Scientific Journal Impact Factor (SJIF): 4.364
International Journal of Modern Trends in Engineering and Research
ISSN (ONLINE) : 2349 - 9745
ISSN (PRINT) : 2393 - 8161

CRASH ANALYSIS OF A BUMPER WITH OUTER FRAME OF CAR CHASSIS

A.VENKATA DINESH[1], NARESH KUMAR.P[2] AND TARIKU ACHAMYELEH[3]
[1,2,3] Department of Mechanical Engineering, Debre Tabor University, Debre Tabor, Ethiopia

Abstract- In automobile design, crash and structural analysis are the two most important engineering processes in developing a high quality vehicle. Computer simulation technologies have greatly enhanced the safety, reliability, and comfort, environmental and manufacturing efficiency of today's automobiles. This significant achievement was realized with the advanced software and powerful computers that have been available in the last twenty years. Computer simulation technologies have greatly enhanced the safety, reliability, and comfort, and manufacturing efficiency of today's automobiles. To determine static and dynamic behavior many methods have been employed, out of them generally used methods are FEA and multi-body simulation methods. In automotive system there is an increasing request for lightweight construction and safety which leads to an intensified use of high-strength multi-phase steel such as DP or TRIP. The deflection of the bumper is varying from 0.503577mm to 0.978934mm for various thickness of bumper under applied load condition, which is rigid. Hence based on the rigidity the design is safe. By increasing the bumper made of thickness the stress induced are reduced from 531.523MPa to 127.317MPa. Hence the bumper design is safe based on strength criteria.

Keywords- Automobile, Computer simulation, Bumper Strength

I. INTRODUCTION

The primary concern for drivers and passengers is safety. Governments have responded to this key concern and expectation with an increasing number of regulations. Although the details may vary slightly from country to country, the fundamental requirements are almost similar. A vehicle is expected to provide adequate protection to drivers and passengers in a not so serious accident. To protect the occupants of a car, there are many new tangible safety features such as airbags; ABS control brakes, traction control The three most commonly used software's for crash analysis are LS-DYNA from LSTC, Pam-Crash from ESI, and RADIOSS from Mecalog. Although these are three distinctively different programs, their core technology can be traced back to the original DYNA3D from Lawrence Livermore National Laboratory. The original DYNA3D was developed for impact simulation with possible applications in weapon research. These three programs are now customized for civilian applications such as crash simulation, metal forming simulation, and product impact simulation. All three programs were launched within five years in the mid-eighties. The crash models used in the industry today are lumped parameter (LP) models, hybrid models, and finite element models. The FE models include heuristic models and continuum mechanics models.

II. EXPERIMENTAL PROCEDURE

2.1 MODELLING:

The model of the chassis frame has been created using the primitives approach using CATIA software.

Model is a Unibody structure. A work plane is established (xy plane).By using different geometrical primitives basic geometry is obtained By using different modeling options of CATIA for several times model is created according to shown in figure

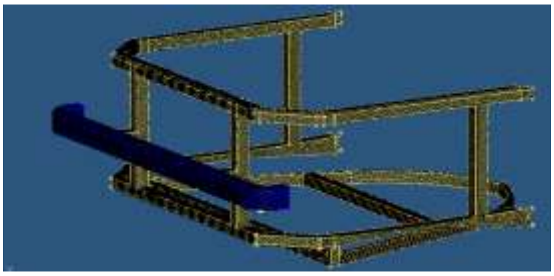

Figure 2.1 CATIA model of car front part

2.2 MESHING

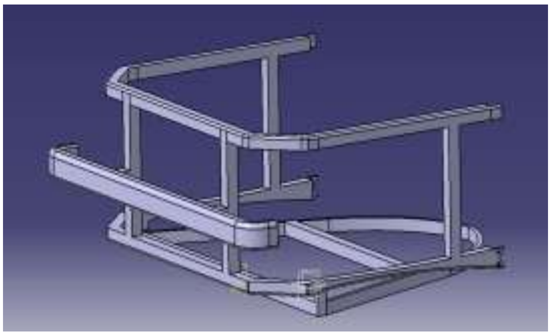

Figure 2.2 Final Mesh of Car Front part

For creating two types of mesh (solid and shell) one collector was created to move one set of surfaces to be meshed separately

Surfaces can be divided using lines to separate elements so that half element of vertical beam will not present in horizontal and vice versa. By separating surfaces into different collectors there may be chance of getting open volumes which cannot be meshed. Those open volumes can be filled with areas using ruled option in tools. Now meshing is done for both solid and shell element.

elements can be classified as solid and shell only based on type of meshing done. If only 2d meshing is done then it is shell element, else if 3d meshing is also done then it is called solid element. After 2d meshing is done all elements should be checked for whether free edges or T-connectors are present or not, if present those should be removed.

From the hyper mesh software we can observe following in our model

- The total number of elements is 133758
- The total number of nodes is 40374

III. RESULTS AND DISCUSSION

Results for Static Analysis:

From the static analysis the deflections and stresses are obtained in the bumper under a load of 0.1MPa.

From figures 3.1 it is observed that that the maximum deflection is 0.978934mm for the bumper made of thickness 3mm at the point which is indicated as MX which is at node 63982. The minimum deflection is obtained at the constrained points which are indicated as MN.

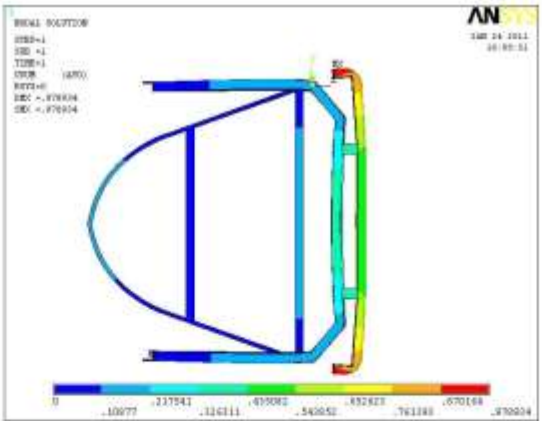

Figure 3.1: Vector plot of deformation for shell thickness 3mm under static load

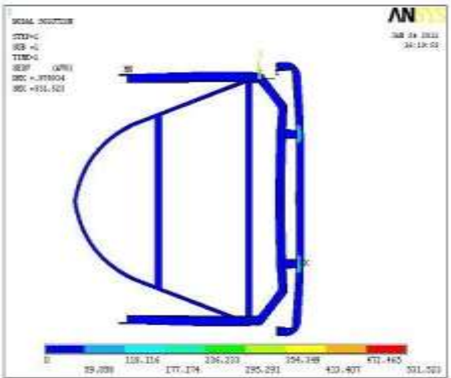

Figure 3.2: Vector plot of Von-mises stress for shell thickness 3mm under static load.

Volume 04, Issue 1, [January– 2017] **ISSN (Online):2349–9745; ISSN (Print):2393-8161**

From Figure 3.2 it is observed that that maximum stress for the bumper made of thickness 3mm is found out to be 531.523MPa at the node 1030 which is indicated as MX in the figure.

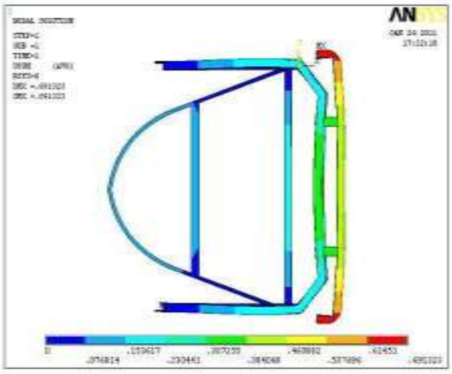

Figure 3.3: Vector plot of deformation for shell thickness 4mm under static load

From the figure 6.3 it is observed that that the maximum deflection is 0.691323mm for the bumper made of thickness 4mm at the point which is indicated as MX which is at node 61358. The minimum deflection is obtained at the constrained points which are indicated as MN.

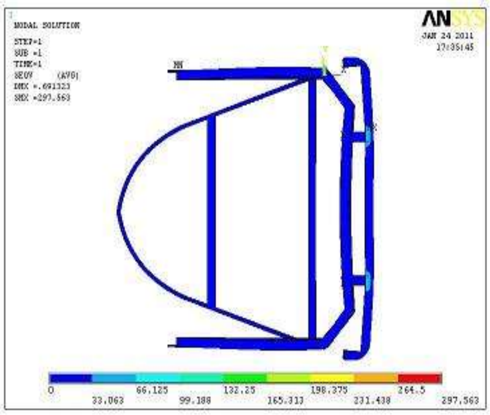

Figure 3.4: Vector plot of Von-mises stress for shell thickness 4mm under static load

From the figure 6.4 it is observed that that maximum stress for the bumper having thickness 4mm is found out to be 297.563MPa at the node 1030 which is indicated as MX in the figure.

Table 3.1: Natural Frequencies of Car Frame by Varying Thickness of the Bumper

Mode No	Natural Frequencies, Hz	
	Bumper Thickness, mm	
	3	4
1	0.531657	0.526162
2	0.587794	0.587785
3	1.236	1.232
4	1.729	1.695
5	2.144	2.141
6	2.458	2.43
7	2.502	2.48
8	2.682	2.682
9	2.929	2.925
10	3.131	3.12

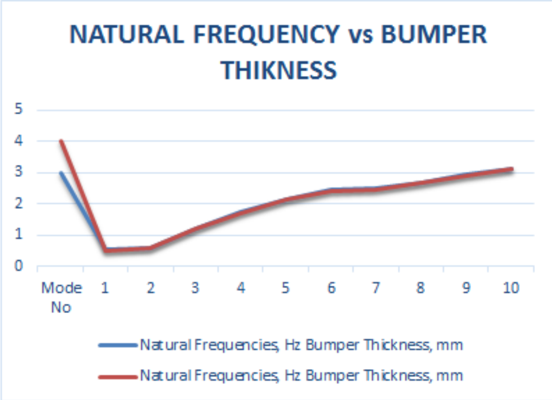

Figure 3.5: Natural Frequency vs Bumper Thickness .

IV. CONCLUSIONS

1. The deflection of the bumper is varying from 0.503577mm to 0.978934mm for various thickness of bumper under applied load condition, which is rigid. Hence based on the rigidity the design is safe.
2. By increasing the bumper made of thickness the stress induced are reduced from 531.523MPa to 127.317MPa. Hence the bumper design is safe based on strength criteria.
3. The natural frequencies estimated for various thicknesses are very low as the structure is stationary.
4. When the impact takes place on the bumper, the maximum amplitude is 0.7611mm under transient loading. This value is very less than 2.8mm as per the **Ford Motor Vehicle Design Test (FMVDT)**. Hence the bumper design is safe based on dynamic conditions.

REFERENCES

[1] Lutjens, J : Hunkel, M;Scholl,U;Wallmersperger, T; Schiling,R; Sihling,D; Wiegand, K : using coupled FEM simulations for improvement of crash-relevant automotive parts by local hardening. SCT 2008, International Conference on steels in Cars and Trucks, June 1-5, 2008, Wiesbaden

[2] Lutjens J., Hunkel M., Hoffmann F., Zoch H,-W.: FEM Simulation of carbon diffusion and phase transformations in dual phase and TRIP Steels, Proc.15th IFHTSE Congress and the 20th Surface Modification Technology Conference(2006)

[3] Lutjens j., Hunkel M., Hoffmann F., Zoch H,-W.: FEM Simulation of local treatment on dual phase and TRIP Steel parts , HTM 61 (2006) 5, S. 249-253

[4] Schilling,R.; Lanzerath, H.; Paas, M.; Wesemann, J.: Numerische Analysen Zum Einsatzpotential neuer werkstoffe in der Passiven Sicherheit. VDI Berichte Nr. 1559, Wurzburg, September 2000

[5] Schilling , R.; Lanzerath, H.; Ghouati, O.: Coupled Stamping-Crash-Analysis. In: E. Dehning & k. Wolf (ED.): Why do multi-physics Analysis? NAFEMS 2006

[6] Lonny L.Thompson, et al, Clifford C. Chou, Bahig B. Fileta, Tawfik B. Khalil, Albert King, Hikmat F. Mahmood, Harold J. Mertz, Jac Wismans, Vehicle Crashworthiness and Occupant Protection, Sponsored by: Automotive Applications Committee, American Iron and Steel Institute, Southfield, Michigan. 2001.

[7] Oijier, F., "Development and Validation of High Fidelity Vehicle Crash Simulation Models,"

[8] Cicek karaoglu et al., Stress analysis using riveted joints, International Symposium on Heavy Vehicle Weights & Dimensions, Delft, NL

[9] Tso-Chien Pan et al, "Dynamic Vehicle Element Method - Present and Future Developments," The International Conference on Vehicle Structural Mechanics.

[10] Arborio,N.,et al, Future trend of automobile and the high strength sheet steel. Tetsu-to-Hagane. vol. 68, no. 9, Japan 1982.

[11] North American Bumper System Market, 1997-2001 Model Years, September 19, 1997, private study, CSM Corporation, 2365 Woodlake Drive, Suite 150, Okemos, MI 48864

[12] David W. Dickinson, Final Report to AISI BUMper group , Bumper Component welding State-of-the-Art Survey, December 31,2000.

[13] Xichan Z (2002) An analysis on automotive frontal crash test regulation and its tendency, Automot Engg 24(1): 1-5

[14] Sabah U, Stan G, Chris A, et al (1998) A study of commercial vehicle safety alliance's out-of-service criteria. Accident Anal Prevent 30(1):61-67
[15] Haug E, Clinckemaillie J, Ni X (1998) Recent trends and advances in crash simulation and design of vehicles. Chin Mech Eng 34(1):1-6
[16] Hallquist JO (1998) LS-DYNA3D theorectical manual. LSTC, Livermore
[17] Zhihua Z (1994) Finite element method in crashworthiness analysis of automobiles. Automot Eng 16(1):1-6
[18] Thacker JG, Reagan SW, Pellettiere JA et al (1998) Experiences during development of a dynamic crash response automobile model. Finite Element Anal Des 30:279-295
[19] Song D, Mack P, Tarriere C et al (1993) Finite element simulation of the occupant/belt interaction: chest and pelvis deformation, belt sliding and submarining. SAE 933108
[20] National Standard of PRC : (1999) The stipulation
[21] Viano DC , Atepally S (1990) Assessing the safety performance of occupant restraint systems, SAE paper 902328
[22] Mcnay GH et al (1988) Numerical modeling of tube crash with experimental comparison. SAE , 88089
[23] Schuster , Dr. Peter, "Current trends in Bumper design for pedestrian impact", December 31, 2004.
[24] Stuart F. Brown, Welding's big New Bag of Tricks, Fortune Magazine, January 13, 1997