Recent Advances on Higher Order 27-node Hexahedral Element in LS-DYNA®

Hailong Teng
Livermore Software Technology Corp.

Abstract

This paper presents recent advances of second-order hexahedral element developed for explicit/implicit analysis in LS-DYNA. Several benchmark problems are studied to demonstrate the performance of the higher order element. The results obtained in modeling practical applications involving large deformations, nearly incompressible materials, severe distortions, bending, and contact-impact are also encouraging. Compared to standard 8-node brick element, the high order element is computationally expensive, but it is found to be competitive with other element types due to its much higher accuracy and higher convergence rate. Furthermore, high order element naturally contains the linear strain field and is capable of modeling bending and curved shape accurately without using either hourglass control or introducing incompatible modes. From a user viewpoint, what is gained is versatility in modeling a wide variety of geometries including three-dimensional or plate/shell geometries, and simplicity since only displacement degrees of freedom are used. This paper also present two techniques that transfer 8-node hexahedral model to higher order hexahedral model.

Introduction

In many applications, one needs to analyze a body comprising of one or multiple shells connected to a solid body. As the shell elements cannot transfer rotational reaction forces to the brick elements, special techniques are required to match the rotational degrees of freedom of the shell elements with the translational degrees of freedom of the solid elements. Ideally, in such a case, it is convenient to model the entire structure using elements that involve only displacement degrees of freedom as in a standard displacement-based brick element. However, modeling shell parts with standard solid elements would require a huge number of elements (3~5 elements through thickness) to prevent locking and leads to prohibitive computational costs. Furthermore, modeling thin structures with standard solid elements often leads to elements with high aspect ratios, which degrades the accuracy of the solution.

Based on the above considerations, what one would like to have is an element which work well not only when used to model three-dimensional geometries, but also relatively immune from locking when used to model plate/shell geometries. This is the motivation for the development of the 27-node brick elements in this work. Second-order 27-node elements can naturally represent curved shapes and model bending accurately for thick to moderately thin plate/shells without using an artificial hourglass control or by adding any incompatible modes [4]. While performing well as flexural elements, they still maintain their versatility as solid elements. Compared to 8-node brick elements, 27-node element are expensive, buy they provide an important modeling alternative in LS-DYNA and they do have distinct advantages for certain applications.

Definition of 27-node element

The element formulation for 27-node element in LS-DYNA is ELFORM=24. Each element contain 27 nodes: 8 corner, 12 edge-center, 6 face-center and 1 body-center node. The node numbering is shown in Figure 1.

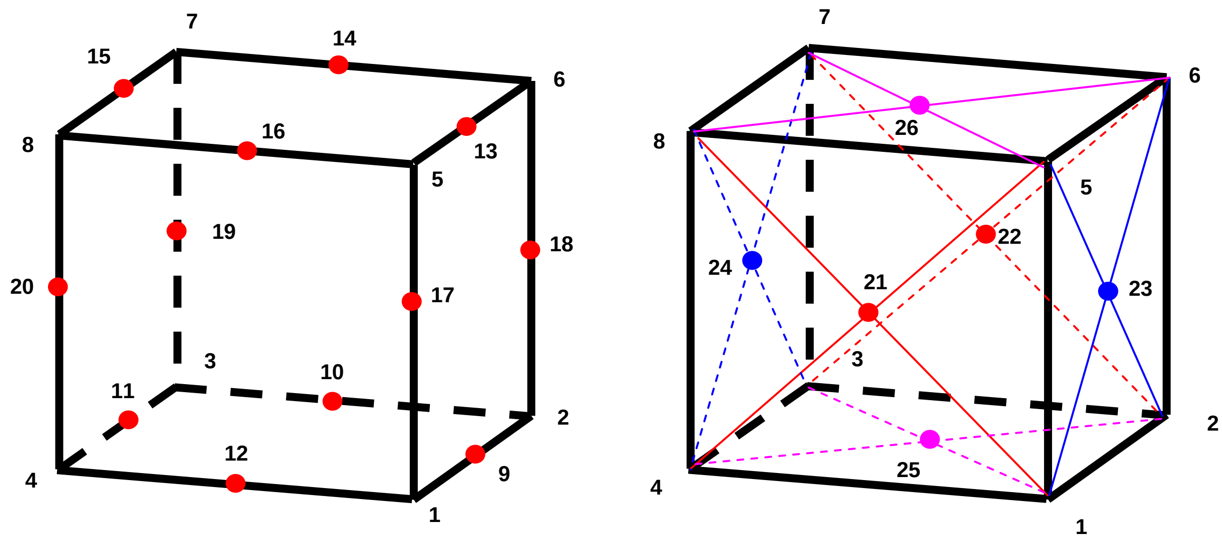

Figure 1 Node numbering

For the 27-node element considered herein, a row summation scheme is evaluated for mass lumping, this produce a non-uniform mass lumping. Relative nodal values for a unit cube is given in Table 1. The lumping is highly non-uniform, as the body-center node has nearly 1/3 of the total mass. Face-center node and edge-center node each have 16 and 4 times the mass of corner node.

Node Type	Relative mass
Corner	1/216
Edge-center	1/54
Face-center	2/27
Body-center	8/27

Table 1 Relative nodal mass for a unit cube

In LS-DYNA, there are two ways to define element connectivity. The first one is user directly define element connectivity by *ELEMENT_SOLID_H27, the nodal numbering follow the rule defined in Figure 1, the following shows an example

```
*ELEMENT_SOLID_H27
      1       1       0
      1       2       3       4       5       6       7       7       9      10
     11      12      13      14      15      16      17      18      19      20
     21      22      23      24      25      26      27
```

LS-PrePost® also provides the function that transfers a 8-node brick element to a 27-node brick element and generate corresponding node coordinate and element connectivity.

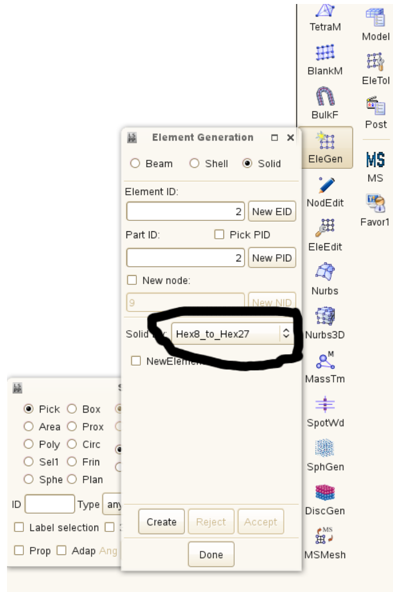

Figure 2 using LS-PrePost to transfer 8-node brick element to 27-node brick element

This procedure might be tedious to transfer many current models to 27-node element model. LS-DYNA also provides another option *ELEMENT_SOLID_H8TOH27 to automatically transfer a current model to 27-nodel brick model:

```
*ELEMENT_SOLID_H8TOH27
     1     1     1     2     3     4     5     6     7     8
```

However, one need to be careful about the boundary conditions for the newly added nodes if *ELEMENT_SOLID_H8TOH27 is utilized. If BC's of existing node are defined with *NODE, for example, as shown in the following

```
*NODE
$#   nid          x           y           z         tc    rc
     2        3.000      -2.000    -1.0000000       7     3
```

Then appropriate boundary condition are applied to the newly added node based on the BC's of its neighbor node.

If BC's are defined to a node set, *BOUNDARY_PRESCRIBED_MOTION_ SET for example, and all the neighbor node of a newly added node belong to this node set, then the BC's are also applied to the newly added node.

If BC's are defined directly to existing node, *BOUNDARY_PRESCRIBED_MOTION_NODE, for example, then LS_DYNA does not know what kind of BC's should be applied to the newly added node, as such, free BC's are applied to the newly added node.

*ELEMENT_SOLID_H8TOH27 option not only transfers a 8-node brick element to a 27-node second order element, this option also read the information of 4-node tetrahedron element , 6-

node pentahedron element or even 5-node pyramid element and generating corresponding of 15-node tetrahedron element, 21-node pentahedron element and 19-node pyramid element. At current stage, these element are treated as degenerated 27-node element.

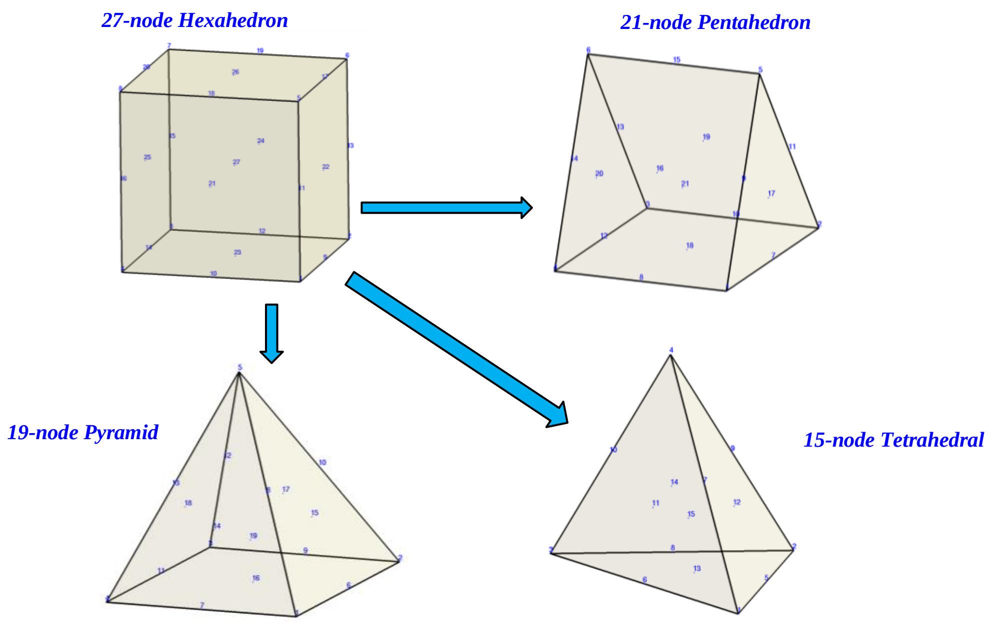

Figure 3 27-node Hexahedron element and degenerated 21-node pentahedron, 19-node pyramid, and 15-node tetrahedron element

Numerical Tests

The first numerical example is 6 element straight cantilever beam. This test was first proposed in [1] and is useful to test the accuracy of the element formulation in bending dominant regime as well as for distorted element configuration.

The cantilever of length l = 6.00, height h = 0.20 and depth d = 0.10 is discretized with six elements. Three mesh types with regular, trapezoidal and parallelepiped shapes are used (see Figure 4). The cantilever has a tip unit load (0.25 for each tip node) and the numerical results for extension, in plane and out of plane shear loading at the tip are evaluated and normalized with the analytical values taken from the Bernoulli beam theory. An isotropic elastic material with Poisson's ratio ν=0.3 and Young's modulus E=1.0E+7 is used. Theoretical values for the deflection are u_{ext} = 3.0E-5 in extension, u_{ip} = 0.1081 for in plane shear load and u_{op} = 0.4321 for out of plane shear load. Results are given in Table 2.

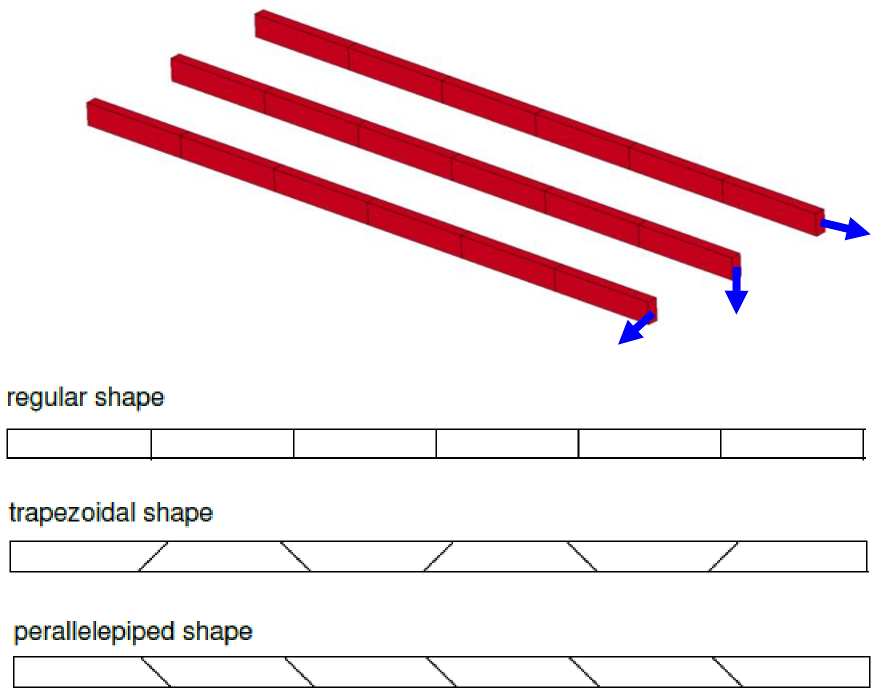

Figure 4 Straight cantilever beam test. Loading: unit forces at free end

Type	Ext	In-Plane	Out of Plane
Regular	1.0	0.9899	0.9766
Trapezoidal	1.0	0.9852	0.9674
parallelogram	1.0	0.9852	0.9674

Table 2 Straight cantilever beam test, normalized displacement

The second test is curved beam test. This example, proposed in [1] tests the accuracy of elements in bending dominant regime in presence of a curved initial geometry. Combination of the principle deformation modes are evoked by a single in-plane or out-of-plane shear load at tip. The curved beam as shown in Figure 5 is discretized with six elements. Note also that the element shape is not quite rectangular, which also test the effect of slightly irregularity. The beam has an inner radius ri = 4.12 and an outer radius ro = 4.32, an arc length of 90◦ and a thickness t = 0.10. One end is clamped and the other end is loaded with a unit force. An isotropic elastic material with a Young's modulus of E = 1.0 · 10+7 and a Poisson's ratio ν = 0.00 is assumed. Theoretical solution of tip displacement along force direction are in the case of in plane shear loading u_{ip} = 0.08734 and in the case of out of plane shear loading u_{op} = 0.50220. Results are given in Table 3. The approximations are with a few percent of the analytical solutions, even for this coarse mesh.

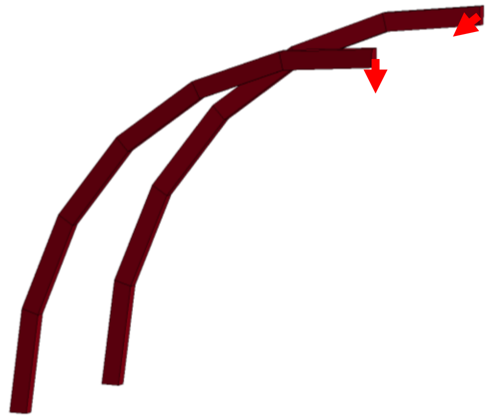

Figure 5 Curved beam test. Loading: unit forces at tip

	In-plane	Out-of-plane
Exact	0.08734	0.5022
Numerical	0.0898	0.466
Numerical (Normalized)	1.0213	0.928

Table 3 Curved beam test, numerical and normalized displacement

The twisted beam tests proposed in [1] tests the effect of warp on plate elements. The tip loaded 90◦ twisted cantilever beam, shown in Figure 6, is a three-dimension bending example. The twist between the two faces of each element along the length is 7.5◦. A vertical unit tip load is applied instantaneously and held constant in magnitude and orientation. Twelve elements are used along the 12.0-in length, two elements spans the 1.1-in width and one element spans the 0.32-in thickness. An isotropic elastic material with Poisson's ratio ν=0.22 and Young's modulus E=2.9E+7 is used. One end is clamped, with unit forces at tip: in-plane shear, and out-of-plane shear loading. Theoretical values for the deflection are u_{ip} = 5.424E-3 for in plane shear load and u_{op} = 1.754E-3 for out of plane shear load. Results are given in Table 4.

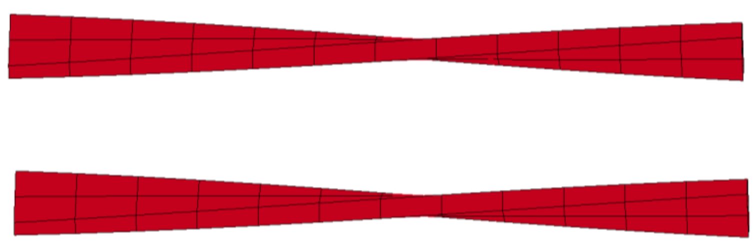

Figure 6 Twisted beam test

	In-plane	Out-of-plane
Exact	0.08734	0.5022
Numerical	0.0898	0.466
Numerical (Normalized)	1.0213	0.928

Table 4 Twisted beam test, numerical and normalized displacement

Tayler bar impact problem uses a quarter symmetric mesh of a cylindrical impact a rigid wall at 227m/s. The bar is 3.23E-2m long with a 3.2E-3m radius. The isotropic and kinematic hardening plasticity model use a modulus of elasticity of 1.17E11Pa, Poisson's ratio of 0.33, tangent plastic modulus of 1.0E8Pa, yield stress of 4.0E8 Pa and mass density of 8930kg/m^3.

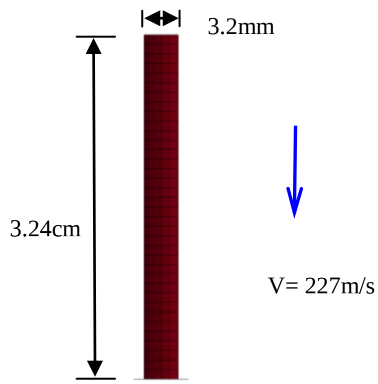

Figure 7 Taylor model description

The bar impact problem is modeled using 972 selective reduced 8-node element (Elform2), 972 27-node elements (Elform24), and 7776 selective reduced 8-node element (Elform2). Figure 8 compares the prediction of three simulations. The final predicted length of the bar for 27-node element agree with the selective reduced 8-node elements and compare reasonably with experimental predictions of Wilkins and Guinan [2].

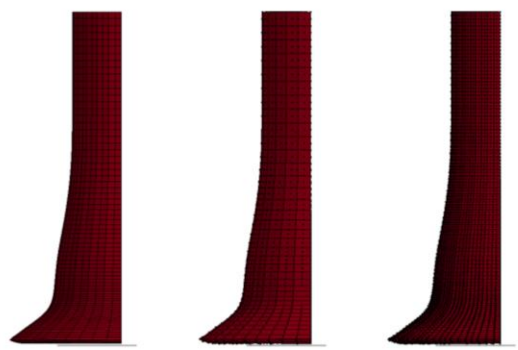

Figure 8 Deformed model for bar impact problem using a) 972 Elform2 element b) 972 Elform24 Element c) 7776 Elform2 element

In term of CPU time, 27-node element is much more expensive than 8-node element. Generally speaking, the CPU time of 27-node element is about 30 times of selective reduced 8-node element with same number of elements. However, we believe that instead of comparing the computational cost for a given number of degrees of freedom, the comparison ought to be between computational costs incurred to achieve a given level of accuracy. From this viewpoint, since relatively far lesser number of 27-node elements are required, they might not be so bad in term of CPU time. For the bar impact problem, the 27-node elform 24 with 972 elements uses about 35% more CPU time than 8-node elform 2 with 7776 element.

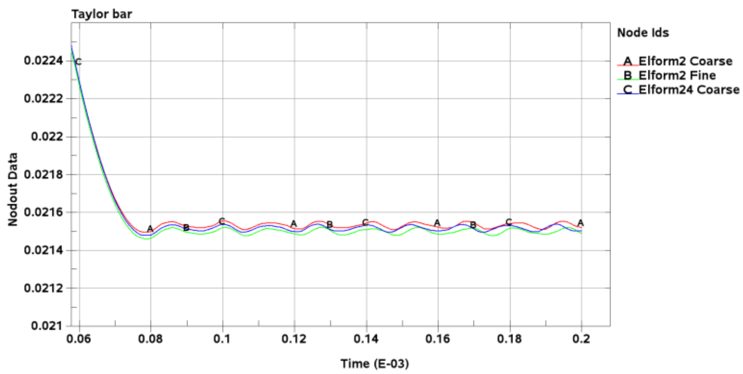

Figure 9 Displacement comparison between a) 972 Elform2 element b) 972 Elform24 Element c) 7776 Elform2 element

The following model is an implicit model and taken from [3], where a clamped plate of dimensions 10x5x1 mm is subjected to 1Nm torque at the free end. The Young's modulus is E=210 Gpa. Analytical solution for end tip deflection is 0.57143mm. The problem is modeled by 5 different discretization with fixed aspect ratio 5:1.

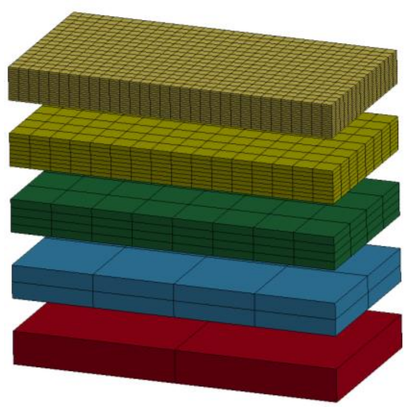

Figure 10 Clamped plate subjected torque

Results are given in Table 5. Also compare the results using other element formulation from [3].

Discretization	Elform2	Elform-2	Elform-1	Elform24
2x1x1	0.0564(90.1%)	0.6711(17.4%)	0.6751(18.1%)	0.5525(3.3%)
4x2x2	0.1699(70.3%)	0.5466(4.3%)	0.5522(3.4%)	0.5534((3.1%)
8x4x4	0.3469(39.3%)	0.5472(4.2%)	0.5500(3.8%)	0.5541(3.0%)
16x8x8	0.4820(15.7%)	0.5516(3.5%)	0.5527(3.3%)	0.5543(3.0%)
32x16x16	0.5340(6.6%)	0.5535(3.1%)	0.5540(3.1%)	0.5545(3.0%)

Table 5 End tip deflection for different mesh discretization and element types, error in parenthesis.

Finally, the axial crushing and crashing of thin-walled high-strength steel tubes is performed using 27-node second order element. The Crush Box is a thin-walled structure attached between the vehicle bumper structure and the side rail. The need of the Crush box is quite most important for absorbing the energy of impact. This kind of problem is usually modelled with shell element, in this study, the crush box is modeled with 27-node brick element. Only one layer or element is required over thickness (Figure 11). For convergence study, the model is discretized with 4mm, 2mm, and 1mm element (Figure 12). The piecewise linear isotropic plasticity model use a modulus of elasticity of 207Gpa, Poisson's ratio of 0.3, tangent modulus 2.0Gpa, yield stress of 200M Pa and mass density of 7830kg/m$^{3.}$

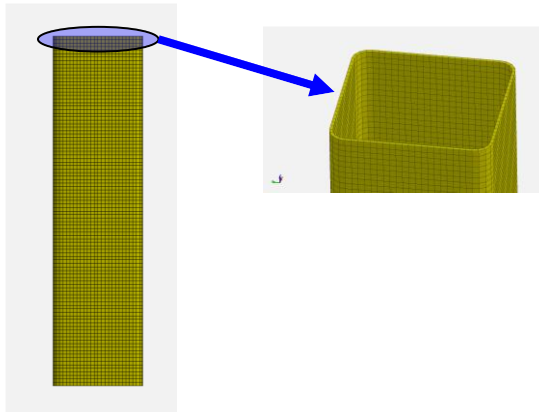

Figure 11 Model set up for crush box, one layer of mesh over thickness

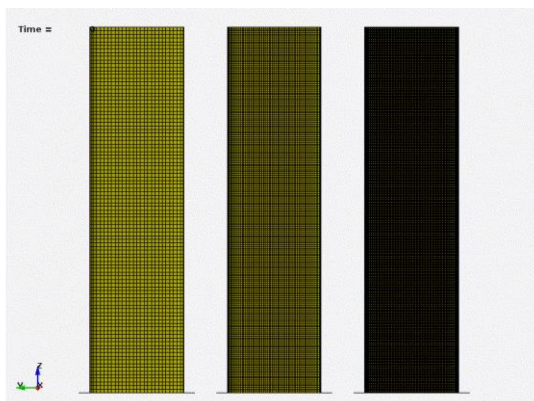

Figure 12 Model discretization with 4mm, 2mm, and 1mm mesh

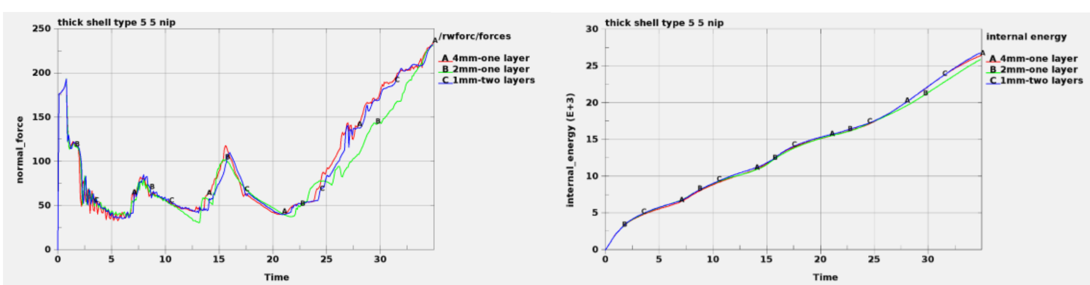

Figure 13 Contact force and internal energy curve

The contact force history and internal energy history curve are shown in Figure 13. It can be seen, even with 4mm coarse mesh, the 27-node element already get converged results and agree with fine mesh very well.

Summary

The paper present a newly implemented 27-node second order element in LS-DYNA for both explicit and implicit analysis. Several numerical tests demonstrate that the proposed element work well for shell/plate as well as fully three-dimensional problems, for the material is compressible or nearly incompressible, for the mesh is regular or distorted. Even no special strategies have been used to eliminate shear locking, the proposed elements yield acceptably accurate results without the use of artificial hourglass control or incompatible modes, and the convergence with mesh refinement is very rapid. The computational cost as compared to stand 8-node elements is higher, but what is gained is versatility in modeling a wide variety of geometries including shell/plate and three-dimensional ones, and simplicity from a user viewpoint since only displacement degrees of freedom are used.

References

1. MacNeal, R. H. et al.," Proposed Standard Set of Problems to Test Finite Element Accuracy", Finite Elements in Analysis and Design, 1985.
2. Wilkins, ML et al.," Impact of cylinders on a rigid boundary", Journal of Applied Physics, 1973.
3. Tobias Erhart, "Review of Solid Element formulations in LS-DYNA", LS-DYNA Forum 2011, Entwicklerforum. Stuttgart 2011
4. Kent T. Danielson, James L. O'Daniel, "Reliable second-order hexahedral elements for explicit methods in nonlinear solid dynamics", International Journal for Numerical Methods in Engineering 85(9):1073 - 1102 · 2011