LS-DYNA^Ó's Linear Solver Development — Phase 2: Linear Solution Sequence

Allen T. Li[1], Zhe Cui[2], Yun Huang[2]

1Ford Motor Company
2Livermore Software Technology Corporation

Abstract: This paper continues with the last one from the same authors on validating LS-DYNA's linear solver development on elements (Phase1: Element Validation).

In this paper, a simple plate model is used as the benchmark example for validation on linear solution sequence. The linear solutions from NASTRAN: SOL101 (static analysis), SOL103 (normal mode analysis), SOL108 (direct frequency response), SOL109 (direct transient response), SOL111 (modal frequency response) and SOL112 (modal transient response) are performed on this model. Equivalent linear analysis functions from LS-DYNA (static analysis, normal mode analysis, SSD, modal transient dynamics, etc.) are also performed. The results such as displacements, natural frequencies and stresses from NASTRAN and LS-DYNA are compared.

The validation shows that there is good match between LS-DYNA and NASTRAN in most cases. For LS-DYNA, the linear solution sequences those are well established and those need further development are summarized in the paper. This paper can provide the guidance on selection of the solution sequence for the users who need to run linear analysis with LS-DYNA.

1. Introduction

Each type of analysis available in NASTRAN is called a solution sequence. A solution is a feature selected by a SOL executive control command. In this paper, the linear solutions from NASTRAN: SOL101 (static analysis), SOL103 (normal mode analysis), SOL108 (direct frequency response), SOL109 (direct transient response), SOL111 (modal frequency response) and SOL112 (modal transient response) are performed for validations.

LS-DYNA provides equivalent linear analysis functions including static analysis, normal mode analysis, mode based steady state dynamics (SSD) analysis, direct SSD analysis, direct transient analysis and modal transient dynamics.

A simple plate model is used to validate the response of different solution sequences. The figure 1 shows the model, and the parameters of the model are, elastic modulus E = 2.07E+05 (MPa), Poison Ratio NU = 0.3, and density RHO = 7.831E-09 (Ton/mm3). The finite element model is shown in figure 2.
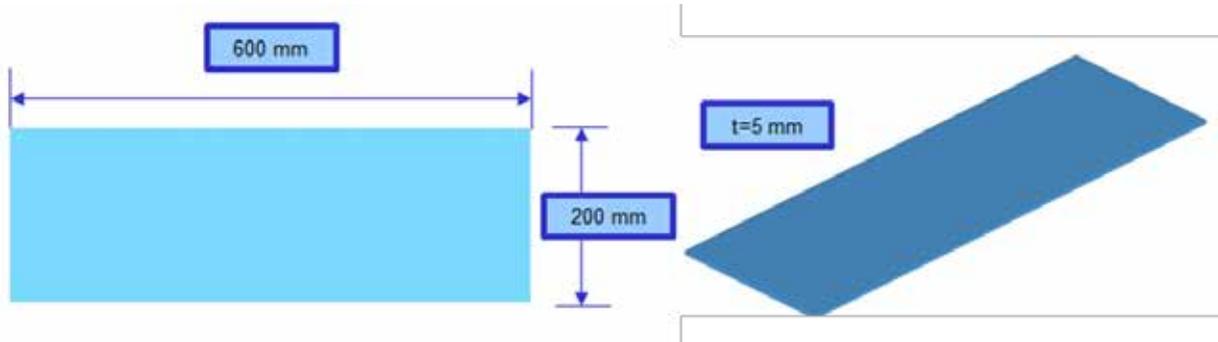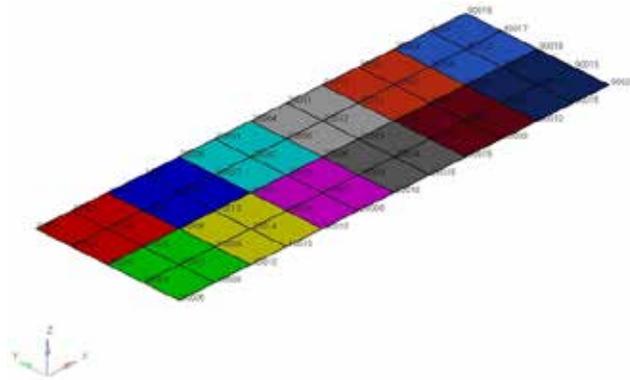
Figure 1 Simple plate

Figure 2 FEM model of simple plate

2. Solution sequence examples

2.1 Static analysis

In NASTRAN, static analysis is requested by the command SOL 101. LS-DYNA Static analysis is done using an implicit solver, the IMASS of keyword *CONTROL_IMPLICIT_DYNAMIC is equal to 0. Two types of static loads (pressure and nodal force) are applied to the plate simultaneously, one edge of which is fixed (figure 3).
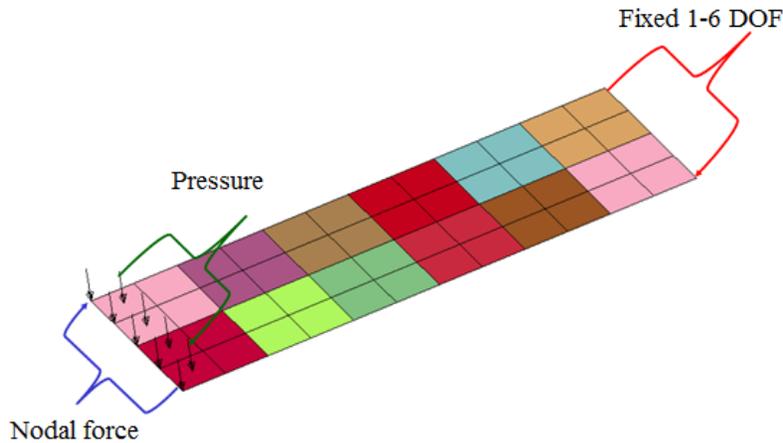

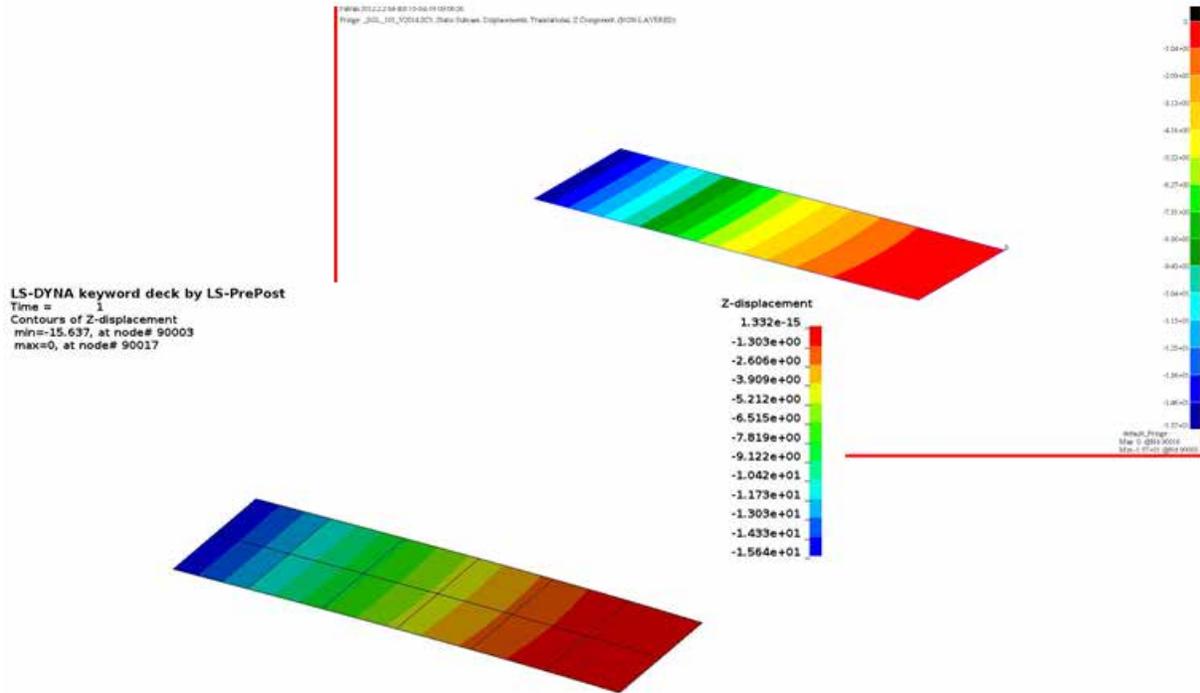Figure 3 Loads and boundary condition of plate model

Figure 4 Contour of Z-displacement in static analysis by NASTRAN and LS-DYNA
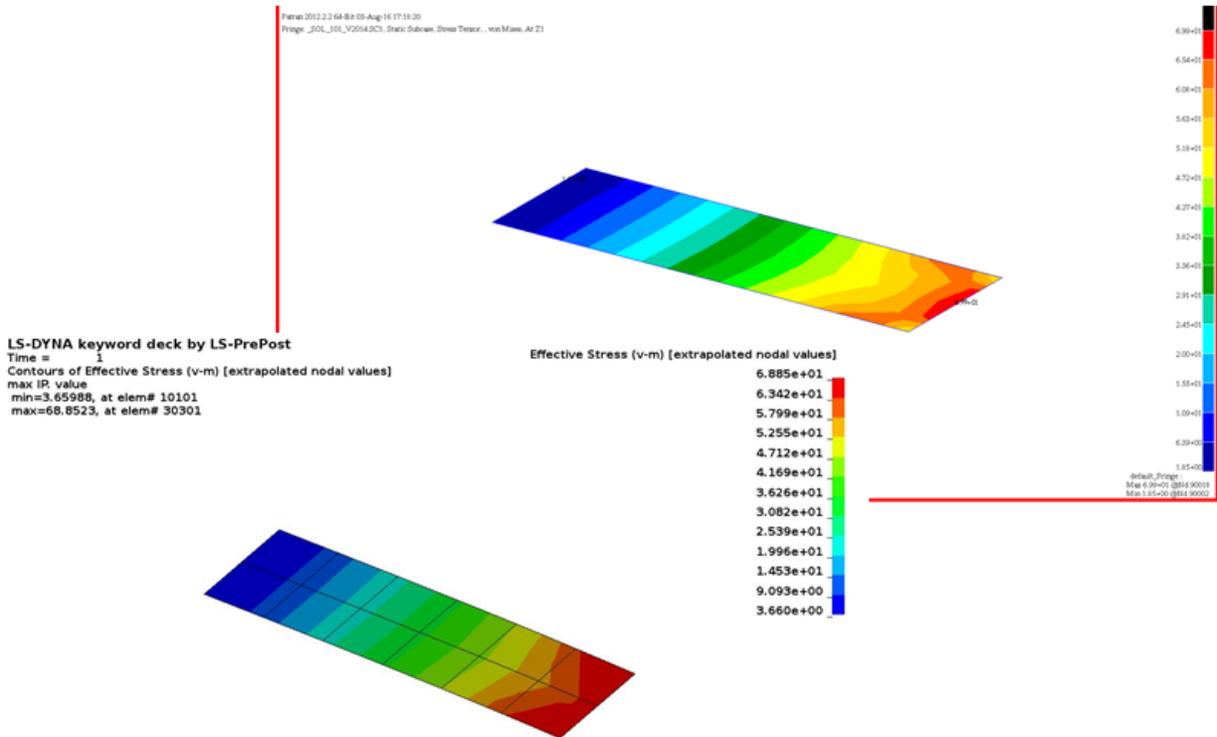

Figure 5 Contour of Von-Mises stress in static analysis by NASTRAN and LS-DYNA

Figure 4 and 5 are displacement and Von-Mises stress contour for comparison. The maximum displacement of LS-DYNA is 15.637mm and with N ASTRAN it is 15.7mm. The maximum Von-Mises stress of LS-DYNA is 68.85 MPa and with NASTRAN it is 69.90 MPa. Both codes give similar results for static analysis.

2.2 Normal mode analysis

Usually the first step in performing a dynamic analysis is determining the natural frequencies and mode shapes of the structure with damping neglected. These results characterize the basic dynamic behavior of the structure and are an indication of how the structure will respond to dynamic loading. NASTRAN normal modes analysis uses command SOL 103. If the parameter COUPMASS is great than zero, it requests the generation of coupled rather than lumped mass matrices for elements with coupled mass capability. The default value (less than zero) causes the generation of lumped mass matrices. The default setting in LS-DYNA is using lumped mass matrices. The consistent mass matrix method in LS-DYA (*CONTROL_IMPLICIT_CONSISTENT_MASS=1) can be used to match with NASTRAN coupled mass matrix.

The sample model (figure 1) is used, and the free boundary condition is used. The table 1 shows results of both codes. There are good match for the normal modes computation bye the two codes.

Table 1 Normal modes analysis comparison

	Lumped mass			Coupled mass		
	NASTRAN	LS-DYNA	Diff	NASTRAN	LS-DYNA	Diff
Mode # (Hz)						
1	9.022E-05	3.204E-04	0.000	2.524E-05	3.611E-04	0.000
2	5.721E-05	2.912E-04	0.000	8.796E-05	3.381E-04	0.000
3	4.136E-05	2.601E-04	0.000	1.084E-04	2.764E-04	0.000
4	8.746E-05	9.228E-05	0.000	1.1823E-04	9.799E-05	0.000
5	1.262E-04	1.525E-04	0.000	2.104E-04	1.103E-04	0.000
6	1.986E-04	1.651E-04	0.000	2.560E-04	1.868E-04	0.000
7	7.218E+01	7.211E+01	0.085	7.426E+01	7.418E+01	0.102
8	1.269E+02	1.243E+02	2.015	1.353E+02	1.326E+02	2.042
9	1.976E+02	1.970E+02	0.268	2.103E+02	2.097E+02	0.303
10	2.657E+02	2.593E+02	2.384	2.884E+02	2.814E+02	2.411
11	3.828E+02	3.810E+02	0.478	4.263E+02	4.240E+02	0.526
12	4.268E+02	4.136E+02	3.084	4.781E+02	4.631E+02	3.126

2.3 Direct frequency response

In direct frequency response, structure response is based on the global stiffness, mass and damping matrices of the system and no modal analysis is needed. NASTRAN direct frequency analysis uses command SOL 108. The keyword *FREQUENCY_DOMAIN_SSD_DIRECT is used to run the direct frequency response solver in LS-DYNA. Since the LS-DYNA direct frequency response solver was implemented recently, there is no validation at this time.

Comparing the direct SSD and the mode-based SSD, the main difference is the unknown variables. For direct SSD, the physical nodal displacements u are used as the primary variables. As the consequence it may result in a large system of equations. For the mode-based SSD, the primary variables are the model coefficients (or generalized coordinates). So the number of unknown variables is equal to the number of eigenmodes adopted in the simulation and it is usually moderate (considering that for many structures, hundreds of eigenmodes can be sufficient for most applications). Besides, with mode-based SSD, the system of equations is decoupled using the orthogonality property of the eigenvectors. So, the mode-based SSD can be cheaper than direct SSD, if the system has a large number of DOF.

Figure 6 compares the displacement of plate model given by direct SSD and mode-based SSD in LS-DYNA. A reasonable match can be found for the results by the two methods.
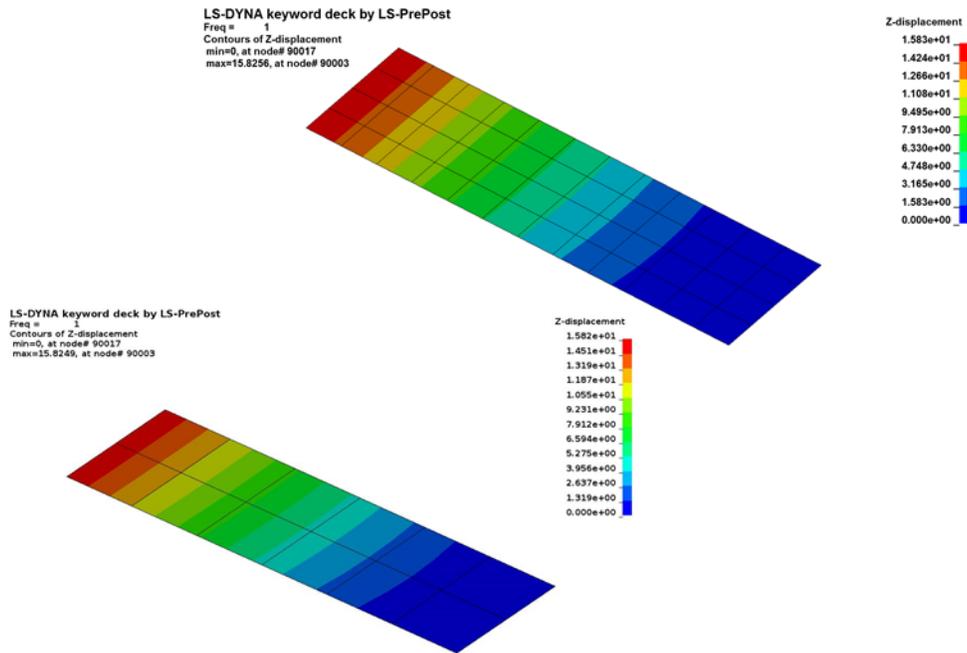
Figure 6 Contour of Z-displacement by mode based SSD and direct SSD in LS-DYNA

2.4 Modal frequency response

Frequency response analysis is a method used to compute structural response to steady-state oscillatory excitation. In frequency response analysis the excitation is explicitly defined in the frequency domain. All of the applied forces are known at each forcing frequency. Forces can be in the form of applied forces and/or enforced motions (displacements, velocities, or accelerations).

In NASTRAN, modal frequency response analysis is requested by the command SOL 111. The keyword *FREQUENCY_DOMAIN_SSD is used to run the modal frequency response solver in LS-DYNA. Two types of harmonic loads (pressure and nodal force) are applied to the plate simultaneously, one edge of which is fixed (figure 3).
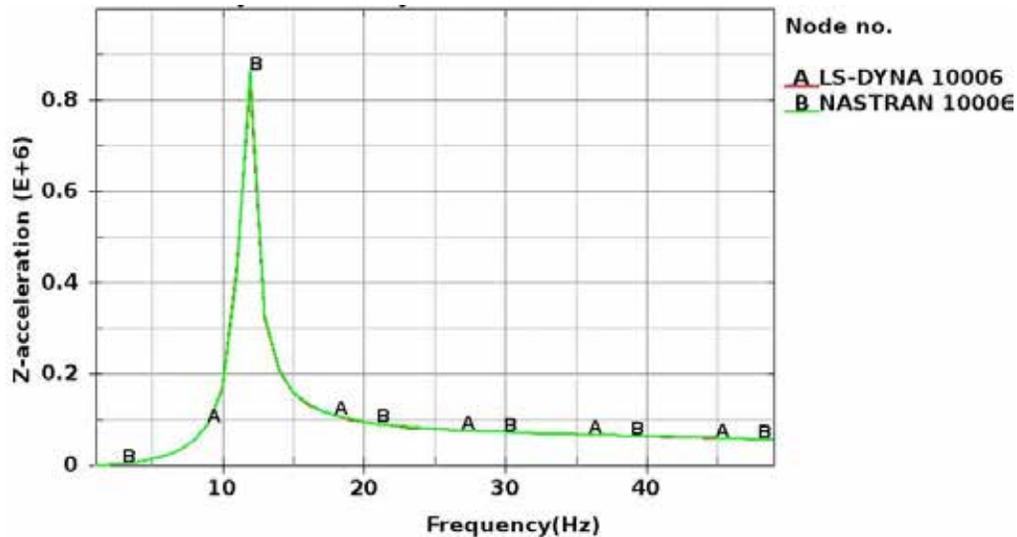

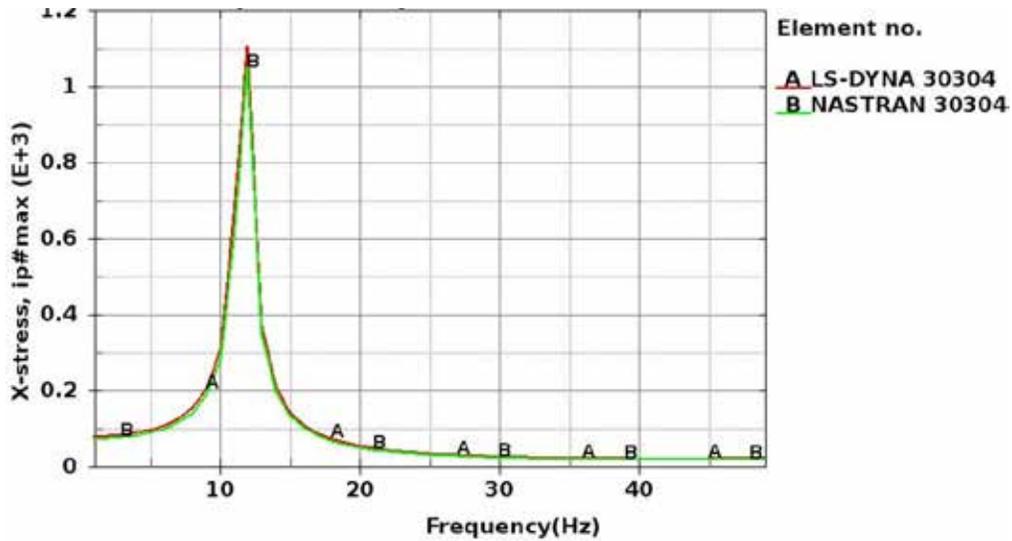Figure 7 Z-acceleration of nodal 10006

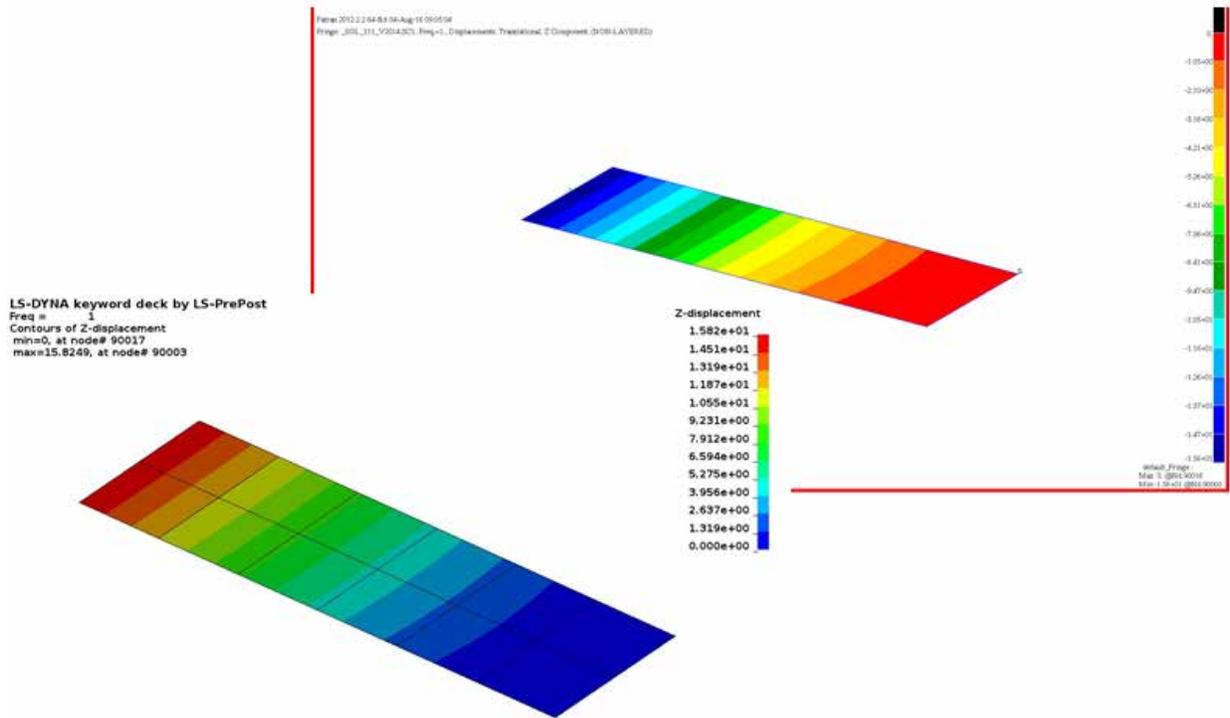Figure 8 X stress of element 30304

Figure 9 Contour of Z-displacement in modal frequency response by NASTRAN and LS-DYNA

Figure 7 compares Z acceleration of the nodal 10006 and figure 8 compares the X stress of element 30304. Figure 9 and 10 are displacement and X stress contour of comparison. The maximum displacement by LS-DYNA is 15.8249mm and it is 15.8mm by NASTRAN. The maximum X stress by LS-DYNA is 84.0976 MPa and it is 79.3 MPa by NASTRAN. We get similar results of modal frequency analysis by the two codes.
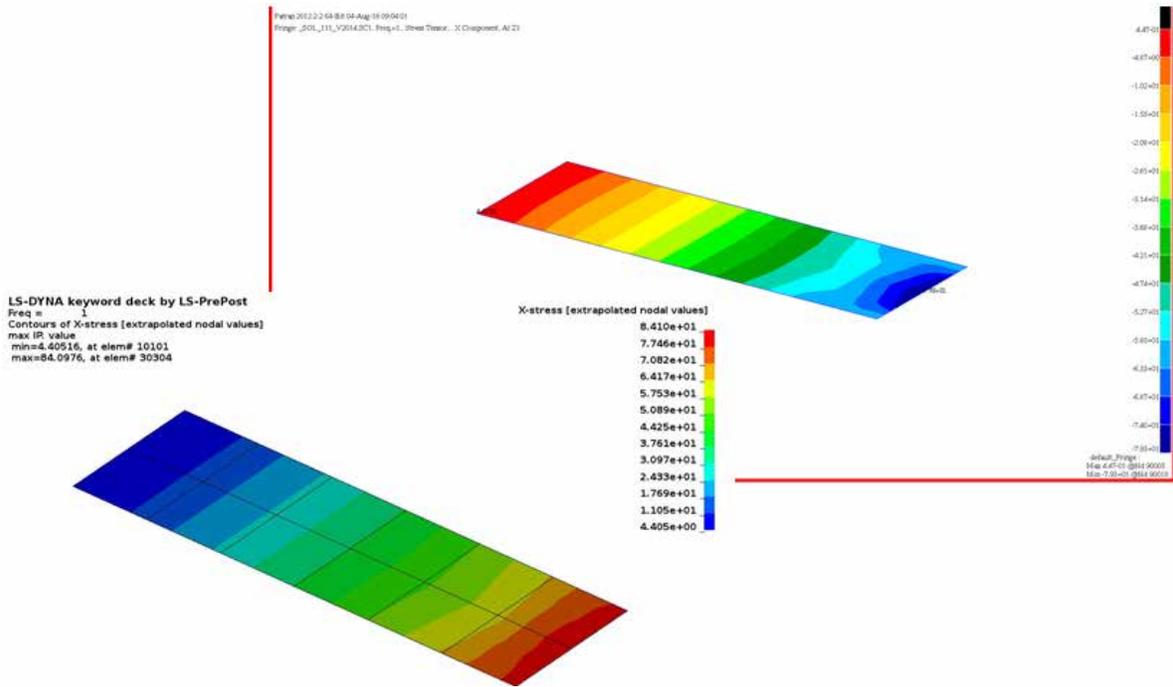
Figure 10 Contour of X stress in modal frequency response by NASTRAN and LS-DYNA

2.5 Direct transient response

In direct transient response, structural response is computed by solving a set of coupled equations using direct numerical integration. The fundamental structural response is solved at discrete times, typically with a fixed integration time step.

In NASTRAN, direct transient response analysis is requested by the command SOL 109. The NSOLVR of keyword *CONTROL_IMPLICIT_SOLUTION is set to 1 to do implicit linear analysis (direct transient response) in LS-DYNA. Two types of dynamic time loads (pressure and nodal force) are applied to the plate simultaneously, one edge of which is fixed (figure 3).
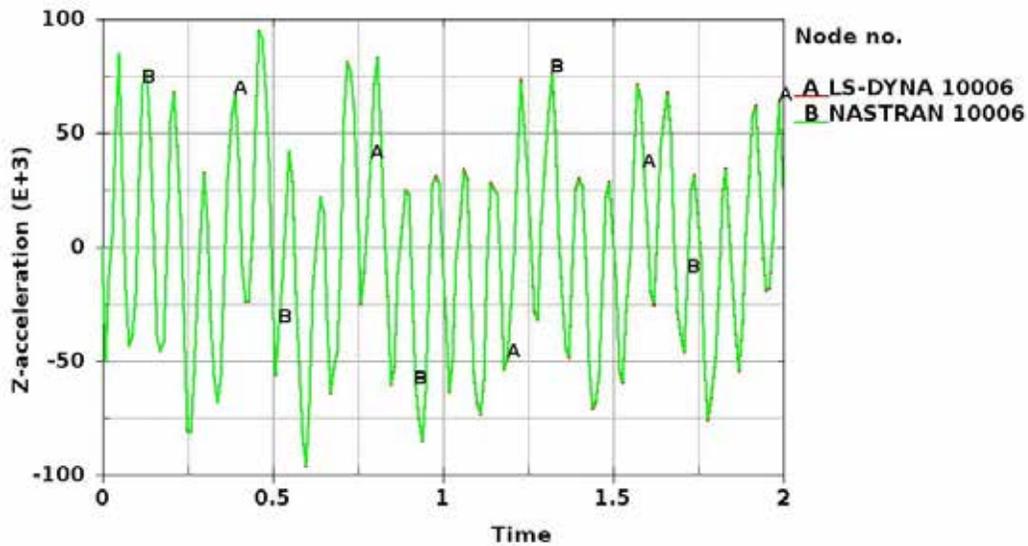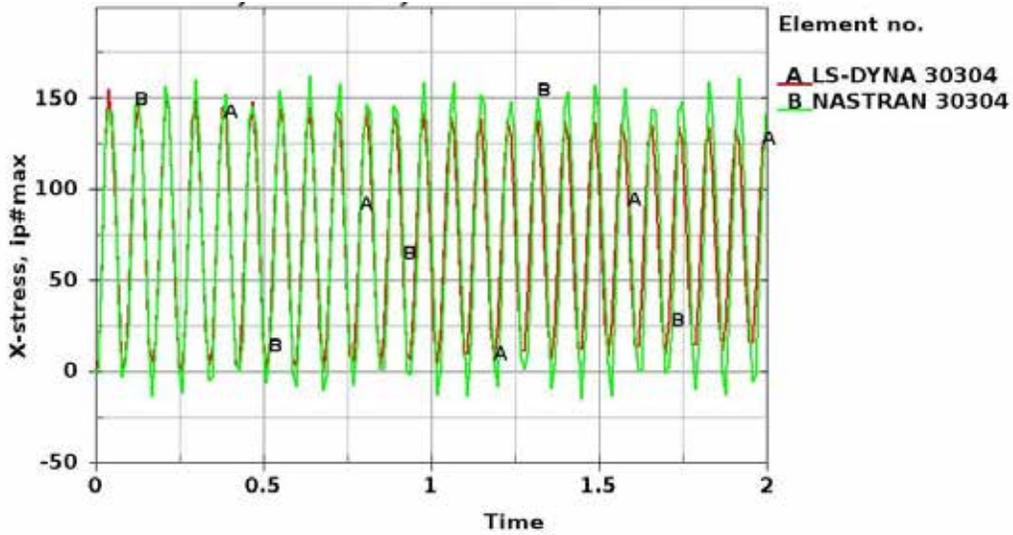

Figure 11 Z-acceleration of nodal 10006

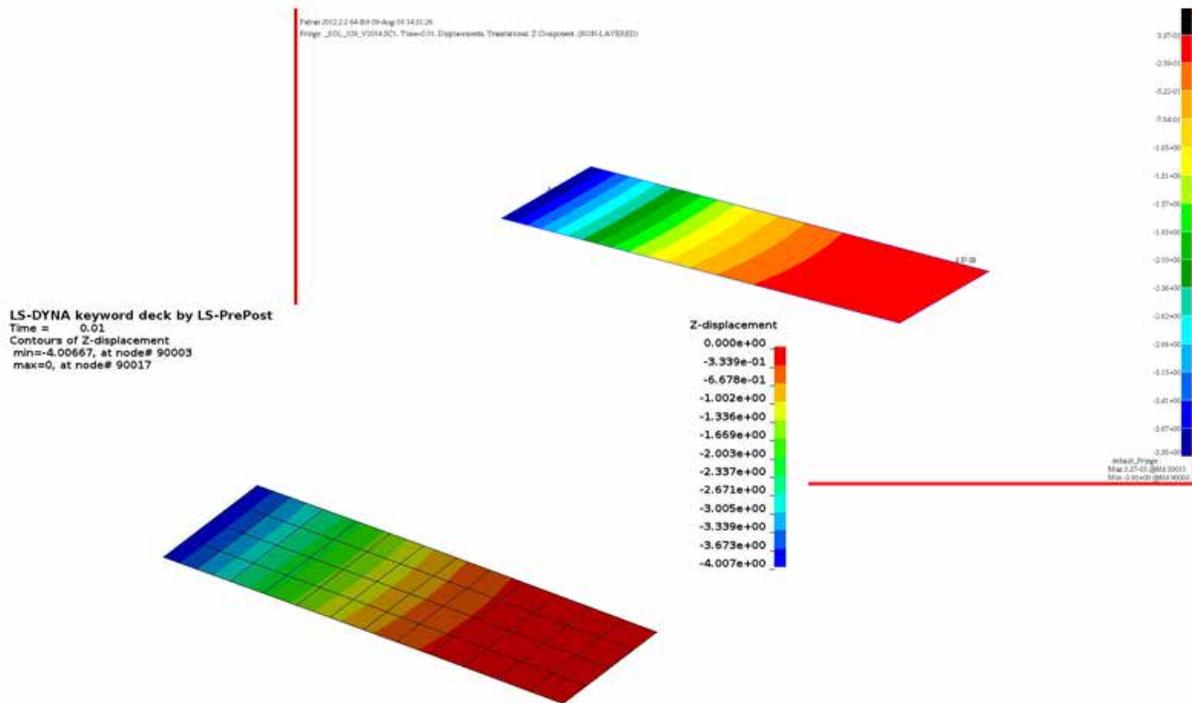Figure 12 X stress of element 30304

Figure 13 Contour of Z-displacement in direct transient response by NASTRAN and LS-DYNA

Figure 11 compares Z acceleration of the nodal 10006 and figure 12 compares the X stress of element 30304. Figure 13 and 14 are displacement and X stress contour for comparison. The maximum displacement by LS-DYNA is 4.007mm and with NASTRAN it is 3.93mm. The maximum X stress by LS-DYNA is 17.63 MPa and with NASTRAN it is 19.0 MPa. We get similar results of direct transient analysis by the two codes.
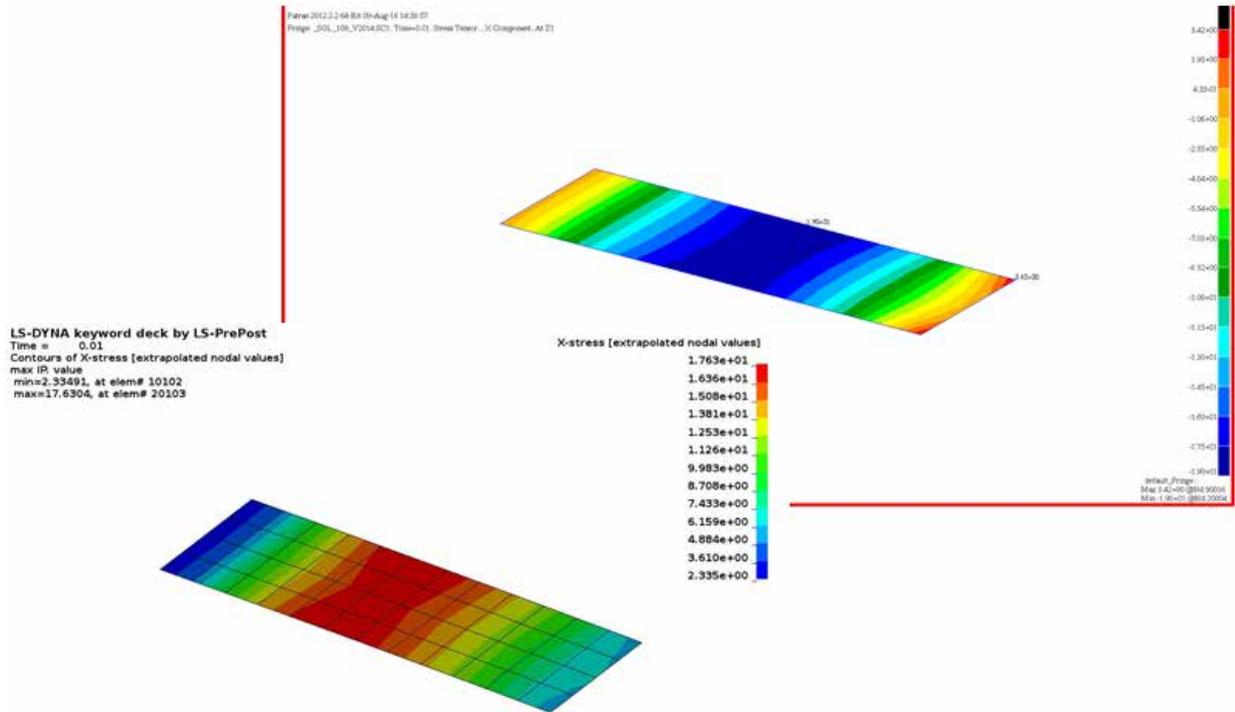
Figure 14 Contour of X stress in direct transient response by NASTRAN and LS-DYNA

2.6 Modal transient response

Modal transient response is an alternate approach to computing the transient response of a structure. This method uses the mode shapes of the structure to reduce the size, uncouple the equations of motion (when modal or no damping is used), and make the numerical integration more efficient. Since the mode shapes are typically computed as part of the characterization of the structure, modal transient response is a natural extension of a normal modes analysis.
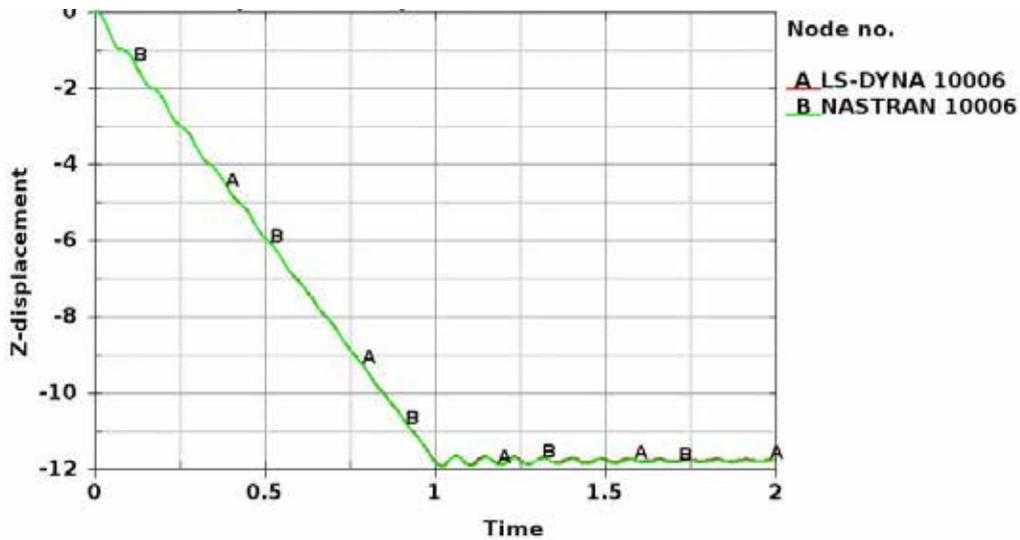

Figure 15 Z-displacement of Nodal 10006

In NASTRAN, modal transient response analysis is requested by the command SOL 112. The keyword *CONTROL_IMPLICIT_MODAL_DYNAMIC is used to do modal transient response analysis in LS-DYNA. Two types of dynamic time loads (pressure and nodal force) are applied to the plate simultaneously, one edge of which is fixed (figure 3).

Figure 15 compares Z displacement of the nodal 10006. Figure 16 is displacement contour for comparison. The maximum displacement by LS-DYNA is 1.079mm and with NASTRAN it is 1.08mm. We get similar results of modal transient analysis by the two codes.
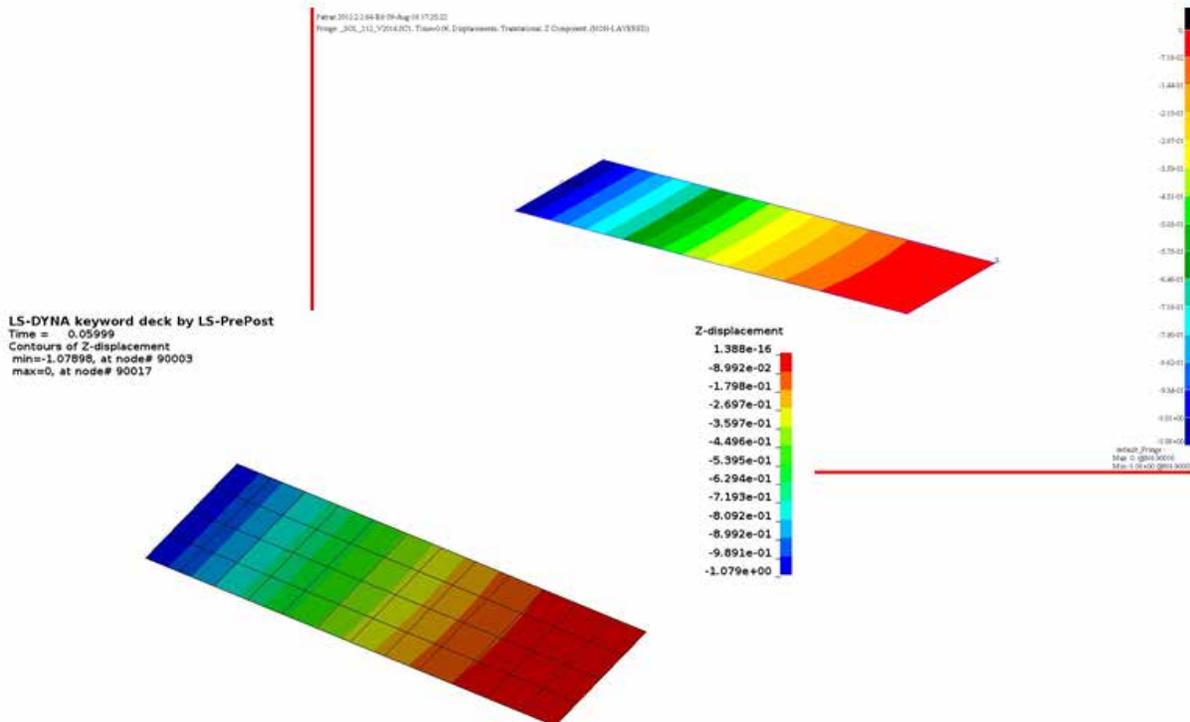

Figure 16 Contour of Z-displacement in modal transient analysis by NASTRAN and LS-DYNA

3. Summary

A summary for the cross validation of the solution sequences in LS-DYNA and NASTRAN is provided in the paper. For most cases, a good match between LS-DYNA and NASTRAN are reached. The area where LS-DYNA needs further development and improvement is also pointed out. This paper can provide the guidance on selection of the solution sequence for the users who need to run linear analysis with LS-DYNA.

With this project, we will continue to work to improve the functions, accuracy, and efficiency of the linear solvers within LS-DYNA, including more types of elements, such as rigid, damping…, and spring elements, and special features, such as residual vectors, ground motion…etc.

References

[1]   Livermore Software Technology Corporation: "LS-DYNA Keyword User's Manual", 2016
[2]   Handbook, MSC/NASTRAN and/or Altair/OptiStruct