ISSN 2395-1621

Crash Analysis By Cad/Cam Tools

#1Vidyasagar.S.Gavali, #2 Dipak.K.Dond

1 gavali2611@gmail.com
2 dpkdnd@gmail.com

#12 MET,BKC,IOE,Nasik,Maharashtra,India

ABSTRACT

The development of automobiles with regards to crashworthiness behavior depends strongly on virtual testing and simulation. Thus development work based on cost intensive prototype construction and testing has been extensively reduced for the body in white as well as for the exterior and interior trim. The dramatic shortening of the total development time during the last years needs a much more front loaded development process which has been realized by numerical simulation. The numerical simulation accompanying the design of the car may be divided into three main phases – the concept, the series, the series development and the optimization phase.During the concept the passive phase the passive safety concept and its needed packaging space have to be determined. The series development is finished by prototype testing which should confirm the virtual development is finished in an ideal case. Optimization work should close the development before the car launch. During these phases the focus of the numerical simulation changes from a more global view to a very detailed local analysis. In response to growing concerns about incompatibilities in collisions between car and light trucks (i.e pickups and SUVs), representatives from automobile manufacturers, the insurance institute for highway safety organization agreed in 2003 to develop collaborative recommendation to improve vehicle crash compatibility.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

The development process of automobiles with regard to crashworthiness behavior depends strongly on virtual testing and simulation. Thus, development work based on cost intensive prototype construction and testing has been extensively reduced for the body in white as well as for the exterior and interior trim. The dramatic shortening of the total development time during the last years needs a much more front loaded development process which has been realized by numerical simulation. The numerical simulation accompanying the design of a car may be divided into three main phases – the concept, the series development and the optimization phase.

During the concept phase the passive safety concept and its needed packaging space have to be determined. The series development is finished by prototype testing which should confirm the virtual development in an ideal case. Optimization work should close the development before the car's launch. During these phases the focus of the numerical simulation changes from a more global view to a very detailed local analysis.

2. PRESENT THEORIES & PRACTICES

Within the passive safety simulation a lot of different load cases are tested numerically. During the last years the number of load cases has been increased significantly. The variety of applications might be divided into three main topics.

Structural Crashworthiness of Whole Car: When frontal, rear and side crash computations are widely used in the development process, compatibility is a growing application. Usually, these models represent most of the body in white, chassis, driveline and exterior trim components. Interior trim components are modeled when the influence on the overall load paths has to be considered, like seats for side crash analysis. These components might be discretized or their masses are smeared over neighbouring parts.

Occupant Protection: The finite element method is mainly used for occupant protection analysis during side impact. The optimization of structural response, interior trim deformation, and airbag behaviour is done in a parallel and iterative process. Because of the high number of needed simulations, a major number of analyses are done in so called substructure runs which uses boundary conditions of the whole car analysis. The same car model as for the side crash
computation of the whole car is used. Interior trim and a dummy model are added.

Crashworthiness of Components: A wide range of applications have to be covered by simulation, some of which are mentioned here. These analyses can be performed independently of the whole car crash simulations, but same meshes are used when possible. The discretization of the components is in general much finer than in whole car analysis but the number of represented components is smaller.

The numerical development influences the design of an increasing number of components. When body-in-white (BIW) components have been analyzed for long time, the design of exterior trim and interior trim is mainly driven by virtual work today, figure 4. Three main development directions are observed: analysis models are getting much more complex, new or numerically not very well described materials are introduced and, even in full car crash simulations, very local results are needed. For example, the door panel made of reinforced thermoplastic material including the detailed rib and retainer design is today analyzed by using complex side impact load cases.

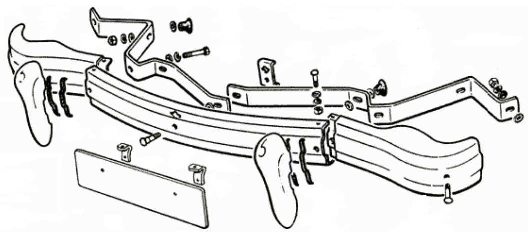

Fig.1 General components and arrangement of front bumper system

3. METHODOLOGY:

This study will use commercial FEA tools such as ANSYS, Abaqus, LSDYNA, etc for carrying out FE analysis limited to the Bumper as a single component (though, the boundary conditions would take into consideration its fitment with the mating parts in the assembly).

- **Finite Element Analysis [14]**
  Finite element method is used to analyze structures by computer simulations and therefore it helps to reduce the time required for prototyping and to avoid numerous test series. The modeling and analysis will be done using Finite element Analysis software.
- **Steps for finite element analysis:**
  FEA is mainly divided into three following stages:
  - Preprocessing
    - Creating the model.
    - Defining the element type
    - Defining material properties
    - Meshing
    - Applying loads
    - Applying boundary conditions
  - Solution: Solving the pre-processed geometry using a suitable Solver
  - Post processing: Review of results such as deformation plot, stress plot, etc

For ***alternative method*** of computation/ validation, the component would be tested over a physical setup by the Sponsoring Company. The results so obtained would be shared by the Company via a Test Certificate issued upon request.

4. PROPOSED WORK

The study will focus on modifying few of above stated parameters to suggest improvements in existing bumper of passenger car/ SUV present in Indian markets. First, study will focus on studying existing design and based on observations, design improvements will be suggested. Modified front bumper design will be tested using FEM software for deflection, impact force and stress distribution. Results of modified bumper will be compared against existing design.
The aim of this work is to study front bumper of one of the existing passenger car/ SUV in Indian market. Design modifications can be suggested or tried out on following basis:

- Performance related parameters of bumper
- Deflection/ Plastic strain induced to be within the limits specified
- Thickness or geometry to be manipulated for effecting compliance

FEM is backbone of today's automotive industry. In recent times FE analysis is widely used to validate the complex designs like bumper. Use of FEA not only reduces product development time but also saves lot of cost. Hence, this work proposes FE analysis of bumper to validate the design modifications in from bumper of car.

Overall objectives of this study are as stated below:

- To study existing passenger car front bumper in Indian market for possible design modifications as discussed earlier
- Carry out design improvements in front bumper of car
- Carry out impact analysis using CAE to study performance of modified bumper against existing front bumper
- Document the results and plan to improvise further

Typical Steps involved in this project work are mentioned as below:
Following steps will be performed to execute this project.

5. FACILITIES AVAILABLE

- Library Facilities, ____(name of the college)
- Typical CAE software like Nastran, LS Dyna, Abaqus, Marc or any other suitable software as decided by the Sponsoring Company and/or its sister concern
- Completion certificate and/or Experimental Test certificate from the sponsoring company

6. NAME OF MATERIAL=NYLON 66,30%GLASS FIBER USED
TYPE=BUMPER (FRONT LOWER) MARUTI A3
MODULUS OF ELASTICITY(E) = 2500 MPa
SPECIFICATION=

Part Name	Material	Wt. in gms.	No. of parts	Wt. per car (grams.)
Instrument Panel	Filled PP	4000	1	4000
Glove box	Filled PP	350	1	350
Glove box lid	Filled PP	200	1	200
Garnish cowl ventilator	Filled PP	210	2	420
Boot component	Filled PP	500	2	1000
Bumper (Front upper)	PP modified	780	1	780
Bumper (Front lower)	PP modified	2600	1	2600
Bumper (Rear upper)	PP modified	730	1	730
Bumper (Rear lower)	PP modified	3200	1	3200

Typical Product Properties of Glass filled-PP

Property	Units	Base PP Homo polymer	20 % glass	30% glass	40% glass filled PP
Tensile strength at yield	MPa	350	355	**460**	370
Density	10^3Kg/m^3	0.90	1.06 - 1.08	**1.75**	1.19-1.21
Tensile strength at break	MPa	230	325	**500**	360
Elongation at break	%	40	30	**15**	25
Flexural strength	MPa	330	440	**700**	450
HDT 66 psi	deg. C.	75	90	**97**	112

CALCULATION FOR V= 4 Km/ hr
3. POINT BEND TEST CALCULATIONS
Input data :
Mass of bumper = 2.6 Kg
Mass of rigid frame = 10 Kg
Total mass of assembly = 12.6 Kg
Deacceleration time = 0.005 sec
Initial velocity of bumper = 4 Km/hr
= 4*1000/3600
V= 1.111 m/sec
Final velocity of bumper = 0 Km/hr
Deacceleration of bumper = a = V_2-V_1/.005
= 0-1.111/.005
a = 222.2 m/sec^2

Force on front bumper : F= m*a
=12.6*222.2
= 2.8*10^3 N
F = 2.8 KN
Answer archived on the hypermeshsoftwere

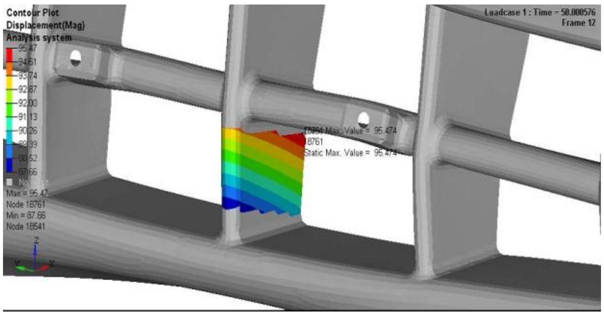

Fig. 2
Hypermesh result for first trial

3 POINT BEND TEST CALCULATIONS
Material: Nylon 30% glass filled
Modulus of Elasticity, E = 2500 MPa
Applied force, F = 2.8 KN
Length of Span, L = 70 mm
Width of Component, W = 41.7 mm
Thickness of Component, T = 4 mm
Distance from neutral axis, y = T/2 = 2 mm
Maximum Moment (M_{max}) = F*L/4
= (2.8*10^3*70)/4
M_{max} = 49*10^3 N-mm
Moment of Inertia (I) = W*T^3/12
= 41.7*4^3/12
I = 222.4 mm^4
Maximum bending stress σ_b = M_{max} * y/ I
= 49*10^3 *2 / 222.4
σ_b = 440.64 MPa

Maximum Deflection (y_{max}) = F*L^3/48EI
= (2.8*10^3*70^3) / (48*2500*222.4)
y_{max} = 35.98 mm
CALCULATION FOR V= 10 Km/ hr
COMPRESSION TESTING
Input data :
Mass of bumper = 2.6 Kg
Mass of rigid frame = 10 Kg

Total mass of assembly = 12.6 Kg
Deacceleration time = 0.005 sec
Initial velocity of bumper = 10 Km/hr
= 10*1000/3600
V= 2.777 m/sec
Final velocity of bumper = 0 Km/hr
Deacceleration of bumper = a = V_2-V_1/.005
= 0-2.777/.005
a = 555.4 m/sec^2

Force on front bumper : F= m*a
=12.6*555.4
= 6.999*10^3 N
F = 7 KN
Analysis of bumper for force: F = 2.8 KN

M_{max} = w*l/4
= 2.8*10^3*70/4
M_{max} = 49*10^3 N-mm

σ_c/Y= M/I
= M*Y/I
I = 1/12 * b*d^3
I = 1/12 0*4.17*4^3
I = 222.4 mm^4

σ_c = M*Y_c /I
= 49*10^3 * 2 / 222.4
σ_c = 440.64N/mm^2

Y_{max} = PL^3/ 48 EI
= 2.8*10^3*70^3/48*2500*222.4
Y_{max} = 35.98 mm
Elasticity of given material :**E = 2500 Mpa**
E = 2500 N/mm^2
σ=E*e
e= σ_c / E
= 440.64/2500
= 0.1762
e = 17.62 %
% Elongation safe zone= 15 %
We have to change thickness of bumper for reducing plastic strain
Analysis of bumper for F = 7 KN
t = 4 mm
F = 7 KN
Gauge length – 190 mm
Total length = L = 1350 mm
Support length =L_1= 3.5 feet
=3.5*12*25
L_1=1097.28 mm
M_{max} = w*L_1/4
= 7*10^3*190/4
M_{max} = 332.5*10^3 N-mm
σ_c/Y= M/I
= M*Y/I
I = 1/12 * b*d^3
I = 1/12 0*280*4^3
I = 1.493*10^3 mm^4

σ_c = M*Y_c /I
= 332.5*10^3 * 2 / 1.493*10^3
σ_c = 445.41N/mm^2

Y_{max} = PL^3/ 48 EI
= 7*10^3*190^3/48*2500*1.493*10^3
Y_{max} = 267.98 mm
Elasticity of given material : E = 2500 Mpa
E = 2500 N/mm^2
σ=E*e
e= σ_c / E
= 445.41/2500
= 0.1781
e = 17.81 %
% Elongation safe zone= 15 %
The answer achieved on the hyper mesh softwere

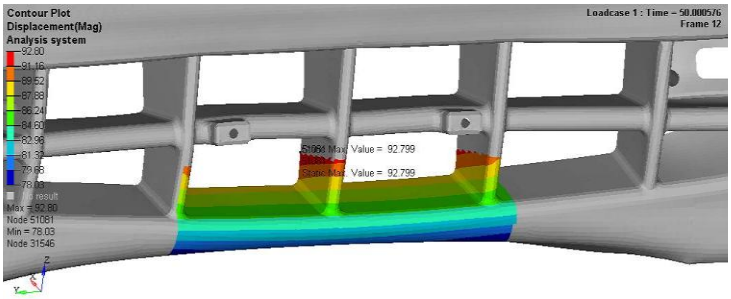

Fig. 3
Hypermesh result for final trial

We have to change thickness of bumper for reducing plastic strain
Calculation for mass;-
1.Density of bumper=ρ=1.75*10^3 Kg/m3
=1.75*10^{-6} Kg/mm3
m= ρ* V
= 1.75*10^{-6} *1330*280*4
m =2.60Kg
Energy absorb=1/2 mV^2
=1/2*2.60*2.77^2
=9.974 J
2.Density of bumper=ρ=1.75*10^3 Kg/m^3
=1.75*10^{-6} Kg/mm^3
m= ρ* V
= 1.75*10^{-6} *1330*280*4.5
m =2.93Kg
Energy absorb=1/2 mV^2
=1/2*2.93*2.77^2
=11.24 J
3.Density of bumper=ρ=1.75*10^3 Kg/m3
=1.75*10^{-6} Kg/mm3
m= ρ* V
= 1.75*10^{-6} *1330*280*5
m =3.25Kg
Energy absorb=1/2 mV^2
=1/2*3.25*2.77^2
=12.46 J

RESULT

VELOCITY	ITERATION	BUMPER THIKNESS	MAXIMUM CENTER POINT DEFLECTION	PLASTIC STRAIN(%)
10 Km/hr Type= 3 Compression testing	1	4 mm	267.98 mm	17.81%
	2	4.5mm	---	Below 16%
	3	5 mm	---	Below 15%
4 Km/hr Type= 3 Point bending testing	1	4 mm	35.98 mm	17.62%
	2	4.5mm	---	Below 16%
	3	5 mm	---	Below 15%

VELOCITY	ITERATION	BUMPER THIKNESS	MASS OF BUMPER	PLASTIC STRAIN(%)	ENERGY ABSORB
10 Km/hr Type= 3 Compression testing	1	4 mm	2.6Kg	17.81%	9.974 J
	2	4.5mm	2.9Kg	Below 16%	11.24 J
	3	5 mm	3.23Kg	Below 15%	12.46 J

By Analytically and Verified by Hypermesh Software:

VELOCITY	ITERATION	BUMPER THIKNESS	PLASTIC STRAIN(%)	MASS OF BUMPER	MAXIMUM CENTER POINT DEFLECTION
10 Km/hr Type= 3 Compression testing	1	4 mm	17.81	2.6Kg	267.98 mm
	2	4.5mm	Below 16%	2.9Kg	200.00mm
	3	5 mm	Below 15%	3.23Kg	150mm

CONCLUSION

After taking crash testing on different thickness of bumper by manually and on software it is shown that on above table that above design is safe for 4.5 mm thickness and it can be used as final result.

REFERENCES

1. Jovan Obradovic, SimonettaBoria, Giovanni Belingardi“Lightweight design and crash analysis of composite frontal impact energy absorbing structures.”

2. Ping Zhu , Yu Zhang , Guanlong Chen,“Metamodeling development for reliability-based design optimization of automotive body structure”.

3. Xin Yang , YongXia, QingZhou, Pei-ChungWang, KathyWang,“ Modeling of high strength steel joints bonded with toughened adhesive for vehicle crash simulations”.

4. RecepGmruk, Sami Karadeniz,“The influences of the residual forming data on the quasi-static axial crash response of a top-hat section”

5. O.G. Lademo,T. Berstad, M. Eriksson, T. Trylandb, T. Furu, O.S. Hopperstad, M.Langseth, “A model for process-based crash simulation”

6. F.İncea, H.S. Türkmena, Z. Mecitoğlua, N. Uludağb, İ Durgunb, E. Altınokb, H.Örenelb, “A numerical and experimental study on the impact behavior of box structures”

7. Tso-Liang Tenga, Fwu-An Chang, Yung-Sheng Liu, Cheng-Ping Peng, “Analysis of dynamic response of vehicle occupant in frontal crashusingmultibody dynamics method”.