1. Abstract

The objective of this report is to analyze the behavior of the AJ250 Aerojet Mill when subjected to Dynamic loading w.r.t Pressure. A finite element analysis (FEA) is conducted to determine the Stresses, Deformed Shape and Displacement.

Keywords**: Dynamic Analysis, Contact Analysis, Isotropic Material, Hypermesh, Abaqus**

2. Introduction

The failure modes in structural material, the initiation of a crack/fracture does not indicate ultimate failure. Generally, a stable crack propagation stage, associated with steady increase in external load, precedes a catastrophic failure. This is frequently observed in structural applications of material. In this case, analytical and experimental investigation of stiffeners is studied. Design and assessment of such structures require adequate consideration of strength and fracture toughness.

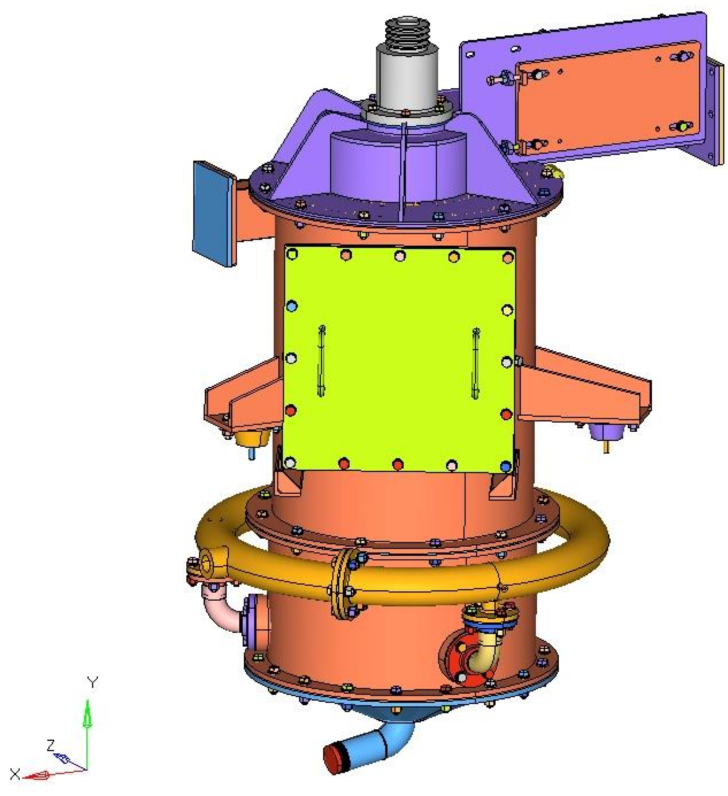

Figure 1: AJ250 Aerojet Mill

These bolts load the Bracket through washer plates, which high loadings (combination of tensile / compression, flexure and torsional loads), the design of supporting structure in this region is critical. However, because of complex loading, this region is often over-designed to prevent failure. Thus it is necessary to focus on the structural design of Inlet duct structure to minimize human and material losses, which can be achieved by understanding the tensile and shear stress distribution and their effect on failures. This determines the load-carrying capacity and damage mechanism of AJ250 Aerojet Mill.

3. Discussion of FEA approaches

The Finite Element Model was generated with shell (At mid surface) and tetra elements (2nd Order). Abaqus Element types of S3, S4, C3D10 and Kinematic Coupling were used for generating the FE Model of AJ250 Aerojet Mill. Fasteners were simulated using rigid elements (Kinematic Coupling), Rotor assembly was simulated as a mass element at the CG location of Rotor. The Non-Linear Contact (Tie) has been modeled between the Rubber Mounts (Slave) and Main Body (Master) to avoid the penetration. This contact has been provided at all the three Rubber Mount locations. British Rema Engineering experimentally and numerically conducted these three tests and studied stress distribution using ABAQUS.

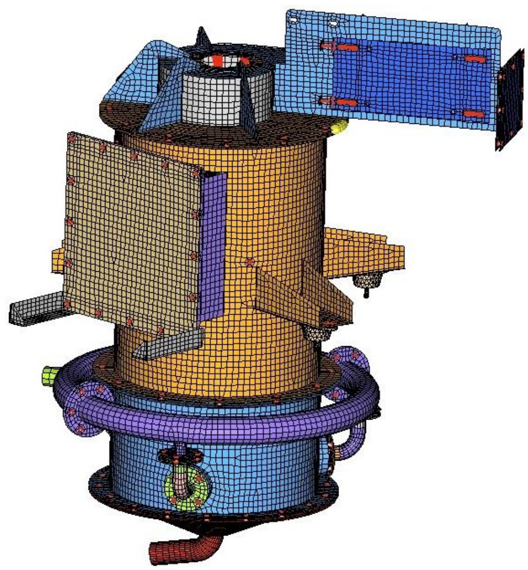

Figure 2: FE Model of an AJ250 Aerojet Mill

4. Numerical simulation

Finite element simulation of a curved is carried out using the commercially available finite element package ABCQUS 6.12. This solver is capable of performing linear and static or dynamic framework. In order for it to be employed in this simulation procedure, the solver is equipped with additional isotropic/orthotropic material models for the description of materials to perform the dynamic static analysis of the structural model.

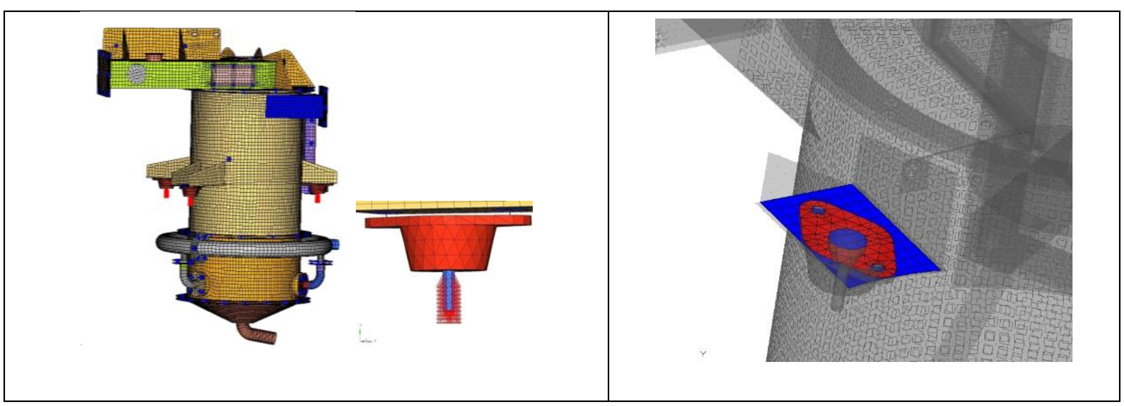

Figure 3: Boundary Conditions and Contact pairs

5. FEA Results

The numerical analysis was conducted using small strain analysis and steel backing plates were modelled with plastic behavior. Displacement & Stress distribution at the handle, just before the first failure is shown in Fig.2

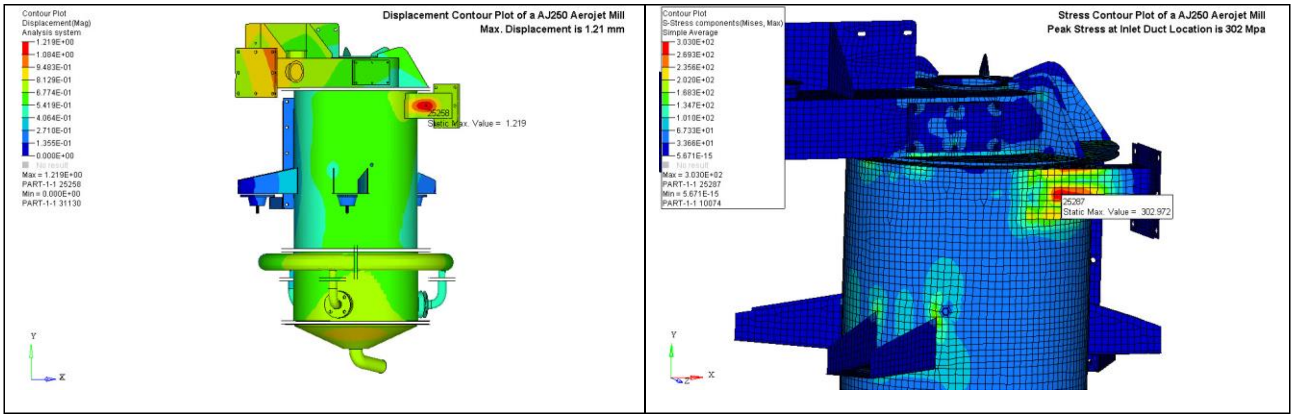

Figure 4: Displacement and Stress Contour Plots of a Dynamic Analysis of AJ250 Aerojet Mill Assembly

6. Conclusion

In this paper, an experimental and analytical investigation of Dynamic Aerojet Mill Assembly that fail due to delamination is presented. It was observed that a better understanding of the Contact behavior and stress distribution makes the design more robust and minimizes material and human catastrophic.

FE analysis of stress distribution around the bends is analyzed. In general the inlet duct is deforming and it was observed that the load is transferred to the bracket, as the mounting pin is overhang structure and it is deforming to an extent.

The inlet duct failed at the curved section due to excessive high stresses. The strength is a resin dependent mechanical property in the through-thickness direction. The limiting factor for failure initiation is the opening of the inlet duct. The predictions using FEA are in reasonable agreement with the experiments. Experimental results include scatter and uncertainty whereas the finite element model is free from manufacturing and testing inaccuracies.