Tomas Skersys
Rimantas Butleris
Rita Butkiene (Eds.)

Communications in Computer and Information Science 403

Information and Software Technologies

19th International Conference, ICIST 2013
Kaunas, Lithuania, October 2013
Proceedings

Springer

Communications in Computer and Information Science 403

Tomas Skersys   Rimantas Butleris
Rita Butkiene (Eds.)

Information and Software Technologies

19th International Conference, ICIST 2013
Kaunas, Lithuania, October 10-11, 2013
Proceedings

Springer

Volume Editors

Tomas Skersys
Rimantas Butleris
Rita Butkiene

Kaunas University of Technology
Studentu g. 50-313a
51368 Kaunas, Lithuania
E-mail: {tomas.skersys; rimantas.butleris; rita.butkiene}@ktu.lt

ISSN 1865-0929 e-ISSN 1865-0937
ISBN 978-3-642-41946-1 e-ISBN 978-3-642-41947-8
DOI 10.1007/978-3-642-41947-8
Springer Heidelberg New York Dordrecht London

Library of Congress Control Number: Applied for

CR Subject Classification (1998): D.2, H.4, H.3, H.2.8, I.2, J.1, K.3, G.1, F.2

Typesetting: Camera-ready by author, data conversion by Scientific Publishing Services, Chennai, India

Printed on acid-free paper

Springer is part of Springer Science+Business Media (www.springer.com)

Table of Contents

Information Systems

Business Intelligence for Information and Software Systems

Software Engineering

Two Scale Modeling of Heterogeneous Solid Body by Use of Thick Shell Finite Elements

Dalia Čalnerytė and Rimantas Barauskas

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

Abstract. An elasticity parameters evaluation for homogeneous material is considered in this paper if parameters of consisting materials are known in micro scale. The thick shell formulation for homogeneous orthotropic material is discussed and total Lagrangian formulation for the 4-node thick shell element in implicit and explicit analysis is described. The results of the thick shell model are compared with the results of 3D model and LS-Dyna shell model with the same loading.

Keywords: Multi-scale modeling, Total Lagrangian formulation, 4-node thick shell element.

1 Introduction

Multi-scale finite element analysis is widely used for modeling and simulation of the physical behavior of materials, the internal structure of which is non-homogeneous and/or architecturally complex. The main idea of multi-scale modeling is to analyze the same physical phenomena or behavior in different length scales. The models of different scales are used to represent the behavior of the same object, however, with different level of minuteness. Appropriate transfer of behavioral features among the models must be ensured. Different assumptions are used for creation of models in each length scale. All the materials traditionally considered as homogeneous are in fact heterogeneous at micro-scale. Traditionally used isotropic, orthotropic, anisotropic behavioral models of materials are based on experimentally known data. As a rule, we consider that they do not require any further analysis at micro-scale. In the real world majority of materials are composites, where the parameters of materials are known only in micro scale. That is the reason why multi-scale modeling is used for evaluation of equivalent material parameters in upper scale. Equivalent parameters are the elasticity parameters of homogeneous body which has the same behavior as heterogeneous body. Sometimes in engineering computations traditional orthotropic models with properly adjusted parameters are employed, however, they may serve only as very rough estimations of the reality.

The equivalent parameters of a material are evaluated according to the rules in [2] and [3]. Obviously this is not the only way to evaluate material parameters – the method using asymptotic homogenization is presented in [9]. Applying this method only

T. Skersys, R. Butleris, and R. Butkiene (Eds.): ICIST 2013, CCIS 403, pp. 322–333, 2013.

one periodic cell is analyzed with specific periodicity boundary conditions and asymptotic expansion of displacement fields. Moreover the material parameters can be evaluated by mechanical approach with respect to material share in the model. All these methods work fine when linear elasticity is analyzed. The problem arises when there is material or geometrical non-linearity.

In this work, we concentrate on elaboration of thick shell finite elements suitable for multi-scale computations. Shell elements are convenient in computations of upper scales if the dimensions of a body are significantly small in one direction compared with others. Three main formulations in analysis of geometrical non-linearity as total Lagrange, updated Lagrange and co-rotational formulations may be employed. In the total Lagrange formulation the reference configuration is the initial state of the element and in the updated Lagrange formulation the reference configuration is the last known state [5, 10]. In the co-rotational formulation the reference configuration translates and rotates with the element [10]. The total Lagrange formulation is used in this paper for the formulation of the thick shell finite element. The stiffness tensor of the material is obtained by means sequential multi-scale coupling. The homogenized mechanical stiffness constants of the structure are obtained by performing the FE analysis of the mechanical behaviour of the micro-cube. Pure stress components of the micro-cube are created by prescribing the necessary displacements of the sides of the micro-cube. The micro-level finite element model is employed for computing stresses within the micro-cube. MATLAB mathematical software environment and finite element software LS-DYNA were employed.

2 Evaluation of Equivalent Parameters for 3D Solid Model

Homogenization of the composite material to linear elastic material is considered, where the elasticity tensor is used in order to relate stresses and strains in accordance with the generalized Hooke's law [1]:

$$\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon}, \tag{1}$$

where $\mathbf{D}$ is 6x6 elasticity matrix, $\boldsymbol{\sigma} = \{\sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx}\}^T$, $\boldsymbol{\varepsilon} = \{\varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx}\}^T$ are the stress and strain tensors in Voigt's notation respectively. In order to evaluate equivalent parameters of the 3D thick shell model it is assumed that there is zero stress in normal direction ($\sigma_z = 0$). Hence the inverse Hooke's law for 3D model may be separated into two systems [2, 3]:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} S_{11} & S_{12} & 0 \\ S_{21} & S_{22} & 0 \\ 0 & 0 & S_{44} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}; \tag{2}$$

$$\begin{Bmatrix} \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} S_{55} & 0 \\ 0 & S_{66} \end{bmatrix} \begin{Bmatrix} \tau_{yz} \\ \tau_{zx} \end{Bmatrix}, \tag{3}$$

where S_{ij} is a component in the *i*th row and *j*th column of the compliance matrix $\mathbf{S} = \mathbf{D}^{-1}$.

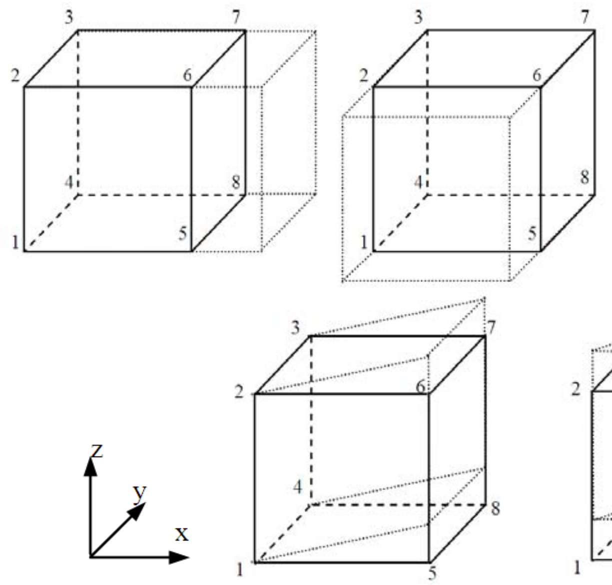

Fig. 1. Schemes for modeling pure stress in LS-DYNA

In order to evaluate the homogenized material parameters of the heterogeneous material pure stresses are simulated according to the schemes in Fig. 1 for the 3D solid finite model of the micro-cube which represents detailed heterogeneous micro-structure.

Table 1. Pure stress assumptions and formulas for parameters evaluation

Assumptions	Formula
$\sigma_x \neq 0,\ \sigma_y = 0, \tau_{xy} = 0,\ \tau_{yz} = 0,\ \tau_{zx} = 0$	$E_x = \dfrac{\sigma_x}{\varepsilon_x},\ \nu_{xy} = -\dfrac{\varepsilon_y}{\varepsilon_x}$
$\sigma_x = 0,\ \sigma_y \neq 0, \tau_{xy} = 0,\ \tau_{yz} = 0,\ \tau_{zx} = 0$	$E_y = \dfrac{\sigma_y}{\varepsilon_y},\ \nu_{yx} = -\dfrac{\varepsilon_x}{\varepsilon_y}$
$\sigma_x = 0,\ \sigma_y = 0,\ \tau_{xy} \neq 0,\ \tau_{yz} = 0,\ \tau_{zx} = 0$	$G_{xy} = \dfrac{\tau_{xy}}{\gamma_{xy}}$
$\sigma_x = 0, \sigma_y = 0, \tau_{xy} = 0,\ \tau_{yz} \neq 0,\ \tau_{zx} = 0$	$G_{yz} = \dfrac{\tau_{yz}}{\gamma_{yz}}$
$\sigma_x = 0, \sigma_y = 0, \tau_{xy} = 0,\ \tau_{yz} = 0,\ \tau_{zx} \neq 0$	$G_{zx} = \dfrac{\tau_{zx}}{\gamma_{zx}}$

If material is isotropic it is enough to know Young's modulus and Poisson's ratio. Orthotropic material for the thick shell element is defined by 6 parameters: E_x, E_y is Young's modulus in index direction, v_{xy} is Poisson's ratio in index plane, G_{xy}, G_{yz}, G_{zx} is shear modulus in index plane (Poisson's ratio v_{yx} is redundant because of the symmetry of elasticity matrix ($v_{xy}E_y = v_{yx}E_x$)):

$$\mathbf{D} = \begin{bmatrix} \frac{E_x}{1-v_{xy}v_{yx}} & \frac{E_x v_{yx}}{1-v_{xy}v_{yx}} & 0 & 0 & 0 \\ \frac{E_y v_{xy}}{1-v_{xy}v_{yx}} & \frac{E_y}{1-v_{xy}v_{yx}} & 0 & 0 & 0 \\ 0 & 0 & G_{xy} & 0 & 0 \\ 0 & 0 & 0 & \kappa G_{yz} & 0 \\ 0 & 0 & 0 & 0 & \kappa G_{zx} \end{bmatrix} \tag{4}$$

Where $\kappa = 5/6$ is a shear correction factor and its purpose is to improve shear displacement approximation.

3 4-node Thick Shell Element

Any shell element can be defined by material properties, nodal point coordinates, shell mid-surface normals and shell thickness at each mid-surface node. The thick shell element is a degenerated three dimensional solid element with integration over its mid-surface. Any point of the thick shell may be related to the top and bottom surfaces of the element:

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum N_k(\xi,\eta) \cdot \left(\frac{1+\zeta}{2} \begin{Bmatrix} \tilde{x}_k \\ \tilde{y}_k \\ \tilde{z}_k \end{Bmatrix}_{top} + \frac{1-\zeta}{2} \begin{Bmatrix} \tilde{x}_k \\ \tilde{y}_k \\ \tilde{z}_k \end{Bmatrix}_{bottom} \right) \tag{5}$$

Where $N_k(\xi,\eta)$ is a shape function of the *k*th node and ζ is a linear coordinate in the thickness direction.

For convenience the previous equation can be rewritten in respect to the mid-surface coordinates and a vector connecting upper and lower points. This vector is a product of shell thickness h_k at the *k*th node and a unit vector $\mathbf{v3}$ in the direction normal to the mid-surface [1]:

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum N_k(\xi,\eta) \cdot \left(\begin{Bmatrix} \tilde{x}_k \\ \tilde{y}_k \\ \tilde{z}_k \end{Bmatrix} + \frac{1}{2}\zeta h_k \mathbf{v3}_k \right) \tag{6}$$

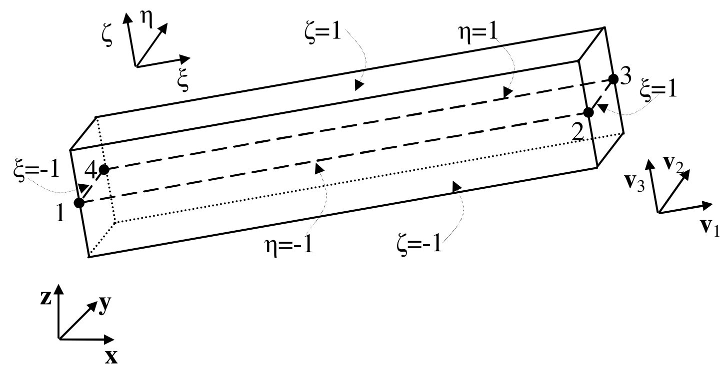

Fig. 2. Thick shell element

The analyzed element is isoparametric – shape functions map every quadrilateral element to square and are used to interpolate the element coordinates and displacements. Shape function for the *k*th node of 4-node thick shell element is bi-linear Lagrange polynomial:

$$N_k(\xi,\eta) = \frac{1}{4}(1+\xi_k\xi)(1+\eta_k\eta),\; k = 1,2,3,4 \tag{7}$$

Due to the fact that the strain in the thickness direction is assumed to be 0, the displacements at each node of the thick shell is uniquely defined by three Cartesian components of the mid-surface node displacement and two rotations about orthogonal directions defined by vectors $\mathbf{v}1$ and $\mathbf{v}2$ normal to $\mathbf{v3}$:

$$\tilde{\mathbf{u}}_k = \{\tilde{u}_k \quad \tilde{\upsilon}_k \quad \tilde{\omega}_k \quad \tilde{\alpha}_k \quad \tilde{\beta}_k\}^T \tag{8}$$

It is evident that a coordinate vector $\mathbf{x}$ in a Cartesian system may be defined by

$$\mathbf{x} = x\mathbf{e}_x + y\mathbf{e}_y + z\mathbf{e}_z \tag{9}$$

Where $\mathbf{e}_x, \mathbf{e}_y, \mathbf{e}_z$ are base vectors. Vectors $\mathbf{v}1$ and $\mathbf{v}2$ in Fig. 2 may be constructed with the following formulas [1]:

$$\mathbf{v1} = \frac{\mathbf{e}_x \times \mathbf{v3}}{|\mathbf{e}_x \times \mathbf{v3}|},\; \mathbf{v2} = \frac{\mathbf{v3}\times\mathbf{v1}}{|\mathbf{v3}\times\mathbf{v1}|} \tag{10}$$

The displacements of any point of the thick shell may be written in respect of the mid-surface displacements [1]:

$$\begin{Bmatrix} u \\ \upsilon \\ \omega \end{Bmatrix} = \sum N_k(\xi,\eta)\cdot\left(\begin{Bmatrix} \tilde{u}_k \\ \tilde{\upsilon}_k \\ \tilde{\omega}_k \end{Bmatrix} + \frac{1}{2}\zeta h_k [\tilde{\mathbf{v}}1_k \quad -\tilde{\mathbf{v}}2_k]\begin{Bmatrix} \tilde{\alpha}_k \\ \tilde{\beta}_k \end{Bmatrix}\right) \tag{11}$$

A 2x2 Gauss integration rule is used for numerical integration of the 4 node element in plane and a 2 point Gauss integration rule is used for numerical integration through thickness.

4 Total Lagrangian Formulation in FEM

Total Lagrangian formulation relates 2nd Piola–Kirchhoff stress to Green–Lagrange strain and all variables of the body are referred to the initial configuration [5].

4.1 Implicit Analysis

Finite element discretization of total Lagrangian formulation for a single element ($\mathbf{R}$ – vector of nodal forces and moments) for *i*th iteration of implicit analysis [5]:

$$\mathbf{K}\Delta\mathbf{u}^{(i)} = \mathbf{R} - \mathbf{F} \tag{12}$$

Where $\mathbf{F} = \int_{V_0} \mathbf{B}_L^T \hat{\mathbf{S}} dV$, $\hat{\mathbf{S}}^T = \{\sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx}\}$, $\mathbf{F}$ is a vector of nodal internal forces and moments, $\Delta\mathbf{u}^{(i)}$ is an increment of nodal displacements in *i*th iteration and stiffness matrix $\mathbf{K}$ is a sum of linear $\mathbf{K}_L$ and non-linear $\mathbf{K}_{NL}$ parts [4]:

$$\mathbf{K}_L = \int_{V_0} \mathbf{B}_L{}^T \mathbf{D} \mathbf{B}_L dV, \; \mathbf{K}_{NL} = \int_{V_0} \mathbf{B}_{NL}^T \mathbf{S} \mathbf{B}_{NL} dV \tag{13}$$

Where $\mathbf{S}^T = \begin{bmatrix} \sigma_{xx}\mathbf{I}_3 & \sigma_{xy}\mathbf{I}_3 & \sigma_{xz}\mathbf{I}_3 \\ \sigma_{xy}\mathbf{I}_3 & \sigma_{yy}\mathbf{I}_3 & \sigma_{yz}\mathbf{I}_3 \\ \sigma_{xz}\mathbf{I}_3 & \sigma_{yz}\mathbf{I}_3 & \sigma_{zz}\mathbf{I}_3 \end{bmatrix}$ and $\mathbf{I}_3$ – 3x3 identity matrix, $\mathbf{D}$ – elasticity tensor, $\mathbf{B}_L$ is a matrix such that $\boldsymbol{\varepsilon} = \mathbf{B}_L \mathbf{u}$ and $\boldsymbol{\varepsilon}$ is Green – Lagrange strain, $\mathbf{u}$ – vector of nodal displacements. Usually $\mathbf{B}_L$ is a sum of two matrices. For non-linear part $\mathbf{B}_{NL}$ can be written:

$$\mathbf{B}_{NL} = \begin{bmatrix} \mid & N'_{k\,x} \cdot \mathbf{I}_3 & \rho_{k,x} \cdot \mathbf{v1} & -\rho_{k,x} \cdot \mathbf{v}2 & \mid \\ \ldots\mid & N'_{k\,y} \cdot \mathbf{I}_3 & \rho_{k,y} \cdot \mathbf{v1} & -\rho_{k,y} \cdot \mathbf{v}2 & \mid\ldots \\ \mid & N'_{k\,z} \cdot \mathbf{I}_3 & \rho_{k,z} \cdot \mathbf{v1} & -\rho_{k,z} \cdot \mathbf{v}2 & \mid \end{bmatrix} \tag{14}$$

Where $\rho_{k,x} = \frac{h}{2}(N'_{k\,x} \cdot \zeta + N_k \cdot \zeta'_x), \rho_{k,y} = \frac{h}{2}(N'_{k\,y} \cdot \zeta + N_k \cdot \zeta'_y), \rho_{k,z} = N_k, k = 1,2,3,4$.

Displacements after *i*th iteration is a sum of displacements after (*i-1*)th iteration and increment of displacements in *i*th iteration:

$$\mathbf{u}^{(i)} = \mathbf{u}^{(i-1)} + \Delta\mathbf{u}^{(i)} \tag{15}$$

4.2 Explicit Analysis

The global system of discretized equations of motion at the *n*th time step is [7]:

$$\mathbf{M}\ddot{\mathbf{u}}_n + \mathbf{K}\cdot\mathbf{u}_n = \mathbf{R}_n \tag{16}$$

Where $\mathbf{u}_n$ is a vector of nodal displacements at the *n*th time step, $\mathbf{M}$ – a mass matrix, $\mathbf{K}$ – stiffness matrix non-linearly dependent on strains, $\mathbf{R}_n$ – vector of nodal (active) forces and moments at the *n*th time step.

The diagonal mass matrix is required in calculations of explicit analysis. Firstly the total element mass is evenly distributed among the four element nodes. Then the rotational nodal masses m_θ are calculated by scaling the translational mass m_t at the node by factor α [8]:

$$m_t = \rho\frac{A}{4}h\,,\ m_\theta = \alpha\cdot m_t\,,\ \alpha = \frac{h^2}{12} \tag{17}$$

Where ρ is the density of material, A is the area of element, h is the thickness of shell, m_t is used for calculating translational accelerations, m_θ is used for calculating rotational accelerations.

Instead of the product of stiffness matrix and displacements the vector of internal forces may be evaluated [7]:

$$\mathbf{K}\cdot\mathbf{u}_n = \mathbf{F}_n = \int_{V_0}(\mathbf{B}_L^T)_n\hat{\mathbf{S}}_n dV \tag{18}$$

Displacements at the $(n+1)$th time step Δt are explicitly computed using central difference formula [7]:

$$\mathbf{u}_{n+1} = \Delta t^2\mathbf{M}^{-1}(\mathbf{R}_n - \mathbf{F}_n) + 2\mathbf{u}_n - \mathbf{u}_{n-1} \tag{19}$$

5 Numerical Experiments

5.1 Evaluation of Material Parameters

The heterogeneous material with periodic microstructure is considered in micro scale. This material consists of two isotropic materials composed as shown in Fig. 3 and called fiber and matrix materials. Each material is defined by Young's modulus and Poisson's ratio and density additionally for the explicit analysis in Table 2. The fibers lie along the x axis in the model.

Table 2. Material parameters of 3D model

	Fiber material	Matrix material
Young‘s modulus, E	$73.1\cdot10^9\,N/m^2$	$3.45\cdot10^9\,N/m^2$
Poisson‘s ratio, v	0.22	0.35
Density, ρ	$1830\,kg/m^3$	$900\,kg/m^3$

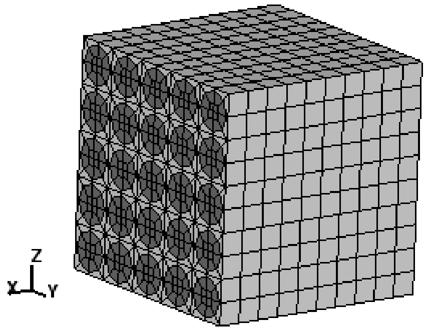

Fig. 3. 3D model used for parameters evaluation

The equivalent parameters for homogeneous material are evaluated by the scheme described in earlier sections. Density of homogeneous material is calculated as weighted mean considering the material share in the model.

Table 3. Material parameters of thick 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	$1432.7 kg/m^3$

5.2 Bending Test

The initial geometry of structure is plane and the one end of the structure is constrained in all directions and rotations. The structure in Fig. 4 is $1m$ length, $0.5m$ width and its thickness (h) is $0.1m$. The out of plane loading is applied in the free end of the structure. It is one of the tests for finite element proposed in [6].

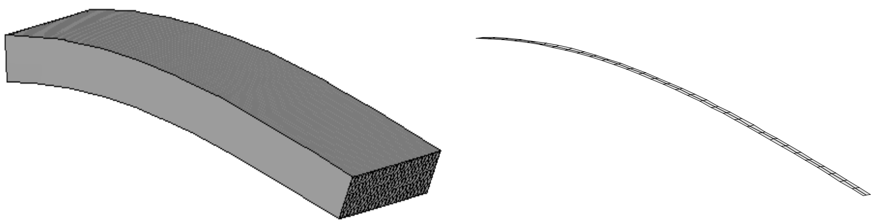

Fig. 4. Deformed configurations of 3D model and thick shell

Implicit Analysis

As it is shown in Table 4 the displacements linearly depend on loaded force when the loading value is small. In this case the relative difference between displacements of 3D model and thick shell elements is constant and displacements of thick shells are greater in absolute value. It should be noticed that displacements were estimated in only one corner mid-surface node of 3D element and the heterogeneity could cause disagreement.

Table 4. Displacements of the free end in z direction (m)

Loading (N)	3D model (m)	Shell (m)	Shell (LS-DYNA) (m)
1e1	1.34e-6	1.74e-6	5.19e-6
1e4	1.34e-3	1.74e-3	5.19e-3
1e5	1.34e-2	1.74e-2	5.19e-2
2e5	2.68e-2	3.47e-2	1.02e-1
4e5	5.35e-2	6.90e-2	1.97e-1
1e6	1.32e-1	1.65e-1	4.10e-1
2e6	2.50e-1	3.00e-1	5.90e-1

Explicit Analysis

In explicit analysis the out-of-plane force evolves linearly according to time.

Table 5. Displacements of the free end in z direction

t (s)	Loading (N)	3D model (m)	Shell (m)	Shell (LS-DYNA) (m)
0.0001	1e4	3.82e-6	6.24e-6	3.87e-6
0.0003	3e4	6.53e-5	8.12e-5	8.07e-5
0.0005	5e4	2.39e-4	2.78e-4	3.20e-4
0.0007	7e4	5.60e-4	6.32e-4	7.71e-4
0.0009	9e4	1.05e-3	1.17e-3	1.46e-3
0.001	1e5	1.37e-3	1.52e-3	1.94e-3

It is obvious that in explicit analysis the displacements do not change linearly though loading force evolves linearly. In addition, displacements of the models differ more than 10% at each moment except for the 3D and LS-DYNA models at the first step. It is important to notice that explicit analysis was performed with a time step $\Delta t = 10^{-5} s$ and the geometrical non-linearity of models was small.

The displacements of models differ more than 15% at the first two moments. Distinct from the **Table 5**, in **Table 6** geometrical non-linearity is large and the displacements of 3D model and both shell models also do not differ more than 15% at last four moments.

Table 6. Displacements of the free end in z direction

t (s)	Loading (N)	3D model (m)	Shell (m)	Shell (LS-DYNA) (m)
0.0001	1e7	4.92e-3	6.23e-3	3.87e-3
0.0003	3e7	6.62e-2	7.73e-2	7.79e-2
0.0005	5e7	2.28e-1	2.22e-1	2.48e-1
0.0007	7e7	4.36e-1	3.94e-1	4.30e-1
0.0009	9e7	6.42e-1	5.87e-1	6.19e-1
0.001	1e8	7.65e-1	7.03e-1	7.34e-1

5.3 Twisting Test

The equal forces in y and z directions are loaded in the free end of the beam as shown in Fig. 5. The length of an arrow is F. It is one of the tests for finite element proposed in [6].

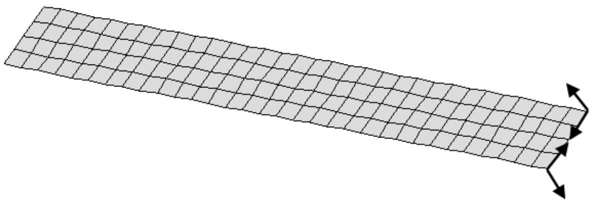

Fig. 5. Loading for twisting test

Implicit Analysis

The initial geometry of beam in Fig. 6 is plane and the one end of the beam is constrained in all directions and rotations. The beam is $1m$ in length, $0.1m$ in width and its thickness (h) is $0.1m$.

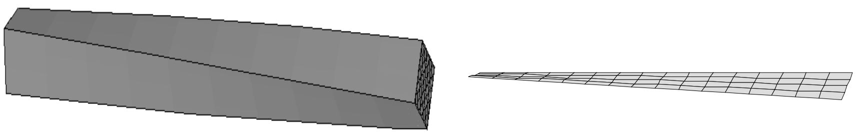

Fig. 6. Deformed configurations of 3D model and thick shell

Table 7. Displacements of the corner of thick shell and midsurface of 3D model

F (N)	3D model (m)		Shell model (m)		Shell (LS-DYNA)(m)	
	y	z	y	z	y	z
1e3	8.54e-6	8.25e-5	1.62e-6	7.19e-5	6.12e-7	7.93e-5
1e4	8.99e-5	8.36e-4	2.08e-5	7.28e-4	1.19e-5	8.06e-4
1e5	1.50e-3	9.36e-3	7.70e-4	7.96e-3	8.89e-4	9.16e-3
2e5	4.95e-3	2.06e-2	3.19e-3	1.70e-2	4.69e-3	2.06e-2

The displacements of twisted models differ significantly in y direction as shown in Table 7 though displacements in z direction differ less than 10% for 3D and LS-DYNA shell models. Like for the bending test here displacements were estimated in only one corner mid-surface node of 3D element. In addition, the loading is sensitive for heterogeneity.

The initial geometry of the structure in Fig. 7 is plane and the one end of the structure is constrained in all directions and rotations. The structure is $1m$ in length, $0.5m$ in width and its thickness (h) is $0.1m$.

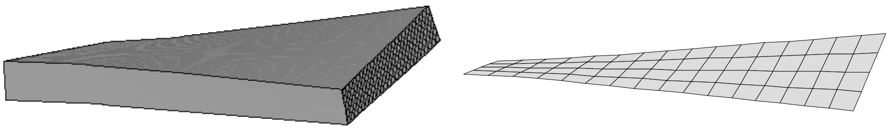

Fig. 7. Deformed configurations of 3D model and thick shell

Table 8. Displacements of the corner of thick shell and midsurface of 3D model

F (N)	3D model (m)		Shell model (m)		Shell (LS-DYNA) (m)	
	y	z	y	z	y	z
1e4	1.08e-4	1.52e-3	3.70e-5	1.54e-3	3.64e-5	1.68e-3
5e4	6.10e-4	7.74e-3	2.77e-4	7.84e-3	2.99e-4	8.65e-3
1e5	1.41e-3	1.59e-2	8.07e-4	1.60e-2	9.28e-4	1.78e-2

The displacements of twisted models differ significantly in y direction as shown in Table 8 though displacements in z direction differ less than 15%.

6 Final Remarks

The homogenized elasticity parameters for heterogeneous material were evaluated in this paper by modeling pure stresses. The 4-node thick shell element with equivalent parameters was implemented. The results of 3D model of heterogeneous structure, thick shell model with 4-node elements and shell model in LS-DYNA were compared.

For bending test the results of shell model in LS-DYNA differed the most compared with the displacements for 3D heterogeneous structure in implicit analysis. Though the differences do not exceed 15% when non-linearity is large in explicit analysis.

Two structures were tested for twisting. Displacements in y direction differed significantly for both structures with all analyzed loadings and difference of displacements in z direction exceeded 20% only with large loading.

In summary evaluation of equivalent elasticity parameters for heterogeneous material described in this paper can be used to analyze behavior of body with composite material only approximately. Moreover thick shell element formulation used for body modeling in upper scale is rather primitive and does not avoid problems such as shear locking. However such element is valuable because of its simple implementation and low computational cost.

References

1. Zienkiewicz, O.C., Taylor, R.L.: The Finite Element Method for Solid and Structural Mechanics. Elsevier, Oxford (2005)
2. Barbero, E.J.: Finite element analysis of composite materials. CRC Press (2008)
3. Kaw, K.A.: Mechanics of Composite Material. CRC Press (2006)
4. Dvorkin, E.N., Bathe, K.J.: A continuum mechanics based four-node shell element for general nonlinear analysis. Eng. Comput. 1, 77–88 (1984)
5. Bathe, K.J.: Finite Element Procedures. Prentice Hall (1996)
6. Macneal, R.H., Harder, R.L.: A Proposed Standard Set of Problems to Test Finite Element Accuracy. Finite Element in Analysis and Design 1, 3–20 (1985)
7. Miller, K., Joldes, G., Lance, D., Wittek, A.: Total Lagrangian explicit dynamics finite element algorithm for computing soft tissue deformation. Communications in Numerical Methods in Engineering 23, 121–134 (2007)
8. Tabiei, A., Tanov, R.: Sandwich shell finite element for dynamic explicit analysis. International Journal for Numerical Methods in Engineering 54, 763–787 (2002)
9. Pinho-da-Cruz, J., Oliveira, J.A., Teixeira-Dias, F.: Asymptotic homogenization in linear elasticity. Part I: Mathematical formulation and finite element modeling. Computational Materials Science 45, 1073–1080 (2009)
10. The, L.H., Clarke, M.J.: Co-rotational and Lagrangian formulations for elastic three-dimensional beam finite elements. Journal of Constructional Steel Research 48, 123–144 (1998)