



Article

Enhancement of Performances of the Simplex Finite Elements by Applying the Strain-Smoothed Element Method

Marija M. Rafailović, Miroslav M. Živković, Vladimir P. Milovanović, Jelena M. Živković, Slobodan R. Savić and Saša T. Milojević



Article

Enhancement of Performances of the Simplex Finite Elements by Applying the Strain-Smoothed Element Method

Marija M. Rafailović , Miroslav M. Živković , Vladimir P. Milovanović , Jelena M. Živković, Slobodan R. Savić and Saša T. Milojević 

Faculty of Engineering, University of Kragujevac, Sestre Janjić 6, 34000 Kragujevac, Serbia; miroslav.zivkovic@kg.ac.rs (M.M.Ž.); vladicka@kg.ac.rs (V.P.M.); jelena.zivkovic@kg.ac.rs (J.M.Ž.); ssavic@kg.ac.rs (S.R.S.)

* Correspondence: marija.rafailovic@kg.ac.rs (M.M.R.); sasa.milojevic@kg.ac.rs (S.T.M.)

Abstract

This paper presents the strain-smoothed element (SSE) method applied to the 4-node tetrahedral finite element. The unique characteristic of the SSE method represents constructing a smoothed strain field for the elements, rather than for the smoothing domains, whereby the constant strains of all adjacent elements are fully utilized for the strain smoothing of the target element. Consequently, a linear strain field within the 4-node tetrahedral finite element is constructed through four Gauss integration points. The method does not require any additional creation of the mesh, and a standard isoparametric formulation of the 4-node tetrahedral finite element is used to formulate it. A numerical example was used to demonstrate the stress prediction improvement achieved as regards the corrected element, as taken from Chaemin Lee and Phill-Seung Lee's publication of the method itself, compared with the linear and quadratic tetrahedral finite elements, which are implemented within the PAK and Simcenter Nastran software packages. The numerical results show that stresses predicted by the strain-smoothed 4-node tetrahedral finite element are comparable to that of a 10-node tetrahedral finite element that incorporates an extrapolation process for the determination of nodal stress values.

Keywords: finite element analysis; low-order finite elements; strain-smoothed element (SSE) method; strain smoothing technique; tetrahedral finite elements

1. Introduction

Displacement finite element method (FEM) models [1,2] have been most commonly used to efficiently solve any practical issues in almost all fields of engineering and physical sciences for the past decades. However, the standard FEM models are generally considered to be too stiff, with an internal strain energy that is not sufficiently estimated [3]. Therefore, the stiffness matrix obtained by the standard FEM method is overestimated, consequently resulting in solutions for displacements that are less than the real ones. As an example, we state the low level of accuracy of predicted stresses during the use of a mesh of linear triangular (3-node triangular finite elements) or linear tetrahedral finite elements (4-node tetrahedral finite elements), i.e., the simplest and the only mesh of finite elements that can be automatically generated in the complex geometry of solids and structures. A classical approach used to deal with these difficulties represents the use of improved hexahedral finite elements instead of linear tetrahedral finite elements. However, the standard FEM method requires the creation of a high-quality mesh of hexahedral finite elements [4,5].



Academic Editor: Qingchao Li

Received: 31 July 2026

Revised: 30 August 2026

Accepted: 31 August 2026

Published: 5 September 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

In addition, mapping and coordinate transformation must be included to ensure the compatibility of the interfaces of hexahedral finite elements, which is why the element cannot be entirely arbitrarily shaped. The use of heavily distorted elements leads to the ill-conditioning of the Jacobian matrix, resulting in very inaccurate solutions and even the termination of the computing process.

In order to provide solutions for the above-mentioned issues, Liu et al. [6] applied the strain smoothing technique of Galerkin meshfree methods [7,8] into the FEM context and formulated a novel method, termed as the smoothed finite element method (SFEM). Introducing the strain smoothing technique, complete with the concept of point interpolation method (PIM), used for constructing meshfree shape functions, into the FEM settings [5], resulted in the development of a class of smoothed finite element methods (S-FEMs or S-FEM) [4,5]. Although the S-FEM methods may be applied to quadrilateral finite elements and hexahedral finite elements, the main concept of S-FEM methods is to enable the use of simple-to-generate, low-order, simplex (triangular, tetrahedral) meshes, keeping the accuracy provided by the higher-order finite elements [9], without introducing additional degrees of freedom (DOFs). The basic idea of S-FEM methods consists of weighted averaging of the compatible strain field over the smoothing domain, which may be found within the specific element cell or spread over a number of elements. Furthermore, by using the simplest local constant smoothing functions, S-FEM methods ensure piecewise constant strain fields through smoothing domains. Therefore, the stiffness matrix in the smoothed model is not computed on the element basis but on the constructed smoothing domain basis. Various schemes for constructing the smoothing domains define various forms of the S-FEM methods, and some of them are the following: the cell-based smoothed finite element method (CS-FEM), originally known as SFEM [6]; the node-based smoothed finite element method (NS-FEM) [10]; the edge-based smoothed finite element method (ES-FEM) [11]; and the face-based smoothed finite element method (FS-FEM) [12]. The theoretical foundation of the S-FEM models consists of the weakened weak form theory (W^2) [5,13]. Therefore, the Gauss divergence theorem is used for its formulation, during which the values of shape functions are necessary but not their derivatives. Hence, the Jacobian matrix is not included in the S-FEM methods, by means of which we develop models that are not sensitive to distortion, which is of particular importance for nonlinear problems of large deformations [5]. For a detailed presentation of the S-FEM method properties, methodology and applications, reference should be made to the review articles [4,5]. It is worth noting that high-order S-FEM methods have been derived for the past few years [14–16]. In these methods, smoothed strains in the smoothing domains are expressed by polynomials of complete order in line with the pick-out technique [17]. In this manner, apart from the constant approximation [16], smoothed strain fields can use the first-order [14–16] and the second-order approximation [14,15]. It should be stated that high-order S-FEM methods go beyond the main concept of the standard S-FEM methods.

The strain smoothing technique is coupled with the extended finite element method (X-FEM) [18] in order to achieve better performance of the low-order finite elements in the crack analysis. Bordas et al. [19] implemented the CS-FEM strain smoothing approach into the X-FEM to solve linear elastic fracture mechanics problems. The formulated method, named the smoothed extended finite element method (SmXFEM) [19], was applied not only to the two-dimensional (2D) but also to the three-dimensional (3D) fracture problems. Afterwards, Chen et al. [9] implemented the ES-FEM method into the X-FEM framework, which resulted in a new numerical method, named an edge-based smoothed extended finite element method (ESm-XFEM). Numerical results show that when compared with the X-FEM, the proposed method considerably improves the accuracy level of the stress intensity factor in the case of using the linear triangular finite elements. The NS-FEM and FS-FEM methods are

coupled with the X-FEM method as well, by means of which superconvergence results are achieved for the linear triangular finite elements [20] and a greater level of accuracy of the linear tetrahedral finite elements [21], respectively. The above-mentioned methods use the advantages of both the S-FEM and X-FEM methods. Namely, due to the transformation of an internal integration (a surface integration for the 2D elements, i.e., a volume integration for the 3D elements) into the integration along the smoothing domain boundaries (a line integration for the 2D elements, i.e., a surface integration for the 3D elements), there is no need to compute neither the derivatives of shape functions nor the ones of enrichment functions, which significantly reduces the level of complexity while evaluating the stiffness matrix. Hence, the derivative of the $1/\sqrt{r}$ term derived from the asymptotic functions does not occur, and in this manner, the integration of a singular term is avoided. By applying the strain smoothing technique, a division of the elements cut by a crack is suppressed in ES-XFEM [9] and a node-based smoothed extended finite element method (NS-XFEM) [20], which simplifies integration of the discontinuous approximation. To additionally increase the level of accuracy of low-order finite elements, Zhao et al. [22] have recently proposed a stable node-based smoothed extended finite element method (SNS-XFEM) for the fracture analysis of structures. The SNS-XFEM method, developed by the coupling of a stable node-based smoothed finite element method (SNS-FEM) [23] and X-FEM method, models the crack-induced discontinuity through the use of enrichment functions and stabilization terms based on strain gradient. Numerical examples indicate that the SNS-XFEM method achieves considerably better results compared with the standard X-FEM method and NS-XFEM method [22].

From another point of view regarding the development of strain smoothing methods, Lee and Lee [24] proposed a new strain smoothing method—the strain-smoothed element (SSE) method for the 4-node tetrahedral finite elements. As with the previously mentioned S-FEM methods, the SSE method does not introduce additional DOFs. However, the proposed method does not require the creation of special smoothing domains, but the strain smoothing process is performed over the entire element by fully utilizing strains of the adjacent elements, sharing common edges with the targeted element. In the SSE method, the important work is to calculate the stiffness matrix of the element, which is now obtained by directly assigning the constructed smoothed strains to the four Gauss integration points. In this manner, the formulated method provides a linear strain field within the 4-node tetrahedral finite element through strain smoothing, whereas the integration of the smoothed strain field is performed over the element. Therefore, the discretization of the domain is completely identical, just like with the standard FEM method, and more accurate solutions are achieved without any additional DOFs, compared with the standard linear formulation of these elements. Subsequently, the SSE method has been applied to the membrane strain field of a 3-node triangular shell finite element (MITC3+) [25]. The strain-smoothed MITC3+ shell finite element has initially been developed for linear analysis, and it considerably enhances the membrane behavior of the MITC3+ finite element while retaining its excellent bending behavior. Likewise, such a formulation has been extended to the nonlinear analysis as well [26], which led to achieving the superior performances of the strain-smoothed MITC3+ finite element in geometrical and material nonlinear analysis. In recent years, the SSE method has been developed for the 4-node quadrilateral finite element [27], as well as for the polygonal finite elements [28]. It should be noted that the application of the SSE method is limited to constant strain finite elements, which is the reason why the strain smoothing process of the 4-node quadrilateral and polygonal finite elements is performed through triangulation with the piecewise linear shape functions employed for each sub-triangle. Hence, to apply the SSE method to the 4-node quadrilateral and polygonal finite elements, additional steps are necessitated within the operative sequence

of procedures invoked by the triangulation, unlike the aforementioned strain-smoothed finite elements [24–26]. More recently, Lee and Kim [29] proposed the formulation of the linear triangular finite element that combined the SSE method, complete with the phantom-node method (PNM) [30] for application in linear elastic fracture mechanics problems. Enhanced accuracy was achieved both in stress intensity factors and strain energy when compared with the linear triangular finite element developed in conjunction with the PNM method. Additionally, the proposed approach demonstrates accuracy comparable to that of the phantom-node approach applied to the quadratic triangular finite element (6-node triangular finite element). The mathematical foundation for the convergence properties of the SSE method is presented in [31], whereas designing preconditioners for the strain smoothing methods has been investigated in [32].

The SSE method has been presented in this paper to analyze the accuracy of the stress predictions obtained using the linear and quadratic tetrahedral finite elements (10-node tetrahedral finite elements), implemented within the PAK [33] and Simcenter Nastran [34] software packages, compared with the solutions for the proposed tetrahedral finite element given in [24]. The analyzed numerical example indicates that the proposed element achieves a significant improvement in the accuracy of the predicted stress compared with the standard formulation of the 4-node tetrahedral finite element. In addition, it is noticed that the newly proposed tetrahedral finite element provides the stress solutions that are more accurate compared with the 10-node tetrahedral finite element, which, taking into account all elements that the considered node appertains to, the nodal stress value calculates as the average of the stresses at the nearest integration points to the observed nodal point. Further on, it is observed that the stress prediction of the corrected tetrahedral finite element is comparable to the 10-node tetrahedral finite element that includes an extrapolation process in order to determine nodal stress values.

This paper has been organized in the following manner: the fundamental equations of the standard linear tetrahedral finite element are presented in Section 2.1, whereas Section 2.2 is devoted to the coupling methodology of the FEM and SSE methods as applied to the linear tetrahedral finite elements. In Section 3, performances of the proposed element are discussed based on a numerical example. Concluding considerations are given in Section 4.

2. The Strain-Smoothed Element Method for the Linear Tetrahedral Finite Elements

An overview of elementary equations of standard formulation for the linear tetrahedral finite element is presented in this section, complete with a brief presentation of the formulation for the strain-smoothed 4-node tetrahedral finite element [24].

2.1. Standard Formulation of the Linear Tetrahedral Finite Elements

Geometry and displacement interpolations of the standard linear tetrahedral finite element have the following forms:

$$\mathbf{x} = \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \mathbf{H}\mathbf{X}, \quad (1)$$

$$\mathbf{u} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} u_x \\ u_y \\ u_z \end{Bmatrix} = \mathbf{H}\mathbf{U}, \quad (2)$$

where the displacement interpolation matrix $\mathbf{H}$ is given by:

$$\mathbf{H} = \begin{bmatrix} \mathbf{H}^1 & \mathbf{H}^2 & \mathbf{H}^3 & \mathbf{H}^4 \end{bmatrix}, \quad (3)$$

and the vectors of nodal point coordinates $\mathbf{X}$ and nodal point displacements $\mathbf{U}$ in the global Cartesian coordinate system are defined as follows:

$$\mathbf{X}^T = \begin{bmatrix} \mathbf{X}^{1T} & \mathbf{X}^{2T} & \mathbf{X}^{3T} & \mathbf{X}^{4T} \end{bmatrix}, \quad (4)$$

$$\mathbf{U}^T = \begin{bmatrix} \mathbf{U}^{1T} & \mathbf{U}^{2T} & \mathbf{U}^{3T} & \mathbf{U}^{4T} \end{bmatrix}. \quad (5)$$

Submatrices $\mathbf{H}^k$, the position vector $\mathbf{X}^k$ and the displacement vector $\mathbf{U}^k$ of the nodal point k have the following forms, respectively:

$$\mathbf{H}^k = \begin{bmatrix} h_k & 0 & 0 \\ 0 & h_k & 0 \\ 0 & 0 & h_k \end{bmatrix}, \quad (6)$$

$$\mathbf{X}^{kT} = \begin{bmatrix} X_1^k & X_2^k & X_3^k \end{bmatrix}, \quad (7)$$

$$\mathbf{U}^{kT} = \begin{bmatrix} U_1^k & U_2^k & U_3^k \end{bmatrix}. \quad (8)$$

Indices 1, 2 and 3 in (7) and (8) refer to the axes x , y and z respectively.

In the case of the 4-node tetrahedral finite element, shape functions $h_k(r, s, t)$ of the standard isoparametric procedure corresponding to the nodal point k are given as:

$$h_1 = 1 - r - s - t, h_2 = r, h_3 = s, h_4 = t. \quad (9)$$

It can be noticed that the shape functions are linear according to the natural coordinates r , s and t which is why this element is referred to as a linear element.

The strain tensor of the tetrahedral finite element contains all six strain components and is expressed as:

$$\mathbf{e}^e = \mathbf{B}^e \mathbf{U}^e = \begin{bmatrix} \mathbf{B}^1 & \mathbf{B}^2 & \mathbf{B}^3 & \mathbf{B}^4 \end{bmatrix} \begin{bmatrix} \mathbf{U}^1 & \mathbf{U}^2 & \mathbf{U}^3 & \mathbf{U}^4 \end{bmatrix}^T, \quad (10)$$

where vectors $\mathbf{U}$ and $\mathbf{U}^k$ are defined by (5) and (8), respectively, whereas $\mathbf{B}$ represents the strain-displacement matrix. Submatrices $\mathbf{B}^k$ for the nodal point k read:

$$\mathbf{B}^k = \frac{1}{6V} \begin{bmatrix} b_k & 0 & 0 \\ 0 & c_k & 0 \\ 0 & 0 & d_k \\ c_k & b_k & 0 \\ 0 & d_k & c_k \\ d_k & 0 & b_k \end{bmatrix}, \quad (11)$$

$$b_k = (-1)^k \begin{vmatrix} 1 & X_2^1 & X_3^1 \\ 1 & X_2^m & X_3^m \\ 1 & X_2^n & X_3^n \end{vmatrix}, \quad (12)$$

$$c_k = (-1)^k \begin{vmatrix} X_1^1 & 1 & X_3^1 \\ X_1^m & 1 & X_3^m \\ X_1^n & 1 & X_3^n \end{vmatrix}, \quad (13)$$

$$d_k = (-1)^k \begin{vmatrix} X_1^l & X_2^l & 1 \\ X_1^m & X_2^m & 1 \\ X_1^n & X_2^n & 1 \end{vmatrix}, \quad (14)$$

where $k, l, m, n = 1, 2, 3, 4$. The coefficients b_k, c_k and d_k are obtained by a cyclic permutation of indices k, l, m, n i.e., $klmn \rightarrow lmkn \rightarrow mnkl \rightarrow nklm$.

2.2. Formulation of the Strain-Smoothed 4-Node Tetrahedral Finite Elements

Constant strains of the adjacent elements, defined by (10), are fully used for the construction of the smoothed strain field of the observed element. Therefore, we observe the linear tetrahedral finite element, the i^{th} edge and adjacent elements that the observed element shares the i^{th} edge with, as shown in Figure 1.

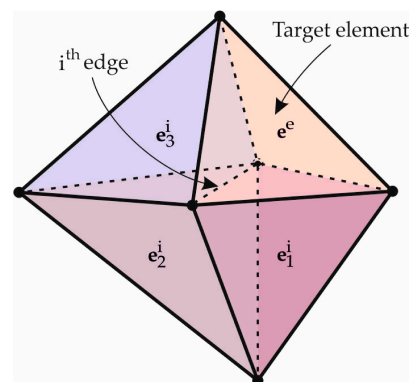


Figure 1. Strain smoothing between the target element and elements in the vicinity of the i^{th} edge of the target element, with $n_i = 3$.

Strain smoothing operation is performed along the six edges of the tetrahedron between the target element and adjacent elements, as expressed in the following equation:

$$\hat{\mathbf{e}}^i = \frac{1}{V^e + \sum_{j=1}^{n_i} V_j^i} \left(V^e \mathbf{e}^e + \sum_{j=1}^{n_i} V_j^i \mathbf{e}_j^i \right), \quad (15)$$

where $i = 1, 2, 3, 4, 5, 6$. Here, n_i represents the number of adjacent elements sharing the i^{th} edge with the target element, $\mathbf{e}^e$ and $\mathbf{e}_j^i$ represent strain tensors of the target element and j^{th} element in the vicinity of the i^{th} edge of the target element, whereas V^e and V_j^i represent their volumes, respectively. The strain tensors $\mathbf{e}^e$ and $\mathbf{e}_j^i$ are determined by the application of the standard formulation of the linear tetrahedral finite elements, i.e., by (10).

It should be stated that the strain smoothing process defined by (15) represents the determination of strain at the middle of the element edges.

The configuration of adjacent tetrahedral finite elements along the six edges of the observed element may differ. In addition, as is the case with the other strain smoothing methods, the effect of strain smoothing of external regions can be less compared with the effect of strain smoothing of internal regions, because external (boundary) elements contain relatively fewer adjacent elements compared with the internal elements. As regards the case when the observed boundary element shares no edges with any other adjacent element, that is, as for the case where $n_i = 0$, the strains shall be equal $\hat{\mathbf{e}}^i = \mathbf{e}^e$.

The four Gauss integration points are used for computing the stiffness matrix of the smoothed tetrahedral finite element, the positions of which a, b, c and d are given in Figure 2.

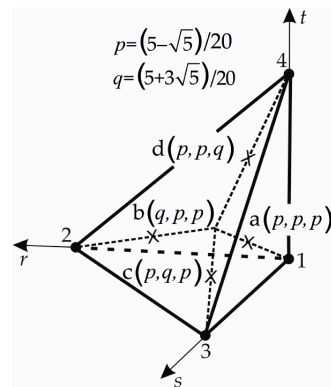


Figure 2. Location of the Gauss integration points in the natural coordinate system of the smoothed tetrahedral finite element (r, s, t) .

The smoothed strain values are directly assigned to the Gauss integration points through the use of the following equations:

$$\bar{\mathbf{e}}^a = \frac{1}{5} (\hat{\mathbf{e}}^1 + \hat{\mathbf{e}}^2 + \hat{\mathbf{e}}^3 + \hat{\mathbf{e}} + \mathbf{e}^e), \quad (16)$$

$$\bar{\mathbf{e}}^b = \frac{1}{5} (\hat{\mathbf{e}}^1 + \hat{\mathbf{e}}^4 + \hat{\mathbf{e}}^6 + \hat{\mathbf{e}} + \mathbf{e}^e), \quad (17)$$

$$\bar{\mathbf{e}}^c = \frac{1}{5} (\hat{\mathbf{e}}^2 + \hat{\mathbf{e}}^4 + \hat{\mathbf{e}}^5 + \hat{\mathbf{e}} + \mathbf{e}^e), \quad (18)$$

$$\bar{\mathbf{e}}^d = \frac{1}{5} (\hat{\mathbf{e}}^3 + \hat{\mathbf{e}}^5 + \hat{\mathbf{e}}^6 + \hat{\mathbf{e}} + \mathbf{e}^e), \quad (19)$$

$$\hat{\mathbf{e}} = \frac{1}{6} \sum_{i=1}^6 \hat{\mathbf{e}}^i. \quad (20)$$

The smoothed strain field and the edge numbering of the observed element are given in Figure 3.

The stiffness matrix of the strain-smoothed 4-node tetrahedral finite element is formed according to the following equation:

$$\mathbf{K} = \sum_{p=a}^d \bar{\mathbf{B}}^p \mathbf{C} \bar{\mathbf{B}}^p |J| w_p, \quad (21)$$

with $\mathbf{C}$ being the observed element constitutive matrix, $\bar{\mathbf{B}}^p$ being smoothed strain-displacement matrices for the four Gauss integration points a–d, $|J|$ being the Jacobian determinant and w_p being weighted coefficients for the four Gauss integration points a–d.

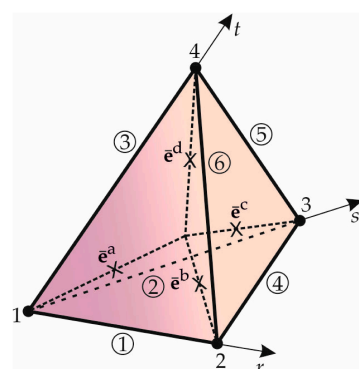


Figure 3. Presentation of the smoothed strain field of the target element and edge numbering.

The smoothed strain-displacement matrices $\bar{\mathbf{B}}^p$ for the four Gauss integration points a–d relate smoothed strains (16)–(19) to the corresponding nodal point displacement vector $\bar{\mathbf{U}}$ by the equation:

$$\bar{\mathbf{e}}^p = \bar{\mathbf{B}}^p \bar{\mathbf{U}}, \quad (22)$$

and are given by:

$$\bar{\mathbf{B}}^p = [\bar{\mathbf{B}}^1 \quad \bar{\mathbf{B}}^2 \quad \dots \quad \bar{\mathbf{B}}^n], \quad (23)$$

where n represents the number of nodes that are involved in the strain smoothing process of the target element.

The corresponding nodal point displacement vector $\bar{\mathbf{U}}$ containing n displacement vectors is given by:

$$\bar{\mathbf{U}}^T = [\mathbf{U}^{1T} \quad \mathbf{U}^{2T} \quad \dots \quad \mathbf{U}^{nT}]. \quad (24)$$

The first four submatrices in (23) and vectors in (24) refer to the four nodes of the target element, while the number n can take values from five to eight, depending on the number of adjacent elements through four element faces of the target element. That is, every adjacent element through an element face of the target element contributes one additional node in the formation of (23) and (24).

3. Numerical Example

A 3D problem, the subject of which is a sphere that is subjected to pressure, shall be analyzed in this section. A numerical analysis shall be conducted in the PAK v2024 and Simcenter Nastran v2412 software packages. The main idea of this section is a comparative representation of the accuracy of the stress predictions obtained for the low-order and high-order tetrahedral finite elements, implemented in the PAK and Simcenter Nastran software packages, compared with the solutions obtained for the corrected element, which are given in the paper itself [24].

PAK (derived from “Program for Structural Analysis” in Serbian) is a software package based on the FEM method that is being developed at the Faculty of Engineering, University of Kragujevac. Evolvement of the PAK program started in 1975 in the field of linear analysis, and it has been developed to encompass nonlinear problems [35]. Today, PAK is an extensive program that can be used to solve problems in the areas of solid mechanics, physical quantity fields (fluids, heat transfer, porous media), biomechanics and coupled problems.

NASTRAN (originated from “NASA STRuctural ANALysis”) is a finite element computer code for structural analysis that was originally developed for NASA in the late 1960s and became available to the public in late 1970 [36]. Simcenter Nastran (formerly NX Nastran) is a program based on the NASTRAN original source code and is today being developed within the company Siemens Digital Industries Software. The program is multidisciplinary and is used to solve problems of linear and nonlinear analysis, dynamic response, vibro-acoustics, aeroelasticity and optimization.

A Hollow Sphere Under Internal Pressure: Lamé Problem

Let us consider the Lamé problem [37], as shown in Figure 4. To create the finite element model, the Simcenter Femap program [34] was used. What is observed is a hollow sphere with an internal radius of $a = 1$ m and the external radius of $b = 2$ m, which is subjected to internal pressure $P = 100$ Pa. Material properties of the problem are $E = 1 \times 10^3$ Pa and $\nu = 0.3$.

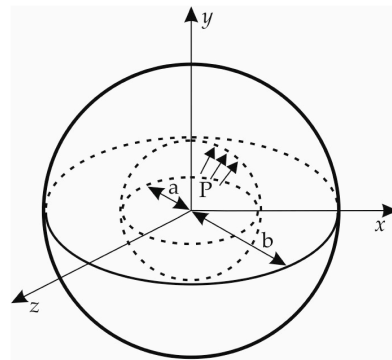


Figure 4. A hollow sphere under internal pressure.

The sphere is divided into three parts, and each one of them is modeled by the mesh the density of which is $N \times N \times N$ ($N = 3, 5, 9, 17$), with N being the number of nodes per edge, as shown in Figure 5. Because the hollow sphere has three axes of symmetry, one-eighth of the model shall be analyzed while bearing in mind the following boundary conditions: $u_z = 0$ for the nodal points on the surface ABEF, $u_x = 0$ for the nodal points on the surface CDEF and $u_y = 0$ for the nodal points on the surface ABCD.

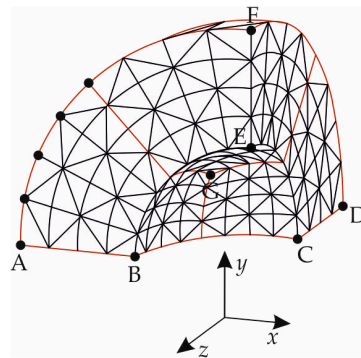


Figure 5. Presentation of the mesh of the Lamé problem, for the number of nodes per edge $N = 5$.

The analytical solution of the Lamé problem is given in the spherical coordinate system (R, ψ, θ) , depicted in Figure 6, in terms of the normal stress σ_R in the radial direction and normal stresses σ_ψ and σ_θ in tangential (ψ and θ) directions (known also as circumferential or hoop stresses), which read [37]:

$$\sigma_R = \frac{Pa^3(b^3 - R^3)}{R^3(a^3 - b^3)}, \quad (25)$$

$$\sigma_\psi = \sigma_\theta = \sigma_t = \frac{Pa^3(2R^3 + b^3)}{2R^3(b^3 - a^3)}, \quad (26)$$

where R represents the radial distance from the centroid of the sphere and the point of interest in the sphere. Note that the hollow sphere expands equally along tangential directions due to the axial symmetry of the problem. From (25) and (26), we can observe that in the case of the hollow sphere subjected to internal pressure only, σ_R is always compressive stress, while σ_t is always tensile stress.

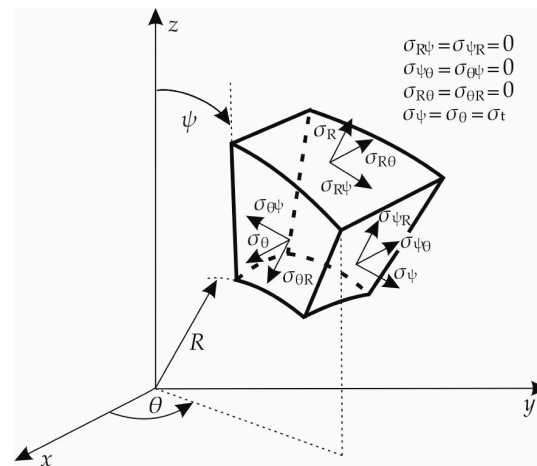


Figure 6. Stress state of the Lamé problem.

Prediction of the numerical solution for the equivalent von Mises stress at the nodal point G of the sphere, as shown in Figure 5, is considered. To analytically calculate the equivalent von Mises stress, axial symmetry should again be taken into account, causing the shear stresses to vanish, meaning that the bases of the spherical coordinate system are also the principal directions of the stress tensor (i.e., $\sigma_1 = \sigma_R$, $\sigma_2 = \sigma_\psi$ and $\sigma_3 = \sigma_\theta$). Written in terms of the principal stress components, the equivalent von Mises stress takes the following form [38]:

$$\sigma_{\text{ref}} = \sqrt{\frac{1}{2} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2]}. \quad (27)$$

Tables 1 and 2 depict the results of linear and quadratic tetrahedral finite elements, obtained in the PAK and Simcenter Nastran software packages, respectively, considering the four different mesh discretizations. For comparison, the numerical results for the corrected element of the Lamé problem, given in paper [24], are also listed in Tables 1 and 2. Relative errors have been computed in the expression $|\sigma_{\text{ref}} - \sigma| / \sigma_{\text{ref}} \times 100$, where σ represents the obtained numerical solution. The corresponding graphical representation of the convergence of the numerical solutions, given in Tables 1 and 2, towards the analytical solution when increasing the mesh density is provided in Figures 7 and 8, respectively, to highlight the convergence rate of all analyzed tetrahedral finite elements. The values in the charts are normalized relative to the analytical solution.

Table 1. Comparative solutions for the equivalent von Mises stress at the nodal point G of the linear and quadratic tetrahedral finite elements obtained by the PAK software package and the corrected tetrahedral finite element adapted from [24].

Number of Nodes per Edge	Linear Tetrahedral Finite Element		Quadratic Tetrahedral Finite Element		Corrected Tetrahedral Finite Element	
N	[Pa]	%	[Pa]	%	[Pa]	%
3	115.6233	32.55	94.6232	44.80	166.7563	2.73
5	129.5844	24.41	139.7400	18.49	166.3370	2.97
9	154.5133	9.87	151.4733	11.64	166.6823	2.77
17	161.4900	5.80	162.7967	5.04	169.5621	1.09
Analytical solution: 171.4286						

Table 2. Comparative solutions for the equivalent von Mises stress at the nodal point G of the linear and quadratic tetrahedral finite elements obtained by the Simcenter Nastran software package and the corrected tetrahedral finite element adapted from [24].

Number of Nodes per Edge	Linear Tetrahedral Finite Element		Quadratic Tetrahedral Finite Element		Corrected Tetrahedral Finite Element	
	[Pa]	%	[Pa]	%	[Pa]	%
3	115.6235	32.55	145.0665	15.38	166.7563	2.73
5	129.5831	24.41	163.9383	4.37	166.3370	2.97
9	154.5146	9.87	168.2156	1.87	166.6823	2.77
17	161.4879	5.80	170.6606	0.45	169.5621	1.09

Analytical solution: 171.4286

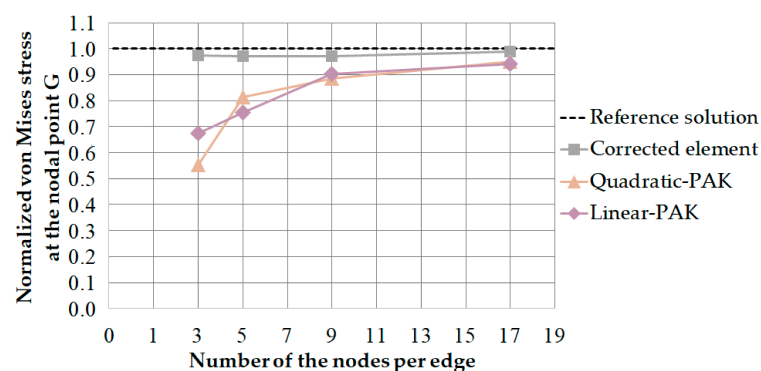


Figure 7. Comparison between convergence towards the analytical solution of the linear and quadratic tetrahedral finite elements, based on the numerical results of the PAK software package, and the corrected tetrahedral finite element adapted from [24]. Convergence curves depict the equivalent von Mises stress at the nodal point G depending on the number of nodes per edge. The results are normalized relative to the analytical solution.

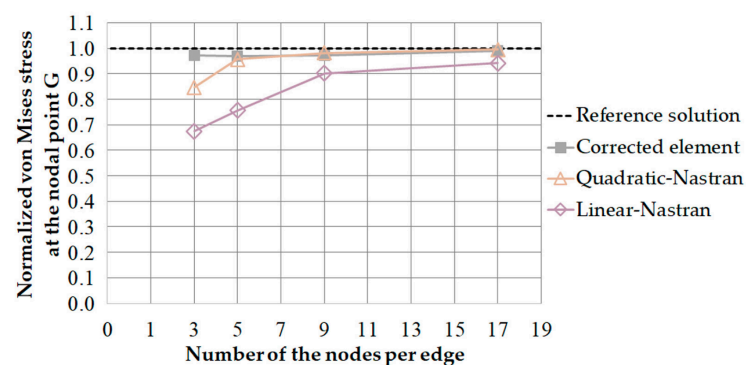


Figure 8. Comparison between convergence towards the analytical solution of the linear and quadratic tetrahedral finite elements, based on the numerical results of the Simcenter Nastran software package, and the corrected tetrahedral finite element adapted from [24]. Convergence curves depict the equivalent von Mises stress at the nodal point G depending on the number of nodes per edge. The results are normalized relative to the analytical solution.

Prior to the analysis of the numerical solutions, let us reflect on the manner of calculating the stress values at nodal point G. Tetrahedral finite element shape functions are C^0 continuous, meaning that the continuity of the unknown (displacement) occurs between elements, but not their slopes (derivatives), which implies that the stress field across element boundaries is discontinuous. In both software packages, the one-point

Gauss quadrature rule (midpoint rule) and four-point Gauss quadrature rule were used for numerical integration over the linear and quadratic tetrahedral finite elements, respectively. Regarding the PAK software package, the nodal stress value of the linear tetrahedral finite element is calculated by averaging the stresses obtained at integration points of all elements to which the observed nodal point belongs. On the other hand, taking into account all elements that the considered node appertains to, the average of the stresses at the nearest integration points to the observed nodal point defines the nodal stress value in the case of the quadratic tetrahedral finite element. The discrepancy in obtaining stress values at nodal points in the PAK and Simcenter Nastran software packages is reflected in the extrapolation to the nodal points of stresses calculated at the integration points, which is applied to the quadratic tetrahedral finite element of the Simcenter Nastran software package [34]. No extrapolation is applied for the constant stress tetrahedral finite element (linear tetrahedral finite element). As regards the corrected element, the stress values are obtained by averaging stresses in the elements to which the nodal point belongs [24].

4. Discussion

Taking into consideration all the results shown, firstly, the ones obtained by the PAK and Simcenter Nastran software packages, we observe good agreement between the numerical solutions of the linear tetrahedral finite elements and numerical discrepancy between the quadratic tetrahedral finite elements, which is justifiable by recalling the fact that the Simcenter Nastran software package calculates the nodal stress value by extrapolating stress values obtained at integration points. While making a comparison between the linear and corrected tetrahedral finite elements, the stress field of which is constant and linear, respectively, we notice a 32.55% error with a very coarse mesh, for $N = 3$, in a linear tetrahedral finite element, compared with the corrected element with an error of 2.73% only. With respect to the numerical solutions of the quadratic and corrected tetrahedral finite elements, it can be observed that increasing mesh density leads to a slightly better stress prediction of the quadratic tetrahedral finite element with an incorporated extrapolation process. However, one should bear in mind that a linear stress field in the quadratic tetrahedral finite element is achieved through the use of second-order shape functions and that it is an element with a higher number of DOFs. It can also be noted that the stress prediction of the corrected element is more accurate in comparison with that of the quadratic tetrahedral finite element, where, taking into consideration all elements that the considered node appertains to, the nodal stress value is defined as the average of the stresses at the nearest integration points to the observed nodal point. It is interesting to point out that the corrected element at a mesh density of $N = 3$ provides more accurate stress prediction than the quadratic tetrahedral finite element in both software packages.

5. Conclusions

The SSE method, which was applied to linear tetrahedral finite elements to reduce their overly stiff behavior, was demonstrated in this work. The proposed element has the smoothed strain field within the element that is constructed by using constant strains of the adjacent elements. The method is based on the standard FEM framework, so the domain discretization and the number of DOFs remain unchanged compared with the standard isoparametric formulation of elements. The strain-smoothed 4-node tetrahedral finite element provides a linear strain field within the element, with the smoothed strain values being directly assigned to the Gauss integration points.

A numerical analysis of a hollow sphere subjected to internal pressure was performed in the PAK and Simcenter Nastran software packages by using the linear and quadratic tetrahedral finite elements. The numerical solutions show that there is a good agreement between the results obtained by the application of linear tetrahedral finite elements. The lack of fit of numerical results obtained in the above-mentioned software packages is noticed in the case of quadratic tetrahedral finite elements, which is explained by the extrapolation that the Simcenter Nastran software package includes when determining the stress values at nodal points, while in the PAK software package the stress values at the integration point are recorded as the stress values at the nearest node.

The numerical example demonstrates that a significant improvement in the accuracy of the stress prediction is achieved by using the corrected element when compared with the linear tetrahedral finite element. It is noticed that the corrected element converges considerably faster towards the analytical solution and that a good fit of the numerical and analytical solutions is achieved even in the case of the relatively coarse mesh of the corrected elements. It is also observed that more accurate stress solutions are provided by the corrected tetrahedral element when compared with those of the quadratic tetrahedral finite element whose nodal stress value is defined as the average of the stresses at the nearest integration points to the observed nodal point, taking into account all elements that the considered node belongs to. The obtained results show that the strain-smoothed tetrahedral finite element ensures the stress solutions, which are comparable to the quadratic tetrahedral finite elements that incorporate extrapolation for the calculation of the nodal stress values. However, one should bear in mind the fact that the quadratic tetrahedral finite element adopts the second-order shape functions, i.e., the ones containing one order more compared with the linear tetrahedral finite element, along with the fact that the number of nodal points of quadratic tetrahedral finite element is greater, leading to the increased number of equilibrium equations of the system that need to be solved. Further research is needed for analysis of bending-dominated problems and development of the SSE method for nonlinear analysis and nearly incompressible materials.

Author Contributions: Conceptualization, M.M.R., V.P.M. and S.T.M.; methodology, M.M.R., M.M.Ž. and V.P.M.; software, M.M.Ž.; validation, M.M.Ž.; formal analysis, M.M.R. and S.R.S.; investigation, M.M.R.; resources, M.M.Ž.; writing—original draft preparation, M.M.R.; writing—review and editing, M.M.R., M.M.Ž., V.P.M., J.M.Ž., S.R.S. and S.T.M.; visualization, M.M.R., V.P.M. and S.T.M.; supervision, M.M.Ž., V.P.M., J.M.Ž. and S.T.M.; project administration, M.M.R.; funding acquisition, M.M.Ž. and S.T.M. All authors have read and agreed to the published version of the manuscript.

Funding: This research was supported by the Science Fund of the Republic of Serbia, #GRANT No 7475, Prediction of damage evolution in engineering structures—PROMINENT, and by the Ministry of Science, Technological Development and Innovation, Republic of Serbia, Agreement No. 451-03-33/2026-03/200378.

Data Availability Statement: The data obtained in this study are included in the article. Analyzed data of the strain-smoothed 4-node tetrahedral finite element are given in paper [24]—“A new strain smoothing method for triangular and tetrahedral finite elements”, from the authors Chaemin Lee and Phill-Seung Lee, published by Elsevier at <https://doi.org/10.1016/j.cma.2018.07.022>.

Conflicts of Interest: The authors declare no conflicts of interest. The funders had no role in the design of the study; in the collection, analyses, or interpretation of data; in the writing of the manuscript; or in the decision to publish the results.

Abbreviations

The following abbreviations are used in this manuscript:

CS-FEM	Cell-based smoothed finite element method
DOFs	Degrees of freedom
ES-FEM	Edge-based smoothed finite element method
ESm-XFEM	Edge-based smoothed extended finite element method
FEM	Finite element method
FS-FEM	Face-based smoothed finite element method
NS-FEM	Node-based smoothed finite element method
NS-XFEM	Node-based smoothed extended finite element method
PIM	Point interpolation method
PNM	Phantom-node method
SFEM	Smoothed finite element method
S-FEMs or S-FEM	Smoothed finite element methods
SmXFEM	Smoothed extended finite element method
SNS-FEM	Stable node-based smoothed finite element method
SNS-XFEM	Stable node-based smoothed extended finite element method
SSE	Strain-smoothed element
X-FEM	Extended finite element method
2D	Two-dimensional
3D	Three-dimensional

References

1. Bathe, K.J. *Finite Element Procedures*, 4th ed.; Prentice-Hall: Upper Saddle River, NJ, USA, 1996.
2. Kojić, M.; Slavković, R.; Živković, M.; Grujović, N. *The Finite Element Method I*, 2nd ed.; University of Kragujevac, Faculty of Mechanical Engineering: Kragujevac, Serbia, 2010. (In Serbian)
3. Liu, G.R.; Nguyen, T.T.; Dai, K.Y.; Lam, K.Y. Theoretical aspects of the smoothed finite element method (SFEM). *Int. J. Numer. Methods Eng.* **2007**, *71*, 902–930. [\[CrossRef\]](#)
4. Zeng, W.; Liu, G.R. Smoothed finite element methods (S-FEM): An overview and recent developments. *Arch. Comput. Methods Eng.* **2018**, *25*, 397–435. [\[CrossRef\]](#)
5. Liu, G.R. The smoothed finite element method (S-FEM): A framework for the design of numerical models for desired solutions. *Front. Struct. Civ. Eng.* **2019**, *13*, 456–477. [\[CrossRef\]](#)
6. Liu, G.R.; Dai, K.Y.; Nguyen, T.T. A smoothed finite element method for mechanics problems. *Comput. Mech.* **2007**, *39*, 859–877. [\[CrossRef\]](#)
7. Chen, J.S.; Wu, C.T.; Yoon, S.; You, Y. A stabilized conforming nodal integration for Galerkin mesh-free methods. *Int. J. Numer. Methods Eng.* **2001**, *50*, 435–466. [\[CrossRef\]](#)
8. Yoo, J.W.; Moran, B.; Chen, J.S. Stabilized conforming nodal integration in the natural-element method. *Int. J. Numer. Methods Eng.* **2004**, *60*, 861–890. [\[CrossRef\]](#)
9. Chen, L.; Rabczuk, T.; Bordas, S.P.A.; Liu, G.R.; Zeng, K.Y.; Kerfriden, P. Extended finite element method with edge-based strain smoothing (Esm-XFEM) for linear elastic crack growth. *Comput. Methods Appl. Mech. Eng.* **2012**, *209–212*, 250–265. [\[CrossRef\]](#)
10. Liu, G.R.; Nguyen-Thoi, T.; Nguyen-Xuan, H.; Lam, K.Y. A node-based smoothed finite element method (NS-FEM) for upper bound solutions to solid mechanics problems. *Comput. Struct.* **2009**, *87*, 14–26. [\[CrossRef\]](#)
11. Liu, G.R.; Nguyen-Thoi, T.; Lam, K.Y. An edge-based smoothed finite element method (ES-FEM) for static, free and forced vibration analyses of solids. *J. Sound Vib.* **2009**, *320*, 1100–1130. [\[CrossRef\]](#)
12. Nguyen-Thoi, T.; Liu, G.R.; Lam, K.Y.; Zhang, G.Y. A face-based smoothed finite element method (FS-FEM) for 3D linear and geometrically non-linear solid mechanics problems using 4-node tetrahedral elements. *Int. J. Numer. Methods Eng.* **2009**, *78*, 324–353. [\[CrossRef\]](#)
13. Wu, S.W.; Liu, G.R.; Jiang, C.; Liu, X.; Liu, K.; Wan, D.T.; Yue, J.H. Arbitrary polygon mesh for elastic and elastoplastic analysis of solids using smoothed finite element method. *Comput. Methods Appl. Mech. Eng.* **2023**, *405*, 115874. [\[CrossRef\]](#)
14. Li, Y.; Liu, G.R. A novel node-based smoothed finite element method with linear strain fields for static, free and forced vibration analyses of solids. *Appl. Math. Comput.* **2019**, *352*, 30–58. [\[CrossRef\]](#)
15. Cui, X.; Duan, S.Y.; Huo, S.H.; Liu, G.R. A high order cell-based smoothed finite element method using triangular and quadrilateral elements. *Eng. Anal. Bound. Elem.* **2021**, *128*, 133–148. [\[CrossRef\]](#)

16. Cui, X.; Liu, G.R.; Li, Z.R. A high-order edge-based smoothed finite element (ES-FEM) method with four-node triangular element for solid mechanics problems. *Eng. Anal. Bound. Elem.* **2023**, *151*, 490–502. [\[CrossRef\]](#)
17. Liu, G.R. A novel pick-out theory and technique for constructing the smoothed derivatives of functions for numerical methods. *Int. J. Comput. Methods* **2018**, *15*, 1850070. [\[CrossRef\]](#)
18. Fries, T.P.; Belytschko, T. The extended/generalized finite element method: An overview of the method and its applications. *Int. J. Numer. Methods Eng.* **2010**, *84*, 253–304. [\[CrossRef\]](#)
19. Bordas, S.P.A.; Rabczuk, T.; Hung, N.X.; Nguyen, V.P.; Natarajan, S.; Bog, T.; Quan, D.M.; Hiep, N.V. Strain smoothing in FEM and XFEM. *Comput. Struct.* **2010**, *88*, 1419–1443. [\[CrossRef\]](#)
20. Vu-Bac, N.; Nguyen-Xuan, H.; Chen, L.; Bordas, S.; Kerfriden, P.; Simpson, R.N.; Liu, G.R.; Rabczuk, T. A node-based smoothed extended finite element method (NS-XFEM) for fracture analysis. *Comput. Model. Eng. Sci.* **2011**, *73*, 331–355. [\[CrossRef\]](#)
21. Jiang, Y.; Tay, T.E.; Chen, L.; Zhang, Y.W. Extended finite element method coupled with face-based strain smoothing technique for three-dimensional fracture problems. *Int. J. Numer. Methods Eng.* **2015**, *102*, 1894–1916. [\[CrossRef\]](#)
22. Zhao, J.W.; Feng, S.Z.; Tao, Y.R.; Li, Z.X. Stable node-based smoothed extended finite element method for fracture analysis of structures. *Comput. Struct.* **2020**, *240*, 106357. [\[CrossRef\]](#)
23. Hu, X.B.; Cui, X.Y.; Feng, H.; Li, G.Y. Stochastic analysis using the generalized perturbation stable node-based smoothed finite element method. *Eng. Anal. Bound. Elem.* **2016**, *70*, 40–55. [\[CrossRef\]](#)
24. Lee, C.; Lee, P.S. A new strain smoothing method for triangular and tetrahedral finite elements. *Comput. Methods Appl. Mech. Eng.* **2018**, *341*, 939–955. [\[CrossRef\]](#)
25. Lee, C.; Lee, P.S. The strain-smoothed MITC3+ shell finite element. *Comput. Struct.* **2019**, *223*, 106096. [\[CrossRef\]](#)
26. Lee, C.; Lee, D.H.; Lee, P.S. The strain-smoothed MITC3+ shell element in nonlinear analysis. *Comput. Struct.* **2022**, *266*, 106768. [\[CrossRef\]](#)
27. Lee, C.; Kim, S.; Lee, P.S. The strain-smoothed 4-node quadrilateral finite element. *Comput. Methods Appl. Mech. Eng.* **2021**, *373*, 113481. [\[CrossRef\]](#)
28. Jung, H.; Lee, C.; Lee, P.S. Strain-smoothed polygonal finite elements. *Struct. Eng. Mech.* **2023**, *86*, 311–324. [\[CrossRef\]](#)
29. Lee, C.; Kim, G. Combined approach using the phantom-node and strain-smoothed element methods for fracture analysis. *Mech. Based Des. Struct. Mach.* **2025**, *53*, 7591–7612. [\[CrossRef\]](#)
30. Hansbo, A.; Hansbo, P. A finite element method for the simulation of strong and weak discontinuities in solid mechanics. *Comput. Methods Appl. Mech. Eng.* **2004**, *193*, 3523–3540. [\[CrossRef\]](#)
31. Lee, C.; Park, J. A variational framework for the strain-smoothed element method. *Comput. Math. Appl.* **2021**, *94*, 76–93. [\[CrossRef\]](#)
32. Lee, C.; Park, J. Preconditioning for finite element methods with strain smoothing. *Comput. Math. Appl.* **2023**, *130*, 41–57. [\[CrossRef\]](#)
33. Kojić, M.; Slavković, R.; Živković, M.; Grujović, N. *PAK-S: Program for FE Structural Analysis*; Faculty of Engineering, University of Kragujevac: Kragujevac, Serbia, 2024.
34. Siemens Digital Industries Software. *Simcenter Femap with Simcenter Nastran*, 2412; Siemens Digital Industries Software: Plano, TX, USA, 2024.
35. Kojić, M. Computer models in engineering and medicine. *Acad. Speeches* **2019**, *2*, 183–201. (In Serbian)
36. NASA Conference Publication 3029. Foreword. In Proceedings of the Seventeenth Nastran Users' Colloquium, San Antonio, TX, USA, 24–28 April 1989.
37. Timoshenko, S.; Goodier, J.N. *Theory of Elasticity*, 2nd ed.; McGraw-Hill Book Company: New York, NY, USA, 1951.
38. Buljak, V.; Ranzi, G. *Constitutive Modeling of Engineering Materials: Theory, Computer Implementation, and Parameter Identification*; Elsevier: Amsterdam, The Netherlands; Academic Press: Cambridge, MA, USA, 2021.

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.