Numerical simulation of flow around a simplified high-speed train model using OpenFOAM

To cite this article: I A Ishak et al 2016 IOP Conf. Ser.: Mater. Sci. Eng. 152 012047

View the article online for updates and enhancements.

Related content
- Numerical simulation of flow past a circular base on PANS methods
  J T Liu, Y Li, Y Gao et al.
- Numerical simulations of flow past a circular cylinder
  Gaurav Chopra and Sanjay Mittal
- Unsteady compressible flows in channel with varying walls
  P Poízková, K Kozel and J Horáek
Numerical simulation of flow around a simplified high-speed train model using OpenFOAM

I A Ishak*, M S M Ali and S A Z Shaikh Salim
Wind Engineering, Malaysia-Japan International Institute of Technology,
UTM Kuala Lumpur, Malaysia

*izuan.amin@gmail.com

Abstract. Detailed understanding of flow physics on the flow over a high-speed train (HST) can be accomplished using the vast information obtained from numerical simulation. Accuracy of any simulation in solving and analyzing problems related to fluid flow is important since it measures the reliability of the results. This paper describes a numerical simulation setup for the flow around a simplified model of HST that utilized open source software, OpenFOAM. The simulation results including pressure coefficient, drag coefficient and flow visualization are presented and they agreed well with previously published data. This shows that OpenFOAM software is capable of simulating fluid flows around a simplified HST model. Additionally, the wall functions are implemented in order to minimize the overall number of grid especially near the wall region. This resulted in considerably smaller numbers of mesh resolution used in the current study compared to previous work, which leads to achievement of much reasonable time simulation and consequently reduces the total computational effort without affecting the final outcome.

1. Introduction
The study of flow around high-speed train (HST) has been, and continues to be, the subject of intense research [1-3]. Rapid developments of HST, especially the ones currently running in Europe and Japan and some other countries, have attracted people's attention. Aggressive improvement in the context of the train's technology expansion has shown a trend to be faster, yet more energy efficient, from the last three decades, hence shaping a new interest among travellers. Many of the latest generation of HST such as Japanese Shinkansen E6, Spanish AVE class 103, China CRH380B, Italy ETR1000, German ICE-3, South Korea KTX Sancheon and France TGV POS reach speeds more than 300 km/h in regular operation. However, modern technologies nowadays are only focusing on the electric motor to speed up the train instead of understanding the flow field around it, which in the end leads to a decreased performance of HST due to large energy losses [4]. At these speeds, the aerodynamic loads become significantly important for the running performance of the train [3]. For instance, turbulence that occurs due to increased speed leads to disturbance of flows around the train, which will then cause the flow energies to be converted to aerodynamic drag, noise and vibration [4].

The alertness on the safety factors concerning the operation of HST, especially the one that relates to crosswind stability for railing vehicles, has grown significantly in the HST community. It has also become an alarming sign to those countries with high-speed rail networks [5]. The lateral stability of a train is an important safety issue as it is largely dependent on aerodynamic loads, which are deduced
from flow field phenomena surround the vehicle. As trains nowadays are often built using lightweight materials in order to achieve higher speed and they are shifting towards more comfortable vehicles in order to cater for high passengers capacity, these trends conclude that it is essential for HST to have a much better handling stability. However, the demands for considerably lighter weight construction and higher driving velocities have shown a conflict with the crosswind stability of a particular HST. The fact that there have been several serious incidents that occurred in recent years where strong winds were thought to be the main cause of the derailment has intensified the importance of further research on flow over a HST [6].

In the context of investigating the flow field around a HST using numerical analysis, the factor that may lead to result discrepancies is due to different shapes of HST that are modelled. Some analysed a more complex geometries with consideration of additional structures such as front spoiler, bogies and pantograph while others define the structure in much more simplified train shapes. Recent study by Hemida et al. [1] used a HST model without underbody complexities or inter-carriage gaps between coaches, and still showed a good comparable result with other previous studies that used complex geometries. Furthermore, shortened train length was also implemented in this study since the execution of numerical simulation for a complete train length requires more advanced computational resources. Based on Khier et al. [7], the reduction in length will not change the important physical features of the flow since the flow characteristic downstream of a certain distance from the nose of the train (less than one coach length) is comparatively constant. Therefore, HST model that is presented in this paper is in the form of a simplified and shortened model in order to optimize the simulation time.

As the external flow of fluid moves around bodies, it is anticipated that there would be changes in pressure and velocity fields. It is important to take into consideration the characteristics of the flow when designing and constructing bodies that are exposed to external flows and computational fluid dynamics (CFD) is a useful tool in understanding these conditions. CFD method is gaining importance nowadays as an option for conducting investigation related to wind engineering. The technique is widely used due to its advantages that include low cost and time consuming, as well as flexibility in measuring various parameters. There are numerous validation and verification studies based on wind tunnel and on-site measurement that have already been carried out to justify the accuracy of the results provided by CFD application. Most of the comparison studies confirmed that the method is reliable enough to accurately predict the flow over a HST.

In this study, the OpenFOAM CFD software package is used to numerically analyse the fluid flow characteristics surrounding the HST. The advantages of OpenFOAM computational toolbox [8] are that it is a free and open software package that is capable of simulating a wide variety of fluid flow processes. With an extensive range of features that are contributed voluntarily by the CFD community around the world, OpenFOAM has the capabilities to simulate a wide range of flow problems. In total, there are over 170 utilities available for grid generation, and pre- and post-processing [9]. This open source CFD package is the alternative to the commercial CFD packages and it is increasingly popular nowadays. In the past few years, several notable studies have been published by OpenFOAM users worldwide in the fields of CFD, computational heat transfer, fluid structure interaction and multiphase flow [9]. The objective of this study is to present a numerical simulation of flow around a simplified HST using the OpenFOAM free software package. The investigation method applied in this work is unsteady simulation using numerical method of unsteady RANS combined with SST k-\omega turbulence model. Two grid resolutions, which represent the coarse grid and the fine grid, are carried out in order to investigate the dependency of the results to the flow simulation. As the grid is refined, grid cells will become smaller. In the meantime, the number of cells in the flow domain increases and the time step is refined (reduced).

This paper is organized as follows. Firstly, the initial set-up inclusive of the model, domain and mesh are described. Next, the solution methodology including the equations used to solve the fluid flow problem is mentioned in details. Then, the result obtained from both coarse and fine meshes are compared with previously published simulated and experimented data. The computing machine for
these simulations are then reported to evaluate the computational cost required to simulate both cases. In the end, the conclusion is given in the last section.

2. Model description
The model used in this study is a simplified HST model, which replicates a similar model investigated in experiment by Sakuma et al. [11] and in simulation by Osth et al. [10], as can be seen in Figure 1, Figure 2 and Figure 3. Design configurations for the train model are as listed below:

- Leading top and side edges on the front are rounded using an elliptical profile with major axis in the ellipse length of 0.07H and the minor axis length of 0.04H as can be seen in Figure 3(a).
- The top and side edges on the rear end of the bluff body are rounded with a circular radius 0.107H as can be seen in Figure 3(b).
- Both front and rear bottom edges are not rounded at all and thus sharp.
- The model is placed on two egg-shaped supports and is lifted 0.41H above the ground in order to replicate the same condition as in the wind tunnel Sakuma et al. [11].
- The length of the train is 7H while the width and height are both equal to H (W = H = 0.56m).

3. Domain description
The dimension of domain cross section is 36H x 21H x 11.41H (length x width x height) whereby H is the height of the train model that equals to 0.56m. The distance from the inlet to the bluff body is 8H and the distance from bluff body to the outlet is 21H. These lengths were found to be sufficient in the previous simulation of flows around a simplified train models by Hemida et al. [12], Krajnovic and Davidson [13], and Hemida and Krajnovic [14]. As for the boundary condition on the inlet, a uniform velocity, \( U_\infty \), is set in the x-direction. On the ground plane, the slip condition is used together with the velocity component in the x-direction equals to inlet velocity, \( U_\infty \), in order to prevent the development of the boundary layer on the ground plane. The homogenous Neumann boundary condition is applied at the outlet. Lastly, on the lateral side and roof, the slip condition is used similar to the ground plane. In this study, the boundary condition setup is similar as in simulation works by Osth et al. [10]. Details of the domain size and its boundary conditions can be seen in Figure 4 and Figure 5.

The Reynolds number (Re) used in this paper is \( 3.7 \times 10^5 \) \( (Re = U_\infty H/\nu) \) based on height of the train \( H \), kinematic viscosity \( \nu \), and free stream velocity \( U_\infty \), respectively. This particular Reynolds
number is selected in order to validate the simulation with previous work by Osth et al. [10]. Based on Alam et al. [15], the effects of Reynolds number can actually be neglected when $Re > 3 \times 10^5$. There were also several other researchers who claimed that the effects of Reynolds number on aerodynamic coefficients are indeed very little. Among them are Orellano and Schober [16], who experimentally studied the influences of changing Reynolds number on aerodynamic coefficients of trains subjected to side winds. They measured the aerodynamic coefficients at three different Reynolds numbers and it was found that the Reynolds number had almost negligible influence. Per-Ake [17], who carried out similar experiments on a bus subjected to crosswind, also arrived at the same conclusion that Reynolds number has insignificant effect on the aerodynamic coefficients.

![Figure 4. Computational domain used in the numerical investigation: side view (not to scale)](image)

4. Mesh description

The cells in the computational domain are constructed using a structured non-uniform Cartesian mesh. Mesh refinement is applied on the train surface and its surrounding areas. Two grid resolutions are carried out, which indicate coarse and fine grids. These different grid resolutions are selected based on grid refinement ratio ($r$). Based on Celik et al. [18], it is desirable that the grid refinement ratio to be greater than 1.3. In this case, $r = 1.4$ is selected. Since the meshes are not uniform, the grid refinement ratio is calculated based on average grid size ($h_{ave}$) as shown in Equation 2 and Equation 3. Details of the grid resolutions can be seen in Table 1. The grid refinement ratio, $r$ and is calculated as follows:

$$r = \frac{h_{ave\ (coarse)}}{h_{ave\ (fine)}}$$  

(1)

where the average cell size ($h_{ave}$) is obtainable using below formulation:

- Three dimensional calculation: $h_{ave} = \left[ \frac{1}{N} \sum_{i=1}^{N} (\Delta V_i) \right]^{\frac{1}{3}}$

(2)

- Two dimensional calculation: $h_{ave} = \left[ \frac{1}{N} \sum_{i=1}^{N} (\Delta A_i) \right]^{\frac{1}{2}}$

(3)

where $\Delta V_i$ is the volume, $\Delta A_i$ is the area of the $i^{th}$ cell and $N$ is the total number of cells used for the computations.
Table 1. Grid parameter for case A and case B

<table>
<thead>
<tr>
<th>CASE</th>
<th>A (Coarse)</th>
<th>B (Fine)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total no. of cells</td>
<td>844,148</td>
<td>2,114,715</td>
</tr>
<tr>
<td>Cell size, $h_{ave}$</td>
<td>0.1215</td>
<td>0.08895</td>
</tr>
<tr>
<td>Averaged $y^+$</td>
<td>100.34</td>
<td>81.76</td>
</tr>
<tr>
<td>Refinement ratio, $r$</td>
<td>1.4</td>
<td>1.4</td>
</tr>
</tbody>
</table>

On the other hand, Figure 6 and Figure 7 show detailed view of grid construction for both Case A (coarse) and Case B (fine) across the simulation domain.

Figure 6. Overall grid structure from front view: (a) Case A (coarse), (b) Case B (fine)

Figure 7. Overall grid structure from side view: (a) Case A (coarse), (b) Case B (fine)

Wall function is used in all the cases to reduce the computational cost by properly treating the cell size near the surface [19]. The distance of the first cell layer to the model surface should be located within the requirement of $y^+ (30 < y^+ < 300)$. Figure 8 shows the details of $y^+$ value acquires for both cases A (coarse) and B (fine) along the mid plane of the train model. As can be seen, the result of $y^+$ value was distributed well in the requirements range.
Stokes and continuity equations. The pressure implicit split operator (PISO) solution algorithm here are Transport (SST) turbulence model.

Flow around the train was considered as incompressible and obtained by solving incompressible form of the Unsteady Reynolds Averaged Navier-Stokes (URANS) equation combined with the help of $k - \omega$ based Shear-Stress-Transport (SST) turbulence model. Actually, there are several selections of turbulence models that can be chosen to solve the fluid flow problems. The simplest is one-equation model such as Baldwin-Barth’s, Spalart-Almaras’ and Prandtl’s models. The solution for this type of models is quick but it does not take into account the convection and diffusion of the turbulent energy. To include these variables, two-equation models can be used [21]. Two most predominant models in two-equation models are $k - \varepsilon$ model and $k - \omega$ model.

Menter [22] introduced the SST model that combines positive features of both $k - \varepsilon$ and $k - \omega$ models. The idea is to employ the wall-bounded advantages through viscous sub-layer of the $k - \omega$ model at near the wall region and the $k - \varepsilon$ model's free stream advantages near the boundary layer edge. This avoids the common $k - \omega$ problem that the model has, which tends to be too sensitive to the inlet free-stream turbulence properties. By taking these benefits, the model was selected in this study due to the fact that severe separation region near the wall bounded area and the wake structure of flow over the HST model in high Reynolds number was anticipated in the simulation. In addition, there was also a similar study in the past by Prime et al. [19] that showed comparable results with the experiment when the SST $k - \omega$ turbulence model was imposed in the simulation method carried out. In this numerical study on flow modelling of interacting prisms, both the flow velocity and turbulence intensities in various positions in the wake were compared with experimental hotwire data measured and the results showed a good agreement for both approaches.

5. Solution methodology

Primitive variables are numerically calculated based on the three dimensional unsteady incompressible Navier-Stokes and continuity equations. The pressure implicit split operator (PISO) solution algorithm [20] with one predictor step and two corrector steps for the pressure-velocity coupling is used in this study. For the discretization scheme, 2nd order backward scheme is used for temporal discretization, 3rd order QUICKV scheme is used for convection term and 2nd order unbounded Gauss linear differencing scheme is used for viscous term. Two different time steps have been used in correspondence to the two different grid cases, such that the Courant-Fredichs-Lewy (CFL) number is always below unity. Table 2 shows time step for pressure, convection and diffusion term, $\Delta t$ for each case.

<table>
<thead>
<tr>
<th>CASE</th>
<th>A (Coarse)</th>
<th>B (Fine)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Delta t$</td>
<td>0.005</td>
<td>0.002</td>
</tr>
</tbody>
</table>

Figure 8. Value of $y^+$ along a line ‘s’ starting at origo along the mid plane of the train model
5.1. Governing equations
The URANS equations are principally obtained from the usual RANS equation but the unsteady term has been maintained [22]. Calculations are carried using two equations: continuity and Navier-Stokes equation for the incompressible flow as follows.

$$\frac{\partial u_i}{\partial x_i} = 0$$  \hspace{1cm} (4)

$$\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = - \frac{\partial p_i}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_j}{\partial x_j} + \frac{\partial u_i}{\partial x_i} \right) \right] + \rho f_i \hspace{1cm} (5)$$

The velocity components, \(u_i\) and the pressure, \(p_i\) are both nonlinear partial differential equations. This means that there is no analytical solution for the problem with arbitrary boundary conditions. The unsteadiness of the flow variables (i.e. velocity and pressure) are decomposed into mean value and fluctuations as follows:

$$u_i = U_i + u'_i \hspace{1cm} (6)$$

$$p_i = P_i + p'_i \hspace{1cm} (7)$$

where \(U_i\) and \(P_i\) are the time averaged terms, \(u'_i\) is the fluctuation terms of velocity, and \(p'_i\) is the fluctuation terms of pressure. Substituting these Reynolds decomposed velocities and pressures into the equation of continuity and Navier-Stokes equation yields the Reynolds Averaged Navier-Stokes equation of motions as shown below:

$$\frac{\partial u_i}{\partial x_i} = 0$$  \hspace{1cm} (8)

$$\frac{\partial u_i}{\partial t} + U_j \frac{\partial u_i}{\partial x_j} = - \frac{1}{\rho} \frac{\partial p_i}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_j}{\partial x_j} - \rho u'_i u'_j \right) \hspace{1cm} (9)$$

5.2. Turbulence model
For a URANS simulation, in order to solve the governing equations, Reynolds stress tensor \(-\rho u'_i u'_j\) must first be determined. The Reynolds stress can be modelled either by additional equations or from the known quantities in order to achieve “closure” for the governing equations. Closure denotes that there is a sufficient number of equations for all the unknowns, including the Reynolds stress tensor resulting from the averaging procedure. The equation used to close the system is depending on the turbulence model being used.

A turbulence model is a computational procedure to close the system of flow equations as derived earlier. Most of them are based on the Boussinesq hypothesis. Based on Boussinesq, Reynolds stress tensor could be linked to the mean rate of deformation. The most widely used concept is as below:

$$-\rho u'_i u'_j = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij} \hspace{1cm} (10)$$

where the turbulent kinetic energy \((k)\) and specific dissipation rate \((\omega)\) are solved using the following equations:

Turbulence Kinetic Energy, \(k\):

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ (v + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right] \hspace{1cm} (11)$$

Specific Dissipation Rate, \(\omega\):

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ (v + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_T) \sigma_{\omega} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial k}{\partial x_i} \hspace{1cm} (12)$$

where \(\nu_T\) is the kinematic eddy viscosity and can be defined as follows:

$$\nu_T = \frac{a_{1k}}{\max (a_{1\omega}, SF_2)} \hspace{1cm} (13)$$
The following closure coefficient is used in this study

\[
F_2 = \tanh \left[ \max \left( \frac{2 \sqrt{E}}{B^* \omega y} \frac{500 \nu}{y^2 \omega} \right)^2 \right]
\]

(14)

where \( y \) is the distance to the next surface.

\[
P_k = \min \left( \tau_{ij} \frac{\partial u_i}{\partial x_j}, 10B^*k \omega \right)
\]

(15)

\[
F_1 = \tanh \left( \min \left[ \max \left( \frac{\sqrt{E}}{B^* \omega y \frac{500 \nu}{y^2 \omega}}, \frac{4 \sigma_{w^*} k}{C_D \omega y^2} \right) \right] \right)
\]

(16)

\[
C_D \omega = \max \left( 2 \rho \sigma_{w^*} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right)
\]

(17)

\[
\phi = \phi_1 F_1 + \phi_2 (1 - F_1)
\]

(18)

\[
\alpha_1 = \frac{5}{9}, \alpha_2 = 0.44
\]

(19)

\[
\beta_1 = \frac{3}{40}, \beta_2 = 0.0828, \beta^* = \frac{9}{100}
\]

(20)

\[
\sigma_{k_1} = 0.85, \sigma_{k_2} = 1, \sigma_{\omega_1} = 0.5, \sigma_{\omega_2} = 0.856
\]

(21)

6. Results and discussions

6.1. Pressure coefficient

For comparison purposes, results obtained from the two grid resolution were compared with numerical data and experimental data studied by [10, 11]. Pressure coefficient is selected and defined as follows:

\[
C_p = (\rho_p - \rho_\alpha/0.5 \rho U^2_\infty)
\]

(22)

The pressure coefficient along a line in the mid plane surface on train model is shown in Figure 9. In general, the pressure follows the same pattern with the result in [10, 11]. However, some discrepancies occurred at separated flow region on the front roof of the train. Recent work seems to underestimate the negative pressure at this specific point. From the simulation, the flow reattached at a bit further downstream as compared to previous simulation and experimental results. Fine mesh resolution (Case B) appeared to produce more consistent result in the separated region compared to coarser mesh (Case A). However, Case B and Case A showed significant similarity in the reattachment region as the flow pass through the roof. In the wake region, the simulation slightly overestimated the value of pressure coefficient. Nonetheless, both meshes still captured the lowest peak value of the pressure coefficient similar to Osth et al. [10].

Figure 9. Pressure coefficient starting at origo along the mid plane of the train model
The discrepancies on the results obtained with previous simulation work by Osth et al. [10] are mainly due to the differences in numerical approach for the simulation model used, as well as the mesh resolution covered. Table 3 shows the differences of the computational setting for Case B (fine mesh) and Osth et al. [10].

<table>
<thead>
<tr>
<th>CASE</th>
<th>Osth et al. [10]</th>
<th>Case B (Fine)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simulation model</td>
<td>PANS (Partially Averaged Navier Stokes)</td>
<td>URANS (Unsteady Reynolds Averaged Navier Stokes)</td>
</tr>
<tr>
<td>No. of extra transport equations</td>
<td>Four model equations $(k - e - \zeta - f)^a$</td>
<td>Two model equations $(k - \omega)^a$</td>
</tr>
<tr>
<td>Algorithm</td>
<td>SIMPLE (Semi-Implicit Method for Pressure Linked Equations)</td>
<td>PISO (Pressure Implicit with Splitting of Operator)</td>
</tr>
<tr>
<td>Time step</td>
<td>0.00038 (CFL &lt; 1)$^b$</td>
<td>0.002 (CFL &lt; 1)$^b$</td>
</tr>
<tr>
<td>No. of cells</td>
<td>12 million</td>
<td>2 million</td>
</tr>
</tbody>
</table>

$^a$where $k$ is the turbulence kinetic energy, $e$ is the dissipation in the flow, $\zeta$ is the scale ratio (e.g. $\zeta_u$ is the velocity scale ratio), $f$ is the ratio (e.g. $f_e$ is the ratio of unresolved dissipation to resolved) and $\omega$ is the specific dissipation rate.

$^b$CFL is the Courant-Fredichs-Lewy condition which must be kept below unity

6.2. Global quantity

Time-averaged drag force coefficient from simulations was compared with that obtained numerically by Osth et al. [10] and experimentally by Sakuma et al. [11]. The result is presented in Table 4. The drag coefficient is defined as follows:

$$C_D = \frac{2F_D}{\rho U_\infty^2 A_x} \tag{23}$$

where $\rho$ is the density of air at 20°C and $A_x = H^2$.

<table>
<thead>
<tr>
<th>CASE</th>
<th>$C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment by Sakuma et al. [11]</td>
<td>0.86</td>
</tr>
<tr>
<td>Numerical by Osth et al. [10]</td>
<td>0.78</td>
</tr>
<tr>
<td>Case A (Coarse)</td>
<td>0.71</td>
</tr>
<tr>
<td>Case B (Fine)</td>
<td>0.73</td>
</tr>
</tbody>
</table>

The drag force coefficient obtained is relatively underestimated compared to numerically attained by Osth et al. [10] by 6% for Case B and 9% for Case A. The under-prediction of the negative pressure that occurred in separated region on the roof might be one of the reasons that resulted to discrepancies in the results of drag force coefficient with the previous works. This is due to the fact that separation region that started from the front leading edges extended much longer, hence deduced much negative pressure that contributed to a decrease in the total drag coefficient of the train model.

6.3. Streamlines of the time-averaged flow

Figure 10 and Figure 11 show a side view streamlines of the time averaged velocity field comparison with the previous data obtained by Osth et al. [10]. Only Case B (fine) is presented here. Generally, the current simulation was able to replicate the flow phenomena as obtained by Osth et al. [10].
In Figure 10, the flow separation occurred at the leading edge at the top and bottom sides, and the vortex produced due to this is denoted as \( V_p \). Only a small discrepancy in the size of vortex appears on the roof of the HST model, which was a bit lower than attained by Osth et al. [10]. This confirmed the earlier argument that the flow reattached further back, which made the drag force coefficient, \( C_D \), larger than the one obtained by Osth et al. [10]. On the other side, a similar flow structure produced at the second half of the train as can be seen in Figure 11(i) and (ii). There were two vortices at the upper and bottom sides that occurred in the wake, denoted as \( V_w \). Simulation result agrees well with previous simulation data with the vortex \( V_w \) extends a distance \( H \) in the streamwise direction from the base.

Figure 12 and Figure 13, on the other hand, illustrate top view streamlines of the time averaged velocity field comparison. In Figure 12, flow separation occurred at the both of side leading edges and produced vortex similar in size. Same as previously stated, the vortex appeared to be slightly extended in the streamwise direction as compared to Osth et al. [10]. For the flow structure formed at the second half of the train as can be seen in Figure 13(i) and (ii), it similarly resembles the same shape as in the previous paper. There were two vortices occurring sideways in the wake. The vortex extended with a distance \( H \) in the streamwise direction from the base.

7. The computing machine
The simulations using OpenFOAM software in this recent work utilized parallel processors. To run the simulation job in parallel, the computational domain needed to be decomposed into several smaller domains. This was set by using `decomposePar` utility, where the total number of processors to run a particular job was selected. With the use of parallel processors, running any OpenFOAM job could be made faster. Table 5 shows the performance of the computer in simulating case A (Coarse) and case B (Fine).
Table 5. Computer performance for different number of processors used in the simulation

<table>
<thead>
<tr>
<th>CASE</th>
<th>A (Coarse)</th>
<th>B (Fine)</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of processors</td>
<td>8</td>
<td>32</td>
</tr>
<tr>
<td>Clock time (hour)</td>
<td>265</td>
<td>325</td>
</tr>
<tr>
<td>Simulation time ($U_\infty/H$)</td>
<td>1000</td>
<td>1000</td>
</tr>
<tr>
<td>Time step ($\Delta t U_\infty/H$)</td>
<td>0.005</td>
<td>0.002</td>
</tr>
</tbody>
</table>

8. Conclusion
The flow around a simplified HST has been simulated numerically with OpenFOAM framework by utilizing Unsteady Reynolds Navier-Stokes (URANS) equation combined with SST k-ω turbulence model for two different mesh resolutions. In this paper, it is well confirmed that, with the usage of OpenFOAM, flow structure and pressure distribution can be captured correctly and the outcomes have a good agreement with other similar studies. Study on two different mesh resolutions shows that the fine grid results in a more accurate data when compared to previous work. With the smaller mesh resolution used in this study compared to that previously used by other examiners, much reasonable time simulation can be achieved and this subsequently reduces the computational cost without effects on the final result.

Acknowledgement
This research was financially supported by Malaysia Ministry of Higher Education (MOHE) under The Fundamental Research Grant Scheme (FRGS PY/2015/05383) and also from the COMSTECH-TWAS 13-272. The authors also acknowledge the High Performance Computer (HPC), Universiti Teknologi Malaysia for the use of their supercomputer facilities.
References