Aerospace Science and Technology 39 (2014) 414–425

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

# Multi-objective optimization for centrifugal compressor of mini turbojet engine

Shuai Guo [a], Fei Duan [a,*], Hui Tang [a], Seng Chuan Lim [b], Mee Sin Yip [b]

[a] School of Mechanical and Aerospace Engineering, Nanyang Technological University, Singapore 639798, Singapore
[b] DSO National Laboratories, Singapore 118230, Singapore

A R T I C L E   I N F O

*Article history:*
Received 27 November 2013
Received in revised form 7 March 2014
Accepted 17 April 2014
Available online 10 May 2014

*Keywords:*
Centrifugal compressor
Mini turbojet engine
Multi-objective optimization

A B S T R A C T

Computational fluid dynamics (CFD) simulation coupled with optimization technique plays a promising role in the turbo-machinery component design. The geometrical optimization has been applied for a mini-centrifugal compressor by using the design of experiments technique. Both the first-order and the second-order regression models are applied with the response surface method (RSM). With the fitting of the regression functions from the least-squares estimator, three compressor geometrical parameters are selected as the input factors. The ideal targets of the optimization problem are for a higher efficiency, a higher pressure ratio and a lower input power. The simulations are conducted in a fully three-dimensional (3D) CFD software – Axcent. The multiple objective optimization problems are solved by an evolutionary algorithm using a Matlab program. Multiple Pareto front solutions can be determined, and the specific optimal solution is selected with the weighted metric methods. The experimental test for the optimal compressor shows a 7.5% increase in pressure ratio. The two-factorial interactions are generated in 3D plots to estimate the effects of different input parameters. The flow analysis for the relative Mach number distribution and the entropy distribution is carried out to explain the changes of the compressor efficiency and pressure ratio.



## 1. Introduction

The centrifugal compressors are applied in mini turbojet engines widely used in the unmanned aerial vehicles, since they satisfy the needs of relative small size, low weight and high specific efficiency. The single-stage centrifugal compressors are composed by one rotated impeller, which generates kinetic energy, and one stationary diffuser, which translates the kinetic energy to pressure power [15]. According to the requirements of application and research development, high efficiency, high pressure ratio and low power input work are the ideal targets of centrifugal compressor design.

After the first invention back in the 1930s, the study of centrifugal compressors experienced from empirical approaches, experimental approaches, to the theoretical approaches [17]. The pioneers of turbo-machinery applied the concept of similarity in the original principle for compressor design. The development of experimental methods made the inner testing possible and researches started to study the detailed flow properties. Recently, the numerical method coupled with higher computational tools allowed the designers simulating and "running" compressors in computers [11,27,29]. The numerical optimization techniques are widely used to find an appropriate combination of design variables to make the objective functions maximal or minimal. Coupled with the computational fluid dynamics (CFD) calculation, the optimization methods can provide an efficient tool to analyze the complex correlations between the geometrical parameters and the compressor performance [2]. Bonaiuti et al. defined an optimization method into three categories: gradient based optimizers, exploratory techniques, and methods based on the concept of function approximation [4]. It is the first method to calculate the gradient of an objective function and move the solution towards the closest local optimum. The exploratory techniques are able to seek the optimal solution in the whole design space and deal with multi-objective problems. The last techniques, for example design of experiments (DOE), are based on the definition of approximated functions which correlate the input parameters with the objective functions statistically. The DOE method is a statistical technique used in the quality control for planning, conducting, analyzing, and interpreting the sets of experiments which aim at making reasonable decisions without incurring too high cost or

* Corresponding author. Tel.: +65 6790 5510; fax: +65 6792 4062.
*E-mail address:* feiduan@ntu.edu.sg (F. Duan).

http://dx.doi.org/10.1016/j.ast.2014.04.014
1270-9638/

taking too long time. The method which most commonly used to construct the approximation model is the response surface method (RSM). The RSM approach is based on experimental design and uses the least-squares estimation methods to construct a polynomial approximation of numerical simulations. It is computationally cheap and easy, and has often been successfully applied in engineering optimizations [1,4,19,23]. A CFD simulation produced by Krain et al. has been used for aerodynamic design tasks [18]. The tool used in their study was originally developed to study steady 3D flows and extended to handle three-dimensional (3D) unsteady flows passing successively rotating and stationary components like rotors and stators, thus offering the possibility to study unsteady effects like rotor/stator interactions. However, the positive qualitative reliability investigation for centrifugal compressors is still rare presently to our knowledge.

The CFD simulation of flows passing through centrifugal compressors has been conducted for design and analysis since the study of Eckardt [10]. Japikse indicated that CFD could provide the detailed flow field for optimum performance of centrifugal compressor since the 3D flows and viscous effects could be taken into account in the design process [15]. Dawes applied a 3D time-resolved simulation with the unstructured mesh and the solution-adaptive Navier–Stokes solver to unsteady flow associated at a splittered centrifugal impeller, and indicated that a strong hub/corner stall could be resulted from the strong spanwise distortion in the swirl angle at inlet [7]. Dickmann et al. analyzed the unsteady flow with transient CFD for the mechanism impeller's vibration in a turbocharger centrifugal compressor stage and pointed out that the integrating the simulation procedure into the design of the compressors would enhance the design quality of the design process [9]. A 3D optimization method was proposed by Benini combining a CFD code and an evolutionary algorithm [3].

Different from the physical experiments, the numerical analysis with the CFD simulation in the aerodynamic design can reduce the random errors, i.e. the results do not change after iterative computation and has a characteristic of certainty. Bonaiuti et al. reported the optimization of the transonic centrifugal compressor [4]. They grouped the impeller parameters into shape parameters and functional parameters. Three impellers were optimized with nine selected parameters; both efficiency and operating range were significantly improved. Wang et al. investigated the application of the Kriging model in the optimization of a centrifugal impeller [26]. They increased the isentropic efficiency of the compressor at the design point by 2.49%, and improved the flow inside the impeller distinctly. Kim et al. reported the design optimization of a centrifugal compressor impeller with four design variables that defined the impeller hub and shroud contours in meridian terms [16]. Cosentino et al. used a genetic algorithm coupled with an artificial neural network method to optimize a 3D radial impeller, which was described by 15 geometrical parameters [6]. However, the optimization research on the mini centrifugal compressor with more than two objectives is quite limited. This study starts with the compressor geometry, the DOE technique coupled with the fully 3D CFD simulation is applied to optimize the mini centrifugal compressor of a mini turbojet engine, SR-30. The multi-objective problems are solved by Pareto front solutions and the Weighted Metric method. In the optimization study, the configuration modification of the compressor resulted in three independent variables to study: pressure ratio, isentropic efficiency, and work input. The goal of this study is to understand the factorial interactions of the input factors, search for an optimal compressor design, and explain the test results on flow analysis after the compressor is fabricated.

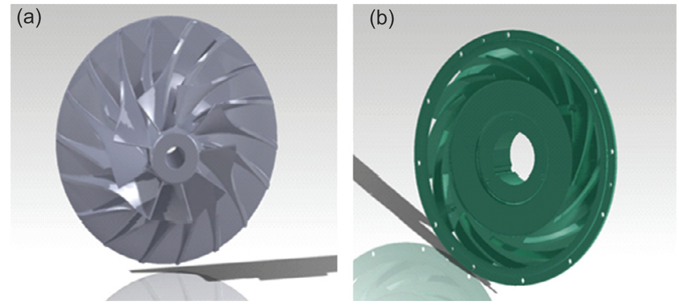


**Fig. 1.** Model of the original centrifugal compressor, (a) impeller; (b) diffuser.

## 2. Methodology

### 2.1. Definition of the problem

The compressor studied herein is the one from the SR-30 mini turbojet engine. Single 110 mm axial length impeller and one radial–axial wedged diffuser compose the stage. The original compressor profile is shown in Fig. 1 [14]. The impeller exit radius is 51.5 mm, with 5 mm gap between the impeller and diffuser. Splitters start from 33.3% of the flow channel depended on main blade. Objectives of this optimization are for a higher isentropic efficiency, a higher pressure ratio, and a lower work input requirement. Based on the screen analysis, three main parameters are selected as the input factors: the impeller exit radius, the impeller exit blade angle, and the diffuser inlet blade angle. The impeller exit radius, $R$, indicates the distance between the impeller axis and the impeller exit hub. It generally determines the whole size of the impeller. The Euler-Turbomachinery equation [5] indicates that the impeller exit radius has important effects on the compressor pressure ratio. Due to the space limitation of the entire diameter of the impeller for further test, the diffuser exit radius is not changed. It means that the diffuser inlet radius changed according to the impeller exit radius. The blade angle distribution follows the Bezier curve [12], which defines a smooth curvature from inlet to exit. The impeller exit blade angle, $\beta_1$, which is also named as the sweep angle, mainly controls the air velocity and angle from impeller. Considering the significant interaction effects between the impeller exit blade angle and the diffuser inlet blade angle, we select the impeller exit blade angle and the diffuser inlet blade angle as the other two input factors. The compressor configuration is designed through the CFD simulation and optimization process.

### 2.2. CFD approaches

The flow analysis is performed in the commercial software – Axcent™, which is based on a fully 3D solver Pushbutten CFD™ (pbCFD). The solver is a hybrid multi-block structured grid full Navier–Stokes solver specially designed for turbomachinery [13,22]. For the fully turbulent computational procedure, the Spalart–Allmaras (S–A) turbulence model [8,24,25] is selected. The model is assumed with the turbulent viscosity being governed by a convection–diffusion equation. The expression is defined for the boundary layer and free shear layer. The transport equation for the internal energy, $e$, can be written as,

$$\frac{\partial(\rho e)}{\partial t} + \nabla\cdot(\rho e v) = \frac{1}{\sigma_v}\nabla\cdot\left[(\mu+\rho e)\nabla e\right] + C_{b2}\rho(\nabla e)^2 + C_{b1}\rho e q - C_{w1}\rho\left(\frac{e}{ny}\right)^2 f_w \qquad (1)$$

where $\rho$ is air density, $U$ is free-stream velocity, $\mu$ is dynamic viscosity, $q$ is local mean vorticity, $f_w$ is wall-damping function.