PROPOSED RESEARCH METHODOLOGY TO DEVELOP THE STATIC & DYNAMIC MODELLING FOR BATTERY TRAYS INCUBATIONS FOR AN ELECTRICAL AUTOMOTIVE VEHICLE

[1]Thara H.N., [2]Preethi K., [3]Dr. Shivappa H.A.

[1]M.Tech. Student (Machine Design) Post-Graduate Student, Department of Mechanical Engineering, Dr. Ambedkar Inst. of Tech., Mallathally, Nr. Jnanabharathi, Bangalore-560056, Karnataka
[2]Assistant Professor & Research Scholar, VTU Research Centre, Department of Mechanical Engineering, Dr. Ambedkar Inst. of Tech., Mallathally, Nr. Jnanabharathi, Bangalore-560056, Karnataka
[3]Assistant Professor Department of Mechanical Engineering, Dr. Ambedkar Inst. of Tech., Mallathally, Nr. Jnanabharathi, Bangalore-560056, Karnataka

***Abstract*: The research work presented in this paper discusses on some of the main objectives of the work are to conduct some simulation studies & carry out deep investigations on battery operated electric vehicles using sophisticated software tools such as Ansys, Catia, Hypermesh, ProE, etc. The following objectives were solved, i.e., a design for the tray to hold the battery of an EV has been proposed, to reduce the maximum deformation as much as possible, while raising the minimum frequency as much as possible, in other words, both the responses - maximum-deformation and base-frequency - are to be treated as objectives. One should be minimized while the other should be maximized, while the proposed design is made out of a single piece of sheet metal, here we use two different finite element models - one for static analysis, under the dead-weight of the batteries, and the other for the normal-modes calculation, to carry out the stress analysis during static & dynamic conditions, check the displacement analysis conducted when the system is moving, i.e., in motion (dynamics), to find out the condition of the EV system when it is static (the EV not moving), determine the stability of the system when the system is subjected to vibrations while the system is in moving condition, to carry out static and dynamic analysis of the batteries when the system is not moving and when the system is moving. Simulations are performed in ANSYS tool & the results are presented.**

***Keywords*: Electric Vehicles, Lithium-ion batteries, cells, materials, dynamic loads, vibrations, Dynamic simulation battery charging, batteries technology, EV charging Finite element, Uncertainties, Mechanical strength, Stress, Strain, Optimization.**

Project Methodology

In this section, the methodology that is being used in the project work is presented. A structured methodology for the project, starting from designing the solution to the implementation phase beginning of the requirement to the end results is presented. This includes simulation, validation, and optimization of products, and processes. The project was built by using the Using Altair Hyper works for Pre and Post Process along with Ansys tool. The simulation process generally includes three main phases [1]:

- Pre-processing – Upload of CAD model, defining the model and environmental factors
- Running solvers applications – Based on simulation analysis algorithms
- Post-processing – Validation of the model and 3D visualization.

3D Modelling & the FE Modelling Process

All the components are modelled by using CATIA V5 R17. The moduled components are exported to IGES format, which can retrieve by Hyper Mesh for the preprocessing of the part. The below-shown flow chart shown in the Fig. 1, which explains the experimental steps for the FE analysis. The accuracy of the FE analysis results solely depends upon the element size and meshing pattern of the parts. Hence, it is essential to give more focus on the meshing of the components. Mesh size is decided based upon the dimension of the critical locations of the parts, which is more important for the analysis. FE meshing is started with the geometry clean-up operation, which is used to make the geometry more appropriate for meshing. It is a customary practice used to combine a number of faces into a single smooth surface. This allows the elements to be created on the entire region at once and prevents unnecessary artificial or accidental edges from being present in the final mesh [2].

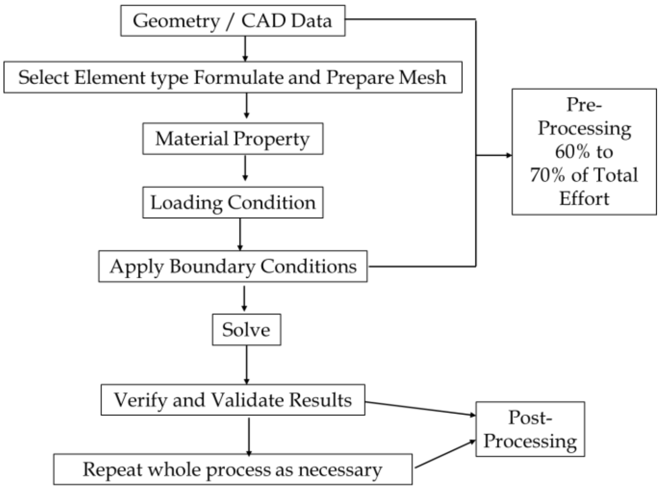

Fig. 1: Flow Chart - Work Flow Chart for FEA Modelling

During shell meshing, trim elements are used for global mesh size, while for critical locations number of elements is increased for accurate stresses at locations with high-stress gradients. The average size of the elements is 3 mm, taking a minimum size of 1 mm & maximum size equal to 5 mm. In the case of connecting rod, the average size of the elements is 2 mm, taking a minimum size of 1 mm & maximum size equal to 4 mm. CAD models that are over-complicated, or have defects that must be repaired, have long been one of the biggest impediments to using CAE software, particularly early in the design process, where it can do the best [3].

It used to be common to hear anecdotal claims that as much as 80% of the time spent on a typical CAE simulation was spent preparing and repairing the simulation model. Tools have gotten better, but the time to prepare the simulation model is still highly variable. Once the simulation model is ready, the next step is to use it to create the mesh. In the case of FEA, the mesh is embedded within, and fitted to, the body of the simulation model. The flow-chart itself gives a brief idea about the how the finite element modelling can be done for the battery vehicle in the form of tray designs [4].

The actual mechanics to create the simulation model and the mesh will depend on whether you're using CAD-integrated CAE software. After shell meshing, checking of the model quality index is performed. The quality parameters such as aspect ratio, warpage, skew, Jacobian, and minimum, maximum angles for the trial elements, and minimum and maximum length are checked. It is essential that the model should qualify through all these criteria. The failure of any one of the parameters leads to failure during 3D meshing or FE analysis. Hence, it is essential to give more focus on the meshing of the components [5].

Mesh size is decided based upon the dimension of the critical locations of the parts, which is more important. FE meshing is started with the geometry clean-up operation, which is used to make the geometry more appropriate for meshing. It is a common practice used to combine a number of faces into a single smooth surface. This allows the elements to be created on the entire region at once and prevents unnecessary artificial or accidental edges from being present in the final mesh. In order to mesh the geometry with smooth meshing, the shell mesh (2D) of the component is done, while for critical locations number of elements is increased for accurate stresses at locations with high-stress gradients [6].

Quality Parameters
After meshing, checking of the model quality index is performed for both Shell and Solid Elements is carried out. The quality parameters inspected are [7]

- Aspect ratio
- Charge ratio
- Warpage
- Skew
- Jacobian
- Minimum and maximum length
- Minimum and Maximum Tria and Quad elements

It is essential that the particular model should qualify through all of these criteria. The failure of any one of the parameters leads to the failure during 2D meshing or FE analysis [8].

Load calculations w.r.t. the battery tray
This section discusses the formulae, methods, and magnitudes of forces that are acting over the battery tray, the main objective of this chapter is to determine the magnitude and direction of the loads that act on the battery tray, which was then used in the FEA over an entire cycle. An analytical approach was used based on 3 DOF moving system, i.e., when the EV automobile is in motion, forces will be acting in all the 3 directions, i.e., the displacement will be there in *x-y-z* motions [9].

Static & Dynamic Analysis in EVs

As you all know that when the EV is in motion, dynamics comes in to picture, in that case, the vehicle as it is moving in the x-y plane (road) will be subjected to upward motion when a hump occurs or a pot hole occurs. Hence, it is necessary for us to perform an indepth static and dynamic analysis and then design the battery tray system for inhousing the batteries. In this chapter, the simulation results are presented along with the analysis and justification of the results that have been obtained using the simulation tools [10].

Dynamic and static characteristics analysis of battery tray

The stress and deformation of the battery tray under typical working conditions are obtained by using the finite element software '*Hypermesh* & *Ansys*'. Battery Tray static analysis is also conducted. All the DOF of the center node at the bolt hole are restrained, and the finite element analysis software & hypermesh is analysed using different calculative methods. Stress and deformation displacement of the battery tray are platted using Hyperview. Modal analysis of battery Tyra is also done. Modal analysis is mainly used to calculate the low-order modes of the battery tray, where the constraint mode of the battery tray is to limit all degrees of freedom of the nodes around the anchor bolt hole. Simulation tools for hybrid electric vehicles (HEVs) can be classified into steady-state and dynamic models, according to their purpose. Tools with steady-state models are useful for system-level analysis. The information gained is helpful for assessing long-term behavior of the vehicle. Tools that utilize dynamic models give in-depth information about the short-term behavior of sublevel components [11]

Materials and method - Establishment of finite element model of battery box

The battery box studied in this project is composed of top cover, side circumference, bottom plate, bracket and supporting foot, etc. Steel DC03 is selected to make battery box top cover, side circumference, bracket and plate, and steel SAPH440 is selected to make supporting foot. In the modelling process, CATIA is mainly used to construct 3D model, and assembly is mainly to constrain the axis and contact surface to achieve the purpose, as shown in Fig. 1 & 2 respectively. The top cover and side circumference are connected by bolts, while the bottom plate and side circumference, supporting foot and side circumference, bracket and bottom plate and other parts are connected by welding. There is no rigid connection between monomer and monomer, and between battery and side enclosure. The battery is positioned by contacting side enclosure. The benchmarked material information has been analyzed and compared based on strength to weight ratio, material cost, and sustainability which is the percentage of CO_2 emission. The potential materials were then selected for design and simulation of concepts. The materials used in the existing SPA2 battery housing were taken as a reference during the selection of materials [12].

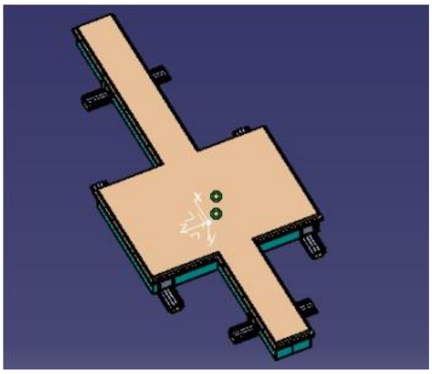

Fig 1 : 3D diagram of battery box (a)

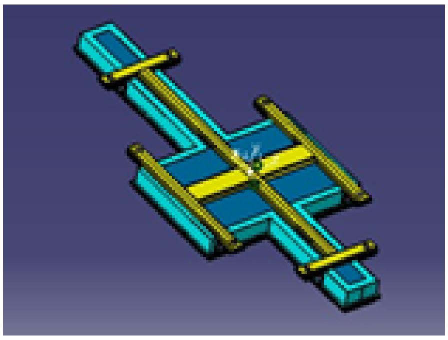

Fig. 2 : 3D diagram of battery box (b)

After the 3D solid model of the battery box is established in CATIA, it is converted into IGES format and imported into Hypermesh software for grid division. Then, it is necessary to carry out geometric cleaning of the model. As the battery box is mainly composed of a series of stamping thin-wall parts, so the shell element should be considered for grid division. The size of the unit should be determined according to the actual shape of the structure [13].

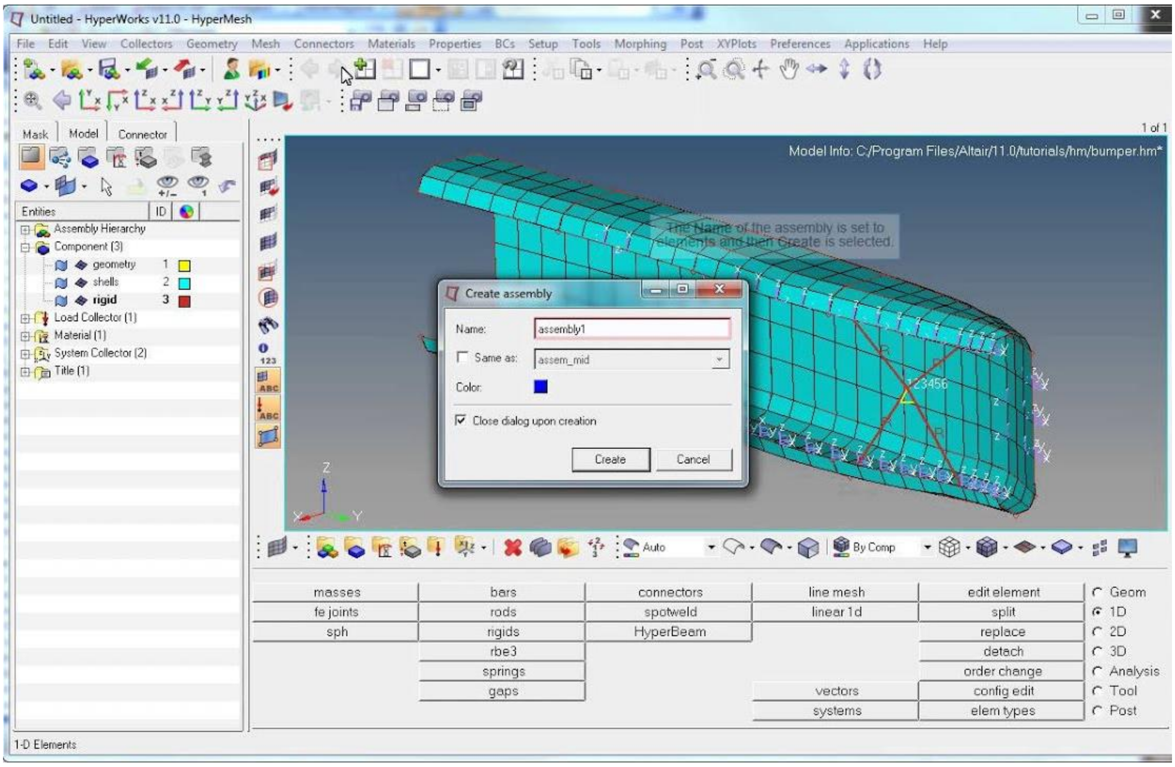

Fig. 3 : Hyper mesh design of the bottom plate of the EV bank battery system

The node coupling method is used to simulate spot welding, and the BEAM element in Hypermesh is used to simulate welding spot during static and modal analysis. The top cover and side circumference are connected by bolts, which can be simulated by using rigid elements, as shown in Fig. 3 in the Ansys tool. The welding seam is usually rigidly connected and simulated by coupling the joints at the welding seam of the two plates together. The simulation mode of welding seam between the supporting foot itself and the supporting foot and the supporting frame is shown in Figs. 6 to 8 respectively. The command window of the Ansys Software is shown in the Fig. 4 which is used for the simulation purposes after writing the program and running it along with hypermesh in Fig. 5 [14] [25].

Ansys
WORKBENCH

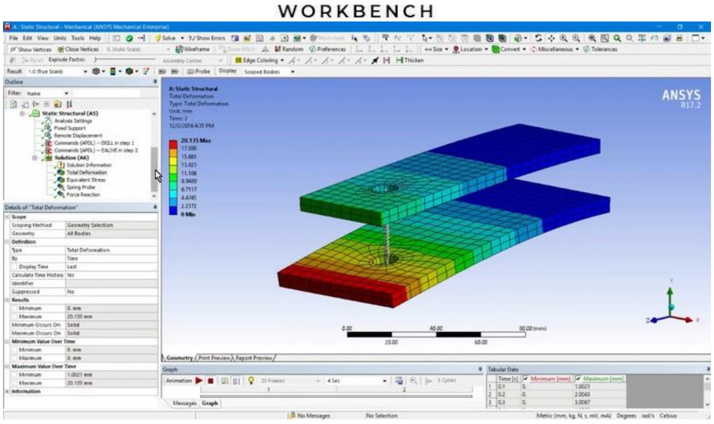

Fig. 4 : Snapshot of the Ansys window

Altair
HyperWorks

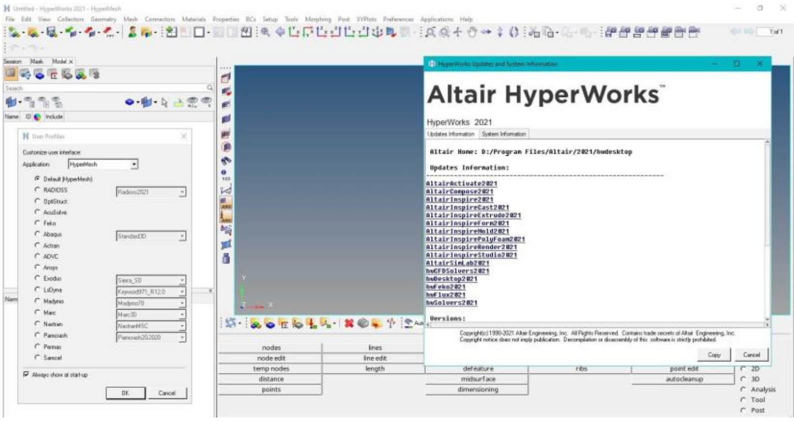

Fig. 5 : Snapshot of the Altair Hyperworks command window

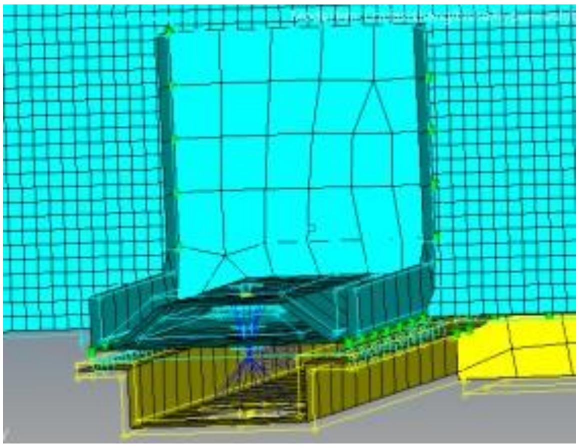
Fig. 6 : Bolt simulation mode

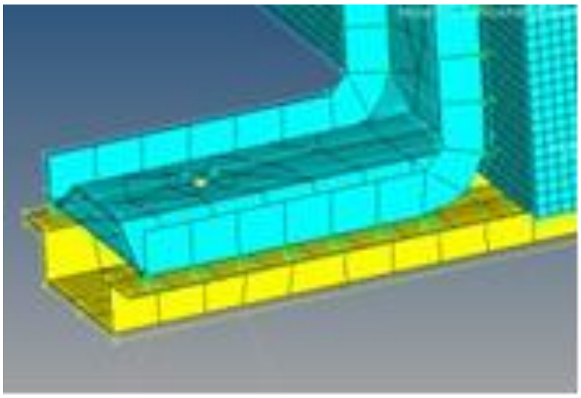
Fig. 7 : Weld simulation mode

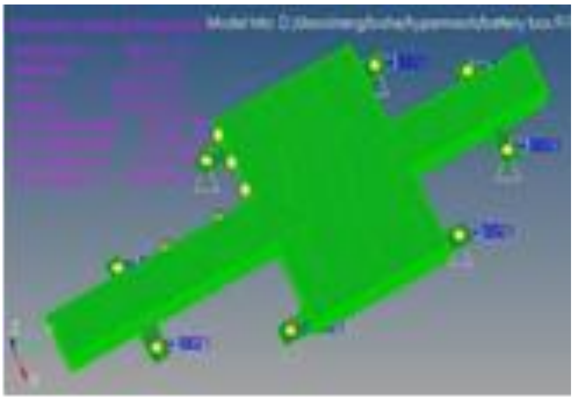
Fig. 8 : Finite element model of the developed battery box using Ansys software

The thickness and material property information of each component are added to their respective finite element models, thus the finite element modelling of the battery box is completed, as shown in Figs 6-8 respectively. Fig. 6 gives the bolt simulation mode along with the weld simulation mode shown in Fig. 7 & the FEM of the developed battery box using Ansys software in Fig. 8 [15] [24].

Dynamic and static characteristics analysis of battery box & Battery box static analysis

The stress and deformation of the battery box under typical working conditions are obtained by using the finite element software **ANSYS** & the results are shown in the further sections. Since the acceleration of the car is small at the beginning, only three conditions, namely road bumping, emergency braking and sharp turning, are considered. When it comes to specific analysis, the combinations of road bumping + emergency braking, road bumping + sharp turning are considered, and the boundary condition is the constraint of all degrees of freedom at the foot. When loading is added, the impact load generated by the battery block is equivalent to static load evenly applied to the inner and bottom nodes of the battery box. The effect of the battery on the soleplate, the side enclosure and the partition are applied to the joints of the soleplate, the side enclosure and the partition in a manner of distributing force [16] [23].

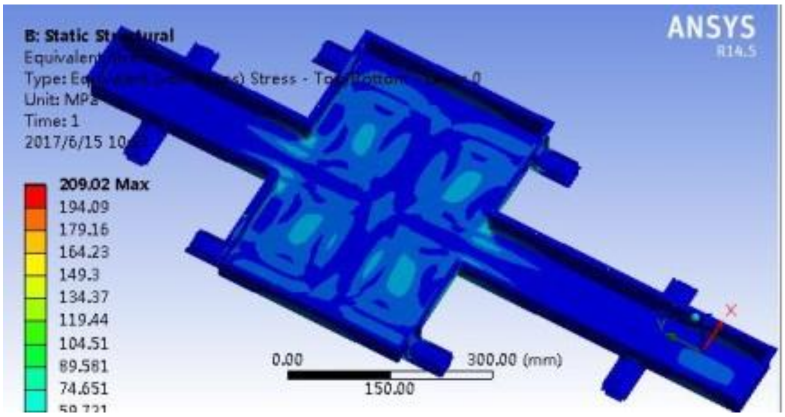

Fig. 9 : Stress cloud diagram of battery box under sudden braking

All the degrees of freedom of the center node at the bolt hole are restrained, and the finite element analysis software ANSYS is submitted for calculation. Fig. 9 shows the stress cloud diagram and deformation displacement of the battery box bottom plate and side enclosure when the car is driving on a bumpy road when braking sharply. Fig. 10 shows the deformation and displacement of battery box under sudden braking. Finally, the simulation result of the deformation and displacement of battery box under sharp turning process is displayed in the Fig. 11 [17] [22].

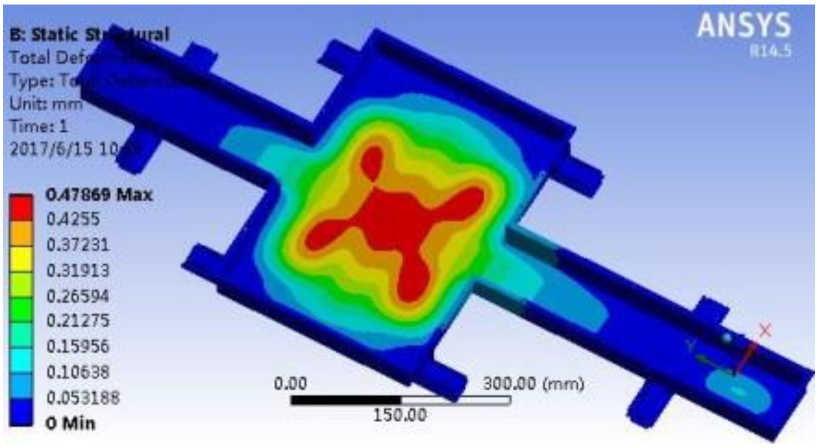

Fig. 10 : The deformation and displacement of battery box under sudden braking

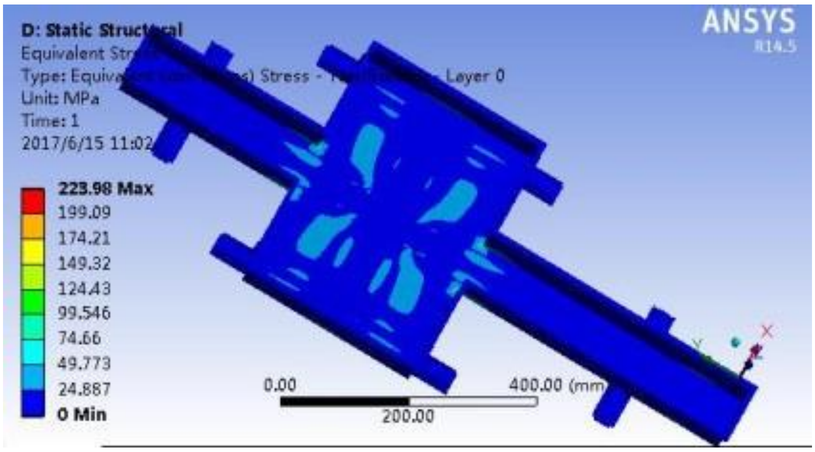

Fig. 11 : Stress diagram model of the battery box under sharp turning

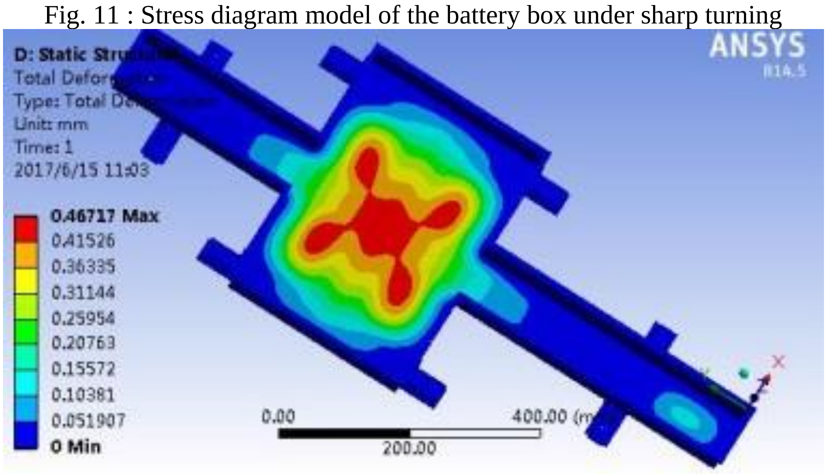

Fig. 12 : The deformation and displacement of battery box under sharp turning

When driving, the maximum stress on the battery box is 209.02 MPa under the condition of abrupt braking on bumpy road, mainly at the corner of the battery box side, and it is 223.98 MPa under the condition of sharp turning on bumpy road. The maximum strain of the battery box is 0.46 mm, which is mainly at the bottom of the box. The maximum stress suffered by the battery box under the above two working conditions is all greater than the yield limit of steel DC03—170 MPa, used as bottom plate and diaphragm material, which needs to be improved and is done herewith [18] [21].

Modal analysis of battery box

Modal analysis is mainly used to calculate the low-order modes of the battery box, where the constraint mode of the battery box is to limit all degrees of freedom of the nodes around the anchor bolt hole. The several parts of the first seven natural modes are shown in Fig. 13-16 respectively. The first seven orders of the battery box are mainly the vibration of the upper cover of the box, which indicates that the stiffness of the structure is relatively low compared with other parts. Generally speaking, the battery box structure is relatively reasonable [19] [20].

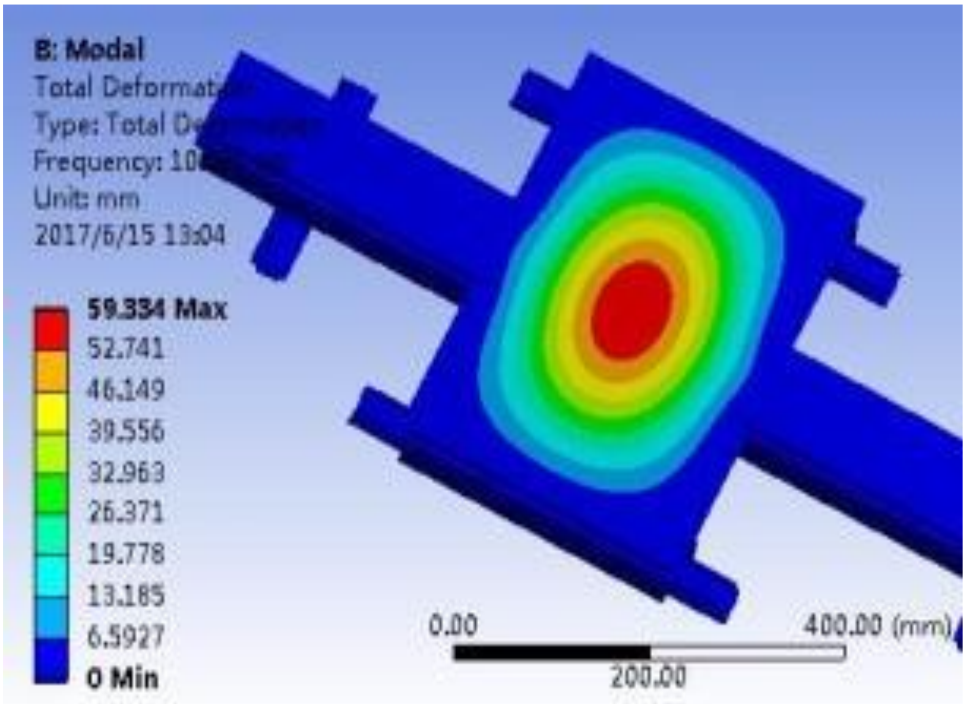

Fig. 13 : Modal shape 1

Conclusions

Project was successfully completed with the conduction of static & dynamic analysis of the battery tray for the electric vehicles. Different types of stress plots were plotted to check the stress levels at different parts of the battery tray when it is subjected to dynamic forces when the vehicle is moving. The displacement plots are also observed which occurs because of the vibration patterns that takes place when the vehicle moves over bumpy roads or pot-hole ridden surface. The different mode shapes are also studies from which we can come to a conclusion that which mode is the more dominant one as it has to be suppressed else the system may go into the resonance mode or reliability of the EV goes on decreasing. The design of the battery pack mounting is also taken care of and seen to it that space is well optimized and left for cooling purposes also. Ansys, Hypermesh was used for the simulation purposes. Models are developed and the analysis is carried out to observe the overall performance of the designed system. All the set objectives are solved herewith and arrived at excellent results in comparision to the works done by other researchers. A design for the tray to hold the battery of an EV has been proposed in order to reduce the maximum deformation as much as possible, while raising the minimum frequency as much as possible. In other words, both the responses - maximum-deformation and base-frequency - are treated and taken care of with. One should be minimized while the other should be maximized. Also, static analysis is conducted when the EV is stationary.

References

[1]. Lee H, Yanilmaz M, Toprakci O, et al. A review of recent developments in membrane separators for rechargeable lithium ion batteries. Energy Environ Sci 2014; 7: 3857–3886.
[2]. Zhang H, Zhou MY, Lin CE, et al. Progress in polymeric separators for lithium ion batteries. RSC Adv 2015; 5: 89848–89860.
[3]. Thorat IV, Stephenson DE, Zacharias NA, et al. Quantifying tortuosity in porous Li-ion battery materials. J Power Source 2009; 188: 592–600.
[4]. Xu H, Zhu M, Marcicki J, et al. Mechanical modeling of battery separator based on microstructure image analysis and stochastic characterization. J Power Sourc 2017; 345: 137–145.
[5]. Somerville L, Hooper JM, Marco J, et al. Impact of vibration on the surface film of lithium-ion cells. Energies 2017; 10: 741.
[6]. Brand MJ, Schuster SF, Bach T, et al. Effects of vibrations and shocks on lithium-ion cells. J Power Sourc 2015; 288: 62–69.
[7]. Zhou M, Yin H, Zhang T, et al. The fatigue life analysis of the battery bracket. In: Proceedings of the 3rd international conference on advances in energy and environmental science (ed P Yarlagadda), Zhuhai, China, 25–26 July 2015, pp. 742–746. Amsterdam, The Netherlands: Atlantis Press.
[8]. Hong SK, Epureanu BI and Castanier MP. Reduced-order modeling method for fatigue life predictions of hybrid electric vehicle batteries. In: Proceedings of the 2012 NDIA ground vehicle systems engineering and technology symposium, Troy, MI, USA, 14–16 August 2012.
[9]. Lu JC. Parametric reduced-order models for the structural dynamics of hybrid electric vehicle batteries. PhD Thesis. Ann Arbor, MI: University of Michigan, 2018.
[10]. Choi Y, Jung D, Ham K, et al. A study on the accelerated vibration endurance tests for battery fixing bracket in electrically driven vehicles. Proc Eng 2011; 10: 851–856.
[11]. Hooper JM and Marco J. Understanding vibration frequencies experienced by electric vehicle batteries. In: Proceedings of the IET hybrid and electric vehicles conference, London, UK, 6–7 November 2013, pp. 1–6. Stevenage, England: IET.

[12]. Lang JF and Kjell G. Comparing vibration measurements in an electric vehicle with standard vibration requirements for Li-ion batteries using power spectral density analysis. IJEHV 2015; 7: 272–286.
[13]. Hooper JM and Marco J. Characterising the in-vehicle vibration inputs to the high voltage battery of an electric vehicle. J Power Sourc 2014; 245: 510–519.
[14]. Shui L, Chen F, Garg A, et al. Design optimization of battery pack enclosure for electric vehicle. Struct Multidisc Optim 2018; 58: 331–347.
[15]. Zhou M, Yin H, Zhang T, et al. The fatigue life analysis of the battery bracket. In: Proceedings of the 3rd international conference on advances in energy and environmental science (ed P Yarlagadda), Zhuhai, China, 25–26 July 2015, pp. 742–746. Amsterdam, The Netherlands: Atlantis Press.
[16]. Hong SK, Epureanu BI and Castanier MP. Reduced-order modeling method for fatigue life predictions of hybrid electric vehicle batteries. In: Proceedings of the 2012 NDIA ground vehicle systems engineering and technology symposium, Troy, MI, USA, 14–16 August 2012.
[17]. Lu JC. Parametric reduced-order models for the structural dynamics of hybrid electric vehicle batteries. PhD Thesis. Ann Arbor, MI: University of Michigan, 2018.
[18]. Choi Y, Jung D, Ham K, et al. A study on the accelerated vibration endurance tests for battery fixing bracket in electrically driven vehicles. Proc Eng 2011; 10: 851–856.
[19]. Hooper JM and Marco J. Understanding vibration frequencies experienced by electric vehicle batteries. In: Proceedings of the IET hybrid and electric vehicles conference, London, UK, 6–7 November 2013, pp. 1–6. Stevenage, England: IET.
[20]. Lang JF and Kjell G. Comparing vibration measurements in an electric vehicle with standard vibration requirements for Li-ion batteries using power spectral density analysis. IJEHV 2015; 7: 272–286.
[21]. Hooper JM and Marco J. Characterising the in-vehicle vibration inputs to the high voltage battery of an electric vehicle. J Power Source 2014; 245: 510–519.
[22]. Wang H, Fu L, Bi J (2011) CO2 and pollutant emissions from passenger cars in China. Energy Policy 39(5):3005–3011
[23]. Wen X, Xiao C (2011) Electric vehicle key technology research in China. In: 2011 international aegean conference on electrical machines and power electronics and 2011 electro-motion joint
[24]. conference (ACEMP), 8 Sep 2011. IEEE, Istanbul, Turkey, pp 308–314, https ://doi.org/10.1109/ACEMP .2011.64906 16
[25]. Wierzbicki T, Sahraei E (2013) Homogenized mechanical properties for the jellyroll of cylindrical Lithium-ion cells. J Power Sour 241:467–476