CASE STUDY : STATIC FEA ANALYSIS OF AUTOMOBILE CHASSIS

INTRODUCTION

Chassis are very important part of automobiles as they are the main load taking member of the vehicle. Apart from dead weight of various functional components, occupants and payload, chassis is subjected to bending moments, torsion and combined bending and torsional loadings while the vehicle is in motion. Since, it's the chassis which is responsible for holding all the systems together as well as safety of occupants, it becomes very important to analyze it for various possible loading combinations. Using FEM for such analysis makes the things lot easier, faster and the designs can be made more reliable, lightweight and cost friendly.

Analyzer has done some advanced simulations for leading global Automobile OEMs for evaluating designs of their chassis in static as well as dynamic conditions. This study puts light on one such project done by Analyzer.

FEA WORKFLOW

- Collection of Chassis geometry and Related material properties from client end.
- Preparing FEA model, which includes simplification and meshing of structure. This requires a lots of structural understanding followed by experience in order to capture all the critical areas without any loss. Also modelling different connections is a great challenge as simplified approach should match with actual structural behavior.
- Solving the FEA model for different loading conditions. The conditions are predicted by considering various parameters like Dead Weight, Payloads, CG Location of vehicle etc.
- Report the results for client's reference.

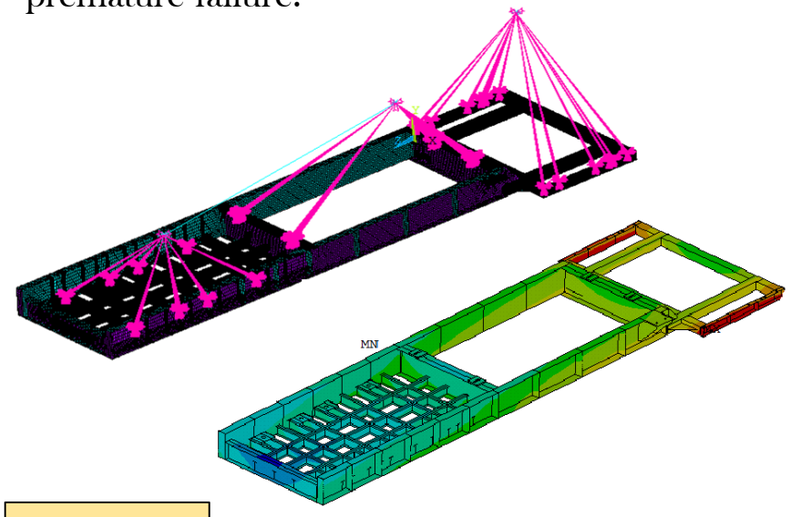
CHALLENGES

- Structural simplification of Chassis geometry.
- Meshing approach for Chassis structure.
- Modelling different Contacts/connections in FEA.
- Predicting the different loading conditions to be analyzed.

THE SOLUTION

The Chassis structure was analyzed using Finite Element Method for different static loading conditions. Due attention was given to consider the effect of various attached components while creating the model. All the welded, riveted and bolted joints were modelled with appropriate assumptions and Element types.

The results reveled that the induced stresses were well within the allowable limits for all the loading combinations. Hence the structure was predicted to behave safe during service life without any premature failure.



BENEFITS

- Prediction of high stress regions so that corrective actions can be taken at early design stage.
- Reduced testing and hence reduced lead time with optimized product.

ANALYZER CAE SOLUTION PVT. LTD, PUNE.
Voice : +91-9561739442
E-mail : dharani@analyzer-cae.com
Web : www.analyzer-cae.com