Numerical and experimental analysis of drag force in medium speed train design

Jean Mario*,1, Beny Halfina2, Dimas Bahtera3, Lukman Shalahuddin4, Agus Windharto5

1,2,3,4 Agency for Assessment and Application of Technology - BPPT, Indonesia
5 Center for Science and Technology – PUI ITS, Surabaya, Indonesia

*Email: jean.mario@bppt.go.id

Abstract. Aerodynamic drag is the highest portion of the overall resistance force in a high speed train. More streamlined design will have lower coefficient of drag. Therefore, improving the mask of car and carbody is essential to minimise the drag force. This study aims to obtain the value of the drag coefficient on the design of Indonesia’s Medium Speed Train which has an operational speed of 160 km/h by the numerical method and then to validate it with the wind tunnel experimental test. In the initial stage, two designs was developed using a 3D CAD software. Subsequently, a CFD simulation was performed to determine the value of the drag coefficient. Two test models were built with a scale of 1:25, then both were tested in a wind tunnel. Based on the test results, the coefficient of drag of both models have relatively similar value. The graph data from the CFD simulation results then compared to the wind tunnel results. In addition, the wind speed variations was conducted to observe the effect of the Reynolds number on the drag coefficient.

1. Introduction
This research was conducted to support the Jakarta-Surabaya Corridor Rail Speed Improvement Project. The results of the feasibility study are the implementation of the Jakarta-Surabaya semi- high-speed train (Medium Speed Train). It would cut the currently 9 hours travel time to only 5.5 hours with an operational speed of 160 km/hour. In fact, train speed in Indonesia is limited by the infrastructure issue, which is narrow gauge of 1067 mm. This condition could lead the possibility of derailment due to instability of train motion. Despite the shorter travel time, speed increase affects the aerodynamics of the train. It is imperative to observe the induced drag, noise generated by the aerodynamic force, and vibrations generated from the wind (Cui, Wang, Hu, Gao, & Yang, 2012). Due to the movement of the train, there is pressure difference between the front area or Mask of Car and the tail end of the train which will cause the drag force (Banga, Zunaid, Ansari, Sharma, & Dungriyal, 2015). On previous research, the air flow velocity in the wind tunnel was determined after observing the Reynolds number consequences on the coefficient of drag (Zhang, Yang, & Liang, 2018). Others, the result between physical and computational modelling techniques, whilst predicting the average value of the force coefficient can be accounted as the approached method and techniques of modelling is close to actual conditions (Gallagher et al., 2018).
To achieve minimum drag resistance, the Mask of car and carbody of the train are the main component that must have good aerodynamic performance. The basic tools in the study of train aerodynamics are CFD calculation and wind tunnel test. The numerical CFD simulation is beneficial to cut down the modelling cost and reduce the time of the model development. It enables the design team to do rapid testing of the drag coefficient of the model and provides guidance for the iterative method of refining the 3D model to get the optimum coefficient of drag. Moreover, the CFD simulation need to be validated using wind tunnel test as the proven method who have been adopted by the aerodynamic experts to assess the train aerodynamic characteristics. In this study, wind tunnel test was conducted to obtain the aerodynamic coefficient, particularly the coefficient of drag. In the case of the study of train aerodynamics, all of these approaches are challenged with difficulties (Baker, 2010). This study aims to determine the value of the drag coefficient on the design of Indonesia’s Medium Speed Train (MST) which has an operational speed of 160 km/h both numerically and experimentally.

2. Method

2.1. 3D CAD Model Preparation

The initial stage of this research is to prepare the 3D CAD Model of MST based on the required specification. The fundamental idea in designing MST that has good aerodynamics is the form of a streamlined mask of car and carbody that will result in easier freestream flow from the upstream to the downstream. The slender shape of the train nose has been used in the modern high-speed train to decrease in the drag that occurs on the body which will increase the aerodynamic efficiency of the train body. However, with the main propulsion being used is a diesel electric which requires a larger space, the mask of car train has limited dimensions to be built with longer nose shape. In this research, there are two different models of MST, Model A and Model B which have different profile of mask of car as shown in Figure 1 and Figure 2.

![Figure 1. 3D CAD MST model A](image)

![Figure 2. 3D CAD MST model B](image)

Figure 3 shows the configuration of Medium Speed Train model for the wind tunnel test which has 3 cars. This configuration allows one to observe the effect of air flow in the head, the carbody and after leaving the tail end of the MST.

![Figure 3. Configuration of medium speed train model for wind tunnel test](image)

2.2. CFD Preparation

Following the design stage in a 3D CAD model, numerical simulations are performed using CFD. From this step, the governing equations which consist of mass conservation, momentum equation and k-e standard turbulent model is solved using computational method (Utomo, Fajar, & Arpriyanto, 2018). However, since the simulation based on a theoretical calculation, the accurate result will always depend on the correct configuration and simulation set up. Based on the actual operating condition, the simulation setup was built. For each model, the setup is similar as illustrated in Figure 4 below.
Figure 4. Computational domain

Figure 4 illustrates the MST model within the computational domain. Boundary condition of the front surface of the train is set as velocity inlet and the downward surface behind the train is set as pressure outlet. The boundary type of boundary condition of frontal surface is set as inlet and has distance of 20,000 mm from the frontal train nose, while the pressure outlet downward the tail end of the MST model is set as 50,000 mm. The width of the enclosure is set to be 5,000 mm from the outer edge of the MST, while the height is set to be 10,000 mm. The space between ground and bogie of the MST is set to be 500 mm. The size of the domain was determined to meet the actual size of the wind tunnel test chamber. Thus, the computational domain is assumed to have similar size and condition with wind tunnel.

Figure 5. Meshing of the air space surrounding the model

Figure 5 shows that default set of meshing was used for wind tunnel area due to the simplification of simulation which is for Model A has maximum aspect ratio 3.303057e.+01 with number of nodes 577,294 and elements 502,455 while Model B has maximum aspect ratio 4.50723e.+01 with number of nodes 660,834 and elements 566,649. For detailed analysis needs, the meshing was fined specifically in the mask of car area which was set to 100 mm. Then the boundary conditions parameters determined as shown in Table 1:

Table 1. Boundary conditions properties
| Parameters     | Value          |
|---------------|---------------|
| Wind Speed    | 44 m/s        |
| Method        | 3 Dimension   |
| Solver type   | Pressure based|
| Model viscous | $k$-epsilon SST |

In the boundary parameters, the wind speed was set at 160 km/hour or 44 m/s and has no increment considering the Reynolds Number check in wind tunnel which will be discussed on the result and discussion, density of air is 1.225 kg/m$^3$ and reference area is 9 m$^2$ based on the geometry of mask of car of MST model. Thus, the CFD simulation determined drag coefficient as the value of air resistance of MST which is calculated using the following equation:

$$C_D = \frac{F_D}{\frac{1}{2} \rho U^2 A}$$  \hspace{1cm} (1)

This coefficient of drag is influenced by the Reynolds numbers (Re). Reynolds numbers are a function of wind speed which is calculated with the formula below:

$$N_{Re} = \frac{\rho v d}{\mu}$$  \hspace{1cm} (2)

2.3. Wind Tunnel Model Preparation

It is important to always validate the result of numerical simulation with the actual condition. Thus, in this study, the wind tunnel test was used to validate the results. The Experiments in wind tunnels are a reliable method and have long been used by aerodynamics experts to study the aerodynamic characteristics of trains. The trainset was modelled with a certain scale adjusting the wind tunnel test section. Several wind flow velocities were blown into the chamber. A railroad study in a wind tunnel is conducted to find the aerodynamic coefficient values of the model, especially the coefficient of drag. In addition to finding the coefficient of drag, the pattern of air flow which forms around the body of the model can be determined by visualizing the flow. The test facility is the Indonesian Low Speed Tunnel (ILST) as shown in Figure 6 equipped with 6 component internal balance, data acquisition and processing system, and a set of train models.

![Figure 6. Indonesian low speed tunnel (ILST)](image1)

ILST is a wind tunnel with a closed test section and has an atmospheric type. The test section has a rectangular shape and measures 4 m x 3 m x 8 m as shown in Figure 7. On the left and right sides of the end of the test section there are air holes to neutralize the air pressure in the test section to the conditions of standard air pressure (standard atmosphere).
The model tested in the ILST wind tunnel is a set of train models that are scaled 1:25 with dimensions of length 2800 mm, width 120 mm and height 120 mm. A set of models consists of three train cars namely head car, middle car and tail car which are equipped with central strut system. The central strut connects the train model to the internal balance mounted under the ILST turntable. In addition, there is an additional obstacle barrier feature (cowcatcher). At full scale conditions, the train is planned to operate at speeds of 160 km/h or around 44 m/s. The configuration of wind tunnel test described in Figure 8.

![Figure 8. Design configuration for wind tunnel test](image)

Sequences of testing the model in a wind tunnel include internal balance check load, ground board and model installation, flow checking on the ground board around the model, checking Reynolds numbers, measurement of drag coefficient on the model, flow visualization on the model. Detailed test configurations are listed in the test run log. The actual model setup inside the wind tunnel is shown in Figure 9, Figure 10, and Figure 11.

![Figure 9. Model A](image)  ![Figure 10. Model B](image)
Figure 11. MST model on test section

3. Result & Discussion

3.1. CFD Result

CFD simulation of the two MST configurations was run until achieving convergence, the drag force and coefficient of drag result is shown in Table 2 below. Parameters in the numerical simulation using CFD are wind speed 160 km/hour or 44 m/s, density of air 1.225 kg/m$^3$ and reference area is 9 m$^2$. After the simulation reached convergence, the drag force result is 9797.41 N with the coefficient of drag (CD) is 0.902.

| Train Configuration | Velocity (m/s) | Density of Air (kg/m$^3$) | Reference Area (m$^2$) | Drag Force (N) | Coefficient of Drag |
|---------------------|----------------|---------------------------|------------------------|----------------|---------------------|
| Single Car          | 44.4           | 1.225                     | 9                      | 7029.63        | 0.647               |
| Triple Cars         | 44.4           | 1.225                     | 9                      | 9797.41        | 0.902               |

Table 2. Result of model A simulation

| Train Configuration | Velocity (m/s) | Density of Air (kg/m$^3$) | Reference Area (m$^2$) | Drag Force (N) | Coefficient of Drag |
|---------------------|----------------|---------------------------|------------------------|----------------|---------------------|
| Single Car          | 44.4           | 1.225                     | 9                      | 5981.81        | 0.550               |
| Triple Cars         | 44.4           | 1.225                     | 9                      | 8520.30        | 0.784               |

Table 3. Result of model B simulation

Table 2. show the two model configurations of Model A which resulted different drag force and drag coefficient depending on the number of cars. Drag force of single car has a value 7029.63 N, while the triple cars have higher value in the similar condition. It is proportional to the drag coefficient value from both of configurations. Single car attained a value 0.647 and triple cars reached 0.902.

Table 3 illustrates two model configurations of Model B which resulted similar trend of drag force and drag coefficient value. The single car of Model B has drag force and drag coefficient sequentially 5981.81 N and 0.550. While the triple cars have drag force and drag coefficient sequentially 8520.3 N and 0.784. This phenomenon reveals that the number of cars has influence to drag force and coefficient of drag, which mean length of train affect the downstream and upstream air flow.

Air velocity distribution and pressure distribution along the trainset are shown in the series of figures below. Figure 12 and Figure 13 show the contour plot for velocity in model A and B while pressure distribution are shown in Figure 14 and Figure 15. The nose region of the MST has the high-pressure area. While relatively lower pressure shown in the window of the train. As the train reached maximum speed the pressure drag escalated and so does the drag force. High-pressure region is also observed at the gaps between cars contributing the total drag resistance of the trainset. Previous research (Utomo et
al., 2018) mentioned that the total drag coefficient in the high-speed train consists of pressure drag coefficient and viscous drag coefficient. However, in the case of a train the viscous drag effect is larger than the pressure due to the geometry of the train which has very long body compared to width or height. From this simulation, the designer can get the insight to improve the design of the Mask of Car. Optimized shape design will improve the aerodynamic performance thus reducing the pressure drag and total drag force which eventually reducing fuel consumption.

Figure 12. Air velocity distribution model A

Figure 13. Air velocity distribution model B

Figure 14. Pressure distribution model A

Figure 15. Pressure distribution model B

More detail image on the difference in the vortices around the train model A and B are shown in Figure 16 and Figure 17 which explain why there is a difference between the aerodynamic forces. Due to more complex profile at the area of windshield in model A, there are more vortices around the windshield and under the nose. Since vortex drag is the main component of aerodynamic drag, the aerodynamic drag of the model A is bigger than those in model B.

The velocity contour shows a blue region at the nose of the train which indicates that when the air strikes the train its velocity rapidly decrease or became stagnant. The air particle gets denser thus result in the higher pressure. According to (Banga et al., 2015), air has a tendency travelling upwards from the high pressure region the low-pressure region on top. Contrast to the nose area, at the top of the train head or above the windshield the velocity contour shown red region which means that the air velocity increases more than 60 m/s.
In the CFD simulation, a pressure-based solver in Fluent was used for the numerical simulation. In the control volumes of the cells, the pressure gradients were calculated. Surface pressure on the train body was mainly concentrated at the front and rear of the train. The plot contour compares the pressure distribution of the train. There is different pressure contour due to the frontal shape of the train as shown in Figure 18 and Figure 19. Model A with relatively complex shape and flat surface of the windshield results in the negative pressure. While in model B, the pressure is only at the tip of the nose.

3.2. Wind Tunnel Result
In the wind tunnel test of MST models, it was conducted the Reynolds Number check which is needed to review the dependence of air velocity to coefficient of drag. In the wind tunnel, the static ground is not like the actual operating condition. There is a boundary layer formed at the static ground affecting the Reynolds Number (Chen, Yao, Lin, & Han, 2017). Theoretically, aerodynamic coefficient value is similar between model and prototype. This coefficient is influenced by Reynolds Number which is the function of air velocity (the formula is shown as Eq. 2). In this wind tunnel test, the aerodynamic coefficient value can be used following the air velocity has slightly affect the coefficient of drag. It means that varying air velocity which are blown, the aerodynamic coefficient will be steady. Reynolds Number check for models used 11, 22, 33, and 44 m/s as shown in the Table 4.

| Air velocity (m/s) | Reynolds Number |
|-------------------|-----------------|
| 11                | 8.74 x 10⁴      |
| 22                | 1.75 x 10⁵      |
| 33                | 2.62 x 10⁵      |
| 44                | 5.5 x 10⁵       |
While the Reynolds Number of full scaled MST on operation speed 44 m/s is $8.74 \times 10^6$. Figure 20 illustrates that there is no effect between Reynolds Number to coefficient of drag of both of models. Thus, the wind tunnel test is be able to measure the coefficient of drag with multiple variation of air velocity with constant result.

![Graph](image)

**Figure 20.** Reynolds number check

The wind tunnel test discovered that the drag coefficient of model A as shown in Figure 21 is 0.99 and drag coefficient of model B as shown in Figure 22 is 0.84.

![Wind tunnel model A](image)

**Figure 21.** Wind tunnel model A

![Wind tunnel model B](image)

**Figure 22.** Wind tunnel model B

Comparison of graph data from CFD simulation results with wind tunnel test results shows relatively small difference of 7%. The errors in the CFD compared to the actual measurement caused by some components, biggest proportion is the front stagnation region (Perzon, Janson, & Höglin, 1999). The wind tunnel test validates the result of numerical simulation.

4. Conclusion

The CFD simulation was run for two model type, single car and three cars configuration. In the single car model, the coefficient of drag of model A and B is found to be 0.647 and 0.550 respectively. The three cars configuration has bigger Cd, 0.902 for model A and 0.784 for model B. The value of CD shown consistently to the model shape regardless of the length of configuration. However, the three cars configuration is closer to the actual trainset. Thus, this configuration is used in the experimental wind tunnel test. The results from the numerical simulation then validated using the wind tunnel test. It is found that there is 7% difference between the numerical simulation and the experimental test. One factor affecting the difference was possibly caused by the ground board where the train model is mounted. It has its own drag coefficient; therefore, the drag coefficient of the wind tunnel test results is bigger. In addition, the support system that connected the model to the balance also contributed to the drag coefficient. There needs to be a change or improvement in the method of testing in the wind tunnel so that the results are more accurate. Nevertheless, the difference is relatively small, which proves that the
numerical simulation has a reliable result and the setting parameters can be used for further development of the models.

Acknowledgment
This research was funded by Badan Pengkajian dan Penerapan Teknologi (BPPT). We thank our colleagues from Balai Besar Teknologi Aerodinamika, Aeroelastika dan Aeroakustika (B2TA3) who provided insight and expertise that greatly assisted the research and thank to our colleagues from Institut Teknologi Sepuluh November (ITS) Surabaya for providing the 3D model of the MST. We also thank to PT. INKA (Persero) who provided the ANSYS 19.0 Software as tool for this research.

Reference
[1] Baker C 2010 The flow around high speed trains Journal of Wind Engineering and Industrial Aerodynamics 98 277–298
[2] Banga S, Zunaid M, Ansari N A, Sharma S and Dungriyal R S 2015 CFD simulation of flow around external vehicle IOSR Journal of Mechanical and Civil Engineering 12 87–94
[3] Chen D, Yao S, Lin P and Han Y 2017 Wind tunnel test study for the influence of reynolds number on the aerodynamic performance of high-speed trains DEStech Transactions on Engineering and Technology Research 131–139
[4] Cui K, Wang X P, Hu S C, Gao T Y and Yang G 2012 Shape optimization of high-speed train with the speed of 500kph Lecture Notes in Electrical Engineering 148 187–197
[5] Gallagher M, Morden J, Baker C, Soper D, Quinn A, Hemida H and Sterling M 2018 Trains in crosswinds – comparison of full-scale on-train measurements, physical model tests and CFD calculations Journal of Wind Engineering and Industrial Aerodynamics 175 428–444
[6] Perzon S, Janson J and Höglund L 1999 On comparisons between CFD methods and wind tunnel tests on a bluff body SAE Technical Papers 724
[7] Utomo T, Fajar B and Arpriyanto H 2018 CFD analysis on the aerodynamics characteristics of Jakarta-Bandung high speed train MATEC Web of Conferences 159
[8] Zhang L, Yang M zhi and Liang X feng 2018 Experimental study on the effect of wind angles on pressure distribution of train streamlined zone and train aerodynamic forces Journal of Wind Engineering and Industrial Aerodynamics 174 330–343