FINITE ELEMENT ANALYSIS ON THE HEAVY DUTY ENGINE EXHAUST GAS REGENERATION UNIT

Final Draft Manuscript
ABAQUS Users Conference 2007

Authors

Srikanth S Mantri
Ameer Ambavaram

MODERN® ENGINEERING, INC.

Author Affiliation

I, Srikanth Mantri graduated from the department of Mechanical and Aerospace at the College of Engineering at West Virginia University, Morgantown WV. Currently, working as structural design Analyst with Modern Engineering, Peoria IL. Modern Engineering Inc., is an engineering firm in Peoria specializing in all aspects of product, manufacturing, supply chain, mechanical and electrical design, and technical staffing.

After having acquired a masters diploma in CAD applications in 2003, I have worked on various engineering projects as a product engineer using Pro/E wildfire for design and ABAQUS CAE for the analysis. Worked with Systems and Electronics Inc., St. Louis MO on the Military truck applications. Worked on key projects for the Lockheed Martin group at Baltimore involving the design of unmanned combat vehicle bridge launching equipment. Currently working on projects involving the structural evaluation of the engine mounting components, design and fatigue strength evaluation.

Abstract

In order to match the more stringent 2007 Environmental Protection Agency (EPA) emission standards, the need for more robust and refined exhaust gas treatment has been recognized by the heavy duty engine manufacturers. This led to the conception of a breakthrough technology in the engine exhaust gas treatment by one of the World's leading heavy duty engine manufacturer. The soul of this technology is vested in the Exhaust Gas Regeneration Unit (EGR) which comprises of various intricate components along with a network of fuel, coolant and air ducts, and distribution head fittings. A prime challenge in the development of a successful design was to come up with a robust routing of these ducts which consisted of various combinations of Hard lines (steel) and Flex lines (tubing) along with compatible end fittings. This configuration of the regeneration unit had to go through various conceptual design modifications before emerging into its current form. Each of the design iterations had to be validated for their design and fatigue strength conformance and this is where ABAQUS claimed its stake as an effective Finite Element Analysis tool with its robust geometry import techniques in conjunction with its advanced mesh-generation algorithms, premier adaptability to design modifications and various other capabilities which yielded highly comparable results on the modal, static and the frequency response analysis. The final test results were precisely comparable to the pre-test predictions based on this analysis result, which stand testimony *to the powerful hand of ABAQUS in predicting the system behavior. The study included a detailed modal and stress analysis of the system along with the frequency response test and a fatigue life analysis.*

***Keywords**: ABAQUS, Ties, Couplings, Heavy duty Engine, Exhaust Gas, Recirculation, Finite Element Analysis, Modal Analysis, Static Analysis, Frequency Response, Fatigue Failure, Steel Fittings, Rubber hoses, gravitational loads, beam elements, convectional shells and base excitation.*

INTRODUCTION:

The benefits of FEA always included increased accuracy, enhanced design and better insight into critical design parameters, virtual prototyping, fewer hardware prototypes, a faster and less expensive design cycle, increased productivity and increased revenue. Add to it a powerful yet easy to use software such as ABAQUS, capable of importing highly complex configurations from different CAD packages, precise import geometry diagnosis and repair techniques, a wide range of materials selection, advanced mesh generation algorithms and a reliable solver and what we have is an unprecedented tool to tackle the current engineering challenges.

Though the traditional design philosophy of component testing is still prominent, Finite Element analysis has rapidly increased its stake in product design and component failure prediction segments in the truck industry. It is primarily through these improved initial prototype designs using FEA that the testing and development stages have been accelerated.

The primary aim of this study was to perform a detailed modal and stress analysis on some of the components considered crucial in the newly conceived engine exhaust gas regeneration unit by one of the leading heavy duty engine manufacturer. As they push for higher-fidelity models to improve the quality and shorten the design cycles, the present day need is to have a robust Finite element replication of the test components with suitable element selections and an effective pre and post-processor, producing close proximate results. The following study showcases the role of ABAQUS in addressing these needs.

DEFINITION

As a result of various vibrations conceived by the heavy duty engines during their operation, numerous small components are forced to inherent them. A group of such components being analyzed in this study are the steel tubes responsible for carrying fuel, coolants or air in and out of the exhaust manifold which is a part of the Exhaust Gas Regeneration unit. A combination of these fuel and coolant transfer lines which are either made of steel tubes or rubber hoses forms a closed loop transferring precise amounts of fuel or coolants. Either ends of these tubes and hoses have steel end fittings which bolt on to the manifold and the regeneration units respectively. Due to the operational loads these steel tubes and hoses are induced with some of the engine vibrations which when resonate with their natural frequencies, expose them to severely amplified cyclic loads and eventually causing them to fail. Failure of such crucial fuel carrying ducts and their end fittings poses a potential threat of fuel leakages and burn out which could be highly hazardous. A modal analysis was to be performed to extract the natural frequencies and mode shapes that characterize the free harmonic oscillations of each of these components in the assembly and see to it that they are above the frequency criteria for that engine at a rated speed.

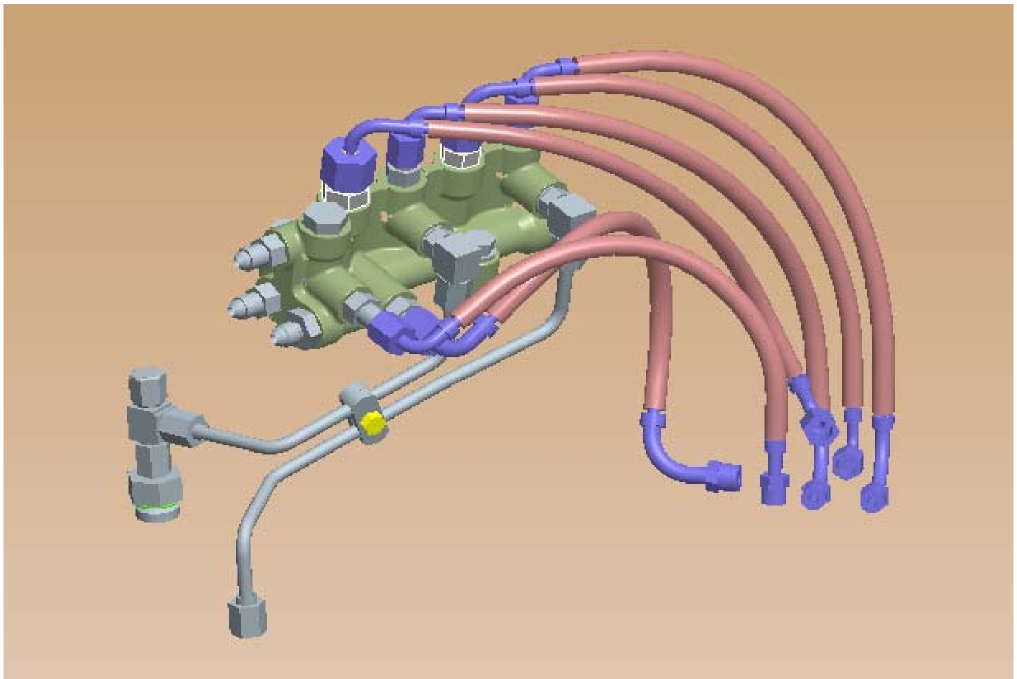

Figure1. Exhaust gas regeneration unit sub-assembly.

<u>FATIGUE FACTOR DETERMINATION</u>

Often, machine members are found to have failed under the action of repeated or fluctuating stresses, though the actual maximum stresses were below the ultimate strength and quite frequently even below the yield strength of the material. The primary cause for such a failure is that the stresses have been repeated a very large number of times over a period of time and this mode of failure is known as the fatigue failure. Fatigue Failure is due to crack formation and propagation. The Modified Goodman diagram is used to define the criteria for the fatigue failure. It has the midrange stress plotted along the abscissa and all other components of stress plotted on the ordinate, with tension in the positive direction.

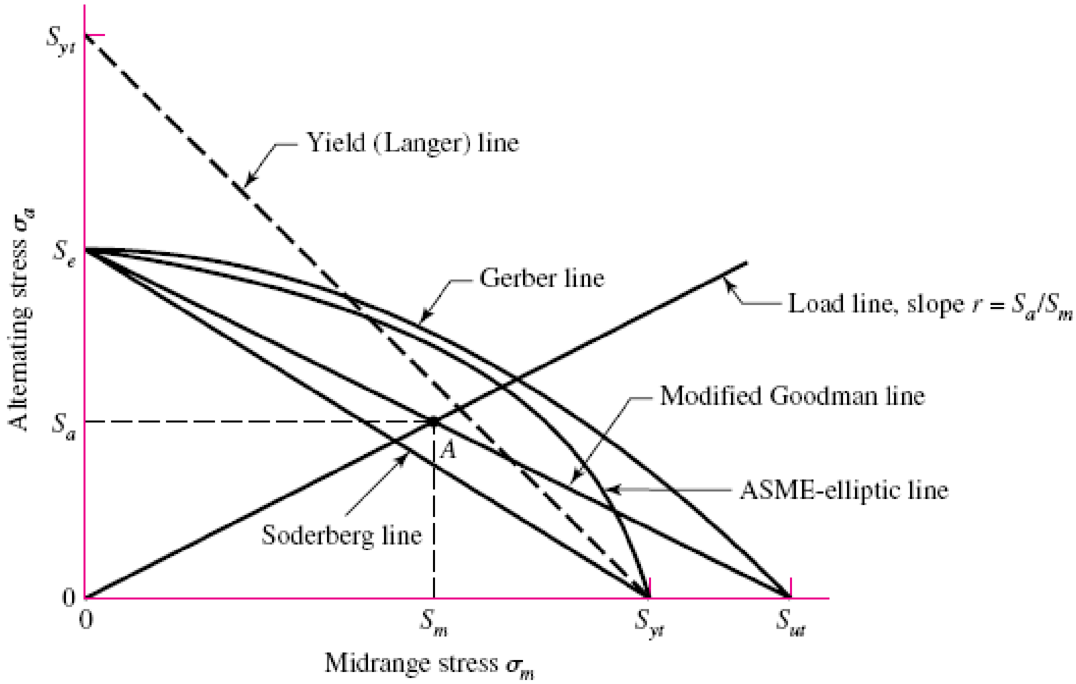

Figure2. Modified Goodman Diagram.

$$\text{Fatigue Factor} = (EnduranceStrength) * \left[\frac{\left(1 - \frac{MeanStress}{UltimateTensileStress}\right)}{Max.AlternatingStress} \right]$$

<u>FINITE ELEMENT MODELING</u>

The CAD models of the assembly were imported into ABAQUS in a neutral file format of STEP files. The import of these models was successful with virtually no loss or approximation in the design features. Some of the imprecise geometries though existed were seamlessly merged using various repair techniques such as repairing of invalid edges and replacing faces.

<u>MESHING</u>

The assembly consisted of varied geometries such as thin and think plates, circular tubes etc. The thin plates were partitioned in order to facilitate hex meshing and apply sweep technique to create a uniform brick mesh. Thick plates have been Tet meshed with solid quadratic elements.

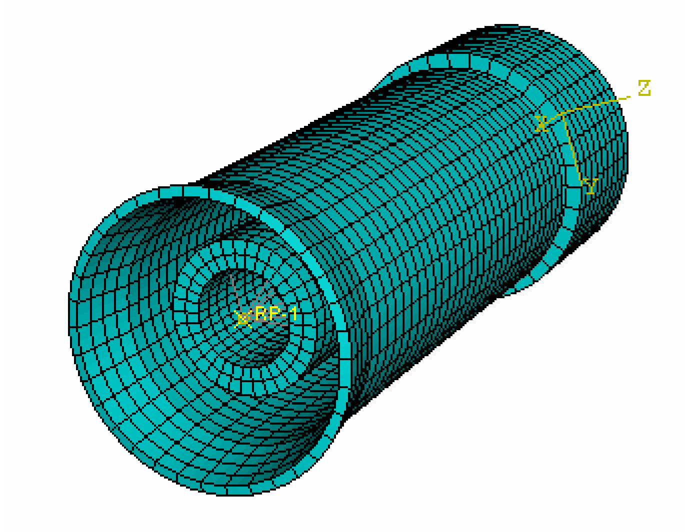

Figure 3. Meshed finite element model of an end fitting.

The imported solid CAD models of the steel tubes were transformed into shells in order to facilitate the convectional shell modeling. The circular hollow steel tubes have been meshed with convectional shell elements where as the more complex rubber hoses have been modeled using linear beam elements incorporating the material properties obtained from the testing of these hoses. The ends of the tubes had steel end fittings attached to them which in turn hooked the tubes to the manifold and the regeneration unit. The manifold and the regeneration units were represented by a mass point at their C.G and coupled to the inside surfaces of the end fittings.

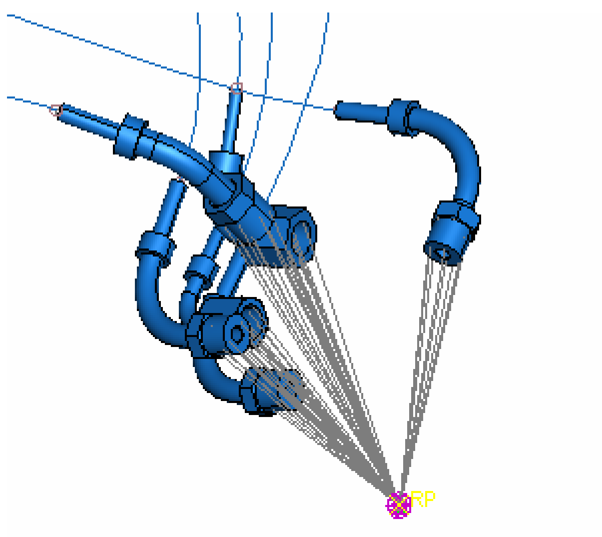

Figure 4. Kinematic coupling of the regeneration unit mass.

EIGEN VALUE EXTRACTION

The modal analysis was performed on the assembly to extract the Eigen values or the natural mode frequencies. The ends of the tubes emerging out of the manifold were assumed to be constrained in all the three translational degrees of freedom. The mass point representing the regeneration unit has been assigned with spring stiffness in the 3 translational degrees of freedom as obtained from experimental testing. The spring stiffness is obtained from the equation: $f = \frac{1}{2\pi}\sqrt{\frac{k}{m}}$ where m is the mass and k is the stiffness.

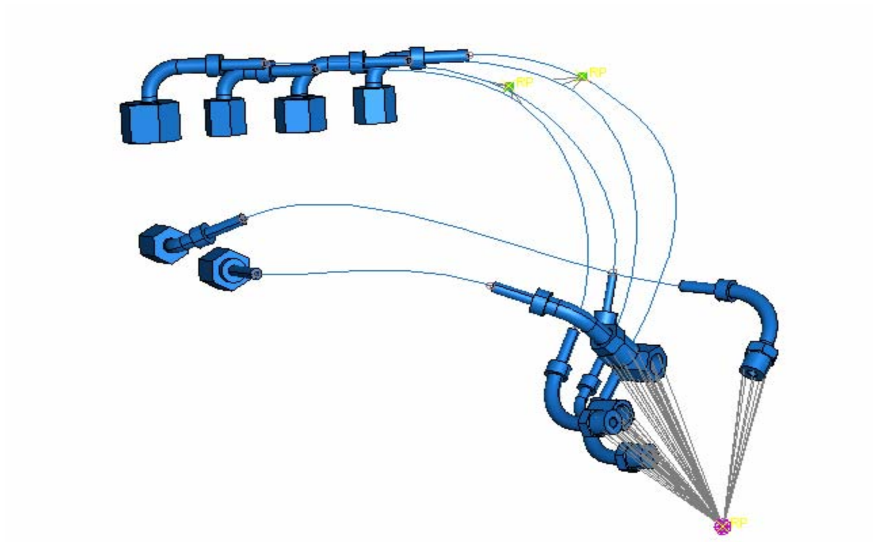

Figure 5. Finite element model of the regeneration unit subassembly.

The First 25 Eigen values were extracted in the modal analysis. This analysis indicates the minimum natural frequency at which a desired component in the assembly vibrates. This frequency is then compared with the frequency criteria of the engine for a rated peak r.p.m and at a given frequency order.

* **Frequency Criteria = [(Engine Order)* (Engine RPM) / 60] + 10 %**

The stress plot from this modal analysis indicates the basic stress distribution pattern for the components and identifies the locations to be concentrated in the Steady State Dynamic analysis.

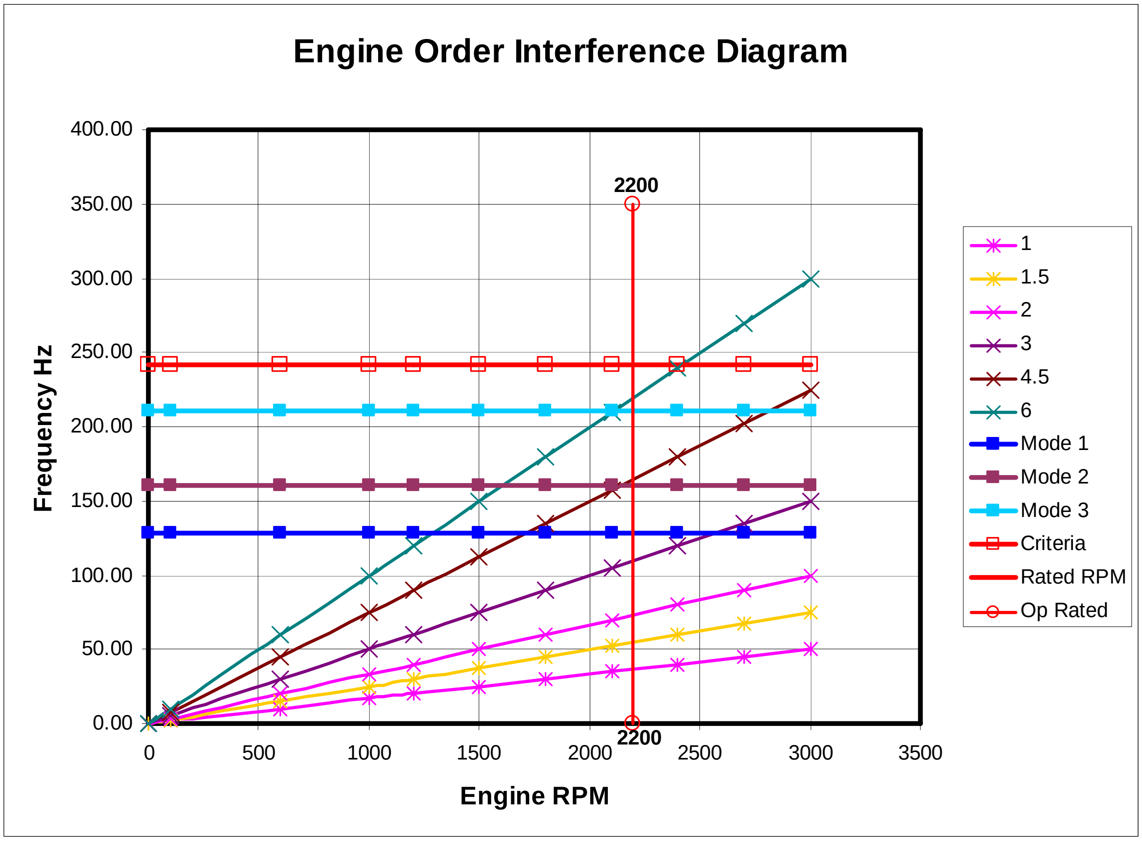

Figure 6. Engine interference diagram.

The above plot depicts the comparison of the modal frequencies of the components with respect to the engine frequencies at various firing orders.

STEADY STATE DYNAMIC ANALYSIS

The steady state dynamic analysis for a linear system is generally employed for the design of such engine mount components over a range of engine operating speeds. Since in this case the system response was to be verified over a specified test based range of excitations, steady state dynamic analysis was inept. The response behavior of the system over each harmonic cycle in time is assumed to be linear. Since the applied loads and the structural response are assumed to vary harmonically with time, the solutions were performed in the frequency domain. A uniform damping coefficient has been included to ensure a bounded solution in the event of an excitation frequency matching with a natural frequency of one of the components.

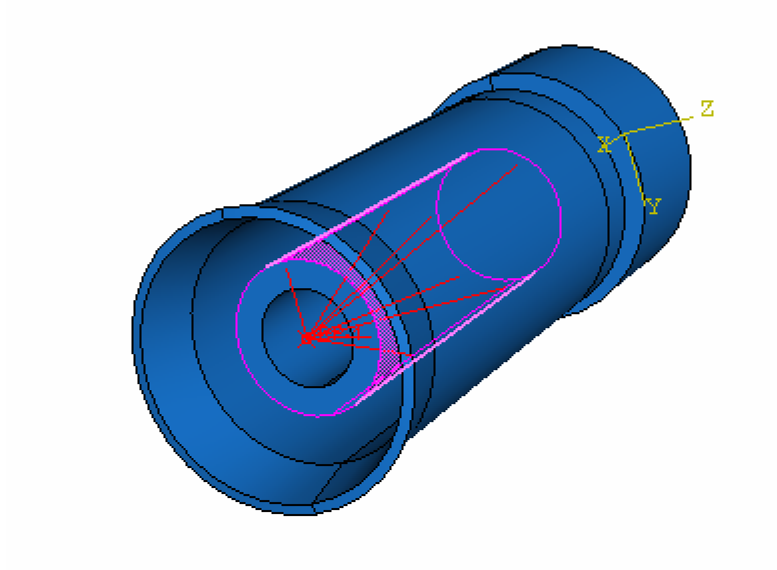

Figure 7. Distributive coupling of the end fitting.

Based on the Modal analysis stress plot the critical stress locations have been identified roughly. A steady state modal dynamic analysis is performed in the next step by superimposing a set of gravitational loads within a given time span. These excitation values were measured on the manifold through experimental testing. Since the flexible tubes are modeled using a wire element with the beam properties, the end of the beam is coupled using a distributive coupling constraint to the inside surface of the steel end fittings. A distributive coupling ensures a load transfer across the junction of the beam and the tube end of the fitting.

CONCLUSIONS

As a result of the detailed Modal analysis performed on the regeneration system, the corresponding vibration modes of the individual components of the assembly were extracted and each checked against the engine excitation frequency criteria for the given order. This analysis also provided strain outputs for the components, helping detect the critical strain location which gives an estimate of the mounting point locations for the experimental gauge set up. The obtained final natural frequencies were interpreted in the Engine interference diagrams and a comparison to different engine order requirements has been drafted.

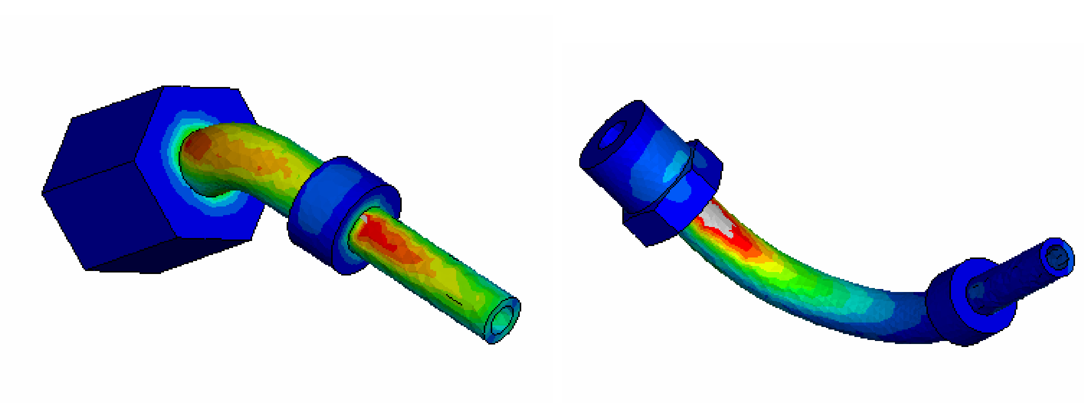

Figure 8. Stress plot for the tube end fittings.

The Frequency response study yielded not only in the determination of the maximum stress locations but also the exact frequencies at which these stresses have been induced into the components. This calls for the need to either optimize the design of the critical components which in this case are the tube end fittings or make better attempts to minimize the excitations caused by the engine vibrations. This analysis also yielded highly precise stress results which formed the basis for the fatigue life prediction of the components under the given set of loading conditions.

REFERENCES

1. **Modern Engineering, Inc.**
2. **ABAQUS 6.6 User Manual**