# Random Vibration and Transient Bump Analysis of Automotive Heavy Vehicle Levelling frame

[1]Asst Prof.B.Anil Kumar, [2]S.Sheshikanth Reddy, [3]Prof. VVRLS Gangadhar, [4]Associate Prof.Anoop Kumar

[1]*Asst Professor, Department of Mechanical Engineering, Princeton College of Engineering & Technology, Hyderabad, India.*
[2]*M.Tech Student, Department of Mechanical Engineering, Princeton College of Engineering & Technology, Hyderabad, India.*
[3]*Professor, Department of Mechanical Engineering, Princeton College of Engineering & Technology, Hyderabad, India.*
[4]*Associate Professor, Department of Mechanical Engineering, JBIET College of Engineering & Technology, Hyderabad, India.*

***Abstract:*** *A heavy vehicle automobile is equipped with body and chassis. The chassis is basic structure of a vehicle. It contains all the engine parts and power systems and the frame is the main portion of chassis on which the entire load will be mounted. For efficient usage of the frame space and to ease the mounting of required equipment and maintain the horizontal leveling of them, a suitable frame is designed. This paper describes design modeling optimization and structural analysis of heavy vehicle levelling frame. Weight reduction is the prime objective of this paper through analysis. In the present work, the existing heavy vehicle levelling frame is studied and redesigned. The new leveling frame is validated through analysis using ANSYS. TATA 715 vehicle is taken for modeling and analysis. The vehicle leveling frame is modeled in SOLID WORKS 2014 and then it is imported to ANSYS 15.0. The analysis is done with the new frame with all the loads being considered. The design constraints are stresses and deformations. The results is studied to validate the optimized leveling frame.*
***Keywords:*** *Levelling frame, Auto CAD 2014, SOLID WORKS 2014, ANSYS 15.0, TATA 715*

## I.    Introduction

Automotive chassis is a French word that was initially used to represent the basic structure. It is a skeletal frame on which various mechanical parts like engine, tires, axle assemblies, brakes, steering etc. are bolted. It gives strength and stability to the vehicle under different conditions. At the time of manufacturing, the body of a vehicle is flexibly molded according to the structure of chassis. Automobile chassis is usually made of light sheet metal or composite plastics. It provides strength needed for supporting vehicular components and payload placed upon it. Automotive chassis or automobile chassis helps keep an automobile rigid, stiff and unbending. It ensures low levels of noise, vibrations and harshness throughout the automobile. Automobile chassis without the wheels and other engine parts is called frame. Automobile frames provide strength and flexibility to the automobile. The backbone of any automobile, it is the supporting frame to which the body of an engine, axle assemblies are affixed. Tie bars that are essential parts of automotive frames are fasteners that bind different auto parts together. Automotive frames are basically manufactured from steel. Aluminum is another raw material that has increasingly become popular for manufacturing these auto frames. In an automobile, front frame is a set of metal parts that forms the framework which also supports the front wheels.

**1.1 Types of frames:**
There are three types of frames
1. Conventional frame
2. Integral frame
3. Semi-integral frame

**1.1.1 Conventional frame** It has two long side members and 5 to 6 cross members joined together with the help of rivets and bolts. The frame sections are used generally. a. Channel Section – Good resistance to bending b. Tabular Section – Good resistance to Torsion c. Box Section – Good resistance to both bending and Torsion

**1.1.2 Integral frame** This frame is used now in most of the cars. There is no frame and all the assembly units are attached to the body. All the functions of the frame carried out by the body itself. Due to elimination of long frame it is cheaper and due to less weight most economical also. Only disadvantage is repairing is difficult.

**1.1.3 Semi – Integral frame** In some vehicles half frame is fixed in the front end on which engine gear box and front suspension is mounted. It has an advantage when the vehicle is met with an accident the front frame can be taken easily to replace the damaged chassis frame. This type of frame is used in American and European cars.

**1.2 Functions of the frame**
1. To carry load of the passengers or goods carried in the body.
2. To support the load of the body, engine, gear box etc.,
3. To with stand the forces caused due to the sudden braking or acceleration.
4. To with stand the stresses caused due to the bad road condition.
5. To with stand centrifugal force while cornering.

**1.3 Various loads acting on the frame**
1. Short duration Load – While crossing a broken patch.
2. Momentary duration Load – While taking a curve.
3. Impact Loads – Due to the collision of the vehicle.
4. Inertia Load – While applying brakes.
5. Static Loads – Loads due to chassis parts.

## II. Problem Specification

Weight reduction is now the main issue in automobile industries. Because if the weight of the vehicle increases the fuel consumption increases. At the same time as the weight of the vehicle increases the cost also increases which becomes a major issue while purchasing an automobile. The frame considered here is an additional frame apart from vehicle chassis frame which will serve the main purpose of levelling and also mounting of all the required equipment. It is manufactured with Structural Steel. The equipment that are mounted on the leveling frame comprises of masts(2 No's) upon which high precision antennae will be mounted, shelter, and leveling jacks(4 No's). The conventional frame is designed with more factor of safety and less regard to the weight and hence the new frame will cater to the load distribution and also keeping acceptable factor of safety limits in to consideration. Structural, Random vibration and Transient analysis are performed on the leveling frame in order to ascertain the factor of safety limits.

**2.1 Specifications of Existing Heavy Vehicle TATA 1109 EX2 Frame**

| Sl. No. | Description | Dimension (mm) |
|---|---|---|
| 1 | Length | 3825 |
| 2 | Width | 2300 |
| 3 | Height | 75 |
| 4 | Weight of conventional frame | 660 Kg. |
| 5 | Material | Structural Steel Mild |

1. Structural Steel : Material Data

**TABLE1: Structural Steel Constants**

| Density | 7850 kg m^-3 |
|---|---|
| Coefficient of Thermal Expansion | 1.2e-005 C^-1 |
| Specific Heat | 434 J kg^-1 C^-1 |
| Thermal Conductivity | 60.5 W m^-1 C^-1 |
| Resistivity | 1.7e-007 ohm m |

**TABLE2: Structural Steel Compressive Yield Strength**

| Compressive Yield Strength Pa |
|---|
| 2.5e+008 |

**TABLE3: Structural Steel Tensile Yield Strength**

| Tensile Yield Strength Pa |
|---|
| 2.5e+008 |

**TABLE4: Structural Steel Tensile Ultimate Strength**

| Tensile Ultimate Strength Pa |
|---|
| 4.6e+008 |

**TABLE5: Structural Steel Strain-Life Parameters**

| Strength Coefficient Pa | Strength Exponent | Ductility Coefficient | Ductility Exponent | Cyclic Strength Coefficient Pa | Cyclic Strain Hardening Exponent |
|---|---|---|---|---|---|
| 9.2e+008 | -0.106 | 0.213 | -0.47 | 1.e+009 | 0.2 |

**TABLE6: Structural Steel Isotropic Elasticity**

| Temperature C | Young's Modulus Pa | Poisson's Ratio | Bulk Modulus Pa | Shear Modulus Pa |
|---|---|---|---|---|
| | 2.e+011 | 0.3 | 1.6667e+011 | 7.6923e+010 |

**TABLE7: Structural Steel Isotropic Relative Permeability**

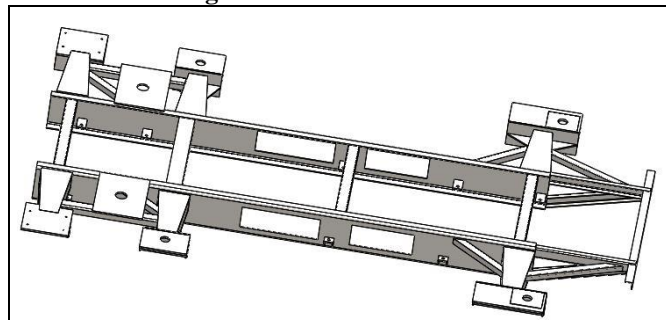| Relative Permeability |
|---|
| 10000 |

**2.1 Figure showing assembly of conventional frame on the Truck**
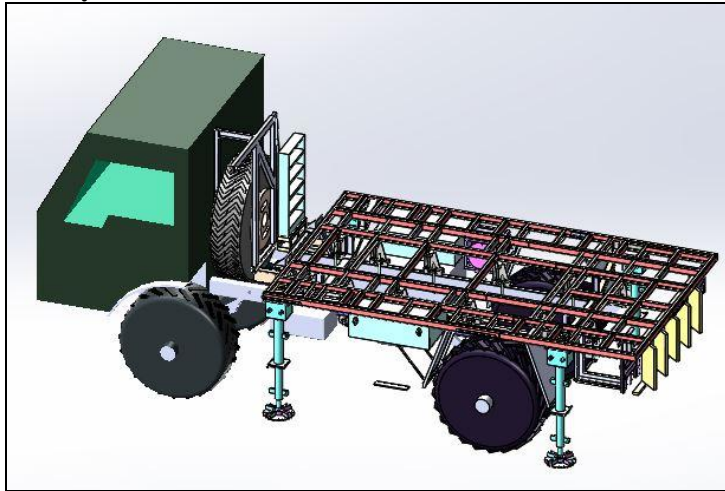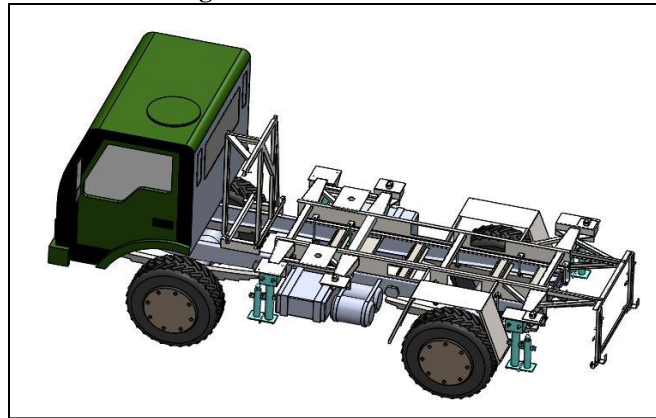


**2.2 Figure showing Conventional frame**
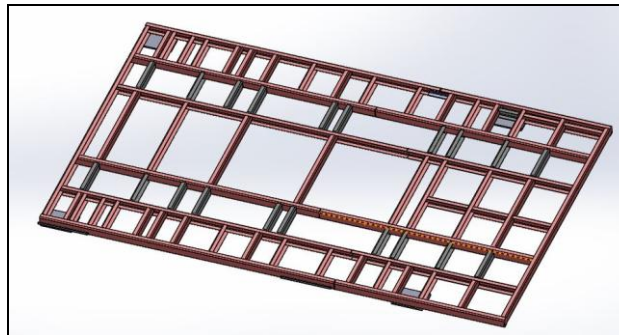


**2.3 Figure showing new modified levelling frame**

**2.4 Figure showing new modified levelling frame assembled on the truck**



## III. Design

Design may be done in two ways one way is the component design which is done by improving the existing ones. The other is conceptual design where there is no reference and creation of new machines. A new or better machine is one which is more economical in the overall cost of production and operation. The process of design is a long and time consuming one. From the study of existing ideas, a new idea has to be conceived. The idea is then studied keeping in mind its commercial success and given shape and form in the form of drawings. In the preparation of these drawings, care must be taken about the availability of resources like money, man power and materials required for the successful completion of the new idea into an actual reality. In designing a machine component, it is necessary to have a good knowledge of many subjects such as Mathematics, Engineering Mechanics, Strength of Materials, Theory of Machines, Workshop Processes and Engineering Drawing. Generally the design of a component involves various steps in it. Initially, the drawings must be drawn in user friendly software and they must be converted into a 3D model. This 3D model must be imported into an analyzing medium where it is structurally or thermally analyzed to sustain the need. Different steps involved in designing a component are

1. Part drawing
2. Modeling
3. Structural analysis

The present frame is divided in to individual components and each component is drawn, modeled and structurally analyzed by using software and its procedure is explained as below.

**3.1 Part Drawing:** It is a document that includes the specifications for a part's production. Generally the part drawings are drawn to have a clear idea of the model to be produced. The part drawing of the entire frame is drawn with all the views in AUTO CAD 2014.
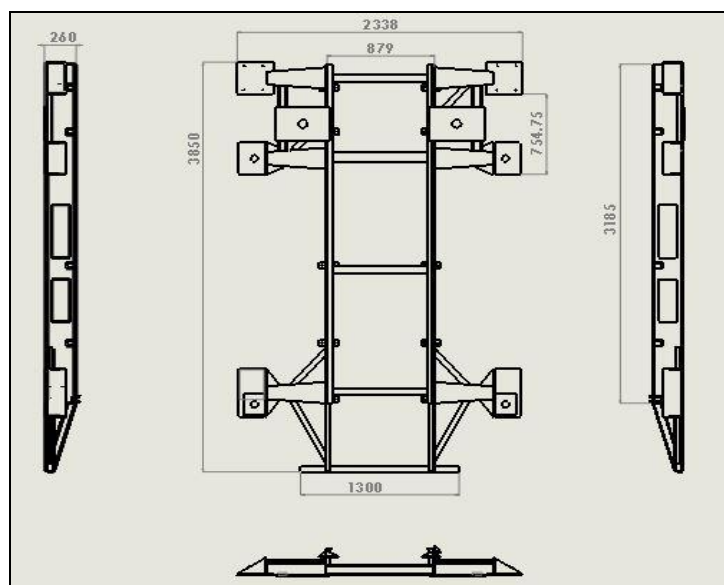
### 3.2 Modeling

It is the process of developing a mathematical representation of any three-dimensional surface of object via specialized software. The product is called a 3D model.

There are three basic types of three dimensional computer geometric modeling methods

1. Wire frame modeling
2. Surface modeling
3. Solid modeling

### 3.2.1 Wire Frame Modeling

A wire frame model is visual presentation of a 3 dimensional or physical object used in 3D computer graphics. It is created by specifying each edge of the physical object where two mathematically continuous smooth surfaces meet, or by connecting an object's constituent vertices using straight lines or curves. The object is projected on to a display screen by drawing lines at the location of each edge. The wire frame format is well suited and widely used in programming tool paths for DNC (Direct Numerical Control) machine tools.

### 3.2.2 Surface Modeling

These models are with no thickness. These models are widely used in industries like automobiles, aerospace, plastic, medical and so on. Surface models should not be confused with thick models that are models having mass properties. The only difference between the soli model and the surface model is solid model will have mass properties. The present frame is modeled by solid modeling because of its ease in construction and realistic profile.

### 3.2.3 Solid Modeling

A solid model of an object is a more complete representation than its surface (wireframe) 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 Solid works and Pro.E. In this project the frame is modeled by using SOLID WORKS 2014. All the parts that are required for constructing the frame are modeled in part module by using different commands like extrude, rotate, loft, fillet, extrude cut etc..,

### 3.3 Assembly

The components that are generated in part module are imported to assembly module and by using 'insert components' command and all these components are mated together to form the required assembly.

### 3.4 Structural Analysis of the frame:

It is the methodology of determining the effects of loads on physical structures and their components. Structures subject to this type of analysis include buildings, bridges, vehicles, machinery, furniture, attire, soil strata, prostheses and biological tissue. Structural analysis incorporates the fields of applied mechanics, materials science and applied mathematics to compute a structure's deformations, internal forces, stresses, support reactions, accelerations, and stability. The results of the analysis are used to verify a structure's fitness for use, often saving physical tests. Structural Analysis is thus a key part of the engineering design of structures. The present frame model is imported to ANSYS Workbench 15.0 through the interface link established in the SOLIDWORKS 2014 software. There are various steps that are to be followed in analyzing a component structurally. They are

1. Mesh generation
2. Fixed supports
3. Application of loads
4. Evaluating result

### 3.4.1 Mesh Generation

The process for generating a mesh consists of three general steps

1. Set the element attributes
2. Set mesh controls (optional)
3. Meshing the model

It is not always necessary to set mesh controls because the default mesh controls are appropriate for many models. If no controls are specified, the program will use the default settings (DESIZE) to produce a free mesh. Alternatively, you can use the advanced Size feature controls to produce a better quality free mesh. ANSYS Meshing technology provides multiple methods to generate a pure hex or hex- dominant mesh.

Depending on the model complexity, desired mesh quality and type, and the time available to perform meshing, ANSYS Meshing provides a scalable solution. Quick automatic hex or hex-dominant mesh can be generated for optimal solution efficiency and accuracy.

Structural analysis is divided in to two parts:
1. Static Structural analysis: Here loads will be static and does not vary with time. i.e., loads are applied on the structure and the steady state response of the structure is observed.
2. Dynamic Structural analysis: Loads in this case vary over a period of time. i.e, both frequency and amplitude of the loads are varied to evaluate the response of the structure. To account for these variations Random and Transient dynamic analysis are performed.

The Dynamic structural analysis performed here are:

**Modal analysis**: For calculating the natural frequencies and mode shapes of the structure.

A modal analysis typically is used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or machine component in the design stage. It also can serve as a starting point for another more-detailed dynamic analysis, such as harmonic response or full transient dynamic analysis

**Random Vibration Analysis**:

Random vibration analyses are used to determine the response of structures to random or time dependent loading conditions, such as earthquakes, wind loads, ocean wave loads, jet engine thrust, rocket motor vibrations, and more. The random vibration analysis can be much less computationally intensive than a full transient dynamic analysis while still providing design-guiding results.

Types of random vibration analyses include:

**Response spectrum**: single and multi-point base excitation in which the results of a modal analysis are used with a known spectrum to calculate the model's displacements and stresses

**Dynamic design analysis method (DDAM):** a technique used to evaluate the shock resistance of shipboard equipment. It is similar to the response spectrum method except the loading is derived from empirical equations and shock design tables.

**Power spectral density**: a statistical-based random vibration method in which the input load histories are specified based on a probability distribution of the loading taking that particular value. An example application is the calculation of the probable response of a sensitive automotive electronic component to the engine and drive train vibrations while the vehicle travels rapidly down a rough road.

**Transient Dynamic Analysis**:

Transient dynamic analysis (sometimes called time history analysis) is a technique used to determine the dynamic response of a structure under the action of any general time-dependent loads. Transient dynamic analyses are used to determine the time-varying displacements, strains, stresses and forces in a structure as it responds to any combination of static and time-varying loads while simultaneously considering the effects of inertia or damping.

Transient dynamic analysis in ANSYS software can be broadly classified as one of two types:

**Rigid dynamics**: In an assembly, all parts are considered to be infinitely stiff, no mesh is required and a special solver is used to drastically reduce the amount of computational resources required. The primary focus of a rigid dynamics simulation is mechanism operation, part velocities and accelerations, as well as joint forces encountered during the range of mechanism motion.

**Flexible dynamics:** Some or all parts of an assembly are meshed and considered flexible based on the materials from which they are made. A flexible dynamics simulation typically is done after a rigid dynamics simulation is used to verify the model setup. Flexible dynamics simulation can provide information about machine performance, such as: • Will a machine or mechanism work adequately with light, more-flexible members, or will stiffer but heavier members be required? • Will the forces transmitted through joints exceed the strength of the materials being used? • At what rotational or translation speed will the mechanism experience plastic deformation and begin to fail? • Will the mechanism's natural frequencies be excited and lead to instability? • Will the repeated loading/unloading lead to fatigue and, if so, where?

Here in this project the transient analysis is performed as BUMP test where the structure is given 25g(g-gravitation acceleration) acceleration force for 6 milli seconds(as per JSS 55555 specifications) . This acceleration force of 25g for 6 milli seconds signifies one BUMP, and the structure is subjected to 3 BUMPS

per one second. The load of 25g acceleration force is normally distributed having a maximum amplitude of 25g at 3 milli seconds. The response of the structure is then observed for further design considerations.

**Random Vibration:**

Random vibration is somewhat of a misnomer. If the generally accepted meaning of the term "random" were applicable, it would not be possible to analyze a system subjected to "random" vibration. Furthermore, if this term were considered in the context of having no specific pattern (i.e., haphazard), it would not be possible to define a vibration environment, for the environment would vary in a totally unpredictable manner. Fortunately, this is not the case. The majority of random processes fall in a special category termed stationary. This means that the parameters by which random vibration is characterized do not change significantly when analyzed statistically over a given period of time - the RMS amplitude is constant with time. For instance, the vibration generated by a particular event, say, a missile launch, will be statistically similar whether the event is measured today or six months from today. By implication, this also means that the vibration would be statistically similar for all missiles of the same design. The assumption of a stationary process is essential in both a technical and legal sense. As previously stated, it would not be possible for a designer to analyze a system, nor for a user to test a system prior to installation in the field, if the vibration excitation were not stationary. Consequently, it would not be possible to develop a legally binding specification. Any vibration is described by the time history of motion, where the amplitude of the motion is expressed in terms of displacement, velocity or acceleration. Sinusoidal vibration is the simplest motion, and can be fully described by straightforward mathematical equations. Figure 1 shows the amplitude time plot of a sinusoidal vibration, and indicates that sinusoidal vibration is cyclic and repetitive. In other words, if frequency and amplitude (or time and amplitude) are defined, the motion can be predicted at any point in time.
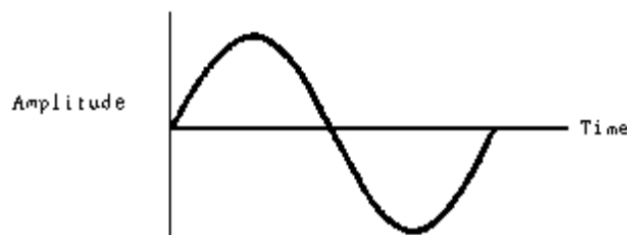


**Figure 1.** Amplitude-Time History of Sinusoidal Vibration

A random vibration is one whose absolute value is not predictable at any point in time. As opposed to sinusoidal vibration, there is no well defined periodicity - the amplitude at any point in time is not related to that at any other point in time. Figure 2 shows the amplitude time history of a random vibration. The lack of periodicity is apparent. A major difference between sinusoidal vibration and random vibration lies in the fact that for the latter, numerous frequencies may be excited at the same time. Thus structural resonances of different components can be excited simultaneously, the interaction of which could be vastly different from sinusoidal vibration, wherein each resonance would be excited separately.
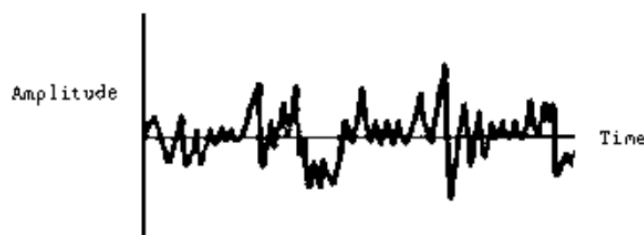


**Figure 2.** Amplitude-Time History of Random Vibration

Although the instantaneous amplitude of a random vibration cannot be expressed mathematically as an exact function of time, it is possible to determine the probability of occurrence of a particular amplitude on a statistical basis.

To characterize a stationary process, an ensemble of possible time histories must be obtained, wherein the amplitude is measured over the frequency range of excitation. Thus, the three parameters of interest are: frequency, time and amplitude. This information would provide the ability to analyze a random process in a statistical sense. The characterization of random vibration typically results in a frequency spectrum of Power Spectral Density (PSD) or Acceleration Spectral Density (ASD), which designates the mean square value of

some magnitude passed by a filter, divided by the bandwidth of the filter. Thus, Power Spectral Density defines the distribution of power over the frequency range of excitation.

The equipment designer is interested in avoiding mechanical failure and equipment malfunction. These may be produced by different mechanisms, requiring different methods of corrective action. To the designer, random vibration could be considered as either:
a) An infinite number of harmonic vibrations with unpredictable amplitude and phase relationships in the frequency domain; or
b) The sum of an infinite number of infinitesimal shocks occurring randomly in the time domain.

In the first case, response at a particular frequency may be the primary concern. For example, when a displacement sensitive device is excited at its natural frequency, relatively large displacements may result in malfunction. In such a case, the malfunction might be corrected by reducing the amplitude of excitation at the particular frequency of concern - the natural frequency of the device. This might be accomplished by inserting a vibration isolator between the source of excitation and the device. Alternatively, displacement might be reduced by adjusting the stiffness of the device, or by increasing damping at the natural frequency of the device.

If the random vibration is considered as an infinite number of infinitesimal shocks, the overall Grms may result in a fatigue related structural failure of a component due to the intermittent shocks associated with the random excitation. In this case, the problem might be corrected by reducing the overall Grms or by increasing the strength of the component.

There is a relationship between these considerations. The nature of the equipment problem will affect the type of corrective action to be investigated.

**Characterization of Input Acceleration And Displacement**
The simplest random excitation to analyze is a band limited white spectrum shown in Figure 4.
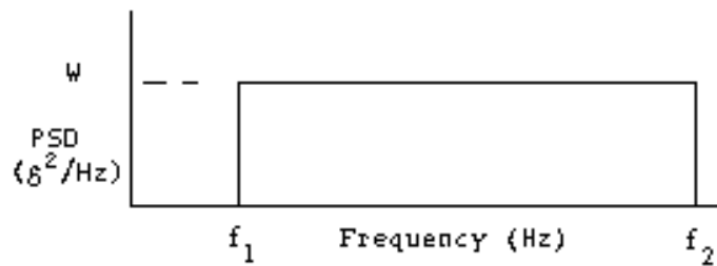


**Figure 4.** Band Limited White Spectrum

The overall input Grms is the square root of the area under the curve, i.e.,

$$Grms = \sqrt{W \, (f_2 - f_1)}$$

This value could be used in Equation 1 to predict the probability of occurrence of instantaneous values of acceleration for a random signal. For design purposes, Grms would generally be multiplied by three to provide three sigma values.
In actual practice, it is not possible for a vibration shaker to generate the instantaneous drop off shown in Figure 4. Consequently, specifications generally show a roll off rate in terms of decibels per octave (dB/Oct.).
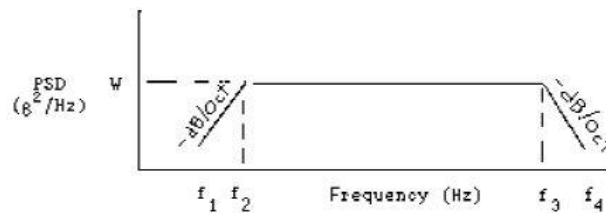Figure 5 shows a possible input vibration spectra including roll-off rates.



Figure 5. Possible Random Vibration Input Sectrum

As in Figure 4, the overall Grms is the square root of the total area under the curve. However, since most curves are plotted as straight lines on log-log paper, calculating the area under the sloped lines is more complicated than for the region of constant PSD. To determine Grms for a spectrum with numerous break points, G2 . for all areas are summed, and the square root of this summation results in overall Grms.
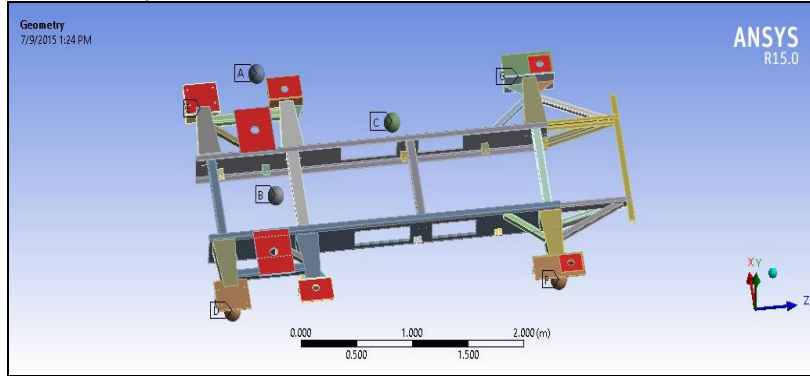
### 3.1 Model imported in to Ansys:



**TABLE8: Geometry - Point Masses**

| Object Name | mast1 'A' | mast2 'B' | shelter load 'C' | jack1 'D' | | jack2 'E' | jack3 'F' | jack4 'G' |
|---|---|---|---|---|---|---|---|---|
| State | Fully Defined | | | | | | | |
| Scope | | | | | | | | |
| Scoping Method | Geometry Selection | | | | | | | |
| Applied By | Remote Attachment | | | | | | | |
| Geometry | 1 Face | | 4 Faces | 1 Face | | | | |
| Coordinate System | Global Coordinate System | | | | | | | |
| X Coordinate | 1.078 m | -0.183 m | 0.4475 m | -0.4815 m | | 1.4774 m | | |
| Y Coordinate | 0.6 m | | | 1.0431e-010 m | -1.2e-002 m | | | |
| Z Coordinate | 0.55975 m | | 1.7 m | 0.13 m | | | | |
| Location | Defined | | | | | | | |
| Definition | | | | | | | | |
| Mass | 150. kg | | 1800. kg | 50. kg | | 45. kg | | |
| Suppressed | No | | | | | | | |
| Behavior | Deformable | | | | | | | |
| Pinball Region | All | | | | | | | |

### 1)   MeshTABLE9: Mesh

| Statistics | |
|---|---|
| Nodes | 59029 |
| Elements | 12761 |
| Mesh Metric | None |
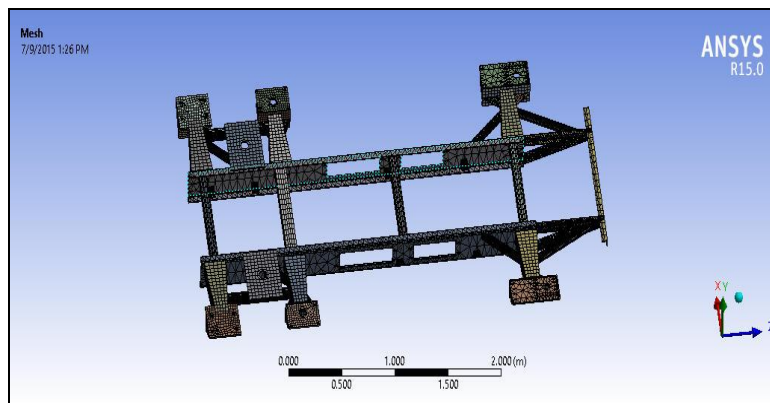
## B. Static Structural (H5)

### TABLE10: Static Structural-Accelerations

| | |
|---|---|
| Object Name | Standard Earth Gravity |
| State | Fully Defined |
| **Scope** | |
| Geometry | All Bodies |
| **Definition** | |
| Coordinate System | Global Coordinate System |
| X Component | 0. m/s² (ramped) |
| Y Component | -9.8066 m/s² (ramped) |
| Z Component | 0. m/s² (ramped) |
| Suppressed | No |
| Direction | -Y Direction |

### TABLE11: Static Structural- Loads

| | |
|---|---|
| Object Name | Fixed Support |
| State | Fully Defined |
| **Scope** | |
| Scoping Method | Geometry Selection |
| Geometry | 2 Faces |
| **Definition** | |
| Type | Fixed Support |
| Suppressed | No |
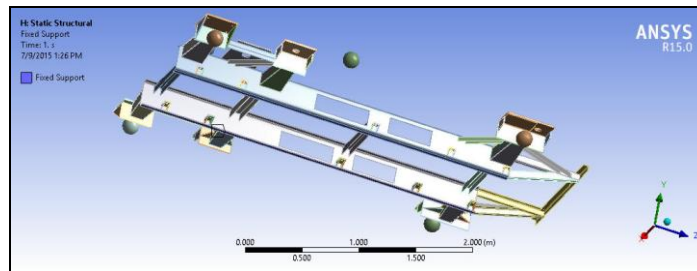


**Results:**

### TABLE12: Static Structural Solution-Results

| Object Name | Total Deformation | Equivalent Stress |
|---|---|---|
| State | Solved | |
| **Scope** | | |
| Geometry | All Bodies | |
| **Definition** | | |
| Type | Total Deformation | Equivalent (von-Mises) Stress |
| **Results** | | |
| Minimum | 0. m | 444.2 Pa |
| Maximum | 1.884e-003 m | 5.9897e+008 Pa |
| Minimum Occurs On | c- runner-1 | |
| Maximum Occurs On | Assem3-1@2721 130 8 TOP PLATE-1 | |
| **Minimum Value Over Time** | | |
| Minimum | 0. m | 444.2 Pa |
| Maximum | 0. m | 444.2 Pa |
| **Maximum Value Over Time** | | |
| Minimum | 1.884e-003 m | 5.9897e+008 Pa |
| Maximum | 1.884e-003 m | 5.9897e+008 Pa |
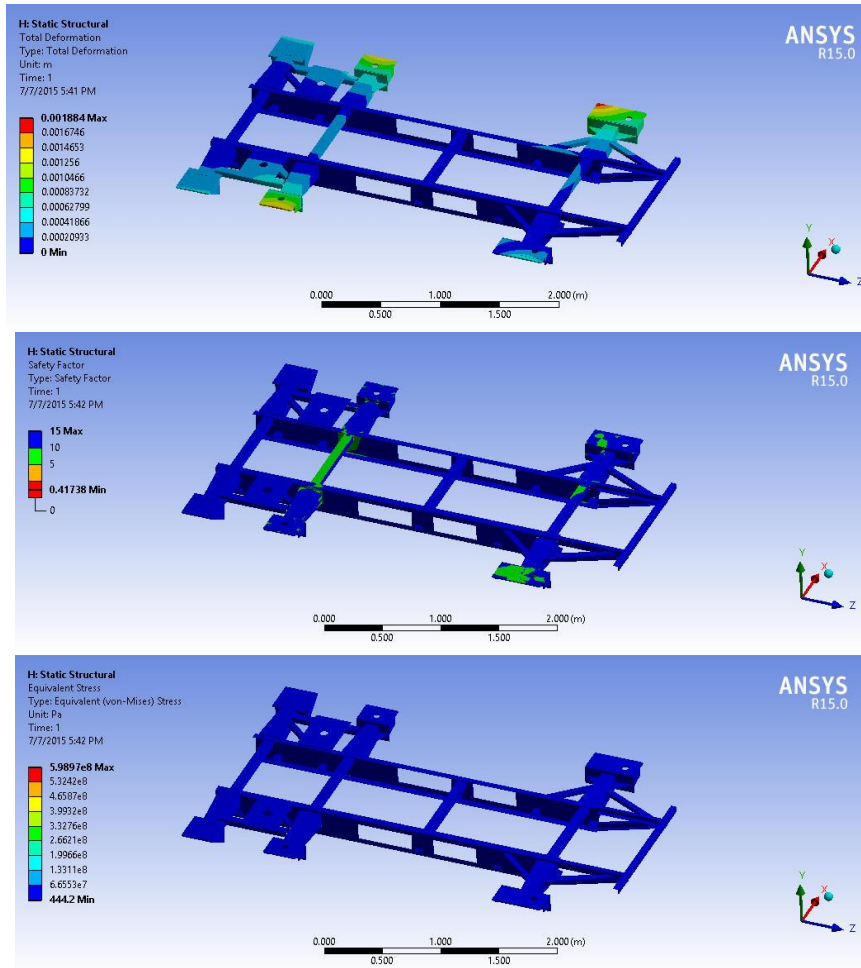
**Modal**

**TABLE13:  Modal- Analysis Settings**

| Object Name | Analysis Settings |
|---|---|
| State | Fully Defined |
| **Options** | |
| Max Modes to Find | 6 |

The following bar chart indicates the frequency at each calculated mode.
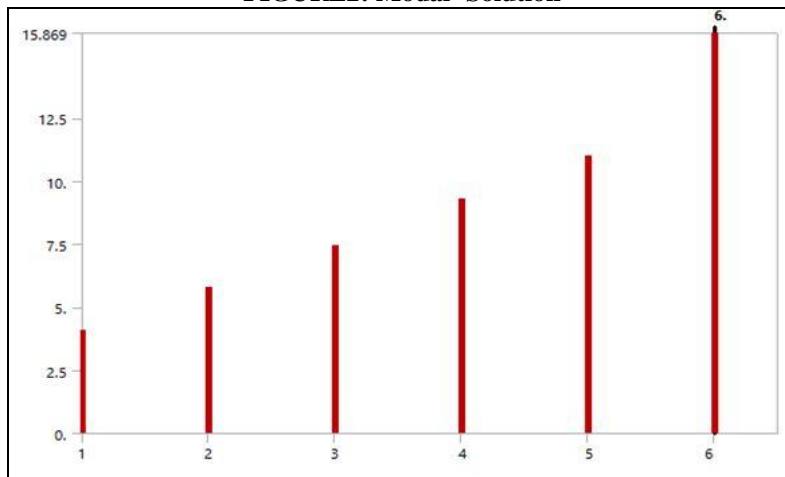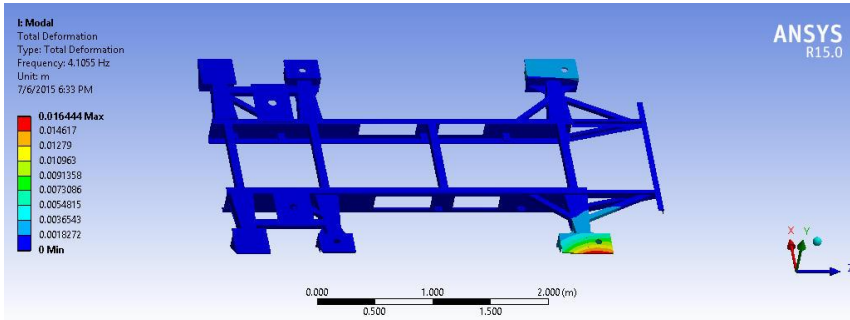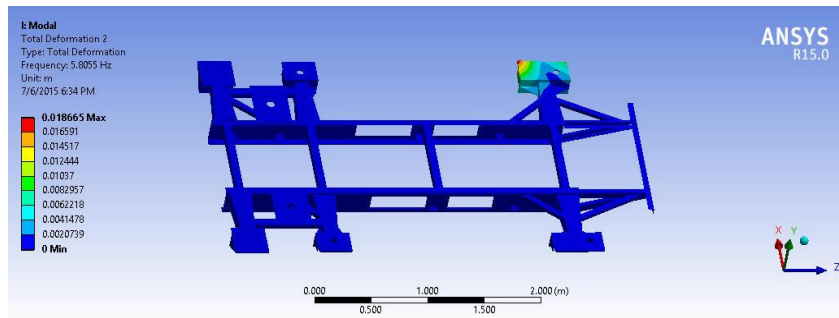
**FIGURE2: Modal -Solution**

**TABLE14: Modal-Solution**

| Mode | Frequency [Hz] |
|------|----------------|
| 1.   | 4.1055         |
| 2.   | 5.8055         |
| 3.   | 7.4631         |
| 4.   | 9.3161         |
| 5.   | 11.024         |
| 6.   | 15.869         |

**Mode 1:**



**Mode:2**



**Mode : 3**
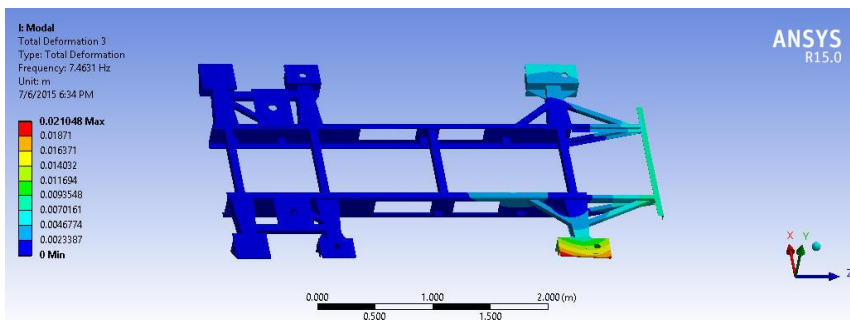


**Mode : 4**
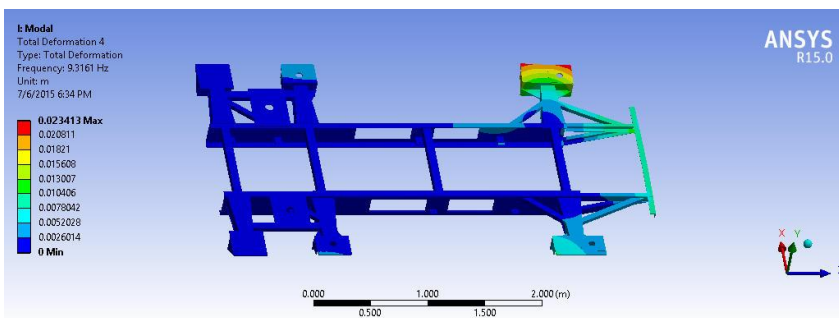


**Mode : 5**
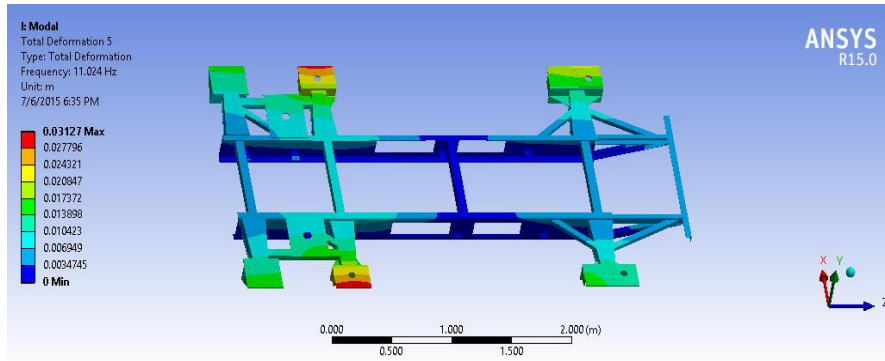
**Mode : 6**
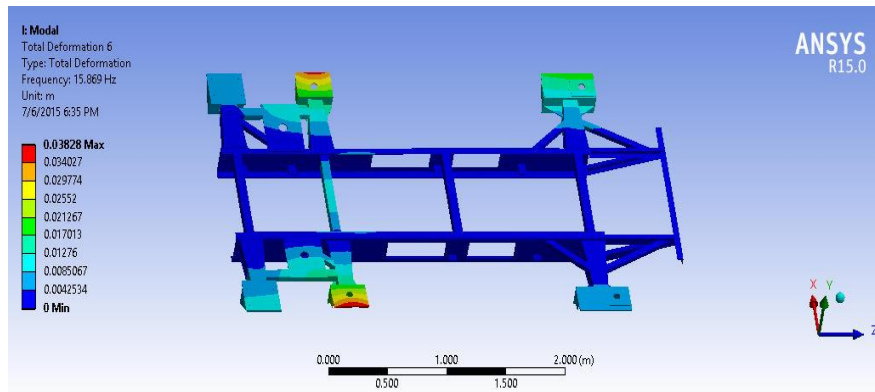


## C. Transient

**TABLE15: Transient Analysis Settings**

| Object Name | Analysis Settings |
|---|---|
| State | Fully Defined |
| **Step Controls** | |
| Number Of Steps | 1. |
| Current Step Number | 1. |
| Step End Time | 1. s |
| Auto Time Stepping | Off |
| Define By | Substeps |
| Number Of Substeps | 1000. |
| Time Integration | On |

**TABLE16:  Transient - Accelerations**

| Object Name | Acceleration |
|---|---|
| State | Fully Defined |
| **Scope** | |
| Geometry | All Bodies |
| **Definition** | |
| Define By | Components |
| Coordinate System | Global Coordinate System |
| X Component | Tabular Data |
| Y Component | Tabular Data |
| Z Component | Tabular Data |
| Suppressed | No |

**FIGURE3: Transient (J5)- Acceleration**
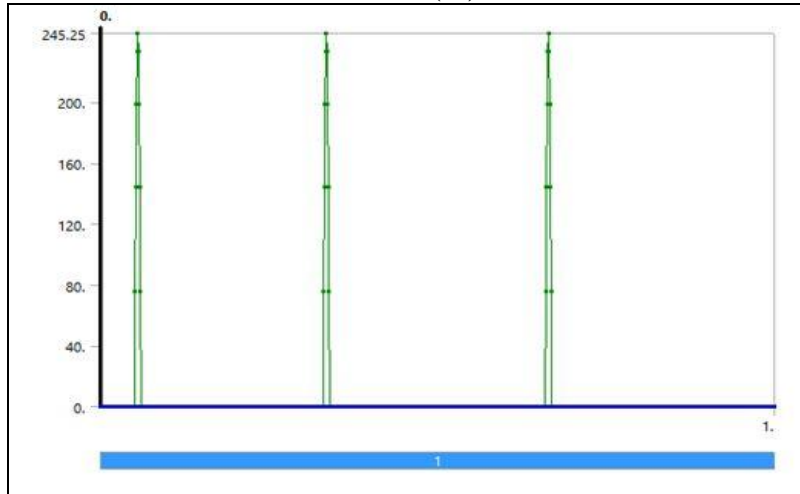


**Solution**

**TABLE17: Transient Solution -Results**

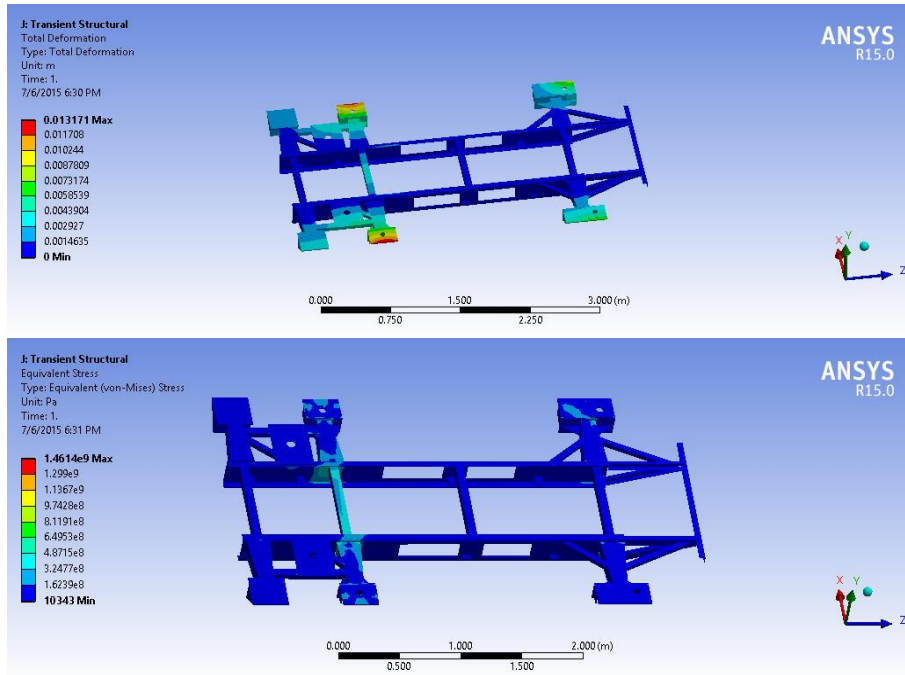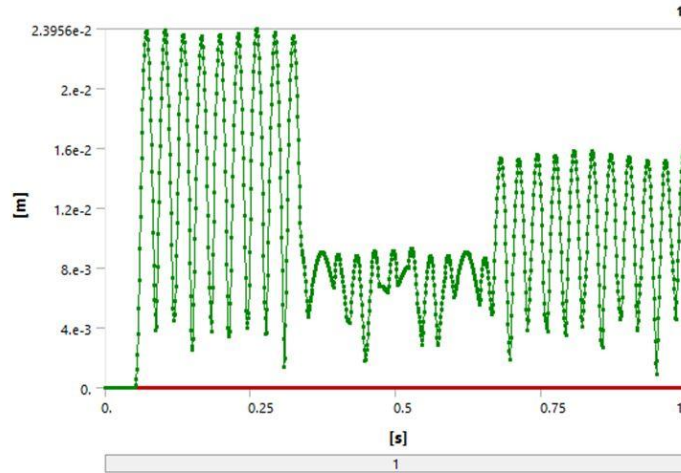| Object Name | Total Deformation | Equivalent Stress |
|---|---|---|
| State | Solved | |
| **Scope** | | |
| Scoping Method | Geometry Selection | |
| Geometry | All Bodies | |
| **Definition** | | |
| Type | Total Deformation | Equivalent (von-Mises) Stress |
| By | Time | |
| Display Time | Last | |
| Calculate Time History | Yes | |
| Identifier | | |
| Suppressed | No | |
| **Results** | | |
| Minimum | 0. m | 10343 Pa |
| Maximum | 1.3171e-002 m | 1.4614e+009 Pa |
| Minimum Occurs On | c- runner-1 | |
| Maximum Occurs On | shelter iso wing front-1@2721 130 8 TOP PLATE front-2 | shelter iso wing front-2@2721 130 8 TOP PLATE front-2 |
| **Minimum Value Over Time** | | |
| Minimum | 0. m | 0. Pa |
| Maximum | 0. m | 15771 Pa |
| **Maximum Value Over Time** | | |
| Minimum | 0. m | 0. Pa |
| Maximum | 2.3956e-002 m | 4.7854e+009 Pa |
| **Information** | | |
| Time | 1. s | |
| Load Step | 1 | |
| Substep | 1000 | |
| Iteration Number | 1000 | |
| **Integration Point Results** | | |

**FIGURE4: Transient  Solution- Total Deformation**



**FIGURE5: Transient Solution -Equivalent Stress**
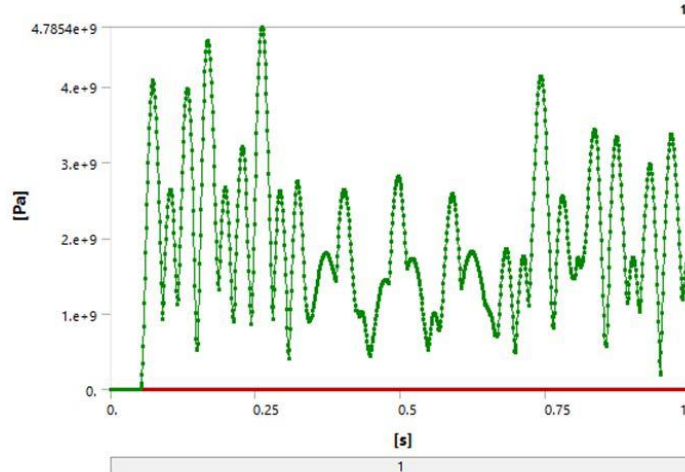
**Random Vibration**

**TABLE18: Random Vibration- Loads**

| Object Name | PSD Acceleration |
|---|---|
| State | Fully Defined |
| **Scope** | |
| Boundary Condition | Fixed Support |
| **Definition** | |
| Load Data | Tabular Data |
| Direction | Y Axis |
| Suppressed | No |

**FIGURE6: Random Vibration- PSD Acceleration**
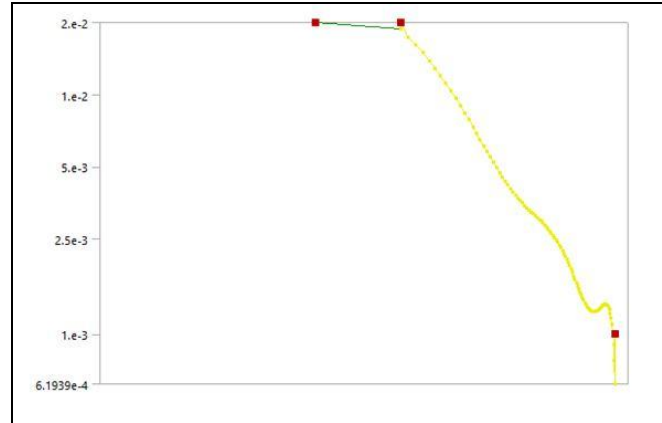


**TABLE19: Random Vibration -PSD Acceleration**

| Frequency [Hz] | Acceleration [(m/s²)²/Hz] |
|---|---|
| 20. | 2.e-002 |
| 50. | |
| 500. | 1.e-003 |

**Solution**

**TABLE20: Random Vibration Solution -Results**

| Object Name | Directional Deformation | Equivalent Stress |
|---|---|---|
| State | Solved | |
| **Scope** | | |
| Scoping Method | Geometry Selection | |
| Geometry | All Bodies | |
| **Definition** | | |
| Type | Directional Deformation | Equivalent Stress |
| Orientation | Y Axis | |
| Reference | Relative to base motion | |
| Scale Factor | 3 Sigma | |
| Probability | 99.73 % | |
| Coordinate System | Solution Coordinate System | |
| Suppressed | No | |
| **Results** | | |
| Minimum | 0. m | 157.34 Pa |
| Maximum | 1.9107e-004 m | 2.8622e+007 Pa |
| Minimum Occurs On | c- runner-1 | u bolt fixing assy- bigger-11@top plate-bigger-1 |
| Maximum Occurs On | shelter iso wing front-1@2721 130 8 TOP PLATE front-2 | Assem3-1@2721 130 8 TOP PLATE-1 |
| **Minimum Value Over Time** | | |
| Minimum | 0. m | 157.34 Pa |
| Maximum | 0. m | 157.34 Pa |
| **Maximum Value Over Time** | | |
| Minimum | 1.9107e-004 m | 2.8622e+007 Pa |
| Maximum | 1.9107e-004 m | 2.8622e+007 Pa |
| **Information** | | |
| Time | 0. s | |

| Load Step | 3 |
|---|---|
| Substep | 1 |
| Iteration Number | 1 |
| **Integration Point Results** | |
| Display Option | | Averaged |
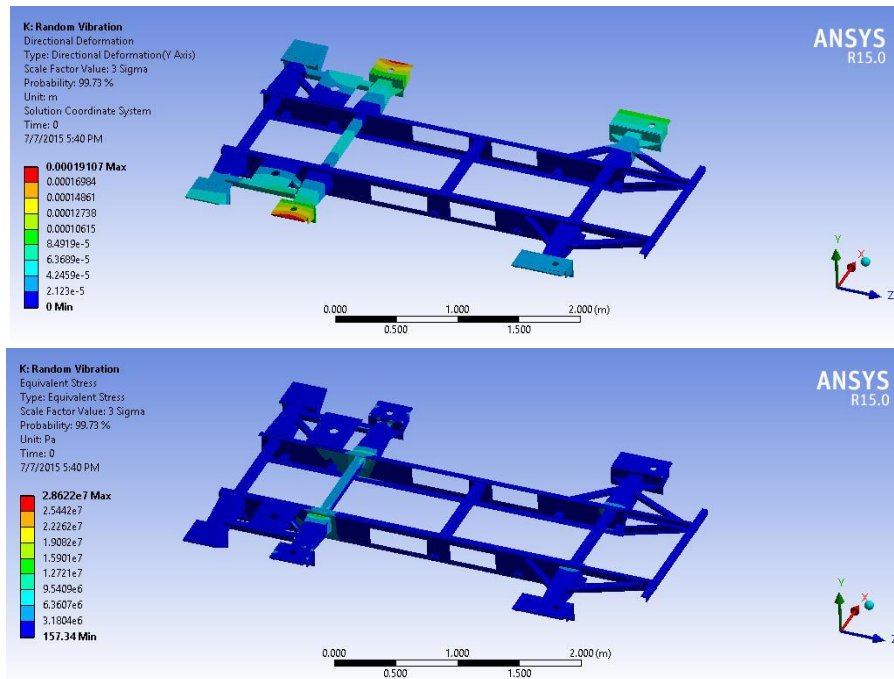




## IV. Conclusion

From the above results it is observed that the stress levels are within acceptable limits and only in transient bump analysis the stresses on the rear member are close to 1.5 factor of safety, which can be minimised by either increasing the thickness of the member or adding the gusset plate at the joining cross members. Hence the overall weight of the Flat levelling frame has come down from 660 Kg(conventional earlier design) to 450 Kg in the new modified design. Also there is a further scope to reduce the weight without compromising on the strength by selecting carbon fibre composite material. Environment sustainability of carbon composite material is much more superior to Mild steel. That is the carbon composite material shows more resistance to extreme and harsh weather conditions comprising freezing temperatures, hot humid conditions, and salty environment conditions.Validation of this environmental sustainability of carbon composite material can be done by conducting Thermal shock analysis tests and salt spray tests in labs .

As the cost of this material is too high, this modification of replacing Mild Steel with Carbon composite material needs to be carried out only under special cases where there is less regard to the cost and much emphasis is placed on the weight reduction and strength enhancement.

## References

[1]. N.V.Dhandapani, Dr. G Mohan Kumar, Dr K.K.Debnath, Static Analysis of Off-High Way Vehicle    Chassis supporting Structure for the effect of various Stress distributions IJART, Vol.2 Issue 1, 2012, 1-8.
[2]. Sairam Kotari, V.Gopinath,"Static and dynamic analysis on Tatra chassis" International Journal of Modern Engineering Research (IJMER), Vol.2, Issue.1, pp-086-094.
[3]. Technical Manual specifications as per JSS 55555.
[4]. Static and Modal analysis of chassis by using FEA -THE IJES by R. Rajappan.
[5]. Stress and dynamic analysis of optimized Trailer chassis by M. Mahmood.
[6]. Finite element analysis in mechanical design using ANSYS: --by WAEL Al-TABEY.
[7]. K.W. Poh, P.H. Dayawansa, A.W. Dickerson, I.R. Thomas, Steel membrane floors for bodies of large rear- dump mining trucks, Finite Elements in Analysis and Design 32, (1999), 141-161.
[8]. Mohd Husaini Bin Abd Wahab, "Stress Analysis of Truck Chassis", Project Report University Malaysia Pahang, 2009
[9]. Ansys Tutorial release -13 :Structural and Thermal analysis.
[10]. H J Beermann, English translation by Guy Tidbury, The Analysis of Commercial Vehicle Structures, Verlag TUV Rheinland GmbH Koln-1989
[11]. T. Irvine, Integration of the Power Spectral Density Function, Vibrationdata.com Publications, 2000.
[12]. RANDOM VIBRATION—AN OVERVIEW by Barry Controls, Hopkinton, MA
[13]. Ibrahim, I.M., Crolla, D.A. and Barton, D.C., "Effect of frame flexibility on the ride vibration of trucks", Engineering Failure Analysis, Vol. 58, pp.709-713, 1994.

[14]. Karaoglu C. and Kuralay, "Stress analysis of a truck chassis with riveted joints", International Journal of Engineering Education, Vol.38, pp.1115-1130, 2000.

[15]. Keiner, Henning. "Static Structural Analysis of a Winston Cup Chassis under a Torsion Load", Engineering Failure Analysis, Vol. 2, pp. 146-157, 1995.

[16]. Kutay Yilmazcoban and Yaar Kahraman Tojsat, "Tuuck chassis structural thickness optimization with the help of finite element technique", Journal of Science and Technology, Vol.1, No.3, 2011.

[17]. Patel Vijay Kumar, Prof. R. I. Patel', "Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction", International Journal of Engineering Research & Technology, Vol.1, pp. 1-6, 2012.

[18]. Romulo Rossi Pinto Filho. "Automotive frame optimization", Review of Mechanical Engineering, SAE Paper, Vol. 67, pp. 146-156, 2003.

[19]. Zehsaz M, Vakili Tahami . and Esmaeili, " The effect of connection plat thickness on stress of truck chassis with riveted and welded joint under dynamic loads", Asian Journal of Applied Science, Vol. 2 No. 1, pp. 22-35, 2009.