Evaluation Of Hydraulic Breaking System In Automobile To Implementing The Performance Of Master Cylinder Using Fea

Muraboena Harikrishna[1], T.R Sydanna, M.Tech [2]

[1]P.G. Scholar, [2] Associate Professor

[1,2] Branch : M.Tech (Machine Design) Mechanical Engineering

[1,2] Sreenivasa College Of Engineering And Technology

Email:- [1]hari.krishna0595@gmail.com, [2]sectknl@gmail.com

ABSTRACT

The demand for a reduction in failures in brake pedal and vibration movement without losing any wastage of raw material and machining cost has play major role for effective breaking system in vehicles. This opportunity occurs at accurate design and stress safe zone must consider in pressurizing the brake fluid inside of the master cylinder. Braking system is a means of converting momentum into heat energy by creating friction in the wheel brakes. The braking system which works with the help of hydraulic break oil is known as hydraulic braking systems. The braking system used most frequently operates hydraulically, by pressure applied through a liquid.

In this article, design of Tandem master cylinder is explained in details. UNIGRAPHICS software is used for developing design. After that designed master cylinder imported in Ansys software for performing structural and vibration analysis. The goals of the structural analysis are to visualize the stress distribution, deformation under static pressure condition. Dangerous stress distribution found on master cylinder and avoided those regions in manufacturing process. The vision of vibration analysis is to finding the free vibration (natural frequency) characters and forced vibration (forced frequency) characteristics of master cylinder using Ansys FEA application.

Keywords: Cylinder, Heat Energy, Hydraulic Pressure, Slave Cylinders

INTRODUCTION
1.1 MASTER CYLINDER

In automotive engineering, the master cylinder is a control device that converts non-hydraulic pressure (commonly from a driver's foot) into hydraulic pressure. This device controls slave cylinders located at the other end of the hydraulic system.

As piston(s) move along the bore of the master cylinder, this movement is transferred through the hydraulic fluid, to result in a movement of the slave cylinder(s). The hydraulic pressure created by moving a piston (inside the bore of the master cylinder) toward the slave cylinder(s) compresses the fluid evenly, but by varying the comparative surface area of the master cylinder and each slave cylinder, one can vary the amount of force and displacement applied to each slave cylinder, relative to the amount of force and displacement applied to the master cylinder.


Fig: 1.1 shows the master cylinder model

The most common vehicle uses of master cylinders are in brake and clutch systems. In brake systems, the operated devices are cylinders inside brake calipers and/or drum

brakes; these cylinders may be called wheel cylinders or slave cylinders, and they push the brake pads towards a surface that rotates with the wheel (this surface is typically either a drum, or a disc, a.k.a. a rotor) until the stationary brake pad(s) create friction against that rotating surface (typically the rotating surface is metal or ceramic/carbon, for their ability to withstand heat and friction without wearing-down rapidly).

In the clutch system, the device which the master cylinder operates is called the slave cylinder; it moves the throw out bearing until the high-friction material on the transmission's clutch disengages from the engine's metal (or ceramic/carbon) flywheel.
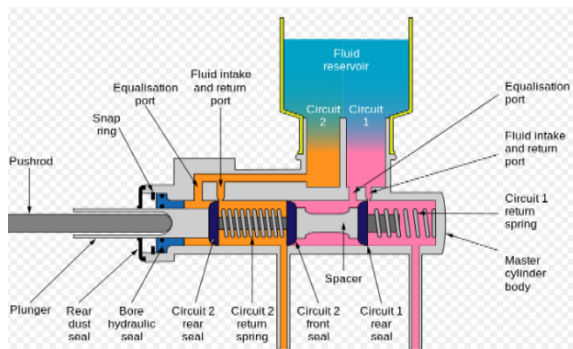
**Fig: 1.2 Inner section of the master cylinder**

For hydraulic brakes or clutches alike, flexible high-pressure hoses or inflexible hard-walled metal tubing may be used; but the flexible variety of tubing is needed for at least a short length adjacent to each wheel, whenever the wheel can move relative to the car's chassis (this is the case on any car with steering and other suspension movements; some drag racers and go-karts have no rear suspension, as the rear axle is welded to the chassis, and some antique cars also have no rear suspension movement).

Fig: 1.3 Assembly model of master cylinder

A reservoir above each master cylinder supplies the master cylinder with enough brake fluid to avoid air from entering the master cylinder (even the typical clutch uses brake fluid, but it may also be referred to as "clutch fluid" in a clutch applr). Each piston in a master cylinder operates a brake circuit, and for modern light trucks and passenger cars, usually a brake circuit leads to a brake caliper or shoe on only two of the vehicle's wheels and the other brake circuit provides brake pressure to slow down and stop the other two wheels.

This is done in a diagonally split hydraulic system. If there is a hydraulic failure in the brake lines served by the master cylinder's secondary piston both pistons will move forward when the brakes are applied, but there is nothing to resist piston travel except the secondary piston spring. This lets the primary piston build up only a small amount of pressure until the secondary piston bottoms in the cylinder bore. Then the primary piston will build enough hydraulic pressure to operate the brakes served by this half of the system. In case of a hydraulic failure in the brake system served by the primary piston, the primary piston will move forward when the brakes are applied but will not build up hydraulic pressure.
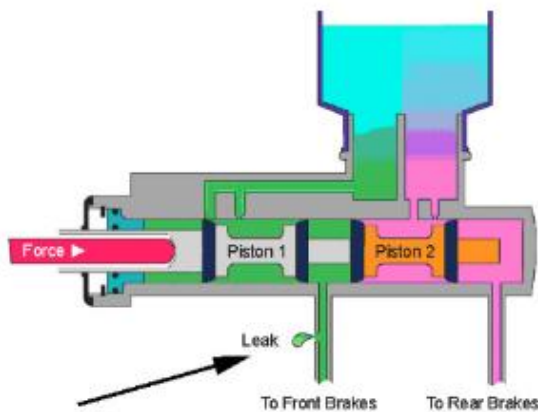
Fig: 1.4 Shows the inner piston arrangement in master cylinder

Fig: 1.5 Shows the twin pistons in master cylinder

In this case very little force is transferred to the secondary piston through the primary piston spring until the piston extension screw comes in contact with the secondary piston. Then, pushrod force is transmitted directly to the secondary piston and enough pressure is built up to operate its brakes. Usually the 2 systems are split into left front and right rear comprising one system and right front and left rear comprising the other system. For safety, this is done so that usually only two wheels lose their braking ability at the same time and that both sides of the car have at least one braking system working and both ends of the car have at least one system working. With only 1 system working there is longer stopping distances and repairs should be done before driving again.

INTRODUCTION TO CAE

In general, there are three phases in any computer-aided engineering task:

- **Pre-processing** – defining the finite element model and environmental factors to be applied to it
- **Analysis solver** – solution of finite element model
- **Post-processing** of results using visualization tools

PRE-PROCESSING

The first step in using FEA, pre-processing, is constructing a finite element model of the structure to be analyzed. The input of a topological description of the structure's geometric features is required in most FEA packages. This can be in either 1D, 2D, or 3D form, modeled by line, shape, or surface representation, respectively, although nowadays 3D models are predominantly used. The primary objective of the model is to realistically replicate the important parameters and features of the real model. The simplest mechanism to achieve modeling similarity in structural analysis is to utilize pre-existing digital blueprints, design files, CAD models, and/or data by importing that into an FEA environment. Once the finite element geometric model has been created, a meshing procedure is used to define and break up the model into small elements. In general, a finite element model is defined by a mesh network, which is made up of the geometric arrangement of elements and nodes. Nodes represent points at which features such as displacements are calculated. FEA packages use node numbers to serve as an identification tool in viewing solutions in structures such as deflections. Elements are bounded by sets of nodes, and define localized mass and stiffness properties of the model. Elements are also defined by mesh numbers, which allow references to be made to corresponding deflections or stresses at specific model locations.

ANALYSIS (COMPUTATION OF SOLUTION)

International Journal of Research

Available at https://journals.pen2print.org/index.php/ijr/

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 09
August 2019

The next stage of the FEA process is analysis. The FEM conducts a series of computational procedures involving applied forces, and the properties of the elements which produce a model solution. Such a structural analysis allows the determination of effects such as deformations, strains, and stresses which are caused by applied structural loads such as force, pressure and gravity.
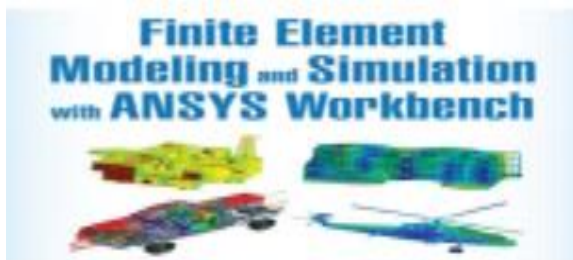
POST-PROCESSING (VISUALIZATION)

These results can then be studied using visualization tools within the FEA environment to view and to fully identify implications of the analysis. Numerical and graphical tools allow the precise location of data such as stresses and deflections to be identified.

Basic Steps in FEA

- Discretization of the domain
- Application of Boundary conditions
- Assembling the system equations
- Solution for system equations
- Post processing the results.

Discritization of the domain: The task is to divide the continuum under study into a number of subdivisions called element. Based on the continuum it can be divided into line or area or volume elements.

Application of Boundary conditions: From the physics of the problem we have to apply the field conditions i.e. loads and constraints, which will help us in solving for the unknowns.

Assembling the system equations: This involves the formulation of respective characteristic (Stiffness in case of structural) equation of matrices and assembly.

Solution for system equations: Solving for the equations to know the unknowns. This is basically the system of matrices which are nothing but a set of simultaneous equations are solved.

Viewing the results: After the completion of the solution we have to review the required results.

The first two steps of the above said process is known as pre-processing stage, third and fourth is the processing stage and final step is known as post-processing stage.

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.

5.4 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) $\sigma1$ or $\sigma2$ or $\sigma3$ (which ever 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) $(\sigma1-\sigma2)$ or $(\sigma2-\sigma3)$ or $(\sigma3-\sigma1)$ = σy (Which ever 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)

$\sigma1-v(\sigma2+\sigma3)$ or $\sigma2-v(\sigma3+\sigma1)$ or $\sigma3-v(\sigma1+\sigma2)$ (which ever 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)

$\sigma1^2+\sigma2^2\sigma3^2-2v(\sigma1\sigma2+\sigma2\sigma3+\sigma3\sigma1) = \sigma y^2$. 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)

$\sigma1^2+\sigma2^2+\sigma3^2-\sigma1\sigma2-\sigma2\sigma3-\sigma3\sigma1 = \sigma y^2$. 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):

$1/n = \sigma m/\sigma y + Kf\, \sigma a/\sigma-1$

$1/n = tm/ty + Kf\, ta/t-1$

Where, σm = mean stress
σy = yield stress
σa = stress amplitude ($\sigma max-\sigma min)/2$
$\sigma-1$ = endurance limit stress
tm = mean shear stress
ty = yield shear stress
$1/n$ = factor of safety

Goodman's equations (for brittle materials)

$1/n = Kt[\sigma m/\sigma u + \sigma a/\sigma-1]$

$1/n = Kt[tm/tu + ta/t-1]$

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.

STRUCTURAL ANALYSIS

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behavior of structures. The subjects of structural analysis are engineering artifacts whose integrity is judged largely based upon their ability to withstand loads; they commonly include buildings, bridges, aircraft, and ships. Structural analysis incorporates the fields of mechanics and dynamics as well as the many failure theories. From a theoretical perspective the primary goal of structural analysis is the computation of deformations, internal forces, and stresses. In practice, structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it.

METHODS OF PERFORMING STRUCTURAL ANALYSIS

To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behaviour.

There are three approaches to the analysis: the mechanics of materials approach (also known as strength of materials), the elasticity theory approach (which is actually a special case of the more general field of continuum mechanics), and the finite element approach. The first two make use of analytical formulations which apply mostly to simple linear elastic models, lead to closed-form solutions, and can often be solved by hand.

The finite element approach is actually a numerical method for solving differential equations generated by theories of mechanics such as elasticity theory and strength of materials. However, the finite-element method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity.

Regardless of approach, the formulation is based on the same three fundamental relations: equilibrium, constitutive, and compatibility. The solutions are approximate when any of these relations are only approximately satisfied, or only an approximation of reality.

DESIGN PROCESS OF MASTER CYLINDER
4.1 UNIGRAPHICS INTRODUCTION
1.6 INTRODUCTION TO UNI-GRAPHICS

Overview of Solid Modeling

The Unigraphics NX Modeling application provides a solid modeling 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 Modeling feature and constraint based solid modeler. 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 modeling 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 Modeling

Solid Modeling 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 Modeling 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 modeling manufactured near-net shape parts
- User-defined features for common design elements (Unigraphics NX/User-Defined Features is required to define them in advance

3.1 General Operation

International Journal of Research

Available at https://journals.pen2print.org/index.php/ijr/

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 09
August 2019

- **Start with a Sketch**

Use the Sketcher to freehand a sketch, and dimension an "outline" of Curves. You can then sweep the sketch using Extruded Body or Revolved 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**

Modeling 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 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.

- **Additional Capabilities**

Other Unigraphics NX applications can operate directly on solid objects created within Modeling without any translation of the solid body. For example, you can perform drafting, engineering analysis, and NC machining functions by accessing the appropriate application. Using Modeling, you can design a complete, unambiguous, three dimensional model to describe an object. You can extract a wide range of physical properties from the solid bodies, including mass properties. Shading and hidden line capabilities help you visualize complex assemblies. You can identify interferences automatically, eliminating the need to attempt to do so manually. Hidden edge views can later be generated and placed on drawings. Fully associative dimensioned drawings can be created from solid models using the appropriate options of the Drafting application. If the solid model is edited later, the drawing and dimensions are updated automatically.

- **Parent/Child Relationships**

If a feature depends on another object for its existence, it is a *child* or *dependent* of that object. The object, in turn, is a *parent* of its child feature. For example, if a HOLLOW (1) is created in a BLOCK (0), the block is the parent and the hollow is its child.A parent can have more than one child, and a child can have more than one parent. A feature that is a child can also be a parent of other features. To see all of the parent-child relationships between the features in your work part, open the Part Navigator.

3.2 Creating A Solid Model

Modeling provides the design engineer with intuitive and comfortable modeling techniques such as sketching, feature based modeling, and dimension driven editing. An excellent way to begin a design concept is with a sketch. When you use a sketch, a rough idea of the part becomes represented and constrained, based on the fit and function requirements of your design. In this way, your design intent is captured. This ensures that when the design is passed down to the next level of engineering, the basic requirements are not lost when the design is edited.

The strategy you use to create and edit your model to form the desired object depends on the form and complexity of the object. You will likely use several different methods during a work session. The next several figures illustrate one example of the design process, starting with a sketch and ending with a finished model. First, you can create a sketch "outline" of curves. Then you can sweep or rotate these curves to create a complex portion of your design.

3.3 Introduction to Drafting

The Drafting application is designed to allow you to create and maintain a variety of drawings made from models generated from within the Modeling application. Drawings created in the Drafting application are fully associative to the model. Any changes made to the model are automatically reflected in the drawing. This associatively allows you to make as many model changes as you wish. Besides the powerful associatively functionality, Drafting contains many other useful features including the following:

- An intuitive, easy to use, graphical user interface. This allows you to create drawings quickly and easily.
- A drawing board paradigm in which you work "on a drawing." This approach is similar to the way a drafter would work on a drawing board. This method greatly increases productivity.
- Support of new assembly architecture and concurrent engineering. This allows the drafter to make drawings at the same time as the designer works on the model.

- The capability to create fully associative cross-sectional views with automatic hidden line rendering and crosshatching.
- Automatic orthographic view alignment. This allows you to quickly place views on a drawing, without having to consider their alignment.
- Automatic hidden line rendering of drawing views.
- The ability to edit most drafting objects (e.g., dimensions, symbols, etc.) from the graphics window. This allows you to create drafting objects and make changes to them immediately.
- On-screen feedback during the drafting process to reduce rework and editing.
- User controls for drawing updates, which enhance user productivity.
  Finally, you can add form features, such as chamfers, holes, slots, or even user defined features to complete the object.

- **Updating Models**

    A model can be updated either automatically or manually. Automatic updates are performed only on those features affected by an appropriate change (an edit operation or the creation of certain types of features). If you wish, you can delay the automatic update for edit operations by using the Delayed Update option. You can manually trigger an update of the entire model. You might, for example, want to use a net null update to check whether an existing model will successfully update in a new version of Unigraphics NX before you put a lot of additional work into modifying the model. (A net null update mechanism forces a complete update of a model, without changing it.)

The manual methods include:

- The Unigraphics NX Open C and C++ Runtime function, UF_MODL_update_all_features, which logs all the features in the current work

part to the Unigraphics NX update list, and then performs an update. See the *Unigraphics NX Open C and C++ Runtime Reference Help* for more information.

- The Playback option on the Edit Feature dialog, which recreates the model, starting at its first feature. You can step through the model as it is created one feature at a time, move forward or backward to any feature, or trigger an update that continues until a failure occurs or the model is complete.

The Edit during Update dialog, which appears when you choose Playback, also includes options for analyzing and editing features of the model as it is recreated (especially useful for fixing problems that caused update failures).Methods that users have tried in the past that has led to some problems or is tricky to use:

- One method uses the Edit Feature dialog to change the value of a parameter in each root feature of a part, and then change it back before leaving the Edit Feature dialog. This method produces a genuine net null update if used correctly, but you should ensure that you changed a parameter in every root feature (and that you returned all the parameters to their original values) before you trigger the update.

- Another method, attempting to suppress all of the features in a part and then unsuppressed them, can cause updates that are not net null and that will fail.The failures occur because not all features are suppressible; they are left in the model when you try to suppress all features. As the update advances, when it reaches the point where most features were suppressed, it will try to update the features that remain (this is like updating a modified version of the model). Some of the "modifications" may cause the remaining features to fail. For these reasons, we highly recommend that

you do not attempt to update models by suppressing all or unsurprising all features. Use the other options described here, instead.

4.2 DESIGN PROCEDURE
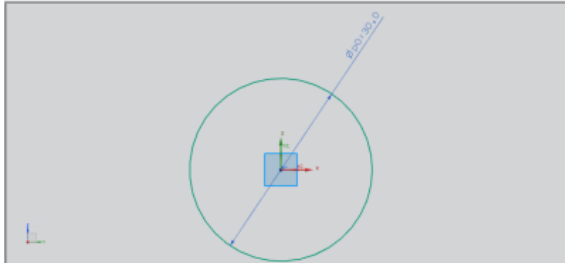
Fig: 4.1 sketch of master cylinder
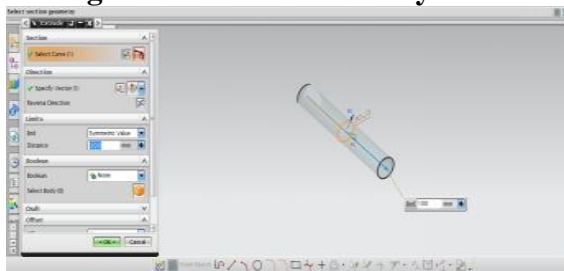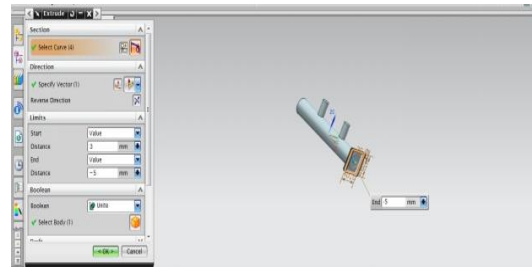

Fig: 4.2 extrusion of 2D sketch
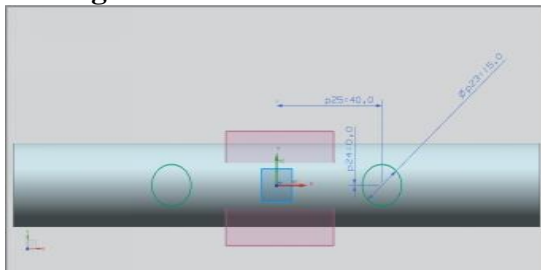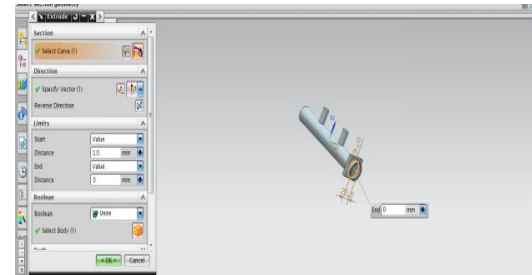

Fig: 4.3 piston sketch on the solid model
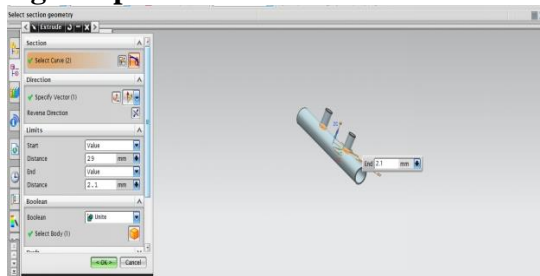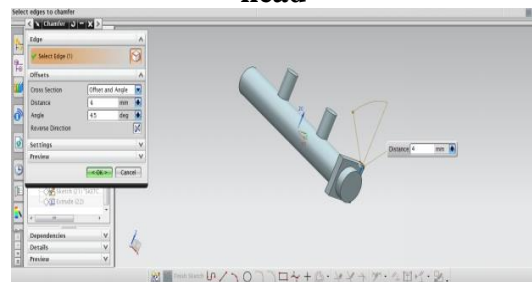

Fig: 4.4 extrusion of piston sketch on solid body
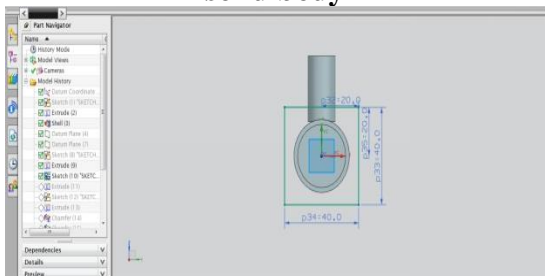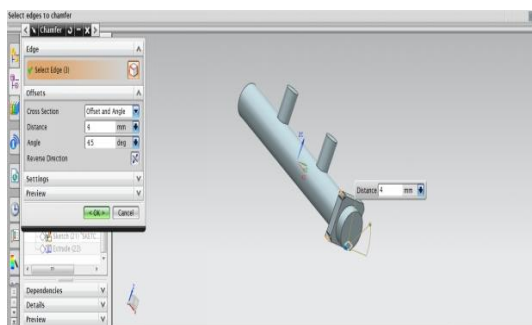

Fig: 4.5 sketch of master cylinder head
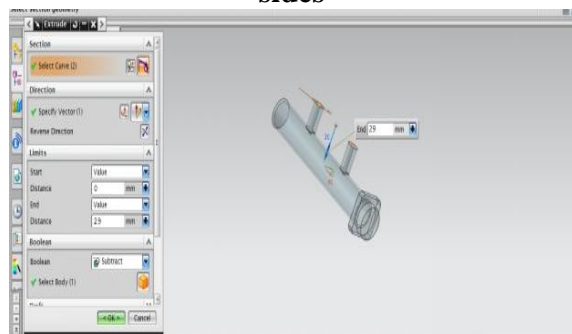

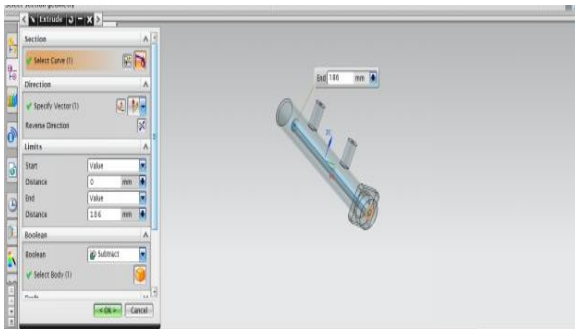**Fig: 4.5 extrusions of master cylinder head**

Fig: 4.6 making inner hallow for cylinder head

Fig: 4.7 Thicknesses for the inner part

Fig: 4.8 forming champers for corner sides

Fig: 4.9 forming champers for corner sides

Fig: 4.10 material removal from the head
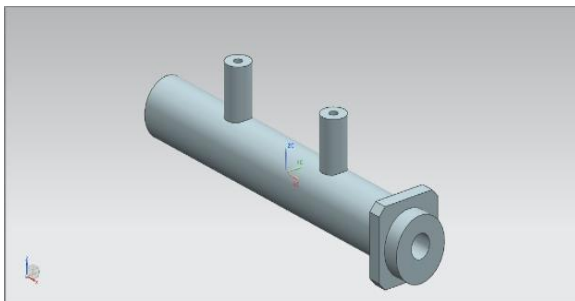
Fig: 4.11 Final model of master cylinder

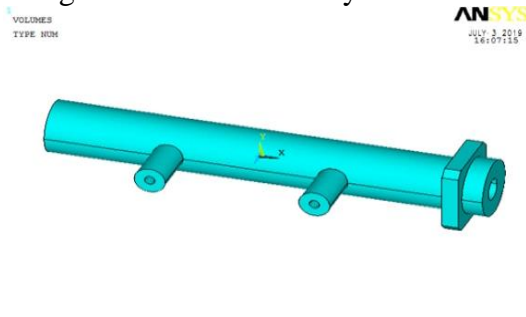**ANALYSIS OF MASTER CYLINDER**
5.1 STRUCTURAL ANALYSIS OF MAASTER CYLINDER
MATERIAL:
Aluminum A360:
Density -2.65 g/cm3
Ultimate Tensile strength -600MPa
Yield tensile strength -480MPa
Modulus of elasticity -71GPa
Poisson ratio - 0.33
Specific heat -963J/kg-K
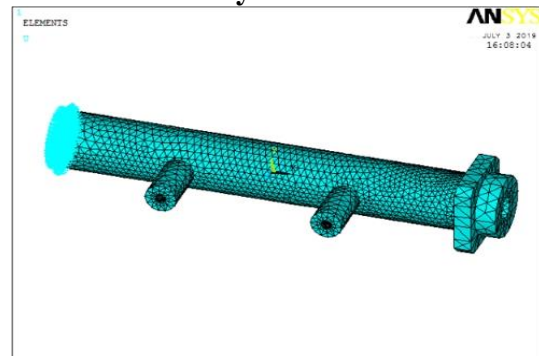Thermal conductivity -113W/m-K
LOADING CONDITION:
Internal fluid pressure 0.7056MPa applied inside of master cylinder. And bolting location is fixed at cylinder face.
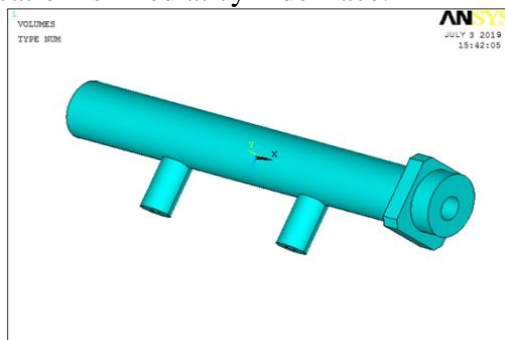
Fig: 5.1 imported model of master cylinder
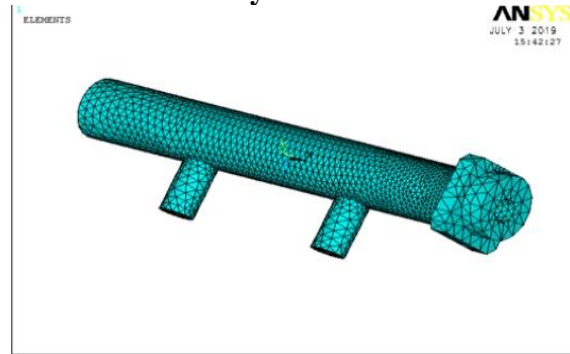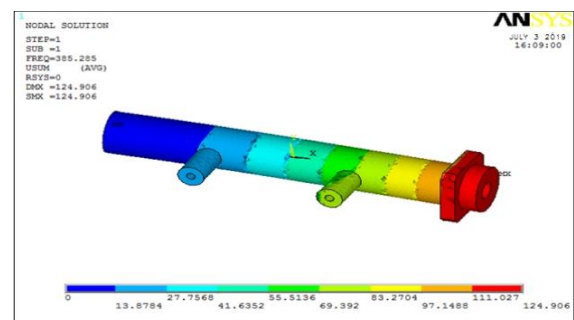

Fig: 5.2 meshing model of the master model
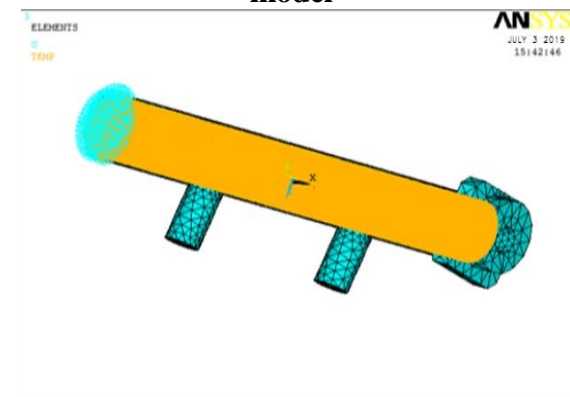
Fig: 5.3 Boundary condition of the master model
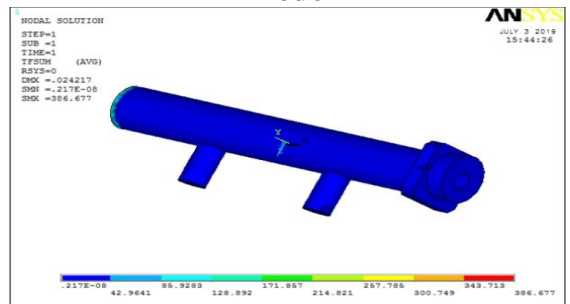
Fig: 5.6 Stress on master cylinder

5.2 MODAL ANALYSIS OF MASTER CYLINDER
Modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic analysis, or a spectrum analysis. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. You can also perform a modal

analysis on a pre-stressed structure, such as a spinning turbine blade.

If there is damping in the structure or machine component, the system becomes a damped modal analysis. For a damped modal system, the natural frequencies and mode shapes become complex.

For a rotating structure or machine component, the gyroscopic effects resulting from rotational velocities are introduced into the modal system. These effects change the system's damping. The damping can also be changed when a Bearing is present, which is a common support used for rotating structure or machine component. The evolution of the natural frequencies with the rotational velocity can be studied with the aid of Campbell Diagram Chart Results.

A Modal analysis can be performed using the ANSYS, Samcef, or ABAQUS solver. Any differences are noted in the sections below. Rotordynamic analysis is not available with the Samcef or ABAQUS solver.

MATERIAL:
Aluminum A360:
Density -2.65 g/cm3
Ultimate Tensile strength -600MPa
Yield tensile strength -480MPa
Modulus of elasticity -71GPa
Poisson ratio - 0.33
Specific heat -963J/kg-K
Thermal conductivity -113W/m-K
LOADING CONDITION:
Bolting location is fixed at cylinder face.

Fig: 5.6 Imported modal of master cylinder
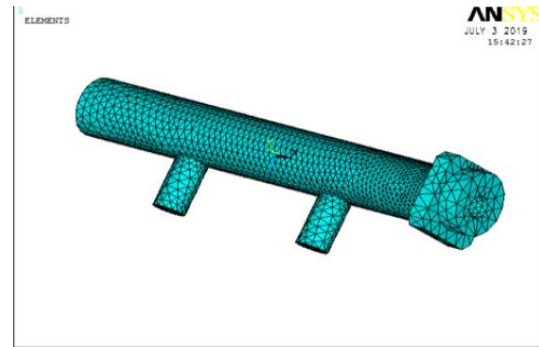
Fig: 5.6 Meshing modal of master cylinder

Fig: 5.7 Boundary conditions of master cylinder

Fig: 5.8 shows the 1st modal frequency

International Journal of Research

Available at https://journals.pen2print.org/index.php/ijr/

e-ISSN: 2348-6848
p-ISSN: 2348-795X
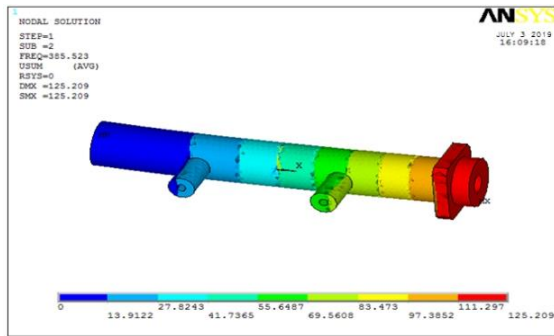Volume 06 Issue 09
August 2019

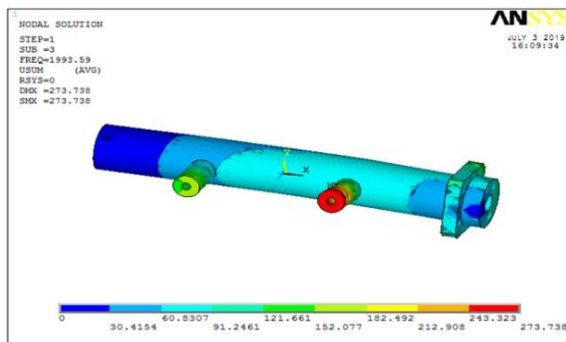**Fig: 5.9 shows the 2nd modal frequency**

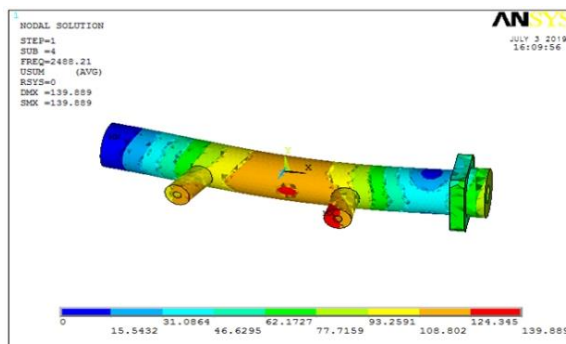**Fig: 5.10 shows the 3rd modal frequency**
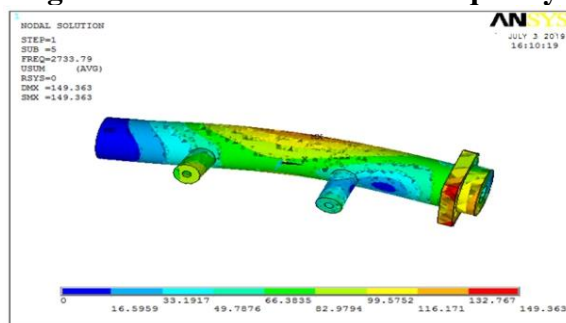

Fig: 5.11 shows the 4th modal frequency

Fig: 5.12 shows the 5th modal frequency

5.3 HARMONIC ANALYSIS OF MASTER CYLINDER
MATERIAL:
Aluminum A360:
Density -2.65 g/cm3

Ultimate Tensile strength -600MPa
Yield tensile strength -480MPa
Modulus of elasticity -71GPa
Poisson ratio - 0.33
Specific heat -963J/kg-K
Thermal conductivity -113W/m-K

LOADING CONDITION:
Internal fluid pressure 0.7056MPa applied inside of master cylinder. And bolting location is fixed at cylinder face.
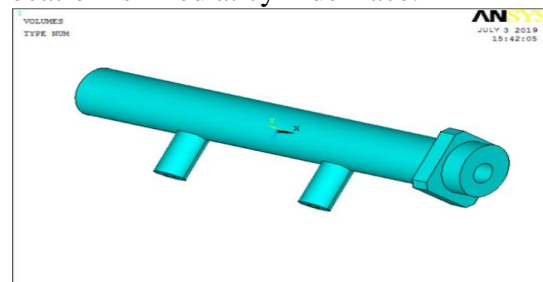
Fig: 5.13 shows the imported modal for harmonic analysis
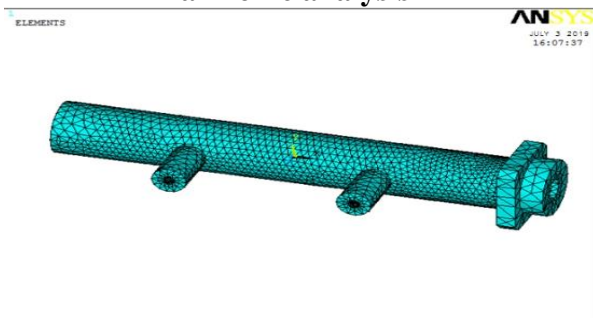
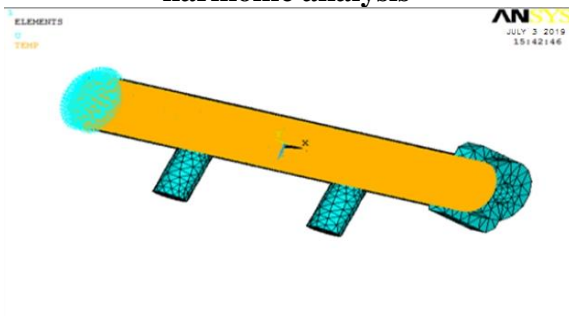**Fig: 5.14 shows the meshing modal for harmonic analysis**

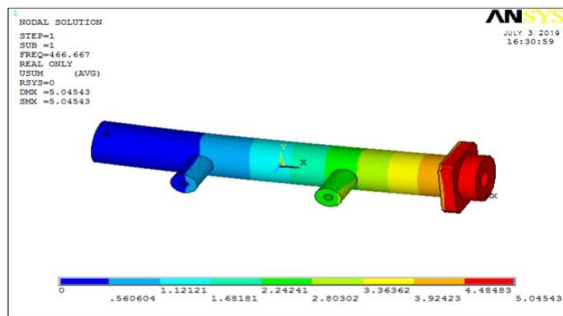

Fig: 5.15 Boundary condition of the master model

Fig: 5.18 shows the 2nd frequency stress results
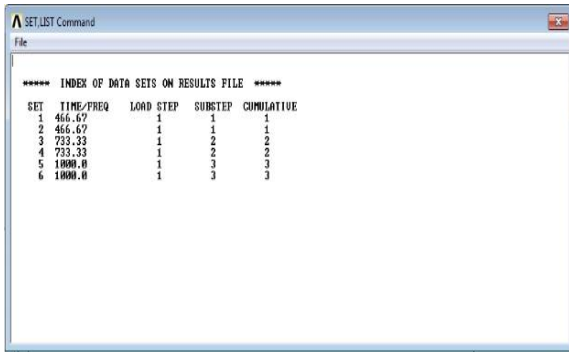
Fig: 5.16 shows the 1st frequency deformation results
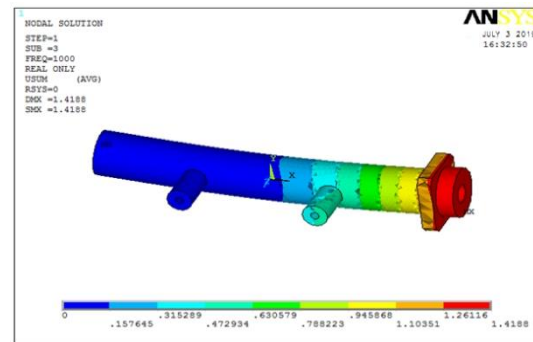
Fig: 5.19 shows the 3rd frequency deformation results

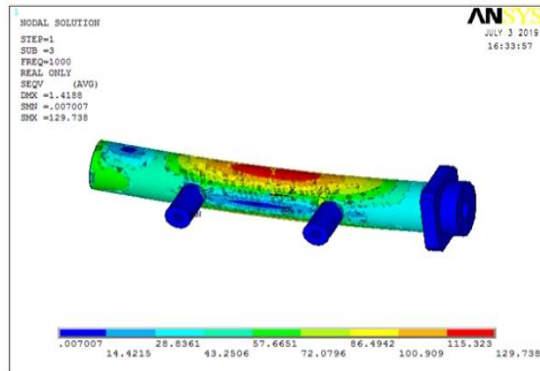**Fig: 5.16 shows the 1st frequency stress results**

Fig: 5.20 shows the 3rd frequency stress results

RESULTS AND CONCLUSION

> ➢ In this project we have designed the master cylinder using nx cad.
> ➢ Later imported into to Ansys software.
> ➢ Conducted structural, model and harmonic analysis and found different results.

THERMAL ANALYSIS RESULTS:

1	Stress (MPa)	386.677

Fig: 5.17 shows the 2nd frequency deformation results

International Journal of Research

Available at https://journals.pen2print.org/index.php/ijr/

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 09
August 2019

MODAL ANALYSIS (385HZ TO 2733Hz) DISPLACEMENT RESULTS

1	1st Frequency results	124.90
2	2nd Frequency results	125.209
3	3rd Frequency results	273.738
4	4th Frequency results	139.889
5	5th Frequency results	149.363

HARMONIC ANALYSIS (466HZ TO 1000Hz) DISPLACEMENT RESULTS

1	1st Frequency results	5.045
2	1st Frequency results	355.892
3	2nd Frequency results	1.89605
4	2nd Frequency results	116.123
5	3rd Frequency results	1.4188
6	3rd Frequency results	129.738

From analysis results observed that master cylinder formed stress 386.6MPa which is less than of yield strength of material(480MPa)and formed natural and forced frequency results less than 4000Hz. So designed master cylinder is safe under loading condition.

REFERENCES:

[1] Heinz-Heisler "Advance vehicle technology", second Edition, 2002, ISBN 07506 51318, Elsevier Butterworth- Heinemann pp. 494-496

[2] R.K.Rajput(2007), "A textbook of Automobile Engineering", First Edition, Laxmi Publication (P) Ltd., New Delhi

[3] Julian happian- smith "An introduction to modern vehicle design", First Edition, 2002, P.C. Brooks and D.C. Barton, ISBN 07506 50443, Elsevier Butterworth- Heinemann pp .157

[4] R.Chandrupatla Tirupathi and Belegundu Ashok D., "Introduction to Finite Element in Engineering", Third Edition, 2002, Prentice-Hall India Ltd.

[5] Jack Erjavec , "Automobile Technology", 5th edition, Cengage learning, Canada.

[6] Douglas M. Vidler, "Automobile Engine performance", Third edition, Delmar, United states.