RESEARCH ARTICLE OPEN ACCESS

Finite Element Method for Stress Analysis of Passenger Car Floor.

Abhijeet Shinde*, Prof. Dhananjay Thombare**
*(M. Tech Automobile Student at Department of Automobile Engineering, Rajarambapu Institute of Technology, Sakharale, Sangali, India)
** (Professor at Department of Automobile Engineering, Rajarambapu Institute of Technology, Sakharale, Sangali, India)

ABSTRACT
Simple structural surfaces (SSS) method of analysis is comparatively easy to understand. This technique may be used as a preliminary step to carry out Finite Element Analysis. This method is particularly useful in assessing possible applied load paths. Finite element analysis (FEA) involves solution of engineering problems using computers. Finite element method provides a high level of accuracy for many types of load case and can even give a reasonable prediction of accident damage. Now a day's there are various computational solvers used for analysis. In this paper only review for element type used, constraint used, load and boundary conditions which used for static strength analysis of floor of passenger car that is pre-processing step in finite element method. This paper will help for how pre-processing is done in static stiffness analysis of the car body.
***Keywords* -** Constraint, FEA, Finite element method, NASTRAN, Simple structural surfaces

I. INTRODUCTION TO FINITE ELEMENT METHOD

An intermediate analysis method was proposed in 1964 by Pawlowski is called 'Simple Structural Surfaces' (SSS)[2]. This method can be used to determine the loads on and then the stresses in the main structural elements of an integral structure. The procedure is to devise a simple representation of a structure using a small number of flat sheets (and beams). Sheets can represent the body floor, roof structure, bulkheads that is wall for safety, and side panels etc. We can calculate loads transmitted between each SSS and then the stress in each and their deflections can be found.

Finite Element Method (FEM) is standard method in the automotive industry for the prediction of vibrational and acoustical response of vehicles. Finite element method includes problems like structural analysis, fluid flow, heat transfer, mass transfer, and electromagnetic potential. Process of modeling a body by dividing it into an equivalent system of smaller bodies or units (finite elements) interconnected at points common to two or more elements (nodal points or nodes) and/or boundary lines and/or surfaces is discretization. In this paper how computational finite element analysis method is used is explained.

Evolution of FEA is tied with the development in computer technology. The enhancement in computer speed and storage capacity, FEA has become a very valuable engineering tool. NASA is credited by their development of comprehensive FEA software in 1960's, known as NASTRAN. In this paper there is description related to the how finite element method is used for analysis and three main steps in it. Mesh quality parameters, spot welding method, load and boundary condition application while doing static analysis of passenger car floor. Main dissertation work belongs to doing modeling of the underbody of XX car model modified and existing [1]. After preparation of model step comes of pre and post processing for solving that for stress analysis using NASTRAN as solver.

II. STRESS ANALYSIS OF PASSENGER CAR FLOOR

Finite element analysis (FEA) involves solution of engineering problems using computers as computational methods. Engineering structures that have simple as well as complex geometry and load application, are very difficult to analyze or have no theoretical solution. However, using FEA, a structure of this type can be easily analyzed within lesser time. Commercial FEA programs are written so that a user can solve a complex engineering problems without knowing the governing equations or the mathematics; the user is required only to know the geometry of the structure and applied boundary conditions to it. FEA software's provides us a complete solution including deflections, stresses, reactions, etc[3].

FEA technique facilitates an easier and a more accurate analysis solutions. In this technique given structure is divided into very small but finite size elements (hence called as finite element analysis). We can get individual behavior of these elements and, based on this behavior of the entire structure is determined. Passenger car floor is made up of various components and of varying thickness of various parts. These all parts are discretized in small elements for getting the input to FEA. All the components are connected to each other by spot weld. It is easy after using FEM for getting stress levels after applying loads to under body structure.

III. THREE STEPS OF FINITE ELEMENT ANALYSIS PROCESS:

3.1 Pre-processing:

Using a CAD program that the structure is modeled. With CAD tool surfacing has to be done as parts are made up of sheet metals and only surface without thickness are easy to modeled. The final FEA model consists of several elements that collectively represent the entire structure. The elements not only represent segments in the structure given, they also simulate their mechanical behavior and properties. Regions where geometry is complex require increased number of elements to accurately represent the shape; whereas, the regions with simple geometry can be represented by coarser mesh.

In the pre-processing phase, the geometry of the structure as well as the constraints, loads and mechanical properties of the structure are defined. Thus, in pre-processing, the entire structure is completely defined by the geometric model. The structure represented by nodes and elements is called "mesh".

3.2 Solving:

In this step, the geometry, constraint positions, material, properties and loads are applied to generate matrix equations for each element. After that these equations are assembled to generate a global matrix equation of the structure. The individual equations and also the structural equation is always in form of,

$$\{F\} = [K]*\{u\} \qquad \dots (1)$$

Where,

{F} = External force matrix.

[K] = Global stiffness matrix

{u} = Displacement matrix

The equation (1) is then solved for deflections. Using the value of deflection from above equation strain, stress, and reactions are calculated. All the results are stored then can be used to create graphic plots and charts in the post analysis.

3.3 Post processing:

This is the last step in a finite element analysis. Results obtained in solving step are usually in the form of raw data and difficult to interpret. In post processing means post analysis, a computer aided designing program is utilized to manipulate the data for generating deflected shape of the structure. We can also get stress plots and also animation result, etc. A representation in graphical form of the results is very useful in understanding behavior of the structure.

Here pre-processing is explained that is building finite element model.

IV. FLOW CHART

From the figure1 flow chart shows how preprocessing is done using the Finite element method is given in this paper.

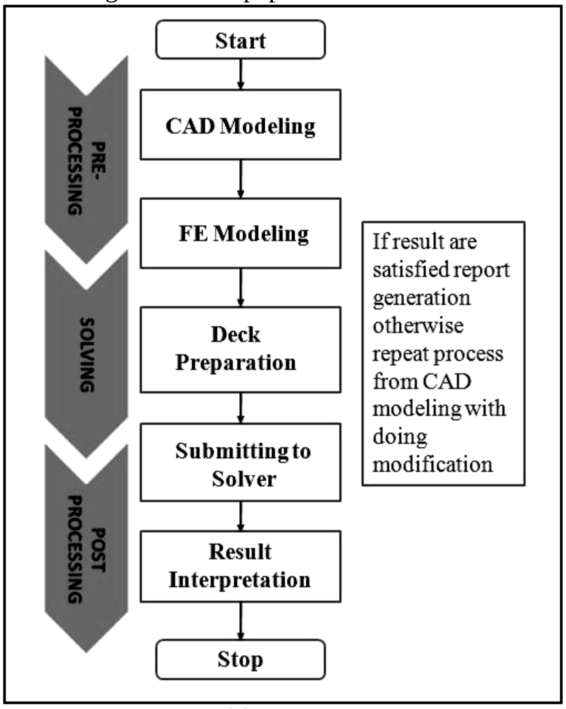

Fig1. Flow chart of finite element analysis steps

V. ELEMENT USED FOR MESHING

Finite element modeling mainly involves the descretization of the structure into elements or domains that are defined by nodes which describe the elements.

There are different types of elements for discretization. Different types of elements used for meshing are:

1. 0 D Element: Scalar element.
2. 1D Element: Rod, Bar, Beam element.
3. 2D Element: Shell, Membrane, Plane stress element, Plane strain element.
4. 3D Element: Solid element.

2D Shell Elements are mainly used for meshing of sheet metal components because of their applicability for thin structures. Element type selection is based on Geometry size and shape, type of analysis, time allotted for project and hardware configuration. In this we have used 2D meshing used with shell element.

Using shell elements for FE modeling of underbody of XX car model. Meshing point of view also, shell mesh results with a better quality mesh and shell elements also has 6 degree of freedom. Shell elements are of relevance when we speak about structural elements and in that two dimensions are much greater than the third one. Shell elements are planer elements. Shell elements are 4 noded and 8 noded elements. 4 noded elements used for plane or flat structure and 8 noded elements used for volume structure.

VI. QUALITY PARAMETER FOR MESHING

Acceptance criteria of model quality are considered when meets the body mesh model Quality check list concerning the various mesh quality parameters like skew, aspect ratio, Jacobian etc. are the measures of how far a given element deviates from ideal shape.

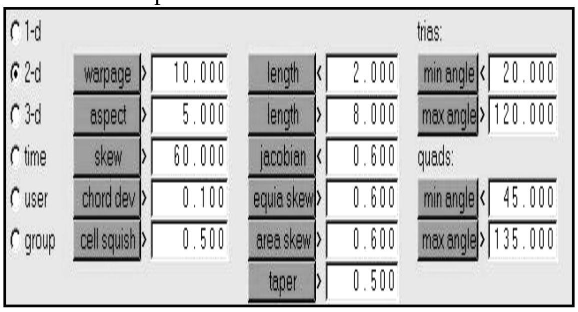

Fig2. Element check for 2D From Hypermesh.

Some of the qualities checks are based on angles (like skew, included angles) while others on side ratios & area (like aspect, stretch). Some of the terms used when checking element quality include:

6.1 Aspect Ratio: The aspect ratio of a rectangular shell element is defined as length over width. Many finite element programs have a restriction on the aspect ratio. For example,

$$1/20 < length/width < 20 \qquad \dots (2)$$

The reason for this restriction is that if the element stiffness in two directions is very different the structural stiffness matrix has both very large numbers and almost zero numbers on the main diagonal. The computed displacements and stresses may have little accuracy.

However, in modern software this is not a problem because high accuracy number representations are used. Sometimes, we can use aspect ratios of 1000 and this does not need to give accuracy problems. Element size used is 5. As this also applicable for crash analysis also. Range of element size is kept between 2 to 7 mm in this report.

6.2 Warpage: Warpage in two-dimensional elements is calculated by splitting a quad into two trias and finding the angle between the two planes which formed by the trias. The quad is then split again using the opposite corners and which forms the second set of trias. The angle between the two planes formed by the trias is then found. The maximum angle between the planes is the warpage of the element.

It is a measure of how to close a QUAD element is to being planer. A perfect planer element will have the warpage of zero. Warpage of up to fifteen degrees is generally acceptable for structural analysis.

6.3 Skew: The angles between the lines join opposite midsides. It Measure the angle created as square is turned into parallelogram or rhombus. Typical required values are to have less than 45 degree or 60 degree.

6.4 Jacobian: The ratio of the maximum determinant of the Jacobian to the minimum determinant of the Jacobian is calculated for each element in the current group in the active viewport. The Jacobian test of element shape can be used to identify elements with interior corner angles far from 90 degrees or high order elements which having midside nodes misplaced. The ratio equal to 1.0 is desired value for best quality. Value used is 0.6 for meshing used here for check.

6.5 Quad angle: The angle between two sides of a quad element should be 90 degree as much as possible. Typical required values are to have all angles between 45 degree and 135 degree.

6.6 Tria angle: The Angle between two sides of a tri element should be 60 degree as much as possible. Typical required values are to have all tria angles between 20 degree to 120 degree. Some time smaller angles are require to model geometry with small angle.[4]

VII. SPOT WELD FOR STRUCTURE

This model uses a solid element as nugget at the position of the spot weld. Its node points are not directly integrated in the shell mesh but are interpolated on close mesh points using RBE3 elements that interpolate the loads on the nodes using weight factors assigned to each node throughout the patch.

As the brick element does not have rotational degrees of freedom, the moments acting on these degrees of freedom of the shell elements are transferred as forces on the brick element. All spot welds were represented using a rigid element placed in the middle of the weld flange with an average spacing of 50 -60 mm.

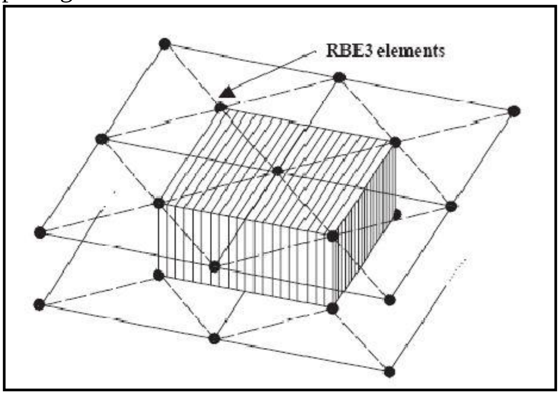

Fig3. ACM spot weld connecting shell elements with RBE3 element.

The name Area Contact Model 2 (ACM 2) stems from the characteristic that the connection is not realized through one single node point but by several shell mesh points, which are called the patch area.

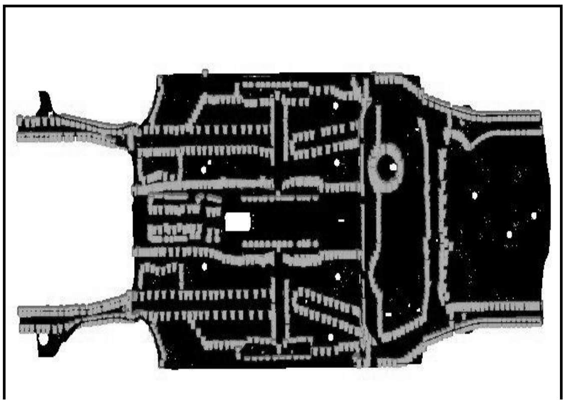

Fig4. Underbody with spot weld

VIII. Constraint position

Constraint points are used for fixing the structure of the car model. Then we can apply loads at loading points.

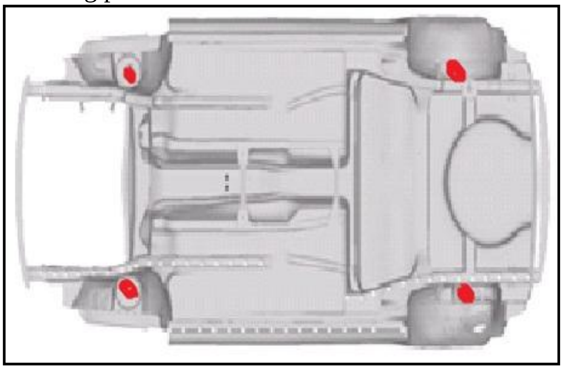

Fig5. Constraint points position.

For the static bending stiffness analysis the body-in-white is constrained at the upper rear axle mounts and the vertical translation of the upper front axle mounts are also completely blocked as shown in figure5 [7].

IX. Loading Points, material and properties

Different load cases are available. During a global bending test, forces are applied at the front seat locating points and the body is constrained at front and rear shock towers as shown in figure6.For this paper we are looking for loads acted by the seat as well as passengers. Load of side panel is also considered and that is applied on the side panel area of the underbody.

The static bending stiffness results from the ratio of the applied load to the maximum deflection along the rocker panel and tunnel beams. Load can be applied in mass or force application at loading points.

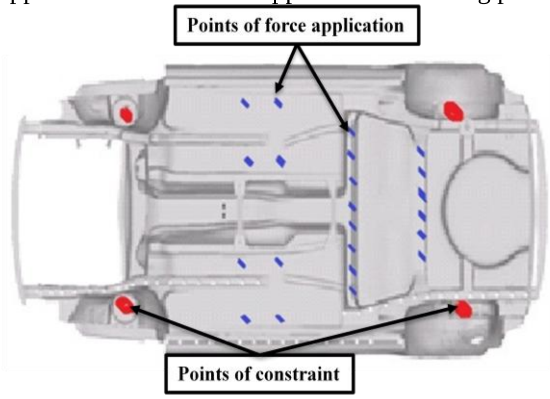

Fig6. Points of force application.

Assigning material to various parts with values of Young modulus, density, Poisson's ratio in SI unit. Normally young modulus for underbody parts come in same range but there yield strength limit is different and which can be used for comparing von mises stress in post processing.

Assigning the properties means assigning thickness to the various parts. Thickness of the underbody parts comes in various thickness range like 0.80 to 3.00 mm.

After this pre-processing step next step is solving. That is now our model is ready for analysis.

X. Conclusion

This paper describes use of finite element method. Mainly pre-processing step is explained here for doing static analysis of underbody of the BIW of car model. Application of this paper when somebody want to do static analysis only of the underbody of car then that person can follow the step of preprocessing explained here. In this paper we have gone through pre-processor phase, where along with pre processing the geometry of the structure, the constraints, loads and mechanical properties of the structure are defined.

Advantages of this paper is of element used for meshing, spot weld type used, constraint point position, load application points to be considered all are explained so that will easy to do analysis. In this paper work following software like Hypermesh for meshing, Nastran as a solver and Hyperview for visualized result are used.

XI. ACKNOWLEDGEMENTS

I would like to thanks to my Institute guide Prof. D. G. Thombre for his support and his suggestion about project and techniques to be use for dissertation work.

REFERENCES

Journal Papers:

[1] Lyu N. and Saitou K., *Decomposition-based assembly synthesis of a 3D body-in-white model for structural stiffness, IMECE2003-43130, ASME International Mechanical Engineering Congress, Washington, D.C., November 15–21, 2000.*

Books:

[2] J. Powloski, *Vehicle body Engineering, Business book limited, 1970.*

[3] M. J. Fagen, *Finite Element Analysis Theory and Practice, Longman Scientific and Technology, 1992.*

[4] Nitin S Gokhale, Sanjay Deshpande, Sanjeeve V. Badekar, Anand N. Thithe, *practical finite element analysis* 1st Edition, ISBN978-81-906195-0-9.

[5] Users Reference Manual for the Hypermesh.

[6] Users Reference Manual for the NASTRAN General Purpose Finite Element Structural Analysis Computer Program (Nov 2011).

Proceedings Papers:

[7] J. Helsen ,L. Cremers ,P. Mas , P. Sas, *Global static and dynamic car body stiffness based on a single experimental modal analysis test proceedings of ISMA2010 including USD2010.*