Design, Modeling and Analysis of VSA Bracket

Er. N. Balasubramanyam[1] Prof. Smt. G. Prasanthi[2] Er. P.Gnana Prakash[3]
[1,2,3]Assistant Professor
[1,2,3]Department of Mechanical Engineering
[1,3]Sreenivasa Institute of Technology and Management Studies, Chittoor Andhra Pradesh India [2]JNTUA College of Engineering (Autonomous) Anantapuramu-515002, Andhra Pradesh, India

Abstract— VSA modulator bracket is used to hold the modulator, which gives stability for the vehicle. The aim of the project is to design a low cost sheet metal and weight optimized bracket from the existing casting bracket. The functionality of this bracket is to withstand the external loads and to reduce the transfer stiffness for frontal crash. Design and optimize a low cost stamped steel VSA bracket meeting all the criteria for assembly, packaging, manufacturing, Structural behavior / functional performance being optimized for weight and cost. This project work involves the modeling of using CATIA; the preprocessing is done in HYPERMESH and analysis of using different materials in ANSYS software's.

Key words: VSA, Casting bracket, CATIA

I. INTRODUCTION

VSA is designed to assist the driver in maintaining control during cornering, acceleration and sudden man oeuvres by applying brake force to the right or left front wheel as necessary and managing the throttle and ignition systems. At its simplest level, VSA has a traction control function whereby it detects wheel slip under acceleration and coordinates the use of braking and retardation of the ignition to regain traction. In situations where the driven wheels are on surfaces with different levels of traction - such as a partially wet road surface - the system applies braking action to the wheel that's slipping, allowing the tyre with better grip to move the vehicle. In addition, the system reduces engine output to minimize wheel spin. By monitoring input from a series of vehicle sensors, VSA calculates a predicted range of vehicle response while constantly monitoring the vehicle's actual response and the driver's control inputs. If the actual response is outside the predicted response range, as when cornering forces exceed the tyres' performance, VSA automatically intervenes with corrective action. In the case of over steer (which may lead to a spin) VSA applies braking to the outside front wheel to counter the yawing tendency. On the other hand, if under steer is detected, VSA applies braking to the inside rear wheel and reduces engine power to turn the car back on to the intended course. Unlike some vehicle stability systems - which can seem intrusive to enthusiast drivers - the Accord's VSA is calibrated to add to stability and predictability, without stifling the driving enjoyment of the carefully engineered chassis. An indicator light flashes on the instrument panel while the system is actively enhancing the stability of the vehicle. A cockpit switch is provided to disable the VSA.

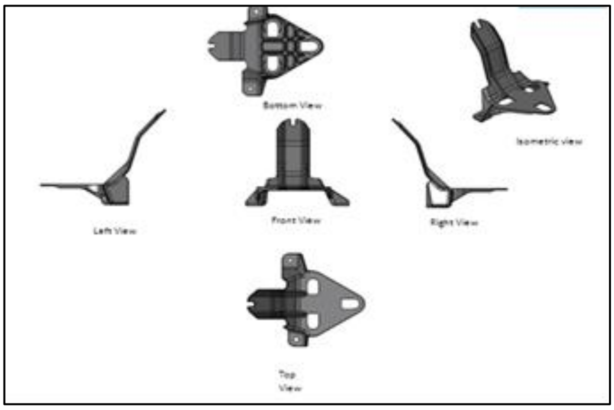


Fig 1: VSA Bracket Views

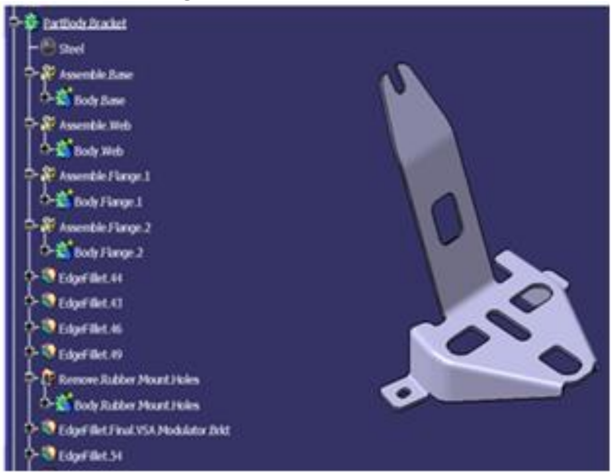


Fig. 2: Shows the Modeling Design in CATIA software.

II. DECISION TREE DIAGRAM

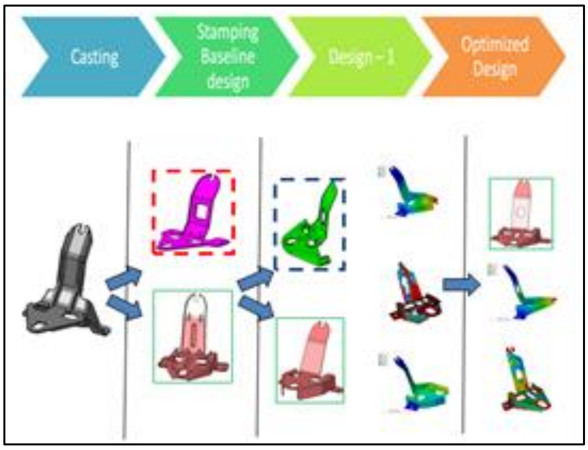
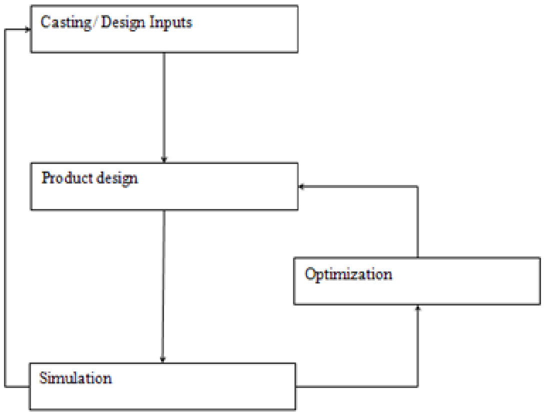


Fig 3: Shows the tree diagram for stages of a VSA Bracket

- Material Aluminum
- Material: Steel
- Thickness Alternatives considered (1.31 / 1.6 / 2.0)
- Optimized design

A. Approach



Product design: It is concerned with the efficient and effective generation and development of ideas through a process that leads to new products

Optimization methodologies: It can be applied during the product development stage to ensure that the finished design will have the high performance, high reliability, low weight, and/or low cost. Alternatively, optimization methods can be applied to existing products to identify potential design improvements.

Simulation: Simulation or computer aided engineering (CAE) using finite element analysis (FEA) is a method of virtually proving a product or design.

III. Introduction to FEA

Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It can be used to analyze either small or large-scale deflection under loading or applied displacement. It can analyze elastic deformation, or "permanently bent out of shape" plastic deformation. The computer is required because of the astronomical number of calculations needed to analyze a large structure. The power and low cost of modern computers has made Finite Element Analysis available to many disciplines and companies. In the finite element method, a structure is broken down into many small simple blocks or elements. The behavior of an individual element can be described with a relatively simple set of equations. Just as the set of elements would be joined together to build the whole structure, the equations describing the behaviors of the individual elements are joined into an extremely large set of equations that describe the behavior of the whole structure. The computer can solve this large set of simultaneous equations. From the solution, the computer extracts the behavior of the individual elements. From this, it can get the stress and deflection of all the parts of the structure. The stresses will be compared to allowed values of stress for the materials to be used, to see if the structure is strong enough. The term "finite element" distinguishes the technique from the use of infinitesimal "differential elements" used in calculus, differential equations, and partial differential equations. The method is also distinguished from finite difference equations, for which although the steps into which space is divided are finite in size, there is little freedom in the shapes that the discreet steps can take. Finite element analysis is a way to deal with structures that are more complex than can be dealt with analytically using partial differential equations.

IV. Requirement of Finite Element Analysis

Finite Element Analysis makes it possible to evaluate a detailed and complex structure, in a computer, during the planning of the structure. The demonstration in the computer of the adequate strength of the structure and the possibility of improving the design during planning can justify the cost of this analysis work. FEA has also been known to increase the rating of structures that were significantly overdesigned and built many decades ago. In the absence of Finite Element Analysis (or other numerical analysis), development of structures must be based on hand calculations only. For complex structures, the simplifying assumptions required to make any calculations possible can lead to a conservative and heavy design. A considerable factor of ignorance can remain as to whether the structure will be adequate for all design loads. Significant changes in designs involve risk. Designs will require prototypes to be built and field tested. The field tests may involve expensive strain gauging to evaluate strength and deformation. With Finite Element Analysis, the weight of a design can be minimized, and there can be a reduction in the number of prototypes built. Field testing will be used to establish loading on structures, which can be used to do future design improvements via Finite Element Analysis.

V. Areas Where FEA is Applicable

The finite element method is a very important tool for those involved in engineering design, it is now used routinely to solve problems in the following areas:

- Structural strength design
- Structural interation with fluid flows
- Analysis of Shock (underwater & in materials)
- Acoustics
- Thermal analysis
- Vibrations
- Crash simulations
- Fluid flows
- Electrical analyses
- Mass diffusion
- Buckling problems
- Dynamic analyses
- Electromagnetic evaluations
- Metal forming
- Coupled analyses

Nowadays, even the most simple of products rely on the finite element analysis for design evaluation. This is because contemporary design problems usually cannot be solved as accurately & cheaply using any other method that is currently available. Physical testing was the norm in years gone by, but now it is simply too expensive.

VI. Difference Between FEM & FEA

The terms 'finite element method' & 'finite element analysis' seem to be used inter changeably in most documentation, so the question arises is there a difference between FEM & FEA The answer is yes, there is a difference, albeit a subtle one that is not really important enough to loose sleep over. The finite element method is a mathematical method for solving ordinary & elliptic partial differential equations via a piecewise polynomial interpolation scheme. Put simply, FEM evaluates a differential equation curve by using a number of polynomial curves to follow the shape of the underlying & more complex differential equation curve. Each polynomial in the solution can be represented by a number of points and so FEM evaluates the solution at the points only. A linear polynomial requires 2 points, while a quadratic requires 3. The points are known as node points or nodes. There are essentially three mathematical ways that FEM can evaluate the values at the nodes, there is the non-variation method (Ritz), the residual method (Galerkin) & the variation method (Rayleigh-Ritz). FEA is an implementation of FEM to solve a certain type of problem. For example if we were intending to solve a 2D stress problem. For the FEM mathematical solution, we would probably use the minimum potential energy principle, which is a variation solution. As part of this, we need to generate a suitable element for our analysis. We may choose a plane stress, plane strain or an axisymmetric type formulation, with linear or higher order polynomials. Using a piecewise polynomial solution to solve the underlying differential equation is FEM, while applying the specifics of element formulation is FEA, e.g. a plane strain triangular quadratic element.

A. *Baseline Design*

1) Casting to stamped (sheet metal) component
2) Side flange removed based on manufacturing limitations
3) Used beads and stiffeners as per the casting product in order to increase strength
4) Maintained mountings and hole patterns same as base product
5) Maintained packaging clearances as per the input data

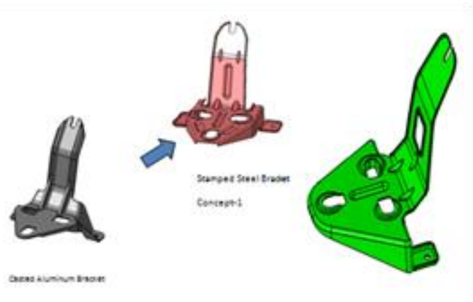


Fig. 4: The Concepts between 1 and 2

B. *Comparison between Concept-1 and Concept-2*

1) Manufacturing criticality
2) Increasing manufacturing cost
3) More material required compare to concept-1
4) The direction of leg is same as the direction force acting on it

1) *First iteration concept-1*

1) Reduced bead to fix manufacturing feasibility
2) Removed flanges to reduce cost
3) Removed flanges for base element to fix manufacturing feasibility

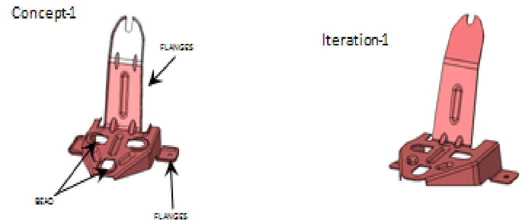


Fig. 5: Shows the Concept of Design 1

2) *Second iteration concept-1*

1) Optimized the product based on the analytical report
2) Maintained packaging clearances as per the in put data
3) Covered all the manufacturing feasibilities
4) Maintained mountings and hole patterns same as per the input

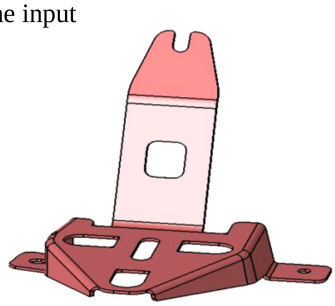


Fig 6: Shows the Final Component

VII. FE MODEL AND ANALYSIS

1) Shell Mesh at the Mid plane of Stamping
2) Element Quality criteria as reference
3) Static Analysis with 4.4G Load in h-dir with Mass of Modulator at CG of Modulator weighing 2.6kg
4) Non-Parametric topology Optimization with stamping constraints and Weight reduction objective
5) Transverse stiffness evaluation by applying a unit load in t-direction at the CG of Modulator
6) Free-Free analysis to understand the dynamic stiffness in the transverse bending modes

A. *Material Properties of Aluminium:*

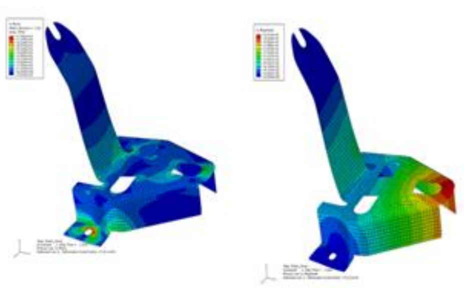
Density = 2800 kg/m3
Young's Modulus (EX) = 130 Gpa
Poisson's Ratio (PRXY) = 0.5

B. *Material Properties of Steel:*

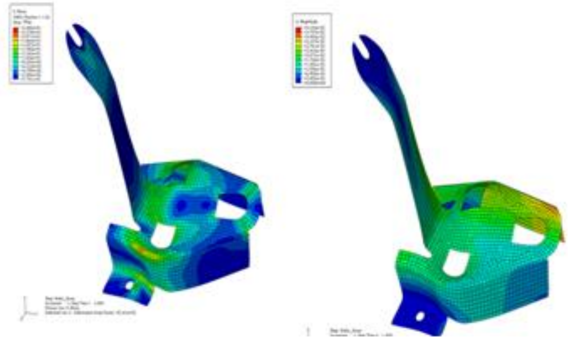
Young's Modulus (EX) = 210 Gpa
Poisson's Ratio (PRXY) = 0.30,
Density = 8000 kg/m3

VIII. STATIC ANALYSIS VSA MODULATOR BRACKET

A. *Base Model 1: Thickness = 2mm*

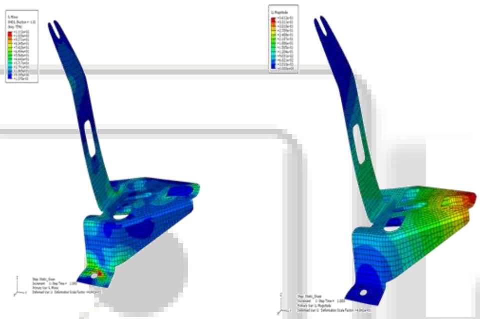


B. Base Model 1: Thickness = 2mm



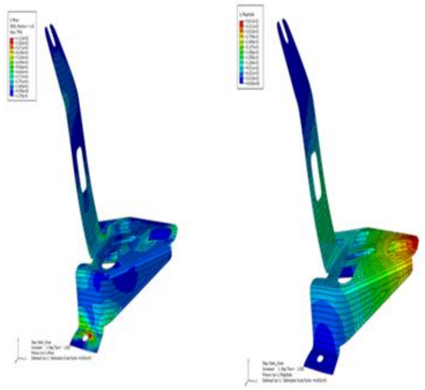
1) Von-Mises Stress contour
Von-Mises Stress contour
4.5G load in h-direction (Z – Axis)
Max Stress = 79.62 MPa
2) Displacement Contour
100 N Loading in t-direction (X – axis)
100 N Loading in t-direction (X – axis)
Max Displacement = 0.04 mm
Max Displacement = 0.04 mm

C. Redesigned Model 2: Thickness = 2mm



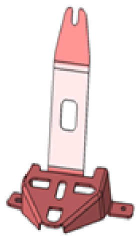
1) Von-Mises Stress contour
4.5G loading in h-direction (Z – Axis)
Max Stress 112.2 MPa
2) Displacement Contour
4.5G Loading in h-direction (Z – axis)
Max Displacement = 0.36 mm

D. Redesigned Model 2: Thickness = 2mm

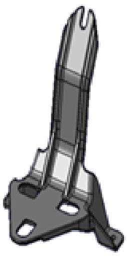


1) Von-Mises Stress contour
100 N Loading in t-direction (X – Axis)
Max Stress 248 MPa
2) Displacement Contour
100 N Loading in t-direction (X – axis)
Max Displacement = 0.04 mm

Should costing for Sheet Metal Part	
Tooling Cost	$ 25,000.00
Components produced	1000000
Amortized cost of tooling	$ 0.03
Component weight	0.3878Kgs
Material rate	$ 1.00per Kg
Material cost	$ 0.39
Total component cost	**$ 0.41**



Should costing for Aluminum casting	
Tooling Cost	$ 15,000.00
Components produced	1000000
Amortized cost of tooling	$ 0.02
Component weight	0.2674Kgs
Material rate	$ 7.00 per Kg
Material cost	$ 1.87
Total component cost	**$ 1.89**



Description	**Baseline Stamping Design**	**Re-designed model**
Sheet Thickness	**2**	**2**
Max Von Mises Stress (Mpa)	**79.62**	**112.2**
Max. Deflection(mm)	**0.5174**	**0.36**

Table 1: Shows the values of defection and stress

IX. CONCLUSION

We suggest Cold rolled 2mm thick for this stamped component. Based on criteria below

- Nesting options
- Life / Durability
- Based on part design and shape
- Based on draw-ability and formability
- Based on surface finish and dimensional accuracy

All the other materials have higher strength and provide larger margin of safety but to since there is a thickness constraint we do not suggest the same.

REFERENCES

[1] British Standards Institution (1987). Measurement and evaluation of human exposure to whole-body mechanical vibration and repeated shock, BS 684.
[2] Griffin, M.J. (1996). Handbook of Human Vibration. (1st Ed.), Elsevier Academic Press, ISBN: 0-12-303041-2.
[3] International Organization for Standardization (1997). Mechanical vibration
[4] Bazrgari, B.; Shirazi-Adl, A.; and Kasra, M. (2008). Seated whole body vibrations with high-magnitude accelerations-relative roles of inertia and
[5] Stein, G.J.; Zahoransky, R.; Gunston, T.P.; Burstrom, L.; and Meyer, L. (2008). Modelling and simulation of a fore-and-aft driver's seat suspension system with road excitation. International Journal of Industrial Ergonomics, 38(5-6), 396-409.
[6] Lee, T.K.; and Kim, B.S. (2008). Vibration analysis of automobile tire due to bump impact. Applied Acoustics, 69(6), 473-478