Numerical Study of the Turbulent Air Flow through the Turbocharger Compressor Using Different Rotor Shapes

Abdelmadjid Chehhat, Mouna Maache
Dept. of Mechanical Eng. Abes Laghrour University
Khenchela, Algeria
achehhat@gmail.com, mounamaache@yahoo.fr

Mohamed Si-Ameur
Dept. of Mechanical Eng. University of Batna 2
Batna, Algeria
msiameur@yahoo.fr

Abstract- **The turbulent air flow through the turbocharger is studied, considering the full stage of the centrifugal compressor, consisting of an air inlet, leading to an impeller that discharges the air radially through a vaneless diffuser in the volute. The objective of this study is the prediction of the influence of impeller blades geometry on the flow local characteristics (velocity and pressure) and on the local characteristics of turbulence represented by the standard k-ε model, for this, three types of impellers that have the same size and the same number of blades are considered, they differ only in inlet and exit blade angle. All simulations are carried out for the same rotational speed of 100000 rpm, and several points of the operating range, it is chosen for representing the results one of these points at flow rate of 0.47 kg/s, the velocity fields is presented in the axial plane, and transverse plan, whereas for the turbulence intensity we show the diffuser with each type of impeller to give an idea about the distribution of turbulence intensity in the diffuser. It is found from the computational fluid dynamics analysis that the rotor design presents a considerable effect not only on the fluid flow field and turbulence characteristics, but also on the performances and operating range of the turbocharger compressor.**

Index Terms – Centrifugal Compressor. Compressible Flow. Turbocharger. Turbulence.

I. INTRODUCTION

Increasing the specific power of the engine for a given amount of energy consumed, as well as torque at low revs has always been one of the biggest challenges for car manufacturers. This goal can be achieved by using a turbocharger. An automotive turbocharger is a small component driven by engine exhaust gases through a radial turbine that mechanically transfers kinetic energy to the centrifugal compressor. The centrifugal compressor delivers compressed air to the cylinders for better combustion. The technology has reached a high level of refinement, and many improvements have been studied and in some cases put into production. A wide range of internal combustion engines and turbochargers are available, but only a good match between these two components can produce an efficient supercharged engine. To carry out the calculations that make it possible to adapt the engine to the turbocharger, it is necessary to have an accurate representation of each component of the turbocharger and an ability to determine the behavior of the entire system with all of its components [1]. However, the turbocharger compressor performance is limited by the engine operating conditions. In this context and in the recent years, that is why most of researchers have been analyzed the centrifugal compressor performance alteration with its elements design.

Variable angle in vanned diffuser study have shown that the open angle diffuser lead to high efficiency and stable operating range [2]. The investigation of the air flow through the turbocharger compressor with dual volute design, revealed the dual volute design improve a stable operating range comparing with a turbocharger with a single volute [3].

The study dealing with impeller-diffuser -interaction in a centrifugal compressor, that four diffuser configurations were studied, by changing the radial gap between the impeller and diffuser[4]. It is shown that the effect of such interaction on the compressor stage efficiency is significant. Two different volutes with the same impeller are studied numerical computation and the experimental measurements reveal that the compressor performances are affected [5].

It also shown that the turbulence and flow field studies, may give a good idea on the pressure variation in the centrifugal compressor as presented in a detailed flow measurements at inlet plane of a centrifugal compressor shows that several phenomenon related to high turbulent kinetic energy and shear region are identified [6].

A comparison of turbulent methods in computational fluid dynamics analysis of radial turbo machines is proposed and the best way to choose turbulence parameters when using FLUENT software is introduced [7]. Some studies are focused on the design and the surge phenomenon of centrifugal compressor [8], [9], [10].

These numerical studies were dealing with the stationary part of the centrifugal compressor (diffuser or volute), seeing the complexity of impeller geometry, not many numerical studies are aimed at the influence of the rotational part (impeller) of the compressor design on the flow field and turbulence. We have studied the impeller blade angle influence on the performance and fluid flow but in the centrifugal pump [11].

Therefore, in order to analyze the effect of the rotor shape on the centrifugal turbocharger compressor performance, three different impeller are designed and studied in this paper with the same vaneless diffuser and volute (simple impeller, curved inducer impeller, and backward-curved inducer impeller) which have the same dimensions, they only differ in their blade geometry.

978-1-6654-1319-0/21/$31.00 ©2021 IEEE