

LS-DYNA[®]'s Linear Solver Development — Phase 1: Element Validation

Allen T. Li¹, Zhe Cui², Yun Huang²

¹Ford Motor Company

²Livermore Software Technology Corporation

Abstract

LS-DYNA is a well-known multi-purpose explicit and implicit finite element code. It is mainly used to analyze the nonlinear response of structures. To answer increasing requests from users, LSTC is taking a big effort to develop and improve the linear solution capabilities in LS-DYNA[®]. As part of this endeavor, a joint project was launched between Ford and LSTC to validate the linear solvers in LS-DYNA. In this project, a bunch of benchmark examples are tested using LS-DYNA and commercial code NASTRAN and the results are compared. The NASTRAN results are provided by Ford Motor Company. The purpose of this study is to: 1) validate the linear solution provided by LS-DYNA; and 2) identify the corresponding elements, material models, boundary conditions, loading types, and solution types in LS-DYNA which have the best match with the counterparts in NASTRAN. This provides us also a great opportunity to check which solvers of linear analysis are available in LS-DYNA and are ready for the users, and which are still missing and need further development.

As the first part of the joint project, we focus on comparing the elements between LS-DYNA and NASTRAN. In this paper, the structural elements including line elements (CBEAM, CBEAM3, CBAR, CBEND, CROD, CONROD and CTUBE), surface elements (CTRIA3, CTRIA6, CQUAD4, and CQUAD8) and solid elements (CTETRA4, CTETRA6, CPENTA6, CPENTA15, CHEXA8 and CHEXA20) are investigated. For line elements, a simple cantilever beam model is adopted to validate the solution from two types of analysis (static analysis and normal mode analysis). For surface and solid elements, several academic problems are used to validate the solution from the same two types of analysis (static analysis and normal mode analysis). The results for validations include displacements, natural frequencies, mode shapes, and stresses.

A summary for the validation on the elements 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 work allows users to get better understanding on the current status of LS-DYNA's linear solvers as well as provides suggestions of how to select the types of elements and parameters in LS-DYNA for the users who need to translate their finite element models between NASTRAN and LS-DYNA.

Line elements

In general, the line elements are used for two dimensions geometry. There are CBAR, CBEAM, CBEAM3, CBEND, CROD, CONROD and CTUBE, etc. line elements available in NASTRAN. The keyword *SECTION_BEAM is used to define beam element in LS-DYNA, which includes different element formulations.

A simple cantilever beam model is used to validate the response from two simple analyses, static analysis and normal mode analysis. The validate response data is from handbook of MSC/NASTRAN and/or Altair/OptiStruct. The figure 1 shows the model, and the parameters of the model are, length $L = 1000$ (mm), elastic modulus $E = 210000.0$ (MPa), Poisson Ratio $\nu = 0.3$, and density $\rho = 7.85E-09$ (Ton/mm³).

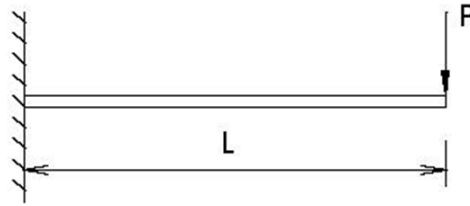


Figure 1 Simple cantilever beam

CBAR element

The bar element is defined with a CBAR entry and its properties are defined with a PBAR entry. The bar element is a one dimensional bending element which is prismatic, and for which the elastic axis, gravity axis, and shear center all coincide.

Three cases are investigated for BAR element comparison. In NASTRAN model, the PBARL entry is used to define properties of beam element (CBAR entry) by cross-sectional dimensions for first case (“ROD”, “TUBE”, “TUBE2”, “I”, “CHAN”, “T”, “BOX”, “BAR”, “CROSS”, “H”, “T1”, “I1”, “CHAN1”, “Z”, “CHAN2”, “T2”, “BOX1”, “HEXA”, “HAT”, “HAT1”, “DBOX”). Both the 2nd and the 3rd cases use PBAR entry to define properties of beam element. The difference is the PBARu¹ case uses default value (infinite) of area factor for shear (K1, K2), while the PBARn² case specifies the value of area factor for shear.

The LS-DYNA beam element definition is different; the beam type 2 is used to define properties of beam element by cross-sectional dimensions, which is used to compare with PBARL case. And beam type 13 is used to compare with PBARn and PBARu cases. Please note that LS-DYNA has only one parameter (SA) for area factor of shear, $SA = K1 \text{ or } K2 * \text{Area}$. For PBARu case (NASTRAN default is infinite), it can set a large number to match with NASTRAN.

All types of cross-section and corresponding beam properties for PBARn and PBARu are compared between LS-DYNA and NASTRAN. Only part of the results is shown in this paper due to paper size limitation. One of the channel shape comparison is shown below. Figures 2a and 2b are geometry definition in two codes.

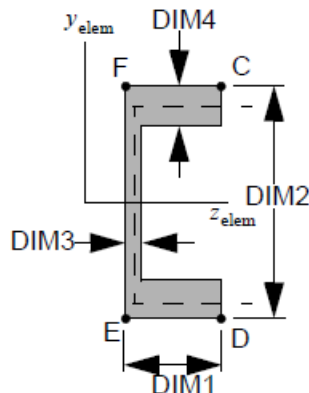


Figure 2a NASTRAN CHAN

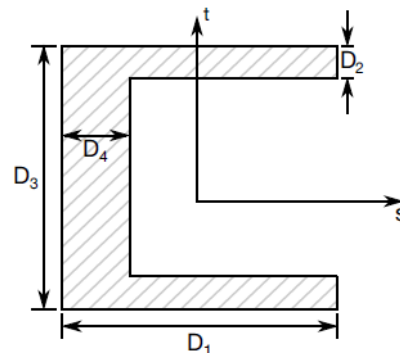


Figure 2b LS-DYNA Channel (Section_02)

1. PBARu, PBEAMu means using the values from user direct input to PBAR and PBEAM cards.
 2. PBARn, PBEAMn means using the calculated values of PBARL and PBEAML from NASTRAN solver into PBAR and PBEAM cards.

Table 1 shows the results comparison of PBARL, PBARn and PBARu cases. We need to point that the geometry definition in the two codes is different. In this example, the input parameters are D1= DIM1, D2= DIM4, D3= DIM2, D4= DIM3. The ISS and ITT value (area moments of inertia) in LS-DYNA should be reversed from NASTRAN I1, I2 value (area moments of inertia) due to the different local coordinate definition. Table 2 is a summary of the section type of corresponding beam (CBAR) element.

Table 1 Channel shape comparison of CBAR element

	PBARL			PBARn			PBARu		
	NASTRAN	LS-DYNA	Diff	NASTRAN	LS-DYNA	Diff	NASTRAN	LS-DYNA	Diff
Dz (mm)	-280.45	-281.29	-0.30%	-280.45	-280.47	-0.01%	-280.39	-280.39	0.00%
Mode # (Hz)									
1	9.20	9.19	0.16%	9.20	9.20	0.01%	9.20	9.20	0.00%
2	23.16	23.14	0.07%	23.16	23.16	0.00%	23.19	23.19	0.00%
3	57.61	57.56	0.10%	57.61	57.58	0.06%	57.67	57.66	0.02%
4	144.03	144.71	0.47%	144.03	144.01	0.01%	145.32	145.31	0.01%
5	161.09	161.11	0.01%	161.09	160.89	0.13%	161.48	161.43	0.03%
6	315.04	315.57	0.17%	315.04	314.35	0.22%	316.43	316.31	0.04%
7	398.44	403.81	1.35%	398.44	398.35	0.02%	406.91	406.82	0.02%
8	519.44	521.39	0.38%	519.44	517.69	0.34%	523.09	522.83	0.05%
9	767.50	778.39	1.42%	767.50	767.23	0.03%	781.40	780.93	0.06%
10	773.49	787.39	1.80%	773.49	769.78	0.48%	797.38	797.11	0.03%

Table 2 Corresponding shape for the beam (CBAR) element

NASTRAN	LS-DYNA	NASTRAN	LS-DYNA
01-ROD	08 Circular	12-I1	15 I-Shape3 01 I-Shape (symmetric)
02-TUBE	09 Tubular	13-CHAN1	16 Channel2
03-TUBE2	09 Tubular	14-Z	06 Z-Shape
04-I	10 I-Shape2 01 I-Shape (symmetric)	15-CHAN2	17 Channel3
05-CHAN	02 Channel	16-T2	18 T-Shape3
06-T	04 T-Shape	17-BOX1	19 Box-Shape 2
07-BOX	05 Box-Shape	18-HEXA	20 Hexagon
08-BAR	11 Solid Box	19-HAT	21 Hat Shape
09-CROSS	12 Cross	20-HAT1	22 Hat Shape 2(limited)
10-H	13 H-shape	21-DBOX	N/A
11-T1	14 T-Shape2	N/A	07 Trapezoidal-Shape

It can see that there is a good match between LS-DYNA and NASTRAN. PBARu case results are best matched as the input parameters are almost same. If the K1 is not equal to K2, it may affect results at some eigen frequencies for PBARn case. There is some difference in PBARL case due to the fact that input parameters are different in some shapes.

CBEAM and CBEAM3 elements

The beam element is defined with a CBEAM entry and its properties are defined with a PBEAM, PBCOMP, or PBEAML entry. The beam element includes extension, torsion, bending in two perpendicular planes, and the associated shears. Similar to bar element, the three cases (PBEAM L, PBEAMn², PBEAMu¹) are investigated using same simple cantilever beam model (figure 1).

Regarding the NASTRAN manual, one needs to use the CBEAM element instead of the CBAR element if any of the following features is important: the neutral axis and shear center do not coincide, the effect of cross-sectional warping on torsional stiffness is critical and the difference in the mass center of gravity and the shear center is significant.

The K1 and K2 default value are 1.0 for CBEAM element, so LS-DYNA uses 1.0*Area for SA in PBEAMu case. The same beam shape results are shown in table3 (PBEAML, PBEAMn and PBEAMu). Table 4 is a summary of section types of the corresponding beam (CBEAM) element.

Table 3 Channel shape comparison of CBEAM element

	PBEAML			PBEAMn			PBEAMu		
	NASTRAN	LS-DYNA	Diff	NASTRAN	LS-DYNA	Diff	NASTRAN	LS-DYNA	Diff
Dz (mm)	-280.45	-281.29	-0.30%	-280.44	-280.47	-0.01%	-280.41	-280.42	0.00%
Mode # (Hz)									
1	9.20	9.19	0.12%	9.20	9.20	0.01%	9.20	9.20	0.02%
2	22.65	23.10	1.98%	22.65	23.20	2.42%	23.18	23.20	0.09%
3	57.61	57.60	0.02%	57.61	57.60	0.02%	57.65	57.60	0.08%
4	88.15			88.15	90.40	2.56%	90.41	90.40	0.01%
5	136.96	145.00	5.87%	136.96	144.00	5.14%	144.93	145.00	0.05%
6	161.09	161.00	0.06%	161.09	161.00	0.06%	161.31	161.00	0.19%
7	253.31			253.31	271.00	6.99%	271.25	271.00	0.09%
8	315.04	316.00	0.30%	315.05	314.00	0.33%	315.84	316.00	0.05%
9	348.11			348.11	398.00	14.33%	404.31	404.00	0.08%
10	425.95	404.00	5.15%	425.95	452.00	6.12%	452.16	452.00	0.04%

Table 4 Corresponding shape for the beam (CBEAM) element

NASTRAN	LS-DYNA	NASTRAN	LS-DYNA
01-ROD	08 Circular	12-I1	15 I-Shape3 01 I-Shape (symmetric)
02-TUBE	09 Tubular	13-CHAN1	16 Channel2
03-TUBE2	09 Tubular	14-Z	06 Z-Shape
04-I	10 I-Shape2 01 I-Shape (symmetric)	15-CHAN2	17 Channel3
05-CHAN	02 Channel	16-T2	18 T-Shape3
06-T	04 T-Shape	17-BOX1	19 Box-Shape 2
07-BOX	05 Box-Shape	18-HEXA	20 Hexagon
08-BAR	11 Solid Box	19-HAT	21 Hat Shape
09-CROSS	12 Cross	20-HAT1	22 Hat Shape 2(limited)
10-H	13 H-shape	21-DBOX	N/A
11-T1	14 T-Shape2	N/A	07 Trapezoidal-Shape
		22-L	03 L-shape

It can see that there is a good match between LS-DYNA and NASTRAN. PBEAMu case results are best matched as the input parameters are almost same. If the K1 is not equal to K2, it may affect results at some eigen frequencies for PBEAMn case. There is some difference in PBEAML case due to the fact that input parameters are different in some shapes. LS-DYNA torsional modes results are not good, especially for PBEAML and PBEAMn cases. When we started to do validations, LS-DYNA did not get torsional modes; then have been just implemented, and we still needs to improve the accuracy of the torsional modes calculation.

The CBEAM3 is a general three-node beam element that has been implemented as a curved one-dimensional Timoshenko beam element so that both the initial curvatures of beam reference axis and the cross-section shears are included in the formulation of the linear strain-displacement relations. LS-DYTNA does not have general three-node beam element at this time.

CBEND and ROD (CROD, CONROD, CTUBE) elements

The bend element is defined with a CBEND entry and its properties are defined with a PBEND entry. The BEND element is a one-dimensional bending element with a constant radius of curvature. The CBEND element may be used to analyze either curved beams or pipe elbows. The bend element includes extension, torsion, bending in two perpendicular planes, and the associated transverse shear. LS-DYNA beam element 14 is elbow (straight or curved pipe) integrated tubular beam element. The input parameters of bend element are different in two codes, it needs figure out the corresponding parameters in future for further validation.

The rod element is defined with a CROD entry and its properties with a PROD entry. The rod element includes extensional and torsional properties. The CONROD entry is an alternate form that includes both the connection and property information on a single entry. The tube element is a specialized form that is assumed to have a circular cross section. The tube element is defined with a CTUBE entry, and its properties with a PTUBE entry. LS-DYNA beam element 3 (truss) can be used to compare with NASTRAN ROD element.

Table 5 shows comparison of ROD (CROD, CONROD, and CTUBE) element between NASTRAN and LS-DYNA. One can see there is a good match between the two codes.

Table 5 ROD element comparison

	CROD			CONROD			CTUBE		
	NASTRAN	LS-DYNA	Diff	NASTRAN	LS-DYNA	Diff	NASTRAN	LS-DYNA	Diff
Dx (mm)	1.52E-03	1.52E-03	0.01%	1.52E-03	1.52E-03	0.01%	1.52E-03	1.52E-03	0.01%
Mode # (Hz)									
1	1293.05	1290.91	0.17%	1293.05	1290.91	0.17%	1293.05	1290.91	0.17%
2	3879.15	3873.05	0.16%	3879.15	3873.05	0.16%	3879.15	3873.05	0.16%
3	6465.24	6456.14	0.14%	6465.24	6456.14	0.14%	6465.24	6456.14	0.14%
4	9051.34	9040.81	0.12%	9051.34	9040.81	0.12%	9051.34	9040.81	0.12%
5	11637.43	11627.70	0.08%	11637.43	11627.70	0.08%	11637.43	11627.70	0.08%
6	14223.51	14217.44	0.04%	14223.51	14217.44	0.04%	14223.51	14217.44	0.04%
7	16809.57	16810.66	0.01%	16809.57	16810.66	0.01%	16809.57	16810.66	0.01%
8	19395.60	19408.00	0.06%	19395.60	19408.00	0.06%	19395.60	19408.00	0.06%
9	21981.59	22010.10	0.13%	21981.59	22010.10	0.13%	21981.59	22010.10	0.13%
10	24567.51	24617.59	0.20%	24567.51	24617.59	0.20%	24567.51	24617.59	0.20%

For the ROD elements, there is good match between LS-DYNA and NASTRAN.

Surface elements

Four types of surface elements in NASTRAN are investigated and compared with LS-DYNA. CTRIA is a triangular element with optional coupling of bending and membrane stiffness. CTRIA6 is a triangular element with optional coupling of bending and membrane stiffness and optional mid-side nodes. CQUAD4 is a quadrilateral element with optional coupling of bending and membrane stiffness. CQUAD8 is a quadrilateral element with optional coupling of bending and membrane stiffness and optional mid-side nodes. LS-DYNA linear elements of shell are element 18, 20 and 21. Shell type 24 is 6-node quadratic triangular shell and shell type 23 is 8-node quadratic quadrilateral shell. These shell elements can be used to compare with NASTRAN shell elements. Table 6 shows the corresponding for the shell element between NASTRAN and NASTRAN.

Table 6 Correspondence for the shell element

NASTRAN	LS-DYNA
CTRIA3	Shell 18,20 and 21
CTRIA6	Shell 24
CQUAD4	Shell 18,20 and 21
CQUAD8	Shell 23

A simple plate model is used to validate the response from two simple analyses, static analysis and normal mode analysis. The validate response data is from handbook of MSC/NASTRAN and/or Altair/OptiStruct. The figure 3 shows the model, and the parameters of the model are, elastic modulus $E = 210000.0$ (MPa), Poison Ratio $\nu = 0.3$, and density $\rho = 7.91E-09$ (Mg/mm³). The force $P=40N$ is applied on the edge at the location A. Boundary are fixed in all directions at location C. Figure 4 is finite element model shell elements.

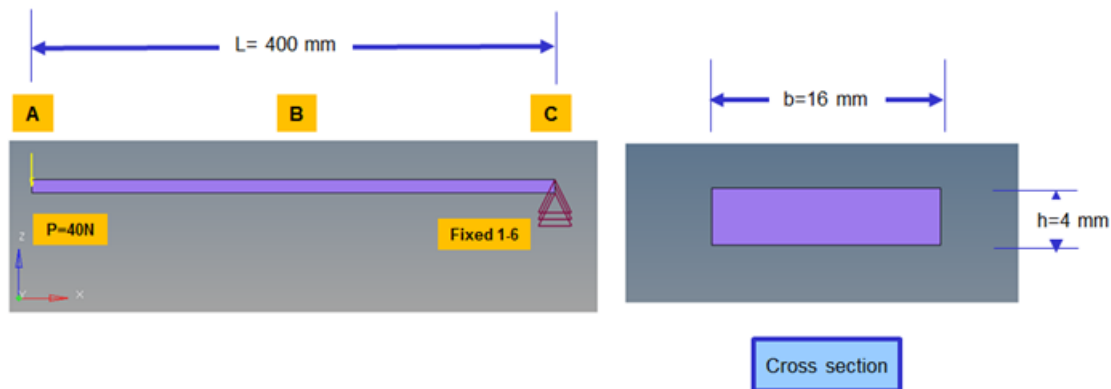


Figure 3 Simple plate models

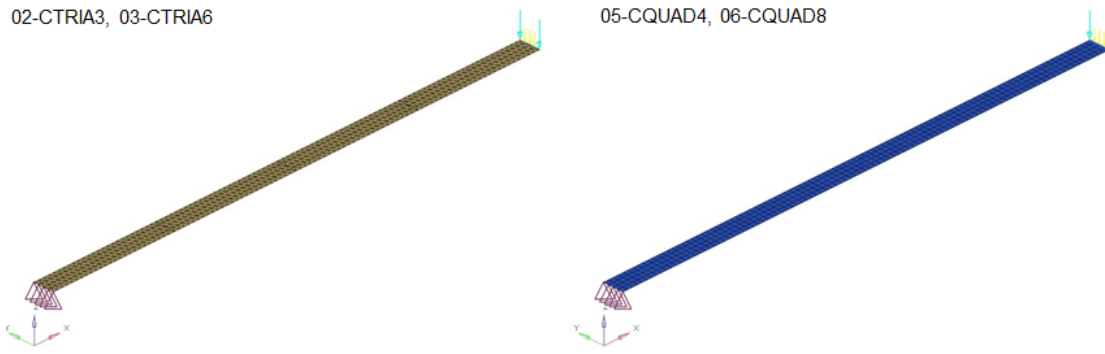


Figure 4 Simple plate FEM models of shell elements

Table 7 is static and modal analysis comparison of triangular elements (CTRIA 3 and 6) between LS-DYNA and NASTRAN. For CTRIA 3 element comparison, only the shell type 20 results of LS-DYNA are shown in this paper.

Table 7 Comparison of triangular elements

	CTRIA3			CTRIA6		
	NASTRAN	LS-DYNA 20	Diff	NASTRAN	LS-DYNA 24	Diff
Dz (mm)	-47.4624	-47.4617	-0.001%	-47.4849	-47.4777	-0.015%
Mode # (Hz)						
1	20.8669	20.7631	0.497%	20.8604	20.8623	0.009%
2	88.3231	88.4559	0.150%	83.2266	83.2339	0.009%
3	130.7269	130.0598	0.510%	130.6804	130.6977	0.013%
4	365.9040	363.9614	0.531%	365.7288	365.8046	0.021%
5	549.3415	550.1863	0.154%	517.7407	517.7508	0.002%
6	716.6975	712.6790	0.561%	716.2107	716.4403	0.032%
7	945.4971	911.7630	3.568%	928.0434	918.6217	1.015%
8	1184.1670	1177.0640	0.600%	1182.9780	1183.5570	0.049%
9	1520.0080	1522.4410	0.160%	1433.0080	1432.9120	0.007%
10	1767.8440	1756.4100	0.647%	1765.4180	1766.6230	0.068%

Table 8 Comparison of quadrilateral elements

	CQUAD4			CQUAD8		
	NASTRAN	LS-DYNA 20	Diff	NASTRAN	LS-DYNA 23	Diff
Dz (mm)	-47.4769	-47.4736	-0.007%	-47.4835	-47.4771	-0.013%
Mode # (Hz)						
1	20.8626	20.8632	0.003%	20.8608	20.8623	0.007%
2	83.2402	83.5015	0.314%	83.2280	83.2343	0.008%
3	130.7130	130.7083	0.004%	130.6838	130.6923	0.007%
4	365.9359	365.8822	0.015%	365.7407	365.7740	0.009%
5	518.0620	519.6969	0.316%	517.7523	517.7132	0.008%
6	716.9801	716.7585	0.031%	716.2390	716.3465	0.015%
7	937.0877	912.2967	2.646%	924.0383	907.2819	1.813%
8	1185.0670	1184.4520	0.052%	1183.0350	1183.3200	0.024%
9	1435.0200	1439.5960	0.319%	1433.0410	1432.6690	0.026%
10	1770.0790	1768.7090	0.077%	1765.5210	1766.1620	0.036%

Table 8 is static and modal analysis comparison of quadrilateral elements (CQUAD 4 and 8) between LS-DYNA and NASTRAN. The shell type 20 results of LS-DYNA are shown in this paper for CQUAD 4 element comparison.

For the shell elements, there is good match between LS-DYNA and NASTRAN.

Solid elements

Nastran includes three different solid polyhedron elements which are defined on the following bulk data entries: CTETRA -- Four-sided solid element with 4 to 10 grid points; CPENTA -- Five-sided solid element with 6 to 15 grid points; and CHEXA -- Six-sided solid element with 8 to 20 grid points. LS-DYNA uses *SECTION_SOLID to define solid elements. Solid type 10, 16 and 17 are tetrahedron elements. Solid type 15 is pentahedron element. Solid type 18 and 23 are hexagon elements. These solid elements can be used to compare with NASTRAN shell elements. Table 9 is one to one correspondence for the shell element between NASTRAN and NASTRAN.

A simple plate model (figure 3) is used to validate the response from two simple analyses, static analysis and normal mode analysis. The parameters of the plate model are same as the shell model. The finite element models are shown in figure 5.

Table 9 Correspondence for the solid element

NASTRAN	LS-DYNA
CTETRA4	Solid 10
CTETRA10	Solid 16 and 17
CPENTA6	Solid 15
CPENTA15	N/A
CHEXA8	Solid 18
CHEXA20	N/A



Figure 5 Simple plate FEM models of solid elements

Table 10 is static and modal analysis comparison of Four-sided solid elements (CTETRA 4 and 10) between LS-DYNA and NASTRAN. For CTETRA 10 element comparison, only the solid type 16 results of LS-DYNA are shown in this paper.

Table 10 Comparison of Four-sided solid elements

	CTETRA4			CTETRA10		
	NASTRAN	LS-DYNA 10	Diff	NASTRAN	LS-DYNA 16	Diff
Dz (mm)	-18.63804	-18.63800	0.000%	-48.17959	-48.17960	0.000%
Mode # (Hz)						
1	33.2550	33.5053	0.753%	20.6594	20.6791	0.096%
2	89.3804	90.0539	0.753%	82.0754	82.1535	0.095%
3	208.4084	209.9997	0.764%	129.4367	129.5605	0.096%
4	555.8498	560.0637	0.758%	362.2688	362.6156	0.096%
5	583.6141	588.0961	0.768%	510.9936	511.4776	0.095%
6	1139.2660	1148.0150	0.768%	709.4455	710.1253	0.096%
7	1537.8000	1549.4570	0.758%	902.8652	903.4024	0.059%
8	1572.4480	1579.5940	0.454%	1171.7590	1172.8830	0.096%
9	1891.6660	1906.3560	0.777%	1415.4800	1416.8100	0.094%
10	2814.6370	2837.2200	0.802%	1748.5480	1750.2260	0.096%

Table 11 compares the static and modal analysis results of 5-sided solid elements (CPENTA 6) between LS-DYNA and NASTRAN. For LS-DYNA, a 15-node solid element is missing and needs to be implemented.

Table 11 Comparison of Five-sided solid elements

	CPENTA6		
	NASTRAN	LS-DYNA 15	Diff
Dz (mm)	-47.40669	-46.9242	1.018%
Mode # (Hz)			
1	20.8828	21.1442	1.252%
2	89.0364	89.5456	0.572%
3	130.8285	132.4522	1.241%
4	366.2072	370.6620	1.216%
5	553.7140	556.9436	0.583%
6	717.3573	725.3578	1.115%
7	934.7210	1118.9560	19.710%
8	1185.3970	1201.3460	1.345%
9	1531.8210	1540.9160	0.594%
10	1769.9740	1791.8410	1.235%

Table 12 shows static and modal analysis results comparison of Six-sided solid elements (CHEXA 8) between LS-DYNA and NASTRAN.

Table 12 Comparison of Six-sided solid elements

	CHEXA8		
	NASTRAN	LS-DYNA 18	Diff
Dz (mm)	-47.41147	-47.40660	0.010%
Mode # (Hz)			
1	20.8814	20.8824	0.005%
2	83.3226	83.3214	0.001%
3	130.8375	130.8438	0.005%
4	366.3220	366.3418	0.005%
5	518.5689	518.5616	0.001%
6	717.8517	717.8992	0.007%
7	926.1385	926.2946	0.017%
8	1186.7640	1186.8630	0.008%
9	1436.4120	1436.3930	0.001%
10	1773.0960	1773.2830	0.011%

Summary

A summary for the validation on the elements is provided in the paper. For most cases, a good match between LS-DYNA and NASTRAN are reached. LS-DYNA needs further development the torsional modes calculation to compare NASTRAN CBEAM elements, area factor for shear (K1, K2), general three nodes beam element and high order solid elements.

This work allows users to get better understanding on the current status of LS-DYNA's linear solvers as well as provides suggestions of how to select the types of elements and parameters in LS-DYNA for the users who need to translate their finite element models between NASTRAN and LS-DYNA.

References

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