the best objective functions, a new experimental prototype was constructed. The static pressure and voluminal flowrate are quantified for the experimental prototype and compared to the numerical results. The SCF is connected to a pipe as in the computational domain (Fig.2). The diameter and length of the pipe are equal to 0.10 m and 0.92 m, respectively. The experimental procedure consists of measuring the generated flowrate by fixing the rotation speed of the fan and the pipe outlet pressure. The outlet pressure was controlled by varying the effective surface at the exit. The outlet pressure and velocity are measured by a manometer with an accuracy of $\pm 1\%$ and $\pm 3\%$, respectively.

Figure 9 represents the experimental and numerical values of the pressure drop $\Delta p$ for different flowrates at a constant rotating speed of the fan (900 rpm). $\Delta p$ is the average pressure difference between the outlet and the inlet. At the design point,
A good agreement is observed between the simulated and the experimental data with a discrepancy of 3.4%. The maximum error between the numerical approach and the experimental results is 5.1% for 90 cfm.

V. CONCLUSION

This paper reported numerical results of the optimization process of an SCF, using opensource libraries. A complete automatic optimization loop was developed using Dakota, Salome for the geometry and mesh, and OpenFOAM for the numerical simulations. LHS and metamodels were selected to reduce the design points and expand the design space. Three configurations were tested with up to 7 design parameters.

The optimal design was obtained for Configuration 3 based on the EGO approach. By adding the two design parameters $D_2$ and $R_c/D_2$, the efficiency was improved by 4.1% compared to Configuration 2. Using a conical shape of the hub leads to a decrease and increase by 6% and 2% of the efficiency and $F_{xy}$, respectively. The distributions of the average velocity magnitude $U$ showed the existence of a strong vortex between the cut-off and the impeller. The generated vortex leads to a total pressure decrease and forces the flow separation on the blades. The parietal static pressure confirmed the flow separation on multiple blades in Configurations 1 and 2. However, in Configuration 3, the separation region is mainly concentrated in the tip region.

Future works should integrate the variation of the blade profile along the span and the extension of the objective function by adding the radial forces and generated sound.

ACKNOWLEDGMENT

All calculations have been performed using the HPC facilities of the Compute Canada network, which is here gratefully acknowledged.

REFERENCES


