

A Practical Simulation Procedure using CFD to Predict Flow Induced Sound of a Turbocharger Compressor

Weiguo Zhang and Mac Lynch
FCA US LLC

Robert Reynolds
CD-adapco

ABSTRACT

A turbocharger is currently widely used to boost performance of an internal combustion engine. Generally, a turbocharger consists of a compressor which typically is driven by an exhaust turbine. The compressor will influence how the low frequency engine pulsation propagates in the intake system. The compressor will also produce broad-band flow induced sound due to the turbulence flow and high frequency narrowband tonal sound which is associated with rotating blade pressures. In this paper, a practical simulation procedure based on a computational fluid dynamics (CFD) approach is developed to predict the flow induced sound of a turbocharger compressor. In the CFD model of turbocharger compressor, the unsteady, moving wheel, detached eddy simulation (DES) approach are utilized. In this manner, both the broad-band and narrow-band flow induced sound are directly resolved in the CFD computation. Two operating points in the compressor performance map are selected and the sound pressure level (SPL) across the compressor is computed at these operating points. Several observation and conclusions are drawn based on the simulation results.

CITATION: Zhang, W., Lynch, M., and Reynolds, R., "A Practical Simulation Procedure using CFD to Predict Flow Induced Sound of a Turbocharger Compressor," *SAE Int. J. Passeng. Cars - Mech. Syst.* 8(2):2015, doi:10.4271/2015-01-0662.

INTRODUCTION

The concept of engine down-sizing has become a trend in current automotive industry. Almost all the automotive and industrial diesel engines and the majority of high performance SI engines are turbocharged. Noise from turbocharger is increasingly becoming an issue in vehicle noise, vibration and harshness (NVH) development.

A turbocharger generally consists of a compressor which normally is driven by an exhaust turbine. A turbocharger compressor is a centrifugal compressor consisting of wheel, diffuser and housing. As the compressor wheel rotates, ambient air is drawn in axially through an air induction system, accelerated to high speed and then expelled in a radial direction to the outlet. The geometry of a compressor is shown in Figure 1. Both the compressor and the turbine will influence how the low frequency engine pulsation propagates in the intake and exhaust systems [1, 2]. The turbocharger will also produce the broad-band and narrow-band flow induced sound [3, 4]. Due to the presence of several after treatment devices and efficient silencer systems on the exhaust side, noise problems from turbocharger are usually associated with the intake side.

A 1D CFD simulation tool has been utilized to study the phenomenon of turbocharger whoosh and its impact on powertrain and vehicle level NVH [5]. The operating points are picked from 1D engine simulation tool. A practical CAE procedure using 1D CFD tool was

developed and validated in Chrysler for air induction noise prediction [6]. This procedure can predict the low frequency strong order content across the compressor for small turbo charged engine but cannot predict the flow induced noise and the low frequency order noise for large turbo engine. An improved 1D compressor model with real compressor geometry in 0D and 1D elements was developed. It improves the prediction of the pressure waves transmitted and reflected by the compressor [7].

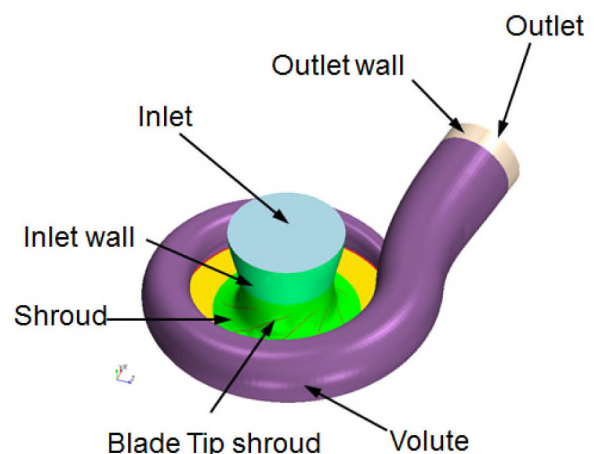


Figure 1. The geometry of a compressor



Documents

Document Sharing Hub

Nevertheless, for the broad-band and narrow-band flow induced sound, 3D CFD simulation tools are needed. Different CFD simulation tools has been utilized to study whistle noise in air intake system [3] and compressor flow induced sound[4].

In this paper, the methodology in reference [4] is utilized to develop a practical simulation procedure to predict the flow induced sound of a turbocharger compressor. First, the CFD model and the solution procedures to generate the flow induced sound by the turbocharger compressor are described. Then, the operating points are selected from the performance map of compressor and the sound pressure levels at the inlet side and outlet side of the compressor and across the compressor are computed for these operating points. Finally some conclusions are drawn based on the simulation results.

DESCRIPTION OF CFD MODEL

Transient CFD methodologies are necessary to capture the spectral contents of the aeroacoustic sources [4]. Because much of the noise coming from a turbocharger is due to unsteadiness in the blade wakes, the Detached Eddy Simulation (DES) turbulence modeling has been utilized which is proven to be the most effective model to capture both narrow-band and broad-band flow excitations. The accurate acoustic solution requires that the rotation of the compressor wheel is represented by a physically moving computational grid surrounding the wheel (i.e. rigid body rotation). In addition, the necessary time resolution should be that required to resolve 1-2 degrees of rotation. Finally, in order to resolve the necessary turbulent structures and near-wall effects, the grid resolution required for a DES solution is relatively fine. At least 20 cells per acoustical wavelength are needed to resolve the acoustics of the system.

The section cut plot of the compressor analyzed is shown in [Figure 2\(a\)](#). A 1-meter duct is added in both the inlet side and outlet side of the compressor to represent the flow domain comes in and out of the compressor. The whole CFD computation domain is shown in [Figure 2\(b\)](#). The acoustical behavior is characterized as the sound pressure level at the discrete points in the computation domain. Two points are selected in the inlet and outlet duct, respectively, which represent the locations near the compressor and far away to the compressor. The arrangement can study the noise across the compressor and acoustic damping the sound propagating in the duct. The boundary conditions are applied at the inlet and outlet duct which will be described in the next section.

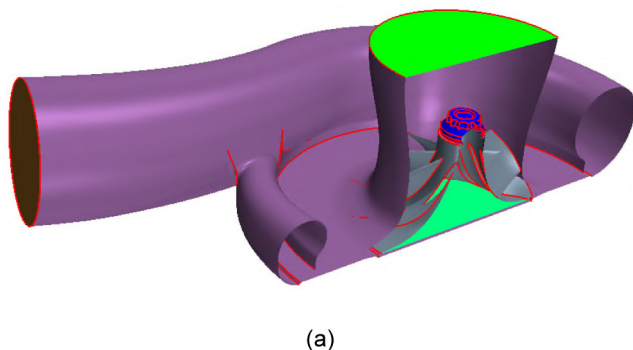


Figure 2.

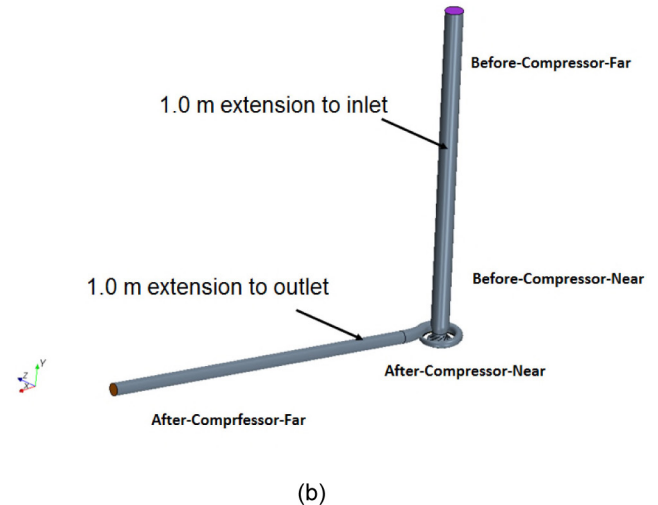


Figure 2. (cont.) CFD computation domain of the compressor (a) section cut of the compressor (b) compressor with inlet and outlet ducts

SOLUTION PROCEDURES

Step-1 Build Coarse and RANS Solution

In order to create an as accurate initial flow field as possible to reduce the computational expense of the transient aeroacoustic solution, a steady RANS model is built first. The wheel rotation is modeled via a moving reference frame (MRF).

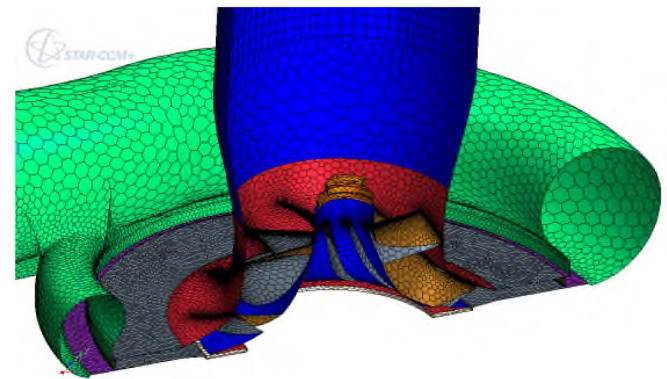


Figure 3. Coarse CFD Mesh for RANS solution

A relatively coarse grid model is shown [Figure 3](#) which has 2.5 million cells. The stagnation inlet is applied at the inlet side and static pressure outlet with target mass flow option enabled is applied in the outlet end. The resulting solution will be mapped onto the finer, aeroacoustic grid which provides accurate boundary values for the non-reflective boundary conditions needed in the aeroacoustic solution.

Step-2 Generate the Non-Reflective BC's

Since the boundary types will be changed in the aeroacoustic model to avoid reflections on the compressor inlet, the RANS solution needs to be re-converged with the final boundary conditions. The compressor inlet boundary type is changed to free stream and applied the boundary values obtained from the previous, RANS solution.