THERMOMECHANICAL FATIGUE ANALYSIS OF STAINLESS STEEL EXHAUST MANIFOLDS

J.David Rathnaraj
Professor in Manufacturing Engineering (PG)
Sri Ramakrishna Engineering College
Coimbatore – 641 022, India

***Abstract*— Automotive engine operates under severe thermo mechanical loading condition, the operating temperature increases up to 800^{o}C from ambient temperature and large thermal stress is induced by temperature gradient and geometrical constraints. This thermo mechanical coupling is one of critical problems in automotive engineering. For instance, most of the crack found in the stainless steel exhaust manifold is caused by out-of-phase thermal fatigue occurring at high temperature. Hence, thermal fatigue analysis should be considered in the design process of the exhaust manifold. Since classical model of plasticity and life prediction is only applicable to isothermal condition, a new methodology for thermo mechanical condition is necessary. In order to reduce the pollutant emission of vehicles, exhaust systems are becoming more advanced which includes advanced technical components such as catalytic converters, particle filters and low weight manifolds. The reduction in fuel consumption, which is another way to limit emissions, leads to hotter exhaust fumes. Thus the durability of exhaust systems, specially corrosion and Thermo Mechanical Fatigue (TMF) resistance, needs to be improved significantly by the use of stainless steel rather than cast iron. In this paper the focus is on the application of constitutive equation to the thermo mechanical condition of a model based on isothermal data. Using the proposed model, the thermal stress analysis and life prediction of exhaust manifold made of 429EM stainless steel is done.**

Keywords- Thermo Mechanical Fatigue (TMF), Xhaust-LIFE, ABAQUS, Constitutive Equation, Continuous Damage Model

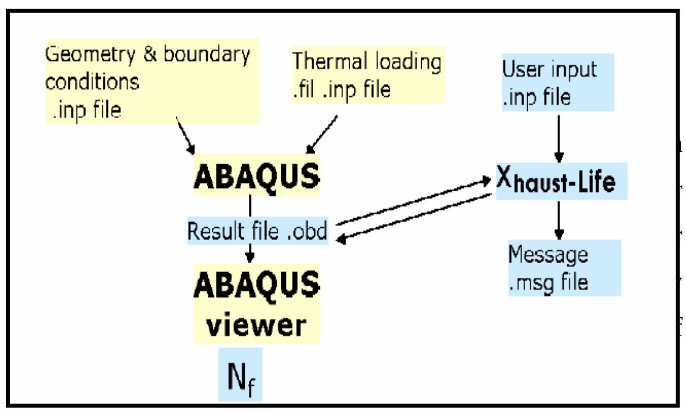
Thermo Mechanical Fatigue (TMF)

Before running the Finite Element Analysis of the manifold under TMF, thermal loading must be defined. This means that the evolution of the temperature at every points of the structure must be computed from gas flows inside and outside the manifold. Based on the thermal loading, the TMF analysis aims to compute the stress and strain response of the structure. More than the cyclic temperature, the simulation, based on the Finite Element Method, needs the geometry and mesh of the part, the boundary conditions and a realistic constitutive model. The last step of the simulation consists of the lifetime prediction. The model used is based on Manson-Coffin curves obtained from non-isothermal tests.

The formatter will need to create these components, incorporating the applicable criteria that follow.

Xhaust-LIFE: A post processor for ABAQUS

A dedicated tool, called Xhaust-LIFE, has been applied to the predictive model after an ABAQUS simulation. The figure shows the links between Xhaust-LIFE and ABAQUS. Xhaust-LIFE consists of reading the data recorded in the "Output Data Base" (ODB) file created by ABAQUS, and finally writes the results in the (ODB) file so that the visualization can be done with ABAQUS Viewer. The calculation can use material parameters defined by the user or materials in the library.



Constitutive Equation

Among the cyclic plasticity models, overlay model have been considered as physically motivated, as many real materials or mechanical systems can be thought of as having a similar structure. For example, most of the engineering materials may have a crystalline structure that is made of a distribution of slip-planes or dislocations of different slip strengths. The overlay model has definite advantages over the classical models. The varieties of effects which cannot be described in the classical models can be done in the overlay model, whereby the description remains fairly simple and qualitative. The basic assumption of overlay model is that material is composed of a series of so-called Jenkin's elements connected in parallel, each of which consists of a linear spring with stiffness.

Continuous Damage Model

The Morrow model is suitable for life prediction of 429EM stainless steel since continual cyclic hardening is observed from isothermal low cycle fatigue. The relation of the isothermal fatigue lifetime with plastic strain energy density is

$$\Delta w_p \left(N_f\right)^m = C$$

Finite Element Analysis

The thermal stress analysis was performed in ABAQUS/Standard. The thermo mechanical overlay model and TMF damage model was implemented into ABAQUS through a UMAT subroutine. The assembly was considered as a four-tube exhaust manifold fastened to an engine head by eight bolts acting on four flanges. The materials of engine head and bolt are aluminum and steel respectively. The bolts and engine head were modeled as elastic materials; the manifold was modeled as an elastic-plastic, temperature-dependent material described by thermo mechanical overlay model. The analysis consists of two parts, heat transfer analysis and thermal stress analysis. The temperature distribution obtained from transient heat transfer analysis was used as an input of the thermal stress analysis. The stress analysis consists of three steps. First, prescribed bolt loads fasten the manifold to the head at room temperature. Then, the assembly is heated to an operating condition, shown in Fig.1, and is cooled to a uniform ambient temperature. After repeating heating cooling cycle two times, the damage contour was obtained as shown in Fig.2 and the critical region was found in the junction of tube. As the calculated damage is strongly depending on the mesh size, the fatigue damage calculated at the centroid of element was used to predict the crack initiation life. As the linear damage evolution was assumed in continuous damage model, it is reasonable to calculate the expected life from the reciprocal of damage. The expected life of exhaust manifold was 5500 ~ 6000 cycles, which is very short when compared to the actual life. The accelerated life may be caused due to the severe loading condition used in the analysis and low resistance to thermal fatigue by 429EM stainless steel.

CONCLUSION

- An accurate elastoviscoplastic behavior model has been identified and is now available to simulate the stresses and strains in a part submitted to thermal fatigue.
- The use of a virtual thermo mechanical fatigue design approach permits to optimize the design of the manifold and limits both the number of prototype and motor bench tests and, finally, reduces the risk of failure.
- In this work a methodology for reliability assessment under thermo mechanical condition using constitutive equation and damage model based on isothermal data has been developed.
- By considering geometrical relation in hysteresis loop and temperature dependence of elastic limit, temperature dependent overlay model was proposed.
- The concept of damage conversion factor was introduced to calculate the TMF damage.
- Using proposed models, thermal stress analysis and life prediction of exhaust manifold was performed and expected life was determined through the simple procedure.

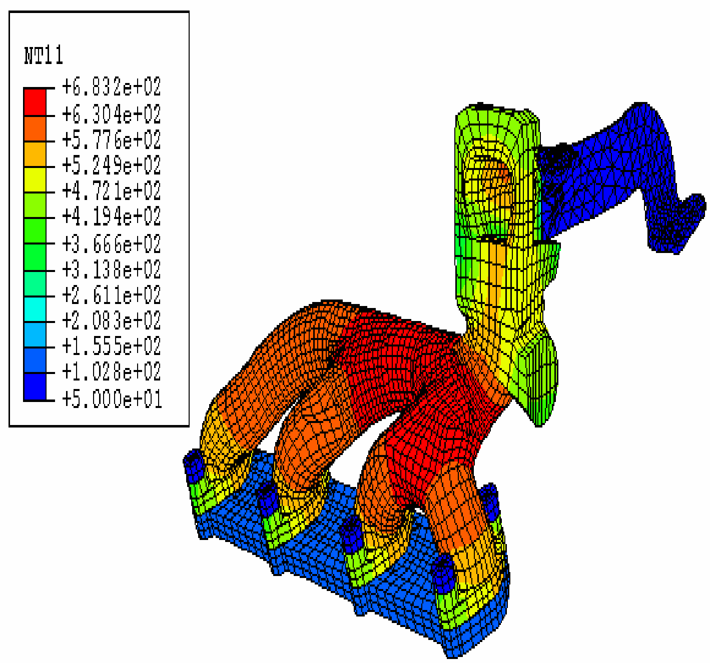


Fig. 1 Temperature distribution at the operating temperature

Reference

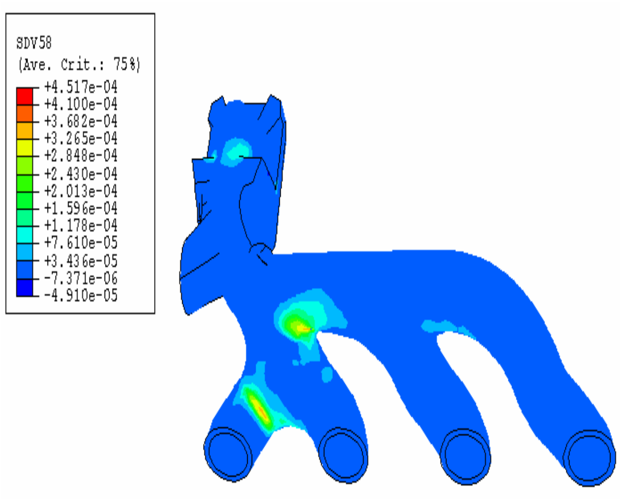


Fig.2 Damage distribution after thermal cycles

[1] Deger.Y., “Simulation of Thermo mechanical Load Cases”,*Sulzer Technical Review*, Vol.3, 2003.

[2] Simperl.B., Schöck.J. Deger.Y, “Thermal Deformation and Stresses in an Exhaust Manifold”,*Technical Report Nr.TB03_0123, Sulzer Innotec*, Oct. 2003, (not public).

[3] ABAQUS User’s Manual, Version 6.7

[4] S.S. Manson;“Behaviour of materials under conditions of thermal stresses”, *Technical report, TN 2933, N.A.C.A*, 1953

[5] N. Mamiya; et al. "Thermal fatigue life of exhaust manifolds predicted by simulation", *SAE Paper no. 01-0854*, 2002

[6] P.O. Santacreu; et al. "Study of thermal fatigue of stainless steels and its application to the life prediction of automotive exhaust line components", *Congress Thermal Stress’99, Cracow, Poland, Skrzypek and R.B.HetnarskiEds*., pp.245,1999.

[7] Sandeep Patel, Nickel Daniel, ”Structural Optimization of Engine Components Using ABAQUS and TOSCA”, *Project report*,2007

Author Profile

The Author J.David Rathnaraj is a Professor in Mechanical Engineering. He has 15 years and more experience in CFD and Computer Aided Simulation of Mechanical Systems. Currently he is working on Computational fluid dynamics and has conducted many conferences and Workshops in CFD