CFD Analysis on Sharp nose Bullet Train

Divyesh Variya, OpenFOAM team, FOSSEE Project, IIT Bombay.
Government Engineering college, Valsad

***Abstract*—This paper aims to explain the computational fluid dynamics (CFD) of a high-speed bullet train with the conventional sharp nose shape. kOmegaSST turbulence model is investigated. Drag and Lift coefficient of the Superior bullet train is determined. After study some additional features in shape is suggested for reducing aerodynamic drag coefficient. The implementation of mesh creation and flow calculation investigated. Three-dimensional effects are discussed to visulize velocity streamlines.**

***Keywords*—Sharp nose, Bullet train, OpenFOAM, simpleFoam.**

I. Introduction

The impact of high-speed train head shape on its aerodynamic performance is quite necessary. Particularly reducing the aerodynamic drag and the lift of the train remains a fundamental issue for high-speed trains. With the evolution of computing technology and computational fluid dynamic in the engineering field, CFD has been successfully applied to the design method of the high-speed train. Computational fluid dynamic optimization is performed to identify the aerodynamic shape which induced the least drag as well as generates the least aerodynamic noise. In the optimization process involve an extensive search at the design parameter over the entire design space and so require a huge fraction of design evaluation which is sometimes expensive as well as time-consuming. In this work evolutionary optimization algorithm utilizing adaptive surrogate pattern is implemented for both optimizations as well as least drag coefficient. Yao et al. proposed optimized nose shape of the train. This optimization reduced the aerodynamic drag at the train by 8.7%. cui et al. Succeed in approximately 20% of drag minimization.

Figure 1: sharp nose Bullet train in service

A. CFD Approach in this project.

These steps followed in the project:

1) Choosing an appropriate OpenFOAM solver.
2) Convert the geometry into a readable format as prescribed by OpenFOAM, i.e., .obj or .stl
3) Create an appropriate blockMeshDict file
4) Modify the snappyHexMeshDict file according to the geometry.
5) Control simulation.
6) Set the initial boundary conditions in the 0 folder.
7) Run the simpleFoam solver, either in serial or parallel computing.
8) Postprocess the results.

B. Geometry

The Geometry of sharp nose Bullet train is made in Siemens NX Software. Dimensions of geometry are the 12m length and 3m * 3m cross-section. Space between lowerWall of train and railway track is 0.4 m. It is taken from approved data accessible on articles.

Figure 2: Front view of the sharp nose bullet train

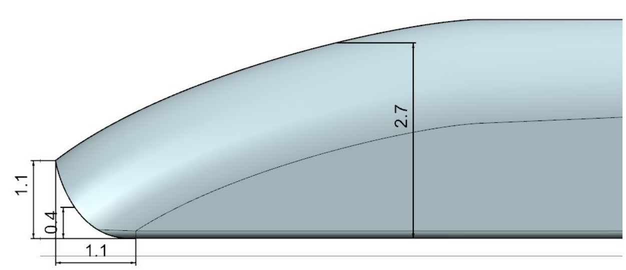

Figure 3: sharp nose bullet train CAD Model

C. Meshing

The meshing for this simulation was done using the OpenFOAM Mesh utilities. BlockMesh and SnappyHexMesh were used to mesh the model. BlockMesh defines a tunnel for bullet train full of air and snappyHexMesh extract the geometry and refines the mesh at CAD features. Results of checkMesh and Final mesh are:

Edge refinement level	6
surface Refinement level	(3 4)
No. of Layers	3

Table 1: snappyHexMeshDict

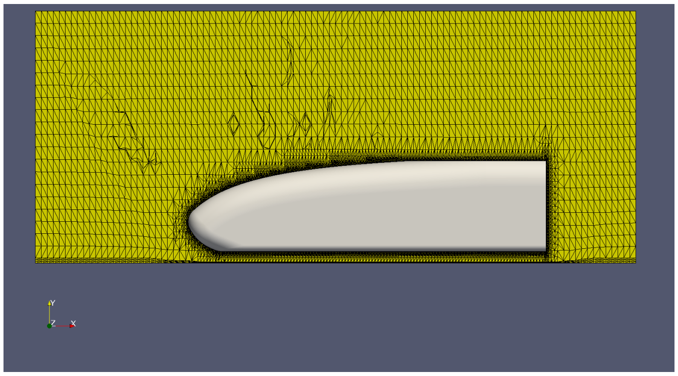

Figure 4: Cut-section of 3D Mesh

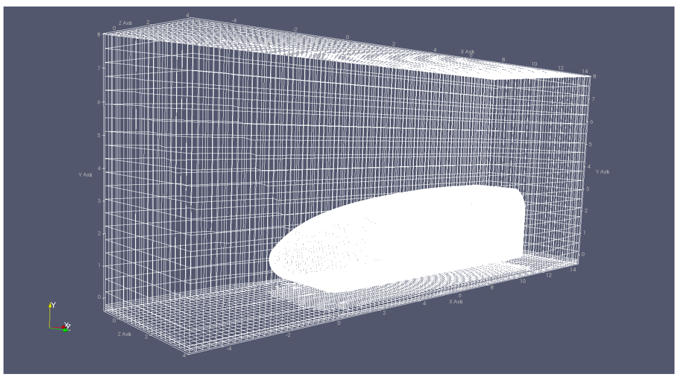

Figure 5: Wireframe 3D Mesh

Domain	(-5 -0.4 -1) (15 8 4)
points	1210680
faces	3176020
internal faces	3002224
cells	986211
faces per cell	6.26463
boundary patches	6
Max Skewness	3.26499
Max Aspect ratio	21.594

Table 2: checkMesh Results

II. ANALYSIS

The CFD analysis of the airflow over the sharp nose bullet train was done using the software OpenFOAM (v-5.0).

A. Boundary Conditions

Airflow enters with 97.22 m/s fixed velocity by inlet patch which is nearby 350 km/hr. Outlet kept inletOutlet as flow velocity 97.22 m/s. Pressure outlet has given a fixed value of 0 atm. front and backplane managed as a slip wall and corresponding velocity given to the lower wall that given at inlet. UpperWall velocity and pressure given as slip as air can leave from the domain. To the train, velocity is given as noSlip and zeroGradient presser applied.

The list of abbreviations used in the following table are:

1) FV: Fixed Value
2) ZG: Zero Gradient
3) IO: Inlet Outlet

Boundary	**U**	**p_rgh**
Inlet	FV(97.22)	ZG
Outlet	IO	FV(0)
frontAndback	slip	slip
lowerWall	FV	ZG
upperWall	slip	slip
train	noSlip	ZG

Table 3: Boundary conditions for U, p_rgh & alpha

B. Turbulence Model

The Reynold's Number was calculated using the freestream velocity and the length of the body. It came out to be $11.11 * 10^7$. The kOmegaSST turbulence model of OpenFOAM is used for this simulation. This model is a combination of $k-\omega$ and $k-\epsilon$ models. The initial values of k and Omega were calculated to be 0.24 and 1.78 respectively. Following are the formulae used in the calculations:

Reynolds Number,

$$Re = \frac{UL}{\nu} \tag{1}$$

where,
U - Maximum velocity of the object relative to the fluid,
L - Characteristic linear dimension,
ν - Kinematic viscosity

Turbulent Energy,

$$k = \frac{3}{2}(UI)^2 \tag{2}$$

where,
U - Mean Flow Velocity
I - Turbulent intensity

Specific Turbulent Dissipation Rate,

$$\omega = \frac{\sqrt{k}}{l} \tag{3}$$

where,
k - Turbulent Energy
l - Turbulent length Scale

Boundary	k	omega	nut
Inlet	FV(0.24)	FV(1.78)	calculated
Outlet	IO	IO	calculated
frontAndback	slip	slip	calculated
lowerWall	kqRWall Function	OmegaWall Function	nutkWall Function
upperWall	slip	slip	calculated
train	kqRWall Function	OmegaWall Function	nutkWall Function

Table 4: Boundary conditions for k, omega & nut

C. simpleFoam solver

Since we want to analyze steady-state turbulent flow for an incompressible fluid, we have used the simpleFoam solver. We do not need to solve the energy equation due to the incompressibility. The SIMPLE(Semi-Implicit Method for Pressure-Linked Equations) algorithm, which the simpleFoam solver is based upon, is solving the momentum equation (Equation 4) and the Poisson pressure equation (Equation 5).

$$(\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_j u_i}{\partial x_j}) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\mu \frac{\partial u_i}{\partial x_j}) + \rho f_i \tag{4}$$

$$\frac{\partial}{\partial x_i}(\frac{\partial p}{\partial x_i}) = -\frac{\partial}{\partial x_i}[\rho u_i u_j] \tag{5}$$

As OpenFOAM utilizes a collocated grid, Rhie-Chow interpolation is used for the pressure-velocity coupling. Following is the SIMPLE algorithm which is the basis of simpleFoam solver:

1) Set the boundary conditions.
2) Solve the discretized momentum equation to compute the intermediate velocity field.
3) Compute the mass fluxes at the cells faces.
4) Solve the pressure equation and apply under-relaxation.
5) Correct the mass fluxes at the cell faces.
6) Correct the velocities on the basis of the new pressure field.
7) Update the boundary conditions.
8) Repeat till convergence.

D. Force coefficients

For studying the airflow over Bullet train, we have to calculate the force coefficients or the aerodynamic coefficients, viz. Co-efficient of Lift(C_L), Co-efficient of Drag(C_D) and Co-efficient of Moment(C_M). A force coefficient function was called in the controlDict file. It was defined as follows:

```
forceCoeffs1
{
type                forceCoeffs;
functionObjectLibs  ( "libforces.so" );
outputControl       timeStep;
timeInterval        1;
log                 yes;
patches             (train);
rhoName             rhoInf;
rhoInf              1;
liftDir             (0 0 1);
dragDir             (1 0 0);
CofR                (6 1.5 1.5);
pitchAxis           (0 1 0);
magUInf             97.22;
lRef                12;
Aref                3.8173;
}
```

III. Results and Discussion

The C_L, C_D and C_M values are key figures in aerodynamics studies. We successfully obtained their values as C_L = -0.5479, C_D = 1.3724, C_M = 662.79 The plot of the force co-efficients against the simulation time is as follows:

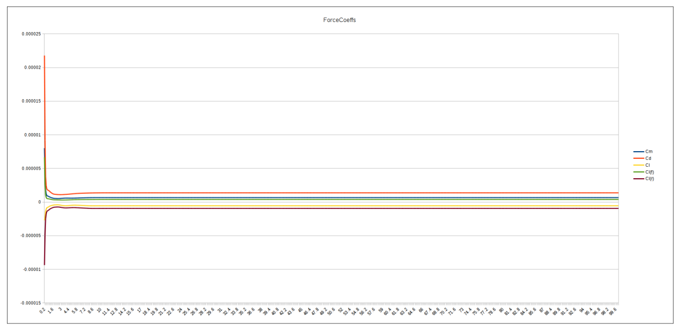

Figure 6: Force coefficients versus Simulation Time

We can visualize the pressure variation over the bullet train from the pressure contour(Figure 7). As can be seen, the pressure is maximum on the front nose of the body. The value of the maximum pressure is $7.2 * 10^3$.

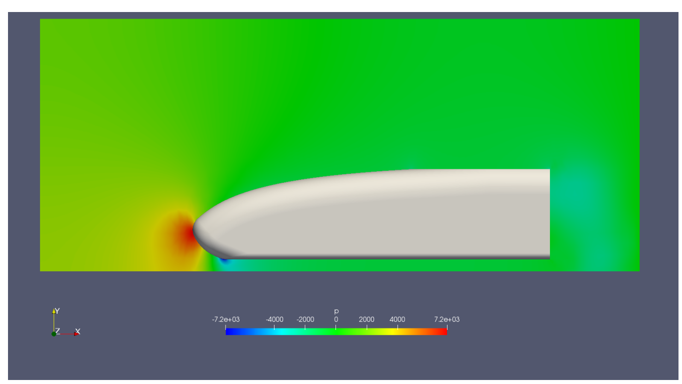

Figure 7: Variation of pressure on sharp nose bullet train

We can also visualize the streamlines due to the flow using the Stream Tracer option in OpenFOAM(Figure 8). Vortices from the trailing edges of the body merge into two counter rotating vortices(one vortex on each side of the symmetry plane).

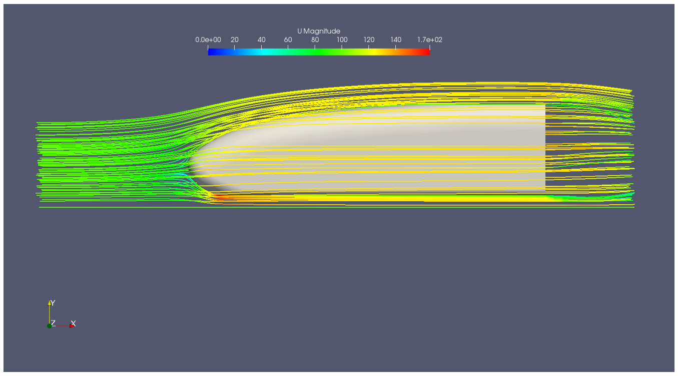

Figure 8a: Streamlines

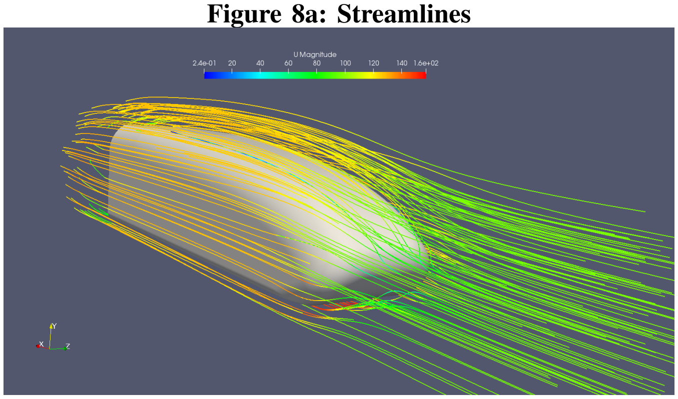

Figure 8b: Streamlines 3D

Figure 9 represents a plot of initial residuals against the timesteps. As we can see, the residuals are decreasing with the increase in timesteps on an average.

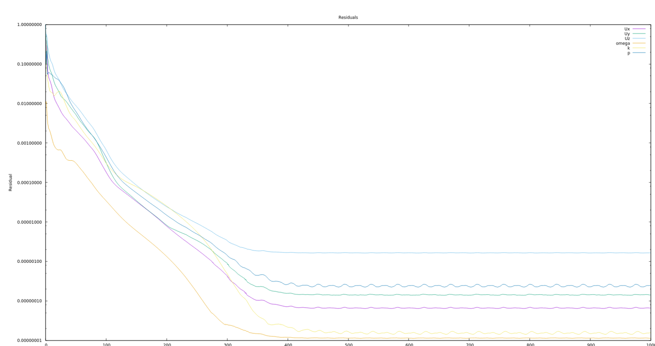

Figure 9: Residuals vs Timesteps

To conclude, the major features of the flow are captured very well by the kOmegaSST turbulence model.

IV. CONCLUSION

From the results, it is clarified that the drag is quite important in the sharp nose shape so, it is required to modify the shape to diminish drag. For minimization of drag, a very traditional procedure is to compose an aerodynamic curved surface at the frontal region of the head. From the streamlines, it is realized that the noise in the velocity is considerably stable but with some supplementary feature both the points can be managed.

ACKNOWLEDGMENT

The writer would like to thank Government Engineering college, Valsad & Indian Institute of technology, Bombay for making him capable to do such wondrous project. He would also like to extend his gratitude to Mr. Rohit Panday for providing such a nice support and guidance.

REFERENCES

[1] Xu G, Liang X, Yao S, Chen D, Li Z (2017) Multi- Objective Aerodynamic Optimization of the Streamlined Shape of High-Speed Trains Based on the Kriging Model. PLoS ONE 12(1): e0170803. doi: 10.1371/journal.pone.0170803

[2] "Aerodynamic optimization of railway motor coaches" F. Monnoyer, E. Lorriaux, N. Bourabaa Laboratoire de Mécanique et d'Énergétique, University of Valenciennes, France.

[3] "CFD Study of the Flow around a High-Speed Train" Florian GUILLOU. Kungliga Tekniska Högskolan, Stockholm

[4] International Journal of sharp nose Engineering Research (IJMER) `www.ijmer.com` Vol.3, Issue.2, March-April. 2013 pp-1161-1166 ISSN: 2249-6645

[5] Journal of sharp nose Transportation Volume 19, Number 1, March 2011, Page 1-6 Journal homepage: `jmt.swjtu.edu.cn` DOI: 10.1007/BF03325733

[6] Tanghong Liu, Zhengwei Chen, Xisai Zhou & Jie Zhang (2018) A CFD analysis of the aerodynamics of a high-speed train passing through a windbreak transition under crosswind, Engineering Applications of Computational Fluid Mechanics, 12:1, 137-151, DOI: 10.1080/19942060.2017.1360211

[7] Xinwen YANG, Jielong JIN , Guangtian SHI "Preliminary study on streamlined design of longitudinal profile of high-speed train head shape" 13th COTA International Conference of Transportation Professionals (CICTP 2013)

[8] The SIMPLE Algorithm `https://openfoamwiki.net/index.php/OpenFOAM_guide/The_ SIMPLE_ algorithm_ in_ OpenFOAM`

Divyesh Variya (M'97) received the B.E. degree in mechanical engineering from the gujarat technological univercity, in 2018 and also work as intern under **Prof. Shivasubramanian Gopalakrishnan** for FOSSEE (Free and Opensource Software for Education) Project on OpenFOAM in Indian Institute of Technology, Bombay. His research interests include all CAE projects with linear/non-linear structural analysis, computational fluid dynamics, dynamic robotics analysis, failure analysis and prevention, and design development.

Contact:
+91 7777908833
`divyeshvariya7@gmail.com`