

Map-Based and 1D Simulation of a Turbocharger Compressor in Surging Operation

2011-24-0126

Published
09/11/2011Fabio Bozza
Universita di NapoliVincenzo De Bellis
Università di Napoli

Copyright © 2011 SAE International

doi:[10.4271/2011-24-0126](https://doi.org/10.4271/2011-24-0126)

ABSTRACT

One-dimensional (1D) models are commonly employed to study the performances of turbocharged engine. Manufacturers' provided steady turbomachinery maps are usually utilized, although they operate in unsteady conditions as a consequence of pressure pulses propagating into the intake and exhaust systems. This may lead to some inaccuracies in the engine-turbocharger matching calculations, which may be solved through the introduction of proper time-delays (virtual pipe corrections). These drawbacks, however, became more relevant when engine operates under low speed and high load conditions, or during a transient maneuver, because of possibilities of compressor surging. This phenomenon can't be opportunely described employing classical manufacturers' maps: first of all, the flow through the compressor is strongly unsteady in these conditions; moreover, the information about both the unstable and reverse flow regions of compressor map is unavailable, and a somewhat arbitrary map's extrapolation technique must be utilized.

In this paper two numerical procedures have been presented to predict automotive turbocharger compressor performance in surging operation. Firstly, a recently proposed approach is utilized to directly compute the stationary map of the compressor for direct and reverse flow conditions. The detailed 3D geometry of the compressor is reconstructed starting from a reduced set of geometrical data measured on the compressor wheel. Then, the 1D meanline evolution of blade angles, cross section and equivalent diameter are deduced from the 3D geometrical model. The steady 1D flow equations are finally solved along the meanline of stationary

and rotating channels constituting the compressor device. Proper flow loss correlation holding for direct or reverse flow conditions are included in the model, which is applied to predict the extended steady maps of a turbocharger compressor. With reference to stable operating regions, the theoretically derived maps are compared to available manufacturer data and, after a limited loss coefficients tuning, a good agreement has been obtained.

The steady procedure, however, can be utilized under unsteady flow conditions, too. Unsteady terms in flow equations are now accounted for, and mass and energy storages are allowed to occur within each stationary and rotating channel. In this way, compressor operation can be described with a quasi-steady method employing the extended map, or in a fully unsteady approach, as well. In the quasi-steady case, a proper virtual pipe correction is introduced, too. The two methodologies are tested to analyze their potential in characterizing compressor operation under surging conditions. To this aim, a reference compressor is connected to a outlet duct and a plenum. Compressor speed and downstream circuit are managed in order to promote deep surging conditions. Results obtained through the applications of the two methods are discussed in terms of pressure and temperature waves, mass flow rate evolution at compressor inlet and outlet, and Surging loop frequency, as well.

INTRODUCTION

One-dimensional (1D) models are usually employed to compute the engine and turbocharger matching conditions. They however require the availability of several information about the turbine and compressor behavior. A first item is

Table I. Modeling approaches followed to describe a typical compression system

Acronyms	Approach	Compressor	Pipes	Plenum
GM	Greitzer Model	Extended Steady-Map	0D Filling-Emptying	
MB	Map-Based Model	Extended Steady-Map	Unsteady 1D model	0D Filling-Emptying
1DC	1D Compressor Model	Unsteady 1D model	Unsteady 1D model	0D Filling-Emptying

related to available performance maps (provided by the manufacturer or directly measured), that are usually limited to a reduced range of rotational speed, pressure ratio and mass flow rate: this involves extrapolation of performance curves [1,2] introducing inaccuracies in the calculation procedure. Besides, performance maps are generally derived on steady-state test benches, while the turbocharger, when coupled to an automotive internal combustion engine, typically operates under unsteady flow conditions. This unsteadiness is still more important at low speed and high load, when the compressor operates near to surge line. In these conditions, turbocharger speed fluctuations or pressure pulses through intake system can reduce the mass flow rate below the surge line, inducing local unstable operation (soft-surfing), characterized by periodic oscillation of mass flow rate and pressure ratio. During surge operation, compressor efficiency is considerably reduced and high blade loading and casing stress levels may occur.

Many models have been developed to analyze compressor surge. Greitzer model (GM) is widely employed to describe this phenomenon both for axial and centrifugal compressors. It describes the dynamic behaviour of a compression system consisting of a compressor, a duct, a plenum volume and a throttle valve. The model is based on a non-linear lumped parameter theory [3,4,5] and provides mass flow rate and pressure temporal fluctuations both in the plenum and in the duct. In [6,7] GM was further developed including a variable compressor rotational speed obtained by torque balance of turbocharger. This approach still employs compressor maps, but requires the knowledge of both negative mass flow rate and unstable operating regions. The latter are hardly available because of difficulties in experimental measurements. For this reason, an arbitrary extrapolation of stable region of compressor map is generally used to extend compressor performance data to negative mass flow rate domain. GM follows a quasi-steady 0D approach, neglecting pressure wave propagation in the duct. The latter is included in the definition of proper non dimensional parameters (B , G and τ) depending on system geometry. It has been successfully utilized to describe the behavior of typical industrial compression systems [8, 9, 10] and analyze surge-suppression methods [6, 7, 8, 9, 10]. Anyway, its characteristics make it hardly applicable to the calculation of surging occurrence in a turbocharged internal combustion engine.

Following the approach developed in [11], in the present work, an innovative procedure is proposed to investigate a centrifugal compressor behaviour, both for reverse and direct flow conditions. Starting from a reduced set of data, compressor geometry is firstly reconstructed; the performance of the device are then evaluated solving the steady 1D equations in rotating and stationary pipes. Proper loss correlations are employed to take into account the main losses that occur in the device during normal and reverse flow conditions, as well. In this way, a steady 'extended' compressor map is obtained, to be used in a so called Map-Based (MB) approach. On the other hand, a more refined procedure solving fully unsteady 1D flow equations is developed, too, in order to better describe strongly unsteady and unstable operations (1D compressor model - 1DC).

The dynamic behaviour of a simple Greitzer-like compression system - constituted by a compressor, a pipe system and a plenum - is analyzed following the two above described approaches for the simulation of the compressor operation: map-based and 1D compressor models. In both cases, 1D flow equations are solved to compute wave propagation in external pipes, and a 0D filling and emptying technique is employed to describe plenum dynamics. Table I summarizes the modeling technique followed to describe each compression system component, compared to Greitzer's one.

Both MB and 1DC procedures have the potential to describe soft- and deep-surfing operation in an turbocharged internal combustion engine, where filling conditions are largely affected by the unsteady 1D flow evolution in the pipes. In this preliminary paper, however, they are applied, as said, to the simulation of a simple compression system. A throttle valve connected to the plenum volume is managed in order to promote deep surging conditions. The effects of geometrical variation of pipes' length and plenum volume are then analyzed with both MB and 1DC techniques. Differences will be highlighted in terms of instantaneous mass flow rate evolution at plenum inlet and surging cycle frequency.

Despite the absence of a direct experimental validation, the results presented are in good agreement with literature data and physical expectations. It will be shown, moreover, that only through a proper numerical treatment of the extended map, and the contemporary introduction of a 'virtual pipe' correction, the MB approach can give results quite

comparable to the 1DC model, constituting the reference most detailed approach. Due to the reduced computational requirement, a properly treated MB approach is however the most suitable one to be applied in advanced 1D engine models.

STEADY AND UNSTEADY 1D COMPRESSOR MODEL

In the following, the 1D compressor model is recalled, including a brief description of its geometrical schematization, of the 1D flow equations holding in stationary and rotating ducts, and of the boundary conditions and flow losses. A more detailed model discussion can be found in [12].

GEOMETRICAL MODEL

The compressor is schematized as follows (Fig. 1):

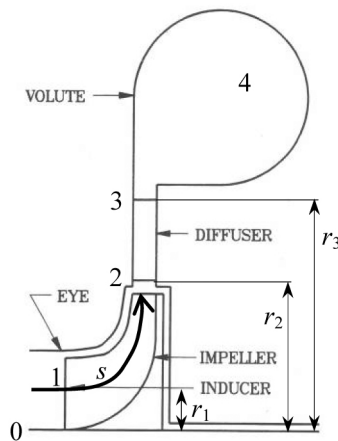


Figure 1. Main compressor stations

- a short inlet pipe (from section 0 to 1); the end section of the inlet pipe is characterized by a sudden area contraction, due to the impeller eye obstruction. This will cause some inlet losses, taken into account in the model;
- the impeller (from section 1 to 2), consisting of Z rotating pipes, Z being the number of blades. The profile of the blade-to-blade duct is assigned in terms of the local blade angle, equivalent and hydraulic diameters and radius. The latter are estimated perpendicularly to the meanline profile. Since upstream and downstream conditions are the same, the flow rate within a single impeller duct is really computed and then multiplied by the number of blades. A numerical procedure has been developed to reproduce the 3D impeller geometry basing on a reduced set of data (inlet and outlet blade angles, diameters, number of blades, etc.), which can be easily measured on the impeller wheel. The comparison between the actual impeller geometry and the CAD reproduced reconstruction is reported in Figure 2. The equivalent 1D profile of blade-to-blade duct, to be employed in 1D flow equations, is then derived from the reconstructed geometry;

- the vaneless diffuser (from section 2 to 3); it is considered as a diverging pipe of constant width in the radial direction;
- the volute (from section 3 to 4); it is schematized as a linearly variable diameter duct, collecting the flow coming from the diffuser and ending with a cone. Volute equivalent diameters at inlet and outlet are derived by diffuser outlet area and cone inlet diameter, respectively. An equivalent length is considered as half the volute meanline length.



Figure 2. Actual impeller geometry (IHI RHF3 compressor) and related 3D reconstruction

1DC FLOW MODEL

The unsteady 1D flow in the stationary ducts is described by the following equations:

$$C_t + [F(C)]_x = S_C$$

$$C = \begin{Bmatrix} \rho \\ \rho c \\ \rho E_C \end{Bmatrix} \quad F(C) = \begin{Bmatrix} \rho c \\ \rho c^2 + p \\ \rho c H_C \end{Bmatrix} \quad S_C = \begin{Bmatrix} \rho c \alpha \\ \rho c^2 \left(\alpha + 2 \frac{f}{d} \frac{c}{|c|} \right) \\ \rho c H_c \alpha - 4 \frac{q}{d} \end{Bmatrix} \quad (1)$$

The terms ρ , c , p , $E_C = c_v T + c^2/2$, $H_C = c_p T_0 = c_p T + c^2/2$ in the system (1) respectively represent density, absolute velocity, static pressure, total energy and enthalpy per unit mass. The source term, S_C , accounts for the duct area variation, $\alpha = 1/\Omega \, d\Omega/dx$, the friction coefficient, f , and the rate of heat exchange, q .

Impeller Flow Model

The system (1) is solved for the inlet pipe, the volute and the outlet duct, while, within the impeller, the following balance equations apply [13]:

$$\mathbf{W}_t + [\mathbf{F}(\mathbf{W})]_s = \mathbf{S}_W$$

$$\mathbf{W} = \begin{Bmatrix} \rho \\ \rho w \\ \rho E_w \end{Bmatrix} \quad \mathbf{F}(\mathbf{W}) = \begin{Bmatrix} \rho w \\ \rho w^2 + p \\ \rho w H_w \end{Bmatrix}$$

$$\mathbf{S}_W = - \begin{Bmatrix} \rho w \alpha \\ \rho w^2 \left(\alpha + \frac{2f_i}{d_i} \frac{w}{|w|} \right) + \frac{2f_s}{d_s} \rho c^2 \frac{c_w}{|c|} - \rho u^2 \delta \\ \rho w H_w \alpha - 4\rho \left(\frac{q_h}{d_h} + \frac{q_s}{d_s} \right) - \rho w u^2 \delta \end{Bmatrix} \quad (2)$$

In the above equations, w is the relative velocity, $u = \omega r$ is the tangential velocity (blade speed) and $E_w = c_v T + w^2/2$, $H_w = c_p T_{0w} = c_p T + w^2/2$ are the total energy and enthalpy in the relative motion. The source term $\mathbf{S}_W$ includes additional contributions arising as a consequence of pipe rotation. They are related to the centrifugal forces acting on the fluid particle and are computed as a function of the radius variation along a streamline: $\delta = 1/r \, dr/ds$. Friction forces are subdivided in two contributions, the first related to flow interaction with the impeller surface and the second concerning friction with the case inner surface. Also heat exchange is subdivided in two terms, the first depending on impeller/flow interaction and the second arising from heat exchange on the case.

Diffuser Flow Model

Along a flow streamline in the vaneless diffuser, a variation of both the radial, c_r , and the tangential velocity components, c_u , occurs, determining a quasi-1D flow. The system of equations solved for the vaneless diffuser is hence:

$$\mathbf{D}_t + [\mathbf{F}(\mathbf{D})]_r = \mathbf{S}_D$$

$$\mathbf{D} = \begin{Bmatrix} \rho \\ \rho c_r \\ \rho c_u \\ E_D \end{Bmatrix} \quad \mathbf{F}(\mathbf{D}) = \begin{Bmatrix} \rho c_r \\ \rho c_r^2 + p \\ \rho c_r c_u \\ \rho c_r H_D \end{Bmatrix}$$

$$\mathbf{S}_D = - \begin{Bmatrix} \rho c_r \alpha \\ \rho c_r^2 \alpha - \rho c_u^2 \delta + \frac{f}{b} \rho c^2 \frac{c_r}{|c|} \\ \rho c_r c_u \alpha + \rho c_r c_u \delta + \frac{f}{b} \rho c^2 \frac{c_u}{|c|} \\ \rho c_r H_D \alpha - \frac{2\rho q}{b} - \rho c_r c_u^2 \delta - \frac{f}{b} \rho c_u^2 \frac{c_u}{|c|} \end{Bmatrix} \quad (3)$$

being $E_D = c_v T + c_r^2/2$ and $H_D = c_p T_0 = c_p T + c_r^2/2$. With this definition of E_D and H_D , the energy balance equation shows an additional term deriving from tangential component of friction force. In this case, along the radial direction: $\delta = 1/r$.

The numerical solution of the partial derivative systems (1), (2), (3) is realized through a 2nd order accurate upwind scheme with a TVD (Total Variation Diminishing) flux correction technique [14]. Temporal integration follows a Runge-Kutta 4th order method.

STEADY FLOW MODEL

In the steady procedure, temporal derivatives in eqs. (1), (2) and (3) are obviously neglected. Computation starts with a fixed value of mass flow rate at compressor inlet duct. Flow equations are discretized in finite differences and numerically integrated with an iterative procedure. The steady flow model will be applied to computed the steady compressor map for map-based calculations (MB approach).

DIRECT FLOW BOUNDARY CONDITIONS (BCs)

The boundary conditions in the nodes of adjacent ducts are specified by applying a classical quasi-steady pipe-to-pipe junction problem. Mass and energy equations at the junction are solved. In the IDC model, in order to preserve the characteristics of wave propagation phenomena, the latter are coupled to the compatibility relations, properly modified to include the additional source terms. The momentum equation is indeed substituted by a total pressure loss relation between adjacent sections of the connected pipes. Formally, the total

pressure loss can be specified as a function of a Mach number expressing the total head loss, as:

$$p_{0ex} = p_{0in} \left(1 - \frac{\gamma - 1}{2} M_{loss}^2 \right)^{\frac{\gamma}{\gamma - 1}} \quad (4)$$

Loss related Mach number is opportunely specified according to the junction and the considered loss mechanism. Exactly the same pressure loss correlation are introduced in the steady procedure, too.

In direct flow operations, the present model takes into account all the main losses occurring in the compressor and slip effect, too. Incidence losses at impeller inlet, secondary impeller losses and volute losses are considered, as already reported in detail in [12]. Compared to previous works, an improvement is here included in the modeling of losses due to diffusion in the volute; in particular, a critical flow angle is defined as a function of inlet and outlet volute area. Below this critical value, the kinetic energy associated to the tangential velocity component is assumed to be partly lost at constant pressure. The radial velocity component is indeed supposed to be completely lost.

REVERSE FLOW BCs

This particular flow situation was not considered in the previous model version. More details concerning the modeling of reverse flow compressor operation will hence be given in the following. When mass flow rate is negative, proper velocity triangles at impeller inlet and outlet (relative to flow direction) have to be considered to describe the transition from absolute to relative motion. Moreover, incidence losses and flow direction in the passage from volute to diffuser have to be opportunely modeled. Computation starts at the volute outlet, where the flow is characterized by the absolute velocity c_4 , depending on upstream conditions and mass flow rate (section 4 in [fig. 3](#)).

Diffuser inlet (section 3)

The flow angle at diffuser inlet, α_3 , is set basing on the hypothesis of angular momentum and volumetric flow rate conservation (negligible density variations):

$$\begin{aligned} c_4 r_4 &= c_{3u} r_3 & c_4 \Omega_4 &= c_{3r} \Omega_3 \\ \tan \alpha_3 &= \frac{c_{3r}}{c_{3u}} = \frac{\Omega_4 r_3}{\Omega_3 r_4} \end{aligned} \quad (5)$$

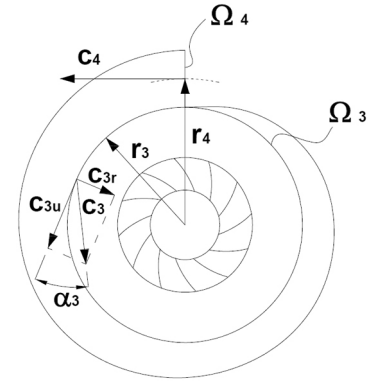


Figure 3. Velocity triangle at volute inlet and outlet in reverse flow condition

Impeller inlet (section 2)

Flow equations in the diffuser also define the radial and peripheral velocity components at the diffuser outlet. Friction losses in the diffuser are taken into account, too, with the same correlation holding for direct flow operation. Once the velocity c_2 is known, the relative velocity, w_2 , can be computed from the velocity triangle at the junction between the diffuser outlet and the impeller inlet, as typically shown in [Fig. 4](#).

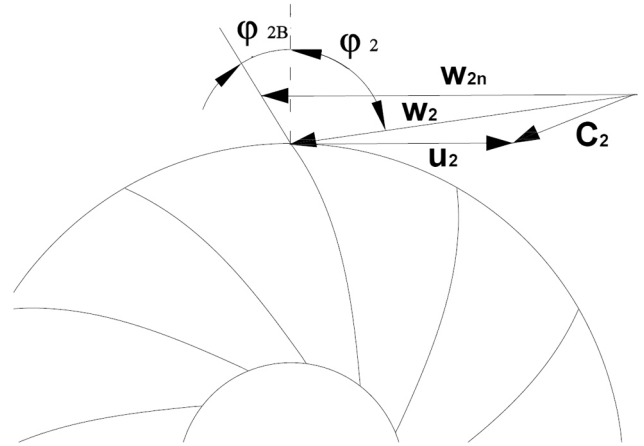


Figure 4. Velocity triangle at impeller inlet in reverse flow condition.

The relative velocity at the impeller inlet defines the flow angle φ_2 characterizing the direction of the approaching flow. Since the flow angle φ_2 greatly differs from the blade angle, φ_{2B} , a consistent percentage of the kinetic energy associated with the flow component w_{2n} ([Fig. 4](#)) is dissipated at a constant pressure. According to NASA Shock [15] incidence loss model, w_{2n} can be calculated as:

$$w_{2n} = c_{2u} - u_2 + c_{2r} \tan \varphi_{2B} \quad (6)$$