

Experimental Investigation and 1D Simulation of a Turbocharger Compressor Close to Surge Operation

Vincenzo De Bellis and Fabio Bozza
Univ of Naples

Silvia Marelli and Massimo Capobianco
Univ of Genoa

ABSTRACT

Downsizing is widely considered one of the main path to reduce the fuel consumption of spark ignition internal combustion engines. As known, despite the reduced size, the required torque and power targets can be attained thanks to an adequate boost level provided by a turbocharger. However, some drawbacks usually arise when the engine operates at full load and low speeds. In fact, in the above conditions, the boost pressure and the engine performance is limited since the compressor experiences close-to-surge operation. This occurrence is even greater in case of extremely downsized engines with a reduced number of cylinders and a small intake circuit volume, where the compressor works under strongly unsteady flow conditions and its instantaneous operating point most likely overcomes the steady surge margin.

In the paper, both experimental and numerical approaches are followed to describe the unsteady behavior of a small in-series turbocharger compressor. Measurements are carried out at the test facility of the University of Genoa. The compressor is included in the in-series intake circuit and a pulsating flow is generated by a motor-driven cylinder head fitted with a variable valve actuation system. Different rotational speeds and various valve opening strategies, characterized by different opening durations, are considered. Among the tested operating conditions, close-to-surge operation is also analyzed. The unsteady compressor behavior is investigated in terms of instantaneous inlet and outlet static pressures, and mass flow rate.

The numerical activity is performed at the University of Naples. The experimental test-rig is schematized in a commercial 1D code. The compressor is described following an enhanced map-based approach, where proper capacities are placed upstream and downstream the compressor to take into account the mass and energy storage effects. The model is validated for various rotational speeds and valve lift strategies, the numerical/experimental comparisons denoting a quiet good agreement.

After being validated, the methodology is employed to analyze the effect of the equivalent engine speed and valve strategy on the surge resistance. The results show that, for the higher speeds, the compressor only experiences soft-surge phenomena. On the contrary, below a specific speed, flow reversal within the intake system occurs and typical deep-surge cycles arise. In addition, valve lift strategies characterized by shorter opening durations seem to most likely promote the surge onset.

CITATION: De Bellis, V., Bozza, F., Marelli, S., and Capobianco, M., "Experimental Investigation and 1D Simulation of a Turbocharger Compressor Close to Surge Operation," *SAE Int. J. Engines* 8(4):2015, doi:10.4271/2015-01-1720.

INTRODUCTION

Modern internal combustion engines (ICEs) have to comply with more and more stringent constraints in terms of pollutant emissions and CO₂ production [1]. In addition, the costumers are devoting an increasing attention towards the fuel economy, without renouncing to the performance.

The engine turbocharging is one of the solutions employed to address the above objectives both for compression ignition (CI) and spark ignition (SI) engines. As well known, in case of CI engines, the turbocharging allows to attain high specific power levels, while

maintaining a competitive fuel consumption and reducing the pollutant emissions [2]. In case of SI ICEs, the turbocharging is usually applied to units of reduced size with the aim of reducing the fuel consumption at part loads, while preserving the maximum power thanks to an adequate boost level (*downsizing*) [3,4,5].

However, the engine performance at low speeds is usually limited by the attainment of unstable operating conditions for the compressor, widely known as surge. The latter phenomenon bounds the maximum boost level and, consequently, limits the delivered torque (and torque).

The compressor maps are usually utilized to derive information about the surge limit. They are measured at proper test rigs in which, however, steady state operations are realized. On the other hand, the compressor experiences unsteady operations when coupled to an ICE, because of the opening and closure of the intake valves. The latter occurrence is even more true in case of small engines, constituted by a reduced number of cylinders and a small intake circuit volume.

The unsteadiness induced by the engine strongly affects compressor behavior [6] and, in particular, its capability to locally operate close to the surge [7,8]. Besides, the adoption of VVA systems can cause a more prominent deviation of instantaneous compressor performance from the steady state operation [9].

As known, one-dimensional (1D) models are commonly used to compute the engine and turbocharger matching conditions. The classical quasi-steady approach, based on the employment of compressor and turbine characteristic maps, is usually followed [10]. To better take into account the effects of the pulsating flow, the turbomachine geometry can be synthesized in terms of 1D equivalent ducts [11,12].

However, since typical steady maps do not include information about the unstable and reverse flow operation of the device, their employment within a 1D code hardly limits the possibility to describe surge phenomena. A dedicated sub-model should be hence developed to this aim.

Among the proposals which can be found in the current literature, the Greitzer model (GM) is widely employed to describe the surge both for axial and centrifugal compressors. The GM follows a quasi-steady 0D approach, neglecting pressure wave propagation in the ducts upstream and downstream the compressor. In particular, it describes the dynamic behavior 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 [13,14,15] and provides mass flow rate and pressure temporal fluctuations both in the plenum and in the ducts. A proper time delay is applied in the access to the compressor map, with the aim of reproducing a certain shift in the device response induced by the pressure waves propagation. The GM has been successfully utilized to analyze the operation of typical industrial compression systems [16,17,18].

However, this approach still employs compressor maps and, in addition, requires the knowledge of both negative mass flow rate and unstable operating regions. The latter are usually unavailable because of difficulties in their experimental measurements [19].

In a standard GM, an arbitrary extrapolation of the stable compressor map region is generally used to extend compressor performance data to negative mass flow rate domain. However, because of its “lumped parameters” nature, the GM cannot be directly coupled to a 1D simulation of a whole engine. A possible solution [7,20] is the introduction of a certain damping in the device response by means of a first order variation model. This approach, still employing the

steady state map, has the aim to take into account the fluid inertia within the device. However, also in this case, the availability of an extended map, including negative branches, is still mandatory.

In this work, a small in-series turbocharger compressor has been experimentally characterized under both steady and unsteady flow operation at a test rig operating at the University of Genoa [9,19,21]. The test facility consists of the entire intake line of a commercial twin-cylinder engine, including the cylinder head and a fully flexible VVA system to control the intake valves. The latter is used to promote unsteady operation. The compressor behavior is analyzed for various rotational speeds and valve strategies. Some experimental tests are performed controlling the device as close as possible to the compressor surge limit, but without incurring in flow reversal.

The above intake system is then modeled within the well-known GT-Power software. The compressor is described following an enhanced map-based approach, in which two “properly sized” capacities are included upstream and downstream the compressor-junction, to account for the mass and energy accumulation phenomena. An extended compressor map is computed by a 1D compressor model (1DCM) developed in the recent years by the University of Naples [22,23]. This innovative methodology is based on the solution of the 1D steady state flow equations within the stationary and rotating channels constituting the device, starting from a reduced set of geometrical data. A direct modelling of main phenomena and losses is performed, also referring to correlations from the literature. The compressor model provides as an output a complete map, also extended to the unstable and negative mass flow rate regions. The extended map and the capacities included in the compressor model represent the basis for the development of an innovative approach aiming to improve the surge description and its onset detection.

In a first stage, the model is validated through comparisons of instantaneous pressures upstream and downstream the compressor, and in terms of downstream mass flow rate, the numerical results denoting a satisfactory agreement with the experimental outcomes. In a second stage, the model is used to investigate the influence of the valve opening frequency and of the VVA strategy with respect to compressor surge resistance capability.

EXPERIMENTAL SETUP

The experimental investigation is carried out at the turbocharger test facility of the University of Genoa, fully described in previous papers [9,19,21]. The test bench is a hot and cold-air apparatus (up to 750 °C) which allows to perform investigations on intake and exhaust automotive components under both steady and unsteady flow conditions. One of the main feature of the facility is the availability of different pulse generator systems which allow to study turbine and compressor behavior under unsteady flow condition, typically occurring in automotive application. Measured quantities presented in the paper are referred to the experimental set-up reported in [Figure 1](#). Tested turbocharger compressor is coupled to an in-series engine intake circuit and to a motor driven cylinder head fitted with a VVA system (MultiAir by Centro Ricerche Fiat).

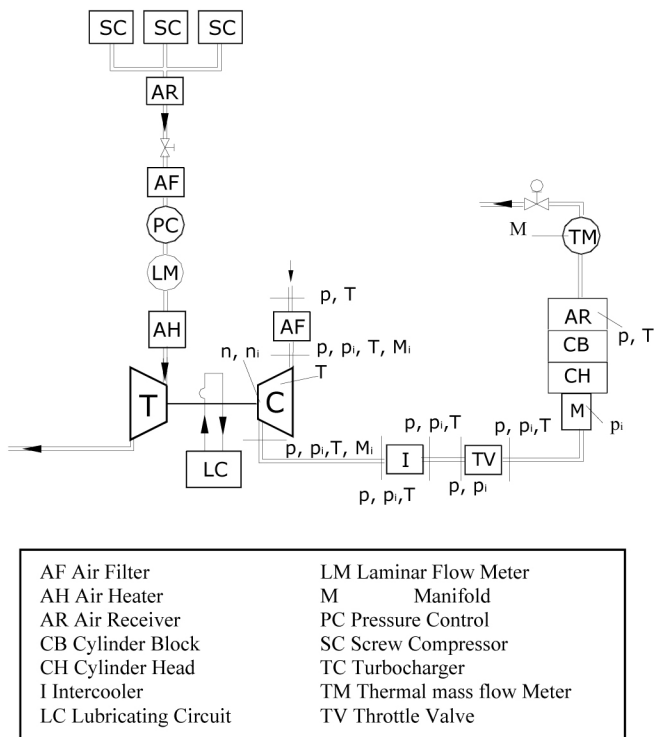


Figure 1. Experimental setup at University of Genoa turbocharger test rig

Different measurements are performed upstream and downstream the compressor. Average and instantaneous pressure levels are measured through strain-gauge transducers (accuracy of 0.14% of full range output), while the average temperature is measured by using platinum resistance thermometers (accuracy of $\pm 0.15^\circ\text{C} + 0.2\%$ of measured value). The average compressor mass flow rate is measured through a thermal mass flow meter (accuracy of 0.9% of measured value and 0.05% of the full scale). Instantaneous compressor mass flow rate is detected by using a hot wire fiber film probe, mounted at the outlet. The sensor is calibrated against the thermal mass flow meter under steady flow conditions over the whole operating range experienced during unsteady flow tests, also considering the effect of temperature variation [9, 21]. The turbocharger rotational speed is recorded through an eddy current probe (with a full-scale frequency precision of 0.009%) mounted close to the compressor wheel. A pick-up probe faced to a camshaft connected phonic wheel is adopted for the measurement of the camshaft speed, used to control the VVA system. A Hall sensor installed in front of the driving camshaft, phased 40 crank angle degrees after TDC related to cylinder #1 overlap, is used to trigger the data acquisition. Two high speed synchronized data acquisition cards, managed by interactive procedures, are used to record instantaneous signals. All measured signals reported in the paper are an ensemble average of several complete cycles.

The unsteady flow investigation is carried out referring to a Mitsubishi turbocharger, with a single entry nozzle-less radial flow waste-gated turbine, matched to a downsized twin-cylinder SI engine with a capacity of approximately 0.9 liters.

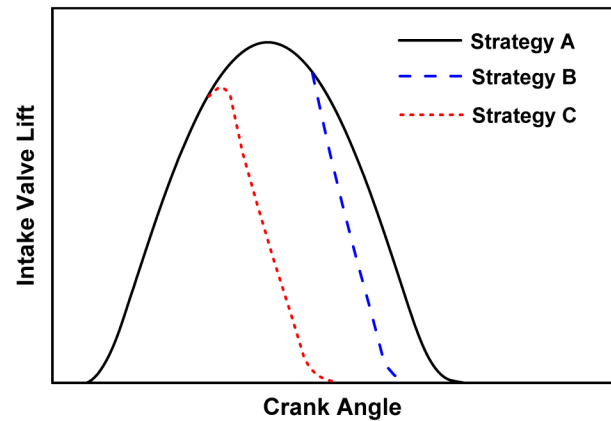


Figure 2. Valve lift strategies

In a first step of the experimental investigation, the steady flow characteristic curves are measured at the test facility for various corrected rotational speeds. An unsteady flow investigation is then performed on compressor coupled to the in-series engine intake circuit and the motor driven cylinder head fitted with the MultiAir system. Different constant levels of corrected compressor rotational speed (80000, 100000, 120000 and 140000 rpm) are considered, taking into account mass flow operating conditions from the stable zone up to the surge line. Two equivalent engine rotational speed of 1500 and 1750 rpm are reproduced. For each operating condition, three valve actuation strategies are considered, corresponding to the full lift profile and two different early closing lifts. Figure 2 qualitatively reports the valve strategies employed (referred as A, B and C).

1D COMPRESSOR MODEL (1DCM)

The compressor model includes a geometrical module that provides the data required to solve the 1D flow equations inside the different ducts composing the device. In particular, the geometrical schematization is constituted by four pipes: inlet duct, impeller, diffuser and volute. Figure 3 draws the main stations along the fluid path.

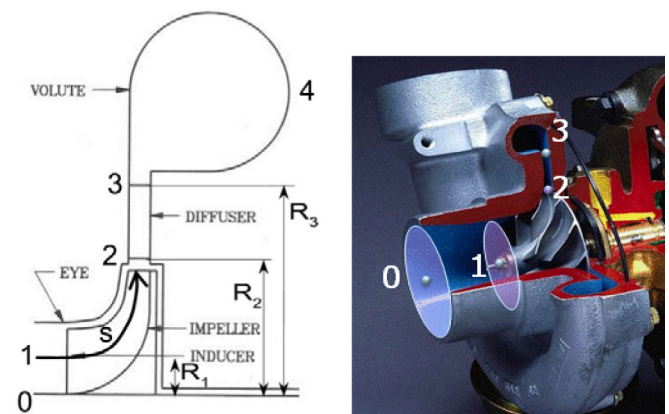


Figure 3. Main compressor stations

In the following, the 1D compressor model is recalled, including a brief description of the geometrical schematization, the 1D flow equations holding in stationary and rotating ducts, and the boundary conditions and flow losses. A more detailed illustration of the model is reported in [22,23].

Geometrical Schematization

The compressor is schematized as follows (Figure 3):

- A short inlet pipe (from section 0 to 1) of constant diameter and length; the inlet pipe end section is characterized by a sudden area contraction, due to the impeller eye obstruction.
- The impeller (from section 1 to 2), consisting of a Z rotating pipes, Z being the number of blades. A sub-module rebuilds the 3D impeller geometry based on a reduced set of data (inlet and outlet blade angles, diameters, number of blades, etc.), which can be easily measured on the real component. The procedure provides as an output the cross section area, the hydraulic diameter, the local blade/rotation angles and the radius evolution along the curvilinear abscissa.
- The vaneless diffuser (from section 2 to 3); it is considered as a diverging pipe of constant width along the radial direction.
- The volute (from section 3 to 4); it is schematized as a constant diameter duct, collecting the flow coming from the diffuser and ending with a cone. The volute equivalent diameter is derived by the diffuser outlet area. An equivalent length is considered as half the volute centerline length.

1D Flow Model

Fixed Ducts

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

$$[\mathbf{F}(\mathbf{C})]_x = \mathbf{S}_C \quad (1)$$

$$\mathbf{C} = \begin{Bmatrix} \rho \\ \rho c \\ \rho E_C \end{Bmatrix} \quad \mathbf{F}(\mathbf{C}) = \begin{Bmatrix} \rho c \\ \rho c^2 + p \\ \rho c H_C \end{Bmatrix} \quad (2)$$

$$\mathbf{S}_C = - \begin{Bmatrix} \rho c \alpha \\ \rho c^2 \left(\alpha + 2 \frac{f}{d} \frac{c}{|c|} \right) \\ \rho c H_C \alpha - 4 \frac{q}{d} \end{Bmatrix} \quad (3)$$

The terms ρ , c , p , $H_C = c_p T_0 = c_p T + c^2/2$ in the system (1), (2), (3) represent the density, the absolute velocity, the static pressure, the total enthalpy per unit mass, respectively. The source term, $\mathbf{S}_C$, accounts for the friction coefficient, f , the rate of heat exchange, q , and the duct area variation, $\alpha = 1/\Omega \, d\Omega/dx$. The latter is provided by the previously described geometrical module at various locations along the duct.

Rotating Ducts

The system (1), (2), (3) is solved for the inlet pipe and the volute, while, within the impeller, the following balance equations apply [24]:

$$[\mathbf{F}(\mathbf{W})]_s = \mathbf{S}_W \quad (4)$$

$$\mathbf{W} = \begin{Bmatrix} \rho \\ \rho w \\ \rho E_W \end{Bmatrix} \quad \mathbf{F}(\mathbf{W}) = \begin{Bmatrix} \rho w \\ \rho w^2 + p \\ \rho w H_W \end{Bmatrix} \quad (5)$$

$$\mathbf{S}_W = - \begin{Bmatrix} \rho w \alpha \\ \rho w^2 \left(\alpha + \frac{2f_i}{d_i} \frac{w}{|w|} \right) + \frac{2f_s}{d_s} \rho c^2 \frac{c_w}{|c|} - \rho u^2 \delta \\ \rho w H_W \alpha - 4 \rho \left(\frac{q_h}{d_h} + \frac{q_s}{d_s} \right) - \rho w u^2 \delta \end{Bmatrix} \quad (6)$$

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

Diffuser flow

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

$$[\mathbf{F}(\mathbf{D})]_r = \mathbf{S}_D \quad (7)$$