Frequency Domain Analysis for Isogeometric Element in LS-DYNA®

Liping Li[1], Yun Huang[1], Zhe Cui[1], Stefan Hartmann[2], David J. Benson[3]

[1]Livermore Software Technology Corporation, Livermore, CA, USA
[2]DYNAmore GmbH, Stuttgart, Germany
[3]Professor for Structural Engineering, UC San Diego, CA, USA

Abstract

In the past few years numerous researches have been done in the area of Isogeometric Analysis (IGA), as the simulation model is exactly the desired geometry in this method. Frequency domain analysis is a cheap and fast alternative for time domain analysis. It is particularly suitable for vibration and acoustic analysis, which are very important topics for the design and research of automotives. This paper gives some frequency domain analysis for isogeometric elements, and the results are compared with the finite element analysis (FEA) results.

1. Introduction

The Finite Element Method (FEM), as we know it today, was introduced in the 1950's. Due to the development of computer technology and finite element software, it becomes more and more widely used to solve various engineering problems. Yet many difficulties encountered with FEM due to its approximation, polynomial based geometry, such as, for example, mesh generation, mesh refinement, sliding contact, flows about aerodynamic shapes, etc. It seems that it is the time to look for more powerful descriptions of geometry to provide a new basis for analysis. Hughes [1] had proposed a numerical method base on the geometry in Computer Aided Design (CAD), and it is called the IGA. It adopts the same mathematical description for the geometry as in the CAD to replace the piecewise continuous Lagrangian polynomial which is the traditional interpolation function used in the FEM. At the same time, it can use the same framework of numerical method as in FEM. In the context of IGA many research activities has been focused on the usage of Non-Uniform Rational B-Splines (NURBS) as basis functions. The NURBS are very well suited for computational analysis and can give more accurate results in comparison with standard FEM. In LS-DYNA, the continuous development of IGA has been added in the last few years, especially for shell element ([2], [3], [4], [5], [6]). The IGA for solid element has also been implemented into LS-DYNA starting from 2014.

On the other hand, frequency domain analysis is an important branch in CAE which concerns the response of structures in a range of frequencies. It has wide application in the areas where frequency is an important factor in determining the structural behaviors under various excitations, such as vibration, acoustics and fatigue analysis. It is a cheap and effective alternative to time domain analysis in many cases especially when the structure is subjected to cyclic or time harmonic loading. In LS-DYNA, a series of frequency domain analysis features have been implemented to meet the need from customers, especially those from auto NVH industries. These features include FRF (frequency response function), SSD (steady state dynamics), random vibration, response spectrum analysis, and acoustics. They were all based on FEM, e.g. using FEM to get the eigen-frequencies and eigenvectors, or setting up the equation systems based on FEM. With the emergence of IGA technology, a new window has been opened for the world of frequency domain analysis.

This paper will give an introduction of the frequency domain analysis for isogeometric element. Some examples are given to illustrate this application. The results are compared with the finite element results.

2. Isogeometric analysis with NURBS

2.1 B-spline basis function and curve

B-splines are piecewise polynomial curves composed of linear combinations of B-splines basis functions. The coefficients are points in space, called control points, while the basis function are composed of knot vectors. A knot vector is a set of non-decreasing real numbers, shown as:

$$U = \{\xi_1, \xi_2, \dots, \xi_{n+p+1}\} \quad (1)$$

where p is the order of the B-spline and n is the number of basis functions. Given the knot vector, the basis function can be defined recursively as follows:

when p=0：

$$N_{i,0} = \begin{cases} 1, & \text{if } \xi_i \le \xi \le \xi_{i+1} \\ 0, & \text{otherwise} \end{cases} \quad (2)$$

when p>0：

$$N_{i,p}(\xi) = \frac{\xi-\xi_i}{\xi_{i+p}-\xi_i} N_{i,p-1}(\xi) + \frac{\xi_{i+p+1}-\xi}{\xi_{i+p+1}-\xi_{i+1}} N_{i+1,p-1}(\xi) \quad (3)$$

Figure 1 is an example of basis function with second order given U={0,0,0,1,2,3,4,4,5,5,5}.

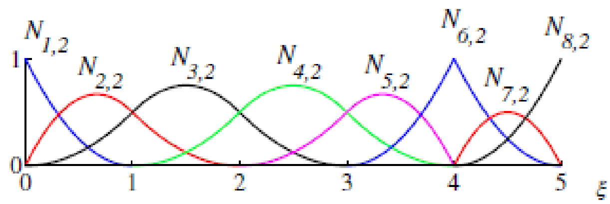


Figure 1: bais funciton with second order, and U={0,0,0,1,2,3,4,4,5,5,5}.

B-spline can be represented as:

$$C(\xi) = \sum_{i=1}^{n} N_{i,p}(\xi) B_i \quad (4)$$

where B_i is the control points. By using the knot vector shown in Figure 1 and the control points (•) in the following figure, the B-spline curve can be plotted as in Figure 2.

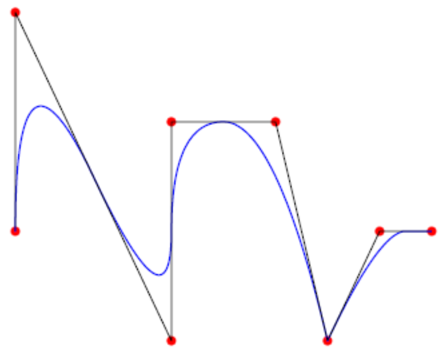


Figure 2: B-spline curve with control points given by (•).

2.2 NURBS curve

NURBS curve can be easily obtained by augmenting the spatial coordinates of the control points with weights:

$$C(\xi) = \frac{\sum_{i=1}^{n} N_{i,p}(\xi) B_i w_i}{\sum_{i=1}^{n} N_{i,p}(\xi) w_i} = \sum_{i=1}^{n} R_{i,p}(\xi) B_i \tag{5}$$

where B_i is the control point, w_i is the weight factor, and $R_{i,p}(\xi) = \frac{N_{i,p}(\xi) w_i}{\sum_{i=1}^{n} N_{i,p}(\xi) w_i}$.

2.3 NURBS surface and body

Given a net of control points $\{B_{i,j}\}$, i=1,2,…n, j=1,2,…m, and two knot vectors $U^1 = \{\xi_1, \xi_2, \dots, \xi_{n+p+1}\}$ and $U^2 = \{\eta_1, \eta_2, \dots, \eta_{m+q+1}\}$, the NURBS surface can be constructed as:

$$S(\xi,\eta) = \frac{\sum_{i=1}^{n} \sum_{j=1}^{m} N_{i,p}(\xi) N_{j,q}(\eta) B_{i,j} w_{i,j}}{\sum_{i=1}^{n} \sum_{j=1}^{m} N_{i,p}(\xi) N_{j,q}(\eta) w_{i,j}} = \sum_{i=1}^{n} \sum_{j=1}^{m} R_{ij,pq}(\xi,\eta) B_{i,j} \tag{6}$$

where w_i is the weight factor, and $R_{ij,pq}(\xi,\eta) = \frac{N_{i,p}(\xi) N_{j,q}(\eta) w_{i,j}}{\sum_{i=1}^{n} \sum_{j=1}^{m} N_{i,p}(\xi) N_{j,q}(\eta) w_{i,j}}$.

Similarly, given a net of control points $\{B_{i,j,k}\}$, i=1,2,…n, j=1,2,…m, k=1,2,…,l, and three knot vector $U^1 = \{\xi_1, \xi_2, \dots, \xi_{n+p+1}\}$, $U^2 = \{\eta_1, \eta_2, \dots, \eta_{m+q+1}\}$ and $U^3 = \{\zeta_1, \zeta_2, \dots, \zeta_{l+r+1}\}$, the NURBS solid can be constructed as:

$$S(\xi,\eta,\zeta) = \frac{\sum_{i=1}^{n} \sum_{j=1}^{m} \sum_{k=1}^{l} N_{i,p}(\xi) N_{j,q}(\eta) N_{k,r}(\zeta) B_{i,j,k} w_{i,j,k}}{\sum_{i=1}^{n} \sum_{j=1}^{m} \sum_{k=1}^{l} N_{i,p}(\xi) N_{j,q}(\eta) N_{k,r}(\zeta) w_{i,j,k}} = \sum_{i=1}^{n} \sum_{j=1}^{m} \sum_{k=1}^{l} R_{ijk,pqr}(\xi,\eta,\zeta) B_{i,j,k} \tag{7}$$

where $w_{i,j,k}$ is the weight factor, and $R_{ijk,pqr}(\xi,\eta,\zeta) = \frac{N_{i,p}(\xi) N_{j,q}(\eta) N_{k,r}(\zeta) w_{i,j,k}}{\sum_{i=1}^{n} \sum_{j=1}^{m} \sum_{k=1}^{l} N_{i,p}(\xi) N_{j,q}(\eta) N_{k,r}(\zeta) w_{i,j,k}}$.

3. Implementation in LS-DYNA

In LS-DYNA, the keyword *ELEMENT_SHELL_NURBS and *ELEMENT_SOLID_NURBS are used to define the shell and solid NURBS element. In both cards, the information of knot vector, polynomial order and weight factors are defined. LS-DYNA will automatically calculate the basis functions and give the geometry of NURBS surface or body. To run the frequency domain analysis, user also need to set up the keywords like *FREQUENCY_DOMAIN_SSD, *FREQUENCY_DOMAIN_RANDOM_VIBRATION, etc. Those keywords specify the range of frequencies under consideration, the type or location of loading, boundary conditions, response type and directions, etc. The results of eigen-frequencies and eigenvectors from IGA simulation will be transferred to the frequency domain solvers and provide the basis for further frequency domain analysis.

4. Steady State Dynamic Response of IGA

4.1 Shell element case

Figure 3 shows a model of one part from a complicated structure. It was modeled by isogeometric elements and finite elements separately. A y-direction load is applied to a node on one edge as shown in the figure. The keyword *FREQUENCY_DOMAIN_SSD is then used to study the steady state dynamic response. The stress and displacement results of isogeometric element and finite element are compared and given in the following figures. For isogeometric element, the numbers of control points are 41 and polynomial orders are 3 in both r- and s-directions. The y-displacement and z-stress distributions at frequency 0.1 are shown in Figure 4. The maximum y-displacement and z-stress are 1.591 and 5.605, respectively. For finite element, three different mesh sizes, from coarse to fine, are used and the results on the finest mesh will be chosen as reference. Figure 5 shows the y-displacement at frequency 0.1 on the three finite element models and the element numbers are also given under each sub-figure. It seems that the displacement distributions are very similar and the maximum displacement increases with the element number. Figure 6 gives how the displacement and stress changes with the change of element number. For finite element model (the blue curves), both y-displacement and z-stress will increase as the element number increases. For isogeometric element model, only one case was studied as shown in Figure 4, so a constant value was plotted (red dotted lines) to show how the y-displacement and z-stress compare to the finite element results. It seems the isogementric results are very close to the results of finite element model with larger element number (12996 elements case).

Table 1 gives the CPU time using LS-DYNA SMP Double precision and it shows the isogeometric model cost is very competitive with the finite models.

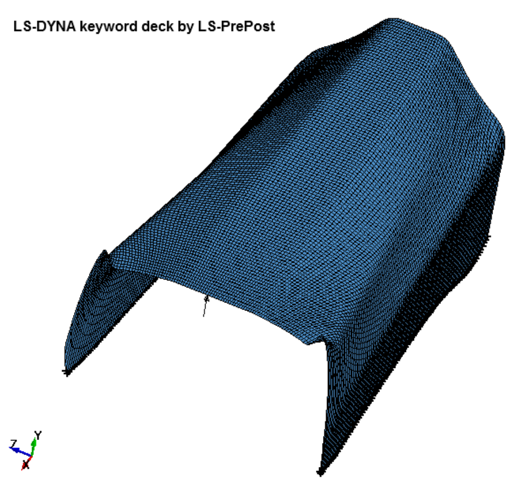
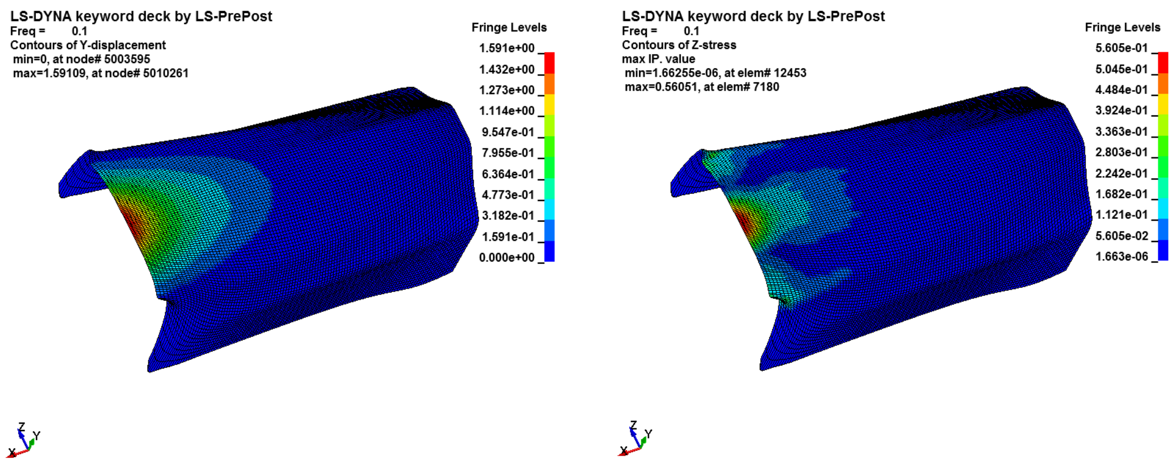
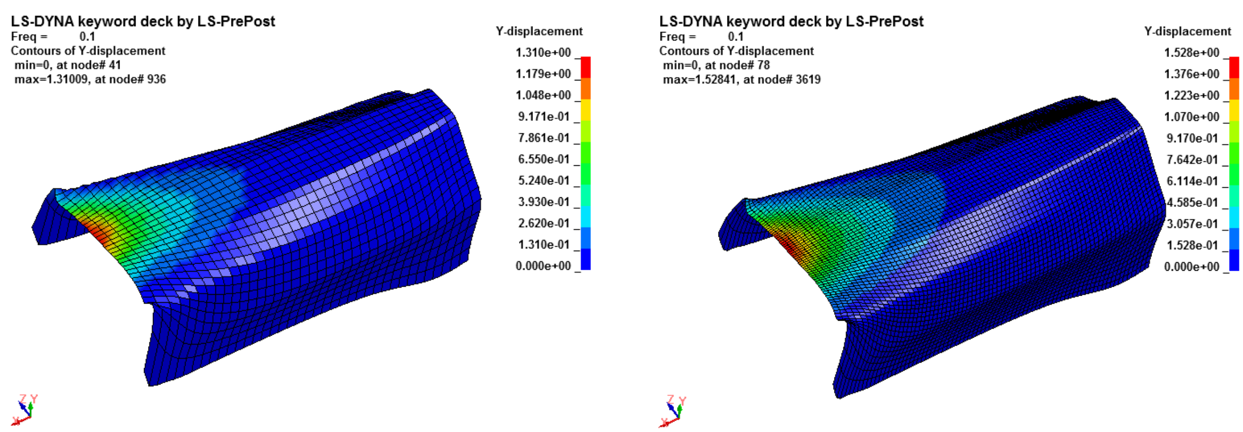


Figure 3 shell element model

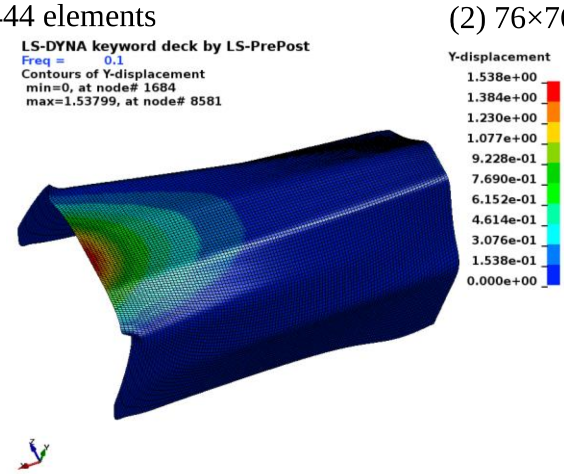


(1) y-displacement (2) z-stress

Figure 4 results for Isogeometric element models

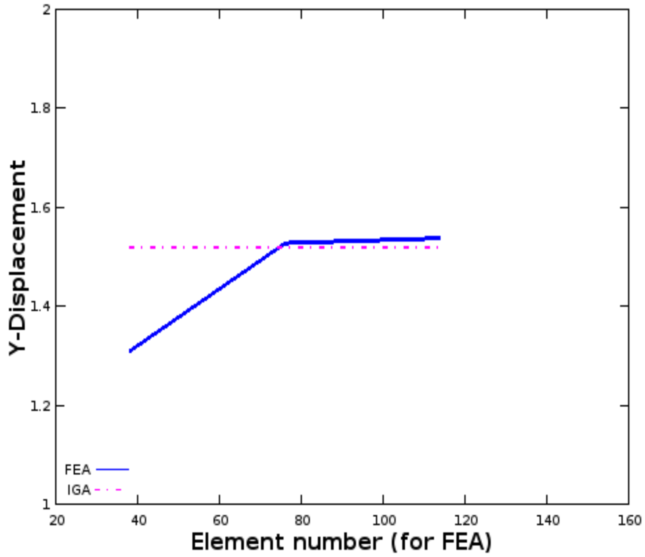


(1) 38×38=1444 elements (2) 76×76=5776 elements

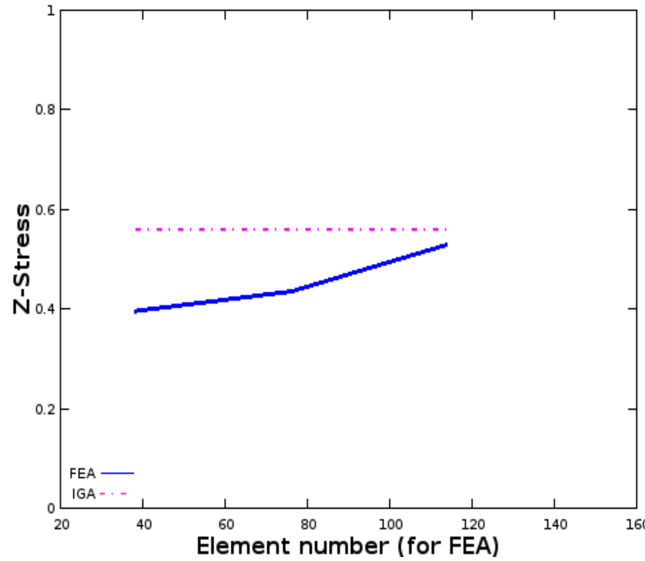


(3) 114×114=12996 elements

Figure 5 y-displacements for finite element models



(1) y-displacement



(2) z-stress

Figure 6 y-displacement and z-stress changes as the element number

Table 1 CPU time using LS-DYNA SMP Double precision (s)

Case	FEA Case 1	FEA Case 2	FEA Case 3	IGA
Time	6	23	54	47

4.2 Solid element case

In the following we will show the steady state dynamic study for solid models with isogeometric element and finite element, respectively. A z-direction load is applied as shown in the figure 7. The keyword *FREQUENCY_DOMAIN_SSD is then used the same way as for the shell models. The x-stress and z-displacement at frequency 0.01 for the two types of element are compared in Figure 8 and 9, respectively. It shows that both the stress and displacement are very similar for these two types of element.

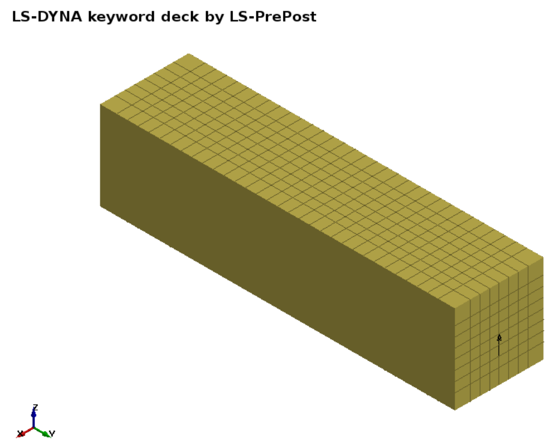
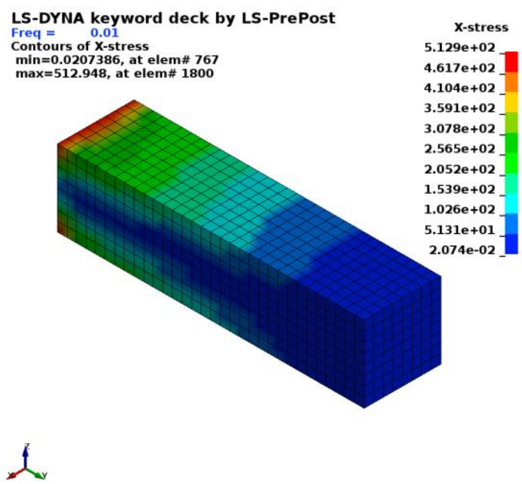
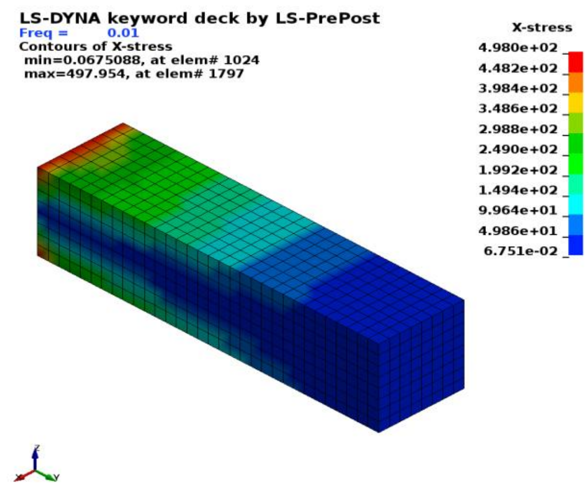


Figure 7 solid element model

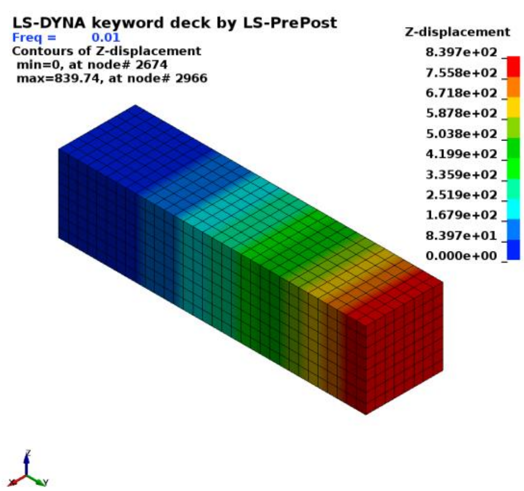


(1) Isogeometric element

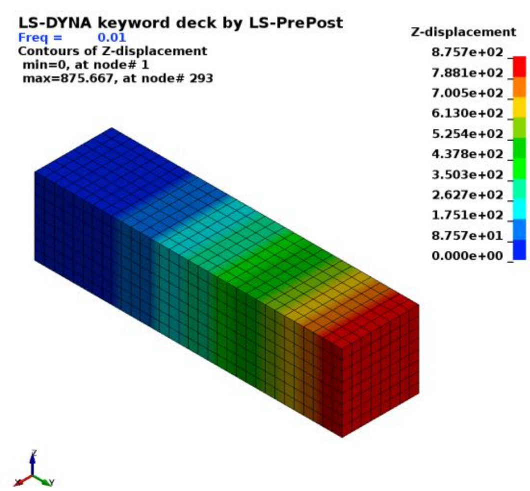


(2) finite element

Figure 8 x-stress comparison



(1) Isogeometric element



(2) finite element

Figure 9 z-displacement comparison

5. Conclusion

The basic ideas of IGA have been introduced along with a brief introduction of B-spline and NURBS. The keywordS *ELEMENT_SHELL_NURBS, *ELEMENT_SOLID_NURBS and *FREQUENCY_DOMAIN_SSD are used to study the steady state dynamics for isogeometric elements and the results are compared with that using traditional finite elements. These preliminary studies show that the IGA and FEA can give very similar results. IGA is also cost competitive for the models studied in this paper.

A lot more work need to be done to figure out the areas in which IGA could be the first choice. Many features that available for standard finite element in LS-DYNA need to be adopted for the IGA in the near future.

6. References

[1] Hughes T J R, Cottrell J A, Bazilevs Y. Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement. Computer Methods in Applied Mechanics and Engineering, 2005, 194(39-41):4135-4195

[2] Benson D J, Bazilevs Y, Hughes T J R. Preliminary Results for an Isogeometric Shell. 10th International LS-DYNA Users Conference. Dearborn, Michigan, June 2008.

[3] Benson D J. Isogeometric Anslysis in LS-DYNA. 11th International LS-DYNA Conference. Dearborn, Michigan, June 2010.

[4] Hartman S, Benson D J, Lorenz D. About Isogeometric Anslysis the new NURBS-based Finite Element in LS-DYNA. 8th European LS-DYNA Conference. Strasbourg, France, May 2011.

[5] Nagy A P, Hartman S, Benson D J. Isogeometric Anslysis in LS-DYNA. 13th International LS-DYNA Conference. Dearborn, Michigan, June 2014.

[6] Benson D J, Bhalsod D, Hartman S, Ho P, Li L, Li W, Nagy A P, Yeh I. Isogeometric Anslysis in LS-DYNA: Using CAD-Geometry for Numerical Simulation. 10th European LS-DYNA Conference. Würzburg, Germany, June 2015.