Journal of Computational and Applied Mathematics 291 (2016) 112–126

Contents lists available at ScienceDirect

Journal of Computational and Applied Mathematics

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

Effect of the numerical scheme resolution on quasi-2D simulation of an automotive radial turbine under highly pulsating flow

J. Galindo, H. Climent, A. Tiseira, L.M. García-Cuevas [*]

CMT-Motores Térmicos, Universitat Politècnica de València, Valencia 46022, Spain

ARTICLE INFO

Article history:
Received 20 October 2014
Received in revised form 9 February 2015

Keywords:
Turbocharger
MUSCL
Pulsating flow
Radial turbine simulation
Numerical scheme selection

ABSTRACT

Automotive turbocharger turbines usually work under pulsating flow because of the sequential nature of engine breathing. However, existing turbine models are typically based on quasi-steady assumptions. In this paper a model where the volute is calculated in a quasi-2D scheme is presented. The objective of this work is to quantify and analyse the effect of the numerical resolution scheme used in the volute model. The conditions imposed upstream are isentropic pressure pulsations with different amplitude and frequency. The volute is computed using a finite volume approach considering the tangential and radial velocity components. The stator and rotor are assumed to be quasi-steady. In this paper, different integration and spatial reconstruction schemes are explored. The spatial reconstruction is based on the MUSCL method with different slope limiters fulfilling the TVD criterion. The model results are assessed against 3D U-RANS calculations. The results show that under low frequency pressure pulses all the schemes lead to similar solutions. But, for high frequency pulsation the results can be very different depending upon the selected scheme. This may have an impact in noise emission predictions.

1. Introduction

Nowadays internal combustion engines, ICE, are facing two main problems, the pollutants emission and the fuel consumption reduction, in order to fulfil new regional regulations such as the European norm Euro VI [1,2] while maintaining the engine performance. The new engine design paradigm used to reach these objectives is based on a reduction of the engine size while incrementing the inlet pressure, an action known as downsizing. This is usually done using a turbocharger placed in the intake and in the exhaust line, and engine efficiency is highly affected by the turbocharger efficiency.

0-D models can be used to compute the turbine behaviour coupled with an engine. These models can predict the flow characteristics at low engine regimes and pulse frequencies, when wave effects are small and the main effects are due to mass and energy accumulation in the volute, as shown in [3] and [4]. At higher engine regimes and pulse frequencies, however, wave effects become important and 0-D models fall sort in accuracy, so one-dimensional codes are used instead. Engine manufacturers are growing their usage of one-dimensional codes during engine development, as they provide accurate results while keeping their computational costs low enough to be used during intensive and broad simulation campaigns. As pulsating flow becomes more important with further engine downsizing and urban driving emission regulations become more stringent, the importance of one-dimensional accurate predictions of turbocharger performances under high

---

[*] Corresponding author. Tel.: +34 963877650; fax: +34 963877659.
E-mail address: luiga12@mot.upv.es (L.M. García-Cuevas).
URL: http://www.cmt.upv.es (L.M. García-Cuevas).

http://dx.doi.org/10.1016/j.cam.2015.02.025

Nomenclature

A	Area
CFD	Computational fluid dynamics
CFL	CourantFriedrichsLewy
CPU	Central processing unit
$\boldsymbol{C}$	Source vector
c_v	Specific heat capacity at constant volume
ECU	Electronic control unit
e_t	Specific total internal energy
FLOP	Floating point operation
$\boldsymbol{F}$	Flux vector
HIL	Hardware-in-the-loop
i	Cell number
$\dot{m}$	Mass flow rate
MUSCL	Monotonic Upstream-Centered Scheme for Conservation Laws
ODE	Ordinary differential equation
RANS	Reynolds-averaged Navier–Stokes
SIMD	Single instruction, multiple data
TVD	Total variation diminishing
T	Temperature
t	Time
U-RANS	Unsteady Reynolds-averaged Navier–Stokes
u	Flow speed
V	Volume
$\dot{W}$	Power
$\boldsymbol{w}$	State vector

Subscripts

$left$	Left-travelling wave
$model$	Model results
$RANS$	Reynolds-averaged Navier–Stokes
$right$	Right-travelling wave
$turb$	Turbine
u	Flow speed
0	Domain inlet
1	Turbine inlet
2	Stator inlet
3	Stator outlet
4	Rotor outlet
5	Turbine outlet
6	Domain outlet

Greek letters

Δ	Difference
ε	Error
ν	Courant number
ϕ	Velocity potential
ρ	Density

amplitude and frequency boundary conditions grow in importance. In one-dimensional codes, the main wave-action effects are supposed to happen in the volute, as it is the largest element of the turbine, just as is observed in CFD simulations [5]. The volute is solved as an equivalent one-dimensional duct of a given length and area distribution, what can be called a classical one-dimensional volute model. The main philosophy behind these models is shown in [6], where the volute is modelled using two tapered pipes. The first tapered pipe represents the turbine inlet duct, from the very beginning of the turbine to the volute tongue, with a length, inlet diameter and outlet diameter equal to the real ones. The second duct had a length equal to the length of the volute from the tongue to a point at 180°, passing through the central point of each section, setting the duct area to get the correct volute volume. This length selection was done supposing that half the mass flow enters the rotor at this