DESIGN OF AUTOMOBILE FRONT BUMPER FOR COLLISION ENERGY ATTENUATION

1. Bharat P. Patil, 2. Prashant N. Ulhe

1. M. E. Student, Mechanical Dept. SSBT's COET, Bambhori, Jalgaon, Maharashtra, India.
2. Assistant Professor, Mechanical Dept. SSBT's COET, Bambhori, Jalgaon, Maharashtra, India.

***

Abstract – *While designing the bumper in the automobile considering an impact during collision is important. The design must improve the ability to absorb more impact load, reducing stress, and increase the protection of the front car component. The method have been employed was study the front bumper system, design using CATIA software, and analyze the alternative front bumper material using ANSYS software.*

The aim of this work is to reduce the degree of damage to passengers, and vehicle's body caused by vehicle collisions. Crash phenomena involving road vehicle were studied for the purpose of developing an impact attenuation design that can withstand specific speed.

Key Words: *front bumper;composites;collision;ANSYS.*

I. INTRODUCTION

In an automobile's structure bumper is the front-most or rear-most part. It is used to sustain an impact and reduce or prevent damage to the passengers, vehicle's frame or safety systems of car during low speed collision. Generally steel and Aluminum has been the dominant material for vehicle body in old days. But, now a days the front of car bumper is made by composite that suit for the commercial front bumper (1). The bumper is use to cover the bumper beam that connected to the car's frame.

Even it could prevent a small damage for the car, the front bumper important as an aesthetical value. Composite are the best materials for bumper which are aesthetically pleasant, lighter weight and offer many more substantial advantages (2).

II. PROBLEM STATEMENT

Now a days accidents are frequent and cause damage to a vehicle's body and became threat to human life. Hence crashworthiness of automobile bumper is essential. Also the cost for replacing bumper is quite expensive especially if surrounding area or part also damaged.

Customer also blame to the manufacture that the bumper easily gets damaged although the collision was slow. The material of the bumper should be analyzed to find the alternative material that can improve the crashworthiness, also have toughness.

III. LITERATURE SURVEY

Until now many research and experiment has done on bumper design. Ramin Hosseinzadeh et al. [1] (2004), has studied on parametric research of automotive composite bumper beams subjected to low-velocity impacts and concluded that fuel efficiency and emission gas regulations are the main causes for reducing the weight of passengers' cars by the use of composite material structures. Hence the composite will probably the best material to produce an effective bumper.

As per Marzbanrad J. M. et.al [2] the important parameters considered for design and analysis of an automotive front bumper beam to improve the crashworthiness design in low-velocity impact are material, thickness, shape and impact condition.

Gintautas Dundulis et al. [3], Static analytical and experimental research of shock absorber to safe guard the nuclear fuel assemblies. Ping Yang et al. [4], Design, test and modeling evaluation approach of a novel Si-oil shock absorber for protection of electronic equipment in moving vehicles.

In their study M. M. Davoodi et al. [5] studied on conceptual design of polymer composite automotive bumper energy absorber. It was observed that, major injury due to impact velocity of around 20-30 km/h was affected to the knee ligament. They proposed fiber reinforced epoxy composite material for bumper as a pedestrian energy absorber. The energy absorption capacity was sufficient for pedestrian impact and it could possible to use as substitute for the existing materials for low impact collision.

IV. SOLID MODELING

A solid model is better because for an object it provides more complete representation than its surface model. It provides more topological information in addition to the geometrical information which helps to represent the solid unambiguously. There are different software that are used for generating these solid models like CATIA and Pro-E. In this project the frame is modeled by using CATIA V5. All the parts that are required for constructing the frame are modeled in part module by using different commands like extrude, loft, fillet, shell etc. as shown in Fig. 1.

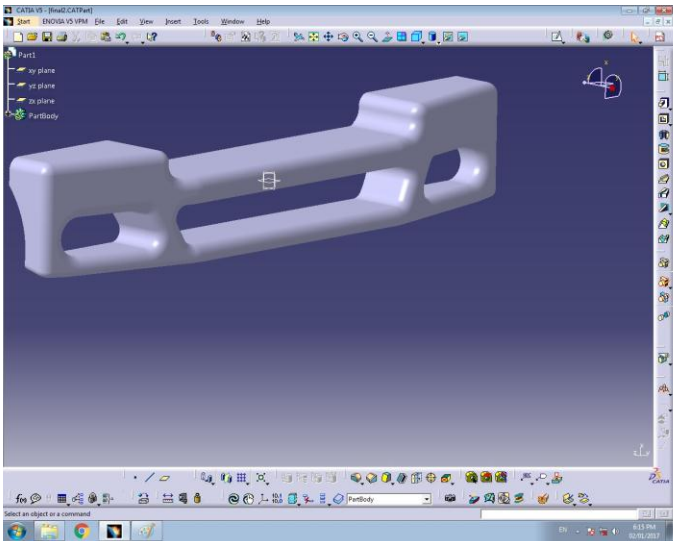

Figure - 1: Modeling of components in CATIA V5

V. IMPACT ANALYSIS OF BUMPER

A. Organization Of The Ansys Program

For analysis of bumper we are using ANSYS software. The begin level acts as a gateway in to and out of the ANSYS program. It is also used for certain global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files. When we first enter the program, we are at the begin level.

At the processor level, several processors are available; each processor is a set of functions that perform a specific analysis task. For example, the general preprocessor is where we build the model, the solution processor is where we apply loads and obtain the solution, and the general postprocessor is where we evaluate the results and obtain the solution. An additional post processor enables us to evaluate solution results at specific points in the model as a function of time.

B. Performing typical ANSYS Analysis

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, dynamic analysis. In this project we have done dynamic analysis of bumper, using bumper model designed in CATIA software. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analysis for different engineering disciplines. A typical ANSYS analysis has three distinct steps:

The following table shows the brief description of steps followed in each phase.

Table - 1. Steps in ANSYS analysis

PRE-PROCESSOR	SOLUTION PROCESSOR	POST-PROCESSOR
Assigning element type	Analysis definition	Read results
Geometry definition	Constant definition	Plot results on graphs
Assigning real constants	Load definition	View animated results
Material definition	Solve	
Mesh generation		
Model display		

The solid model is used in ANSYS software to simulate impact analysis as shown in fig. 2. It is considered that bumper is striking against fixed support. E.g. Pole. At a velocity of 10 m/s.

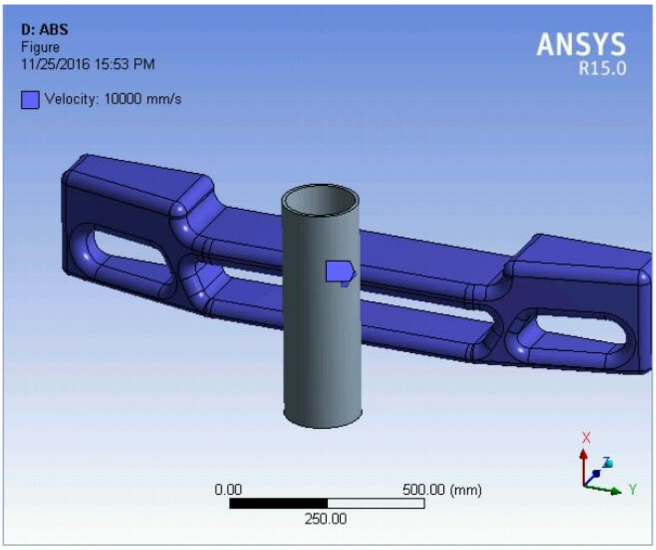

Figure - 2: Simulation of Impact phenomenon in ANSYS

In this way impact analysis of bumper has been done using following materials.

D. Material properties

The suitable materials are selected on the basis of previous research data which are as follows.

Table - 2. ABS Plastic Material Properties

Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa	Density
2300	0.33	1916.7	884.62	1040 Kg/m^3

Table - 3. Carbon fiber reinforced Plastic Material Properties

Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa	Density
350000	0.3	291670	134620	1800 kg/m^3

Table - 4. Glass fiber reinforced Plastic Material Properties

Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa	Density
60000	0.3	50000	23077	1200 kg/m^3

VI. RESULTS

After doing Impact analysis of bumper in ANSYS software we got following results –

A. ANSYS Results for ABS Plastic

Considering Bumper material as Acrylonitrile Butadiene Styrene Plastic (ABS Plastic) and pole material as Structural Steel as shown in Fig. 3

Equivalent stress in ABS Plastic

Maximum Equivalent stress = 39.18 MPa

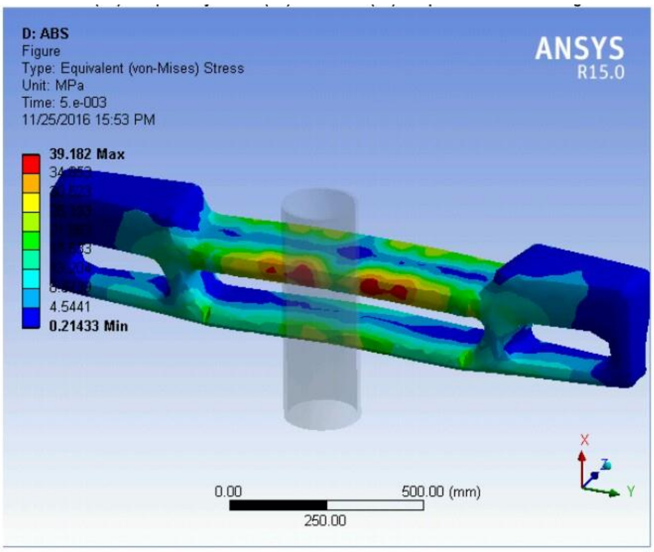

Figure - 3: Equivalent Stress Distribution in ABS Plastic

Deformation in ABS Plastic

Maximum Total Deformation = 38.158 mm

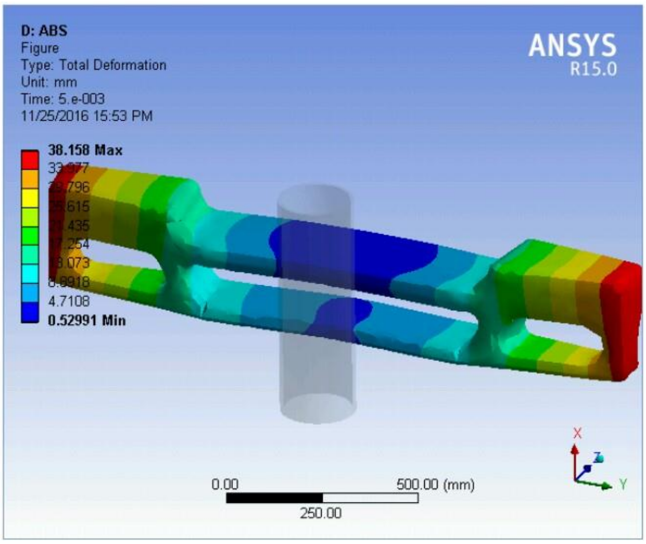

Figure - 4: Deformation in ABS Plastic

B. ANSYS Results for carbon fiber reinforced composites

Considering Bumper material as Carbon fiber reinforced composite and pole material as Structural Steel as shown in Fig. 5

Equivalent stress in Carbon fiber reinforced Plastic

Maximum Equivalent stress = 256.21 MPa

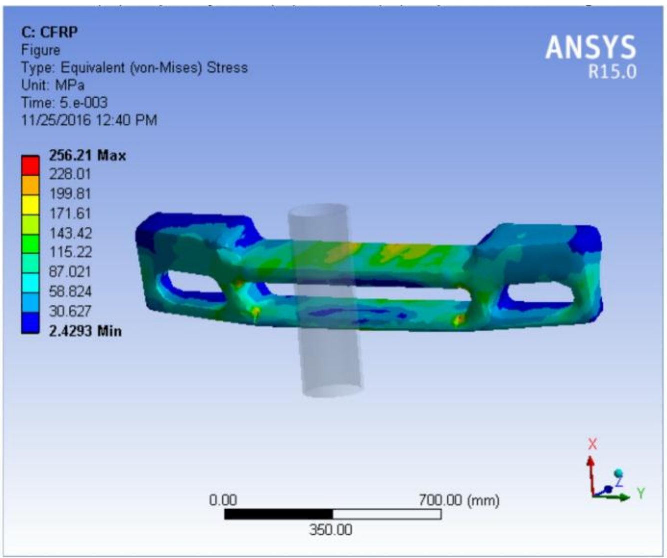

Figure - 5: Equivalent Stress Distribution in Carbon fiber reinforced Plastic

Deformation in Carbon fiber reinforced Plastic

Maximum Total Deformation = 24.842 mm

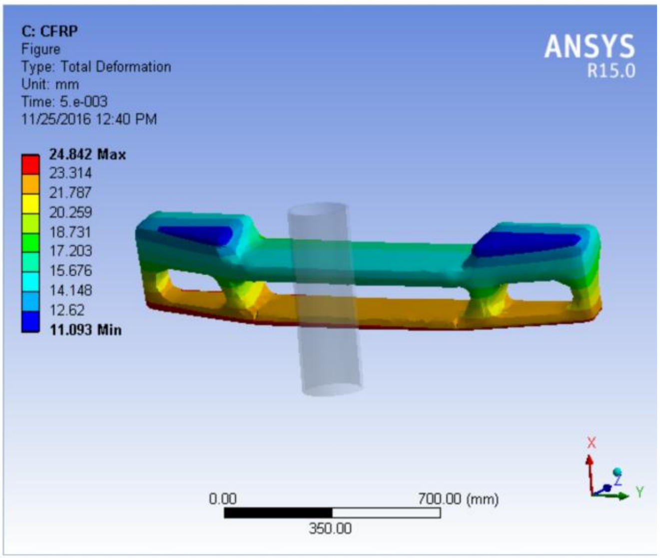

Figure - 6: Deformation in Bumper with Carbon fiber reinforced Plastic

C. ANSYS Results for Glass fiber reinforced Plastic

Considering Bumper material as Glass fiber reinforced composite and pole material as Structural Steel as shown in Fig. 7

Equivalent stress in Glass fiber reinforced Plastic

Maximum Equivalent stress = 152.4 MPa

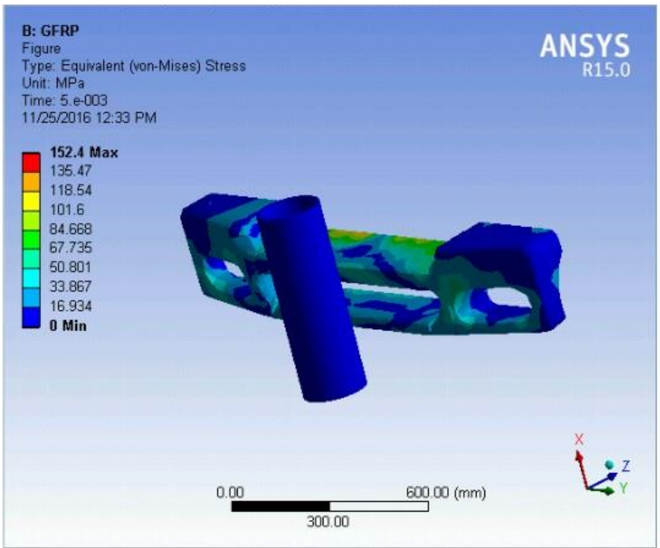

Figure -7: Equivalent Stress Distribution in Glass fiber reinforced Plastic

Deformation in Glass fiber reinforced Plastic

Maximum Total Deformation = 22.514 mm

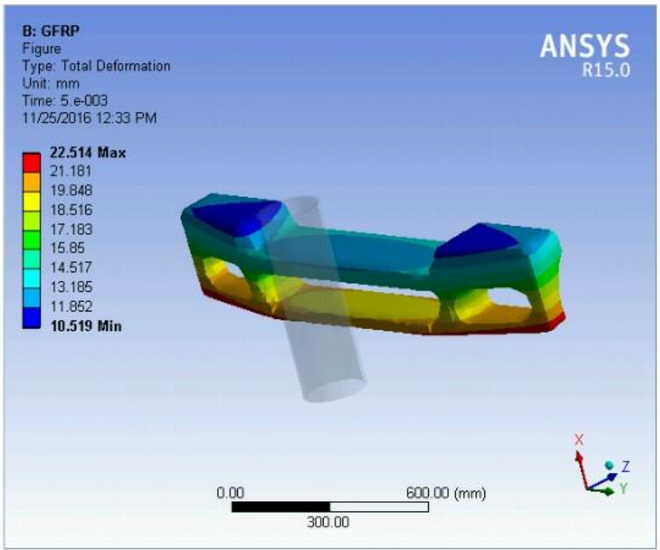

Figure - 8: Deformation in Bumper with Glass fiber reinforced Plastic.

D. Mathematical modeling and Results

Parameters:

Let,

M = Mass of Bumper in Kg,

S = Velocity of Bumper in m/s,

V = Volume of Bumper in m^3,

E = Modulus of Elasticity of Bumper in N/m^2,

σ = Stress generated in Bumper in N/m^2,

KE = Change in Kinetic Energy of Bumper in J,

SE = Change in Strain Energy of Bumper in J,

Assumptions Made:

1) The collision is perfectly elastic so that Momentum is conserved.
2) The system is loaded within elastic limit to apply theories of failure.
3) The time for collision is very less, that stresses generated rapidly and distribute in fraction of second.
4) The whole kinetic energy of Bumper is converted into strain energy of Bumper only.
5) The friction between Pole and Bumper is neglected.

6) Neglecting effect of bending and shearing of Pole and Bumper for analysis.

Formulation:

Kinetic energy of bumper before collision,

KE = ½ $M*S^2$…………………… (Equation 1)

Strain Energy of bumper after collision

SE = ½ $\sigma^2 V/E$ ………………………. (Equation 2)

Since the total kinetic energy of Bumper is converted into strain energy of Bumper, so according to law of conservation of energy, total Energy of the system is constant. So, balancing energies of the Bumper.

KE = SE

i.e. ½ $M*S^2$ = ½ $\sigma^2 V/E$ ………………………… (Equation 3)

From equation 3, we can find stress generated in Bumper.

So, Stress generated by bumper is given by,

$\sigma^2 = M*S^2*E/ V$

$\sigma = \sqrt{(M* S^2*E/ V)}$……………… (Equation 4)

$\sigma = K_f* \sqrt{(M* S^2*E/ V)}$…………… (Equation 5)

Where, K_f = Ks* Ksz * Kc * Ke.

K_f = Resultant Factors corresponding to shape, size, stress concentration, and endurance limit as shown in following table.

Table 5. Values of Design Stress Factors for Materials

	Shape Factor (K_s)	Size Factor (K_{sz})	Stress Concentration Factor (K_c)	Endurance Limit Factor (K_e)	Resultant Stress Factor (K_f)
ABS Plastic	1	1	2.5	1	2.5
CFRP	1	1	1.1	1	1.1
GFRP	1	1	1.8	1	1.8

Now, Applying Material conditions for Bumper,

1] For ABS Plastic

Mass, M = 9.2672 Kg,

Velocity, S = 10 m/s,

Volume, V = $8.9108*10^{-3}$ m^3,

Modulus of Elasticity, E = $2300*10^6$ N/m^2,

Resultant Stress Factor, K_f = 2.5

So, from equation 5,

$\sigma = K_f* \sqrt{(M* S^2*E/ V)}$…………… (Equation 5)

$\sigma = 2.5* \sqrt{(9.2672 * 10^2*2300*10^6 / 8.9108*10^{-3})}$

σ = 38665229.86 N/m^2,
σ = 38.66522986 MPa

2] CFRP

Mass, M = 16.039 Kg,

Velocity, S = 10 m/s,

Volume, V = $8.9108*10^{-3}$ m^3,

Modulus of Elasticity, E = $3.5*10^{11}$ N/m^2,

Resultant Stress Factor, K_f = 1.1

So, from equation 5,

$\sigma = K_f* \sqrt{(M* S^2*E/ V)}$…………… (Equation 5)

$\sigma = 1.1 * \sqrt{(16.039 * 10^2*3.5*10^{11} / 8.9108*10^{-3})}$

σ = 276097808.8 N/m2,

σ = 276.0978088 MPa

3] GFRP

Mass, M = 10.693 Kg,

Velocity, S = 10 m/s,

Volume, V = $8.9108*10^{-3}$ m^3,

Modulus of Elasticity, E = $60000*10^6$ N/m^2,

Resultant Stress Factor, K_f = 1.8

So, from equation 5,

$\sigma = K_f* \sqrt{(M* S^2*E/ V)}$……………(Equation 5)

$\sigma = 1.8 * \sqrt{(10.693 * 10^2* 60000*10^6 / 8.9108*10^{-3})}$

σ = 152735350.41 N/m^2,

σ = 152.73 MPa

Comparison of results for Numerical Impact Stress and ANSYS Impact Stress –

Table 6. Comparison of results

Sr. No.	Material	Numerical Impact Stress	ANSYS Impact Stress	Percentage Error

1	ABS Plastic	38.6652	39.18	-1.3313
2	CFRP	276.0978	256.21	7.2031
3	GFRP	152.7350	152.40	0.2193

From above table we can say that ANSYS results are confirming to mathematical model and maximum error is +7.2%.

VII. CONCLUSION

From this report it can be concluded that computer model can be used to predict impact analysis of bumper and stress pattern within bumper system has been studied. Also mathematical model results provides validation for ANSYS simulation model results. Depending on outcome, composite materials for bumper are selected for impact strength analysis. From, impact analysis of bumper it is suggested that Glass fiber reinforced composite bumper is suitable from repairing cost point of view, because it has minimum deformation. But Acrylonitrile Butadiene Styrene Plastic (ABS Plastic) is having minimum stress, hence it is best material from impact point of view and can absorb maximum collision energy.

VIII. ACKNOWLEDGMENT

The authors expresses their deepest gratitude to Department of Mechanical Engineering of SSBT's COET, Bambhori, Jalgaon, for providing their valuable time and full-fledged support to accomplish this research work.

REFERENCES

1. Hosseinzadeh RM, Shokrieh M, and Lessard LB, "Parametric study on automotive composite bumper beams subjected to low-velocity impacts", 2005.

2. Marzbanrad JM, Alijanpour M, and Kiasat MS, "Design and analysis of automotive bumper beam in low speed frontal crashesh", Thin Walled Struct., P P 902-911, 2009.

3. Gintautas Dundulis et al, Static analytical and experimental research of shock absorber to safe guard the nuclear fuel assemblies. *Elsevier, Nuclear engineering and design, 239,* 2009, 1-8.

4. Ping Yang et al, Design, test and modeling evaluation approach of a novel Si-oil shock absorber for protection of electronic equipment in moving vehicles. *Mechanism and Machine Theory, 43,* 2008, 18–32.

5. M. M. Davoodi, S. M. Sapuan and R. Yunus "Conceptual design of polymer composite Automotive bumper energy absorber" (2007).