EasyChair Preprint

№ 5766

A Experiment & FEA of Cross Sectional Area of Automotive Center Console Mounting Bracket

Sudarshan Bhalshankar

EasyChair preprints are intended for rapid dissemination of research results and are integrated with the rest of EasyChair.

June 11, 2021

A Experiment & FEA Of Cross Sectional Area Of Automotive Center Console Mounting Bracket .

Sudarshan Dilip Bhalshankar.
M.E[CAD/CAM]
PradnyaSurya Engineering Works,Pvt.Ltd. Solapur, Maharashtra, India.
sudarshanbhalshankar111@gmail.com

Abstract

Floor consoles or Center consoles are an indispensable pan of Automotive Cockpit systems in modem passenger vehicles. It occupies space between the front scats in the car and has a lot of utilities and functionalities. The center console design can be very simple as just providing an enclosure for the gear shifter and parking brake and as complex as having storage bins with armrest which can slide. During the genuine working condition, the most extreme load is moved from lower side to upper side which makes plausibility of failure in the ***floor console/ center console,*** *Henceforth it is fundamental to concentrate on the stress (****Bending & Shear****) investigation of* ***Automotive Floor Console Mounting Bracket*** *to improve and alter the current design. In present research design of automotive center console is done in* ***CATIA software. ANSYS19 software*** *was also used for analyze the structural strength(Bending & Shear) and* ***optimize*** *the parts weight along with modal analysis to determine* ***cross sectional area*** *of the bending* ***cantilever beam*** *of* ***I section*** *with mode shape and validate the results with* ***UTM Machine*** *tested. The comparison between existing design and modified design ,Material are used in modified design is* ***Plastic & Aluminum****. Also compared the bending performance of Cantilever Beam. The target of the new design was recommended the* ***"I – Section" for cantilever beam****, Attached to the* ***Plastic Center Console To The Aluminum Underbody****. The new design was 12% weight reduction from the existing part (Plastic Center Console To The Steel Underbody) fabricated using steel material.* ***Topology optimization*** *strategy is applied to optimize the material on its Steel Floor Console mounting bracket. In the optimization of housing bracket,* ***finite element analysis*** *software was used for calculating the stress and fatigue life and then the weight was optimized. Experimental validation will be perform on UTM machine. Results and conclusion will be drawn by comparing analytical and experimental values. Testing and validation of new design using UTM machine is done and it is found that average percentage error in experimental and theoretical analysis of Steel and Aluminum material is around 48 % .*

Keywords - I- Section , Cantilever Beam , Floor Console , FEA , UTM machine , Bending Stress.

I. Introduction.

The center console in an Automobile refers to the control-bearing surfaces in the center of the front of the vehicle interior, the area beginning in the dashboard and continuing beneath it. and often merging with the transmission tunnel which runs between the front driver's and passenger's scats of many vehicles. Increasingly. center consoles include a wide variety of storage compartments, cup holders, ash trays and some of them with a refrigerator. Figure I shows a typical automobile center console assembly. Generally, plastic materials are used to manufacture the center console frames (console frame). Designing a console frame is always a

challenging task, because with-in a design package space it should be robust enough to with-stand the various design and abusive loads. Optimization is a procedure used to make the effective design and reduce the material where ever it is not required, resulting material savings.

The modelling of the existing bracket is done in CatiaV5 where meshing is done in Hypermesh and Abacus solver is used for analysis. The results of existing bracket obtained with high stress and displacement. So, there is necessity to design the new bracket which can withstand for high stress and displacement. The modelling of the modified bracket is done in CatiaV5 while the meshing is done in Hypermesh and analysis is done in Abacus. Comparing the results of existing bracket and modified bracket, best is selected for the desired application.

II. Literature Review.

The main objective of the project is value engineering and value analysis of the part in the air suspension system without affecting the performance and the quality of the product. The existing model of rear air spring bracket is higher in cost and weight. So many considerations are kept in mind cited value engineering as an effective tool in refining the existing design of product towards the goal of design of new product with the stress analysis and resultant displacement. Value engineering concerned with new products and applied during product development and it does not influence the project costs and quality, but proved to have positive impacts on the weight reduction. It was found that a project can significantly save a cost and improve performance of project functioning by using the value engineering process at right time. This project summarizes the benefits and effectiveness of the value engineering methodology.[1] A mounting bracket for an automotive console of an automotive vehicle having at least one sidewall which extends upwardly from the floor of a vehicle. The mounting bracket includes a base adapted for connection to the floor of the vehicle. A central leg extends upwardly from the base and has a top attached to its upper end. The top includes a portion which lies in substantially horizontal plane and is adapted for connection to the console. In addition, a side flange is attached to and extends outwardly from the top and abuts against the console sidewall to provide additional lateral support. The mounting bracket is of a one-piece construction.[2] Disclosed herein is a hinge unit of a console armrest for a vehicle. Which includes an upper bracket including an arm rest installation part coupled to an armrest. And upper hinge parts armed at both sides of the armrest installation part , each having a first upper hinge hole, a lower bracket including a console installation part coupled to a console. And lower hinge parts formed at both sides of the console installation part, each having a first lower hinge hole, an upper bracket cover configured to surround at least a portion of each of the upper hinge parts and to have a second upper hinge hole formed at a position corresponding to the first upper hinge hole, and a lower bracket cover configured to surround at least a portion of each of the lower hinge parts and to have a second lower hinge hole.[3] A console assembly for use in a passenger seating area of a vehicle includes a console housing. Having a storage area defined therein. An armrest is pivotally connected to the con sole housing and is movable between a first position extend hinge at least partially over a top Surface of the. Console housing and at least one second position. The armrest includes a closeout section, a Receiving portion operatively connected to the closeout section and a console lid sizably connected to the receiving portion. A console accessory tray is adjustably positioned adjacent the console housing between a stored position and a deployed position extending rearward of the console housing. The console accessory tray includes a dis play Surface accessible to occupants in a rear portion of the passenger seating area.[4] Console assembly for a vehicle including a video enter attainment system includes a console blousing having a storage compartment and a

storage compartment closeout section pivotally connected to the console housing position able between a raised position and a lowered position. An armrest is pivotally connected to the closeout section and is pose tunable between a closed position disposed adjacent the closeout section and an open position allowing access to a. Video monitor and monitor mount stored below. A video monitor mount pivotally connects the video monitor to the console assembly and allows for selective adjustment of the video monitor so that the screen is visible to rear seat occupants in the vehicle.[5] A Bracket is a component used to support load or any attachments to structural clement for heavy vehicle. It is subjected to different types of loading while the vehicle is in motion. One of the main common loading is static load which is assumed to be constant throughout the operation. This static load is main causes for the failure of bracket. To reduce the failures stress concentration needs to be reduced at vicinity of the joint hole of the bracket, as the hole is highly stressed region. The objective of the research work is study the static behavior of the truck cross member bracket. Failure analysis and reduction of weight by changing the geometrical features and structural properties. The failure analysis of the cross member bracket is effected by design and analysis approach. The failure analysis has been carried out by using standard FE tools. In this investigation FEA model has been generated for truck cross member with the specified quality criteria and analyzed for the optimized results. The results show the reduction in stress values and leads to sate design. The final geometry of bracket weight is reduced to 61%. [6]

III. Problem Statement

Optimization of weight has been very critical aspects of any design. It has substantial impact on vehicle performance, and in spin minimizes the emissions. This dissertation would focal point on the design gap offered by the element practical even as crucial the scenery and scope of the weight optimization more the areas acknowledged during design optimization.

IV. Need

1. Emission norms of the engines and vehicles.
2. Fuel efficiency
3. Increased Starter Durability or Fatigue life.
4. Increase of material cost.
5. Market demand on compact size and lightweight.

V. Objective

1. Modelling Steel Floor Console mounting bracket in CATIA V5 software.
2. Analyzing for stresses and deformation in Steel Floor Console mounting bracket of vehicle
3. Topological optimization for the model.
4. To perform experimental testing of new optimize Steel Floor Console mounting bracket on UTM.
5. Experimental testing and correlating results.

VI. Methodology

- Review of existing case.

- Secure 3D CAD form of in Steel Floor Console mounting Bracket.
- Use Pre-processor for Meshing.
- Solve problem by using suitable FEA Solver for Structural analysis.
- Solve problem by using appropriate FEA for structural.
- Validation by experimentation for benchmark (existing) & case study.
- Validation by comparing FEA results with results recorded during physical Experimentation.

VII. Finite Element Analysis

Finite element method is the process of solving the complex and simple experimental methodology by converting them into simulation. Hypermesh is the product of Altair Engineering is the preprocessor which is widely used for finite element modeling. FEA consists of a computer model of a material which is stressed and analyzed for specific results. It is not only used in new product design but also existing product refinement. It was decided to do 2D mesh for the CAD model. The modelling is done in CatiaV5 and Hypermesh software is used for meshing and analysis.

Parameters	Value
Yield Strength	220 MPa.
Young's Modulus	$2.1*10^{5}$ MPa.
Possion's Ratio	0.3
Density	$7.8*10^{-9}$ tonnes/mm^3
Element Size	5
Mesh Type	Mixed
No.of Nodes	1080
No.of Element	1016
Weight of the Centre Console	573 gram

Table-1. Material Properties

Parameters	Value
Beam Section (Cross Sectional) Area	I – section
Types of Beam	Cantilever Beam
Permissible Bending Stress	140 N/mm^2
Max. U.D.L	3.217 kN/m
Length of Span	4 m
Shear Force	4 kN
Max. Shear Stress	3.15 N/mm^2
Moment of Inertia	$15.69 * 10^{6}$ mm^4

Table -2. Boundary condition.

Parameters	Value
Flange	100*10 mm (Equal)
Web	150*10 mm

Table – 3. I – Section

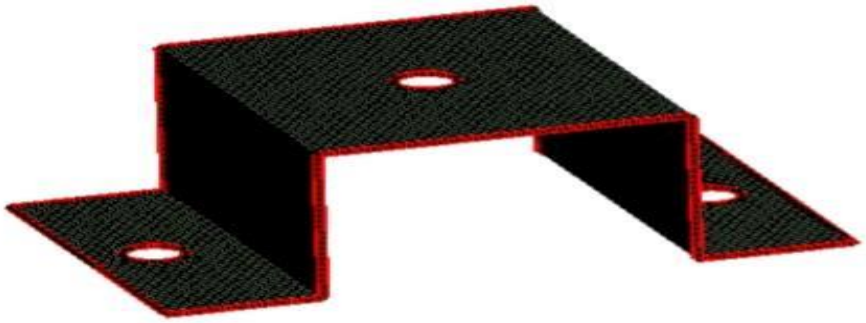


Fig. 1 Meshing of baseline center console for steel.

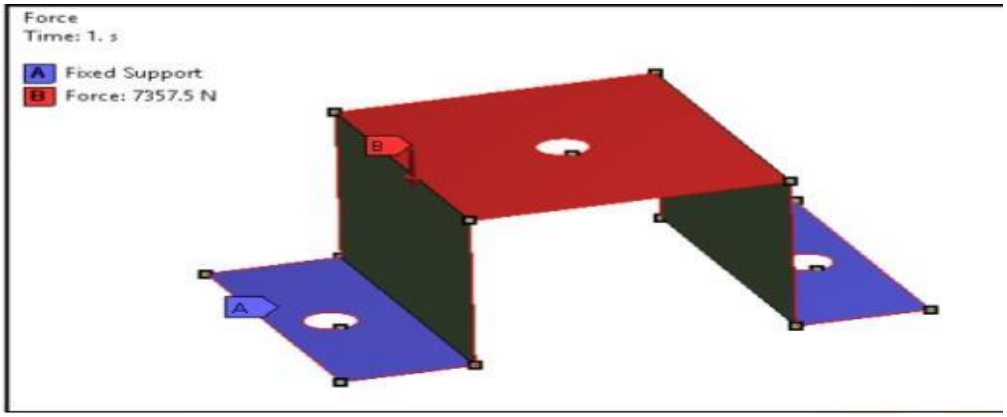


Fig. 2 Fixed boundary condition applied on the bottom side of Center Console and loading.

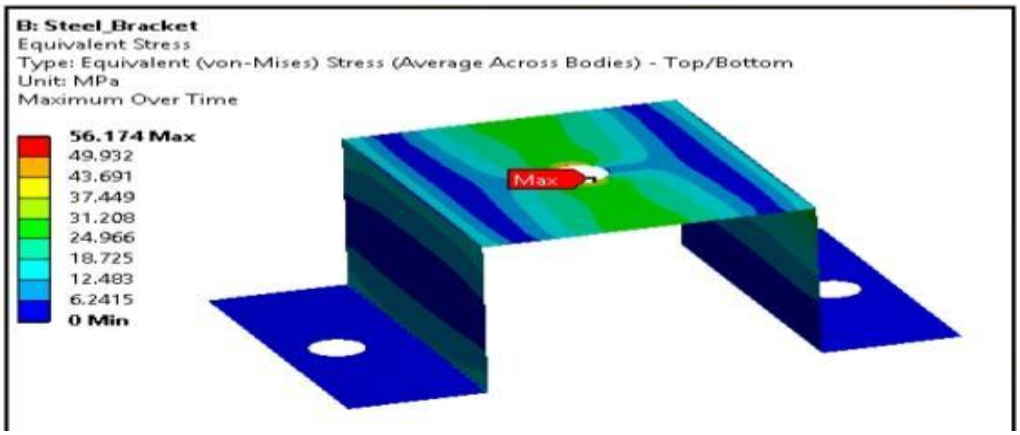


Fig. 3 Equivalent Stress plot for Steel Centre Console.

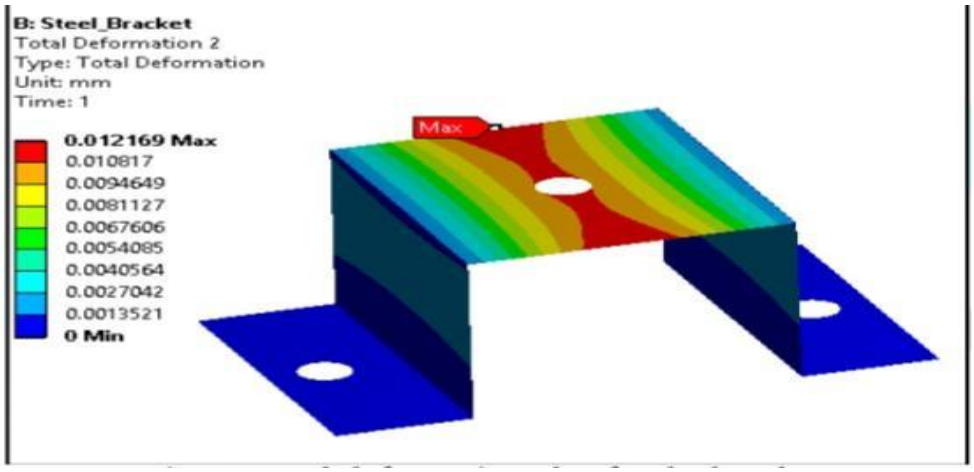


Fig. 4 Total deformation plot for the Centre Console Steel.

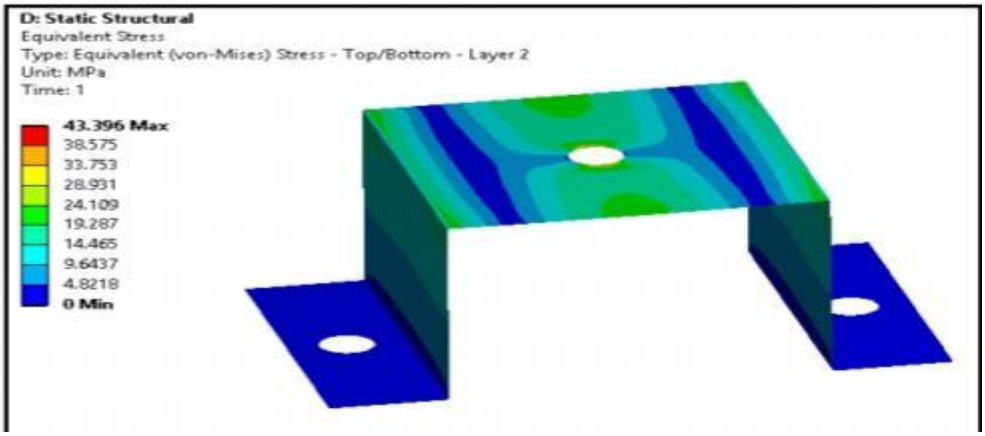


Fig. 5 Maximum Equivalent Stress plot at outermost Center Console for Aluminum.

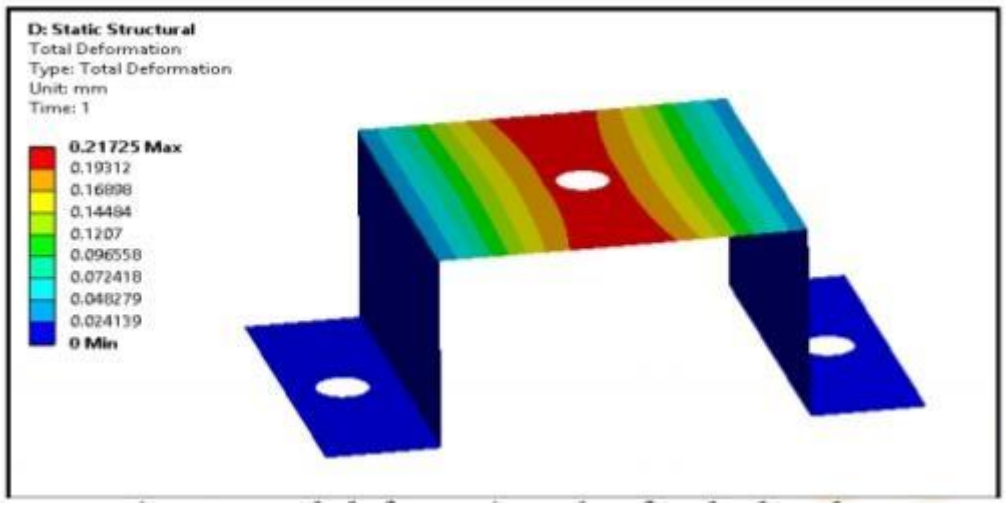
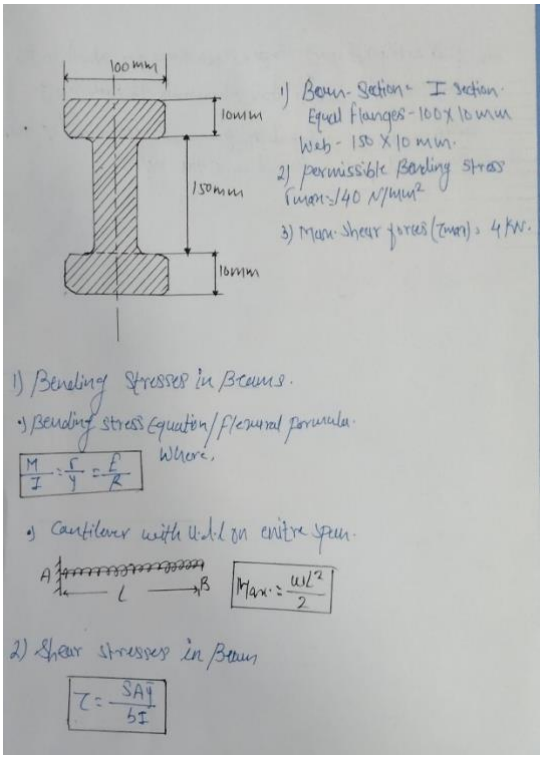
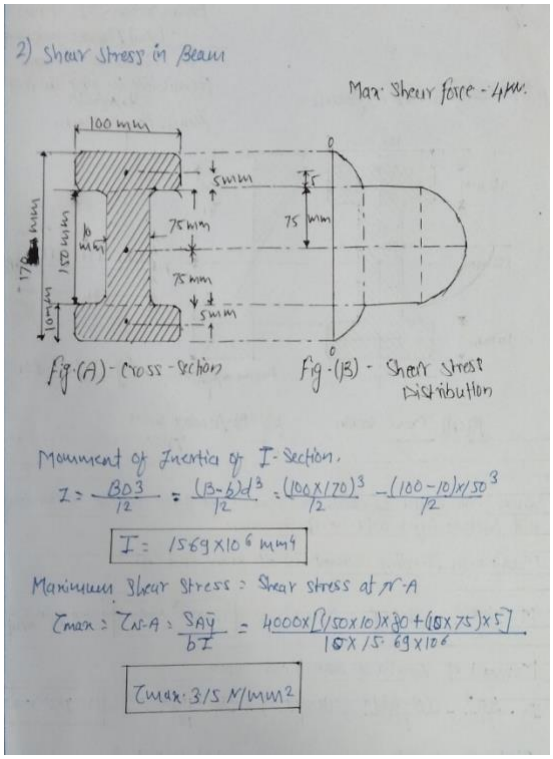


Fig. 6 Total deformation plot for the Centre Console For Aluminum .





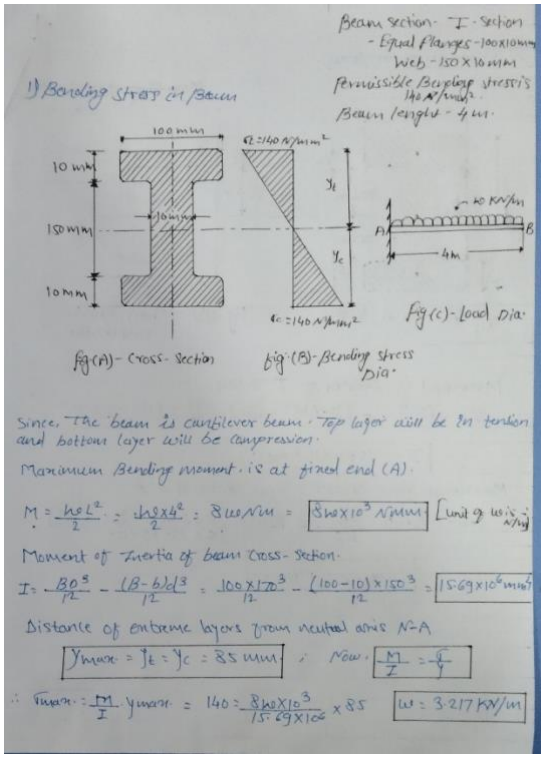


Fig. 7 Theoretical Analysis Of Center Console c/s. For Aluminum.

VIII. Experimental Analysis

The experimentation is done on the Universal testing Machine. A universal testing machine is a machine that is used to perform standard tensile and compressive tests on Materials, components and structures. It works on the Principal of elongation and deformation. The existing bracket is placed in between two plates, there is no need of special fixture for the bracket mounting. The load is applied on the bracket gradually by the speed of 2mm/min. By applying constantly load on existing bracket up to required loading condition i.e. 250 N. The displacement. Obtained in the bracket is shown in the display panel. In this Case for 250 N load we get the displacement of 6.820 mm. We increased the load up to 330 N and the displacement. Obtained is 8.840 mm. The graph of load versus displacement is displayed in the computer screen. Finally we observed that, load applied 250 N and found no cracks or deformation observed. Load applied to sample to 330 N and found free from deformation with no cracks.

The Fig.8 graph shows load versus displacement values such that at 330 N load the displacement obtained is 8.840 mm.

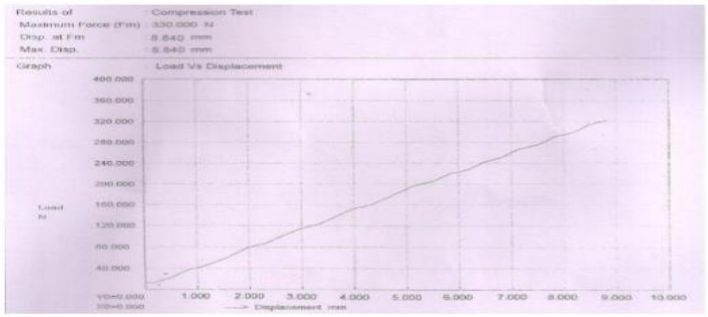


Fig. 8 Graph of load Vs Displacement of Existing Center Console For Steel .

The Fig. 9 graph shows load versus displacement values such that at 283 N load the displacement obtained is 2.210 mm.

Fig. 9 Graph of load Vs Displacement of Modified Center Console For Aluminum .

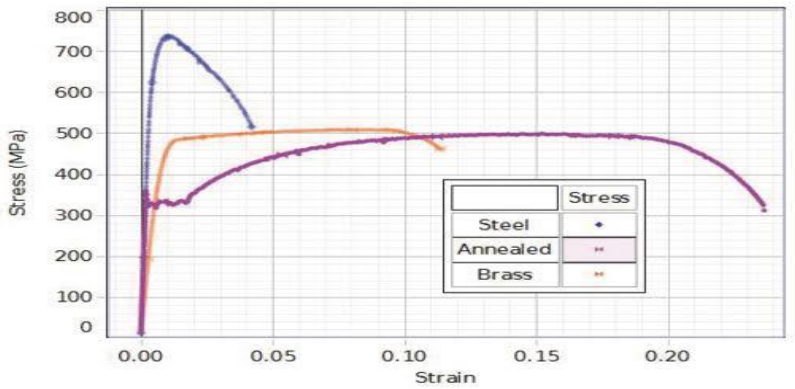


Fig. 10 Graph of material (Plastic , Steel , Aluminum)

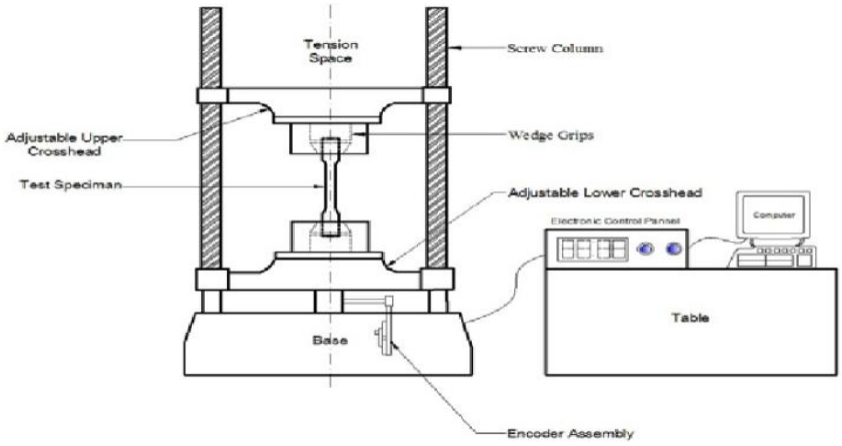


Fig. 11 Block Diagram Of UTM Machine.

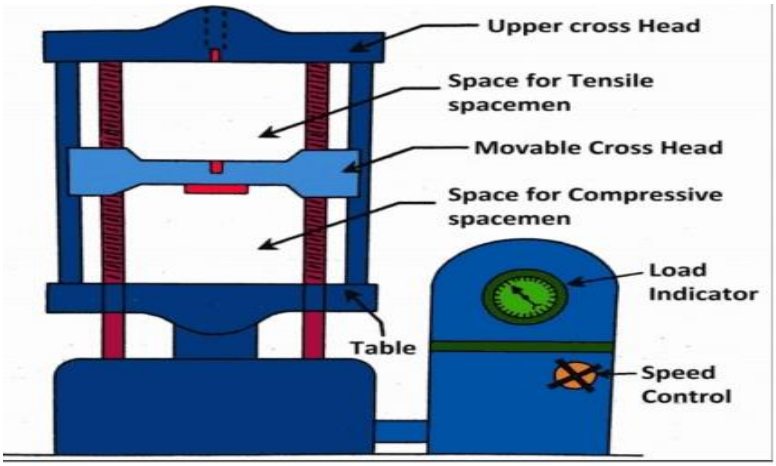


Fig. 12 Working Principle Of UTM Machine.

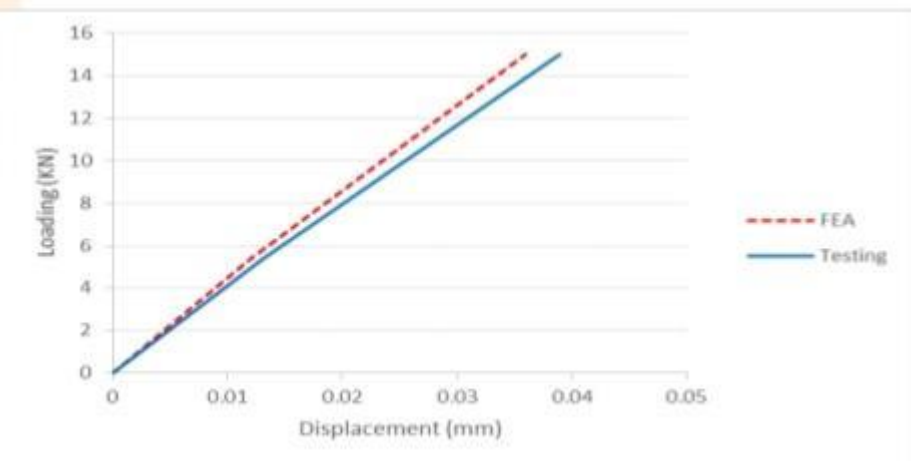


Fig. 13 Load vs. Displacement plot for Steel Centre Console .

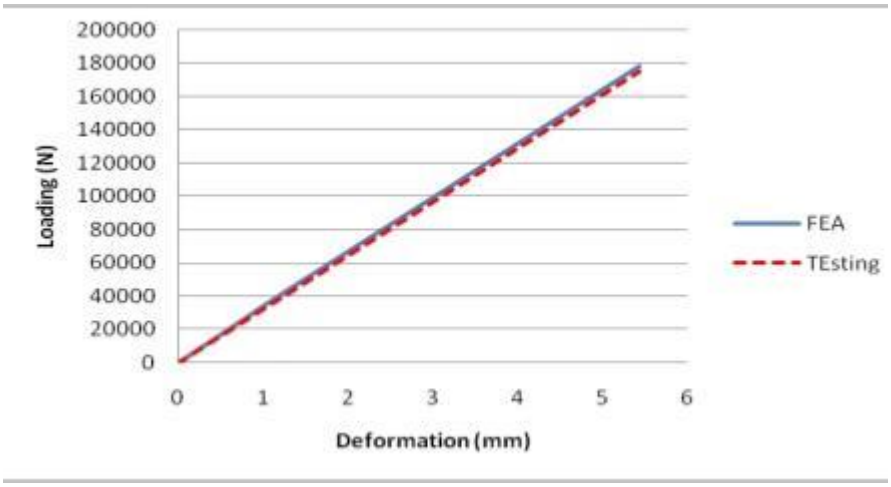


Fig. 14 Load vs. Displacement plot for Aluminum Center Console .

IX. Summary

With the trend of increased adoption of consumer electronics in the automotive industry. The design of the center console is undergoing a major shift. This article covers some of the emerging trends which are tending increasing adoption in the center console. These technologies not only provide a seamless human machine interface for when the passenger uses a cell phone or the car navigation unit. But also enable automobile manufacturers to save money. Improve reliability, and create a shift in the aesthetics of the car. With greater emphasis being placed on the user experience and an increased need for design differentiation, automotive human machine interface (HMI) has gone through a paradigm shift and is taking the user experience to a whole new level. The center console. Located in the center of the front interior of an automobile. Has a display and control switches for Infotainment. HVAC, communication modules. And other functions. It is an essential component of an automobile interior as it consolidates access to these systems and provides the user with all the relevant information required for an enhanced and controlled driving experience.

Traditionally center consoles have been a stack-up of Infotainment and HVAC modules with, Mechanical control switches on relatively flat plastic overlay. These modules work independently of each other. Have different design formats, and utilize separate communication channels. To implement this architecture. There are dedicated companies supporting car manufacturers for one or more of these modules. Consumers also have the option of buying these modules separately from aftermarket distributors.

X. Result And Discussion

In the static analysis generally two parameters are considered i.e. stress and displacement. The table shows the stress and displacement of existing bracket and modified bracket following. From this testing it is observed that load applied to 250 N And found no crack and deformation.

By comparing the results of finite element analysis and Experimental analysis, the displacement of existing bracket Is 13.26mm and 6.82 mm respectively which close to each Other while the displacement of modified bracket is 4.928 Mm and 2 mm which close to each other.

Material Of Design	**Steel**	**Aluminum**
Von Mises Stress FEA (MPa)	56.17	14
Deformation	0.012	0.217
Mass of Components (grams)	590	282
% Weight Reduction	0	12%

Table – 4 Result Summary Of FEA And Calculation .

Sr. No	**Parameter**	**Rectangular Section Beam**	**I Section Beam**	**C Section Beam**
1	Max. Induced Stress	Min.	Max.	Max.
2	Strain	-	Min.	Max.
3	Displacement	Min.	-	Max.
4	Yield Strength	Max.	Max.	Not Considered due to Torsion
5	Distribution Of Stress	-	Max.	-
6	Weight	Max. (Heavy)	Min.	-

Table – 5 Selection Of Cross Section Of Beam .

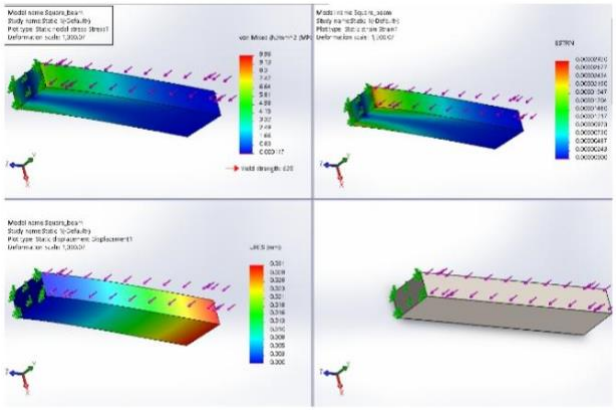


Fig. 15 Rectangular Section.

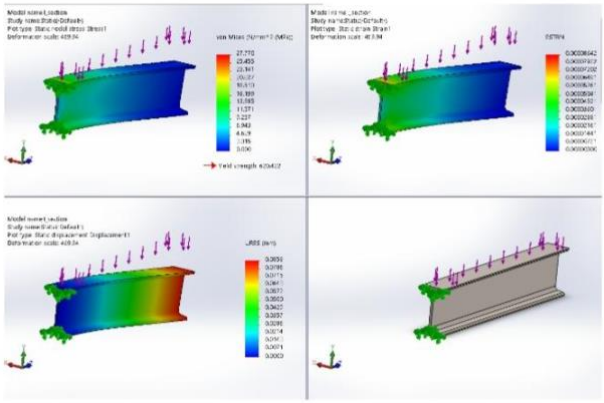


Fig. 16 I – Section

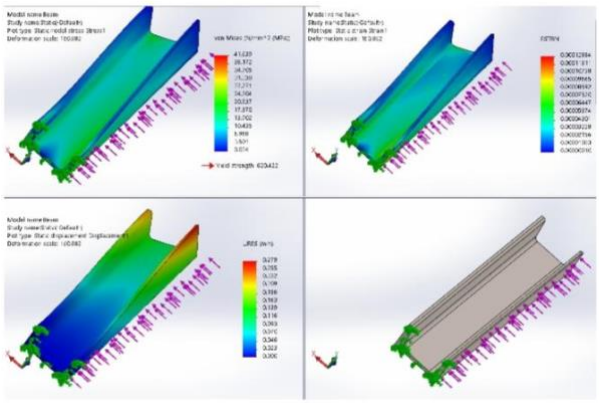


Fig. 17 C – Section.

XI. Conclusion

- In present research existing center console is redesigned with the help of topology optimization algorithm in ANSYS.
- Weight Optimization of 12 % is observed as initial weight is around 1.79 kg. and reduced weight is around 1.50 kg. including overall assembly (plastic center console with Aluminum Underbody). And the specially Center Console weight is 282 grams (Plastic material).
- It is observed that in Bending Stress on cantilever beam along I-section C/s area has been reduced in Optimized design compared to existing C- section C/s area design.
- Experimental UTM machine analysis compression, bending & shear test are nearly identical with numerical obtained analysis with an strength 48% more than existing design material (Steel) .
- Hence, It is confirmed that modified design of the Automotive Center Console Mounting Bracket is safe and validated with Experimental Result

Recommended – Cantilever Beam With I-section C/s , Plastic Center Console With Aluminum Underbody.

XII. References

1. Value engineering and value analysis of rear air spring bracket" by R.Vijayana, T.Thanka Geethab B Nishanthe. M.Tamilarasanc, V.Vijaya Kunare Materials Today: Proceedings 16 (2019) 10751083
2. Automotive console mounting bracket
3. HINGE UNIT OF CONSOLE ARMREST FOR VEHICLE [Seoyon E – Hwa Co.. Ltd Seoul (KR)
4. CONSOLE ASSEMBLY FORAVEHICLE Alan Sturt. West Bloomfield, MI (US): Jason Gamache. Romeo. MI (US)]
5. VEHICLE CONSOLE ASSEMBLY | David M. Emerling. West Bloomfield 1
6. Optimum Design of Automobile Chassis Bracket Based on Topography Optimization IOSR Journal of Mechanical and Civil Engineering (1OSR-IMCE)

Researcher/Author – Er. Sudarshan Dilip Bhalshankar.
Educational Qualification – M.E [CAD/CAM], B.E [Mechanical Engineering], D.A.E[Automobile Engineering].
E- mail – sudarshanbhalshankar111@gmail.com
Founder of – PradnyaSurya Engineering Works, Pvt. Ltd.
Add – Solapur, Maharashtra, India.
E- mail – pradnyasuryaengineerinworks@gmail.com