Random Vibration Fatigue Analysis Model Development from Explicit to Implicit in LS-DYNA®

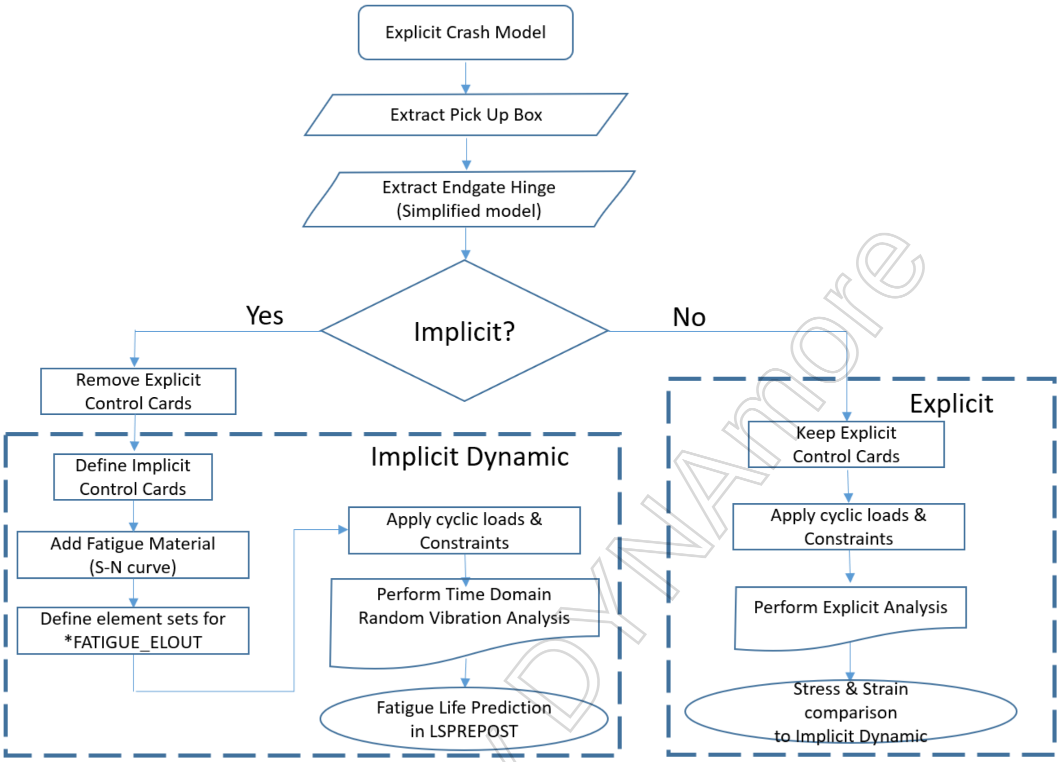
Hwawon Lee, Parvath Police
General Motors, BODY CAE
Amit Nair
LSTC

Abstract

Fatigue damage evaluation on vehicle body and hang-on components is one of the most critical paths for the vehicle development stage. Conventionally, fatigue analysis model has been characterized by linear static or dynamic model in which non-linearity of material and contact among vehicle parts are not properly considered, whereas the crash model using Explicit code takes both factors into account. Recent trend of crash event modeling is to increase number of elements up to several millions of finite elements, which is aided by rapidly improving computing power, enabling full vehicle simulations in a very short period of turnaround time. Currently, more focus is on the automotive industry to create larger FE models in a shorter period of time. This is to minimize or reduce vehicle development time caused by the size of the fatigue evaluation model. There were certain efforts to reduce modeling time by converting Explicit crash model to Implicit fatigue evaluation model without losing model contents in a very short period time. This improvement can be achieved in LS-DYNA.

This paper demonstrates how to build random vibration fatigue analysis models on MAST (Multi Axis Shaking Table) from Explicit crash model and how to predict fatigue life under random vibration cyclic loading. The first model is a full pick-up truck box, and the other one is a simplified end-gate hinge. A series of parameter study has been attempted to achieve a good correlation between the simplified fatigue testing and such parameters including mesh size, shell element formulation, number of thru thickness integration point & forming effect. The most critical parameter affecting damage ratio in the pick-up truck box is identified by comparing the corresponding test and the proposed model to achieve reasonable fatigue life predictions.

Work Flow



Setup Test / Analysis Model

A real-life example of fatigue crack was simulated on Hinge bracket (HSLA 340 grade) of pick-up truck end-gate after MAST (Multi Axis Simulation Table) tests of 200,000 cycles by repeated lateral loads. To address this issue, a simplified component test and analysis models have been developed on the hinge and the surrounding components as shown in Fig. 1. Both sides of the end-gate hinge minor brackets were mounted on two angle stanchions and secured onto the rigid bed plate. The load was applied through the cross beam in the lateral direction. Two strain gages and load cell were installed on the shaking side of the bracket and compared to the corresponding analyses for correlations.

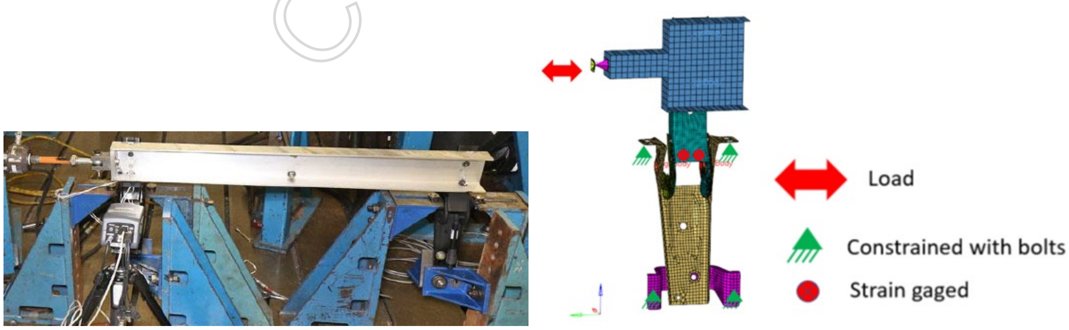


Fig.1 Simplified fatigue test and random vibration analysis model set up

Correlation works: Test / Analysis

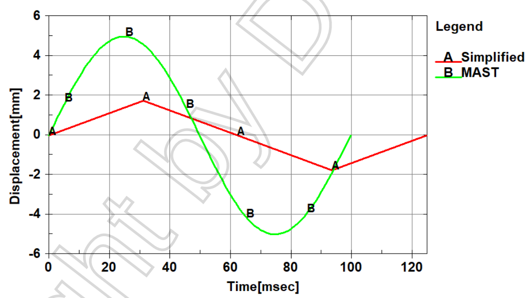
First displacement and loads were developed with trial-and-error method to achieve the same amount of strain measurement in full vehicle test as shown in Table 1.

Table 1. Measured loads and strain comparison

Descriptions		Peak	Valley
Component Test	Displacement [mm]	-1.71	1.75
	Loads[N]	-1557.61	1228.73
	Strain 1	-1286.14	1037.39
	Strain 2	-1058.93	1323.43
Vehicle Test	Strain	-1253.36	1080.85

Explicit vs. Implicit Analysis Model

Explicit analysis model was developed and converted to Implicit analysis model and compared to the test results for von-mises stress, maximum principal strain.
For the calculation of fatigue life in Implicit analysis model, developed SN curve in *MAT_ADD_FATIGUE.
Loads are applied in *BOUNDARY_PRESCRIBED_MOTION_RIGID as shown on Fig. 2.




```
*BOUNDARY_PRESCRIBED_MOTION_RIGID
94014709         2         2       101       1.0
       .      .      .      .      .
       .      .      .      .      .
 *DATABASE_HISTORY_SHELL_SET         .
          1
 *DATABASE_HISTORY_SOLID_SET
          1
 *DATABASE_FREQUENCY_BINARY_D3FTG
          1
 *FATIGUE_ELOUT
 $#   strsn     index     restrt
          0         0         1      4900
 binout0050
 *FATIGUE_MEAN_STRESS_CORRECTION
          1
   10000000     0.370
```

Fig.2 Load curves: Simplified vs. MAST on Pick Up box

Typical von-mises stress contours at integration point #2 (lower or inner surface in the Fig. 3) between Explicit and Implicit are demonstrated on Fig. 4. Measured force in Explicit analysis is 26% higher than that of Implicit run. However, its stress in Explicit is 6% lower than Implicit as shown on Table 2.

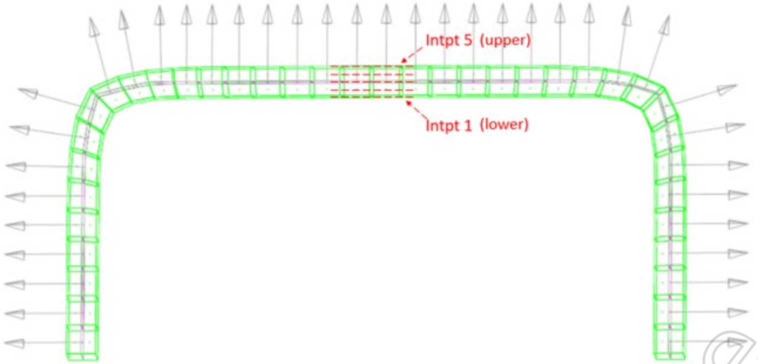
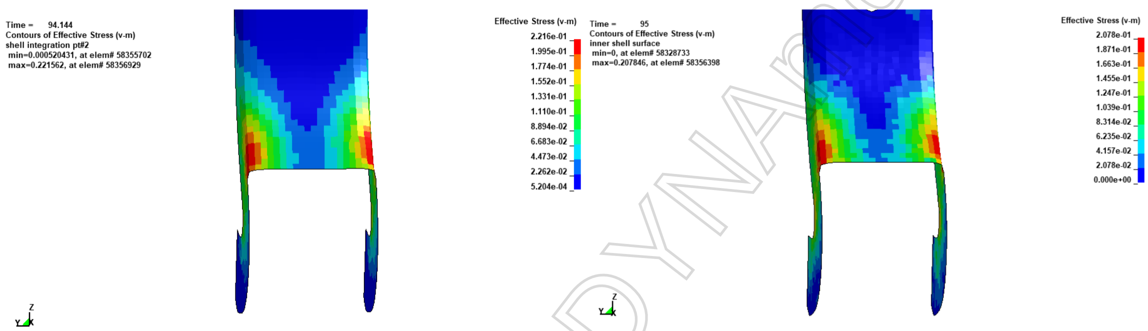


*Fig. 3 Thru-thickness integration points in *SECTION_SHELL*



Implicit Explicit

Fig. 4 Von-mises stress at peak load

Implicit Analysis results vs. Test results: Simplified Models

The following parameters have been studied for test correlation.

(1) Mesh size: 1.5 ,2, 3 & 5mm
(2) Shell element formulation: 2 , 16 & -16
(3) Number of thru thickness integration points: 2 & 5
(4) Forming results: mapped Effective Plastic Strain and Thickness, Thickness only

3 mm mesh sizes were investigated and summarized as the table 2.
Mesh size is linearly proportional to Expected life in log scale as shown on Fig. 5 . 2mm difference in mesh sizes changed about 30% in Expected life while von-mises stress changed 0.4 ~ 1.8%.

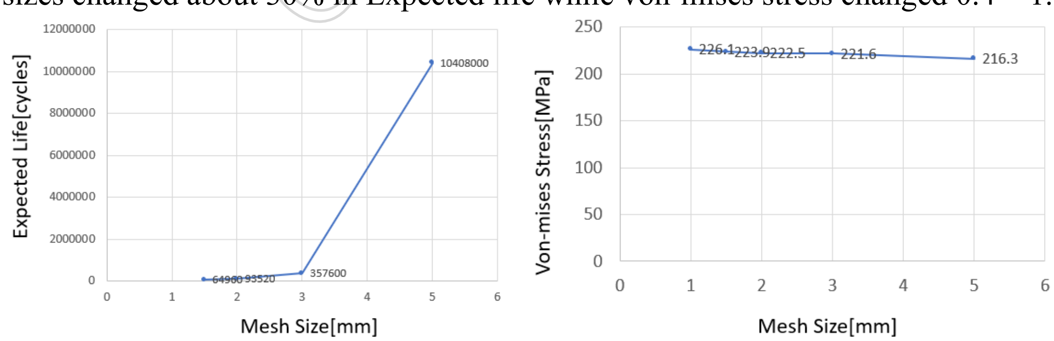


Fig. 5 Expected Life and Von-mises Stress along Mesh size variation

Table 2. parameter study: analysis vs. test results

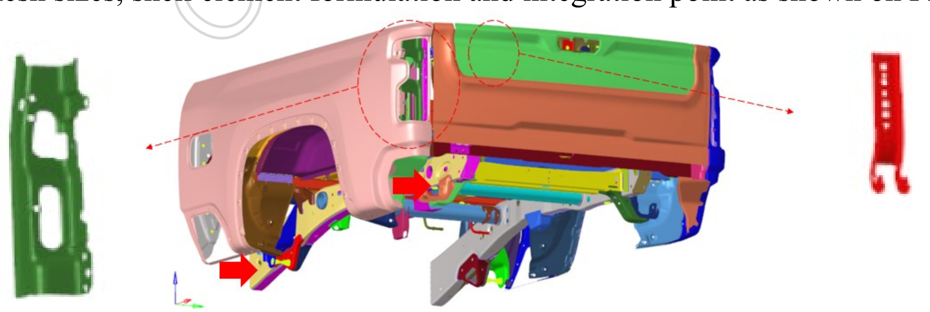
El. Form	NIP	Mesh [mm]	Force [N]	Micro strain				Mises stress [MPa]		life [cycle]
				max		on tested		max		
				Upper	Lower	Upper	Lower	Upper	Lower	
16	5	1.5	1196	1331	1095	1034	951.9	275.5	223.9	64,960
16	5	2	1188	1309	1081	1018	878.7	270.1	222.5	93,520
16	5	3	1197	1243	1077	1056	940.6	258.4	221.6	357,600
*16	5	3	1514	1001	1619	1001	1001	243.3	207.8	na (explicit)
16	5	5	1149	1071	1030	870.5	768.7	222.9	216.3	10,408,000
2	5	3	1367	1218	1133	1079	957.5	250.1	229.2	696,560
-16	5	3	1175	1242	1077	1055	940.9	258.2	221.5	362,240
16	2	3	1174	894.8	717.9	718.1	629.1	182.8	148.5	206,160,000
TEST1			1229			1286	1037			331,473
TEST2			1558			1323	1059			354,423

Shell element formulation 16 and -16 exhibited similar trends in stress, strain and expected life, but formulation 2 yielded nearly double the life when compared to formulation 16. Number of integration point 2 indicated 33% lower stress values than integration point 5 as LS-DYNA calculates and reports the stress close to the surface in integration point 5. Shell element formulation 16, number of integration point 5 and 3mm meshed case showed the best correlation to the test in life expectation as shown on Table 2 and Fig.6.

Fig.6 Test vs. Fatigue Life prediction – Simplified Model

Full Pick Up box Analysis in MAST and Test

Based on the simplified analysis correlation study, full pick up box MAST model has been developed using the same mesh sizes, shell element formulation and integration point as shown on Fig. 7.



Part (A) Part (B)

Fig. 7 Pick Up Box MAST model

Applied displacement along time in the MAST test is shown in Fig.8. One single sine wave with +- 5mm amplitude is introduced in corresponding MAST analysis.

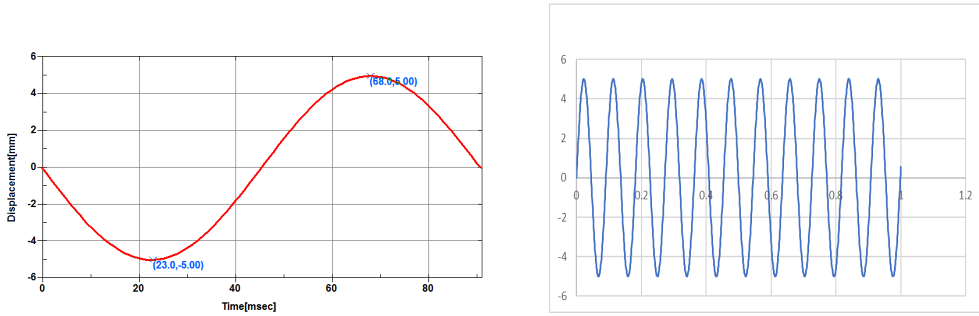


Fig. 8 Applied loads: cyclic displacement - 11 Hz

Two mesh sizes were investigated, one is 3mm and the other one is 5mm on the two components indicated green-Part (A) and red color-Part (B) on Fig. 7.
Fatigue cracks on both Part (A) and (B) are identical to the corresponding physical tests, life predictions on 3mm and 5mm mesh sizes are demonstrated on Fig. 8. The larger mesh sizes, the higher life prediction the same trend as the simplified model.
Overall, the life prediction at Part (A) underpredicts, and overpredicts at Part (B).

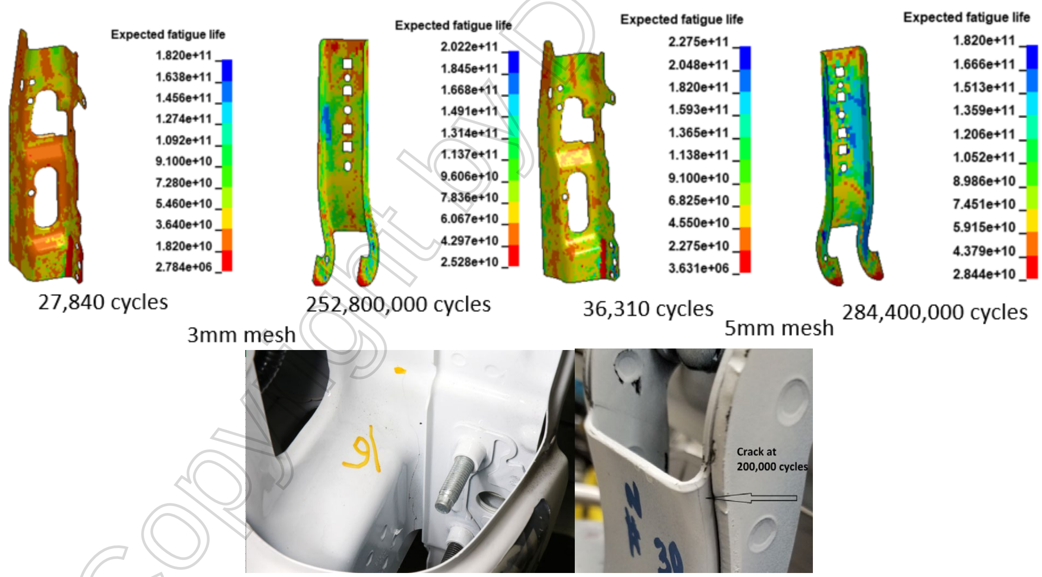


Fig. 8 Expected Fatigue Life : different mesh sizes and Test Photos

Summary

Random vibration fatigue analyses on MAST (Multi Axis Shaking Table) and simplified model under cyclic loading have been performed, and its fatigue life has been successfully predicted. Four (4) parameters including mesh size, shell element formulation, number of thru thickness integration point & forming effect affecting life prediction were studied to achieve a good correlation to the corresponding test. The identified critical parameters affecting damage ratio in the pick-up truck box were mesh size and number of shell integration points from the simplified model. The parameters of 3mm mesh, shell element formulation 16, and number of integration point 5 were proposed to apply MAST on full pick up box model.

Acknowledgements

The authors would like to acknowledge Yun Huang of LSTC through providing valuable advice on this study.

References

1. Yun Huang, Zhe Cui, Advances in Fatigue Analysis with LS-DYNA, Livermore Software Technology Corporation (LSTC), 15th International LS-DYNA Users Conference, June 10-12 2018, Detroit
2. Yun Huang, Zhe Cui, Philip Ho, Chengju Zhang, Application of LS-DYNA in Structural Fatigue Analysis, Proceedings of the 14th International LS-DYNA Users Meeting & Conference 2016, June12-14 2016, Detroit
3. Yun Huang, Zhe Cui, Recent updates in LS-DYNA frequency domain solvers, proceedings of the 10th European LS-DYNA Conference 2015, Wurzburg, Germany
4. LS-DYNA Keyword User's Manual Volume I & II, Livermore Software Technology Corporation (LSTC), 2019