Thermo Mechanical analysis of Cylinder Liner

[1]Shekhar Shinde,[2] Dr. P.V.Jadhav,[3]D.B.Jadhav

Department of Mechanical Engineering, Bharati Vidyapeeth Deemed University, College of Engineering,Pune.

ABSTRACT - Many times it is difficult to analyze the actual working conditions and problems created which may lead to development of stresses and thus may cause failure, also it's difficult to find the stresses developed in each section of component thus the, material required to added or subtracted not easily analyzed. Even material properties can't be considered only by mathematical modeling without virtual analysis.Hence Finite Element Analysis is done by using different software's such as ANSYS and NASTRAN. This makes it easy to design a component for more durability with help of virtual work by knowing its different physical condition in actual working.In this Case study thermo-mechanical (coupled filed) analysis on cylinder liner of 4-stroke diesel engine of SANTRO vehicle of Hyundai Motors performed.

Keywords— Ansys, Nastran, Santro, Thermomechanical, Cylinder, Liner

INTRODUCTION –

FEA is a numerical procedure that can be used to obtain solution to large class of engineering problems involving stress analysis, heat transfer, electro magnetism and fluid flow.A finite element model is a discrete representation of the continuous, physical part that is being analyzed. This discrete representation is created using nodes and elements. Nodes are connected together to form elements. The nodes are the discrete points on the physical part where the analysis predicts the response of the part due to applied loading. This response is defined in terms of nodal degrees of freedom (DOF). For stress analysis, up to six degrees of freedom are possible at each node (three components of translation and three components of rotation). Depending on the element type selected (e.g., beam, plate, 2-D and 3-D elements, etc.), the number of required degrees of freedom at each node is determined.

Phases in Finite Element Analysis:

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

1. Pre-processing: Defining the finite element model and environmental factors to be applied to it.
2. Analysis solver: Solution of finite element model.
3. Post-processing of results using visualization tools.

VEHICLE SPECIFICATION: -

Various engine specifications of SANTRO vehicle required for this case study are as follows: -

Engine Type/Model: -	Hyundai Epsilon
Combustion principle	4-stroke
Engine Displacement: -	999cc
Maximum Horsepower (PS @rpm): -	55 PS @ 5500 rpm
Maximum Torque (Nm @rpm): -	84 Nm @ 2500 rpm
Bore (mm): -	66 mm
Stroke (mm): -	77 mm
Compression Ratio: -	8.9:1

Configuration and No of Cylinders: - In-Line 4Cylinder 12 Valves SOHC

Cylinder Block: - Iron

Cylinder head: - Aluminum

Fuel system: - Multipoint fuel injection

Performance: - Maximum Speed 141 Km/Hour 0-100kmph 16.9 seconds 1/4 Mile 20.6 seconds

Material for Liner-

For increasing some properties of cast iron some material is alloyed as shown in below:

Table 1:Properties of material

Component	Percentage
• Total carbon	3.10-3.40%
• Combined carbon	0.75-0.90%
• Manganese	0.55-0.75%
• Phosphorus	0.20 max.%
• Silicon	1.90-2.10%
• Nickel	1.80-2.20%
• Chromium	0.55-0.75%

Nickel-Chromium Iron - composition: The high nickel content of this alloy provides excellent resistance to chloride-ion stress corrosion cracking and imparts resistance to corrosion by a number of organic and inorganic compounds. chromium gives this alloy its resistance to oxidation at temperatures up to 2150°f (1175°c).combines high strength with desirable workability. it has excellent mechanical properties from sub-zero to elevated temperatures.[7].

CAD Model-

General purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis, heat transfer and fluid problems, as well as acoustic and electromagnetic problems. ANSYS, finite element analysis software enables engineers to perform the following tasks:

• Build computer models or transfer CAD models of structures, products, Components or systems.

• Apply operating loads or other design performance conditions.

• Study physical responses such as stress levels, temperature distributions or electromagnetic fields.

• Optimize a design early in the development process to reduce production costs.

• Do prototype testing in environments, where it otherwise would be undesirable or impossible (for example, biomedical applications).

Import CAD model in a ANSYS software for analysis.

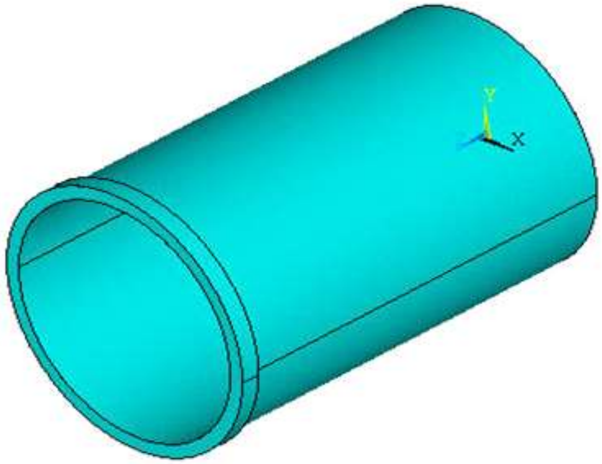


Fig 1: Importing CAD model

Mesh Model-

A given problem is discredited by dividing the original domain into simply shaped elements. All the elements are connected to each other by nodes called as mesh model. Here we are using the Brick solid 45 elements for structural analysis.

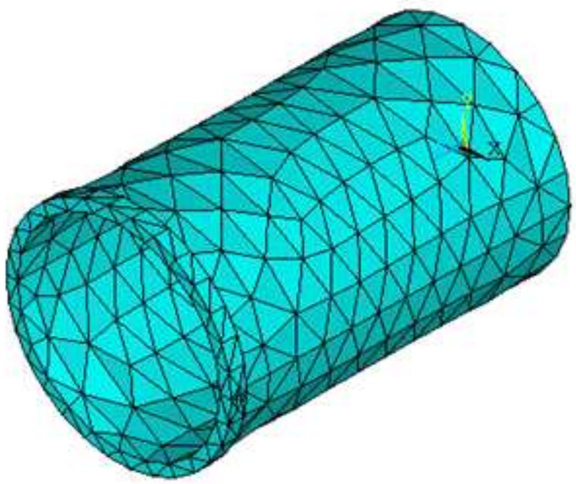


Fig 2: Mesh model

Boundary Conditions-

While performing the analysis the degrees of freedom have to be restricted. Here all degree of freedom is restricted for the structural analysis In the ANSYS the Surface loads are typically pressures for structural element types, convections or heat fluxes for thermal element types, etc.We are considering the 60 bar pressure on inside surface of cylinder liner.

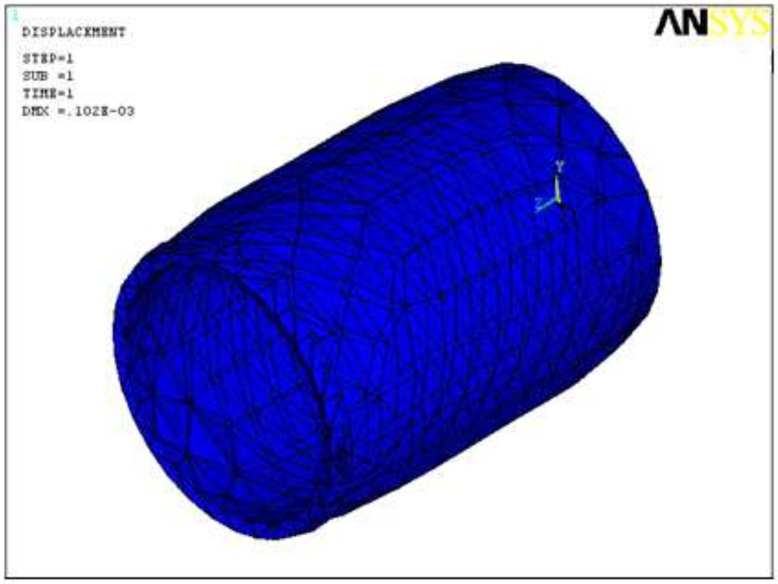


Fig 3:Deformation of liner due to high pressure

The pressure is uniform inside the liner therefore the deformation of liner is 0.102e-3 mm

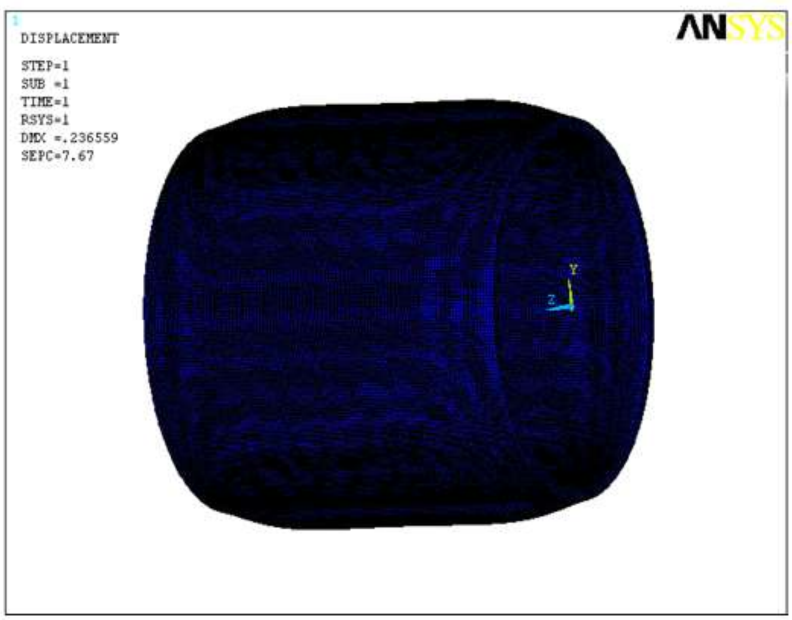


Fig 4:Deformation due to Thermal Stress

Due to temperature difference the deformation occur is 0.236559 mm

OPTIMIZATION OF DESIGN

Optimization deals with the study of minimization or maximization. Design optimization are basically depends upon number of parameters which are cost, size, strength, weight, reliability and performance. In that case if the designer select weight as an optimization criteria then the overall analysis is performed to solve the problem for weight minimization. For that we perform the stress analysis for weight minimization of that part. The stress analysis performed on that part gives idea that if we remove certain amount of structure of that component. While performing these stress analysis we take care of overall factor of safety requirement and load acting on that critical component.

One more key point to keep in mind that engineering system assembled by number of component ,optimizing assembly component that makes up the system did not require lead to an optimize overall system. This study shows basic techniques for design optimization of mechanical system. In this paper we define weight as crucial optimization criteria. Previously the improvement in design initiate from starting with basic design ,analysis on that initial design ,observing on results and conclude whether or not can we modify the initial design. From some previous years the optimization process changes from linear programming to non linear programming techniques.

Optimization Statement and Constraints:

The structural and gas load is major load coming on the cylinder liner hence the objective of the optimization was to minimize the mass of the crankshaft under the effect of static load comprising the peak gas pressure load, such that equivalent stress amplitude are within the limits of allowable stress. The optimized cylinder liner has to be interchangeable with existing one in current engine. Each of these requirements or constraints, are now briefly discussed.

ANALYSIS ON LINER FOR OPTIMIZATION

For optimization two cases are taken as follows: -

Case 1: - In this case the thickness of cylinder liner is reduced up to 1.5 from 2.31 so hoop stress increased to 144.216 N/mm2

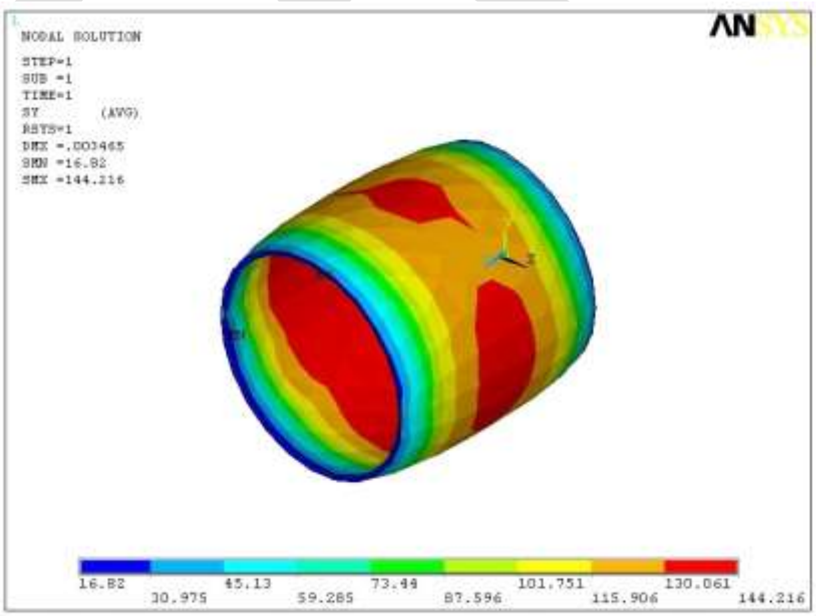


Fig 5: Hoop stress for reduced thickness

In this case study the thickness of cylinder liner is reduced upto 1.5 from 2.31 .the main objective of these case study I(a) was to analyze the various stresses acting on the cylinder liner. The variation of hoop stress, combined stress and maximum thermal stress and effect of these stresses on cylinder liner geometry are observed. The red color indicates the maximum hoop stress impacting area.

So finally from this ANSYS result it is observed that as the thickness of liner reduced the hoop stress acting on cylinder liner increased .In this case the thickness of cylinder liner is reduced up to 1.5 from 2.31 so hoop stress increased to 144.216 N/mm^2.

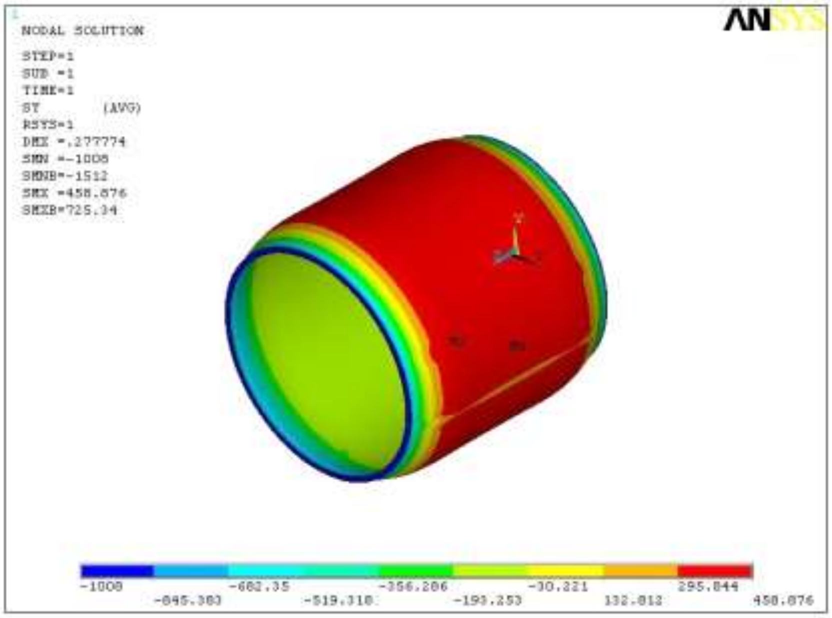


Fig. 6: Combined stress for reduced thickness

In this case study the thickness of cylinder liner is reduced upto 1.5 from 2.31 .the main objective of these case study I(b) was to analyze the various stresses acting on the cylinder liner. The variation of hoop stress, combined stress and maximum thermal stress and effect of these stresses on cylinder liner geometry are observed. The red color indicates the maximum combined stress impacting area. So finally from this ANSYS result it is observed that as the thickness of liner reduced the combined stress acting on cylinder liner increased .In this case the thickness of cylinder liner is reduced up to 1.5 from 2.31 so combined stress increased to 725.34 N/mm^2.

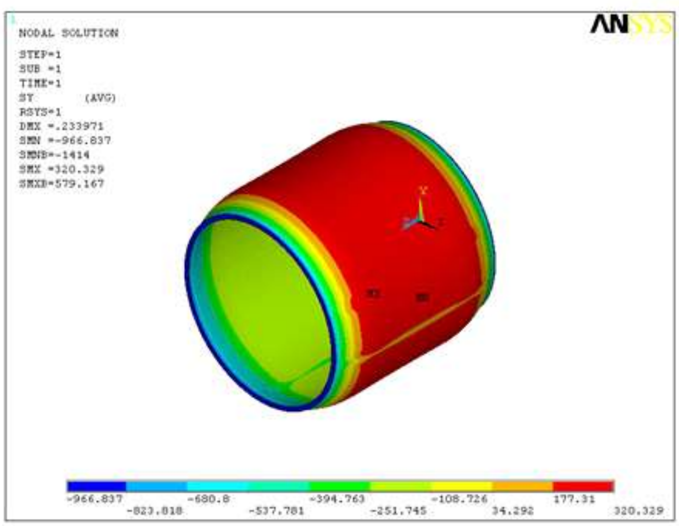


Fig 7 :Max thermal stresses for reduced thickness

In this case study the thickness of cylinder liner is reduced upto 1.5 from 2.31 .the main objective of these case study I(c) was to analyze the various stresses acting on the cylinder liner. The variation of hoop stress, combined stress and maximum thermal stress and effect of these stresses on cylinder liner geometry are observed. The red color indicates the maximum thermal stress impacting area. So finally from this ANSYS result it is observed that as the thickness of liner reduced the maximum thermal stress acting on cylinder liner increased .In this case the thickness of cylinder liner is reduced up to 1.5 from 2.31 so maximum thermal stress increased to 579.167 N/mm^2.

Case 2: Taking thickness equal to 5 mm the results obtained are as follows

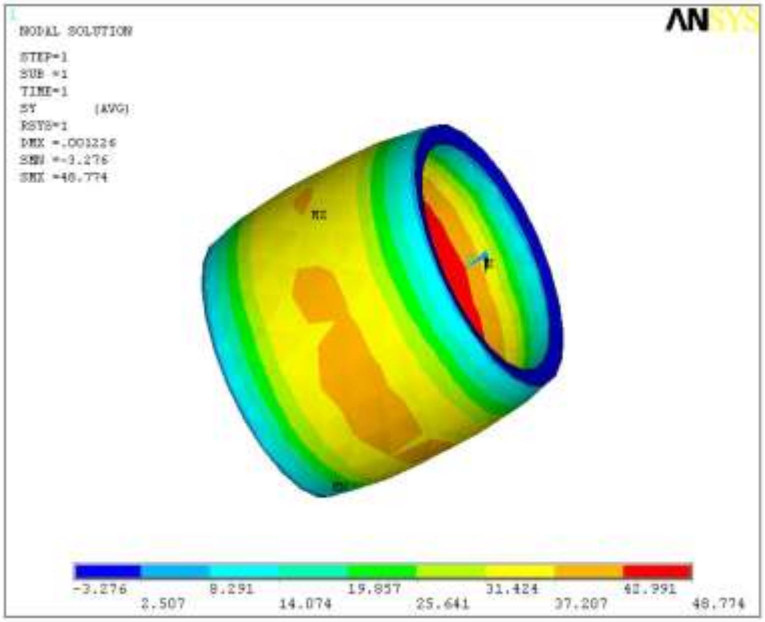


Fig 8 : Hoop stress for thickness 5 mm.

In this case study the thickness of cylinder liner is incresed upto 5mm from 2.31.The main objective of these case study II(a) was to analyze the various stresses acting on the cylinder liner. The variation of hoop stress, combined stress and maximum thermal stress and effect of these stresses on cylinder liner geometry are observed. The red color indicates the hoop stress impacting area. So finally from this ANSYS result it is observed that as the thickness of liner increased the hoop stress acting on cylinder liner decreased .In this case the thickness of cylinder liner is increased up to 5mm from 2.31 so hoop stress decreased to 48.774N/mm^2.

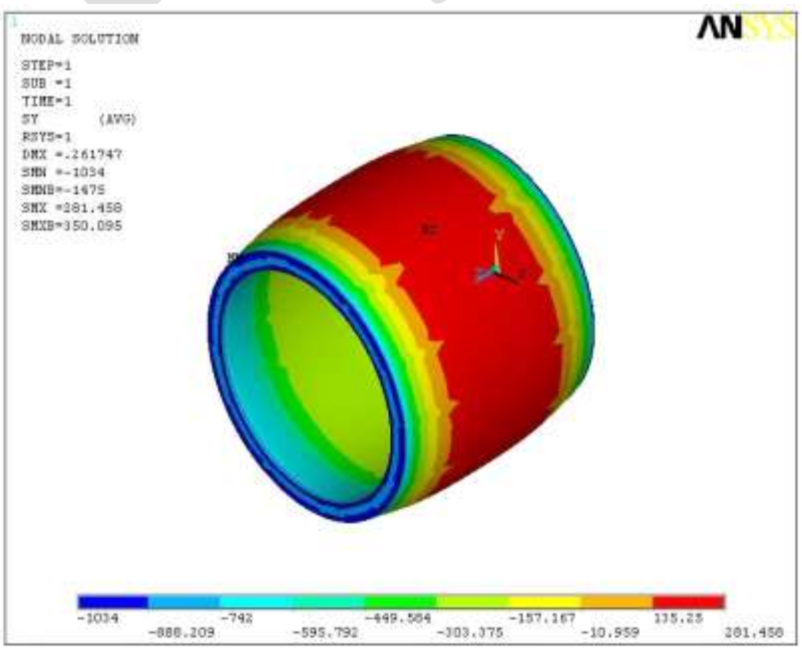


Fig 9 :Max. thermal stress for 5 mm

In this case study the thickness of cylinder liner is incresed upto 5mm from 2.31.The main objective of these case study II(b) was to analyze the various stresses acting on the cylinder liner. The variation of hoop stress, combined stress and maximum thermal stress and effect of these stresses on cylinder liner geometry are observed. The red color indicates the Max. thermal stress impacting area. So finally from this ANSYS result it is observed that as the thickness of liner increased the Max. thermal stress acting on cylinder liner decreased .In this case the thickness of cylinder liner is increased up to 5mm from 2.31 so Max. thermal stress decreased to 281.458N/mm^2.

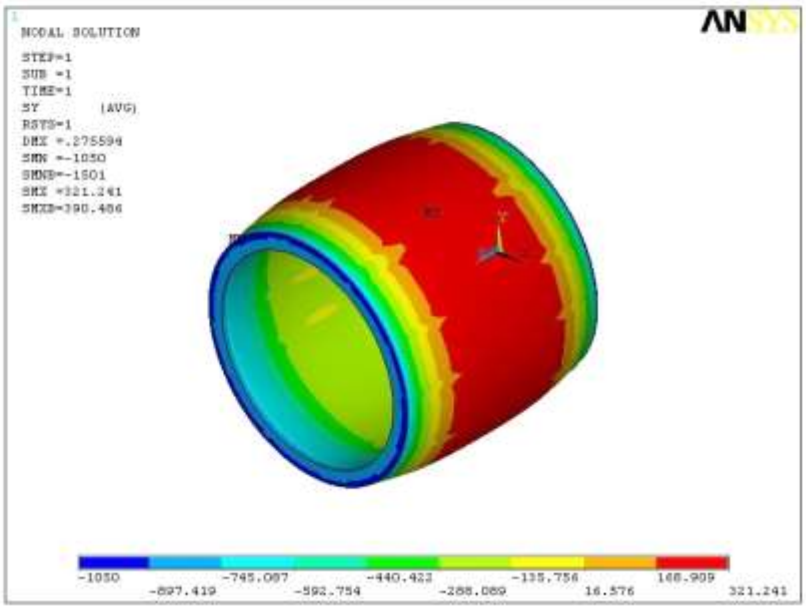
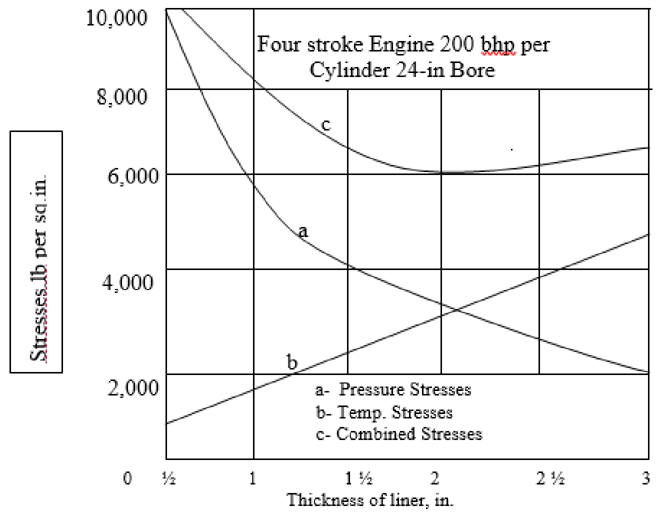


Fig 10 :Combined stress for thickness 5 mm.

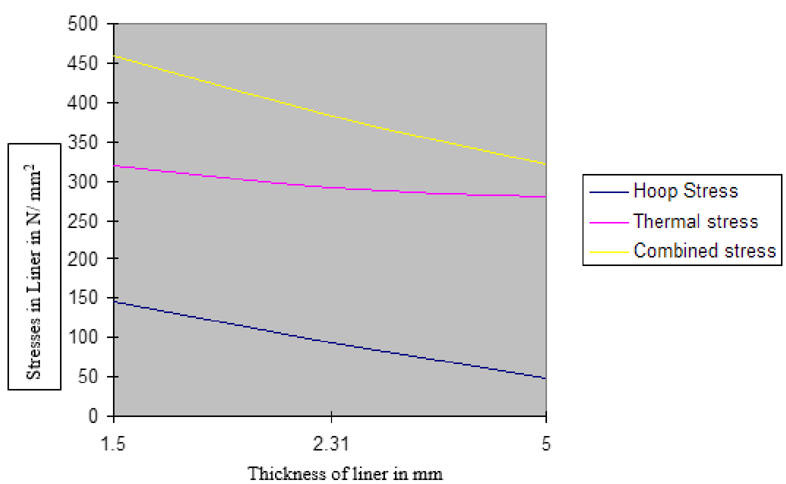
In this case study the thickness of cylinder liner is incresed upto 5mm from 2.31.The main objective of these case study II(c) was to analyze the various stresses acting on the cylinder liner. The variation of hoop stress, combined stress and maximum thermal stress and effect of these stresses on cylinder liner geometry are observed. The red color indicates the Combined stress impacting area. So finally from this ANSYS result it is observed that as the thickness of liner increased the Combined stress acting on cylinder liner decreased .In this case the thickness of cylinder liner is increased up to 5mm from 2.31 so Combined stress decreased to 321.241 N/mm^2.For increased thickness of liner Hoop stress, Maximum thermal stress, and combined stress gets reduced.

International Journal of Engineering Research and General Science Volume 4, Issue 3, May-June, 2016
ISSN 2091-2730

Result



Graph 1Actual graph for stresses in liner



Graph 2 Practical graph for stresses in liner

CONCLUSION

- The thermal analysis of cylinder liner component is done by using:

- Solid 45 brick element → structural analysis
- Solid 70 brick element → thermal analysis
- The outer surface of cylinder liner is under maximum stress
- From case study I it is concluded that

By reducing the thickness of cylinder liner component hoop stress, maximum thermal stress and combined stress increase

- From case study II it is concluded that

For increased thickness of cylinder liner, hoop stress, maximum thermal stress and combined stress gets reduced.

Future scope for the project:

- For the analysis of cylinder liner
    - Solid 45 brick element → structural analysis
    - Solid 70 brick element → thermal analysis
- The future scope is
    - 20 node solid 90 → thermal analysis
    - 20 node solid 92 → structural analysis
- In this work we use linear method because of isotropic material property by assuming alloying elements no linear methods can be used.
- Advanced Materials for Liner can be used such as
- GOE323 (GJL) is a micro alloy cast iron with flake graphite.
- GOE330 (GJV) is a compacted graphite cast iron and belongs to the group of ductile cast irons.
- We have performed the Analysis by using Sequential Coupled Analysis –ANSYS Multi-field solver; it can also be performed by using Direct Coupled-Field Analysis.

ACKNOWLEDGEMENT

The authors would like to acknowledge the management of Bharati Vidyapeeth Deemed University,College of Engineering, Pune for providing academic and technical support throughout this research work. Authers would also like to thanks Mr. Balaji Hawaldar, (Bharat Forge) Pune, Mr. Pradip Mandake, BVDU College of Engineering, Pune for his kind support during the course of this research work.

REFERENCES:

[1] Mangonon PL. Materials selection for engineering design. USA: Prentice Hall, 1999.
[2] American Society for Testing and Materials. Annual book of ASTM standards, Vol. 0301, EA-247. ASTM, 1992.
[3] IIT Research Institute. Failure analysis of metallic materials by scanning electron microscopy. Chicago (USA) 1979.
[4] Branco CM, Infante V, Sousa e Brito A, Martins RF. A failure analysis study of wet liners in maritime diesel engines.Eng Fail Anal 2002;9:403–21.
[5] Hormaza W, Mateus L, Maranon A. Failure analysis of a cylinder sleeve from a turbocharged diesel engine. Eng Fail Anal 2008. doi:10.1016/j.engfailanal.2008.09.010.
[6] Xu XL, Yu ZW. Failure analysis of a diesel engine connecting rod. J Fail Anal Prevent 2007;7:316–20.
[7] Rabb R. Fatigue failure of a connecting rod. Eng Fail Anal 1996;3:13–28.
[8] Griza S, Bertoni F, Zanon G, Reguly A, Strohaecker TR. Fatigue in engine connecting rod bolt due to forming laps. Eng Fail Anal 2008. doi:10.1016/j.engfailanal.2008.10.002.
[9] Silva FS. Fatigue on engine pistons – a compendium of case studies. Eng Fail Anal 2006;13:480–92.
[10] Yu Z, Xu X, Ding H. Failure analysis of a diesel engine piston-pin. Eng Fail Anal 2007;14:110–7.
[11] Heywood JB. Internal combustion engine fundamentals. Singapore: McGraw-Hill Book Company; 1998
[12] L. L. Ting and J. E. Mayer, J. L&r. Tec~no~, 96 (April 1974) 258 - 266.
[13] M. J. Neale and T. S. Eyre, Paper C7/82, 1982, pp. 55 - 64 (Institution of Mechanical Engineers, London).