

Unsteady 1D Simulation of a Turbocharger Compressor

F. Bozza and A. Gimelli
Università di Napoli

Copyright © 2009 SAE International

ABSTRACT

The one-dimensional (1D) modeling of a turbocharged engine requires the availability of the turbine and compressor characteristic maps. This leads to two main problems:

- performance maps of the turbocharger device are usually limited to a reduced number of rotational speeds, pressure ratios and mass flow rates. Extrapolation of maps' data is commonly required;
- performance maps are experimentally derived on stationary test benches, while the turbocharger usually operates under unsteady conditions, when coupled to an internal combustion engine (ICE).

To overcome the above problems, in the present paper the flow inside a rotating pipe of a centrifugal compressor is simulated within a 1D modeling approach, with the aim of predicting its characteristic map. The main improvement with respect to the employment of a steady experimental map consists in the absence of data extrapolation and in the possibility of fully characterizing the unsteady operation of the component. In this way it is also possible to handle on a physical point of view surge phenomena (backflows) which may arise particularly at low engine speed and high load.

To this aim, the actual evolution of the blade-to-blade duct profile is specified through the assignment of proper data which define the duct orientation in the space along its curvilinear abscissa. Flow equations are next generalized to include additional terms arising as a consequence of pipe rotation. The whole compressor is schematized as a number of rotating channels in parallel, linked at the impeller inlet and outlet by a constant

pressure boundary condition. The velocity triangles are employed to handle the transition between the absolute and the relative motion occurring at the impeller inlet and outlet. The presence of a vaneless diffuser is also considered at the exit section of the rotor blades. The procedure includes some correlations to keep into account slip effects, incidence and distributed losses. The above methodology is utilized to directly compute the stationary map of the device, starting from the specification of its geometry and rotating speed. A comparison with experimental data shows a good agreement with the predicted performance curves. The same procedure can be easily extended to the simulation of a radial turbine, too.

INTRODUCTION

Current literature is producing more and more numerous contributions dealing with internal combustion engine turbocharging [1,2].

As an example, complex layouts of the turbocharging system have been recently proposed for heavy-duty diesel engines [3]. High exhaust gas recirculation (EGR) rates are in fact required to meet future emissions demands. This also requires a considerably higher boost pressure to maintain the air excess ratio [4]. A precise managing of the boost level is also needed to extend the operating domain of homogeneous charge compression ignition (HCCI) engines and to control the autoignition time [5].

More recently, manufacturers are devoting a growing attention to the development of modern turbocharging systems for spark-ignition (SI) engines, too.

The revival of turbocharged SI engine is based on the need for fuel economy improvement [6,7]. This can be accomplished by reducing the engine size and limiting the throttle usage and pumping losses (engine downsizing) [8,9]. In order to reach the performance level of larger engines, a turbocharging system is employed, also realizing a higher low-speed torque, a better vehicle drivability and fun-to-drive.

During the development of a new turbocharged engine, the computation of the engine and turbocharger matching conditions is a very common task. This is usually realized through the employment of 1D codes [10], which can also provide the boundary conditions for more refined 3D CFD analyses [11]. These procedures require the specification of the turbine and compressor characteristic maps, provided by the manufacturer. Anyway, in many cases, a limited operating domain of the turbocharger device is really available, and various maps data extrapolation techniques are commonly included within commercial 1D codes [12,13,14].

In addition, due to various phenomena, the actual compressor behavior may differ from the one expressed by the characteristic map. First of all, performance data refer to a steady operation of the device, while the compressor, due to the pressure pulsations caused by the engine [15,16], always operates under unsteady conditions. Due to packaging of the inlet duct, the incoming air may be not uniformly distributed across the inlet section of the compressor wheel, determining a variation of the flow direction at the inlet. Moreover, heat transfer through the casing and heat radiated by the turbine and the engine may affect the map based values of the adiabatic efficiency.

Due to the above considerations, an improvement of the classical map-based quasi-steady matching calculation is required [17]. In the present paper, a 1D simulation procedure is developed to characterize the unsteady behavior of a small centrifugal compressor. Although not included in the present study, the proposed approach has the potential to also describe soft-surge phenomena, which may arise particularly at low engine speed and high load. To reach the above goal, the geometry of the blade-to-blade duct profile is specified through the assignment of proper data which define the duct orientation in the space along its curvilinear abscissa. The whole compressor is schematized as a series of five pipes: the inlet duct, the impeller, the diffuser, the volute and the outlet duct. The impeller itself is considered as a number of rotating channels in parallel.

In the following paragraphs, the 1D flow equations referred to a rotating frame are reported. Moreover, a particular treatment of the angular momentum conservation equations is discussed, which allows reducing the two-dimensional flow inside the diffuser to an equivalent one-dimensional problem. A proper development of the boundary conditions is then described to handle the transition between the absolute and the relative motion occurring at the impeller inlet and

outlet. The unsteady methodology is firstly utilized to compute the behavior of a reference compressor under isentropic flow conditions. The results are compared with the exact solution to verify the reliability of the numerical technique. Then, some correlations are included in the procedure to keep into account slip effects, incidence and distributed losses. The stationary map of a typical turbocharger compressor is next computed under real flow conditions, and the results are compared with experimental performance data. Finally, an example of a fully unsteady calculation is presented.

1D COMPRESSOR MODEL

GEOMETRICAL MODEL – The compressor is schematized as follows (fig. 1):

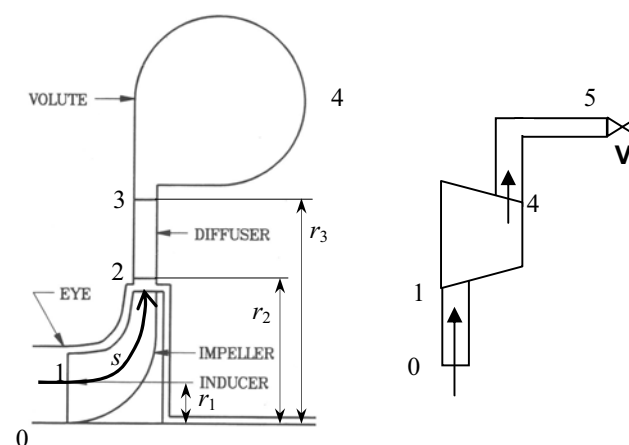


Fig. 1 – Main compressor stations

- a short inlet pipe, (from section 0 to 1); the orientation of the inlet pipe is employed to define the flow direction α_1 at the inducer inlet;
- 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 inlet and outlet blade angles β_{1c} and β_{2c} (fig. 2) and duct length. In order to characterize the flow area, the distribution of the equivalent diameter along its curvilinear abscissa, s , is also assigned. The latter is estimated perpendicularly to the blade profile. In addition, the distribution of the radius variation from r_1 to r_2 is prescribed to compute the local centrifugal forces acting on the fluid;
- the vaneless diffuser (from section 2 to 3), considered as a diverging pipe of constant width in the radial direction. The flow area increase depends on the radius variation and directly affects the pressure recovery related to the radial flow component. The one related to the tangential component is, indeed, included in the flow equations as a source term, as discussed in the following;

- the volute (from section 3 to 4), schematized as a constant area duct, collecting the flow contributions coming from the Z impeller channels. An equivalent length is considered as half the circumference at the radius r_3 .
- the outlet pipe (from section 4 to 5), of constant area, ending with a throttle valve, V , controlling the delivered mass flow rate.

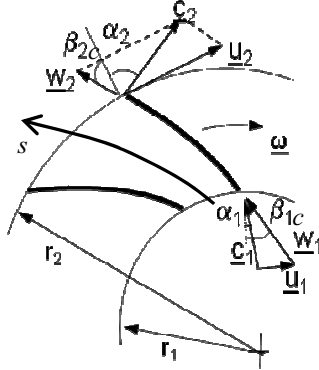


Fig. 2 – Velocity triangles at the impeller inlet and outlet

FLOW MODEL – The 1D flow in the stationary ducts is described by the following equations:

$$\mathbf{C}_t + [\mathbf{F}(\mathbf{C})]_x = \mathbf{S}_C$$

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

$$\mathbf{S}_C = - \begin{Bmatrix} \rho c \alpha \\ \rho c^2 (\alpha + 2f/d c/|c|) \\ \rho c H_C \alpha - 4q/d \end{Bmatrix} \quad (1)$$

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, $\mathbf{S}_C$, 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 [18]:

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

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

$$\mathbf{S}_W = - \begin{Bmatrix} \rho w \alpha \\ \rho w^2 (\alpha + 2f/d w/|w|) - \rho u^2 \delta \\ \rho w H_W \alpha - 4q/d - \rho w u^2 \delta \end{Bmatrix} \quad (2)$$

In the above equations, w is the relative velocity, $u = \omega r$, is the tangential velocity (blade speed) and $E_W = c_v T + w^2/2$, $H_W = c_p T_0 = c_p T + w^2/2$ 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: $\delta = 1/r \, dr/ds$. Coriolis forces, indeed, always being perpendicular to the relative velocity, do not contribute to the shaft work and are not included in the system (2).

Diffuser Flow Model - Along a flow streamline in the vaneless diffuser, a variation of both the radial and the tangential velocity components, c_r , c_u , occurs, determining a 2D flow (fig. 3).

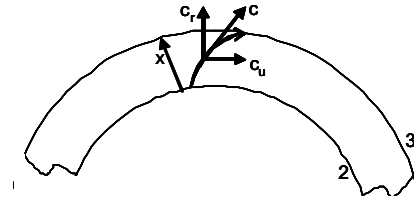


Fig. 3 – Velocities in the vaneless diffuser

In order to retain the overall 1D modeling structure of the procedure, the sole radial component is really integrated, while the other component is directly derived from the conservation of the tangential momentum:

$$c_u \cdot r = \text{const.} \quad (3)$$

Under this hypothesis, the system of equations solved for the vaneless diffuser is:

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

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

$$\mathbf{S}_D = - \begin{Bmatrix} \rho c_r \alpha \\ \rho c_r^2 (\alpha + 2f/d c_r/|c_r|) - \rho c_u^2/r \\ \rho c_r H_D \alpha - 4q/d - \rho c_r c_u^2/r \end{Bmatrix} \quad (4)$$

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 formulation, the pressure recovery related to the diminishing of the tangential component in the diffuser is included as a source term in the system (4). Noting that along the radial direction: $\delta = 1/r$, the system (4) presents exactly the same formal relationships of the system (2).

The numerical solution of the systems (1), (2) and (4) is realized through a 2nd order accurate upwind scheme with a TVD flux correction technique [19].