# Failure Analysis of Body Control Module under Mechanical Shock & Random Vibration Loading

**Mr. Bipin S Dhole[1], Prof. Shailesh.S.Pimpale[2], Dr. Subim.N.Khan[3]**

[1]*Student(Mechanical* Engg. Dept.*) JSPM'SRajarshi Shahu College of Engineering Pune*
[2]*Professor(Mechanical* Engg. Dept.*) JSPM'S Rajarshi Shahu College of Engineering Pune*
[3]*Doctor (Mechanical Engg. Dept.) JSPM'S Rajarshi Shahu College of Engineering Pune*

---

**Abstract -** The Body Control Module design is used for off highway vehicle .In new product development cost and time is very important. Simulations technique is playing important role with testing when concept of new design BCM considered for off highway vehicle, design should stand under high vibration and structural loading

***Key Words***: Body Control Module, Mechanical Shock, Bolt Torque, Engine RPM

## 1. Introduction

The Body Control Module regulates the operation and coordination between the different parts of the car also by using signals of some sort. The various electronic parts of the vehicle are actually controlled by a Body Control Module – from the car light to the simple door locks; each part has a module that controls it. In today's automotive setups, these modules are operating under a single assembly – they are now controlled by the Body Control Module. It may seem like the module performs a very complex function. Hence this Body Control Module has been protected from the damage during handling and vehicle running condition. This Body Control Module is subjected to various loading conditions such as vibration loading, Impact loading, mechanical shock. One of the case studies is doing to modify the existing Body Control Module. These modifications are carried out by using Computer Aided Engineering approach and verified by the experimental results. The software being used for the simulation is ANSYS and Ls-DYNA/explicit method

## 2. LITERATURE REVIEW

### Mechanical Shock Condition:

Seungbae Park, Chirag Shah, Jae Kwak, Changsoo Jang and James Pitarresi studied "Transient dynamic simulation and full-field test validation for a slim-PCB of mobile phone under Drop / Impact". This paper gives description about a numerical model for a drop test has been developed using ANSYS / LS-DYNA that addresses the limitations of traditional time consuming methodologies. To eliminate strike surface uncertainties, the known input pulse measured on the test vehicle is supplied as base acceleration to the entire model. For that, a shaker table has given half-sine impact acceleration. On that shaker table test board is placed and test is carried out. In this, using acceleration as an input the drop test is carried out and the highest stress and most susceptible failure zone is found out. The numerical model has been validated against experimental measurements of acceleration, velocity (derived) and strain, the results show an excellent correlation with all measured data.

### Bolting Joints:

Mike Guo and Shujath Ali studied "Study on Simplified Finite Element Simulation Approaches of Fastened Joints". Source: SAE 2006-01-2626. In this paper, mechanism of fastened joints is described; numerical analyses and testing calibrations are conducted for the possible simplified finite element simulation approaches of the joints; and the best simplified approach is recommended. The approaches cover variations of element types and different ways that the joints are connected. The element types include rigid elements, deformable bar elements, solid elements, shell elements and combinations of these element types. The critical fatigue damage area is located at a bolt joint on the rear bumper beam bracket. Jeong Kim, Joo-Cheol Yoon, Beom-Soo Kang studied "Finite element analysis and modeling of structure with bolted joints". Source: Science Direct 2006. This paper gives information about the bolt model adopted for a structural analysis of a large marine diesel engine consisting of several parts which are connected by long stay bolts. In order to investigate a modeling technique of the structure with bolted joints, four kinds of finite element models are introduced; a solid bolt model, a coupled bolt model, a spider bolt model, and a no-bolt model. All the proposed models take into account pretension effect and contact behavior between flanges to be joined. Among these models, the solid bolt model, which is modeled by using 3D solid elements and surface-to-surface contact elements between head/nut and the flange interfaces, provides the best accurate responses compared with the experimental results. In addition, the coupled bolt model, which couples degree of freedom between the head/nut and the flange, shows the best effectiveness and usefulness in view of computational time and memory usage. Hsiu-Ying Hwang studied "Bolted joint torque setting using numerical simulation and experiments". Source: Journal of Mechanical Science and Technology (Springer) 2012. This paper gives description about a numerical simulation using the finite element method to determine the installation torque for a fastener until the bolt failed. Both numerical simulation and experimental tests were performed to set the installation torque for the joint based on torque-angle signature curves. The component used for a study was a sliding door mechanism of a light commercial minivan. The torque required to install a fastener varies depending on the conditions of contact surfaces, thread types, material properties, etc. For a newly designed joint of a vehicle, making prototypes and performing tests is expensive and time consuming. Numerical simulation can help predict joint behavior and detect potential failure modes prior to hardware testing.

## 3. Theory

**Engine Excitation:** During vehicle running condition the source of engine vibrations are due to mass forces and gas forces which is shown in fig. 1. Further mass forces categorized into two parts, due to unbalanced rotating masses and oscillating masses. The gas forces are produced non uniform gas pressure inside the cylinder.

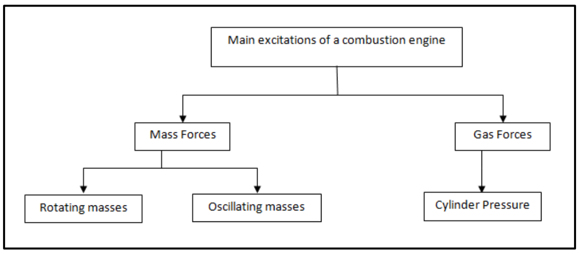



Fig. 1.Engine Excitation.

Mathematically the engine excitation frequency is calculated by using the general equation of motion as;

$$M\ddot{x}(t)+C\dot{x}(t)+Kx(t)=F(t) \qquad (3.1)$$

Where M, C and K are matrices of mass, damping and stiffness, x(t) is the vector of displacement (response), and F(t) is the vector of applied force (excitation).

As the engine running at 3000 rpm, hence its excitation frequency comes 50 Hz. Therefore, the natural frequency of the Body Control Module must be greater than the 50 Hz to avoid the resonance.

Modal analysis is used to determine the natural frequencies and mode shapes of a Body Control Module. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. Modal analysis is also required to do a spectrum analysis. Modal analysis is a linear analysis. Any non-linearities, such as plasticity and contact (gap) elements, are ignored even if they are defined. In the modal analysis, the eigen value represents the natural frequency and eigenvector represents the mode shape of the structure. Theoretically, the natural frequency is calculated by neglecting damping and applied force (excitation) as;

$$M\ddot{x}(t)+Kx(t)=0 \qquad (3.2)$$

**Road Vibration:**

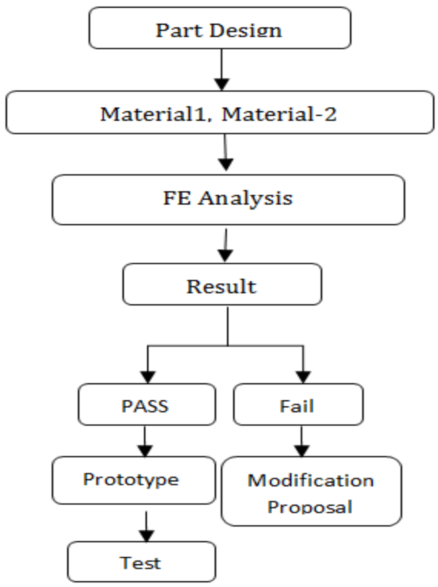
Road vibrations are the random in nature as the vehicle run on the rough road. A Random Vibration Analysis is a form of Spectrum Analysis. The spectrum is a graph of spectral value versus frequency that captures the intensity and frequency content of time-history loads. Random vibration analysis is probabilistic in nature, because both input and output quantities represent only the probability that they take on certain values. This Random Vibration Analysis uses Power spectral density to quantify the loading. Power spectral density (PSD) is a statistical measure defined as the limiting mean-square value of a random variable. It is used in random vibration analyses in which the instantaneous magnitudes of the response can be specified only by probability distribution functions that show the probability of the magnitude taking a particular value.

## 4. Material Properties for BCM

| Sr. No . | Material | Density (T/mm3) | Modulus (MPa) | Poisson's ratio | Max Stress (MPa) | Strain at break (%) |
|---|---|---|---|---|---|---|
| 1 | PP 20% TF Homopolymer | 1.05 e-9 | 3000 | 0.3 | 36 | 14 |
| 2 | PP 20% TF Copolymer | 1.05 e-9 | 800 | 0.3 | 23 | >40 |

Table No:1

## 5. METHODOLOGY



## 6. Geometry

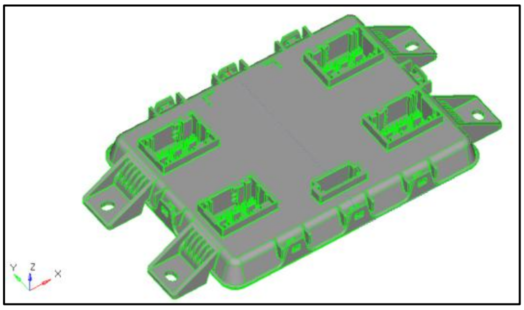


Fig 2.Body Control Module Geometry

The cad model in IGES format is imported in HyperMesh for the preparation of FE model. Then geometry clean-up was done by using options like 'geom. clean-up' and 'defeature' to modify the geometry data and prepare it for meshing operation. This process involves deletion of curvature of very small radius (less than 5mm) which has less structural significance. Mixed type of elements which contains quadrilateral as well as triangular elements, have been used in analysis. All the components of different thickness have been organized in different collectors. The sensitive regions have been re-meshed by manually considering the shape and size of the parts. The Body Control Module is meshed with about 62627 nodes and 61490 elements. Quality check of all the elements has been performed and mesh is accordingly optimized. Three parts, cover, base and PCB of the Body Control Module is shown in the fig.3

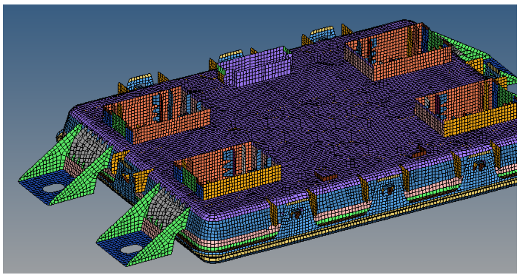


Fig.3 Finite Element Model of Body Control Module.

## 7. Modal Analysis:

The same modal analysis is performed for the modified Body Control Module. The material polypropylene copolymer with 20% talc filled is used for modified Body Control Module. In that also, the fundamental natural frequency obtained is 113 Hz which is also greater than the engine excitation frequency. Hence the modified Body Control Module is also safe in Modal analysis.

| Modes | Frequency(Hz) |
|---|---|
| 1 | 113 |
| 2 | 155 |
| 3 | 180 |
| 4 | 181 |
| 5 | 203 |

Table 2. Natural Frequencies Modified BCM

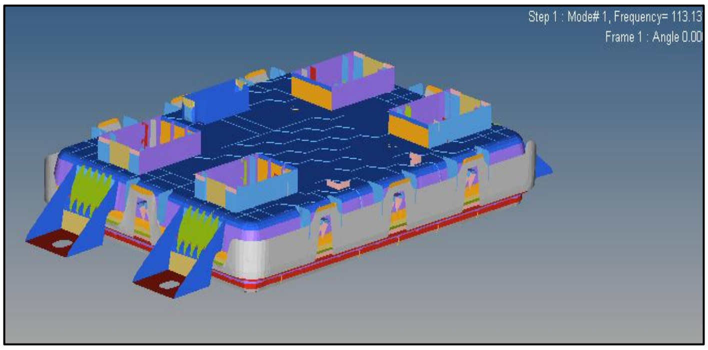



Fig. 5 Natural Frequency of BCM

The above Figure shows the fundamental natural frequency for the modified Body Control Module.

## 8. RANDOM VIBRATION:

The Random Vibration analysis is performed after Modal Analysis is done for the modified Body Control Module. After finishing the Modal Analysis, spectrum analysis is performed. In the spectrum analysis P.S.D. as a sub analysis is chosen because the input data available in terms of the P.S.D. acceleration plot. The results obtained from the Random Vibration Analysis are plotted in the Fig. 6 and Fig. 7

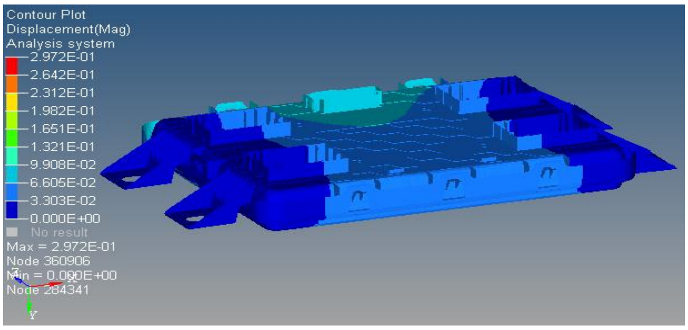



Fig.6 Displacement plot for the Random Vibration Analysis.

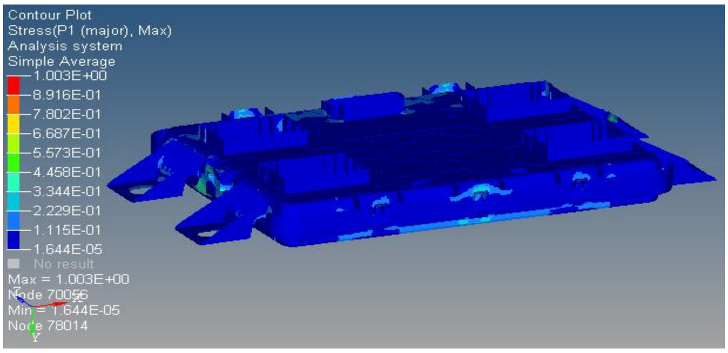



Fig 7. Stress plot for the Random Vibration Analysis.

The deflection occurred in the random vibration is 0.29 mm. The stress induced in the modified Body Control Module during the random vibration is 1MPa which is less than the yield stress of the material. Hence the modified Body Control Module is not damage due to the road vibrations

**9. MECANICAL SHOCK:**

During the fast driving over curb stones and deep potholes, mechanical shock induced on the Body Control Module. The modified Body Control Module which is passed in drop test, further used for the mechanical shock analysis. For this analysis, Body Control Module is to be exposed to half sine mechanical shock pulses, ten in each opposite direction of each perpendicular axis. Analysis is carried in LS-DYNA software in all three perpendicular axes that is positive X, Y, Z and negative X, Y, Z. The single sinusoidal pulse is generated by using equation 5.4. This half sine pulse is further applied to Body Control Module, ten in each perpendicular axis.

This half pulse is calculated by using the equation,

$$\ddot{X}(t) = \ddot{X}_m \sin\Omega t \tag{5.4}$$

Where, $\ddot{X}_m$ is the amplitude of the shock and $\tau$ is the duration.

The pulsation is equal to, $\Omega = \frac{\Pi}{\tau}$.

**Calculation for Half Sine Pulse:**

For the mechanical shock;

$\ddot{X}_m = 500$ m/s2;

$\tau = 11$ ms;

Using these values, a graph is obtained as;

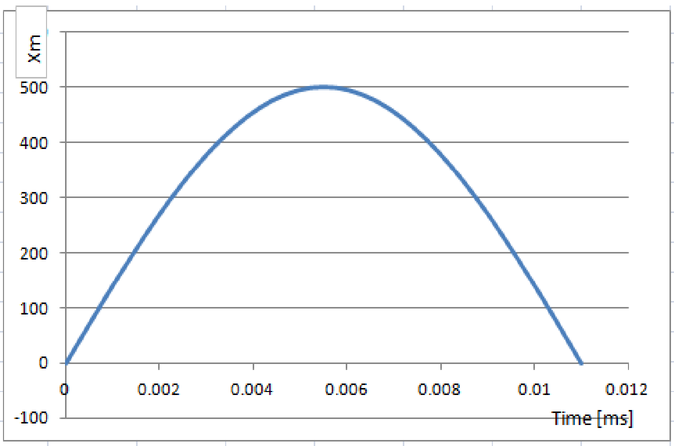



Fig. 7. Half Sine Pulse.

**10. Test Parameters:**

The half sine pulse is given to the Body Control Module, ten times in each axis along both the axis.

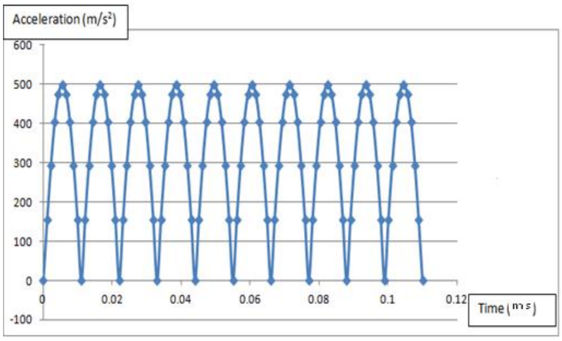



Fig. 8. Sinusoidal Shock Pulse.

**11. Boundary Condition:**

All the degree of freedom (that is three translational and three rotational) are fixed at the mounting location. The Mechanical Shock pulse is applied at the same mounting location. The applied boundary conditions are shown in the Fig. 9

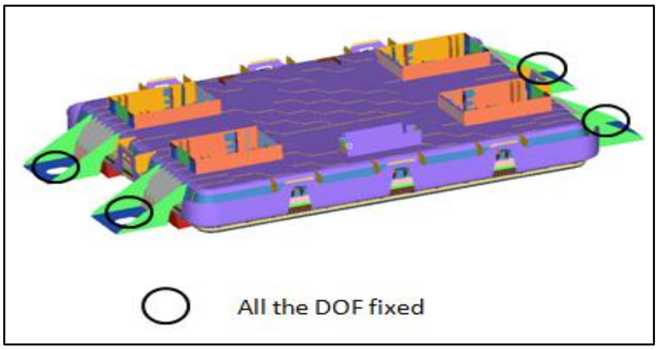



Fig. 9 Boundary Condition for Mechanical Shock Analysis.

**12. Results of Analysis:**

By using above Boundary condition, simulations are performed in the LS-DYNA. The Stress plot for the mechanical shock is shown in Fig. 10 to 15

**Positive X Direction**

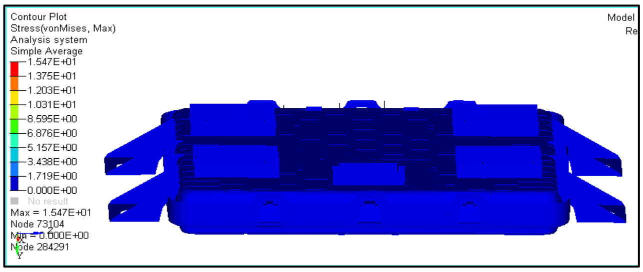



Fig. 10 Mechanical Shock along Positive X direction.

**Negative X Direction**

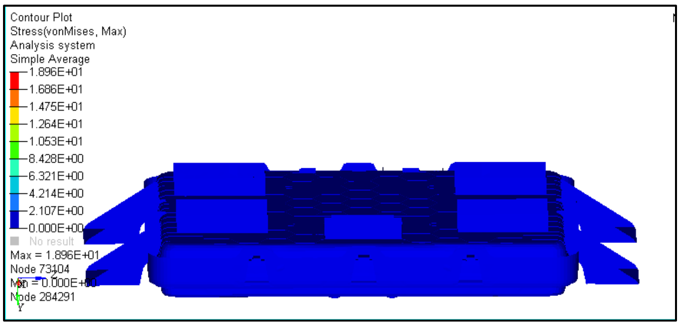



Fig. 11 Mechanical Shock along Negative X direction.

**Positive Y Direction**

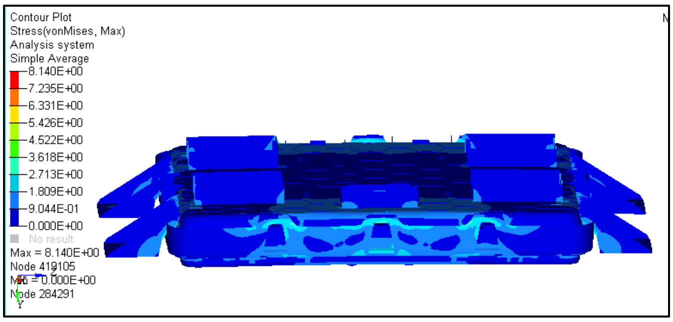



Fig. 12 Mechanical Shock along Positive Y direction.

**Negative Y Direction**

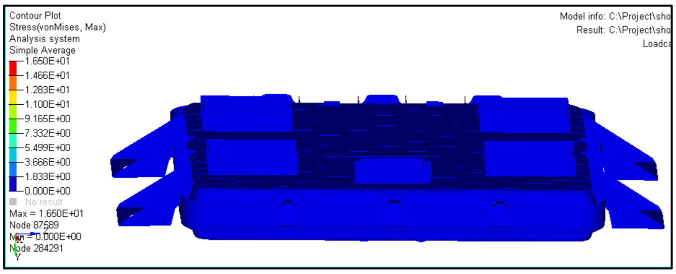



Fig. 13 Mechanical Shock along Negative Y direction.

**Positive Z Direction**

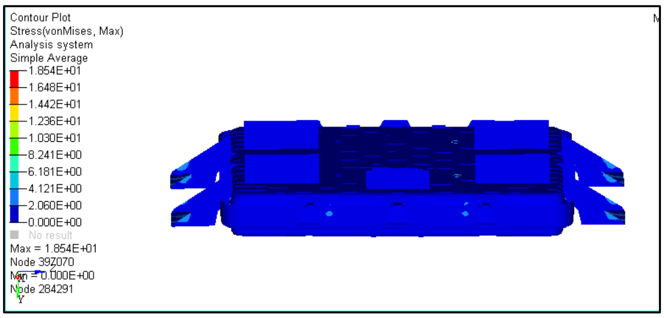



Fig. 14 Mechanical Shock along Positive Z direction

**Negative Z Direction**

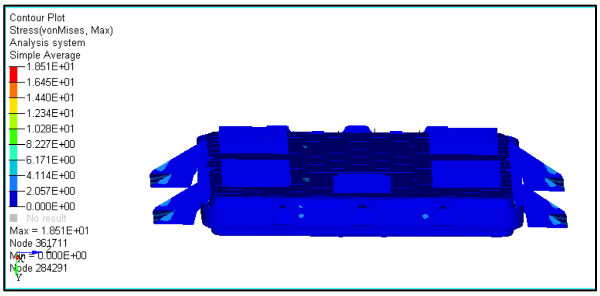



Fig. 15 Mechanical Shock along Negative Z direction.

| Sr.No | Mechanical Shock Direction | Stress N/mm$^2$ | Result |
|---|---|---|---|
| 1 | Positive X Direction | 15.47< 23 | Safe |
| 2 | Positive Y Direction | 8.14< 23 | Safe |
| 3 | Positive Z Direction | 18.54< 23 | Safe |
| 4 | Negative X Direction | 18.96< 23 | Safe |
| 5 | Negative Y Direction | 18.733 < 23 | Safe |
| 6 | Negative z Direction | 19.51< 23 | Safe |

Table 2 Mechanical Shock.

In the Mechanical Shock analysis the modified Body Control Module is safe as there is stress induced in the modified Body Control Module is less than the 23 MPa.

## 13. TIGHTENING TORQUE:

In the automotive industry, fasteners are commonly used to join or secure components. The advantages of using fasteners are ease of assembly/disassembly and low cost. Standard metal fasteners are also used to assemble thermoplastic components. Fasteners can be tightened using simple tools; however, to maintain required lamping force in the joint, the correct torque must be applied.

The fastener installation torque is critical to the integrity of a joint. The torque required to install a fastener varies depending on the conditions of contact surfaces, thread types, material properties, etc. For various joint designs that use the same bolts and nuts, the torque for each joint could be different. During the assembly process, a joint should be tightened with a specified torque. Excessive torque can overload the bolt and damage the components. High torques generally produces high compressive stress. The higher the stress, the greater the stress relaxation. As stress relaxation occurs, the clamping force and torque retention drop and the fastener will loosen.

On the other hand, a joint that is assembled with inadequate torque will not have the clamping force required to maintain the integrity of the joint. This can lead to noise, vibration, and ultimately failure of the joint. To determine the installation torque for a fastener, one should consider the strength, geometry, and contact surface of the joint, along with the installation speed, and the loading conditions in the joint. Proper torque provides the clamping force necessary to prevent joint separation.

Torque simulation is performed for determining the limiting torque at the mounting location. This analysis is performed to avoid the failure of mounting location due to excess applied tightening torque. The bolt used for the simulation is the M-6 size and a washer of 0.5 mm in thickness.

**14. FE Model:**

One leg of the Body Control Module is mesh with the Brick element. The washer is mesh with the shell element with thickness 0.5 mm. 1-D beam is created to apply the boundary conditions. Null elements are provided on the face of mounting location to avoid the penetration and good contact between the brick elements and the shell elements. In between the washer and null element surface to surface contact is provided and interior contact is provided for the mounting location which is brick mesh. The FE Model is as shown in Fig. 5.18.

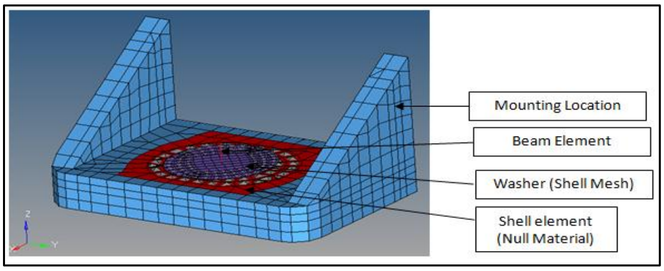



Fig. 5.35 FE Model of the mounting location.

**15. Boundary Conditions:**

Boundary conditions are applied at the 1-D beam. The rotation is applied at the one end of the beam which is $2\pi$ about the Z axis. The displacement is applied at the same location

**16. Result and Discussion:**

The graph, Stress vs Torque is plotted which is shown in Fig. 5.19.

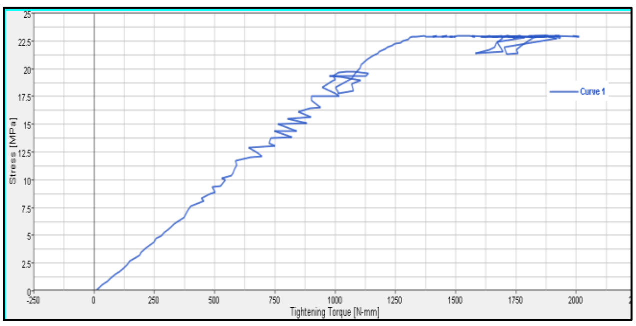



Fig. 5.36 Stress Vs Torque

From the graph, the limiting tightening torque is 2010 N-mm at the stress 23 MPa

**15. Conclusion**

The frequency of the BCM module is meeting the target goal frequency. Maximum stress is under yield limit for mechanical shock in all three principal direction ,hence we can concluded that BCM module is safe for vibration loading.

**References**

[1] J. S. Lin and Kui-Sun Yim, “Application of Random Vibration Test Methods for Automotive Subsystems Using Power Spectral Density (PSD)”, SAE 2000-01-1331.

[2] Hong Su, “Vibration Test Specification for Automotive Products Based on Measured Vehicle Load Data”, SAE 2006-01-0729.

[3] Min-Chun Pan, Po-Chun Chen, “Drop simulation/experimental verification and shock resistance improvement of TFT–LCD monitors”, Microelectronics Reliability 47 (2007) 2249–2259 (Science Direct).

[4] Masaaki TSUTSUBUCHI, Tomoo HIROTA, Yasuhito NIWA, Tai SHIMASAKI, “Application of Plastics CAE: Focusing on Impact Analysis”, Sumitomo Chemical Co., Ltd. Plastics Technical Center.

[5] “Impact and plastics”, Source: Zeus, Technical Newsletter.

[6] Guido MuzioCandido, “Methodology of plastic parts development in the automotive industry”, SAE 2006-01-2626.

[7] Yoshiaki Togawa and TomooHirota, “A Method to Design Plastic Part for Impact Energy Absorption by Optimizing both Material Property and Part Structure”, SAE 2005-01-1682.

[8] Seungbae Park, Chirag Shah, Jae Kwak, Changsoo Jang and James Pitarresi, “Transient dynamic simulation and full-field test validation for a slim-PCB of mobile phone under Drop / Impact”.