Non – Linear Analysis of Transmission Rear Cover Using Optistruct Considering Sequential Multiple Load-Cases

Sonu Paroche
VE Commercial Vehicles Ltd.
102.Industrial Area 1, Pithampur-454775,
smukherjee@VECV.IN

Sayak Mukherjee
VE Commercial Vehicles Ltd.
102.Industrial Area 1, Pithampur-454775,
sparoche@VECV.IN

Abstract

Nonlinear analysis of automobile components is becoming increasingly important at the product development stage as they have to withstand complex loading condition during their operation cycle. The main challenge lies in developing a robust analytical process that makes it possible to conduct analysis of such loading conditions with sufficient degree of accuracy. The present paper focuses on non – linear analysis of transmission rear cover using Altair FEA package Optistruct. The rear cover was analyzed for assembly loads (contact, bolt pre – tension) and structural loading conditions (bearing force and gravitational loads) applied as subsequent load-steps. The results obtained were used for comparative study with the stress and displacement results obtained from other FEA Non-Linear software being used currently and physical testing of the assembly under operational loads.

Introduction

Transmission rear cover is used to prevent the leakage of oil from main transmission housing. It has a slot for mounting of bearing which supports the shaft passing through it. The contact between the transmission housing and rear cover should be air tight so as to prevent any oil spill through the contact area. This calls for proper analysis of contact area between the rear cover and transmission housing.

Thus the main purpose of this paper is to perform a non-linear analysis of the contact surface taking into account the effect of bolt pretension and structural loads comprising of bearing force and gravitational loads in subsequent sub-steps. The FE Model of the Rear Cover – Transmission system is created using HyperMesh™. The meshed model is then solved using Optistruct. Results obtained were analyzed using post-processing software HyperView.

Finite Element Modeling

The 3-D geometry of the rear cover assembly was generated using solid modeling software package Pro-E. The CAD data is then converted to Initial Graphics Exchange Specification (IGES/STP) format and imported into pre-processing software HyperMesh™. 2nd order 3-d tetra elements are used to create FE model of the rear cover assembly. Fig.2 shows the Finite Element Model of the rear cover assembly meshed using HyperMesh^{TM}. The rear - cover assembly considered for FE Analysis consists of the rear – cover, gearbox and 1-D bolts modelled using bar elements to position the joining surfaces.

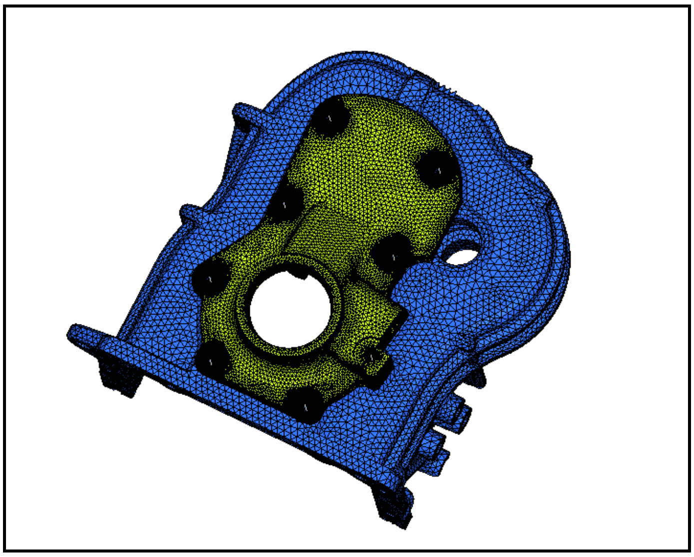

Figure 1: Rear Cover Assembly - FE Model

Material Details

The material considered for the FE Analysis of rear cover is FC25. The material for transmission box is cast iron (CI). The bolts were assigned the material properties of structural steel.

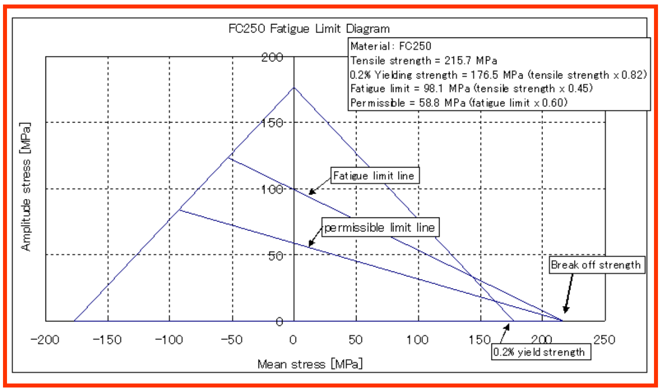

Figure 2: Material details for FC25

Contact Details and Bolt Pretension

The following contacts between different components are considered for the analysis

- Frictional contact behavior was used at the rear cover – gearbox interface.

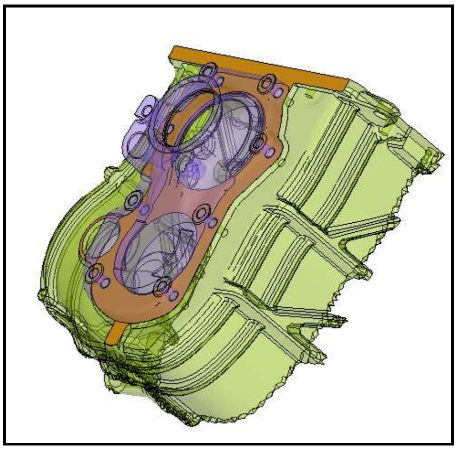

Figure 3: Contact Details

- To simulate the effect of load due to bolt torque, pretension of 22 KN was applied on the eight 1D bolts joining the rear cover and gearbox.

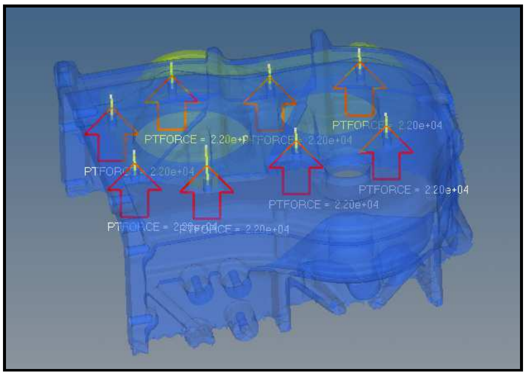

Figure 4: Bolt Pre-Load Details

Loading Conditions

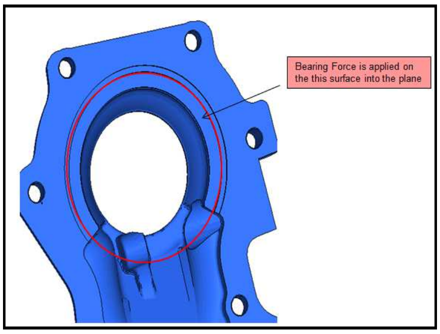

Figure 5: Static Load Details

The static analysis of the connecting rod was carried out in two load steps. The load steps include

- Assembly Loads (Contact + Bolt preload)
- Main Shaft Rear Bearing Force for First Gear and Gravitational Loads

The contact b/w the rear cover and gearbox is applied using Auto Contact tool in Optistruct user profile of HyperMesh. There was no initial penetration b/w the contact surface the MORIENT parameter of the CONTACT card was set to OPENGAP.

The 1-D bolt preload is applied using Pretension Manager tool.

The second load-step was created for structural and gravity loads and the control card for second load-case was set so as to include the results from the previous substep.

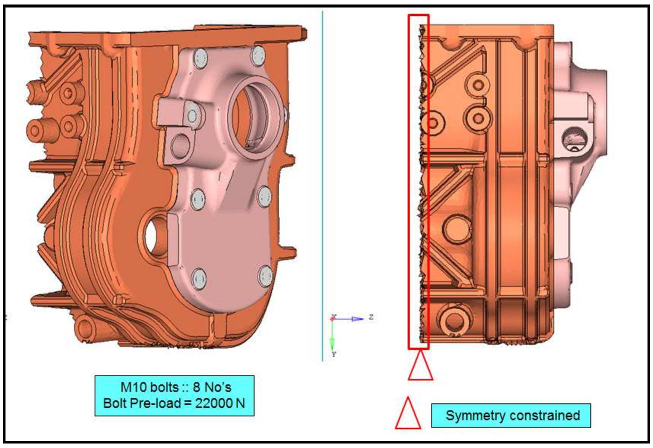

Figure 6: Pretension Loads and Constraints

Non-Linear Analysis Setting

To account for the non-linear nature of the contact a non-linear solution is activated for the subcase by including the NLPARM parameter. If NLPARM is not present in the subcase definition, the FE model is assumed to have linear behavior. In this situation, contact status is determined once at the beginning of the solution, and does not change as the solution progresses. In HyperMesh, the NLPARM parameter was defined by creating a load collector with card image “NLPARM.” The NINC parameter on this card represents the number of equal subdivisions that the total load in a given subcase will be divided into. If NINC is blank, the entire load for a given subcase is applied at once. For the present analysis the total load acting on the system was divided into three subcases.

The MAXITER parameter refers to the limit on number of implicit iterations for each load increment. If reached, the solution is terminated. The MAXITER parameter was set to carry out twenty five iterations at each sub case.

For the second load step CNTNLSUB was set to YES to read the results of previous substep.

After the loadsteps were sequentially defined and analysis control cards were properly set, the FE Model of the assembly was solved using Optistruct.

Results and Discussions

After the results from Optistruct were obtained they were properly analyzed for both the sequential load cases.

a. First Loadstep (Assembly Loads – Contact + Pretension)

Displacement plot for assembly loads were obtained and viewed using post-processing software HyperView.

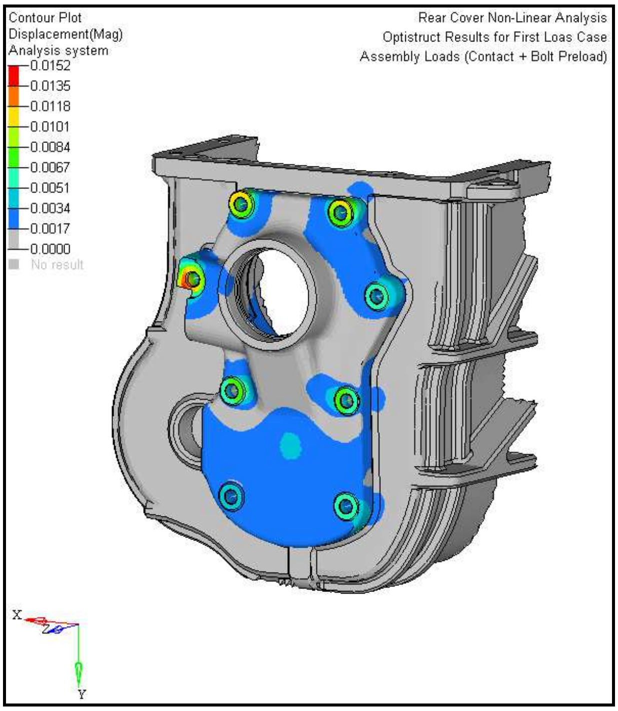

Figure 7: Displacement (in mm) results for assembly loads (Contact + Bolt Pre-tension).

b. Second Loadstep (Assembly + Structural Loads)

The stress and displacement results for the second loadstep were obtained from Optistruct

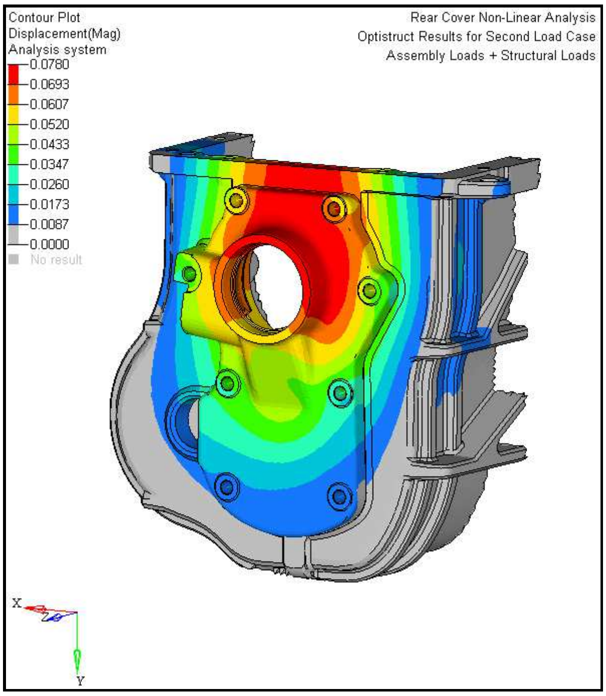

Figure 8: Displacement results (in mm) for Structural loads.

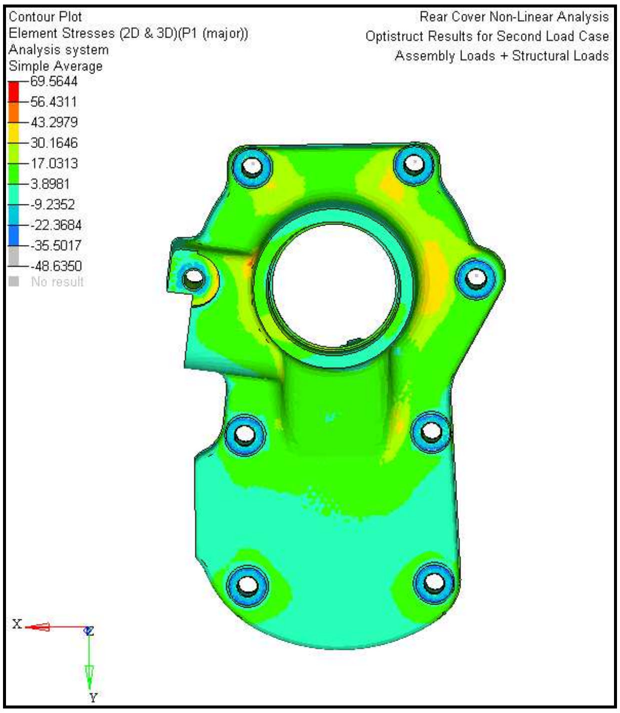

Figure 9: Maximum principal stress (MPa) results for Structural loads

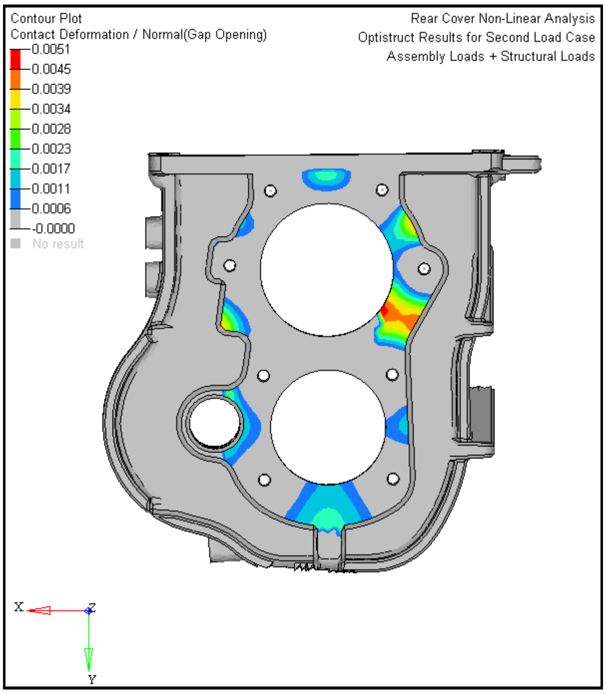

Figure 10: Gap opening

The maximum gap opening in the rear-cover assembly is 5 micron which is less than the target maximum gap opening criteria of 10 micron. Therefore the contact interface b/w the the rear-cover and transmission housing can be considered to be leakproof.

The results obtained from Optistruct were then compared with the results obtained from other FEA softwares with defined procedure currently being used for carrying out non-linear analysis. The results from both the analysis are tabulated below

Solver / Results	Load case	Optistruct	Existing Solver
Displacement (micron)	Assembly Loads	1.5	1.7
Displacement (micron)	Assembly Loads + Structural Loads	7.8	7.6
Max. Principal Stress (Mpa)		69.5	67.4
Gap Opening (Micron)		3	3

Table 1: Comparative Analysis

Conclusion

After performing the above analysis & studying the results it is found that the results from Optistruct are in accord with the results from existing non-linear solver. The displacement results have also been correlated with physical testing results. Therefore the current procedure can be used for solving other non-linear problems involving material non-linearities, contact and structural loads.

Acknowledgments

The authors would like to express their gratitude to Mr. Rajinder S. Sachdeva, Executive Vice-President, Technology Development, VE Commercial Vehicles Ltd, for granting permission to publish this work. Special thanks to Mr. V. Chandrasekhar, GM-CAE for his guidance and encouragement to develop present work. Also, the contributions and support of all CAD and CAE engineers are highly appreciated.

References

1. Altair – Optistruct Help.
2. www.6dof.com (Optistruct forum).

Definitions, Acronyms, Abbreviations

FC25 – Ferro Casting (Grey Cast Iron)

CI – Cast Iron

CAE – Computer Aided Engineering

FEM – Finite Element Method

CAD – Computer Aided Design

FEA – Finite Element Analysis

IT – Iteration

NLPARM – Non-linear Parameter

NINC - Number of implicit load sub-increments

CNTNLSUB - Continue nonlinear solution sequence from a preceding nonlinear subcase

Keywords

Rear Cover, Non-Linear Analysis, Loadstep, Load Case, Contact and Bolt Pretension