Structural Optimization for Support Structures of Electromechanical Devices using Finite Element Approach

Anvith N Kulkarni
M.Tech Mechatronics, SMBS
VIT, university
Chennai, India

Mr Nazeer Azam
Technical Lead, PE R&D
Danfoss IDC
Chennai, India

Vinayagamurthy G
Asst. Professor, SMBS
VIT, university
Chennai, India

Dr Shanmugam Paramasivam
Senior Manager, PE R&D
Danfoss IDC
Chennai, India

***Abstract*—Electromechanical devices are held on Support Structures (plates made of Sheet metal). These support structures are, subjected to static and dynamic loads. Since special provisions are provided on these support plates to access the heat sinks of the electromechanical devices it becomes necessary to ensure the stability of the plates for various loading conditions. Hence, these plates are analyzed for Static and Vibrational loads. Since the deformation and stress are, observed to be considerably high in the Support plates with heat sink access, hence suitable solutions are developed in order to contain the deformation and stresses generated.**

Keywords—Support plate, Static structural analysis, Modal(Free Vibration) analysis and Random vibration analysis

I. INTRODUCTION

Electromechanical devices are mounted on these support plates with the help of bolts. Thus, the weight of the device is transferred onto the plate through bolts. The only static load acting on the plate is weight of the device, which is concentrated at the CG of the device.Being deployed in different environments, they are subjected to dynamic loads.Hence, a necessity arises for them to be analysed for their stability under static as well as dynamic loading condition.

The support plate without heat sink access is stable under both the loads. But when the heat sink access in provided in the support plate the deformation increases considerably when static load[1] is applied to the support plate, and again it increases considerably after the support plate is subjected to the dynamic (Vibrational) [14] loading. Hence a suitable solution was developed in the from a support structure or a stiffener [3]. ANSYS workbench was the tool used for this analysis. The solutions were modelled in PTC Creo 14. The steps followed during analysis Viz,

1. Analyse the model for Static loads and develop the solution
2. Find out Natural frequencies of the support plate without and with solution
3. Analyse the behaviour of both the support plates under Random Vibration specification.

Further details are discussed in section III.

II. LITERATURE REVIEW

A systematic literature survey provides an opportunity to, specifically review various methods used to solve problems on thin plates subjected to Static load.

C B Dolicanin et al. used finite difference method to study the behavior of thin plates under static loads. By dividing the plate into elements and selecting the specific, a partial differential equation represented each node. Equation obtained partial differential equation was the function of deflection, stress, strain. Analogy of beam under static load represented the loading condition of thin plate. Where, length and thickness formed the basis for calculating moment of inertia of the plate. The results obtained from the beam analogy were, substituted in the PDE of each node to obtain the value of stress, strain or deflection at that particular node. Max deformation in a beam is equal to deformation in the and by substituting its value in PDE of each node; distribution of deformation in the plate is calculated. [1]

Ewa Magnucka Blandzi developed mathematical model to analyze the deformation of two plates compressing (sandwiching) a metal foam core. [2]

Deepak Kumar Singh et al. performed FEA analysis of thin square plate in ANSYS Workbench. A 10mm thick 1m*1m thick plate was, subjected to various loads at various boundary conditions. In order to reduce the deflection of the plate, an additional support was, introduced. The thickness of the support matched with the thickness of the plate and its length was, varied. Thus deflection-based stiffening was, carried out in the paper. Thus forming a basis for the solutions provided in the thesis. [3]

N Dalvi et al performed Analysis of flat plate weld joint. This analysis determined the stress in the weld joints plates, for different transverse loads and overlap lengths. Thus determining, the optimum overlap length for the plates. At this length, the stresses formed and deflection of the plates is minimal. [4]

D S Gujar, K B Ladhane performed a parametric study on a clamped circular plate by using classical plate theory and ANSYS APDL (ANSYS Parametric Design Language). They observed that 4NODE SHELL 181 element gives better results with the analytical calculations; Solid 187 is also a better option in case the plate is not uniform or symmetrical.[5]

Abhishek Kumar Tiwari et al Performed static analysis of Disc brake in ANSYS WORKBENCH. In this work, the authors observed the behavior of the braking disc under uniform distributed pressure, and recorded the stress distribution, deformation and strain. [6]

M.Venkatasudhahar et al Analyzed the Fatigue life of a Spot Welded Joint and, assessed the effect of Spot weld diameter and thickness of the plates on the fatigue life of the joint. [7]

Jeong Kim et al performed a comparative study of different models used for simulating the bolted joints. The paper performs a comparison between three types of bolt simulations (a) Solid bolt models (b) Spider bolt models (c) No bolt models. Solid bolt models are accurate but require more computational time. Spider bolt models require moderate computational time and are quite accurate. No bolt models require less computational time but their accuracy depends on the Mesh density. [8]

Yorcun et al performed simulations with bolted connections, and recorded their behavior. Any bolt beam connection will face few irregularities and inelastic behavior of contact surfaces, partial changes in contact area. Thus increasing the complexities, this will affect the accuracy of the simulation. [9]

Dario Croccoloet al performed Static structural analysis of an Articulated Urban Bus Chassis in ANSYS WORKBENCH. The paper sheds light of analysis, of an 18m long bus chassis under the action of various loads and describes its behavior under those loads. The rear end of the chassis encountered higher stresses especially near the welded joints. [10]

Archibald N Sherbourne, Mohammed R Bahari Studied the behavior of bolts and plates for considering the plates to be, bolted at the ends under the action of axial load. They observed the nature of deflection plates and bolts by varying the bolt size. [11]

Tian Wen Chao et al performed modal and random vibration analysis for the on a chip packaging module. The model had developed issues for the vibration in Z direction. Hence, they proposed few solutions to overcome the issues. [12]

Xiaxhon Lie et al performed a static load and modal analysis on a micro milling machine tool. Considering the results from analysis authors developed an optimized column for the Gantry in milling machine. [13]

Xueli Qi performed Random vibration analysis for the on a PCB. They observed the effect of Solder joint parameter on high density PCB when subjected to random vibration loading. [14]

Wu Yong-Hai, Wang Feng performed pre stressed modal analysis on a semi-trailer frame. This analysis helped to assess the inherent dynamic characteristics of the frame thus paving way todevelop solutions that can improve the inherent dynamic characteristics of the frame. [15]

The literature survey consists of the FEA and analytical solutions performed to assess the behavior of bodies such as thin plates, columns, beams. This paper discusses static and vibrational analysis of a plate with large cutouts. It includes the Assessment of plate’s behavior under static eccentric point load and Vibration loads. This analysis of the plates is unique since none of the above-mentioned references has performed any analysis involving a plate with large cutouts and design similar to this plate. The literature survey helped in understanding the behavior of various bodies under static and vibrational loads. In addition, this literature survey helped in developing the solutions in accordance the issues arising after the application of the loads. But None of the papers have emphasized on analyzing a support plate with under action of eccentric point load concentrated at four different points on the plate.

III. METHODOLOGY

Fig 3.1: represents the methodology followed during the project.

Procure the CAD model for both the Support plates (with and without heat sink access). Perform a static structural analysis for both the plates. Set the deformation and stress results of support plate without heat sink access as the reference Consider static analysis results of support plate without heat sink access as the reference. Since, the deformation in the body reduced exceeds beyond permissible limits. There is a necessity to reduce the deformation. Then the nature of deformation is, studied to get a fair idea about the problem. Since the deformation is unidirectional, the solutions should have more material in the direction (axis). The solutions extended along Z-axis (the direction of deformation). Thus stiffening the body in the process this brought about very effective changes in the deformation. Even the resonant frequencies changed after modal analysis.

Initially all the solutions underwent only static analysis. After observing their effectiveness in reducing the deformation, the best solutions were, sorted. Since not all the developed solutions could be, implemented at once, a comparison seemed necessary. Based on DFM DFA and cost of the component a comparison was, performed. A concept was, singled out, after the comparison of concepts in Pugh matrix. This concept had more advantages than the reference concepts. The best concept was, subjected to Vibration analysis. The Natural frequencies of the support plate indicating that the stiffness of the plate had increased; this solution later underwent random vibration analysis.
Deformation of the support had reduced and the stresses reduced.

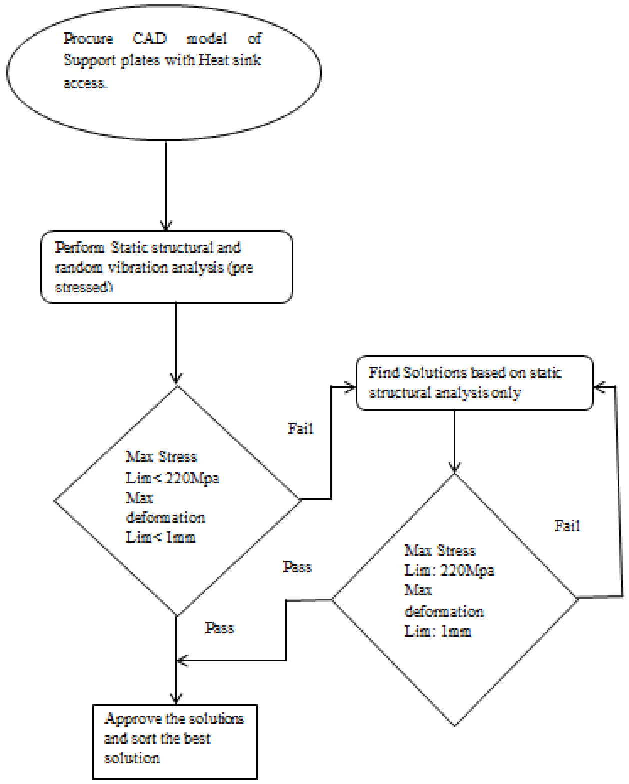

Fig3.1: Methodology

IV. FINITE ELEMENT ANALYSIS OF SUPPORT PLATES.

a) Static Structural analysis: This analysis involves application of static loads (similar to point load)[4]. The loads induce stresses and deformations. These stresses and deformations should not exceed the permissible limits. This procedure helps in analysis the structural strength of the body and loads it can with stand. Figures 4.1, 4.2, 4.3&4.4 represent constraints, load applied, the load equation and the side view of loads of applied.

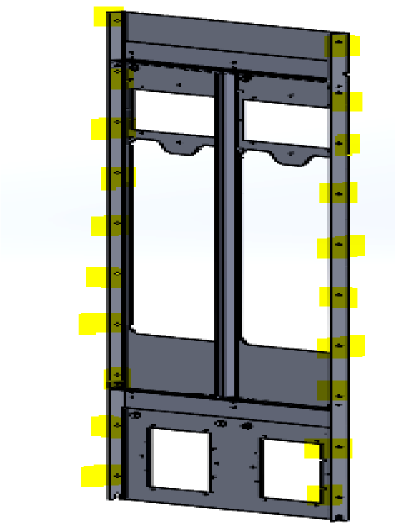

Fig: 4.1fixed support at highlighted holes

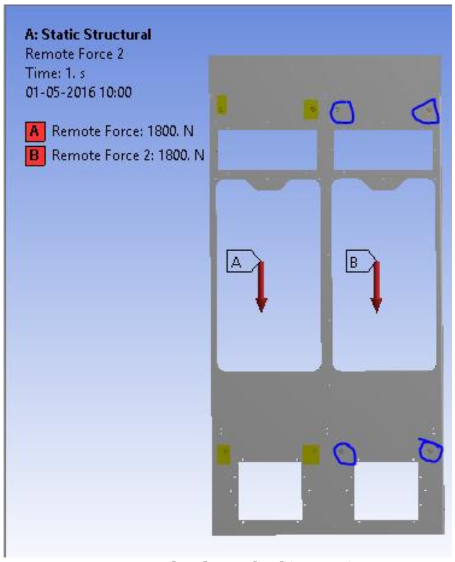

Fig: 4.2 loads applied(A& B)

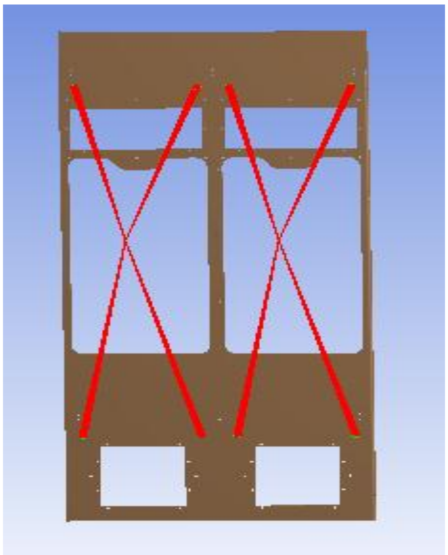

Fig: 4.3 nature of load distribution.

The support plate is analysed for Static and. Material used is Cold rolled steel. Load applied is 1800N approximate weight of a drive, since the weight of the drive is concentrated at its CG acts at an approximate distance of 150mm away from the drive [16]. Since the drives are, mounted only on the bolts, the boltholes will carry the load (Fig 4.2 highlighted bolts for load A and marked holes for load B). The bolts are, idealized as beams [8] using multipoint contacts (Fig 4.6) (constraints). Since the weight of the drive is the load, acting it is, simulated as a concentrated point force acting at a finite distance away from the plate. Aspect ratio of the plate is (2:1).

For analysis material can be, considered as cold rolled steel as far as yield strength of 410 Mpa.Fig 4.3 displays place of application of loads.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 12, December-2016

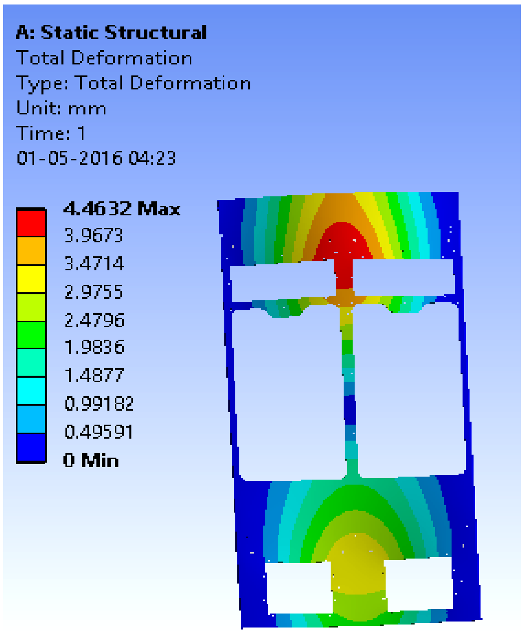

Fig: 4.Total deformation of the support plate subjected to load.

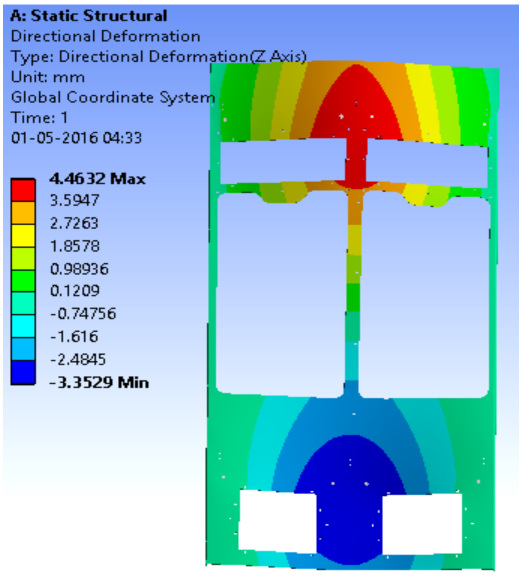

Fig: 4.8 directional deformation of the support plate

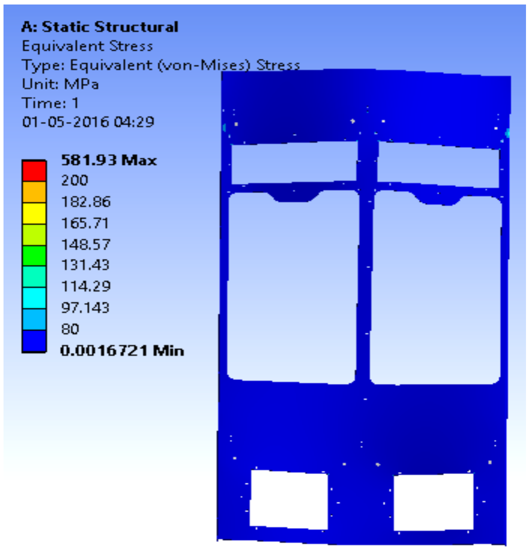

Fig: 4.9Stress concentrations in the support plate.

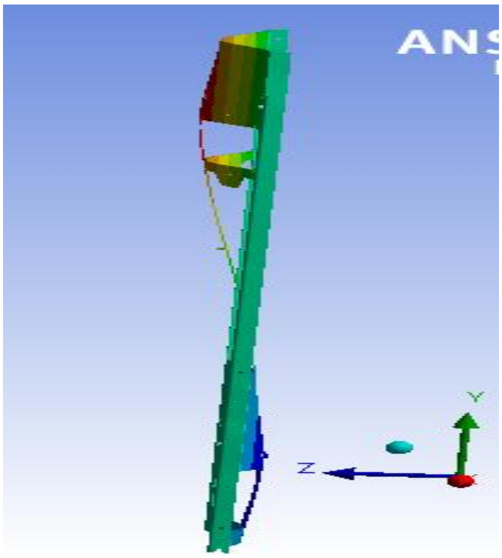

Fig: 4.10 Nature of deformation of the body.

Inferences:The plate deforms only along Z-axisfigure 4.10 confirms the direction of deformation of the body. To constrain this deformation the body an additional body is to be, added on the support plate. The body should be extending along Y and Z-axes respectively. The stress concentration in the body is comparatively low, as the body is encounters stresses beyond permissible limit of 220 MPa (Consider Factor of safety to be 1.8) only in notches, corners and holes. Most part of the body encounters stresses up to 80Mpa. The body stiffness should be, increased to reduce the deformation.

b) Vibration Analysis: This analysis includes free and forced vibration analysis. Free vibration or modal analysis of the body involves determination of natural frequency of the body.

Natural frequency is the frequency the body at which it starts resonating [13]. It is pre stressed modal analysis since static load is included in the modal analysis.

Modal analysis involves determining only first six modes. First six natural frequencies of the body are obtained table 4.1 gives the account of natural frequencies.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 12, December-2016

MODE	Natural frequency (Hz)
1	20.94
2	26.202
3	27.64
4	35.594
5	47.472
6	72.212

Tab 4.11: first six modes of support plate

Random vibration analysis: Random Vibration is Varying Waveform with its intensity defined by the term PSD(Power Spectral Density). In real time application of the system, subjected to Random vibrations. Random Vibrations occur at all frequencies(simultaneously) .That is all the resonance frequencies of the body will be excited simultaneously.

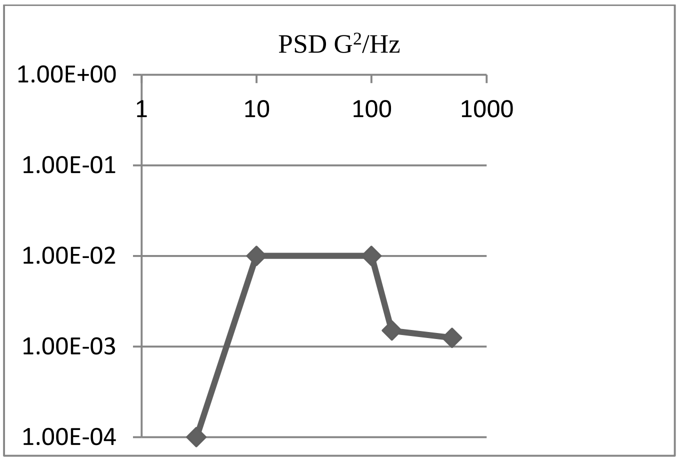

Fig 4.12: x-Axis, PSD values G^2/Hz, y- axis represents frequency

PSD (Power Spectral Density): It can be defined as the ratio of the Square of G to the frequency (acceleration /amplitude provided as per specification) units G^2/Hz or (mm/s) ^{2}/Hz. The input givenis similar to the Fig4.12.

While calculating the energy in time domain is,converted to frequency domain hence involving infinite intervals of time regardless of time a body is, subjected to random vibration PSD value is same and stress induced are same regardless of how long the component has been, exposed to the Random Vibration. Since all the resonance frequencies are excited simultaneously, the body experiences maximum loads. The area under the curve represents G RMS (Acceleration of the body under the action of PSD).

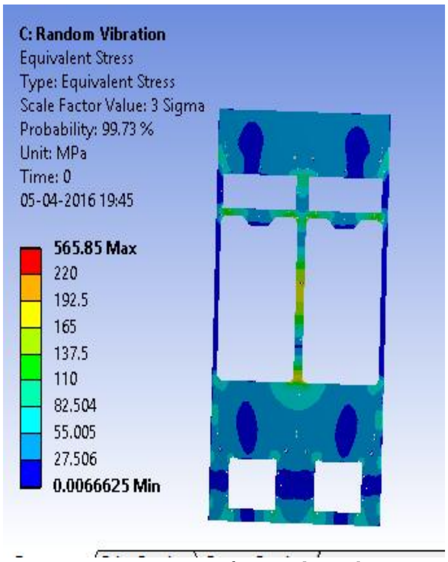

Fig 4.12: Stress concentration after Random vibration analysis.

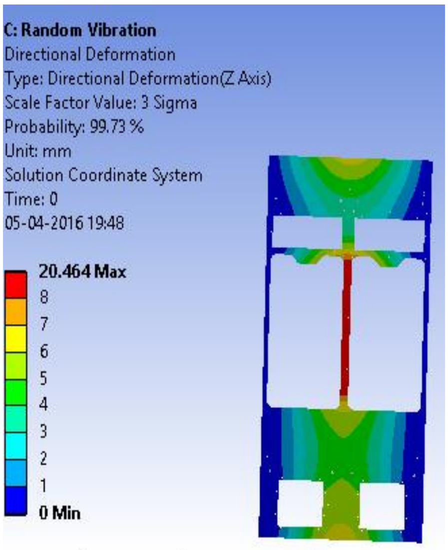

Fig 4.13: Deformation after Random vibration analysis.

The modal results show that body is resonating at lower frequencies (Tab4.1). This also forms a matter for concern since; the body may fail during random vibration analysis. The random results showed that body is deforming at Higher value (Fig 4.13). This deformation had to be constrained to bring the stability in the body. The reduction in the deformation is brought my adding the material in the plane (axis) of deformation, the thickness of the solution should match with the thickness of the support plate [3]. Hence, by including this solution on the support plate the deformation, can be reduced to some extent. Similar to the concept of ribbing, that provides rigidity to the body. Here various factors are to be, considered just providing a solution to reduce the deformation is not accepted, the solution must be feasible enough to be manufactured and then assembled on the support plate. The solution must not only fit in the assembly with ease but also make it look consistent. Inconsistent assembly aesthetically looks awkward.

V. DEVELOPMENT OF SOLUTION.

The deformation led to development of different solutions only thethe best solution out of probable five solutions is, discussed in this paper.

This solution includes involves the addition of three bodies on the rear end of the support plate. Figures 5.1 represents Z Bracket, 5.2 represents L bracket, 5.3 represents placement of the bodies.

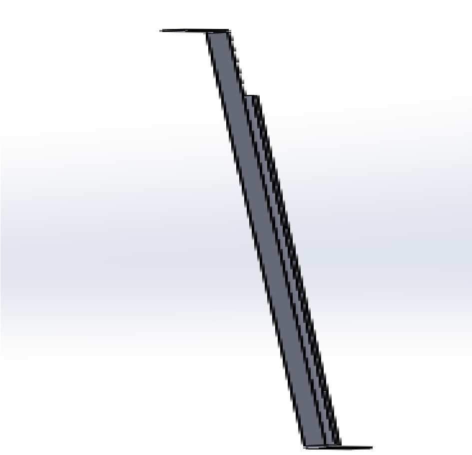

Fig5.1: Z bracket

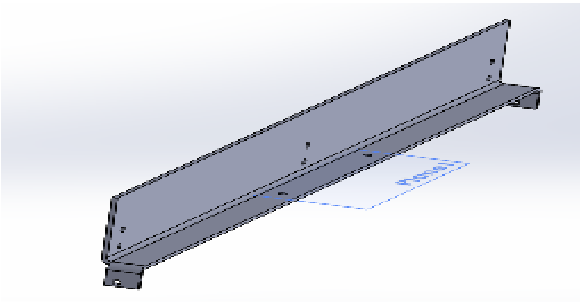

Fig5.2: Lbracket

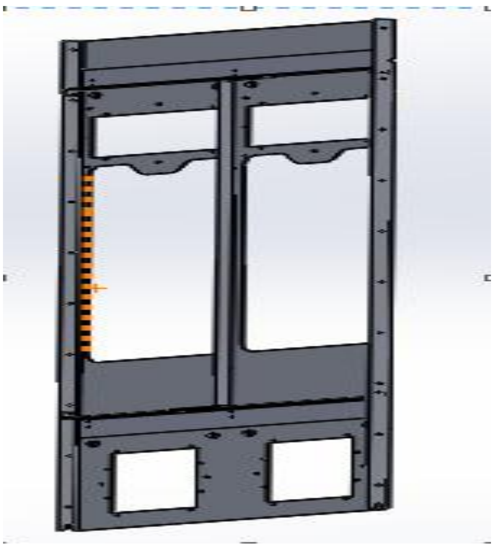

Fig5.2: Placement of the flanges.

Static Structural results:

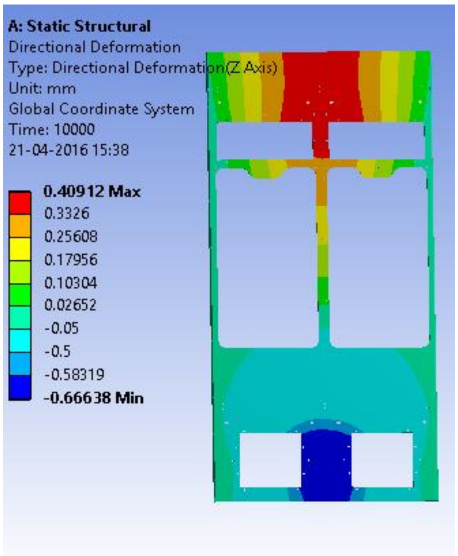

Fig5.4: Maximum deformation 0.6638mm

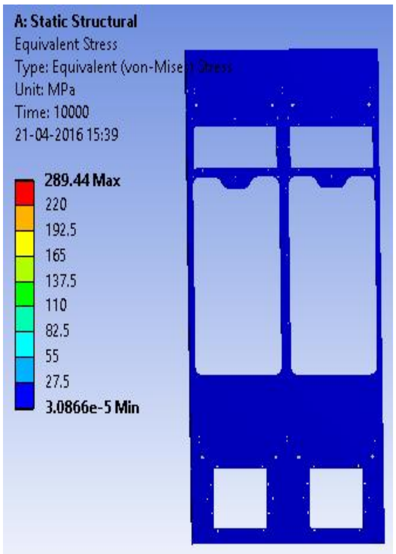

Fig5.5: Max.stress concentration 289.44MPa

The Support plate is more stable because most part of the plate is covered. EspeciallytheCentre of the plateis coveredwhere the material concentration is less. Thus reducing the deformation by good extent (Fig 5.4) .There is consistency in the assembly. The stresses beyond permissible limit are concentrated only in holes and notches (Fig 5.5).

Vibration Results:

Modal: Table 5.1 represents Natural Frequency and modes of the body.

Mode	Natural frequency (Hz)
1	41.264
2	59.159
3	75.833
4	89.598
5	99.79
6	104.78

Tab. 5.1: Natural frequencies of Support plate after the addition of solution

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 12, December-2016

Random vibration: Figures 5.6 and 5.7 represent Deformation and stress results for Random vibration analysis

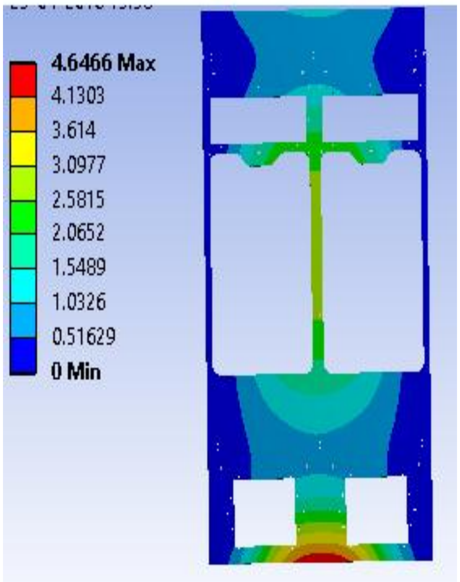

Fig5.6: Maximum deformation 4.6466

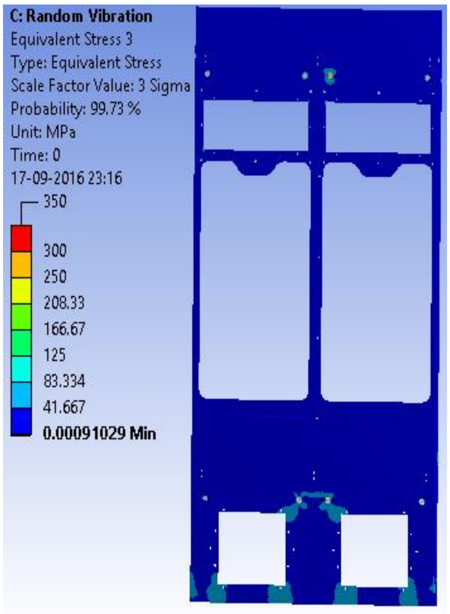

Fig5.7: Max.stress concentration 350 MPa.

Inference: The natural frequencies of the body have changed. The change in natural frequencies of the body is the evidence of Support plate becoming stiffer (Tab5.1). Stresses formed on the body do not exceed 80MPa (Fig 5.7). The stress beyond the permissible limits is, seen only in the corners and holes in the body. The addition of the concept also increases the mass on the support plate. Since concept4 is, assembled on the support plate, the body contact during the assembly may increase the stress concentration to some extent in the corners where there might be a corner-to-corner contact of the assembly. That will already have induced pre stresses. Moreover, when it is, subjected to vibrational loading the stress concentration in such areas will increase in a drastic manner, but such areas are too small and the stress concentration in such areas is, neglected. The reduction in deformation is, displayed in (Fig 5.6)

VI. RESULTS AND DISCUSSION

Static results Comparison:Table 6.1 represents deformation of the plate with and without addition of solution.

Plate type	Max. Deformation (mm)
Support plate without addition of solution	4.462
Support plate after addition of solution	0.6678

Tab.6.1: Difference in maximum deformation before and after addition of the plates

Random Vibration Results:Table 6.2 Table 6.1 represents deformation of the plate with and without addition of solution after the body being subjected to Random vibration loads.

Plate type	Max. Deformation (mm)
Support plate without addition of solution	20.12
Support plate after addition of solution	4.4672

Tab 6.2 Deformation before and after addition of the solution

The addition of the solution to the support plate results in increase in stiffness of the body.From tables 5.2 and 5.3 it is, observed that support plate after the addition of solution has, higher natural frequencies as compared to support plate without addition of solution. Hence, confirming the increase in stiffness of the body. The deformation is constrained by very good extent in the body.

VII. CONCLUSION

This paper covers the behavior of Support plate with two large cuts subjected to eccentric point loads. The Deformation of Support plate with heat sink access (plate with cuts) exceeded beyond the acceptable limits (Fig 4.7). Necessity to contain this excess deformation led to development of solutions (concepts). The introduction of these solutions strengthened the plate, resulting in reduced deformation.. A vibration analysis conducted after inclusion of the best solution, confirmed that the stiffness of support plate improved.

The considerable improvements in the deformation and Stress formations are listed in the table below. The changes in the deflection and stresses are mentioned in terms of percentage.

Reduction in Deformation:

Analysis	Deformation before solution (mm)	Deformationafter solution (mm)	%Reduction
Static	4.4672	0.40912	991
Vibration	20.82	4.6466	348

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 12, December-2016

Reduction in Stress Concentration :

Analysis	Stress before solution (MPa)	Stress after solution (MPa)	%Reduction
Static	581.93	289.44	101.053
Vibration	565.85	350	61.671

VIII. REFERENCES

[1] Dolicanin et al /Application of Finite Difference Method to Study of the Phenomenon in the Theory of Thin Plates, Scientific Publications of the State University of Novi Pazar. pp 25-31.

[2] Ewa Mag-nucka-Blandzi Mathematical Modeling Of a Rectangular Sandwich Plate with A Metal Foam Core, Journal Of Theoretical And Applied Mechanics 49, 2, pp. 439-455, Warsaw 2011

[3] Deepak Kumar Singh et al, Analysis of Stiffened Plates using FEM – A Parametric Study IJERT, Volume: 02 Issue: 04 July-2015 pp. 1650-1656

[4] M V Dalvi et al Determination of Failure Strength of Flat Plate Weld Joint Using Finite Element Analysis, International Journal of Scientific & Engineering Research Volume 3, Issue 12, December-2012 pp.67-72

[5] P. S. Gujar, K. B. Ladhane, Bending Analysis of Simply Supported and Clamped Circular Plate SSRG International Journal of Civil Engineering (SSRG-IJCE) Volume 2 Issue 5–May 2015 pp.69-75.

[6] Abhishek Kumar Tiwari et al Finite Element Analysis Of Disc Brake By ANSYS Workbench, IJREAT International Journal of Research in Engineering & Advanced Technology, Volume 2, Issue 2, Apr-May, 2014 pp.1-6

[7] Jeong Kim et al, Finite element analysis and Modeling of structure withBolted joints, Applied Mathematical Modeling 31 (2007) 895–911, DOI 20.3.2006

[8] M.Venkatasudhahar et al, Finite Element Analysis of Fatigue Life of Spot Welded Joint and the Influence of Sheet Thickness and Spot Diameter, International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS Vol:14 No:06 December 2012 pp.79- 82.

[9] C. Yorgun, et al Finite Element Modeling Of Bolted Steel Connections,Designed By Double Channel, Computers and Structures 82 2004 pp.2563–2571.

[10] Dario Croccolo et al Structural Analysis of an Articulated Urban Bus Chassis via FEM: a Methodology Applied to a Case Study. Strojniški vestnik - Journal of, Mechanical Engineering, 57(2011)11, pp.799-809.

[11] Archibald Sherbourne, Mohammed R Bahari, 3D Simulation of End plate bolted Connections Journal Of Structural Engineering Vol. 120 No. 11 November 1994 pp.3122-3136.

[12] Tian Wenchao et al Anti-vibration Structure Analysis and Optimizing on BGA Packaging Module 2014 15th International Conference on Electronic Packaging technology, pp.524-528.

[13] Xiao Zhou Li et al, Structural Design and Analysis of Micro Milling Machine Tool, in Proceedings of 2014 IEEE International Conference on Mechatronics and Automation August 3 – 6(2014), Tianjin, pp.770-774.

[14] Xueli Qi , Effect of Solder Joint Parameter on Vibration Fatigue Reliability of High Density PCB Assembly Proceedings of 2014 IEEE International Conference on Mechatronics and Automation August 3 – 6(2014), Tianjin, China

[15] Wu Yong-Hai ,Wang Feng, Modal Analysis of a Semi-Trailer Frame based on Ansys Workbench, Fourth International Conference on Information and Computing 2011 pp.346-349.