Design And Coupled Field Analysis Of Exhaust Manifold By Using Different Materials

B.V. Harish Kumar[1], **A.Madhava Rao**[2]
[1]P.G. Scholar, [2]Guide, Asst. Professor
[1,2] Branch: M.TECH, Machine Design (MD)
[1,2] PBR Visvodaya institute of technology & science, Kavali
Mail Id : [1]bvharishkumar@yahoo.com, [2]maddy.2009avula@gmail.com

ABSTRACT:
Exhaust manifold receives the exhaust gases comes out from chamber and passes to environment. They usually assembled with cylinder. The exhaust manifold is mounted on the cylinder head of the engine.

Back pressure can be produced at two places, i.e., when the exhaust valve opens and cam overlap *taking* place. Pressure measurements at the exhaust valve during the start of the exhaust stroke at bottom dead centre (BDC) to cam overlap at the end of the exhaust stroke/beginning of the intake stroke at top dead centre (TDC).The exhaust gases emitted from the cylinder come out at temperatures of nearly 800^{0}C and with pressures ranging from 100 to 500 kPa. The exhaust manifold is subjected to high temperatures and pressures which will lead to thermo mechanical failure.

This project aims in redesigning an exhaust manifold by determining Thermal stresses and deflections exhibited under various operating conditions with different materials and temperatures. The objective is to ensure the suitability of the design for a particular material from the view point of reliability and serviceability. High end cad cam software such as Unigraphics and Ansys is used for modeling and analysis. The 3d Model of exhaust manifold is subjected to thermal and structural loads and results are tabulated according to the procedure for the Exhaust manifold.
CAD TOOL: UNIGRAPHICS
CAE TOOL: ANSYS

Keywords:- Exhaust manifold, Cast iron, Metal matrix, Turbulence effect, Thermal stress

INTRODUCTION

1.1 EXHAUST MANIFOLD

Exhaust manifold receives the exhaust gases comes out from chamber and passes to environment. The usually assembled with cylinder. Cast iron is material used for exhaust manifolds. Factors to be considered during the design and development of exhaust manifold

A. Runner length This is arguably one of the most important factors. First would be to make sure that the runners are as equal length as possible. The idea being that the exhaust pulses will be spaced out evenly and arriving at the turbine wheel on the turbo at their own time in the firing order.

B. Runner volume Runner volume needs to be considered when building a turbo manifold. While a larger runner diameter does facilitate lower exhaust backpressure for better flow on the top-end, it does cause a lower exhaust velocity. A lower exhaust velocity will cause longer spool times, and less transient response out of the turbo.

C. Collectors A collector's job is to tie all of the cylinder's pipes together in one common place and send them into a single exit pipe.A collector is generally a conglomeration of pipes all merged together, allowing for a smooth transition from the primaries or secondaries into the rest of the exhaust.

D. Back Pressure Back pressure can be produced at two places, i.e., when the exhaust valve opens and cam overlap taking place. Pressure measurements at the exhaust valve during the start of the exhaust stroke at bottom dead centre (BDC) to cam overlap at the end of the exhaust stroke/beginning of the intake stroke at top dead centre (TDC)

The mounted exhaust manifolds on top of the cylinder head of the engine collect the gas exhausted from the engine, and sends it to a catalyst converter. The performance of the Engine depends upon the design of the exhaust manifold. Principally, the efficiency of the emission and fuel utilization are strongly related with the exhaust manifold

As the exhaust gases are very hot, the pipe must be heat-resistant. The pipe should be able to send away the toxic gases away from the user. The exhaust pipes are different for different types of engines. A stationary engine may have a chimney serving as an exhaust pipe. For motor cycles the exhaust pipes depends on the type of the engine it has. In case of trucks, the exhaust system is horizontal and sometimes may have vertical exhaust pipes. Some trucks are provided with flexible ducting between the engine and silencer. This arrangement will avoid the vibrations being transferred to the exhaust system. In a two stoke engine, the exhaust pipe is provided with a bulge known as expansion chamber. More air and fuel is made to enter the engine cylinder with the exhaust pressure of the gases in the chamber. This improves the power and fuel efficiency. This effect of using the momentum of the exhaust gases to create a pressure drop in the cylinder and assisting more air and fuel to enter into the cylinder is called Kadenacy effect. Careful design of the inlet and exhaust pipes will maximize the Kadenacy effect.

PROBLEM FORMULATION & SOLUTION METHODOLOGY

PROBLEM IDENTITY

In recent years the engine operating temperatures of cars, vans and heavy goods vehicles have been increasing because of environmental legislation on emissions and the need to improve engine efficiency. The motor industry worldwide is highly competitive, operating on small margins and large volumes. Therefore, the profitability is highly geared to reductions in design, development and manufacturing costs. There are very significant economic and environmental benefits from using existing materials.

The exhaust gases emitted from the cylinder come out at temperatures of nearly 800^0C and with pressures ranging from 100 to 500 kPa. The exhaust manifold is subjected to high temperatures and pressures which will lead to thermo mechanical failure.

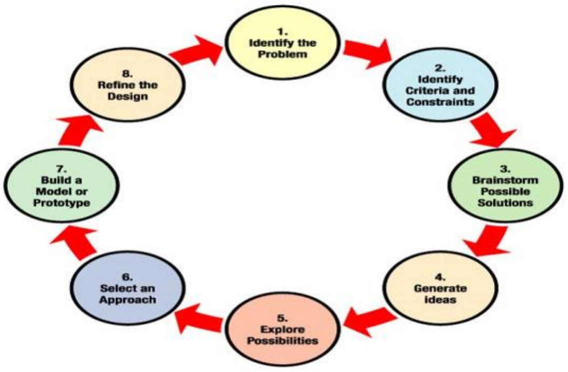
METHODOLOGY

This project aims in redesigning an exhaust manifold by determining Thermal stresses and deflections exhibited under various operating conditions with different materials and temperatures. The objective is to ensure the suitability of the design for a particular material from the view point of reliability and serviceability. High end cad cam software such as Unigraphics and Ansys is used for modeling and analysis.

MODELING OF EXHAUST MANIFOLD

COMPUTER AIDED DESIGN (CAD)

Computer-aided design (CAD), also known as computer-aided design and drafting (CADD), is the use of computer systems to assist in the creation, modification, analysis, or optimization of a design.



Computer-aided drafting describes the process of creating a technical drawing with the use of computer software. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print or machining operations. CAD software uses either vector based graphics to depict the objects of traditional drafting, or may also produce raster graphics showing the overall appearance of designed objects.

CAD often involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.

CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) space.

CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more. CAD is also widely used to produce computer animation for special effects in movies, advertising and technical manuals. The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s. Because of its enormous economic importance, CAD has been a major driving force for research in computational geometry, computer graphics (both hardware and software), and discrete differential geometry.

STEPS INVOLVED IN 3D MODELLING OF EXHAUST MANIFOLD:

3D model is designed by using NX cad software.

Sketching:

Below is the sketch required to obtain the 3D model of the exhaust manifold.

Below image shows the sketch of the exhaust manifold.

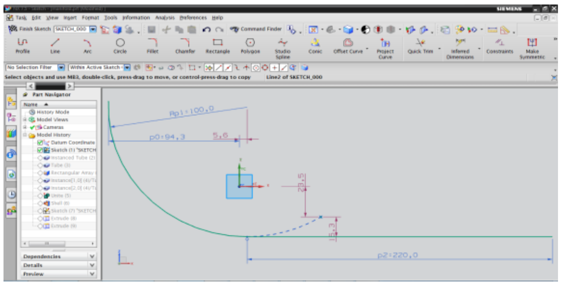


Fig.4.2.3 sketch of the exhaust manifold.

Below image shows the tube option.

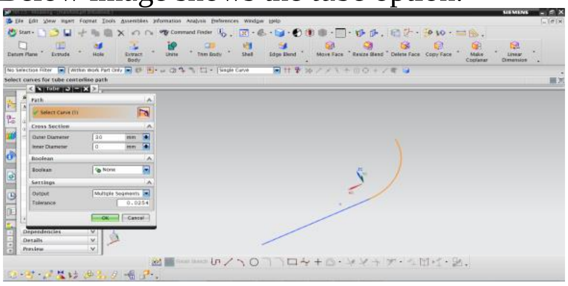


Fig.4.2.4 Tube option

Below image shows the Instant geometry option.

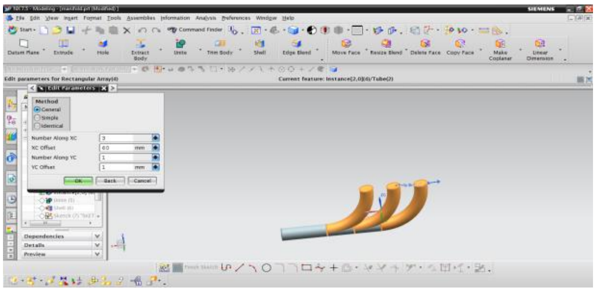


Fig. 3.2.5 Instant geometry option

Below image shows tube along Bridge curve option

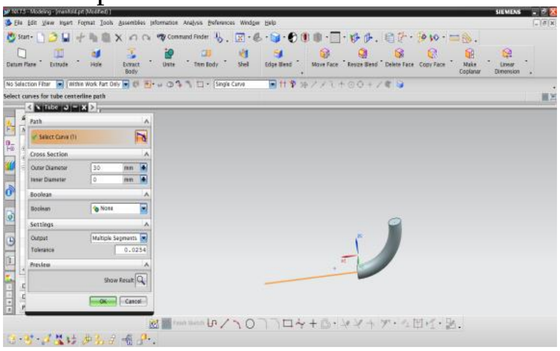


Fig. 4.2.7 tube along Bridge curve option

Below image shows the shell option

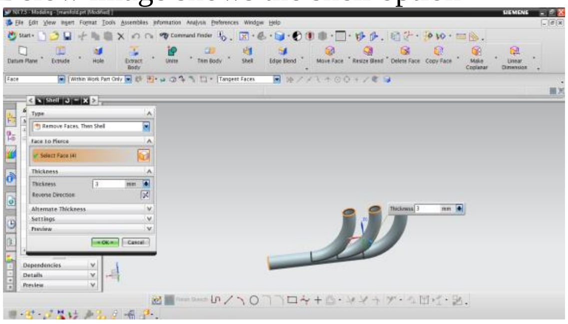


Fig. 3.2.8 shell option

Below image shows Sketch of exhaust house

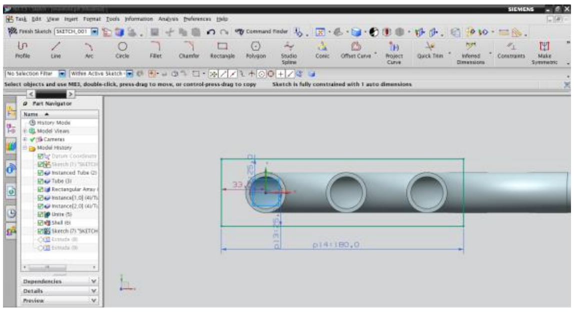


Fig. 3.2.9 extrude option for exhaust house

Below image shows the Extrude of exhaust house

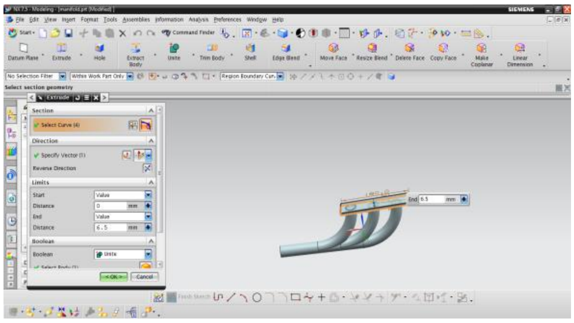


Fig. 3.2.10 Extrude option for Exhaust house option

Below image shows the Extrude option

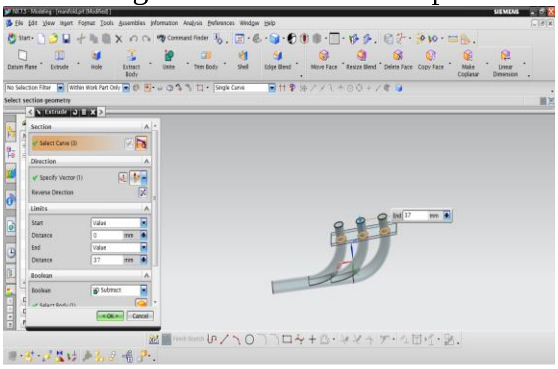


Fig. 3.2.12.sketch option for Extrude option

Final component

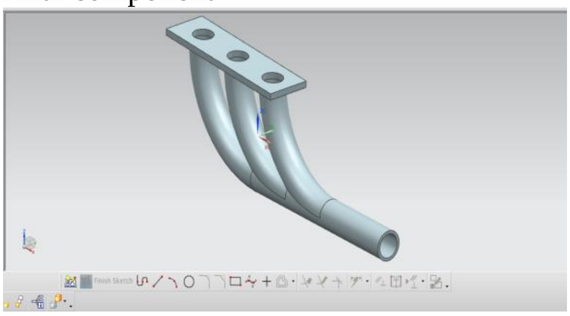


Fig. 3.2.15 final part

3D MODEL OF EXHAUST MANIFOLD:

Front view of exhaust manifold:

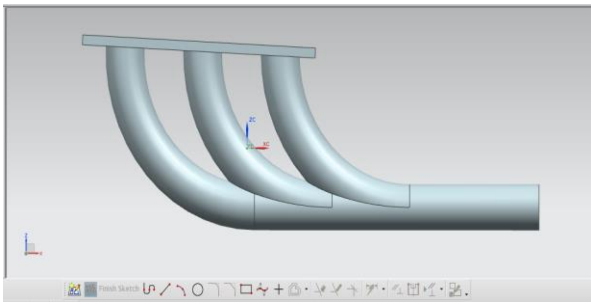


Fig. Shows the Front view of exhaust manifold

Top view of exhaust manifold:

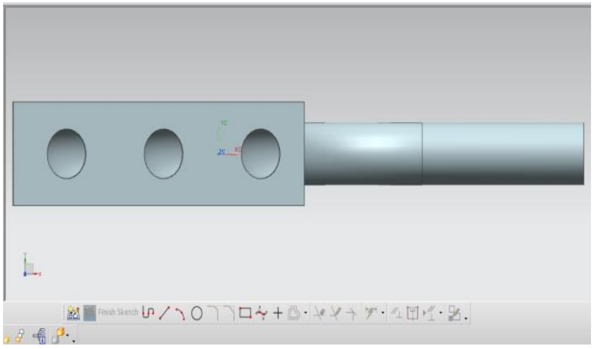


Fig. Shows the Top view of exhaust manifold

Right view of exhaust manifold:

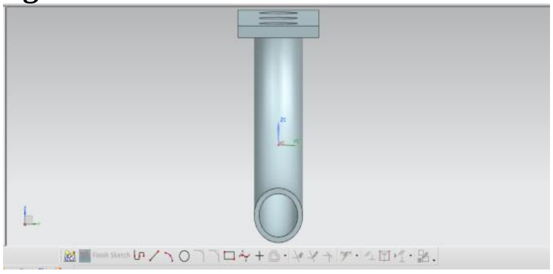


Fig. Shows the Right view of exhaust manifold

Bottom view of exhaust manifold:

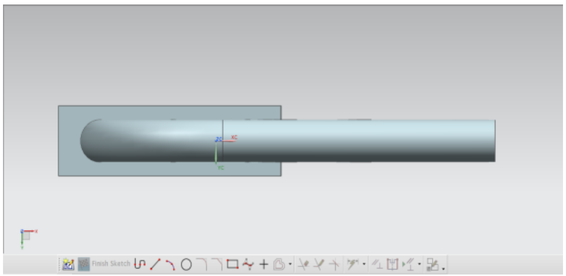


Fig. Shows the Isometric view of exhaust manifold

FINITE ELEMENT ANALYSIS OF EXHAUST MANIFOLD

5.1 INTRODUCTION

The Basic concept in FEA is that the body or structure may be divided into smaller elements of finite dimensions called "Finite Elements". The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called "Nodes" or "Nodal Points". Simple functions are chosen to approximate the displacements over each finite element. Such assumed functions are called "shape functions". This will represent the displacement with in the element in terms of the displacement at the nodes of the element.

The Finite Element Method is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problems that can be represented in differential equations form. The applications of FEM are limitless as regards the solution of practical design problems.

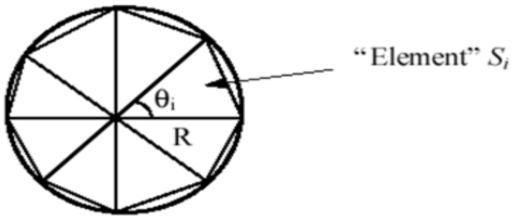
Due to high cost of computing power of years gone by, FEA has a history of being used to solve complex and cost critical problems. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an automobile or an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

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

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

- Structural analysis
- Thermal analysis

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



Basic Concepts: The Finite Element Method is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life as well as engineering. The philosophy of FEA can be explained with a small example such as measuring the area of a circle. Area of one Triangle: Si = ½ * R2* Sin θI

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

Where N = total number of triangles (elements)

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

Available Commercial FEM software packages

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

More about FEA

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

FEA consists of a computer model of a material or design that is loaded and analyzed for specific results. It is used in new product design, and existing product refinement. A design engineer shall be able to verify the proposed design, which is intended to meet the customer requirements prior to the manufacturing. Things such as, modifying the design of an existing product or structure in order to qualify the product or structure for a new service condition. Can also be accomplished in case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

The Basic Steps Involved in FEA

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

Brief Over View of Structural Static Analysis:

Static analysis is one in which the loads/boundary conditions are not the functions of time and the assumption here is that the load is applied gradually. The most common application of FEA is the solution of stress related design problems.

Typically in a static analysis the kind of matrix solved is

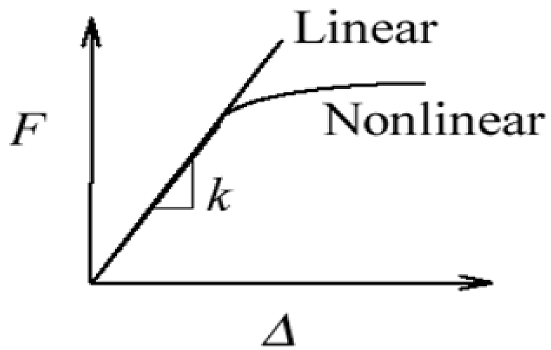
$$[K] * [X] = [F]$$

Where K is called the stiffness matrix, X is the displacement vector and F is the load matrix. This is a force balance equation. Some times, the K matrix is the function X. Such systems are called non-linear systems.

Nodal Displacements	ui, uj
Nodal Forces	fi, fj
Spring constant	k

Spring force displacement relationship

$F = k\,\Delta$ with $\Delta = uj - ui$



K = F/Δ (>0) is the force needed to produce a unit stretch

Consider the equilibrium forces for the spring. At node i, we have

$fi = -F = -k(uj - ui) = kui - kuj$

And at node j,

$F = k(uj - ui) = - kui + kuj$

Element Quality Requirements:

There are certain parameters that determine the quality of the results. The engineer has to ensure that these parameters maintained within the acceptable limits of the software for obtaining the good results. These are called mesh quality parameters.

Warp age: Warp age occurs only on Quad and Hexa elements. Since three points define a plane, one of the four nodes being in different plane by an angle causes warp age. The perfect warp age value desired is zero.

Aspect Ratio: It checks the ration of the length of the longest side of an element to the shortest length of the same element. A perfect value is one. The quickest way is to correct high aspect ratio is to increase the length of the shortest side or decrease the length of the longest side

NOTES ON ANSYS ELEMENT MANUAL

General element Features:

Element Input: Many features that are common to all ANSYS elements in the element library are described here.

Element Name: An element type is identified by a name, such as BEAM3, consisting a group label (BEAM) and a unique, identification number (3).

Nodes: The nodes associated with the element are listed as I, J, K, etc. The node order determines the element coordinate system orientation for some element types.

Degrees of Freedom: Each element type has DOF set, which constitutes the primary nodal unknowns to be determined by the analysis. They may be displacements, rotations, temperatures, pressures, voltage, etc. DOF are not defined on the nodes explicitly by the user, but rather are implied by the element types attached to them

Real Constants: This data is required for the calculation of element matrix, but which cannot be determined from node locations or material properties, are input as "real constants". Typical real constants include area, thickness, diameter and etc.

Material Properties: Typical material properties include Young's modulus, density, thermal conductivity and etc.

Surface Loads: These loads are typically pressures for structural element types, convictions and heat fluxes for thermal element type and etc.

Body Loads: Body loads are typically temperatures for structural element type, heat generation rates for thermal element type and etc.

Special Features: The keywords in the special features list indicate that certain additional capabilities are available for the element. Most often these features make the element the element nonlinear and require an iterative solution be done.

KEY Opts: These are the switches, used to turn various element options on or off. KEYOPT options include stiffness formulation choices, printout controls, element co-ordinate system choices and etc.

Solution Output: The output from the solution consists of the nodal solution (or the primary DOF solution) and the element solution (or the derived solution). Each of these solutions are described here. Solution output is written to output file "jobname.out" and the results are "jobname.rst". The output file can be viewed through the GUI, while the database and results file data can be postprocessor.

Nodal Solution: The nodal solution form of analysis consists of (1) the degree of freedom solution, such as nodal displacements, temperatures, and pressures and (2) the reaction solution calculated at constrained nodes, such as forces at displacement constraints, heat flows at temperature DOF constraints, fluid flows at pressure DOF constraints, and so on.

Element Solution: The element output items are shown along with the element type description. Not all of the items shown in the output table will appear at all the times for the element. Items not appearing are either not applicable to the solution or have all zero results and are suppressed to save the space. The output is, in some cases, dependent on the input. For example, for thermal elements accepting either surface convection or nodal heat flux, the output will be either in terms of convection or heat flux.

Linear Material Properties:

The material properties used by the element type are listed under "Material Properties" in the input table for each element type. These properties are called linear properties, because typical non-thermal solutions with these properties require only a single iteration. Properties such as stress-strain data are called nonlinear properties, because an analysis with these properties requires an iterative solution. For orthotropic materials, the X, Y, and Z part of the label refers to the direction that the particular property acts in. Poisson's ratio may be input in either major or minor form, but not both for a particular material. The major form is converted to the minor form during the solve operation.

Data Tables:

A data table is a series of constants that are interpreted when they are used. Data tables are always associated with a material number and are most often used to define nonlinear material data. For some element types, the data table is used for special element input data other than material properties. The form of data table depends upon the data being defined. Where the form is peculiar to only one element type.

Node and Element Loads:

Loadings are defined to be two ways: nodal and element. Nodal loads are defined at the nodes and are not directly related to the elements. These nodal loads are associated with DOF at the nodes. Element loads are surface loads, body loads and inertia loads. Element loads are always associated with a particular element type. Certain elements may also have "flags". Flags are not actually loads, but are used to indicate that a certain type calculation is to be performed. For example, when the FSI(fluid structure interaction) flag is turned on, a specified face of an acoustic element is treated as an interface between a fluid portion and a structural portion of the model.

Triangle, Prism and Tetrahedron Elements:

Degenerated elements are elements whose characteristic face shape is quadrilateral, but is modeled with at least one triangular face. For example, PLANE42 triangles, SOLID45 wedge, and SOLID$% tetrahedral are all degenerated shapes. Degenerated elements are often used for modeling transition regions between fine and course meshes, for modeling irregular and warped surfaces, etc. Degenerated elements formed with quadrilateral and brick elements without midsize nodes are much less accurate than those formed from elements with midside nodes and should not be used in high stress gradient regions. An exception where triangular shell elements are preferred is for severely skewed or warped elements. Warping occurs when the four nodes of a quadrilateral shell element are not in the same plane. Warp age is measured by the relative angle between the normal to the face at the nodes. A flat face has all normal parallel to i.e. a zero relative angle.

Degenerated triangular 2-D solid and shell elements may be formed from four noded quadrilateral elements by defining duplicate node numbers for the third and fourth(K and L) node locations. The node pattern then becomes I, J, K, K. If the L node is not input, it defaults to node K. If extra shape functions are included in the element, they are automatically suppressed (degenerating the element to lower order). Element loads specified on nodal basis should have the same loads specified at the duplicate node locations.

Degenerated triangular prism elements may be formed from eight noded 3-D solid elements by defining duplicate node numbers for the third and fourth and the seventh and eighth node locations. The node pattern then becomes I, J,K,K,M,N,O,O.

Shear Deflection:

Shear deflection effects are often significant in the lateral deflection of short beams. The significance decreases as the ratio of the radius of gyration of the beam cross section to the beam length becomes small compared to unity. Shear deflection effects are activated in the stiffness matrices of ANSYS beam element by including a nonzero shear deflection constant (SHEAR) in the real constant list for the element type.

The shear deflection constant is defined as the ratio of the actual beam cross sectional area to the effective area resisting shear deformation. The shear constant should be equal to or greater than zero. The element shear stiffness decreases with increasing the value of shear deflection constant. A zero shear deflection constant may be used to neglect shear deflection.

THEORIES OF FAILURE:

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

Maximum principal stress theory (rankine's) $\sigma 1$ or $\sigma 2$ or $\sigma 3$ (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) ($\sigma 1$-$\sigma 2$) or ($\sigma 2$-$\sigma 3$) or ($\sigma 3$-$\sigma 1$) = σ y (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)

σ1-ν (σ2+σ3) or σ2-ν (σ3+σ1) or

σ3-ν (σ1+σ2) (which ever is maximum) = σ y. According to this theory, failure of the material is deemed to have taken place when the maximum principal strain reaches a value calculated from a simple tensile test.

Maximum strain energy theory (Beltrami-Haigh's)

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

Octahedral or distortion energy theory (von mises-hencky)

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

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

$1/n= \sigma_m/\sigma_y + Kf\ \sigma_a/\sigma\text{-}1$

$1/n= t_m/t_y + Kf\ ta/t\text{-}1$

Where, σm =mean stress

σ_y = yield stress

σ_a = stress amplitude (σmax-σmin)/2

σ-1 = endurance limit stress

t_m = mean shear stress

t_y = yield shear stress

1/n = factor of safety

Goodman's equations (for brittle materials)

$1/n=Kt\ [\sigma_m/\sigma_u +\sigma_a/\sigma\text{-}1]$

$1/n=Kt[t_m/t_u +ta/t\text{-}1]$

Where, σ_u = 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.

The ANSYS program has a compressive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS Program. Users can input data using a mouse, a keyboard, or a combination of both. A graphical user interface is available throughout the program, to guide new users through the learning process and provide more experienced users with multiple windows, pull-down menus, dialog boxes, tool bar and online documentation

The following table shows the brief description of steps followed in each phase:

PRE-PROCESSOR	SOLUTION PROCESSOR	POST-PROCESSOR
Assigning element type	Analysis definition	Read results
Geometry definition	Constant definition	Plot results on graphs
Assigning real constants	Load definition	View animated results
Material definition	Solve	
Mesh generation		
Model display		

SOLUTION PROCESSOR

Here we create the environment to the model, i.e, applying constraints &loads. This is the main phase of the analysis, where the problem can be solved by using different solution techniques. Her three major steps involved:

- Solution type required, i.e. static, modal, or transient etc., is selected
- Defining loads. The loads may be point loads, surface loads; thermal loads like temperature, or fluid pressure, velocity are applied.
- Solve FE solver can be logically divided in o three main steps, the pre-solver, the mathematical-engine and post-solver. The pre-solver reads the model created by pre-processor and formulates the mathematical representation of the model and calls the mathematical-engine, which calculates the result.

POST –PROCESSOR:

Post processing means the results of an analysis. It is probably the most important step in the analysis, because we are trying to understand how the applied loads affects the design, how food your finite element mesh is, and so on.

The analysis results are reviewed using postprocessors, which have the ability to display distorted geometries, stress and strain contours, flow fields, safety factor contours, contours of potential filed results; vector field displays mode shapes and time history graphs. The postprocessor can also be used for algebraic operations, database manipulators, differentiation, and integration of calculated results. Response spectra may be generated from dynamic analysis. Results from various loading may be harmonically loaded axis metric structures

REVIEW THE RESULTS:

Once the solution has been calculated, we can use the ANSYS postprocessor to review the results. Two postprocessors are available: POST1 and POST 26. We use POST 1, the general postprocessor to review the results at one sub step over the entire model or selected portion of the model. We can obtain contour displays, deform shapes and tabular listings to review and interpret the results of the analysis. POST 1 offers many other capabilities, including error estimation, load case combination, calculation among results data and path operations.

We use POST 26, the time history post processor, to review results at specific points in the model over all time steps. We can obtain graph plots of results, data vs. time and tabular listings. Other POST 26 capabilities include arithmetic calculations and complex algebra.

In the solution of the analysis the computer takes over and solves the simultaneous set of equations that the finite element method generates, the results of the solution are

- Nodal degree of freedom values, which form the primary solution
- Derived values which form the element solution

MESHING:

Before meshing the model and even before building the model, it is important to think about weather 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.

Compare to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. If we want this type of mesh, we must build the geometry as series of fairly regular volumes and/or areas that can accept a mapped mesh.

FREE MESHING:

In free meshing operation, no special requirements restrict the solid model. Any model geometry even if it is regular, can be meshed. The elements shapes used will depend on whether we are meshing areas or volumes. For area meshing, a free mesh can consist of only quadrilateral elements, only triangular elements, or a mixture of the two. For volume meshing, a free mesh is usually restricted to tetrahedral elements. Pyramid shaped elements may also be introduced in to the tetrahedral mesh for transitioning purposes.

STRUCTURAL STATIC ANALYSIS:

A static analysis calculates the effects of study loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis can however include steady inertia loads and time varying loads that can be approximated as static equivalent loads. Static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed, i.e. the loads and the structure's responses are assumed to vary slowly with respect to time. The kinds of loading that can be applied in static analysis include:

- Externally applied forces and pressures.
- Steady state inertial forces
- Imposed displacement
- Temperatures
- Fluences (for nuclear swelling)

Finite element analysis has been carried out on the modified exhaust manifold to record the stresses and deflections. Initially thermal analysis is carried out to find the temperature distribution and later couple field analysis is carried out to find the structure behavior due to coupled thermal and structural loads. The 3D model and the FE model of the modified exhaust manifold used for analysis is shown in the below figures.

THERMAL ANALYSIS

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

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions)

Some types of coupled-field analyses, such as thermal-structural and magnetic-thermal analyses, can represent thermal effects coupled with other phenomena. A coupled-field analysis can use matrix-coupled ANSYS elements, or sequential load-vector coupling between separate simulations of each phenomenon.

Steady-State Thermal Analysis:

The ANSYS Multiphysics, ANSYS Mechanical, ANSYS FLOTRAN, and ANSYS Professional products support steady-state thermal analysis. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineer/analysts often perform a steady-state analysis before performing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis, performed after all transient effects have diminished.

You can use steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convections
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Including radiation effects also makes the analysis nonlinear.

HEAT TRANSFER

The discipline of heat transfer in the simplest terms is concerned with only temperature and flow of heat. The amount of thermal energy available is represented by temperature meanwhile the movement of thermal energy from pace to place is represented by heat flow.

Thermal energy on a microscopic scale is related to the kinetic energy of molecules. A greater thermal agitation of material constituent molecules is cause by the greater temperature of it. Conduction, convection and radiation are three aspects of heat rate. Temperature difference is the driving force for all the three modes in which the direction of heat transfer is from the high temperature to the low temperature.

$$Q=Q_{\text{conduction}}+Q_{\text{convection}}+Q_{\text{radiation}}$$

Conduction

It is mode transfer of energy through solid. The rate of heat transfer is over a cross sectional area A with temperature difference is inversely proportional to the thickness X.

$Q_{cond}=\frac{kA(\Delta T)}{x}$; (W)

Where, k= thermal conductivity (W/m K)

A = cross sectional area (m^2)

ΔT = temperature different(K)

X =thickness(m)

Convection

Two types of convection are, free and forced convection. Free convection motion is set up by the temperature of the fluid via natural circulation.

Meanwhile, forced convection is forced the fluid to and enhanced the rates of heat transfer between the flowing fluid and solid surface.

The rate of heat transfer over cross sectional area A and temperature difference is proportional to the surface of heat transfer coefficient

$$\dot{Q}_{conc}=hA(\Delta T); \qquad (W)$$

Where, h = convection coefficient(W/m^2K)

A=cross sectional area (m^2)

ΔT= temperature different (K)

5.8.3 Radiation

The heat transfer of heat input is the lumped mass of consequential at temperature rise with specific heat.

$$\dot{Q}_{rad} = mC_p \Delta T ; \qquad (W)$$

Where m=weight (m)
C_p =specific heat (kJ/kgK)
ΔT= temperature difference (K)

COUPLED FIELD ANALYSIS OF MANIFOLD USING CAST IRON

MATERIAL PROPERTIES OF EXHAUST MANIFOLD (Cast Iron):

Thermal conductivity, K (w/m k)-		50
Density, (kg/m^3)	-	8100
Specific heat, c (J/Kg k)	-	1.88
Poisson's ratio, v	-	0.3
Thermal expansion, á (1/ k)	-	0.3
Elastic modulus, E (GPa)	-	210
Coefficient of friction, µ	-	0.2
Bulk temperature	-	26

SOLUTION PROCESSOR

Here we create the environment to the model, i.e, applying constraints &loads. This is the main phase of the analysis, where the problem can be solved by using different solution techniques. Her three major steps involved:

- Solution type required, i.e. static, modal, or transient etc., is selected
- Defining loads. The loads may be point loads, surface loads; thermal loads like temperature, or fluid pressure, velocity are applied.
- Solve FE solver can be logically divided in o three main steps, the pre-solver, the mathematical-engine and post-solver. The pre-solver reads the model created by pre-processor and formulates the mathematical representation of the model and calls the mathematical-engine, which calculates the result. The result return to the solver and the post solver is used to calculate strains, stresses, etc., for each node within the component or continuum.

STRUCTURAL STATIC ANALYSIS:

A static analysis calculates the effects of study loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis can however include steady inertia loads and time varying loads that can be approximated as static equivalent loads. Static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed, i.e. the loads and the structure's responses are assumed to vary slowly with respect to time. The kinds of loading that can be applied in static analysis include:

- Externally applied forces and pressures.
- Steady state inertial forces
- Imposed displacement
- Temperatures
- Fluences (for nuclear swelling)

Finite element analysis has been carried out on the modified exhaust manifold to record the stresses and deflections. Initially thermal analysis is carried out to find the temperature distribution and later couple field analysis is carried out to find the structure behavior due to coupled thermal and structural loads. The 3D model and the FE model of the modified exhaust manifold used for analysis is shown in the below figures.

THERMAL ANALYSIS

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

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions)

Coupled-Field Analyses:

Some types of coupled-field analyses, such as thermal-structural and magnetic-thermal analyses, can represent thermal effects coupled with other phenomena. A coupled-field analysis can use matrix-coupled ANSYS elements, or sequential load-vector coupling between separate simulations of each phenomenon.

Steady-State Thermal Analysis:

The ANSYS Multiphysics, ANSYS Mechanical, ANSYS FLOTRAN, and ANSYS Professional products support steady-state thermal analysis. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineer/analysts often perform a steady-state analysis before performing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis, performed after all transient effects have diminished.

You can use steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convections
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Including radiation effects also makes the analysis nonlinear.

HEAT TRANSFER

The discipline of heat transfer in the simplest terms is concerned with only temperature and flow of heat. The amount of thermal energy available is represented by temperature meanwhile the movement of thermal energy from pace to place is represented by heat flow.

Thermal energy on a microscopic scale is related to the kinetic energy of molecules. A greater thermal agitation of material constituent molecules is cause by the greater temperature of it. Conduction, convection and radiation are three aspects of heat rate. Temperature difference is the driving force for all the three modes in which the direction of heat transfer is from the high temperature to the low temperature.

$$Q = Q_{\text{conduction}} + Q_{\text{convection}} + Q_{\text{radiation}}$$

Conduction

It is mode transfer of energy through solid. The rate of heat transfer is over a cross sectional area A with temperature difference is inversely proportional to the thickness X.

$$Q_{cond} = \frac{kA(\Delta T)}{x}; \text{ (W)}$$

Where, k= thermal conductivity (W/m K)
A= cross sectional area (m^2)
ΔT = temperature different(K)
X =thickness(m)

5.8.2 Convection

Two types of convection are, free and forced convection. Free convection motion is set up by the temperature of the fluid via natural circulation.

Meanwhile, forced convection is forced the fluid to and enhanced the rates of heat transfer between the flowing fluid and solid surface.

The rate of heat transfer over cross sectional area A and temperature difference is proportional to the surface of heat transfer coefficient

$$\dot{Q}_{conc} = hA(\Delta T); \qquad \text{(W)}$$

Where, h = convection coefficient(W/m^2K)

A=cross sectional area (m^2)

ΔT= temperature different (K)

5.8.3 Radiation

The heat transfer of heat input is the lumped mass of consequential at temperature rise with specific heat.

$$\dot{Q}_{rad} = mC_p\Delta T; \qquad (W)$$

Where m=weight (m)

C_p =specific heat (kJ/kgK)

ΔT= temperature difference (K)

COUPLED FIELD ANALYSIS OF MANIFOLD USING CAST IRON

MATERIAL PROPERTIES OF EXHAUST MANIFOLD (Cast Iron):

Thermal conductivity, K (w/m k)	-	50
Density, (kg/m^3)	-	8100
Specific heat, c (J/Kg k)	-	1.88
Poisson's ratio, v	-	0.3
Thermal expansion, á (1/ k)	-	0.3
Elastic modulus, E (GPa)	-	210
Coefficient of friction, µ	-	0.2
Bulk temperature	-	26

ELEMENT DESCRIPTIONS:

NODE SOLID87:

SOLID87 is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis.

10 NODE SOLID187:

SOLID187 has a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems). See Solid 187 for a 20-node brick shaped element. The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The parasolid file is imported into ansys and is meshed with 10 node thermal solid 87 element type. The structure, number of nodes and input summary of the element is given below.

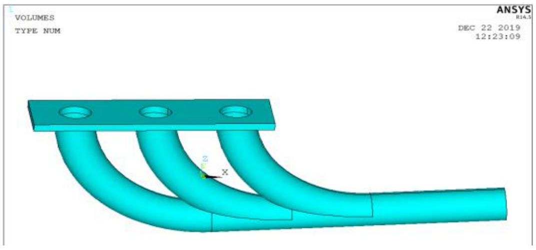


Fig. shows the Infinite model of the modified exhaust manifold

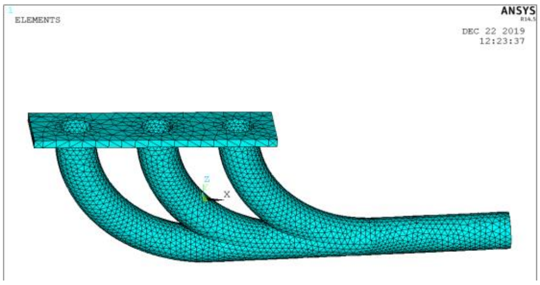


Fig. shows the finite model of the modified exhaust manifold

APPLYING THE BOUNDARY CONDITIONS

In thermal analysis of modified exhaust manifold, we have to apply thermal loads. Temperature 1073K is applied inside of exhaust pipes (4), and convection is applied on modified exhaust manifold.

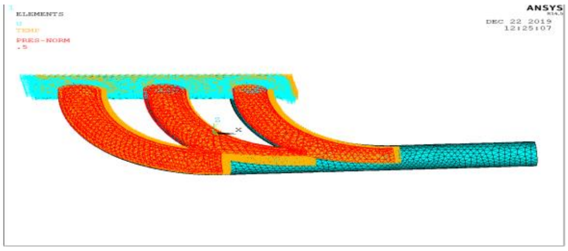
STRUCTURAL LOADING CONDITIONS

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 structural analysis of modified exhaust manifold, we have to apply structural and thermal loads. The bolts are arrested in all Dof, and pressure load 500000Pa is applied inside of the exhaust pipes (4). Temperature distribution is applied as Thermal loads on modified exhaust manifold from the thermal analysis

LOADS:

- Pressure = 500KPa= 500000Pa
- Temperature from engine that is 1073K applied inside of manifoldtubes..
- The bolting locations are arrested in all Dof for modified exhaust manifold.



RESULTS:

Nodal displacements:

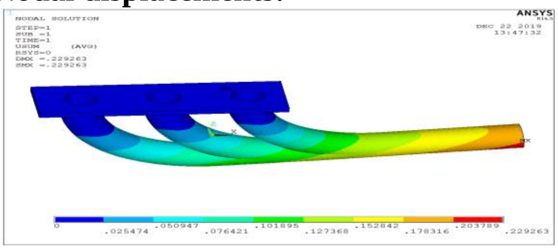


Fig. Shows the Nodal displacements on exhaust manifold

Nodal temperature:

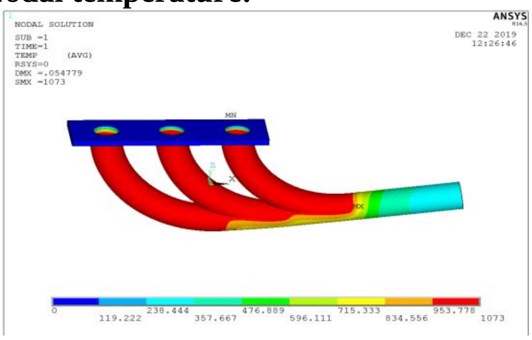


Fig. Shows the temperature distribution on exhaust manifold

Von Mises stress:

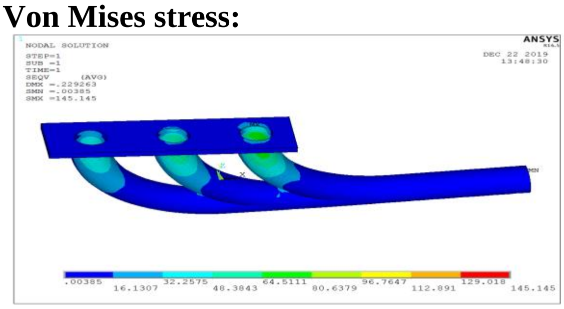


Fig. Shows the Von Misses stress of exhaust manifold

From the above results it is observed that:

- The Max Deflection 0.22mm observed on the exhaust manifold for operating loading conditions.
- The Max Avg. VonMises Stress observed 145.145Mpa on the segway assembly for operating loading conditions. And the Yield strength of the materials cast iron is 600Mpa.

5.8 COUPLED FIELD ANALYSIS OF MANIFOLD USING ALUMINIUM MATERIAL PROPERTIES OF EXHAUST MANIFOLD (AL):

Thermal conductivity, K (w/m k)-		113
Density, (kg/m3)	-	2650
Specific heat, c (J/Kg k)	-	960
Poisson's ratio, v	-	0.3
Elastic modulus, E (GPa)	-	7
Bulk temperature	-	26

ELEMENT DESCRIPTIONS:

10 NODE SOLID87:

SOLID87 is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis.

NODE SOLID187:

SOLID187 has a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems). See Solid 187 for a 20-node brick shaped element. The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The parasolid file is imported into ansys and is meshed with 10 node thermal solid 87 element type. The structure, number of nodes and input summary of the element is given below.

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 07 Issue 02
February 2020

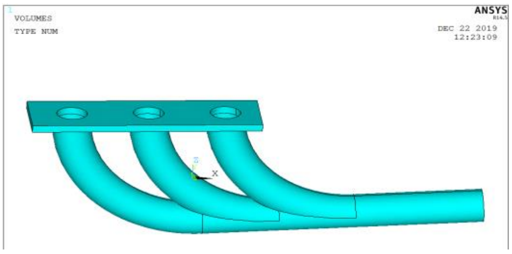


Fig. shows the Infinite model of the modified exhaust manifold

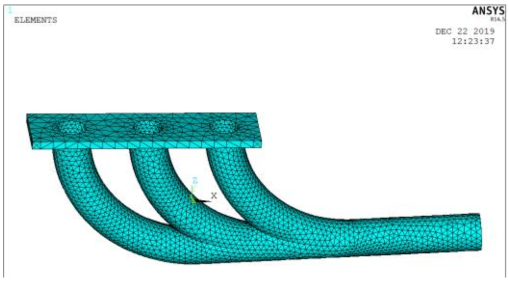


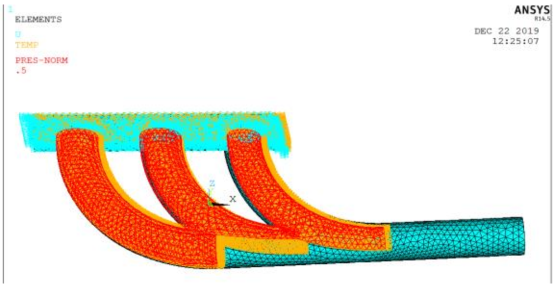
Fig. shows the finite model of the modified exhaust manifold

APPLYING THE BOUNDARY CONDITIONS

In thermal analysis of modified exhaust manifold, we have to apply thermal loads. Temperature 1073K is applied inside of exhaust pipes (4), and convection is applied on modified exhaust manifold.

LOADS:

- Pressure = 500KPa= 500000Pa
- Temperature 1073K apply inside of manifold.
- The bolting locations are arrested in all Dof for modified exhaust manifold.



RESULTS:

Nodal displacements:

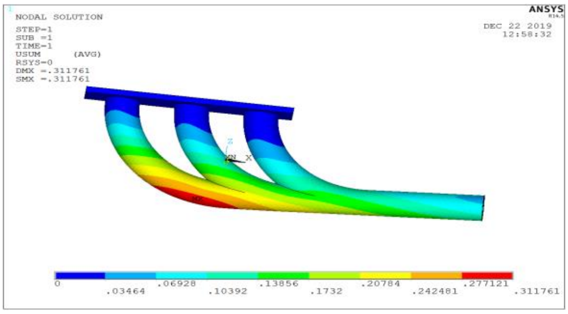


Fig. Shows the Nodal displacements on exhaust manifold

Nodal temperature:

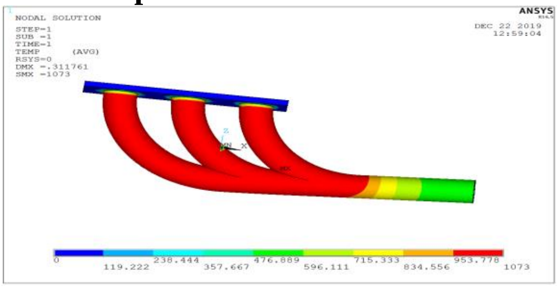


Fig. Shows the temperature distribution on exhaust manifold

Von Mises stress:

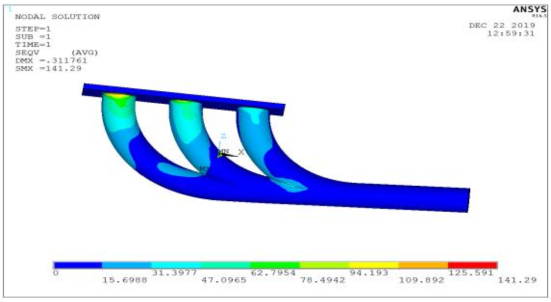


Fig. Shows the Von Mises stress of exhaust manifold

From the above results it is observed that:

- The Max Deflection 0.054mm observed on the exhaust manifold for operating loading conditions.
- The Max Avg. VonMises Stress observed 141.29Mpa on the exhaust manifold for operating loading conditions. And the Yield strength of the materials Aluminium is 180Mpa .

5.9 COUPLED FIELD ANALYSIS OF MANIFOLD USING CARBON STEEL ASTM A148

Thermal conductivity, K (w/m k)- 37
Density, (kg/m3) - 7850
Specific heat, c (J/Kg k) - 470

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 07 Issue 02
February 2020

Poisson's ratio, v	-	0.29
Elastic modulus, E (GPa)	-	190
Melting point temperature	-	2733K

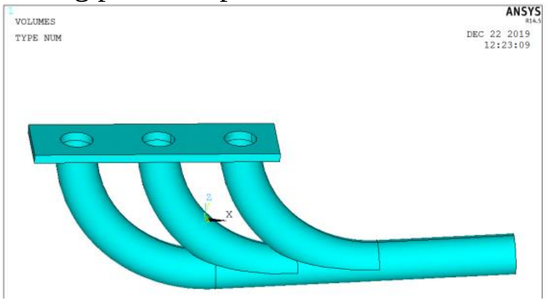


Fig. shows the Infinite model of the modified exhaust manifold

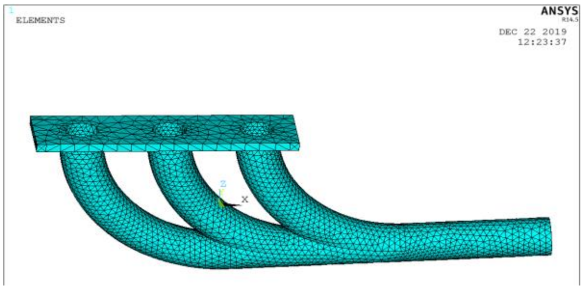


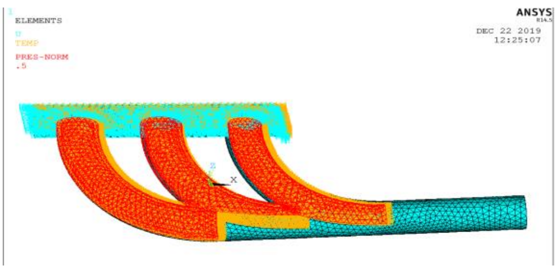
Fig. shows the finite model of the modified exhaust manifold

APPLYING THE BOUNDARY CONDITIONS

In thermal analysis of modified exhaust manifold, we have to apply thermal loads. Temperature 1073K is applied inside of exhaust pipes (4), and convection is applied on modified exhaust manifold.

LOADS:

- Pressure = 500KPa= 500000Pa
- Temperature from engine that is 1073K applied inside of manifoldtubes..
- The bolting locations are arrested in all Dof for modified exhaust manifold.



RESULTS:

Nodal displacements:

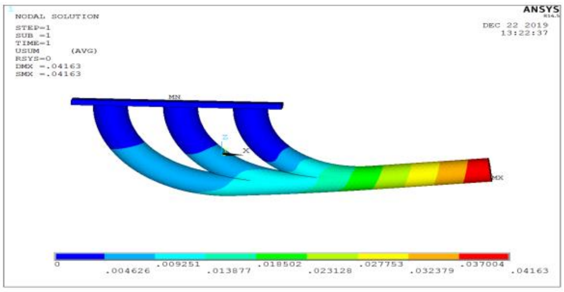


Fig. Shows the Nodal displacements on exhaust manifold

Nodal temperature:

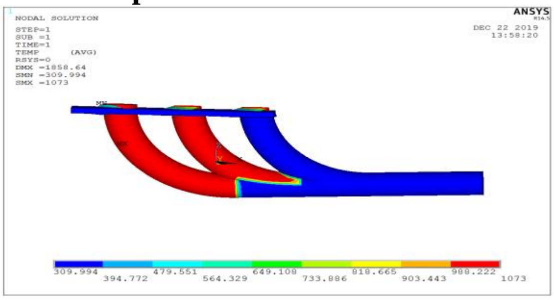


Fig. Shows the temperature distribution on exhaust manifold

Von Mises stress:

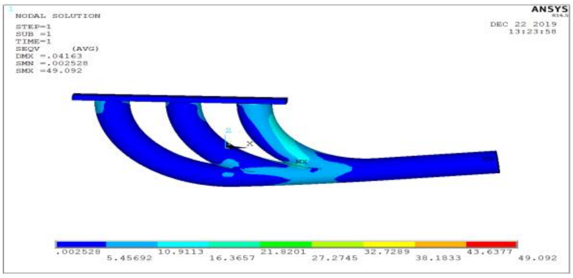


Fig. Shows the Von Misses stress of exhaust manifold

From the above results it is observed that:

- The Max Deflection 0.054mm observed on the exhaust manifold for operating loading conditions.

  The Max Avg. VonMises Stress observed 49.092Mpa on the exhaust manifold for operating loading conditions. And the Yield strength of the materials Carbon steel is 600MPa

RESULTS AND CONCLUSION

In this project we studied couple field analysis of exhaust manifold at maximum temperature and pressure condition by using aluminium and castiron materials. Results are given below:

PARAMETR	MANIFOLD WITH CAST IRON	MANIFOLD WITH ALUMINIUM	MANIFOLD WITH CARBON STEEL
Deformation(mm)	0.22	0.311	0.04
Stress(MPa)	145.145	141.29	49.09
Yield stress(MPa)	600	180	630

Thermal vonmises stress observed 141.29Mpa on the exhaust muffler for operating loading conditions using Aluminium. And the Yield strength of the material Aluminium is 180Mpa . Here 21.5 % stress less formed for Aluminium that of yield strength. Thermal vonmises stress observed 145.145Mpa on the exhaust manifold for operating loading conditions using cast iron. And the Yield strength of the material cast iron is 600Mpa. Here 75.8 % stress less formed for cast iron that of yield strength.

Thermal vonmises stress observed 49.09Mpa on the exhaust manifold for operating loading conditions using carbon steel. And the Yield strength of the material carbon steel is 630Mpa . Here 93.8 % stress less formed for carbon steel that of yield strength. Compare all results, carbon steel formed less thermal stresses during loading conditions on manifold. Thats why carbon steel is a best material for manifold.

REFERENCES

[1] Swathi Satish, Mani Prithiviraj and Sridhar Hari, “Comparison of predictions obtained on an exhaust manifold analysis using conformal and indirect mapped interface”, International Congress on Computational Mechanics and Simulation, IIT Hyderabad, 10 –12 December 2012.

[2] J.DavidRathnaraj, “Thermo mechanical fatigue analysis of stainless steel exhaust manifolds”, IRACST – Engineering Science and Technology, Vol 2, No. 2, April 2012, PP 265 – 267.

[3] M.Rajasekhar Reddy, DrK.Madhava Reddy, “Design and Optimization of Exhaust Muffler In Automobiles”, International Journal of Engineering Research and Applications, Vol 2, Issue 5, Sept-Oct 2012, PP 395 – 398.

[4] Sweta Jain, AlkaBaniAgrawal, “Coupled Thermal – Structural Finite Element Analysis for Exhaust Manifold of an Off-road Vehicle Diesel Engine”, International Journal of Soft Computing and Engineering, Vol 3, Issue 4, September 2013, PP 226-230.

[5] Bin Zou, Yaqian Hu, ZhienLiu, Fuwu Yan and Chao Wang, “The Impact of Temperature Effect on Exhaust Manifold Thermal Modal Analysis”, Research Journal of Applied Sciences Engineering and Technology, August 20, 2013, PP 2824-2829.

[6] A.K.M. Mohiuddin, AtaurRahamn and Mohd. Dzaidin, “Optimal design of automobile exhaust system using gt-power”, International Journal of Mechanical and Materials Engineering, Vol 2 No. 1, 2007, PP 40 – 47.

[7] Modeling and Design of Exhaust Manifold under Thermo mechanical Loading by K.H. Park, B.L. Choi, K.W. Lee, K.-S.Kim, and Y.Y. Earmme.

[8] Masahiro Kawasaki, Masashi Morikawa, “Multiobjective Design Optimization of Merging Configuration for an Exhaust Manifold of a Car Engine”, Parallel Computational Fluid Dynamics, International Conference parallel, Kyoto, japan , may 2002