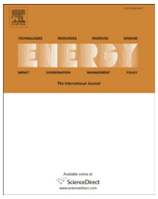
Energy 123 (2017) 511–520

Contents lists available at ScienceDirect

Energy

journal homepage: www.elsevier.com/locate/energy



# Effect of splitter leading edge location on performance of an automotive turbocharger compressor

S. Abolfazl Moussavi, Ali Hajilouy Benisi*, Mohammad Durali

*Department of Mechanical Engineering, Sharif University of Technology, Tehran, Iran*

A R T I C L E   I N F O

*Article history:*
Received 8 May 2016
Received in revised form
1 November 2016
Accepted 2 February 2017
Available online 5 February 2017

*Keywords:*
Centrifugal compressor
Splitter blades
Leading edge position
Performance optimization
Surge improvement
Incidence loss

A B S T R A C T

Design of centrifugal compressors can be accomplished using methods provided in text books and open literature. In this general approach, after determination of dimensions in meridional plane, blade profiles are designed using aerodynamic methods. Then, some blades are trimmed at the beginning to avoid inlet blockage. But there is no clear guidance on how this trimming must be performed. With this widely implemented method, splitter blades have the same profile as main ones.

In this research, blade profiles and dimensions of the compressor impeller are not changed while the effect of changes in the position and angle of splitters leading edge are investigated. An optimization scheme is performed to find the best configuration over the complete operating curve of the compressor using genetic algorithm and considering structural aspect of the problem. CFD codes with experimental support are used to predict the compressor performance and flow field specifications with the original impeller as well as the modified one. After taking into account some geometrical details, a new impeller is manufactured based on the modified design and tested in a wide range of operating conditions. Results show 2.7 points improvement in efficiency at the nominal point and 5% decrease in impeller moment inertia. No change in choke behavior, but notable improvement in surge margin of the modified compressor is observed. This is an important achievement in all applications of centrifugal compressors, especially in turbochargers.

## 1. Introduction

Design of centrifugal compressors impellers has been investigated vastly in industry and academic literature. A number of codes and algorithms have been developed leading to optimum designs according to initial assumptions and applications [1–3]. There are also several well-known text books, discussing the design procedure of these compressors [4–6]. In this method, after determining general dimensions of the impeller by iteration of 1D thermodynamic and empirical models, blade meridional profiles of the impeller are obtained by optimization. Then the number of blades is estimated by relations provided by Stepannof [7], Eckert [8] and Came [9]. To avoid inlet flow blockage due to blades thicknesses, some blades are trimmed constituting splitter blades. With this in mind, splitter blades have the same profile as main ones. This is the design procedure being practiced by manufacturers and observed in commercial products. But there is no clear guidance on how this trimming must be performed.

Measurements carried out by Eckart [10] and Krain [11] in the impeller of centrifugal compressors, revealed that due to presence of Coriolis forces, the flow does not follow blade profile and severe deviation is observed. These results show the necessity of having different treatments for splitter blades or revising the trimming location when manufacturing costs prevent independent profiles for blades. As stated by Japikse [4], design of centrifugal compressors is a state of the art rather than just theories. This demonstrates the importance of geometrical optimization when no theoretical constraint is imposed on impeller details.

Design and optimization of compressor impeller blades has been investigated in the literature [12,13]. However, less works have addressed splitters specifically. The concept of implementing two splitters between each pair of main blades is examined in the work of Ibaraki [14] and Layth [15] in which some advantages over usual designs was reported.

Lohmberg et al. [16] studied the effect of changing the leading edge position of splitter blades in a refrigeration transonic compressor at working point condition. Their main objective was to

---

* Corresponding author.
*E-mail address:* Hajilouy@sharif.ir (A. Hajilouy Benisi).

http://dx.doi.org/10.1016/j.energy.2017.02.011
0360-5442/© 2017 Elsevier Ltd. All rights reserved.

ensure that the splitter blade leading edge was placed sufficiently far back in the passage to avoid choking in the splitter passages. Due to proprietary restrictions, no precise geometry was provided in their work.

Oana et al. [17] performed an optimization with the aim of equating the flow rate on either sides of splitters in a transonic centrifugal compressor. Some performance improvements were achieved but the final geometry was not presented. Erdmenger and Michelassi [18] analyzed 10 different shapes for the leading edge of main blades and 3 ones for splitter blades. The studies conducted on the splitter leading edge profile indicate that aft sweep may help to increase the operating range of the impeller analyzed by up to 16% while maintaining similar pressure ratio and efficiency characteristics of the impeller. However, since the study was performed only numerically on few limited cases, and no logic was provided for the derivation of suggested shapes, it cannot be deduced that an optimum geometry is obtained.

We believe that, more care must be taken in implementing splitter blades in the impeller of a centrifugal compressor and simple trimming of main blades does not result in optimum design. So, the concept of having splitter blades with independent shape and profile from main blades, was successfully implemented and presented in references [19,20]. In order to avoid complexity in manufacturing process, we are intended to perform a more in depth examination of the position of leading edge of splitters without changing their profile and explore its effect on compressor performance over a wide range of working mass flow, rather than a single working point. This work is distinguished by consideration of all aspects of the problem including structural issues and experimental analysis.

The procedure is started with an experimentally validated numerical code and continued by implementing a scheme to find an optimized configuration for the impeller with efficiency improvement as the main objective. The modified impeller is simulated numerically and the compressor performance is compared with the original design. After taking into account structural considerations, the optimized impeller is manufactured and an experimental validation is performed. Results approve the improvement of performance in a wide range of working speeds, which is an essential feature for a compressor implemented in a turbocharger.

## 2. The model

### 2.1. Geometry

In this work, a typical turbocharger compressor is selected as the base design. The compressor specifications are given in Table 1. To extract the exact geometry of the compressor a Multi-Slice CT scan of volute and impeller is performed. The resulted point cloud is processed from 3D image obtained by CT scan (Fig. 1), which is used for CAD geometry construction.

**Table 1**
Geometrical parameters of the compressor impeller.

| Parameter | Value |
|---|---|
| N | 6 |
| $\beta_{1s}$ | 60 Degree |
| $\beta_{2s}$ | 30 Degree |
| $r_{1s}$ | 56 mm |
| $r_{1h}$ | 22 mm |
| $r_{2h}$ | 82 mm |
| $w_2$ | 5.5 mm |
| $\dot{m}_{nom}$ | 287 g/s |
| $PR_{nom}$ | 2.18 |
| $\omega_{nom}$ | 92000 rpm |

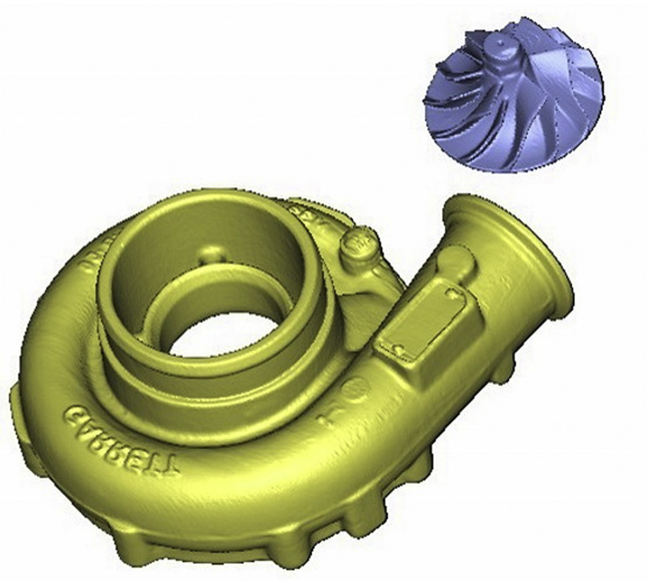


**Fig. 1.** CAD model of the compressor.

### 2.2. Flow simulation model

The geometrical model is meshed using a structured hexahedral grid which is later used by a commercial CFD software to simulate the flow inside the compressor. This grid implements near wall layers with a thickness of $50\mu$ leading to a y+ of less than 100. Hence it is able to catch flow details in passage with sufficient precision and thus improving the accuracy of simulations. This mesh uses an O-grid scheme for leading edges and an H-grid for trailing edge. Due to large number of simulations needed for the optimization of impeller geometry, the volute is excluded from the model and only one passage is considered by assuming periodic boundary conditions on domain sides. By performing a mesh independency analysis, it is observed that a minimum of 450000 elements per passage is needed (Fig. 2).

In simulations the compressor inlet is assumed to have a total pressure of 1 bar and total temperature of 300 K. The outlet is set to have a specified mass flow rate. Frozen rotor method is used to transfer fluid properties when passing from a rotating reference frame to a fix one. Also the SST model is implemented for the

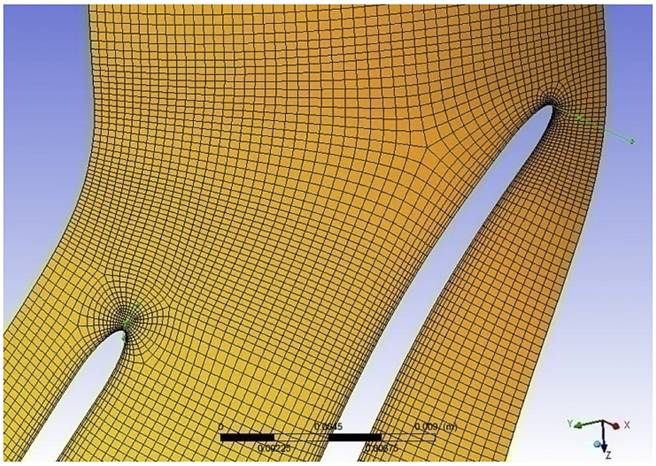


**Fig. 2.** Generated mesh for the impeller at mid span.