LS-DYNA®'s Linear Solver Development — Phase1: Element Validation Part II

Allen T. Li[1], Zhe Cui[2], Yun Huang[2]
[1]*Ford Motor Company*
[2]*Livermore 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 Part I).

In this paper, the R-type elements and bushing elements are investigated. The R-type elements include both rigid (RBE2, etc.) and interpolation elements (RBE3, etc.), which are very popularly used elements. The bushing (generalized spring and damper) elements consist of the CBUSH and CBUSH1D. Several benchmark examples are studied to perform cross-validation of the R-type and bushing elements in LS-DYNA and NASTRAN, in different types of analysis such as static, normal mode and SSD analysis.

The validation shows that there is a good match for the R-type elements and bushing elements in most cases, between LS-DYNA and NASTRAN. This paper can provide guidance for the users who need to translate their FEM models between NASTRAN and LS-DYNA.

R-type elements

The R-type elements include both rigid and interpolation elements, which consist of RBAR, RBE1, RBE2, RROD, RTRPLT, RBE3, and RSPLINE. In this paper, the RBE2 and RBE3, which are the most commonly used of R-type elements are investigated. The RBE2 and RBE3 elements are all mainly based on a linear displacement relationship, not an elastic relationship. They are not dictated by stiffness, mass or force. They all follow the small displacement theory. The RBE2 and RBE3 elements have different definitions for dependent and independent nodes or grid points. The stiffness, mass and loads at the dependent degree-of freedoms are transferred to the independent degree-of-freedoms.

The RBE2 is a rigid body connected to an arbitrary number of grid points. The independent degrees-of-freedom are the six components of motion at a single grid point. The dependent degrees-of-freedom at the other grid points all have the same user-selected component numbers. The RBE3 defines a constraint relation in which the motion at a "reference" grid point is the least square weighted average of the motions at other grid points. The element is useful for loads and masses from a "reference" grid point to a set of grid points.

In LS-DYNA, the corresponded keyword to RBE2 is ***CONSTRAINED_NODAL_RIGID_BODY,** which defines a nodal rigid body consisting of the defined nodes. The first card is shown in Figure 1; the 2nd card is used for SPC option; and the 3th-5th card are used for INERTIA option; the 6th card is required if a local coordinate system is used to specify the inertia tensor when the INERTIA option is set.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	CID	NSID	PNODE	IPRT	DRFLAG	RRFLAG	
Type	I	I	I	I	I	I	I	
Default	none	none	none	0	0	0	0	

Figure 1. The keyword format (1st card) of ***CONSTRAINED_NODAL_RIGID_BODY**

The ***CONSTRAINED_INTERPOLATION** is doing the same feature as RBE3 in LS-DYNA, which defines an interpolation constraint. With this constraint type, the motion of a single dependent node is interpolated from the motion of a set of independent nodes. This option is useful for the redistribution of a load applied to the dependent node by the surrounding independent nodes. This load may be a translational force or a rotational moment. This keyword is typically used to model shell-brick and beam-brick interfaces.

Figure 2 shows the input parameters of ***CONSTRAINED_ INTERPOLATION.** The card 2 is used to define independent node card sets.

Card 1	1	2	3	4	5	6	7	8
Variable	ICID	DNID	DDOF	CIDD	ITYP			
Type	I	I	I	I	I			
Default	0	0	123456	optional	0			

Card 2	1	2	3	4	5	6	7	8
Variable	INID	IDOF	TWGHTX	TWGHTY	TWGHTZ	RWGHTX	RWGHTY	RWGHTZ
Type	I	I	F	F	F	F	F	F
Default	0	123456	1.0	TWGHTX	TWGHTX	TWGHTX	TWGHTX	TWGHTX

Figure 2. The keyword format of ***CONSTRAINED_ INTERPOLATION**

A simple HAT section model as shown in Figure 3 is used to do the validation. There are totally 12,614 nodes and 12,046 shell elements and 30 solid elements in this model.

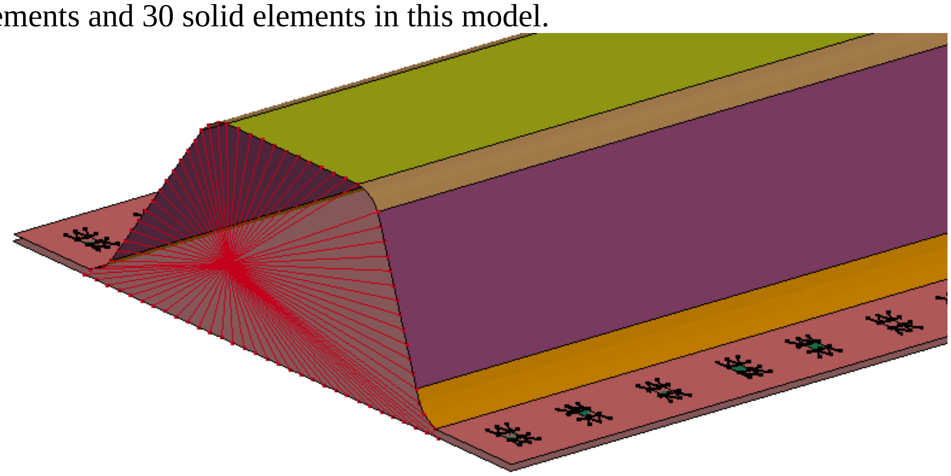

Figure 3. The HAT section model

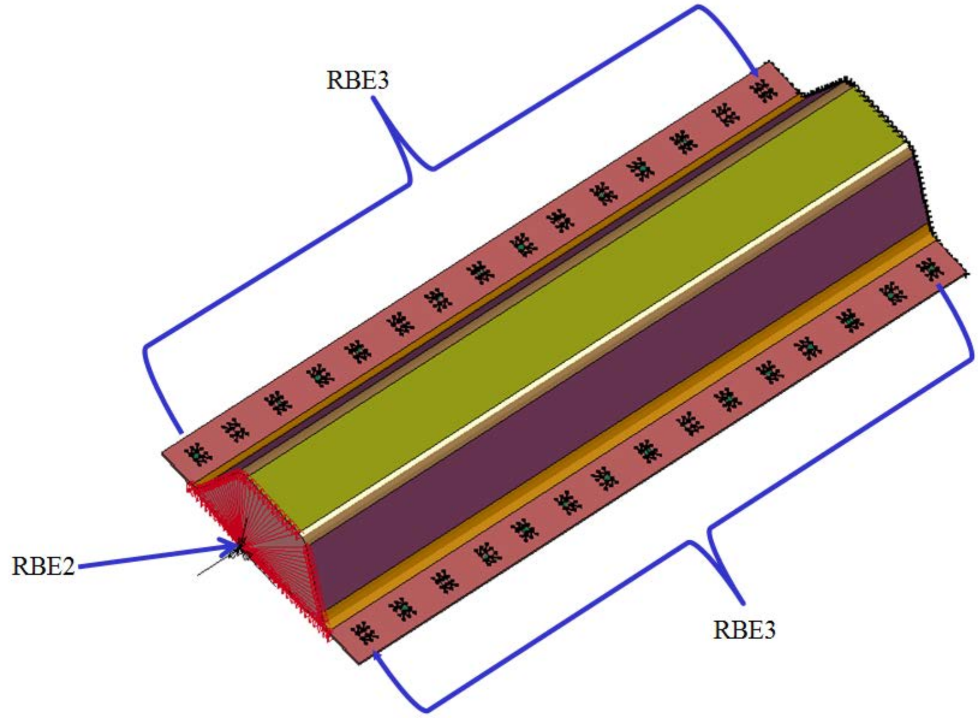

Figure 4. The HAT section model with both RBE2 and RBE3 elements

Figure 4 shows the HAT section model with both RBE2 and RBE3 elements. Four load cases are investigated. The first 3 cases are static analysis where a nodal force of 100 N is applied to the structure at X, Y and Z direction separately in each case. The displacement of node 101 is calculated by NASTRAN and LS-DYNA and the results are compared. The 4th case is about normal mode analysis. The first 10 modes are computed for comparison. One can see from Table 1 that the results by the two codes match very well.

Table 1. The comparison of hybrid RBE2 and RBE3 elements case

Load case ID	Displacement (mm)	NASTRAN	LS-DYNA	Diff
101	X	3.751E-06	3.750E-06	0.024%
102	Y	1.039E-04	1.038E-04	0.078%
103	Z	-4.798E-04	-4.795E-04	-0.049%
201	Mode # (Hz)			
	1	263.72	263.72	0.001%
	2	420.49	420.24	0.059%
	3	495.03	494.84	0.038%
	4	548.38	548.55	0.030%
	5	603.25	603.19	0.010%
	6	722.84	723.30	0.064%
	7	763.12	763.26	0.019%
	8	907.38	907.64	0.029%
	9	916.79	916.73	0.006%
	10	979.52	979.85	0.034%

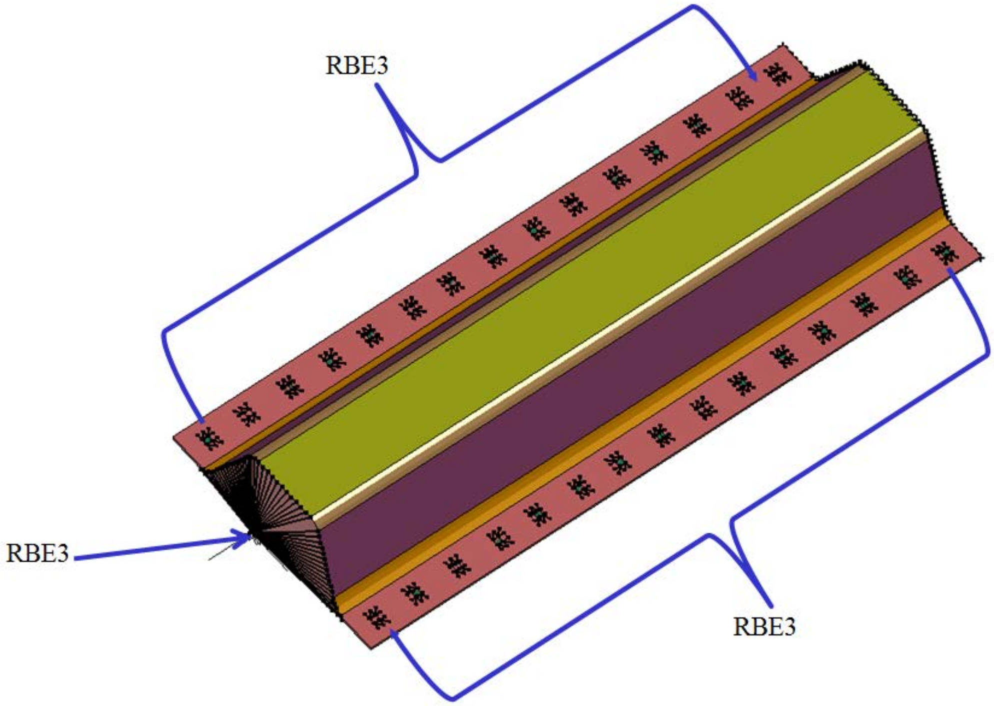

Figure 5. The HAT section model with RBE3 element

Figure 5 shows the HAT section model with RBE3 element. Again, there are 4 load cases investigated. The first 3 cases are static analysis where a nodal force of 100 N is applied to the structure at X, Y and Z direction separately in each case. The displacement of node 101 is calculated by NASTRAN and LS-DYNA and the results are compared. The 4th case is about normal mode analysis. The first 10 modes are computed for comparison. One can still see from Table 2 that the results by the two codes match very well.

Table 2. The comparison of RBE3 elements case

Load case ID	Displacement (mm)	NASTRAN	LS-DYNA	Diff
101	X	3.778E-06	3.777E-06	0.010%
102	Y	1.564E-04	1.564E-04	0.012%
103	Z	-8.420E-04	-8.425E-04	-0.059%
201	Mode # (Hz)			
	1	254.63	254.64	0.003%
	2	414.81	414.52	0.069%
	3	447.25	447.33	0.016%
	4	448.79	448.65	0.031%
	5	503.95	503.68	0.054%
	6	624.02	623.87	0.024%
	7	673.75	674.07	0.047%
	8	710.67	710.92	0.035%
	9	785.10	785.17	0.009%
	10	900.46	900.39	0.007%

Bushing elements in SSD analysis

The bushing (generalized spring and damper) elements consist of the following: CBUSH and CBUSH1D.

The CBUSH is a structural scalar element connecting two non-coincident grid points, or two coincident grid points, or one grid point with an associated PBUSH entry. This combination is valid for any structural solution sequence. In modal frequency response, the basis vectors (system modes) will be computed only once in the analysis and will be based on nominal values of the scalar frequency dependent springs. In general, any change in their stiffness due to frequency will have little impact on the overall contribution to the structural modes. The stiffness matrix for the CBUSH element takes the diagonal form in the element system.

The BUSH1D is a one dimensional version of the BUSH element, which is defined with the CBUSH1D and a PBUSH1D entry. The user may define several spring or damping values on the PBUSH1D property entry. It is assumed that springs and dampers work in parallel. The element force is the sum of all springs and dampers. The BUSH1D element has axial stiffness and axial damping. The element includes the effects of large deformation. The elastic forces and the damping forces follow the deformation of the element axis if there is no element coordinate system defined. The forces stay fixed in the x-direction of the element coordinate system if the user defines such a system. Arbitrary nonlinear force-displacement and force velocity functions are defined with tables and equations. A special input format is provided to model shock absorbers.

In LS-DYNA, there are three material models with damping, which can be applied in frequency domain SSD analysis. These damping materials are very useful to model viscous damping or local damping in frequency domain SSD analysis. The material type 66 **(*MAT_LINEAR_ELASTIC_DISCRETE_BEAM)** is defined for simulating the effects of a linear elastic beam by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length b beam, or offset to give a finite length beam. It can be used with discrete beam element type 6 in frequency domain SSD analysis. The material type 74 (***MAT_ELASTIC_SPRING_DISCRETE_BEAM**) permits elastic springs with damping to be combined and represented with a discrete beam element type 6. Linear stiffness and damping coefficients can be defined, and, for nonlinear behavior, a force versus deflection and force versus rate curves can be used. The material type S02 (***MAT_DAMPER_VISCOUS**) for discrete elements **(*ELEMENT_DISCRETE**) provides a linear translational or rotational damper located between two nodes. Only one degree of freedom is then connected.

The ***MAT_66** (***MAT_LINEAR_ELASTIC_DISCRETE_BEAM**) corresponds to NASTRAN's CBUSH element; the ***MAT_74** (***MAT_ELASTIC_SPRING_DISCRETE_BEAM**) and ***MAT_S02** (***MAT_DAMPER_VISCOUS**) correspond to NASTRAN's CBUSH1D element, which can be used to model viscous local damping in frequency response analysis.

To active the material damping in frequency domain SSD analysis, two flags in the keyword input files are required. First, one needs to set "DMPFLG" equal to 1 in the keyword ***FREQUENCY_DOMAIN_SSD** (Figure 6) to use local damping. Second, the "EVDUMP" should be smaller than 0 (e.g. -1) in the keyword ***CONTROL_IMPLICIT_EIGENVALUE** (Figure 7) to request output of eigenvalues and eigenvectors using a binary format.

Card 2	1	2	3	4	5	6	7	8
Variable	DAMP	LCDAM	LCTYP	DMPMAS	DMPSTF	DMPFLG		
Type	F	I	I	F	F	I		
Default	0.0	0	0	0.0	0.0	0		

Figure 6. Card 2 of ***FREQUENCY_DOMAIN_SSD**

Card 2	1	2	3	4	5	6	7	8
Variable	ISOLID	IBEAM	ISHELL	ITSHELL	MSTRES	EVDUMP	MSTRSCL	
Type	I	I	I	I	I	I	F	
Default	0	0	0	0	0	0	0.001	

Figure 7. Card 2 of ***CONTROL_IMPLICIT_EIGENVALUE**

A simple beam model with attached damper at one end is used for the cross-validation between LS-DYNA and NASTRAN. Figure 8 shows the finite element model of the beam with the damper. The frequency response analysis, which is the SSD in LS-DYNA and SOL 111 in NASTRAN are preformed on this model. Three cases with different damper models are investigated.

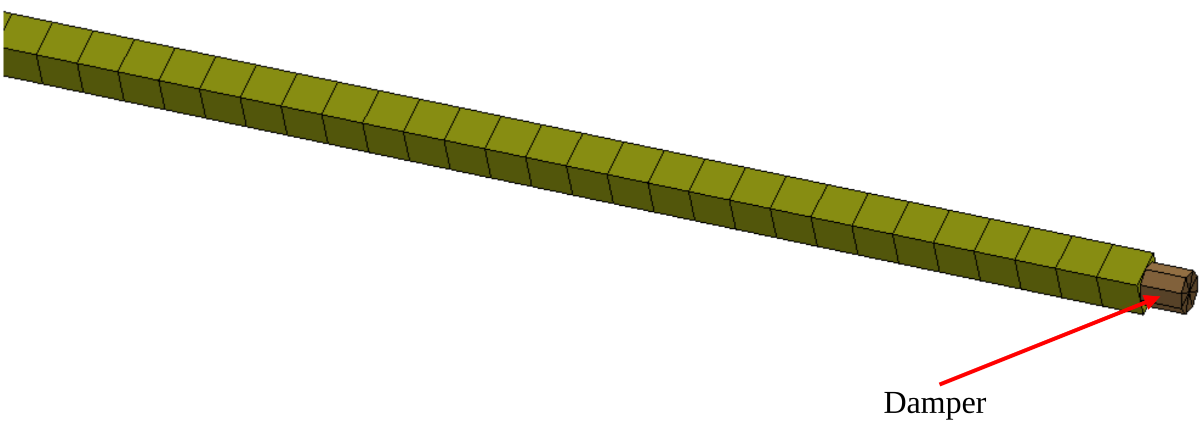

Figure 8. A simple beam model with an attached damper

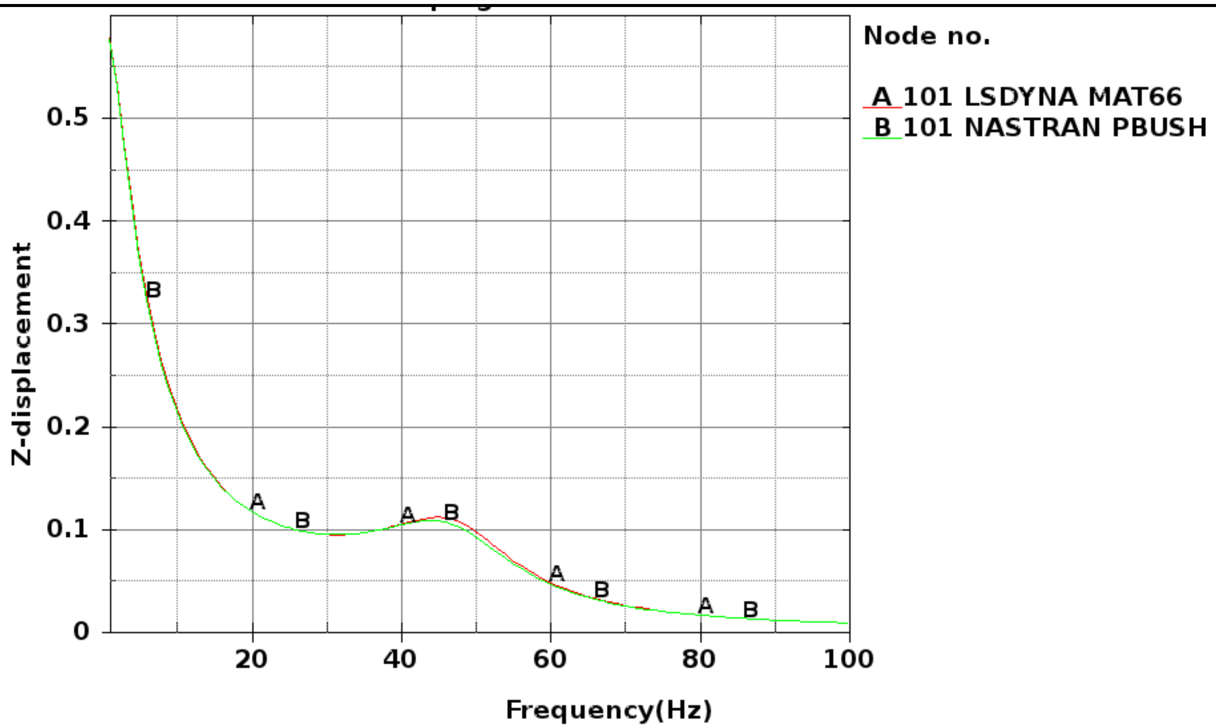

Figure 9. Z displacement comparison

Figure 9 shows the Z displacement results at node 101. MAT66 is used in LS-DYNA and PBUSH is used in NASTRAN. One can see that there is a very good match between LS-DYNA and NASTRAN. Figure 10 shows the Z displacement results at node 101, for which MAT74 and MATS02 are used in LS-DYNA and PBUSH1D is used in NASTRAN. One can see that the results by LS-DYNA and NASTRAN match very well.

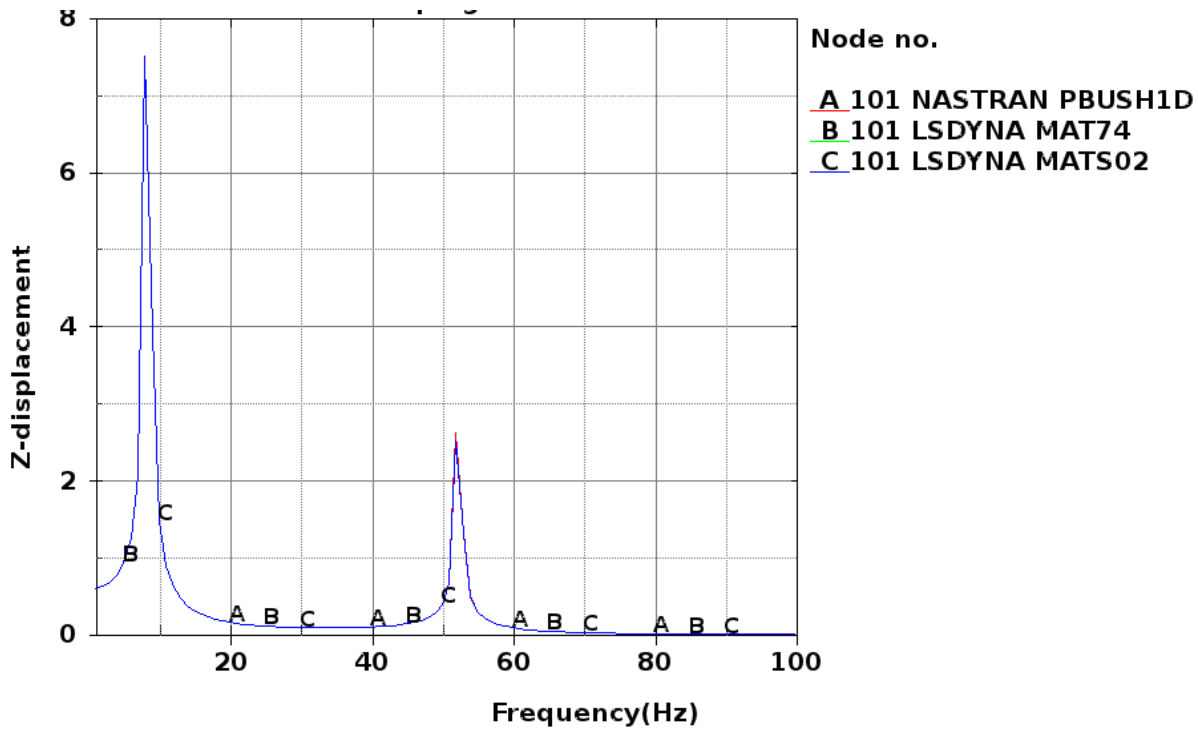

Figure 10. Z displacement comparison

Summary

In this paper, two commonly used R-type elements: RBE2 and RBE3, and bushing elements in NASTRAN are investigated. For RBE2, its corresponding keyword in LS-DYNA is ***CONSTRAINED_NODAL_RIGID_BODY**. For RBE3, its corresponding keyword in LS-DYNA is ***CONSTRAINED_INTERPOLATION**. For the bushing elements, one can also find corresponding material models in LS-DYNA. For CBUSH element, the corresponding material model in LS-DYNA is ***MAT_66** (***MAT_LINEAR_ELASTIC_DISCRETE_BEAM**). For CBUSH1D element, the corresponding material models in LS-DYNA are ***MAT_74** (***MAT_ELASTIC_SPRING_DISCRETE_BEAM**) and ***MAT_S02** (***MAT_DAMPER_VISCOUS**).

As demonstrated by the examples, a good match between the results by LS-DYNA and NASTRAN for the R-type elements and bushing elements can be reached.

Cross-validations of more element types will be performed in the future.

References

[1] LS-DYNA Keyword User's Manual, Livermore Software Technology Corporation, 2017.