Non Linear Quasi-Static Analysis of Rear Under-Run Protection Device

HARSHAL DESHPANDE[1], ANIRUDH SENGER[2], ANIL LASHKARI[3], SAGAR VERMA[4]

[1]IICAE, Indore

--***--

Abstract - *Whenever the passenger car meet with accident to the heavy duty truck from rear, then it will tend to penetrate under the truck bed. This caused is responsible for the thousands of accidents, which causing severe injuries and spot deaths. This is mostly due to the lack of effective guarding system. The rear underrun protection device (RUPD) is used to offer safety against rear impact and rear underrunning from the passenger vehicle to heavy and medium duty trucks. Therefore the bureau of Indian standards adopted Indian standard to ensure the proper strength of RUPD's. This paper is focusing on the design of RUPD should follow the safety standards and meet the test criteria. As per this standard the RUPD subjected to 5 consecutive loads P1RH, P1LH P2RH, P2LH & P3 transversely with the given distance. The RUPD design can approved only when it shows sufficient strength in the test procedure. In order to optimize the RUPD design under the test criteria the FEM (finite element method) is used. In this study the FE model of RUPD is developed in HYPERWORKS with L.S DYNA non-linear material properties, then the FE model of RUPD is subjected in to the test or loading conditions and captured the effective plastic strain, and compared with allowable maximum percentage elongation of the material to check the strength of the RUPD model.*

***Key Words*: Rear under-run protection device(RUPD), Finite element method(FEM), Non linear quasi-static analysis, Effective plastic strain, Max. percentage elongation, Hyperworks & L.S dyna.**

1. INTRODUCTION

Rear under-run protection device provides safety in the rear impact of heavy duty vehicles with a passenger car. Any passenger vehicle can impact with heavy duty vehicle either from front rear and side, the maximum probability of damage only occurs in a passenger vehicle.

Assume a case where passenger vehicle impact with heavy duty truck and there is no under-run protection device in this condition entire vehicle gone under Neath the heavy duty truck, the entire kinetic energy of the passenger vehicle applied on the front pillar of the passenger vehicle and they are not capable to take such load that's why the entire structure got crumbled (as shown in figure).

From IS-14812 2005 regulation which derived from ECE 58 regulation. RUPD design should follow the safety standard rules and meet the test criteria.

More than 250 fatalities recorded in northern Sweden between 1995 to 2005 due to passenger vehicle accidents to the heavy duty trucks. According to a published report, more than 40,000 traffic fatalities occur every year in the European Union. India is also not untouched with this problem, about 8,304 people died in 2 years (2009-2011) due to under-run types of accidents. The Under Run Protection Devices (UPD) can be used to minimize the probability and fatalities of such accidents. There are mainly three types of under-run protection devices, namely Front, Side, and Rear under-run protection devices. Based on their name, each of these devices is designed to protect the pedestrian or small vehicle occupants from the direct impact of the trucks. This study is mainly focused on Rear Under-run Protection Device (RUPDs).

Test procedure

The test procedure for rear under-run protection device is mentioned below which is from IS-14812-2005 regulation. The order in which forces are applied may be specified by the manufacturer.

a) The device should be fitted as close to the rear of the vehicle as possible.
b) When the vehicle is unladen the lower edge of the device should not be more than 550mm above the ground.

c) The width of the device should not exceed the width of the rear axel.
d) The section height of the cross-member should not less than 100mm.

Loading:-

The specified force should be applied by a ram which is suitable articulated (for example by means of the universal joint) and should be parallel to the median longitudinal plane of the vehicle via surface not more than 250mm in height and 200mm wide. The center of the surface is placed successively at point P1, P2, & P3
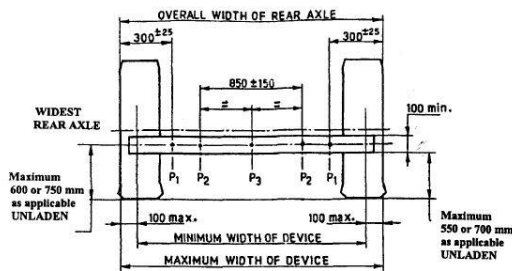
NOTE — The exact transverse location of points P_1 and P_2 is to be specified by the manufacturer within the limits shown.

- Point P, are located 300 + 25 mm from the longitudinal planes tangential to the outer edges of the wheels on the rear axle; point P2 which are located on the line joining point P1, is symmetrical to the median longitudinal plane of the vehicle at a distance from each other of 700 to 1000 mm inclusive, the exact position being specified by the manufacturer. The height above the ground of points P, and P2 (see Fig.) should be defined by the vehicle manufacturer within the lines that bound the device horizontally. The height shall not, however, exceed 600 mm when the vehicle is unladen. P3 is the center point of the straight line-joining point P2.

  Note:- consider the length of RUPD bar is equal to real axel so the distance of P1 loading can measure from the outer most edge of the RUPD bar

- A horizontal force equal to 12.5 percent of the maximum technically permissible weight of the vehicle but not exceeding 25 kN should be applied successively to both points P, and to point P3.
- A horizontal force equal to 50 percent of the maximum technically permissible weight of the vehicle but not exceeding 100 kN shall be applied successively to both points P2.

Methodology:-

Detailed 3D model of RUPD assembly and truck chassis (half segment) were developed using PTC CREO, and the assembly were imported in HYPERMESH for the preparation of FE model of RUPD and the truck chassis assembly. Fig shows the detailed 3D model of the assembly
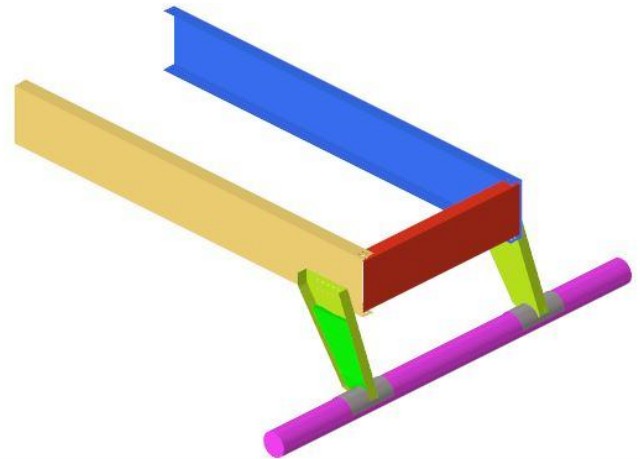
In Finite Element Analysis (FEA) mainly three processes are being followed:

1. Pre-process (ALTAIR HYPERMESH)
2. Solving (L.S DYNA)
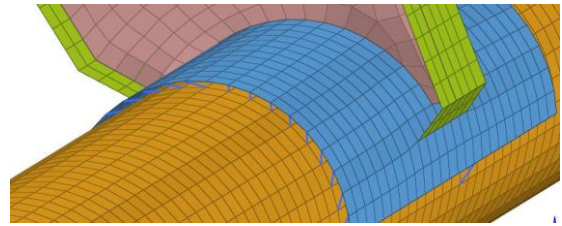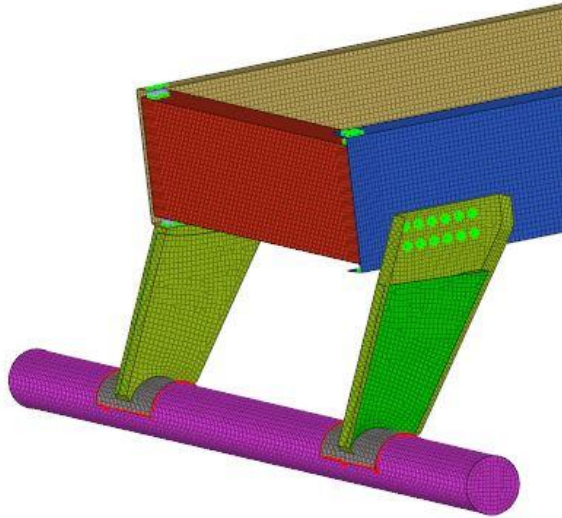3. Post-process (ALTAIR HYPERVIEW)

PRE-PROCESSING

Preprocessing is also called Meshing is the first step in solving a problem in Finite Element Analysis. Here the entire domain is discretized (divided) into meaningful divisions often called "Elements". These elements form the building block on which the Boundary conditions and external effects are specified.

- **Mashing**
-

The basic theme of FEA is to make the calculation at an only limited finite number of point's and then interpolate the results for the entire domain (surface or volume). Any continuous object has an infinite degree of freedom and its just not possible to solve the problem in this format. Finite element method reduces the degree of freedom from infinite with the help of discretization *i.e.* mashing (nodes& elements).

Mathematically element thickness (specified by the user) is assigned half element top & half element bottom. Hence for appropriate representation of geometry via 2D mesh, it's necessary to extract midsurface and generate nodes and elements on the midsurface.

Shell element were created on the extract mid surface on all components of the assembly (long member, cross member bracket, & RUPD bar) as shown in the figure with satisfying following element quality criteria
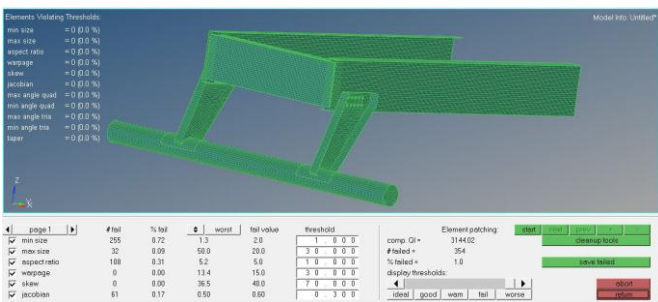

a) Warpage angle: it is defined as the angle between normal to row planes Max. Angle put the possibilities is reported as warpage angle.
b) Aspect ratio: max, element edge length to the minimum element edge length.
c) Jacobian: Jacobian is a scale factor arising because of the transformation of co-ordinate system. The element is transformed from global co-ordinate to local co-ordinate system.

In order to bolting between the cross member and long member similarly long member and brackets as shown in the figure we can use 1D element, assign beam section define solid circle 10mm Dia.

For rigid connection between bracket and RUPD bar 1D element are also used with the help of rigid element.

Material properties:

Linear and non-linear material are used in our model. Non-linear material namely E250 BSK46 & domex are assigned their different components. In order to assign non-linear material type MATL024 (PIECE_WISE_LINEAR_PLASTICITY) is used for L.S. DYNA. Linear material is assigned for bolt, and type MATL001 is used for that.

We do not want to any kind of deformation or other disturbance in the ramp so that the ramp would behave like a rigid therefore the rigid material type MATL020 are used in L.S DYNA.

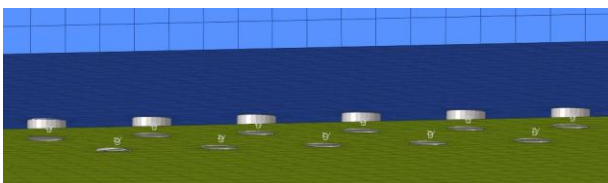| Material	Modulus of elasticity (in Mpa)	Poisson's ratio	Density (in ton/mm)	Yeald stress (in Mpa)	Ultimate stress (in Mpa)	Max percentage elongation
BSK46	210000	7.8e-009	0.3	560	720	10%
DOMEX	210000	7.8e-009	0.3	550	760	17%
E250	210000	7.8e-009	0.3	250	410	23

Properties:

In properties the thickness has been assigning to the component different components has different thicknesses are as follow.

Component name	Thicknesses (in mm)
Chassis frame	7.0
Bracket	5.0
RUPD bar	3.0

Contact interference:-

contact definition allow the modeling of intersection between one or more part's in a simulation model and have become a necessary in any small or large deformation problem, the main objective of the contact interference is to eliminate any overlap or penetration between the interaction surfaces and they accomplished this by first detecting the amount of penetration then applying a force to remove them.

The various type of contact interference is available in L.S DYNA, surface to surface type contact between ramp and RUPD assembly.

Boundary and loading conditions:-

Boundary conditions are applied such that it will be the same as the physical test and it will not any numerical error in the analysis. The loads are applied as per standard's IS14812-2005

➢ Boundary conditions:

The nodes at chassis are constrained in all the direction (see Figure). The chassis member is very critical and its deformation may lead to severe structural damage as well. It would be also impossible to change the chassis member as it is a major component in the heavy commercial vehicles.
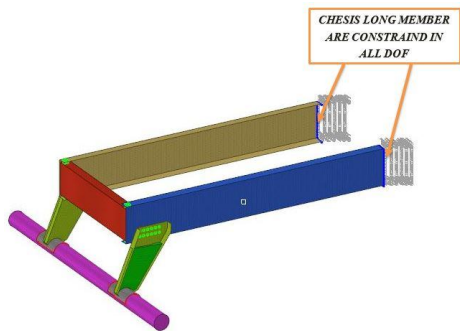

➢ Loading conditions:

The load was applied using RAM as shown in the figure. The model of the ram has using shell element with required dimensions (just like plats) as shown in the figure. Now both the center nodes of both plates are joint together with the help of contact interference (EXTRA NODE), then FE joint (spherical joint) are created between that both nodes their for these model has behaved like a universal (ball) joint (acc. to IS14812).
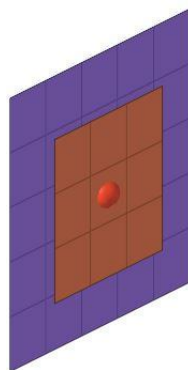
Five different loads were applied sequentially as shown in the figure. The location of each load application is shown in the figure, 6. 12.5% of the GVW was used for P1 and P3 while 25% of the GVW load was applied at P2 location. In order to apply this type of loads, a different type of card images are available in L.S DYNA. LOAD_RIG_BOD card are used. Part ID, load curve and degree of freedom are required input.

P1 LH which carries 25 KN load first het the RUPD bar. This give some effective plastic strain. Now the deformed body is import in again hypermesh and the loading of P1 RH which also carries 25 KN load is prepared and some simulation result has occurred. Now the deformed structure due to this loading is again import in hypermesh and P2LH loading which carries 100 KN is prepared and again some simulation result is occurred due to this loading. Similarly, loading P2 RH & P3 are applied respectively.
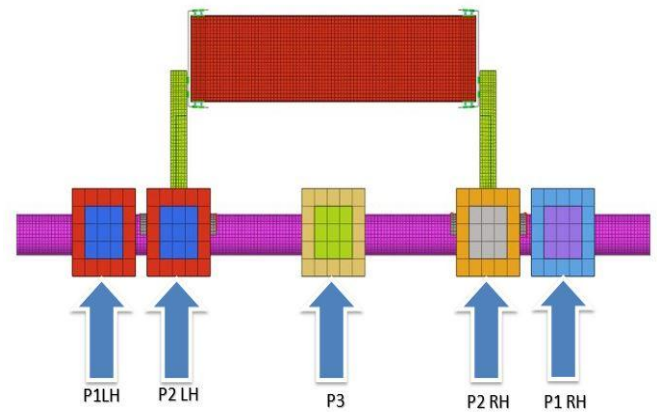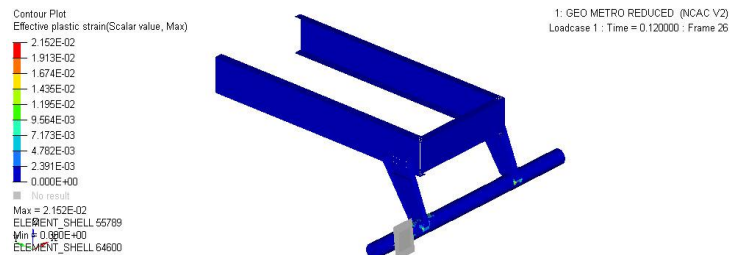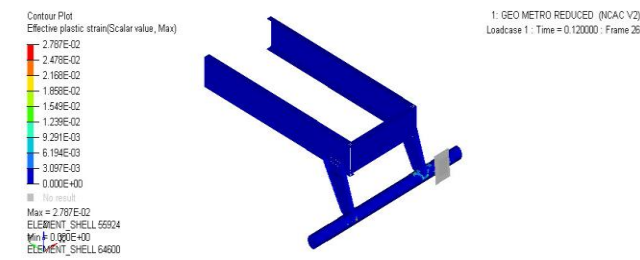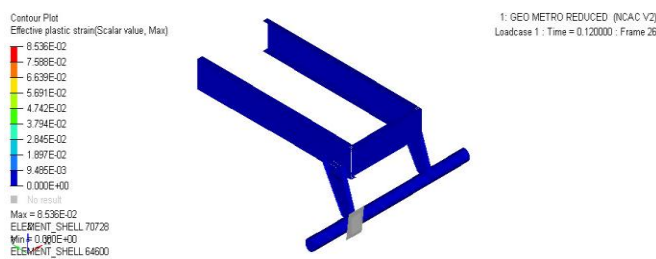

✓ Loading sequence

RESULTS:

P1LH: The effective plastic strain is shown in figure 2.15% peak strain was found in case of the P1 LH load case.
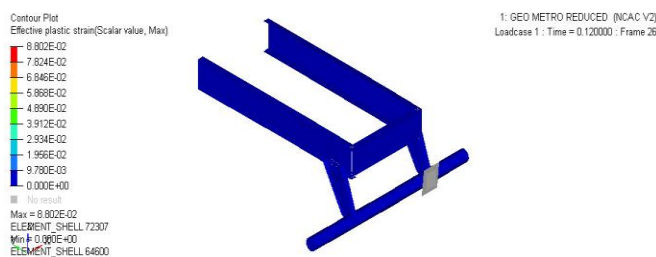
P1 RH: The effective plastic strain is shown in figure 2.78% peak strain was found in case of the P1 RH load case.
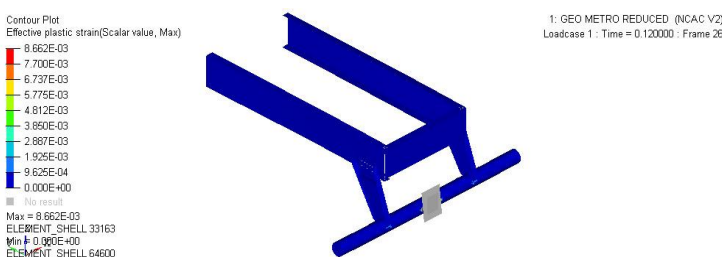

P2 LH: The effective plastic strain is shown in figure 8.53% peak strain was found in case of the P2 LH load case.

P2 RH: The effective plastic strain is shown in figure 8.80% peak strain was found in case of the P2 RH load case.

P3: The effective plastic strain is shown in figure 0.08% peak strain was found in case of the P3 load case.

CONCLUSION

Through study and analysis, Rear Under-Run Protection devices protect road users such as pedestrians and cyclists from slipping from the rear under the wheels of trucks and trailers. The basic objective is to improve the safety of the car and the occupants by designing the RUPD. The choice of material and the structural design are the two major factors for quasi-static analysis of RUPD in sequential loading. It is important to know the material & mechanical properties and failure mechanism during the impact. From the above results, the effective plastic strain for P1 load, P2 load, and P3 load is 2.15%, 2.78%, 8.53%, 8.80%, and 0.08% respectively which less than max. Percentage elongation of the material. As our Truck, the under-run protection device is within the limit.

REFERENCES

1. **IS-14812-2005** AUTOMOTIVE VEHICLES REAR UNDERRUN PROTECTIVE DEVICE GENERAL REQUIREMENTS (*First Revision)*

2. Dimensions of heavy duty vehicle from **EICHER PRO-5025.**

3. Practical aspects of Finite Element Analysis (Author **Nitin S Gokhale**).

4. ISO 6892, Metallic Materials-Tensile Testing. Road Accidents in India-2015, 'Transport research wing', Ministry of Road transport and Highways, Government of India, New Delhi, 2015.

5. Sumit Sharma, Shailesh Pawar, Dharmendra Patel, Sandeep Sharma, 'Finite Element Analysis of Rear Under-Run Protection Device (RUPD) With Sequential Loadings and Strain Mapping Method on Radioss', Altair Technology Conference, India, 2015.