EFFECT OF ROTATING CYLINDER ON THE DRAG FORCE OF A ROAD TRUCK VEHICLE

Najdat N. Abdulla
Prof./Dr./ University of Baghdad

Alaalddin A. Ismail
Mechanical Engineer

ABSTRACT
The effect on aerodynamic drag of a truck by controlling the boundary layer separation using a rotating cylinder on leading edge of the truck-trailer is investigated numerically. The flow was assumed to be steady, incompressible, turbulent, and two-dimensional passing over the top surface of the truck. The boundary condition for all the boundaries of the truck was set as well as the cylinder was treated as a moving wall with a specific rotational velocity. The developed computational algorithm is tested for the flow over a flat plate (8m) long with various free stream inlet relative velocity ($U_\infty$) which are considered the same as truck velocity and has the values (40, 60, 90, and 120) km/h. The effect of cylinder diameter (10, 20, 30, and 40 cm), rotational speed (1000-5000 r.p.m) and free stream velocity on the aerodynamic drag and pressure distribution of the flow field were investigated. The governing equations which used are the continuity, momentum, and the (K-$\varepsilon$) turbulence model. These equations are approximated by using a finite volume method, with staggered grid and modified SIMPLE algorithm. A computer program in FORTRAN 90 is built to perform the numerical solution. The numerical results show that, the optimum cases for inlet free stream condition for all the boundaries of the truck was set as well as the cylinder was treated as a moving wall with a specific rotational velocity (1000-5000 r.p.m) and free stream velocity on the aerodynamic drag and pressure (

Comparison of the results with the available previous published experimental and fluent program results was investigated.
INTRODUCTION

Boundary layer separation takes place under adverse pressure gradient conditions when viscous effects are no longer confined to a thin layer but affect the overall flow pattern drastically, Singh (2005). It can be seen that an adverse pressure gradient, \( (\partial p / \partial x > 0) \), is a necessary condition for separation. It does not mean that if \( (\partial p / \partial x > 0) \), a separation will occur, but rather it is reasoned that separation cannot occurs unless \( (\partial p / \partial x > 0) \). The point on the boundary where \( [\partial u / \partial y]_{x=0}=0 \) is called the point of separation. Just downstream from the point of separation, the flow direction in the separated region is actually opposite to the main flow direction. The low energy fluid in the separated region is forced back upstream by the increased pressure downstream Robert (1973, 1978). The most common application to date has been the flow around an aerofoil wing of an aircraft. Flow separation takes place on the upper surface of the aerofoil at a large angle of attack resulting in a drastic fall in lift. Several methods, such as mass injection either by blowing or by suction, coating of the wall, or transition to turbulent flow etc. have been practiced with varying degrees of success. The concept of delaying boundary layer separation has also been used in road vehicles to reduce aerodynamic drag. However, the use of momentum injection using a moving wall for boundary layer control is still in its nascent stages. Momentum injection using rotating cylinders has been applied to airfoils in order to improve their lift characteristics like applied the concept of moving surface boundary layer control to a joukowsky airfoil using rotating cylinders at the leading and trailing edges of the airfoil. However, the use of rotating cylinders to reduce aerodynamic drag on tracks is a new concept Singh (2005). At 70 mph, a common highway speed today, overcoming aerodynamic drag represents about 65% of total energy expenditure for a typical heavy truck vehicle. Reduced fuel consumption for heavy vehicles can be achieved by altering truck shapes to decrease the aerodynamic resistance or drag. It is conceivable that present day truck drag coefficients might be reduced by as much as 50%. This reduction in drag would represent approximately a 25% reduction in fuel use at highway speeds Rose (2004).

Singh et al (2005), investigated aerodynamic drag effect using a model of a truck to control the boundary layer separation by the momentum injection method using a rotating cylinder. They used an experiments coupled with computational fluid dynamics (CFD) analysis to validate the theory of momentum injection. Modeling of the truck has been done on the software (GAMBIT); the best suitable turbulence model was selected by comparing the results with the experimental results. Steady state Navier Stokes equations are taken as the governing equations for the fluid motion, and appropriate boundary conditions are used. The rotational speed and radius of the cylinder are varied to establish the effect of momentum injection on aerodynamic drag. The coefficient of drag reduces by approximately 35 percent from an initial value of 0.51- 0.32 for a cylinder of radius 1cm with rotational speed of 4000 r/min. Robson et al (2005), used a generic rotating ventilator as a form of low energy drag reducer for use on bluff bodies such as transport vehicles. An experimental investigation of a rectangular bluff body was conducted to demonstrate the viability of this novel concept. The ventilator is expected to produce an effect on the flow characteristics similar to a rotating cylinder but with the added advantage that it can be operated using natural wind without the need for any power input by the engine or any auxiliary unit. The bluff body was modeled as a rectangular box; the ventilator was mounted on the front of the model with the axis of rotation parallel to the front surface and bottom edge. Open circuit open jet wind tunnel was used to conduct the experiments. The test Reynolds numbers were selected to lie between \( 3.5 – 6.5 \times 10^5 \),measurements were taken at yaw angles of 0, 5, 10, 15 degrees. The results indicate that the device decreases drag on the body significantly by approximately 50% at Reynolds numbers of \( 3.5 \times 10^5 \) and \( 6.5 \times 10^5 \). The
incorporation of this device also makes the drag of the body less Reynolds number dependent. Consequently the proposed configuration is more efficient at higher speed in comparison with the conventional configuration which did not have the device attached to it.

Salam Al-Taee (2005), studied the separation control on the NACA 0012 & NACA 0018 airfoils by using a rotating cylinder based on the computation of Reynolds-average Navier-Stokes equations. A numerical model based on collocated finite volume method is developed to solve the governing equations on a body-fitted grid. To ensure the accuracy of the code first, second and third order differencing schemes, with and without flux-limiters, have been implemented and tested. The systematical investigation of the rotating cylinder is conducted on the same NACA 0012 airfoil in the range of attack angles from 10° up and beyond the stall angle at Re=10^6. The influence of some parameters associated with using rotating cylinder, such as its radius, location, and the speed ratio (U_c/U_∞) strength on the performance of the airfoils have been studied. The result shows that the rotating cylinder is affected in controlling the separation and the lift coefficient of circulation control airfoil is increased with the angle of attack.

Robert Clark (2006), presented a combination of wind tunnel test, computational fluid dynamic modeling, and real-world testing to determine the effects of the external additions devices and systems on aerodynamic drag and fuel economy. These devices include (cab enclosure, vortex traps, side strakes, and side skirts). The results showed a reduction on the aerodynamic drag by 23% for tractor-semitrailer systems.

Johan Hoffman (2007), considered the problem of computational simulation of flow past a wheel of a vehicle. As a model he used a rotating cylinder in contact with a flat surface moving with the same velocity as the uniform free stream. In particular he was interested in the importance of including the effect of rotation in the model to accurate compute the drag force. He compared two different models: (i) a cylinder rolling around ground, and (ii) a stationary cylinder in free stream, corresponding to a simple wind tunnel testing. He compared the drag for (i) and (ii) at Reynolds number of 10000. He used CFD method to investigate the importance of including (i)-(ii) in a model. As computational model he considered the flow about cylinder rolling along a flat surface. CFD concerns the computational solution of the Navier-Stokes equations, which are considered to model both laminar and turbulent flow. In this paper study the flow of turbulent boundary layer over a flat plate, study the effect of a rotating cylinder (in order to inject momentum) on the aerodynamic drag, utilizes CFD method and standard \(\kappa-\varepsilon\) model to analyze the effect of momentum injection on the aerodynamic drag, use different values of rotation speeds to demonstrate the effect of rotation on the drag coefficient and use different values of diameter size to demonstrate the effect of diameter size on the drag coefficient.

**MATHEMATICAL MODEL**

The starting point of any numerical method is the mathematical model, i.e. the set of partial differential or integral-differential equations and boundary conditions. These equations are based on the conservation of mass and momentum. A finite volume method (FVM) is presented for the solution of two-dimensional Navier-Stokes equations in a Cartesian coordinate. The \((k-\varepsilon)\) model is utilized to describe the turbulent flow on the upper surface of the truck. The flow was assumed to be steady, two-dimensional and incompressible over a flat plate with a rotating cylinder in the leading edge of the plate.

**Geometry and Coordinate System:**

The usual system of Cartesian coordinates is adopted which represents the x-axis being along the wall and the y-axis being at right angle to it, as shown in Fig. (a).

![Figure.( a).](image)

**The Governing Equations:**

The equations of motion for steady state, two-dimensional and incompressible flows over flat plate can be written as follows .Robert(1973,1978):-

(i) **Continuity Equation**

\[
\frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) = 0
\]

(ii) **Momentum Equations**
X– Direction (u – momentum)
\[
\frac{\partial}{\partial x}(\rho u u) + \frac{\partial}{\partial y}(\rho u v) = -\frac{\partial p}{\partial x} + \mu \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right]
\] (2)

Y-Direction (v- momentum)
\[
\frac{\partial}{\partial x}(\rho u v) + \frac{\partial}{\partial y}(\rho v v) = -\frac{\partial p}{\partial y} + \mu \left[ \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right]
\] (3)

The effects of fluctuation can be introduced by replacing the flow variable \(u, v, \rho\) and \(p\) by the sum of mean and fluctuating components. Thus, Versteeg (1995):
\[
\begin{align*}
u &= \bar{U} + u' \\
v &= \bar{V} + v' \\
P &= \bar{P} + p'
\end{align*}
\] (4)

With some simplification the momentum equations become, Awbi(1991):
\[
\begin{align*}
\rho U \frac{\partial U}{\partial x} + \rho V \frac{\partial U}{\partial y} &= \frac{\partial p}{\partial x} + \mu \left[ \frac{\partial U}{\partial x} \right] + \frac{\partial U}{\partial y} \\
\rho V \frac{\partial V}{\partial x} + \rho U \frac{\partial V}{\partial y} &= \frac{\partial p}{\partial y} + \mu \left[ \frac{\partial V}{\partial x} \right] + \frac{\partial V}{\partial y} \\
\end{align*}
\] (5)

Turbulence Model:

Standard K-\(\epsilon\) Model
The general modified form of (K-\(\epsilon\)) model can be written as follows, Versteeg (1995):
\[
\begin{align*}
\rho U \frac{\partial K}{\partial x} + \rho V \frac{\partial K}{\partial y} &= \frac{\partial}{\partial x} \left[ \mu \frac{\partial K}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \mu \frac{\partial K}{\partial y} \right] + \\
\mu_l \left[ \left( \frac{\partial U}{\partial x} \right)^2 + \left( \frac{\partial V}{\partial y} \right)^2 + \left( \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right)^2 \right] - \rho \epsilon
\end{align*}
\] (6)

\[
\begin{align*}
\rho U \frac{\partial \epsilon}{\partial x} + \rho V \frac{\partial \epsilon}{\partial y} &= \frac{\partial}{\partial x} \left[ \mu \frac{\partial \epsilon}{\partial x} / \sigma_k \right] + \frac{\partial}{\partial y} \left[ \mu \frac{\partial \epsilon}{\partial y} / \sigma_k \right] + \\
\mu_l \left[ \left( \frac{\partial U}{\partial x} \right)^2 + 2 \left( \frac{\partial V}{\partial y} \right)^2 + \left( \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right)^2 \right] - \rho \epsilon
\end{align*}
\] (7)

Further Calculation:
The most important parameters for boundary layer flow, is drag coefficient \(C_{Df}\):
\[
\text{Drag} = 1/2 \rho V^2 A C_{Df}
\]
Where \(C_{Df}\) coefficient of skin friction drag.
\(A\): is the surface area.
\(\rho\): is the density of the air.
\(V\): is the speed of the vehicle relative to the air.

COMPUTATIONAL TECHNIQUE
The solution of the continuity and momentum equations was performed by using a finite volume method (FVM) to obtain the discretization form for these equations. These discretization equations are solved by using SIMPLE algorithm with hybrid scheme. A computer program based on this algorithm and uses FORTRAN 90 language was built to meet the requirements of the problem. The SIMPLE algorithm was based on the staggered grid in which the velocities staggered midway between the grid intersections is used to obtain the numerical results.

\[
\begin{align*}
\frac{\partial}{\partial x} (\rho u \Phi) + \frac{\partial}{\partial y} (\rho v \Phi) &= 0 \\
\frac{\partial}{\partial x} \left[ \Gamma \frac{\partial \Phi}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \Gamma \frac{\partial \Phi}{\partial y} \right] + S\Phi
\end{align*}
\] (8)

Where \(\Phi, \Gamma, S\) define in table (1) as flow

Table (1) Source terms in the transport equation.

<table>
<thead>
<tr>
<th>Equation</th>
<th>(\phi)</th>
<th>(\Gamma)</th>
<th>(S)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
\[
\begin{align*}
\text{U-momentum} & : \quad U \quad \mu_e \quad \begin{bmatrix}
\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left[ \mu_e \frac{\partial U}{\partial x} \right] + \\
\frac{\partial}{\partial y} \left[ \mu_e \frac{\partial V}{\partial x} \right] + \\
\frac{\partial}{\partial y} \left[ \mu_e \frac{\partial V}{\partial y} \right]
\end{bmatrix} \\
\text{V-momentum} & : \quad V \quad \mu_e \quad \begin{bmatrix}
\frac{\partial P}{\partial y} + \frac{\partial}{\partial x} \left[ \mu_e \frac{\partial U}{\partial y} \right] + \\
\frac{\partial}{\partial y} \left[ \mu_e \frac{\partial V}{\partial y} \right] + \\
\frac{\partial}{\partial y} \left[ \mu_e \frac{\partial V}{\partial y} \right]
\end{bmatrix} \\
\text{Kinetic energy} & : \quad K \quad \Gamma_K \quad G - \rho e \\
\text{Dissipation rate} & : \quad \varepsilon \quad \Gamma_\varepsilon \quad C_v \rho \frac{\varepsilon}{K} G - C_s \rho \frac{\varepsilon}{K}
\end{align*}
\]

Where:
\[
G = \mu_e \left( \begin{array}{c}
2 \left( \frac{\partial U}{\partial x} \right)^2 + 2 \left( \frac{\partial V}{\partial y} \right)^2 \\
\left( \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right)^2
\end{array} \right)
\]
(10)

\[
\Gamma_K = \frac{\mu_e}{\sigma_K}, \quad \Gamma_\varepsilon = \frac{\mu_e}{\sigma_\varepsilon}
\]
(11)

\[
\Gamma_\varepsilon = \frac{\mu_e + \mu_i}{\sigma + \sigma_i}
\]
(12)

\section*{Grid Generation:}

To avoid the instability in the solution, and unrealistic pressure field, the staggered grid technique has been adopted. Staggered grids imply that different dependent variables are evaluated at different grid points. The scalar variables are stored at the main grid points, while the velocity components are stored at staggered grids (the velocities are defined at the faces of control volumes). Hence, the scalar variables including pressure are stored at the main grid nodes denoted by small circles, Versteeg (1995).

\section*{Discretization of the general form of Equations:}

The final discretised algebraic equation, Versteeg (1995):
\[
A_p \phi_p = A_x \phi_x + A_y \phi_y + A_s \phi_s + A_s \phi_s + S_p
\]
(13)

Where:
\[
A_p = A_e + A_w + A_s + A_s - S_p
\]
(14)

\[
A_e = D_e - \frac{F_x}{2}, \quad A_w = D_w + \frac{F_x}{2}, \\
A_s = D_s - \frac{F_x}{2}
\]
(15)

Were the above equations can be expressed in general form for different types of schemes as Patankar (1980):
\[
A_e = D_e A \begin{bmatrix} |Pe| \end{bmatrix} + |F_x,0|, \quad A_w = D_w A \begin{bmatrix} |Pe| \end{bmatrix} + |F_x,0|, \\
A_s = D_s A \begin{bmatrix} |Pe| \end{bmatrix} + |F_x,0|
\]
(16)

Here, A \begin{bmatrix} |Pe| \end{bmatrix} is a function concerned with various types of schemes. Expressions for A \begin{bmatrix} |Pe| \end{bmatrix} for different schemes are listed in Table (4-2) in which |Pe| is peclet number (Pe = Re Pr) at the control volume faces which may be expressed as the ratio of the strength of convection term (F) to that of diffusion term (D). The operator [[A …]] appearing in equations refers to the maximum value of the quantities contained within it.

| Scheme              | Formula for A(|Pe|)        |
|---------------------|---------------------------|
| Central difference  | 1-0.5|Pe|               |
| Upwind              | 1                           |
| Hybrid              | [[0,1-0.5|Pe|]]           |
| Power law           | [[0,(1-0.1|Pe|) 5]]       |
| Exponential         | Pe/|exp(|Pe|-1)          |

\section*{RESULTS AND DISCUSSION}

Fig. (1) Shows the effect of varying the mesh size on the drag coefficient at rotation speed \( \omega = 2000 \) and cylinder diameter D=40 cm. A grid independency test is performed to check the validity of the numerical technique, the numbers of nodes which had been taken are (2800, 3200, 3600, 4000). It is clear that the influence of the mesh size on the drag coefficient was small, so (3200) nodes was selected for all calculations.
The parameters that had a direct influence on the amount of reduction in the drag coefficient are the rotation speed ($\omega$), and diameter size (D), thus the effect of each parameter was studied in separate cases according to the value of vehicle speed ($U_\infty$) as will shown in the figures.

**Fig. (2.a-d)** show the effect of variation of cylinder rotation speeds on the drag coefficient ($C_D$) and wall shear stress ($\tau_w$) with vehicle speed ($U_\infty$) equal to 60 km/h. The cases are classifying according to diameter size. When the cylinder rotates, the fluid starts to inject over the surface, this action creates a low pressure zone directly after the cylinder and this causes an appearing of a streamwise vortices Solberg (1989). This streamwise jet help the air flow to cling more closely to the upper surface {1}, and this is leading to delay the separation zone and reduce the drag coefficient. The amount of reduction depends on rotation speed as shown in the figures. For the case of D=10, 20 cm, it can be seen that the drag coefficient ($C_D$) increases along distance as the rotation speed increases, but in the other cases of D=30, 40 cm they are started to decrease after the rotation speed ($\omega$) became 3000 r.p.m. This behavior is due to diameter size and the high vehicle speed ($U_\infty$) velocity. For small cylinder diameter the momentum injected of the flow is small and will not create enough streamwise vortices to help the flow to cling closely to the surface which has a reverse influence on the flow.

**Fig. (3.a-d)** show the effect of variation of cylinder rotation speeds on the drag coefficient ($C_D$) with vehicle speed ($U_\infty$) equal to 90 km/h. The cases also classified according to diameter size. For the cases of D=10, 20 cm, it can be seen that the drag coefficient ($C_D$) increases along distance as the rotation speed increases, but in the other cases of D=30, 40 cm they are started to decrease after the rotation speed ($\omega$) became 3000 r.p.m. This behavior is due to diameter size and the high vehicle speed ($U_\infty$) velocity. For small cylinder diameter the momentum injected of the flow is small and will not create enough streamwise vortices to help the flow to cling closely to the surface which has a reverse influence on the flow.

**Fig. (4.a-d)** show the effect of diameter sizes on the drag coefficient ($C_D$) with vehicle speed of ($U_\infty$) equal to 120 km/h. For the cases of $\omega$=1000, 2000 r.p.m the figures showed a little reduction in the drag coefficient, while the case of case of $\omega$=3000 r.p.m showed a significant reduction in the drag coefficient, Furthermore for the cases of $\omega$ =4000, 5000 r.p.m it can be seen a huge reduction in the drag coefficient. The amount of reduction depends on the diameter size, as shown in the figures and the amount of reduction increases by increasing the diameter. This is due to the significant amount of injected flow, and thus the streamwise vortices significantly increases.

**Fig. (5)** show the predicted distribution of velocity field vector over the surface with and without rotation, $\omega_0$=0, 3000, 5000 r.p.m for each vehicle speed, $U_\infty$= 90 to demonstrate the influence of rotation. It can be seen that at $W=0$ there is no vortices, but as the rotation starts, the effect of streamwise vortices appears and perform a significantly influence on the flow field.

**CONCLUSIONS**

For the case of vehicle speed ($U_\infty$) equal to (40km/h), the maximum reduction in the drag coefficient obtained by using a rotation speed ($\omega$) equal to (5000) r.p.m and a diameter equal to (40) cm. For the case of vehicle speed ($U_\infty$) equal to (60km/h), the maximum reduction in the drag coefficient obtained by using a rotation speed ($\omega$) equal to (5000) r.p.m and a diameter equal to (40) cm. For the case of vehicle speed ($U_\infty$) equal to (90km/h), the maximum reduction in the drag coefficient obtained by using a rotation speed ($\omega$) equal to (5000) r.p.m and a diameter equal to (40) cm. For the case of vehicle speed ($U_\infty$) equal to (120km/h), the maximum reduction in the drag coefficient obtained by using a rotation speed ($\omega$) equal to (5000) r.p.m and a diameter equal to (40) cm.
coefficient obtained by using a rotation speed ($\omega$) equal to (5000) r.p.m and a diameter equal to (40) cm. For the case of vehicle speed ($U_\infty$) equal to (120km/h), the maximum reduction in the drag coefficient obtained by using a rotation speed ($\omega$) equal to (5000) r.p.m and a diameter equal to (40) cm.

Using a rotating cylinder creates stream vortices and this helps in attachment of the flow to the wall and hence delaying the separation of boundary layer separation.

REFERENCES

- Rose C.McCallen, Larry J.Dechant, and David Pointer W. "DOS’s Effort to Reduce Truck Aerodynamic Drag – Joint Experiments and Computations Lead to Smart Design “34th AIAA Fluid Dynamics Conference and Exhibit, Portland, or, United States, June 28, 2004 through July 1, (2004).
(2.c) $D=30\text{cm.}$

Figure. (2) Effect of cylinder rotation on drag coefficient, for vehicle speed $U_\infty=60\text{ km/h.}$

(3.a) $D=10\text{cm.}$

(3.b) $D=20\text{cm.}$

(3.c) $D=30\text{cm.}$

(3.d) $D=40\text{cm.}$
Figure. (3) Effect of cylinder rotation on drag coefficient, for vehicle speed $U_\infty=90$ km/h.

(4.a) $\omega=1000$ r.p.m.

(4.b) $\omega=2000$ r.p.m.

(4.c) $\omega=3000$ r.p.m.

(4.d) $\omega=4000$ r.p.m.

Figure. (4.a-d) Effect of cylinder diameter on drag coefficient, for vehicle speed $U_\infty=120$ km/h.
Fig. (5) Velocity field vectors over upper surface at different values of $\Omega$ for vehicle speed $U_\infty=90$ km/h and $D=40$ cm.
NOMENCLATURE

A = Convection-diffusion coefficient (kg/s)

$C_D$ = coefficient of skin friction drag

$C_\mu C_1 \sigma C_2 \varepsilon$ = Constants in turbulence model

D = Diffusion term (kg/s)

F = Convection term (kg/s)

P = Pressure (Pa)

Pe = Peclet number (Pe = Re. Pr)

Re = Reynolds number (Re = $\rho U L / \mu$)

$U_\infty$ = Relative Free stream velocity (m/s)

U, V = Mean velocity components in x, y directions (m/s)

X, Y = Cartesian coordinates (m)

$S_\phi$ = Source term

$\omega$ = Rotational speed (r.p.m)

$\Gamma$ = Diffusion coefficient

$\varepsilon$ = Dissipation rate of turbulent kinetic

$\mu$ = Laminar viscosity (kg/m.s)

$\mu_t$ = Turbulent or eddy viscosity (kg/m.s)

$\mu_e$ = Effective eddy viscosity (kg/m.s)

$\rho$ = Density (kg/m³)

$\lambda$ = Thermal conductivity (W/m.k)

$\phi$ = Dependent variable

$\upsilon$ = Kinematic viscosity (m²/s)

$\upsilon_t$ = Turbulent kinematic viscosity (m²/s)

$\sigma_K$ = Constants for the K-\varepsilon model

$\sigma = \frac{\mu_c}{\lambda}$

$\sigma_t$ = Turbulent Prandtl number ($\sigma_t = \frac{\mu_t c_p}{\lambda}$)

ABBREVIATIONS

CFD = Computational Fluid Dynamics

FVM = Finite Volume Method

SIMPLE = Semi-Implicit Method for Pressure Linked Equations

TDMA = Tri-Diagonal Matrix Algorithm