https://doi.org/10.1016/j.finel.2021.103697

Peer reviewed version
License (if available):
CC BY-NC-ND
Link to published version (if available):
10.1016/j.finel.2021.103697

Link to publication record in Explore Bristol Research
PDF-document

This is the accepted author manuscript (AAM). The final published version (version of record) is available online via Elsevier at https://doi.org/10.1016/j.finel.2021.103697. Please refer to any applicable terms of use of the publisher.

University of Bristol - Explore Bristol Research
General rights
This document is made available in accordance with publisher policies. Please cite only the published version using the reference above. Full terms of use are available: http://www.bristol.ac.uk/red/research-policy/pure/user-guides/ebr-terms/
A geometrically nonlinear variable-kinematics continuum shell element for the analyses of laminated composites

Bristol Composites Institute,
University of Bristol, Queen’s Building, University Walk, Bristol, BS8 1TR, UK

Abstract
To facilitate further gains in structural efficiency, the use of composite materials in engineering structures is on the rise. Simultaneously, a drive for thinner components is leading to structural behaviour that is governed by elastic nonlinearities such as large deflections and instabilities. For efficient and reliable design, numerical models must predict the nonlinear displacements, as well as the corresponding stress and strain responses, both accurately and at minimal computational cost. In this work, we present a novel tensor-based variable kinematics continuum shell (VKCS) formulation that is geometrically nonlinear in a total Lagrangian sense. The key contribution is the development and validation of a nonlinear continuum shell model that is completely general in terms of its geometric and kinematic descriptions. The governing equations are derived and presented in tensorial form, which enables a straightforward spatial mapping for models with complex curvatures. The ‘variable-kinematics’ capability means that the element field variables can be refined in a hierarchical and orthotropic manner, i.e. the in-plane and through-thickness displacements can be independently discretised using any polynomial functions with arbitrary orders of expansion. With this feature, the model configurations can be tailored for specific nonlinear problems, whilst also achieving fast solution convergence rate through the use of higher-order basis functions. For validation, the VKCS model has been benchmarked against existing nonlinear problems in the literature that feature large displacements with complex equilibrium paths. In addition, we have proposed two new benchmarks to investigate the 3D Cauchy stress in a snapped shallow roof, and the postbuckling behaviour of a wind turbine blade section. The VKCS formulation is shown to be a versatile tool that allows the user to easily switch between a multitude of model configurations, and can thus accommodate the varying fidelity of analyses required across different design stages. Furthermore, our benchmarks have demonstrated that the variable-kinematics model requires fewer degrees of freedom and run time to track complex 3D stresses when compared to conventional low-order continuum elements.

Keywords: Geometric nonlinearity, variable-kinematics, continuum shell finite element, 3D stress fields, laminated shell structures

*Corresponding author.
Email address: aewis.hii@bristol.ac.uk (A.K.W. Hii)
1. Introduction

Across the aerospace and automotive industries, lightweight composite structures have gained significant attention from the industrial and research communities. The high stiffness- and strength-to-weight ratios of composite materials allow the design of slender primary structures with much higher load-bearing capacity than is possible with metallic materials.

Due to their slenderness and anisotropic material properties, composite structures exhibit high degrees of geometric nonlinearity, such as inter-rib panel buckling of aircraft wings and ‘breathing’ of wind turbine blade trailing edge (WTB TE) panels [1]. Furthermore, large deformations in slender structures can cause localised buckling [2, 3] and Brazier collapse [4]. These phenomena can induce high magnitudes of 3D stress states in the laminate, and initiate a sequence of events leading to catastrophic failure. It goes without saying that, for safe design, numerical models must accurately predict the nonlinear structural deformations as well as the corresponding 3D stresses.

Large structures can develop localised deformations and instabilities that may be confined to small regions or propagate globally. In any case, it is nontrivial to determine which part of the analysis domain requires refined model kinematics. Therefore, when the user’s experience and know-how can no longer inform the locations and sequences of nonlinear phenomena, a refined model must be used everywhere. In the studies by Yuan [5] and Liang et al. [6] on the nonlinear behaviour of thick laminates and sandwich structures, the authors demonstrate that it is indeed essential to explicitly model the through-thickness distortion and include all geometrically nonlinear terms to capture the nonlinear deformation responses of thick laminates accurately. As noted by Krause et al. [7], the iterative procedures for nonlinear solutions require much more computational effort than linear problems. Hence, it is beneficial for any model to have the lowest discretisation and solution error for a given number of degrees of freedom (dofs).

The kinematic refinement of conventional finite elements often increases the $p$-levels in an ‘isotropic’ manner. For example, when going from a linear to a quadratic continuum element, the order of interpolation increases from 1 to 2 in every direction. In the case where the gradients of the state variables are sharp only in a particular direction, this refinement strategy unnecessarily takes up a significant amount of computational resources. In recent years, a large body of literature on a new class of finite element formulations, known as the variable kinematics models, has emerged. These formulations are rooted in the so-called Unified Formulation by Carrera [8] (CUF). In essence, CUF provides a generalised framework to build finite elements models, by allowing users to customise the type and order of the expansion functions along different directions in the element. The variable kinematics models make it possible to refine finite elements in an orthotropic manner, by only prescribing higher-order basis functions in specific directions to capture certain sharp gradients, thereby significantly reducing the total model dofs.

Many studies [9, 10], including our previous work [11], have shown that this class of models is particularly effective at capturing accurate linear elastic three-dimensional stresses in laminated composites with complex geometry. More specifically, owing to the use of higher-order interpolating functions and the orthotropic refinement strategy, the variable kinematics models can acquire similar level of convergence in 3D stresses with much fewer dofs when compared to the linear and quadratic solid elements commonly found in commercially available finite element codes.

From a design perspective, the numerical analysis of slender structures is usually multi-step in nature. The first steps generally involve stiffness and modal analyses using shell finite elements with Kirchhoff-Love or Reissner-Mindlin kinematics. Although these analyses are accurate in
modelling nonlinear deflections and in-plane stresses of thin shells, their interlaminar stresses 
are inaccurate due to the simplifying kinematic assumptions. The following design steps would
involve modelling the 3D stresses in the critical regions using high-fidelity solid elements to 
check for stress concentrations and local sources of failure. The creation of these models is 
labour intensive, and any subsequent model refinements (for instance, a finer discretisation in 
the thickness direction) may require remeshing of the whole domain. Furthermore, most commercial 
finite element (FE) libraries only offer low-order solid elements that have low convergence rates 
in 3D stresses, hence requiring significant computational resources for accurate stress analysis.

A variable kinematics model offers several advantages compared to this multi-step approach. 
Due to the generalised descriptions of displacement fields, the model fidelity along different di-
rections is a free parameter that can be controlled without changes to the pristine mesh [12]. 
This is a major advantage as mesh reconstruction is labour intensive for complex models. Many 
 hierarchical models in the literature also share similar advantages, for instance the geometrically 
linear hierarchical plate/shell elements in References [13–16]. In terms of geometrically nonlinear 
formulations, Surana et al. [17] proposed a series of models with hierarchical \textit{p}-refinements  
(with Lagrange polynomials) in the thickness direction, as well as in both thickness and the in-
plane directions [12]. In addition, Başar et al. [18] proposed a general higher-order nonlinear 
shell element that allows any polynomial function for the in-plane interpolation, whilst assum-
ing a quadratic distribution for the through-thickness field variables. Other examples include Yu 
et al. ’s work [19] on a hierarchical mixed spectral/\textit{hp}-solid element formulation using a general 
Jacobi polynomial, and Han et al. ’s work [20] on a geometrically nonlinear plate element, where 
the in-plane and through-thickness displacement fields are independently described with Rodrigues’ form of Legendre orthogonal polynomials. In Han’s models, the interpolation functions 
can have different polynomial orders about all three principal directions. In general, hierarchi-
cal finite element models are different from a variable kinematics model, in that the latter offers 
additional flexibility where any type/form of expansion functions can be used.

Existing works on geometrically nonlinear 1D and 2D CUF finite elements can be found in 
Pagani [21] and Wu et al. [22] respectively, both of which have demonstrated good performance 
of variable kinematics models for nonlinear applications. In these models, the governing equa-
tions are derived in the so-called orthogonal curvilinear coordinate system, where the covariant 
basis vectors are written in terms of the principal curvatures and therefore are always orthogonal 
to one another. In doing so, the expressions for the differential areas and Green-Lagrange strains 
are simplified substantially [23]. Nonetheless, in order to obtain a pair of orthogonal surface 
covariant basis vectors, the parametric curves on a surface must be chosen such that they align 
with the principal lines of curvatures. In the context of finite element discretisation, it is trivial 
to create a shell mesh where the orthogonality condition is satisfied for simple geometries such 
as cylinder, sphere and surfaces of revolution. However, for a complex arbitrary surface with 
 spatially varying curvatures, the principal lines of curvatures can vary in a pointwise manner 
within an element. As such, it is non-trivial and impractical to create a shell mesh that guaran-
tees the orthogonality of the covariant basis vectors everywhere in the domain. Generally, mesh 
lines are non-orthogonal and distorted on complex surfaces, and as a result, the covariant basis 
vectors computed from the finite element discretisation are often non-orthogonal. When using 
shell models that utilise orthogonal curvilinear systems, one must evaluate the second funda-
mental forms at each integration point and transform the local basis vectors to align with the 
lines of principle curvatures. This is cumbersome and causes a great deal of inconvenience from 
an implementation point of view when applied to the modelling of complex surfaces.

In this paper, we present a tensor-based variable-kinematics continuum shell formulation
that can model general arbitrary surfaces in a straightforward manner, with a view to address the aforementioned challenges in the existing nonlinear CUF shell models [22]. In our derivation, the discretisation of the finite element geometry follows the so-called tensor-based continuum shell formulations in References [24–26]. The strain energy and their variations are expressed in tensorial forms with no restrictions made on the shell basis vectors. When compared to existing works on nonlinear CUF models [21, 22], the distinguishing feature in our shell element is that the principal radii terms are not considered, and hence do not enter the expressions for the differential areas and the strain-displacement matrices. The governing finite element equations are written completely in terms of the covariant basis vectors and the spatial derivatives using CUF notation. The resulting model is a nonlinear shell element that has a generalised kinematic and geometric descriptions. Here, we also note that the present model is a geometrically nonlinear extension of our previous work [11]. The novelty is in the development of a nonlinear continuum shell element with generalised descriptions of both the geometric and kinematic variables. To the best of our knowledge, it is the first of its kind in the open literature of variable kinematics finite element models. We refer to this formulation as the Variable-Kinematics Continuum Shell (VKCS) model hereafter. In this paper, we demonstrate how the VKCS model can be used as an effective tool during early design stages. The model offers convenience as with a single implementation it encompasses a large library of possible configurations to cover the various levels of model fidelity that may be required in design analysis. The users can switch between different model configurations with minimal effort since the fidelity along different directions is effectively a free parameter. This 'all-in-one' modelling paradigm is in contrast to a conventional setting where significant labour costs are dedicated to building separate models for different design stages.

On a side note, we note that the derivation of the proposed shell model is based on the well-known Ahmad-Irons-Zienkiewicz shell formulation [27], as an alternative to the more classical resultant based shell models [28, 29]. In the former, the material/integration points lie within the continuum shell body; whereas the latter has integration points on the shell mid-surface, and utilises surface membrane, curvatures and transverse shear strains, in which the corresponding material stiffness matrices are obtained through explicit through-thickness integration. Our reasoning in doing so, is that the expressions for the generalised variable kinematics models are already complex in nature, and for ease of implementation, it is beneficial to bypass the complex mappings of quantities between shell mid-surface and material points as required in resultant based formulations. Furthermore, in this work, the through-thickness expansion functions are chosen specifically to yield shell models with only displacement dofs, as opposed to rotational dofs often featured in classical shells. Our motivation in doing so, is to circumvent the complexities often associated with shell rotations [30], i.e. the linearisation of rotation vector/tensor in a consistent manner, as well as the choice of appropriate constitutive laws for drilling rotations.

In Section 2, the derivation of the nonlinear VKCS formulation is outlined. In Section 3.1, the element is validated against a selection of literature benchmarks that feature thin shells with large displacements/rotations. In Section 3.2, a literature benchmark with a complex equilibrium path is considered, where the effects of h- and p-refinements in both in-plane and through-thickness directions on solution convergence and solver performance are discussed. In Section 3.3 and 3.4, we propose two new benchmarks, namely the tracking of nonlinear 3D Cauchy stresses in a thick shallow roof under point load, and the postbuckling response of WTB TE panels.
Figure 1: To construct a patch on a surface, a function $\phi : \xi \mapsto \hat{X} \subset \mathbb{R}^3$ is used to map the curvilinear coordinates $(\xi^1, \xi^2)$ onto a point $(\hat{X}_1, \hat{X}_2, \hat{X}_3)$ on the patch in the Cartesian basis. Naturally, the curvilinear curves are always tangent to the local surfaces. An additional linear function $\Phi : \hat{X} \mapsto X \subset \mathbb{R}^3$ is required to offset the midsurface to produce a shell volume.

2. Governing equations

2.1. Conventions

The following conventions are adopted in this paper: subscripts in Roman and Greek characters range from 1 to 3 and 1 to 2, respectively, unless otherwise stated; the transpose operator is written as superscript ‘T’, i.e. $v^T$. The Roman superscript ‘T’ as in $K^T$ is reserved to denote the tangential stiffness matrix. The outer product is denoted as $\otimes$, the inner product is denoted as $(\cdot)$, and the double inner product is denoted as $(\cdot)$. Einstein notation is implied over repeated indices. Vector components are denoted through superscripts in parenthesis, i.e. $\vec{v} = \{v^{(1)}, v^{(2)}, v^{(3)}\}$.

Unless otherwise stated, tensor and indicial notations are used throughout the paper.

2.2. Geometric and kinematic description of a continuum

The shell geometry is discretised using finite elements with the natural coordinates $\xi = (\xi^1, \xi^2, \xi^3)$. The in-plane domain is parameterised with $\xi^1$ and $\xi^2$, whereas the through-thickness domain is parameterised with $\xi^3$, see Figure 1. The position vector in the reference configuration is

$$\tilde{X}(\xi^1, \xi^2, \xi^3) = N_i(\xi^1, \xi^2) \left( \hat{X}_i + \frac{h(\xi^1, \xi^2)}{2} \xi^3 \vec{D}_i \right) \quad \text{where} \quad i = 1, 2, ..., i_{\text{total}},$$

where $i$ corresponds to the $i$-th node in a 2D element, $i_{\text{total}}$ is the total number of nodes, $N_i$ are the planar interpolation functions, $\hat{X}_i$ are the midsurface position vectors and $\vec{D}_i$ are the directors erected from the midsurface nodes. The covariant basis vectors are defined as

$$G_a(\xi^1, \xi^2, \xi^3) = \frac{\partial \tilde{X}}{\partial \xi^a}$$
where vectors $\vec{G}_1$ and $\vec{G}_2$ are the tangents to the midsurface. The $\vec{G}_3$ is parallel to the local director vector and requires the computation of $\vec{D}_i$.

The director field must be carefully defined to accurately account for complex 3D shell geometry. For simple surfaces, i.e. flat plates or singly curved shells, $\vec{D}_i$ are defined analytically as $\vec{D}_i = \hat{D}(\xi^1, \xi^2)$. For complex geometries, the $\vec{D}_i$ can be approximated numerically as $\vec{D}_i = \frac{\vec{G}_1 \times \vec{G}_2}{|\vec{G}_1 \times \vec{G}_2|}$. (3)

A differential line in the element is $d\vec{X} = d\vec{X}_1 + d\vec{X}_2 + d\vec{X}_3 = \vec{G}_1 d\xi^1 + \vec{G}_2 d\xi^2 + \vec{G}_3 d\xi^3$ (4) or in matrix form,

$$d\vec{X} = \begin{bmatrix} \vec{G}^{(1)}_1 & \vec{G}^{(1)}_2 & \vec{G}^{(1)}_3 \\ \vec{G}^{(2)}_1 & \vec{G}^{(2)}_2 & \vec{G}^{(2)}_3 \\ \vec{G}^{(3)}_1 & \vec{G}^{(3)}_2 & \vec{G}^{(3)}_3 \end{bmatrix} \begin{bmatrix} d\xi^1 \\ d\xi^2 \\ d\xi^3 \end{bmatrix} = \mathbf{J} d\vec{\xi},$$

(5)

where $\mathbf{J}$ is the Jacobian matrix, and the differential volume of the element is $\text{det}(\mathbf{J}) = J$.

2.3. Unified formulation in the Finite Element Method

The primary field variables are written in a hierarchical manner to allow for variable kinematics in the element. Conventionally, the primary variables in a continuum element are interpolated as $\vec{q}(\xi^1, \xi^2, \xi^3) = N_i(\xi^1, \xi^2, \xi^3) \tilde{\vec{q}}_i$, (6) where $i$ denotes the degree of freedom at each finite element node, $N$ is any set of admissible 3D interpolation functions, and $\tilde{\vec{q}}$ denotes the primary variables of the system at the nodes. According to the Unified Formulation, we can arrive at the same interpolations by summing the products of two independent sets of 1D and 2D functions, as follows,

$$\vec{q}(\xi^1, \xi^2, \xi^3) = \sum_{i=1}^{n} \sum_{\tau=1}^{m} N_i(\xi^1, \xi^2) F_{\tau}(\xi^3) \tilde{q}_{i\tau} = N_i(\xi^1, \xi^2) F_{\tau}(\xi^3) \tilde{q}_{i\tau},$$

(7)

where $\tilde{\vec{q}}_{i\tau}$ is a displacement vector that denotes the contribution from expansion terms $\tau$ and $i$. The summation sign has been dropped due to the implied indices summation. The derivatives of the primary variables can be evaluated as

$$\frac{\partial \tilde{\vec{q}}}{\partial \vec{\xi}^i}(\xi^1, \xi^2, \xi^3) = \frac{\partial (N_i(\xi^1, \xi^2) F_{\tau}(\xi^3))}{\partial \vec{\xi}^i} \tilde{q}_{i\tau},$$

(8)

In the context of a CUF shell element, the indices $i$ and $\tau$ denote the dofs, whilst $n$ and $m$ denotes the total number of dofs in the planar and through-thickness domains, respectively.

As discussed previously, any type and order of functions can be used in place of $N_i$ and $F_{\tau}$. An exposition on variable kinematics model with different choices of expansion function can be found in Cinefra et al. [31]. In this paper, we choose to use Lagrange polynomials for both $N_i$ and $F_{\tau}$, effectively yielding a variable-kinematics solid-shell model with variable number of nodes in the planar and through-thickness directions, the various possible configurations are illustrated...
in Figure 2. With this approach, we can utilise layerwise shell theory by simply stacking the solid-shell elements along the through-thickness direction. This choice is based on convenience and it shall be noted that the expressions provided in this paper remain valid for any choice of polynomial functions.

For a displacement-based model, the stiffness matrix corresponding to each CUF index \(i, j, \tau, s\) has size of \(3 \times 3\). The indices \((j, s)\) correspond to the variation components of \((i, \tau)\), respectively. With the compact notation, the user has full control over the type and order of functions to interpolate the planar and through-thickness field variables independently, hence the terminology orthotropic refinement scheme.

2.4. Nonlinear stress and strain measures for incremental solution

The VKCS tangential stiffness matrix and the force vector are derived in this section. The element is formulated based on the Principle of Virtual Work (PVW), where the internal strain energy conjugates are the covariant components of the Green-Lagrange (GL) strain tensor and the contravariant components of the Second Piola-Kirchhoff (2PK) stress tensors written in contravariant and covariant bases, respectively. The strain tensor is:

\[
E_{kl} = \frac{1}{2} (\vec{G}_k \cdot \vec{u}_l + \vec{G}_l \cdot \vec{u}_k + \vec{u}_k \cdot \vec{u}_l),
\]

where \(\vec{G}_k\) are the displacement vectors and the quadratic terms \((\vec{u}_k \cdot \vec{u}_l)\) account for large displacements. \(\vec{G}_k\) are the contravariant basis vectors as a dual basis to \(\vec{G}^l\). Hooke’s law is applied assuming a linear material response:

\[
S = C : E.
\]

Figure 2: By using Lagrange polynomials in both the in-plane and through-thickness basis functions, the VKCS model can be used as a solid-shell element with variable number of planar and through-thickness nodes.
where $S$ and $C$ denote the 2PK stress and the constitutive tensors, respectively. They are both written in a covariant basis, i.e.

$$C = C^{klmn} \vec{G}_k \otimes \vec{G}_l \otimes \vec{G}_m \otimes \vec{G}_n,$$

$$S = S^{kl} \vec{G}_k \otimes \vec{G}_l,$$

with $k, l, m, n = 1, 2, 3$. The components of $S$ and $C$ are

$$S^{kl} = C^{klmn} E_{mn},$$

$$C^{klmn} = T_{ka} T_{lb} T_{mc} T_{nd} C_{abcd}.$$

where $T_{kl} = \vec{G}_k \cdot \hat{e}_l$ where $\vec{G}_k$ is the contravariant basis vector, $C_{abcd}$ are the components of the material tensor in a local Cartesian coordinate system, and $\hat{e}_k$ refer to the local Cartesian material axes.

### 2.5. Strain energy variation

In a static large displacement analysis, the energy functional consists of the virtual internal and external work as

$$\delta \Pi_{\text{int}} - \delta \Pi_{\text{ext}} = 0.$$  \hspace{1cm} (13)

In a total Lagrangian formulation, all the kinematic quantities in the strain energy refer to the reference configuration with volume $B^0$. The governing equations are obtained by linearising the virtual work, which is equivalent to the application of two consecutive variations as follows,

$$\delta \Pi_{\text{int}} = \int_{B^0} S : \delta E \; dB^0$$

$$\Delta(\delta \Pi_{\text{int}}) = \int_{B^0} \Delta S : \delta E \; dB^0 = \int_{B^0} S : \Delta(\delta E) \; dB^0.$$  \hspace{1cm} (14)

For the sake of clarity, the first and second variations are indicated by $\delta$ and $\Delta$, respectively.

There are two parts to the strain energy in Equation (14), namely the material and geometric components, denoted as $\Delta(\delta \Pi_{\text{int}})^m$ and $\Delta(\delta \Pi_{\text{int}})^l$ respectively. The material part can be written in terms of the following tensor components

$$\Delta(\delta \Pi_{\text{int}})^m = \int_{B^0} \Delta S^{kl} \delta E_{kl} \; dB^0.$$  \hspace{1cm} (15)

and from Equation (9), the first strain variation is

$$\delta E_{kl} = \frac{1}{2} \left( \vec{G}_k \cdot \delta \vec{u}_l + \vec{G}_l \cdot \delta \vec{u}_k + \delta \vec{u}_k \cdot \vec{u}_l + \delta \vec{u}_l \cdot \vec{u}_k \right)$$

$$= \frac{1}{2} \left( (\vec{G}_k + \vec{u}_k) \cdot \delta \vec{u}_l + (\vec{G}_l + \vec{u}_l) \cdot \delta \vec{u}_k \right),$$

where the commas denote partial derivatives. Assuming a linear constitutive relation, the variation in the components of 2PK stresses is

$$\Delta S^{mn} = C^{mnkl} \Delta E_{kl}.$$  \hspace{1cm} (17)
The geometric component of the strain energy in Equation (14) is written in terms of the following tensor components,

$$\Delta(\delta\Pi_{\text{int}})_{\text{g}} = \int_{\Omega^0} S^{ij} \Delta(\delta E_{\text{int}}) \, d\mathbf{B}^0, \quad (18)$$

with

$$\Delta(\delta E_{\text{int}}) = \frac{1}{2} \left[ \mathbf{G}_k \cdot \Delta(\delta \mathbf{u}_j) + \mathbf{G}_l \cdot \Delta(\delta \mathbf{u}_k) + \Delta(\delta \mathbf{u}_k) \cdot \mathbf{u}_j + \Delta(\delta \mathbf{u}_j) \cdot \mathbf{u}_k + \Delta(\delta \mathbf{u}_k) \cdot \Delta(\delta \mathbf{u}_j) \right]. \quad (19)$$

For a consistent linearisation of the first variation of the internal strain energy, the higher-order variations of the displacement vector can be neglected [32, 33], such that \( \Delta(\delta \mathbf{u}_k) = \mathbf{0} \). Thus

$$\Delta(\delta E_{\text{int}}) = \frac{1}{2} \left[ \Delta \mathbf{u}_k \cdot \Delta \mathbf{u}_j + \Delta \mathbf{u}_j \cdot \Delta \mathbf{u}_k \right]. \quad (20)$$

Substituting Equation (20) into the second part of Equation (14), the geometric part of the strain energy variations is

$$\Delta(\delta\Pi_{\text{int}})_{\text{g}} = \frac{1}{2} \int_{\Omega^0} \left[ \left( \frac{\partial \Delta \mathbf{u}_k}{\partial \mathbf{x}^i} \cdot \frac{\partial \mathbf{u}_l}{\partial \mathbf{x}^j} + \frac{\partial \mathbf{u}_l}{\partial \mathbf{x}^i} \cdot \frac{\partial \Delta \mathbf{u}_k}{\partial \mathbf{x}^j} \right) S^{i1} + \left( \frac{\partial \Delta \mathbf{u}_k}{\partial \mathbf{x}^i} \cdot \frac{\partial \Delta \mathbf{u}_l}{\partial \mathbf{x}^j} \right) S^{i2} + \frac{\partial \mathbf{u}_l}{\partial \mathbf{x}^i} \cdot \frac{\partial \mathbf{u}_k}{\partial \mathbf{x}^j} \right) S^{12} \right] \, d\mathbf{B}^0. \quad (21)$$

### 2.6. Finite element discretisation with the Unified Formulation

In this section, the strain energy expressions are discretised using the finite element method. More specifically, to introduce variable kinematics, the element displacement fields and their derivatives are written using the CUF notation in Equations (7) and (8), respectively. Substituting the CUF notations into Equation (16) and collecting like-terms, the variation of Green-Lagrange strain becomes

$$\delta E_{\text{int}} = \frac{1}{2} \left[ \mathbf{g}_k \cdot \frac{\partial \mathbf{F}_l}{\partial \mathbf{x}^i} \Delta(\delta \mathbf{u}_i) + \mathbf{g}_l \cdot \frac{\partial \mathbf{F}_k}{\partial \mathbf{x}^i} \Delta(\delta \mathbf{u}_i) \right] = \frac{1}{2} \left( \frac{\partial \mathbf{F}_l}{\partial \mathbf{x}^i} \mathbf{g}_k + \frac{\partial \mathbf{F}_k}{\partial \mathbf{x}^i} \mathbf{g}_l \right) \cdot \Delta \mathbf{u}_i. \quad (22)$$

where \( \mathbf{g}_k \) is the covariant basis vector in the deformed configuration. Similarly,

$$\Delta E_{\text{int}} = \frac{1}{2} \left( \frac{\partial \mathbf{F}_l}{\partial \mathbf{x}^i} \mathbf{g}_k + \frac{\partial \mathbf{F}_k}{\partial \mathbf{x}^i} \mathbf{g}_l \right) \cdot \Delta \mathbf{u}_i. \quad (23)$$

Substituting Equation (22) and (23) into Equation (15), the material part of the strain energy variation is rewritten in matrix notations as

$$\Delta(\delta\Pi_{\text{int}})_{\text{m}} = \delta \mathbf{u}^T_i \int_{\Omega^0} \mathbf{B}^T \mathbf{C} \mathbf{B} \, d\mathbf{B}^0 \Delta \mathbf{u}_i = \delta \mathbf{u}^T_i \mathbf{K}_{j\ell} \Delta \mathbf{u}_j. \quad (24)$$
where $K^i_{jrs}$ is the material tangent stiffness. Again, both the displacement vector and strain-displacement matrix are indexed using $(\tau, i)$ and $(s, j)$ for the first and second variations, respectively. From Equation (22), the explicit matrix form of $B_{ri}$ is

$$B_{ri} = \begin{pmatrix}
\frac{\partial(F,N_i)}{\partial s^1} g_1^{(1)} & \frac{\partial(F,N_i)}{\partial s^2} g_1^{(2)} & \frac{\partial(F,N_i)}{\partial s^3} g_1^{(3)} \\
\frac{\partial(F,N_i)}{\partial s^1} g_2^{(1)} & \frac{\partial(F,N_i)}{\partial s^2} g_2^{(2)} & \frac{\partial(F,N_i)}{\partial s^3} g_2^{(3)} \\
\frac{\partial(F,N_i)}{\partial s^1} g_3^{(1)} & \frac{\partial(F,N_i)}{\partial s^2} g_3^{(2)} & \frac{\partial(F,N_i)}{\partial s^3} g_3^{(3)}
\end{pmatrix}, \quad (25)$$

where the superscript on $g^{(i)}$ denotes the component of the covariant vector in the current configuration. The matrix $B_{ri}$ is obtained by simply replacing the indices $(\tau, i)$ with $(s, j)$.

For the geometric part of the strain energy variation, by substituting CUF notations into Equation (20), we yield

$$\Delta(\delta E_{\text{int}}) = \frac{1}{2} \left( \frac{\partial(F,N_i)}{\partial s^1} \delta u_{ri}, \frac{\partial(F,N_i)}{\partial s^2} \Delta \vec{u}_{sj} + \frac{\partial(F,N_i)}{\partial s^3} \Delta \vec{u}_{sj} \right). \quad (26)$$

Then, the CUF notations are substituted into Equation (21), so that the geometric part of (14) becomes

$$\Delta(\delta \Pi_{\text{int}})_{h} = \delta u_{ri} \int_{g_0} \frac{1}{2} \left\{ \left( \frac{\partial F_{N_i}}{\partial s^1} \frac{\partial F_{N_j}}{\partial s^1} \right) S^{11} + \left( \frac{\partial F_{N_i}}{\partial s^2} \frac{\partial F_{N_j}}{\partial s^2} \right) S^{22} + \left( \frac{\partial F_{N_i}}{\partial s^3} \frac{\partial F_{N_j}}{\partial s^3} \right) S^{33} + \left( \frac{\partial F_{N_i}}{\partial s^2} \frac{\partial F_{N_j}}{\partial s^3} \right) S^{23} \right\} \partial B^{0} \Delta \vec{u}_{sj}. \quad (27)$$

Since Equation (27) is inconvenient for FE development, it is rearranged into the following FE matrices,

$$\Delta(\delta \Pi_{\text{int}})_{h} = \delta u_{ri} \int_{g_0} \begin{pmatrix}
\frac{\partial(F,N_i)}{\partial s^1} \frac{\partial(F,N_j)}{\partial s^1} \\
\frac{\partial(F,N_i)}{\partial s^2} \frac{\partial(F,N_j)}{\partial s^2} \\
\frac{\partial(F,N_i)}{\partial s^3} \frac{\partial(F,N_j)}{\partial s^3}
\end{pmatrix} \begin{pmatrix}
S^{11} \\
S^{22} \\
S^{33} \\
S^{23}
\end{pmatrix} \partial B^{0} \Delta \vec{u}_{sj} = \delta u_{ri}^T \begin{pmatrix}
B_{Ni}^T S I & dB^0 \Delta \vec{u}_{sj}
\end{pmatrix}$$

$$= \delta u_{ri}^T K^i_{jrs} \Delta \vec{u}_{sj}, \quad (28)$$
where $I$ is the $3 \times 3$ identity matrix and $K^G$ is the geometric part of the tangential stiffness matrix. In matrix notation, the full tangent stiffness matrix of the VKCS model is therefore

$$\delta \Delta (\Pi_{\text{int}}) = \Delta (\delta \Pi_{\text{int}})_A + \Delta (\delta \Pi_{\text{int}})_B$$

$$= \delta \mathbf{u}^T_{\text{tr}} (K^t + K^G) \Delta \mathbf{u}_{\text{tr}}$$

$$= \delta \mathbf{u}^T_{\text{tr}} K^T \Delta \mathbf{u}_{\text{tr}}, \quad (29)$$

where $K^T$ is the tangential stiffness matrix.

Nonlinear solvers require the computation of nodal forces, which can be expressed in terms of matrix notation as

$$f_{sj} = \int_{B^0} B_{sj}^T S \, dB^0, \quad (30)$$

where $B_{sj}$ is evaluated from Equation (25).

2.7. Post-processing of Cauchy stress in the current configuration

Often, the rotations of material axes in a small strain, large displacement analysis are not explicitly computed in a total Lagrangian formulation, because all quantities refer to the initial undeformed configuration. However, when criteria-based failure initiation analysis is of interest, the Cauchy stresses must be transformed into the current fibre/matrix directions for the evaluation of damage and failure criteria; hence the orientations of the material triads must be explicitly computed for every configuration throughout the loading history. In commercial finite element codes, the orientation of material axes in the deformed configuration is often computed via Hughes and Winget [34], which provides an effective approximation at low computational cost. In this work, the rotation matrix that transports the material axes between the reference and current configuration is computed directly through polar decomposition of the deformation gradient tensor. The procedures are briefly described here.

First, the deformation gradient tensor is computed in the usual way,

$$F = \bar{g}_i \otimes \bar{G}^i, \quad (31)$$

where $\bar{g}_i$ is the covariant basis vector in the deformed configuration and $\bar{G}^i$ is the contravariant basis vector in the reference configuration.

$$\bar{G}^i = G^i_j \bar{G}_j, \quad \text{where } G^i_j = (G_{ij})^{-1} \text{ and } G_{ij} = \bar{G}_i \cdot \bar{G}_j. \quad (32)$$

The Cauchy stress tensor is then calculated from the 2PK stress tensor as follows

$$\sigma = \frac{1}{\det(\mathbf{F})} \mathbf{F} \hat{\mathbf{S}} \mathbf{F}^T. \quad (33)$$

In this work, the Cauchy stress components are first expressed in Cartesian coordinates ($X_1, X_2, X_3$) and then mapped to the current material orientations.

The deformation gradient can be decomposed into the product of an orthogonal rotation tensor, $\mathbf{R}$ and the right stretch tensor $\mathbf{U}$ as $\mathbf{F} = \mathbf{RU}$. Since $\mathbf{R}^T \mathbf{R} = I$, we can write

$$\mathbf{F}^T \mathbf{F} = (\mathbf{R} \mathbf{U})^T (\mathbf{R} \mathbf{U})$$

$$= \mathbf{U}^T \mathbf{U}. \quad (34)$$
The right stretch tensor \( \mathbf{U} \) is solved for by a spectral decomposition of the symmetric tensor \( \mathbf{F}^{\top}\mathbf{F} \) such that
\[
\mathbf{U} = \sqrt{\mathbf{W}_i} \mathbf{Q}_i \otimes \mathbf{Q}_i \tag{35}
\]
where each \( \mathbf{Q}_i \) is an eigenvector of \( \mathbf{F}^{\top}\mathbf{F} \), and \( \sqrt{W_i} \) is the corresponding eigenvalue. Due to the symmetric nature of \( \mathbf{F}^{\top}\mathbf{F} \), all eigenvectors \( \mathbf{Q}_i \) are orthogonal to each other. Subsequently, the rotation matrix can be solved for via
\[
\mathbf{R} = \mathbf{F} \mathbf{U}^{-1}. \tag{36}
\]
The Cauchy stresses about the current material axes are therefore
\[
\sigma_c = \mathbf{R}^{\top} \mathbf{\sigma} \mathbf{R}, \tag{37}
\]
with \( \mathbf{\sigma} \) expressed in the global Cartesian basis.

3. Numerical benchmarks and discussions

In this section, several studies are conducted to validate the nonlinear VKCS model. Section 3.1 validates the nonlinear formulation with two thin shell examples from the literature. Then, Section 3.2 showcases the VKCS model capability to switch between different configurations at ease, i.e. with varying orders of planar and through-thickness kinematics, followed by a discussion of the effects of an orthotropic kinematic refinement scheme on the solver performance.

The following conventions are used to indicate the kinematic settings in the VKCS element: the models are denoted as ‘inplane-\( N_1 \), ESL-\( N_2 \)’ or ‘inplane-\( N_1 \), LW-\( N_2 \)’. The parameters \( N_1 \) and \( N_2 \) refer to the polynomial orders of the planar and through-thickness approximations functions, respectively. Suffixes ESL and LW indicate Equivalent Single Layer and Layer-Wise expansions, respectively.

In Equivalent Single Layer models, the through-thickness material properties are accounted for in a homogenised sense, via weighted averaging of all the layers. In Layer-Wise models, each material layer is explicitly discretised with finite elements and the displacement fields are separately treated for each ply or ply blocks.

3.1. Large displacement analyses of thin shells

Two popular thin shell literature benchmarks that feature large displacements and rotations are studied, namely the pull-out of an isotropic cylindrical shell [35] and a pinched laminated hyperboloid shell [36]. The isotropic cylindrical shell has radius \( R = 4.953 \), length \( L = 10.35 \) and thickness \( h = 0.094 \), with material properties \( E_{11} = 10.5 \times 10^6 \) and \( \nu = 0.3125 \).

The geometric parameterisation of the hyperboloid shell is
\[
\mathbf{X}(r_1, r_2) = \begin{bmatrix} R_0 \cos (r_1 \pi/2) \\ R_0 \sin (r_1 \pi/2) \\ L r_2 \end{bmatrix} \quad \text{where} \quad 0 \leq r_1 \leq 1 \text{ and } 0 \leq r_2 \leq 1,
\]
\[
R_0(r_2) = R_1 \sqrt{1 + \left( \frac{L r_2}{C} \right)^2} \quad \text{where} \quad R_1 = 7.5, \quad C = \frac{20}{\sqrt{3}}, \quad L = 20 \quad \text{and} \quad h = 0.04,
\]
with a layup of \( (90^\circ/0^\circ/90^\circ) \), where the \( 0^\circ \) fibres are in the hoop direction. The material properties are \( E_{11} = 40 \times 10^6, \ E_{22} = E_{33} = 1 \times 10^6, \ G_{12} = G_{13} = G_{23} = 0.6 \times 10^6, \ \nu_{13} = \nu_{12} = 0.25 \) and \( \nu_{23} = 0.25 \).
The loading conditions for both cases are shown in Figure 3(a) and 4(a), respectively. The applied loads increase monotonically in both benchmarks, therefore the nonlinear systems can be solved using a modified Newton-Raphson method. Higher-order in-plane functions are used to circumvent locking, as well as to capture the significant bending deformation in both cases.

The model settings are inplane-4, ESL-2 with a mesh density of $10 \times 10$, and inplane-3, ESL-2 with a mesh density of $20 \times 20$ for the pulled-out cylinder and the pinched hyperboloid, respectively. The solutions agree well with the reference solution from the literature, as shown in Figure 3(b) and Figure 4(b), respectively.

### 3.2. Convergence properties of models orthotropic h- and p- kinematic refinement

The first part of this section aims to showcase the versatility of the VKCS model in its ability to switch between different kinematic configurations easily. The second part will investigate the effects of the refinement scheme on the nonlinear solver performance.

The test case is a thin laminated shallow roof subjected to a point load, which was also studied by Payette et al. [25] and Rivera et al. [26]. The geometrical parameters for the shallow roof are $R = 2540$, $z = 508$, $\alpha = 0.2$ and $h = 6.25$, as shown in Figure 5. The layup is ($-45^\circ/45^\circ/-45^\circ/45^\circ$) from the bottom to the top, where the 1-direction corresponds to the hoop direction. The material properties are $E_{11} = 3300$, $E_{22} = E_{33} = 1100$, $G_{12} = G_{13} = 660$, $G_{23} = 440$ and $\nu_{13} = \nu_{12} = \nu_{23} = 0.25$.

With a width-to-thickness ratio of $> 80$, the through-thickness distortion has a negligible contribution to the global deformation; hence all the analyses use ESL models with quadratic through-thickness expansion function. This model is denoted as ESL-2, where the 2 indicates quadratic thickness function. Pinned-pinned boundary conditions are applied to the two straight edges of the shallow roof. For a shell model that uses a quadratic (3-noded) Lagrange function for its through-thickness expansion, this boundary condition can be conveniently enforced by

---

Figure 3: (a) Pull-out loads applied on the thin isotropic cylinder. (b) Comparisons of force-displacement responses at Locations A and B. The reference solutions are from Sze et al. [35]. The colour contour indicates the magnitude of displacements.
Figure 4: (a) Pinching loads on the hyperboloid shell. (b) Comparisons of force-displacement responses at Location C. The reference solution is from Payette et al. [25]. The colour contour indicates the magnitude of displacements.

Figure 5: A laminated shallow cylindrical panel subjected to a central point load.
setting the displacements \((u_1, u_2, u_3)\) of the mid-thickness nodes to zero. This model is solved using a cylindrical arc-length solver [37] as there are several limit points in the equilibrium path.

In terms of solver settings, we scale the size of arc-length automatically according to \((I_0/I_{des})^2\), where \(I_0\) and \(I_{des}\) denote the total number of iterations in the previous increment and desired number of iterations per increment, respectively. Also, if five consecutive increments require the same number of iterations for convergence, the solver will quadruple the size of arc-length and initiate a full-newton procedure for the subsequent step, where the tangent matrices are recomputed for every iteration. The L-2 norm convergence criterion for the solution vector is \(\epsilon = 1 \times 10^{-3}\).

The parametric study looks at models with \(p\)-level of 1–6, where the mesh densities are increased gradually for each in-plane polynomial order until the solution converges. Note that in the absence of analytical solution for the nonlinear problem, it is non-trivial to measure the errors objectively using the reference numerical solution. In this study, the solution is considered converged when the solution has shown good overall correlation with the reference solution, especially in the parts of equilibrium path with high degree of nonlinearity, i.e. between displacement = 12.5 to 17.5 in Figure 6.

The converged mesh density for each in-plane order can be identified. For instance, converged mesh densities for in-plane orders 1, 2, 3, 4, 5 and 6 are 196 \times 196, 16 \times 16, 8 \times 8, 6 \times 6, 4 \times 4 and 2 \times 2, respectively. The total dofs, number of increments, number of iterations and CPU runtime are shown in Table 1 for comparison in the numerical performances of these models.

There are several key observations. Firstly, the linear elements require very fine mesh and computational cost for convergence due to locking. The required model dofs reduce as the in-plane order increases, i.e. the inplane-6 model requires only 1443 dofs, which is 7 times fewer than the inplane-2 model that needs 9603 dofs for convergence. However, the reduction in the model dofs does not necessarily translate into computational savings. The inplane-2 and inplane-3 models appear to be the most computationally efficient for convergence in displacement, despite the higher model dofs compared to inplane-6 model, as shown in Table 1. In a static geometrically nonlinear analysis, most of the computational cost is due to the re-assembly of the tangential stiffness matrix for every increment, followed by the computation of elastic nodal forces for every iteration. The factors that determine the cost of these operations are the number of elements, integration points per element and operations per integration point. The number of integration points and the number of operations per integration point increase with the element order, drastically increasing the computational cost per element.

For general analysis, the VKCS model provides a convenient framework—to conduct the parametric study as demonstrated in this section—in search of model configurations that offer a good trade-off between accuracy and computational cost given a nonlinear problem. It is especially useful for structural optimisation problems where large number of simulations are required.

The bottom-most right plot in Figure 6 shows the structural deformations at selected points on the equilibrium path. The shallow roof shows bend-twist coupling due to the asymmetric layup. The asymmetric deformations become more pronounced between the peak and troughs of the force-displacement curve. These modes are difficult to observe in real experiments because the equilibrium points are unstable. For a more detailed exposition of unstable equilibrium paths, the reader is referred to Groh and Pirrera [38].

3.3. 3D Cauchy stress tracking in a thick laminated roof

In a large displacement analysis, a structure must be checked for material failures throughout the operational equilibrium path. The localised 3D stresses arising from the local instability can
Figure 6: The model responses for VKCS model with in-plane polynomial order 1–6 at various mesh sizes. All the models have ESL through-thickness kinematics with $p$-level of 2. The structural configurations are shown for selected points on the equilibrium paths for inplane-6, $2 \times 2$. The reference solution is obtained from Payette et al. [25]. The contour indicates the displacement magnitudes.
Table 1: Number of incremental steps and the total dofs of models with converged solution in the thin cylindrical panel benchmark.

<table>
<thead>
<tr>
<th>Order</th>
<th>Mesh</th>
<th>Increments</th>
<th>Iterations</th>
<th>Dofs</th>
<th>CPU time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>196 × 196</td>
<td>160</td>
<td>633</td>
<td>114072</td>
<td>68754</td>
</tr>
<tr>
<td>2</td>
<td>16 × 16</td>
<td>161</td>
<td>632</td>
<td>9603</td>
<td>3852</td>
</tr>
<tr>
<td>3</td>
<td>8 × 8</td>
<td>161</td>
<td>631</td>
<td>5475</td>
<td>4098</td>
</tr>
<tr>
<td>4</td>
<td>6 × 6</td>
<td>162</td>
<td>635</td>
<td>5475</td>
<td>6745</td>
</tr>
<tr>
<td>5</td>
<td>4 × 4</td>
<td>160</td>
<td>624</td>
<td>3843</td>
<td>9325</td>
</tr>
<tr>
<td>6</td>
<td>2 × 2</td>
<td>161</td>
<td>628</td>
<td>1443</td>
<td>5642</td>
</tr>
</tbody>
</table>

$Z$

$Y$

$X$

$p = 250 \times 10^3$

$h = 150 \times 10^{-6}$

Figure 7: Schematic of the geometry, loading and boundary conditions of the semi-infinite laminated flat plate.

Figure 8: The linear and nonlinear $\sigma_{13}$ solutions of the semi-infinite flat plate subjected to uniformly distributed load, at two different locations. The reference solution is obtained from Hartman et al. [39].
initiate local and structural failures, and hence must be accurately tracked throughout nonlinear analyses. To obtain accurate interlaminar stresses, models with refined kinematics often must be used everywhere since it is difficult to qualitatively predict the locations where local instability may initiate. In some cases, this class of problems is analysed first using low fidelity models, followed by the ‘indirect’ evaluation of interlaminar stresses as a post-processing step. For instance, Park et al. [40] utilises a nonlinear predictor-corrector approach as a post-processing step to recover accurate transverse stresses using the equilibrium equations. Lee et al. [41] utilises a non-iterative approach to recover transverse stresses by using the in-plane and transverse shear stresses. They first approximate a transverse stress field, whose error is reduced via minimisation of a least square functional at the element level. Later, Hartman et al. [39] proposed an iterative stress recovery technique with application to laminated plates, which is then extended to include inertial effects in a later publication [42]. These works demonstrated considerable savings in the computational time for accurate interlaminar stresses. However, the recovered stresses cannot be used as inputs to update the geometric stiffness matrix, and as a result present difficulties in the analysis of very thick laminates or sandwich structures where the interlaminar stresses contribute significantly to the global responses. In these instances, it is necessary to use models that output accurate 3D stresses directly from the get-go.

Firstly, the 3D stress accuracy of the VKCS model is validated with a simple benchmark from Hartman et al. [39], as shown in Figure 7. It shall be noted that the benchmarks for nonlinear Cauchy stresses are scarce in the literature, and mostly focus on laminated flat plates [39–42] that are 2D in nature (plane stress/strain).

Hartman et al. ’s benchmark is a semi-infinite flat plate with 8 plies, subjected to uniformly distributed transverse pressure across a band in the centre. The plate is simply supported along the edges. The material properties are $E_{11} = 155.0 \times 10^9$, $E_{22} = 12.1 \times 10^9$, $E_{33} = 12.1 \times 10^9$, $G_{23} = 3.2 \times 10^9$, $G_{13} = 4.4 \times 10^9$, $G_{12} = 4.4 \times 10^9$, $\nu_{23} = 0.458$, $\nu_{13} = 0.248$ and $\nu_{12} = 0.248$. The laminate layup is ($0^\circ/90^\circ/0^\circ/90^\circ$)$_8$, where the $0^\circ$ plies align with the $Y$ axis. The infinite width along $Y$ indicates a plane strain condition, where $\frac{\partial u}{\partial y} = 0$. As an approximation, the condition is enforced by setting $u_y = 0$ for every node in the model. The VKCS solution is obtained using 20 elements in the width ($X$-direction), with model inplane-4, LW-2. The $\sigma_{13}$ are measured through-thickness at locations $X = 0.1$ and $X = 0.9$. As shown in Figure 8, VKCS solution is in good agreement with the benchmark.

A new benchmark is also proposed to showcase the accuracy of the VKCS model, namely a thick laminated roof under large displacement. The shell geometry, except the thickness, is the same as the one described in Section 3.1 in Figure 5. The material properties are $E_{11} = 3300$, $E_{22} = E_{33} = 1100$, $G_{12} = G_{13} = 660$, $G_{23} = 440$ and $\nu_{12} = \nu_{13} = \nu_{23} = 0.25$. The shell thickness is 25.4 units with bottom-to-top layup of ($0^\circ/90^\circ/0^\circ/90^\circ$), where $0^\circ$ denotes the hoop direction. It is subjected to a central point load of 20 000, and has pinned-pinned boundary conditions along the straight edges. The benchmark is designed to exhibit no limit point, therefore only a Newton-Raphson solver is required to trace the equilibrium path. To improve the robustness of the solution procedure, the modified Newton-Raphson solver switches to a full Newton scheme under two conditions: when the number of iterations exceeds 10, or when the L-2 convergence criterion $\epsilon \geq 1$ at any iteration. Total increment steps of 20 are prescribed. To provide a benchmark solution, a ‘twin’ model is created in ABAQUS using 40 000 quadratic fully integrated solid elements. This level of discretisation is determined by gradually increasing the solid mesh density in both in-plane and through-thickness directions, until the through-thickness distributions of all the Cauchy stresses show no significant overall changes at the selected sampling location.
For convergence in the displacement responses, the VKCS model with inplane-5, ESL-2 is sufficient. Again, since the reference solution comes from a numerical model, it is non-trivial to know exactly when a set of solution is converged. Here, the VKCS solution is considered converged when it shows good correlation with the reference solution on the force-displacement figures. In Figure 9, the model kinematics in both directions are varied from 1 to 6 and 1 to 4 for both ESL and LW expansions, respectively. The mesh density is $2 \times 2$. For this particular test case, there are no significant differences between the model solutions from LW-1, LW-2, ESL-2 and ESL-4, because the through-thickness deformation does not contribute significantly to the global response.

The converged Cauchy stress solutions for $P = 4000$, $P = 10000$, $P = 16000$ and $P = 20000$ at the location $X_1 = R \cos(-0.05)$, $X_2 = R \sin(-0.05)$, $X_3 = -0.25 L$ are shown in Figure 10. It should be noted that each of the stress components converges at different model kinematics. For mesh density of $10 \times 10$, $\sigma_{13}$ and $\sigma_{23}$ converge at in-plane order 4 and layer-wise expansion order 2, whereas $\sigma_{33}$ converges at in-plane order 6 and layer-wise expansion order 3. The $\sigma_{33}$ solutions for different models at $P = 20000$ are shown in Figure 11. A comparison between the LW-3 models at inplane-4, inplane-5 and inplane-6 shows that accurate $\sigma_{33}$ solutions also require high continuity in the in-plane domain. On the other hand, the $\sigma_{33}$, $\sigma_{13}$ and $\sigma_{23}$ do not converge at all for any order of ESL kinematics since ESL models do not satisfy the $C^0$ condition [43].

Overall, the VKCS solutions show good agreement with the reference Abaqus solution across all the load levels. Due to the orthotropic higher-order kinematic refinement, the VKCS model obtains accurate stresses at a much smaller model size as compared to the solid 3D-FEM. The Abaqus model requires dof of 517,599 for convergence in stresses, as compared to higher-order variant of the VKCS model with kinematics inplane-4, LW-3 and total dofs of 55,449 at a mesh density $10 \times 10$. In this benchmark, the signs of $\sigma_{11}$, $\sigma_{22}$ and $\sigma_{33}$ reverse going from $P = 4000$ to $P = 10000$, due to the inversion of the shell curvature past the ‘zero-position’, where the tension and compression sides reverse through the thickness of the laminate. Nevertheless, the overall stress distribution does not change considerably throughout the analysis.

Also, since localised nonlinearities do not develop, global bending dominates the nonlinear deformation mode of the thick shell. In such cases, analysts only need to carry out a linear analysis to check for appropriate level of discretisation, because once the stress solution has converged for one load step, it will also converge for all other load steps, similar remarks were also made by Angioni et al. [44].

The number of iterations in each step required by selected models that have converged displacements are shown in Figure 12. Interestingly, all discretisations require almost the same number of iterations for each step. Therefore, the $h$- and $p$-levels in either direction and the expansion type do not affect the solver’s performance.

3.4. Post-buckling of a wind turbine blade substructure

This section studies the post-buckling response of a wind turbine blade (WTB) sub-structure, i.e. the trailing edge (TE) panels. WTBs undergo complex combinations of aeroelastic and inertial loads under real operating conditions. The loads can be simplified into compressive loads applied in the spanwise direction, which eventually cause the panels to buckle (also known as ‘breathing’) and to lose load-carrying capability. In the literature, most modelling works on TE panel breathing do not disclose material properties, layups, and detailed blade geometry. The proposed benchmark in this section is designed to show comparable responses to experimental results reported in the literature. The geometry, loading and boundary conditions used in the benchmark are based on the works from Danmarks Tekniske Universitet (DTU) [45–47].
Figure 9: Force displacement response of the VKCS models at mesh density of $2 \times 2$ for the problem in Section 3.3. The reference solution is obtained using Abaqus.

Figure 10: Through-thickness 3D stress solution at the location $X_1 = R \cos(-0.05), X_2 = R \sin(-0.05), X_3 = -0.25L$. The reference solution is obtained from Abaqus. The VKCS solution is obtained from the model with mesh density $10 \times 10$, inplane-6, LW-3.
Figure 11: The convergence of $\sigma_{33}$ in the shallow roof stress benchmark at mesh density of $10 \times 10$. The load level is at $P = 20\,000$. The reference solution is obtained from an ABAQUS model.

Figure 12: Number of iterations in each step at different converged discretisation.
The WTB model is a prismatic airfoil section with the top and bottom panels adhesively bonded at the trailing edge. Real WTBs have different layups across the panels, and their thickness reduce drastically at the bondline area. Some simplifications are made in the proposed model, such that the same layups are assumed everywhere and the panel thickness linearly tapers along $X_1$: from $h = 0.025$ at $X_1 = 0.3928$ to $h = 0.005$ at $X_1 = 1.0$. The section length is 1 unit, and the airfoil profile is FFA-W3-241, obtained from Björck [48]. Assuming that the box spar provides rigid support in edgewise compression, the model only includes the airfoil section beyond 40% chord length. Also, the panels are clamped along the span at 40% chord length. Numerical experiments show that the direct application of compressive pressure at locations $X_3 = 0$ and $X_3 = 1$ can cause premature panel buckling. Hence, the model has wood clamp boundary conditions on 10% either ends of the section to restrict any out-of-plane displacements, where the mid-surface nodes have constraints $u_1 = u_2 = 0$, whilst $u_3$ is free.

The bondlines are assumed rigid. There are two modelling options [45], i.e. tying the top and bottom panels with Multi-Point-Constraint (MPC), or explicitly modelling the bondline with solid elements. The proposed model assumes the former since the elastic properties of the bondlines are not readily available. For each mid-surface node on the bottom panel in the bondline region, a rigid MPC is imposed between itself and the mid-surface node directly above it on the top panel. The technique is similar to a static condensation procedure, where the tied dofs are ‘lumped’ via a transformation matrix. It is based on a Master-Slave MPC approach. An example is shown below to demonstrate the technique. Assuming that $u_2$ is a master to $u_3$, in matrix notation:

$$
\begin{bmatrix}
u_1 \\
u_2 \\
u_3
\end{bmatrix} =
\begin{bmatrix}
1 & 0 \\
0 & 1 \\
0 & 1
\end{bmatrix}
\begin{bmatrix}
\hat{u}_1 \\
\hat{u}_2 \\
\hat{u}_3
\end{bmatrix}.
$$

(38)

The condensed stiffness matrix and force vector are obtained with a congruential transformation as follows:

$$
\hat{K} = T^T K T,
\hat{f} = T^T f,
$$

(39)

where $K = K^T + K^G$ and $f = f_{NL}$ as in Equation (29) and (30) respectively. The system of equations to solve are

$$
\hat{u} = \hat{K}^{-1} \hat{f}.
$$

(40)

where Equation (38) recovers the full displacement vector.

A ‘twin’ model is created in *Abaqus*, where the panels are modelled with S8R elements. As for the bondline, each pair of mid-surface nodes on the top and bottom panels in the bondline region is tied using the MPC-Pin setting. This option only ties the three displacement dofs, hence their rotational dofs are unconstrained. A snapshot of the MPC constraints in the *Abaqus* model is shown in Figure 14. Numerical experiments showed that the nonlinear response is very sensitive to which through-thickness dofs are tied in the bondline. The tie constraints are applied only to the mid-surface displacements in both the *Abaqus* and VKCS models to simplify the model setup.

The VKCS model is first verified with *Abaqus* in a linear analysis with unit load. The VKCS mesh configuration is shown in Figure 15(c) with ESL-2 expansion; whereas the *Abaqus* S8R model has mesh density $200 \times 100$ and assumes first-order shear deformation. Figure 16 com-
Figure 13: Model setup of the trailing edge panels. The loading and boundary conditions applied to different regions are colour-coded.

pare the spanwise out-of-plane displacement \( u_2 \) along \( X_1 = 0.6052 \) and \( X_1 = 1.0 \), where good agreements are found between both models.

The nonlinear problem is solved using an arc-length solver with the same settings as in Section 3.2. Two mesh densities, i.e. Mesh 1 and Mesh 2 as shown in Figure 15(a) and 15(b), respectively are used in the convergence study. The in-plane orders are parametrically varied from 1 to 6 for both meshes. The VKCS models converge at in-plane order 6 for Mesh 1, and in-plane order 5 for Mesh 2. The model dofs are 14 640 and 17 250 respectively. The ABAQUS model has total dofs of 139 389. The dofs comparison between VKCS and ABAQUS is consistent with our findings in Section 3.2, where models with higher \( p \)-level converge in the nonlinear analyses at much lower model dofs compared to models with lower \( p \)-level.

The proposed benchmark has comparable responses to experimentally measured data reported by Lahuerta et al. [47]: the linear response is followed by a global stiffness reduction (in compression) in the post-buckling regime. The deformation modes at different loading stages are shown in Figure 17. The global bending mode dominates the linear response on one side of the panel. Approaching the limit point, buckling waves with high amplitudes appear in the centre of the panels. As the structure passes through the limit point, the high amplitude buckling waves eventually migrate to the trailing edge.

4. Summary and future work

The overarching aim of our work is to develop a general shell element whose kinematics in the in-plane and thickness domains can be tailored for the modelling of a wide range of structures, thereby enabling accurate nonlinear displacement and 3D Cauchy stress responses to be captured with minimally required dofs. In this paper, we have derived a Variable-Kinematics Continuum Shell (VKCS) formulation that is nonlinear in a total Lagrangian sense. The novelty
Figure 14: Schematic of a ‘twin’ model created in Abaqus. Assuming a rigid bondline, each pair of nodes (aligned in the Y-direction) is tied using the MPC-Pin function.

Figure 15: The mesh configurations used in the WTB trailing edge panels post-buckling model.
Figure 16: Spanwise out-of-plane displacements of the trailing edge panels under a unit load in a linear analysis. All models have through-thickness kinematics ESL-2. The reference solution is obtained using S8R in ABAQUS.

Figure 17: Nonlinear force-displacement responses of the wind turbine blade trailing edge panel subjected to compressive edge pressure, for VKCS models with different kinematic settings. All models have ESL-2 through-thickness kinematics. The reference solution is obtained using S8R in ABAQUS.
is in the application of CUF notations to a tensor-based continuum shell formulation that makes no simplifying assumptions on the element geometry. The resulting model is a shell element with completely generalised geometric and kinematic descriptions. The nonlinear VKCS model offers a convenient framework, such that the users can control the discretisation and modelling errors in the in-plane and through-thickness domains by simply tuning some free parameters. Within the VKCS framework, the kinematic configurations of the analysis models are free parameters that can be customised on a case-by-case basis, so as to create models that can capture the physics in the problems at hand with minimal computational effort. Through this work, we are advocating for an ‘all-in-one’ modelling paradigm using the present variable-kinematics formulation, in order to flatten the hierarchies of models in design analysis. Essentially, with a variable-kinematics formulation, the same model can be used throughout the design process as it already encapsulates many of the low and high fidelity configurations that can be accessed by specifying the orders of the planar and thickness basis functions. This, in turn eradicates the need to build a multitude of models for different design stages, and thus has the potential to reduce labour costs significantly.

Selected thin shell benchmarks in the literature are studied to validate the accuracy of the VKCS model. This study demonstrates that generally higher-order VKCS models are more computationally efficient compared to the low-order models, in terms of fewer model dofs and shorter computational runtime. However, the computational advantages can diminish for model orders 4, 5 and 6, as the computational cost per integration point in higher-order elements overcomes their faster convergence rates.

This paper also proposed a new nonlinear 3D stress benchmark, where the VKCS model is used to track the evolution of 3D Cauchy stresses in a thick laminated shell. The benchmark showed that the higher-order VKCS models are accurate in 3D stresses compared to the reference for all the load levels, and require much fewer total dofs for solution convergence. Lastly, the formulation is applied to a more industrially relevant benchmark, namely the post-buckling response of the trailing edge panels of a wind turbine blade. The resulting nonlinear response correlates qualitatively well with the experimental results reported in the literature.

5. Acknowledgement

The first author A.K.W. Hii would like to acknowledge the funding support by Rolls-Royce plc and the EPSRC through the EPSRC Centre for Doctoral Training in Advanced Composites for Innovation and Science [Grant No. EP\L016028\1]. S. Minera was funded by The H2020 Marie Skłodowska-Curie European Training Network [Grant No. R117251-101]. R.M.J. Groh was funded by the Royal Academy of Engineering under the Research Fellowship scheme [Grant No. RF\201718\17178]. A. Pirrera was funded by the EPSRC through the Early-Career Fellowship scheme [Grant No. EP\M013170\1] and the Wind Blades Research Hub (WBRH), a joint collaboration between the University of Bristol and ORE Catapult. L.F. Kawashita was supported by Rolls-Royce plc.

6. Bibliography

References