Turbulence Models Comparison for the Numerical Study of the Air Flow, an Internal Combustion Engine

Ioannis Pantokratoras, Charalampos Arapatsakos

Abstract—Numerical flow studies of air flowing into the cylinder of an internal combustion, four stroke, two valve engine have been undertaken. The studies relating to the phase of intake stroke, when the piston movement of the engine creates pressure lower than the atmospheric in the cylinder. Results were obtained with the following turbulent models provided by commercial CFD code ANSYS v15.0: a) Standard k-ε, b) Standard k-ω, c) RNG k-ε, d) Sst k-ω. Three dimensional, steady state flow conditions considered, for valve opening and piston position at 90° degrees of crankshaft revolution after top dead center.

Key-Words—Turbulence, Four Stroke Engine, Intake Stroke, Computational Fluid Dynamics

I. INTRODUCTION

It is very important for the efficiency, fuel economy and ideal working characteristics of an internal combustion engine, to be able to predict and model the air/fuel mixture and exhaust gases flow dynamics. Throughout the years of engine development, various experimental techniques have been developed for this reason. But the cost and time needed and also the limited potential that experiments provided, made these techniques obsolete for engine design and development[6].The evolution and increase of the computational capabilities resulted to the use of Computational Fluid Dynamics (CFD) codes. CFD codes define a solution to a set of equations of fluid flow by advancing the solution through space and time and provide a numerical description of the flow field.CFD nowadays has applications in almost all fields of engineering. Very complicated phenomena can be accurately simulated, while cost and time are saved without the need of prototyping. It is not difficult to understand why CFD became so popular for modern engine design. Various intake manifolds and exhaust geometries can be accurately simulated fast and with comparatively low cost[4,5,7,9]. CFD workflow consists of: a) Geometry design, b) Grid generation, c) Setup the model, d) Solution, e) Results evaluation.CFD codes use the finite volume method (FVM) to discretize the geometry of the flow. This discretization is created upon an integral form of the partial differential equations (conservation of mass, momentum and energy) to be solved. The geometry is divided into cells as small control volumes and the governing equations are solved for each control volume.

II. TURBULENCE

During intake of the air into the engine cylinder and also during the exhaust of the gases to the atmosphere, highly turbulent flows are developed. Also, when the mixture enters the engine cylinder, jet, swirl and tumble motions are generated[8][10]. In turbulent flows the species which constitute the fluid, follow random streamlines, creating vortices and the velocity fields are fluctuating. As a result the transported quantities such as momentum and energy fluctuate as well. The vortices created enhance the momentum exchange in all 3 directions. These fluctuations can be of small scale and high frequency and they are highly computational expensive to be precisely simulated. A very effective way to reduce computational cost and maintain reliable and usable results is Reynolds Averaged Navier-Stokes (RANS) approach.

In this method additional terms are incorporated in the governing equations that need to be modeled in order to reach the solution. The RANS equations govern the transport of the averaged flow quantities, with the whole range of the scales of turbulence being modeled. This approach reduces the required computational effort and resources, and is effective for practical engineering applications, where there is no need of highly detailed modeling of the flow. The modified equations contain additional unknown variables, and turbulence models are needed to determine these variables. The governing equations that ANSYS code uses to compute the flow quantities are the continuity equation for the conservation of mass and the governing equations of momentum. The general form of the mass conservation equation (continuity equation) is given by:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = S_m$$

The source $S_m$ is the mass added to the continuous phase from the dispersed second phase and user defined sources.

The conservation of momentum equation:
\[ \frac{\partial}{\partial t} \left( \rho \bar{v} \right) + \nabla \cdot (\rho \bar{v} \bar{v}) = -\nabla p + \nabla \cdot (\mathbf{\tau}) + \rho \bar{g} + \bar{F} \]

Where \( p \) is the static pressure, \( \mathbf{\tau} \) is the stress tensor, \( \rho \bar{g} \) is the gravitational body forces and \( \bar{F} \) the external body forces.

As said, in turbulent flows additional terms are added to these equations. These terms are calculated by the different turbulence model techniques.

### III. TURBULENCE MODELS

Many turbulence models have been developed in order to solve the RANS governing equations. At the beginning were one equation models (Spalart-Allmaras) and subsequently more effective two-equation models. No one of them, unfortunately, is widely accepted as being superior to others for all cases. The choice of turbulence model depends on considerations such as the geometry of the field, physics in the flow, the level of accuracy required, the available computational resources and the amount of time available for the simulation. Two-equation models are considered nowadays the best way to calculate the unknown variables emerging from the RANS approach. In these models the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. Two of the most important of them are the \( k-\epsilon \) model and the \( k-\omega \) model. Later, optimized versions of these models, like RNG \( k-\epsilon \) and SST \( k-\omega \), were developed[2,3].

**Standard \( k-\epsilon \) model:**
The standard \( k-\epsilon \) model was proposed by Launder and Spalding. It became very popular because it provides robust results, computational economy, and acceptable accuracy for a wide range of turbulent flows.

The standard \( k-\epsilon \) model is characterized as a semi-empirical model based on model transport equations for the turbulence kinetic energy (\( k \)) and its dissipation rate (\( \epsilon \)). In the derivation of the \( k-\epsilon \) model, the acceptance is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. It is considered very accurate for free stream flows.

**Transport Equations:**
The turbulence kinetic energy, \( k \), and the dissipation rate, \( \epsilon \), arising from the following transport equations[1]:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k \bar{u}_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\epsilon} \right) \frac{\partial k}{\partial x_j} \right] + \frac{\rho_k}{\epsilon} \left( G_k + G_\delta - \rho \bar{c}_M + \bar{S}_k \right)
\]

\[
\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho \epsilon \bar{u}_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{\mu \epsilon} \frac{\epsilon}{k} \left( G_k + G_\delta \right) - C_{\mu} \rho \bar{c}_M^2 + \bar{S}_\epsilon
\]

where:

\[
\alpha_k, \alpha_\epsilon
\]

are the inverse effective Prandtl numbers for \( k \) and \( \epsilon \).

\[
\begin{align*}
\alpha &= \frac{1}{C_{\mu \epsilon} k} \\
\alpha &= \frac{1}{C_{\mu} k} \\
\alpha &= \frac{1}{C_{\mu \epsilon} k}
\end{align*}
\]

\[
\begin{align*}
\alpha &= 1.3929 \\
\alpha &= 0.5321 \\
\alpha &= 0.3679
\end{align*}
\]

\[
\frac{\mu_t}{\mu_{\text{eff}}} = \frac{\mu_{\text{mol}}}{\mu_{\text{eff}}}
\]

where:

\[
\begin{align*}
\mu_{\text{mol}} &= \mu_{\text{eff}} \\
\mu_{\text{mol}} &= \mu_{\text{eff}}
\end{align*}
\]

RNG \( k-\epsilon \) model:
The RNG \( k-\epsilon \) model is identical to the Standard \( k-\epsilon \) model but includes specific refinements. It was formed using renormalization group methods. That resulted in constants different than those in the Standard \( k-\epsilon \) model and also additional terms and functions[1].

**Transport Equations:**

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k \bar{u}_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\epsilon} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_\delta - \rho \bar{c}_M + \bar{S}_k
\]

\[
\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho \epsilon \bar{u}_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{\mu \epsilon} \frac{\epsilon}{k} \left( G_k + G_\delta \right) - C_{\mu} \rho \bar{c}_M^2 + \bar{S}_\epsilon
\]

where:

\[
C_{\mu \epsilon} = 0.0845, \quad C_{k} = 1.42, \quad C_{\epsilon} = 1.68
\]

\[
Pr_\epsilon = 1/\alpha
\]

\[
\alpha_k, \alpha_\epsilon
\]

are the inverse effective Prandtl numbers for \( k \) and \( \epsilon \).
\( \alpha_0 = 1.0 \)

At first it has an additional term in the \( \varepsilon \) equation that makes the model more efficient for strained flows. Moreover it is more accurate predicting swirling flows, as it includes the effect of swirl and it gives the user a detailed form of turbulent Prandtl numbers. Finally, the RNG \( k-\varepsilon \) model is more reliable at low-Reynolds number situations, because it offers an enhanced differential formula for effective viscosity. These characteristics result in more accurate and reliable results for a wider class of flows.

**Standard \( k-\omega \) model:**

The standard \( k-\omega \) model in ANSYS is based on the Wilcox model and includes modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. It can estimate free shear flow spreading rates, mixing layers, different type of jets and is thus applicable to wall-bounded flows and free shear flows. The standard \( k-\omega \) model is an empirical model based on model transport equations for the turbulence kinetic energy \( (k) \) and the specific dissipation rate \( (\omega) \).

**Transport Equations:**

The turbulence kinetic energy, \( k \), and the specific dissipation rate, \( \omega \), are arising from the following transport equations[1]:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k
\]

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega
\]

where:

\[
G_k = -\rho \omega u_j \frac{\partial u_j}{\partial x_i} \Gamma_k, \quad G_\omega = \alpha^* \frac{\omega_k}{k} G_k
\]

\[
\alpha = \alpha^{\infty} \left( \frac{\alpha_0 + \text{Re}_k / \text{Re}_\omega}{1 + \text{Re}_k / \text{Re}_\omega} \right),
\]

\[
\alpha^* = \alpha^{\infty} \left( \frac{\alpha_0 + \text{Re}_k / \text{Re}_k}{1 + \text{Re}_k / \text{Re}_\omega} \right)
\]

\[
\text{Re}_k = \frac{\rho k}{\mu \omega}, \quad \Gamma_k = \mu + \frac{\mu_k}{\sigma_k}, \quad \Gamma_\omega = \mu + \frac{\mu_\omega}{\sigma_\omega}
\]

\[
\mu_k = \alpha^* \frac{\rho k}{\omega}, \quad Y_k = \rho \beta^* f_\beta \omega, \quad k \omega
\]

\[
\beta^* = \beta_0^* \left[ 1 + \zeta^* (M_t)^{4/15} \right] \quad \beta_0^* = 4\left( \frac{4/15 + (\text{Re}_k / \text{Re}_\omega)^{4/15}}{1 + (\text{Re}_k / \text{Re}_\omega)^{4/15}} \right)
\]

\[
\beta = \frac{1}{\beta_0^*} \left[ 1 - \beta_0^* \zeta^* F(M_t) \right]
\]

\[
F(M_t) = \begin{cases} 
0 & \text{if } M_t \leq M_{t0} \\
\text{Re}_t^2 - M_{t0}^2 & \text{if } M_t > M_{t0}
\end{cases}
\]

\[
M_t^2 = \frac{2k}{\alpha^2}, \quad M_{t0} = 0.25
\]

\[
\alpha^{\infty} = 1, \quad \alpha_0 = 0.52, \quad \alpha_0 = 0.1, \quad \beta_0^* = 0.09, \quad \beta_0 = 0.072, \quad \beta_0 = 8
\]

\[
R_k = 6, \quad R_\omega = 2.95, \quad \zeta^* = 1.5, \quad M_0 = 0.25, \quad \sigma_k = 2.0, \quad \sigma_\omega = 2.0
\]

are constants.

\( \sigma_k \) and \( \sigma_\omega \)

are the turbulent Prandtl numbers

\( S_k, S_\omega \)

are user defined.
The k-ω model is considered to provide high accuracy in the near wall region and it is recommended for wall bounded flows.

**Shear stress transport (SST) k-ω model:**
The shear-stress transport k-ω model was developed by Menter in order to combine the accuracy of the Standard k-ω model in the near wall region with the effectiveness of the k-ε model in the free stream region of the flow. It is based on the Standard k-ω model including refinements[1]:

**Transport equations:**
\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \Gamma_k \frac{\partial k}{\partial x_j} \right] + \tilde{G}_k - Y_k + S_k
\]
\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + D_\omega + S_\omega
\]
where:
\[
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k}
\]
\[
\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega}
\]
\[
\mu_t = \frac{\rho k}{\omega} \frac{1}{\max \left( \frac{1}{\sigma^k}, \frac{S_k}{\delta^k} \right)}
\]
\[
\tilde{G}_k = \min \left( G_k, 10 \rho \beta^k \kappa \omega \right)
\]
\[
G_\omega = \frac{\alpha}{\nu_t} \tilde{G}_k
\]
\[
\alpha_\infty = F_1 \alpha_{\infty,1} + \left( 1 - F_1 \right) \alpha_{\infty,2}
\]
\[
\alpha_{\infty,1} = \frac{\beta_{\kappa,1}}{\beta_{\kappa,\infty}} - \frac{\kappa^2}{\sigma_{w,1} \sqrt{\beta_{\kappa,\infty}}}
\]
\[
\alpha_{\infty,2} = \frac{\beta_{\kappa,2}}{\beta_{\kappa,\infty}} - \frac{\kappa^2}{\sigma_{w,2} \sqrt{\beta_{\kappa,\infty}}}
\]
\[
Y_k = \rho \beta^k \kappa \omega
\]
\[
Y_\omega = \rho \beta \omega^2
\]
\[
\beta_1 = F_1 \beta_{1,1} + \left( 1 - F_1 \right) \beta_{1,2}
\]
\[
F_1 = \tanh \left( \Phi_1 \right)
\]
\[
\Phi_1 = \min \left[ \max \left( \frac{\sqrt{k}}{0.09 \omega y}, \frac{50 \mu}{\rho y^2 \omega} \right), \frac{4 \rho \kappa}{\sigma_{w,2} D_{\omega} y^2} \right]
\]
\[
D_{\omega}^+ = \max \left[ 2 \rho \frac{1}{\sigma_{w,2} \omega} \frac{\partial \omega}{\partial x_j} \frac{10^{-10}}{\partial x_j} \right]
\]
\[
\sigma_{k,1} = 1.176, \quad \sigma_{\omega,1} = 2.0, \quad \sigma_{k,2} = 1.0, \quad \sigma_{\omega,2} = 1.168
\]
\[
\alpha_{1,1} = 0.31, \quad \beta_{1,1} = 0.075, \quad \beta_{1,2} = 0.0828
\]
are constants.

The standard k-ω model is used in the near wall region, while the Standard k-ε is used in the free stream. This is achieved by multiplying the model by a blending factor which takes the values “0” away from the boundary and “1” close to it. Also the SST model uses a cross diffusion derivative term in the \( \omega \) equation and the constants are different. These features result in a more accurate and reliable model for more flow cases such as airfoils, shock waves etc, than the standard k-ω model.

**IV. SOLUTION METHODS-RESULTS**

The computational grid consists of an unstructured mesh of 1,034,000 tetrahedron and hexahedron elements. Five layer inflation was created at all boundaries. This particular grid was used because it consists of high quality elements and after evaluation it was proved to provide grid independent results.

![Figure 1. Section view of the computational grid](image)

The geometry boundary conditions consist of Pressure Inlet, Walls and Pressure Outlet. Atmospheric pressure considered at the inlet and 90,000 Pascal in the cylinder. Standard wall functions were used in the k-ε models and compressibility effect was activated in all cases, because at certain areas (such as the valve opening region) the flow becomes compressible. COUPLED algorithm was used for pressure-velocity coupling. The COUPLED algorithm is a pressure-based solver that solves the
momentum and pressure-based continuity equations at the same time. Coupling is achieved through an implicit discretization of pressure gradient terms in the momentum equations, and an implicit discretization of the face mass flux. It is considered more effective comparing to SIMPLE or SIMPLEC solvers. Second-order upwind spatial discretization schemes were used for momentum and density, providing high accuracy.

The four different turbulence models were used in order to compute the flow quantities. Velocity streamlines, surface velocity distribution, pressure values and also velocity profile results for critical areas, such as the valve body, after valve area and runner, are demonstrated.
Figure 6. Top view of Velocity contours around the valve body for a) $k-\varepsilon$, b) $k-\omega$, c) RNG $k-\varepsilon$, d) Sst $k-\omega$

Figure 7. Top view of Pressure contours around the valve body for a) $k-\varepsilon$, b) $k-\omega$, c) RNG $k-\varepsilon$, d) Sst $k-\omega$ (the values are negative from the atmospheric pressure).

Figure 8. Velocity profiles at the valve body, for different turbulence models

Figure 9. Velocity profiles after the valve, for different turbulence models

Figure 10. Velocities at the centerline of the inlet
manifold, through the valve opening and until the surface of the piston

Figure 11. Pressure drop from the inlet of the manifold, through the valve opening and until the surface of the piston

V. CONCLUSION

The turbulent flow through the intake runner and inside the cylinder of a four stroke engine, two valve engine was investigated numerically using the commercial CFD code FLUENT v15.0. The RANS method was used and the performance of four different turbulence models is compared.

The turbulence models \( k-\varepsilon \), \( k-\omega \), RNG \( k-\varepsilon \) and SST \( k-\omega \) provide results that have some rate of divergence between them. It is obvious that the \( k-\varepsilon \) model differ more than the other three. It is important to notice also that the \( k-\varepsilon \) model was the less computational expensive between them. The theory and factors that every model use, can guide the user to choose between them for the purpose being sought. Also the computational cost along with the accuracy searched for, could be factors of choice. The detailed evaluation of these results would be possible if experimental values had been taken and compared with these results.

REFERENCES