

Development of a $K-k-\epsilon$ Phenomenological Model to Predict In-Cylinder Turbulence

Navin Fogla and Michael Bybee
 Gamma Technologies LLC

Mohsen Mirzaeian and Federico Millo
 Politecnico di Torino

Syed Wahiduzzaman
 Gamma Technologies LLC

ABSTRACT

The turbulent flow field inside the cylinder plays a major role in spark ignition (SI) engines. Multiple phenomena that occur during the high pressure part of the engine cycle, such as early flame kernel development, flame propagation and gas-to-wall heat transfer, are influenced by in-cylinder turbulence. Turbulence inside the cylinder is primarily generated via high shear flows that occur during the intake process, via high velocity injection sprays and by the destruction of macro-scale motions produced by tumbling and/or swirling structures close to top dead center (TDC). Understanding such complex flow phenomena typically requires detailed 3D-CFD simulations. Such calculations are computationally very expensive and are typically carried out for a limited number of operating conditions. On the other hand, quasi-dimensional simulations, which provide a limited description of the in-cylinder processes, are computationally inexpensive. They require only a small fraction of the computational resources needed for CFD calculations and can be carried out for a large number of cases. Such simulations typically use zero dimensional (0D) phenomenological sub-models to simulate the various in-cylinder phenomena such as heat transfer, combustion, flow variations etc.

The current study presents a newly developed sub-model, available as a part of the software GT-SUITE, which governs the evolution of the mean and turbulent flow inside the cylinder. Within a 0D context, the accurate knowledge of in-cylinder turbulence levels close to ignition is essential to reliably model turbulent flame propagation. However, engines running with stratified charge and/or early spark timings also require the accurate estimation of turbulence levels over the intake and compression strokes. The new model aims to accurately predict in-cylinder turbulence levels over the entire engine cycle. It utilizes a $K-k-\epsilon$ approach, where K and k are the mean and turbulent kinetic energy and ϵ is the turbulent dissipation rate. The study shows that the model, once calibrated for a particular engine using 3D-CFD results, has the capability to predict the temporal evolution of in-cylinder flow quantities and also responds well to changes in the operating conditions of the engine such as variations in speed, valve lift and timing.

CITATION: Fogla, N., Bybee, M., Mirzaeian, M., Millo, F. et al., "Development of a $K-k-\epsilon$ Phenomenological Model to Predict In-Cylinder Turbulence," *SAE Int. J. Engines* 10(2):2017, doi:10.4271/2017-01-0542.

INTRODUCTION

In a 0D setting, most in-cylinder flow models found in the literature follow either a $K-k$ energy cascade approach or a $k-\epsilon$ approach. In the energy cascade method, the mean kinetic energy and turbulent kinetic energy are modeled via two differential equations, one each for K and k . Algebraic equations are used to model the turbulence dissipation rate and integral length scale. Mean kinetic energy, generated primarily due to valve flow, leads to the production of turbulence via shear stresses which are modeled through a production source term. In the $k-\epsilon$ method turbulent kinetic energy and its dissipation are modeled via two differential equations. The effects of mean fluid motion are added via source terms (representing intake flow,

turbulence production via shear, etc.) in the turbulent kinetic energy equation. In this approach the integral length scale is obtained directly from k and ϵ without requiring any additional modeling. One of the earliest 0D flow models was proposed by Borgnakke, Arpacı and Tabaczynski [1]. They modeled in-cylinder turbulence via a 0D $k-\epsilon$ model which incorporated the unsteady, anisotropic and compressible nature of the flow field. In a subsequent study [2] the proposed model was improved by including the effects of swirl and squish motions on the flow field. This was achieved by solving additional equations governing swirl motion, modeled as a macro-vortex undergoing solid body rotation. Morel and Keribar [3] proposed a more resolved flow model, using the $k-\epsilon$ approach to modeling turbulence together with equations for swirl and squish

motions, which described the key fluid motions responsible for convective heat transfer. Mansouri, Heywood and Radhakrishnan [4], followed by Poulos and Heywood [5] were the proponents of the zero dimensional energy cascade approach or the $K-k$ model. Assanis and Heywood [6] included an additional source term in the equation for k in the $K-k$ model that accounted for turbulence production by compression. A recent study by Bordet, Caillol and Higelin [7] presented a modified $K-k$ model which accounted for the influences of swirl motion as well as high speed fuel injection into the cylinder.

It is generally recognized that tumble motion contributes to the enhancement of combustion in SI engines. Therefore, significant effort was directed towards developing zero dimensional flow models that account for its effects. One of the earliest models was presented in the works by Benjamin [8, 9] where the tumble motion was modeled as an elliptical macro-vortex with varying dimensions undergoing compression and decaying to produce turbulence. Tumble motion was governed by an additional equation for the angular momentum of the macro-vortex and its contributions to turbulence were modeled as source terms to the 0D $k-\epsilon$ equations. Similar developments were made by Dai, Newman and Davis [10]. An indirect validation of their model was attempted by comparing global burn rates obtained from simulation with those deduced from in-cylinder pressure measurements. It was shown that their model could predict early burn duration to within 1-2 deg of experimental data. A recent model proposed by Grasreiner et al. [11], specifically aimed at engines with fully variable valvetrains, accounted for the effects of tumble motion within a $k-\epsilon$ framework. Comparison of the 0D model predictions with 3D-CFD results at various operating points showed very good agreement between the two. The works of Bozza et al. [12, 13, 14] have made significant improvements to the $K-k$ family of models, especially in terms of accounting for the effects of tumble motion and variable valve lifts/timings. They compared the predictions of their models to results from 3D-CFD simulations and demonstrated good agreement between the two.

The current study presents a new 0D in-cylinder flow model developed by Gamma Technologies as a part of the commercial code GT-SUITE. The existing flow model in the code, developed by Morel and Keribar [3], utilized a $k-\epsilon$ approach to model in-cylinder turbulence. The predictions of this model were indirectly validated via comparison of combustion and heat transfer quantities predicted by simulation with those measured experimentally [15]. Good agreement of the predicted and measured quantities were observed. However, with the advancement of computing technology over the years, 3D-CFD analyses of in-cylinder flow structure evolution have become very commonplace in the engine design process. Therefore, it is essential to provide users with a 0D model that whose predictions of in-cylinder turbulence quantities compare well with 3D-CFD results. This is the main motivation behind the development of the model presented here.

The new flow model attempts to combine the existing $K-k$ and $k-\epsilon$ approaches into a $K-k-\epsilon$ model. Typically in the $k-\epsilon$ approach, the contribution of the mean kinetic energy is imposed via algebraic closure and in the $K-k$ models, the integral length scale of turbulence is approximated by either relating it to cylinder volume or functional fitting together with tuning constants, which must then be tuned using

3D-CFD [16, 17]. The proposed model aims to overcome these limitations by providing a mean kinetic energy equation which appropriately handles the decay of mean kinetic energy to turbulence eliminating the need for arbitrary source terms and an equation for dissipation rate which, together with turbulence kinetic energy, can be used to calculate the integral length scale (using equation (11)) without resorting to any additional modeling. The current model also accounts for the effect of tumble motion on the flow by treating it as a macro-vortex undergoing solid body rotation and tracks its evolution using an equation for angular momentum.

The paper is organized as follows. The next section presents the governing equations of the flow model. That is followed by the description of the engine geometry and the setup of 3D-CFD runs that were used to calibrate and validate the proposed model. The Results and Discussion section presents the comparison between the predictions of the 0D model and the results of 3D-CFD simulations. The final section presents some conclusions of the current study and directions for future investigations.

0D IN-CYLINDER FLOW MODEL

In this model the $K-k$ and the $k-\epsilon$ approaches have been combined via three differential equations. The equations are as follows:

$$\frac{d(mK)}{dt} = C_{in}(1 - \alpha_{in})E_{in} + K\dot{m}_{out} - P_k \quad (1)$$

$$\frac{d(mk)}{dt} = C_{in}\alpha_{in}E_{in} + k\dot{m}_{out} + P_k + C_{tumb}T - m\epsilon \quad (2)$$

$$\begin{aligned} \frac{d(m\epsilon)}{dt} = & C_{in}E_{in}\frac{\sqrt{k}}{L_g} + \epsilon\dot{m}_{out} + P_\epsilon + C_{tumb}T\frac{\sqrt{k}}{L_g} \\ & - 1.92\frac{m\epsilon^2}{k}. \end{aligned} \quad (3)$$

The equations shown above govern the evolution of the following flow quantities:

1. Mean kinetic energy $K = (1/2)U^2$, where U is the mean velocity inside the cylinder.
2. Turbulent kinetic energy $k = (3/2)u'^2$, where u' is the intensity of the turbulent field inside the cylinder, assumed to be homogeneous and isotropic.
3. Turbulent dissipation rate ϵ .

The term m multiplying the flow quantities is the in-cylinder mass.

The first term on the right hand side of each equation describes the production of each quantity due to flow entering the cylinder. The term E_{in} is calculated as

$$E_{in} = (1 - C_T) \frac{1}{2} \dot{m}_{in} v_{in}^2 \quad (4)$$

where $\dot{m}_{in}$ and v_{in} are the mass flow rate and isentropic velocity of the flow entering the cylinder respectively. C_T is the tumble coefficient associated with the valves, typically measured on a steady state flow bench as a function of the valve lift and provided as an input to the 0D model. The tumble coefficient provides a measure of the fraction of inflow energy that is imparted to the tumble macro-vortex. This coefficient takes on values from 0 to 1. Further details about this coefficient can be found in the [Appendix](#). The term $L_g = C_{len} \times \min(s, 0.5B)$, B being the bore of the cylinder and s the instantaneous piston stroke, is representative of a geometric length scale. The model parameter $C_{len} = 0.19 C_3$ modulates the value of this length scale.

The coefficient α_{in} is indicative of the fraction of inflow energy that directly enters the cylinder as turbulence and is not generated by the cascade process. It tries to model turbulence generated in the cylinder as soon as the valves open causing significant shearing flow. It was set to a fixed value of 0.1. The term $C_{in} = 0.18 C_1$ is a model parameter which is used to account for the actual flow velocities through the valves, which are not equal to the isentropic values. C_1 is a tuning constant which can be used to adjust the magnitude of the inflow source term. Exhaust backflow into the cylinder, which is included in the inflow source term, typically occurs for a shorter period of time compared to the intake flow. Also, this high velocity backflow is typically localized to a smaller region of the cylinder. This localized source of energy and its high rate of dissipation can be well resolved by 3D-CFD. Owing to its localization, it makes a small contribution to the values of energy that are averaged over the entire cylinder. However, the 0D model does not have the required spatial resolution to capture the localized effects and hence energy coming in the cylinder via exhaust backflow was scaled differently to incorporate the effect of localization. A scaling constant equal to $0.3 C_1$ was found to produce appropriate results.

The second term in the equations describes the energy flowing out of the valves, where $\dot{m}_{out}$ is the mass flow rate of the flow exiting the cylinder.

The terms P_k and P_ϵ respectively model the production of turbulent kinetic energy and dissipation rate from the large scale mean flows via the energy cascade process. These terms are calculated as follows

$$P_k = C_\beta \nu_T \frac{2mK}{L_g^2} - \frac{2}{3} mk \left(\frac{\dot{\rho}}{\rho} \right) - \frac{2}{3} m \nu_T \left(\frac{\dot{\rho}}{\rho} \right)^2 \quad (5)$$

$$P_\epsilon = \frac{\epsilon}{k} \left[5.76 C_\beta \nu_T \frac{mK}{L_g^2} - 2mk \left(\frac{\dot{\rho}}{\rho} \right) - \frac{2.64}{3} m \nu_T \left(\frac{\dot{\rho}}{\rho} \right)^2 \right] \quad (6)$$

where $\nu_T = C_\mu k^2 / \epsilon$ is the turbulent viscosity ($C_\mu = 0.09$ is a standard $k-\epsilon$ model constant), ρ and

$\dot{\rho}$ are the density and rate of change of density of the charge inside the cylinder respectively and $C_\beta = 0.38$ C_2 is a model parameter. The tuning constants C_2 and C_3 can be used to adjust the magnitudes of the production source terms. The forms of the production terms shown in [equations \(5\) and \(6\)](#) are motivated by their forms in the $k-\epsilon$ equations for compressible flows [18]. Their development is explained in the [Appendix](#).



Figure 1. Illustration of the evolution of the tumble macro-vortex.

The terms with the quantity T model the production of turbulence due to the decay of the tumble macro-vortex during the compression stroke and C_{tumb} is a tuning constant that controls the intensity of this process.

The last term on the right hand side of each equation acts as a sink term for that respective quantity. The mean kinetic energy is converted into turbulent kinetic energy via turbulent dissipation modeled by the term P_k , the turbulent kinetic energy is converted into heat via viscous dissipation $m\epsilon$ and the dissipation of the turbulent dissipation rate is given by the term $1.92m\epsilon^2/k$. The constant 1.92 is taken from the $k-\epsilon$ equations for compressible flows [18].

The rotational component of the mean flow, i.e. the tumble, is modeled as a single macro-vortex, as shown in [Figure 1](#), which undergoes stretching and compression during the intake and compression strokes and is finally destroyed to produce turbulence. Its evolution is described via an equation for its angular momentum $L = mr_t^2 \omega$ given as

$$\frac{dL}{dt} = \dot{L}_{in} + \dot{L}_{out} - Lf(s/B) \frac{\sqrt{k}}{r_t} \quad (7)$$

The first term on the right hand side represents the tumble production by the incoming charge and is defined as

$$\dot{L}_{in} = C_T \dot{m}_{in} |v_{in}| r_t, \quad (8)$$

where r_t is the radius of the tumble macro-vortex and is modeled as $r_t = (1/4)\sqrt{B^2 + s^2}$. The second term accounts for flow out of the cylinder and is given by

$$\dot{L}_{out} = \dot{m}_{out} r_t^2 \omega \quad (9)$$

where ω is the angular speed of the tumble macro-vortex. The third term accounts for the decay of the tumble vortex during compression and its conversion to turbulence close to TDC. It is modeled in a

fashion similar to Grasreiner et al. [11]. The decay is proportional to the turbulence levels in the cylinder and is modulated by the term $f(s/B)\sqrt{k}/r_t$. This term seeks to intensify the tumble decay as the piston approaches TDC. The tumble decay function $f(s/B)$ is plotted versus (s/B) in Figure 2. The contribution of tumble decay to turbulence is modeled via the term T given by

$$T = \frac{1}{2} L \omega f(s/B) \frac{\sqrt{k}}{r_t}. \quad (10)$$

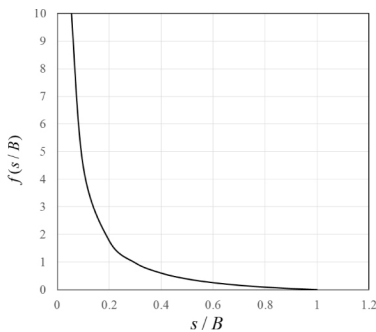


Figure 2. Tumble decay function.

Finally, since both turbulent kinetic energy and dissipation rate are being modeled in the proposed model, evolution of the integral length scale over time can be easily be obtained via the following equation

$$\ell = C_\mu^{3/4} \frac{k^{3/2}}{\epsilon}. \quad (11)$$

The proposed flow model can be calibrated to match 3D-CFD results by varying the following tuning constants:

- C_1 - controls the levels of mean and turbulent kinetic energies during inflow into the cylinder.
- C_2, C_3 - control the production of turbulence from the mean flow.
- C_{tumb} - controls the contribution of tumble decay to turbulence production.

Note that the tuning constants C_1, C_2, C_3 influence the model parameters C_{in}, C_β, L_g respectively. The effects of varying these tuning constants individually on the turbulent kinetic energy output from the model are shown in Figure 3.

3D-CFD SETUP

Two engines were used in the current study. Engine A is a typical European four-cylinder turbocharged SI engine. The main characteristics of the engine are listed in Table 1.

Engine B is also a SI engine, but with a different bore to stroke ratio and a very different piston shape compared to Engine A. All further details about Engine B are confidential data of the engine manufacturer and hence cannot be presented.

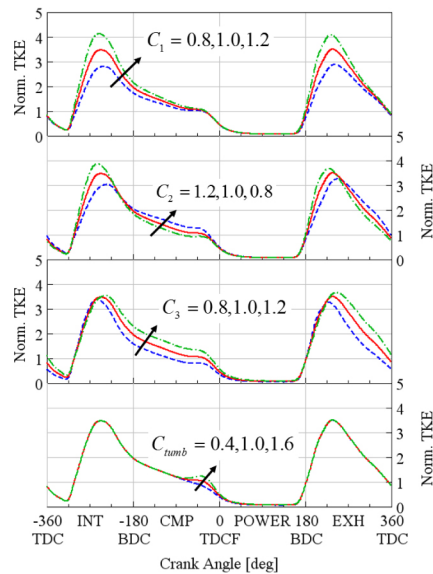


Figure 3. Effects of varying the tuning constants of the 0D model on turbulent kinetic energy output.

Table 1. Main characteristics of Engine A

Cylinders	4
Arrangement	Inline
Valves	4
Displacement	1368 cm ³
Bore	72 mm
Stroke	84 mm
Compression Ratio	9.8 : 1

CFD models of one cylinder of both engines mentioned above were setup in the commercial software CONVERGE [19]. The computational domain included the combustion chamber together with the intake and exhaust ports in the cylinder head. Figure 4 shows the computational domain for Engine A. The base grid was set to 4 mm, with additional mesh refinements that resulted in a local grid size of the order of 0.5 mm during the gas-exchange process. The total number of cells were around 500,000 at Bottom Dead Center (BDC) and 100,000 at Top Dead Center (TDC). The Navier-Stokes equations were solved using the Reynolds Averaging (RANS) approach together with the RNG k - ϵ model for turbulence.

The 3D-CFD simulations were carried out for the motoring case and included the intake-compression-expansion-exhaust strokes. The simulations were run for at least two cycles in order to minimize the effects of initial conditions. The time dependent pressure boundary conditions at the intake and exhaust ports were obtained from an equivalent 1D model of the setup in GT-SUITE as shown in Figure 5.

The in-cylinder quantities from the 3D simulation that were used in the present study, for calibrating and validating the 0D model, were the time history of the tumble number, defined as the ratio of the angular speed of the flow about the center of mass in the y-direction to the angular speed of the crankshaft and the mass averaged flow quantities namely mean kinetic energy (K), turbulent kinetic energy (k), turbulent dissipation rate (ϵ), integral lengthscale (ℓ).

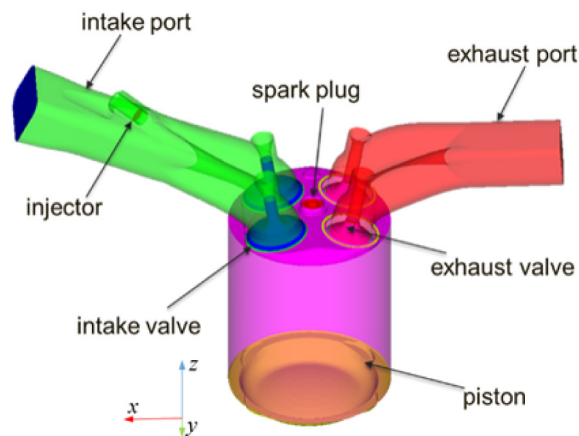


Figure 4. CFD 3D computational domain of Engine A.

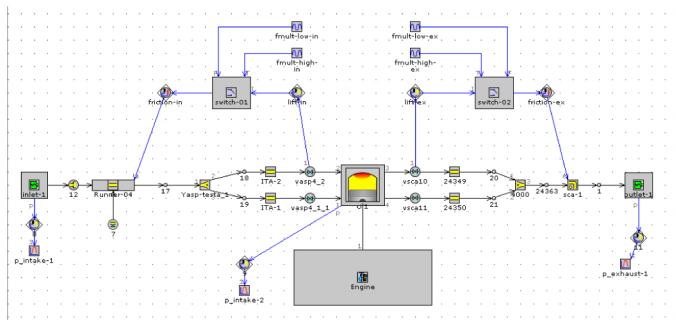


Figure 5. 1-D model of Engine A in GT-SUITE.

RESULTS AND DISCUSSION

This section presents the predictions of the proposed 0D flow model at various operating conditions and compares them to the results obtained via 3D-CFD simulations. The operating conditions considered for Engine A are the following:

- A sweep on 4 engine speeds - 2500, 3500, 4500 and 5500 rpm.
- An intake valve lift sweep on 3 valve lifts - 100%, 75% and 50% lift.
- A scaled intake valve lift profile with reduced lift and early closure.

For Engine B the following operating conditions were simulated:

- A sweep on 3 engine speeds - 1200, 4000 and 5200 rpm.

Results for Engine A

This section compares the simulation results of 0D and 3D-CFD for Engine A. The first step in using the flow model is to calibrate the 0D model via the tuning constants using 3D-CFD results. In the current study the model was calibrated manually through a simple trial and error procedure to obtain a good match of all quantities of interest. The values obtained for the constants are listed in [Table 2](#).

Table 2. Values of flow model tuning constants for Engine A

Model Parameter	Tuning constant	Value
C_{in}	C_1	1.0
C_{β}	C_2	1.0
C_{len}	C_3	0.85
	C_{tumb}	2.3

The terms in the column next to the tuning constants indicate the model parameters that they influence. The values of the tuning constants were kept fixed for all the operating conditions mentioned above without any operating condition specific tuning. In all the results presented below, the values of mean and turbulent kinetic energies have been normalized by $0.5u_p^2$ where u_p is the mean piston speed. The values of integral length scales have been normalized by the bore B and turbulent dissipation rates by the factor u_p^3/B .

Figures 7, 8, 9, 10, 11, 12 present the comparison between the predictions of the 0D model and results from 3D-CFD for 4 values of engine speed. Figure 7 compares the mass flow rates into the cylinder. The good agreement between the two approaches provides an additional validation that the models were setup correctly with the appropriate boundary conditions. Figure 8 presents a comparison of the tumble numbers. The definition of this quantity in 3D-CFD has been explained earlier. Within the 0D context it is defined as the ratio of the angular speed of the tumble macro-vortex to the angular speed of the engine. During the intake stroke, the tumble numbers obtained from 3D-CFD show large negative values indicative of tumble motion in the opposite direction. Examining the in-cylinder flow field immediately after IVO reveals that it is very complex and cannot be described using a single macro-vortex only. Hence, the tumble numbers obtained from 0D simulation are not expected to match the results from 3D-CFD during this period. The results from the compression stroke indicate that the model of a single macro-vortex spinning up and decaying captures the tumble development and decay very well at 2500 and 3500 rpm. However, at higher speeds of 4500 and 5500 rpm this model predicts higher levels of tumble motion than 3D-CFD. This discrepancy is also clearly visible in the plots of turbulent kinetic energy for 5500 rpm in Figure 10, close to the end of the compression stroke. The reasons for this discrepancy will be discussed further below.

Figures 9, 10, 11, 12 compare the mean kinetic energy and the turbulence quantities, namely turbulent kinetic energy, integral length scale and turbulent dissipation rates respectively. The evolution of the mean and turbulent kinetic energies and the integral length scales are well predicted by the 0D model over the intake and compression strokes.