International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

OPTIMIZED DESIGN OF PASSENGER CAR & ANALYSIS OF DRAG FORCE USING CFD

G.SUDHAKAR *gsudhaarmgm@gmail.com*
K.BHUVANA CHANDRA *CHNADRA.kurra22@gmail.com*
A.NARASIMHAREDDY *anarasimhareddy111@gmail.com*
D.BHARATH KUMAR REDDY *dbharathkumarreddy111@gmail.com*

Department of Mechanical Engineering, Dr.K.V.Subba Reddy Institute of Technology, Kurnool-518218 (AP), India

Abstract

Volvo buses travel about 250 to 350 km in a trip and usually are of sleeper coach mode. The exterior styling, drag force reduction and aerodynamically efficient design for reduced fuel consumption are the three essential factors for a successful operation in the competitive world. The bus body building companies prioritizes the exterior looks of the bus and ignores the aerodynamic aspect. Scientific design of sleepers for increased comfort of the passengers is seldom seen. The overall aim of this project was to redesign a Volvo bus with enhanced exterior styling, reduced aerodynamic drag and increased comfort for the passengers.

An operating Volvo bus was benchmarked and analyzed for styling, aerodynamic performance and comfort.Fluent, a commercial CFD code was used to evaluate the aerodynamic performance. Principles of product design were used to analyze the styling and comfort.

I. INTRODUCTION

Buses are used as means for transporting large amount of people from one place to other. All the states governments are having its own intercity bus fleet in India which provides mobility for the people at a reasonable cost. Huge numbers of private bus firms are also in operation and are efficient in reducing the dependency on trains. Indian road conditions are significantly improved for the past 10 years and intercity bus travel time is reduced as they can travel with high speeds. In order to keep a low operating cost these buses have to deliver high efficiency at these speeds. Rising fuel prices and government regulations, force the vehicle manufactures and operators to produce and operate fuel efficient buses. The power generated in the engine is mainly used to overcome the rolling resistance, aerodynamic drag and climbing resistance. Out of these three components aerodynamic drag increases with respect to the vehicle speed. At high speeds at about 100 Km/hr the drag force exceeds the power spend on overcoming the rolling resistance. So reducing the aerodynamic drag is of prime importance to achieve fuel efficiency. Vehicle aerodynamics deals with the study of forces acting on a vehicle body when it moves through air.

Drag and lift are the two main phenomena observed on the vehicle body due to the effect of the wind. About 90% of drag is due to the pressure difference created between the various areas of the vehicle. The automotive industry is a large user of commercial CFD packages. The advantage of CFD results in better designs and reduced time for the automotive manufacturers. CFD is not only used to improve the aerodynamics of vehicles, but also for the optimization of domains such as engine cooling, brake cooling, airbags, lighting and fuel system. During the development of new vehicles, understanding the flow phenomena and how aerodynamic forces are influenced by changes in body shape are very important. A large variety of complex flow properties such as three-dimensional turbulent boundary layer on the body surfaces, longitudinal vertices induced by three-dimensional separation, recirculation flows caused by separation and the ground plane boundary layer and their interaction are important to be understood. However, using CFD is a good way for designers to obtain results in a shorter time.

1.2 CONCEPT OF THE PROJECT

Drag force acting on the vehicle depends on frontal projected area and the coefficient of drag value of the vehicle. Any reduction in these values will directly reduce drag force experienced by the vehicle. Frontal projected area of the intercity bus is decided by the interior packaging of the bus. Coefficient of drag value is determined by the shape of the vehicle. These two factors influence the exterior styling of the vehicle. Exterior styling of the vehicle is important due to the fact that the vehicle has to attract customers. The vehicle should project its performance and comfort capabilities through its exterior design. Finding harmony with the aerodynamic requirements and customer oriented styling will lead to a successful vehicle with low fuel consumption.

This research is aimed to deliver an aerodynamically improved bus design with user oriented exterior styling. The popular Volvo 9400 bus was evaluated for its aerodynamic performance and guidelines for better aerodynamics were collected from literature survey. Based on these guidelines and user study concepts were generated. The model was analyzed using fluent and improvements in drag values were predicted.

II. BACKGROUND INFORMATION

AIM OF THE PROJECT

Decreasing the fuel consumption of road vehicles, due to environmental and selling arguments reasons, concerns bus manufacturers. Consequently the improvement of the aerodynamics of bus shapes, more precisely the reduction of their drag coefficient, becomes one of the main topics of the automotive research sectors. Designing a vehicle with a minimized Drag resistance provides economical and performance advantages. Decreased resistance to forward motion allows higher speeds for the same power output, or lower power output for the same speeds.

The main aim for reducing drag resistance is:

- Fuel consumption reduction and
- Performance increasing

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

1.4 SCOPE OF THE PROJECT

The results of the redesigned exterior body showed a reduction of C_d from 0.53 to 0.29 and overall aerodynamic drag reduction by 60% due to combined effect of reduced C_d and frontal area. The redesigned interior was found to be at the satisfaction of commuters.

1.5 STEPS IN THE PROJECT

- Plan
- Cad design
- Prototype
- Evaluation
- Analysis
- Final design
- Production

1.6 COMPUTATIONAL FLUID DYNAMICS

1.6.1 INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS

"Computational fluid dynamics" (CFD) is the use of computers and numerical techniques to solve problems involving fluid flow. CFD has been successfully applied in a huge number of areas, including many of interest to civil engineers (highlighted below). Examples include:

- Aerodynamics of aircraft and automobiles;
- Hydrodynamics of ships;
- Engine flows – IC engines and jet engines;
- Turbo machinery – pumps and turbines;
- Heat Transfer – heating and cooling systems;
- Combustion;
- Process Engineering – mixing and reacting chemicals;

1.6.2 BASIC PRINCIPLES OF CFD

The approximation of a continuously-varying quantity in terms of values at a finite number of points is called discretization.

The fundamental elements of any CFD simulation are:

(1) The **fluid continuum is discretized**, i.e. field variables (u, v, w, p, etc) are approximated by their values at a finite number of nodes.

(2) The **equations of motion are discretized, i.e.** approximated in terms of values at nodes:

Differential or integral equations → algebraic equations
(Continuum) (Discrete)

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

(3) The **system of algebraic equations is solved** to give values at the nodes.

1.6.3 WORKING OF CFD CODE
1.6.3.1 PRE-PROCESSOR

Input of a flow problem to a CFD program:

- Definition of the geometry of the region of interest: the computational Domain.
- Grid generation: the sub-division of the domain into a number of smaller, Non overlapping sub-domains, i.e., a grid (or mesh) of cells (or control Volumes or elements)
- Selection of the physical and chemical phenomena that needs to be modeled.
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary.

The accuracy of a CFD solution is governed by the number of cells in the grid. Optimal meshes are often non-uniform: finer in areas where large variations occurs from point to point and coarser in regions with relatively little change. Over 50% of the time spent in industry on a CFD project is devoted to the definition of the domain geometry and grid generation.

1.6.3.2 SOLVER

Three distinct streams of numerical solution techniques:

a) Finite Difference Method:

- Describe the unknown φ of the flow problem by means of point samples at the node points of a grid of co-ordinate lines.
- Truncated Taylor series expansions are often used to generate finite difference approximations of derivatives of φ in terms of point samples of φ at each grid point and its immediate neighbors.
- The derivatives appearing in the governing equations are replaced by finite differences yielding an algebraic equation for the values of φ at each grid point.

b) Finite Volume Method:

- It is a special finite difference formulation and is central to main commercial CFD codes. The numerical algorithms:
- Formal integration of the governing equations of fluid over all the (finite) control volumes of the solution domain.
- Convert the integral equations into a system of algebraic equations.
- Solution of the algebraic equations by an iterative method.
- The most popular solution procedures are the TDMA line-by-line solver of the algebraic equations and SIMPLE algorithm to ensure correct linkage between pressure and velocity.

c) Finite Element Method:

- Use simple piecewise functions (e.g. linear or quadratic) valid on elements to describe the local variation of unknown flow variable φ.
- The governing equation is precisely satisfied by the exact solution φ.
- If the piecewise approximating functions for φ are substituted into the equation it will not hold exactly and a residual is defined to measure the errors.
- The residuals are minimized in some sense by multiplying them by a set of weighting functions and integrating.
- A set of algebraic equations for the unknown coefficients of the approximating functions.
- functions and integrating.
- A set of algebraic equations for the unknown coefficients of the approximating functions.

1.6.3.3 POST-PROCESSOR

The raw output of the solver is a set of numbers corresponding to the values of each field variable (*u*, *v*, *w*, *p*, etc.) at each point of the mesh. This huge quantity of numbers must be reduced to some meaningful subset and, usually, manipulated further to obtain the desired predictive quantities. For example, a set of surface pressures and cell-face areas is required to compute a drag coefficient or a set of velocities and areas to determine a flow rate. Commercial packages often provide post-processing facilities to plot , interpolate or simply extract quantities from the output dataset. A key component of post-processing is being able to visualize complex flows – either to indicate important features of the flow or, unfortunately, sometimes to establish why a calculation is diverging.

III. MAIN RESULT

3. AERODYNAMIC DEVICES TO REDUCE DRAG OF SEMITRAILER

An experimental study of aerodynamic optimization of semitrailer unit was performed. Attention has been paid to the reduction of base-drag by means of rear-end tapering panels, reduction of water spray behind the vehicle and reduction of semitrailer-under-body drag using side fairings of various forms. Experiments have been performed with a scaled 1:15 model in a wind-tunnel. Comparison of some results with measurements taken with the vehicle in natural size in 1:1 wind-tunnel is mentioned. Next to force measurements, PIV technique has been employed for flow-field analysis in the wake downstream the semitrailer. Results of PIV-measurements are discussed in comparison with numerical simulation of flow around the semitrailer in the case of semitrailer rear-end tapering. Tractor-semitrailer units play nowadays a decisive role in goods transportation, especially thanks to the flexibility and quickness which are the virtues railroad transportation is not able to offer. On the other hand, road delivery of one ton to the distance of one km is in comparison with railroad more energy demanding, though. The reduction of fuel consumption of heavy road vehicles is therefore in connection with still increasing number of cars getting to be a very important issue. At the end of 20th century it has been estimated, that keeping the annual growth of oil consumption the world's known resources of oil would last for some more 70 years only. Aerodynamic drag of heavy road vehicles significantly participates on the overall energy efficiency of vehicles and thus represents a challenge for its optimization. If we consider a tractor-semitrailer unit of 40t weight and a drag coefficient *cD* of 0,6 (coefficient of rolling resistance *fr* = 0,0055), the aerodynamic drag-part of the overall driving resistance exceeds 50% at the speed of about 90 km/h. Considerable effort since the 80s of the last century has already been invested to the aerodynamic optimization of tractor units. Compared to this, semitrailers are even nowadays having aerodynamically still very ineffective form.

CONSTRUCTION

The Volvo 9400 intercity bus was selected as bench marked model. It is the latest model in the Indian market and is very popular in the segment. Engineering parameters of this model was kept as same for the new bus design and was selected as the baseline for studying the aerodynamic performances. This vehicle model was used to understand the flow behavior Pressure distribution, Coefficient of drag value, Contribution of different parts, Drag force acting at different speeds, Flow separation and pressure stagnation areas. Three dimensional model of the baseline model was created using Alias studio tools and Catia as shown in fig 3. Small details and gaps in the vehicle body were eliminated as the purpose of the analysis was to understand the overall aerodynamic performance of the basic bus shape

3.1 DEVELOPMENT OF 3D MODEL

I) Details of Semi Sleeper Coach used for Analysis

The type of bus used for study is VOLVO, MJT Travels with the Chassis length of 11 m, width of 2.6 m, and height of 2.2 m. In this experiment, we obtain data of Pressure and Velocity of Air flow around the bus from front to rear end when the object is subjected to moving. Our experimental data is to be shared as a standard model to assess the environment within Automobiles.

II) Details of Sleeper Coach used for Analysis

The type of coach used here is VOLVO 9400, MJT Sleeper Coach with the Chassis length of 12m, width of 2.6m, and height of 2.3m. In this experiment, we obtain data of Pressure and Velocity of Air flow around the bus from front to rear end when the object is subjected to moving. Our experimental data is to be shared as a standard model to assess the environment within Automobiles.

The 3D model was developed using the dimensions shown in the Figure3.1 using Pro e Wildfire 4.0

Fig.3.1. Model of Existing Design

Parameters	Regulations
Width of the bus	Shall not exceed 2.6 m
Length of the bus	Maximum 12 meters for transport vehicle with rigid frame having two or more axles,
Gangway	Minimum of 1800 mm height and 300 mm wide
Service doors	Minimum 1
Width of door	Minimum 650 mm
Height of service door	Minimum 1650 mm
Width of windows	Minimum 550 mm (sliding type except for ACX)
Emergency exit	2 numbers (1 at front half opposite to service door next one at rear with area not less than 4000 cm^2)
Height of first step	425 mm maximum
Height of second steps	350 mm maximum
Intrusion above seat	100 mm at height 1350 from floor
Wheel arch intrusion	200 mm from the seat front

Table. 3.1. Design Parameters

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

3.2 IMPORTING THE 3D MODEL INTO GAMBIT

The starting point for the meshing process is the import of some type of surface data in the form of either a mesh or geometric description of some kind. The Graphical User Interface while opening GAMBIT is shown. The method of importing 3D model is shown.

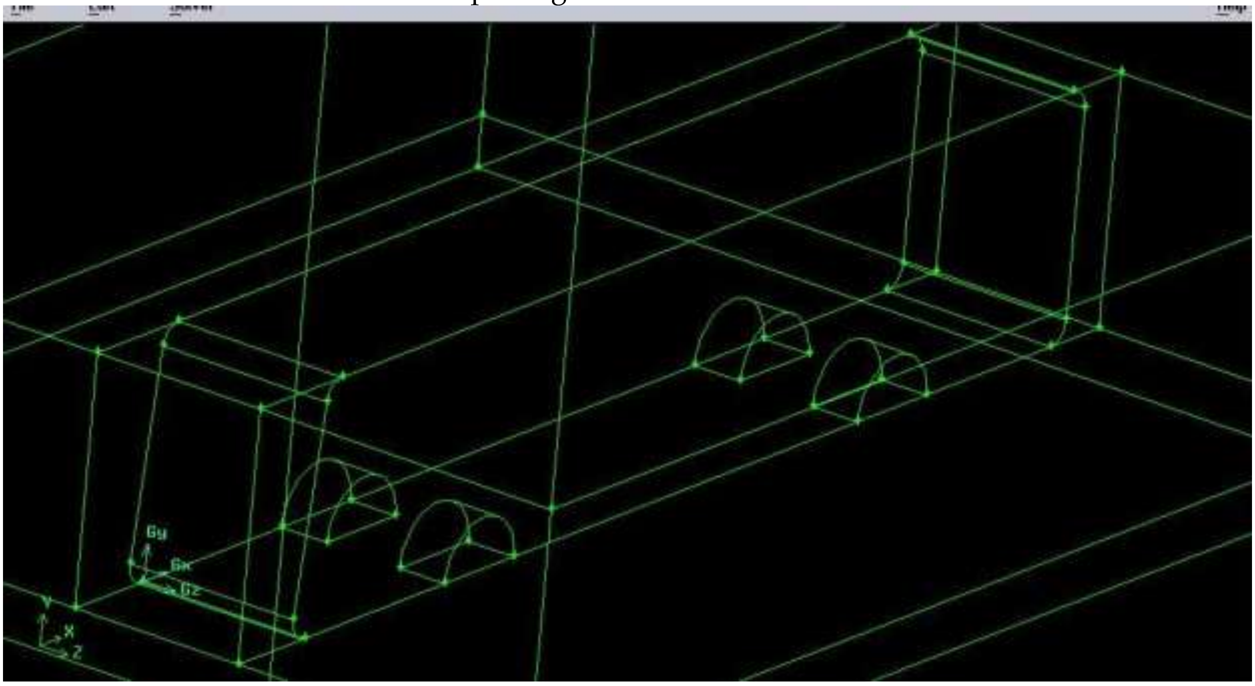


Fig. 3.2 Existing design in GAMBIT

3.3 GENERATING MESH

GAMBIT mesh generation tool offer the capability to parametrically create meshes from geometry in numerous formats.

3.3.1 MESHING MODULES

- **Hexagonal Mesh**

  The semi-automated meshing modules present rapid generation of multi-block structured or unstructured hexahedral volume meshes. Hexa represents a new approach to grid generation where the operation performed experts are automated and made available at the touch of button.

- **Tetra Mesh**

  A tetra mesh is the collection of meshing models that are used to generate the surface and volume mesh for the input geometry representing the individual regions to be used for the simulation. A mesh node has its own properties and pop-up menu.

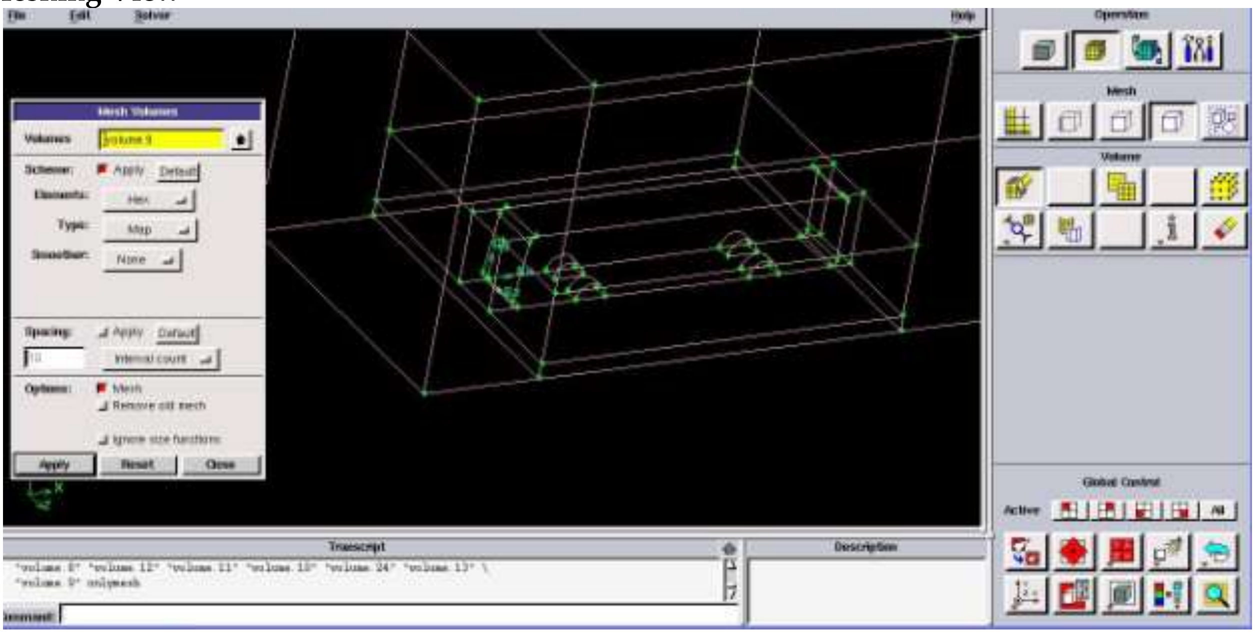
- **Hybrid Mesh**

  This mesh can generate meshing models which are very irregular in shape and size. All

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

the irregular features can be meshed with this type of Mesh.

Meshing View



IV. Results

Static pressure contours of the bus model at vehicle speed of 100 km/hr are shown in fig 11. The plot shows considerable reduction in pressure stagnation area in the front of the vehicle. The static pressure value also reduced compared to the baseline model. Static pressure plot at the rear end of the vehicle reveals an increase in pressure. The pressure difference between the front and the rear area was reduced which reduces the pressure drag acting on the body.

Large numbers of vortices were generated and flow separation was observed at the rear of the baseline model fig 12. In the new design these were brought to a minimum value. It is evident from the fig 13 that the low velocity area behind the new bus design is considerably reduced due to the effect of roof tapering, roof lowering boat tailing and diffuser at the rear. Flow is directed at the rear to minimize the low pressure area behind the vehicle. No flow separation is occurring at the front of the vehicle as due to the improvement in the front corner radii and elimination of the rear view mirror

V. CONCLUSION

A detailed investigation of the present bus in the field of styling and aerodynamics was carried out. The results of these investigations were used to come up with an aerodynamically efficient user friendly intercity bus design. The flowing conclusions are drawn from the studies.

- The present high floor sleeper coach was modified to a low floor version with substantially reduced aerodynamic drag.
- The drag coefficient of 0.53 of present bus was found to be reduced to 0.29 in the modified design.
- The exterior was redesigned giving emphasis to both aerodynamics and aesthetics.
- The interior was also modified to improve the comfort of the commuters.
- The proposed concept was well received by the commuters.
- Comparatively New Design Proves to be less Drag Values compared to the Existing Design.

REFERENCES

[1] Dindayal Mahto, Danish Ali Khan and Dilip Kumar Yadav, "Security Analysis of Elliptic Curve Cryptography and RSA", Proceedings of the World Congress on Engineering 2016 Vol I, WCE 2016, June 29 - July 1, 2016.

[2] Geetha G and Padmaja Jain, "Implementation of Matrix based Mapping Method Using Elliptic Curve Cryptography", International Journal of Computer Applications Technology and Research Vol. 3, Issue 5, 312 - 317, 2014.

[3] Naresh S. Badve, "Cryptography using Elliptic Curve with Matrix Scrambling", Journal for Research, Vol. 01, Issue 01, March 2015.

[4] Fatima Amounas, "On enhancement of Elliptic Curve Encryption of Amazigh Text using Graph Theory", International journal of Computer Science & Network Solutions, Vol. 4, No.3, pp. 1-10, 2016.

[5] D. Sravana Kumar, Ch. Suneetha and A. Chandrasekhar, "Encryption of Data Using Elliptic Curve Over Finite Fields", International Journal of Distributed and Parallel Systems, vol. 3, no. 1, 2012.

[6] William Stallings, "Cryptography and network security principles and practice", Prentice Hall, 5th Edition, 2011.

[7] Padma Bh, D.Chandravathi, P.Prapoorna Roja, "Encoding And Decoding of a Message in the Implementation of Elliptic Curve Cryptography using Koblitz's Method", International Journal on Computer Science and Engineering, Vol. 02, No. 05, pp. 1904-1907, 2010.

[8] O.Srinivasa Rao and S. Pallam Setty, "Efficient Mapping methods for Elliptic Curve Cryptosystems", International Journal of Engineering Science and Technology, Vol. 2(8), pp.3651-3656, 2010.

[9] B. Ravikumar, G. Naga Lakshmi, T. Surendra, A. Chandra Sekhar. "A Study on group theoretic Elliptic curves - An m implementation to Cryptography", International Journal of Science and Advanced Technology, Vol 1, No 6, 2011.

[10] V.Kamalakannana and S.Tamilselvan, "Security Enhancement of Text Message Based on Matrix Approach Using Elliptical Curve Cryptosystem", 2nd International Conference on Nanomaterials and Technologies, Procedia Materials Science 10, pp. 489-496, 2015.

[11] Vishal Kumar, Ratnesh, Mashud A. And Monjul Saika, "Multiple Encryption using ECC and it's Time Complexity Analysis, International Journal of Computer Engineering In Research Trends, Vol. 3, Issue 11, pp. 568-572, 2016.

[12] Ameur M., Bouhjar A., Boukhris F., Boukous A., Boumalk A., Elmedlaoui M., Iazzi E., Souifi

H. Initiation à la langue amazighe. IRCAM, Rabat, Maroc, 2004.
[13] Lazzi E., Outahajala M. Amazigh Data Base. In Proceedings of HLT & NLP Workshop within the Arabic world: Arabic language and local languages processing status updates and prospects, Marrakech, Morocco, pp. 36-39, 2008.
[14] Andries P. Unicode 5.0 en pratique, Codage des caractères et internationalisation des logiciels et des documents. Dunod, France, Collection InfoPro. 2008.
[15] Fatima Amounas and El Hassan El Kinani, "Fast Mapping Method based on Matrix Approach for Elliptic Curve Cryptography", International Journal of Information & Network Security , Vol.1, No.2, pp. 54-59, 2012.
[16] Geetha G, Padmaja Jain ,"Implementation of Matrix based Mapping Method Using Elliptic Curve Cryptography",International Journal of Computer Applications Technology and Research Volume 3– Issue 5, 312 - 317, 2014.
[17] R. Balamurugan, V. Kamalakannan, D. Rahul Ganth and S. Tamilselvan, "Enhancing Security in Text Messages Using Matrix based Mapping and ElGamal Method in Elliptic Curve Cryptography", International Conference on Contemporary Computing and Informatics, IEEE, pp.103-106, 2014.