

1D Simulation and Experimental Analysis of a Turbocharger Compressor for Automotive Engines under Unsteady Flow Conditions

2011-01-1147

Published
04/12/2011

Fabio Bozza and Vincenzo De Bellis
Universita' di Napoli

Silvia Marelli and Massimo Capobianco
Universita' di Genova

Copyright © 2011 SAE International

doi:[10.4271/2011-01-1147](https://doi.org/10.4271/2011-01-1147)

ABSTRACT

Turbocharging technique will play a fundamental role in the near future not only to improve automotive engine performance, but also to reduce fuel consumption and exhaust emissions both in Spark Ignition and diesel automotive applications. To achieve excellent engine performance for road application, it is necessary to overcome some typical turbocharging drawbacks i.e., low end torque level and transient response. Experimental studies, developed on dedicated test facilities, can supply a lot of information to optimize the engine-turbocharger matching, especially if tests can be extended to the typical engine operating conditions (unsteady flow). Different numerical procedures have been developed at the University of Naples to predict automotive turbocharger compressor performance both under steady and unsteady flow conditions. A classical 1D approach, based on the employment of compressor characteristic maps, was firstly followed. A different and more refined procedure has been recently proposed. The new approach is based on the solution of the 1D unsteady flow within the stationary and rotating channels constituting the compressor device, starting from a reduced set of geometrical data. The refined methodology can be utilized to directly compute the stationary map of the compressor but also to reproduce the unsteady flow behavior of the device.

A specialized components test rig (particularly suited to study automotive turbochargers) has been operating since several years at the University of Genoa. The test facility also allows to develop studies under unsteady flow conditions both on

single components and subassemblies of engine intake and exhaust circuit.

In the paper the results of a preliminary experimental study developed on a turbocharger compressor for gasoline engine application under unsteady flow conditions are presented. Instantaneous inlet and outlet static pressure and mass flow rate are compared with the corresponding numerical data supplied by simulation codes. The numerical results showed a good agreement with experimental data. In addition, the comparison between the classical and the refined procedure results highlighted the potential of the performed unsteady 1D calculation, especially in specific compressor operating conditions.

The integration of the experimental activity with the numerical analysis represents a methodology that can be helpfully employed during the design process of internal combustion engine intake systems.

INTRODUCTION

Engine downsizing is becoming a successful way of improving fuel consumption in automotive applications. As regards spark ignition (SI) engines, this technique, associated with charge boosting and other state-of-the art technologies (such as gasoline direct injection and fully flexible intake valves control), can help to substantially reduce CO₂ emissions, thus making this propulsion system competitive with the diesel engine in terms of fuel consumption, while keeping exhaust pollutant emissions very low by using the three-way catalyst [1, 2].

Therefore, engine turbocharging, today used in all diesel applications, will be extensively applied also to SI automotive units as a way to reach the required performance level while downsizing the engine [3]. However, an effective management of the turbocharging system to achieve a suitable engine torque curve and an excellent vehicle driveability is a very difficult task [4], especially in the case of small SI engines fitted with a reduced number of cylinders (often lower than four in recent units). This is due to the wide load and speed range experienced in road applications and to the magnitude of unsteady flow effects in the intake and exhaust circuit.

One-dimensional (1D) models are usually employed to compute the engine and turbocharger matching conditions. However, 1D modelling of an automotive turbocharged engine requires the availability of several information about the turbine and compressor behavior. A first item is related to available performance maps, that are usually limited to a reduced range of rotational speed, pressure ratio and mass flow rate: this involves extrapolation of performance curves 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.

It is therefore apparent that the integration of experimental activity with numerical procedures able to take into account the unsteady behavior of the turbocharger, represents a methodology that can be helpfully employed in the design process of the intake and exhaust systems of turbocharged automotive engines.

As above mentioned, the extension of turbocharger turbine and compressor maps to a broad operating range and the provision of turbocharger behavior when operating under severe unsteady flow conditions are currently fundamental requirements for improving the accuracy of turbocharged engine calculations. To this end, the availability of highly flexible test rigs can allow all these aspects to be investigated without the limits imposed by the engine, thus providing more general and transferable information concerning turbocharger operation. Since several years, a specialized components test facility (particularly suited to study automotive turbochargers) has been operating at the University of Genoa [5]. The test rig is fitted with specific devices which allow investigations to be performed under unsteady flow operation both on single components and subassemblies of engine intake and exhaust circuit. Several experimental studies on automotive turbocharger turbines, also fitted with different regulating devices (waste-gate valve and variable geometry systems), have been performed using this facility [6, 7, 8, 9, 10].

Different numerical procedures have been developed in order to predict automotive turbocharged engine performance both under steady and unsteady flow conditions. A classical quasi-steady approach, based on the employment of compressor and turbine characteristic maps (provided by the manufacturers or directly measured) is usually followed [11, 12]. Anyway, in many cases, a limited operating domain of the turbocharger device is available, and extrapolation of maps is commonly required [12, 13]. In addition, due to various phenomena, the actual turbocharger behavior may differ from that expressed by the characteristic maps. First of all, performance data refer to a steady operation of the device, while the turbocharger, due to the pressure pulsations caused by the engine, always operates under unsteady conditions [5, 10]. A first attempt to better take into account the effects of the pulsating flow, is the discretization of the compressor or turbine in terms of equivalent lengths and volumes, together with their steady flow characteristics [14]. However, other inaccuracies related to the employment of steady maps can be due to the variation of the flow direction at the compressor or turbine inlet, depending on the actual packaging of the engine intake and exhaust circuits [27]. Moreover, heat transfer through the casing and heat radiated by the turbine and the engine may considerably affect the map-based values of the adiabatic efficiency. In addition, in a turbocharged engine with a long-route Exhaust Gas Recirculation device [15], the compressor operates with a variable mixture of air and recycled burned gases, while gas composition effects are usually neglected in a map-based approach.

Due to the above considerations, an improvement of the classical map-based quasi-steady matching calculation is required [16]. Concerning a small turbocharger compressor, a different and more refined procedure has been recently proposed [17] to characterize its behavior both under steady and unsteady flow conditions. The new approach is based on the solution of the 1D unsteady flow within the stationary and rotating channels constituting the compressor device, starting from a reduced set of geometrical data. A direct modeling of main phenomena and losses is performed, also referring to correlations from the literature. A tuning procedure allows a better definition of the compressor steady flow map by calibrating the constants included in the various loss correlations. Due to the calculation approach, the procedure allows to reproduce both steady and unsteady flow behavior of the compressor. In the paper the results of a preliminary experimental study developed at the University of Genoa on a turbocharger compressor for gasoline engine application under steady and unsteady flow conditions are presented. Measured performance are compared with the corresponding numerical data supplied by the simulation procedure developed at the University of Naples.

In a first step the model was applied to calculate the compressor steady flow map, referring both to the standard value of the friction and slip correlation multipliers and to a

tuning of the same constants. The theoretical procedure was then used to predict instantaneous levels of inlet and outlet static pressure and mass flow rate, following both a classical approach, based on the employment of the tuned steady flow compressor map, and a more refined procedure, based on the direct time integration of model equations.

The numerical results showed a good agreement with experimental data. In addition, the comparison between the classical and the refined procedure results highlighted the advantages of the performed unsteady 1D calculation, especially in specific engine operating conditions.

EXPERIMENTAL ANALYSIS

The experimental investigation is carried out on the University of Genoa (UNIGE) test facility (Fig. 1), fully described in previous papers [8, 10]. Three electrical screw compressors provide a total mass flow rate of about 0.6 kg/s at a maximum pressure of 8 bar. To prevent water from condensing and freezing, a 54 kW electrical heater can increase air temperature up to 400 K. Two independent feeding lines allow to supply turbocharger turbine and compressor with air at controlled pressure thus allowing to measure steady flow maps over an extended range.

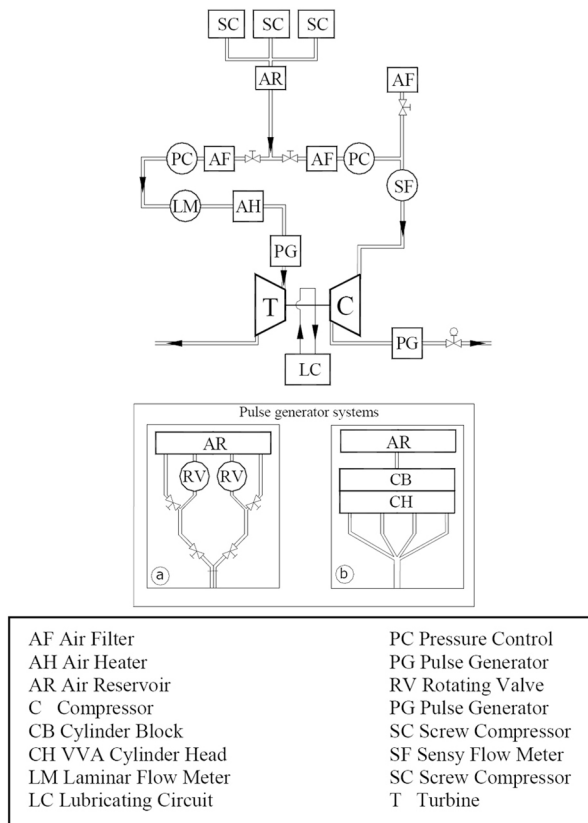


Figure 1. Schematic layout of UNIGE turbocharger test facility.

The main feature of the test rig is the availability of two different systems to generate the unsteady flow typically occurring in the intake and exhaust automotive engine circuit. In a first configuration, diametral slot rotating valves are adopted in order to perform parametric studies by properly controlling the main characteristics of generated pressure pulses (i.e. mean value, amplitude, shape and pulse frequency). The second layout, including a Variable Valve Actuation cylinder head (Fiat Multiair system) makes it possible to analyze the interactions between the engine circuit and both turbocharger turbine and compressor.

Measurement of different parameters under steady and unsteady flow conditions is performed by an automatic data acquisition system using dedicated procedures developed in Lab VIEW® environment.

The experimental investigation was performed on a small automotive turbocharger (IHI RHF3) matched to a small-size spark ignition engine. Initially, the reference compressor steady flow maps, expressed in terms of both total-to-total pressure ratio and total-to-total efficiency versus corrected mass flow rate, were measured at different corrected rotational speed levels. Secondly, the compressor performance was evaluated under unsteady flow conditions by integrating the turbocharger in a current production automotive engine intake circuit (Fig. 2). Results reported in the paper are referred to a constant level of corrected compressor rotational speed (about 100000 rpm), taking into account three different mass flow operating conditions (near the choking area, in an intermediate zone and near the surge line, respectively referred later as CASES A, B and C). The unsteady flow pulse frequency considered corresponds to an equivalent engine rotational speed of 1500 rpm (camshaft speed of 750 rpm).

The average and instantaneous levels of several parameters were recorded in different measuring section located along the intake circuit (i.e. at the inlet and outlet section of both compressor and intercooler, downstream the throttle valve, etc., see Fig.2). Average and instantaneous static pressure were measured through strain gauge transducers characterized by a high frequency response (about 25kHz) and an accuracy of 0.14% of the full scale. Platinum resistance thermometers with an uncertainty of ± 0.3 °C were used to measure the average temperature values. The instantaneous temperature levels were evaluated referring to an adiabatic process of an ideal gas, starting from measured mean inlet pressure and temperature and instantaneous static pressure. Instantaneous mass flow rate was estimated by using a hot wire fiber film probe, mounted upstream of the compressor. The sensor was calibrated against a thermal mass flow meter (with an accuracy of $\pm 0.3\%$ of measured level) in steady state over the entire operating range experienced under unsteady flow conditions. The turbocharger rotational speed

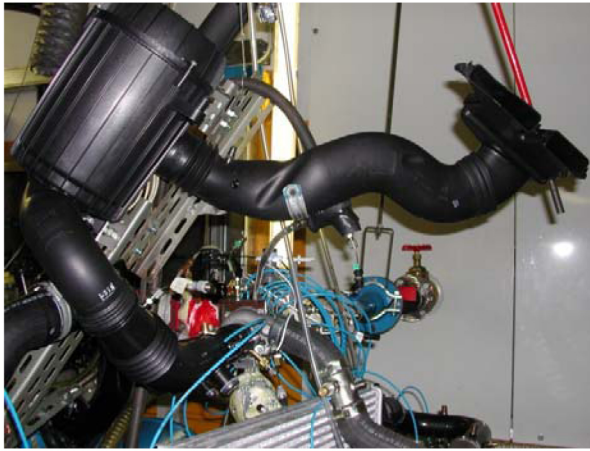
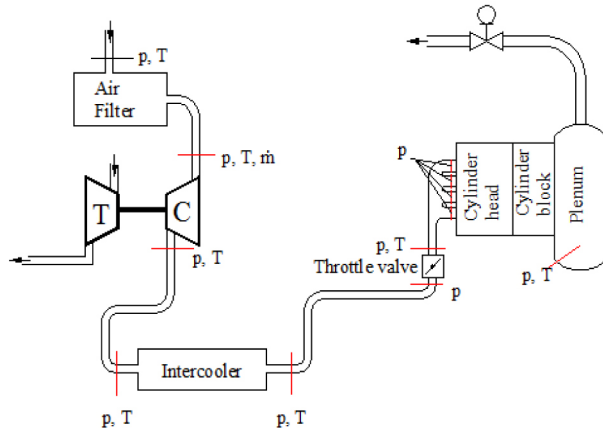


Figure 2. Intake engine circuit and related measuring sections



was evaluated by using an eddy current probe (with a precision of 0.009% of the full scale) mounted close to the compressor wheel. Two high speed synchronized data acquisition cards were used to record instantaneous signals. A Hall sensor mounted on the driving cam shaft of the cylinder head provided a trigger signal, corresponding to 40 crank angle degrees after TDC related to cylinder no. 1 overlap.

All measured signals reported in the paper are an ensemble average of several complete cycles.

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 [17].

GEOMETRICAL MODEL

The compressor is schematized as follows (Fig. 3):

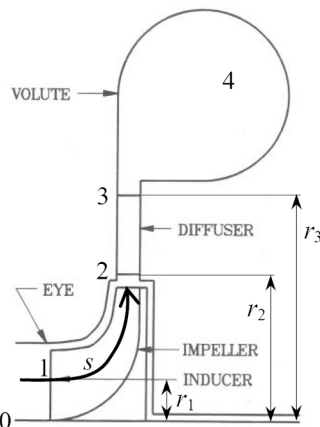


Figure 3. Main compressor stations

- a short inlet pipe (from section 0 to 1); the end section of the inlet pipe is characterized by a sudden area contraction, taking into account the impeller eye obstruction. This will cause some inlet losses, as explained in the following;

- 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 flow angles, diameters, number of blades, etc.), which can be measured easily on the impeller wheel. The comparison between the actual impeller geometry and the CAD reproduced reconstruction is reported in Figure 4. The equivalent 1D profile of blade-to-blade duct, to be employed in 1D flow equations, is then easily derived from the 3D CAD model;

- 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), schematized as a constant equivalent area duct, collecting the flow coming from the diffuser. An equivalent length is considered as half the circumference at the radius r_3 . The volute ends with an outlet cone of assigned dimensions.

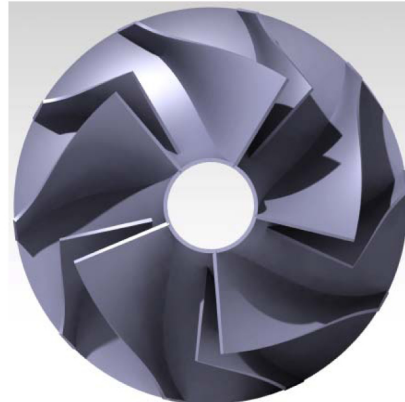


Figure 4. Actual impeller geometry (IHI RHF3 compressor) and related 3D reconstruction

UNSTEADY FLOW MODEL

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

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

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

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]:

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

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

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:

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

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

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

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 [19]. 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.

BOUNDARY CONDITIONS

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 unsteady procedure, 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.

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

Loss related Mach number is opportunely specified according to the junction and the considered loss mechanism. Exactly the same pressure loss correlations are introduced in the steady and unsteady procedures.

SLIP EFFECT

Even under ideal (inviscid) conditions, the flow leaving the impeller is said to 'slip'. Secondary flows (relative eddies) determine an increase of the exit flow angle, respect to the blade angle. As a consequence, the tangential component (c_{2u}) varies with respect to the ideal value c_{2u} , determining a decrease of the energy transfer, quantified by the slip factor, σ :

$$\sigma = 1 - \frac{c_{2u}' - c_{2u}}{u_2} \quad (5)$$

The evaluation of the slip coefficient was performed through the correlation proposed in [20]:

$$\sigma = 1 - c_{sl} \left[\frac{Fs_2\Phi_2 \cos \varphi_{2c}}{Z} - \frac{Fs_2\Phi_2}{4 \cos \varphi_{2c}} \left(\frac{d\varphi}{dm} \right)_2 + \frac{F\Phi_2 s_2 \sin \varphi_{2c}}{4\rho_2 b_2} \left(\frac{d\rho b}{dm} \right)_2 \right] \quad (6)$$

c_{sl} , s_2 , Φ_2 , b_2 , F , being, respectively, a tuning multiplier, span at impeller exit, flow coefficient, meridional blade to blade duct height and shape factor. F is defined as:

$$F = 1 - 2 \sin \left(\frac{\pi}{Z} \right) \sin \left(\frac{\pi}{Z} + \varphi_{2c} \right) \cos \varphi_{2c} - \frac{s_p}{s_2 \cos \varphi_{2c}} \quad (7)$$

FRICTION LOSSES

Friction losses are due to the presence of viscous forces in the boundary layer and are directly taken into account in the flow equations through the definition of a friction coefficient, depending on the local absolute or relative velocity based on Reynolds Number. The well known Poiseuille, Blasius or Nikuradse expressions are employed in both inlet, volute and outlet pipe. Concerning impeller, drag coefficient is evaluated separately for rotating surfaces and fixed ones and it is corrected according to [21]. As regards the diffuser, the approach proposed in [22] is followed:

$$f = x_{fat} 0.01 \left(\frac{1.8e5}{Re_L} \right)^{0.2}$$

$$Re_L = \frac{c L_r \rho}{\mu} \quad L_r = (r - r_2) \frac{c}{c_r} \quad (\log \text{ spiral length}) \quad (8)$$