Fatigue Analysis of Light Commercial Vehicle by Inertia Relief method

S.S.Sane  
Sr.Vice President  
Engineering and R&D  
Piaggio Vehicles Private Limited  
Baramati, Pune, India

Vyankatesh Madane  
Manager, Design  
Piaggio Vehicles Private Limited  
Baramati, Pune, India

Anandaraj.H  
Executive, Design  
Piaggio Vehicles Private Limited  
Baramati, Pune, India

Keywords: Stress, Finite element method, fatigue analysis, inertia relief

Abstract

This paper gives low cost and quick, durability virtual verification method which is validated by testing to get correlation. Durability analysis of frame and cabin is quite vast. As frame and cabin is suspended on suspension, ideally they are not fixed. In finite element analysis if model is not constrained it does not solve. If the frame is constrained at suspension attachment point and load applied, high stresses will be at attachment points which are not realistic. Normally frame/cabin tested on four posters by giving suspension forces at suspension mount while keeping vehicle free. To know co-relation of finite element analysis with testing, finite element analysis need to be done as similar to testing. In new vehicle development program, finite element analysis of frame and cabin is done by inertia relief method. Complete mesh model is made along with comm2 masses for seat, driver, engine, gear box, fuel tank and payload etc. In inertia relief analysis no need to fix the model, the SUPORT and SUPORT1 bulk data entries are used to define up to six reaction degrees of freedom of the free body. Inertia relief boundary conditions may be generated automatically by using PARAM, INREL, -2. Once model is ready, unit load (1kN) applied separately (different sub-cases) at all load transfer points like front-rear spring mountings, shock absorber mountings, rubber bushes on both side.

In detail durability analysis, stress result of this unit load sub cases is linear superimpose with the time series force data from track and damage is calculate for durability target (nos of repeats of track). However taking such time series forces is costly so in this analysis, stress results of unit loads were multiplied by maximum suspension forces and compare with endurance limit of material to see any risk of the crack. Few areas of potential crack identified in the analysis were matched with testing on track.

Introduction:

Past few decades product existence duration in the market has become shorter because of which companies are launching new models every year. In this scenario, it is necessary to reduce product development time. In the product development cycle, large amount of time gets consumed in fatigue testing. Elimination of some prototypes is a significant way of reducing fatigue testing time. To eliminate prototypes best way is to do virtual testing (FEA) similar to real fatigue testing. During past new developments in CAE field, one of the significant tool available is “INERTIA RELIEF ANALYSIS”. In real fatigue testing of a complete vehicle, it is free to move and force/displacement inputs are through wheels. Inertia relief analysis can do exactly same, apply forces at suspension mountings with no constrains in the model. Technical detail of inertia relief analysis is as in next paragraph.
Inertia relief analysis is an advanced option that allows to simulate unconstrained structures in a static analysis. A typical application of inertia relief includes modeling an aircraft in flight, an automobile on a test track, or a satellite in space. In Static analysis it is assumed that the model contains no mechanisms and may not move as a rigid body (strain free). If either of these conditions exists in a conventional finite element analysis, the stiffness matrix for the model becomes singular.

Consequently, conventional finite element static analysis cannot be performed on unconstrained structures. However, a method called inertia relief is provided in RADIOSS for analyzing these conditions. A simple description of inertia relief is that the inertia (mass) of the structure is used to resist the applied loadings, that is, an assumption is made that the structure is in a state of static equilibrium even though it is not constrained. In an inertia relief analysis, the applied loads are balanced by a set of translational and rotational accelerations generated automatically by the software. These accelerations provide forces, distributed over the structure in such a way that the sum of the applied forces on the structure is zero.

The SUPORT and SUPORT1 bulk data entries are used to define up to six reaction degrees of freedom of the free body. If all possible rigid body motion is not described on the SUPORT entry, then the stiffness matrix is singular, and the problem either fails in decomposition or gives unreasonable answers.

**Process Methodology:**

An inertia relief analysis has been carried out in RADIOSS to determine the stress and displacement levels. From the results obtained for initial design, the potential crack areas were identified. Simultaneously two vehicles with initial design were tested in pave track. The failure areas in simulation were matched with the test vehicles. To avoid failures different iterations were carried out by adding stiffener, increasing the thickness of the component, modified design of the components, etc. The final modifications were implemented in the vehicle as per the simulation and the vehicle was retested to confirm the strength. The step-by-step procedure is shown below.
Figure 1: Overview of chassis analysis.
Numerical Simulation Approach:

Mesh Model:

A chassis component has been meshed with fine elements to capture all the details and critical areas while cabin, load tray & other parts were coarse meshed. Sheet metal parts and CO2 welds were modeled by thin shell (quad-4, tria-3) elements with average element size of 8mm. Spot-weld, bolt connections and other links modeled by 1-D beam and rigid elements in HYPERMESH. The quality of the mesh is of major importance for the final result. Therefore quite some time was spent to verify the connectivity, warpage and aspect ratio of the elements.

![FE model of chassis](image)

Boundary Conditions & Loading Condition:

Inertia relief boundary conditions may be defined in the bulk data section of the input deck or they may be determined automatically by the solver. The SUPORT and SUPORT1 bulk data entries are used to define up six degrees of freedom of the free body. SUPORT entries will be used in all relevant subcases and therefore do not need to be referenced in the subcase information section. SUPORT1 entries need to be referenced by SUPORT1 data statement for use within a subcase. Inertia relief boundary conditions may be generated automatically by using PARAM, INREL, -2. In this analysis the boundary condition was generated automatically.

There are a number of forces acting on several places. All these forces have different magnitude. Therefore forty two different load cases were created, each case with its own importance of load. All these forces applied on suspension mountings. These different load cases represent the forces when braking, cornering, accelerating, vehicle goes on ditch etc. The magnitude of the applied forces in each point was 1KN. In HYPERVIEW depends upon the severity of the load at the corresponding point the 1KN was multiplied by scale factor. To simulate some of the load case like front LH wheel in ditch, rear LH wheel in ditch, front both wheels in ditch, rear both wheels in ditch, front LH & rear LH wheels in ditch and front LH & rear RH wheels in ditch, linear superimpose was done in HYPERVIEW.
Results & Discussion:

Stress results for unit load were multiplied by relevant scale factor in HyperView. These scale factors were maximum loads during vehicle running on track. As loads acting simultaneously on vehicle, a stress result multiplies with scale factor and linear superimposed in HyperView. Final stress results were compared with endurance strength of material to check any possibility of crack. For different grades of steel, different endurance strength values were used. Finally critical locations/parts were identified. Design modifications & FE iterations were done to avoid failures. As per final iteration one proto vehicle was made and re-tested for the final confirmation.

Correlation of CAE V/S Testing:

It is very essential for a CAE engineer to establish correlation between physical test and the simulation at very early stage of analysis. The correlation of numerical simulation and the actual test were established by comparing the failure areas in simulation & testing. In simulation five failure areas were found and in the testing four failures were found as similar to simulation. The failure location in simulation & testing are compared as shown below

<table>
<thead>
<tr>
<th>Failure location</th>
<th>CAE</th>
<th>Testing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Body mounting bracket</td>
<td>Failed</td>
<td>Failed</td>
</tr>
<tr>
<td>Cross member</td>
<td>Failed</td>
<td>Failed</td>
</tr>
<tr>
<td>Cross member stiffener</td>
<td>Failed</td>
<td>Failed</td>
</tr>
<tr>
<td>Chassis &amp; Cabin joining area</td>
<td>Failed</td>
<td>Failed</td>
</tr>
<tr>
<td>Suspension bracket</td>
<td>Failed</td>
<td>Not failed</td>
</tr>
</tbody>
</table>

*Table1: Correlation between CAE & Testing*

Due to confidentiality, only one failure location is shown in figure-3. This figure shows the failure in cabin & chassis joining location in initial design.

*Figure 3: failure location (cabin & chassis joining area)*
Benefits Summary:
The failures were avoided in the final iteration by adding stiffeners, changing the design of the components, etc.
Testing iterations and thus development time reduced on chassis design by using FEA.

Future Plans:
Extracting time series force data from track either by measurement or by MBD system to do damage calculation.

Conclusions:
Numerical simulation methodology discussed here is very effective in predicting the failure location in light commercial vehicles chassis. Failure locations in simulation & testing were same. With the help of CAE it is possible to reduce number of physical tests and thereby reducing design cycle time.

REFERENCES
[1] Altair HyperWorks 10.0 user manual
[2] Nitin S. Gokhale "Practical Finite Element Analysis", Finite to Infinite