

# Shear Locking and Hourglassing in MSC Nastran, ABAQUS, and ANSYS

Eric Qiuli Sun

## Abstract

A solid beam and a composite beam were used to compare how MSC Nastran, ABAQUS, and ANSYS handled the numerical difficulties of shear locking and hourglassing. Their tip displacements and first modes were computed, normalized, and listed in multiple tables under various situations. It was found that fully integrated first order solid elements in these three finite element codes exhibited similar shear locking. It is thus recommended that one should avoid using this type of element in bending applications and modal analysis. There was, however, no such shear locking with fully integrated second order solid elements. Reduced integration first order solid elements in ABAQUS and ANSYS suffered from hourglassing when a mesh was coarse. If there was only one layer of elements, the reported first mode of the beam examples from ABAQUS and ANSYS was excessively smaller than the converged solutions due to hourglassing. At least four layers of elements should, therefore, be used in ABAQUS and ANSYS. MSC Nastran outperformed ABAQUS and ANSYS by virtually eliminating the annoying hourglassing of reduced integration first order 3D solid elements because it employed bubble functions to control the propagation of non-physical zero-energy modes. Even if there was only one layer of such elements, MSC Nastran could still manage to produce reasonably accurate results. This is very convenient because it is much less prone to errors when using reduced integration first order 3D solid elements in MSC Nastran.

## Introduction

In finite element analysis (FEA), shear locking and hourglassing are two major numerical problems because they may cause spurious solutions in certain situations. They are, however, usually not discussed in the conventional college finite element textbooks (Logan, 1992). It is, therefore, necessary to have a solid understanding of how major commercial FEA codes handle these numerical difficulties before performing any serious engineering analysis.

Start with a cantilever beam example (Fig. 1).

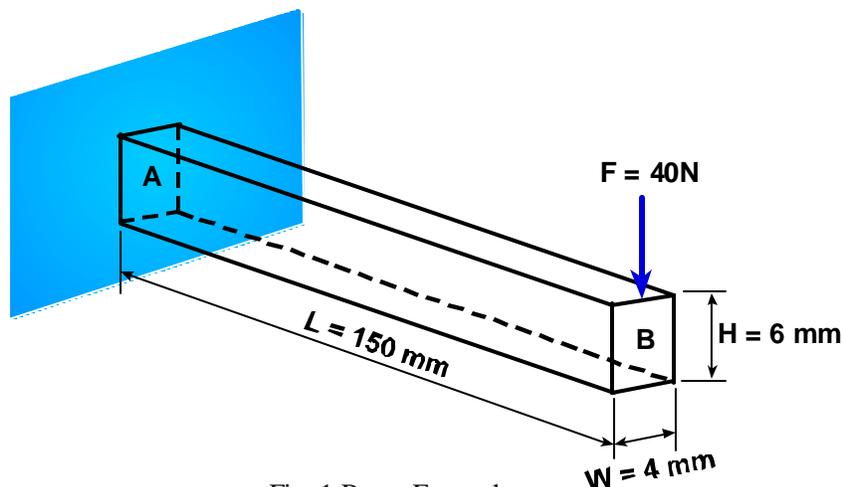


Fig. 1 Beam Example

The dimensions of the beam are  $L \times W \times H = 150\text{mm} \times 4\text{mm} \times 6\text{mm}$ . The beam is fixed at the end A and a force of 40N is applied at the free end B. The material properties (Matthews and Davies, 2003) are

$$E_{11} = 146.9\text{GPa} = 146.9 \times 10^3 \text{ N/mm}^2$$

$$E_{22} = E_{33} = 10.89\text{GPa} = 10.89 \times 10^3 \text{ N/mm}^2$$

$$G_{12} = G_{13} = 10.89\text{GPa} = 10.89 \times 10^3 \text{ N/mm}^2$$

$$G_{23} = 6.4\text{GPa} = 6.4 \times 10^3 \text{ N/mm}^2$$

$$\nu_{12} = \nu_{23} = 0.38$$

$$\nu_{23} = 0.776$$

$$\rho = 1.5 \times 10^3 \text{ kg/m}^3 = 1.5 \times 10^{-9} \text{ ton/mm}^3$$

Beam elements could be used to conduct the analysis of this example, which was actually much simpler and easier. However, it was desirable to know how solid elements perform in this particular case. The beam was meshed such that  $H \times L \times W = 4 \times 10 \times 1$ , which means there were 4 elements in the height, 10 elements in the length, and 1 element in the width.

Since the material was not isotropic, classical beam theory was not appropriate for computing the deflection at the tip. It was found through convergence study that the deflection at the tip was 4.281 mm and that the first mode of the beam was 284.0 Hz. Table 1 and Table 2 are a collection of normalized solutions from Nastran, ABAQUS, and ANSYS.

Table 1 Normalized Results from First Order Brick Elements with Full Integration

FEA code	Nastran	ABAQUS	ANSYS
Element	CHEXA (8-node)	C3D8	SOLID45 (8-node)*
Tip Deflection	0.6774	0.6933	0.6772
First mode	1.429	1.075	1.435

\* Extra displacement shapes were suppressed.

Table 2 Normalized Results from First Order Brick Elements with Reduced integration

FEA code	Nastran	ABAQUS	ANSYS
Element	CHEXA (8-node)	C3D8R	SOLID45 (8-node)
Tip Deflection	0.990	1.063	1.064
First mode	0.9965	0.04225	0.03027

For first order brick elements, it is apparent from the Table 1 and Table 2 that the solutions do not agree with each other when using different integration schemes even with the same FEA code. In particular, the solutions from ABAQUS, ANSYS and those from Nastran for the first mode of the beam are dramatically different with a reduced integration scheme. Nastran is capable of doing a much better job because its results are close to the converged solutions. All of the three codes gave poor results for the deflection and the first mode with a full integration scheme.

## Shear Locking

Based on Table 1, fully integrated linear brick elements of Nastran, ABAQUS, and ANSYS are overly stiff in bending applications and modal analysis. This numerical problem is called shear locking. Fully integrated first order elements, such as solid elements, Timoshenko beam elements, Mindlin plate

elements, may suffer from the locking (Prathap, 2005). FEA codes could therefore give false results when this type of element is used.

In an ideal situation, a block of material under a pure bending moment experiences a curved shape change (Fig. 2). Suppose there are straight dotted lines on the surface of the block. Under the bending moment, horizontal dotted lines and edges bend to curves while vertical dotted lines and edges remain straight. The angle  $A$  remains at 90 degrees after bending, just as predicted by classical beam theory (Gere and Timoshenko, 1997).

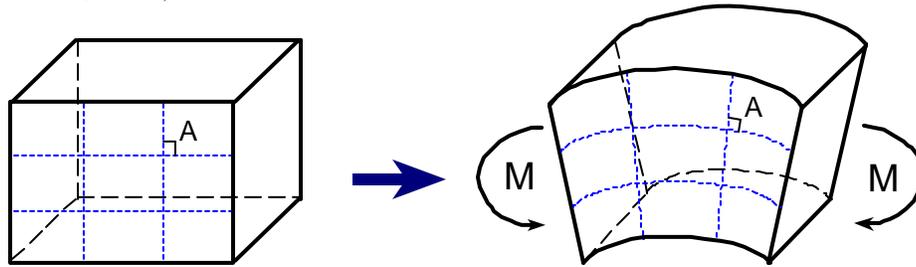


Fig. 2 Shape Change of the Material Block under the Moment in the Ideal Situation

To correctly model the ideal shape change, an element should have the ability to assume the curved shape. The edges of the fully integrated first order element are, however, not able to bend to curves. The linear element will develop a shape shown in Fig. 3 under a pure bending moment. The top surface experiences tensile stress, and the lower surface experiences compressive stress. All dotted lines remain straight. But the angle  $A$  can no longer stay at 90 degrees.

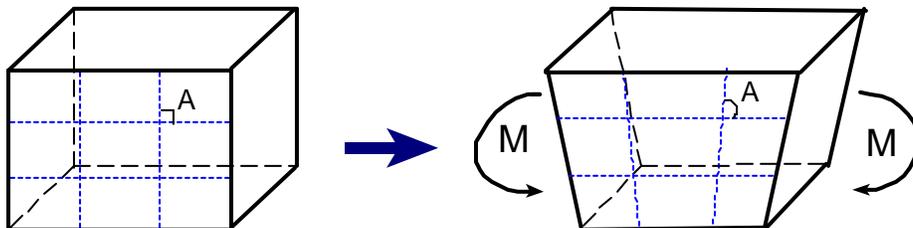


Fig. 3 Shape Change of the Fully Integrated First Order Element under the Moment

To cause the angle  $A$  to change under the pure moment, an incorrect artificial shear stress has been introduced. This also means that the strain energy of the element is generating shear deformation instead of bending deformation. The overall effect is that the linear fully integrated element becomes locked or overly stiff under the bending moment. Wrong displacements, false stresses, and spurious natural frequencies may be reported because of the locking.

The fully integrated second order element behaves differently since its edges are able to bend to curves. Under a bending moment, the shape change of the element will correctly assume that of the material block (Fig. 4). The angle  $A$  continues to remain at 90 degrees after the bending. No artificial shear stress is introduced and the element can correctly simulate the behaviors of the material block. There is no shear locking associated with this type of element.

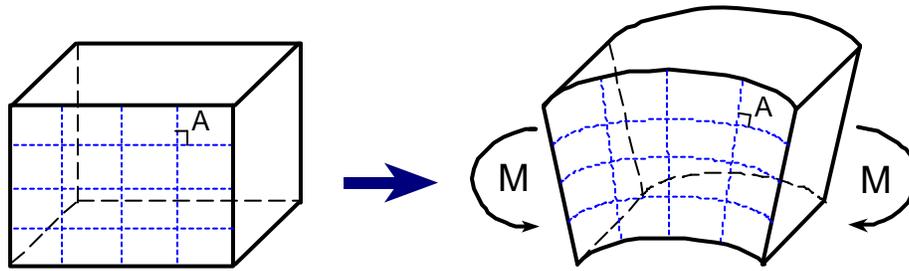


Fig. 4 Shape Change of the Fully Integrated Second Order Element under the Moment

## Hourglassing

To address the shear locking and to increase computational efficiency, a reduced integration scheme is proposed and widely implemented in FEA codes. For example, for the reduced-integration first order 8-node brick element, a single integration point scheme is used while its fully integrated version employs eight integration points. For the reduced-integration, second-order, 20-node brick element, an 8-integration-point scheme is used while its fully integrated version in ABAQUS needs 27 integration points. In addition, the reduced integration element is tolerant of shape distortions, which is significantly beneficial in finite element modeling.

However, nothing is perfect. The reduced integration first order element suffers from its own numerical difficulty called hourglassing since it tends to be excessively flexible. The hourglassing has to be properly controlled. If not, the results from this type of element are often not usable.

Figure 5 demonstrates the deformation of such an element under a bending moment. To visualize the deformation, notice that vertical and horizontal dotted lines and the angle  $A$  remain unchanged. This means that normal stresses and shear stresses are zero at the integration point and that there is no strain energy generated by the deformation. This zero-energy mode is a nonphysical response, which may propagate when a coarse mesh is used. The propagation of such a mode may therefore produce meaningless results. The results often indicate that the structure is excessively flexible. In order to make the reduced integration elements useful, the FEA codes provide default hourglassing control internally. The user may be able to adjust control parameters.

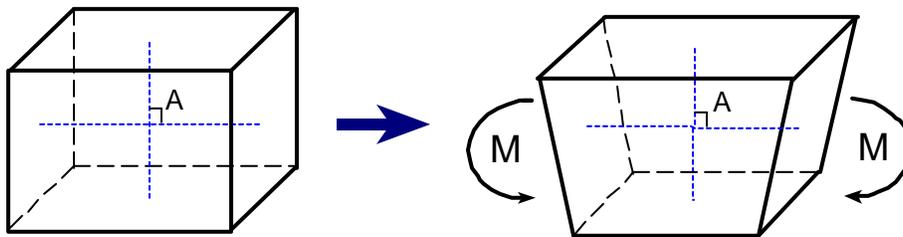


Fig. 5 Shape Change of the Reduced Integration Element under the Moment

For the second order solid element with reduced integration, it may also suffer from hourglassing when only one layer of elements is used. But this rarely causes numerical problems because it virtually vanishes with two layers of elements. No special technique is needed to control it.

## Beam Example

Since shear locking and hourglassing are major numerical problems in FEA, special techniques were developed to control them. However, commercial FEA codes may use different techniques. It is, therefore, likely for FEA codes to give different results with the same problem under the same mesh. There is, however, only one correct answer for the same problem. Thus it is critical to have a solid understanding of how the FEA codes may perform in various situations.

Consider the beam shown in Fig. 1 as an example to demonstrate the differences. As it is mentioned previously, convergence study has shown that the deflection of the tip of the beam under a load of 40N was 4.281 mm and that its first mode (first natural frequency) was 284.0 Hz. Default values of hourglass control were used with the reduced integration elements in ABAQUS and ANSYS. The bubble functions were employed in Nastran with the reduced integration first order elements.

The following tables were normalized so that it is easy to see how far the solutions are away from the converged solutions.

Table 5 and Table 6 show the normalized tip deflections and the normalized first modes from fully integrated 8-node brick elements. It is noticeable that all three codes gave overly stiff results and that the refinement of the elements did not help in a significant way. Therefore, one must be very cautious when applying this type of element.

ABAQUS makes efforts to overcome the issue of shear locking by providing incompatible mode elements such as C3D8I and CPS4I. They are fairly accurate if there are no distortions (Table 5 and Table 6). However, they are sensitive to element distortions, which may make them much too stiff and less useful because it is difficult not to distort the elements in real-life finite element modeling practices. Similarly, ANSYS allows the user to include extra displacement shapes in its fully integrated 8-node brick element to produce more accurate results.

Table 5 Normalized Tip Deflection with Fully Integrated 8-node Brick Elements

Element	Mesh Size (Height x Length x Width)					
	1x3x1	2x5x1	4x10x1	6x20x1	7x30x1	8x40x1
CHEXA(Nastran)	0.1615	0.3471	0.6774	0.8904	0.9463	0.9675
C3D8(ABAQUS)	0.1859	0.3665	0.6933	0.9021	0.9556	0.9743
C3D8I(ABAQUS)	0.9682	0.9769	0.9900	0.9956	0.9974	0.9984
SOLID45(ANSYS)*	0.1615	0.3471	0.6774	0.8904	0.9463	0.9675
SOLID45(ANSYS)**	0.9673	0.9766	0.9900	0.9956	0.9974	0.9981

Table 6 Normalized First Mode with Fully Integrated 8-node Brick Elements

Element	Mesh Size (Height x Length x Width)					
	1x3x1	2x5x1	4x10x1	6x20x1	7x30x1	8x40x1
CHEXA (Nastran)	3.518	2.272	1.429	1.129	1.062	1.037
C3D8(ABAQUS)	3.280	2.052	1.075	0.5965	0.4514	0.3866
C3D8I(ABAQUS)	0.9717	0.9893	0.9965	0.9982	0.9985	0.9986
SOLID45 (ANSYS)*	3.518	2.272	1.435	1.129	1.062	1.038
SOLID45(ANSYS)**	1.011	1.005	1.001	0.9996	0.9993	0.9992

\* Extra displacement shapes were suppressed.

\*\* Extra displacement shapes were included.

Table 7 and Table 8 demonstrate the normalized tip deflections and the normalized first modes from fully integrated 20-node brick elements. Even with a very coarse mesh, the results from this type of element were close to the converged solutions.

Table 7 Normalized Tip Deflection with Fully Integrated 20-node Brick Element

Element	Mesh Size (Height x Length x Width)					
	1x3x1	2x5x1	4x10x1	6x20x1	7x30x1	8x40x1
CHEXA(Nastran)	0.9724	0.9900	0.9967	0.9988	0.9998	1.00
C3D20 (ABAQUS)	0.9724	0.9900	0.9967	0.9988	0.9998	1.00
SOLID95 (ANSYS)	0.9729	0.9900	0.9967	0.9988	0.9998	1.00

Table 8 Normalized First Mode with Fully Integrated 20-node Brick Element

Element	Mesh Size (Height x Length x Width)					
	1x3x1	2x5x1	4x10x1	6x20x1	7x30x1	8x40x1
CHEXA (Nastran)	1.021	1.008	1.002	1.001	1.000	1.000
C3D20 (ABAQUS)	1.025	1.008	1.002	1.001	1.000	1.000
SOLID95 (ANSYS)	1.024	1.008	1.002	1.001	1.000	1.000

Table 9 and Table 10 illustrate that the performance of different codes may be drastically different. When a mesh was coarse, ABAQUS and ANSYS had difficulty in obtaining reasonably accurate tip deflections. However, once the mesh was refined, they both could provide good results. ABAQUS and ANSYS gave the incorrect first mode of the beam no matter whether the mesh of height and length was coarse or fine. Because there was only one layer of elements in the width, the FEA model was excessively flexible in that direction due to the reduced integration. ABAQUS and ANSYS gave much smaller first modes than the converged one. The default hourglassing control was used in ABAQUS and ANSYS. For example, if the beam was meshed such that  $H \times L \times W = 6 \times 20 \times 6$ , the normalized first mode in ANSYS was 0.9844, which was close to the converged solution.

Nastran did a superior job in both the tip deflection and the first mode. Even though the mesh was very coarse, it could report reasonably accurate results. From the Nastran manual, it is apparent that Nastran employs a special technique, bubble functions, to work with the reduced elements. This technique makes it possible produce reliable results and reduces the human errors in avoiding hourglassing. However, the use of bubble functions is more expensive computationally.

Table 9 Normalized Tip Deflection with Reduced 8-node Brick Element

Element	Mesh Size (Height x Length x Width)					
	1x3x1	2x5x1	4x10x1	6x20x1	7x30x1	8x40x1
CHEXA (Nastran)	0.9680	0.9771	0.9900	0.9958	0.9974	0.9984
C3D8R (ABAQUS)	534.2	1.316	1.063	1.028	1.021	1.016
SOLID45 (ANSYS)	1049	1.327	1.064	1.028	1.021	1.016

Table 10 Normalized First Mode with Reduced 8-node Brick Element

Element	Mesh Size (Height x Length x Width)					
	1x3x1	2x5x1	4x10x1	6x20x1	7x30x1	8x40x1
CHEXA (Nastran)	1.013	1.005	0.9965	0.9993	0.9986	0.9986
C3D8R (ABAQUS)	0.0412	0.04225	0.04225	0.04225	0.04261	0.04261
SOLID45 (ANSYS)	0.02946	0.03027	0.03027	0.03033	0.03034	0.03035

Table 11 and table 12 shows that reduced 20-node brick elements can usually reported accurate results of the tip deflection and the first mode. When only one layer of elements was used, ABAQUS and ANSYS simply failed to produce any results because the hourglass mode has no stiffness. When there were two layers of elements, ABAQUS and ANSYS were successful in giving accurate results. Nastran, however, provide accurate results with only one layer of elements.

Table 11 Normalized Tip Deflection with Reduced 20-node Brick Elements

Element	Mesh Size (Height x Length x Width)					
	1x3x1	2x5x1	4x10x1	6x20x1	7x30x1	8x40x1
CHEXA (Nastran)	0.9848	0.9925	0.9967	0.9991	0.9998	1.000
C3D20R (ABAQUS)	No sol.	0.9981	1.002	1.016	1.036	1.065
SOLID95 (ANSYS)	No sol.	0.9981	1.002	1.016	1.036	1.065

Table 12 Normalized First Mode with Reduced 20-node Brick Elements

Element	Mesh Size (Height x Length x Width)					
	1x3x1	2x5x1	4x10x1	6x20x1	7x30x1	8x40x1
CHEXA (Nastran)	1.011	1.005	1.002	1.000	1.000	1.000
C3D20R (ABAQUS)	No sol.	1.002	1.001	1.000	1.000	1.000
SOLID95 (ANSYS)	1.024	1.002	1.001	1.000	1.000	0.9998

## Composite Beam Example

The numerical difficulties of shear locking and hourglassing were further demonstrated with a composite beam example (Fig. 6). The beam is a four-ply symmetrical laminate with a  $(0/90^D)_s$  stacking sequence. The material properties are the same as that of the beam example.

Through the convergence study, it was found that the deflection at the tip was 4.832 mm and that the first mode was 209.1 Hz.

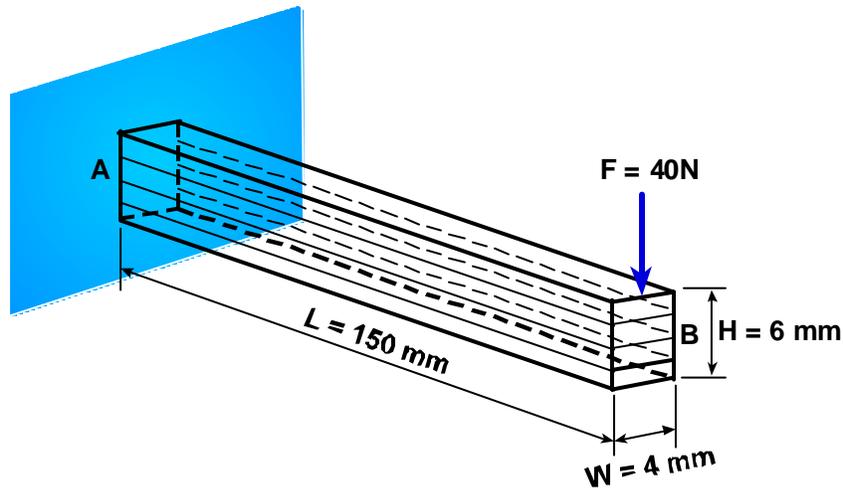


Fig. 6 Composite Beam Example

Table 13 Normalized Tip Deflection with 8-node Brick Elements

Element	Mesh Size (Height x Length x Width)			
	4x3x1	4x5x1	4x10x1	4x20x1
CHEXA (Full, Nastran)	0.1771	0.3725	0.6993	0.8976
C3D8 (Full, ABAQUS)	0.1779	0.3756	0.7092	0.9131
Solid45 (Full, ANSYS)*	0.1771	0.3725	0.6993	0.8976
CHEXA (Reduced, Nastran)	0.9501	0.9768	0.9921	0.9981
C3D8 (Reduced, ABAQUS)	1.014	1.032	1.040	1.042
Solid45 (Reduced, ANSYS)	1.014	1.033	1.040	1.043

\* Extra displacement shapes were suppressed.

Table 13 and table 14 demonstrate the overly stiff issue of the fully integrated first-order brick elements in the bending application and modal analysis. The tables also indicate the results from reduced integration versions in ABAQUS and ANSYS were excessively flexible. For example, the first mode produced by ABAQUS and ANSYS was much smaller than the converged solution when there was only one layer of element in the width. Nastran gave reasonably accurate solutions due to the use of bubble functions.

Table 14 Normalized First Mode with 8-node Brick Elements

Element	Mesh Size (Height x Length x Width)			
	4x3x1	4x5x1	4x10x1	4x20x1
CHEXA (Full, Nastran)	4.321	2.902	1.710	1.223
C3D8 (Full, ABAQUS)	4.311	2.764	1.447	0.8041
Solid45 (Full, ANSYS)*	4.498	2.947	1.717	1.225
CHEXA (Reduced, Nastran)	0.9813	0.9947	0.9976	0.9971
C3D8 (Reduced, ABAQUS)	0.05610	0.05715	0.05763	0.05772
Solid45 (Reduced, ANSYS)	0.04002	0.04079	0.04112	0.0412

\* Extra displacement shapes were suppressed.

## 2D Solid Elements

3D brick elements were used as examples to demonstrate the concept of shear locking and hourglassing. It is well understood that 2D solid elements also have similar problems.

In Nastran, CQUAD, a fully nonlinear plane strain element, and the plane strain version of CQUAD4 have the tendency of shear locking. One must be careful in bending applications and modal analysis due to this numerical difficulty.

In ABAQUS, fully integrated first order 2D elements such as CPS4 and CPE4 may manifest shear locking, and their reduced versions such as CPS4R and CPE4R may develop hourglassing.

In ANSYS, PLAE42 and PLANE182 with full integration may exhibit shear locking in bending applications and modal analysis. If it does happen, the reduced integration versions of PLANE42 and PLANE182 with hourglass control may be used.

## Summary

Nastran, ABAQUS, and ANSYS each have unique ways to handle the numerical difficulties of shear locking and hourglassing. In the bending applications and the modal analysis, the fully integration first order solid elements in all three commercial FEA codes exhibited shear locking. One must pay extra attention to this possibility and consider not using them in this case. To avoid the locking, the reduced integration versions or second order solid elements are recommended. If element distortions are not severe, the user can employ incompatible mode elements as alternative options in ABAQUS.

The reduced integration first order solid elements in ABAQUS and ANSYS suffered from hourglassing when a mesh was coarse. It is recommended that at least four layers of elements should be used in order to obtain reasonably accurate results. With the bubble function technique, Nastran outperformed ABAQUS and ANSYS by virtually getting rid of the annoying hourglassing of the reduced integration first order 3D elements. This is extremely convenient because the user may be much less prone to introduce modeling errors. In addition, reduced integration elements are computationally inexpensive and insensitive to element distortions.

## Bibliography

- (1) Logan, D. L., *A First Course in the Finite Element Method*, 2<sup>nd</sup> ed., PWS Publishing Company, Boston, 1992
- (2) Matthews, F. L., Davies, G. A. O., Hitchings D., and Soutis, C., *Finite Element Modeling of Composite Materials and Structures*, Woodhead Publishing Limited, Cambridge, England, 2003
- (3) Prathap, G., *Finite Element Analysis as Computation*, India, 2005
- (4) *ABAQUS Documentation*, version 6.3-3
- (5) MSC Nastran *Reference Manual*, version 2001
- (6) *ANSYS Documentation*, version 5.7
- (7) Gere, J. M. and Timoshenko, S. P., *Mechanics of Materials*, 4<sup>th</sup> ed., PWS Publishing Company, Boston, 1997