International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

ELSEVIER
SSRN

IJIEMR Transactions, online available on 18 Oct 2020.

Link :

https://papers.ssrn.com/sol3/JELJOUR_Results.cfm?form_name=journalbrowse&journal_id=3206923&dgcid=EngRN-PIP-IJIEMR_email_netann

Title:- **DESIGN AND STRUCTURAL ANALYSIS OF COMPOSITE MOUNTING BRACKET ASSEMBLY.**

Volume 09, Issue 10, Pages: 60 - 73.

Paper Authors

[1]MD SHOAIB IMRAN, [2]TR SYDANNA.

Department of Mechanical Engineering Sreenivasa College Of Engineering and Technology.

USE THIS BARCODE TO ACCESS YOUR ONLINE PAPER

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

DESIGN AND STRUCTURAL ANALYSIS OF COMPOSITE MOUNTING BRACKET ASSEMBLY

[1]MD Shoaib Imran, [2]TR SYDANNA
[1] M.tech Student, Department of Mechanical Engineering Sreenivasa College Of Engineering and Technology, (AP).
[2]Assistant Professor, Department of Mechanical Engineering Sreenivasa College Of Engineering and Technology, (AP).
[1]shoaibimran28@gmail.com, [2]sectknl@gmail.com

ABSTRACT:
In an automobile industry while designing the components, the most critical aspect considered is the compactness and the weight of the component. According to the Newton's 2 law of motion the energy required to accelerate the vehicle depends upon the mass of the automobile.
In the structural point of view the automotive materials should have more strength to weight ratio. One of the structural automotive vehicles and which are more in number are the mounting brackets. The mounting brackets are meant for supporting the structural component and electronic components such as batteries, seats, cabin, chassis, rear body and also it should support the external load such as passengers. Since it is very much needed to reduce the weight and therefore it is to be redesigned or optimized for minimum weight without sacrificing the functionality. The most common material used for the structural components is structural steel. Substituting composite structures for conventional metallic structures has many advantages because of higher specific stiffness and strength of composite materials.
Analysis will be performed for static and modal. The most important factors that are concentrated are stress distribution, deflection and natural frequencies for two materials. The design structure is carried through Unigraphics software. The reviewed design is analyzed through Ansys software.

INTRODUCTION

MOUNTING BRACKET

Brackets are meant for carrying loads, supports structures, bearings that support the shaft.

Brackets Are Of Different Types

Wall Brackets

Wall brackets are those fixed to wall for the purpose supporing bearings, which may either be cast with the brackets.

Pillar Brackets

It is used to support a horizontal shaft from a pillar where there is no wall in the way of wheels or pulleys on the shaft

Mounting Bracket

These are mainly used in automotive and aerospace industries for supporting the structures such as electronic goods like batteries, sensors in automobile for fixing the body to the chassis, for fixing the auxiliaries and for fixing the Cabin to the Chassis.

Applications of Mounting Brackets

1. Automobile industry
2. Aerospace industry
3. Electronic industry
4. Machine tool industry
5. Structural application

The shapes of the mounting brackets are different and are particularly varied according to the application. They may be hanging and supporting type. In dimensionality these mounting brackets differed according to the weight i.e. to be carried or of the size of the object to be carried.
Material selection for the brackets also depends on the application point of view. Most of the brackets are made up of cast iron in automobile industry as the load carrying capacity is more and good in compression and absorbs the disturbances in the traveling. In aerospace industry these are made up of aluminum as the weight is considered as critical factor and at the same time the strength to weight ratio is needed to be more.

The criterion for the design of the mounting bracket is depends on the loading also. If any eccentric loading acts on the brackets the design criteria is based on the shear and flexural moment. If the loads act transverse the design is based on the shear on the bending stress. These mounting brackets are fixed to the structures by bolts or pins.

DEFINITION OF THE PROBLEM

In any design problem the output of the design depends on the input given.i.e. The design parameters, which affects the design adversely and some of the definitions are to be mentioned clearly. Some of the general inputs are mentioned

- What is the importance of the part?
- What are the design parameters?
- What are the loads that are acting on the designed component?

The unambiguous definition of the problem makes the designer work easy. Probably the most critical step in the design process is the definition of the problem. The true problem is not always what it seems to be at first glance because this first step requires such a small part of the total time to create the final design, its importance is often overlooked.

This project deals with the design of the mounting bracket which are mostly used in many areas like automotive, aerospace etc… the mounting bracket, which is designed and analyzed is been used in a automobile (Load-King Pride). It is a field problem. Design is failing under the worst loading (Road-Load condition). The objective of this work is to provide optimum design to face such problems.

As the project deals with the design of mounting bracket, the existing design is analyzed and the results are taken as the reference for the present project work. The main problem is that, the mounting bracket analyzed for its minimum weight and also be taken care that stress should not exceed allowable or yield stress given for the composite material. Since the minimum weight is the one of the objective in the design, composites are also considered along with the present material. Composite that is considered in the present work is Graphite Epoxy Composite.

DESCRIPTION ABOUT THE PROBLEM

The present bracket that is being used in the vehicle weighs about 0.8kg. Four mounting brackets are used to support the entire cabin assembly. Total weight of the cabin assembly is 510 kg including three passengers. Two brackets are positioned at the front side of the cabin and remaining two are positioned at the rear side.

In this design I have done static analysis of the bracket. In the static analysis I had considered only the weight of the cabin including passengers.

Fig 1.1 Field problem of the mounting bracket.

Fig:1.1 show the practical problem of the mounting bracket. The red mark shows the mounting bracket failure

Fig1.2 Field problem of the mounting bracket.

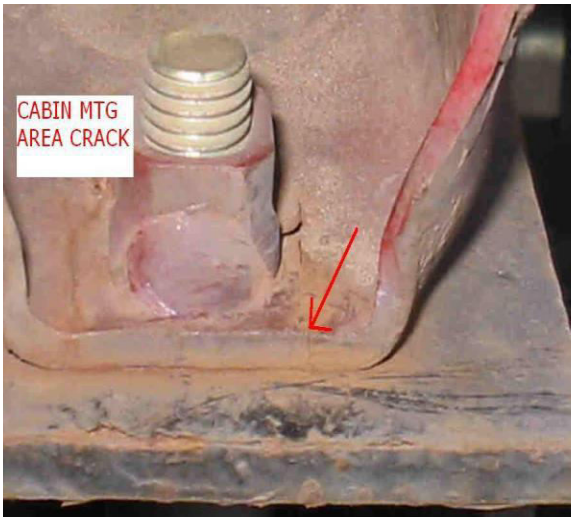


Fig:1.2 this figure shows the crack, which is developed on the mounting bracket after traveling 50000 kmph.

Fig 1.3 Field problem of the mounting bracket.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

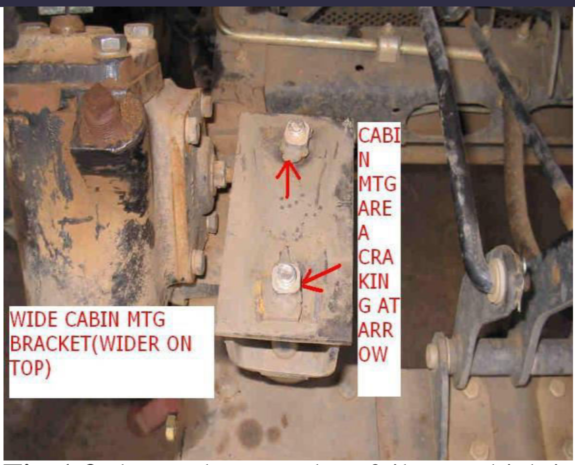


Fig:1.3 shows the complete failure, which is developed on the mounting bracket after traveling 75000 kmph.

COMPOSITE MATERIALS:

A composite material is made by combining two or more materials – often ones that have very different properties. The two materials work together to give the composite unique properties. The biggest advantage of modern composite materials is that they are light as well as strong. By choosing an appropriate combination of matrix and reinforcement material, a new material can be made that exactly meets the requirements of a particular application. Composites also provide design flexibility because many of them can be moulded into complex shapes. The downside is often the cost. Although the resulting product is more efficient, the raw materials are often expensive.

Composite material a combination of a matrix and a reinforcement, which when combined gives properties superior to the properties of the individual components. In the case of a composite, the reinforcement is the fibers and is used to fortify the matrix in terms of strength and stiffness. The reinforcement fibers can be cut, aligned, placed in different ways to affect the properties of the resulting composite. The matrix, normally a form of resin, keeps the reinforcement in the desired orientation. It protects the reinforcement from chemical and environmental attack, and it bonds the reinforcement so that applied loads can be effectively transferred.

TYPE OF COMPOSITES:

The term 'composite' can be used for a multitude of materials. Composites UK uses the term composite, or reinforced polymers to encompass:

1. Carbon fibre-reinforced polymers (CFRP)
2. Glass fibre-reinforced polymers (GFRP)
3. Aramid products (e.g. Kevlar)
4. Bio-derived polymers (or biocomposites)

MODELING OF MOUNTING BRACKET

3.1 UNIGRAPHICS INTRODUCTION

The NX Modelling application provides a solid modelling system to enable rapid conceptual design. Engineers can incorporate their requirements and design restrictions by defining mathematical relationships between different parts of the design.

Design engineers can quickly perform conceptual and detailed designs using the Modelling feature and constraint based solid modeller. They can create and edit complex, realistic, solid models interactively, and with far less effort than more traditional wire frame and solid based systems. Feature Based solid modelling and editing capabilities allow designers to change and update solid bodies by directly editing the dimensions of a solid feature and/or by using other geometric editing and construction techniques.

Advantages of Solid Modelling

Solid Modelling raises the level of expression so that designs can be defined in terms of engineering features, rather than lower-level CAD geometry. Features are parametrically defined for dimension-driven editing based on size and position.

Features

- Powerful built-in engineering-oriented form features-slots, holes, pads, bosses, pockets-capture design intent and increase productivity
- Patterns of feature instances-rectangular and circular arrays-with displacement of individual features; all features in the pattern are associated with the master feature

Blending and Chamfering

- zero radius
- Ability to chamfer any edge
- Cliff-edge blends for designs that cannot accommodate complete blend radius but still require blends

Advanced Modelling Operations

- Profiles can be swept, extruded or revolved to form solids
- Extremely powerful hollow body command turns solids into thin-walled designs in seconds; inner wall topology will differ from the outer wall, if necessary
- Fixed and variable radius blends may overlap surrounding faces and extend to a Tapering for modelling manufactured near-net shape parts
- User-defined features for common design elements (NX/User-Defined Features is required to define them in advance

General Operation

Start with a Sketch Use the Sketcher to freehand a sketch, and dimension an "outline" of Curves. You can then sweep the sketch using pad Body or shaft Body to create a solid or sheet body. You can later refine the sketch to precisely represent the object of interest by editing the dimensions and by creating relationships between geometric objects. Editing a dimension of the sketch not only modifies the geometry of the sketch, but also the body created from the sketch.

Creating and Editing Features

Feature Modeling lets you create features such as holes, slots and grooves on a model. You can then directly edit the dimensions of the feature and locate the feature by dimensions. For example, a Hole is defined by its diameter and length. You can directly edit all of these parameters by entering new values. You can create solid bodies of any desired design that can later be defined as a form feature using User Defined Features. This lets you create your own custom library of form features.

Associativity

Associatively is a term that is used to indicate geometric relationships between individual portions of a model. These relationships are established as the designer uses various functions for model creation. In an associative model, constraints and relationships are captured automatically as the model is developed. For example, in an associative model, a through hole is associated with the faces that the hole penetrates. If the model is later changed so that one or both of those faces moves, the hole updates automatically due to its association with the faces. See Introduction to Feature Modeling for additional details.

Positioning a Feature

Within Modeling, you can position a feature relative to the geometry on your model using Positioning Methods, where you position dimensions. The feature is then associated with that geometry and will maintain those associations whenever you edit the model. You can also edit the position of the feature by changing the values of the positioning dimensions.

Reference Features

You can create reference features, such as Datum Planes, Datum Axes and Datum CSYS, which you can use as reference geometry when needed, or as construction devices for other features. Any feature created using a reference feature is associated to that reference feature and retains that association during edits to the model. You can use a datum plane as a reference plane in constructing sketches, creating features, and positioning features. You can use a datum axis to create datum planes, to place items concentrically, or to create radial patterns.

Expressions

The Expressions tool lets you incorporate your requirements and design restrictions by defining mathematical relationships between different parts of the design. For example, you can define the height of a boss as three times its diameter, so that when the diameter changes, the height changes also.

Boolean Operations

Modelling provides the following Boolean Operations: Unite, Subtract, and Intersect. Unite combines bodies, for example, uniting two rectangular blocks to form a T-shaped solid body. Subtract removes one body from another, for example, removing a cylinder from a block to form a hole. Intersect creates a solid body from material shared by two solid bodies. These operations can also be used with free form features called sheets.

Undo

You can return a design to a previous state any number of times using the

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Undo function. You do not have to take a great deal of time making sure each operation is absolutely correct, because a mistake can be easily undone. This freedom to easily change the model lets you cease worrying about getting it wrong, and frees you to explore more possibilities to get it right.

3.2 DESIGN PROCEDURE

Geometric models of the design is shown in the following figure 3.1 respectively.

Fig 3.1 Existing design of the mounting bracket.

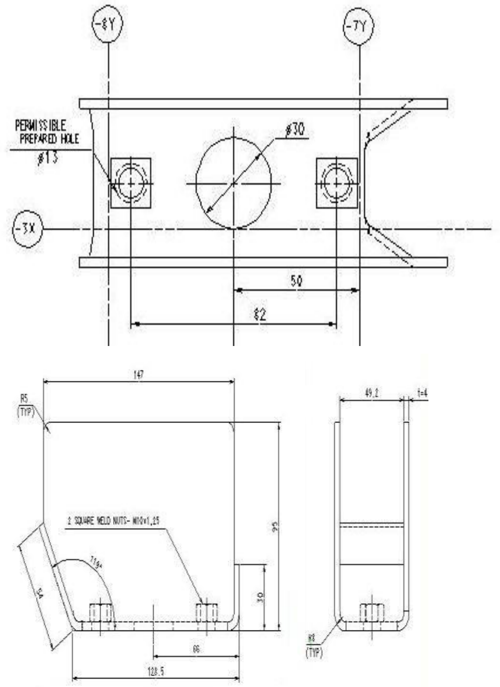


Fig: 3.1The above figure shows the detail description of the mounting bracket of the existing design.

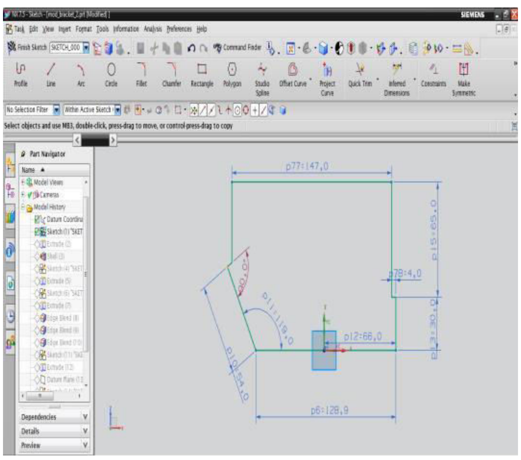


Fig.3.2 2D sketch of mounting bracket

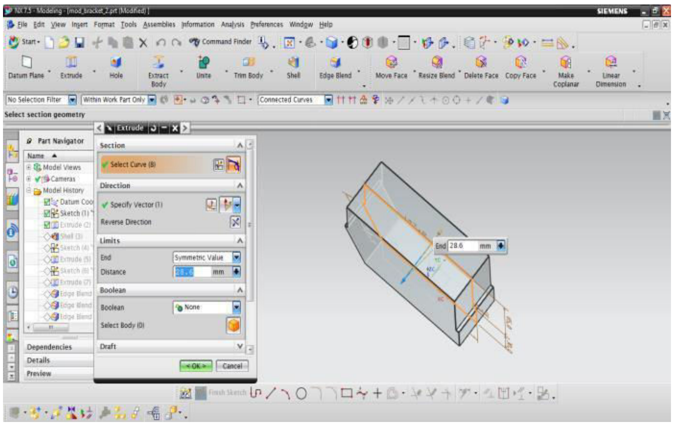


Fig.3.3 Extrude of mounting bracket

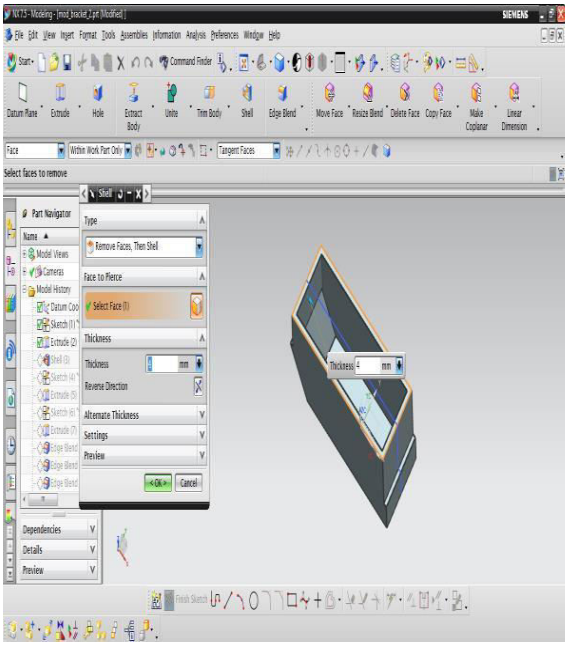


Fig.3.3 Shell of mounting bracket

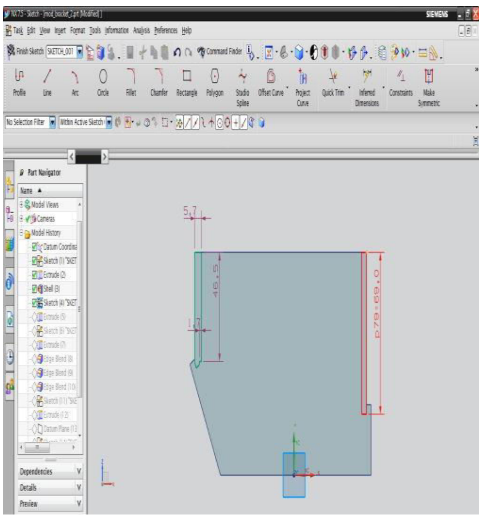


Fig.3.4 2D sketch of mounting bracket

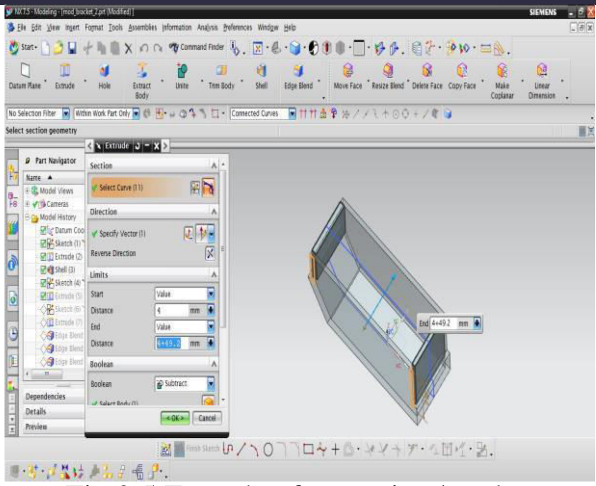


Fig.3.5 Extrude of mounting bracket

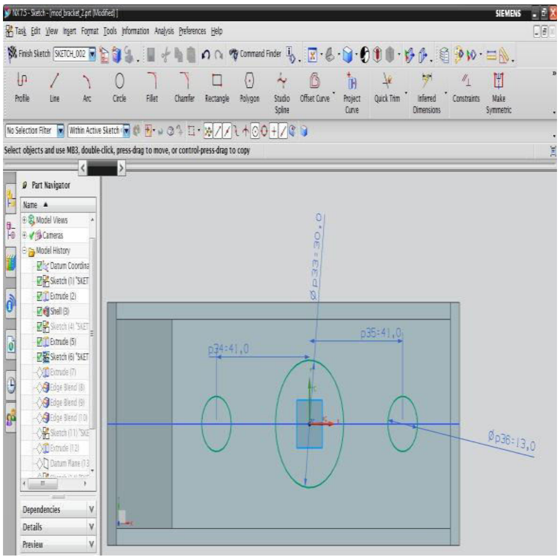


Fig.3.6 2D sketch of mounting bracket

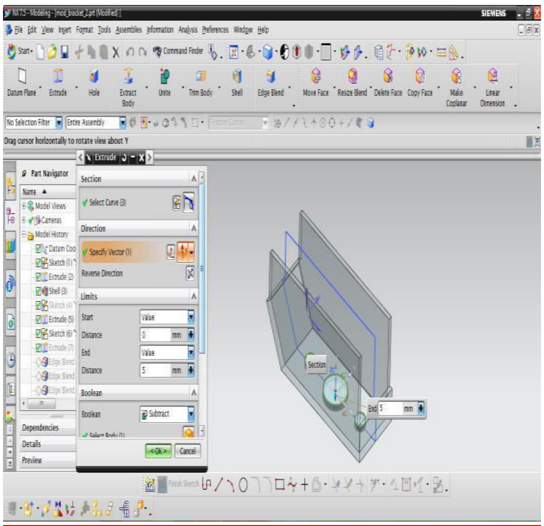


Fig.3.7 Extrude of mounting bracket

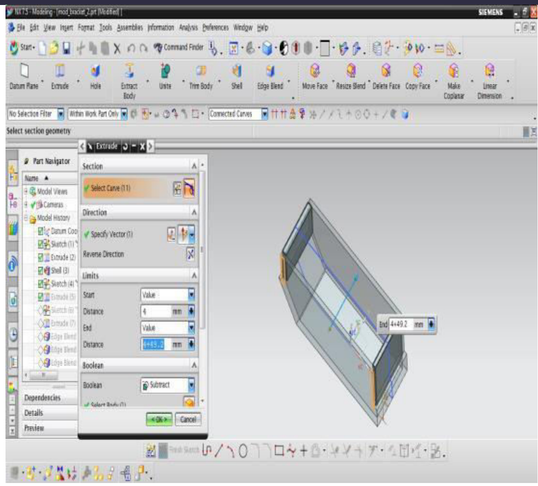


Fig.3.8 Edge blend of mounting bracket

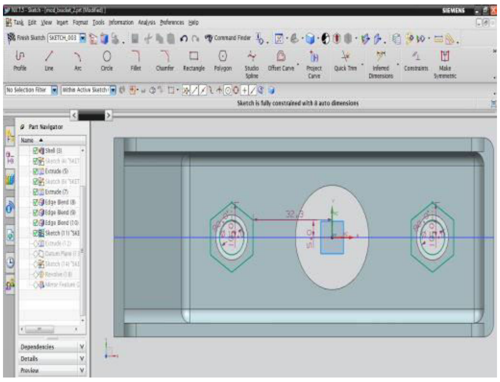


Fig.3.9 2D sketch of mounting bracket

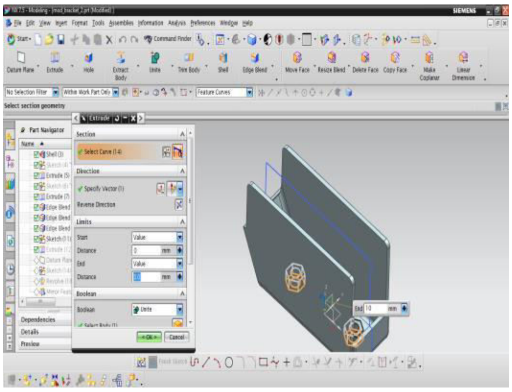


Fig.3.10 Extrude of mounting bracket

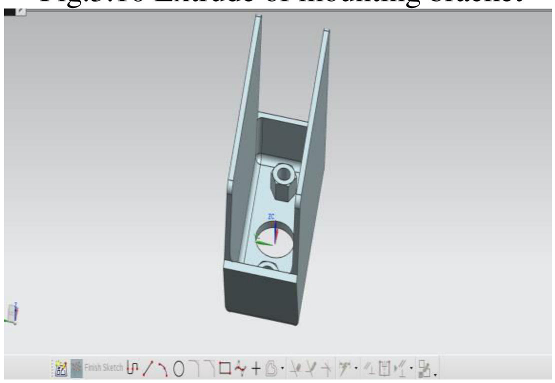


Fig.3.11 Final mounting bracket

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Fig.3.12 Final mounting bracket

FINITE ELEMENT METHOD

INTRODUCTION ABOUT FEM

The Basic concept in FEM is that the body or structure may be divided into smaller elements of finite dimensions called “Finite Elements”. The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called “Nodes” or “Nodal Points”. Simple functions are chosen to approximate the displacements over each finite element. Such assumed functions are called “shape functions”. This will represent the displacement within the element in terms of the displacement at the nodes of the element. The Finite Element Method is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problems that can be represented in differential equations form. The applications of FEM are limitless as regards the solution of practical design problems.

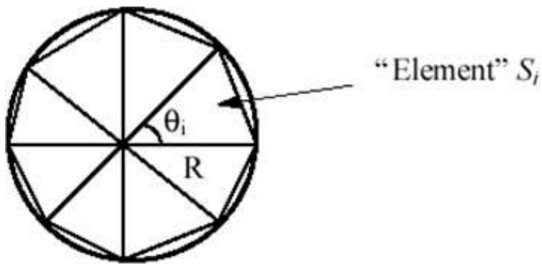
Due to high cost of computing power of years gone by, FEM has a history of being used to solve complex and cost critical problems. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an automobile or an aircraft. In the recent years, FEM has been universally used to solve structural engineering problems. The departments, which are heavily relied on this technology, are the automotive and aerospace industry. Due to the need to meet the extreme demands for faster, stronger, efficient and lightweight automobiles and aircraft, manufacturers have to rely on this technique to stay competitive.

FEA has been used routinely in high volume production and manufacturing industries for many years, as to get a product design wrong would be detrimental. For example, if a large manufacturer had to recall one model alone due to a hand brake design fault, they would end up having to replace up to few millions of hand brakes. This will cause a heavier loss to the company.

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 analysis
- Thermal analysis
- Vibrations and Dynamics
- Buckling analysis
- Acoustics
- Fluid flow simulations
- Crash simulations
- Mold flow simulations

Nowadays, even the most simple of products rely on the finite element method 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 the years gone by, but now it is simply too expensive and time consuming also.



Basic Concepts: The Finite Element Method is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life as well as engineering. The philosophy of FEA can be explained with a small example such as measuring the area of a circle.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Area of one Triangle: Si = ½ * R2* Sin I

Area of the Circle: SN = ½ * R2 * N * Sin (2 / N) R2 as N

Where N = total number of triangles (elements)

If one needs to evaluate the area of the circle without using the conventional formula, one of the approaches could be to divide the above area into a number of equal segments. The area of each triangle multiplied by the number of such segments gives the total area of the circle.

Brief history of the fem:

Who

The reference credited is to Courant (Mathematician), Turner (air craft industry), clough (California university), Martin (air craft industry), argyris (German university),…. However, it was probably established by several pioneers independently.

When

- Initial idea in mathematical terms was put in 1940s.
- Application to simple engineering problems in 1950s.
- Implementation in large computer is 1960s.
- Development of pre and post processors in 1980s.
- Analysis of large structural problems in 1990s.

Where

Implementation and application were mainly in aircraft industry and automobile sectors (large and fast computers were available only in these industries)

What

Field problems in the form matrix methods of organizing large numbers of algebraic equations are used and matrix equations are solved. Differential equations are transformed into an algebraic form. Blocks with different geometry are hooked together for creating complex geometry of the engineering problem

Why

The advantage of doing FEM analysis is that it is fairly simple to change the geometry, material and loads recomputed stresses for modified product rather than build and test. The method can be used to solve almost any problem that can be formulated as a field problem. The entire complex problem can be cast as a larger algebraic equation by assembling the element matrices within the computer and solved.

Available Commercial FEM software packages

- ANSYS (General purpose, PC and workstations)
- SDRC/I-DEAS (Complete CAD/CAM/CAE package)
- NASTRAN (General purpose FEA on mainframes)
- LS-DYNA 3D (Crash/impact simulations)
- ABAQUS (Nonlinear dynamic analysis)
- NISA (A General purpose FEA tool)
- PATRAN (Pre/Post processor)
- HYPERMESH (Pre/post processor)

More about FEA

Finite Element Analysis was first developed for use in the aerospace and nuclear industries where the safety of the structures is critical. Today, the growth in usage of the method is directly attributable to the rapid advances in computer technology in recent years. As a result, commercial finite element packages exist that are capable of solving the most sophisticated problems, not just in structural analysis. But for a wide range of applications such as steady state and transient temperature distributions, fluid flow simulations and also simulation of manufacturing processes such as injection molding and metal forming.

FEA consists of a computer model of a material or design that is loaded and analyzed for specific results. It is used in new product design, and existing product refinement. A design engineer shall be able to verify the proposed design, which is intended to meet the customer requirements prior to the manufacturing. Things such as, modifying the design of an existing product or structure in

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

order to qualify the product or structure for a new service condition. Can also be accomplished in case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

The Basic Steps Involved in FEA

Mathematically, the structure to be analyzed is subdivided into a mesh of finite sized elements of simple shape. Within each element, the variation of displacement is assumed to be determined by simple polynomial shape functions and nodal displacements. Equations for the strains and stresses are developed in terms of the unknown nodal displacements. From this, the equations of equilibrium are assembled in a matrix form which can be easily be programmed and solved on a computer. After applying the appropriate boundary conditions, the nodal displacements are found by solving the matrix stiffness equation. Once the nodal displacements are known, element stresses and strains can be calculated.

Basic Steps in FEA

1. Preprocess

Create geometrical model by either hyper mesh or ansys software or any cad software.

Import a cad model in ansys software if geometrical model was done by cad software.

Select analysis type such as static, modal, transient dynamic, thermal analysis.

Insert element type either 2-Dimensional(ex: Trusses, Beams) or 3-Dimensional (ex: Solid, Shell, Plate)

What is an Element?

Element is an entity, into which a system under study can be divided into. An element definition can be specified by nodes. The shape (area, length, and volume) of the element depends upon the nodes with which it is made up of.

What are Nodes?

Nodes are the corner points of the element. Nodes are independent entities in the space. These are similar to points in geometry. By moving a node in space an element shape can be changed. This is a volume element, can take the shape of a Hexahedron or a Wedge or a Tetrahedron order elements. For linear elements the edge is defined by a linear function called shape function whose degree is one. For the elements having mid side nodes on the edge quadratic function called shape function whose degree is two is used. The higher order elements when over lapped on geometry can represent complex shapes very well within few elements. Also the solution accuracy more with the higher order elements. But higher order elements will require more computational effort and time.

- Apply material properties (Young's modulus, Poisson ratio, Thermal conductivity, Density values) based on types of materials.
- Apply meshing to FE model.
- Also Apply boundary conditions and Loading conditions.

Solution

- Get nodal solutions such as displacement values at each node or temperature values at each node by solving linear or non linear algebraic equations at each node.
- These solutions formed based on matrix formulation for system of equations. That formula noted as

$$[K]*[x] = [F]$$

[K] is system stiffness matrix

[x] is displacement vector matrix

[F] is force vector

3.Postprocessing

Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution.

Theories of failure:

Determining the expected mode of failure is an important first step in analysing a part design. The failure mode will be influenced by the nature of load, the expected response of the material and the geometry and constraints. In an engineering sense, failure may be defined as the occurrence of any event considered to be unacceptable on the basis of part performance. The modes of failure considered here are related to mechanical loads and structural analysis. A failure may include either an unacceptable response to a temporary load involving no permanent damage to the part or an acceptable response which does involve permanent, and sometimes catastrophic, damage. The purpose of theories of failure is to predict what combination of principal stresses will result in failure. There are number of theories to describe failure criteria, of them these are the widely accepted theories.

Maximum principal stress theory (rankine's) $\sigma 1$ or $\sigma 2$ or $\sigma 3$ (whichever is maximum) = σy. According to this theory failure of the material is assumed to have taken place under a state of complex stresses when the value of the maximum principal stress reaches a value equal to that of the elastic limit stress (yield stress) as found in a simple tensile test.

Maximum shear theory (guest's or coulomb's) $(\sigma 1-\sigma 2)$ or $(\sigma 2-\sigma 3)$ or $(\sigma 3-\sigma 1) = \sigma y$ (Whichever is maximum). According to this theory the failure of the material is deemed to have taken place when the maximum shear stress exceeds the maximum shear stress in a simple tension test.

Maximum principal strain theory (St.Venant's)
$\sigma 1-\nu(\sigma 2+\sigma 3)$ or $\sigma 2-\nu(\sigma 3+\sigma 1)$ or
$\sigma 3-\nu(\sigma 1+\sigma 2)$ (whichever is maximum) = σy.

According to this theory, failure of the material is deemed to have taken place when the maximum principal strain reaches a value calculated from a simple tensile test.

Maximum strain energy theory (Beltrami-Haigh's)
$\sigma 1^2+\sigma 2^2\sigma 3^2-2\nu(\sigma 1\sigma 2+\sigma 2\sigma 3+\sigma 3\sigma 1) = \sigma y^2$. According to this theory failure is assumed to take place when the total strain energy exceeds the strain energy determined from a simple tensile test.

Octahedral or distortion energy theory (von mises-hencky)
$\sigma 1^2+\sigma 2^2+\sigma 3^2-\sigma 1\sigma 2-\sigma 2\sigma 3-\sigma 3\sigma 1 = \sigma y^2$. According to this theory failure is assumed to take place when the maximum shear strain energy exceeds the shear strain energy in a simple tensile test. This is very much valid for ductile material; in this the energy which is actually responsible for the distortion is taken into consideration.

Soderberg's equations (recommended for ductile materials only):

$1/n = \sigma m/\sigma y + Kf\, \sigma a/\sigma\text{-}1$
$1/n = tm/ty + Kf\, ta/t\text{-}1$
Where, σm =mean stress
σy = yield stress
σa = stress amplitude $(\sigma max-\sigma min)/2$
$\sigma\text{-}1$ = endurance limit stress
tm = mean shear stress
ty = yield shear stress
1/n = factor of safety

Goodman's equations (for brittle materials)
$1/n=Kt\,[\sigma m/\sigma u + \sigma a/\sigma\text{-}1]$
$1/n=Kt[tm/tu + ta/t\text{-}1]$
Where, σu = ultimate stress
K = stress concentration factor

Choice of the theories of failure:
Well documented experimental results by various authors on the various theories of failure, indicate that the distortion energy theory predicts yielding with greatest accuracy. Compared to this maximum shear stress theory predicts results which are always on safer side. Maximum principal stress theory gives conservative results only if the sign of the two principal stresses is the same (2-D case). Therefore, the use of maximum principal stress theory for pure torsion is ruled out where the sign of the two principal stresses are opposite.

When the fracture of a tension specimen loaded up to rupture is examined, it shows that for ductile materials, failure occurs along lines at angles 45 degrees with the load axis. This indicates a shear failure. Brittle materials on the other hand, rupture on planes normal to the load axis, indicating that maximum normal stress determines failure. Because of the above mentioned observations, it is universally accepted that for a brittle materials, the maximum normal stress theory is the most suitable. For ductile materials, the maximum shear stress theory gives conservative results and it is simpler to use as compared to distortion energy theory, so it is universally accepted as the theory of failure for ductile materials. But, where low weight is desired, the distortion energy theory is recommended.

In brief:
Ductile material

Under combined static loading, the machine parts made of ductile material will fail by yielding. The working or allowable stress is therefore, passed on the yield point stress. The maximum shear stress theory will be used for the design because it is conservative and easy to apply.

Brittle materials

Failure in brittle materials, takes place by fracture. Brittle materials do not have a distinct yield point and so, the ultimate strength is used as the basis for determining the allowable or design stress. Separate design equations should be used in tension and compression, since for materials like cast iron; the ultimate compressive strength is considerably greater than the ultimate tensile strength. The maximum principal stress theory will be used for the design. Due consideration will be given to the sign of principal stresses. If both the principal stresses (2-D case) are of the same sign, the effect of the smaller stress is neglected. If the two principal stresses are of opposite sign, then the maximum principal stress theory does not give conservative results. In that case another equation should be used.

This gives information about the methods that are used for obtaining the optimized solution. Here in this problem, static analysis was carried out to know the strength of the mounting bracket. Static analysis shows the strength of the mounting bracket when the vehicle is in the rest position. Here the constraint on thickness is, it should not exceed 5mm. The increase in thickness will add considerable weight to the bracket. With the previous parameters, i.e. the dimensional parameters the structure is modeled in UNIGRAPHICS modeling software. The models can be meshed for further analysis using ANSYS. Meshing of the component plays an important role in analysis, as it is the basis for analyzing the component in any software package, which supports finite element techniques. As the brackets are used in aerospace vehicles they should be light in weight. Therefore according to the requirement aluminum alloy considered. The material prosperities of STEEL HR 11DD are tabulated in the table 4.1.

Table 4.1 Material properties of steel hr 11dd

MATERIAL PROPERTIES	MAGNITUDES
DENSITY	7850e-12 tons/mm3
YOUNG'S MODULUS	210000 Mpa
IN PLANE POISSON'S RATIO	.3

RESULTS AND DISSCUSSIONS

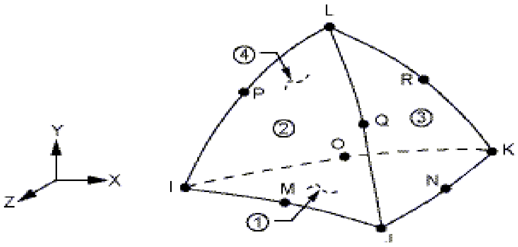
The finite element modeling and analysis is used to study the stress variation at different locations of the mounting bracket and also the deflection at various locations of the mounting bracket having various thicknesses with single material. Topology optimization is also carried out to extract the best design and to reduce the weight of the mounting bracket.

Two parameter are studied in this thesis work, one is the thickness and the other one is the weight of the mounting bracket to rectify the field problem. Table 5.1 furnishes the results summary of various mounting brackets with different thick nesses 3.2 and 4 .It can be seen that even though thickness is varying, the applied load does not vary. This table furnishes the deformations, von mises stresses and the material reduction in each design against the maximum loading condition. The 3d model is imported into ansys and meshed with solid 92 element type. The description of solid 92 is given below.
No of Nodes: 10
No. of dof: 3 (ux, uy, uz)

SOLID187 Element Description

SOLID187 has a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org



SOLID187 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure "SOLID187 Geometry".

Beside the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems.

Element loads are described in Node and Element Loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure "SOLID187 Geometry". Positive pressures act into the element. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T (I) defaults to TUNIF. If all other temperatures are unspecified, they default to T (I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

The force of 750N is applied on the two bolting locations using mass 21 element type.The description of mass 21 is given below.

LOAD CALCULATIONS

The static load is calculated by using the weight of the cabin. The calculations are as follows.
Total weight of the cabin =285 kgf
Three passengers = 3*75 =225 kgf
Total weight acting on the four mounting brackets = 285+225 = 510 kgf

When the vehicle is on the rest position 40 % of the total weight is going to act on the two front mounting brackets and 60% of the total weight is going to act on the rear two mounting brackets.

40% of the total weight
= 204 kgf
60% of the total weight
= 306 kgf
40% of the total weight on each bracket
= 102 kgf
60% of the total weight on each bracket
= 153 kgf
Total force acting on the each bracket
= 153*9.8 = 1500 N.

This load is acting along the vertical Y direction. Each bracket consists of two bolts. Total load is going to act on these two bolts. So each bolt takes half amount of the total load. Constraints were considered at the welding region.
Total load acting on the each bolt = 750 N

STATIC ANALYSIS OF MOUNTING BRACKET USING COMPOSITE E-GLASS/EPOXY

3d model of the existing bracket:

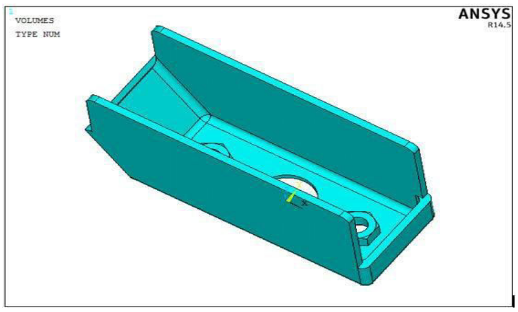
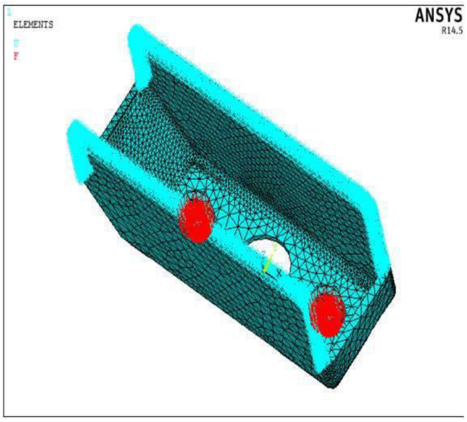


Fig5.1: Imported 3d model of assembly mounting bracket in ansys software

FE model of the existing bracket:

Results:Boundary Conditions on the existing model:

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

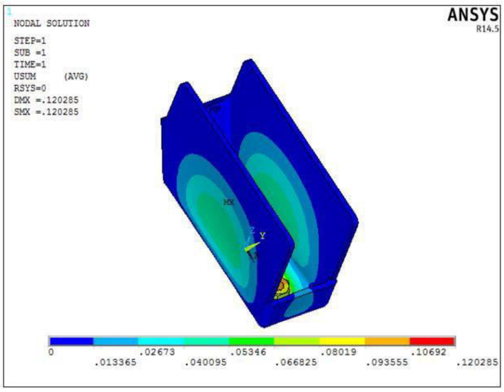


Material properties of Eglass/epoxy:

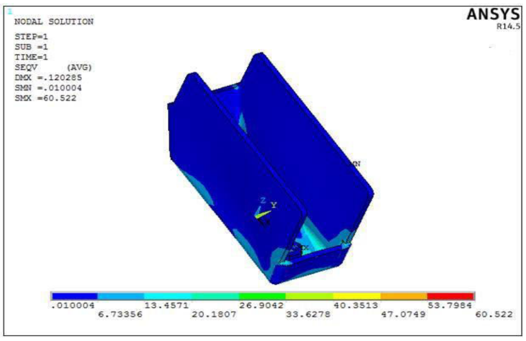
Property	Glass/epoxy
Young's modulus in fiber direction, E_1 (GPa)	53.8
Young's modulus in transverse direction, E_2 (GPa)	17.9
Shear modulus, G_{12} (GPa)	8.96
Major Poisson's ratio, υ_{12}	0.25
Minor Poisson's ratio, υ_{21}	0.08
Strength in the fiber direction, X_L (MPa)	1.03×10^3
Strength in the transverse direction, X_T (MPa)	27.58
Shear strength, S (MPa)	41.37

Results of Eglass/epoxy:

Resultant deflection:



VonMises Stress



From the above results the VonMises stress observed on the bracket with Eglass/epoxy material is 60.52Mpa

STATIC ANALYSIS OF MOUNTING BRACKET USING STEEL MATERIAL

3d model of the existing bracket:

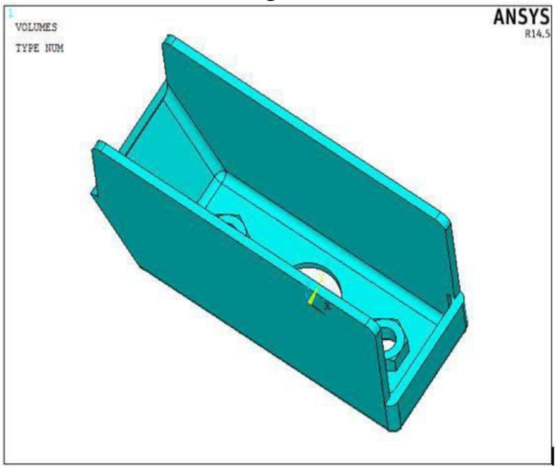
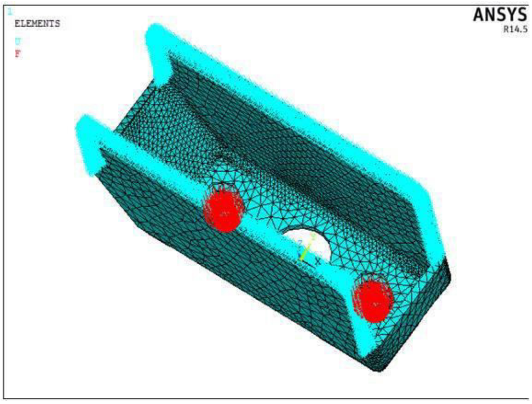


Fig: Imported 3d model of assembly mounting bracket in ansys software

FE model of the existing bracket:

Results: Boundary Conditions on the existing model:



Material properties of Steel material

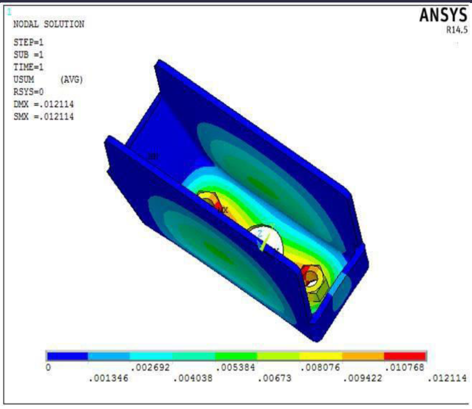
YOUNG'S MODULUS (E)= 200GPa

Poisson ration= 0.3

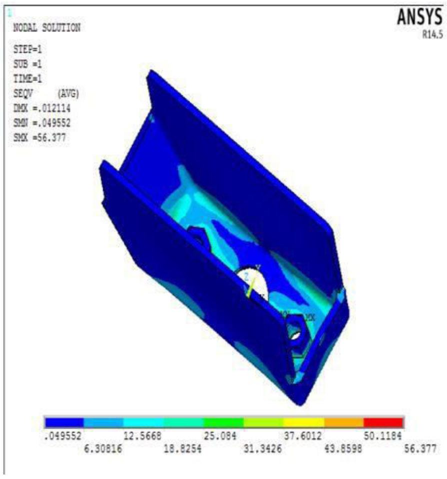
Density = 7800Kg/m^3

Results of steel:

Resultant deflection:



VonMises Stress



From the above results the VonMises stress observed on the bracket with steel material is 56.377Mpa

CONCLUSION

Two different materials were studied under static loading conditions and attempts were made to reduce the stresses on the bracket and minimize weight.

Table shows results summary of various mounting brackets

SI. No	Design	(mm) Deforma-tion	(Mpa) Von - Mises stress(MPa)
1	E glass/epoxy material	.04	83.4
2	steel material	0.12	56.3

Modal analysis was also performed to understand the dynamic behavior of the Mounting bracket. Based on the results, we can forecast the possibility of mutual interference between the mounting bracket and other parts. The resonance vibration of system can be avoided effectively by appropriate structure design .It is also observed the stiffness is more for Eglass/epoxy material. From the above results we can conclude E-Glass/epoxy material can be used for mounting brackets to have more strength to weight ratio.

REFERENCES

1. C.C. Menzemer, L. Fei, T.S. Srivatsan, Design criteria for bolted connection elements in Aluminium Alloy 6061, Transactions of ASME, Vol.121, Sep 1991, p348-357.

2. J. Newnham, L. Curley, Bolted joints under tensile loads: The effect of torsion in the Bolt on in-service clamping loads, Transactions of ASME, vol.115, March 1993, p36-40.

3. Nina Lauterstajn-S and Alf Samuelsson: Further discussion on four-node isoparametric quadrilateral elements in plane bending, International Journal for Numerical Methods in Engineering, vol. 47, pp. 129-140, (2000).

4. Mats Ander and Alf Samuelsson: Finite Element analysis of Geometrically Non-Linear Structures Using Translational Variables, International Journal for Numerical Methods in Engineering, Vol. 46, pp.1367-1383, (November 1999).

5. Kjell Mattiasson, Lars Bernspong and Alf Samuelsson: Solution of quasi-static, force-driven problems by means of a dyanamic-explicit approach and an adaptive loading procedure, Engineering Computations, vol.13, pp. 172-189, (1996).

6. Alf Samuelsson and Nils-Erik Wiberg: Finite element adaptivity in dynamics and elastoplasticity". Volume "The finite element method in the 1990's, Springer-Verlag, 11 pp. (1991).

7. Magnus Kjellman and Alf Samuelsson: A high-precision adaptive procedure for solving Kirchhoff plates, in Discretization Mehtods in Structural Dynamics, (Eds. Kuhh, G. and Mang H.), Springer, pp. 125-133, (1990).