

Analysis of Coach Driver Door with Hinge – A Numerical Study

Parmashwar, EjazKittur, P.S. Shivakumar Gouda

Department of Mechanical Engineering, SDM College of Engineering and Technology, Dharwad, Karnataka, India

Department of Design, TATA Marcopolo Motors Ltd, Belur Industrial Area Garag Road, Dharwad Karnataka, India

ABSTRACT-: Generally coaches have various types of doors. The coach driver door analysis is more essential in terms of driver safety by protecting humans from side crashes. In the present work initially coach door geometry is modeled using Solid Works modeling software. The stress analysis is done using the commercially available FEA packages. For the FE modeling pre-processor tool is used which prepares the model by applying the real time loading conditions. Further, a finite element analysis is carried out with a load of 500N to 1500N to determine the stress limits in door panel and its Hinges. Finally, the stresses, in the door panel and hinges were compared with chosen material properties for Finite Element analysis for the better door design.

KEYWORDS: Finite element analysis, Coach Door, Static analysis, Solidworks, HyperMesh, Optistruct.

I. INTRODUCTION

A coach driver door is, typically hinged, but sometimes attached by other mechanisms such as tracks, in front of an opening which is used for entering and exiting a vehicle. A bus driver door can be opened to provide access to the opening, or closed to secure it. These doors can be opened manually. Unlike other types of doors, the exterior side of the bus driver door contrasts sharply from its interior side (the interior side is also known as the car door panel): The exterior side of the door is designed of galvanized steel like the rest of the bus driver door exterior. In addition, its decorative appearance, typically coloured with a design, is intended to match with the rest of the bus driver door exterior. On the other hand, the vehicle door's interior side is typically made up of a variety of materials and the choice of cover materials depends on the rest of the styles used in the vehicle's inner body like the dashboard, carpet, seats, etc. However, unlike the material used on the exterior side of the vehicle door, the material on the interior side serves a greater purpose other than just aesthetic appeal.

While opening the bus driver door to enter into to the bus commonly the lower hinge will fail. This is because of maximum stress occurs at the lower hinge of the bus driver door. To avoid that maximum stress on the lower hinge handle is provided on the bus body nearer to the bus driver door. So that driver can enter into the bus with the help of both driver door handle as well as handle provided on the bus body and hence maximum stress can be minimized on the lower hinge of the driver door.

FEA is based on various formulations checking up numerical methods used is essential for estimation of validity of results [5, 6]. FEA is a powerful computational tool for analyzing complex structures like doors and window frames etc. If it is compared to experimental work, it saves much time and costs. Various types of tests are essential for the validation of the coach door [7]. Dynamic modal analysis involved determination of the lowest fundamental natural frequencies of the door. An Optistruct FEA solver is used to carry out the modal analysis. This is a linear dynamic analysis which utilizes a less is computational timethan a forced frequency response, analysis. This helps us to estimate the vibrational characteristics of the door model. In modal analysis, if the operating frequencies are known which may coincide with the natural frequencies of the door may cause resonance or failure of the door joints, the design can be modified to avoid these frequencies.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

The strength of driver door by applying forces carefully at critical points on various components of each door for extracting the displacement and stresses. The effect of force on the whole door is analyzed while in the later, certain closures in the door are also analyzed. Stress results are compared with yield and ultimate strength of the material of the component. It must be lower than it with a certain factor of safety. Displacement results are used for calculating stiffness of the component at a specified point through dividing the applied force with resulting displacement and the calculated stiffness values are compared to target. If results of some tests do not match targets, modifications must be suggested to some components such as web addition or component thickness increasing or some related parameters changing.

II. METHODOLOGY

The flow chart shown in Fig.1 reveals the different steps involved in the analysis of coach driver door with hinge support.

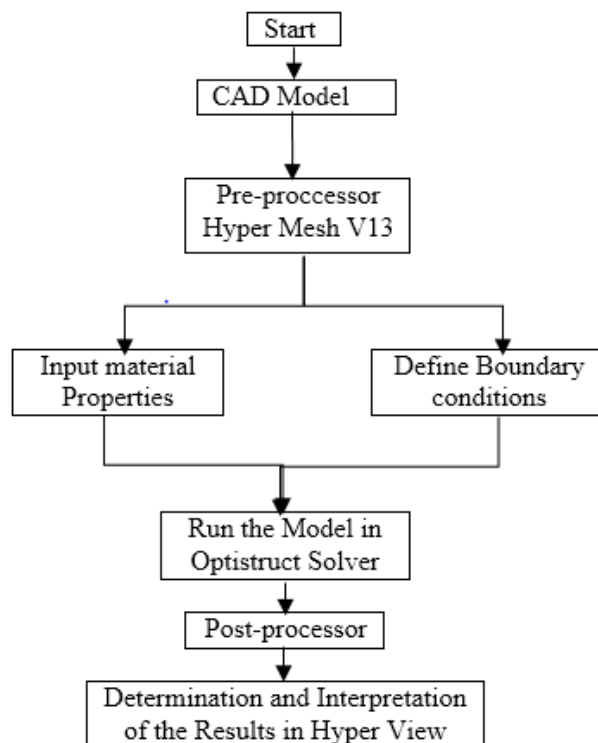


Figure 1: Flowchart showing different steps in the analysis of coach door.

III. GEOMETRICAL MODELING

The CAD software Solid Works is a mechanical design automation application that lets designers quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings.

A Solid Works model consists of 3D geometry that defines its edges, faces, and surfaces. Solid Works uses a 3D design approach. As you design a part, from the initial sketch to the final result, you create a 3D model. From this model, we can create 2D drawings or mate components consisting of parts or subassemblies to create 3D assemblies. We can also create 2D drawings of 3D assemblies. When designing a model using Solid Works, you can visualize it in three dimensions, the way the model exists once it is manufactured.

The design process usually involves the following steps:

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

- Identify the model requirements.
- Conceptualize the model based on the identified needs.
- Develop the model based on the concepts.
- Analyze the model.
- Prototype the model.
- Construct the model.
- Edit the model, if needed.

Initially the rough design model was created and it is helpful to plan out a method of how to create the actual model. After it needs to be isolated the appropriate concepts, then one can develop the model:

Sketches – Create the sketches and decide how to dimension and where to apply relations.

Features – Select the appropriate features, such as extrudes and fillets, determine the best features to apply, and decide in what order to apply those features.

Assemblies – Select the components to mate and the types of mates to apply

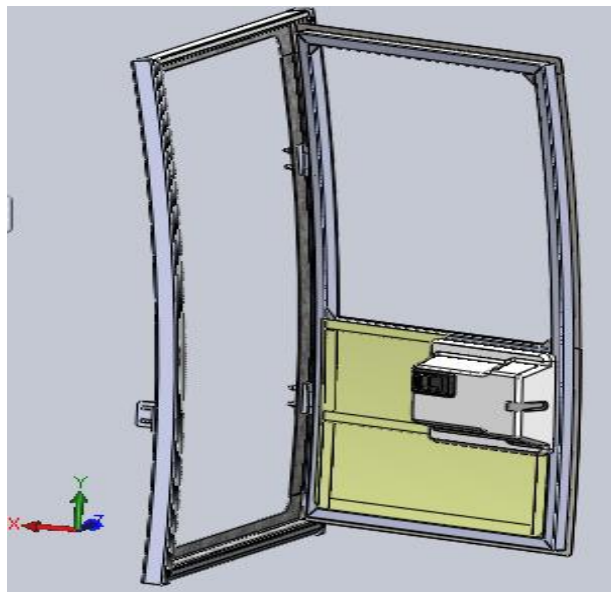


Figure 2:3-D Model of the coach driver Door

IV. FINITE ELEMENT MODELING

HyperMesh -V13 is the pre-processor tool used to carry out the meshing of the imported geometry from Solid works, hyper mesh is user friendly tool used for meshing complex geometrical shapes in the industry as it provides more and better meshing techniques. Hyper Mesh is a high-performance finite element pre-processor and it is used for major finite element solvers, which allows engineers to analyze design conditions in a highly interactive and visual environment. Hyper Mesh's user-interface is easy to learn and supports the direct use of CAD geometry and existing finite element models, providing robust interoperability and efficiency. Advanced automation tools within HyperMesh allow users to optimize meshes from a set of quality criteria, change existing meshes through morphing, and generate mid-surfaces from models of varying thickness. Following are the benefits of the Hypermesh

- Reduce time and engineering analysis cost through high-performance finite element modeling and post-processing
- Reduce learning time and improve productivity with an intuitive user-interface and best-in-class functionality
- Reduce redundancy and model development costs through the direct use of CAD geometry and existing finite element models
- Simplify the modeling process for complex geometry through high-speed, high-quality auto-meshing

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

- Support numerous commercial solvers by providing direct interfaces to a wide array of analysis codes ensuring the best code can be used for specific situations

Usually we used the Hexa or brick mesh for following reasons

- The number of elements and nodes generated by a brick mesh are of the order of 1/2 to 1/50 in comparison to a tetra mesh. A brick mesh reduces the solution time and results in the ease of handling the model on a workstation (pre and post display).
- Analysis types like crash or nonlinear give preference to brick mesh due to the number of nodes and mesh flow lines
- The time consumed in brick meshing is more and requires experience, hard work, and a lot of patience too.

Table 1: Details of the element used for meshing

Number of Nodes	98998
Number of Elements	90877
No. of DOF at each Node	6
Element Type	Quad & Hexa

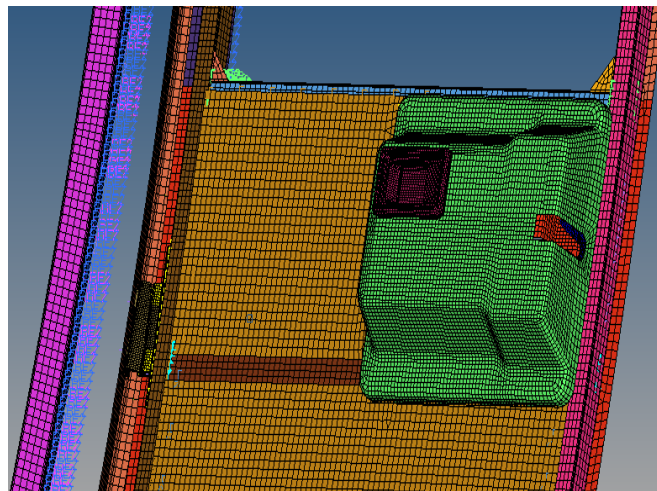


Figure 3:FE meshed Model using Quad elements for door panel

V. FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) is a computer based method of simulating/analysing the behaviour of engineering structures and components under a variety of conditions. It is an advanced engineering tool that is used in design and to augment/replace experimental testing.

The method is comprised of three stages

- Pre-processing:* in which the analyst develops a finite element mesh of the geometry and applies material properties, boundary conditions and loads
- Processing:* during which the program derives the governing matrix equations ($\text{stiffness} \times \text{displacement} = \text{load}$) from the model and solves for the displacements, strains and stresses.
- Post-processing:* in which the analyst obtains results usually in the form of deformed shapes, contour plots etc. which help to check the validity of the solution.

A. Boundary Conditions

The Figure 4 showing boundary conditions imposed on the coach driver door meshed model. The imposed boundary conditions are

Force: Load of 500N applied near to the door handle.

Constraint: Both hinges are fixed in all DOF.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

RBE2: Rigid body element can be used for connecting two nodes to model a 'rigid' welded type connection with all 6 DOFs, or model a pinned connection with 3 translation DOFs.

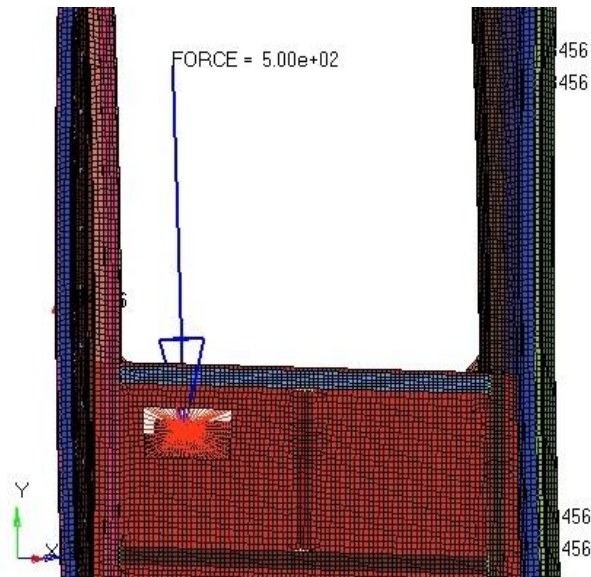


Figure 4: Loading and Boundary condition

B. Displacement in Door

Figure 5 shows the displacement plot in which we can observe that the maximum displacement is 1.271mm and it is close to the bottom of the handle. This is also because of the point load acting near to the handle it causes bending moment of the door about z-axis.

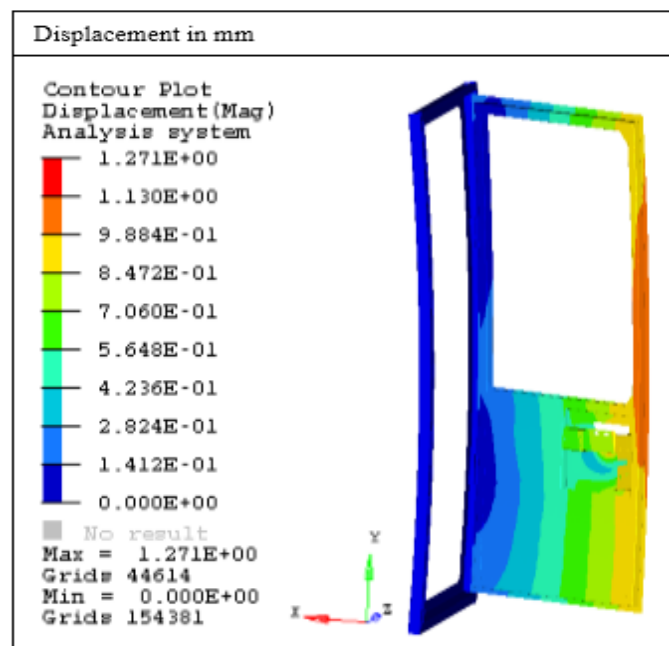


Figure 5: Displacement Plot of the Driver Door

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

C. Stress Distribution in Door

After applying load of 500N near to the door handle, maximum stress occurring at the handle panel corner and also maximum stress occurring at the lower hinge corner as shown in Figure 6(a) and (b). From the stress plot it is absorbed that maximum stress at the handle panel corner is 157.5MPa and maximum stress at the lower hinge corner is 153.9MPa. Numerical results of finite element analysis of the driver door are within the elastic stress limit (assuming Galvanized steel of strength 203.94MPa) means design is safe for applied load.

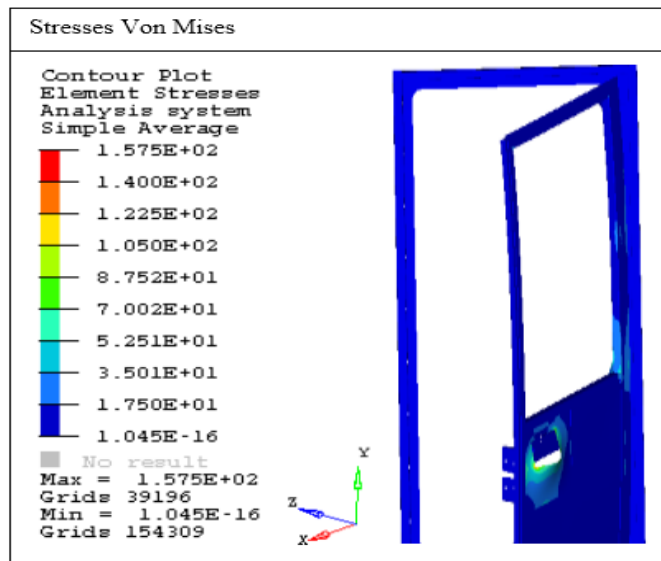


Figure 6(a): Stress plot on door panel when the load at the door handle

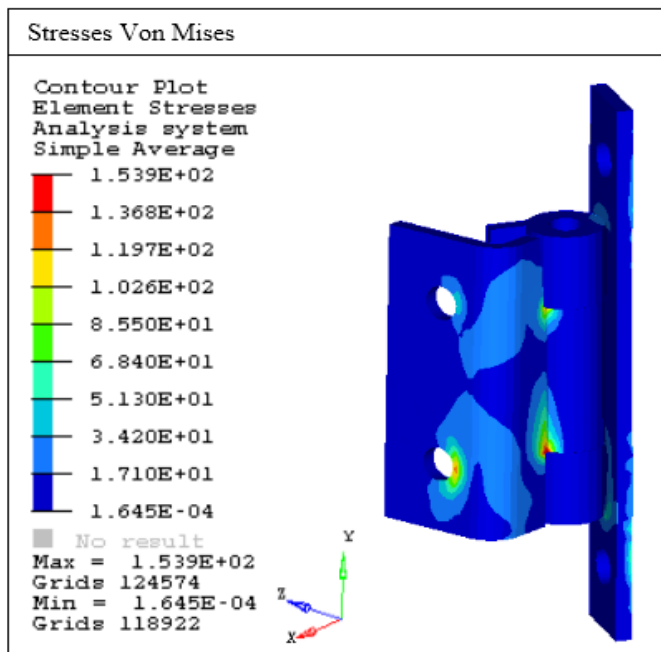


Figure 6 (b): Stress plot for on lower hinge when the load at the door handle

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

D. Load Capacity of the Hinge

After applying load of 500N on the door hinge maximum stress is 60.35MPa occurring very near to the hinge plate hole and also maximum displacement of the hinge is 0.01445mm as shown in Figure 7 and Figure 8. Stresses on the hinge are lower as compare to yield strength of the galvanized steel material. Hence design is safe for a load of 500N. Similarly we have analysed for different loads on the hinge to find out maximum capacity of the hinge. But for load of 2000N maximum stress is 301.7MPa occurring very near to the hinge hole and also at the hinge corner. To avoid failure near to the hinge hole, slightly hole should be away from the corner fillet of the hinge. Also to avoid tearing at the sharp corner of the hinge proving round fillet at the hinge corner.

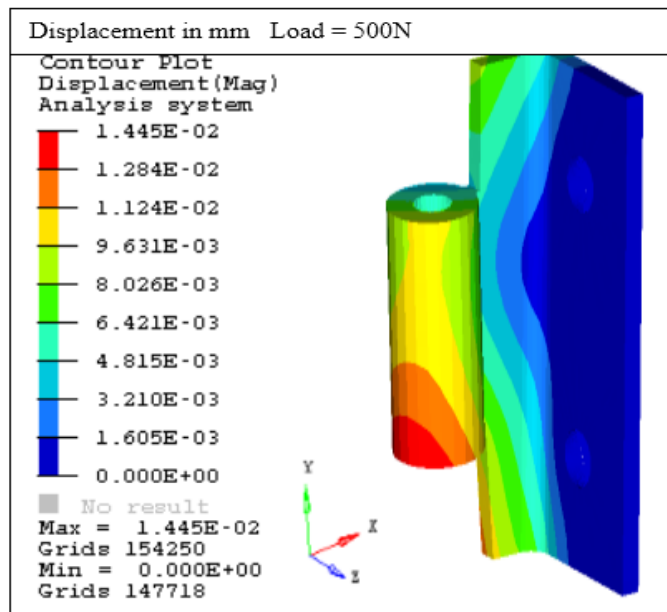


Figure 7: Displacement plot for load capacity of the hinge

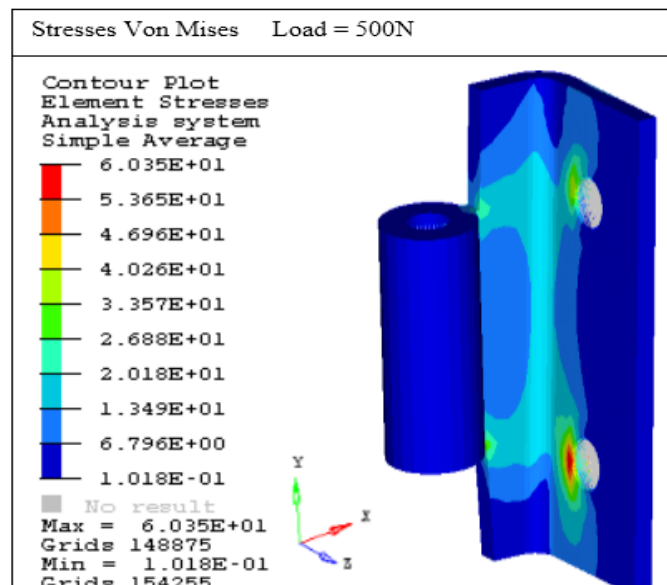


Figure 8: Stress plot for load capacity of the hinge

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

Table 2:Maximum Load Capacity of the Hinge

Load (N)	Max. Displacement (mm)	Max. Stress (MPa)
200	0.005779	24.14
500	0.01445	60.35
1000	0.02289	130.2
1500	0.04334	181
2000	0.07223	301.7

The Table 2 summarizes maximum stress in the Hinge for varying loads in the range of 200N to 2000N. It is absorbed that maximum load capacity of the hinge is 1500N, for which the stress is 181MPa. This is lower than the yield strength of hinge material 203.94MPa. Further increase in load to 2000N, the maximum stress observing is 301.7MPa and which is above than the yield strength. Hence design is not safe for a load of 2000N. To carry load of 2000N hinge thickness must be increase from 4mm to 5mm. Also providing one more extra hole on the hinge plate to increase the load capacity of the hinge. To provide extra hole on the hinge plate the length of the hinge plate must also be increase.

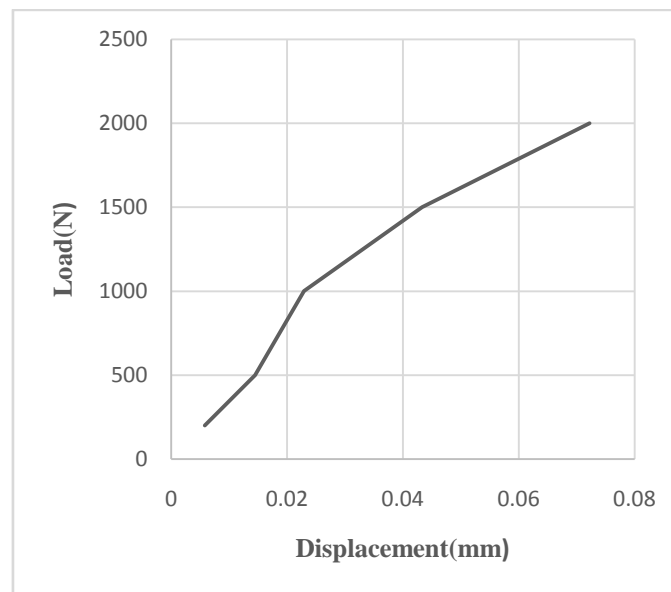


Figure 9: Load versus Displacement Curve

E. Modal Analysis

Modal analysis is the study of the dynamic properties of structures under vibrational excitation. Modal analysis is the field of measuring and analysing the dynamic response of structures. Figure 10 (a) and (b) shows the first two modes that are fundamental modes while the second two modes are global modes as shown in Figure 11 (a) and (b). The fundamental modes with natural frequencies 90.73 and 110.7 Hz respectively and oscillates mainly inner surface while the global modes with natural frequencies 114.56 (target >114Hz) and 128.4 Hz respectively and oscillates mainly outer surface of the door. The first mode of the global modes is out of target and so, indicates that it needs to increase its stiffness. This can be achieved by inserting metal partition inside the opening, to increase the local stiffness near lower slide fixing.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

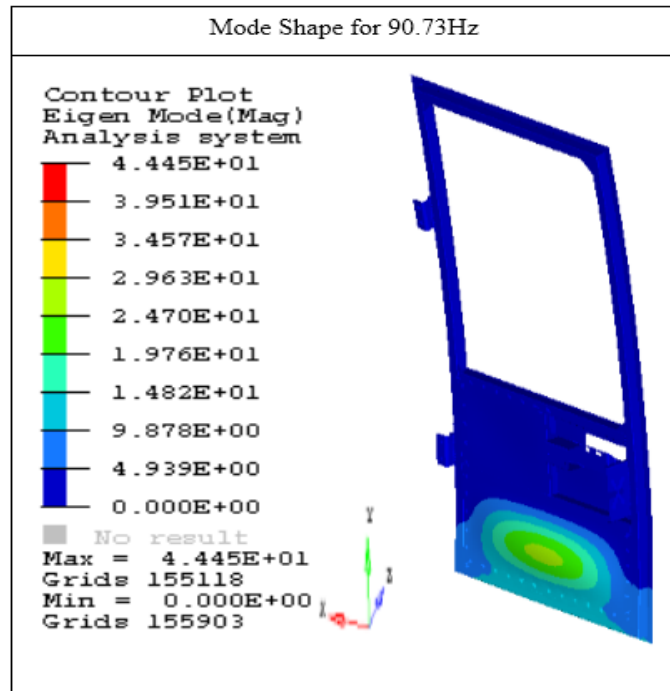


Figure 10(a): Local mode shapes of the driver door

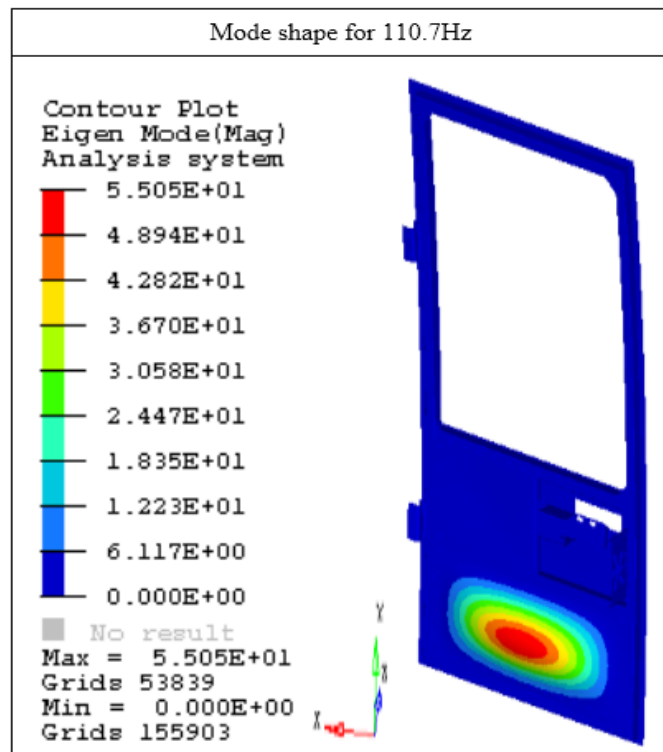


Figure 10(b): Local mode shapes of the driver door

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

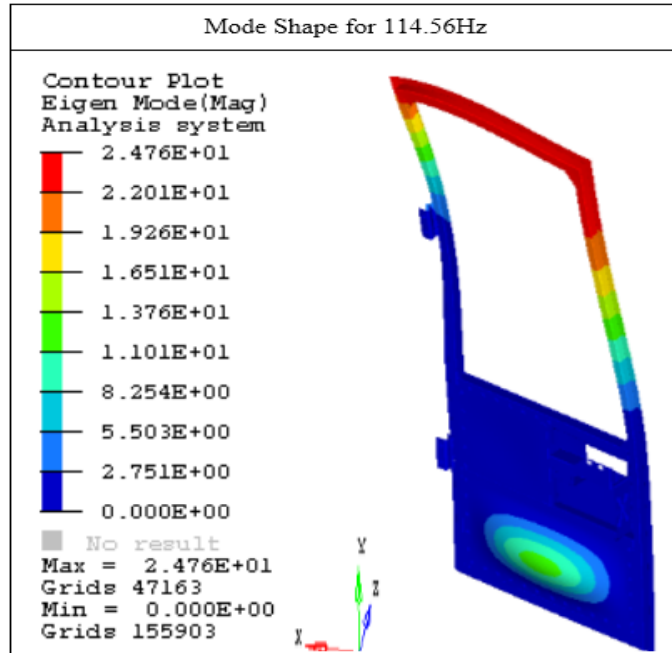


Figure 11(a): Global mode shapes of the driver door

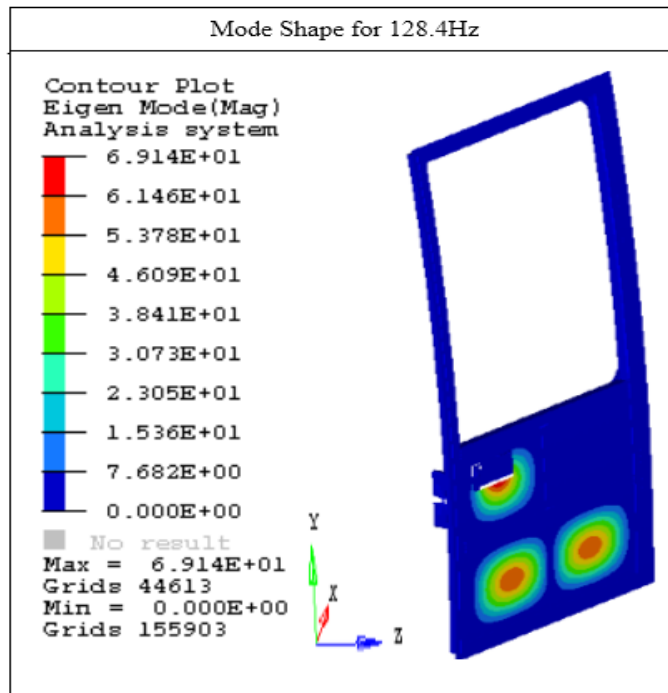


Figure 11(b): Global mode shapes of the driver door

F. Sag Analysis of the Door

For the door component load case “Door Sag” the door is rigidly constrained at the mounting points for the hinges. All degrees of freedom are constraint except for the rotation around the hinge axis. A force of 500N is applied in vertical

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

direction at the door lock. The point of application of load is constrained in transversal direction in order to suppress a global rotation of the door around the hinge axis. Sag analysis is done to simulate the bending of the door about the perpendicular axis to the hinge axis. The door will sag due to continuous closing and opening of the door and also due to weight of the door is continuously acting on the hinge. Commonly upper hinge will undergo elongation which causes the door to sag. The scalar stiffness value for the load case “Door Sag” is calculated by dividing 500N by the total displacement at the force application point.

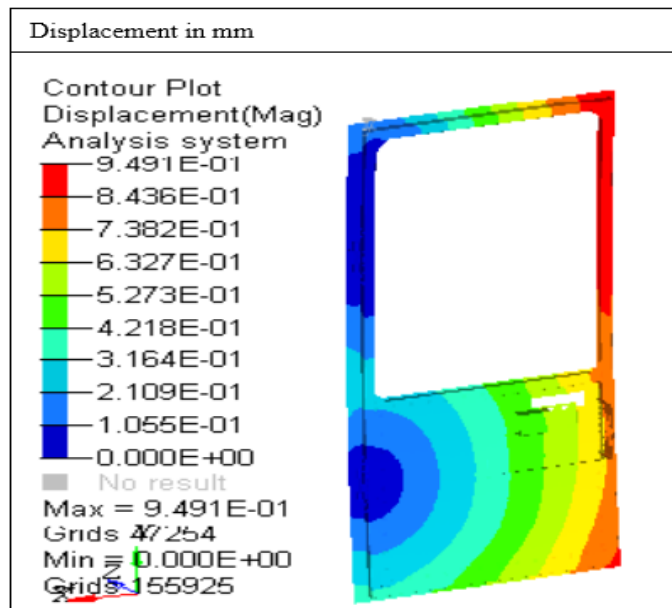


Figure 12: Displacement Plot Due Load on the Latch

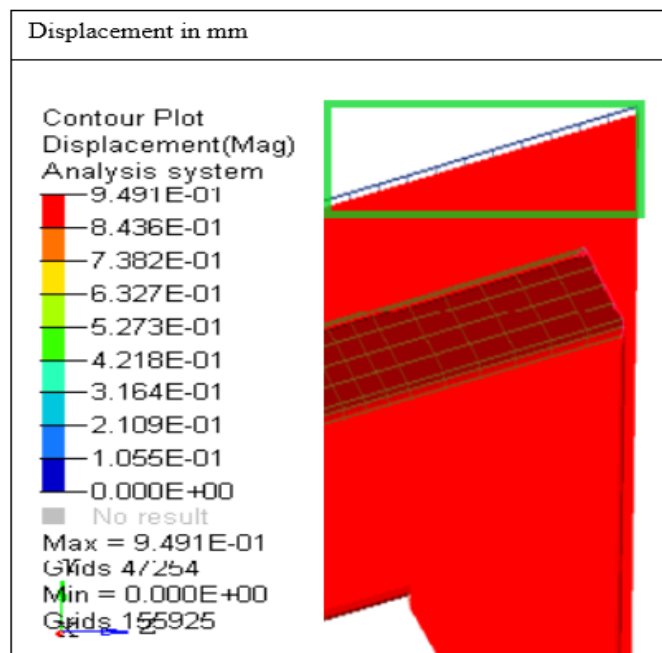


Figure 13: Door Sag due to Load on the Latch

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

From the Figure 15 it is absorbed that maximum stress is 75MPa on the upper hinge of the door which is lower than the yield strength of the door hinge material. Hence design is safe for a given load. The maximum stress occurring at the upper hinge because of continuous loading on the door and also because of all degrees of freedom are constraint except for the rotation around the hinge axis. To avoid maximum stress on the hinge we can use pre-stressed door hinge.

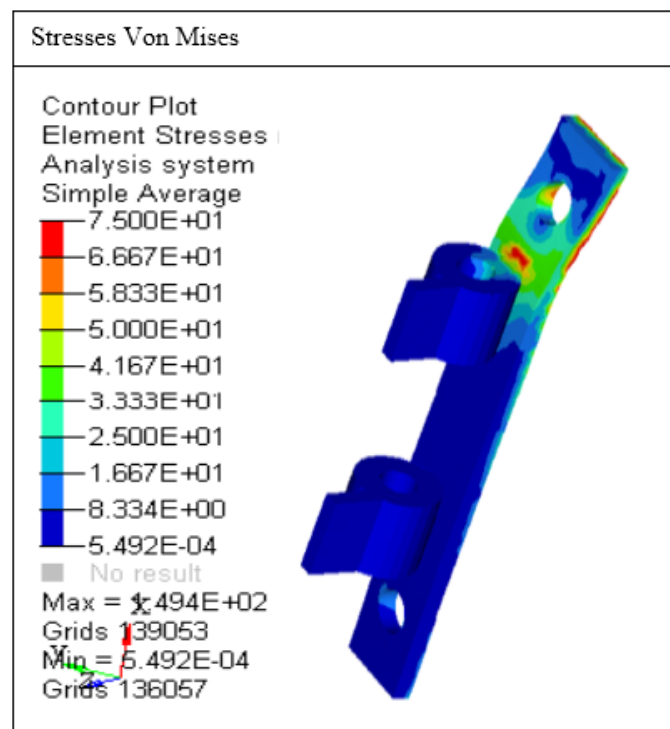


Figure 14: Stress plot on upper hinge due to door sag

VI. CONCLUSION

Following are the conclusion drawn from the above work is

1. The stress at the door handle panel corner is 157.5MPa and stress at the lower hinge corner is 153.9MPa. These stress values are lower as compared to the yield strength of the door material. Hence it is concluded that the design is safe for applied load of 500N.
2. The maximum load capacity of the hinge is 1500N with stress of 181MPa, which is lower than the yield strength of the hinge 203.94MPa. Further increase in load to 2000N, the stress observed is 301.7MPa which is above the yield strength. By looking at the above stress values for 1500N and 2000N, it is concluded that the maximum load carrying capacity of 1500N is safe for the given hinge material.
4. The fundamental natural frequencies for the given door material is 90.73 and 110.7Hz respectively. It is observed from mode shapes that the door panel oscillates mainly at inner surface. Whilst with natural frequencies of 114.56 (target >114Hz) and 128.4 Hz the door panel oscillates mainly at outer surface of the door. To reduce the door panel oscillation by inserting metal partition inside the opening and to increase the local stiffness near lower slide of the door.
5. The door sag analysis shows that the upper frame is undergoing slight bending including the inner vertical frame and slight local buckling of the door panel. Further, the maximum stress is 75MPa on the upper hinge of the door which is lower than the yield strength of the door hinge material. Hence design is safe for a given load. To avoid maximum stress on the hinge we can use pre-stressed door hinge.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

ACKNOWLEDGEMENT

The authors express their thanks to Dr.S. B.Vanakudre, Principal SDMCET Dharwad and Facilities provided to carry out this work by TATA Marcopolo Motors Limited Dharwad, India, and Prof.V.K.Heblikar Head, Mechanical Engineering Department also to the staff of Mechanical Engineering Department, S.D.M College of Engineering and Technology, Dharwad, India.

REFERENCES

- [1] Said Darwish, H. M. A. Hussein and Ahmad Gemeal, "Numerical Study of Automotive Doors" International Journal of Engineering & Technology IJET -IJENS Vol: 12 No: 04 August 2012.
- [2] Chandru B. T, Maruthi B. H, Dileep Kumar, Vinay. V. Nesaragi, "NVH Characteristics Study of Automotive Car Door by Numerical Method" International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 3 Issue 6, June – 2014.
- [3] M. Raghuvver, Ganti Satya Prakash, "Design and Impact Analysis of a Car Door" International Journal of Modern Engineering Research (IJMER), ISSN: 2249-6645, Vol. 4 Iss. 9 Sept. 2014.
- [4] Ika, "Stiffness Relevance and Strength Relevance in Crash of Car Body Components", Public version of official report 83440, 2ed. May 2010.
- [5] M. B. Kanchi, Matrix Methods of Structural Analysis, New Delhi: Wiley Eastern Limited. 2nd ed., 1993.
- [6] R. D. Cook, D. S. Malkus and M. E. Plesha, Concepts and Applications of Finite Element Analysis, 3 ed., New York: John Wiley & Sons, 1989.
- [7] M. Grujicic, G. Arakere, V. Sellappan and J. C. Ziegert, "MultiDisciplinary Design Optimization of a Composite Car Door for Structural Performance, NVH, Crashworthiness, Durability and Manufacturability," Journal of Materials Processing Technology, 2008.
- [8] R. W. Clough and J. Penzien, Dynamics of Structures, 2 ed., Singapore: McGraw Hill Book Co., 1999.
- [9] C. Lanczos, "An Iteration Method for the Solution of Eigen Value Problem of Linear Differential and Integral Operators," J. Natn. Bur. Stand., vol. 45, pp. 255-282, 1987.
- [10] T. J. Hughes, "The Finite Element Method—Linear Static and Dynamic Finite Element Analysis," Prentice-Hall, Englewood Cliffs, 1987.
- [11] A. E. Inc., "HyperStudy, User Manual," Troy, MI, 2007.
- [12] J. Fang, C. Hoff, B. Holman, F. Muelle and D. Wallerstein, "Weld Modelling with MSC. Nastran," in 2nd MSC Worldwide Automotive Conference, Dearborn, MI, 2000.
- [13] V. 7. MD Nastran R3 Quick Reference Guide, "MSC," Costa Mesa, Los Angeles, USA, 2008.
- [14] M. Palmonella, J. E. Mottershead and M. I. Friswell, "Guidelines for the implementation of the CWELD and ACM2 spot weld models in structural dynamics," Finite Elements in Analysis and Design, vol. 41, no. 2, Nov. 2004.
- [15] M. Palmonella, M. Friswell, J. Mottershead and A. Lees, "Finite element models of spot welds in structural dynamics: review and updating," Comput. Struct, vol. 83, no. 8-9, pp. 648-661, 2005.
- [16] Y. Chao, "Ultimate strength and failure mechanism of resistance spot weld subjected to tensile, shear or combined tensile/shear loads," J. Eng. Mater. Tech.-Trans., pp. 125-132, 2003.
- [17] H. P. Mlejnek and R. Schirmacher, "An engineer's approach to optimal material distribution and shape finding," Computer Meth. in Appl. Mechanics and Engng, vol. 106, p. 26, 1990.
- [18] S. I. Song, J. S. Im, Y. M. Yoo, J. K. Shin, K. H. Lee and G. J. Park, "Automotive door design with the ultra-light steel auto body concept. Using structural optimization," in 20th International Congress on Theoretical and Applied Mechanics, Chicago, Illinois, 2000.