

Smoothed finite element method for the resultant eight-node solid shell element analysis

Xavier Élie-Dit-Cosaque¹, Augustin Gakwaya¹, Julie Lévesque¹, Michel Guillot¹

¹ Département de génie mécanique, Université Laval, Québec, Canada

Abstract: A local finite element strain smoothing process also known as smoothed finite element method (SFEM) is proposed for the resultant eight-node solid shell element applied in non linear analysis. Only displacement degrees of freedom are used to define the element kinematics. The strain smoothing process of membrane and bending terms is presented in a local Cartesian basis and results in modified strain operators which contain the product of element shape functions and normal vectors to the edges of the resultant quadrilateral element in the shell mid-surface. Cartesian derivatives of shape functions are thus no more required and the integration is performed on the resultant element along the edges of the subdividing cells located in the mid-surface of the initial eight-node hexahedral element. Hence all the smoothing process as well as the integration on cell boundaries are performed in the mid-surface. The assumed natural strain (ANS) method is used to calculate the transverse shear and the through-thickness terms of the stiffness matrix without any locking. The new element formulation has been implemented in Abaqus via the facilities provided by the User Element (UEL) subroutine. The element capability was validated analytically and successfully compared with other equivalent shell elements in various common path tests. This element seems to be robust and gives good results for both regular and distorted mesh in the considered benchmark problems.

Keywords: Solid shell element, smoothed finite element method, geometrical non linearity.

1. Introduction

Accurate analysis of complex shell structures which is nowadays accomplished by means of the finite element method is one of the most important demands of design engineers in industries. To achieve reasonable accuracy, meshes with sufficient resolution often require many hours of computer time, even with explicit methods. Hence to speed up the design process and reduce the computational cost of these simulations, the efficiency of finite elements is of crucial importance. Over the last two decades, considerable progress has been achieved in developing fast and reliable elements. In many engineering applications, e.g. metal forming problems, high standards of finite element technology are required because the work pieces undergo very large deformations and the material is plastically incompressible. Especially in these situations standard low order finite element formulations exhibit the undesirable features of locking. Too high stress values and an underestimation of the deformation are some consequences of this problem. Obviously, if the finite element analysis is expected to support the production process by means of quantitatively reliable results, the locking phenomena must be eliminated. In order to avoid locking problems

encountered in many structural mechanics problems, various efficient plate and shell elements have been developed based on mixed formulations or enhanced assumed strain (EAS) methods (see e.g. (Bathe & Dvorkin 1985), (Bathe & Dvorkin 1986), (Gruttmann & Wagner 2004), (Cardoso & Jeong-Whan Yoon 2006), (Cardoso & Jeong-Whan Yoon 2005) and (Cardoso et al. 2006)). Stabilization methods such as EAS or the Assumed Natural Strain (ANS) consist in adding, in the deformation field, a field of internal variables which creates additional modes of deformation as presented e.g. in (J.C Simo & Rifai 1990). However, these stabilization methods increase the size of the stiffness matrix, decreasing at the same time the computational efficiency. Hence several authors (see, e.g. (Belytschko & Bindeman 1993), (Puso 2000), (Legay & Alain Combescure 2003) and (Reese 2005)) have worked on the problem to transfer the enhanced strain method into finite element formulations based on reduced integration with hourglass stabilization. However, in the context of integrated computer aided solid modeling and finite element analysis of real-life structures, the coexistence of three-dimensional and structural zones is quite common, and both types of elements must be used simultaneously. Then in order to avoid both arbitrary definitions of separation zones (e.g. continuum/structural) and the intricacies of connecting different types of elements (e.g. shell/continuum), elements that behave well in both continuum and structural applications considerably simplify the modeling of such structures. Hence much effort has been devoted to the development of continuum-based shell elements because they offer many advantages: the use of general three-dimensional constitutive models, the avoidance of complex shell-type kinematics, the direct calculation of thickness (strain) variations, easy treatment of large rotations along with simple updating of configurations, straightforward connection with three-dimensional elements since displacements are the only degrees of freedom, and natural contact conditions on both sides of the structure. However, the performance of solid shell elements deteriorates rapidly as the thickness becomes smaller, due to the locking phenomena. Consequently, the development of accurate, stable and robust solid-shell elements becomes more challenging and demanding than that of the degenerated shell elements. Most of the methods developed earlier were based on EAS fields, and consisted in either using a conventional integration scheme with appropriate control of all locking phenomena or in the application of reduced integration with hourglass control. Both approaches have been extensively investigated and evaluated in various structural applications, as reported in the work of (Dvorkin & Bathe 1984) and (Belytschko & Bindeman 1993). (Hauptmann et al. 2001) developed a 'solid-shell' element with only displacement degree of freedoms for linear and non-linear analysis. The locking free behavior was obtained by employing the ANS and the EAS methods and no stabilization is required. A resultant eight-node solid shell element having only displacement degrees of freedom proposed by (Kim et al. 2005)(Sze & Yao 2000) by extending the classical resultant shell theory, presented by (J. C. Simo et al. 1993). In-plane membrane and bending behavior is fully integrated and to avoid shear locking, the commonly used ANS scheme is to truncate and interpolate the natural transverse shear strains evaluated at mid-edge of the Q4 element defined in the shell mid-surface. To avoid trapezoidal locking, the commonly used ANS scheme is to interpolate the thickness strain at the four corner node on the shell mid-surface. Several eight-node solid shell elements with reduced integration and hourglass stabilization have been proposed amongst which the non linear SHB8PS shell element presented in (Abed-Meraim & A. Combescure 2002), (Legay & Alain Combescure 2003) and (Desroches 2005) and the eight-node shell element presented in (Sousa et al. 2006) in linear and (Sousa et al. 2005) in non linear cases can be mentioned. In line with the current trends of multiscale, coupled mechanical problems, the ever-increasing demands of non-linear applications have brought new challenges for finite element

development. Finite strain, bending-dominated problems are quite common, inducing locking in most low-order continuum-type elements, together with high mesh distortion. Solutions to these problems should be found while maintaining low-order integration due to efficiency requirements as well as compatibility with contact algorithms. Moreover with elastic-plastic material models encountered e.g. in metal forming processes, associated incompressibility problems also contributes to undesirable locking phenomena. All these issues have motivated the recent development of finite element technology combining the advantages of both solid and shell elements. One such recent development is the smoothed finite element method (SFEM) that was first introduced in (G. R. Liu et al. 2006), (G. R. Liu et al. 2007) and then further developed by (N. Nguyen-Thanh et al. 2008). This method is rooted in mesh free stabilized conforming nodal integration (Chen et al. 2001) and is based on the gradient (strain) smoothing technique. It was shown to provide a suite of finite elements with a range of interesting properties that depend on the number of smoothing cells employed within each finite element (see (S. P. A. Bordas & Natarajan 2009) for a review of recent developments and properties) and include (i) Improved dual accuracy and super convergence; (ii) Softer than the FEM; (iii) Relative insensitivity to volumetric locking and (iv) Relative insensitivity to mesh distortion. These advantages are well illustrated in (K. Y. Dai & G.R. Liu 2006), (K.Y. Dai & G.R. Liu 2007), (T. Nguyen-Thanh et al. 2007), (Nguyen-Xuan et al. 2008) and (S. P.A. Bordas et al. 2009). In SFEM techniques, each element is divided into a number of smoothing cells and the use of divergence theorem then allows the integration to be transferred on the cell boundary. Its essential feature is that no isoparametric mapping is required, which implies that the approximation can be defined in the physical space directly, thereby providing freedom in the selection of the element geometry. SFEM has already been applied to plate and flat shell element mostly in the small strain range and for degenerated shell element (N. Nguyen-Thanh et al. 2008). In this paper the SFEM formulation is applied to the resultant eight-node solid shell element as described in (Kim et al. 2005) for geometrically nonlinear formulation based on the updated Lagrangian formulation. In this preliminary development phase, the SFEM technique is applied only to the membrane and bending effects while ANS technique is still applied to avoid locking associated with trapezoidal and transverse shear effects. Moreover, only implicit formulation has been tasted. The new formulation has been implemented as Abaqus/User Element and its preliminary capabilities are assessed through various typical benchmark problems.

2. Geometric description

In order to represent a shell like structure by solid shell element we introduce the standard hexahedral eight-node isoparametric element. In addition to the standard solid element parameterization comprising the intrinsic coordinates (ξ, η, ζ) and the global Cartesian coordinates (X, Y, Z) , we introduce a set of co-rotational local orthogonal coordinate systems (r, s, t) , which is set up in the element mid-surface as in (Kim et al. 2005). The eight-node solid-shell element shown in Fig. 1 can thus be described by the relation between global (X, Y, Z) , local (r, s, t) and natural co-ordinates (ξ, η, ζ) . In global coordinates, the shell geometry and kinematics i.e. the position and displacement of any point within the element can be defined as:

$$\bar{\mathbf{X}}_q(\xi, \eta, \zeta) = \sum_{i=1}^4 N_i(\xi, \eta) \cdot ((1+\zeta) \cdot \mathbf{X}_p^U + (1-\zeta) \cdot \mathbf{X}_p^L), \quad (-1 \leq \xi, \eta, \zeta \leq 1) \quad (2.1)$$

$$U_q(\xi, \eta, \zeta) = \sum_{i=1}^4 N_i(\xi, \eta) \cdot ((1+\zeta) \cdot u_p^U + (1-\zeta) \cdot u_p^L), \quad (-1 \leq \zeta \leq 1) \quad (2.2)$$

In the previous equations, $\mathbf{X}_p^L$ and $\mathbf{X}_p^U$ are respectively the position of the bottom and the top surface nodes in the global Cartesian coordinate basis (X, Y, Z) and, u_p^L and u_p^U are respectively the displacement of the bottom and the top surface nodes in the global Cartesian coordinate.

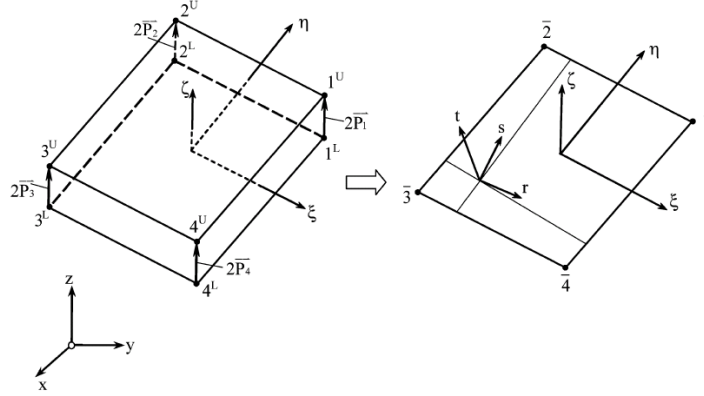


Figure 1: Eight-node shell element (Kim et al. 2005)

In the resultant solid shell theory, equations (2.1) and (2.2) can be written as

$$\bar{\mathbf{X}}_q(\xi, \eta, \zeta) = \sum_{i=1}^4 N_i(\xi, \eta) \cdot (\bar{\bar{\mathbf{X}}}_{pi} + \zeta \cdot \Delta \bar{\mathbf{X}}_i), \quad (-1 \leq \zeta \leq 1) \quad (2.3)$$

$$\bar{\mathbf{u}}_q(\xi, \eta, \zeta) = \bar{\bar{\mathbf{u}}}(\xi, \eta) + \zeta \cdot \Delta \bar{\mathbf{u}}(\xi, \eta), \quad (-1 \leq \zeta \leq 1) \quad (2.4)$$

Where $N_i(\xi, \eta)_{i=1,4}$ are the standard bilinear shape functions of a Q4 element as presented in (Batoz & Gouri 1992). $\bar{\bar{\mathbf{X}}}_{pi} = \frac{\bar{\mathbf{X}}_{i+4} + \bar{\mathbf{X}}_i}{2}$ are the position vectors of the mid-surface and $\Delta \bar{\mathbf{X}}_i = \frac{\bar{\mathbf{X}}_{i+4} - \bar{\mathbf{X}}_i}{2}$ are the nodal vectors pointing from lower to upper surface. In the previous equations, $\bar{\mathbf{X}}_{i,i=1,4}$ and $\bar{\mathbf{X}}_{i+4,i=1,4}$ are the position vectors of respectively the bottom and the top surface nodes in the local Cartesian coordinate basis $(\mathbf{r}, \mathbf{s}, \mathbf{t})$. Similar definition hold for $\bar{\bar{\mathbf{u}}}$ and $\Delta \bar{\mathbf{u}}$.

3. Kinematics of shell deformation

Adopting a total or updated lagrangian formulation, starting from the deformation gradient $\vec{F} = \vec{1} + \vec{\nabla}_x u$ or in component form $F_{i\alpha} = \delta_{i\alpha} + \frac{\partial u_i}{\partial X_\alpha} = \frac{\partial x_i}{\partial X_\alpha}$, the natural Green-Lagrange strain can be written as

$$\vec{E} = \frac{1}{2}(\vec{F}^T \vec{F} - \vec{I}) = \frac{1}{2} \left\{ \left(\vec{\nabla}_x u \right) + \left(\vec{\nabla}_x u \right)^T + \left(\vec{\nabla}_x u \right)^T \cdot \left(\vec{\nabla}_x u \right) \right\} \quad (3.1)$$

In local Cartesian components (corotational frame), equation (3.1) can then be written as

$$\varepsilon_{ij} = \frac{1}{2} \cdot \left[\underbrace{\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)}_{\text{Linear}} + \underbrace{\frac{\partial \bar{u}_k}{\partial x_j} \times \frac{\partial \bar{u}_k}{\partial x_i}}_{\text{Non linear}} \right] + \zeta \cdot \left[\underbrace{\left(\frac{\partial \Delta u_i}{\partial x_j} + \frac{\partial \Delta u_j}{\partial x_i} \right)}_{\text{Linear}} + \underbrace{\frac{\partial \Delta u_k}{\partial x_j} \times \frac{\partial \Delta u_k}{\partial x_i}}_{\text{Non linear}} \right] \quad (3.2)$$

The ANS technique is used to interpolate the transverse shear and the thickness strain fields (Hannachi 2007).

4. Smoothed strain field FEM formulation

In SFEM, the smoothed strain is obtained using a strain smoothing operation defined in (K.Y. Dai & G.R. Liu 2007) or (N. Nguyen-Thanh et al. 2008):

$$\tilde{\varepsilon}_{ij}(x_c) = \int_{\Omega_c} \varepsilon_{ij}(x) \cdot \varphi(x - x_c) d\Omega \quad (4.1)$$

Where φ is a smoothing function with $\varphi(x - x_c) = \begin{cases} 1/A_c & x \in \Omega_c \\ 0 & x \notin \Omega_c \end{cases}$ and $A_c = \int_{\Omega_c} d\Omega$ the area of

the smoothing cell Ω_c . This operation is very similar to the mean dilatation procedure used to deal with the incompressibility in non linear mechanics and it has been used in weak-form mesh less method based on nodal integration. By applying the divergence theorem, the smoothed strain formulation is then expressed as an integration of the strain matrix on the cell boundary. Following equation (3.1) the smoothed Green-Lagrange strain can be obtained from the smoothed deformation gradient which can be written as

$$\tilde{F}_{ij}(\mathbf{X}_c) = \frac{1}{A_c^x} \int_{\Omega_c^x} \left[\frac{\partial u_i^h}{\partial X_j} + \delta_{ij} \right] d\Omega = \tilde{e}_{ij}(\mathbf{X}_c) + \delta_{ij} \quad (4.2)$$

Where

$$\tilde{e}_{ij}(\mathbf{X}_c) = \frac{1}{A_c^x} \int_{\Gamma_c^x} (u_i^h \cdot n_j) d\Gamma = \sum_I \tilde{b}_{ji}^c d_{il} \quad (4.3)$$

With $\tilde{b}_{ji}^c = \frac{1}{A_c^x} \int_{\Gamma_c^x} (N_i \cdot n_j) d\Gamma$.

Hence the Green strain E can be obtained using the smoothed deformation gradient as

$$\vec{\vec{E}}_{ij} = \frac{1}{2} (\tilde{F}_{ik} \tilde{F}_{kj} - I_{ij}) = \frac{1}{2} [(\tilde{e}_{ik} + \delta_{ik})(\tilde{e}_{kj} + \delta_{kj}) - \delta_{ij}] \quad (4.4)$$

Or after simplification

$$\vec{\vec{E}}_{ij} = \frac{1}{2} [\tilde{e}_{ij} + \tilde{e}_{ji} + \tilde{e}_{ki} \tilde{e}_{kj}] \quad (4.5)$$

Introducing the local displacement field $\bar{\mathbf{u}}_q(\xi, \eta, \zeta) = \bar{\bar{\mathbf{u}}}(\xi, \eta) + \zeta \cdot \Delta \bar{\mathbf{u}}(\xi, \eta)$, then we can write:

$$\tilde{e}_{ij}(\mathbf{X}_c) = \frac{1}{A_c^x} \int_{\Gamma_c^x} (\bar{u}_i^h \cdot n_j + \zeta \Delta u_i^h n_j) d\Gamma = \sum_I (\tilde{b}_{ji}^c + \zeta \Delta \tilde{b}_{ji}^c) d_{il} \quad (4.6)$$

Separating the linear and non linear strain components, the linear part becomes

$$\vec{\vec{E}}_{ij}^L = \frac{1}{2} [\tilde{e}_{ij} + \tilde{e}_{ji}] = [\tilde{\mathbf{B}}_{il} + \zeta \cdot \Delta \tilde{\mathbf{B}}_{il}] \cdot \{\hat{u}_n\} \quad (4.7)$$

Where $\tilde{\mathbf{B}}_{il}$ is a smoothed strain operator defined as in equation (5.3).

The smoothing technique integration is here applied to the membrane and bending parts of the strain field thus resulting in integration in the mid-surface boundary for the resultant eight-node solid shell element. Hence, separating into membrane-bending, transverse shear and transverse normal components, we can write

$$\tilde{E}_{ij}^L = \tilde{E}_{ij}^{mbL} + \tilde{E}_{ij}^{sL} + \tilde{E}_{ij}^{zL} \quad (4.8)$$

Where $\tilde{E}_{ij}^{mbL} = [\tilde{\mathbf{B}}_{il}^m + \zeta \cdot \Delta \tilde{\mathbf{B}}_{il}^b] \cdot \{\hat{u}_n\}$.

The non linear part of the smoothed strain field can thus be written as:

$$\begin{aligned} \vec{\vec{E}}_{ij}^{NL} &= \frac{1}{2} [\tilde{e}_{ik} \cdot \tilde{e}_{kj}] \\ &= \sum_I \langle d_{il} \rangle \cdot \left(\frac{1}{2} \cdot (\tilde{b}_{kl}^c \cdot \tilde{b}_{kl}^{cT} + \zeta \cdot (\Delta \tilde{b}_{kl}^c \cdot \tilde{b}_{kl}^{cT} + \tilde{b}_{kl}^c \cdot \Delta \tilde{b}_{kl}^{cT}) + \zeta^2 \cdot \Delta \tilde{b}_{kl}^c \cdot \Delta \tilde{b}_{kl}^{cT}) \right) \cdot d_{jl} \end{aligned} \quad (4.9)$$

Which becomes after introducing the classical A and G operator [(Stegmann & Lund 2001)],

$$\tilde{E}_{ij, NL} = \frac{1}{2} \cdot \tilde{\mathbf{B}}_{ij, NL} \cdot \mathbf{u} = \frac{1}{2} \cdot \tilde{\mathbf{A}} \cdot \tilde{\mathbf{G}} \cdot \mathbf{u}$$

So that the membrane part can be written as:

$$\tilde{E}_{ij, NL}^{mb} = \frac{1}{2} \cdot \tilde{\mathbf{B}}_{ij, NL}^{mb} \cdot \mathbf{u} = \frac{1}{2} \cdot \tilde{\mathbf{A}}^{mb} \cdot \tilde{\mathbf{G}} \cdot \mathbf{u} \quad (4.10)$$

5. Tangent stiffness matrix

5.1 Material tangent stiffness matrix

The membrane and bending smoothed tangent material stiffness component is can be written as

$$\begin{aligned} \tilde{\mathbf{K}}_{T, mat, L+NL}^{mb} &= \sum_{c=1}^{n_c} \left[\left(\tilde{\mathbf{B}}_{c_i, L+NL}^{mb, 0} (x_c) \right)^T \cdot \mathbf{A} \mathbf{H} \cdot \left(\tilde{\mathbf{B}}_{c_i, L+NL}^{mb, 0} (x_c) \right) + \dots \right. \\ &= \dots + \left. \left(\tilde{\mathbf{B}}_{c_i, L+NL}^{mb, w} (x_c) \right)^T \cdot \mathbf{D} \mathbf{H} \cdot \left(\tilde{\mathbf{B}}_{c_i, L+NL}^{mb, w} (x_c) \right) \right] \cdot (A_c) \end{aligned} \quad (5.1)$$

The expression of the strain matrix is the sum of the linear and the non linear parts

$$\tilde{\mathbf{B}}_{c_i, L+NL}^{mb} (x_c) = \tilde{\mathbf{B}}_{c_i, L}^{mb} (x_c) + \tilde{\mathbf{B}}_{c_i, NL}^{mb} (x_c) \quad (5.2)$$

Where the linear strain matrices are

$$\tilde{\mathbf{B}}_{c_i, L}^{mb, 0} (x_c) = \frac{1}{A_c} \cdot \sum_{b=1}^{nb} \begin{bmatrix} N_i(x^G) \cdot n_r & 0 & 0 & 0 & 0 & 0 \\ 0 & N_i(x^G) \cdot n_s & 0 & 0 & 0 & 0 \\ N_i(x^G) \cdot n_s & N_i(x^G) \cdot n_r & 0 & 0 & 0 & 0 \end{bmatrix} \cdot l_b^c \quad (5.3)$$

And

$$\tilde{\mathbf{B}}_{c_i, L}^{mb, w} (x_c) = \frac{1}{A_c} \cdot \sum_{b=1}^{nb} \begin{bmatrix} 0 & 0 & 0 & N_i(x^G) \cdot n_r & 0 & 0 \\ 0 & 0 & 0 & 0 & N_i(x^G) \cdot n_s & 0 \\ 0 & 0 & 0 & N_i(x^G) \cdot n_s & N_i(x^G) \cdot n_r & 0 \end{bmatrix} \cdot l_b^c \quad (5.4)$$

With i is the mid-surface node label from 1 to 4; x^G is the gauss point and l_b^c the length of Γ_b^c ; n_r and n_s are the components of the sub cell border outward normal vectors; A_c is the area of the c sub cell; N_i , the studied field corner shape functions; b is the mid-point label of cell c going from 1 to 4; and $\mathbf{A} \mathbf{H}$ and $\mathbf{D} \mathbf{H}$ are constitutive matrices given in (Kim et al. 2005).

And where the non linear matrices are

$$\tilde{\mathbf{B}}_{c_i, NL}^{mb,0}(\chi_c) = \tilde{\mathbf{A}}_c^{mb} \cdot \tilde{\mathbf{G}}_c^0 \quad (5.5)$$

$$\tilde{\mathbf{B}}_{c_i, NL}^{mb,w}(\chi_c) = \tilde{\mathbf{A}}_c^{mb} \cdot \tilde{\mathbf{G}}_c^w \quad (5.6)$$

With $\tilde{\mathbf{A}}_c^{mb}$ and $\tilde{\mathbf{G}}_c$ are the derivative matrices introduced in equation (4.10).

5.2 Geometric tangent stiffness matrix

As presented by (Stegmann & Lund 2001), the membrane and bending part of the geometric stiffness matrix is,

$$\mathbf{k}_\sigma^{mb} = \int_V \mathbf{G}^T \cdot \mathbf{H}_\sigma^{mb} \cdot \mathbf{G} dV \quad (5.7)$$

Where $\mathbf{H}_\sigma^{mb}$ is the membrane and bending part of $\mathbf{H}_\sigma$ initial stress matrix defined as

$$\mathbf{H}_\sigma^{mb} = \begin{bmatrix} S_{11} \cdot \mathbf{I} & S_{12} \cdot \mathbf{I} & 0 & 0 \\ S_{12} \cdot \mathbf{I} & S_{22} \cdot \mathbf{I} & 0 & 0 \\ 0 & 0 & S_{44} \cdot \mathbf{I} & S_{45} \cdot \mathbf{I} \\ 0 & 0 & S_{45} \cdot \mathbf{I} & S_{55} \cdot \mathbf{I} \end{bmatrix} \quad (5.8)$$

Where

$\mathbf{S} = \{S_{11}, S_{22}, S_{12}, S_{44}, S_{55}, S_{45}\}$ are the stress matrix components expressed in the local Cartesian basis.

$\mathbf{I} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$ is the identity matrix.

6. Benchmark problems

The present smoothed solid shell element has been implemented in Abaqus/Standard using a User element facilities and was programmed in Fortran. For paper understanding, let's call our eight-node solid shell element SH8-Mist1, SH8-Mist2 and SH8-Mist4 as in (G. R. Liu et al. 2006) depending on the number of smoothing cells dividing its mid-surface, respectively 1, 2 and 4. The capabilities of this element are compared with other eight-node elements while dealing with standards patch tests. The mesh distortion used in some cases is obtained using the same technique as the one proposed in (G. R. Liu et al. 2006). In the following benchmark problems graphics, the reference value corresponds to the normalized exact solution of the problem.

6.1 Cook's membrane problem

In this example, a flat skew plate is clamped on its left side and subjected to a shear load on the right side as presented in figure 2(a). This standard example is usually used to observe the membrane capability of an element. The exact reference vertical displacement of point C is

$U_y = 23.91$. Even though ABAQUS element SC8R and our element (still underdevelopment) do not use the same theory, they are however compared, in order to see how accurate the new SFEM element is.

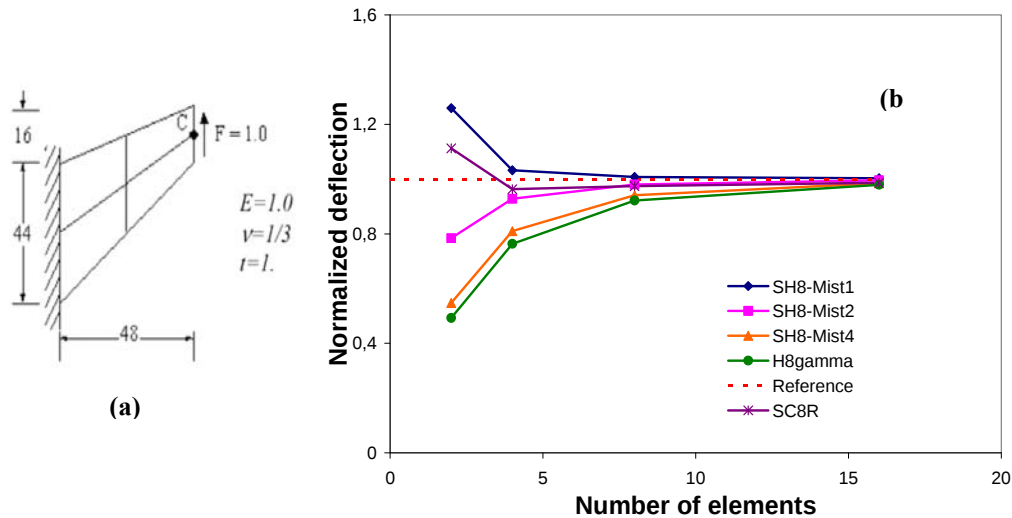


Figure 2: Cook's membrane (a) problem (Kim et al. 2005) and (b) results.

Good results for the presented element are observed. Besides it appeared that the SC8R element gave similar results as exposed in figure 2(b) and table 1.

Element size	Normalized solutions				
	H8 gamma	SC8R	SH8-Mist1	SH8-Mist2	SH8-Mist4
2*2*1	0,493	1,112	1,259	0,784	0,547
4*4*1	0,763	0,963	1,032	0,928	0,810
8*8*1	0,922	0,974	1,008	0,980	0,941
16*16*1	0,979	0,987	1,003	0,995	0,985

Table 1. Cook membrane problem results.

6.2 Scordellis-lo roof problem

In the Scordellis-Lo roof problem a curved area is clamped on the two curved edges and free on the two straight one as resented in figure 3(a). This standard problem is very useful to verify the membrane and bending behaviour of an element. The vertical displacement of point A is observed and compared to the reference vertical displacement of $U_y = 0.3024$.

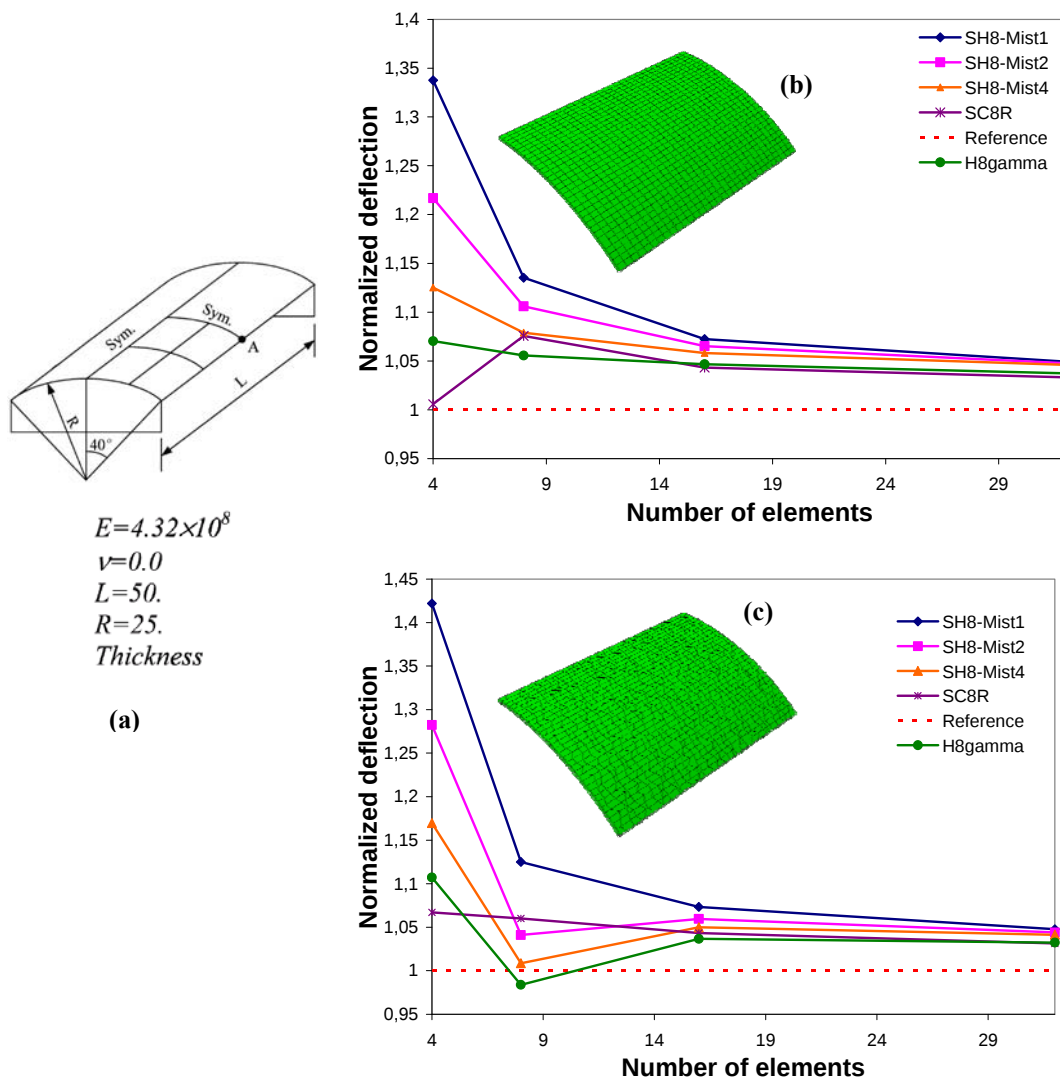


Figure 3: Scordellis-lo roof problem (Kim et al. 2005) (a) and results, (b) regular and (c) distorted mesh.

The presented element gave comparable results as the SC8R for a thin regular or distorted mesh. However its results seemed to be inaccurate for a coarse mesh. Nevertheless the SH8-MIST elements convergences were better than the original H8 gamma element in the case of a distorted mesh as shown in table 2 and figures 3(b) and 3(c).

Element size	Normalized solutions, regular mesh (a)				
	H8 gamma	SC8R	SH8-Mist1	SH8-Mist2	SH8-Mist4
4*4*1	1,07	1,006	1,338	1,217	1,125
8*8*1	1,056	1,076	1,135	1,106	1,079
16*16*1	1,047	1,043	1,072	1,065	1,058
32*32*1	1,037	1,033	1,049	1,048	1,046

Element size	Normalized solutions, distorted mesh (b)				
	H8 gamma	SC8R	SH8-Mist1	SH8-Mist2	SH8-Mist4
4*4*1	1,112	1,067	1,422	1,282	1,17
8*8*1	0,985	1,060	1,125	1,041	1,009
16*16*1	1,037	1,043	1,073	1,06	1,05
32*32*1	1,032	1,032	1,048	1,044	1,041

Table 2. Scordellis-lo roof problem results, (a) regular and (b) distorted mesh.

6.3 Pinched cylinder with end diaphragm problem

A cylinder with rigid diaphragms at both ends subjected to a concentrated load at its center top surface A is considered. This standard example is generally used to evaluate a shell element capability in term of unintentional bending modes and complex membrane states as presented in figure 4(a). The vertical displacement of point A is observed and its exact value is $U_y = 1.8248 e^{-5}$.

The presented element gave similar results as the SC8R element with a comparable precision for a coarse as well as a refined mesh as shown in table 3 and figures 4(b) and 4(c).

Element size	Normalized solutions, regular mesh (a)				
	H8 gamma	SC8R	SH8-Mist1	SH8-Mist2	SH8-Mist4
4*4*1	0,407	0,453	0,541	0,441	0,417
8*8*1	0,774	0,785	0,838	0,807	0,782
16*16*1	0,953	0,946	0,979	0,970	0,957
32*32*1	1,011	0,996	1,019	1,017	1,012

Element size	Normalized solutions, distorted mesh (b)				
	H8 gamma	SC8R	SH8-Mist1	SH8-Mist2	SH8-Mist4
4*4*1	0,407	0,459	0,591	0,445	0,418
8*8*1	0,764	0,784	0,836	0,798	0,773
16*16*1	0,953	0,953	0,991	0,976	0,958
32*32*1	1,010	1,000	1,019	1,016	1,012

Table 3. Pinched cylinder problem results, (a) regular and (b) distorted mesh.

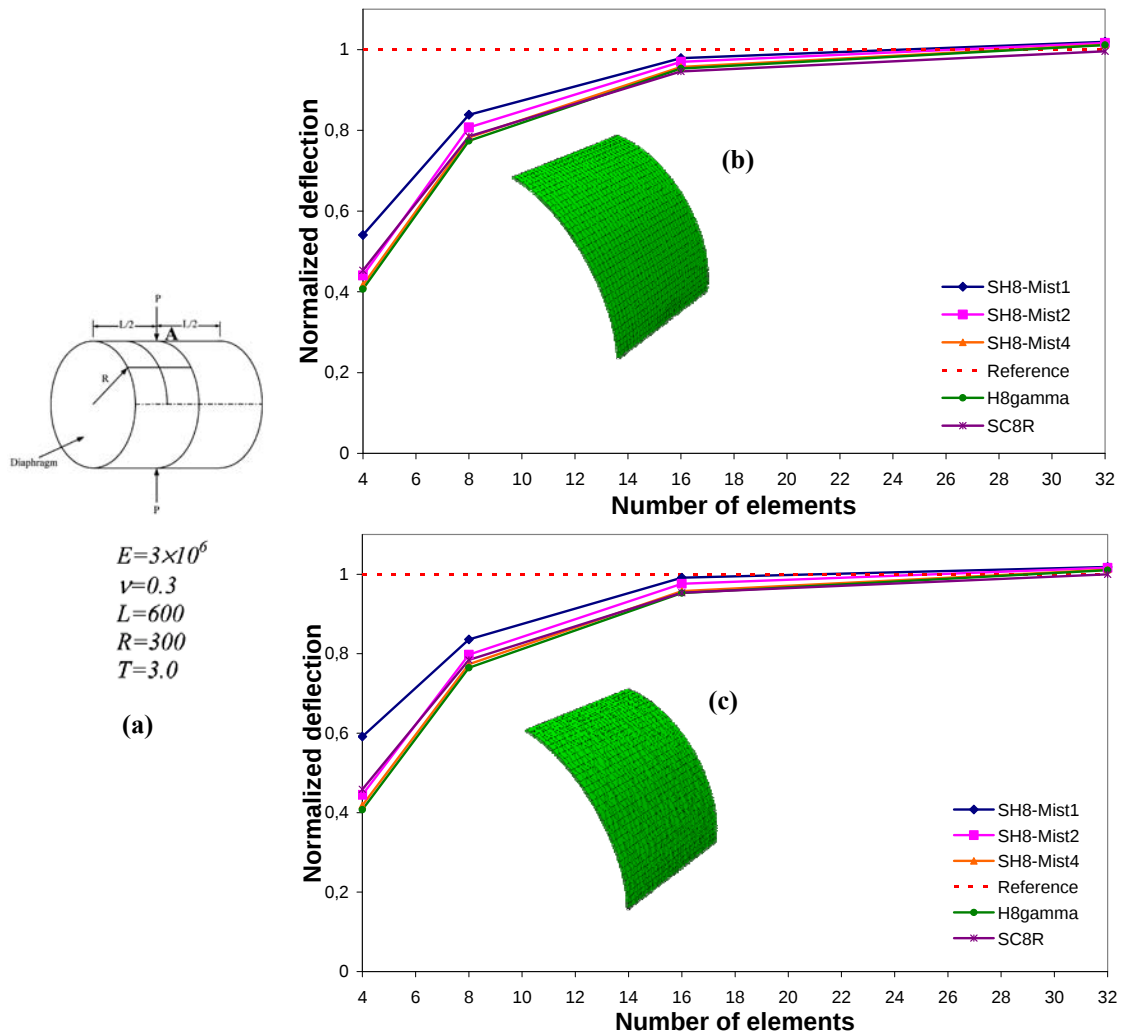


Figure 4: Pinched cylinder with end diaphragm problem (Kim et al. 2005) (a) and results, (b) regular and (c) distorted mesh.

6.4 Pinched cylinder with end diaphragm problem overloaded

The same cylinder as in the previous section is used. However, the concentrated load at its center top surface is increased up to $8e^5$ to consider the non linear aspect on this problem. Some results (not normalized) with the mesh density 32×32 are presented in fig. 5(a) and a picture deformation is presented in figure 5(b). It then appeared that SH8-Mist elements handle geometric non linearities as well as the H8-Gamma and SC8R elements.

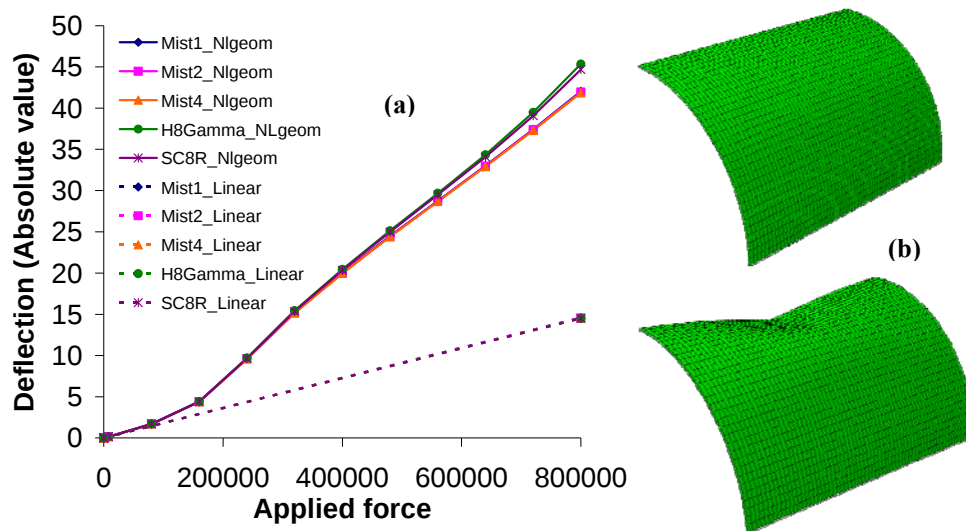


Figure 5: Pinched cylinder problem, non linear aspect (load = $8e^5$) results (a) and initial and deformed configurations (b).

7. Conclusions

An eight-node solid-shell element based on the resultant stress formulation has been successfully implemented on Abaqus/User element using the smoothing technique to calculate the membrane and bending stiffness matrix. One of the interests of this method is that all the degrees of freedom are displacements which simplify the formulation. Another one is the fact that the membrane and bending parts of the stiffness matrix are calculated on the borders of the element mid-surface which does not require any shape function derivative calculations on the one hand and which makes the element accurate even with a distorted mesh. Further work is underway in order to extend the SFEM techniques in the dynamics range.

8. References

1. Abed-Meraim, F. & Combescure, A., 2002. SHB8PS--a new adaptative, assumed-strain continuum mechanics shell element for impact analysis. *Computers & Structures*, 80(9-10), p.791-803.
2. Bathe, K. & Dvorkin, E.N., 1986. A formulation of general shell elements—the use of mixed interpolation of tensorial components. *International Journal for Numerical Methods in Engineering*, 22(3), p.697-722.

3. Bathe, K. & Dvorkin, E.N., 1985. A four-node plate bending element based on Mindlin/Reissner plate theory and a mixed interpolation. *International Journal for Numerical Methods in Engineering*, 21(2), p.367-383.
4. Batoz, J. & Gouri, D., 1992. *Modélisation des structures par éléments finis: coques* Les Presses de l'Université Laval,.
5. Belytschko, T. & Bindeman, L.P., 1993. Assumed strain stabilization of the eight node hexahedral element. *Computer Methods in Applied Mechanics and Engineering*, 105(2), p.225-260.
6. Bordas, S.P.A. & Natarajan, S., 2009. On the approximation in the smoothed finite element method (SFEM). *International Journal for Numerical Methods in Engineering*, p.n/a-n/a.
7. Bordas, S.P. et al., 2009. Strain smoothing in FEM and XFEM. *Computers & Structures*, In Press, Corrected Proof.
8. Cardoso, R.P.R., Yoon, J.W. & Valente, R.A.F., 2006. A new approach to reduce membrane and transverse shear locking for one-point quadrature shell elements: linear formulation. *International Journal for Numerical Methods in Engineering*, 66(2), p.214-249.
9. Cardoso, R.P.R. & Yoon, J., 2005. One point quadrature shell elements for sheet metal forming analysis. *Archives of Computational Methods in Engineering*, 12(1), p.3-66.
10. Cardoso, R.P.R. & Yoon, J., 2006. One point quadrature shell elements: a study on convergence and patch tests. *Computational Mechanics*, 40(5), p.871-883.
11. Chen, J. et al., 2001. A stabilized conforming nodal integration for Galerkin mesh-free methods. *International Journal for Numerical Methods in Engineering*, 50(2), p.435-466.
12. Dai, K.Y. & Liu, G., 2006. *Smoothed Finite Element Method*.
13. Dai, K. & Liu, G., 2007. Free and forced vibration analysis using the smoothed finite element method (SFEM). *Journal of Sound and Vibration*, 301(3-5), p.803-820.
14. Desroches, X., 2005. *Éléments finis de coques volumiques*, EDF-R&D/AMA.
15. Dvorkin, E.N. & Bathe, K., 1984. A continuum mechanics based four-node shell element for general non-linear analysis. *Engineering Computations*, 1(1), p.77-88.
16. Gruttmann, F. & Wagner, W., 2004. A stabilized one-point integrated quadrilateral Reissner-Mindlin plate element. *IJNME*, 61(13), p.2273-2295.
17. Hannachi, M., 2007. *Formulation d'éléments finis volumiques adaptés à l'analyse, linéaire et non linéaire, et à l'optimisation de coques isotropes et composites*. Compiègne: UTC.
18. Hauptmann, R. et al., 2001. Solid-shell elements with linear and quadratic shape functions at large deformations with nearly incompressible materials. *Computers & Structures*, 79(18), p.1671-1685.
19. Kim, K.D., Liu, G.Z. & Han, S.C., 2005. A resultant 8-node solid-shell element for geometrically nonlinear analysis. *Computational Mechanics*, 35(5), p.315-331.
20. Legay, A. & Combescure, A., 2003. Elastoplastic stability analysis of shells using the physically stabilized finite element SHB8PS. *International Journal for Numerical Methods in Engineering*, 57(9), p.1299-1322.

21. Liu, G.R., Dai, K.Y. & Nguyen, T.T., 2006. A Smoothed Finite Element Method for Mechanics Problems. *Computational Mechanics*, 39(6), p.859-877.
22. Liu, G.R. et al., 2007. Theoretical aspects of the smoothed finite element method (SFEM). *International Journal for Numerical Methods in Engineering*, 71(8), p.902-930.
23. Nguyen-Thanh, N. et al., 2008. A smoothed finite element method for shell analysis. *Computer Methods in Applied Mechanics and Engineering*, p.165–177.
24. Nguyen-Thanh, T. et al., 2007. Selective Smoothed Finite Element Method. *Tsinghua Science & Technology*, 12(5), p.497-508.
25. Nguyen-Xuan, H., Bordas, S. & Nguyen-Dang, H., 2008. Smooth finite element methods: Convergence, accuracy and properties. *International Journal for Numerical Methods in Engineering*, 74(2), p.175-208.
26. Puso, M.A., 2000. A highly efficient enhanced assumed strain physically stabilized hexahedral element. *International Journal for Numerical Methods in Engineering*, 49(8), p.1029-1064.
27. Reese, S., 2005. On a physically stabilized one point finite element formulation for three-dimensional finite elasto-plasticity. *Computer Methods in Applied Mechanics and Engineering*, 194(45-47), p.4685-4715.
28. Simo, J.C., Armero, F. & Taylor, R.L., 1993. Improved versions of assumed enhanced strain tri-linear elements for 3D finite deformation problems. *Computer Methods in Applied Mechanics and Engineering*, 110(3-4), p.359-386.
29. Simo, J. & Rifai, M., 1990. A class of mixed assumed strain methods and the method of incompatible modes. , 29, p.1595-1638.
30. Sousa, R.J.A.D. et al., 2006. A new one-point quadrature enhanced assumed strain (EAS) solid-shell element with multiple integration points along thickness - part II: nonlinear applications. *International Journal for Numerical Methods in Engineering*, 67(2), p.160-188.
31. Sousa, R.J.A.D. et al., 2005. A new one-point quadrature enhanced assumed strain (EAS) solid-shell element with multiple integration points along thickness: Part I - geometrically linear applications. *International Journal for Numerical Methods in Engineering*, 62(7), p.952-977.
32. Stegmann, J. & Lund, E., 2001. Notes on structural analysis of composite shell structures, Aalborg University, Institute of Mechanical Engineering.
33. Sze, K.Y. & Yao, L., 2000. A hybrid-stress ANS solid-shell element and its generalization for smart structure modelling - part I: solid-shell element formulation. *International Journal for Numerical Methods in Engineering*, 48, p.545-564.

9. Acknowledgment

The authors would like to thank the following institutions for their support: the department of mechanical engineering of Laval University, the Consortium for Research and Innovation in Aerospace in Quebec (CRIAQ), the Martinique Regional Council associated with the European Social fund and the Aluminum Research Center (REGAL).

2011 SIMULIA Customer Conference