International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

Analysis on the Effect of Temperature Distribution on Exhaust Manifold using FEA Tools

Akshay Pujar M
7th Semester, Department of Automobile Engineering
BVBCET
Hubli, India

Tejas G Shirali
7th Semester, Department of Automobile Engineering
BVBCET
Hubli, India

Chetan Kulkarni
7th Semester, Department of Automobile Engineering
BVBCET
Hubli, India

Suneel Katti M
7th Semester, Department of Automobile Engineering
BVBCET
Hubli, India

Abstract—**An exhaust system carries waste gases and other combustion products away from an automobile engine. It allows the vehicle to operate with minimal noise, smoke and pollution transmitted to the environment. The exhaust discharges the gases at a very high temperature, thus it is important to study the distribution of temperature along the whole exhaust for its effective working.**

In this study, the impact of temperature effect on the exhaust of an automotive is scrutinized. The objective of the analysis is to find the stresses induced in the manifold due to thermal growth. Firstly, the distribution of temperature giving rise to thermal stresses which are encountered due to varying load conditions in the manifold is analyzed using ANSYS 10.0with temperature field boundary conditions. Secondly, the same is analyzed analytically and after comparison it is found that the error which lies between the two results in less than 10%.

Keywords-*Temperature Distribution; thermal stress; ANSYS 10.0;*

I. INTRODUCTION

Exhaust manifold is a part of automotive engines which are required to collect the exhaust gases from the cylinder head and send it to the exhaust system. The exhaust manifold plays an important role in the performance of an engine system. Particularly, the efficiencies of emission and the fuel consumption are nearly related to the exhaust manifold.Exhaust Manifolds are affected by thermal stresses and deformations due the temperature distribution, heat accumulation or dissipation and other related thermal quantities.Finite Element Analysis (FEA) involves the solution of simultaneous and algebraic equations. The algebraic equation result from subdividing a complex shape into many discrete, interconnected, simple shapes. Finite elements allow us to simulate a wide variety of physical phenomena, encompassing mechanical, electrical and even chemical circumstances. Some examples of the phenomena, one can simulate using FE methods include Structural, Fluid, Dynamic and Thermal.The ideal analysis consists of two phases, a technique validation phase and an optimization phase.The technique validation phase verifies that the modellingdeformation, high strains and impact loading. Additionally, the physical testing may require careful refinement to represent actual operating conditions. The optimization phase simply establishes a baseline, and iterates on design changes until it meets the specified design criteria. Although it should be a goal, the full analysis process may not be realistic for many design cycles. Many times the technique validation phase is skipped. This can affect the confidence of a particular solution, thus affecting the test requirements and factors of safety. Additionally, multiple iterations are not timely enough for some schedules. What is left is a single iteration analysis with no validation and no reference point. This is undesirable, but it is better to perform one analysis than none. Ultimately, each omission of the total ideal FE analysis cycle results in lost confidence.

II. THE PROCESS OF ANALYSIS

The finite element method deals easily with rather general material properties and with both mechanical and thermal loading. Here in the paper an attempt has been made to find the critical regions where the stress concentration is more due to temperature distribution on the model resulting to thermal stresses which in turn influence these mechanical stresses and lead to stress concentration at a particular region resulting in a fissure which slowly and steadily propagates to cause the failure of the complete component.In the current work the model of interest is an exhaust manifold of 3 cylinder dieselengine.Here the manifold is analyzed with reference to stresses developed at various regions and the temperature distribution over the surface of the manifold, which is being compared with the analytical approach. The primary failures of the manifold are found to be caused by thermal growth and include yielding the manifold.As with any analysis, many assumptions must be made and referenced when interpreting the results. The following are some of the assumptions made for this analysis:

1. Thin shell elements are adequate for this thin wall geometry
2. Steady state heat transfer
3. Linear static analysis
4. Temperature dependent isotropic material properties

The key to any analysis is the loading and boundary conditions. This is a structural analysis in which the only loads on the structure are due to thermal growth. The difficulty in this case has been applying an accurate distribution of temperatures to the model. Here the model is being generated in CATIA V5R19 then it is being imported to the ANSYS 10.0 for analysis, to know the behavior of the model with the varying thermal conditions. The manifold is meshed finely on critical regions using element type Shell 131. First the initial thermal analysis is done, and then the results of the thermal analysis are used in structural analysis to find out the uniaxial stresses, stress concentration areas, and the temperature distribution in the model.Temperature dependent material properties including thermal conductivity were added to the model. Convection and radiation effects were excluded from the analysis. Due to the large range of temperatures that the manifold is subjected to, temperature dependent properties were critical to this analysis. Although the data on the exact alloy used in the manifold design was found out from the ASTM handbook, wherein the alloy was identified to be an annealed condition of wrought iron which was used for this analysis. A density of 7.6687 g/mm^3 was included for all temperatures. Temperature dependent Young's modulus, Poisson's Ratio, Shear Modulus, and coefficient of thermal expansion have been utilized in the structural analysis. Temperatures applied from the thermal analysis were used to determine the uniaxial stresses during the structural analysis of the manifold.

Temp 'T' (K)	**Young's Modulus 'E' (N/mm^2)**	**Poisson's ratio 'v'**	**Shear Modulus 'G' (N/mm^2)**	**Coefficient of thermal expansion 'α' (mm/mm-K)**	**Thermal Conductivity 'k' (N-mm/s-mm-K)**
294.2	2.036e5	0.28	7.926e4	7.03e-6	155.769
588.7	1.961e5	0.29	7.038e4	1.099e-5	276.927
699.8	1.715e5	0.30	6.594e4	1.119e-5	282.116
922.0	1.346e5	0.31	5.179e4	1.17e-5	320.19
1033	1.168e5	0.32	4.407e4	1.206e-5	285.578
1116	0.8951e5	0.33	3.32e4	1.24e-5	259.618

The temperature distribution in a part can cause thermal stress effects. These thermal stress effects can be simulated by coupling a heat transfer analysis (steady-state or transient) and a structural analysis (static stress with linear or non-linear material models). The process consists of two basic steps:

1. A heat transfer analysis is performed to determine the temperature distribution.
2. The temperature results are directly input as loads in a structural analysis to determine the stress and displacement caused by the temperature loads.

Thus, the ability to couple heat transfer and structural analysis capabilities provides an easy and convenient way to simulate thermal stress effect. Here a couple field analyses are performed wherein the thermal loads are initially computed in the thermal analysis and therein it is subsequently applied into the structural analysis to calculate different results. In this analysis the material properties for temperatures 450 K, 500K, 550K and 600K was found by the interpolation method and then the analysis was carried out in ANSYS 10.0 using the interpolated values which was obtained after interpolation for the desired temperature as described above.

III. RESULTS

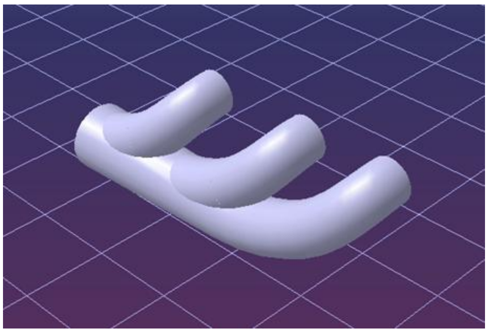

Fig1. Model generated in CATIA V5 R19.

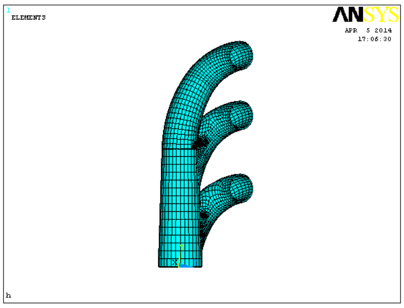

Fig2. Elements created after meshing

TABLE I. CASE1.

Temperature at 1st runner	350
Temperature at 2nd runner	400
Temperature at 3rd runner	450

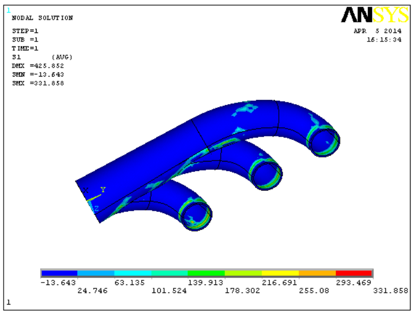

Fig3. Uniaxial stress distribution in the manifold

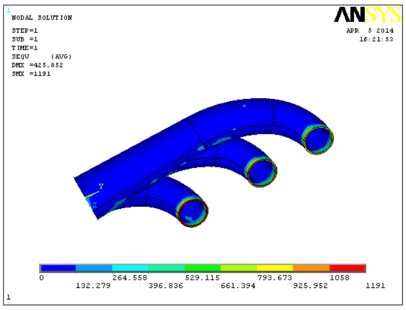

Fig4. Von Mises stress of the manifold

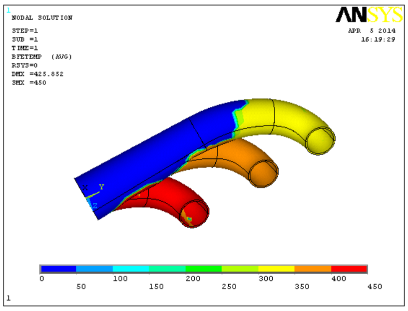

Fig5. Temperature Distribution

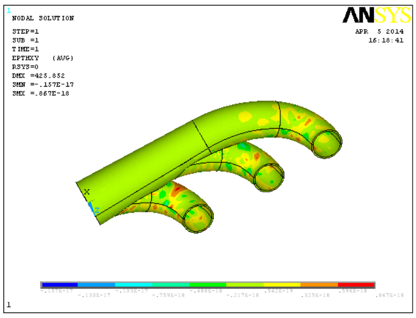

Fig6. Thermal Shear stress

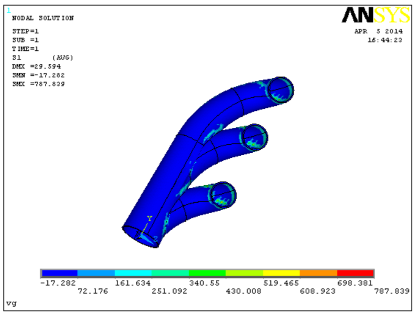

Fig7. Uniaxial stress distribution in the manifold

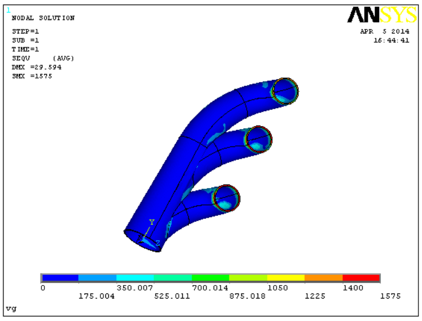

Fig8. Von Mises stress of the manifold

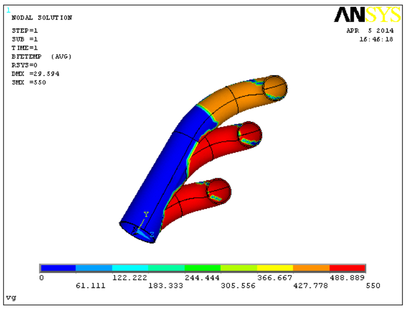

Fig9. Temperature Distribution

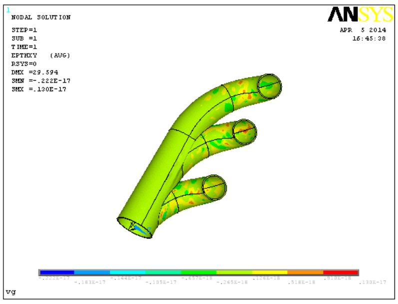

Fig10. Thermal Shear stress

TABLE II. CASE 2

Temperature at 1st runner	450
Temperature at 2nd runner	500
Temperature at 3rd runner	550

IV. ANALYTICAL APPROACH

Analytical approach for finding uniaxial stress:

1. Maximum uniaxial thermal stress for 1st case:

$$\sigma_{x,1} = \frac{E\alpha}{1-\nu}(T_{f,1-}T_i) \qquad (1)$$

$$= \frac{2.036*10^5*7.03*10^{-6}(450-294.26)}{1-0.28}$$

$= 311.711 \text{N/mm}^2$

E= Young's Modulus

T_i = Initial temperature

T_f = Final temperature

2. Maximum uniaxial thermal stress for 2nd case:

$$\sigma_{x,2} = \frac{E\alpha}{1-v}\left(T_{f,2} - T_i\right) \qquad (2)$$

$$= \frac{1.97*10^5*1.035*10^{-5}(550-294.26)}{1-0.28}$$

$$= 724.22 \text{ N/mm}^2$$

V. VALIDATION OF RESULTS

The results obtained from ANSYS 10.0 and by Analytical approach are clearly stated in the table given below.

	ANSYS 10.0 Results (N/mm^2)	**Results from Analytical approach (N/mm^2)**
1st iteration results for uniaxial stress	331.858	311.71
2nd iteration results for uniaxial stress	787.839	724.22

The results from the above thermal analysis are said to be accurate, when we compared these results with analytical results. And the error, which lies between these two results, is within the limit, i.e. it is about less than 10% error between two results. The error may be due to the assumptions, which we made in the analysis, or due to constraints in the software.So from the above discussion it is clear that maximum stress induced in the manifold is due to structural as well as thermal loads, hence by choosing lower weight materials, increasing the fillet radius, runner lengths should be increased and using materials having lower thermal conductivity. The cracks may develop on the manifold because of maximum temperature distribution. It can be reduced by increasing the thickness or by proper design of the manifold. All these can be taken care at the time of optimization of the model.

VI. CONCLUSION

1. From the above analysis the temperature distribution on the manifold and maximum uniaxial stress on the manifold are identified.
2. Moreover the critical regions are clearly identified from the above Couple field analysis.
3. The various places in the manifold were identified where stress concentration was more. The regions which are mostly affected are the runners and at the areas that are fixed to the bracket. These were found to be critical sections as they are constrained in all degrees of freedom.

VII. FUTURE SCOPE

1. The analysis, which has been carried out presently, is static linear analysis; because of this errors are found between two results.
2. If the same is carried under transient analysis (time dependent), the error between the results may reduce.
3. Many design changes can be performed by including changes in material, wall thickness, runner length and the inclusion of ribs and connecting bars.
4. A combination of increasing the runner length, wall thickness and fillet radii is expected to help in reducing stress levels.
5. The optimization of the manifold design can be done for reducing the stresses and maximum temperature distribution in the manifold by using ANSYS 10.0software.

REFERENCES

[1]Young Warren C, "Roark's Formulas for stress and strain" , 6th Ed., McGraw Hill Company, 1989.

[2] Zienkiewicz, O. C., "The Finite Element Method" by McGraw Hill Book Company (U.K.) Ltd., London, 1977.

[3] Shigley, Joseph E., Mischke, Charles R. "Standard Handbook of Machine Design" by, McGraw Hill Inc, 1986.

[4] Cook, Robert D.,"Concepts and Application of Finite Element Analysis"by. McGraw Hill, 1989.

[5] SrinathL.S.,"Advanced Solid Mechanics" by, Tata McGraw Hill Inc, 1998.

[6] Giri N.K., "Automotive Mechanics" by, Khanna Publications, 1986.

[7] "SAE Technical Papers Series", 2003-01-0940.

[8] "ASTM Metals Handbook", by Brady and George Stuart, McGraw Hill publications, 1998.