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

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

[1]Ford Motor Company
[2]Livermore Software Technology Corporation

Oct. 2017

Content

- Introduction
- Validation examples of solution sequence
- Summary

Introduction

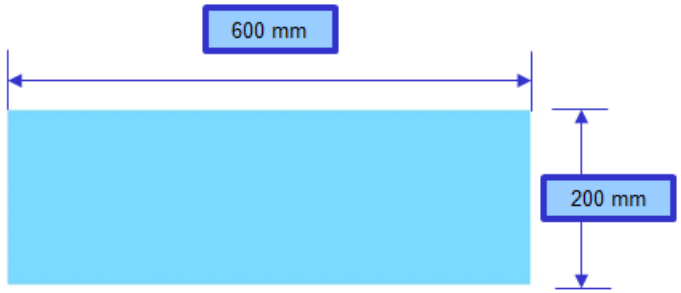
Each type of analysis available in NASTRAN is called a solution sequence. A solution is a feature selected by a SOL executive control command.

- SOL101 (static analysis)
- SOL103 (normal mode analysis)
- SOL108 (direct frequency response)
- SOL109 (direct transient response)
- SOL111 (modal frequency response)
- SOL112 (modal transient response)

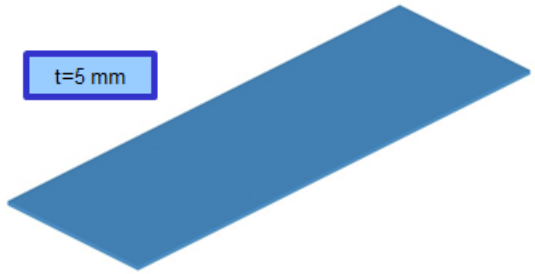
LS-DYNA provides equivalent linear analysis functions

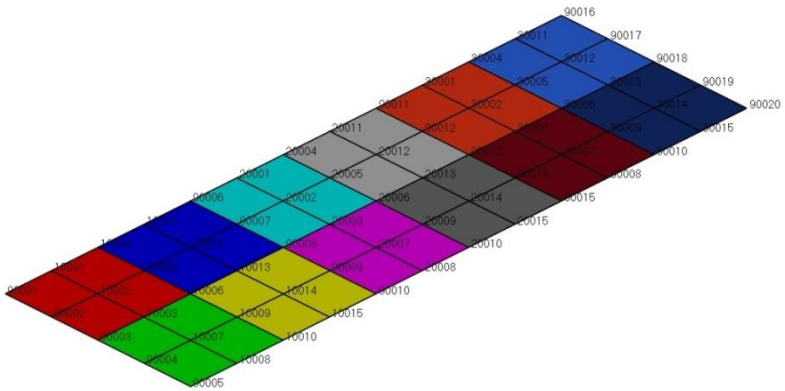
- Static analysis
- Normal mode analysis
- Direct SSD analysis
- Direct transient analysis
- Mode based steady state dynamics (SSD) analysis
- Modal transient dynamics

Problem Statement



- Material Properties
  - Yang's modulus 2.07E+05 MPa
  - Poisson ratio 0.3
  - Mass Density 7.831E-09 Mg/mm^3
- CAE Model

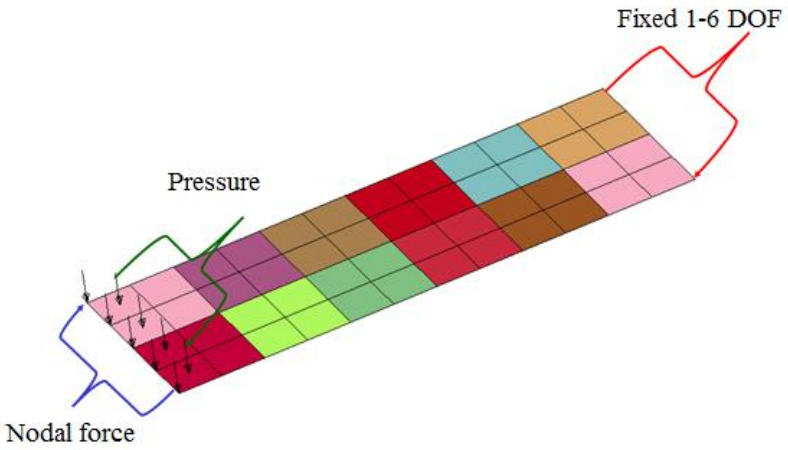




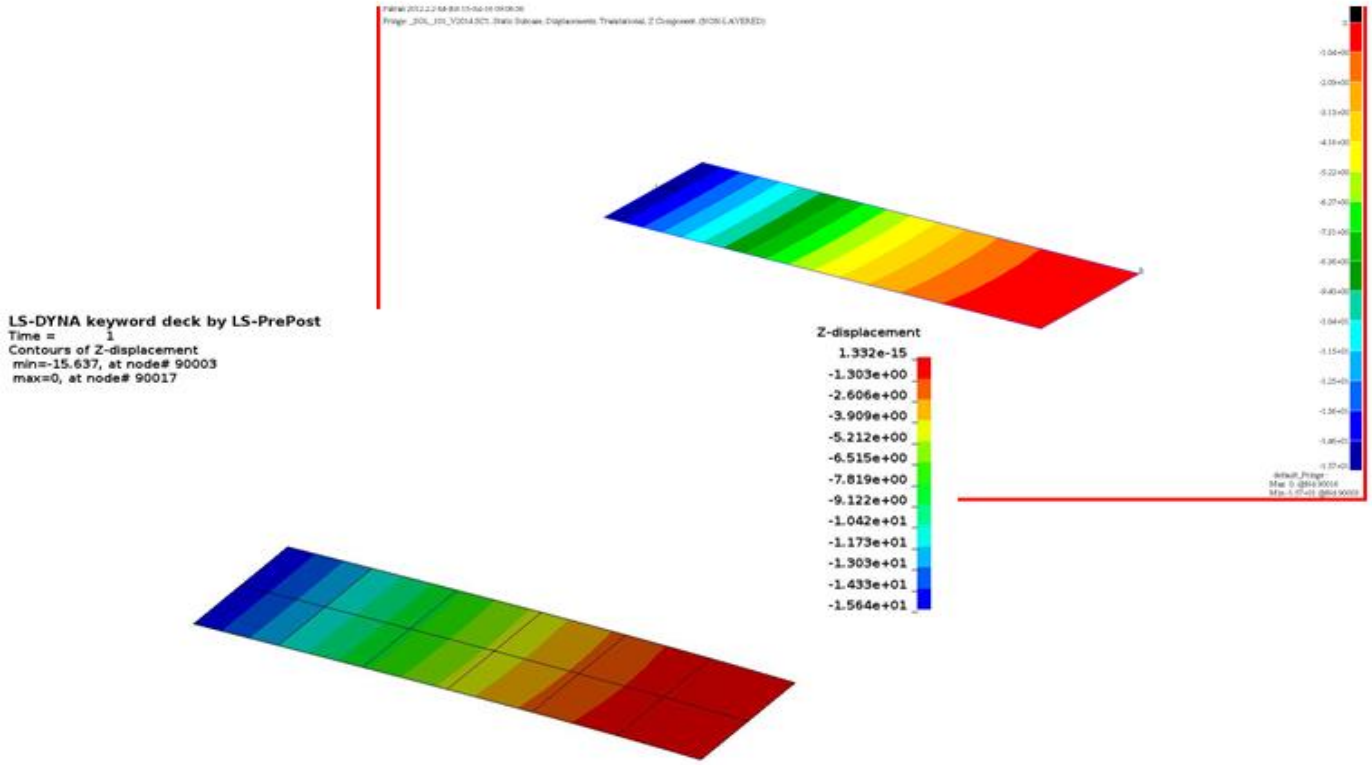
Static analysis

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.

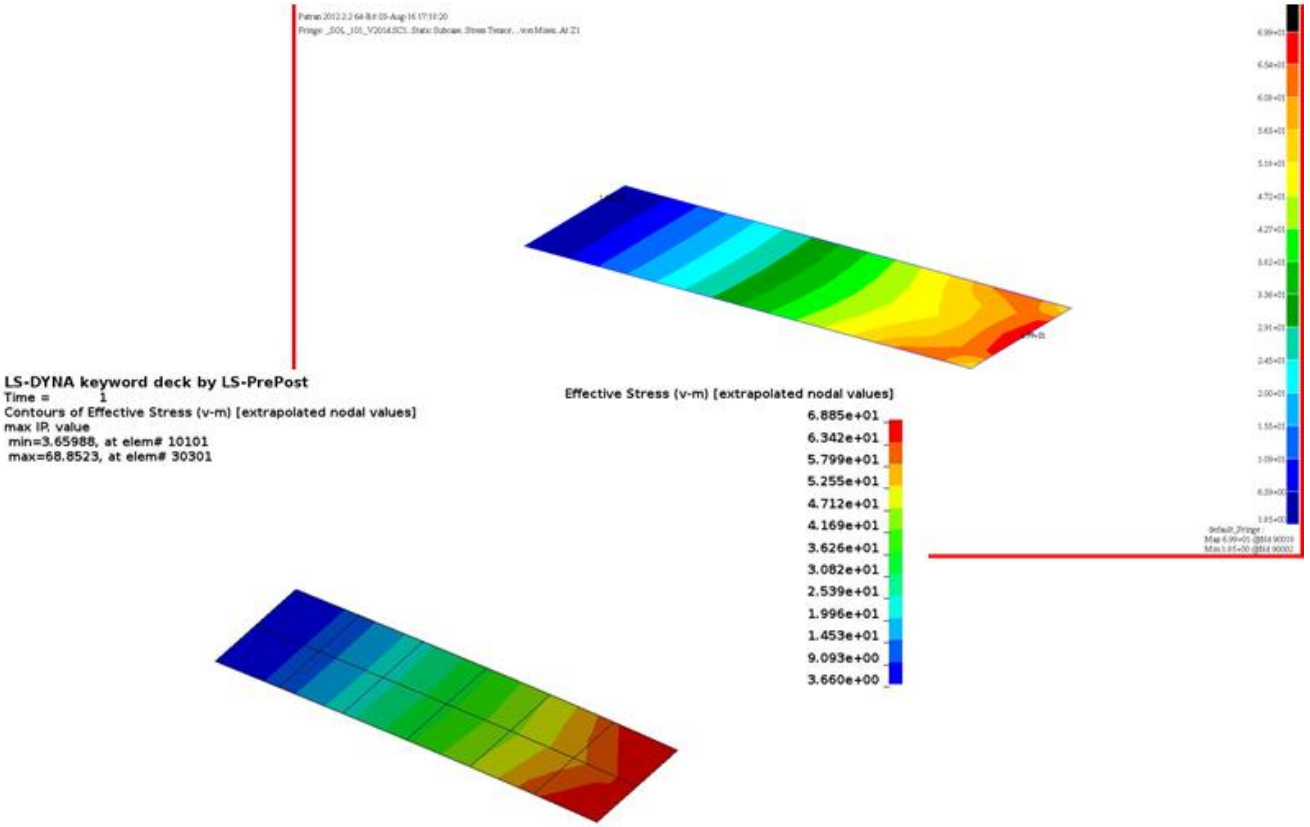


Static analysis



The maximum displacement of LS-DYNA is 15.637mm.
NASTRAN it is 15.7mm.

Static analysis



The maximum Von-Mises stress of LS-DYNA is 68.85 MPa.
NASTRAN it is 69.90 MPa.

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.

- COUPMASS >0 means coupled mass matrices.
- COUPMASS <0 means lumped mass matrices (default).

LS-DYNA is using *CONTROL_IMPLICIT_EIGENVALUE

- The default is lumped mass matrices.
- *CONTROL_IMPLICIT_CONSISTENT_MASS setting to 1 is used to match with NASTRAN coupled mass matrices.

Normal mode analysis

	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%

Direct frequency response

NASTRAN direct frequency analysis uses command SOL 108.

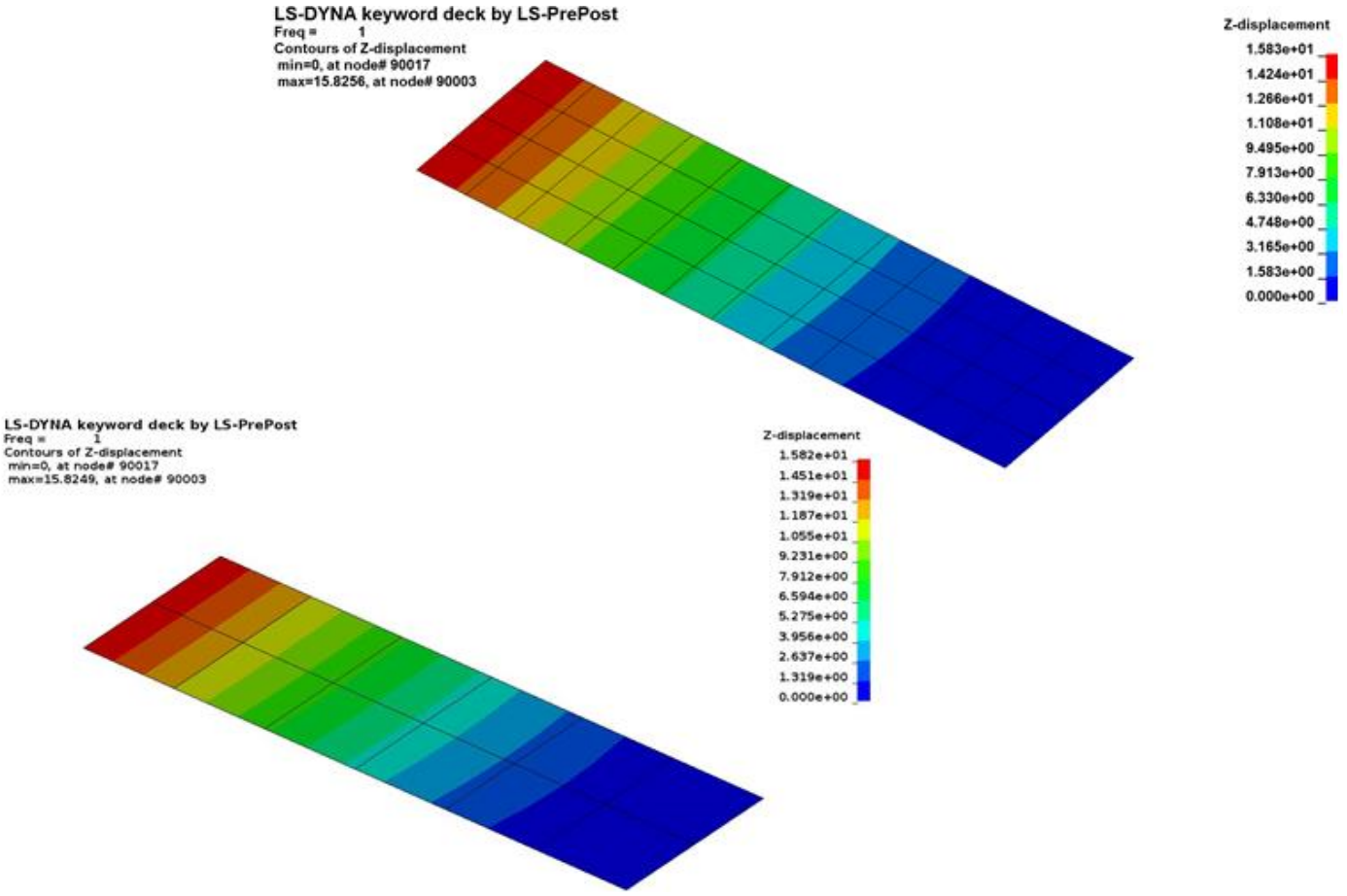
The keyword ***FREQUENCY_DOMAIN_SSD_DIRECT** is used in LS-DYNA

For direct SSD, structure response is based on the global stiffness, mass and damping matrices of the system and no modal analysis is needed. The physical nodal displacements 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).

Since the LS-DYNA direct frequency response solver was implemented recently, there is no validation at this time.

Direct frequency response



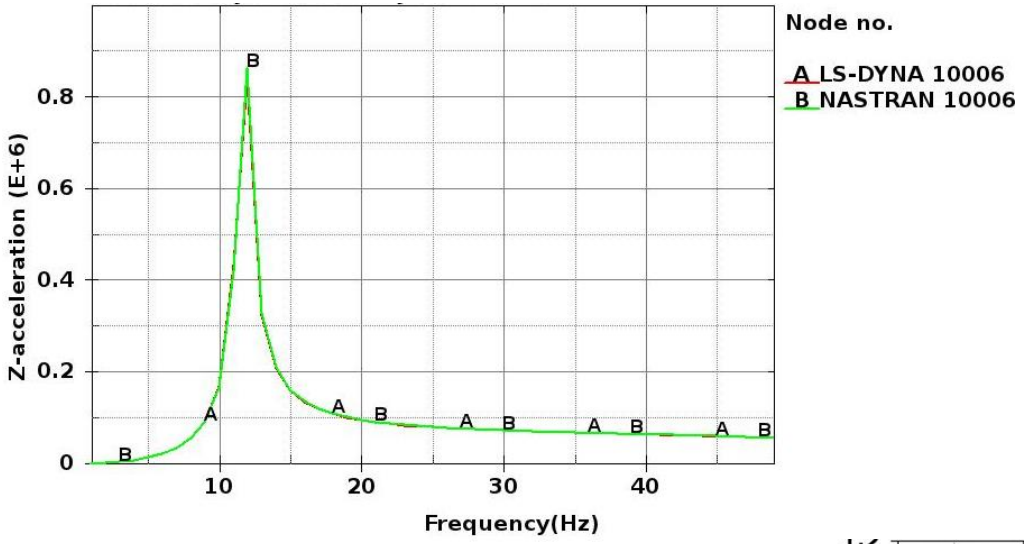
Modal frequency response

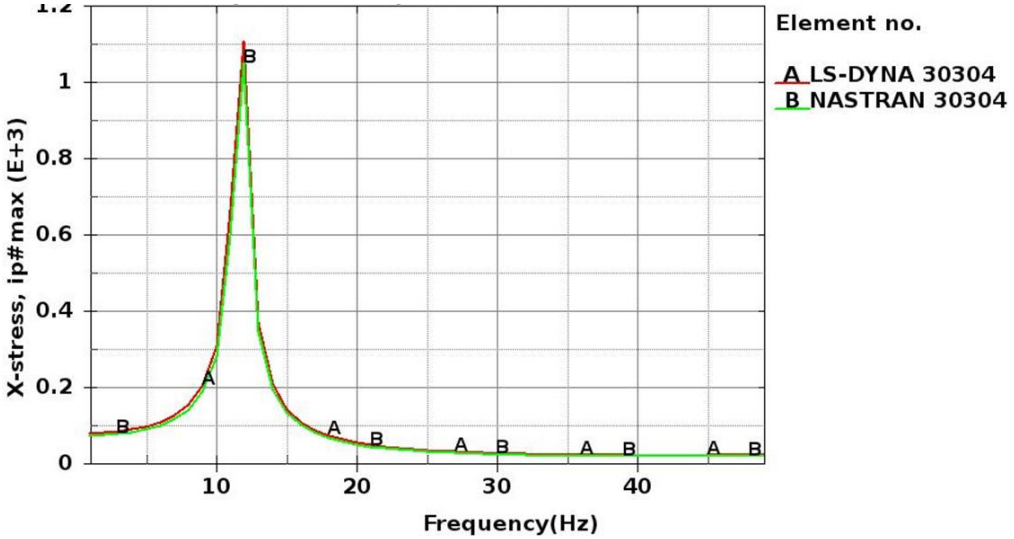
Frequency response analysis is a method used to compute structural response to steady-state oscillatory excitation. In frequency response analysis the excitation is explicitly defined in the frequency domain. All of the applied forces are known at each forcing frequency. Forces can be in the form of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

NASTRAN modal frequency analysis uses command SOL 111.

The keyword ***FREQUENCY_DOMAIN_SSD** is used in LS-DYNA

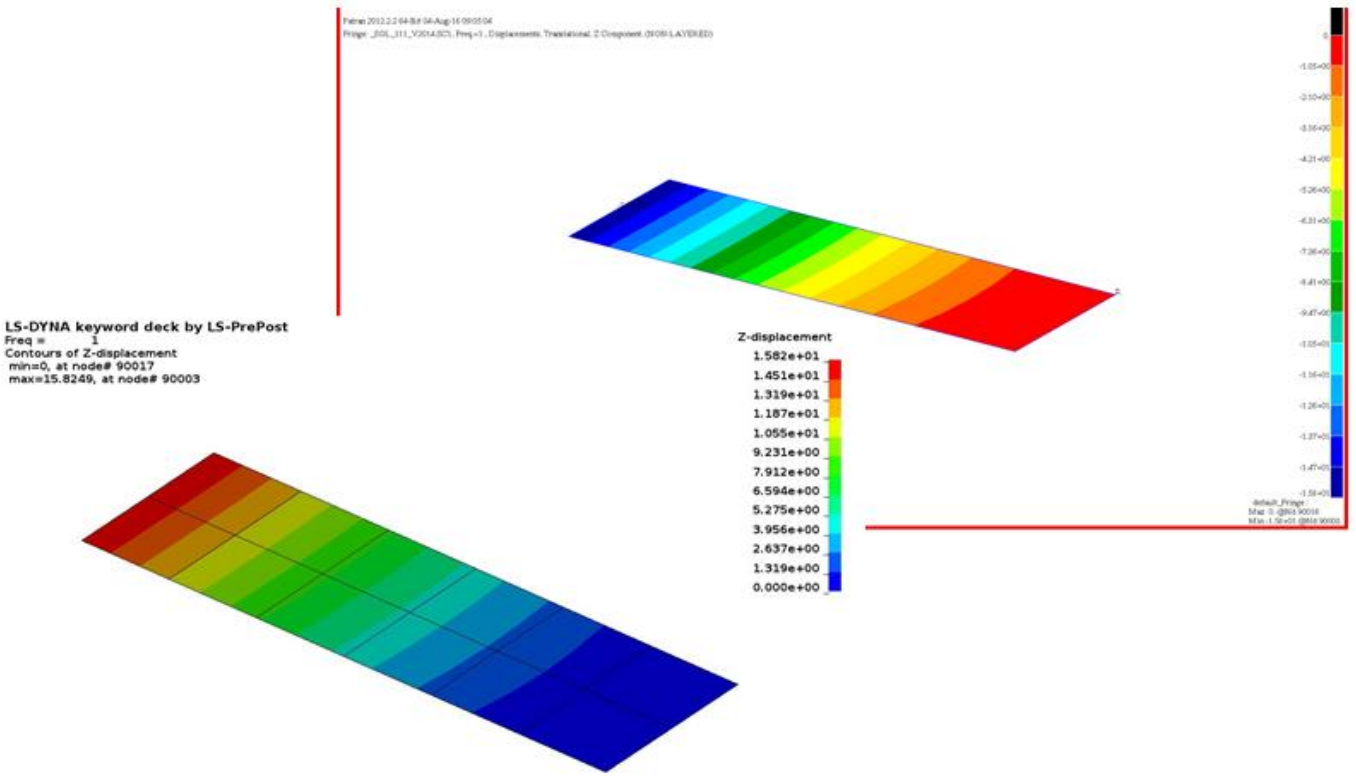
Modal frequency response





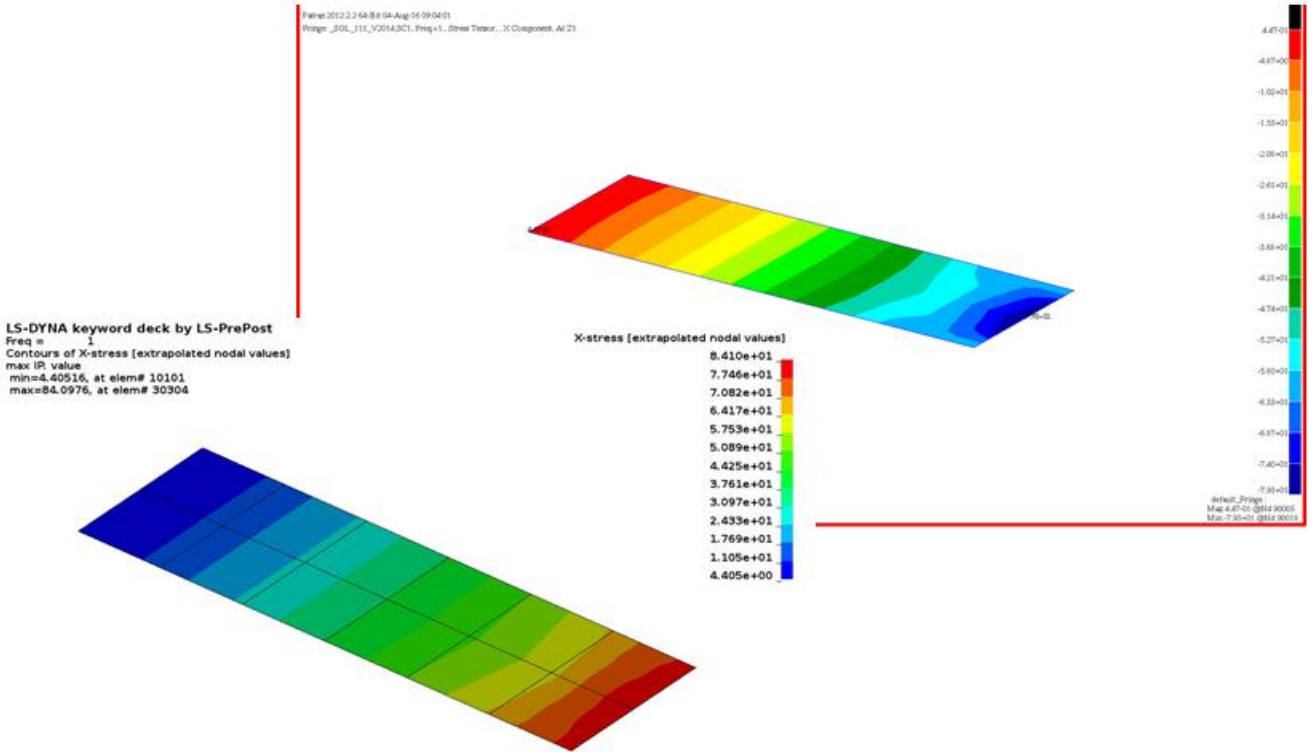
Modal frequency response

LS-DYNA



The maximum displacement by LS-DYNA is 15.8249mm.
It is 15.8mm by NASTRAN.

Modal frequency response



The maximum X stress by LS-DYNA is 84.0976 Mpa.
It is 79.3 MPa by NASTRAN.

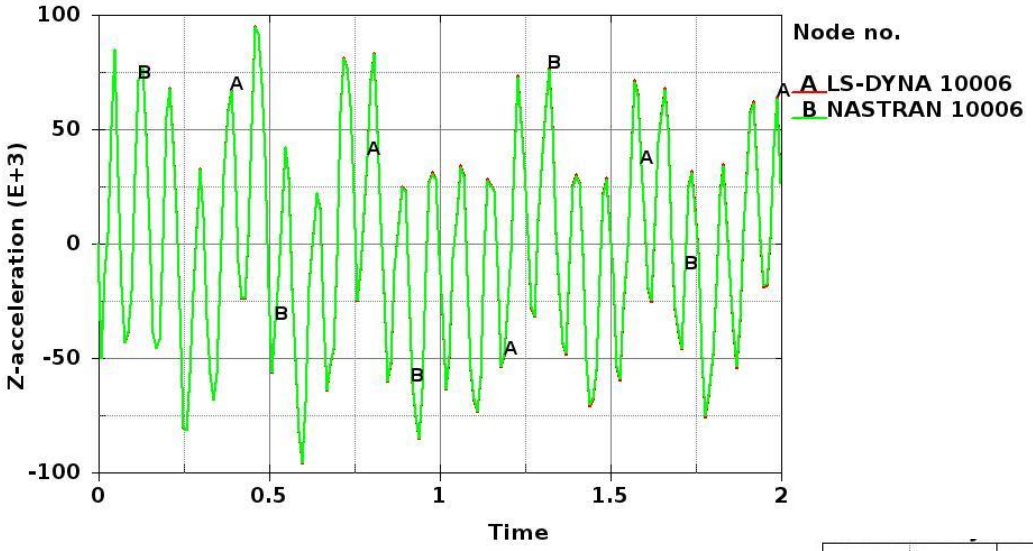
Direct transient response

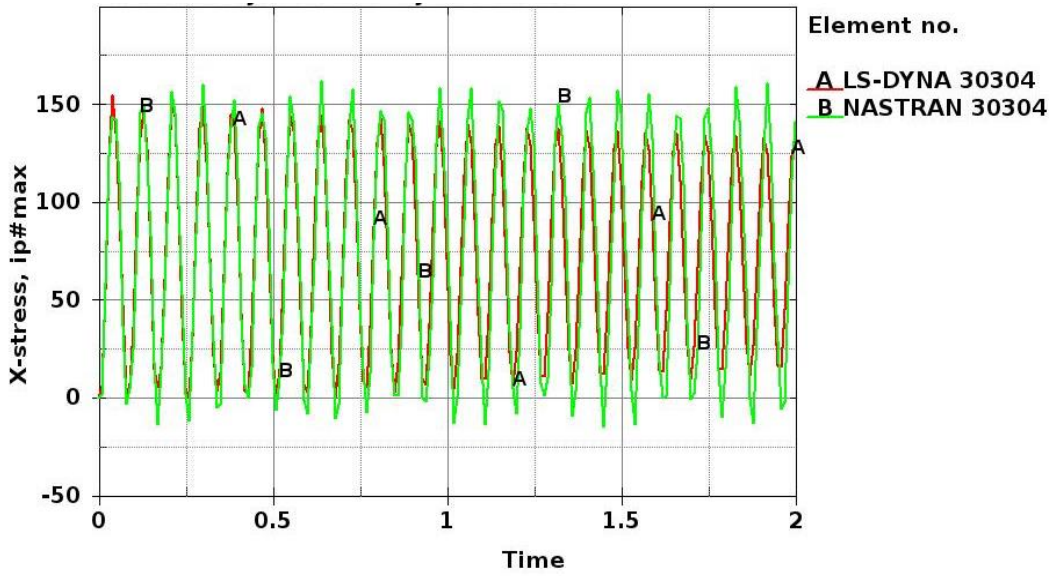
In direct transient response, structural response is computed by solving a set of coupled equations using direct numerical integration. The fundamental structural response is solved at discrete times, typically with a fixed integration time step.

NASTRAN direct transient analysis uses command SOL 109.

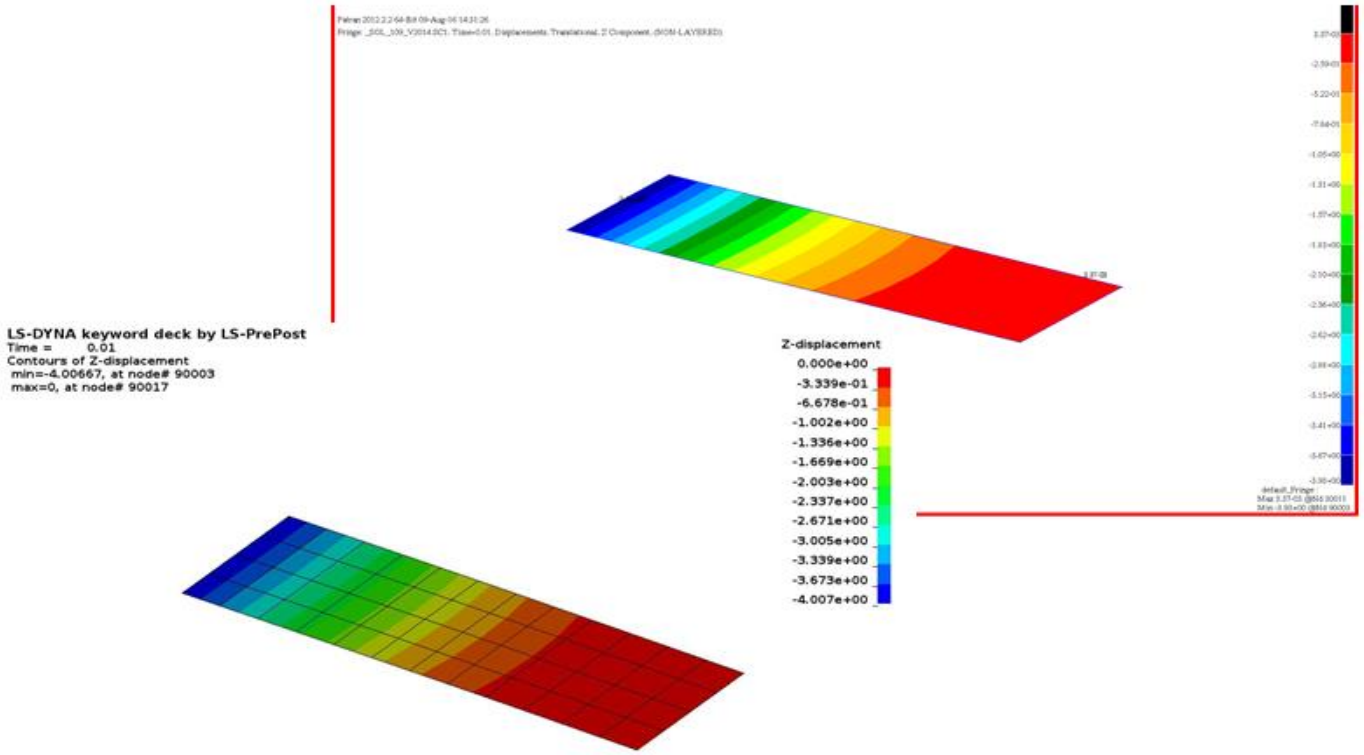
The NSOLVR of keyword *CONTROL_IMPLICIT_SOLUTION is set to 1 to do implicit linear analysis and the IMASS of *CONTROL_IMPLICIT_DYNAMIC is equal to 1 for dynamic analysis (direct transient response) in LS-DYNA.

Direct transient response



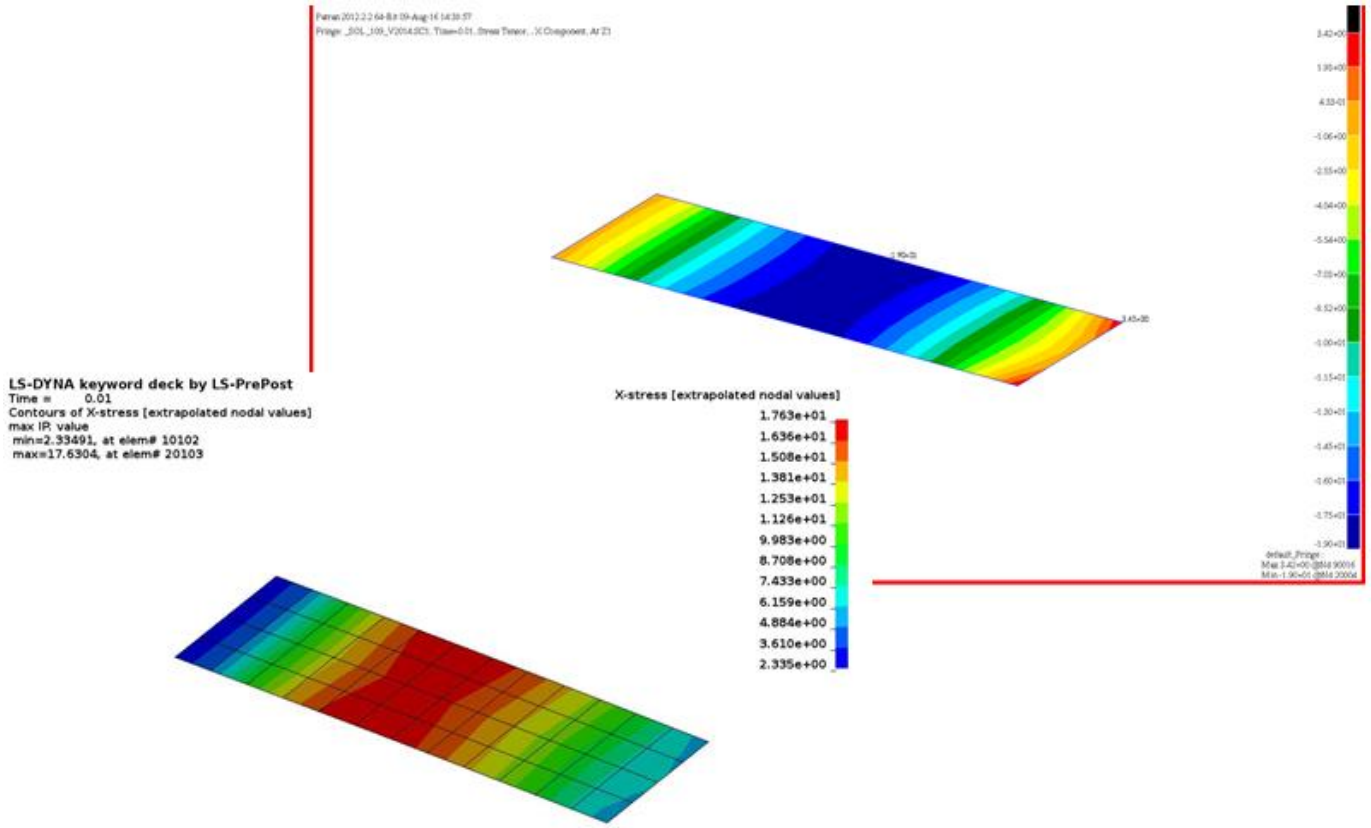


Direct transient response



The maximum displacement by LS-DYNA is 4.007mm.
NASTRAN it is 3.93mm.

Direct transient response



The maximum X stress by LS-DYNA is 17.63 MPa.
NASTRAN it is 19.0 MPa.

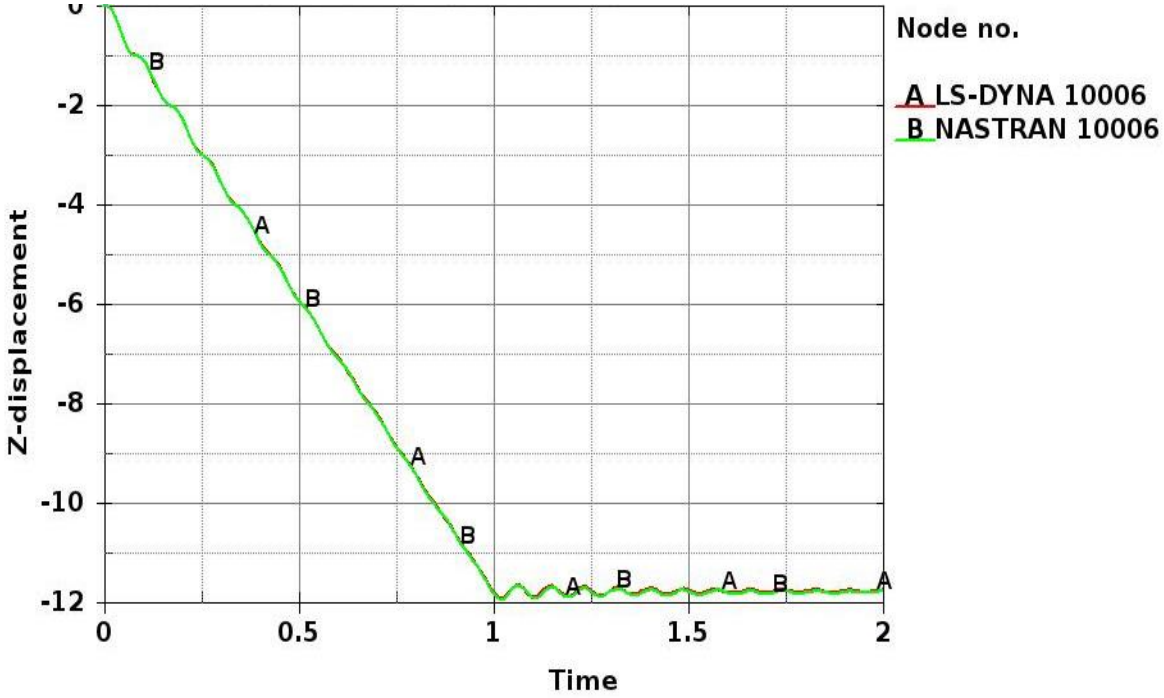
Modal transient response

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.

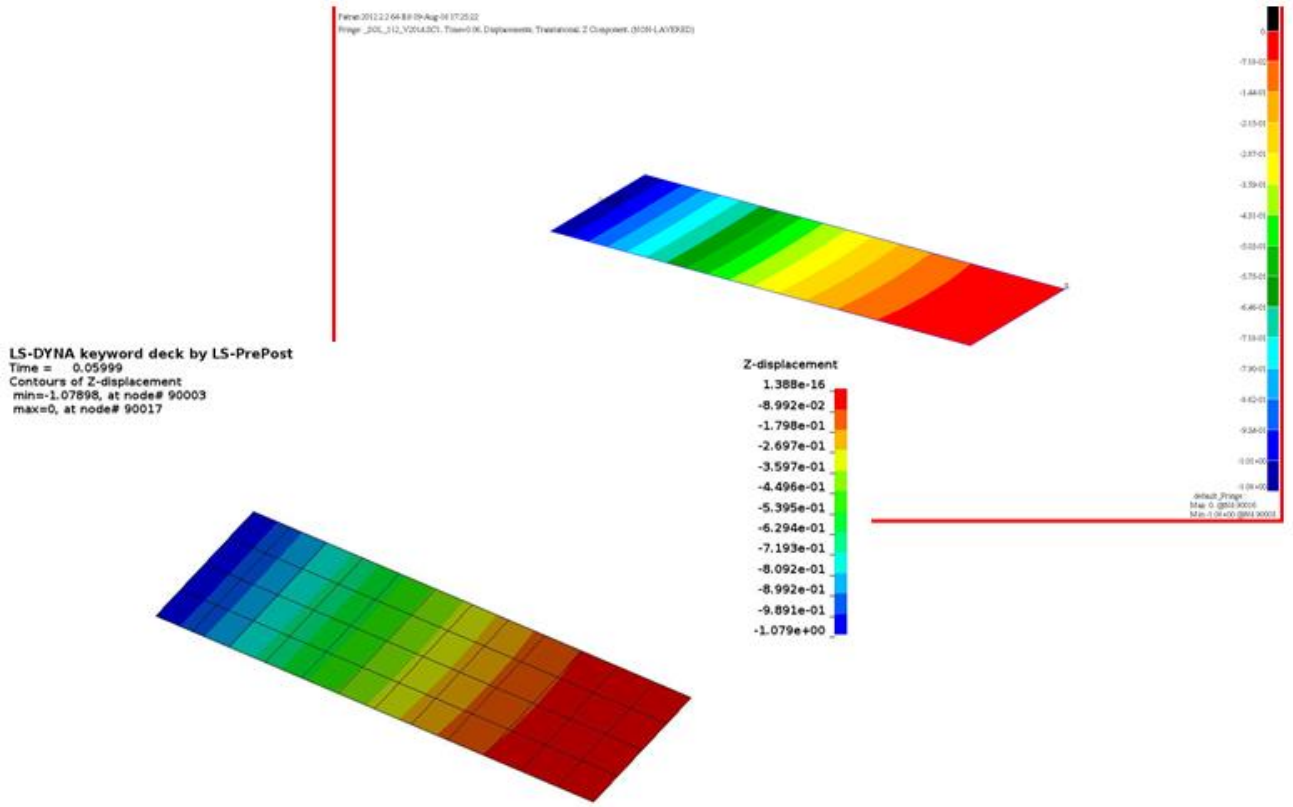
In NASTRAN, modal transient response analysis is requested by the command SOL 112.

The keyword *CONTROL_IMPLICIT_MODAL_DYNAMIC is used to do modal transient response analysis in LS-DYNA.

Modal transient response



Modal transient response



The maximum displacement by LS-DYNA is 1.079mm.
NASTRAN it is 1.08mm

Summary

- A good match between LS-DYNA and NASTRAN are reached for most of cases.
- Pointed out the area where LS-DYNA needs further development and improvement.
- Help users to get better understanding on the current status of LS-DYNA's linear solvers
- Provides the guidance on selection of the solution sequence for the users who need to run linear analysis with LS-DYNA

Thank you