Shell Failure Simulation Using Master-Slave and Penalty Methods

Dalia Čalnerytė and Rimantas Barauskas

Kaunas University of Technology, Faculty of Informatics, Kaunas, Lithuania
dalia.calneryte@ktu.edu, rimantas.barauskas@ktu.lt

Abstract. Failure simulation for the flat structure assembled of the degenerated shell elements and referred as unidirectional composite is considered in this paper. Master-Slave and penalty methods are implemented in order to describe connections between elements. The results are compared for 2D simulation by both methods if structure is loaded in longitudinal and transverse directions.

Keywords: Degenerated shell element, failure criteria, Master-Slave method, penalty method.

1 Introduction

Finite element method has been used for linear elasticity problems since the origins of the finite element analysis. Nowadays it is also applied for non-linear elasticity problems and failure simulation.

The principal problems of nonlinear shell analysis are damage simulation and delamination. A zero-thickness rigid bar connecting master and slave nodes which belong to different layers of the shell is used in [7] to simulate delamination. Similar approach is used to simulate damage evolution in this article.

The general theory of element failure criteria is described in [10]. Failure criteria used for fiber reinforced unidirectional composites are summarized in [5]. Failure criteria varies from simple forms such as maximum stress (element fails if stress in the principal material direction reaches critical value), maximum strain (element fails if strain reaches critical value) to more complex criteria such as Tsai-Wu or Hashin-Rotem. Same criteria can be used to simulate failure of the layer and delamination between layers. One of the most popular criteria used in simulation is the Hashin criteria used in this paper. This criteria is used in [9] to predict damage evolution properties and failure strengths of composite laminates. The same criteria is also used in [11] to predict layer failure with additional component for delamination of the layers.

Two methods used for connecting shell elements are compared in this work. Flat structure is divided to degenerated shell elements. Each element has 4 individual nodes. The elements are connected by fictitious bar elements. In this work, we use two different methods to constrain connections between elements. Master/Slave and penalty methods are implemented and results for longitudinal and transverse loads are compared. The orthotropic material corresponding unidirectional fiber reinforced

G. Dregvaite and R. Damasevicius (Eds.): ICIST 2014, CCIS 465, pp. 408–418, 2014.

composite in mezzo scale is used in simulation. Total Lagrangian formulation is employed in dynamic analysis where all variables are referred to the initial configuration of the finite element model.

2 Explicit Solution

The global system of discretized equations of motion at the nth time step is given by [9]:

$$\mathbf{M}\ddot{\mathbf{u}}_n + \mathbf{C}\dot{\mathbf{u}}_n + \mathbf{K} \cdot \mathbf{u}_n = \mathbf{R}_n \tag{1}$$

Where $\mathbf{M}$ is the mass matrix, the damping matrix $\mathbf{C} = \alpha\mathbf{M}$, α – damping constant, $\mathbf{K}$ is the tangential stiffness matrix, $\mathbf{u}_n$ is the displacement vector at the moment after n time steps, $\mathbf{R}_n$ is a vector of the external forces. In the nonlinear explicit analysis component $\mathbf{K} \cdot \mathbf{u}_n$ is replaced with a vector of the internal forces $\mathbf{F}_n$ in order to simplify calculations [8]:

$$\mathbf{F}_n = \int_{V_0} (\mathbf{B}_L^T)_n \hat{\mathbf{S}}_n dV \tag{2}$$

Where $\mathbf{B}_L$ is a matrix such that $\boldsymbol{\varepsilon} = \mathbf{B}_L\mathbf{u}$ and $\boldsymbol{\varepsilon}$ is Green – Lagrange strain, $\hat{\mathbf{S}}$ is a vector corresponding the 2nd Piola–Kirchhoff stress. This follows from the total Lagrangian formulation that relates the 2nd Piola–Kirchhoff stress to the Green–Lagrange strain and where all variables of the element are referred to the initial configuration [1].

Displacements at the $(n+1)\Delta t$ moment are explicitly computed using formula

$$\mathbf{u}_{n+1} = \Delta t^2 \mathbf{M}^{-1}(\mathbf{R}_n - \mathbf{F}_n - \mathbf{C}(\mathbf{u}_n - \mathbf{u}_{n-1})/\Delta t) + 2\mathbf{u}_n - \mathbf{u}_{n-1} \tag{3}$$

3 Degenerated Shell Element (Reissner-Mindlin Assumptions)

Degenerated shell element also called basic shell model was developed from solid model in order to represent in-plane and bending behavior and corresponds to the plate element of the Reissner-Mindlin mathematical model [3, 4]. Basic shell element is developed with the plane stress assumption $\sigma_{33} = 0$ which is a contradiction to a consequence of the Reissner-Mindlin kinematic assumption that the strain component $e_{33} = 0$. These assumptions are substantiated by considering kinematical assumptions with additional thickness variable at each node for the higher order elements [4].

Any shell element is defined by material properties, nodal point coordinates, shell mid-surface normal and shell thickness at each mid-surface node. For convenience this relation can be written in respect to midsurface coordinates and a vector connecting upper and lower points:

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum N_k(\xi,\eta)\cdot\left(\begin{Bmatrix} \tilde{x}_k \\ \tilde{y}_k \\ \tilde{z}_k \end{Bmatrix} + \frac{1}{2}\zeta h_k \mathbf{v}_k\right) \tag{4}$$

Where $N_k(\xi,\eta)$ is a shape function of the kth node and ζ is a linear coordinate in the thickness direction, h_k is thickness of the shell at the kth node and $\mathbf{v}$ is a unit vector in the direction normal to the mid-surface [13]. The displacements at each node of the degenerated shell are uniquely defined by three components of the mid-surface node displacement and two rotations about orthogonal directions normal to $\mathbf{v}$. To simplify equations, additional degree of freedom (rotation about z axis) is included and constrained [2].

4 Element Connections

The flat structure is assembled of the 4-node degenerated shell elements. Each node belongs to one shell element but the initial coordinates are identical for the nodes which connect two or four neighboring elements, e.g. in Fig. 1 the initial coordinates of nodes 6, 7, 10, 11 (connecting 4 elements) are identical. Moreover, each connection has an indication which describes the direction the elements are connected, e.g. the connection 5-9 has indication 1 which means that elements are connected in the fiber direction of the unidirectional composite and connection 2-3 has indication 2 which means that elements are connected in the transverse direction.

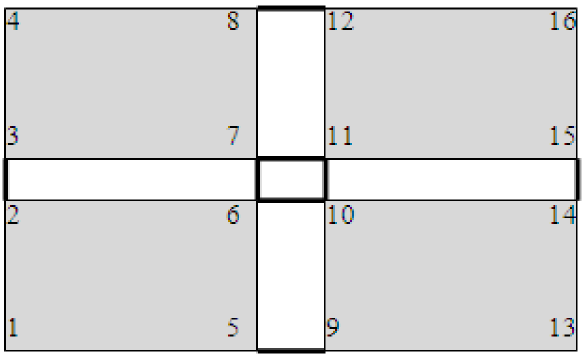

Fig. 1. Element connections and node numbering

Element connections are constrained using Master-Slave or penalty methods. Both methods are described in [6] as methods used for multifreedom constraints. The main idea of both methods is to modify stiffness equations by applying constraints.

4.1 Master – Slave Method

As mentioned above, there are nodes that have identical initial coordinates. One node of this group is chosen as a master node (typically, the one with highest global number) and the other nodes are called slaves. The slave nodes are eliminated and modified equations are used to evaluate displacements at the master nodes. Due to simplicity, it is assumed that constraints are homogeneous, that is the displacements of the one master-slaves group are identical.

The Master-Slave method is implemented by constructing transformation matrix $\mathbf{T}$. First of all, all nodes are classified into independent nodes (nodes 1, 4, 13, 16 in Fig. 1), masters (nodes 3, 9, 11, 12, 15 in Fig. 1) and slaves (others). Matrix $\mathbf{T}$ is a sparse matrix which has $n \times m$ form, where n is a number of degrees of freedom in the structure and m is a number of degrees of freedom of the independent and master nodes. Each row in the matrix $\mathbf{T}$ has only one element that is equal to 1 in the column that corresponds to:

- the degrees of freedom of the independent node if the row corresponds the degrees of freedom of the independent node;
- the degrees of freedom of the master node, if the row corresponds the degrees of freedom of the master node;
- the degrees of freedom of the master node of the group with the slave node, if the row corresponds the degrees of freedom of the slave node.

Matrix $\mathbf{T}$ has $n \times m$ form, where n is a number of degrees of freedom in the structure and m is a number of degrees of freedom of the independent and master nodes.

Then displacements of all nodes $\mathbf{u}$ are written in the matrix form:

$$\mathbf{u} = \mathbf{T}\hat{\mathbf{u}} \tag{5}$$

Where $\hat{\mathbf{u}}$ is a displacement vector of the modified system which consists of the independent and master nodes. All matrices and vectors in (3) are transformed by the rule:

$$\hat{\mathbf{M}} = \mathbf{T}^T\mathbf{M}\mathbf{T}\,,\ \ \hat{\mathbf{f}} = \mathbf{T}^T\mathbf{f} \tag{6}$$

Where vectors and matrices with circumflex accent define vectors and matrices of the modified system.

If failure criteria in the specified direction is satisfied and connection between elements is deleted, nodes are re-classified and matrix $\mathbf{T}$ is re-arranged.

4.2 Penalty Function Method

The degenerated shell elements in Fig. 1 are connected at two nodes (ith and jth) by fictitious bar elements called penalty elements. There are 6 degrees of freedom at each node. The length of bar is zero at the initial moment and the axial stiffness of the element is ω and called penalty weight. The stiffness equations for the penalty element are:

$$\omega\begin{bmatrix} \mathbf{I}_{6\times6} & -\mathbf{I}_{6\times6} \\ -\mathbf{I}_{6\times6} & \mathbf{I}_{6\times6} \end{bmatrix}\begin{bmatrix} \mathbf{u}_i \\ \mathbf{u}_j \end{bmatrix} = \begin{bmatrix} \mathbf{0} \\ \mathbf{0} \end{bmatrix} \tag{7}$$

Then the additional equations of penalty elements are added to the initial system of equations. This method is direct (all displacements are computed at once) and no node reduction is used.

However, the basic problem of the method is the penalty weight selection. To maintain the integrity of the structure, we choose the penalty weight equal to Young's modulus in x or y direction respectively to the indication of the degree of freedom.

If failure criteria in the specified direction is satisfied, the penalty weight is changed. In the simplest case, the penalty weight becomes zero. However, more natural case is not considered in this paper when penalty weight decreases if length of the penalty element increases.

5 Failure Criteria

General formulation of failure criteria used to evaluate loads that cause failure of the individual layer of the unidirectional composite is described in [10]. If layer is modeled with the shell elements, it is enough to know stresses in principal material directions. Then failure criteria is described specifying the combination of stresses that cause fracture:

$$F(\sigma_1, \sigma_2, \tau_{12}) = 1 \tag{8}$$

σ_1, σ_2 stresses in principal directions and τ_{12} is shear stress. This means that elements work without failure if $F < 1$, fails if $F = 1$ and is deleted if $F > 1$.

In this article, Hashin failure criteria is employed [12]:

- Tensile fiber mode ($\sigma_1 \geq 0$):

$$\left(\frac{\sigma_1}{X_t}\right)^2 - 1 = \begin{cases} \geq 0, \textit{connection fails} \\ < 0, \textit{connection remains} \end{cases} \tag{9}$$

- Compressive fiber mode ($\sigma_1 < 0$):

$$\left(\frac{\sigma_1}{X_c}\right)^2 - 1 = \begin{cases} \geq 0, \textit{connection fails} \\ < 0, \textit{connection remains} \end{cases} \tag{10}$$

- Tensile matrix mode ($\sigma_2 \geq 0$):

$$\left(\frac{\sigma_2}{Y_t}\right)^2 + \left(\frac{\tau_{12}}{S}\right)^2 - 1 = \begin{cases} \geq 0, \textit{connection fails} \\ < 0, \textit{connection remains} \end{cases} \tag{11}$$

- Compressive matrix mode ($\sigma_2 < 0$):

$$\left(\frac{\sigma_2}{Y_c}\right)^2 + \left(\frac{\tau_{12}}{S}\right)^2 - 1 = \begin{cases} \geq 0, \textit{connection fails} \\ < 0, \textit{connection remains} \end{cases} \quad (12)$$

Where X_t and X_c are respectively longitudinal tension and compression strengths, Y_t and Y_c are respectively transverse tension and compression strengths and S is shear strength.

Failure criteria is evaluated at the node with stresses equal to the average of the stresses of the connecting nodes. Penalty element with indication 1 is deleted if criteria in fiber tensile or compressive mode is satisfied and penalty element with indication 2 is deleted if criteria in matrix tensile or compressive mode is satisfied.

6 Numerical Examples

6.1 Longitudinal Load (Fiber Direction)

The material of the model is orthotropic and the structure is assumed as unidirectional fiber composite where fibers lie along the x axis. Material properties are defined by the parameters listed in Table 1.

Table 1. Material parameters of the degenerated shell model

Young‘s modulus, E_x	$44.3 \cdot 10^9 N/m^2$
Young‘s modulus, E_y	$14.4 \cdot 10^9 N/m^2$
Poisson‘s ratio, v_{xy}	0.32
Shear modulus, G_{xy}	$4.43 \cdot 10^9 N/m^2$
Shear modulus, G_{yz}	$4.05 \cdot 10^9 N/m^2$
Shear modulus, G_{zx}	$4.94 \cdot 10^9 N/m^2$
Density, ρ	$1432.7 kg/m^3$

Penalty weight ω changes in steps and is equal to w - arbitrary value equal to Young’s modulus in x or y direction respectively to the indicator of the degree of freedom if failure criteria $F <= 1$.

$$\omega = \begin{cases} w, F <= 1 \\ 0, \textit{otherwise} \end{cases} \quad (13)$$

The flat structure in Fig. 2 is analyzed with linear loads in the x direction. Nodes that lie on the x axis are constrained in the y direction and rotation about the x axis and nodes that lie on the y axis are constrained in the x direction and rotation about the y axis. These constraints are employed in order to simulate only a quarter of the structure and maintain the symmetry of a structure.

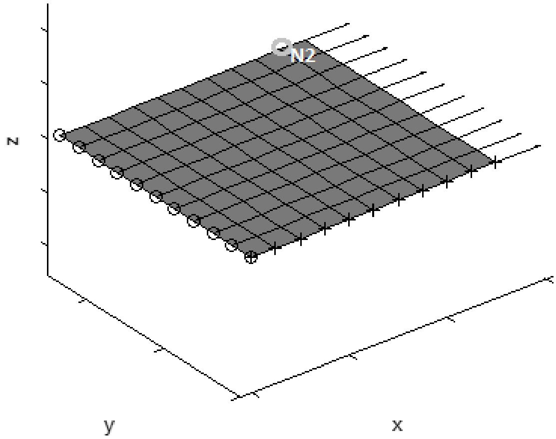

Fig. 2. Shell model used for damage simulation and loads along the x axis

Displacements of the node group N2 (Fig. 3) are compared below. The node group N2 corresponds to nodes 8, 12 in Fig. 1 that are labeled $N2_1$, $N2_2$. Due to the kinematic effects, failure criteria is satisfied for the first row of fictitious elements in the load direction as displayed in Fig. 3. As mentioned above, if failure criteria in tensile fiber mode is satisfied, connection of the elements is deleted and group N2 is disjointed to the nodes $N2_1$ and $N2_2$.

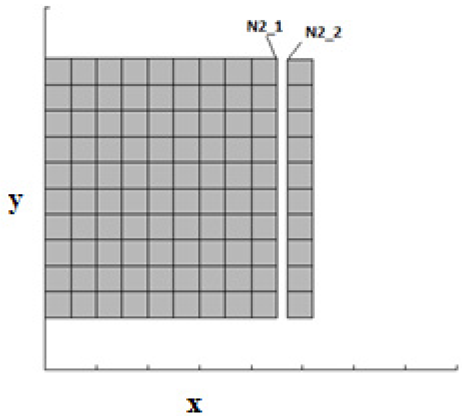

Fig. 3. Damaged structure at the final moment (Displacements multiplied by 10)

The x-displacements at the nodes $N2_1$ and $N2_2$ evaluated by different methods are displayed in Fig. 4. The difference between x-displacements is insignificant and appears because of computational errors. X-Displacements at both nodes are equal for Master-Slave method or differ minutely for penalty method if failure criteria is not satisfied as displayed in Fig. 5. After failure criteria is satisfied, x-displacements at the nodes differ significantly. This is caused by increasing linear load on the edge of the structure.

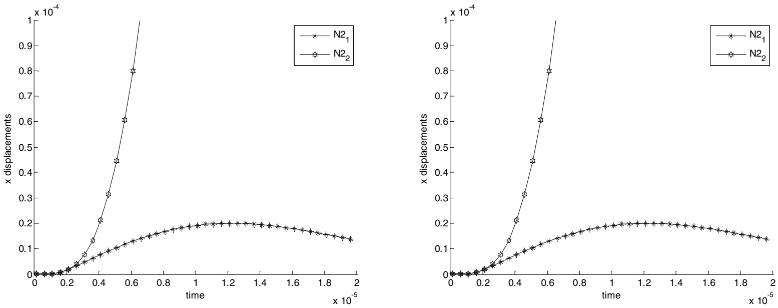

Fig. 4. X-Displacements of the $N2_1$ and $N2_2$ evaluated by Master-Slave (left) and penalty (right) methods

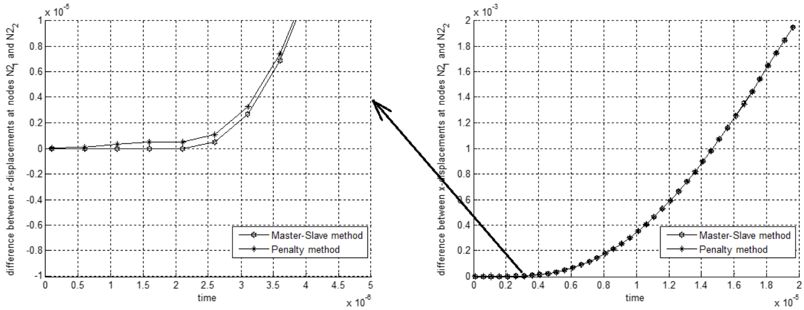

Fig. 5. Difference between displacements at the nodes $N2_1$ and $N2_2$ in the x direction

6.2 Transverse Load

The structure from the previous section is analyzed with linear load of the same magnitude in the transverse direction. Displacements of the node group N1 are compared below. Node group N1 corresponds to nodes 14, 15 in Fig. 1 and are labeled $N1_1$, $N1_2$.

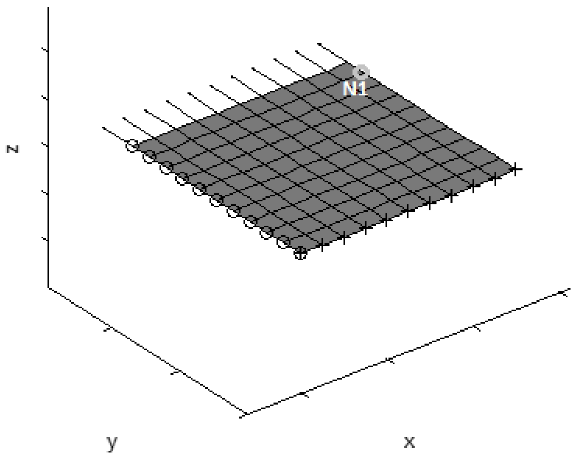

Fig. 6. Shell model used for damage simulation and loads along the y axis

Due to kinematic effects, failure criteria is satisfied for the first row of fictitious elements in the load direction as shown in Fig. 6. If failure criteria in tensile matrix mode is satisfied, connection with indication 2 is deleted and group N1 is disjointed to the nodes $N1_1$ and $N1_2$ (Fig. 7).

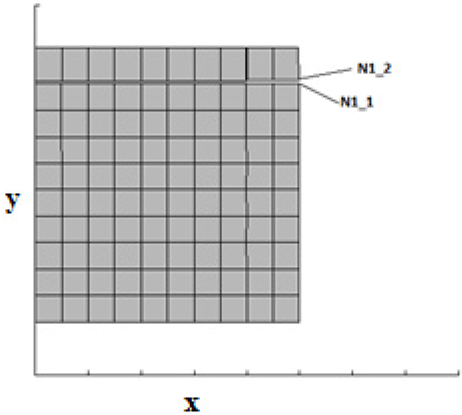

Fig. 7. Damaged structure at the final moment (Displacements multiplied by 10)

As in the previous section, displacements in the load direction (y) of the nodes $N1_1$ and $N1_2$ evaluated by Master-Slave and penalty methods displayed in Fig. 8 differ insignificantly and differences appear due to the computational errors.

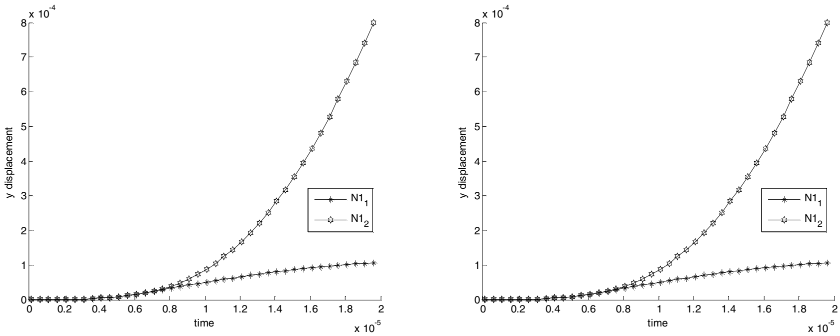

Fig. 8. Y-Displacements of the N1 group evaluated by Master-Slave (left) and penalty (right) methods

Difference between displacements between the nodes $N1_1$ and $N1_2$ also referred as a length of penalty element is zero for Master-Slave method and is small compared with shell element length for penalty method if failure criteria is not satisfied. Like in the previous section, the difference between nodes evolves after failure criteria is satisfied. This is caused by increasing linear load on the edge of the structure.

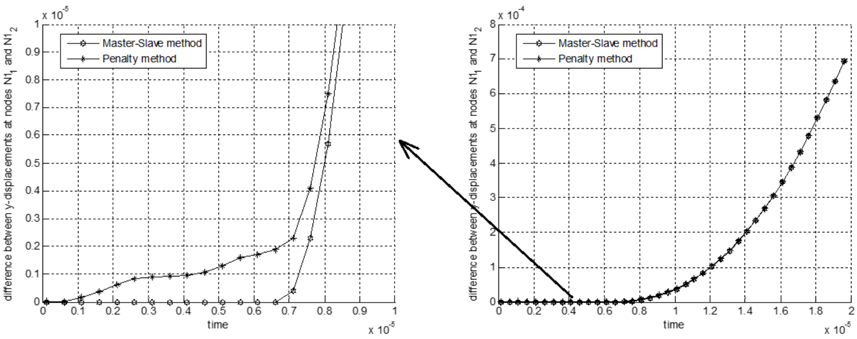

Fig. 9. Difference between displacements at the nodes $N1_1$ and $N1_2$ in the y direction

7 Conclusions

The failure of the flat structure assembled of the degenerated shell elements simulated by Master-Slave and Penalty methods is discussed in this paper. The orthotropic material was used for simulation and penalty weights were selected with respect to material parameters. As expected, for the loads of the same magnitude, structure fails for longitudinal load approximately 3 times faster than for the transverse load because of the orthotropicity of the material.

The obvious advantage of penalty method is a simple implementation. Contrary to the penalty function method, assembling of transformation matrix used in Master-Slave method is rather complex because of the master node selection and rearranging equations if the connection between the elements is deleted. The main advantage of the Master-Slave method is that it reduces the number of unknowns and is similar to the usual assembly process used in FEM. Moreover, all parameters in the Master-Slave method are determined contrary to the penalty method where the main problem is penalty weight selection. The results of the both methods show decent agreement for the considered examples.

References

1. Bathe, K.J.: Finite Element Procedures. Prentice Hall (1996)
2. Benson, D.J., Bazilevs, Y., Hsu, M.C., Hughes, T.J.R.: Isogeometric shell analysis: The Reissner – Mindlin shell. Comput. Methods Appl. Mech. Engrg 19, 276–289 (2010)
3. Carrera, E.: Theories and Finite Elements for Multilayered, Anisotropic, Composite Plates and Shells. Arch. Comput. Meth. Engng 9, 87–140 (2002)
4. Chapelle, D., Bathe, K.J.: The Finite Analysis of Shells – Fundamentals, 2nd edn. Springer (2011)
5. Cui, W.C., Wisnom, M.R., Jones, M.: A comparison of failure criteria to predict delamination of unidirectional glass/epoxy specimens waisted through the thickness. Composites 23, 158–166 (1992)
6. Felippa, C.A.: Introduction to Finite Element Methods (2011)

7. Li, B., Li, Y., Su, J.: A combined interface element to simulate interfacial fracture of laminated shell structures. Composites: Part B 58, 217–227 (2014)
8. Li, M., Zhan, F.: The Finite Deformation Theory for Beam, Plate and Shell. Part IV. The FE Formulation of Mindlin Plate and Shell Based on Green–Lagrangian strain. Comput. Methods Appl. Mech. Engrg. 182, 187–203 (2000)
9. Liu, P.F., Xing, L.J., Zheng, J.Y.: Failure analysis of carbon fiber/epoxy composite cylindrical laminates using explicit finite element method. Composites: Part B 56, 54–61 (2014)
10. Morozov, E.V., Vasiliev, V.V.: Advanced Mechanics of Composite Materials, 2nd edn. Elsevier Science (2007)
11. Spotswood, S.M., Palazotto, A.N.: Progressive failure analysis of a composite shell. Composite Structures 53, 117–131 (2001)
12. Schweizerhof, K., Weimar, K., Munz, T., Rottner, T.: Crashworthiness Analysis with Enhanced Composite Material Models in LS-DYNA – Merits and Limits. In: LS-DYNA World Conference, Detroit, MI, USA (1998)
13. Zienkiewicz, O.C., Taylor, R.L.: The Finite Element Method for Solid and Structural Mechanics. Elsevier (2005)