COUPLED STRUCTUAL / THERMAL ANALYSIS OF DISC BRAKE

Guru Murthy Nathi[1], T N Charyulu[2], K.Gowtham[3], P Satish Reddy[4]

[1] *M.tech student, Mechanical Department, NOVA College of Engineering and technology, Andhra Pradesh, India,*
[2] *Associate Professor, Mechanical Department NOVA College of Engineering and technology, Andhra Pradesh, India,*
[3] *Assistant Professor, Mechanical Department, KIMS College of Engineering, Andhra Pradesh, India,*
[4] *Assistant Professor, Mechanical Department, PRASIDDHA College of Engineering, Andhra Pradesh, India,*
murthy408@gmail.com, tnchoney@gmail.com, gautam_universalpal@yahoo.co.in, satish2436@gmail.com

Abstract

The motive of undertaking this project of "Coupled Structural / Thermal Analysis of Disc Brake" is to study and evaluate the performance under severe braking conditions and there by assist in disc rotor design and analysis. This study is of disc brake used for cars. ANSYS package is a dedicated finite element package used for determining the temperature distribution, variation of stresses and deformation across the disc brake profile. In this present work, an attempt has been made to investigate the effect of stiffness, strength and variations in disc brake rotor design on the predicted stress and temperature distributions. By identifying the true design features, the extended service life and long term stability is assured. A transient thermal analysis has been carried out to investigate the temperature variation across the disc using axisymmetric elements. Further structural analysis is also carried out by coupling thermal analysis.

An attempt is also made to suggest a best combination of material and flange width used for disc brake rotor, which yields a low temperature variation across the rotor, less deformation, and minimum vonmises stress possible.

Index Terms: *Disc Brake, ANSYS, Thermal analysis, Structural analysis, and Transient thermal analysis*

---***---

1. INTRODUCTION

1.1 Introduction:

A brake is a device by means of which artificial frictional resistance is applied to moving machine member, in order to stop the motion of a machine.

In the process of performing this function, the brakes absorb either kinetic energy of the moving member or the potential energy given up by objects being lowered by hoists, elevators etc. The energy absorbed by brakes is dissipated in the form of heat. This heat is dissipated in to the surrounding atmosphere.

1.2 Braking requirements:

1. The brakes must be strong enough to stop the vehicle with in a minimum Distance in an emergency.
2. The driver must have proper control over the vehicle during braking and the vehicle must not skid.
3. The brakes must have good antifade characteristics i.e. their effectiveness should not decrease with constant prolonged application.
4. The brakes should have good anti wear properties.

Classification of brakes (based on transformation of energy):

- Hydraulic brakes.
- Electric brakes.
- Mechanical brakes.

The mechanical brakes according to the direction of acting force may be sub divided into the following two groups:
1. Radial brakes.
2. Axial brakes.

1.3 Radial brakes:

In these brakes the force acting on the brake drum is in radial direction. The radial brake may be subdivided into external brakes and internal brakes.

1.4 Axial brakes:

In these brakes the force acting on the brake drum is only in the axial direction.
e.g. Disc brakes, Cone brakes.

Disc brakes:

A disc brake consists of a cast iron disc bolted to the wheel hub and a stationary housing called caliper. The caliper is connected to some stationary part of the vehicle, like the axle casing or the stub axle and is cast in two parts, each part containing a piston. In between each piston and the disc, there is a friction pad held in position by retaining pins, spring plates etc. passages are drilled in the caliper for the fluid to enter or leave each housing. These passages are also connected to another one for bleeding. Each cylinder contains rubber-sealing ring between the cylinder and piston. A schematic diagram is shown in the figure.

Principle:

The principle used is the applied force (pressure) acts on the brake pads, which comes into contac with the moving disc. At this point of time due to friction the relative motion is constrained.

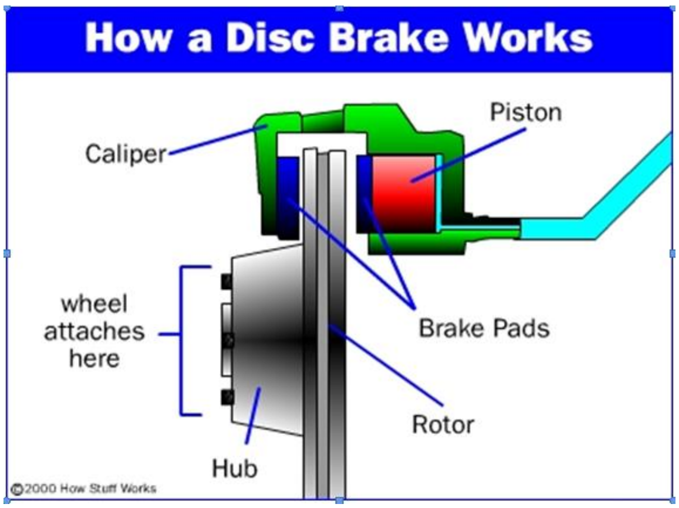


Fig -1: working principle of disc brake

Working:

When the brakes are applied, hydraulically actuated pistons move the friction pads in to contact with the disc, applying equal and opposite forces on the later. On releasing the brakes the rubber-sealing ring acts as return spring and retracts the pistons and the friction pads away from the disc.

The main components of the disc brake are:

- The Brake pads
- The caliper, which contains the piston
- The Rotor, which is mounted to the hub

Most car disc brakes are vented as shown in the below figure:

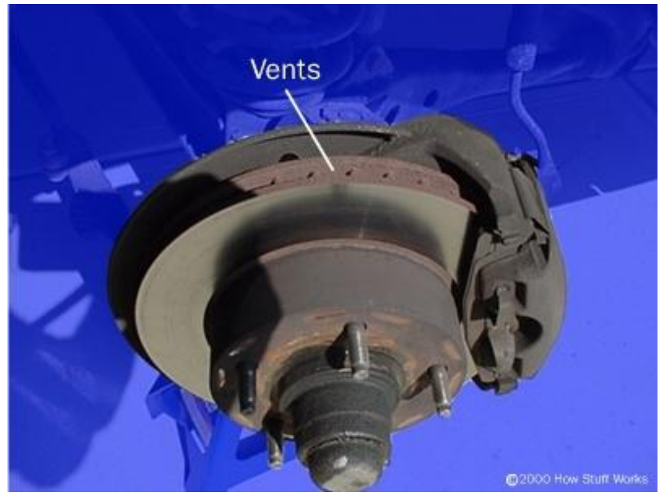


Fig -2: Vents provided on Disc Brakes

Vented disc brakes have a set of vanes, between the two sides of the disc that pumps air through the disc to provide artificial cooling.

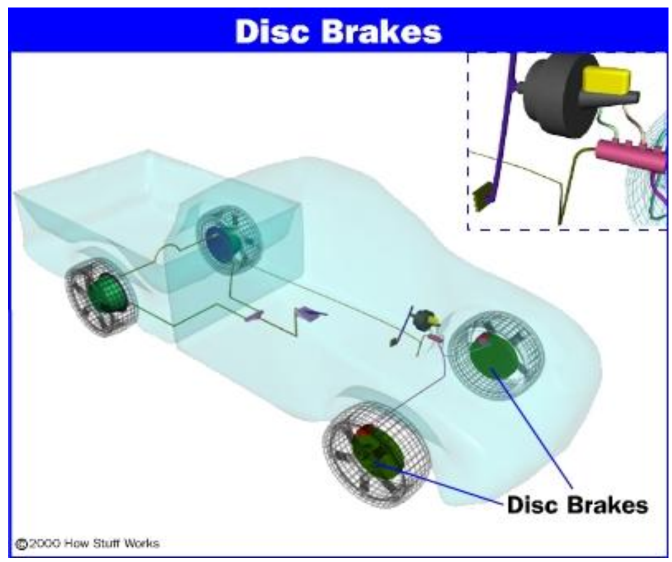


Fig -3: Location of Disc Brake in a car

Objectives of the project:

The present investigation is aimed to study:

1. The given disc brake rotor for its stability and rigidity (for this Thermal analysis and coupled structural analysis is carried out on a given disc brake rotor).
2. Best combination of parameters of disc brake rotor like Flange width and material there by a best combination is suggested. (for this three different combinations in each case is analyzed)

2. FINITE ELEMENT METHOD

2.1. Introduction to finite element method*:*

The finite element method is a powerful tool to obtain the numerical solution of wide range of engineering problem. The method is general enough to handle any complex shape or geometry, for any material under different boundary and loading conditions. The generality of the finite element method fits the analysis requirement of today's complex engineering systems and designs where closed form solutions of governing equilibrium equations are usually not available. In addition, it is an efficient design tool by which designers can perform parametric design studies by considering various design cases, (different shapes, materials, loads, etc.) and analyze them to choose the optimum design.

The method originated in the aerospace industry as a tool to study stress in a complex airframe structures. It grows out of what was called the matrix analysis method used in aircraft design. The method has gained increased popularity among both researchers and practitioners. The basic concept of finite element method is that a body or structure may be divided into small 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.

2.2. History of finite element method:

The concept of finite element method has been used several centuries back, although it has named recently. Basic ideas of the finite element method originated from the aircraft structural analysis, the concept of finite element method was first used by courant in 1943. He used the principle of stationary potential energy and piece-wise polynomial interpolation over triangular sub regions, to study the torsion problems. In 1956 Turner and Clough derived stiffness matrices for truss, beam and other elements in their presentation. The term finite element was first coined and used by Clough in 1960. Paper presented by the Clough and Turner presents the application of simple finite elements for the analysis of aircraft structure and is considered as one of the key contribution in the development of the finite element method. The digital computer provided performs many calculations rapidly involved in the finite element analysis and make the method practical viable. Along with development of high-speed digital computers, the application of the finite element method also progressed at a very impressive rate. Large general-purpose finite element computer program emerged during the late 1960s and early1970s.

In the early 1960s, engineers used the method for approximate solution of problems in stress analysis, fluid flow, heat transfer and other areas. The book by Prezemieniecki presented the finite element method as applied to the solution of stress analysis problems. Zienkiewicz and Cheung presented the broad interpretation of the method and its applicability to any general field problem. In the late 1960s and early 1970s finite element analysis was applied to non-linear problems and large deformations. In 1963, the finite element method gained popularity, when it was recognized as having a sound mathematical foundation. In 1970s study is focused on new element development and convergence studies. With this broad interpretation of the finite element method, it has been found that using a weighted residual method such as Galerkin method or least squares approach can also derive finite element equations. With all the progress, today the finite element method or least squares approach is considering as one of the well-established and convenient analysis tool by engineers and applied scientists.

2.3. General procedure of finite element method:

The finite element method is a method of piecewise approximation in which the structure or body is divided into small elements of finite dimensions called finite elements and then the original body or the structure is considered as an assemblage of these elements connected at finite number of joints called nodal points or nodes. Since the actual variation of field variables like displacement, stress, temperature, pressure or velocity inside the continuum are not known, the variation of the field variable inside a finite element can be approximated by a simple function. These approximation functions called interpolation models are defined in terms of the values of the field variables of the nodes. The nodal values of the field variable are obtained by solving the field equations, which are generally in the form of matrix equations. Once the nodal values are known, the approximating functions define the field variable throughout the assemblage of elements.

The solutions of general continuum problems by the finite element method always follow an orderly step-by-step process.

The step-by-step procedure for static structural problem can be stated as follows:

Step 1:- Description of Structure (Domain).
The first step in the finite element method is to divide the structure of solution region in to sub divisions or elements.
Step 2:- Selection of proper interpolation model.
Since the displacement (field variable) solution of a complex structure under any specified load conditions cannot be predicted exactly, we assume some suitable solution, within an element to approximate the unknown solution. The assumed solution must be simple and it should satisfy certain convergence requirements.
Step 3:- Derivation of element stiffness matrices (characteristic matrices) and load vectors.
From the assumed displacement model the stiffness matrix [K(e)] and the load vector P(e) of element 'e' are to be derived

by using either equilibrium conditions or a suitable Variation principle.

Step 4:- Assemblage of element equations to obtain the equilibrium equations.

Since the structure is composed of several finite elements, the individual element stiffness matrices and load vectors are to be assembled in a suitable manner and the overall equilibrium equation has to be formulated as

$[K]\varphi = P$

Where [K] is called assembled stiffness matrix,
Φ is called the vector of nodal displacement and
P is the vector or nodal force for the complete structure.

Step 5:- Solution of system equation to find nodal values of displacement (field variable)

The overall equilibrium equations have to be modified to account for the boundary conditions of the problem. After the incorporation of the boundary conditions, the equilibrium equations can be expressed as,

$[K]\varphi = P$

For linear problems, the vector 'φ' can be solved very easily. But for non-linear problems, the solution has to be obtained in a sequence of steps, each step involving the modification of the stiffness matrix [K] and 'φ' or the load vector P.

Step 6:- Computation of element strains and stresses.

From the known nodal displacements, if required, the element strains and stresses can be computed by using the necessary equations of solid or structural mechanics.

In the above steps, the words indicated in brackets implement the general FEM step-by-step procedure.

2.4. Convergence requirement:

The finite element method provides a numerical solution to a complex problem. It may therefore be expected that the solution must converge to the exact formulation of the structure. Hence as the mesh is made finer the solution should converge to the correct result and this would be achieved if the following three conditions were satisfied by the assumed displacement function.

- The displacement function must be continuous within the element. Choosing polynomials for the displacement model can easily satisfy this condition.
- The displacement function must be capable of representing rigid body displacement of the element. This is when the nodes are given such displacement corresponding to a rigid body motion; the element should not experience and hence leads to zero nodal forces. The constant terms in the polynomials used for displacement models would usually ensure this condition.
- The displacement function must be capable of representing constant strain states within the element. The reason for the requirement can be understood if we imagine the condition when the body or structure is divided in to smaller and smaller elements. As these elements approach infinitesimal size the strain in each element also approach constant strain states. For one, two and three-dimensional elasticity problems the linear terms present in the polynomials satisfy the requirement. However, in constant curvature instead of constant strains.

2.5. Advantages of fem:

The properties of each element are evaluated separately, so an obvious advantage is that we can incorporate different material properties for each element. Thus almost any degree of non-homogeneity can be included. There is no restriction on to the shape of medium; hence arbitrary and irregular shapes cause no difficulty like all numerical approximations FEM is based on the concept of description. Nevertheless as either the variations or residual approach, the technology recognizes the multidimensional continuous but also requires no separate interpolation process to extend the approximate solution to every point with the continuum.

One of the important advantages of FEM is that it makes use of boundary conditions in the form of assembled equations. This is relatively an easy process and requires no special technology. Rather than requiring every trial solution to satisfy boundary conditions, one prescribes the conditions after obtaining the algebraic equations for individual's finite elements.

2.6. Limitations in fem:

FEM reached high level of development as solution technology; however the method yields realistic results only if coefficient or material parameters that describe basic phenomena are available.

The most tedious aspects of use of FEM are basic process of sub-dividing the continuum of generating error free input data for computer.

2.7. Applications of fem:

The finite element method was developed originally for the analysis of aircraft structures. However, the general nature of its theory makes it applicable to wide variety of boundary value problem in engineering. A boundary value problem is one in which a solution is sought in domain or region of a body subject to the satisfaction of prescribed boundary conditions. Finite element method is the best tool in investigation of aircraft structures involving static analysis of wings, structures of rockets and missiles, dynamic analysis, response to random loads and periodic loads. In mechanical design, stress concentration problems, stress analysis of pressure vessels, dynamic analysis of mechanical linkages can be effectively dealt using finite element method.

The specific application of the finite element method in the three major categories of boundary value problems, namely equilibrium of steady state or time independent problems, Eigen value problems, and propagation or transient problems. In the equilibrium problems steady state displacement or stress distribution is found for a solid mechanics problem, temperature or heat flux distribution in the case of heat transfer problem. Referring to Eigen value problems in solid mechanics or structural problem, natural frequencies, buckling loads and mode shapes are found, stability of laminar flows is found if it is a fluid mechanics problem and resonance characteristics are obtained if it is an electrical circuit problem, while for the propagation or transient problem, the response of the body under time varying force is found in the area of solid mechanics.

Finite element method finds its application in the field of civil engineering in carrying out the static analysis of trusses, frames and bridges. The dynamic analysis of the structure is to obtain natural frequencies, modes and response of the structures to periodic loads. Nuclear engineering also uses finite element method concept in the static and dynamic characterization of its systems such as nuclear pressure vessels, containment structure and dynamic response of reactor component containment structures. Even the Bio-medical engineering applies finite element method, for impact analysis of skulls. Finite element method can be applied to analysis of excavation, underground openings and dynamic analysis of dam reservoir systems, which come under Geo-mechanics.

3. FEA SOFTWARE – ANSYS

3.1 Introduction to ANSYS Program:

Dr. John Swanson founded ANSYS. Inc in 1970 with a vision to commercialize the concept of computer simulated engineering, establishing himself as one of the pioneers of Finite Element Analysis (FEA). ANSYS inc. supports the ongoing development of innovative technology and delivers flexible, enterprise wide engineering systems that enable companies to solve the full range of analysis problem, maximizing their existing investments in software and hardware. ANSYS Inc. continues its role as a technical innovator. It also supports a process-centric approach to design and manufacturing, allowing the users to avoid expensive and time-consuming “built and break” cycles. ANSYS analysis and simulation tools give customers ease-of-use, data compatibility, multi platform support and coupled field multi-physics capabilities.

3.2 Evolution of ANSYS Program:

ANSYS has evolved into multipurpose design analysis software program, recognized around the world for its many capabilities. Today the program is extremely powerful and easy to use. Each release hosts new and enhanced capabilities that make the program more flexible, more usable and faster. In this way ANSYS helps engineers meet the pressures and demands modern product development environment.

3.3 Overview of the program:

The ANSYS program is flexible, robust design analysis and optimization package. The software operates on major computers and operating systems, from PCs to workstations and to super computers. ANSYS features file compatibility throughout the family of products and across all platforms. ANSYS design data access enables user to import computer aided design models in to ANSYS, eliminating repeated work. This ensures enterprise wide, flexible engineering solution for all ANSYS user.

User Interface:

Although the ANSYS program has extensive and complex capabilities, its organization and user-friendly graphical user interface makes it easy to learn and use.

There are four graphical methods to instruct the ANSYS program:

1. Menus.
2. Dialog Boxes
3. Tool bar.
4. Direct input of commands.

Menus:

Menus are groupings of related functions or operating the analysis program located in individual windows. These include:

- Utility menu
- Main menu
- Input window
- Graphics window
- Tool bar
- Dialog boxes

Dialog boxes:

Windows that present the users with choices for completing the operations or specifying settings. These boxes prompt the user to input data or make decisions for a particular function.

Tool bar:

The tool bar represents a very efficient means for executing commands for the ANSYS program because of its wide range of configurability. Regardless of how they are specified, commands are ultimately used to supply all the data and control all program functions.

Output window:

Records the ANSYS response to commands and functions

Graphics window:

Represents the area for graphic displays such as model or graphically represented results of an analysis. The user can adjust the size of the graphics window, reducing or enlarging it to fit to personal preferences.

Input window:

Provides an input area for typing ANSYS commands and displays program prompt messages.

Main menu:

Comprise the primary ANSYS functions, which are organized in pop-up side menus, based on the progression of the program.

Utility menu*:*

Contains ANSYS utility functions that are mapped here for access at any time during an ANSYS session. These functions are executed through smooth, cascading pull down menus that lead directly to an action or dialog box.

Processors:

ANSYS functions are organized into two groups called processors. The ANSYS program has one pre-processor, one solution processor; two post processors and several auxiliary processors such as the design optimizer. The ANSYS pre-processor allows the user to create a finite element model to specify options needed for a subsequent solution. The solution processor is used to apply the loads and the boundary conditions and then determine the response of the model to them. With the ANSYS post processors, the user retrieves and examines the solutions results to evaluate how the model responded and to perform additional calculations of interest.

Database:

The ANSYS program uses a single, centralized database for all model data and solution results. Model data (including solid model and finite element model geometry, materials etc) are written to the database using the processor. Loads and solution results data are written using the solutions processor. Post processing results data are written using the post processors. Data written to the database while using one processor are therefore available as necessary in the other processors.

File format:

Files are used, when necessary, to pass the data from part of the program to another, to store the program to the database, and to store the program output. These files include database files, the results file, and the graphics file and so on.

3.4 Reducing the design and manufacturing costs using ANSYS (FEA):

The ANSYS program allows engineers to construct computer models or transfer CAD models of structures, products, components, or systems, apply loads or other design performance conditions and study physical responses such as stress levels, temperature distribution or the impact of vector magnetic fields.

In some environments, prototype testing is undesirable or impossible. The ANSYS program has been used in several cases of this type including biomechanical applications such as hi replacement intraocular lenses. Other representative applications range from heavy equipment components, to an integrated circuit chip, to the bit-holding system of a continuous coal-mining machine.

ANSYS design optimization enables the engineers to reduce the number of costly prototypes, tailor rigidity and flexibility to meet objectives and find the proper balancing geometric modifications.

Competitive companies loom for ways to produce the highest quality product at the lowest cost. ANSYS (FEA) can help significantly by reducing the design and manufacturing costs and by giving engineers added confidence in the products they design. FEA is most effective when used at the conceptual design stage. It is also useful when used later in manufacturing process to verify the final design before prototyping.

Program availability:

The ANSYS program operates on Pentium based PCs running on Wndows95 or Windows NT and workstations and super computers primarily running on UNIX operating system. ANSYS Inc. continually works with new hardware platforms and operating systems.

Analysis types available:

1. Structural static analysis.
2. Structural dynamic analysis.
3. Structural buckling analysis.
    - ➢ Linear buckling
    - ➢ Non linear buckling
4. Structural non linearities.
5. Static and dynamic kinematics analysis.
6. Thermal analysis.
7. Electromagnetic field analysis.
8. Electric field analysis
9. Fluid flow analysis
    - ➢ Computational fluid dynamics

- Pipe flow
10. Coupled-field analysis
11. Piezoelectric analysis.

3.5 Procedure for ANSYS analysis:

Static analysis is used to determine the displacements, stresses, strains and forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds of loading that can be applied in a static analysis include externally applied forces and pressures, steady state inertial forces such as gravity or rotational velocity imposed (non-zero) displacements, temperatures (for thermal strain).
A static analysis can be either linear or non linear. In our present work we consider linear static analysis.

The procedure for static analysis consists of these main steps:

1. Building the model.
2. Obtaining the solution.
3. Reviewing the results.

3.6. Build the model:

In this step we specify the job name and analysis title use PREP7 to define the element types, element real constants, material properties and model geometry element types both linear and non-linear structural elements are allowed. The ANSYS element library contains over 80 different element types. A unique number and prefix identify each element type.
E.g. BEAM 3, PLANE 55, SOLID 45 and PIPE 16

3.7. Material properties:

Young's modulus(EX) must be defined for a static analysis .If we plan to apply inertia loads(such as gravity) we define mass properties such as density(DENS).Similarly if we plan to apply thermal loads (temperatures) we define coefficient of thermal expansion(ALPX).

3.8 Obtain the solution:

In this step we define the analysis type and options, apply loads and initiate the finite element solution. This involves three phases:

- Pre – processor phase
- Solution phase
- Post-processor phase

3.8.1. Pre – Processor:

Pre processor has been developed so that the same program is available on micro, mini, super-mini and mainframe computer system. This slows easy transfer of models one system to other.

Pre processor is an interactive model builder to prepare the FE (finite element) model and input data. The solution phase utilizes the input data developed by the pre processor, and prepares the solution according to the problem definition. It creates input files to the temperature etc., on the screen in the form of contours.

3.8.1.1. Geometrical definitions:

There are four different geometric entities in pre processor namely key points, lines, areas and volumes. These entities can be used to obtain the geometric representation of the structure. All the entities are independent of other and have unique identification labels.

3.8.1.2. Model generations:

Two different methods are used to generate a model:

- Direct generation.
- Solid modeling

With solid modeling we can describe the geometric boundaries of the model, establish controls over the size and desired shape of the elements and then instruct ANSYS program to generate all the nodes and elements automatically. By contrast, with the direct generation method, we determine the location of every node and size, shape and connectivity of every element prior to defining these entities in the ANSYS model. Although, some automatic data generation is possible (by using commands such as FILL, NGEN, EGEN etc) the direct generation method essentially a hands on numerical method that requires us to keep track of all the node numbers as we develop the finite element mesh. This detailed book keeping can become difficult for large models, giving scope for modeling errors. Solid modeling is usually more powerful and versatile than direct generation and is commonly preferred method of generating a model.

3.8.1.3. Mesh generation:

In the finite element analysis the basic concept is to analyze the structure, which is an assemblage of discrete pieces called elements, which are connected, together at a finite number of points called Nodes. Loading boundary conditions are then applied to these elements and nodes. A network of these elements is known as Mesh.

3.8.1.4. Finite element generation:

The maximum amount of time in a finite element analysis is spent on generating elements and nodal data. Pre processor allows the user to generate nodes and elements automatically at the same time allowing control over size and number of elements. There are various types of elements that can be mapped or generated on various geometric entities.

The elements developed by various automatic element generation capabilities of pre processor can be checked element characteristics that may need to be verified before the finite element analysis for connectivity, distortion-index, etc. Generally, automatic mesh generating capabilities of pre processor are used rather than defining the nodes individually. If required, nodes can be defined easily by defining the allocations or by translating the existing nodes. Also one can plot, delete, or search nodes.

3.8.1.5. Boundary conditions and loading:

After completion of the finite element model it has to constrain and load has to be applied to the model. User can define constraints and loads in various ways. All constraints and loads are assigned set 1D. This helps the user to keep track of load cases.

3.8.1.6. Model display:

During the construction and verification stages of the model it may be necessary to view it from different angles. It is useful to rotate the model with respect to the global system and view it from different angles. Pre processor offers this capability. By windowing feature pre processor allows the user to enlarge a specific area of the model for clarity and details. Pre processor also provides features like smoothness, scaling, regions, active set, etc for efficient model viewing and editing.

3.8.1.7. Material definitions*:*

All elements are defined by nodes, which have only their location defined. In the case of plate and shell elements there is no indication of thickness. This thickness can be given as element property. Property tables for a particular property set 1-D have to be input. Different types of elements have different properties for e.g.

Beams: Cross sectional area, moment of inertia etc
Shells: Thickness
Springs: Stiffness
Solids: None

The user also needs to define material properties of the elements. For linear static analysis, modules of elasticity and Poisson's ratio need to be provided. For heat transfer, coefficient of thermal expansion, densities etc are required. They can be given to the elements by the material property set to 1-D.

3.8.2. Solution:

The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer and finally displacements and stress values are given as output. Some of the capabilities of the ANSYS are linear static analysis, non-linear static analysis, transient dynamic analysis, etc.

3.8.3. Post – Processor:

It is a powerful user-friendly post-processing program using interactive colour graphics. It has extensive plotting features for displaying the results obtained from the finite element analysis. One picture of the analysis results (i.e. the results in a visual form) can often reveal in seconds what would take an engineer hour to asses from a numerical output, say in tabular form. The engineer may also see the important aspects of the results that could be easily missed in a stack of numerical data. Employing state of art image enhancement techniques, facilities viewing of:

- Contours of stresses, displacements, temperatures, etc.
- Deform geometric plots
- Animated deformed shapes
- Time-history plots
- Solid sectioning
- Hidden line plot
- Light source shaded plot
- Boundary line plot etc.

The entire range of post processing options of different types of analysis can be accessed through the command/ menu mode there by giving the user added flexibility and convenience.

4. DISC BRAKE CALCULATIONS:

4.1 Disc Brake Calculations:

Given Data:
Velocity of the vehicle = 70 m.p.h = 112.65408 kmph = 31.2928 m/s
Time for stopping the vehicle = 4 seconds
Mass of the vehicle = 1400 kg.

Step-1:
Kinetic Energy (K.E) = ½ * m * υ2
= ½ * 1400 * 31.29282
= 685467.5323 Joules

The above said is the Total Kinetic Energy induced while the vehicle is under motion.

Step-2:
The total kinetic energy = The heat generated
Qg = 685467.5323 Joules

Step-3:
The area of the rubbing faces
A = Π * (0.1802 – 0.10362)
= 0.068069 m2

Step-4:
Heat Flux = Heat Generated / Second / rubbing area
= 685467.5323 / 4 /0.068069
= 2517546.652 Watts / m2

The analysis is done by taking the distribution of braking torque between the front and rear axle is 70:30

Thus

Heat Flux on each front wheel = (2517546.652 * 0.7)/2

=881141.3283 Watts / m2

For 5 Seconds of Breaking:

Heat Flux = Heat Generated / Second / rubbing area

= 685467.5323 / 5 /0.068069

= 2014037.322 Watts / m2

The analysis is done by taking the distribution of braking torque between the front and rear axle is 70:30

Thus

Heat Flux on each front wheel = (2014037.322 * 0.7)/2

=704913.0626 Watts / m2

For 6 Seconds of Breaking:

Heat Flux = Heat Generated / Second / rubbing area

= 685467.5323 / 6 /0.068069

= 1678364.435 Watts / m2

The analysis is done by taking the distribution of braking torque between the front and rear axle is 70:30

Thus

Heat Flux on each front wheel = (1678364.435 * 0.7)/2

=587427.5522 Watts / m2

4.2 Material properties:

4.2.1 The Material Properties of Cast Iron:

Thermal co-efficient of expansion (Kxx) = 1.7039e-5 /°C
Thermal conductivity (K = 54.0 W / m k
Specific heat (Cp) = 586.0 J/Kg k
Density of cast iron (ρ) = 7100 kg/m3
Young's Modulus (E) = 125e9 N/m2
Poisson's ratio (υ) = 0.25

4.2.2 The Material Properties of Stainless Steel:

(Stainless Steel 302 Annealed)

Density of Stainless Steel (ρ) = 7860 kg / m3
Thermal conductivity (k) = 16.2 Watts / m k
Specific heat (Cp) = 500 J / kg k
Young's Modulus (E) = 193e9 N / m2
Poisson's ratio (ν) = 0.29
Coefficient of Thermal Expansion (Kxx) = 1.72e-5 m/m °C

4.2.3 The Material Properties of Aluminum:

(Aluminum 2014-T6)

Density of Aluminum (ρ) = 2800 kg / m3
Thermal conductivity (k) = 155 Watts / m k
Specific heat (Cp) = 880 J / kg k
Young's Modulus (E) = 72.4e9 N / m2
Poisson's ratio (ν) = 0.33
Coefficient of Thermal Expansion (Kxx) = 2.3e-5 m/m °C

4.3 Assumptions:

1. The analysis is done taking the distribution of the braking torque between the front and rear axle is 70:30
2. Brakes are applied on all the four wheels.
3. The analysis is based on pure thermal loading .The analysis does not determine the life of the disc brake.
4. Only ambient air-cooling is taken in to account and no forced convection is taken.
5. The kinetic energy of the vehicle is lost through the brake discs i.e. no heat loss between the tyres and the road surface and the deceleration is uniform.
6. The disc brake model used is of solid type and not the ventilated one.
7. The thermal conductivity of the material used for the analysis is uniform throughout.
8. The specific heat of the material used is constant throughout and does not change with the temperature.
9. Heat flux on each front wheel is applied on one side of the disc only.
10. Displacement in axial direction on flange is constrained in one side of the disc.

4.4 Different cases of analysis:

In our present study the following different cases are considered:

Case I: Coupled Structural/Thermal Analysis is carried out for 4 seconds of braking on a cast iron disc with 10mm flange thickness.

Case II: Coupled Structural/Thermal Analysis is carried out for 4,5,6 seconds of braking on a cast iron disc with 10mm flange thickness and results are compared.

Case III: Coupled Structural/Thermal Analysis is carried out for 4seconds of braking, using 10mm flange width for different materials (cast iron, steel, aluminum) and results are compared.

Case IV: Coupled Structural/Thermal Analysis is carried out for 4seconds of braking on cast iron disc with different flange thickness (8mm, 10mm, 12mm) and results are compared.

5. THERMAL ANALYSIS

5.1 Introduction:

A Thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities are:

1. The temperature distributions
2. The amount of heat lost or gained
3. Thermal fluxes

Types of Thermal Analysis:

1. A Steady State Thermal Analysis determines the temperature distribution and other thermal quantities under steady state loading conditions. A steady state loading condition is a situation where heat storage effects varying over a period of time can be ignored.
2. A Transient thermal analysis determines the temperature distribution and other thermal quantities under conditions that vary over a period of time

5.2 Definition of problem:

Due to the application of brakes on the car disc brake rotor, heat generation takes place due to friction and this temperature so generated has to be conducted and dispersed across the disc rotor cross section. The condition of braking is very much severe and thus the thermal analysis has to be carried out.

The thermal loading as well as structure is axis-symmetric. Hence axis-symmetric analysis is performed which is an exact representation for this thermal analysis.

Linear thermal analysis is performed to obtain the temperature field since conductivity and specific heat of the material considered here are independent of temperature. The analysis performed here is transient thermal analysis as temperature distribution varies with time. (The time for thermal analysis is taken as 4, 5 and 6 seconds of braking)

5.3 Element considered for thermal analysis and its description:

According to the given specifications the element type chosen is PLANE 55.It can be used as a plane element or as an axisymmetric ring element with a two-dimensional thermal conduction capability. The element has four nodes with a single degree of freedom, temperature, at each node. The element is applicable to a two-dimensional, steady-state or transient thermal analysis. The element can also compensate for mass transport heat flow from a constant velocity field. If the model containing the temperature element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as PLANE42). The following Figure shows the schematic diagram of the 4-noded thermal solid element.

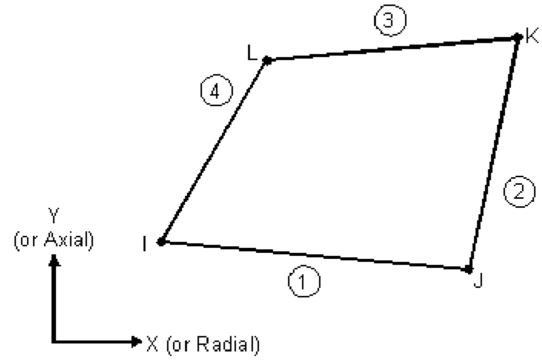


Fig -4: Thermal Solid

5.4 Mesh Generation:

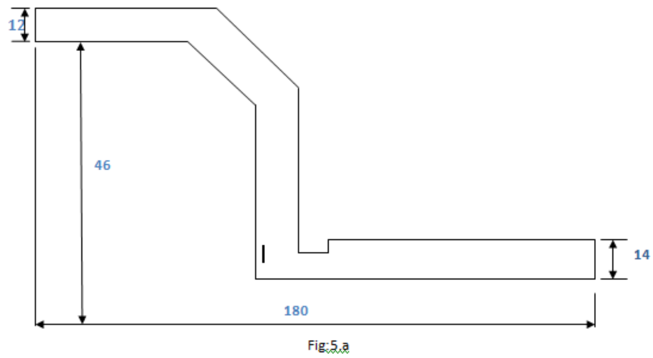
Before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes and has no specified pattern applied to it.

Compared to the free mesh, a mapped mesh is restricted in terms of the element shape it contains and pattern of the mesh. A mapped mesh contains either only quadrilateral or only triangular element, while a mapped volume mesh contains only hexahedral elements. In addition, a mapped mesh typically has a regular pattern, with obvious rows of elements.

For mapped mesh, we must build the geometry as a series of fairly regular volumes and/or areas that can accept a mapped mesh. The type of mesh generation considered here is a mapped mesh.

5.5 Thermal Loads:

Heat flux (q) = 881141.3283 W/ m2
(for 4 seconds of braking)
Convection film co-efficient (h) = 5.0 W/ m2 k
The temperature fixed at the hub bore grinds = 35°C



Flange Width = 14 mm

Disc Radius = 180 mm

Thickness = 12 mm

Height = 46 mm

Fig 5. 1 Dimensions of Disk Brake

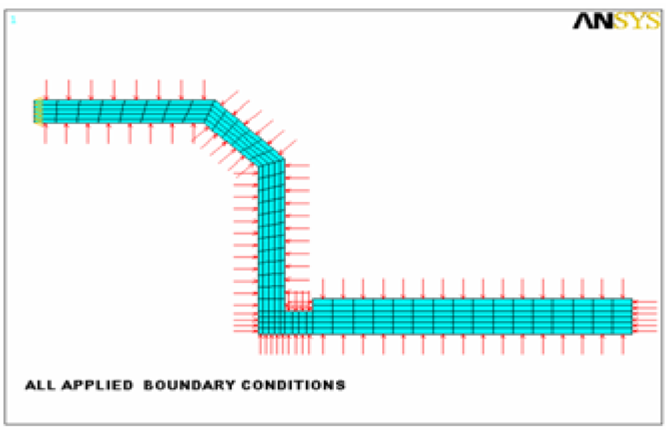


Fig 5. 2 All applied boundary conditions

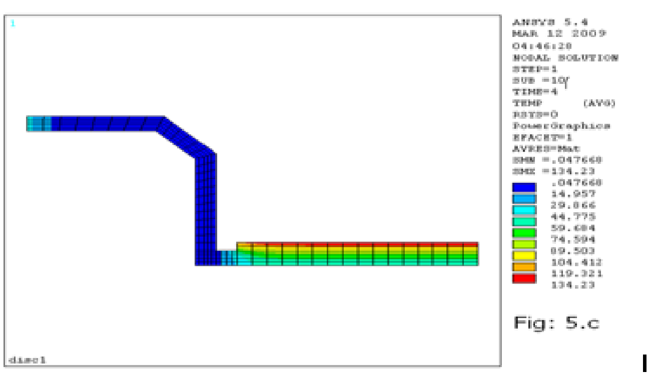


Fig 5. 3 Temp distribution of CAST IRON at 4 seconds(10 mm thick)

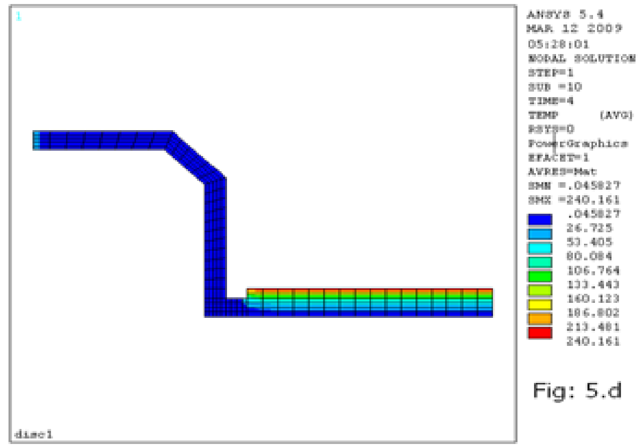


Fig 5. 4 Temp distribution of STEEL at 4 seconds(10 mm thick)

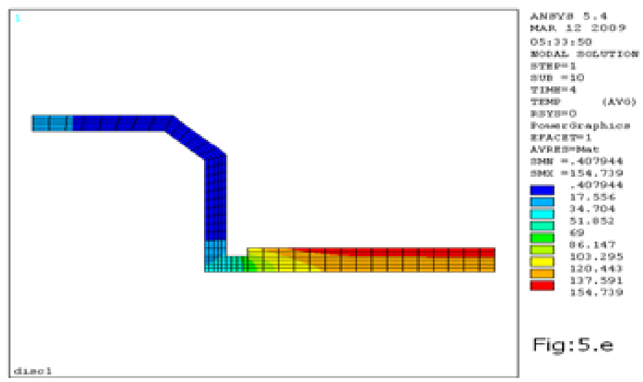


Fig 5.5 Temp distribution of ALUMINUM at 4 seconds (10 mm thick)

6. STRUCTURAL ANALYSIS

6.1 Introduction:

Structural analysis is the most common application of the finite element method. The term structural (or structure) implies civil engineering structures such as bridges and buildings, but also naval, aeronautical and mechanical structures such as ship hulls, aircraft bodies and machines housings as well as mechanical components such as pistons, machine parts and tools.

Types of structural analysis:

There are seven types of structural analyses available in ANSYS. One can perform the following types of structural analyses.

1. Static analysis
2. Modal analysis
3. Harmonic analysis
4. Transient dynamic analysis

5. Spectrum analysis
6. Buckling analysis
7. Explicit dynamic analysis

6.2 Definition of the problem:

Due to the application of brakes on the car disc brake rotor heat generation takes place due to friction and this temperature so generated has to be conducted away and dispersed across the disc brake cross section. The condition of braking is very severe and thus thermal analysis is carried out and with the above load structural analysis is also performed for analyzing the stability of the structure.

The basic analysis equation is

[K] [Q] = F

Where K = global stiffness matrix.

F = Load vector

Q = Displacement vector

6.3 Boundary conditions:

Geometric Boundary conditions:

All the nodes in the hub radius are fixed. So the nodal displacements in the hub bore become Zero i.e. both in radial and axial direction and nodes on flange are also fixed in Y-direction.

6.4 Results and discussion:

CASE 1:

The following table illustrates the deflection and thermal stresses for 4 seconds of braking:

Table 6.1: deflection and thermal stresses for 4 seconds of braking

Deflections	Deflection in X Direction U_x	0.120e-03
Thermal Stresses	Principle Stress 1 Principle stress 2 Stress in X direction S_x Stress in Y direction S_y Von Mises Stress	0.585e+08 0.140e+08 0.585e+08 0.172e+08 0.115e+09
Temperature	All dof	134.23

From the table we can observe that the maximum deflection induced is 0.12 mm which is less than the allowable deflection i.e. 0.5 mm. Hence the design is safe based on the rigidity.

The stresses in both x and y directions respectively with magnitude of 58.5 Mpa and 17.2 Mpa and variation of principle stresses along the both principal direction respectively with magnitude of 58.5 Mpa and 14 Mpa. The variation of von mises stresses with magnitude of 115 Mpa.

The maximum induced von mises stress is far below the working stress i.e. 375 M pa according to the manufacturer's specifications.

Thus the brake disc parameters that constitute the design are in turn safe based on the strength and rigidity.

CASE 2:

The following table illustrates the deflection and temperature distribution for 4,5,6 seconds of braking on a cast iron disc brake with 10 mm flange width.

Table 6.2: deflection and temperature distribution for 4,5,6 seconds of braking on a cast iron disc brake with 10 mm flange width

Time of braking	Deflection in X Direction U_x	Temperature
4 sec	0.120e-03	134.23
5 sec	0.120e-03	123.644
6 sec	0.120e-03	116.496

From the table we can observe that the temperature distribution decreases as the time of braking increasing.

CASE 3:

Now for establishing a best material for the car disc brake rotor for the present application ,three different materials which are commonly used for the disc brakes namely cast iron(CI),stainless steel 302 annealed (S.S),alluminium-2014-T6(AL) were taken and analysis is done using their properties on the disc brake rotor with 6.5mm wall thickness and 10mm flange width.

The comparison of different parameters for the three materials used for the disc brake under consideration in the below table 6.3

Table 6.3: Comparisions between disc brakes with different materials keeping other conditions same for 4 seconds braking

PARAMETER	CAST IRON	STAINLESS STEEL	ALUMINIUM
Temperature distribution(◦C)	134.23	240.161	154.739
Deformation(DMX)meters	0.120e-03	0.128e-03	0.273e-03
Stress in X-direction (M.pa)	0.585e+08	0.224e+09	0.160e+08
Stress in Y –direction (M.pa)	0.172e+08	0.597e+08	0.133e+08
1st principal stress(M.pa)	0.585e+08	0.224e+09	0.160e+08
2nd principal stress(M.pa)	0.140e+08	0.579e+08	0.564e+07
Von mises stress(M.pa)	0.115e+09	0.518e+09	0.602e+08

Discussion from the table:

Comparing the all together we may suggest that cast iron as the best material for the present case which has temperature distribution of 134.23 and deformation of 0.120e-03 and a von mises stress of 115 M Pa.

CASE 4:

Car disc brake rotor with different flange widths is analyzed using the three different car disc brake rotor axis-symmetric models. Here the aim is to establish a best flange width for the present application.

A best flange width is the one using which we get moderate temperature distribution, less deformation and the von mises stress as minimum as possible .

In the present case three different flange widths applied to car disc brake rotor models were analyzed. They are disc brake rotor with 8mm flange width, 10mm flange width and 12mm flange width.

The comparison of different parameters for the three flange widths in consideration is shown in below table 6.2

Table 6.4 The comparison of different parameters for the three flange widths in consideration

PARAMETER	8mm flange width	10mm flange width	12mm flange width
Temperature distribution(°C)	143.818	134.23	130.231
Deformation(DMX)meters	0.147e-03	0.120e-03	0.101e-03
Stress in X-direction (M.pa)	0.474e+08	0.585e+08	0.679e+08
Stress in Y –direction (M.pa)	0.172e+08	0.172e+08	0.172e+08
1st principal stress(M.pa)	0.474e+08	0.585e+08	0.679e+08
2nd principal stress(M.pa)	0.108e+08	0.140e+08	0.162e+08
Von mises stress(M.pa)	0.930e+08	0.115e+09	0.135e+09

Now observing the above table 6.2, it can be seen that temperature distribution is decreasing from 143.818 for 8mm flange width to 130.231 for 12mm flange width. The deformation is decreasing as the flange width is increasing and the von mises stress is increasing as the flange width is increasing.

Now comparing all these three cases we may conclude that the disc brake rotor with 10 mm flange width is most suitable one which has the temperature distribution of 134.23 and a deformation of 0.120e-03 meters and a von mises stress of 115 M pa.

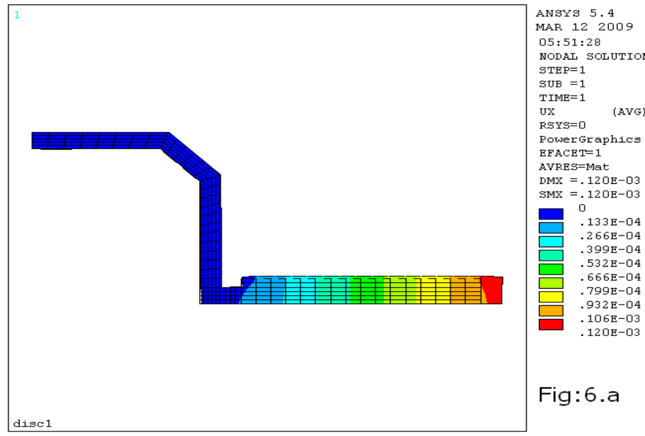


Fig6.1 Deflection in X direction (maximum)

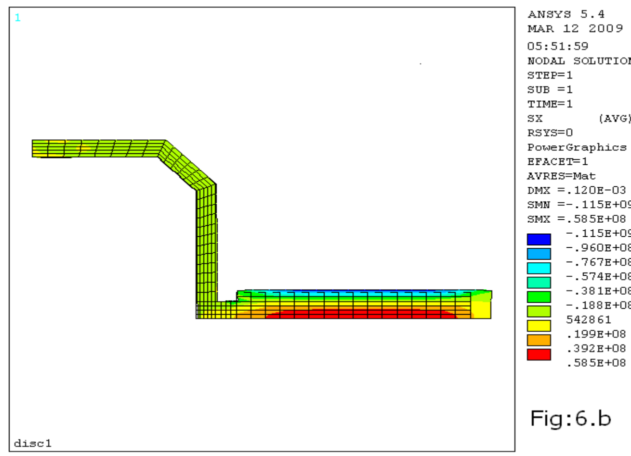


Fig6.2 Principal Stress1

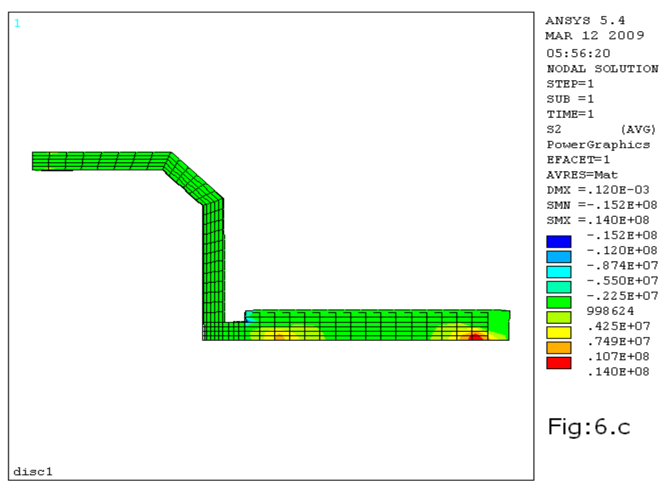


Fig6.3 Principal Stress2

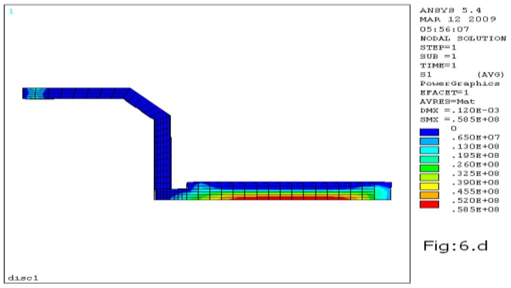
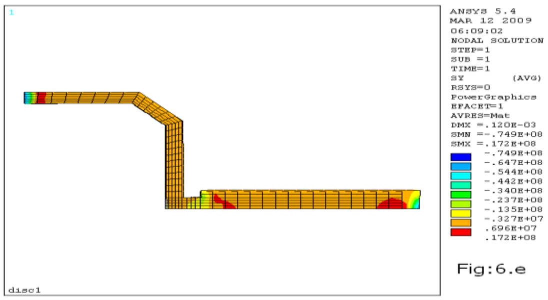


Fig6.3 Stress in X direction



Stress in Y direction

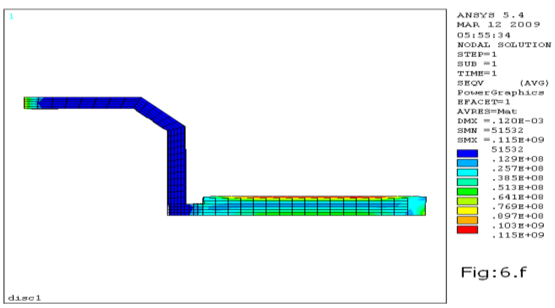


Fig6.4 Vonmises stress

7. MACRO FOR ANALYSIS

APPENDIX – I

```
/title,disc1
/filname,disc11
/prep7
et,1,plane55
keyopt,1,3,1
UIMP,1,DENS, , ,7100,
UIMP,1,NUXY, , ,0.25,
UIMP,1,KXX, , ,54,
UIMP,1,C, , ,586,
K, 1,,,,
K,2 ,37/1000,66.5/1000,,
K, 3,37/1000,60/1000,,
K,4 ,80.4/1000,66.5/1000,,
K, 5,77.95/1000,60/1000,,
K, 6,96.95/1000,49.95/1000,,
K, 7,90.45/1000,47.5/1000,,
K, 8,90.45/1000,,,
K, 9,96.95/1000,6.5/1000,,
K,10 ,103.6/1000,10/1000,,
K,11 ,103.6/1000,6.5/1000,,
K,12 ,180/1000,10/1000,,
K, 13,180/1000,,,
K,14 ,96.95/1000,,,
K, 15,103.6/1000,,,
K, 16,90.45/1000,6.5/1000,,
K,17 ,180/1000,6.5/1000,,
LSTR, 2, 3
LSTR, 2, 4
LSTR, 4, 6
LSTR, 6, 9
LSTR, 9, 11
LSTR, 11, 10
LSTR, 10, 12
LSTR, 12, 13
LSTR, 3, 5
LSTR, 5, 7
LSTR, 7, 16
LSTR, 16, 8
LSTR, 8, 14
LSTR, 15, 14
LSTR, 15, 13
LSTR, 15, 11
LSTR, 9, 14
LSTR, 16, 9
LSTR, 7, 6
LSTR, 4, 5
LSTR, 11, 17
LSTR, 17, 12
LSTR, 13, 17
AL,1,2,3,4,18,11,10,9
A,8,14,9,16
AL,17,14,16,5
AL,6,21,22,7
AL,16,15,23,21
ASBL,1,19
ASBL,7,20
LESIZE,1,,,5
LESIZE,20,,,5
LESIZE,19,,,5
LESIZE,18,,,5
LESIZE,13,,,5
LESIZE,12,,,4
LESIZE,17,,,4
LESIZE,23,,,4
LESIZE,14,,,4
LESIZE,5,,,4
LESIZE,2,,,8
LESIZE,9,,,8
LESIZE,3,,,6
LESIZE,10,,,6
LESIZE,4,,,12
LESIZE,11,,,12
LESIZE,7,,,16
LESIZE,21,,,16
```

```
LESIZE,15,,,16
LESIZE,6,,,2
LESIZE,22,,,2
AMESH,ALL
physics,write,thermal1
physics,clear
```

APPENDIX – II

```
etchg,tts
mp,ex,1,125e9
mp,prxy,1,0.25
mp,alpx,1,1.7039e-5
mp,c,1,586
physics,write,strctural1
```

APPENDIX – III

```
/solu
antype,0
physics,read,thermal1
flst,2,6,1,order,2
fitem,2,1
fitem,2,-6
d,p51x,temp,35
sfl,7,hflux,881141.3283
flst,2,14,4,order,6
fitem,2,2
fitem,2,-6
fitem,2,9
fitem,2,-15
fitem,2,22
fitem,2,-23
sfl,p51x,conv,5,,35
antype,4
time,4
deltim,0.4,,,on
kbc,1
solve
finish
physics,read,strctural1
/solu
nsel,,loc,y,0
nplot
d,all,uy
nsel,all
d,1,all,,,6,1
nplot
ldread,temp,,,,,disc11,rth
solve
/post1
plnsol,u,x,1
```

CONCLUSIONS

The following conclusions are drawn from the present work.

1. An Axis-symmetric analysis of disc brake has been carried out using Plane 55 and Plane 42 through ANSYS 5.4 (F.E.A) software.
2. Transient thermal analysis is carried out using the direct time integration technique for the application of braking force due to friction for time durationof 4,5 and 6 seconds.
3. The maximum temperature obtained in the disc is at the contact surface and is observed to be 240.161°C.
4. Static structural analysis is carried out by coupling the thermal solution to the structural analysis and the maximum Von Mises stress is observed to be 518 M Pa.
5. The Brake disc design is safe based on the Strength and Rigidity Criteria.
6. To arrive at a best combination of parameters of the Disc Brake like Flange width and Material, Transient Thermal and Structural Analysis for three different combinations in each of the three different analyses are carried out separately and the results were compared.
7. Comparing the different results obtained from the analysis, it is concluded that disc brake with 10 mm flange width, 6.5 mm Wall Thickness and of material Cast Iron is the Best possible combination for the present application.

FUTURE SCOPE OF THE PROJECT

In the present investigation of Thermal analysis of disc brake, a simplified model of the disc brake without any vents with only ambient air cooling is analyzed by FEM package ANSYS.

As a future work, a complicated model of Ventilated disc brake can be taken and there by forced convection is to be considered in the analysis.
es complicated by considering variable thermal conductivity, variable specific heat and non uniform deceleration of the vehicle. This can be considered for the future work.

REFERENCES

[1]. Introduction to Finite Elements in Engineering,Tirupathi, R.Chandrupatla Ashok D.Belegundu.
[2]. Finite Element Procedures, Klaus-Jurgen Bathe.
[3]. Finite Element Method, J.N.Reddy.
[4]. User Guide for ANSYS version 5.4.
[5]. Internet web site http://auto.howstuffworks.com/disc-brake.htm.
[6]. Design Data Book,PSG college of technology.
[7]. Heat and Mass Transfer Data Book,C P Kothandaraman, S Subramanyan.