



Politecnico di Bari

Repository Istituzionale dei Prodotti della Ricerca del Politecnico di Bari

Development and validation of new adaptive finite elements for complex aerospace problems

This is a PhD Thesis

Original Citation:

Development and validation of new adaptive finite elements for complex aerospace problems / Rubino, Andrea. - ELETTRONICO. - (2026).

Availability:

This version is available at <http://hdl.handle.net/11589/299080> since: 2026-03-25

Published version

DOI:

Publisher: Politecnico di Bari

Terms of use:

(Article begins on next page)



Department of Mechanics, Mathematics and Management

AEROSPACE SCIENCE AND ENGINEERING

Interuniversity Ph.D. Program - Cycle XXXVII

Coordinator of Ph.D. Program: Prof.ssa Caterina Ciminelli

SSD: ING-IND/04 – AEROSPACE STRUCTURES AND DESIGN

Final Dissertation

Development and validation of new adaptive finite elements for complex aerospace problems

Ph.D. Candidate:

Andrea Rubino

Supervisor:

Prof.ssa Maria Cinefra

La ricerca è stata realizzata con il cofinanziamento dell'Unione europea - POC Puglia 2014-2020
- Asse X - Azione 10.4 "Interventi volti a promuovere la ricerca e per l'istruzione universitaria"
Approvazione Avviso pubblico n. 1/POC/2021 "Dottorati di ricerca in Puglia XXXVII Ciclo"



Acknowledgments

Abstract

In aerospace engineering, the structural analysis of aircraft, spacecraft, and related components requires highly accurate numerical models to predict behaviour under various operational conditions. Traditional finite element methods (FEM) rely on standard mesh structures, which may not always provide the optimal geometry for capturing the complexity of certain shapes or stress distributions. To address this, adaptive meshes offer a way to design non-conventional, flexible elements that conform more effectively to the geometry and loading conditions of aerospace structures. These meshes are not automatically generated but are specifically designed to adapt to the particular requirements of the problem, offering greater accuracy and efficiency.

The ability to tailor the mesh geometry to fit complex structural forms or specific stress regions is crucial for optimizing simulations in aerospace applications. By employing non-conventional mesh designs, engineers can better capture critical phenomena such as stress concentrations around joints, load-bearing members, or complex curvatures found in modern aerospace structures. This approach allows for a more precise representation of the physical model without a substantial increase in computational resources.

In this research, we will incorporate Carrera Unified Formulation (CUF), a versatile and innovative framework for structural modelling. CUF allows for the creation of custom, adaptable elements that respond to the structural complexities unique to aerospace components. By combining CUF with adaptive mesh strategies, the objective is to develop mesh configurations that are highly specialized, efficiently representing regions with different requirements, such as thin shells or complex junctions, where traditional meshes would fail to deliver the needed precision.

The development of adaptive meshes in aerospace structural analysis offers significant advantages across a variety of applications. In aircraft fuselage and wing structures, these meshes can conform to complex curvatures and junctions, improving the accuracy of stress distribution analysis, particularly in critical areas such as wing-fuselage connections. For landing gear components, non-conventional elements can be designed to adapt to the highly loaded and geometrically intricate parts, enhancing predictions for fatigue life and overall structural performance.

In spacecraft structures, adaptive meshes can be tailored to the lightweight and complex designs typical in space applications, ensuring precise load distribution during both launch and orbital conditions. Additionally, for composite materials, adaptable mesh elements can better account for the heterogeneous nature of these materials, providing more accurate predictions of failure points and material behavior, which is essential for ensuring the reliability and safety of aerospace components. Another key application lies in fluid-structure interaction (FSI), where adaptive meshes can dynamically conform to the evolving interface between fluids and structures, capturing the complex interplay of aerodynamic loads on structural components, thus improving the accuracy of simulations in scenarios such as wing flutter, aerodynamic shape optimization, and structural response under varying flow conditions.

This research aims to push the capabilities of structural simulation by using adaptive mesh strategies that are tailored to aerospace-specific challenges. By developing more efficient and accurate models, it will contribute to safer and more advanced aerospace structures, capable of meeting the stringent performance requirements of the industry.

Contents

1	Introduction	1
1.1	Context and Motivation	1
1.1.1	Importance of Structural Analysis in Aerospace Engineering	2
1.1.2	The Need for Highly Accurate Numerical Models in Aerospace Structural Analysis	3
1.1.3	Limitations of Traditional Finite Element Methods (FEM) with Standard Mesh Structures	5
1.1.4	Introduction to Adaptive Meshes and Their Advantages in Structural Analysis	7
1.1.5	Research Objectives	7
1.2	Literature review	8
1.2.1	Main Trends and Open Challenges	23
2	Theoretical framework	25
2.1	Introduction	25
2.2	1D and 2D models based on Carrera Unified Formulation	27
2.2.1	Taylor Expansions	29
2.2.2	Legendre Expansions	30
2.2.3	Lagrange Expansions	32
2.3	Finite Element formulation	33
2.3.1	Geometrical Relations and Constitutive Equations	33
2.3.2	Finite Element Approximation of Displacements	34
2.3.3	Node-Dependent Kinematics (NDK) Approach	36
2.3.4	3D modelling for non-orthogonal geometries	37
2.3.5	Jacobian computation for Node-Dependent Kinematic	42
2.3.6	Governing Equations	48
2.3.7	Finite Element Discretization and Structural Modeling	50
2.3.8	Computational Advantages	51

3	Numerical Results and Discussion	52
3.1	Computational Setup and Reference Solvers	52
3.2	Overview of the Investigated Problems	56
3.3	Adaptive Mesh: Preliminary Free-Vibration Assessment	57
3.3.1	The model	57
3.3.2	Results	58
3.4	Cross-Model Free-Vibration Validation and Scaling Assessment . . .	61
3.4.1	CUF–NDK Reduced-Order Elements versus Classical 3D Solids	62
3.4.2	Cross-Model Frequency Comparison	67
3.4.3	Scaling Indicator S_k and Cross-Model Regime Transition . .	82
3.5	Static Analyses	88
3.5.1	Benchmark Plates under Concentrated Loads	88
3.5.2	Reduced and Adaptive Mesh Strategies	95
3.5.3	Influence of Mesh Distortion	99
3.6	Dynamic Analyses	107
3.6.1	Free-Vibration Analysis of Plates	107
3.6.2	Square plate with distorted mesh	110
3.6.3	Free-Vibration Analysis of Variable-Thickness Rotors	116
3.7	Comparative Discussion	131
3.7.1	Static vs Dynamic Behaviour	131
3.7.2	Efficiency in Terms of Accuracy and Degrees of Freedom . .	131
3.8	Concluding Remarks on the Numerical Results	132
3.8.1	Additional exploratory investigations	133
4	Conclusions	135
4.1	Summary	135
4.1.1	Key Insights and Contributions	136
4.1.2	Future works	137
	References	139

List of Figures

3.3.1 Comparison for the first mode	59
3.4.1 1D finite element mesh with bidimensional cross-sectional expansion.	65
3.4.2 2D finite element mesh with monodimensional through-thickness expansion.	66
3.4.3 Classical 3D volumetric solid discretization.	66
3.4.1 Frequency comparison: 1D_element_GENE vs 1D_element_NONE (cases 1–6).	71
3.4.2 Frequency comparison: 1D_element_GENE vs 1D_element_NONE (case 7).	72
3.4.3 Frequency comparison: 2D_element_GENE vs 2D_element_NONE (cases 1–6).	73
3.4.4 Frequency comparison: 2D_element_GENE vs 2D_element_NONE (case 7).	74
3.4.5 Frequency comparison: 2D_element_GENE vs 1D_element_GENE (cases 1–6).	75
3.4.6 Frequency comparison: 2D_element_GENE vs 1D_element_GENE (case 7).	76
3.4.7 Frequency comparison: 2D_element_GENE vs 3D_element (cases 1–6).	77
3.4.8 Frequency comparison: 2D_element_GENE vs 3D_element (case 7).	78
3.4.9 Frequency comparison: 1D_element_GENE vs 3D_element (cases 1–6).	79
3.4.10 Frequency comparison: 1D_element_GENE vs 3D_element (case 7).	80
3.4.1 Coefficient of variation (CoV) of S_k across geometries for 1D_element_GENE, 2D_element_GENE and 3D_element	83
3.4.2 Heatmaps of S_k across geometries (rows) and mode index k (columns) for 1D_element_GENE, 2D_element_GENE and 3D_element	85
3.4.3 Median S_k split by aspect ratio group (AR=1 vs AR>1) for 1D_element_GENE, 2D_element_GENE and 3D_element	86
3.4.4 S_k versus aspect ratio for selected mode indices, shown for 1D_element_GENE, 2D_element_GENE and 3D_element	87
3.5.1 Square plate with concentrated force	90
3.5.2 Square plate	91

3.5.3 Square plate - Comparison between Nastran and MUL2 code with Q16 plate elements and cubic expansion along thickness	92
3.5.4 Square plate - Percentage error between Nastran and MUL2 code	93
3.5.5 Razzaque's skew plate	94
3.5.6 Razzaque's skew plate - Displacement field from Nastran	94
3.5.1 B3 beam element (black nodes along beam axis) connecting Q9 and Q16 plate elements (white nodes on beam cross-section)	95
3.5.2 Reduced mesh of quarter-plate model	96
3.5.3 Square plate - Comparison between full and reduced mesh	96
3.5.4 Reduced mesh of Razzaque's skew plate	97
3.5.5 Razzaque's skew plate - Displacement comparison between full and reduced mesh	98
3.5.6 Razzaque's skew plate - Percentage displacement error	99
3.5.1 Quarter plate approximation	101
3.5.2 Quarter plate - original mesh	102
3.5.3 Quarter plate - 2xQ4 1xB2	103
3.5.4 Quarter plate - 2 different configurations	103
3.5.5 Quarter plate - distorted element configurations	104
3.5.6 Quarter plate - distorted mesh defined by parameter s	104
3.5.7 Variation of the ratio U_z/U_z^0 for CC quarter plate	105
3.5.8 Variation of the ratio U_z/U_z^0 for SS quarter plate	106
3.6.1 Quarter plate - modal analysis for different configurations	108
3.6.2 Quarter plate CC - modal analysis for different values of s	109
3.6.3 Quarter plate SS - modal analysis for different values of s	110
3.6.1 Square plate - original regular meshes	111
3.6.2 Square plate - original regular meshes	111
3.6.3 Square plate - distorted meshes	113
3.6.4 Square plate - static analysis	114
3.6.5 Square plate - static analysis	115
3.6.6 Square plate - modal analysis	116
3.6.1 Disk geometry	118
3.6.2 Axis discretization	118
3.6.3 Cross-section discretization	119
3.6.4 Proposed solution for rotor discretization by Filippi et al.	120
3.6.5 New proposed solution with smoother profile	121
3.6.6 Static analysis - Displacements	122
3.6.7 Reference meshes for comparison	123
3.6.8 Non-null modal shapes	125
3.6.9 First 2 non-null modes for Case 1 and Case 2	126
3.6.10 3D model DOFs - 2203 TET10 vs 4xB2 90xQ9	129

3.6.1 From complexity to simplicity	130
---	-----

List of Tables

2.2.1 Taylor polynomials	29
3.3.1 First 10 natural frequencies and DOFs	60
3.3.2 Error on the first natural frequency	60
3.5.1 Tip displacement	106
3.6.1 Natural frequencies and errors	127
3.6.2 Natural frequencies and errors - Case 1 & Case 2	127

Chapter 1

Introduction

1.1 Context and Motivation

The aerospace industry continually seeks advancements in structural analysis and design methodologies to enhance performance, safety, and efficiency. A pivotal tool in this endeavor is the Finite Element Method (FEM), a numerical technique that approximates solutions to complex structural problems. FEM has become indispensable in predicting how aerospace components respond to various loads and environmental conditions, thereby reducing the reliance on physical prototypes and accelerating the design process.

In aerospace manufacturing, FEM is employed to predict the behavior of parts under structural and thermal loads, ensuring that designs meet stringent safety and performance standards. By discretizing structures into smaller, manageable elements, engineers can analyze stress distributions, deformations, and potential failure points with high precision.

A critical application of FEM in aerospace is the analysis of fluid-structure interactions (FSI), particularly concerning phenomena like flutter—a dynamic instability caused by the interaction between aerodynamic forces and structural elasticity. Accurate modeling of FSI is essential for predicting and mitigating flutter, which can have catastrophic consequences if not properly addressed. Advanced numerical methods have been developed to simulate these interactions with high accuracy and efficiency, enabling the design of safer and more reliable aircraft structures.

The integration of FEM with computational fluid dynamics (CFD) allows for comprehensive simulations that capture the complex interplay between aerodynamic forces and structural responses. This multidisciplinary approach is crucial for optimizing designs to withstand various operational conditions while minimizing weight and material usage. Moreover, the development of adaptive mesh refine-

ment techniques and advanced interpolation methods has enhanced the accuracy and efficiency of these simulations, providing detailed insights into the behavior of aerospace structures under fluid-structure interactions.

In summary, the application of FEM in aerospace engineering has revolutionized the way structures are analyzed and designed. By enabling detailed simulations of complex interactions between structural components and aerodynamic forces, FEM contributes significantly to the development of safer, more efficient, and innovative aerospace structures.

1.1.1 Importance of Structural Analysis in Aerospace Engineering

Structural analysis plays a crucial role in aerospace engineering as it ensures that aircraft, spacecraft, and related components can withstand the complex operational loads they encounter throughout their lifespan. The primary objective of structural analysis is to predict the response of aerospace structures to various loading conditions, such as aerodynamic forces, gravitational loads, thermal effects, and impact stresses [1]. Accurate structural analysis is essential for achieving the optimal balance between weight and strength, which directly affects fuel efficiency, performance, and safety.

Aerospace structures, unlike conventional mechanical structures, are subjected to extreme environmental conditions. These include rapid changes in temperature, high dynamic loads, and pressure variations, all of which necessitate a highly detailed understanding of material behavior and structural integrity [2]. The ability to model and predict these factors enables engineers to improve the performance and reliability of aerospace vehicles, ultimately contributing to advancements in the industry.

Structural Health Management (SHM) for aerospace systems increasingly relies on advanced computational methods to ensure real-time monitoring, damage detection, and control of structural integrity during operation. Central to these advancements are two finite-element-based approaches: the Smoothing Element Analysis (SEA) and the inverse Finite Element Method (iFEM). These methods are designed to process discrete sensor data—typically strain measurements from in-situ sensors such as Fiber Bragg Gratings (FBGs)—and reconstruct full-field deformation and internal load information with high accuracy and efficiency. This capability is vital for enabling active structural control in flexible aerospace structures, as well as for detecting and characterizing damage, including delaminations in composite materials.

SEA is a finite element formulation derived from a Penalized Discrete Least Squares (PDLS) variational principle. It enables the smoothing of noisy experi-

mental or numerical data while computing first-order derivatives of field variables, such as strain or displacement, over arbitrary geometries. The method enforces C^1 continuity through penalty terms and regularization, allowing for accurate reconstruction of smoothed fields even when sensor coverage is sparse or uneven. This makes SEA suitable for pre- and post-processing tasks in structural diagnostics and optimal sensor layout studies [3, 4].

The iFEM, by contrast, aims to reconstruct displacement fields from measured strains by minimizing a least-squares functional. It employs standard finite element interpolation of kinematic variables based on Mindlin plate and shell theory, ensuring that all strain-displacement compatibility relations are satisfied. Unlike direct FEM, iFEM uses no material constitutive relations in reconstructing displacements, allowing it to derive shape information independently of material properties. Once displacements are reconstructed, stresses and internal loads can be recovered through classical constitutive relations [5].

These methods are particularly suited for integration with fiber-optic sensing technologies, such as FOSS (Fiber Optic Strain System), which allow dense, lightweight, and electromagnetic-interference-immune sensor networks. When combined with iFEM or SEA, such sensor systems enable full-field shape sensing and structural anomaly detection in complex aerospace structures, including those with non-uniform geometries or embedded defects. This approach has been validated in experimental settings, showing high accuracy in reconstructing deformation fields and in detecting localized damage features like holes or delaminations, even in the presence of noise [6, 7].

1.1.2 The Need for Highly Accurate Numerical Models in Aerospace Structural Analysis

Given the complexity of aerospace structures, numerical methods have become indispensable for analyzing their behavior under different operating conditions. Computational techniques such as the Finite Element Method (FEM) provide engineers with powerful tools to simulate stresses, deformations, and failure mechanisms without the need for costly and time-consuming experimental testing [8]. However, achieving high accuracy in these simulations is critical, as even minor discrepancies can lead to significant performance issues or structural failures.

The development of FEM has its roots in early 20th-century mathematical methods, including the Ritz method [9], which approximated solutions by minimizing potential energy through trial functions, and the work of Courant [10], who introduced the use of piecewise linear functions over triangular elements. These foundational ideas were further advanced by structural engineers such as Hrennikoff [11] and McHenry [12], who used lattice and truss analogies to model

continuum behavior. The formalization of the method occurred in the 1950s with contributions from Turner [13], who applied stiffness matrix formulations to complex structures, and Clough [14], who coined the term “finite element” in the context of plane stress analysis. The method gained momentum with the rise of digital computing, which made it possible to solve the large algebraic systems resulting from domain discretization [15, 16].

The standard procedure in FEM involves several key steps: defining the governing equations and boundary conditions; discretizing the domain into finite elements; deriving local element equations based on the chosen interpolation (shape) functions; assembling the global system of equations; applying boundary conditions; solving the resulting system; and post-processing the results. Each of these stages contributes to the accuracy and reliability of the final solution. Special care must be taken during discretization to ensure that critical regions—such as those with high stress gradients or abrupt changes in geometry—are adequately resolved by the mesh.

FEM introduces several types of numerical errors that must be considered. Iteration errors occur in iterative solvers and should diminish as the solution converges. Approximation errors arise from simplifying a continuous problem using discrete elements and can be reduced by refining the mesh or using higher-order elements. Round-off errors result from the finite precision of digital computation and become significant when subtracting large, nearly equal numbers [17, 18]. Validation against known solutions or experimental data remains an essential step in assessing the accuracy of FEM results.

Among its advantages, FEM allows the analysis of domains with arbitrary shapes, supports mixed-element meshes, and can handle a wide range of loading and support conditions within a unified computational framework. Its modular structure allows different element types and formulations to be combined, and the resulting models closely resemble physical systems, improving interpretability and trustworthiness [19].

The scope of FEM continues to expand, especially in areas involving multi-physics problems such as fluid–structure interaction, thermo-mechanical and electro-mechanical coupling, and biomedical applications. Recent advancements include its use in the simulation of additive manufacturing (e.g., 3D printing), which involves complex interactions between heat transfer, phase transitions, and mechanical deformation. Moreover, cloud-based platforms like SimScale now allow users to perform FEM and CFD simulations directly through web browsers, lowering the barrier to access and enabling broader collaboration [20, 21, 22].

Traditional numerical models rely on discretization techniques to represent complex geometries and stress distributions. The accuracy of such models depends largely on the quality of the mesh used in the analysis. A finely refined mesh can

capture localized stress variations and complex deformations more effectively than a coarse mesh, but this often comes at the cost of increased computational expense [23]. Therefore, a balance must be struck between computational efficiency and model fidelity.

1.1.3 Limitations of Traditional Finite Element Methods (FEM) with Standard Mesh Structures

The conventional FEM approach employs structured or unstructured mesh grids to approximate the geometry and physical behavior of aerospace structures. These mesh structures typically consist of uniform elements, which, while effective in many cases, may not always provide an optimal representation of the actual stress distribution, especially in regions with high gradients, discontinuities, or complex boundary conditions [24].

One of the primary limitations of standard FEM meshing techniques is their inability to adapt dynamically to local variations in stress or strain. When a conventional uniform mesh is used, high levels of refinement are required throughout the entire structure to achieve the desired accuracy, leading to excessive computational costs and inefficiencies. Additionally, standard meshing strategies struggle to model certain geometric complexities found in aerospace structures, such as sharp edges, curved surfaces, and stress concentration points around joints and fasteners [23].

As highlighted by Hatheway [25], Finite Element Analysis (FEA) is a numerical tool designed to evaluate the mechanical behavior of structures, yet its effectiveness is intrinsically tied to the quality and completeness of the underlying structural model. It does not inherently construct or validate the model; instead, it operates on a pre-defined system of geometry, boundary conditions, loading, and material properties. Therefore, the accuracy of its predictions relies not on the method itself, but on the appropriateness of the assumptions embedded in the model. Errors in geometry representation, oversimplified support conditions, or incorrect material definitions can propagate through the analysis, yielding results that may appear numerically sound but lack physical validity.

A critical limitation lies in the tendency to accept the numerical output of FEA as inherently reliable. The method is deterministic in nature because it always produces a result, regardless of the physical meaningfulness of the model. This characteristic presents a considerable risk when the analyst lacks sufficient insight into the structural system being modeled. The false confidence generated by a complete set of output data may conceal fundamental flaws in the formulation, leading to incorrect engineering decisions.

Boundary conditions deserve particular attention, as they often introduce sig-

nificant modeling errors. These conditions must accurately reflect the physical constraints of the system rather than replicate simplified or idealized notions of support. Improper constraint definitions can drastically alter stress distributions and displacement patterns, undermining the validity of the entire analysis [26].

From a computational standpoint, the discretization of a continuous system into finite elements introduces approximation errors that are sensitive to mesh density, especially in regions of high stress gradients, geometric discontinuities, or localized loading. Inadequate refinement in these areas can result in the underestimation or overestimation of stresses, particularly near singularities such as holes, sharp corners, or abrupt material transitions [27, 28]. Furthermore, complex or poorly conditioned systems may suffer from numerical instabilities or round-off errors during matrix operations, especially in large-scale models with high degrees of freedom [29].

Validation of FEA results is an essential yet often impractical step. Comparison with experimental data remains the most effective means of ensuring the reliability of numerical predictions; however, access to such data is not always feasible due to cost, technical constraints, or the unavailability of physical prototypes. In the absence of validation, reliance on FEA alone should be tempered by a critical evaluation of the model and its assumptions [30].

Another important limitation concerns computational cost. High-fidelity simulations, such as those involving nonlinear material behavior, contact phenomena, or transient dynamics, can become prohibitively expensive in terms of memory and processing time. This often forces a compromise between model detail and computational feasibility, which may affect the resolution of key physical phenomena.

Also, the study emphasizes the flexibility of CUF in managing multi-theory formulations and highlights the robustness of Legendre polynomial-based kinematics in achieving exponential convergence with respect to the expansion order [31, 32, 33, 34]. These findings build upon foundational studies in refined beam and plate theories [6–8] and reinforce the role of CUF as a versatile platform for modeling advanced composite structures. Overall, this work provides a robust and efficient methodology for structural analyses where both global behavior and local accuracy are required.

Furthermore, in cases involving material heterogeneities—such as composite structures commonly used in aerospace applications—standard FEM meshes often fail to capture the local variations in stiffness and strength that are critical for accurate failure predictions. This limitation highlights the need for advanced meshing techniques capable of adjusting to the specific requirements of aerospace simulations.

1.1.4 Introduction to Adaptive Meshes and Their Advantages in Structural Analysis

To overcome the challenges associated with traditional FEM meshing, adaptive meshing techniques have been developed to enhance the accuracy and efficiency of numerical simulations. Unlike standard meshes, adaptive meshes adjust their element density and distribution dynamically based on the characteristics of the structure being analyzed [35]. This adaptive approach allows for greater refinement in regions with high stress gradients while maintaining coarser elements in areas where lower accuracy is acceptable, thereby optimizing computational resources.

One of the major benefits of adaptive meshing is its ability to conform more effectively to complex geometries. In aerospace engineering, where structures often feature intricate designs such as aerodynamic fairings, composite laminates, and lightweight truss configurations, adaptive meshes can provide a more precise representation of the actual physical model without significantly increasing the computational burden [36]. This is particularly beneficial in applications such as:

- **Stress Concentration Regions:** Around joints, rivets, and load-bearing members, where accurate stress prediction is essential for durability assessment
- **Thin-Walled Structures:** Such as aircraft fuselage panels, where capturing bending and buckling phenomena requires high local mesh density
- **Fluid-Structure Interaction (FSI):** Where aerodynamic loads influence structural response, necessitating dynamic mesh refinement in high-pressure zones [37]
- **Composites and Multi-Material Interfaces:** Where adaptive meshing can better capture anisotropic behavior and damage progression

Adaptive meshing also integrates well with advanced structural modeling techniques, such as the Carrera Unified Formulation (CUF), which allows for a more flexible and higher-order representation of structural elements [38]. The combination of CUF with adaptive meshing strategies can further improve the precision of simulations by enabling localized refinements in thin shells, complex junctions, and multi-layered composite components.

1.1.5 Research Objectives

This study aims to address the limitations of conventional FEM meshing by implementing adaptive mesh strategies tailored to aerospace applications. Specifically, the objectives of this research include:

- Developing adaptive meshing techniques that enhance simulation accuracy while optimizing computational efficiency
- Investigating the integration of adaptive meshes with Carrera Unified Formulation (CUF) to improve structural modeling capabilities
- Evaluating the effectiveness of these techniques through the assessment of specific structural elements, including concave, convex, and generally distorted geometries, as well as through modeling approaches that incorporate the combination of non-classical elements, which are not typically allowed in conventional FEM, in various aerospace structural components such as fuselage structures, wing sections, landing gear, and spacecraft assemblies
- Laying the foundations for future research on the application of adaptive meshing in fluid-structure interaction (FSI) problems, where dynamic refinements are necessary for capturing aerodynamic-structural coupling, rather than directly exploring these applications in the current study

By achieving these goals, this research will contribute to the advancement of numerical simulation methodologies in aerospace structural analysis, facilitating more reliable, efficient, and high-fidelity modeling approaches that meet the stringent performance requirements of modern aerospace engineering.

1.2 Literature review

Structural modeling has always played a central role in engineering analysis, offering insight into the behavior of components under various loading and boundary conditions. The widespread adoption of the Finite Element Method (FEM) has revolutionized this domain by enabling accurate simulation of structures with arbitrary shapes. Nonetheless, FEM encounters critical limitations when applied to innovative materials, such as metamaterials with microarchitected topologies [39, 40], or when modeling components with complex geometrical features. In such cases, traditional formulations often demand dense 3D meshes, resulting in excessive computational cost and reduced feasibility for large-scale or parametric studies.

Typical assumptions inherent to classical beam and plate theories—such as constant cross-sections orthogonal to the axis for beams, or flat midsurfaces with uniform thickness for plates—are rarely met in practical scenarios. When such geometric idealizations fail, classical lower-dimensional models lose their predictive power, and full 3D simulations become unavoidable. The need for computationally efficient, yet geometrically versatile, modeling tools becomes evident.

Historically, structural elements have been modeled using simplified kinematic assumptions: Euler [41] introduced the fundamental beam theory for slender structures under bending, followed by the Timoshenko beam model [42], which accounted for shear deformations. Similarly, the classical Kirchhoff plate theory [43] and the Reissner–Mindlin First-Order Shear Deformation Theory [44, 45] laid the groundwork for modeling plate-like structures. Despite their usefulness, these theories exhibit rigid limitations, particularly in addressing higher-order phenomena such as cross-sectional warping, in-plane distortions, and coupled field effects.

To address these shortcomings, numerous Higher-Order Theories (HOTs) have been developed [46, 47], and critical comparative studies, such as those by Kapania and Raciti [48], have highlighted their relative performance in terms of shear accuracy, vibrational analysis, and wave propagation.

In response to the need for a more flexible and unified modeling environment, the Carrera Unified Formulation (CUF) was introduced [49, 50]. CUF enables the formulation of structural theories using a hierarchical structure, where the kinematics can be arbitrarily refined according to the problem requirements. Unlike traditional theories that adopt fixed displacement fields, CUF allows for the construction of models with variable-order expansions.

Originally conceived for two-dimensional problems, CUF was rapidly extended to plate and shell applications in multi-field contexts [51, 52, 53, 54]. The extension to beam theories was achieved through the introduction of Taylor Expansion (TE) models [55], in which the mechanical fields are approximated using Maclaurin series. These models offer enhanced kinematic representations while keeping computational demands moderate [56, 57, 58, 59].

A major advancement within CUF was the incorporation of Lagrange Expansion (LE) models [60]. These models utilize Lagrange polynomials to define the through-thickness distribution of displacement fields, offering increased geometric adaptability. LE-based CUF elements have shown strong capabilities in reproducing full 3D solutions while operating with significantly reduced degrees of freedom. Their utility has been further confirmed in multi-component simulations, where the unknowns are directly assigned to the actual surfaces of the components, avoiding the need for theoretical reference axes or midsurfaces [61, 62, 63].

One of the most promising innovations in the CUF framework is the introduction of Node-Dependent Kinematics (NDK) [64, 65, 66]. This technique enables the variation of the kinematic order at the nodal level, providing a mechanism for localized model refinement. In essence, NDK allows for high-fidelity modeling in critical regions, while coarser approximations can be adopted elsewhere—an approach particularly beneficial in global–local simulations [67].

Furthermore, NDK has been applied effectively in the modeling of laminated and smart structures containing embedded devices such as piezoelectric actuators

[51]. It avoids the need for dense 3D meshes while preserving the accuracy required to capture local phenomena [66].

To further extend the applicability of CUF to structures with complex geometrical features, the present work introduces a novel class of CUF finite elements that integrates NDK and is specifically designed to model non-orthogonal configurations. These elements maintain the 1D or 2D kinematic foundations typical of CUF, yet the computation of the governing matrices is performed through integrals expressed in full 3D form. The Jacobian and the approximating functions are tailored to the true geometry of the physical domain, enabling accurate modeling of inclined, curved, or tapered structural elements.

The implementation relies on Lagrange polynomial expansions, though the methodology is compatible with other hierarchical bases, including Taylor and Legendre polynomials. By incorporating geometric blending functions, the kinematic description can be aligned with the physical morphology of the component.

These new finite elements offer a dual benefit: on the one hand, they allow local mesh refinement in geometrically or mechanically critical regions; on the other, they enable an effective reduction in computational cost by avoiding full 3D discretization of the entire structure. Conceptually, these elements behave like 3D models with anisotropic order—where the degree of expansion can be independently controlled in each spatial direction—allowing high-resolution modeling without the typical computational penalties.

In the subsequent chapters, these elements will be applied and validated in representative case studies, demonstrating their effectiveness and accuracy in modeling real-world aerospace and mechanical components.

The work of Carrera [49] presents a thorough analysis of the theoretical and numerical approaches used for the modeling of multilayered, anisotropic composite plates and shells. The study is divided into three main parts, each focusing on different aspects of the subject.

The first part provides an overview of the various theoretical models applied to composite structures. It explores three-dimensional elasticity theories, continuum-based methods, and two-dimensional theories derived from axiomatic and asymptotic approaches. One of the key challenges in modeling multilayered composites is the need to properly capture transverse deformation effects, discontinuities in displacement fields across layers, and interlaminar stress continuity, often referred to as the $C0_z$ requirements. The author discusses how traditional plate and shell theories, originally formulated for isotropic materials, must be extended and refined to accommodate the unique characteristics of anisotropic laminates.

The second part of the study is dedicated to two-dimensional theories, which are categorized into equivalent single-layer (ESL) theories and layer-wise mod-

els (LWM). Carrera reviews classical approaches such as the classical lamination theory and first-order shear deformation theory, along with higher-order theories that provide improved accuracy in stress distribution calculations. A significant portion of the discussion is devoted to zig-zag theories, which address the displacement discontinuities at layer interfaces. Several notable zig-zag formulations, including those proposed by Lekhnitskii–Ren, Ambartsumian–Whitney–Rath–Das, and Reissner–Murakami–Carrera, are analyzed in terms of their effectiveness in capturing interlaminar behavior.

The third part of the paper focuses on the finite element implementation of these theories. The author examines the development of finite element models for multilayered composite structures, highlighting the transition from classical elements used for homogeneous materials to advanced elements tailored for layered configurations. The study discusses the role of Reissner’s Mixed Variational Theorem in improving numerical accuracy and introduces hybrid and global-local approaches that enhance computational efficiency. Different numerical strategies are compared, and the advantages of refined finite element formulations in capturing complex multilayered behavior are emphasized.

Throughout the paper, Carrera underscores the importance of advanced modeling techniques in accurately predicting the mechanical response of composite structures. The review provides valuable insights into the limitations of classical models and the need for refined approaches that account for shear deformation, interlaminar stress continuity, and multilayered effects. This work serves as a foundation for future advancements in the field of aerospace and mechanical engineering, where composite materials play an increasingly critical role.

The study by Cinefra [68] explores the development of non-conventional one-dimensional (1D) and two-dimensional (2D) finite elements based on the Carrera Unified Formulation (CUF) for the analysis of structures with non-orthogonal geometries. Traditional finite element methods (FEM) are widely used in structural mechanics but often suffer from excessive computational costs when applied to complex geometries, such as non-orthogonal beams and plates or materials with intricate microstructures. To address these challenges, this work introduces an advanced approach that exploits the flexibility of CUF in describing structural kinematics while maintaining computational efficiency.

The paper begins with an overview of the limitations of classical beam and plate theories. Traditional 1D models, such as those based on Euler-Bernoulli or Timoshenko beam formulations, assume a-priori fixed kinematics, limiting their applicability to specific structural cases. Similarly, classical plate theories, including the Kirchhoff and Reissner-Mindlin approaches, impose constraints on displacement fields, making them unsuitable for non-conventional geometries. These

limitations necessitate the use of refined 3D finite elements, which, while accurate, dramatically increase the number of degrees of freedom and computational costs. The Carrera Unified Formulation has been proposed as a generalized framework that enables the derivation of refined beam and plate models through systematic kinematic expansion.

A key innovation in this study is the Node-Dependent Kinematics (NDK) approach, which allows different kinematic assumptions to be assigned at individual finite element nodes. This technique enables the local refinement of the displacement field, allowing high-order kinematics in critical regions while maintaining simpler assumptions in less sensitive areas. The application of NDK within CUF-based finite elements facilitates the analysis of structures with variable cross-sections, curved geometries, and non-orthogonal boundary conditions without requiring excessively fine 3D meshes. Previous research has demonstrated the effectiveness of CUF in modeling laminated composites [38], piezoelectric structures [51], and aeroelastic components [65], but this work extends its applicability to a broader class of problems.

The formulation of the new finite elements is based on Lagrange polynomial expansions, allowing the order of approximation to vary along different spatial directions. Unlike conventional 3D finite elements, which enforce uniform interpolation orders across all directions, the proposed elements allow independent refinement along the beam axis, thickness, and width. This flexibility leads to a significant reduction in computational cost while preserving the accuracy of 3D models. The governing equations of these elements are derived using the Principle of Virtual Displacements, ensuring compatibility with classical FEM approaches while incorporating the advantages of hierarchical kinematic descriptions.

To validate the effectiveness of the proposed elements, the paper presents numerical results from a series of free vibration analyses. Several benchmark problems are analyzed, including isotropic plates with non-orthogonal edges, beams with variable cross-sections, and plate-like structures with non-uniform thickness. The results are compared against classical 3D finite elements, demonstrating that the new CUF-based elements achieve comparable accuracy with significantly fewer degrees of freedom. In particular, a reduction of up to an order of magnitude in computational cost is observed when using NDK-enhanced CUF elements, making them highly suitable for practical engineering applications.

The study concludes by emphasizing the potential of CUF and NDK to revolutionize structural modeling. The ability to efficiently analyze non-orthogonal geometries without resorting to computationally expensive 3D meshes represents a major advancement in finite element analysis. Future research directions include the extension of the proposed methodology to curvilinear coordinate systems [69] and the incorporation of mixed interpolation techniques to mitigate locking effects

in thin-walled structures.

The study by Bell et al. [70] presents an adaptive mesh refinement (AMR) algorithm designed for solving hyperbolic conservation laws in three-dimensional space. The primary motivation behind this work is the challenge posed by high-resolution three-dimensional simulations, which often require substantial computational resources. The AMR method proposed in this paper addresses this issue by utilizing local grid patches superimposed on a coarse grid, allowing for an efficient and dynamic allocation of computational effort.

The algorithm extends the two-dimensional AMR methods developed in previous research to three dimensions. It employs a hierarchical grid structure where finer grids are recursively embedded within coarser ones, ensuring that regions requiring higher resolution are dynamically refined. A key feature of the method is its error estimation procedure, which automatically determines areas of the solution requiring additional resolution. Grid generation routines then create or remove refined patches as necessary, and special difference equations are employed at the interface between coarse and fine grid patches to maintain conservation.

The authors highlight several successful applications of AMR in two-dimensional fluid dynamics problems. The additional resolution provided by AMR has enabled the study of phenomena such as Kelvin-Helmholtz instabilities, Mach reflections, and shock interactions with interstellar clouds. When extended to three dimensions, AMR offers the potential to make computationally intractable problems feasible. However, challenges remain in terms of grid clustering, computational cost, and numerical stability.

Alternative approaches to adaptive meshing, including moving grid methods and unstructured grid techniques, are discussed. While unstructured grids offer flexibility in point placement, they present difficulties in terms of wave propagation accuracy and computational overhead. In contrast, AMR's use of locally uniform refined patches retains the benefits of structured grid solvers while reducing computational costs.

The paper also describes the implementation details of the three-dimensional AMR algorithm, focusing on improvements in grid generation and error estimation. A "smart bisection" algorithm is introduced to cluster refinement regions efficiently, improving computational performance. Additionally, a mixed spatial and truncation error-based approach enhances the refinement criteria, ensuring that key flow structures such as shock waves and contact discontinuities are adequately resolved.

A key numerical example presented in the study is the interaction of a supersonic Mach 1.25 shock wave with an ellipsoidal cloud of Freon-22. The simulation, performed on a Cray-2 supercomputer, demonstrates the effectiveness of AMR

in capturing fine-scale vortex structures and complex fluid dynamics interactions while significantly reducing computational costs compared to a uniform grid approach. The results show that AMR can achieve a computational speedup of more than a factor of 20 while maintaining high accuracy.

The study concludes that AMR provides a highly effective means of reducing computational costs for three-dimensional hyperbolic conservation laws. While the method remains problem-dependent, it offers substantial advantages over traditional fixed-grid approaches. The authors emphasize the potential for further optimization and the integration of AMR with other numerical techniques to improve efficiency and expand its applicability in computational fluid dynamics.

The study by Sorgente et al. [71] explores the impact of mesh quality on the performance of the Virtual Element Method (VEM). The authors present a comprehensive analysis of how different mesh characteristics influence the accuracy, stability, and computational efficiency of the VEM. The research is conducted within the framework of the ERC Project CHANGE and focuses on understanding the correlation between mesh quality metrics and VEM performance indices.

The paper begins by discussing the challenges associated with the discretization of geometrically complex domains, which are frequently encountered in engineering applications. Traditional finite element methods (FEM) rely on structured meshes consisting of triangular, quadrilateral, tetrahedral, or hexahedral elements. However, these approaches impose stringent constraints on element shapes, leading to potential inefficiencies in capturing complex geometries. To overcome these limitations, polytopal element methods (PEM) have been developed, allowing numerical approximations on polygonal and polyhedral meshes without requiring an explicit definition of basis functions. Among these methods, the VEM stands out as a powerful alternative due to its flexibility in handling arbitrary polygonal elements.

A key contribution of the paper is the examination of mesh quality and its effect on VEM convergence. The authors conduct an extensive numerical study where they evaluate VEM performance under conditions that violate conventional shape regularity assumptions. Several datasets of polygonal meshes are analyzed, each exhibiting different geometric irregularities such as high aspect ratios, non-convexity, and extreme element distortions. The study introduces a novel mesh quality indicator that quantifies the degree of deviation from ideal mesh properties, allowing for an assessment of how these deviations influence numerical accuracy.

The findings indicate that while VEM maintains optimal convergence rates under standard geometric assumptions, its performance deteriorates in the presence of highly irregular meshes. However, the method demonstrates remarkable robustness, maintaining reasonable accuracy even when theoretical conditions are not strictly satisfied. This suggests that the commonly imposed shape regularity

constraints in FEM may be relaxed in VEM without significant loss of precision. The research also reveals that specific geometric features, such as the presence of hanging nodes and non-star-shaped elements, can negatively impact solution stability.

The study further explores the design of polytopal meshing algorithms that optimize VEM performance. By analyzing the correlation between mesh quality metrics and VEM error estimates, the authors provide insights into the geometric properties that should be prioritized in mesh generation. The results emphasize the need for adaptive meshing strategies tailored to the specific requirements of VEM-based solvers.

In conclusion, the research conducted by Sorgente et al. highlights the critical role of mesh quality in determining the effectiveness of VEM. The study not only provides a deeper understanding of VEM behavior on irregular meshes but also lays the groundwork for future advancements in polytopal meshing techniques. The findings are expected to contribute significantly to the development of more efficient numerical methods for solving partial differential equations on complex geometries.

The study by Gain et al. [72] presents a detailed investigation of the Virtual Element Method (VEM) applied to three-dimensional linear elasticity problems on arbitrary polyhedral meshes. The motivation for this work stems from the challenges associated with mesh generation in complex geometries, where traditional finite element methods (FEM) struggle due to constraints on element shapes and interpolation functions. VEM, as a recent development, circumvents these issues by allowing for discretization on general polyhedral elements without explicitly computing shape functions within the elements.

The paper introduces the core mathematical framework underlying the first-order VEM for three-dimensional elasticity. Unlike standard FEM, which requires explicit basis functions, VEM defines the approximation space through projection operators that decompose the strain energy into its essential components. This decomposition is facilitated by local projection maps that extract rigid body motion and constant strain components. By enforcing the exact representation of linear deformations, the method guarantees the satisfaction of the patch test and ensures optimal convergence.

One of the key advantages of VEM highlighted in this work is its ability to handle general polyhedral meshes without the need for complex meshing algorithms. The authors discuss how mesh refinement and coarsening can be seamlessly integrated, making VEM particularly suited for adaptive simulations. Additionally, the study compares VEM with other numerical techniques, such as Mimetic Finite Differences (MFD) and Finite Volume Methods (FVM), showing that VEM retains

the best properties of both approaches while avoiding their respective limitations.

The paper also details the implementation of VEM, deriving explicit expressions for element stiffness matrices and the projection maps used to construct the discrete bilinear forms. The authors emphasize that the formulation is purely geometric in nature, relying on element topology rather than explicit interpolation functions. This makes VEM highly robust against mesh distortions and applicable to problems where classical FEM encounters difficulties, such as those involving non-convex elements and irregular boundaries.

A series of numerical studies are presented to validate the accuracy and efficiency of VEM. The results confirm that VEM maintains optimal convergence rates even on highly irregular meshes, demonstrating its potential for practical applications in structural mechanics and computational engineering. In particular, the method is tested on benchmark problems in elasticity, including cantilever beams and torsional deformations, where it exhibits excellent agreement with analytical solutions.

The study concludes by discussing potential future developments, including the extension of VEM to nonlinear elasticity and its integration with adaptive meshing techniques. The authors suggest that the method's ability to operate on arbitrary polyhedral grids positions it as a promising tool for high-fidelity simulations in engineering and scientific computing.

The study by Funken and Schmidt [73] presents an efficient implementation of adaptive mesh refinement (AMR) techniques in two-dimensional domains using Matlab. The primary motivation behind this work is to provide an accessible and computationally efficient approach to AMR that can be used in research and education. The authors emphasize the importance of flexibility in mesh refinement, allowing both triangular and quadrilateral grids, while minimizing computational overhead through optimized data structures and vectorization.

The paper begins by discussing the significance of mesh refinement in various applications, such as solving partial differential equations (PDEs) using the finite element method (FEM) and rendering in computer graphics. Many existing refinement tools, whether commercial software or open-source implementations, present challenges in terms of accessibility and ease of integration with other numerical solvers. Commercial packages often function as "black boxes," limiting user control, while open-source solutions are frequently too complex for a broader audience. The authors address these limitations by developing a lightweight yet powerful refinement package in Matlab.

The study introduces several well-established refinement strategies, including red-green (RG) refinement as proposed by Bank and Sherman [74], newest vertex bisection (NVB) first introduced by Sewell [75], and red-green-blue (RGB)

refinement following Carstensen [76]. The research extends these classical strategies by incorporating quadrilateral refinement techniques such as red refinement by quadrissection [77], red-green refinement [74], and the red-blue (RB) method inspired by Kobbelt [78]. Additionally, the authors introduce the red2-green2 (RG2) refinement method, which extends previous techniques by allowing division into nine smaller quadrilaterals per refinement step.

A key feature of this work is the structured and efficient implementation of AMR through a standardized procedure consisting of four main steps: marking elements for refinement, ensuring grid closure, creating new nodes, and generating refined elements. The marking phase involves identifying edges that require refinement based on predefined criteria, while the closure step ensures that hanging nodes are either removed or appropriately managed to maintain grid consistency. The implementation focuses on minimizing computational cost by utilizing hash maps, virtual elements, and vectorized operations to streamline the refinement process.

The authors also address the challenge of mesh coarsening, an essential component of adaptive simulations that allows for dynamic adjustment of the computational mesh. While hierarchical refinement strategies enable straightforward coarsening, non-hierarchical methods require additional considerations. The study presents an approach to non-hierarchical local coarsening, extending previous work by Chen and Zhang [79] to handle various refinement patterns without requiring extensive data storage.

To validate their implementation, Funken and Schmidt conduct numerical experiments demonstrating the efficiency and versatility of their Matlab code. The results confirm that their AMR framework maintains near-linear computational complexity, even for large-scale refinements involving up to millions of elements. The study further provides a series of practical examples, including refinement along circular boundaries and adaptive finite element simulations, showcasing the adaptability of the proposed methodology.

The paper concludes by highlighting the contributions of the work in making adaptive mesh refinement more accessible and computationally efficient. The developed Matlab package serves as a valuable tool for both researchers and educators, bridging the gap between theoretical AMR concepts and practical implementation. The authors suggest future extensions to three-dimensional refinement and further optimization of data structures to enhance performance in large-scale simulations.

The work by Ortiz and Quigley [80] presents an adaptive meshing technique specifically designed for strain localization problems in solids. Strain localization poses a significant challenge in computational mechanics due to the presence of

multiple length scales: a global scale, corresponding to the macroscopic dimensions of the solid, and a local scale, associated with the shear band thickness. The microscale often collapses over time, leading to difficulties in resolving strain localization accurately using conventional finite element methods. The authors propose an adaptive refinement strategy that equidistributes variations in the velocity field across the mesh, offering an efficient way to capture the evolving localization patterns.

A key challenge in strain localization problems is the loss of ellipticity in the governing equations at the onset of localization. Traditional adaptive mesh refinement (AMR) methods for linear elasticity rely on elliptic estimates for error indicators, which break down in strain localization cases. The authors argue that an alternative refinement criterion, based on interpolation error minimization, is better suited for these problems. Specifically, they propose using variations in the velocity field as an adaptation indicator, leading to a resolution-driven refinement strategy. This approach ensures that regions experiencing rapid variation in strain receive a finer mesh resolution, while less critical areas remain coarser, optimizing computational efficiency.

The study employs Delaunay triangulation for mesh generation, a technique well-suited for adaptive refinement due to its ability to maintain well-shaped elements during mesh subdivision. When an element requires refinement, new nodes are inserted at the midpoints of its edges, and the connectivity is updated dynamically. The authors utilize Sloan’s triangulation algorithm [81], which has an average computational complexity of $O(N^{5/4})$, where N is the number of nodes. This ensures that the adaptive meshing process remains computationally feasible even for large-scale simulations.

One of the most critical aspects of adaptive meshing in history-dependent materials is the transfer of state variables between different mesh resolutions. Unlike linear elasticity problems, where the solution can be recomputed from scratch after each refinement step, strain localization problems require a transfer operator to interpolate stress and strain history variables onto the new mesh. The authors derive a canonical transfer operator using the Hu-Washizu variational principle, ensuring that state variables are transferred consistently across mesh updates. This minimizes numerical artifacts and preserves solution accuracy during refinement.

To validate their method, the authors present numerical examples demonstrating its effectiveness. The first case study involves plane-strain uniaxial tension, where a rectangular solid undergoes deformation leading to the formation of shear bands. The refinement process is triggered when velocity field variations exceed a predefined threshold, leading to progressive mesh refinement around the shear bands. The results confirm that the adaptive approach captures localization patterns efficiently, producing well-defined bands misaligned with the initial mesh

directions.

The second case study examines plane-strain compression of a frictional solid, modeled using a Drucker-Prager yield criterion with a non-associated flow rule. Despite the absence of material softening, strain localization still occurs due to a lack of normality in the flow rule. The adaptive method successfully refines the mesh along evolving localization zones, revealing distinct interior and boundary shear bands with different orientations. The computed band angles closely match analytical predictions from previous theoretical studies [82].

The study concludes by emphasizing the advantages of adaptive meshing in strain localization problems. The method enables simulations to accurately resolve evolving shear bands without excessive computational cost. By dynamically refining the mesh in critical regions while maintaining coarser elements elsewhere, the approach avoids artificial numerical diffusion and locking effects commonly observed in traditional finite element simulations. The authors suggest further research into three-dimensional extensions of the method and its integration with higher-order finite element formulations.

The work by Salazar de Troya and Tortorelli [83] presents a novel framework for topology optimization incorporating adaptive mesh refinement (AMR) under stress constraints. Traditional topology optimization approaches have predominantly focused on compliance minimization, often neglecting the importance of stress constraints, which are critical for ensuring structural integrity. The authors address this limitation by integrating AMR into the optimization process to enhance accuracy in stress field computations and improve the resolution of optimized structures.

One of the primary challenges in stress-constrained topology optimization is the singularity problem, where gradient-based optimization algorithms struggle to reach global optima due to stress constraints being active in lower-dimensional subspaces. Previous works, such as Cheng and Guo [84] and Bruggi [85], have introduced relaxation techniques to mitigate this issue. The present study employs a stress relaxation strategy based on a RAMP-like stress formulation, which effectively regularizes the optimization problem while maintaining numerical stability.

A second challenge arises from the computational complexity associated with enforcing local stress constraints. In finite element discretization, the number of stress constraints often scales with the number of elements, making the optimization process computationally expensive. To overcome this, the authors employ an aggregation technique using the Kreisselmeier-Steinhauser (KS) function [86], which approximates the maximum stress over the entire domain. This method significantly reduces the number of active constraints, facilitating a more efficient optimization process.

Another major issue in stress-constrained topology optimization is the inaccuracy of stress field computations on non-conforming meshes. The traditional density-based method projects the design onto a fixed finite element grid, leading to blurred material boundaries and unreliable stress estimates. To address this, the study integrates AMR techniques to dynamically refine the mesh in regions of high stress concentration. The authors employ an error-driven refinement strategy, leveraging a posteriori error estimators to guide mesh refinement and coarsening. This ensures that stress computations remain accurate while avoiding unnecessary computational costs.

The implementation of AMR within the topology optimization framework follows a hierarchical strategy. Initially, a coarse mesh is used to define the design domain, with refinements introduced iteratively based on stress distribution. The authors validate their approach using benchmark problems, including a 2D L-bracket under stress constraints. Their results demonstrate that AMR leads to more accurate stress predictions, improved design resolution, and substantial reductions in computational expense compared to uniform mesh refinement.

The paper also discusses alternative refinement strategies, including methods based on geometric proximity to the material boundary [87], residual error indicators [85], and metric-based anisotropic refinement [88]. The authors highlight the advantages of their approach in handling arbitrary stress distributions while maintaining computational efficiency.

In conclusion, the study presents a robust methodology for integrating AMR into stress-constrained topology optimization. By combining error-driven mesh refinement with efficient constraint aggregation and stress relaxation techniques, the proposed framework enhances the accuracy and efficiency of stress-constrained designs. The findings suggest promising avenues for future research, including three-dimensional extensions and the integration of AMR with higher-order finite element formulations.

The study by Carrera [32] introduces a novel approach for developing two-dimensional (2D) finite element (FE) models of plate structures featuring arbitrary kinematic expansions for each displacement component along the thickness direction. Classical plate models such as the Thin Plate Theory (TPT) and the First-Order Shear Deformation Theory (FSDT) have historically been used to simplify 3D elasticity problems, but they often fall short in capturing local or higher-order effects in complex structures. To address these limitations, the paper employs the Carrera Unified Formulation (CUF), which allows the construction of both classical and higher-order theories within a unified mathematical framework.

A key innovation in this work is the independent definition of the polynomial expansion order for each displacement variable—longitudinal, transverse, and lat-

eral—unlike traditional models which impose uniform expansions. This flexibility is achieved by replacing the conventional 3×3 Fundamental Nucleus (FN) used in CUF with a scalar FN, thereby enabling tailored reductions in the number of degrees of freedom (DOFs) without compromising accuracy. The formulation is derived using the Principle of Virtual Displacements, and the Mixed Interpolation of Tensorial Components (MITC) method is adopted to mitigate shear locking in thin plates.

The paper presents a comprehensive review of classical and higher-order theories, followed by the development of reduced and non-uniform models. Taylor series expansions up to fourth-order are considered, with selective inclusion of terms to study their influence on the solution quality. Numerical benchmarks—including simply-supported plates under bi-sinusoidal or localized loading, cylindrical bending, and cantilevered configurations—demonstrate the proposed method’s ability to replicate 3D-like accuracy with reduced computational effort. For example, reduced models with strategically chosen expansion terms achieve results close to full models with significantly fewer DOFs.

The authors also propose a roadmap for future developments, including the integration of Node-Dependent Kinematics (NDK) and Asymptotic-Axiomatic Methods (AAM), potentially assisted by machine learning techniques to identify optimal kinematic models based on problem-specific parameters. This work sets the foundation for an efficient and flexible generation of refined plate theories in structural analysis.

The study by Carrera [31] presents a refined class of two-dimensional shell finite element models based on the Carrera Unified Formulation (CUF), integrating Legendre-based Node-Dependent Kinematics (NDK) for the global-local analysis of both homogeneous and layered structures. Classical shell theories such as the Thin Shell Theory (TPT) or First-Order Shear Deformation Theory (FSDT) are efficient for simple configurations but inadequate for capturing complex local behaviors in layered composites. To overcome these limitations, the paper proposes a flexible modeling strategy that allows for different kinematic assumptions at the element node level.

A key innovation is the use of hierarchical Legendre expansions (HLE) for through-the-thickness approximations, enabling higher-order theories to be applied selectively in critical regions while simpler models are used elsewhere. This is achieved without additional mathematical coupling, thanks to the hierarchical structure of CUF. The NDK method permits assigning different polynomial orders—and even different types of polynomials such as Taylor, Legendre, or Zig-Zag enriched Taylor expansions (TEZ)—to individual nodes, providing local refinement capabilities within the same element. The finite element discretization is based on

nine-node Lagrangian shell elements, and numerical integration is performed using the Mixed Interpolation of Tensorial Components (MITC) to prevent locking phenomena.

Two numerical benchmarks are analyzed to validate the approach: a simply supported cylindrical shell under pinching loads and a three-layered spherical shell subjected to bi-sinusoidal local pressure. For each case, different combinations of global-local kinematic models are compared, including uniform higher-order models and NDK-enhanced variants. Results show that models using local HLE or TEZ expansions yield accuracy close to full 3D solutions with significantly fewer degrees of freedom. In particular, combining refined and coarse models within the same element reduces computational cost while maintaining the precision necessary to capture local effects such as shear stresses and displacement discontinuities.

The study confirms that Legendre-based NDK shell elements can effectively merge the computational efficiency of Equivalent Single Layer (ESL) approaches with the accuracy of Layer-Wise (LW) models. Furthermore, the ability to use different structural theories within the same FE model without resorting to coupling strategies makes this method suitable for a wide range of applications in structural mechanics, especially in aerospace and composite engineering.

The work by Pagani [34] introduces a unified and flexible modeling approach for laminated composite structures based on Lagrange polynomial expansions within the Carrera Unified Formulation (CUF). Traditional theories like Equivalent Single Layer (ESL) or Layer-Wise (LW) models each have strengths and limitations: ESL ensures low computational cost but fails in capturing interlaminar effects, while LW captures such details at the expense of computational efficiency. This work proposes a variable-kinematic method that combines both approaches using Lagrange interpolation points to describe the through-thickness behavior in beams, plates, and shells.

The formulation allows the user to a priori select regions to be treated as ESL or LW by controlling the placement of Lagrange points (LPs). Displacement continuity across layers is enforced naturally at LPs, avoiding the need for coupling strategies such as Lagrange multipliers. The CUF framework enables both 1D and 2D finite element models with variable-order polynomial expansions to describe mechanical fields across the structure. The derived governing equations are based on the Principle of Virtual Displacements, leading to a consistent assembly of stiffness matrices across beam, plate, and shell elements.

To validate the proposed modeling approach, several numerical benchmarks are presented. These include an eight-layer laminated structure, a simply supported multilayered plate under transverse loading, a nine-layer plate with sinusoidal pressure, and a cylindrical shell under transverse pressure. In all cases, comparisons

are made among Taylor-based models, LW models, and various ESL strategies. Results show that the proposed variable-kinematics approach can accurately capture both global and local responses, such as displacements and interlaminar shear stresses, while significantly reducing the number of degrees of freedom compared to full LW formulations.

Notably, the study demonstrates that proper placement and distribution of Lagrange points is critical to achieving high accuracy at low computational cost. ESL models with localized LW-like refinements achieve near-identical accuracy to full LW models for interlaminar stress evaluations, making the approach ideal for global-local analyses of complex composite structures. The work underscores the strength of CUF in enabling hierarchical and customizable structural theories without sacrificing computational efficiency.

1.2.1 Main Trends and Open Challenges

In modern computational mechanics, several strategies have been developed to achieve adaptive meshing, allowing for improved accuracy and efficiency in numerical simulations. One prominent approach is the Virtual Element Method (VEM), which generalizes the Finite Element Method (FEM) by enabling arbitrary polygonal/polyhedral elements while maintaining high-order accuracy and stability. Another widely used technique is the p -method, which refines the solution by increasing the polynomial degree of basis functions within elements rather than modifying the mesh topology. Additionally, h -adaptive methods adjust the mesh density locally by refining or coarsening elements based on error estimators. Hybrid approaches, such as the hp -method, combine both strategies, dynamically adapting element size (h -refinement) and polynomial order (p -refinement) to optimize computational performance. More recent developments include the use of isogeometric analysis (IGA), which integrates CAD-based representations with numerical simulations to enhance smoothness and accuracy, as well as neural-network-driven adaptive meshing, which leverages machine learning to optimize mesh refinement patterns dynamically. These adaptive meshing techniques are crucial in aerospace and structural applications, where capturing complex stress distributions and fluid-structure interactions is essential for predictive accuracy [89, 90].

The development and validation of the new adaptive finite elements presented in this PhD thesis represents a compelling alternative to the aforementioned approaches, offering significant advantages in terms of implementation, usability, and computational efficiency. In particular, when compared to the Virtual Element Method (VEM), which, despite its flexibility in handling arbitrary polygonal and polyhedral elements, requires a complex mathematical formulation and intricate computational procedures, the proposed method stands out for its simplicity

and ease of use. The VEM necessitates the construction of specific projection operators and stabilization terms to ensure convergence and accuracy, which can lead to non-trivial implementation challenges, especially when dealing with high-order approximations or three-dimensional applications. Conversely, the adaptive finite elements introduced in this work rely on a more straightforward formulation that naturally integrates into existing finite element frameworks without the need for elaborate modifications or additional stabilization techniques. This inherent simplicity not only facilitates the implementation process but also enhances the robustness and adaptability of the method across different engineering applications. Furthermore, the reduced computational overhead associated with the proposed approach makes it an attractive choice for large-scale problems where efficiency is a key concern. Ultimately, these characteristics position the newly developed adaptive finite elements as a highly practical and accessible alternative for researchers and engineers seeking an effective yet easily deployable meshing strategy [91, 92, 93].

Chapter 2

Theoretical framework

2.1 Introduction

The Finite Element Method (FEM) is a well-established and widely adopted technique that provides highly accurate results for the structural analysis of geometrically complex components. Nevertheless, when applied to novel materials—such as metamaterials characterized by intricate architected microstructures [94, 69]—or to structural components exhibiting unconventional beam or plate geometries, the computational effort required can become prohibitively high. Specifically, standard beam models fail to be effective when dealing with beams featuring variable cross-sections or sections that are not perpendicular to the longitudinal axis. Similarly, classical plate theories are inadequate for plates with non-uniform thickness or edges not orthogonal to the midsurface. In such scenarios, researchers are often compelled to employ highly refined three-dimensional (3D) meshes, which introduce the following major limitations:

- the substantial computational cost associated with conventional 3D elements, which can render certain analyses impractical
- the inherent difficulty of coupling different scales when analyzing large-scale structural components

To address these challenges, finite elements based on the Carrera Unified Formulation (CUF) have emerged as a powerful alternative, offering 3D-level accuracy at a significantly reduced computational cost compared to traditional 3D elements. One-dimensional (1D) models, commonly referred to as beam models, are particularly suited for analyzing slender structural components extensively employed in aerospace and civil engineering. These models are advantageous due to their efficiency, as they require considerably lower computational resources than their 2D

and 3D counterparts. Classical beam theories, initially formulated by Euler [95] and later refined by Timoshenko [96], present notable limitations, as they rely on predefined kinematic assumptions and are applicable only to a restricted range of problems (e.g., long beams under bending loads, homogeneous structures, etc.). To capture additional effects such as torsion, warping, or in-plane deformations, more advanced kinematic descriptions must be introduced. On the other hand, two-dimensional (2D) models, also known as plate models, are widely utilized in modern engineering to analyze lightweight structural systems. Several plate theories have been proposed over time, with some gaining extensive adoption in structural analysis. Among these, the Kirchhoff classical theory [97], the First-Order Shear Deformation Theory (FSDT) derived from the Reissner-Mindlin assumption [98, 99], and various Higher-Order Theories [100, 101] have been extensively studied. Numerous refined models have been developed to extend the capabilities of conventional plate theories, allowing for the inclusion of higher-order effects while maintaining computational efficiency. A comprehensive discussion of various beam and plate theories was conducted by Kapania and Raciti [102] [103], who focused on shear deformations, vibration response, and wave propagation phenomena.

The Carrera Unified Formulation (CUF) was introduced to overcome the constraints of traditional structural models by expressing the kinematics of structural theories and their governing equations in a unified, hierarchical, and compact manner. Initially proposed by Carrera [104, 105] for two-dimensional (2D) analysis, CUF has since been widely applied in the development of plate and shell theories for structural and multi-field problems, as documented in [106, 107, 108, 109]. The extension of CUF to beam modeling was later undertaken by Carrera and Giunta [110], who introduced hierarchical N-order approximations of primary mechanical variables using Maclaurin polynomial expansions. This class of 1D CUF models, commonly referred to as Taylor Expansion (TE) models, has been extensively utilized in structural research (e.g., [111, 112, 113, 114, 115]). A comprehensive review of the most significant CUF-based beam models was recently provided by Carrera et al. [116].

By leveraging Lagrange polynomial expansions, a novel CUF-based beam formulation was introduced by Carrera and Petrolo [117], demonstrating the enhanced capabilities of Lagrange Expansion (LE) models. In particular, this study highlighted the ability of LE to deliver highly accurate 3D-elasticity solutions for simple and complex structures with minimal computational cost. More recently, LE-based approaches have been extended to analyze multi-component structures [118, 119, 120], where finite element unknowns are directly associated with the physical surfaces of the 3D structural body. This eliminates the need for predefined mathematical reference axes in beam models or reference surfaces in plate models.

A further advancement within the CUF framework has been the introduction of the Node-Dependent Kinematics (NDK) approach, designed to improve numerical efficiency [121, 122, 123, 124, 125]. By defining kinematic assumptions on a node-by-node basis, NDK enables the construction of finite element models with spatially varying kinematics. This methodology is particularly beneficial for concurrent global-local analyses, allowing critical regions—where higher-order kinematics are necessary—to be seamlessly coupled with less critical areas modeled using lower-order theories [126]. Additionally, Lagrange-based models can be effectively applied to laminated structures with surface-mounted or embedded actuators, such as piezoelectric components [127]. In such cases, NDK facilitates numerical modeling by reducing the reliance on 3D elements where high-resolution solutions are required [128].

All the models discussed thus far are based on standard assumptions regarding the initial undeformed configuration of the structure: beams are typically assumed to have a constant cross-section that remains perpendicular to the axis, while plates are considered to have uniform thickness with through-the-thickness segments normal to the midsurface. This research seeks to develop and evaluate advanced CUF finite elements based on the NDK approach for analyzing beam-like and plate-like structures with non-orthogonal geometric features. The proposed elements are formulated based on a straightforward yet effective principle: integrals within the governing equation matrices (stiffness and mass matrices), which involve approximation functions and the Jacobian matrix, are computed in 3D form, despite the structural model being described using 1D or 2D kinematics. While this study focuses on Lagrange expansions, the methodology can be readily extended to Taylor and Legendre expansions by incorporating suitable blending functions to capture geometric variations. The principal advantage of these non-standard 1D and 2D elements lies in their ability to locally refine the computational mesh in regions where geometric complexity or kinematic behavior necessitates it. Ultimately, these elements can be viewed as an extension of 3D elements in which the order of expansion can be independently defined along the three spatial directions. This strategy leads to a significant reduction in computational costs compared to traditional 3D finite elements, as will be demonstrated in the subsequent analysis.

2.2 1D and 2D models based on Carrera Unified Formulation

As extensively discussed in prior research on the Carrera Unified Formulation (CUF), classical structural theories exhibit several inherent limitations. One of the primary issues stems from the need to incorporate higher-order effects while

mitigating physical inconsistencies. In theory, an infinite number of terms should be added to the kinematic description to achieve an exact representation of the displacement field, as indicated by Washizu [129].

From a practical standpoint, however, this is unattainable, as structural theories are generally formulated by truncating the expansion of primary mechanical variables along the smallest dimension of the structural domain at a finite order. The inclusion of additional terms increases the accuracy of the model but also significantly escalates the complexity of both the formulation and the computational resolution of the problem.

To address these challenges, the Carrera Unified Formulation (CUF) [111, 130] provides a unified approach in which the kinematic field is described in a generalized manner, allowing the governing equations to be derived in a compact and systematic way.

Let us consider a generic beam structure whose longitudinal axis coincides with the y -axis in a Cartesian coordinate system, while its cross-section lies in the xz -plane. Within the CUF framework, the displacement field of one-dimensional models is expressed as an expansion of generalized displacement components—when displacement-based theories are considered—through a set of arbitrary functions that depend on the cross-section coordinates:

$$\mathbf{u}(x, y, z) = F_\tau(x, z)\mathbf{u}_\tau(y), \quad \tau = 1, 2, \dots, M \quad (2.2.1)$$

where $\mathbf{u} = \{u_x, u_y, u_z\}$ represents the vector of three-dimensional displacements, while $\mathbf{u}_\tau = \{u_{x\tau}, u_{y\tau}, u_{z\tau}\}$ defines the corresponding generalized displacements. The integer M denotes the total number of terms in the expansion, and τ is the summation index. The functions $F_\tau(x, z)$ determine the specific class of beam theory adopted.

By selecting appropriate $F_\tau(x, z)$ functions, various beam models can be formulated, ranging from classical theories to higher-order formulations.

For plate theories, a similar approach can be adopted. If we assume that the mid-plane of the plate lies in the xy -plane, the CUF expansion can be formulated as follows [105]:

$$\mathbf{u}(x, y, z) = F_\tau(z)\mathbf{u}_\tau(x, y), \quad \tau = 1, 2, \dots, M \quad (2.2.2)$$

Here, the generalized displacements vary as a function of the mid-plane coordinates (x, y) , while the expansion is carried out along the thickness direction z . Again, different theories can be formulated depending on the choice of the expansion functions $F_\tau(z)$.

In this work, we specifically focus on Lagrange expansions. However, the choice of the expansion function F_τ is flexible, allowing for different formulations depending on the problem at hand. Alternative expansions, such as Taylor and Legendre

polynomials, can also be employed to define the kinematic field. Taylor expansions provide a straightforward hierarchical approach based on polynomial series, while Legendre expansions offer an orthogonal basis that can enhance numerical stability. These alternative formulations have been extensively explored in various studies by Carrera et al. [38].

2.2.1 Taylor Expansions

Taylor polynomials provide a convenient and systematic way to approximate functions by expressing them as a sum of their derivatives at a given point. This approach is particularly useful in structural analysis, as it allows for hierarchical refinements of the kinematic field while maintaining computational efficiency. By increasing the order of expansion, higher-order effects can be captured without the need for additional empirical assumptions, making Taylor expansions a powerful tool for developing refined structural models.

One possible choice is the adoption of Taylor polynomials consisting of the base z^i , where i is a positive integer.

Table 2.2.1: Taylor polynomials

N	M	F_τ
0	1	$F_0 = 1$
1	2	$F_1 = z$
2	3	$F_2 = z^2$
3	4	$F_3 = z^3$
$\vdots$	$\vdots$	$\vdots$
N	$N + 1$	$F_N = z^N$

This table presents the compact form of the Taylor polynomials

Table 2.2.1 presents F_τ , as functions of the order of expansion N , and M , which is the number of terms in the expansion. Each row shows the expansion terms of an N th order theory. The second-order model ($N = 2$) exploits a parabolic expansion of the Taylor polynomials.

$$\begin{aligned}
 u_x &= \underbrace{u_{x0}}_{N=0} + z \underbrace{u_{x1}}_{N=1} + z^2 \underbrace{u_{x2}}_{N=2} \\
 u_y &= \underbrace{u_{y0}}_{N=0} + z \underbrace{u_{y1}}_{N=1} + z^2 \underbrace{u_{y2}}_{N=2} \\
 u_z &= \underbrace{u_{z0}}_{N=0} + z \underbrace{u_{z1}}_{N=1} + z^2 \underbrace{u_{z2}}_{N=2}
 \end{aligned} \tag{2.2.1}$$

The 2D model given by Eqn. (2.2.1) has nine displacement variables. The strain components can be derived through partial derivatives, for instance, the shear and normal components are:

$$\begin{aligned}
 \gamma_{xz} &= u_{x,z} + u_{z,x} = u_{x1} + 2z u_{x2} + u_{z0,x} + z u_{z1,x} + z^2 u_{z2,x} \\
 \varepsilon_{zz} &= u_{z,z} = u_{z1} + 2z u_{z2}
 \end{aligned} \tag{2.2.2}$$

The CUF can also deal with *reduced* models by exploiting a lower number of variables, see [117]. An example of a reduced model is given by

$$\begin{aligned}
 u_x &= u_{x0} + z u_{x1} \\
 u_y &= u_{y0} + z u_{y1} \\
 u_z &= u_{z0} + z u_{z1} + z^2 u_{z2}
 \end{aligned} \tag{2.2.3}$$

For instance, the 2D model given in Eqn. (2.2.3) has seven displacement variables: three constants, three linear and one parabolic.

2.2.2 Legendre Expansions

In the case of Layer-Wise Models (LWMs), each layer is treated as an independent entity, with displacement continuity enforced at the interfaces. To achieve this, Lagrange polynomials can be employed, as suggested by Reddy (1997). However, when dealing with composite materials, the use of thickness functions expressed in terms of Legendre polynomials is generally preferred due to their superior numerical stability and orthogonality properties. The following expansion is adopted:

$$\begin{aligned}
 u_x^k &= F_t u_{xt}^k + F_b u_{xb}^k \\
 u_y^k &= F_t u_{yt}^k + F_b u_{yb}^k \\
 u_z^k &= F_t u_{zt}^k + F_b u_{zb}^k
 \end{aligned} \tag{2.2.1}$$

where the superscript k denotes the layer index, and F_t and F_b are the thickness functions corresponding to the top and bottom surfaces of each layer, respectively.

By employing Legendre polynomials, a more robust approximation of the displacement field across the thickness is achieved, making this approach particularly well-suited for modeling layered composite structures.

It is now intended that the subscripts t and b denote values related to the top and bottom layer surface, respectively. These two terms consist of the linear part of the expansion. The thickness functions $F_\tau(\zeta^k)$ are now defined at the k -layer level.

$$F_t = \frac{P_0 + P_1}{2}, \quad F_b = \frac{P_0 - P_1}{2} \quad (2.2.2)$$

in which $P_j = P_j(\zeta^k)$ is the Legendre polynomial of the j th order defined in the ζ^k domain:

$$\zeta^k = \frac{2z^k}{h^k}, \quad -1 \leq \zeta^k \leq 1.$$

For instance, the first five Legendre polynomials are:

$$P_0 = 1, \quad P_1 = \zeta^k, \quad P_2 = \frac{3\zeta_k^2 - 1}{2}, \quad P_3 = \frac{5}{2}\zeta_k^3 - \frac{3}{2}\zeta^k, \quad P_4 = \frac{35}{8}\zeta_k^4 - \frac{15}{4}\zeta_k^2 + \frac{3}{8}.$$

The chosen functions exhibit the following interesting properties:

$$\zeta_k = \begin{cases} 1 : & F_t = 1, \quad F_b = 0, \quad F_r = 0 \\ -1 : & F_t = 0, \quad F_b = 1, \quad F_r = 0 \end{cases} \quad (2.2.3)$$

That allows one to have interface values as unknown variables, therefore avoiding the inclusion of constraint equations to impose C_z^0 requirements. In a unified form

$$u^k = F_t u_t^k + F_b u_b^k = F_\tau u_\tau^k, \quad \tau = t, b \quad (2.2.4)$$

This theory will be denoted as LD1: that is, LW theory with displacement unknowns and first-order expansion. Higher-order LW theories are written by adding higher-order terms:

$$\begin{aligned} u_x^k &= F_t u_{xt}^k + F_b u_{xb}^k + F_2 u_{x2}^k + \cdots + F_N u_{xN}^k \\ u_y^k &= F_t u_{yt}^k + F_b u_{yb}^k + F_2 u_{y2}^k + \cdots + F_N u_{yN}^k \\ u_z^k &= F_t u_{zt}^k + F_b u_{zb}^k + F_2 u_{z2}^k + \cdots + F_N u_{zN}^k \end{aligned} \quad (2.2.5)$$

where

$$F_r = P_r - P_{r-2}, \quad r = 2, 3, \dots, N \quad (2.2.6)$$

or, in a unified form:

$$u^k = F_t u_t^k + F_b u_b^k + F_r u_r^k = F_\tau u_\tau^k, \quad \tau = t, b, \quad r = 2, 3, \dots, N \quad (2.2.7)$$

2.2.3 Lagrange Expansions

Lagrange expansion theories rely on the use of Lagrange-type polynomials as expansion functions, either over the beam cross-section or along the plate thickness. For beam models, the cross-section is partitioned into multiple local expansion sub-domains, with the polynomial degree determined by the type of Lagrange expansion used. Three commonly adopted expansions are bilinear four-node Q4, cubic nine-node Q9, and quartic sixteen-node Q16 polynomials [117]. The interpolation functions for the Q9 expansion, for instance, are given by:

$$\begin{aligned} F_\tau &= \frac{1}{4}(\xi^2 + \xi \xi_\tau)(\zeta^2 + \zeta \zeta_\tau), \quad \tau = 1, 3, 5, 7 \\ F_\tau &= \frac{1}{2}\zeta_\tau^2(\zeta^2 + \zeta \zeta_\tau)(1 - \xi^2) + \frac{1}{2}\xi_\tau^2(\xi^2 + \xi \xi_\tau)(1 - \zeta^2), \quad \tau = 2, 4, 6, 8 \\ F_\tau &= (1 - \xi^2)(1 - \zeta^2), \quad \tau = 9 \end{aligned} \quad (2.2.1)$$

where ξ and ζ are natural coordinates spanning from -1 to $+1$, and ξ_τ and ζ_τ denote the positions of the interpolation nodes. The displacement field for a single-Q9 beam element is thus expressed as:

$$\begin{aligned} u_x &= F_1(\xi, \zeta) u_{x_1} + F_2(\xi, \zeta) u_{x_2} + F_3(\xi, \zeta) u_{x_3} + \dots + F_9(\xi, \zeta) u_{x_9} \\ u_y &= F_1(\xi, \zeta) u_{y_1} + F_2(\xi, \zeta) u_{y_2} + F_3(\xi, \zeta) u_{y_3} + \dots + F_9(\xi, \zeta) u_{y_9} \\ u_z &= F_1(\xi, \zeta) u_{z_1} + F_2(\xi, \zeta) u_{z_2} + F_3(\xi, \zeta) u_{z_3} + \dots + F_9(\xi, \zeta) u_{z_9} \end{aligned} \quad (2.2.2)$$

or, in a compact form, as:

$$u_x = \sum_{\tau=1}^9 F_\tau(\xi, \zeta) u_{x_\tau}, \quad u_y = \sum_{\tau=1}^9 F_\tau(\xi, \zeta) u_{y_\tau}, \quad u_z = \sum_{\tau=1}^9 F_\tau(\xi, \zeta) u_{z_\tau} \quad (2.2.3)$$

Higher-order Lagrange expansions or multi-domain Lagrange interpolations allow for the development of refined beam models. More details on Lagrange-based beam theories can be found in [131, 117, 132, 116].

Lagrange polynomials are also employed in plate theories by using one-dimensional Lagrange expansions along the thickness direction z . A generic displacement field based on these expansions is given by:

$$\begin{aligned} u_x &= F_1(\zeta) u_{x_1} + F_2(\zeta) u_{x_2} + F_i(\zeta) u_{x_i} \\ u_y &= F_1(\zeta) u_{y_1} + F_2(\zeta) u_{y_2} + F_i(\zeta) u_{y_i} \\ u_z &= F_1(\zeta) u_{z_1} + F_2(\zeta) u_{z_2} + F_i(\zeta) u_{z_i} \end{aligned} \quad (2.2.4)$$

or, in a compact form, as:

$$u_x = \sum_{i=1}^{p+1} F_i(\zeta) u_{x_i}, \quad u_y = \sum_{i=1}^{p+1} F_i(\zeta) u_{y_i}, \quad u_z = \sum_{i=1}^{p+1} F_i(\zeta) u_{z_i} \quad (2.2.5)$$

where p denotes the polynomial order.

Within the CUF framework, such displacement fields have been successfully applied in layer-wise models for laminated structures (see, e.g., [106, 107, 130]).

Lagrange expansions have also been utilized extensively for variable kinematics plate and shell theories in a unified framework, particularly in the work of Kulikov and collaborators. Further insights into Lagrange-based 2D models can be found in [133, 134].

2.3 Finite Element formulation

The main advantage of the Carrera Unified Formulation (CUF) lies in its ability to express the governing equations and the associated finite element arrays in a compact and unified manner. This formulation remains formally invariant with respect to the choice of the expansion functions F_τ , ensuring a high degree of generality and adaptability.

The following sections provide a detailed mathematical derivation of the fundamental nucleus of the stiffness matrix for CUF-based 1D and 2D models.

2.3.1 Geometrical Relations and Constitutive Equations

In this section, the notation and reference system previously introduced in Section 2.2 are adopted. The three-dimensional displacement vector is defined as:

$$\mathbf{u}(x, y, z) = \begin{Bmatrix} u_x(x, y, z) \\ u_y(x, y, z) \\ u_z(x, y, z) \end{Bmatrix} \quad (2.3.1)$$

According to classical elasticity theory, stress and strain tensors can be rewritten as six-term vectors without loss of generality:

$$\begin{aligned} \boldsymbol{\sigma}^T &= \{ \sigma_{yy} \quad \sigma_{xx} \quad \sigma_{zz} \quad \sigma_{xz} \quad \sigma_{yz} \quad \sigma_{xy} \} \\ \boldsymbol{\varepsilon}^T &= \{ \varepsilon_{yy} \quad \varepsilon_{xx} \quad \varepsilon_{zz} \quad \varepsilon_{xz} \quad \varepsilon_{yz} \quad \varepsilon_{xy} \} \end{aligned} \quad (2.3.2)$$

Using this notation, the strain-displacement relationships can be compactly expressed as:

$$\boldsymbol{\varepsilon} = \mathbf{D} \mathbf{u} \quad (2.3.3)$$

where, assuming small deformations and infinitesimal rotations, $\mathbf{D}$ is the following

linear differential operator:

$$\mathbf{D} = \begin{bmatrix} 0 & \frac{\partial}{\partial y} & 0 \\ \frac{\partial}{\partial x} & 0 & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \end{bmatrix} \quad (2.3.4)$$

For isotropic materials, the stress-strain relationship is governed by Hooke's law:

$$\boldsymbol{\sigma} = \mathbf{C} \boldsymbol{\varepsilon} \quad (2.3.5)$$

where $\mathbf{C}$ is the isotropic stiffness matrix:

$$\mathbf{C} = \begin{bmatrix} C_{33} & C_{23} & C_{13} & 0 & 0 & 0 \\ C_{23} & C_{22} & C_{12} & 0 & 0 & 0 \\ C_{13} & C_{12} & C_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{66} \end{bmatrix} \quad (2.3.6)$$

The stiffness coefficients depend only on Young's modulus E and Poisson's ratio ν , and are given by:

$$\begin{aligned} C_{11} &= C_{22} = C_{33} = \frac{(1-\nu)E}{(1+\nu)(1-2\nu)} \\ C_{12} &= C_{13} = C_{23} = \frac{\nu E}{(1+\nu)(1-2\nu)} \\ C_{44} &= C_{55} = C_{66} = \frac{E}{2(1+\nu)} \end{aligned} \quad (2.3.7)$$

2.3.2 Finite Element Approximation of Displacements

In the case of one-dimensional (1D) models, the discretization process along the longitudinal axis of the beam is carried out using the finite element method (FEM). This approach allows the structural domain to be divided into a finite number of smaller elements, each of which is characterized by a set of nodal degrees of freedom. Within this framework, the generalized displacement field is approximated as a function of the unknown nodal displacement vector, denoted as $\mathbf{q}_{\tau i}$, and a set of interpolation functions, commonly referred to as shape functions $N_i(y)$. The displacement field for the 1D formulation can thus be expressed as:

$$\mathbf{u}_{\tau}(y) = N_i(y)\mathbf{q}_{\tau i}, \quad i = 1, 2, \dots, n_{elem} \quad (2.3.1)$$

where n_{elem} represents the number of nodes per element. The unknown nodal displacement vector, which encapsulates the degrees of freedom at each node, is defined as:

$$\mathbf{q}_{\tau i} = \left\{ \begin{matrix} q_{u_{x\tau i}} & q_{u_{y\tau i}} & q_{u_{z\tau i}} \end{matrix} \right\}^T \quad (2.3.2)$$

This discretization procedure effectively transforms the continuous problem into a discrete system of algebraic equations, which can be efficiently solved using numerical methods. The selection of appropriate shape functions plays a crucial role in ensuring the accuracy and stability of the finite element formulation.

A similar discretization approach is applied to two-dimensional (2D) models, where the displacement field is approximated over the midsurface of the plate using 2D shape functions $N_i(x, y)$. In this case, the generalized displacement field is given by:

$$\mathbf{u}_{\tau}(x, y) = N_i(x, y)\mathbf{q}_{\tau i}, \quad i = 1, 2, \dots, n_{elem} \quad (2.3.3)$$

where $N_i(x, y)$ are the two-dimensional shape functions, which serve to interpolate the displacement components over the domain of the element. These functions ensure the continuity of the displacement field across adjacent elements and contribute to the overall accuracy of the numerical model.

A variety of polynomial functions can be employed to define the shape functions used in finite element discretization. In this work, Lagrange interpolating polynomials have been selected for constructing both one-dimensional and two-dimensional finite elements. Lagrange polynomials provide a robust and widely used interpolation technique that enables accurate representation of the displacement field across the finite element mesh. The explicit formulation of these polynomials is not provided here for brevity; however, detailed descriptions can be found in the book by Carrera et al. [111], where different types of finite elements are described, including two-node (B2), three-node (B3), and four-node (B4) beam elements, as well as four-node (Q4), nine-node (Q9), and sixteen-node (Q16) plate elements.

By combining the finite element discretization procedure with the kinematic assumptions of the Carrera Unified Formulation (CUF), the three-dimensional displacement field can be expressed in a compact and generalized form as:

$$\mathbf{u} = F_{\tau} N_i \mathbf{q}_{\tau i} \quad (2.3.4)$$

where the expansion functions F_{τ} define the kinematic assumptions of the model, while the shape functions N_i characterize the spatial discretization of the element. The selection of these functions depends on whether the model is based on a beam or plate formulation. This formulation provides a unified framework

that facilitates the development of refined structural theories while maintaining computational efficiency.

2.3.3 Node-Dependent Kinematics (NDK) Approach

In Eq. 2.3.4, the shape functions N_i and the expansion functions F_τ are treated as independent entities, meaning that the interpolation of the generalized displacement field follows a decoupled formulation. However, recent advancements in the Carrera Unified Formulation (CUF) have introduced an alternative approach that establishes a direct coupling between these two function sets. This innovative concept, known as Node-Dependent Kinematics (NDK), was proposed by Carrera et al. [121, 122] as a means to enhance the flexibility and accuracy of finite element models. The NDK approach modifies the classical formulation by incorporating an additional dependence of the expansion functions on the nodal discretization, as expressed by the following equation:

$$\mathbf{u} = F_\tau^i N_i \mathbf{q}_{\tau i} \quad (2.3.1)$$

The key difference between Eq. 2.3.1 and the classical formulation in Eq. 2.3.4 lies in the introduction of the superscript i in the expansion function F_τ^i . This notation signifies that the expansion functions now exhibit an explicit dependency on the nodal discretization, meaning that the kinematic assumptions can vary locally at each finite element node. This novel formulation fundamentally alters the way in which structural theories are applied within the finite element framework, allowing for a spatially adaptable kinematic refinement.

One of the primary advantages of the NDK approach is that it enables localized refinement of the kinematic model without necessitating modifications to the global mesh structure. By selectively increasing the expansion order of F_τ at specific nodes, the accuracy of the displacement field approximation can be enhanced in critical regions while maintaining computational efficiency in less critical areas. This capability is particularly beneficial when analyzing structures with complex geometries, localized stress concentrations, or multi-scale behaviors. The flexibility introduced by NDK eliminates the need for special interface coupling techniques, as different kinematic assumptions naturally blend through the nodal shape functions within each finite element domain. Additionally, since the nodal interpolation functions inherently ensure continuity across the finite element mesh, no additional compatibility constraints are required when employing varying kinematic orders. The NDK methodology has proven to be highly effective in global-local modeling strategies, where a detailed, high-fidelity representation is needed in localized regions while maintaining a computationally efficient lower-order approximation in the surrounding areas. Within this framework, the finite element model in

the critical zone can be progressively refined until the desired level of accuracy is achieved, while the rest of the structure is modeled using standard lower-order kinematic assumptions. This allows for a seamless transition between different structural theories within a single numerical model, avoiding unnecessary computational overhead while preserving solution accuracy where needed.

An additional advantage of NDK is that it facilitates the reusability of finite element meshes for different levels of refinement. Since the kinematic assumptions are locally adjusted at the nodal level, the same mesh grid can be utilized to construct multiple hierarchical models within a family of concurrent global-local analyses. This feature is particularly advantageous for parametric studies and multi-fidelity simulations, where different levels of refinement can be applied to the same structural domain without the need for remeshing.

The effectiveness of the NDK approach has been demonstrated in various applications, particularly in the modeling of laminated composite structures, where different material layers may require distinct kinematic representations. The methodology has been successfully implemented in both 1D structural models (beam-like structures) [121] and 2D models (plate and shell structures) [122, 123, 124]. In these studies, NDK has enabled the precise representation of layer-wise behavior, ensuring accurate predictions of interlaminar stress distributions and localized deformation effects, which are crucial for assessing failure mechanisms in composite materials.

In conclusion, the NDK approach represents a significant advancement in the field of finite element modeling, offering a powerful tool for adaptive kinematic refinement. By introducing node-dependent assumptions, it allows for a fine-tuned balance between computational efficiency and solution accuracy, making it a promising technique for the analysis of complex aerospace and mechanical structures. The ability to dynamically refine the kinematic model at the nodal level provides unprecedented flexibility, eliminating the need for artificial constraints or mesh modifications while naturally integrating different structural theories within a unified computational framework [33, 34, 32, 31].

2.3.4 3D modelling for non-orthogonal geometries

Classical beam and plate theories are inherently limited when dealing with complex geometries, such as beams with variable cross-sections or sections that are not perpendicular to the longitudinal axis. Similarly, conventional plate theories are not well-suited for modeling plates with non-uniform thickness distributions or edges that deviate from an orthogonal configuration relative to the midsurface. These limitations arise because traditional structural theories assume simplified kinematic descriptions that do not account for the variations in geometry.

In the framework of the Node-Dependent Kinematic (NDK) approach, it is possible

to overcome these constraints and extend the Carrera Unified Formulation (CUF) to accommodate such complex geometries. This extension is achieved by integrating the CUF kinematic assumptions with the finite element method (FEM) in a unified three-dimensional (3D) approximation. The resulting formulation enables an accurate and computationally efficient representation of structures exhibiting non-standard geometries. The generalized displacement field in this extended CUF framework is given by:

$$\mathbf{u} = (F_\tau^i N_i) \mathbf{q}_{\tau i} = L_{\tau i}(\xi, \eta, \zeta) \mathbf{q}_{\tau i} \quad (2.3.1)$$

where ξ, η, ζ are the natural coordinates corresponding to the global Cartesian coordinates x, y, z . In this expression, the function

$$L_{\tau i} = (F_\tau^i N_i) \quad (2.3.2)$$

represents a generalized 3D shape function in which the expansion order may differ along different spatial directions. This flexibility allows the kinematic description to be refined in selected regions while keeping the computational cost manageable.

To ensure consistency in the derivation of the governing equations, the virtual variation of the displacement field must be defined in an analogous manner:

$$\delta \mathbf{u} = (F_s^j N_j) \delta \mathbf{q}_{sj} = L_{sj}(\xi, \eta, \zeta) \delta \mathbf{q}_{sj} \quad (2.3.3)$$

where summation indices s and j have been introduced to maintain generality in the numerical implementation.

By adopting this formulation, the traditional approach of decomposing volume integrals into separate one-dimensional (1D) and two-dimensional (2D) components is no longer necessary. Instead, the functions $(N_i F_\tau)$ and $(N_j F_s)$ are treated as conventional 3D shape functions, allowing for a fully three-dimensional treatment of the structural problem. Consequently, the Jacobian matrix associated with the transformation from the natural coordinate system ξ, η, ζ to the global coordinate system x, y, z is computed directly in 3D form, ensuring an accurate mapping of the finite element mesh onto the real geometry.

Examples: 1D B3-Q4 & 2D Q9-B2

To further clarify and demonstrate the proposed methodology, two representative examples are presented in detail. The first is a one-dimensional case (B3-Q4), and the second is a two-dimensional case (Q9-B2). For both cases, the implementation workflow is illustrated step by step, highlighting the key operations required to assemble the Jacobian matrix and compute the fundamental nuclei within the Node-Dependent Kinematic (NDK) framework.

1D Case: B3-Q4 In the B3-Q4 case, the kinematic field is approximated by combining shape functions defined along the structural axis (η) and across the cross-section (ζ). The displacement field is interpolated as:

$$\mathbf{u} = (F_\tau^i N_i) \mathbf{q}_{\tau i} = L_{\tau i}(\eta, \zeta) \mathbf{q}_{\tau i} \quad (2.3.4)$$

Here, $N_i(\eta)$ are the linear Lagrange shape functions along the axis (Q4 has 4 nodes), and $F_\tau^i(\xi, \zeta)$ are the quadratic shape functions in the cross-section (B3 has 3 nodes). The resulting Lagrange functions $L_{\tau i}(\xi, \eta, \zeta)$ are defined as the product:

$$L_{\tau i}(\xi, \eta, \zeta) = F_\tau^i(\xi, \zeta) \cdot N_i(\eta)$$

The explicit expressions for the functions F and N are:

$$\begin{aligned} N_1(\eta) &= \frac{1}{2}\eta(\eta - 1) \\ N_2(\eta) &= 1 - \eta^2 \\ N_3(\eta) &= \frac{1}{2}\eta(\eta + 1) \\ F_1(\xi, \zeta) &= \frac{1}{4}(1 - \zeta)(1 - \xi) \\ F_2(\xi, \zeta) &= \frac{1}{4}(1 + \zeta)(1 - \xi) \\ F_3(\xi, \zeta) &= \frac{1}{4}(1 + \zeta)(1 + \xi) \\ F_4(\xi, \zeta) &= \frac{1}{4}(1 - \zeta)(1 + \xi) \end{aligned}$$

The full set of 12 shape functions $L_{\tau i}(\eta, \zeta) = F_\tau(\eta)N_i(\zeta)$ is:

$$\begin{aligned} L_{1,1} &= \frac{1}{8}\eta(\eta - 1)(1 - \xi)(1 - \zeta) \\ L_{1,2} &= \frac{1}{8}\eta(\eta - 1)(1 + \zeta)(1 - \xi) \\ L_{1,3} &= \frac{1}{8}\eta(\eta - 1)(1 + \zeta)(1 + \xi) \\ L_{1,4} &= \frac{1}{8}\eta(\eta - 1)(1 - \zeta)(1 + \xi) \\ L_{2,1} &= \frac{1}{4}(1 - \eta^2)(1 - \xi)(1 - \zeta) \\ L_{2,2} &= \frac{1}{4}(1 - \eta^2)(1 - \xi)(1 + \zeta) \\ L_{2,3} &= \frac{1}{4}(1 - \eta^2)(1 + \xi)(1 + \zeta) \\ L_{2,4} &= \frac{1}{4}(1 - \eta^2)(1 + \xi)(1 - \zeta) \\ L_{3,1} &= \frac{1}{8}\eta(\eta + 1)(1 - \xi)(1 - \zeta) \\ L_{3,2} &= \frac{1}{8}\eta(\eta + 1)(1 - \xi)(1 + \zeta) \\ L_{3,3} &= \frac{1}{8}\eta(\eta + 1)(1 + \xi)(1 + \zeta) \\ L_{3,4} &= \frac{1}{8}\eta(\eta + 1)(1 + \xi)(1 - \zeta) \end{aligned}$$

The full approximation of a scalar field u within the element is thus given by the double summation:

$$\mathbf{u}(\eta, \zeta) = \sum_{\tau=1}^3 \sum_{i=1}^4 F_{\tau}^i(\xi, \zeta) N_i(\eta) \cdot \mathbf{q}_{\tau i} = \sum_{\tau=1}^3 \sum_{i=1}^4 L_{\tau i}(\xi, \eta, \zeta) \cdot \mathbf{q}_{\tau i} \quad (2.3.5)$$

Each function $L_{\tau i}$ corresponds to a degree of freedom. This configuration allows the displacement field to vary quadratically across the section and linearly along the axis.

2D Case: Q9-B2 In the Q9-B2 case, the kinematic field is approximated by combining shape functions defined in the in-plane directions (ξ, η) and along the thickness direction (ζ) . The displacement field is interpolated as:

$$\mathbf{u} = (F_{\tau}^i(\zeta) N_i(\xi, \eta)) \mathbf{q}_{\tau i} = L_{\tau i}(\xi, \eta, \zeta) \mathbf{q}_{\tau i} \quad (2.3.6)$$

Here, $N_i(\xi, \eta)$ are the bi-quadratic Lagrange shape functions for the Q9 element in the reference plane $[-1, 1] \times [-1, 1]$, while $F_{\tau}(\zeta)$ are the linear Lagrange shape functions used in the B2 formulation along the thickness. The functions are defined as:

$$L_{\tau i}(\xi, \eta, \zeta) = F_{\tau}(\zeta) \cdot N_i(\xi, \eta)$$

Given that the Q9 element has 9 nodes in the (ξ, η) plane and B2 has 2 nodes along ζ , a total of $2 \times 9 = 18$ functions $L_{\tau i}$ are constructed. Each function corresponds to a degree of freedom $\mathbf{q}_{\tau i}$, and allows accurate three-dimensional reconstruction of the kinematic field across the element.

$$F_1(\zeta) = \frac{1}{2}(1 - \zeta), \quad F_2(\zeta) = \frac{1}{2}(1 + \zeta)$$

$$\begin{aligned} N_1 &= \frac{1}{4}\xi(\xi - 1)\eta(\eta - 1) \\ N_2 &= \frac{1}{2}(1 - \xi^2)\eta(\eta - 1) \\ N_3 &= \frac{1}{4}\xi(\xi + 1)\eta(\eta - 1) \\ N_4 &= \frac{1}{2}\xi(\xi - 1)(1 - \eta^2) \\ N_5 &= (1 - \xi^2)(1 - \eta^2) \\ N_6 &= \frac{1}{2}\xi(\xi + 1)(1 - \eta^2) \\ N_7 &= \frac{1}{4}\xi(\xi - 1)\eta(\eta + 1) \\ N_8 &= \frac{1}{2}(1 - \xi^2)\eta(\eta + 1) \\ N_9 &= \frac{1}{4}\xi(\xi + 1)\eta(\eta + 1) \end{aligned}$$

The combination of the 2 functions in ζ and 9 in (ξ, η) yields 18 total shape functions $L_{\tau i}$, explicitly given by:

$$\begin{aligned}
L_{1,1} &= -\frac{1}{8}\xi\eta(\xi-1)(\eta-1)(1-\zeta) \\
L_{1,2} &= \frac{1}{4}\eta(\eta-1)(1-\xi^2)(1-\zeta) \\
L_{1,3} &= -\frac{1}{8}\xi\eta(\xi+1)(\eta-1)(1-\zeta) \\
L_{1,4} &= \frac{1}{4}\xi(\xi-1)(1-\eta^2)(1-\zeta) \\
L_{1,5} &= (1-\xi^2)(1-\eta^2)\frac{1}{2}(1-\zeta) \\
L_{1,6} &= \frac{1}{4}\xi(\xi+1)(1-\eta^2)(1-\zeta) \\
L_{1,7} &= -\frac{1}{8}\xi\eta(\xi-1)(\eta+1)(1-\zeta) \\
L_{1,8} &= \frac{1}{4}\eta(\eta+1)(1-\xi^2)(1-\zeta) \\
L_{1,9} &= -\frac{1}{8}\xi\eta(\xi+1)(\eta+1)(1-\zeta) \\
L_{2,1} &= -\frac{1}{8}\xi\eta(\xi-1)(\eta-1)(1+\zeta) \\
L_{2,2} &= \frac{1}{4}\eta(\eta-1)(1-\xi^2)(1+\zeta) \\
L_{2,3} &= -\frac{1}{8}\xi\eta(\xi+1)(\eta-1)(1+\zeta) \\
L_{2,4} &= \frac{1}{4}\xi(\xi-1)(1-\eta^2)(1+\zeta) \\
L_{2,5} &= (1-\xi^2)(1-\eta^2)\frac{1}{2}(1+\zeta) \\
L_{2,6} &= \frac{1}{4}\xi(\xi+1)(1-\eta^2)(1+\zeta) \\
L_{2,7} &= -\frac{1}{8}\xi\eta(\xi-1)(\eta+1)(1+\zeta) \\
L_{2,8} &= \frac{1}{4}\eta(\eta+1)(1-\xi^2)(1+\zeta) \\
L_{2,9} &= -\frac{1}{8}\xi\eta(\xi+1)(\eta+1)(1+\zeta)
\end{aligned}$$

The full approximation of a scalar field u within the element is thus given by the double summation:

$$\mathbf{u}(\xi, \eta, \zeta) = \sum_{\tau=1}^2 \sum_{i=1}^9 F_{\tau}^i(\zeta) N_i(\xi, \eta) \cdot \mathbf{q}_{\tau i} = \sum_{\tau=1}^2 \sum_{i=1}^9 L_{\tau i}(\xi, \eta, \zeta) \cdot \mathbf{q}_{\tau i} \quad (2.3.7)$$

These functions represent the full three-dimensional interpolation basis used to approximate the kinematic variables in the element, each multiplied by a corresponding degree of freedom $\mathbf{q}_{\tau i}$. This representation, made possible by the CUF-NDK framework, provides a flexible and powerful formulation to capture complex structural behaviors across the full element domain.

As illustrated in the examples, this novel approach offers a significant computational advantage, as it allows for a substantial reduction in the number of degrees of freedom compared to traditional finite element methods that rely on classical 3D elements. Conventional 3D elements typically enforce the same expansion order along all three spatial directions, which imposes geometric constraints on the aspect ratio of the element. In many cases, this restriction leads to unnecessarily dense meshes, particularly in regions where the structural dimensions vary

significantly in one direction. By contrast, the proposed CUF-based formulation with NDK introduces anisotropic kinematic refinements, where different expansion orders can be assigned to different spatial directions based on the specific requirements of the geometry. This capability not only enhances accuracy but also optimizes computational efficiency, as it avoids the excessive refinement of elements in directions where lower-order expansions would suffice.

The advantages of this methodology are particularly evident in applications involving beam-like structures with varying cross-sections, plates with tapered or curved edges, and aerospace components with complex structural configurations. By leveraging the synergy between CUF, FEM, and NDK, the proposed framework enables the accurate modeling of non-orthogonal geometries while maintaining the computational efficiency of reduced-order structural models.

2.3.5 Jacobian computation for Node-Dependent Kinematic

One of the key aspects of the Node-Dependent Kinematic (NDK) model, which combines the Carrera Unified Formulation (CUF) with the Finite Element Method (FEM), is the computation of the Jacobian matrix. The Jacobian plays a fundamental role in the transformation between the global and local coordinate systems, directly affecting the accuracy of the numerical integration and the evaluation of strain-displacement relationships. Moreover, understanding how to compute the Jacobian is essential for deriving the fundamental nuclei of the unified formulation, as it directly influences the evaluation of differential operators and the assembly of elemental matrices.

As expressed in the previous section and further detailed in the following one, the calculation of the Fundamental Nuclei (FN) for both the stiffness and mass matrices requires the evaluation of the following integrals:

$$\text{FN} = C \int_V f(x, y, z) dV \quad (2.3.1)$$

where C is a constant that can be either a mass parameter or a material stiffness constant, and $f(x, y, z)$ is a generic function that involves the product of the derivatives of the shape functions. When the Carrera Unified Formulation is introduced with FEM, using Eqn. 2.3.1, Eqn. 2.3.2 and Eqn. 2.3.3 the expression becomes:

$$\text{FN} = C \int_V (N_i F_\tau)(N_j F_s) dV \quad (2.3.2)$$

which represents the generic FN. When the integrals are explicated with respect to the generalized coordinates, the Jacobian appears:

$$\int_V f(x, y, z) dV(x, y, z) = \int_V f(\xi, \eta, \zeta) |\mathbf{J}(\xi, \eta, \zeta)| dV(\xi, \eta, \zeta) \quad (2.3.3)$$

In the following subsections, the procedure for computing the Jacobian will be detailed step by step. First, the discretization of the element coordinates will be introduced, followed by the derivation of the shape functions and their derivatives. Finally, the formulation of the Jacobian matrix will be presented for different element configurations, including one-dimensional (1D), two-dimensional (2D), and three-dimensional (3D) cases.

Jacobian computation for 1D element

The 1D element coordinates are discretized as follow:

$$\begin{aligned} x &= \sum_{\tau=1}^{N_{\text{CUF}}} F_{\tau}(\xi, \zeta) \cdot x_i \\ y &= \sum_{i=1}^{N_{\text{FEM}}} N_i(\eta) \cdot y_i \\ z &= \sum_{\tau=1}^{N_{\text{CUF}}} F_{\tau}(\xi, \zeta) \cdot z_i \end{aligned} \quad (2.3.4)$$

The axis is considered along the y -direction, while the section lies in the x - z plane. Note that the FEM discretization is performed along the axis, meaning that y_i represents the nodal coordinates.

To obtain a 3D case, two Jacobian will be computed: a 1D Jacobian ($\mathbf{J}^{1D}$) referred to the axis coordinate and a 2D Jacobian ($\mathbf{J}^{2D}$) referred to the section plane coordinates.

First, the partial derivatives are computed:

$$\begin{aligned} \frac{\partial x}{\partial \xi} &= \sum_{\tau=1}^{N_{\text{CUF}}} F_{\tau,\xi}(\xi, \zeta) \cdot x_i & \frac{\partial x}{\partial \zeta} &= \sum_{\tau=1}^{N_{\text{CUF}}} F_{\tau,\zeta}(\xi, \zeta) \cdot x_i \\ \frac{\partial y}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} N_{i,\eta}(\eta) \cdot y_i \\ \frac{\partial z}{\partial \xi} &= \sum_{\tau=1}^{N_{\text{CUF}}} F_{\tau,\xi}(\xi, \zeta) \cdot z_i & \frac{\partial z}{\partial \zeta} &= \sum_{\tau=1}^{N_{\text{CUF}}} F_{\tau,\zeta}(\xi, \zeta) \cdot z_i \end{aligned} \quad (2.3.5)$$

The computation of the 1D Jacobian is straightforward:

$$\frac{\partial}{\partial \eta} = \underbrace{\left[\frac{\partial y}{\partial \eta} \right]}_{\mathbf{J}^{1D}} \frac{\partial}{\partial y} \quad (2.3.6)$$

The computation of the 2D Jacobian is:

$$\begin{pmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \zeta} \end{pmatrix} = \underbrace{\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix}}_{\mathbf{J}^{2D}} \begin{pmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial z} \end{pmatrix} \quad (2.3.7)$$

Jacobian computation for 2D element

The 2D element coordinates are discretized as follow:

$$\begin{aligned} x &= \sum_{i=1}^{N_{\text{FEM}}} N_i(\xi, \eta) \cdot x_i \\ y &= \sum_{i=1}^{N_{\text{FEM}}} N_i(\xi, \eta) \cdot y_i \\ z &= \sum_{\tau=1}^{N_{\text{CUF}}} F_{\tau}(\zeta) \cdot z_i \end{aligned} \quad (2.3.8)$$

The surface is considered in the x - y plane, while the thickness is in the z -direction. Note that the FEM discretization is performed on the plane, meaning that x_i and y_i represent the nodal coordinates.

To obtain a 3D case, two Jacobian will be computed: a 2D Jacobian ($\mathbf{J}^{2D}$) referred to the surface coordinates and a 1D Jacobian ($\mathbf{J}^{1D}$) referred to the thickness coordinate.

First, the partial derivatives are computed:

$$\begin{aligned}
 \frac{\partial x}{\partial \xi} &= \sum_{i=1}^{N_{\text{FEM}}} N_{i,\xi}(\xi, \eta) \cdot x_i & \frac{\partial x}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} N_{i,\eta}(\xi, \eta) \cdot x_i \\
 \frac{\partial y}{\partial \xi} &= \sum_{i=1}^{N_{\text{FEM}}} N_{i,\xi}(\xi, \eta) \cdot y_i & \frac{\partial y}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} N_{i,\eta}(\xi, \eta) \cdot y_i \\
 \frac{\partial z}{\partial \zeta} &= \sum_{\tau=1}^{N_{\text{CUF}}} F_{\tau,\zeta}(\zeta) \cdot z_i
 \end{aligned} \tag{2.3.9}$$

The computation of the 2D Jacobian is:

$$\begin{pmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{pmatrix} = \underbrace{\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix}}_{\mathbf{J}^{2D}} \begin{pmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{pmatrix} \tag{2.3.10}$$

The computation of the 1D Jacobian is:

$$\frac{\partial}{\partial \zeta} = \underbrace{\begin{bmatrix} \frac{\partial z}{\partial \zeta} \end{bmatrix}}_{\mathbf{J}^{1D}} \frac{\partial}{\partial z} \tag{2.3.11}$$

Jacobian computation for 3D element

The novelty of the proposed approach lies in treating the entire element as a 3D element.

In this context, by exploiting the Node-Dependent Kinematic, the coordinates for the 1D element are discretized as follows:

$$\begin{aligned}
 x &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} [N_i(\eta) F_{\tau}(\xi, \zeta)] \cdot x_{\tau i} \\
 y &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} [N_i(\eta) F_{\tau}(\xi, \zeta)] \cdot y_{\tau i} \\
 z &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} [N_i(\eta) F_{\tau}(\xi, \zeta)] \cdot z_{\tau i}
 \end{aligned} \tag{2.3.12}$$

Then, we need to compute all the partial derivatives involved:

$$\begin{aligned}
 \frac{\partial x}{\partial \xi} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\eta) F_{\tau,\xi}(\xi, \zeta) \cdot x_{\tau i} \\
 \frac{\partial x}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\eta}(\eta) F_{\tau}(\xi, \zeta) \cdot x_{\tau i} \\
 \frac{\partial x}{\partial \zeta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\eta) F_{\tau,\zeta}(\xi, \zeta) \cdot x_{\tau i} \\
 \frac{\partial y}{\partial \xi} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\eta) F_{\tau,\xi}(\xi, \zeta) \cdot y_{\tau i} \\
 \frac{\partial y}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\eta}(\eta) F_{\tau}(\xi, \zeta) \cdot y_{\tau i} \\
 \frac{\partial y}{\partial \zeta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\eta) F_{\tau,\zeta}(\xi, \zeta) \cdot y_{\tau i} \\
 \frac{\partial z}{\partial \xi} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\eta) F_{\tau,\xi}(\xi, \zeta) \cdot z_{\tau i} \\
 \frac{\partial z}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\eta}(\eta) F_{\tau}(\xi, \zeta) \cdot z_{\tau i} \\
 \frac{\partial z}{\partial \zeta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\eta) F_{\tau,\zeta}(\xi, \zeta) \cdot z_{\tau i}
 \end{aligned} \tag{2.3.13}$$

If we consider the 2D element, the coordinates are:

$$\begin{aligned}
 x &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} [N_i(\xi, \eta) F_{\tau}(\zeta)] \cdot x_{\tau i} \\
 y &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} [N_i(\xi, \eta) F_{\tau}(\zeta)] \cdot y_{\tau i} \\
 z &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} [N_i(\xi, \eta) F_{\tau}(\zeta)] \cdot z_{\tau i}
 \end{aligned} \tag{2.3.14}$$

and the partial derivatives are:

$$\begin{aligned}
 \frac{\partial x}{\partial \xi} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\xi}(\xi, \eta) F_{\tau}(\zeta) \cdot x_{\tau i} \\
 \frac{\partial x}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\eta}(\xi, \eta) F_{\tau}(\zeta) \cdot x_{\tau i} \\
 \frac{\partial x}{\partial \zeta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\xi, \eta) F_{\tau,\zeta}(\zeta) \cdot x_{\tau i} \\
 \frac{\partial y}{\partial \xi} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\xi}(\xi, \eta) F_{\tau}(\zeta) \cdot y_{\tau i} \\
 \frac{\partial y}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\eta}(\xi, \eta) F_{\tau}(\zeta) \cdot y_{\tau i} \\
 \frac{\partial y}{\partial \zeta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\xi, \eta) F_{\tau,\zeta}(\zeta) \cdot y_{\tau i} \\
 \frac{\partial z}{\partial \xi} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\xi}(\xi, \eta) F_{\tau}(\zeta) \cdot z_{\tau i} \\
 \frac{\partial z}{\partial \eta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_{i,\eta}(\xi, \eta) F_{\tau}(\zeta) \cdot z_{\tau i} \\
 \frac{\partial z}{\partial \zeta} &= \sum_{i=1}^{N_{\text{FEM}}} \sum_{\tau=1}^{N_{\text{CUF}}} N_i(\xi, \eta) F_{\tau,\zeta}(\zeta) \cdot z_{\tau i}
 \end{aligned} \tag{2.3.15}$$

The general matrix form for the 3D Jacobian ($\mathbf{J}^{3D}$) is:

$$\begin{aligned}
 \left\{ \begin{array}{c} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \\ \frac{\partial}{\partial \zeta} \end{array} \right\} &= \underbrace{\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix}}_{\mathbf{J}^{3D}} \left\{ \begin{array}{c} \frac{\partial x}{\partial \xi} \\ \frac{\partial y}{\partial \xi} \\ \frac{\partial z}{\partial \xi} \end{array} \right\}
 \end{aligned} \tag{2.3.16}$$

In Eqn. 2.3.16, it is evident that 9 partial derivatives are involved, compared to only 5 (highlighted in red) in the combination of $\mathbf{J}^{1D}$ and $\mathbf{J}^{2D}$ from the previous cases. This increased number of derivatives is not merely a mathematical refinement but a fundamental characteristic of the Node-Dependent Kinematic (NDK) model, which provides a more comprehensive and flexible description of the element's deformation and kinematics. By incorporating all the necessary transformation terms, this enhanced Jacobian formulation ensures full control over the element, allowing for precise adjustments to the nodal positions. This capability is particularly advantageous in complex study cases and engineering applications where adaptability and accuracy are critical. The ability to reposition nodes dynamically to fit specific structural or computational requirements highlights the true power of the NDK model, making it a robust and versatile tool for high-fidelity finite element analysis.

2.3.6 Governing Equations

The governing equations of the finite element formulation within the Carrera Unified Formulation (CUF) framework are derived based on the principle of virtual displacements. This principle establishes that the virtual variation of the internal work must be equal and opposite to the sum of the virtual variations of the inertial work and the external work, ensuring equilibrium within the structure. Mathematically, this can be expressed as:

$$\delta L_{\text{int}} + \delta L_{\text{ine}} = \delta L_{\text{ext}} \quad (2.3.1)$$

where δL_{int} represents the internal work, δL_{ine} denotes the inertial work, and δL_{ext} corresponds to the external work applied to the structure.

2.3.6.0.1 Internal Work

The internal work is equivalent to the elastic strain energy stored within the domain V :

$$\delta L_{\text{int}} = \int_V \delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} dV \quad (2.3.2)$$

where $\boldsymbol{\varepsilon}$ represents the strain tensor and $\boldsymbol{\sigma}$ is the stress tensor. The internal work equation can be rewritten using the geometrical relations (Eq. 2.3.3), the constitutive law (Eq. 2.3.5), the CUF kinematic assumptions (Eq. 2.2.1), and the finite element discretization (Eq. 2.3.1). Substituting these expressions, the internal work can be rewritten in terms of the generalized displacement components as:

$$\delta L_{\text{int}} = \delta \mathbf{q}_{\tau i}^T \mathbf{K}^{\tau s i j} \mathbf{q}_{s j} \quad (2.3.3)$$

where $\mathbf{K}^{\tau s i j}$ is the fundamental nucleus of the stiffness matrix, a 3×3 matrix whose components are given by the volume integrals of the stiffness coefficients and the derivatives of the shape functions and expansion functions, as presented in Eqns. (2.3.4)

$$\begin{aligned} K_{xx}^{\tau s i j} &= C_{22} \int_V (N_i F_\tau)_{,x} (N_j F_s)_{,x} dV + C_{44} \int_V (N_i F_\tau)_{,z} (N_j F_s)_{,z} dV + \\ &\quad + C_{66} \int_V (N_i F_\tau)_{,y} (N_j F_s)_{,y} dV \\ K_{xy}^{\tau s i j} &= C_{23} \int_V (N_i F_\tau)_{,y} (N_j F_s)_{,x} dV + C_{66} \int_V (N_i F_\tau)_{,x} (N_j F_s)_{,y} dV \\ K_{xz}^{\tau s i j} &= C_{12} \int_V (N_i F_\tau)_{,z} (N_j F_s)_{,x} dV + C_{44} \int_V (N_i F_\tau)_{,x} (N_j F_s)_{,z} dV \\ K_{yx}^{\tau s i j} &= C_{23} \int_V (N_i F_\tau)_{,x} (N_j F_s)_{,y} dV + C_{66} \int_V (N_i F_\tau)_{,y} (N_j F_s)_{,x} dV \\ K_{yy}^{\tau s i j} &= C_{33} \int_V (N_i F_\tau)_{,y} (N_j F_s)_{,y} dV + C_{55} \int_V (N_i F_\tau)_{,z} (N_j F_s)_{,z} dV + \\ &\quad + C_{66} \int_V (N_i F_\tau)_{,x} (N_j F_s)_{,x} dV \\ K_{yz}^{\tau s i j} &= C_{13} \int_V (N_i F_\tau)_{,z} (N_j F_s)_{,y} dV + C_{55} \int_V (N_i F_\tau)_{,y} (N_j F_s)_{,z} dV \\ K_{zx}^{\tau s i j} &= C_{12} \int_V (N_i F_\tau)_{,x} (N_j F_s)_{,z} dV + C_{44} \int_V (N_i F_\tau)_{,z} (N_j F_s)_{,x} dV \\ K_{zy}^{\tau s i j} &= C_{13} \int_V (N_i F_\tau)_{,y} (N_j F_s)_{,z} dV + C_{55} \int_V (N_i F_\tau)_{,z} (N_j F_s)_{,y} dV \\ K_{zz}^{\tau s i j} &= C_{11} \int_V (N_i F_\tau)_{,z} (N_j F_s)_{,z} dV + C_{44} \int_V (N_i F_\tau)_{,x} (N_j F_s)_{,x} dV + \\ &\quad + C_{55} \int_V (N_i F_\tau)_{,y} (N_j F_s)_{,y} dV \end{aligned} \quad (2.3.4)$$

This formulation ensures that the stiffness matrix remains independent of the choice of expansion functions F_τ , which define the structural theory, and the shape functions N_i , which define the finite element interpolation.

2.3.6.0.2 Inertial Work

The inertial work, corresponding to the kinetic energy variation, is given by:

$$\delta L_{\text{ine}} = \int_V \delta \mathbf{u}^T \rho \ddot{\mathbf{u}} dV \quad (2.3.5)$$

where ρ is the mass density of the material and $\ddot{\mathbf{u}}$ is the acceleration field. Applying the CUF kinematic field and the finite element discretization (Eqs. 2.3.1 and 2.3.3), the inertial work can be rewritten as:

$$\delta L_{\text{ine}} = \delta \mathbf{q}_{\tau i}^T \mathbf{M}^{\tau s i j} \ddot{\mathbf{q}}_{s j} \quad (2.3.6)$$

where $\mathbf{M}^{\tau s i j}$ is the fundamental nucleus of the mass matrix, which is also a 3×3 matrix obtained through volume integrals over the shape functions and expansion functions.

2.3.6.0.3 External Work

The external work contribution, representing the work done by external forces, is formulated as:

$$\delta L_{\text{ext}} = \int_V \delta \mathbf{u}^T \mathbf{g} dV \quad (2.3.7)$$

where $\mathbf{g}$ represents the external force field acting on the structure. Following the same discretization procedure applied to the internal work, the external work is rewritten in terms of the nodal displacement vector $q_{\tau i}$ as:

$$\delta L_{\text{ext}} = \delta \mathbf{q}_{\tau i}^T \mathbf{P}^{\tau i} \quad (2.3.8)$$

where $\mathbf{P}^{\tau i}$ is the generalized load vector, defined in terms of the shape functions N_i and the expansion functions F_τ .

2.3.7 Finite Element Discretization and Structural Modeling

A key feature of this formulation is that the fundamental nuclei of the stiffness and mass matrices are independent of the specific choice of expansion functions F_τ and shape functions N_i . This allows for the automatic generation of beam and plate elements by appropriately expanding the fundamental nuclei according to the indices τ, s, i, j .

In this work, linear, quadratic, and cubic Lagrange polynomials are used for both F_τ and N_i , ensuring a versatile and high-fidelity representation of the structural response. Specifically:

- The acronyms B2, B3, and B4 refer to beam elements, where the Lagrange polynomials are applied both for the discretization along the longitudinal axis and for the expansion through the thickness.
- The acronyms Q4, Q9, and Q16 refer to plate elements, where these polynomials are used both for the cross-sectional expansion (beam models) and for the discretization of the midsurface (plate models).

This flexibility allows the kinematic description to be tailored to specific structural configurations, ensuring that different levels of refinement can be applied locally in regions of interest. The formulation naturally supports multi-domain cross-sections, enabling enhanced modeling capabilities for structures with complex geometries or varying material properties.

2.3.8 Computational Advantages

The proposed governing equations, formulated within the CUF framework, provide a generalized and computationally efficient approach for modeling both beam-like and plate-like structures. By integrating CUF with the finite element method, the approach allows:

- The automatic generation of beam and plate elements with different kinematic descriptions.
- The ability to apply local kinematic refinements without modifying the global mesh structure.
- The use of multi-domain expansions, where different polynomial orders can be applied selectively to different regions.
- The ability to efficiently handle non-orthogonal geometries, including beams with variable cross-sections and plates with tapered or curved edges.

These features make the method particularly suitable for applications in aerospace, mechanical, and civil engineering, where structural efficiency and computational accuracy are crucial for large-scale simulations.

Chapter 3

Numerical Results and Discussion

3.1 Computational Setup and Reference Solvers

The numerical framework developed in this work relies on a combination of in-house academic codes and commercial software tools, each contributing to different aspects of the computational workflow. The core numerical routines were implemented using the Fortran programming language, chosen for its efficiency in handling intensive numerical computations and its long-standing reliability in structural mechanics applications. These codes were specifically tailored to address the structural problems under investigation, including customized solvers and post-processing routines developed in a modular fashion to ensure flexibility and extensibility.

Alongside the academic codes, MATLAB was extensively used as a supportive environment for data preprocessing, visualization, and comparative analysis. Its matrix-oriented architecture and built-in plotting capabilities facilitated rapid prototyping of numerical experiments and enabled efficient interpretation of the simulation results. MATLAB also served as a bridge between the different computational modules, allowing for the implementation of custom scripts to manage input/output operations and to perform sensitivity or parametric studies.

For instance, dedicated MATLAB routines were developed to automatically generate structured and perturbed nodal meshes, which served as input to the Fortran-based finite element solver. The following two examples illustrate the generation of a regular and a distorted mesh, respectively `mesh_NODES` and `mesh_NODES_dist`.

```
function [mesh] = mesh_NODES(length_horiz,base_node,n_el)

% Dimension of plate: length_horiz x length_vert x thickness
% Number of nodes for element base: base_node (ex. Q16 -> 4)
% Numer of element per row: n_el (ex. 15x15)
```

```
% Number of nodes per row: n_node_row

n_node_row = base_node + (base_node-1)*(n_el-1);

% Compute distance between two nodes: d
% Horizontal length of plate

d = length_horiz/(n_node_row-1);

% Mesh dimension

mesh_dim = n_node_row^2;

% Creation of input file NODES

mesh = zeros(mesh_dim,7);

% Coloumn 1: element ID

for i = 1:mesh_dim
    mesh(:,1) = [1:mesh_dim];
end

% Coloumn 2: x-coordinate

x_vec_0 = [1:n_node_row:mesh_dim];

for i = 1:n_node_row-1
    mesh(x_vec_0+i,2) = zeros(length(x_vec_0),1) + d*i;
end

% Coloumn 3: y-coordinate

y_vec_0 = [1:n_node_row];

for i = 1:n_node_row-1
    mesh(y_vec_0+n_node_row*i,3) = zeros(length(y_vec_0),1) + d*i;
end

% Coloumn 4: z-coordinate
% They are all zero already

% Coloumn 5: element type
% N.B. This cycle actually doesn't work. It is necessary to open the matrix
% with an editor (Notepad++) and substitute all the values for coloumn 5

for i = 1:mesh_dim
    mesh(i,5) = LE;
```

```
end

% Coloumn 6: material
mesh(:,6) = ones(mesh_dim,1);

% Coloumn 7: kinematics
mesh(:,7) = ones(mesh_dim,1);

end

function [mesh] = mesh_NODES_dist(length_horiz,base_node,n_el,ang)

% Dimension of plate: length_horiz x length_vert x thickness
% Number of nodes for element base: base_node (ex. Q16 -> 4)
% Numer of element per row: n_el (ex. 15x15)

% Number of nodes per row: n_node_row

n_node_row = base_node + (base_node-1)*(n_el-1);

% Compute distance between two nodes: d
% Horizontal length of plate

d = length_horiz/(n_node_row-1);

% Mesh dimension

mesh_dim = n_node_row^2;

% Creation of input file NODES

mesh = zeros(mesh_dim,7);

% Coloumn 1: element ID

for i = 1:mesh_dim
%     mesh(:,1) = [1:mesh_dim];
    mesh(:,1) = [360:1:359+625];
end

% Coloumn 2: x-coordinate

x_vec_0 = [1:n_node_row];

for i = 1:n_node_row-1
```

```
mesh(x_vec_0,2) = [0:d:length_horiz];
mesh(x_vec_0+n_node_row*i,2) = [0:d:length_horiz] + ...
                                d*cos(deg2rad(ang))*i;
end

% Coloumn 3: y-coordinate

y_vec_0 = [1:n_node_row];

for i = 1:n_node_row-1
    mesh(y_vec_0+n_node_row*i,3) = zeros(length(y_vec_0),1) + ...
                                    d*sin(deg2rad(ang))*i;
end

% Coloumn 4: z-coordinate
% They are all zero already

% Coloumn 5: element type
% N.B. This cycle actually doesn't work. It is necessary to open the matrix
% with an editor (Notepad++) and substitute all the values for coloumn 5

for i = 1:mesh_dim
    mesh(i,5) = LE;
end

% Coloumn 6: material

mesh(:,6) = ones(mesh_dim,1);

% Coloumn 7: kinematics

% mesh(:,7) = ones(mesh_dim,1);
mesh(:,7) = 5*ones(mesh_dim,1);

end
```

Once the mesh is defined, the final command exports the nodal information in ASCII format, creating an input file for the Fortran-based solver MUL2:

```
[mesh] = mesh_NODES(100,4,20);

save NODES.dat mesh -ascii -double
```

To further support the numerical investigation and provide a benchmark for validation, commercial finite element tools such as MSC Patran and NASTRAN were employed. These software packages were used to generate high-fidelity reference models and to perform parallel analyses under similar boundary conditions and loading configurations. The comparison between the results obtained from

the in-house codes and those produced by the commercial solvers allowed for a rigorous verification of the implemented numerical schemes, and also highlighted potential limitations or assumptions embedded in the academic models.

3.2 Overview of the Investigated Problems

This chapter collects and discusses the numerical results obtained within the framework of the Carrera Unified Formulation (CUF), with particular emphasis on adaptive and non-conventional finite element strategies enabled by the Node-Dependent Kinematics (NDK) approach. The purpose is to present the outcomes as a coherent chapter of a PhD thesis in aerospace structures, focusing on methodological consistency, verification against reference commercial solvers, and engineering interpretation of the computed responses.

From an aerospace structural perspective, the analysed configurations address representative challenges encountered in practical design and assessment workflows: (i) localized loading and stress concentrations (typical of attachments, fittings, discrete actuators and local load introductions); (ii) geometric non-orthogonality and element distortion (typical of skew panels, swept regions, and irregular partitions in complex assemblies); (iii) the trade-off between accuracy and computational cost (typical of preliminary sizing, design iterations, and model updating); (iv) modal characteristics and vibration behaviour (critical in rotordynamics and in lightweight aerospace structures where dynamic margins and resonances are key).

The investigated cases span both 2D and 1D CUF-based models and include:

- a preliminary study on Adaptive Mesh (AM) for free-vibration analysis, used to highlight the effect of mixed kinematics and 1D–2D couplings;
- benchmark plate problems under concentrated forces (regular and skew geometries), used for validation and baseline comparisons;
- reduced and adaptive discretizations obtained by coupling plate and beam finite elements through non-conventional kinematics, aimed at DOFs reduction while retaining accuracy near critical regions;

Parts of the work discussed in this chapter have been disseminated in: (i) *Mechanics of Advanced Materials and Structures* (available at <https://www.tandfonline.com/doi/abs/10.1080/15376494.2022.2126039>); (ii) *Aeronautics and Astronautics: AIDAA XXVII International Congress* (available at <https://mrforum.com/product/9781644902813-68/>); (iii) *ASME Aerospace Structures, Structural Dynamics, and Materials Conference* (available at <https://asmedigitalcollection.asme.org/ssdm/proceedings-abstract/SSDM2023/87141/1169203>). The present chapter reorganizes, expands and harmonizes the discussion to match a unified thesis narrative.

- distorted meshes, including convex and non-convex element configurations, used to assess robustness with respect to mesh irregularities;
- free-vibration analyses of plates and of a variable-thickness rotating disk, used to assess dynamic accuracy and the sensitivity of natural frequencies and mode shapes to discretization choices.

All the problems are analysed using CUF-based finite elements implemented in the academic code MUL2 and compared with reference solutions obtained from commercial finite element solvers, namely Patran/Nastran and ANSYS. The comparison is carried out in terms of displacement fields, stress distributions, natural frequencies, and computational efficiency, quantified through the number of degrees of freedom (DOFs).

To provide a clear interpretation, the results are grouped according to the type of analysis performed. Static analyses are discussed first, followed by dynamic investigations. This organization supports a direct cross-comparison of modelling strategies under different loading and boundary conditions, and it enables a structured discussion of accuracy versus efficiency, which is particularly relevant in aerospace design and certification-oriented modelling.

3.3 Adaptive Mesh: Preliminary Free-Vibration Assessment

A preliminary analysis is performed to study the Adaptive Mesh (AM) based on the Carrera Unified Formulation (CUF). In particular, the free vibration analysis is considered, which investigates the equilibrium between elastic forces and inertial forces. After choosing a reference solution, two other cases are analyzed and compared with the reference one.

3.3.1 The model

The CUF allows to consider different kinematics assumptions for the Finite Elements (FE), which are considered always as 3D elements. In this way is possible to combine 1D, 2D and 3D elements to create the suitable mesh for the structure. In particular, in this work different types of elements are used to model a plate of dimension $1 \times 1 \times 0.1$ m. The elements adopted are:

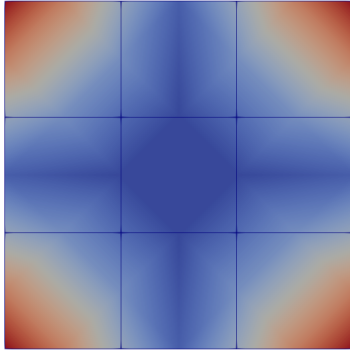
- 1D beam elements: B2 and B3
- 2D plate elements: Q4 and Q9

where the first letter denotes the type of element used and the relative kinematics (B-beam or Q-plate) and the number represents how many nodes there are for each element (for example, Q4 is a plate element with 4 nodes).

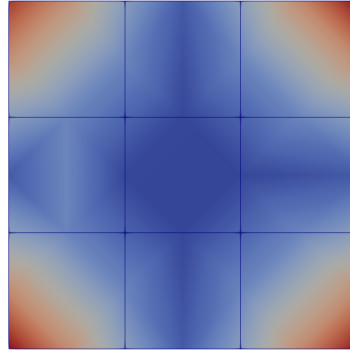
The reference solution is the plate modelled with 9 plate elements Q9 with a kinematics B3 along the thickness (Q9B3). The two cases compared are: a plate modelled with 9 plate elements Q4 with a kinematics B2 along the thickness (Q4B2) and a plate modelled combining all the previous stated elements, in particular 6 plate elements Q4 with B2 for the thickness, 1 plate element Q9 with B3 for the thickness and 2 beam elements with Q4 or Q9 for the kinematics on the section (Mixed Elements).

3.3.2 Results

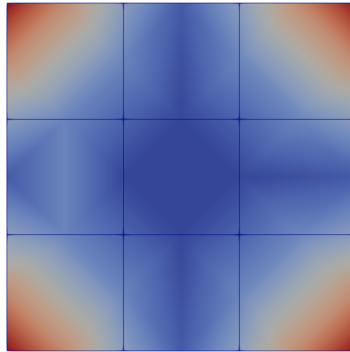
The results are here reported. In particular, in Figure 3.3.1 the comparison among the three adopted models is reported for the first mode.



(a) *Mode 1: Q9B3*



(b) *Mode 1: Mixed Elements*



(c) *Mode 1: Q4B2*

Figure 3.3.1: Comparison for the first mode

Then, a list of the first 10 frequencies for all the models adopted is listed in Table 3.3.1 with the relative number of Degrees Of Freedom (DOF).

Model	Mixed Elements				Q4B2	
Mode	1xQ9	6xQ4	2xB2	B3B2	9xQ4	B2
1				0.0666		0.0674
2				0.2052		0.2478
3				0.2249		0.2621
4				0.2708		0.2862
5				0.2785		0.2862
6				0.4257		0.4389
7				0.4254		0.4429
8				0.4570		0.4741
9				0.4578		0.4639
10				0.4678		0.4639
DOFs	153				96	

Table 3.3.1: First 10 natural frequencies and DOFs

Finally, the percentage error is computed with respect to the reference model in Table 3.3.2.

Mode	Q9B3 vs Mixed Elements	Q9B3 vs Q4B2
1	4.20	5.51

Table 3.3.2: Error on the first natural frequency

This preliminary study anticipates a key message developed throughout the chapter: mixed kinematic descriptions and 1D–2D couplings can offer an effective balance between computational efficiency and fidelity of the dynamic response. In particular, the results show that, even with fewer DOFs, the mixed discretization can preserve the qualitative characteristics of the fundamental mode shape and provide frequency estimates that remain close to the chosen reference.

3.4 Cross-Model Free-Vibration Validation and Scaling Assessment

The preliminary free-vibration assessment presented in the previous section focused on the qualitative comparison between different CUF-based discretizations for a single reference configuration. While that analysis provided initial evidence of the consistency of mixed kinematic descriptions, a more systematic validation procedure is required in order to assess the robustness of the proposed modelling strategies across different geometrical configurations and formulation types.

In structural dynamics, modal quantities represent a particularly stringent benchmark for model verification. Natural frequencies are global indicators of the combined stiffness and mass distribution of the system, and therefore they are sensitive to both geometric scaling and kinematic richness of the adopted finite element formulation [8, 135]. In free-free conditions, where no boundary constraints artificially modify the spectrum, the comparison of elastic modes beyond the rigid-body subspace provides a direct measure of the intrinsic modelling consistency.

Within the framework of the Carrera Unified Formulation (CUF) and Node-Dependent Kinematics (NDK) [121, 122, 123, 124, 125], different structural representations can be constructed by combining 1D and 2D elements with variable expansion orders along selected directions. Such flexibility enables reduced-order models that aim at preserving the essential three-dimensional dynamic behaviour while limiting the number of degrees of freedom. However, this modelling freedom must be accompanied by a rigorous cross-comparison to ensure that no systematic stiffness bias or spurious modal distortion is introduced.

The present section introduces a structured cross-model validation strategy based on two complementary analyses:

- direct comparison of natural frequencies between different modelling variants for a fixed geometry;
- geometry-driven scaling assessment through a dimensionless modal indicator.

The first analysis addresses inter-model consistency, i.e., whether different discretization strategies (e.g., trave, piastra, solido formulations and their respective variants) produce comparable elastic spectra. The second analysis investigates whether the expected geometric scaling behaviour is preserved across the considered formulations.

All simulations are performed under free-free boundary conditions. Consequently, the first six modes correspond to rigid-body motions and are excluded from the discussion. The comparison therefore focuses on the elastic spectrum

starting from mode index $k = 7$, which represents the first deformable mode of the structure.

The following subsection presents the complete set of cross-model frequency comparisons for all the investigated geometries and modelling configurations. The graphical representation adopted allows a direct identification of systematic deviations, clustering behaviour, and mode-dependent discrepancies between the various formulations.

3.4.1 CUF–NDK Reduced-Order Elements versus Classical 3D Solids

The cross-model assessment presented in this section involves three different finite element representations of the same three-dimensional continuum:

1. 1D finite elements enriched by a bidimensional expansion over the cross-section;
2. 2D finite elements enriched by a monodimensional expansion along the thickness;
3. classical 3D continuum solid elements.

The objective of this subsection is not to reiterate the general CUF formulation, which has been extensively discussed in the previous chapters, but rather to clarify how these CUF–NDK elements are constructed in practice and how they differ structurally from a standard 3D discretization.

1D elements with bidimensional cross-sectional expansion

In the first modelling strategy, the structure is discretized along a single reference coordinate (denoted here as x), similarly to classical beam-type finite elements. However, instead of assuming a predefined kinematic hypothesis over the cross-section, the three-dimensional displacement field is reconstructed through a bidimensional expansion in the (y, z) plane.

The displacement approximation adopted for this element follows the general formulation introduced in Section 2.3.2, and is consistent with the reference expression reported in Eq. (2.3.1). In the present case, the expansion is performed over the cross-sectional coordinates, while the generalized unknowns are interpolated along the axial direction through standard 1D finite element shape functions.

From a constructive viewpoint:

- the mesh is one-dimensional (nodes distributed along the reference axis)

- the cross-sectional kinematics is not imposed *a priori* but generated hierarchically through the selected expansion order
- each node may carry a different kinematic order under the Node-Dependent Kinematics (NDK) paradigm

The resulting element have a 1D topological structure while being capable of reproducing three-dimensional effects such as cross-sectional warping, distortion and higher-order bending contributions.

2D elements with monodimensional through-thickness expansion

In the second modelling strategy, the structure is discretized over a two-dimensional reference surface (x, y) using standard 2D finite elements. The three-dimensional behaviour is then reconstructed through a monodimensional expansion along the thickness coordinate z .

The displacement field adopted for this element follows the general formulation introduced in Section 2.3.2, and is consistent with the reference expression reported in Eq. (2.3.3). In the present case, the expansion is performed along the thickness coordinate, while the generalized unknowns are interpolated over the in-plane domain through standard 2D finite element shape functions.

From a constructive perspective:

- the mesh is two-dimensional (surface discretization)
- the through-thickness kinematics is reconstructed hierarchically
- thickness-wise refinement can be locally increased via NDK without modifying the in-plane mesh

This strategy differs fundamentally from classical plate theories, since no fixed kinematic assumption (e.g. linear or quadratic variation) is embedded in the formulation. The thickness behaviour is instead controlled by the selected expansion order.

Classical 3D solid elements

In contrast, classical 3D continuum elements discretize the entire volume of the structure. The displacement field is directly interpolated in (x, y, z) using volumetric shape functions and the mesh must explicitly resolve all geometric dimensions.

Consequently:

- refinement requires volumetric mesh densification

- through-thickness and cross-sectional resolution are achieved only by increasing the number of 3D elements
- no hierarchical separation between topology and kinematic enrichment is available

While fully general, this approach typically involves a significantly larger number of degrees of freedom when thin or slender structures are analysed.

NDK and variable kinematic enrichment

A distinctive feature of the CUF–NDK framework is the possibility of assigning the expansion order on a node-wise basis. This enables non-uniform kinematic enrichment without modifying the underlying mesh topology. Higher-order descriptions can therefore be activated only in regions where three-dimensional effects are relevant, while lower-order kinematics are used elsewhere.

This separation between:

- geometrical discretization (1D or 2D mesh)
- hierarchical kinematic reconstruction

is the key structural difference with respect to classical 3D solid elements and underpins the reduced-order modelling strategy assessed in the following sections.

GENE vs NONE: advanced and adaptive formulation

For both 1D and 2D CUF-based families, two variants are considered: **GENE** and **NONE**. The distinction between the two does not concern the introduction of a shear-locking correction but is focused on the underlying theoretical framework adopted in the element formulation.

The **NONE** variant represents the standard CUF implementation, where the kinematic expansion is uniformly defined over the entire element and the same set of DoFs is assigned to each node. In this configuration, the enrichment level is controlled globally through the expansion order, leading to a hierarchical and structural uniform model.

On the other side, the **GENE** variant is based on the Node-Dependent Kinematics (NDK) approach described in Section 2.3.3. This formulation enhances the classical CUF framework by allowing the kinematic description to be defined at the nodal level. As a consequence, different nodes within the same element may be characterized by different kinematic discretization, enabling a local refined and full adaptive representation of the displacement field.

This innovation significantly increases the flexibility and generality of the formulation. The NDK-based **GENE** elements allow specific kinematic enrichment in regions where higher accuracy is required, without the need for global mesh refinement or uniform increase of the expansion order. Moreover, the modular structure of the formulation improves numerical robustness and creates a more consistent stiffness representation across different geometrical configurations and modal regimes.

Therefore, the **GENE** variant is expected to provide superior predictive capability, particularly in problems characterized by strong local gradients, high-frequency modes or geometrical slenderness variations, while preserving the unified and organised philosophy of the CUF framework.

Schematic comparison of modelling strategies

The conceptual differences between the three modelling approaches are illustrated in Figure 3.4.1, Figure 3.4.2 and Figure 3.4.3.

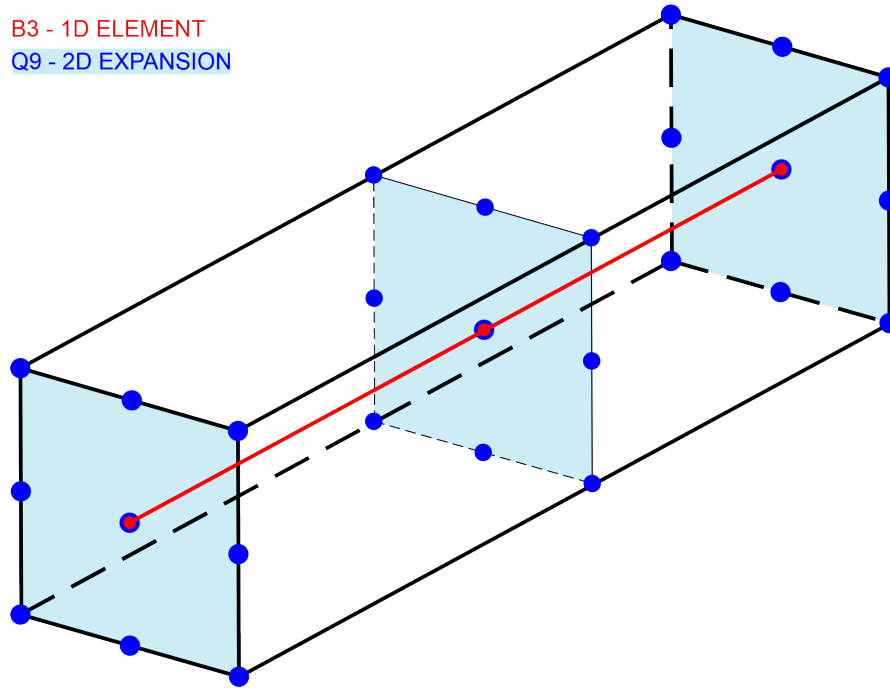


Figure 3.4.1: 1D finite element mesh with bidimensional cross-sectional expansion.

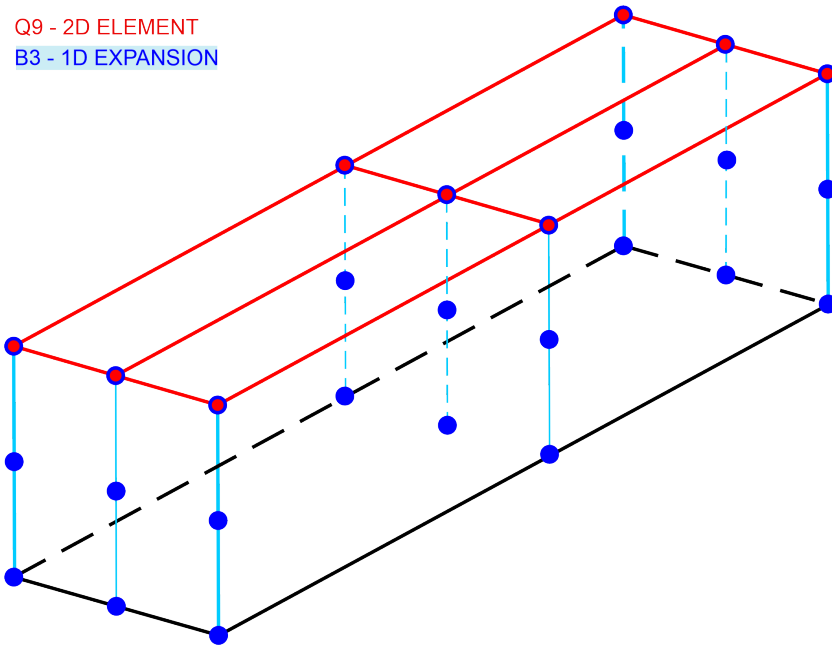


Figure 3.4.2: 2D finite element mesh with monodimensional through-thickness expansion.

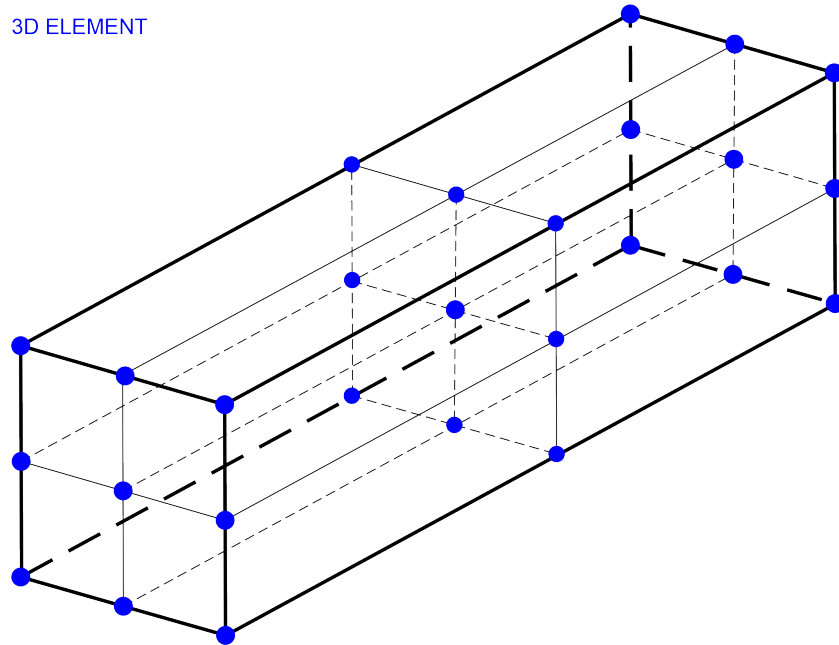


Figure 3.4.3: Classical 3D volumetric solid discretization.

The following subsections exploit these definitions to perform a systematic

cross-model frequency comparison and to assess geometry-driven scaling consistency.

3.4.2 Cross-Model Frequency Comparison

This subsection presents a systematic comparison of the natural frequencies obtained with the different modelling strategies introduced in this work. The objective is:

- to verify the consistency between reduced CUF-based models and full 3D formulations
- to identify possible mode-dependent discrepancies as a function of geometry and kinematic assumptions

For each of the seven geometrical configurations considered, five cross-model comparisons are performed. It is worth clarifying that the terminology “1D element” refers to a 1D finite element enriched by a bidimensional cross-sectional expansion, while “2D element” denotes a 2D finite element enriched by a monodimensional through-thickness expansion. In all cases, the CUF–NDK formulations reconstruct a fully three-dimensional displacement field. The distinction therefore lies exclusively in the topological dimension of the reference mesh (1D, 2D or 3D), not in the dimensionality of the underlying mechanical description, which remains three-dimensional for all analysed models.

- 1D_element_GENE vs 1D_element_NONE
- 2D_element_GENE vs 2D_element_NONE
- 2D_element_GENE vs 1D_element_GENE
- 2D_element_GENE vs 3D_element
- 1D_element_GENE vs 3D_element

In total, $7 \times 5 = 35$ frequency-comparison plots are analysed.

Plot nomenclature

Each frequency-comparison plot follows a compact naming convention of the form

$$\text{pGvtN_x_y_z,}$$

which encodes both the compared modelling families and the geometric configuration.

The adopted notation is defined as follows:

- the first letter identifies the reference model:
 - $\mathbf{t}$ = 1D element with 2D cross-sectional expansion
 - $\mathbf{p}$ = 2D element with 1D through-thickness expansion
 - $\mathbf{s}$ = classical 3D solid element
- the second letter specifies the kinematic variant of the reference model:
 - $\mathbf{G}$ = **GENE**
 - $\mathbf{N}$ = **NONE**
- $\mathbf{v}$ denotes “vs”, separating reference and compared models
- the triplet $\mathbf{x_y_z}$ represents the geometrical dimensions of the analysed configuration along the three spatial directions

For instance, a plot labelled $\mathbf{pGvtN_x_y_z}$ corresponds to the comparison between a 2D element with **GENE** correction (reference dataset) and a 1D element with **NONE** correction (compared dataset), for a geometry characterized by dimensions (x, y, z) .

Material properties and nondimensional formulation

All the analyses discussed in this subsection are performed under the assumption of linear elastic, homogeneous and isotropic material behaviour. In order to isolate purely geometric and kinematic effects, the material parameters are intentionally normalized as:

$$E = 1, \quad \nu = 0.25, \quad \rho = 1, \quad (3.4.1)$$

where E is the Young’s modulus, ν is the Poisson ratio and ρ is the mass density.

With this choice, the eigenvalue problem associated with free vibration,

$$\mathbf{K}\Phi = \omega^2 \mathbf{M}\Phi, \quad (3.4.2)$$

can be interpreted in a normalized (nondimensional) form [8], since both the stiffness matrix $\mathbf{K}$ and the mass matrix $\mathbf{M}$ are constructed using unit elastic modulus and unit density. Consequently, the computed circular frequencies ω (and the corresponding frequencies $f = \omega/2\pi$) represent normalized quantities, scaled with respect to the characteristic stiffness-to-mass ratio of the structure.

More generally, for an isotropic continuum, the natural frequencies scale as:

$$f \propto \sqrt{\frac{E}{\rho}}, \quad (3.4.3)$$

while the influence of ν affects the internal distribution of stiffness but does not introduce dimensional scaling. By setting $E = 1$ and $\rho = 1$, the factor $\sqrt{E/\rho}$ becomes unity, so that differences observed between models are exclusively attributable to:

- geometric parameters (in-plane dimensions, thickness, aspect ratio)
- kinematic enrichment (GENE vs NONE)
- modelling family (1D, 2D, 3D)

This normalization ensures that the comparison is not biased by material scaling and that the frequency discrepancies directly reflect differences in structural representation.

Graphical representation and interpretation

For each pair of models, the elastic natural frequencies are plotted in a log-log diagram starting from mode $k = 7$. As the structure is analysed under free-free boundary conditions, the first six modes correspond to rigid-body motions and are therefore associated with null eigenvalues. The comparison thus begins with $k = 7$, which represents the first deformable mode.

In the adopted representation:

- the x -axis reports the reference dataset (“Original”)
- the y -axis reports the compared dataset (“Avatar”)

The dashed diagonal line $y = x$ represents perfect agreement between the two models.

A point lying:

- exactly on the diagonal indicates identical prediction of that modal frequency
- above the diagonal indicates that the compared model predicts a stiffer behaviour (higher frequency)
- below the diagonal indicates a softer behaviour (lower frequency)

The log-log representation is intentionally adopted because:

1. the frequencies span several orders of magnitude across geometries and modes
2. relative deviations are visually uniform across low and high frequencies
3. multiplicative discrepancies (typical in stiffness scaling) appear as systematic vertical shifts

For each configuration, quantitative error indicators are computed, namely:

$$\text{MRE} = \frac{1}{N} \sum_k \left| \frac{f_k^{(A)} - f_k^{(B)}}{f_k^{(A)}} \right|, \quad (3.4.4)$$

$$\text{RMSE}_{\log_{10}} = \sqrt{\frac{1}{N} \sum_k \left(\log_{10} f_k^{(A)} - \log_{10} f_k^{(B)} \right)^2}, \quad (3.4.5)$$

which measure, respectively, the mean relative error and the logarithmic dispersion between spectra.

These metrics allow distinguishing between:

- uniform stiffness bias (global shift)
- mode-dependent dispersion (shape distortion of the spectrum)

It is also worth noting that, as the mode index increases, mode families may reorder across geometries (mode crossing), so that the same index k does not necessarily correspond to the same deformation mechanism for all configurations [8, 135].

3.4.2.1 1D_element: GENE vs NONE

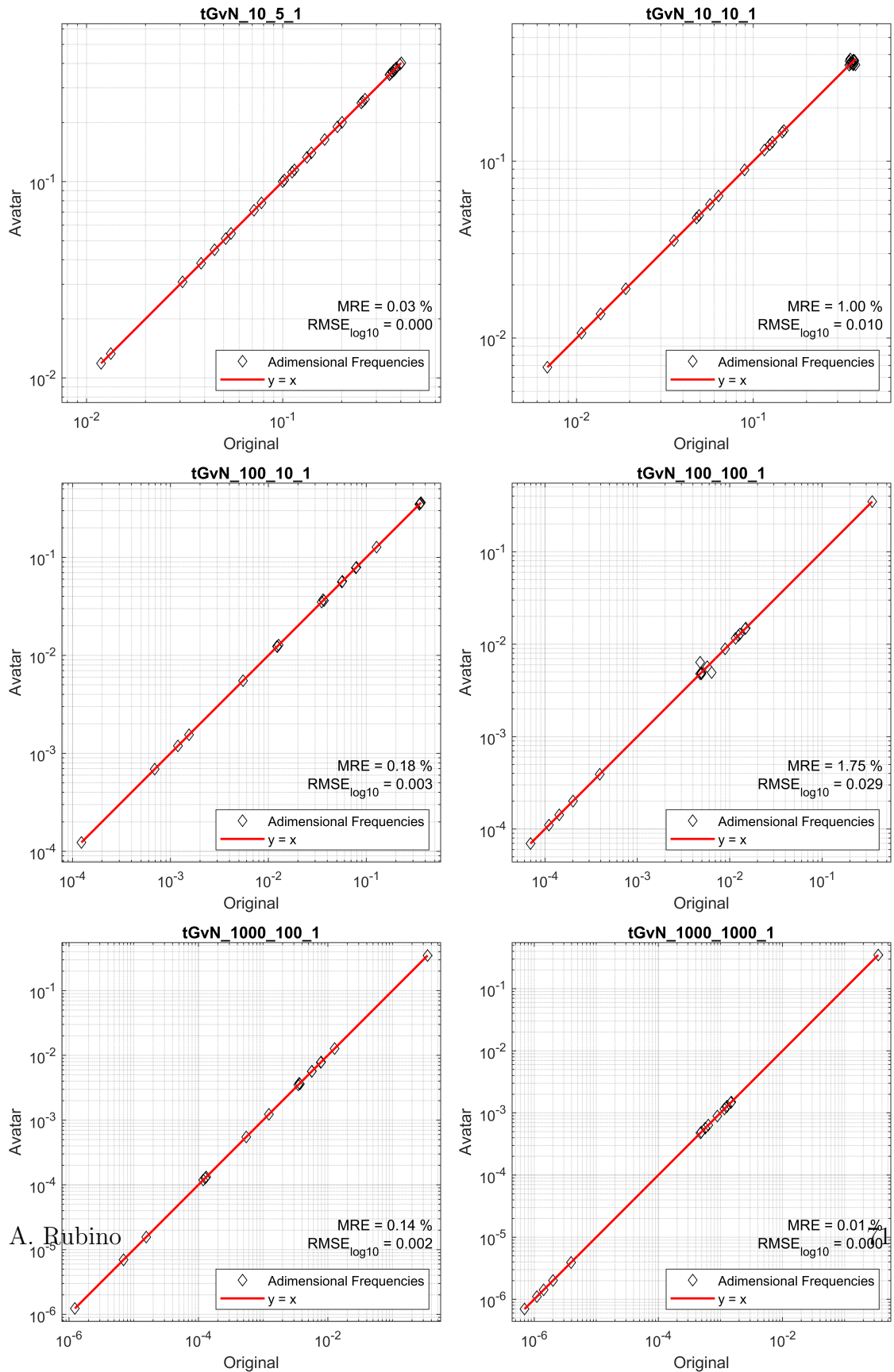


Figure 3.4.1: Frequency comparison: 1D_element_GENE vs 1D_element_NONE (cases 1–6).

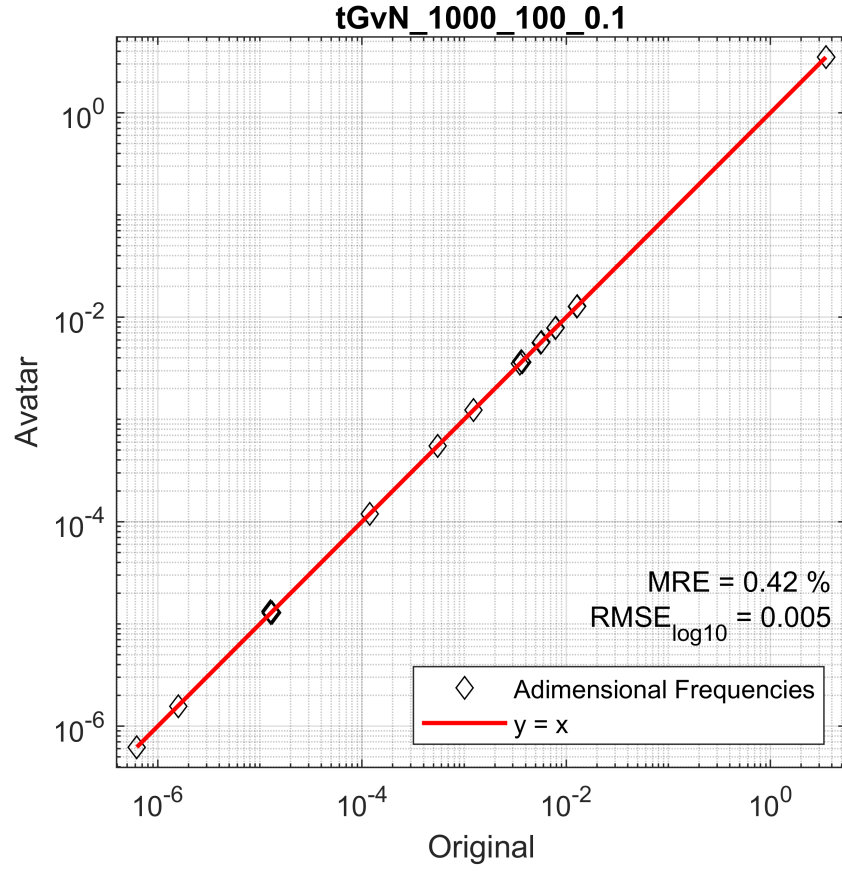


Figure 3.4.2: Frequency comparison: 1D_element_GENE vs 1D_element_NONE (case 7).

The comparison between 1D_element_GENE and 1D_element_NONE evaluates the influence of the adopted kinematic enrichment within the same 1D-based modelling family. The results show that:

- for lower modes, the points cluster close to the diagonal, indicating similar global stiffness representation
- discrepancies tend to increase with mode index, revealing higher sensitivity of short-wavelength modes to kinematic refinement

The deviations remain structured rather than random, indicating that the NONE formulation introduces a consistent approximation shift rather than numerical instability.

3.4.2.2 2D_element: GENE vs NONE

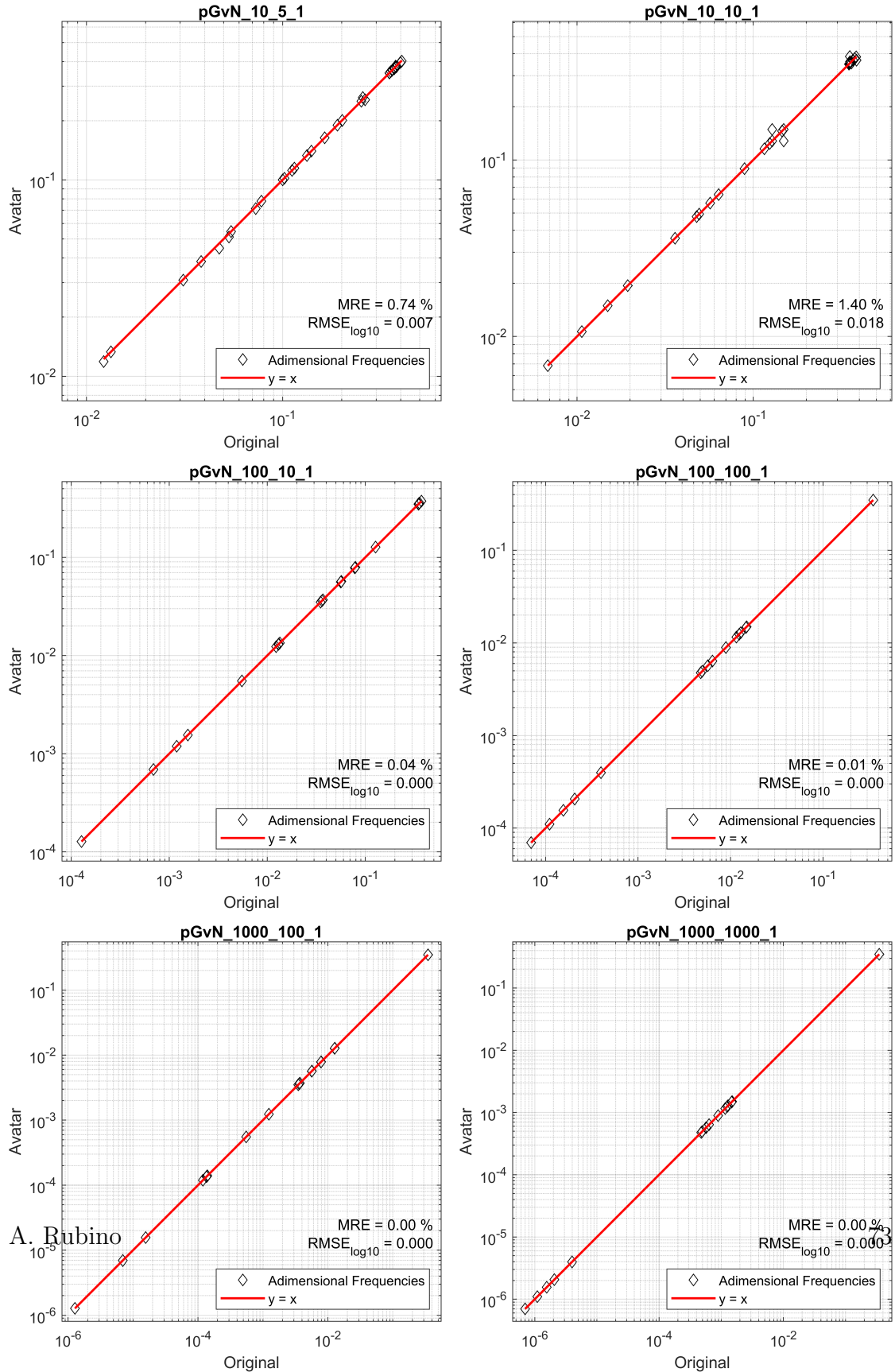


Figure 3.4.3: Frequency comparison: 2D_element_GENE vs 2D_element_NONE (cases 1–6).

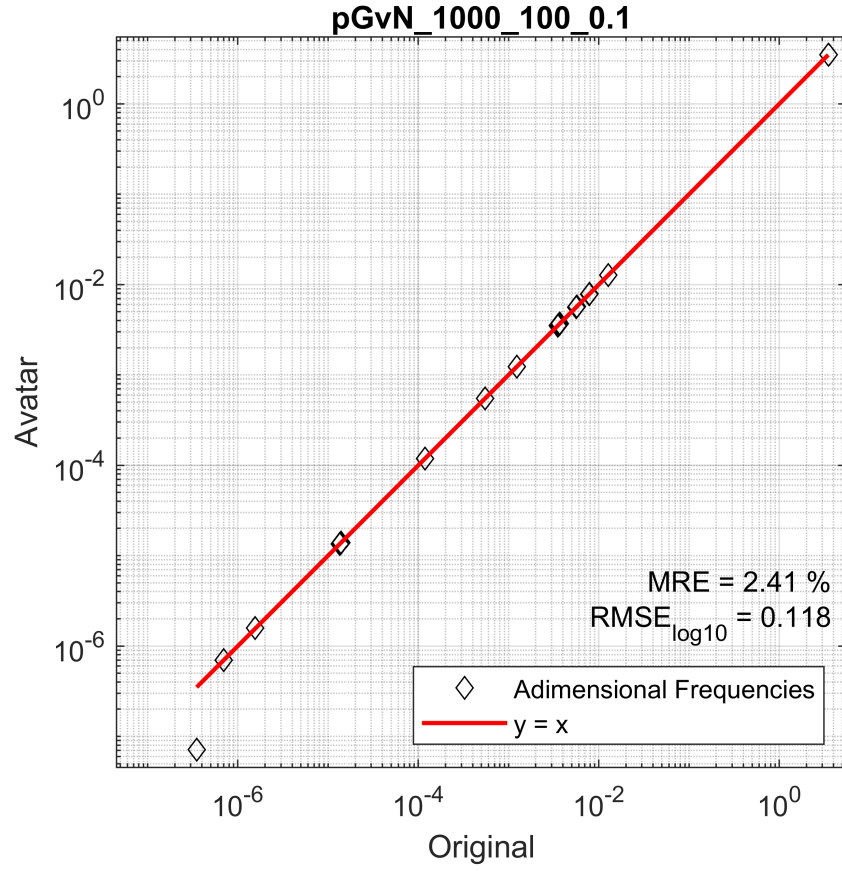


Figure 3.4.4: Frequency comparison: 2D_element_GENE vs 2D_element_NONE (case 7).

A similar trend is observed within the 2D-based formulations. However, the dispersion is typically lower compared to the 1D_element case, suggesting that in-plane enriched kinematics capture bending-dominated behaviour more robustly even at moderate orders.

The absence of systematic divergence at high frequencies indicates that the NONE configuration does not suffer from artificial stiffening.

3.4.2.3 2D_element: GENE vs 1D_element: GENE

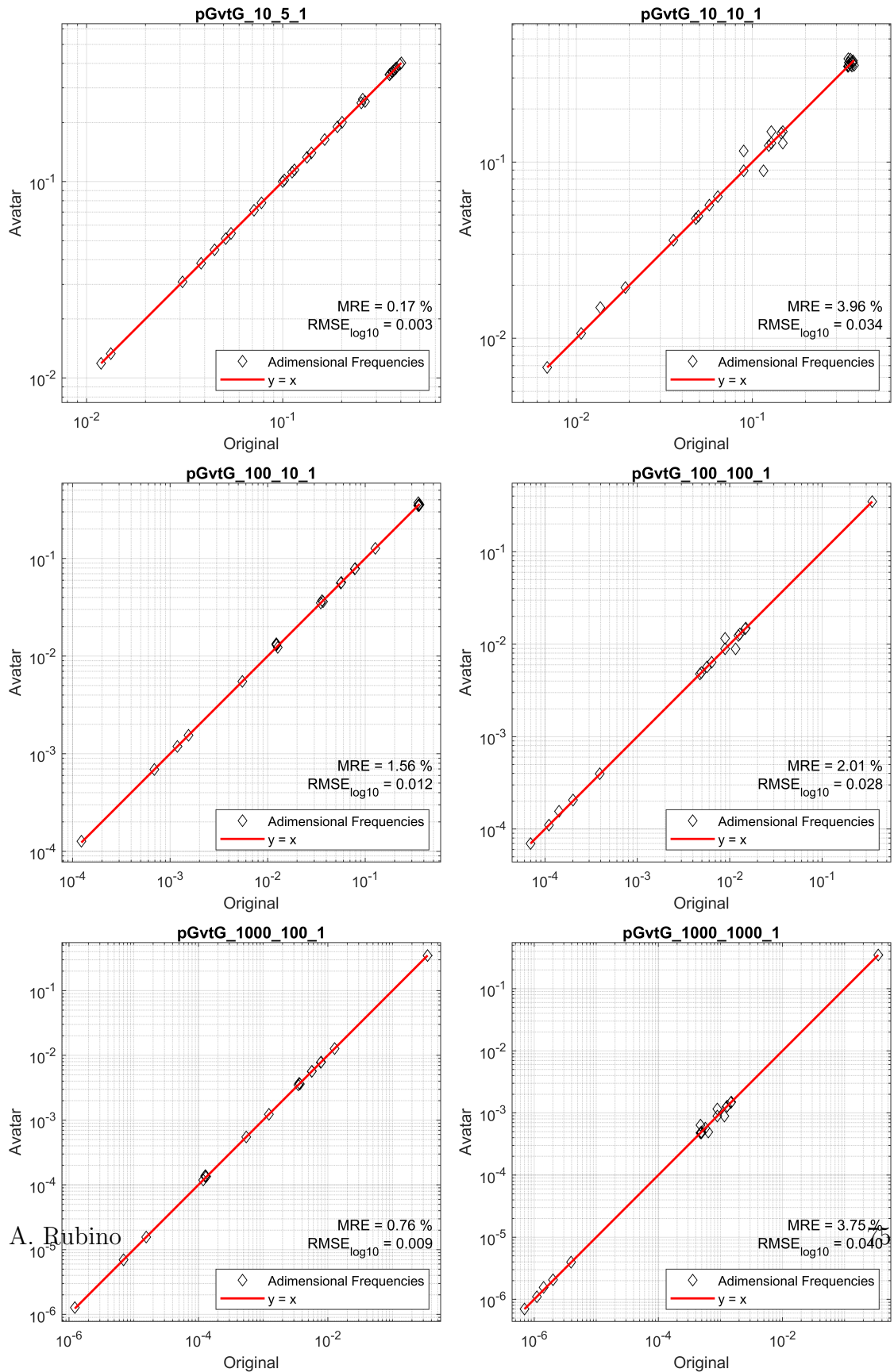


Figure 3.4.5: Frequency comparison: 2D_element.GENE vs 1D_element.GENE (cases 1–6).

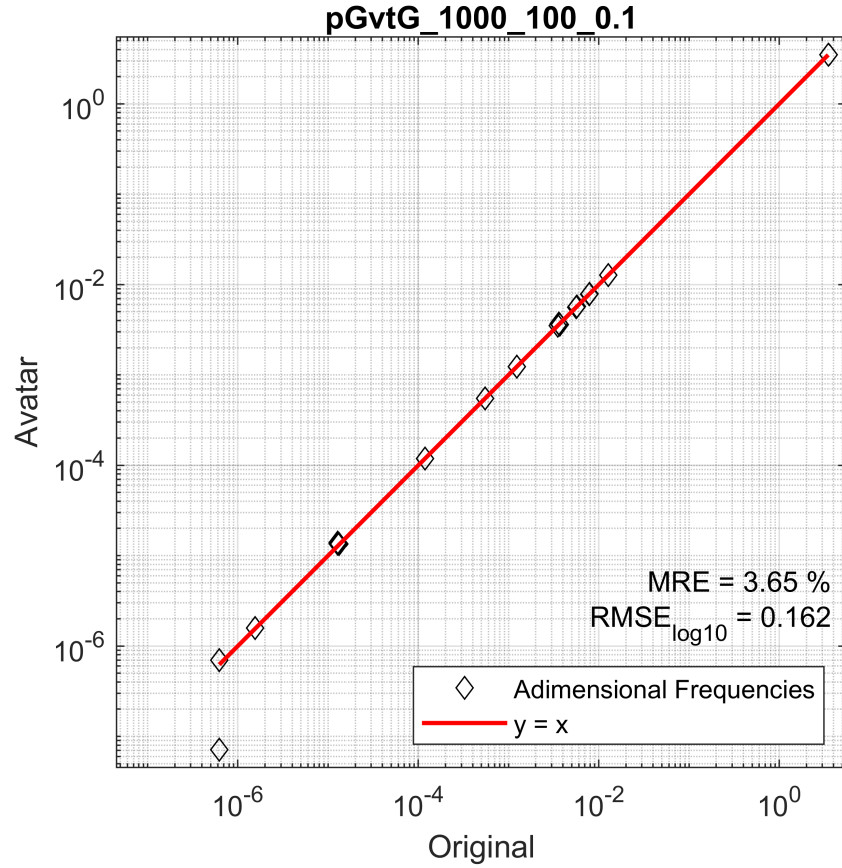


Figure 3.4.6: Frequency comparison: 2D_element_GENE vs 1D_element_GENE (case 7).

This comparison is structurally more significant, as it contrasts two different reduced modelling topologies (2D vs 1D meshes) within the CUF/NDK framework. The observed behaviour typically shows:

- strong agreement in lower modes, dominated by global bending
- increasing divergence for higher modes, where cross-sectional and through-thickness deformation patterns become more relevant

The absence of erratic scattering confirms that both models converge toward a consistent three-dimensional representation.

3.4.2.4 Against the 3D reference model: 2D_element_GENE and 1D_element_GENE

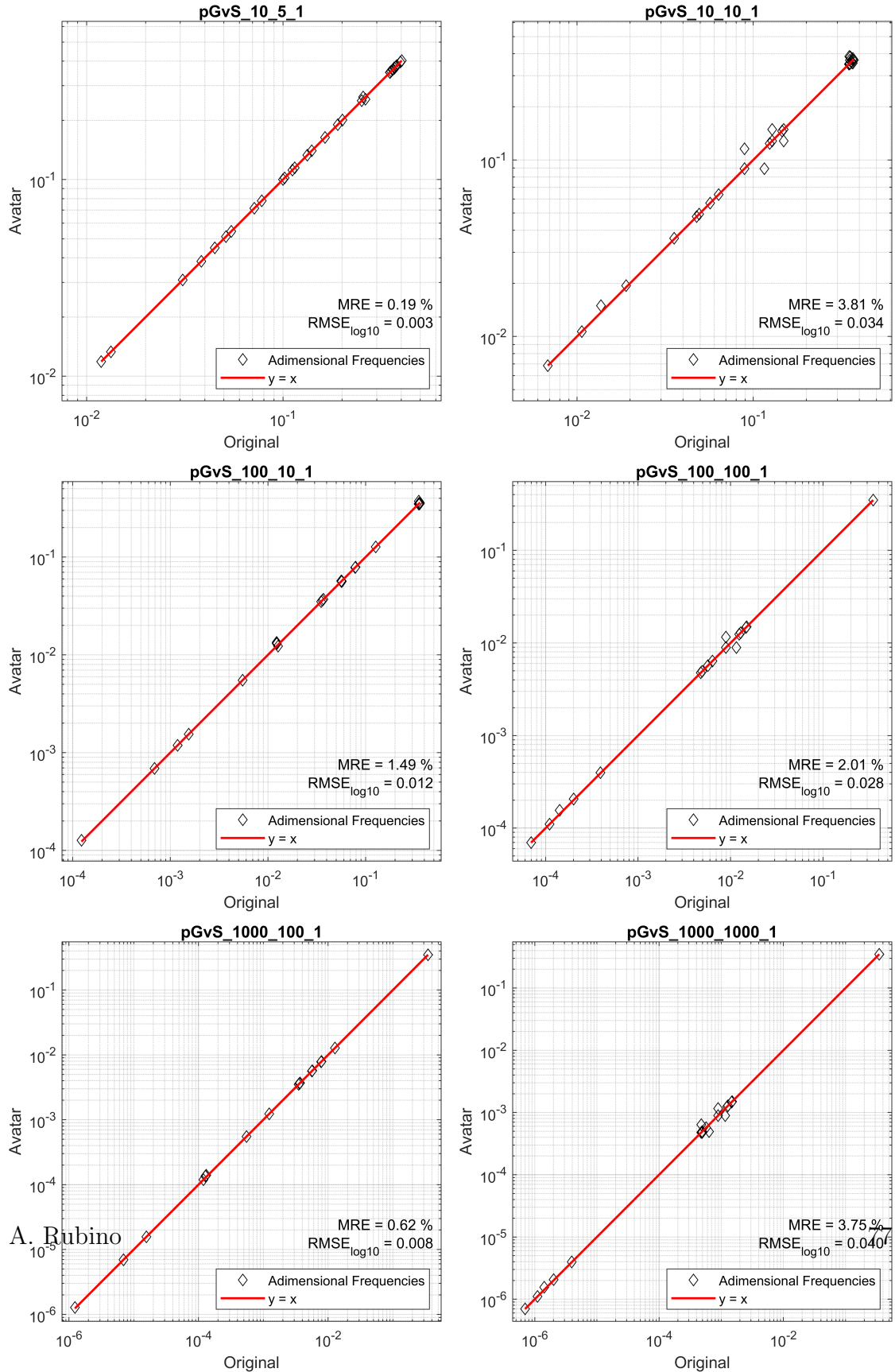


Figure 3.4.7: Frequency comparison: 2D_element_GENE vs 3D_element (cases 1–6).

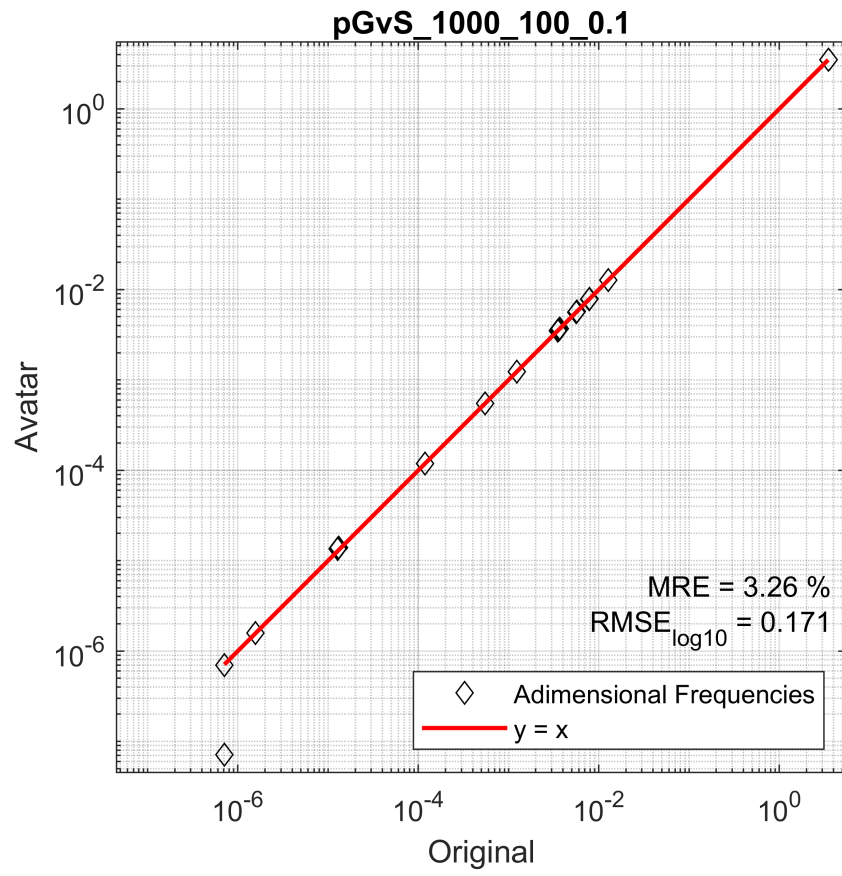


Figure 3.4.8: Frequency comparison: 2D_element_GENE vs 3D_element (case 7).

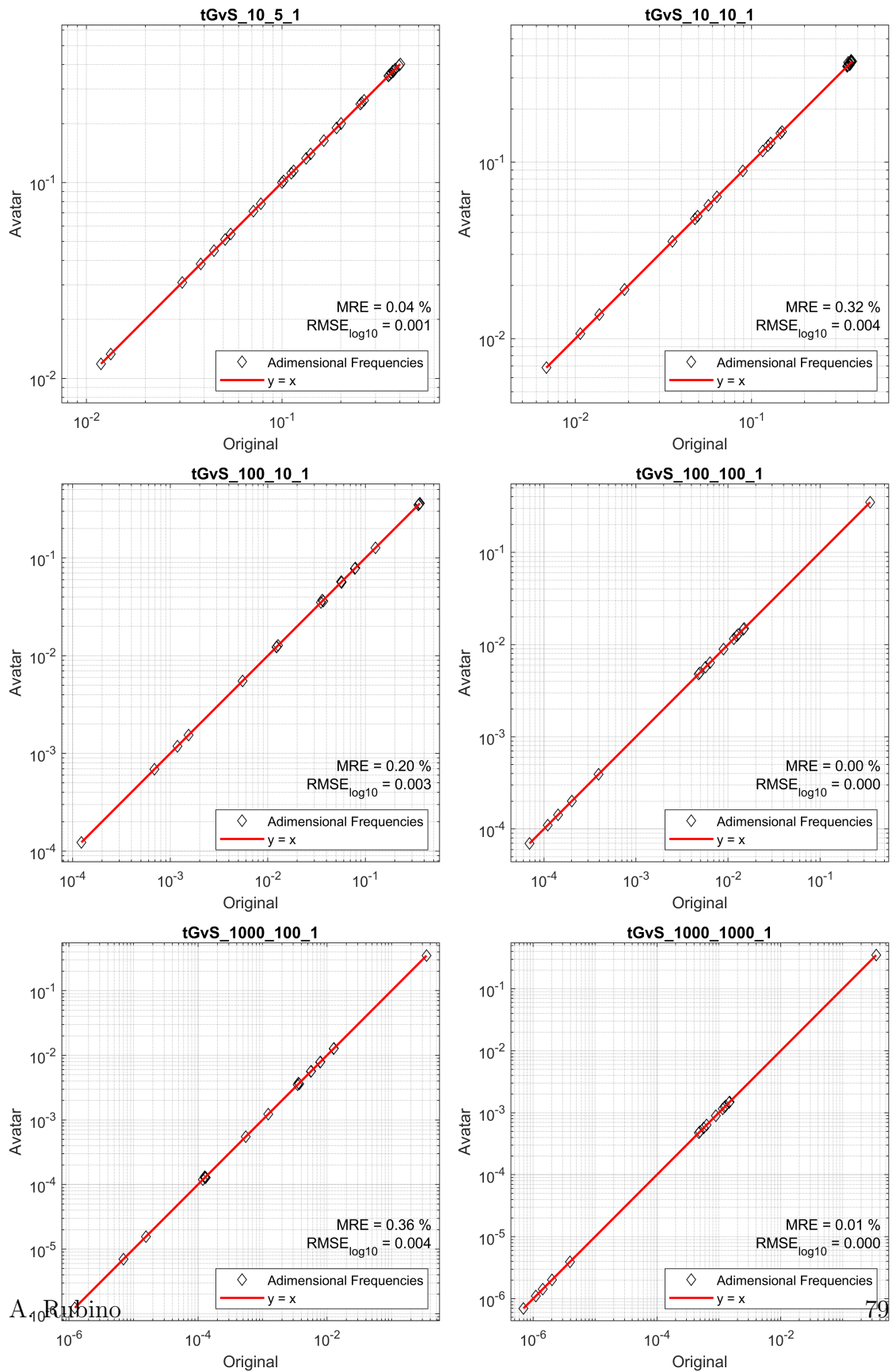


Figure 3.4.9: Frequency comparison: 1D_element_GENE vs 3D_element (cases 1–6).

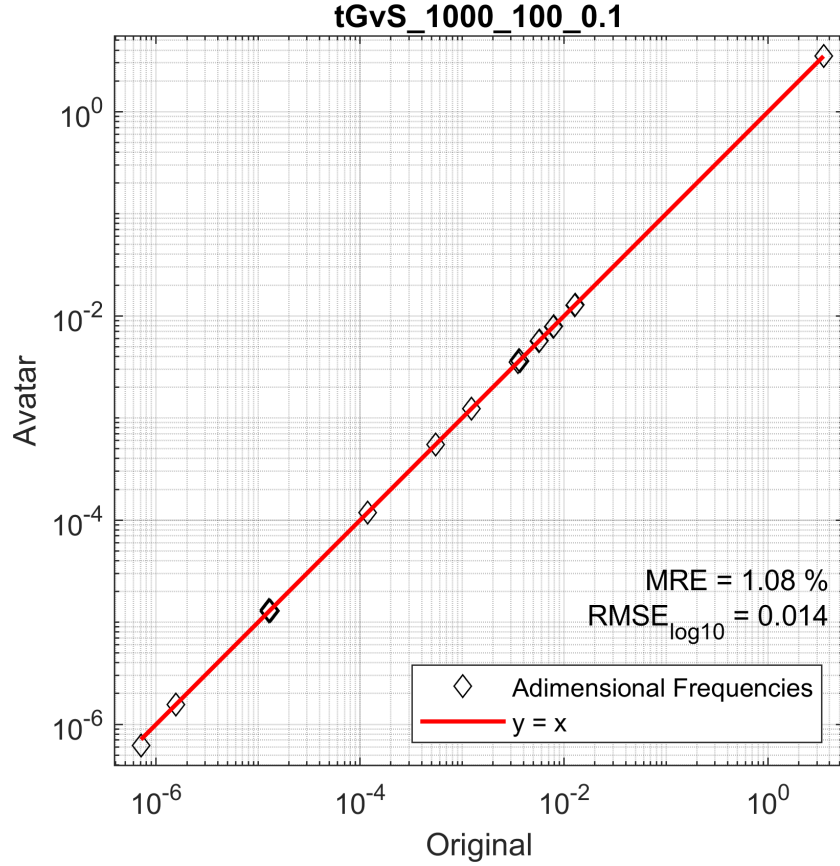


Figure 3.4.10: Frequency comparison: 1D_element_GENE vs 3D_element (case 7).

The comparison against the 3D_element model represents the most stringent validation benchmark. Across geometries, the following consistent features emerge:

- lower elastic modes exhibit near-perfect alignment along the diagonal, confirming that the reduced CUF models correctly reproduce the global stiffness and inertia distribution
- deviations increase for higher modes, as expected, due to the richer local deformation patterns captured by the full 3D discretization [8, 135]
- no evidence of erratic scattering is observed, suggesting that discrepancies are primarily attributable to systematic kinematic truncation rather than numerical instability

Global interpretation

From a structural-dynamics perspective, the results demonstrate that:

1. CUF-based reduced models preserve the global spectral structure across all tested geometries
2. differences between modelling families become more pronounced at higher modal indices, where localized deformation dominates
3. the 3D_element reference does not reveal hidden stiffness artefacts in the reduced models

Therefore, the cross-model comparison confirms that the proposed CUF-based strategies achieve spectral consistency across geometrical scales while maintaining computational efficiency.

3.4.3 Scaling Indicator S_k and Cross-Model Regime Transition

A direct cross-model comparison of natural frequencies provides an essential verification step but it does not fully assess whether the different formulations preserve the same geometric scaling structure across families of geometries. To address this point, a dimensionless scaling indicator is introduced and analysed comparatively for 1D_element_GENE, 2D_element_GENE, and 3D_element.

Under the material normalization adopted in this study ($E = 1$, $\rho = 1$, $\nu = 0.25$), the modal spectrum is governed primarily by geometry. For dominating bending plate-like behaviour, classical theory predicts a characteristic dependence on in-plane length and thickness and it is therefore meaningful to define an indicator of the form

$$S_k = f_k \frac{L^2}{z}, \quad (3.4.1)$$

where f_k is the normalized natural frequency, L is the dominant in-plane dimension and z is the thickness. In an ideal bending-dominated regime, S_k is expected to remain approximately stable across geometries sharing comparable deformation mechanisms, while systematic departures indicate either a change of dominant deformation regime (bending–membrane and/or shear contributions) or model-dependent stiffness bias [137, 138, 139].

The following results are presented by plot type, so that each figure provides an immediate comparison among 1D_element_GENE, 2D_element_GENE and 3D_element for the same metric. This organization is essential to separate physical trends (shared by all models) from formulation-specific effects.

The CoV plots in Fig. 3.4.1 provide a compact measure of the dispersion of the scaling indicator across the seven geometries at each mode index. Here, CoV denotes the coefficient of variation, defined as

$$\text{CoV}(S_k) = \frac{\sigma_{S_k}}{\mu_{S_k}}, \quad (3.4.2)$$

where σ_{S_k} and μ_{S_k} represent respectively the standard deviation and the mean value of S_k computed across the considered geometries at fixed mode index k . This dimensionless metric quantifies the relative spread of the indicator and enables a direct comparison of variability across modes.

A marked increase in dispersion is observed around $k \approx 28$ for all three modelling families. The fact that the same transition is visible in 1D_element-, 2D_element- and 3D_element-based formulations strongly suggests that this behaviour is primarily geometry-driven rather than induced by kinematic reduction.

From a mechanics standpoint such a transition is consistent with the progressive activation of in-plane (membrane) and/or transverse shear contributions

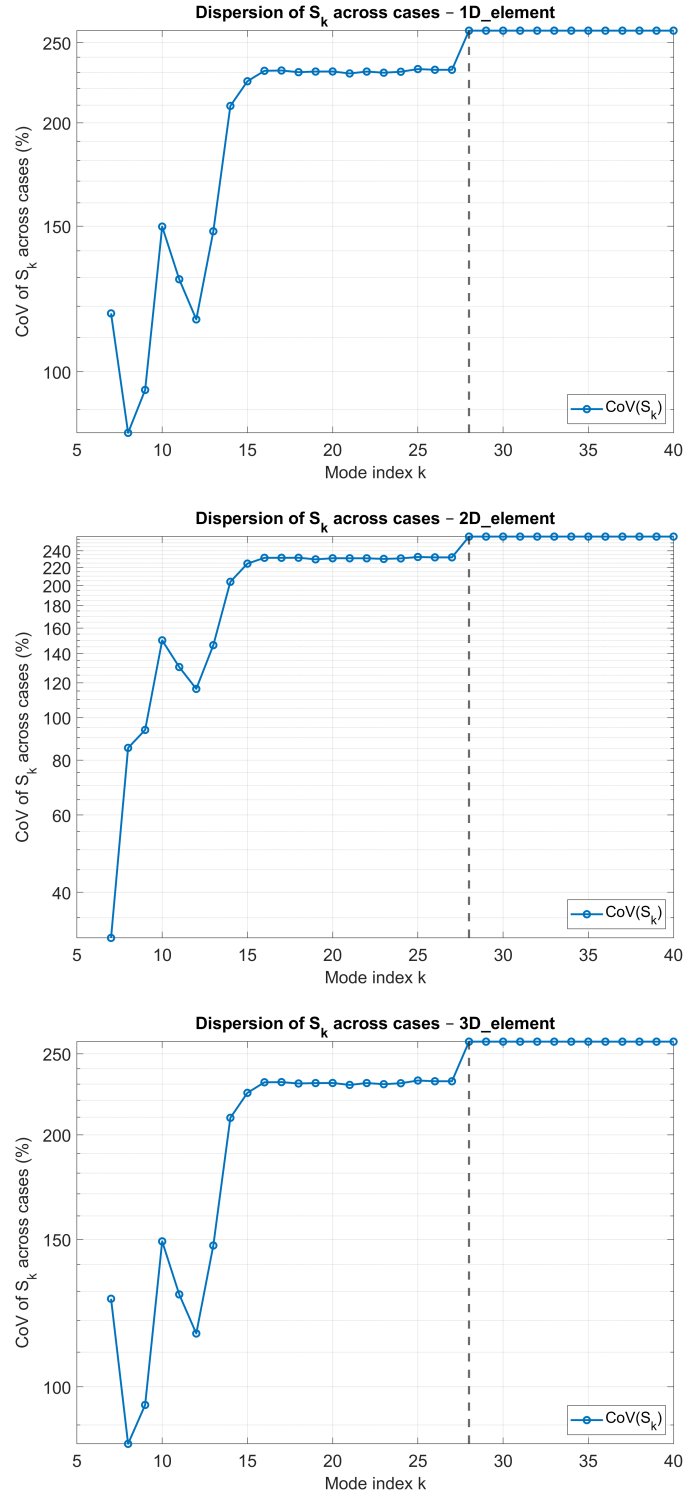


Figure 3.4.1: Coefficient of variation (CoV) of S_k across geometries for 1D_element_GENE, 2D_element_GENE and 3D_element

at increasing modal index and with the increased sensitivity of higher modes to thickness and aspect ratio [137, 139, 138]. The following plots are used to confirm that the same transition is observable in the spatial distribution of S_k and in its dependence on aspect ratio.

The heatmaps in Fig. 3.4.2 explain where the dispersion measured in Fig. 3.4.1 originates. For low and intermediate modes, the distributions remain relatively homogeneous across geometries, whereas beyond the transition region (around $k \approx 28$) the contrast between cases increases sharply.

Importantly, this qualitative change is consistent across the three modelling families: the 3D_element model exhibits smoother gradients (as expected from full volumetric resolution), yet the same transition is still visible. This further supports the interpretation that the jump in CoV is not a numerical artefact specific to CUF reductions but a physical change in the modal sensitivity to geometry.

The median trends in Fig. 3.4.3 provide a direct confirmation of the regime transition indicated by Fig. 3.4.1. In all models, square-like cases ($AR=1$) remain comparatively stable over a wider range of modes, while rectangular/slender configurations ($AR>1$) exhibit a pronounced growth of S_k after the transition region.

This behaviour is consistent with the physical interpretation that higher modes increasingly involve deformation patterns characterised by shorter wavelengths and stronger sensitivity to directional stiffness imbalance [140, 138]. The agreement among 1D_element, 2D_element and 3D_element medians indicates that the CUF-based reductions capture the same aspect-ratio amplification observed in the full three-dimensional discretization.

The scatter plots in Fig. 3.4.4 explicitly show how the aspect-ratio dependence evolves with mode index. Lower elastic modes exhibit limited dependence on AR , whereas higher modes display a strong amplification with increasing slenderness. The appearance of this behaviour is consistent with the transition identified at $k \approx 28$ in Fig. 3.4.1 and with the post-transition contrasts visible in the heatmaps (Fig. 3.4.2).

Overall, the key outcome is not that the three modelling families produce identical values of S_k at all modes but that they reproduce the same structural signature: a coherent low-mode scaling regime and a mode-dependent transition toward stronger aspect-ratio sensitivity. Since the same regime transition is observed in the 3D_element model, the CUF-based reduced formulations can be considered physically consistent and not affected by spurious stiffness artefacts. This provides a consistent geometry validation complementing the direct frequency comparisons discussed in the previous subsection.

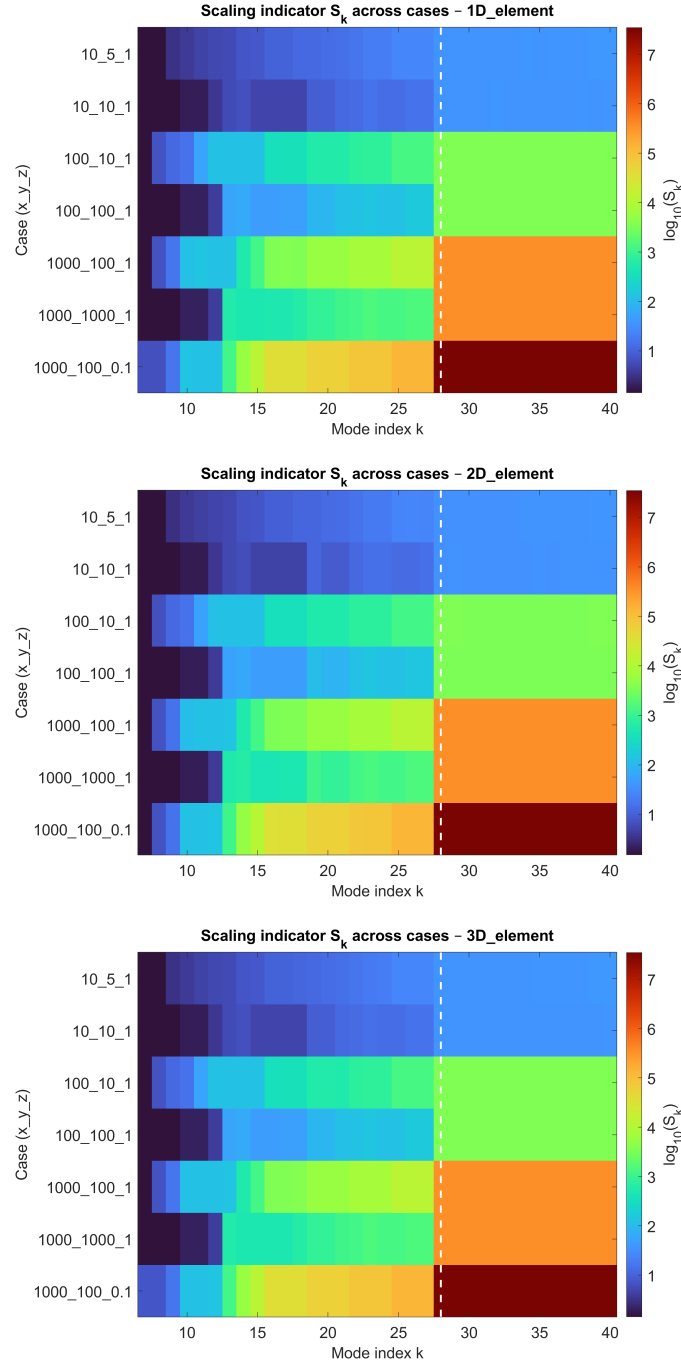


Figure 3.4.2: Heatmaps of S_k across geometries (rows) and mode index k (columns) for 1D_element_GENE, 2D_element_GENE and 3D_element

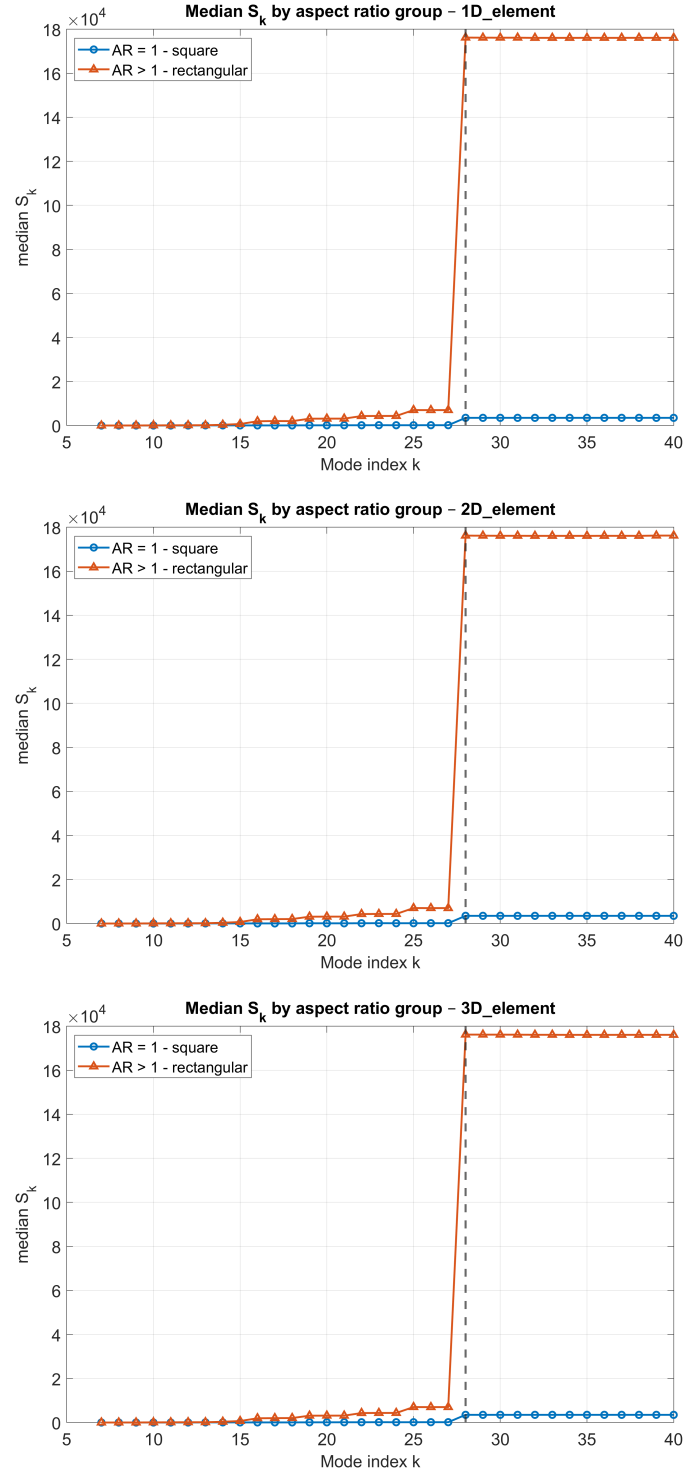


Figure 3.4.3: Median S_k split by aspect ratio group (AR=1 vs AR>1) for 1D_element_GENE, 2D_element_GENE and 3D_element

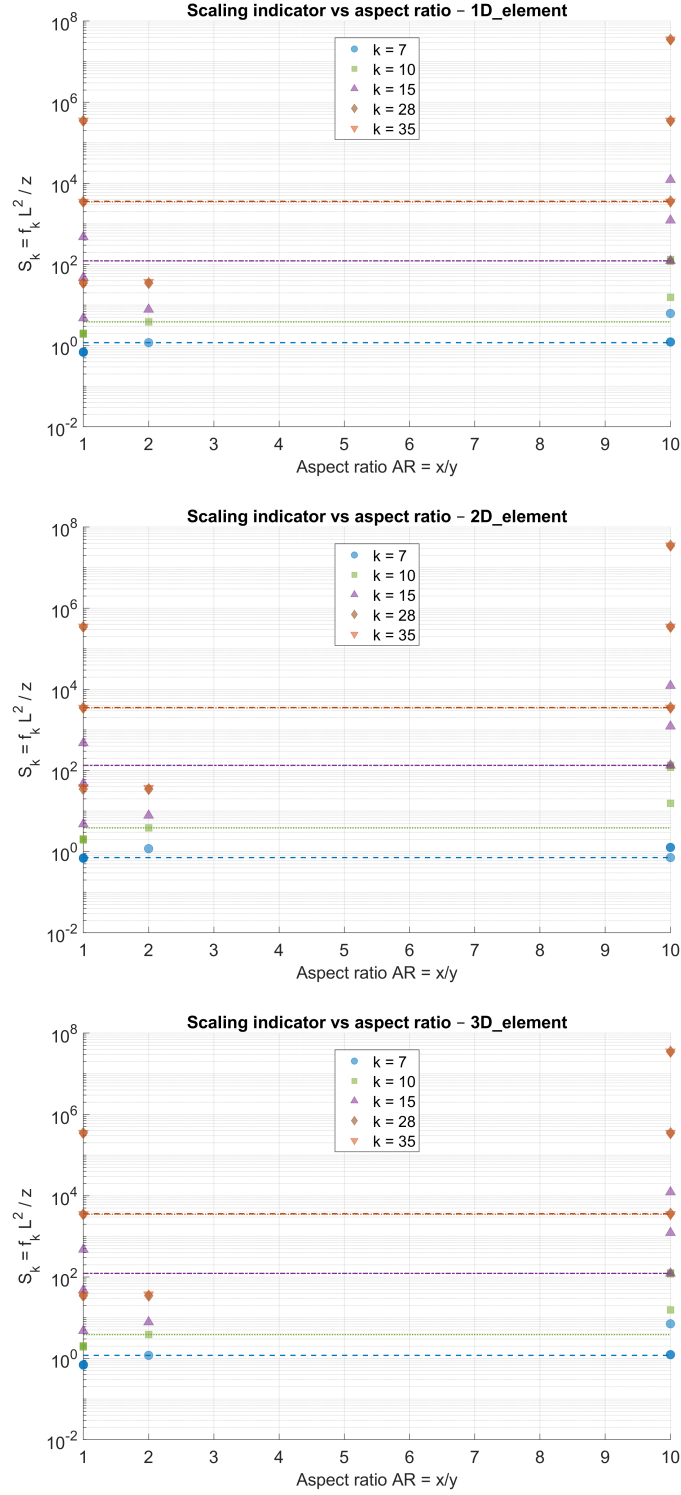


Figure 3.4.4: S_k versus aspect ratio for selected mode indices, shown for 1D_element_GENE, 2D_element_GENE and 3D_element

3.5 Static Analyses

Static analyses represent a primary benchmark for assessing the performance of adaptive CUF-based finite elements. They allow evaluating the capability of the proposed models to capture localized effects induced by concentrated loads, boundary constraints, and mesh distortion, while maintaining numerical stability and accuracy. In aerospace structural applications, static verification is typically the first step in design loops, and it directly supports sizing, margin assessment, and preliminary trade studies.

The static results discussed in this section refer to:

- benchmark plates subjected to concentrated forces (square plate and Razzaque's skew plate);
- reduced and adaptive mesh strategies, based on coupling of plate and beam elements within CUF/NDK;
- distorted mesh investigations, including controlled and random distortions, to quantify robustness.

3.5.1 Benchmark Plates under Concentrated Loads

In engineering applications, the Finite Element Method (FEM) has become very popular due to the prediction power that has on simulations of a variety of physical problems, such as structural analysis, fluid dynamics simulations and so on. This method is able to give accurate results by constructing a proper mesh, which means the more enriched is the mesh, the more precise is the computation. In this way, in order to be sure to capture all the properties of interest from the simulation, it is necessary to have a very dense mesh in some region of the domain analyzed and due to the complexity of the studied physics, the number of degrees of freedom can reach the order of magnitude of millions. The main drawback is the high computational cost and a common practice to overcome this problem is to focus the presence of high order elements in the regions of interest and to relax the elements in number and accuracy in the others regions of the domain, namely the Adaptive Mesh (AM).

The aim of this chapter is to demonstrate that the 1D and 2D finite elements modelled under the assumption of the Carrera Unified Formulation (CUF) are able to offer a great alternative to the usual and advanced techniques used for the Adaptive Mesh. As a matter of fact, the Carrera Unified Formulation, applied both to 1D and 2D finite elements, provide very accurate 3D-like solutions with much lower computational cost than the classical 3D elements, becoming an important tool for the analysis of structural problems. Recently, in the framework of CUF, a

novel approach, called Node-Dependent Kinematics (NDK), has been proposed to further increase the numerical efficiency of the models [121, 122, 123, 124, 125]. By relating the kinematic assumptions to the chosen nodes, FE models with variable nodal kinematics can be built conveniently. Naturally, the critical zone with higher-order assumptions can be bridged to the less-critical area modeled with adequate lower-order theories [126], in such a way the numerical modeling procedure can further be simplified.

The Node-Dependent Kinematics approach open the possibility to do a proper mesh refinement when it is necessary to save degrees of freedom for computational cost reasons, but not neglecting accuracy in results [64]. Indeed, using this approach under CUF assumptions, it is quite simple to construct a proper mesh considering elements with less nodes in regions far from loads and gradually enrich the mesh using not only homogeneous and symmetric elements, but also the non-conventional finite elements with a number of nodes different along the thickness or in-plane, by combining 1D and 2D elements. In particular, it is possible to construct elements with peculiar shapes that can adapt to the edges of the domain considered or to adjacent portions which share one or more sides, when the geometrical constraints or the kinematic behaviour require that.

3.5.1.1 Numerical Results: Reference static solutions

In this section, two study-cases are presented: the first one is a square plate with a concentrated force at the centre of the top surface and the second one is a Razzaque's skew plate with concentrated forces at the mid-line of the top surface. In each study case, linear static analyses are performed with an academic code MUL2 and compared with classical 3D finite elements analysis using the commercial software Patran/Nastran of MSC Software Corporation. Different boundary conditions are applied to the structures and the material is isotropic in both cases, with different properties.

3.5.1.2 Validation

First of all, the present elements are validated through regular plate geometries. The results are compared with the solution obtained with classical 3D elements using the commercial software and with regular and homogeneous finite elements using full mesh, for which a convergence study has been performed to ensure the highest accuracy for these and following analyses.

3.5.1.2.1 Square plate with concentrated force

The in-plane dimensions of the square plate (SQ) considered are $1 \times 1 \text{ m}^2$, while the thickness is 0.1 m. The boundary conditions applied are all fixed edges and

a concentrated force at the centre of the top surface equal to $q=1$ N, as shown in Figure 3.5.1. The material considered is isotropic with the following properties: $E=1$ Pa and $\nu=0.25$. By using a mesh with $30 \times 30 \times 3$ H8 hexagonal elements with 8 nodes for element in Nastran, the convergence 3D solution is reached, computed with Nastran. The plate is then meshed with the present plate finite elements used in the MUL2 code: 10×10 Q16 elements are used in the plane with 1 $B4$ discretization along the thickness; in this case, the convergent solution is reached with 11532 degrees of freedom (DOFs).

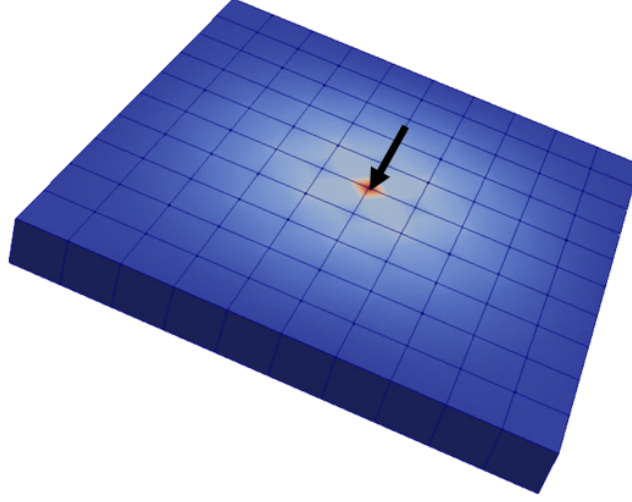
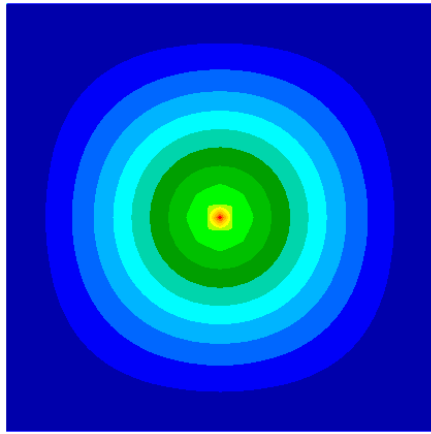


Figure 3.5.1: Square plate with concentrated force

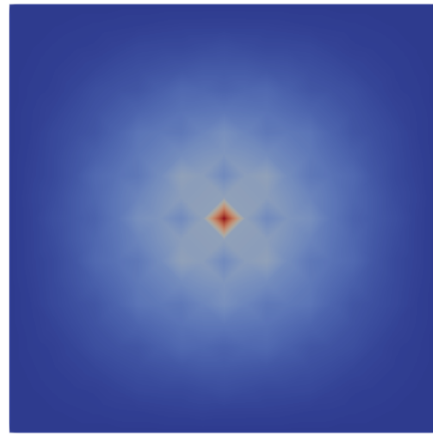
Due to the symmetry of the problem, with symmetric geometry and symmetric boundary conditions, the quarter-plate (QP) finite element model is used to simplify the analysis: the full plate is divided into four equal sub-plates and the study is conducted only on one of them. In particular, the bottom-left quarter is chosen and the following boundary conditions are applied, considering as u , v and w the displacements along x , y and z respectively : bottom and left edges are fixed, on the top edge $v=0$, on the right edge $u=0$ and on the top-right corner a force of $q=0.25$ N is applied. Then, the resulting quarter plate mesh is 5×5 Q16 elements used in the plane and 1 $B4$ discretization along the thickness. The displacement field computed with Nastran is shown in Figure 3.5.2 and the comparison between displacements and stresses with the developed MUL2 code is reported in Figure 3.5.3. In order to evaluate the accuracy of the analysis, the results obtained are compared in terms of percentage error, compute as follow:

$$err\% = 100 \times \left| \frac{u_{sol,ref} - u_{sol}}{u_{sol,ref}} \right| \quad (3.5.1)$$

where $u_{sol,ref}$ is the reference solution, in this case computed with Nastran, and u_{sol} is the solution computed with the MUL2 code. The results are comparable and the different peak at the centre, where the force is applied, is due to a different approximation of the infinite between the two codes: in fact, as illustrated in Figure 3.5.4, the computed percentage error is below 1% if the peak value is not accounted for, in great agreement with the common practice which consider acceptable this type of displacements error in structural analysis.



(a) Displacement field from Nastran



(b) Displacement field from MUL2

Figure 3.5.2: Square plate

Chapter 3 – Numerical Results and Discussion

Comparison: Patran vs MUL2

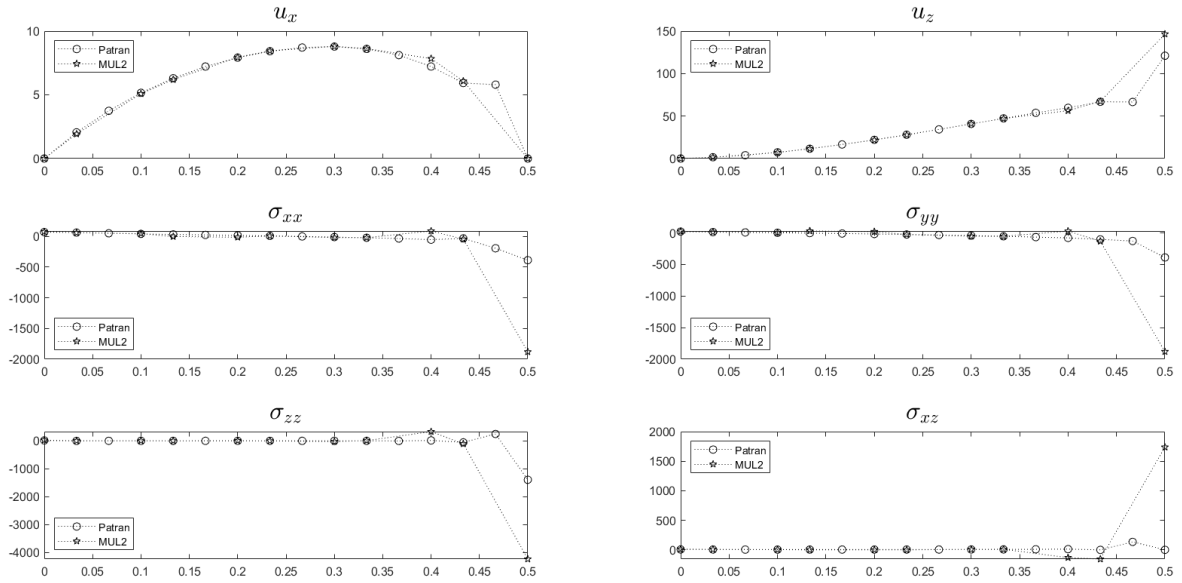


Figure 3.5.3: Square plate - Comparison between Nastran and MUL2 code with Q16 plate elements and cubic expansion along thickness

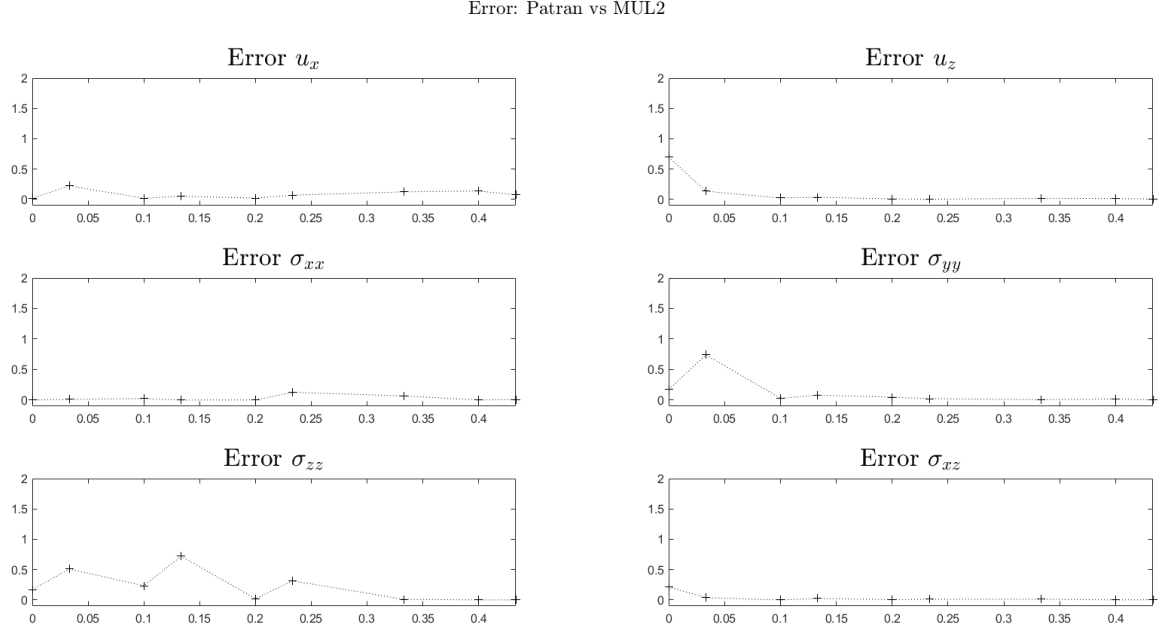


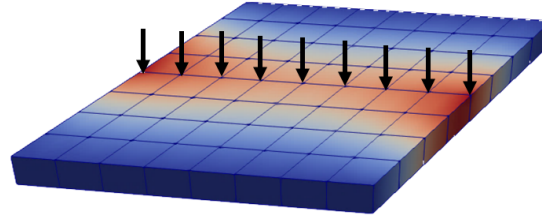
Figure 3.5.4: Square plate - Percentage error between Nastran and MUL2 code

Due to the strong localization induced by the concentrated force, this benchmark is representative of local load introduction and the resulting gradients typically found in aerospace structures near fittings and attachments. The validation highlights that CUF-based plate elements capture the global response with high accuracy and that discrepancies are confined near the singular zone. This behaviour is consistent with standard FEM practice, where the singularity cannot be represented exactly and where mesh refinement strategies are commonly employed near the load application region.

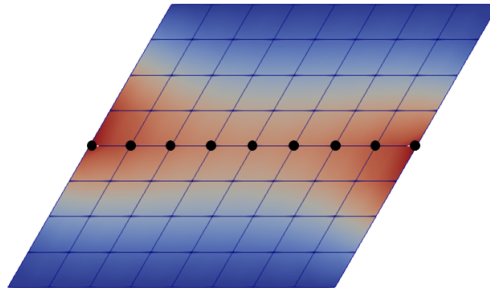
3.5.1.2.2 Razzaque's skew plate

In order to study distorted elements, starting from the work in [68], the Razzaque's skew plate (RSP) is considered, which is a square plate with two opposite edges skewed by an angle with respect to the other two edges, in such a way the resulting geometry is non-orthogonal. The in-plane dimensions of the plate considered are $100 \times 100 \text{ m}^2$, the thickness is 10 m and the angle is $\theta=60^\circ$. The boundary conditions applied are top and bottom fixed edges while the remaining edges are free and a set of concentrated force at the mid-line of the top surface, all equals to $q=1 \text{ N}$, as shown in Figure 3.5.5. The material considered is isotropic with the following properties: $E=1.092 \text{ MPa}$ and $\nu=0.30$. By using a mesh with $20 \times 20 \times 2$ T10 tetrahedral elements with 10 nodes for element in Nastran, the

convergence 3D solution is reached, computed with Nastran. The skew plate is then meshed with the present plate finite elements used in the MUL2 code: 8×8 Q16 elements are used in the plane with 1 $B4$ discretization along the thickness; in this case, the convergent solution is reached with 7500 degrees of freedom (DOFs). The displacement field computed from Nastran is shown in Figure 3.5.6.



(a) 3D view



(b) Top view

Figure 3.5.5: Razzaque's skew plate

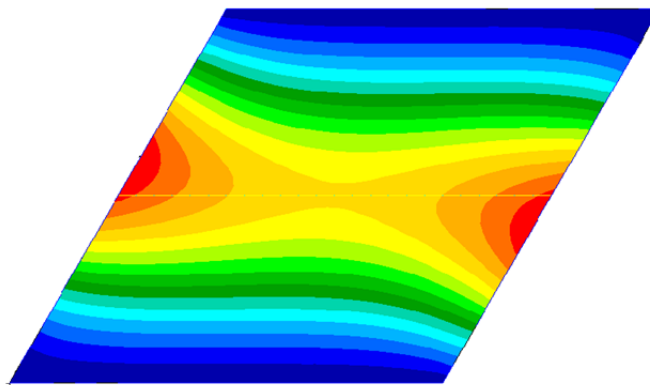


Figure 3.5.6: Razzaque's skew plate - Displacement field from Nastran

This benchmark is relevant for aerospace panels and components where skewness and non-orthogonal partitions arise due to sweep, taper, or interface constraints. In such contexts, distorted meshes are often unavoidable, and the reference solution provides the baseline needed to evaluate reduced and adaptive meshes in a controlled manner.

3.5.2 Reduced and Adaptive Mesh Strategies

After the validation of the mesh using the proposed plate finite elements, a reduced mesh is sought in order to decrease the DOFs and the computational cost requested for the analysis. A convenient reduced model is reached by combining plate and beam finite elements using the CUF, for example using beam finite element B3 with different number of nodes in the cross-section, as shown in Figure 3.5.1.

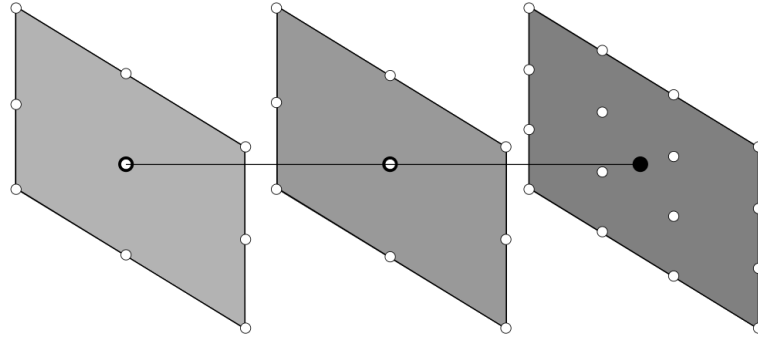


Figure 3.5.1: B3 beam element (black nodes along beam axis) connecting Q9 and Q16 plate elements (white nodes on beam cross-section)

3.5.2.1 Square-plate model with concentrated force

Starting from the quarter-plate model of the square plate with a concentrated force, having an homogeneous plate finite element mesh Q16 in-plane and B4 along the thickness, a suitable reduced mesh is reached by connecting plate finite elements with different in-plane kinematics with beam finite elements having different cross-sectional kinematics at tips. In particular, the area near the applied force is not changed, and the elements remain with the coupled kinematics Q16 and B4; the elements adjacent to the fixed edges are meshed with plate finite elements having Q9 in-plane and B3 along the thickness; the connecting elements are beam finite elements B3 with Q16 cross-sectional mesh at tips nodes in contact with the plate finite elements Q16 and with Q9 cross-sectional mesh at middle and tips nodes in contact with the plate finite elements Q9, as shown in Figure 3.5.2. The results are presented in Figure 3.5.3 in terms of displacements and stresses. The final

number of DOFs for the full plate, considering this reduced mesh, is 6600, which means saving the 42.77% of the number of DOFs considering the full mesh.

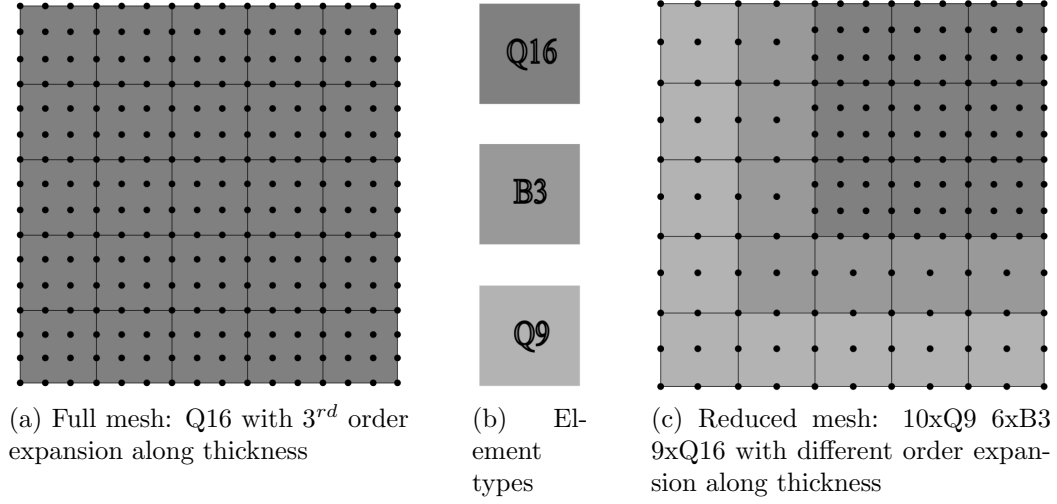


Figure 3.5.2: Reduced mesh of quarter-plate model

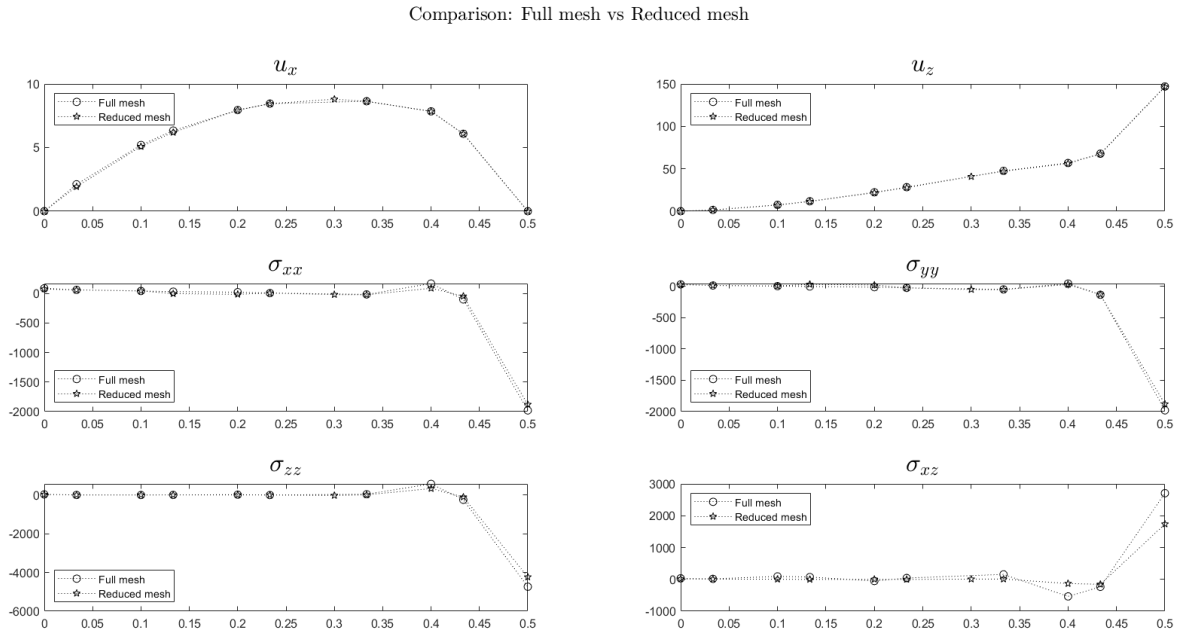


Figure 3.5.3: Square plate - Comparison between full and reduced mesh

3.5.2.2 Razzaque's skew plate

Starting from a mesh with plate finite elements with Q16 and B4 kinematics for in-plane and along the thickness respectively, the reduced mesh is achieved considering the area near the applied load not changed for the in-plane mesh, as in the previous case, in order to ensure high accuracy elements capable to reproduce displacements in the stressed zone, although the mesh along the thickness is B2 for all the nodes except those in correspondence of the beam finite element B3, which are B4; the elements near the fixed edges are plate finite elements with Q9 in-plane and B3 along the thickness mesh; the connecting elements are beam finite elements B3 with Q16 cross-sectional mesh at middle and tips nodes near the Q16 plate elements and Q9 cross-sectional mesh at tips nodes near the Q9 plate elements, as shown in Figure 3.5.4. The results are presented in Figure 3.5.5 as displacements. As shown in Figure 3.5.6, the error remains below the 1%. The final number of DOFs for the full plate, considering this reduced mesh, is 4593, which means saving the 38.76% of the number of DOFs considering the full mesh.

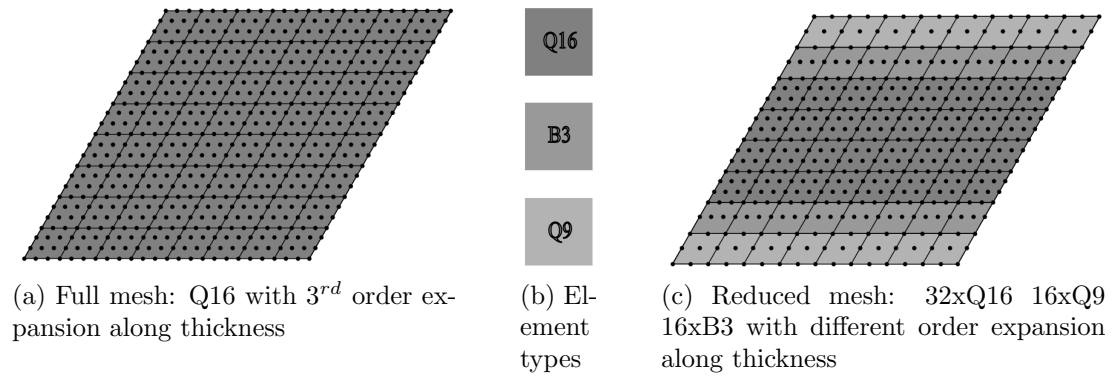


Figure 3.5.4: Reduced mesh of Razzaque's skew plate

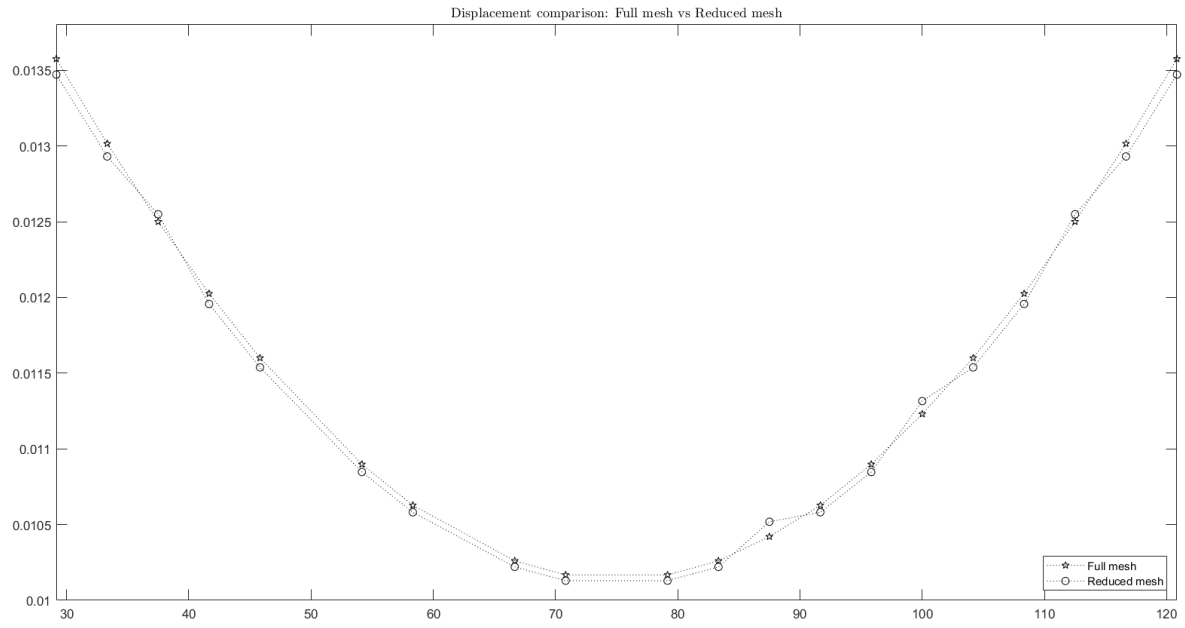


Figure 3.5.5: Razzaque's skew plate - Displacement comparison between full and reduced mesh

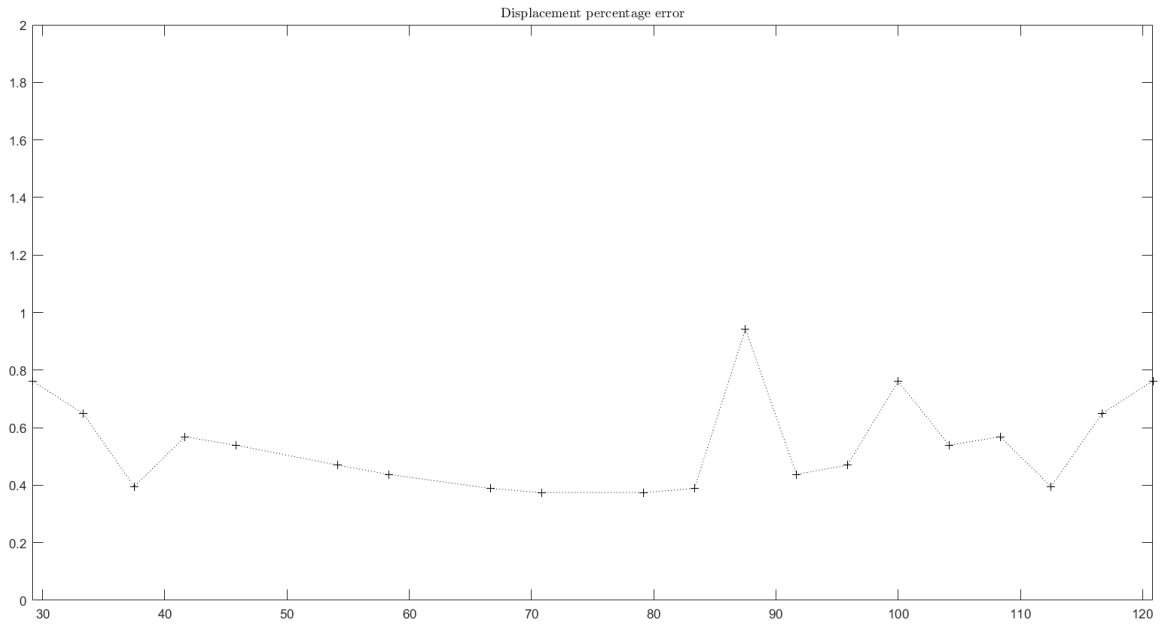


Figure 3.5.6: Razzaque's skew plate - Percentage displacement error

3.5.3 Influence of Mesh Distortion

The meshing technique represents the capability to discretize the domain of interest, to fit the real physical continuum in the best possible way. In the engineering field, meshing is a crucial milestone in creating the model of the problem to solve, it is the starting point on which the simulation is performed and it is important that it can represent in the more accurate way the analyzed domain. The physical area investigated is not so regular, in general, and the meshing should account for it.

The most used technique is the Finite Element Method, which relies on a quite simple mathematical model capable of describing the field under investigation (structural and/or fluid dynamic) up to a certain level of accuracy. This method allows the construction of 2D and 3D regular elements with a prescribed number of nodes each [8]. The main issues of this method are represented by: the constraint to regular presetting of elements; the difficulties managing different types of elements on the same domain, perhaps with a different number of nodes, therefore a different approximation; the high computational cost due to the high number of elements used, to adapt when the real domain it is not geometrically regular or when enrichment of the mesh is required because it is in the area of interest.

To alleviate meshing issues due to FEM, more advanced techniques can be used, resorting to numerical methods that are designed from the very beginning to provide arbitrary order of accuracy on more generally shaped elements. These techniques are based, for instance, on the Virtual Element Method (VEM) or Polytopal Element Method (PEM) [71]. In fact, in contrast to FEM in which elements are typically triangular and quadrilateral in 2D or tetrahedral and hexahedral in 3D, the VEM permits arbitrary two-dimensional polygonal and three-dimensional polyhedral elements [92], having many aspects closely connected with Finite Volume Methods (FVM) and in particular with Mimetic Finite Differences (MFD) [90]. This allows the problem domain to be discretized by elements represented by arbitrary polygons, which can be concave and convex. Moreover, different polynomial consistency is allowed within the method and non-conforming discretizations can be handled, mainly for local refinement and so on, representing the key aspect of this method [91].

The newly proposed Adaptive Elements, which are Finite Elements based on Carrera Unified Formulation (CUF), represent more agile and manageable elements, based on a well-known technique (FEM), easy to implement and can produce the same benefits obtained using VEM. As a matter of fact, the Carrera Unified Formulation, applied both to 1D and 2D finite elements, provides very accurate 3D-like solutions with much lower computational cost than the classical 3D elements, becoming an important tool for the analysis of structural problems.

Recently, in the framework of CUF, a novel approach called Node-Dependent Kinematics (NDK) has been proposed to further increase the numerical efficiency of the models [121, 122, 123, 124, 125]. By relating the kinematic assumptions to the chosen nodes, FE models with variable nodal kinematics can be built conveniently. Naturally, the critical zone with higher-order assumptions can be bridged to the less-critical area modeled with adequate lower-order theories [126], in such a way the numerical modeling procedure can further be simplified.

The Node-Dependent Kinematics approach, along with the 3D modelling of non-orthogonal geometries, opens the possibility to create elements with a non-conventional number of nodes, with non-conventional geometrical shapes, and with different polynomial approximations along different directions, leading to a certain freedom in constructing proper elements. These elements are formulated considering that the integrals inside the matrices of the governing equations (stiffness and mass matrices), which involves the approximating functions, and the Jacobian matrix of the elements are computed in 3D form, although 1D and 2D CUF models are still employed. In conclusion, 3D elements can be recovered with a non-regular shape that can adapt to the edges of the domain considered or to adjacent portions which share one or more sides, when the geometrical constraints or the kinematic behaviour require that.

3.5.3.1 Patch Test

This standard test is classically used in order to investigate the mesh distortion sensitivity in plate bending problems. The in-plane dimensions of the square plate considered are $100 \times 100 \text{ m}^2$, while the thickness is 5 m . The boundary conditions applied are: Clamped-Clamped (CC), all edges are fixed; Simply-Supported (SS), all edges have no displacement in the vertical direction and the slope normal to the edge must be zero. The load is expressed as a concentrated force at the centre of the top surface equal to $q = 4 \text{ N}$. The material considered is isotropic with the following properties: $E = 10.92 \text{ Pa}$, $\nu = 0.30$ and $\rho = 1 \text{ kg/m}^3$. The starting meshes are a 4×4 Q4 elements in the plane with 1 B4 element along the thickness, and a 4×4 Q9 elements in the plane with 1 B3 element along the thickness.

Due to the symmetry of the problem, with symmetric geometry and symmetric boundary conditions, the quarter-plate finite element model is used to simplify the analysis: the full plate is divided into four equal sub-plates and the study is conducted only on one of them, as shown in Figure 3.5.1. In particular, the bottom-left quarter is chosen and the following boundary conditions are applied, considering as u , v and w the displacements along x , y and z respectively: bottom and left edges are fixed or simply-supported, on the top edge $v = 0$, on the right edge $u = 0$ and on the top-right corner a force of $q = 1 \text{ N}$ is applied.

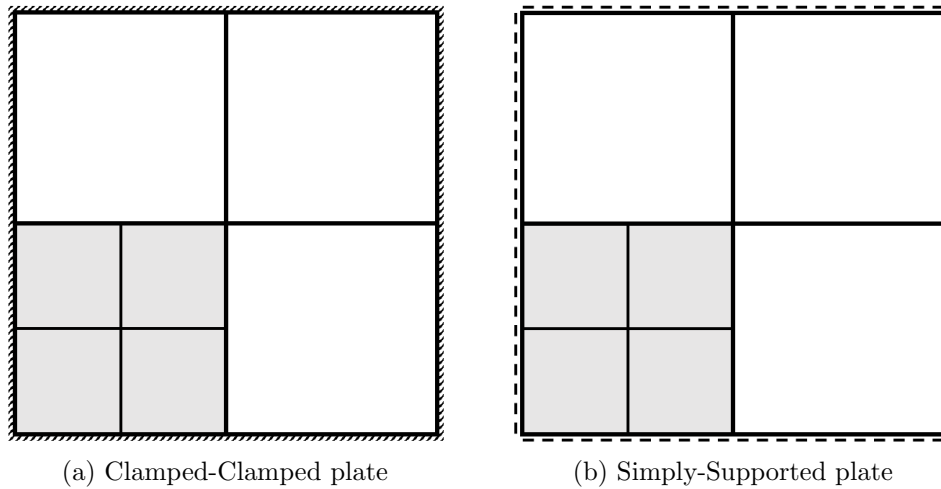


Figure 3.5.1: Quarter plate approximation

3.5.3.2 Mesh distortion sensitivity

The resulting quarter plate mesh is 2×2 based on Q4 or Q9 elements with B3 discretization along the thickness, see Figure 3.5.2. Note that from now on, the

B2 element is colored heavenly, the B3 element is colored yellow, the Q4 element is colored light blue and the Q9 element is colored orange. Then, three different configurations are analyzed:

- 2xQ4 1xB2: in this configuration, the previous two elements at the top are substituted by a single 1D element with 2 nodes, having a Q9 approximation on the cross-section;
- 2xQ4 1xB3: as in the previous case, the two elements at the top are substituted by a single 1D element but having 3 nodes, where the cross-section is expressed as a Q9 element;
- 2xQ4 1xQ9: the number of nodes is identical with respect to the previous case, but the discretization of the two elements at the top is different, accounting for a single Q9 element having a B3 discretization along the thickness.

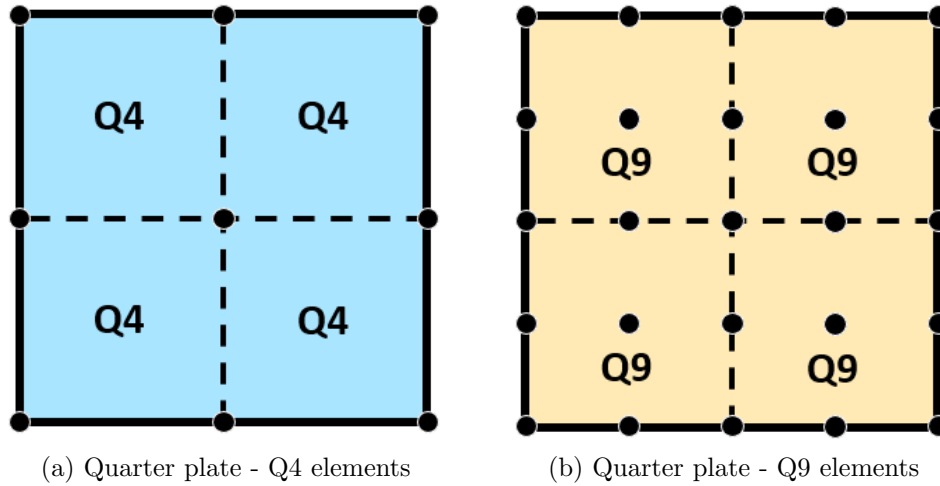


Figure 3.5.2: Quarter plate - original mesh

In Figure 3.5.3 and Figure 3.5.4 it is possible to visualize the described configurations: the black dots are the nodes used during the implementation, while the white dots represent the nodes added considering the thickness and the cross-section of the plate and the beam elements, respectively.

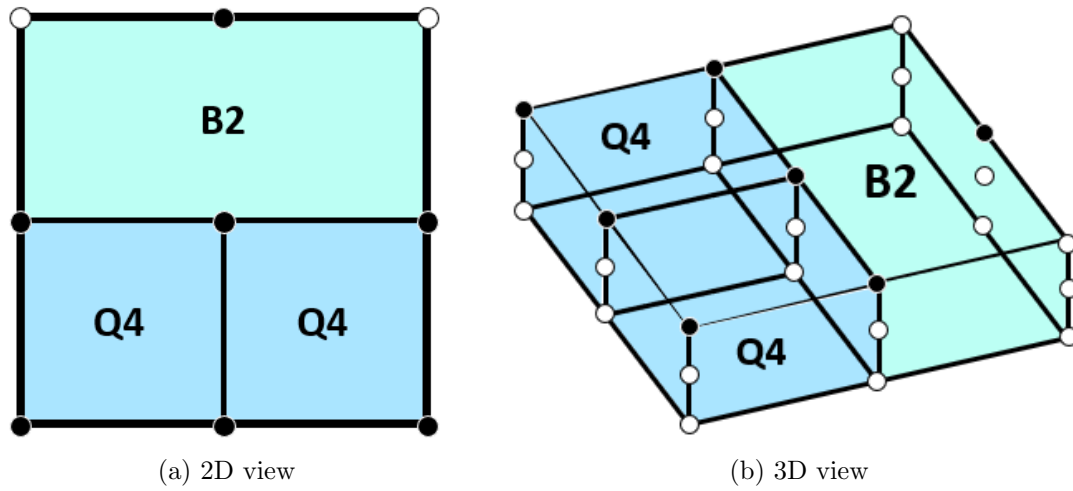


Figure 3.5.3: Quarter plate - 2xQ4 1xB2

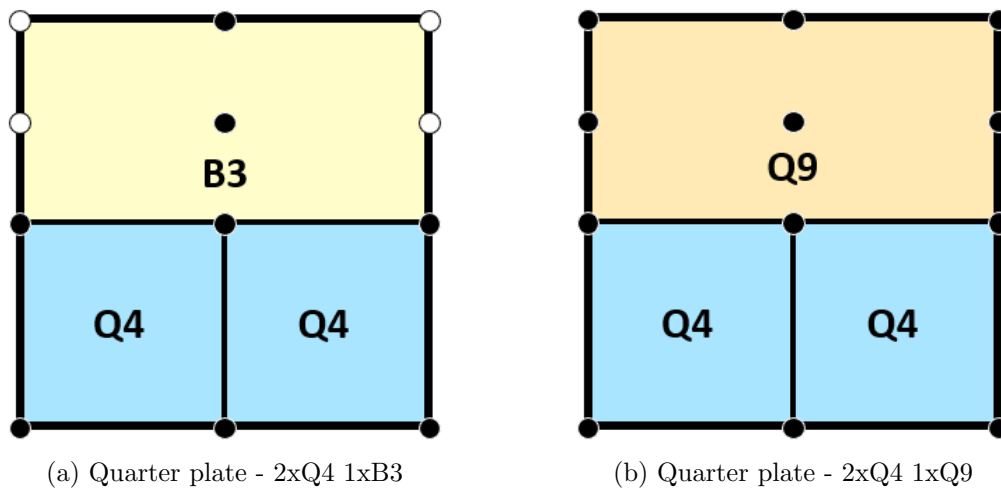


Figure 3.5.4: Quarter plate - 2 different configurations

In Figure 3.5.5 the emphasis is focused on the beam elements and on how is possible to freely move the nodes associated to them. In particular, the original configuration is compared with the new one, indicated with a dash-dotted line.

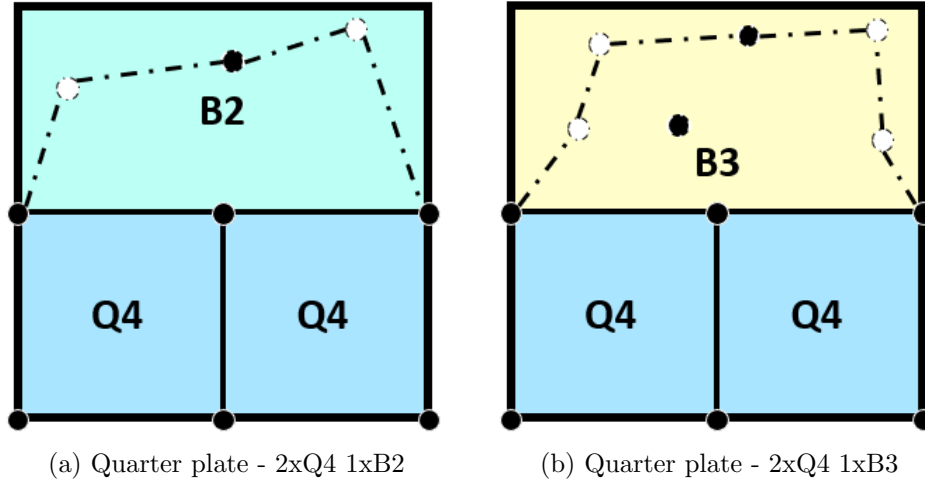
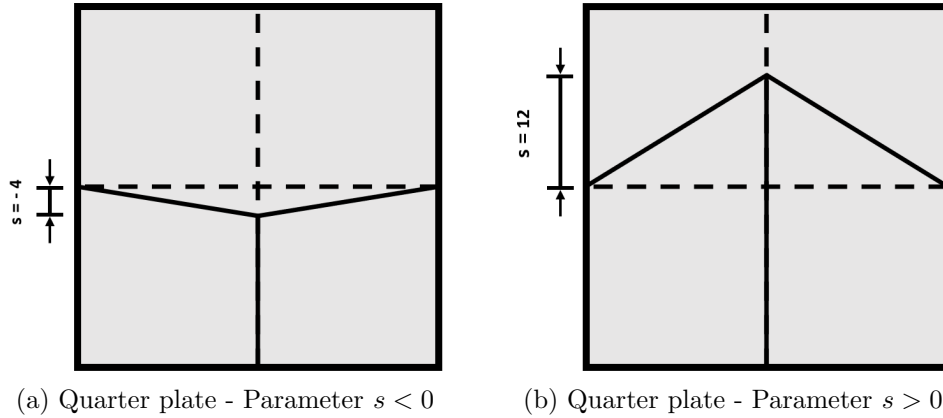


Figure 3.5.5: Quarter plate - distorted element configurations

The mesh distortion is characterized by the parameter $s \in \{-12, -10, -6, 0, 6, 10, 12\}$, which defines the coordinates of the mid-node of the quarter plate. Note that the parameter s can have negative and positive values both: this is not common in literature and, particularly in this work, it is used to control the property of the element: in fact, as shown in Figure 3.5.6, if the parameter s is negative, the considered element is convex; if the parameter s is positive, the element becomes non-convex.


 Figure 3.5.6: Quarter plate - distorted mesh defined by parameter s

Note that the y -coordinate of the mid-node changes according to s : defining a as the length of the quarter plate, the coordinate varies as $y = (a/2) + s$, where in

Figure 3.5.6a $s = -4$, so the coordinate is $y = (a/2) - 4$; in Figure 3.5.6b $s = 12$, so the coordinate is $y = (a/2) + 12$.

3.5.3.2.1 Static and modal analysis

The three different configurations are used to perform a static and modal analysis, as done in [136]. Regarding the static analysis, the results are reported in terms of transverse displacement $U_z = u_z(a, a, 0)$ at plate's center, normalized with respect to the reference value U_z^0 calculated for the regular mesh ($s = 0$). The results are reported in Figure 3.5.7 and Figure 3.5.8, where different boundary conditions are considered. It is evident that the configuration having B2 element with a Q9 discretization along the cross-section has almost the same trend with respect to the other configurations, with the benefit of having fewer Degrees Of Freedom.

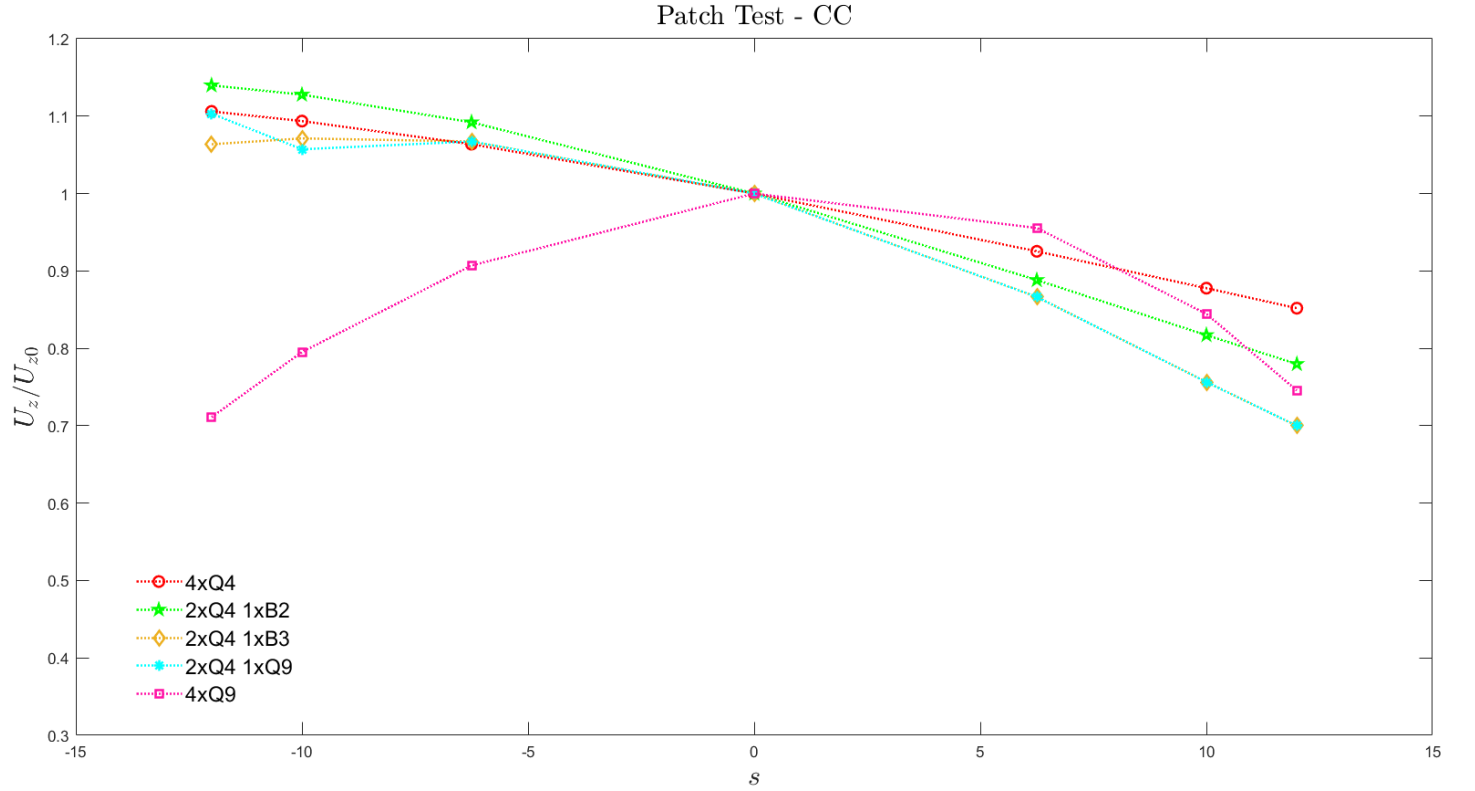


Figure 3.5.7: Variation of the ratio U_z/U_z^0 for CC quarter plate

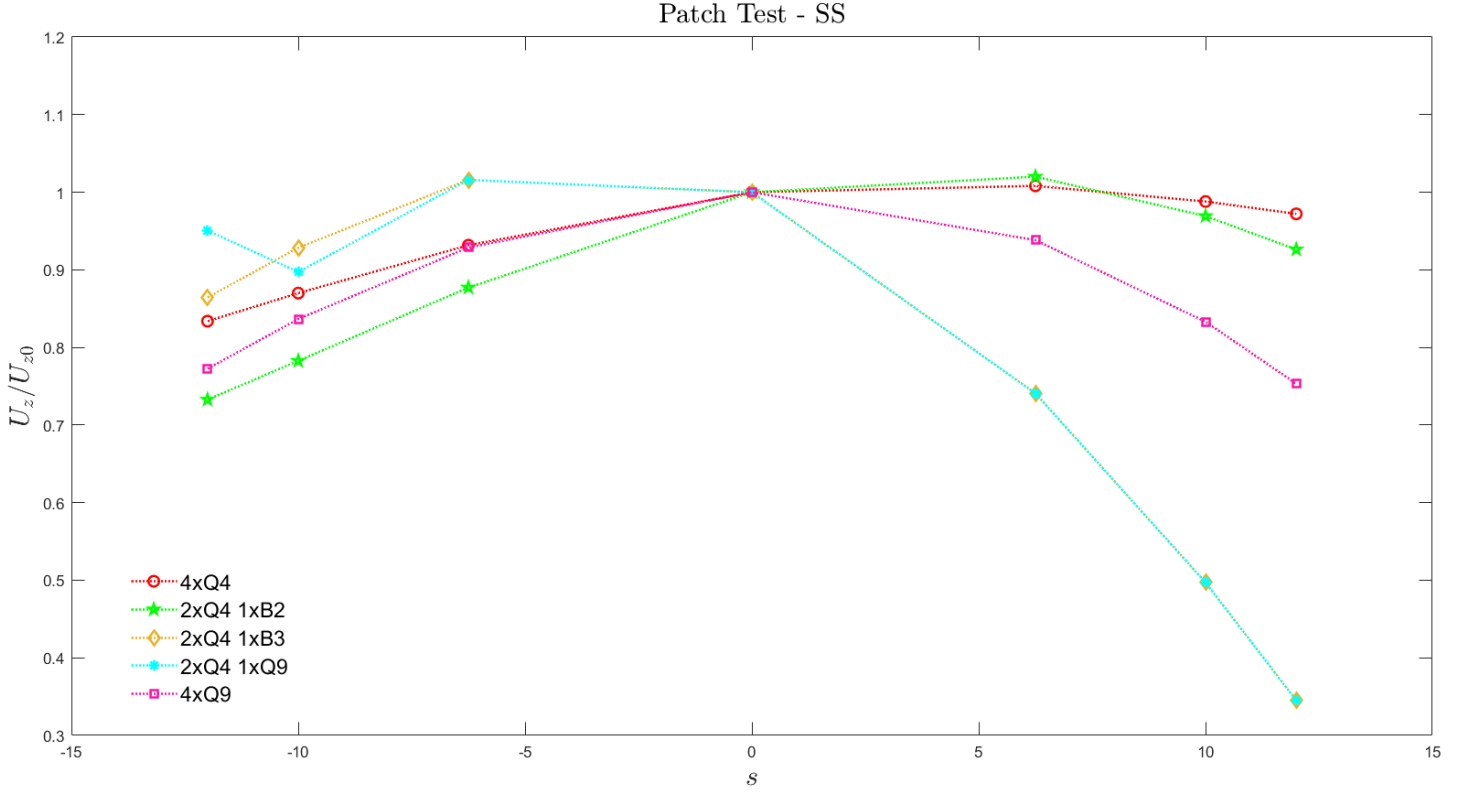


Figure 3.5.8: Variation of the ratio U_z/U_z^0 for SS quarter plate

	Configuration	U_z^0	DOFs
CC	4xQ4	0.2784	81
	2xQ4 1xB2	0.3276	81
	2xQ4 1xB3	0.4883	108
	2xQ4 1xQ9	0.4883	108
	4xQ9	1.5804	225
	Nastran [®]	1.6612	513
SS	4xQ4	0.7542	81
	2xQ4 1xB2	1.0207	81
	2xQ4 1xB3	2.4678	108
	2xQ4 1xQ9	2.4678	108
	4xQ9	3.5733	225
	Nastran [®]	3.6003	513

Table 3.5.1: Tip displacement

3.6 Dynamic Analyses

Dynamic analyses provide a complementary and more stringent assessment of the proposed modelling strategies, as natural frequencies and mode shapes are sensitive to stiffness and mass distributions. In aerospace structures, dynamic margins, resonance avoidance, and vibro-acoustic coupling are key design drivers. In rotating machinery, rotordynamic stability and critical speeds are fundamental to ensure operational integrity.

In this section, free-vibration analyses are discussed for:

- distorted plate configurations (quarter plate and full plate);
- a variable-thickness disk representative of a rotor component.

3.6.1 Free-Vibration Analysis of Plates

Then, a free-vibration analysis is performed considering the three configurations with the same properties except for the boundary conditions: in this case, no boundary conditions are applied, so free edges are considered. The results are shown in Figure 3.6.1 in terms of natural frequencies associated to the first 16 modes: because no constraints are acting on the structure, the first 6 modes are null, so the curves show the first 10 non-zero modes. The plots indicate the family of curves associated to the sign of the parameter s for the three configuration and the last picture summarize the results: it is evident that the error is low and it is independent on the element property (concave or convex), also for severe distortion ($|s| = 12$). The Figure 3.6.2 and Figure 3.6.3 compare the different configurations for a selected value of the parameter s .

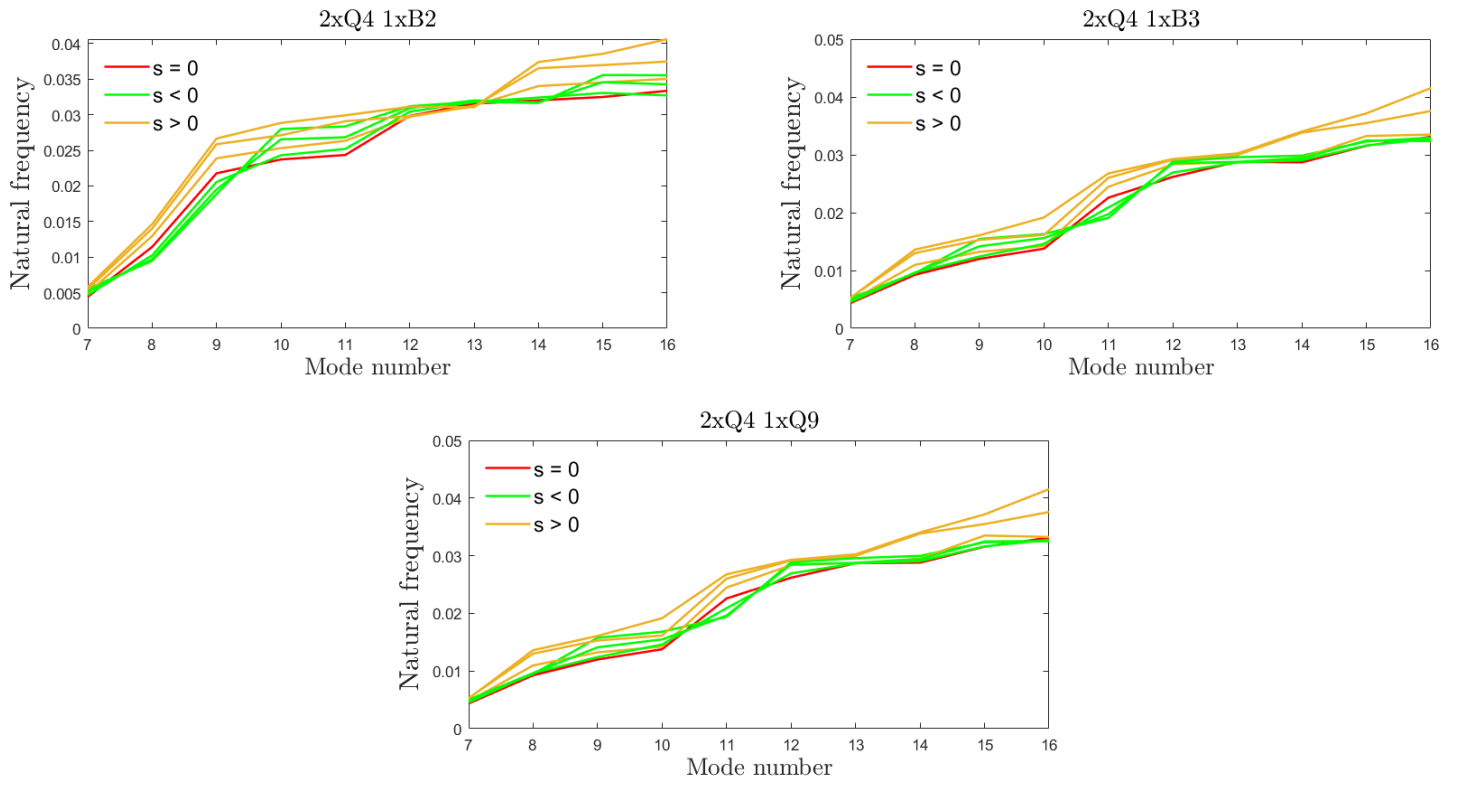


Figure 3.6.1: Quarter plate - modal analysis for different configurations

Modal analysis - CC

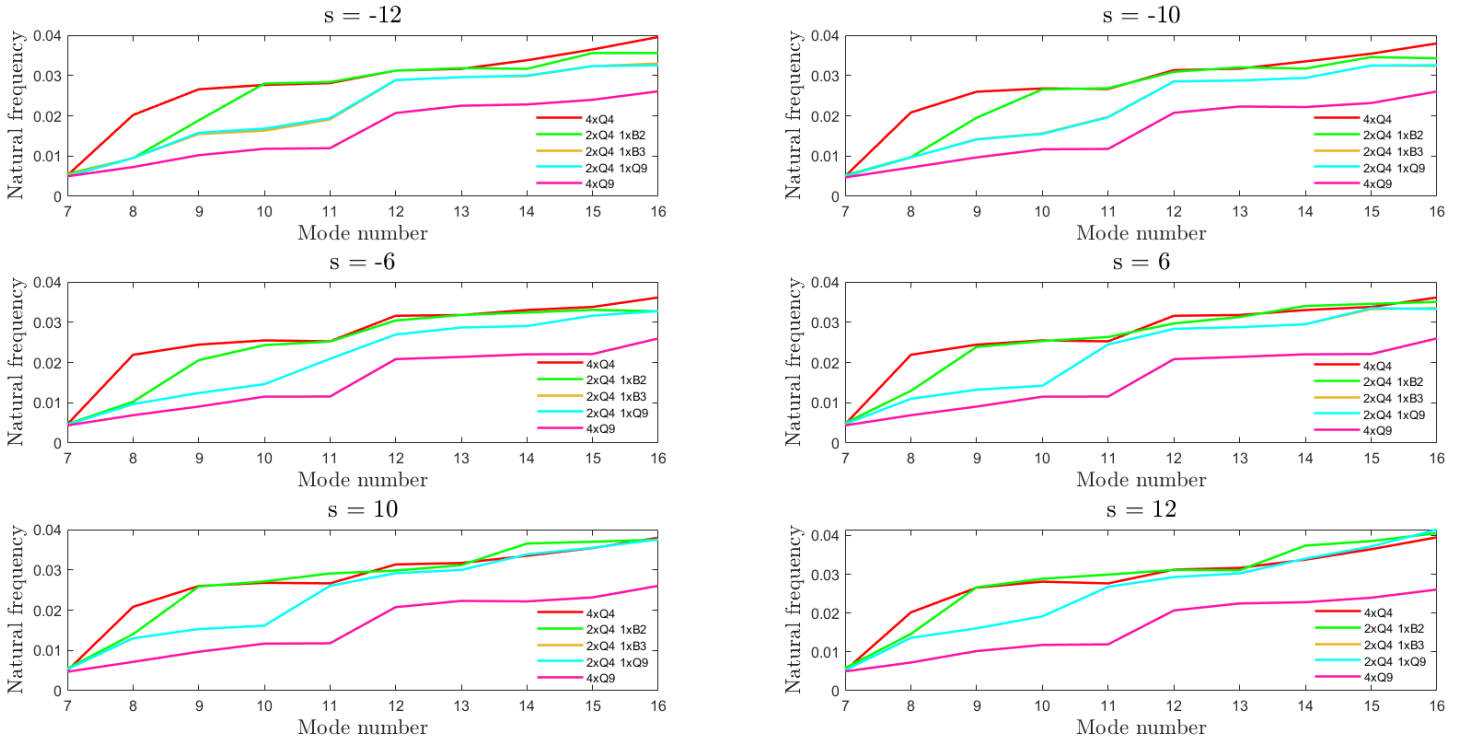


Figure 3.6.2: Quarter plate CC - modal analysis for different values of s

Modal analysis - SS

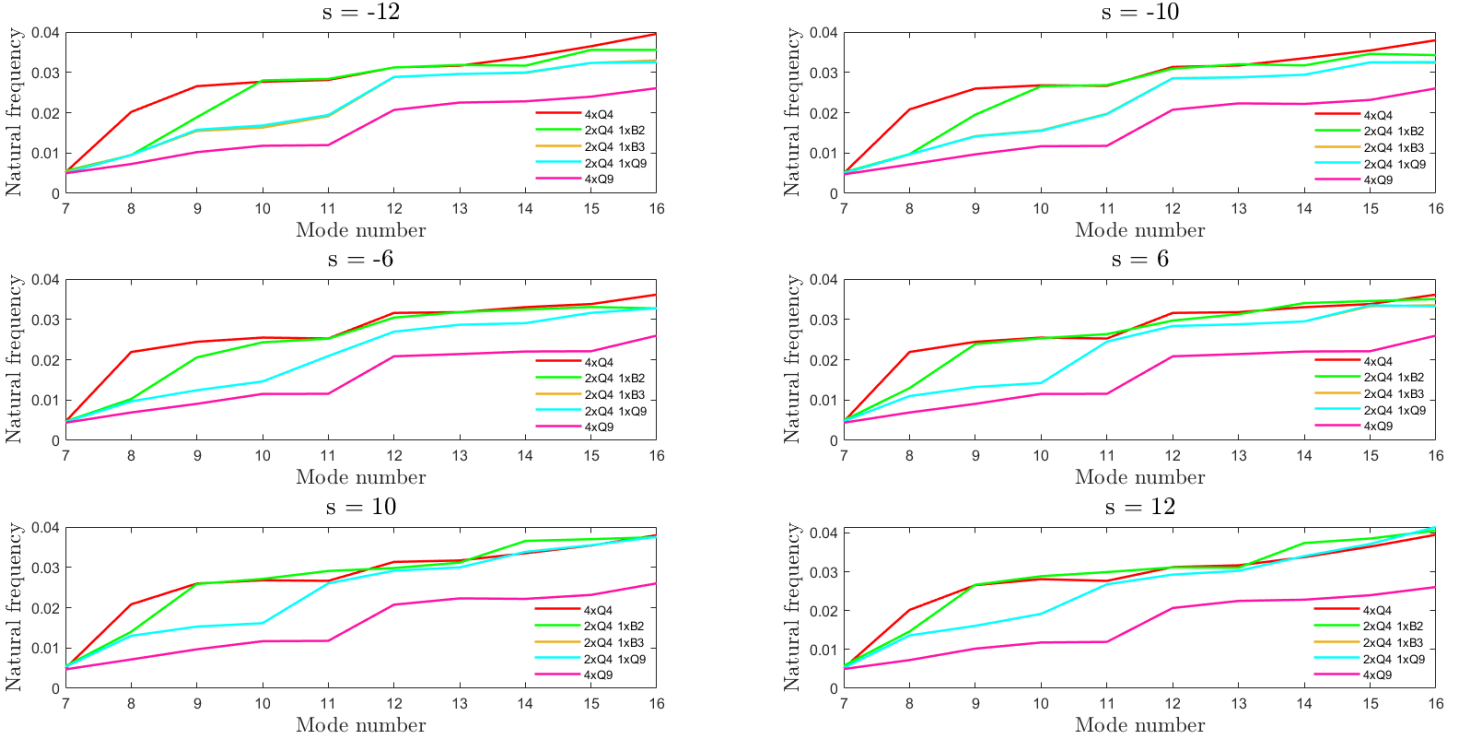


Figure 3.6.3: Quarter plate SS - modal analysis for different values of s

3.6.2 Square plate with distorted mesh

A full square plate is now considered to perform the static and modal analysis. The in-plane dimensions of the plate are $100 \times 100 \text{ m}^2$, while the thickness is 10 m . The boundary condition applied is a clamped edge (the top edge, as illustrated in Figure 3.6.1a) and the load is expressed as a concentrated force at the right-bottom corner equal to $q = 5 \cdot 10^8 \text{ N}$, again represented in Figure 3.6.1a as a crossed circle. The material considered is isotropic with the following properties: $E = 70 \text{ GPa}$, $\nu = 0.33$ and $\rho = 2.81 \cdot 10^3 \text{ kg/m}^3$. Note that a reference problem is analyzed with the commercial software Nastran[®] and illustrated in Figure 3.6.1b. The starting meshes are a 4×4 Q4 elements in the plane with 1 B3 element along the thickness (DOF = 225), and a 4×4 Q9 elements in the plane with 1 B3 element along the thickness (DOF = 729), see Figure 3.6.2 .

In this case, because of non-regular distorted elements are used, no quarter plate approximation is done.

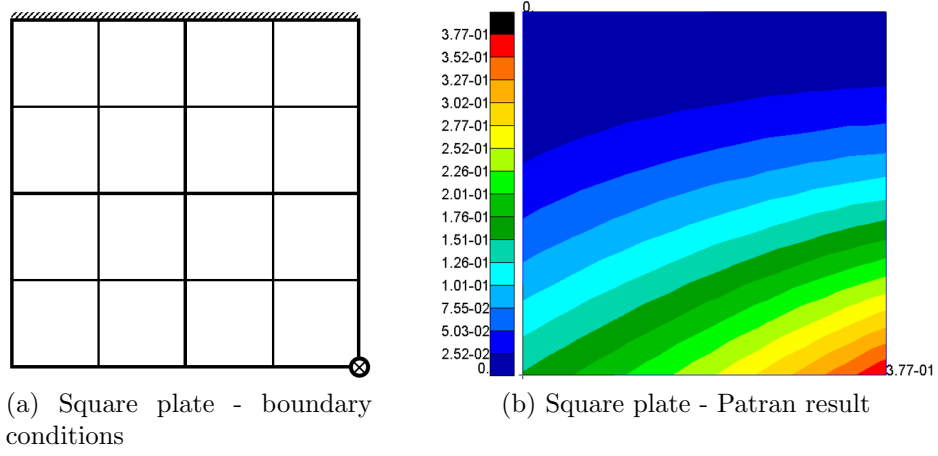


Figure 3.6.1: Square plate - original regular meshes

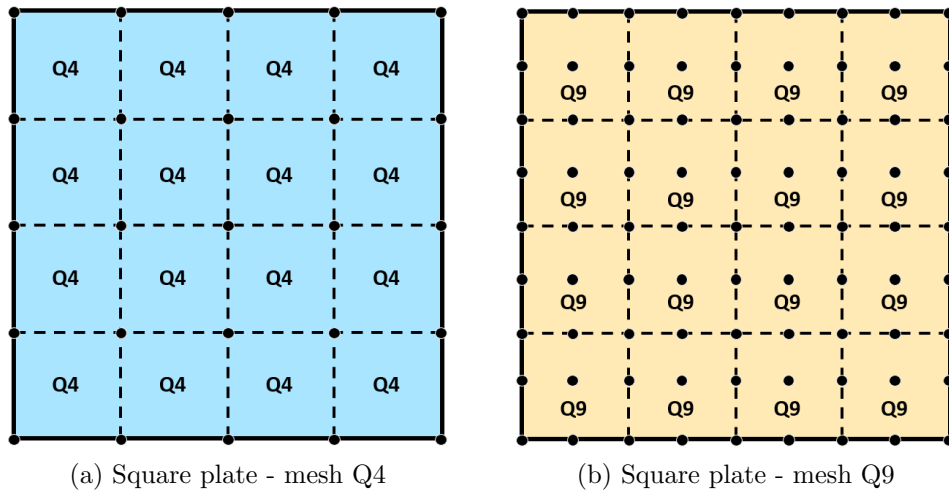


Figure 3.6.2: Square plate - original regular meshes

Starting from these regular meshes, the elements are gradually distorted to demonstrate the capabilities of the presented Adaptive Elements. The nodes coordinates of some elements are varied in a random way, considering a Gaussian distribution having the mean equal to the original coordinate and the standard deviation increasing little by little. Stating that x_0 is the original coordinate of a node, σ_{dev} the standard deviation and μ the mean value, the new coordinates are obtained as follow:

$$x_0 = x_0 + \sigma_{dev} \cdot randn(\mu) \quad (3.6.1)$$

where the Matlab[®] function `randn` returns pseudorandom values drawn from the standard normal distribution, $\mu = 1$ and the standard deviation varies in a range equal to $\sigma_{dev} \in \{1, 2, 3, 5\}$.

Moreover, in other examples, the nodes are moved intentionally to obtain specific shapes (for instance, a pentagon, which is a benchmark example for the VEM). As a final example, a complete non-regular distorted mesh is considered. For all of these cases, a combination of plate and beam elements is exploited.

The previous meshes are reported in Figure 3.6.3, where:

- Case A: in Figure 3.6.3a Mesh 9xQ9 1xQ4 6xB2 (DOF = 387) - the mesh is obtained using a standard deviation equal to $\sigma_{dev} = 3$;
- Case B: in Figure 3.6.3b Mesh 3xQ9 3xQ4 10xB2 (DOF = 330) - the mesh is obtained creating some pentagon elements;
- Case C: in Figure 3.6.3c Mesh 3xQ9 3xQ4 10xB2 v2 (DOF = 330) - an upgraded version of the previous mesh is obtained to highlight the presence of pentagon elements, leading to a more distorted mesh with the same number of DOF;
- Case D: in Figure 3.6.3d Mesh 8xB3 9xQ9 (DOF = 765) - a complete distorted mesh is obtained and modelled using high order accuracy in the approximation of the kinematics (quadratic interpolation);
- Case E: in Figure 3.6.3e Mesh 10xQ4 7xB2 (DOF = 315) - a similar mesh to the previous one is considered, using a lower order accuracy of the kinematics approximation (linear interpolation), to investigate if it is possible to obtain acceptable results with fewer Degrees Of Freedom.

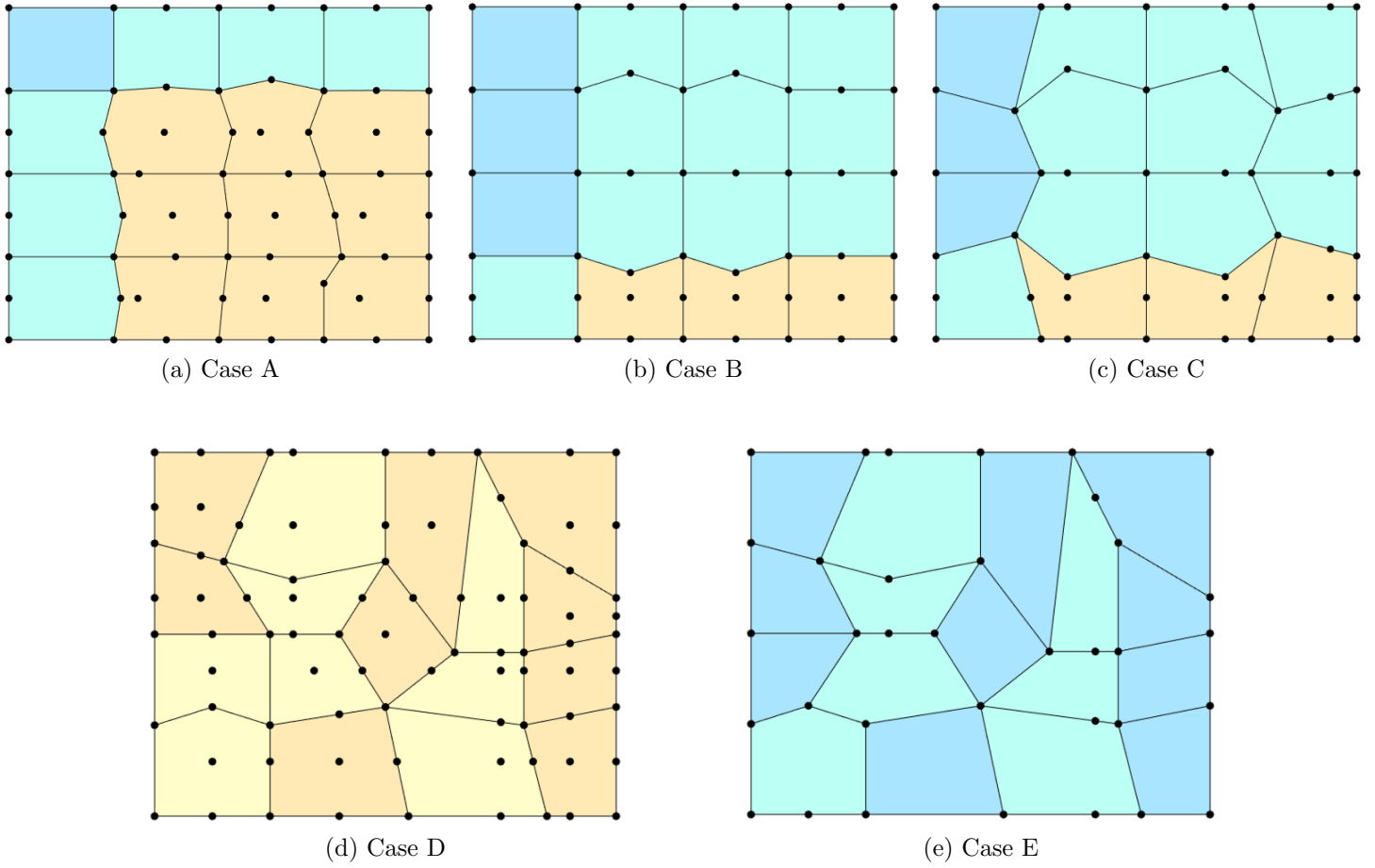
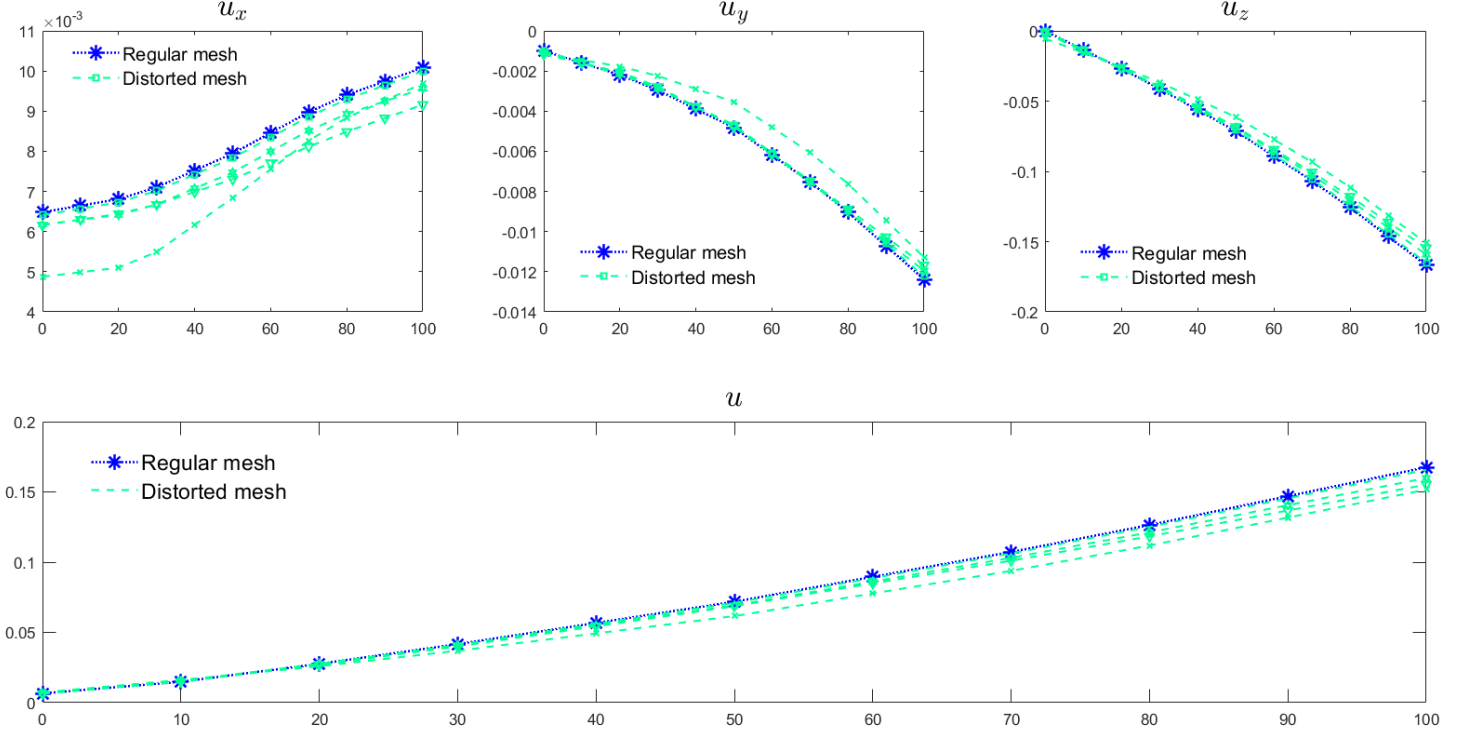


Figure 3.6.3: Square plate - distorted meshes

3.6.2.1 Static and modal analysis

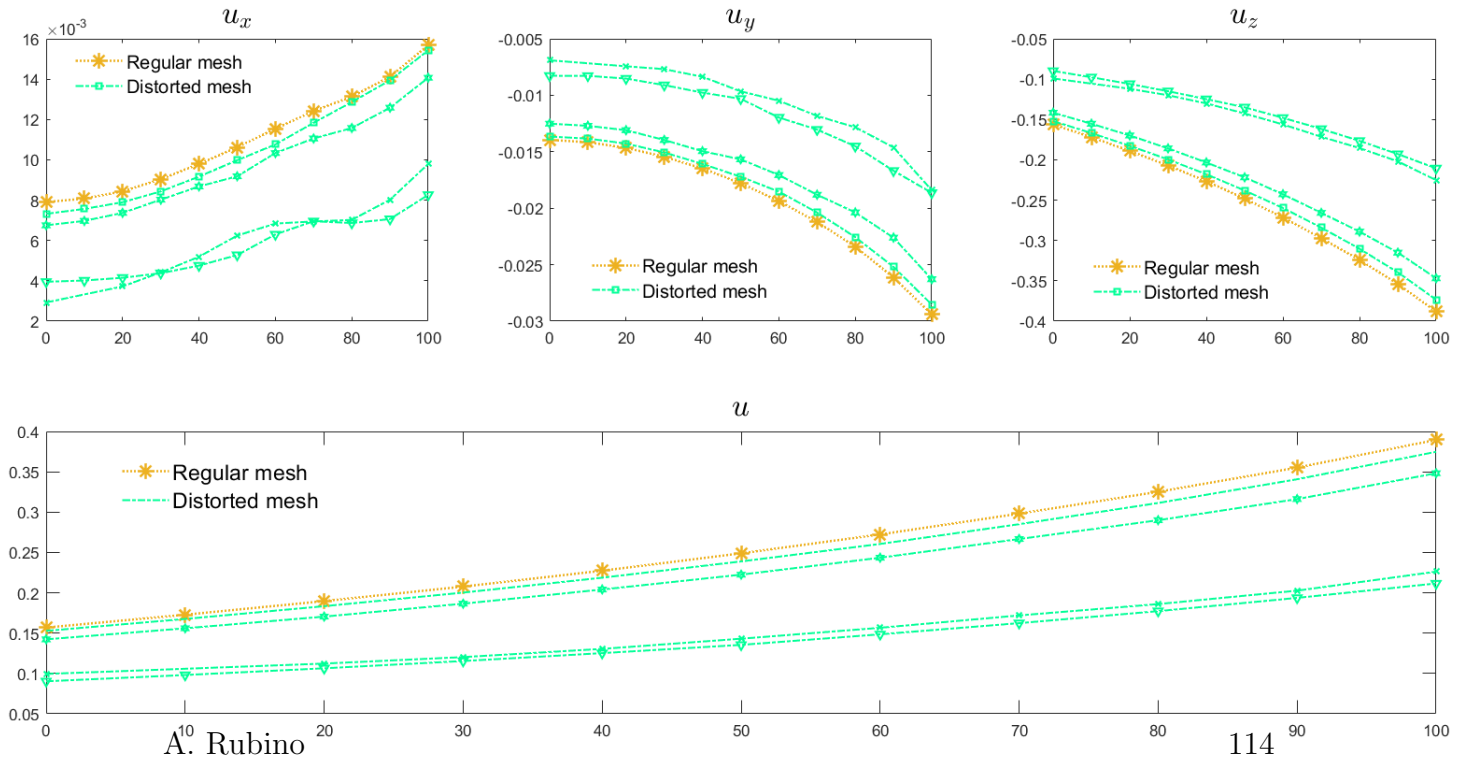
The different meshes are investigated using static analysis and the results are reported in terms of displacements along the bottom free edge (the edge where the corner on which the load is applied belongs to). In the following figures, the three components of the displacements and the relative magnitude is shown. In particular, in Figure 3.6.4, the results obtained with the distorted mesh (considering different values of standard deviations σ_{dev}) are compared with the solution obtained with the regular one: the error between the data is very low. Note that in Figure 3.6.4b some curves referring to the distorted mesh are quite far from the solution of the regular mesh: they refer to a $\sigma_{dev} = 5$, which is too high for the number of nodes in the selected element Q9, leading to a non-legal intersection of the nodes (physically, there is a permeation of material, which is not possible).

Distorted mesh - Q4



(a) Square plate - mesh Q4

Distorted mesh - Q9



(b) Square plate - mesh Q9

Figure 3.6.4: Square plate - static analysis

Then, each mesh is analyzed and compared with the family of curves related to the Q4 and Q9 elements. In Figure 3.6.5 is shown the Case A in which there is almost a balance between elements with high order accuracy (Q9) and elements with a lower order of accuracy (Q4, B2). In case B the predominant elements are similar to the Q4 discretization; then, the Case C is similar to the previous case, with the difference that now the pentagons are immersed in a more distorted mesh. Because of the severe distortion of the elements, in both Cases D and E, the results tend to align to the reference solutions related to Q9 distorted mesh with $\sigma_{dev} = 3$ for the first case and related to Q4 distorted mesh with $\sigma_{dev} = 5$ for the second case. In conclusion, it is evident that the results are between the two reference curves of Q4 and Q9, although the presence of the distorted elements.

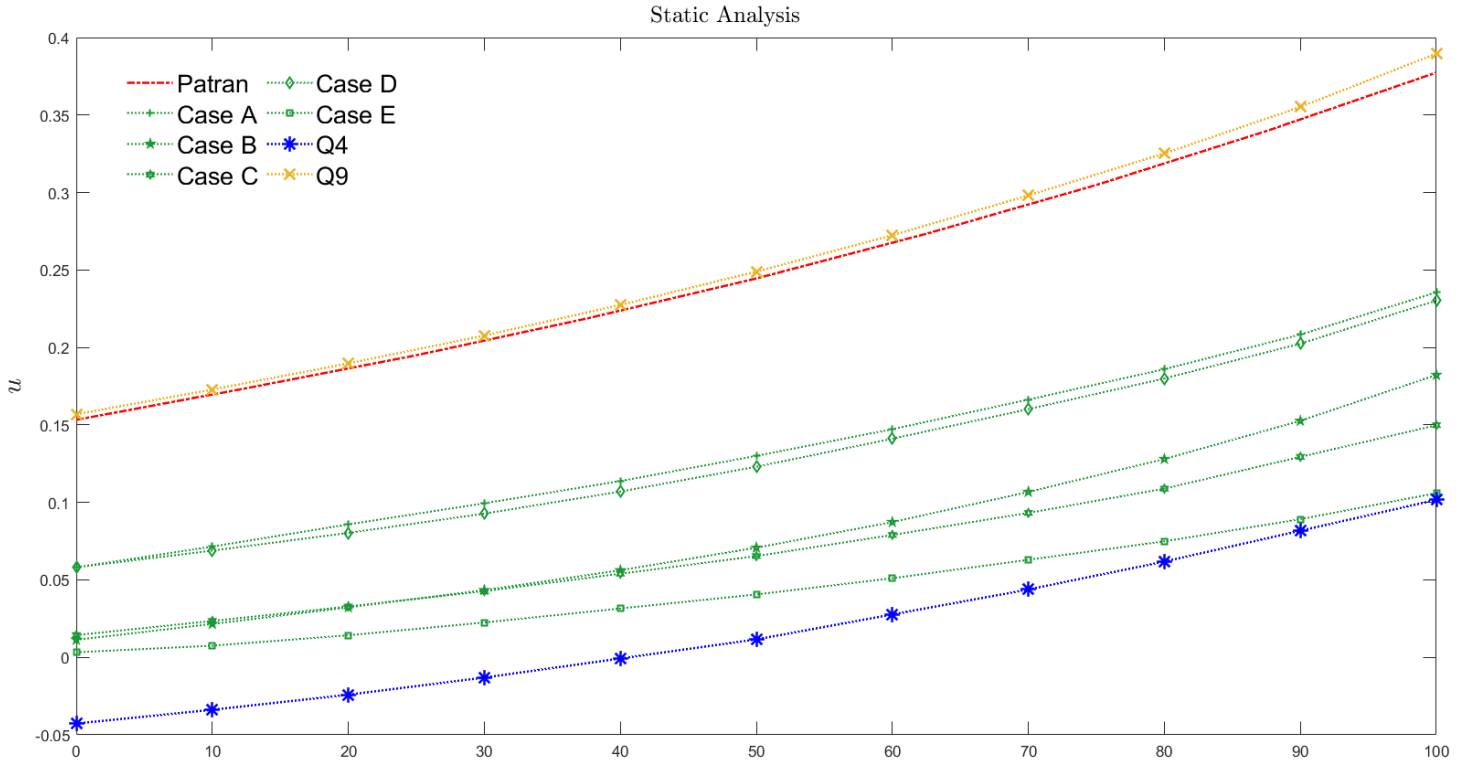


Figure 3.6.5: Square plate - static analysis

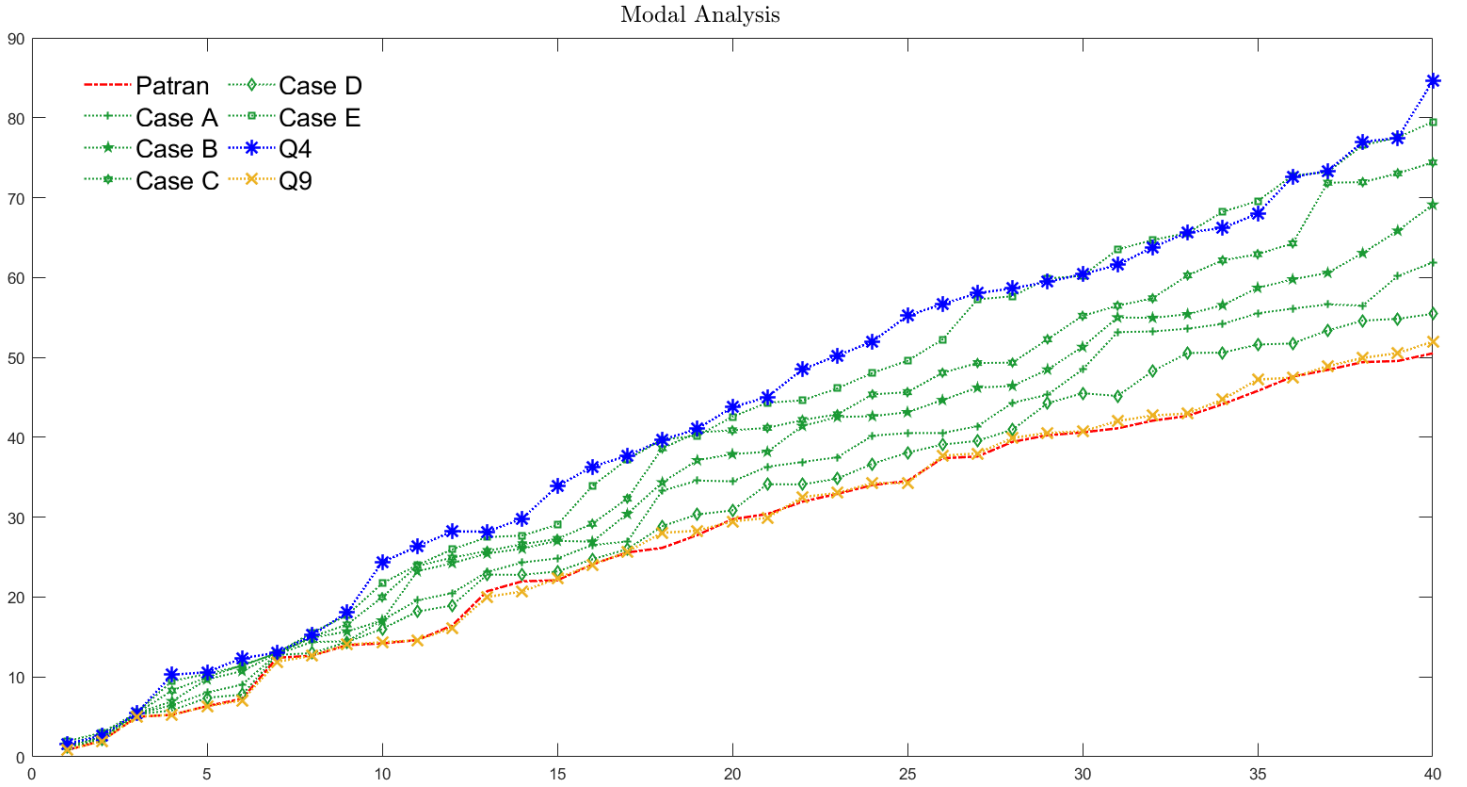


Figure 3.6.6: Square plate - modal analysis

In a similar manner as done for the static analysis, also for the modal investigation the frequencies are compared with respect to the solutions related to the Q4 and Q9 elements families, as shown in Figure 3.6.6. Note that the frequencies of the distorted elements are perfectly bounded by those of the regular mesh, in particular the Case D is very close to the Q9 solution.

3.6.3 Free-Vibration Analysis of Variable-Thickness Rotors

The rotordynamics represents a fundamental branch in industrial sector. The physical behaviour of rotors and rotating machines is strongly affected by vibrations, which are enhanced by the rotation and the structure itself. Assemblies of perfectly symmetric rotors show different resonance frequencies as a function of the angular velocity, but imperfections and imbalances could excite these frequencies in complex modes. During the design of machines with rotating parts, an effective way is necessary to take into account these behaviours, optimizing operations and performances [141].

Usually, the main part of a rotating machine is composed by a shaft and one or more rotating disks [58, 142]. In practical problems (such as rotor turbines for aerospace industry or for energy production), those disks have a complex profiles, which can be designed as a combination of concave, convex and linear sections. Because of the complexity and the presence of components such as spacers, blades and air seals, the analytical solution of the modelled problem is available only for very simple cases, with simple geometry under axisymmetric thermal and mechanical loads. Composite real design can be analyzed only using numerical approaches, including the Finite Difference Method (FDM), the Finite Element Method (FEM) and the Boundary Element Method (BEM) [143].

In this part of the chapter, a refined model of a disk with complex shape is presented. Using a refined model based on FEM combined with the Carrera Unified Formulation (CUF), a free vibration analysis is performed. In particular, the Node-Dependent Kinematic (NDK) approach allows to discretize the rotating disks in a smooth way, accounting for the variable thickness of the considered elements [68]. The results are compared and validated with the solution obtained from the commercial software Nastran®.

3.6.3.1 Problem description

An illustrative example of a rotor with complex shape is analyzed. To demonstrate the ability of the method to analyze rotor disks with arbitrary profile, an annular disk with hyperbolic profile may be considered, as done in [144]. In particular, the disk profile is assumed to be symmetric with respect to the middle plane perpendicular to the axis. The inner and outer radii are $r_{in} = 0.05 \text{ m}$, $r_{out} = 0.2 \text{ m}$, while the thickness of the disk from the value of $h_{in} = 0.06 \text{ m}$ in the inner radius varies as $h(r) = Cr^\alpha$ to the value of $h_{out} = 0.03 \text{ m}$ in the outer radius, where $C = 0.0134$ and $\alpha = -0.5$, as shown in Fig. 3.6.1. The material of the disk is assumed to be steel with $E = 207 \text{ GPa}$, $\nu = 0.28$ and $\rho = 7860 \text{ kg/m}^3$.

In order to derive a finite element formulation of the problem, a 1D refined FEM in the CUF framework is employed. In this approach, the rotor may be assumed as a beam along its axis so that each cross-section of the beam is defined in any plane perpendicular to the axis. The structure, therefore, can be discretized into a finite number of 1D beam elements along the axis.

In the present study, two different options are considered for both the axis and the cross-section discretization. These options are then combined to assess the performance of the proposed formulation.

Regarding the axis discretization, 1D elements are employed. The first option consists of a series of B2 elements, where the interpolation between the two nodes of each element is linear. The second option considers a single B4 element, where the interpolation among the nodes is cubic. These two configurations are illustrated

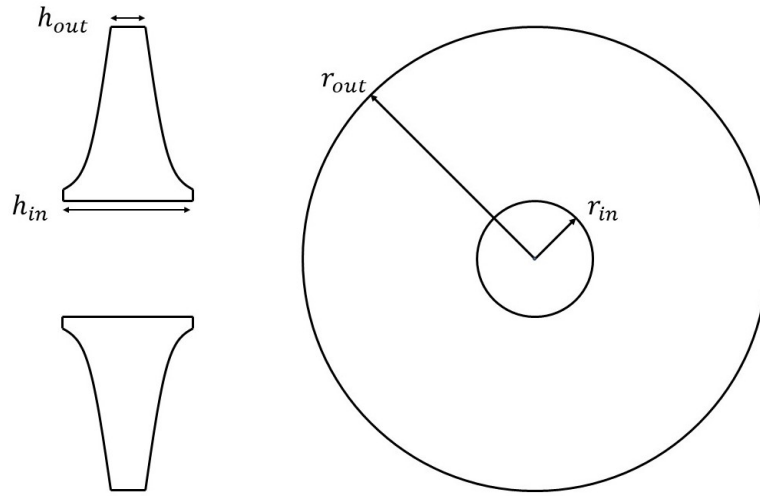


Figure 3.6.1: Disk geometry

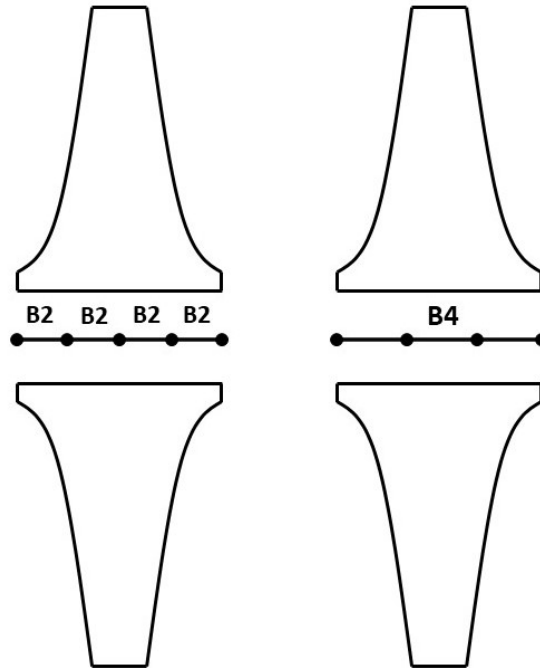
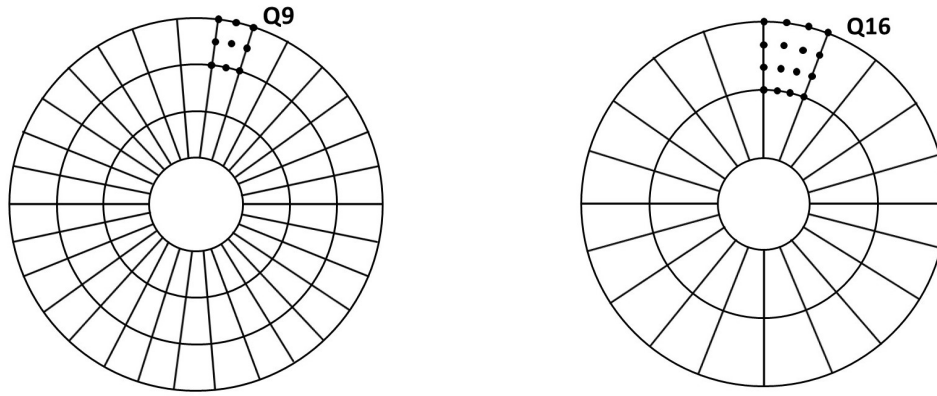


Figure 3.6.2: Axis discretization

in Fig. 3.6.2, where the left side depicts the series of four B2 elements, while the right side shows the single B4 element.

Concerning the cross-section discretization, 2D elements are employed. The first option is to consider a series of Q9 elements, where the interpolation is parabolic; then, the second option is to consider a series of Q16 elements, with a cubic interpolation among the nodes. These options are illustrated in Fig. 3.6.3, where both series are along the radius but with a different number of elements.



(a) 90 elements - 3xQ9 in radial direction

(b) 40 elements - 2xQ16 in radial direction

Figure 3.6.3: Cross-section discretization

In [144], different discretization methods for the disk modeling the rotor have been investigated, considering a total of eight approaches. The proposed solutions are illustrated in Fig. 3.6.4.

In contrast, Fig. 3.6.5 presents the discretizations adopted in the present work, where a smoother transition is evident compared to the stepwise discretization.

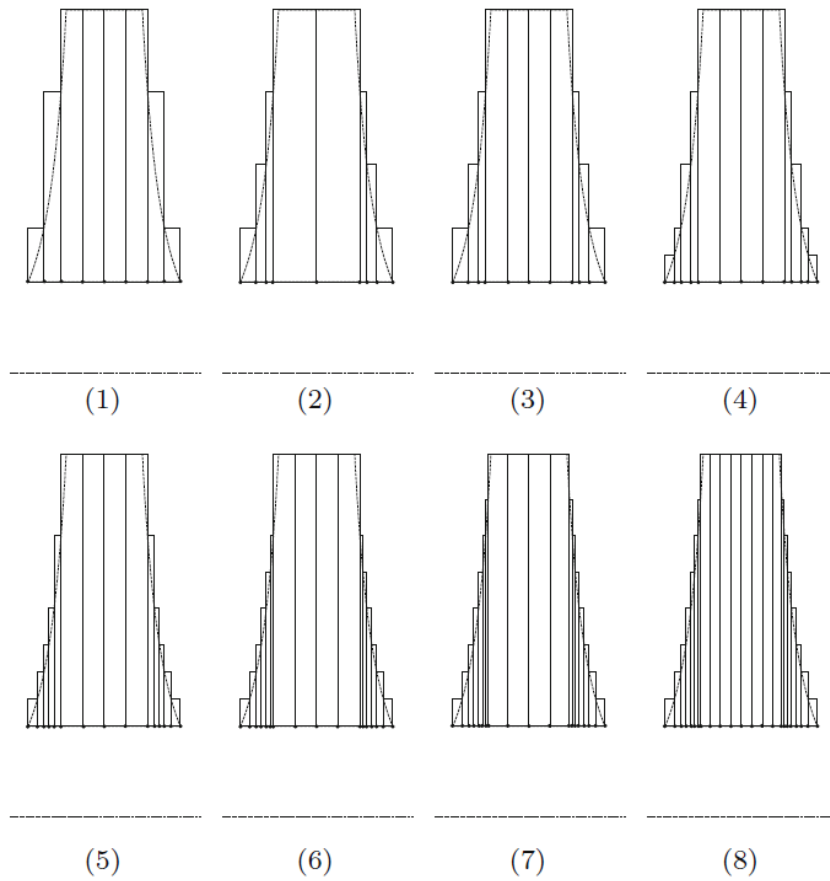


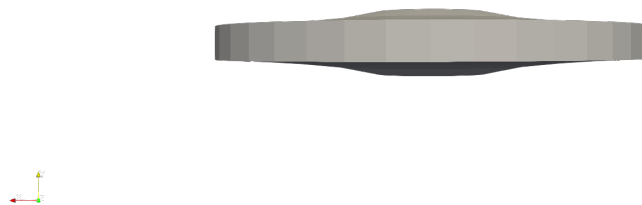
Figure 3.6.4: Proposed solution for rotor discretization by Filippi et al.



(a) First view



(b) Second view



(c) Third view

Figure 3.6.5: New proposed solution with smoother profile

Focusing on cases (1) and (8) in Fig. 3.6.4, it can be observed that the discretization in case (1) is performed using a series of 8 B2 elements with 3 different cross-sections (8 B2, 3 s). Conversely, case (8) is obtained by employing 22 B2 elements with 8 different cross-sections (22 B2, 8 s).

3.6.3.2 Numerical results

A static analysis was conducted to evaluate the displacement values across the disk. The results indicate a consistent trend, with displacement values increasing progressively with distance from the center of the disk. This behavior aligns well with theoretical expectations, confirming the model's accuracy in capturing the physical response of the structure under static loading conditions. The gradual increase in displacement towards the disk's outer regions highlights the structural characteristics and load distribution, reinforcing the validity of the chosen modeling approach.

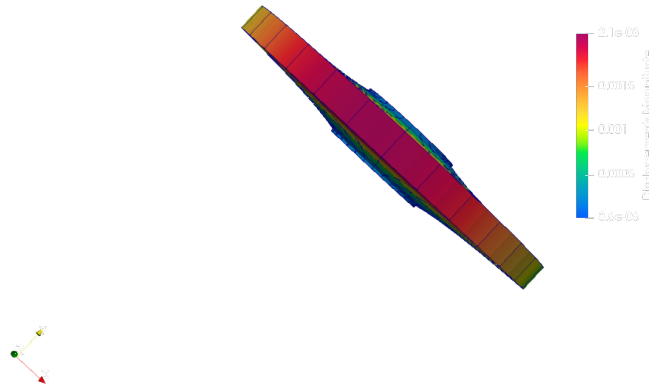


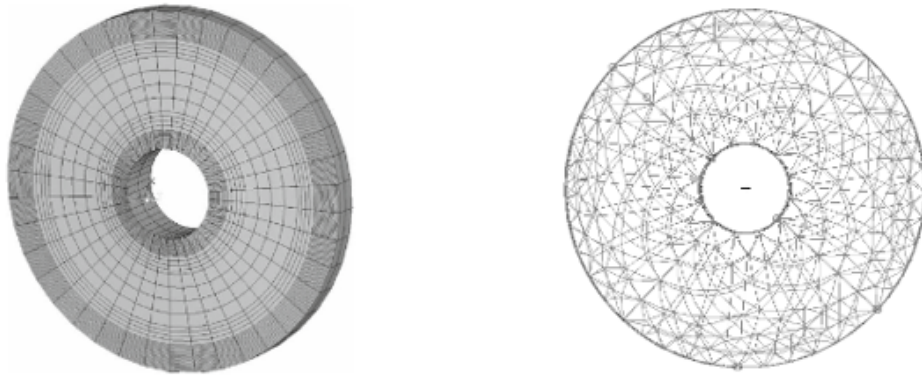
Figure 3.6.6: Static analysis - Displacements

Then, the results of the performed free-vibration analysis are illustrated. The modal analysis is conducted using the commercial software Nastran® and the academic code MUL2, developed using the theory of CUF with the exploitation of the Node-Dependent Kinematics. In both cases, the first 20 natural frequencies are considered.

A comparison of the natural frequencies obtained with MUL2 was conducted against both ANSYS and NASTRAN, performing a dual verification process. In the first comparison, ANSYS results were obtained using 3,120 8-node SOLID185

elements, leading to a total of 14,400 degrees of freedom (DOFs). The SOLID185 elements are 3D continuum elements with eight nodes, each possessing three translational degrees of freedom, making them particularly suitable for structural analysis under large deformations and nonlinear behaviors, as illustrated in Fig. 3.6.7a. In the second comparison, NASTRAN results were obtained using 2,203 10-node TET10 elements, resulting in a total of 11,934 DOFs. The TET10 elements are tetrahedral quadratic elements with ten nodes, offering improved accuracy in capturing curved geometries compared to linear tetrahedral elements. These elements are commonly employed in finite element models where complex geometries require higher-order interpolation functions, as illustrated in Fig. 3.6.7b.

The convergence with the academic code MUL2 is reached using 4xB2 elements along the axis and 90xQ9 elements for the cross-section, with a percentage error almost below 1%, as illustrated later in the chapter. From now onward, this configuration is stated as the Reference Case (RC).



(a) ANSYS - 3120 8-node elements SOLID185 (b) NASTRAN - 2203 10-node elements TET10

Figure 3.6.7: Reference meshes for comparison

As stated in the previous section, using different combinations of discretization, different problems can be analyzed. In particular, starting from the reference case RC, two other cases are investigated:

- Case 1 (C1): 1xB4 1D elements and 90xQ9 2D elements, same discretization along the cross-section but different one along the axis
- Case 2 (C2): 4xB2 1D elements and 40xQ16 2D elements, same discretization along the axis but different one along the cross-section

3.6.3.3 Model validation

The model of the disk with non-uniform thickness is analyzed, and the modal shapes of the first 20 frequencies are illustrated in Fig. 3.6.8. Among these, particular attention is given to the first two nonzero modes of both models under investigation, as they provide crucial insights into the dynamic behavior of the structure. These modes are of fundamental importance as they typically correspond to the dominant deformation patterns influencing the overall response of the system.

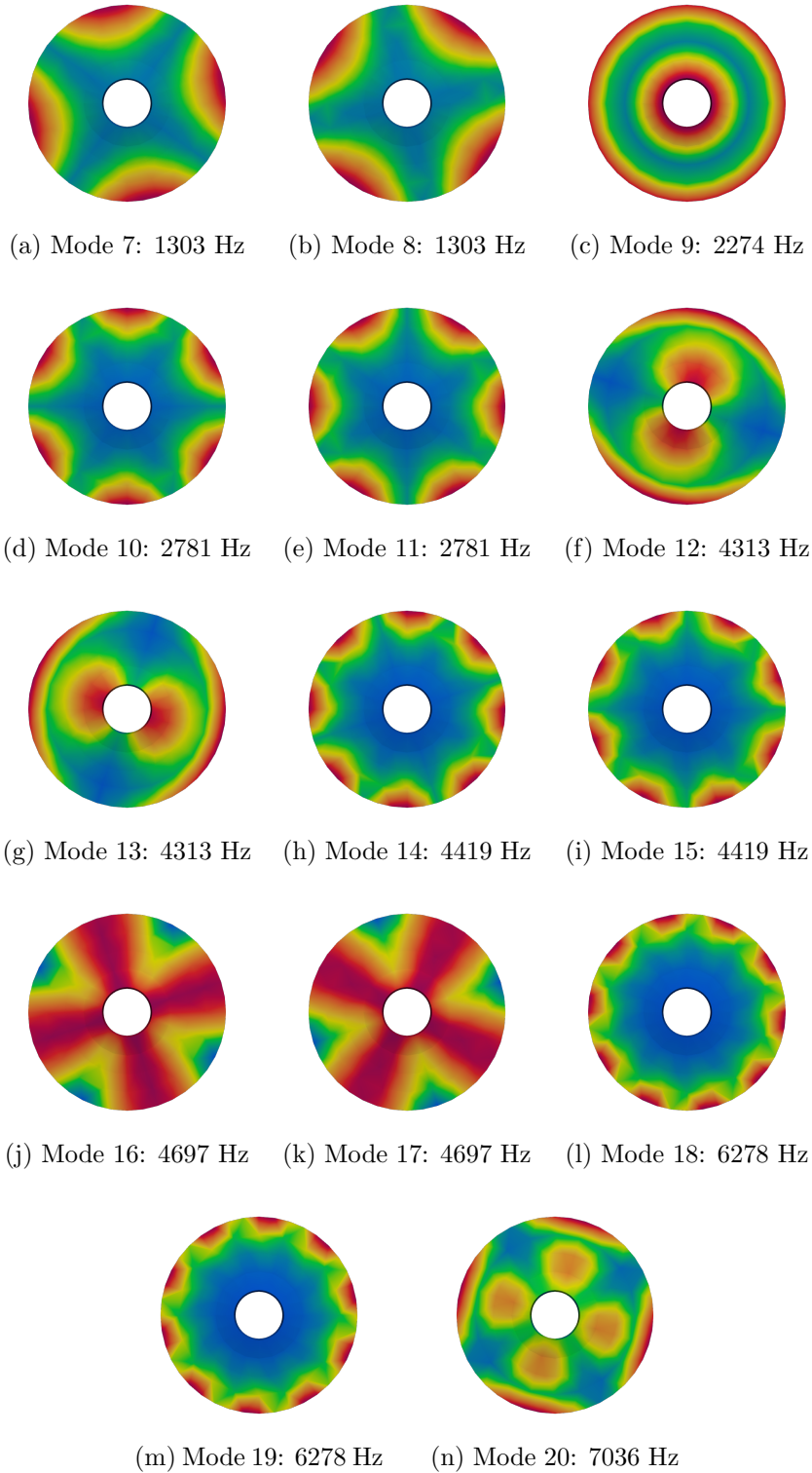


Figure 3.6.8: Non-null modal shapes

A detailed examination of these first two nonzero modal shapes is provided in Fig. 3.6.9, highlighting the differences between the two modeling approaches. Comparisons are made in terms of mode shape patterns, nodal displacement distributions, and frequency deviations to assess the accuracy and reliability of the models. The results will also be correlated with the Reference Case (RC) to evaluate the impact of the different element formulations on the computed modal characteristics.

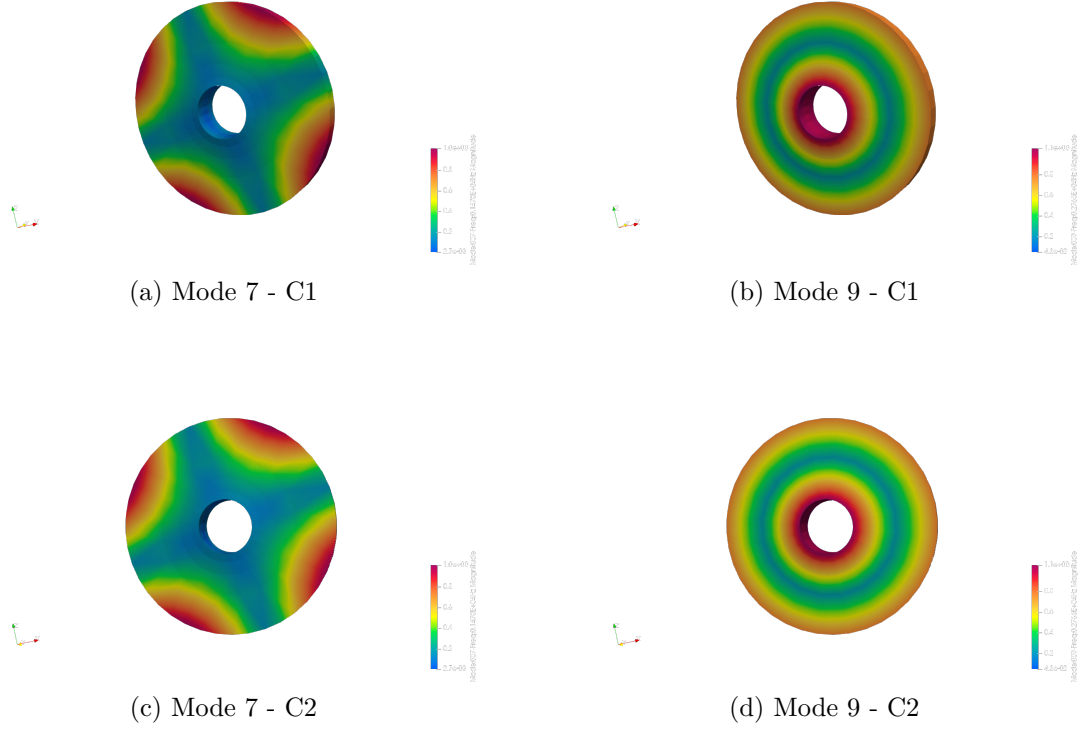


Figure 3.6.9: First 2 non-null modes for Case 1 and Case 2

The results are in agreement with those obtained with the commercial software. A proof of that is obtained computing the percentage error between the two solution in the following way:

$$err_{\%} = 100 \times \frac{|f_{n,ref} - f_n|}{f_{n,ref}} \quad (3.6.1)$$

where $f_{n,ref}$ is the natural frequency computed with Nastran®[®], representing the reference value and f_n is the natural frequency computed with MUL2. The frequencies values and the relative errors are reported in Tab. 3.6.1

Table 3.6.1: Natural frequencies and errors

Mode	$f_{Nastran} - \text{Hz}$	$f_{MUL2} - \text{Hz}$	Percentage error – %
7	1305.4	1303.0	0.1823
8	1305.8	1303.0	0.2130
9	2270.4	2274.4	0.1774
10	2780.4	2781.0	0.0232
11	2782.6	2781.0	0.0562
12	4251.5	4313.4	1.4576
13	4270.1	4313.4	1.0154
14	4428.8	4419.6	0.2073
15	4443.9	4419.6	0.5450
16	4728.7	4697.8	0.6526
17	4743.5	4697.8	0.9625
18	6328.7	6278.4	0.7937
19	6341.9	6278.5	0.9983
20	6917.9	7036.2	1.7104

Table 3.6.2: Natural frequencies and errors - Case 1 & Case 2

Mode	$f_{C1} - \text{Hz}$	$f_{C2} - \text{Hz}$	$err_{C1} - \%$	$err_{C2} - \%$
7	1307.4	1303.0	0.3362	0.0014
8	1307.4	1303.0	0.3363	0.0013
9	2252.0	2274.4	0.9861	0.0012
10	2804.0	2781.0	0.8253	0.0016
11	2803.9	2781.0	0.8221	0.0012
12	4482.8	4313.4	3.9256	0.0016
13	4483.0	4313.4	3.9304	0.0013
14	4275.9	4419.6	3.2517	0.0003
15	4275.9	4419.6	3.2531	0.0017
16	4714.5	4697.8	0.3546	0.0007
17	4714.5	4697.8	0.3545	0.0008
18	6412.6	6278.5	2.1364	0.0005
19	6412.9	6278.4	2.1392	0.0029
20	7024.8	7036.2	0.1623	0.0003

Notice that the first 6 modes are null because are related to the rigid-body motion, in fact the disk is not constrained.

Moreover, it is worth noting that the Degrees Of Freedoms (DOFs) of the Nas-tran® model is much higher with respect to those of the FEM-CUF-NDK model. An immediate comparison is appreciable in Fig. 3.6.10, where the DOFs of Nas-tran® model is 11934 and the DOFs of the proposed model is 5040.

Another comparison can be done considering the previous work by Filippi et al. in [144] and the proposed one. Referring to Fig. 3.6.4, in the extreme cases corresponding to the simplest and most complex configurations (1) and (8), the number of DOFs was 6240 and 14496, respectively. In the proposed approach, at most 5040 DOF are employed, achieving comparable results.

3.6.3.4 Comparison between different modeling

Now the analysis is focused on the design of the disk. The Reference Case is compared with Case 1 and Case 2. Notice that for all the cases the number of Degrees Of Freedoms is the same, in this way the attention is focused only on the adopted model. The frequency values and the error with respect to the Reference Case are reported in Tab. 3.6.2.

It has been already verified that in a disk problem, an increasing number of discretized elements through the radial direction and/or an higher order of interpolation in the used elements, compared to the circumferential direction, is more effective in improving the accuracy of the solution. This is evident in Tab. 3.6.2, where the error of the model related to the Case 2 is noticeably lower compared to the error of the Case 1, where the discretization is enriched in the circumferential direction.

3.6.3.5 Concluding remarks on rotor dynamics and modelling implications

During the design of rotors and rotating parts, due to the complexity of the system, the numerical model need to be the more accurate as possible. Classical numerical methods (FDM,FEM,BEM) often are not able to reproduce some particularities, such as a complex profile. In this work, using the CUF with the NDK, a feasible model is presented which is able to reproduce in an accurate manner the behaviour of the investigated part, together with a significant reduction of the DOFs. This is a starting point to study in dept the rotordynamics. Moreover, the complexity of the problem can be overcome in future works using the adaptive meshing technique suggested by [145], taking care also of the computational cost and the number of DOFs.

From an aerospace engineering standpoint, this outcome is particularly relevant, as early-stage design and iterative optimisation workflows often require rapid yet

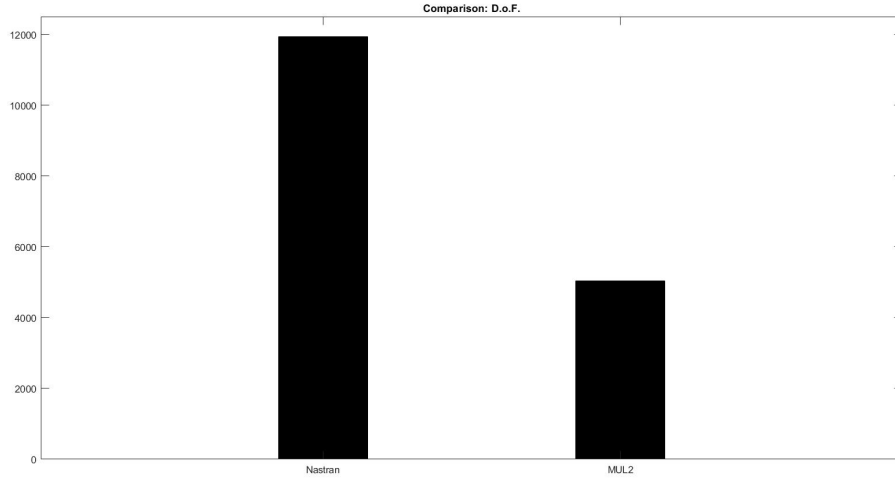


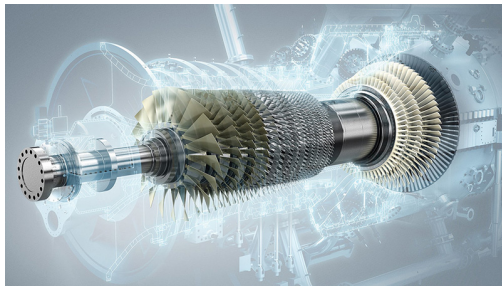
Figure 3.6.10: 3D model DOFs - 2203 TET10 vs 4xB2 90xQ9

reliable evaluations of the dynamic response. The proposed CUF–NDK approach effectively addresses these requirements by enabling:

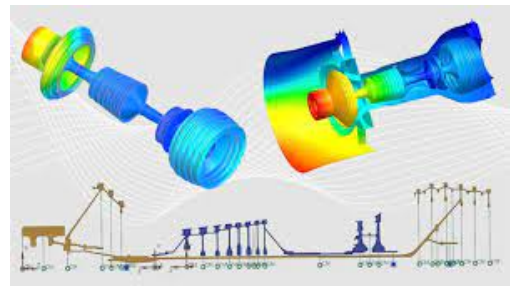
- a smooth and consistent representation of variable-thickness profiles, which are characteristic of turbine disks and rotating components in aerospace and energy applications;
- a reduced-order modelling strategy in which high accuracy is preserved in the regions governing stiffness and inertia distributions, while limiting the overall computational cost;
- a direct and flexible control of the discretization strategy along both the axial and cross-sectional directions, allowing an effective balance between numerical fidelity and efficiency.

The comparisons carried out against high-fidelity three-dimensional commercial finite element models confirm that the modal response of the structure can be accurately reproduced with a significantly reduced computational footprint. This capability is particularly advantageous for parametric studies and preliminary certification-oriented assessments, where repeated modal analyses are required within constrained computational budgets.

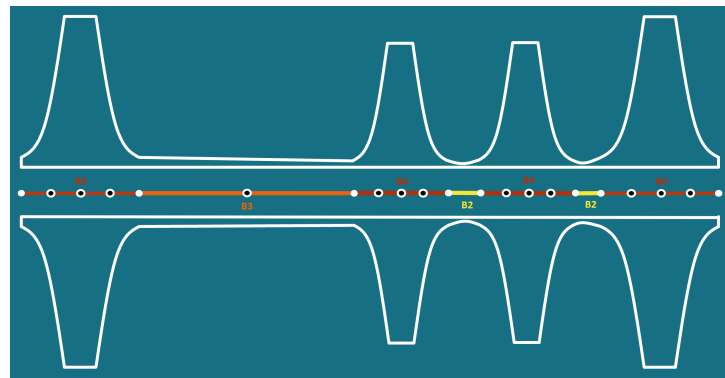
The goal is to develop a powerful tool for the early stages of design, enabling a transition from complexity to simplicity while assisting in meeting industry requirements, as represented in Fig. 3.6.11.



(a) Siemens®



(b) Siemens®



(c) Total rotor model using the suggested approach

Figure 3.6.11: From complexity to simplicity

3.7 Comparative Discussion

3.7.1 Static vs Dynamic Behaviour

The results collected in this chapter allow a cross-comparison between static and dynamic analyses under the common framework of CUF and NDK. Static benchmarks (square plate, skew plate, distorted plate) emphasize the ability of the formulation to capture displacement and stress patterns in presence of localized loads and mesh irregularities. In these cases, numerical discrepancies tend to concentrate near singular or highly localized regions (e.g., concentrated forces), whereas the global response is accurately predicted.

Dynamic benchmarks (plate and rotor free-vibration) provide a complementary perspective: natural frequencies and mode shapes depend on the global stiffness and mass distribution. The observed boundedness of modal frequencies with respect to reference families and the low sensitivity to distortion indicate that the proposed adaptive/non-conventional discretizations do not introduce non-physical stiffness artifacts. This is a key requirement for aerospace applications, where dynamic constraints may govern design margins and where accurate modal prediction is necessary for vibration qualification and rotordynamic critical speed avoidance.

3.7.2 Efficiency in Terms of Accuracy and Degrees of Freedom

A recurring theme across all cases is the trade-off between accuracy and computational cost. In the plate benchmarks, reduced meshes achieved significant DOFs savings (e.g., 42.77% for the square plate and 38.76% for the skew plate) while maintaining displacement and stress fields comparable to those of the full mesh. In the distorted mesh investigations, solutions remained bounded between reference discretizations, highlighting robustness and practicality in non-ideal meshing conditions.

In the rotor benchmark, the CUF-based discretization achieved agreement with commercial 3D models while employing substantially fewer DOFs (5040 versus 11934 in Nastran®, and even larger DOFs in ANSYS). The comparison between C1 and C2 further indicates that, for variable-thickness disks, discretization enrichment in the radial direction and higher-order cross-sectional interpolation lead to improved accuracy more efficiently than increasing the circumferential discretization.

Overall, these outcomes support the use of CUF/NDK adaptive strategies as a feasible modelling approach for aerospace structural applications requiring iterative analyses, parametric studies, or reduced-order modelling while retaining

fidelity in critical regions.

3.8 Concluding Remarks on the Numerical Results

During the design of aerospace structures and rotating components, due to the complexity of the system, the numerical model need to be the more accurate as possible. Classical numerical methods often are not able to reproduce some particularities, such as a complex profile, without resorting to heavy 3D discretizations and high computational costs. In this chapter, CUF-based models exploiting Node-Dependent Kinematics have been assessed on a set of static and dynamic problems, showing that refined kinematic descriptions and non-conventional element couplings can provide accurate and robust results with a reduced number of DOFs.

The static investigations carried out in this work confirmed the following key aspects:

- CUF-based plate elements are able to accurately reproduce converged three-dimensional reference solutions obtained using high-fidelity commercial finite element models;
- reduced meshes combining plate and beam elements can achieve substantial reductions in the number of Degrees Of Freedom, while preserving the accuracy of the structural response in terms of displacements and stresses;
- highly distorted meshes, including non-standard element shapes, can be handled in a robust manner, with results remaining bounded with respect to the reference solution families, provided that element validity conditions are preserved.

The dynamic investigations led to the following conclusions:

- modal trends remain stable under mesh distortion, including both convex and non-convex configurations, indicating a low sensitivity of the proposed adaptive elements to geometric irregularities;
- natural frequencies remain bounded and do not exhibit spurious stiffening or softening effects induced by the adaptive discretization strategy;
- variable-thickness rotor disks can be accurately modelled using CUF-based formulations with a significantly reduced number of Degrees Of Freedom, and the discretization strategy can be guided by clear physical considerations, such as the radial distribution of stiffness and inertia.

These results represent a consistent basis for extending the proposed approach to more complex aerospace configurations, including shell structures and curvilinear geometries and for integrating additional numerical treatments (e.g. mixed formulations) where required to mitigate locking or improve robustness. The chapter therefore provides both verification evidence and engineering-level interpretation supporting the adoption of CUF/NDK adaptive strategies in aerospace structural modelling.

3.8.1 Additional exploratory investigations

In addition to the numerical results presented in this chapter, several further study cases were investigated during the development of this research activity. These analyses are not reported in detail, as they require additional validation steps and more extensive parametric investigations to ensure the same level of robustness achieved for the results discussed in the previous sections. Nevertheless, they are briefly outlined here to provide a complete picture of the scope of the work and to contextualize potential extensions of the proposed methodologies.

A first class of additional investigations concerns L-shaped structures, which are widely recognized in computational structural mechanics as benchmark configurations. L-shaped plates and shells are characterized by the presence of re-entrant corners, which induce localized stress and strain concentrations and represent a critical challenge for standard finite element discretizations. From an aerospace engineering perspective, such geometries are highly relevant, as they idealize structural junctions commonly found in brackets, stiffener–skin connections, wing–fuselage interfaces, and load-carrying fittings. Preliminary static analyses were performed on L-shaped plates subjected to concentrated loads applied at the free end of one arm, with the objective of assessing the capability of CUF-based adaptive elements to capture displacement fields and stiffness redistribution in the presence of geometric singularities. Different thickness configurations were explored, varying both the in-plane thickness of the arms and the out-of-plane thickness along the transverse direction, in order to investigate bending-dominated and shear-influenced regimes. Although the initial results confirmed physically consistent trends, a systematic mesh-convergence study and a dedicated comparison with reference solutions are required before these results can be presented as fully validated benchmarks.

A second line of exploratory activity focused on the potential extension of the proposed adaptive CUF framework to fluid–structure interaction (FSI) problems. In this context, the emphasis was placed on developing a progressive modeling strategy, starting from simplified aerodynamic representations and gradually increasing the level of fidelity of the fluid solver. Initial investigations considered low-order aerodynamic models, such as vortex-lattice methods (VLM), coupled

with refined structural models formulated within the CUF and Node-Dependent Kinematics framework. This approach is particularly appealing in preliminary aerospace design phases, where rapid evaluations of aeroelastic behavior are required with limited computational cost. The long-term objective of this line of research is to enable a seamless transition from low-fidelity aerodynamic models to more advanced formulations, including unsteady panel methods or Navier–Stokes-based solvers, while preserving the flexibility and efficiency of the structural discretization. However, the development of stable and reliable coupling strategies, as well as the validation against canonical aeroelastic benchmarks, remains an essential prerequisite before such results can be fully documented.

Overall, these additional investigations highlight the versatility of the proposed numerical framework and its potential applicability to a broader class of aerospace-relevant problems. At the same time, they reflect a deliberate methodological choice adopted throughout this thesis, namely to present only those results that have undergone thorough verification and validation, while clearly identifying ongoing developments and exploratory studies as a basis for future, more comprehensive analyses.

Chapter 4

Conclusions

4.1 Summary

This study has explored the application of refined finite element methodologies based on the Carrera Unified Formulation (CUF) to address computational challenges in structural mechanics, particularly in cases where traditional Finite Element Method (FEM) approaches become computationally expensive due to the necessity of fine 3D meshes. The work has demonstrated the effectiveness of novel 1D and 2D adaptive elements in optimizing mesh refinement, reducing the number of degrees of freedom (DOFs) while maintaining high accuracy in the analysis of displacements and stresses.

The implementation of the Node-Dependent Kinematics (NDK) approach within CUF has allowed for an advanced adaptation of mesh refinement strategies. By utilizing Lagrange expansion functions, the presented methodology enables the localization of fine mesh regions where critical conditions, such as geometric discontinuities or load constraints, occur. This approach significantly reduces computational costs compared to conventional 3D FEM, making it particularly advantageous in structural applications requiring efficiency without compromising accuracy. Several numerical study cases have validated the effectiveness of these elements, confirming their ability to provide reliable results with a reduced computational burden.

Beyond standard FEM discretization techniques, this research has introduced Adaptive Elements, which offer a more flexible and efficient meshing approach. The ability to handle arbitrary polygonal elements, both concave and convex, extends the adaptability of FEM-based models. The capacity of these elements to accommodate non-conforming discretizations allows for localized refinement with-

out imposing constraints on element continuity. The implementation of patch tests and numerical studies with highly distorted meshes has demonstrated the robustness of this technique, confirming its potential for practical engineering applications.

Furthermore, the study has extended the application of refined FEM formulations to the dynamic analysis of rotating disks with variable thickness, a critical aspect in the design of turbomachinery components such as turbojet engines and steam turbines. The CUF-based beam model, combined with the NDK approach, has been effectively employed to analyze vibration characteristics in such structures. The introduction of 1D beam elements, enhanced by Lagrange-like polynomial expansions, has provided an accurate representation of the displacement field across the rotor cross-section, while allowing for smooth discretization of the variable thickness distribution. Numerical results obtained using Hamilton's Principle and FEM computations have demonstrated the feasibility of this approach, establishing a solid foundation for future investigations in rotordynamics.

The findings presented in this research pave the way for further advancements in computational structural mechanics. Future developments may focus on integrating additional physical effects, such as geometric stiffening, spin softening, and Coriolis contributions, into the CUF framework for rotordynamic applications. Moreover, the implementation of higher-order interpolation schemes and multi-domain modeling strategies could further enhance the efficiency and accuracy of adaptive elements. The potential applications of these techniques extend beyond aerospace and mechanical engineering, encompassing a wide range of fields where structural optimization and computational efficiency are of paramount importance.

In conclusion, this work has demonstrated that the Carrera Unified Formulation, in combination with the Node-Dependent Kinematics approach and refined finite elements, provides a powerful and flexible computational tool for structural analysis. By significantly improving the balance between accuracy and computational cost, the proposed methodologies represent a valuable contribution to modern finite element modeling, enabling more efficient and accurate simulations of complex engineering structures.

4.1.1 Key Insights and Contributions

The primary objective of this research was to enhance simulation capabilities by leveraging adaptive meshes to address complex challenges in aerospace engineering. By implementing advanced meshing techniques and refined finite element methodologies, this work has demonstrated significant improvements in both ac-

curacy and computational efficiency.

One of the most impactful benefits of the proposed approach is its efficiency. The use of adaptive mesh refinement allows for high-fidelity structural analysis while maintaining a manageable computational cost. This ensures that accurate results can be obtained without the excessive burden of traditional 3D finite element models. Additionally, the safety of aerospace components is significantly improved, as the refined mesh strategies enable a more precise capture of critical stress distributions and structural behaviors, ultimately enhancing the reliability of the designs. Furthermore, this work fosters innovation by introducing methodologies that align with stringent industry performance requirements, paving the way for more optimized and robust aerospace structures.

The overall impact of this research extends beyond computational gains. By pushing the boundaries of simulation techniques and refining numerical modeling approaches, the proposed framework contributes to the development of safer and more advanced aerospace technologies. The insights gained from this study can be instrumental in optimizing structural designs, reducing material waste, and improving the overall performance of next-generation aerospace systems.

4.1.2 Future works

The methodologies and numerical approaches developed in this research provide a strong foundation for further advancements in computational structural mechanics. Several directions can be explored to enhance the applicability of the proposed techniques in aerospace engineering, addressing both structural complexity and multidisciplinary interactions.

One promising avenue is the extension of these methods to aircraft structures, particularly in the modeling of fuselage, wings, and wing-fuselage connections. These components exhibit intricate geometries and require highly accurate numerical simulations to predict stress distributions, dynamic behavior, and potential failure modes. The refined finite element strategies presented in this work can be adapted to optimize the structural integrity of such systems while maintaining computational efficiency.

Another crucial application concerns landing gear systems, which represent one of the most heavily loaded components in aerospace structures. Due to their complex mechanical interactions and significant impact loads during landing, advanced modeling techniques are required to capture both structural deformations and contact forces. The integration of adaptive mesh refinement and high-fidelity

finite elements can provide an improved understanding of the stress states and fatigue life of these components.

The proposed methodologies also hold great potential for spacecraft applications, where lightweight and high-strength structures must be designed to withstand extreme launch and orbital conditions. The ability to simulate precise load distributions across multi-component spacecraft structures is essential for ensuring their reliability. Future work may involve refining numerical models for deployable structures, thermal stress effects, and micro-vibration damping mechanisms.

In the context of composite materials, the formulation developed in this study can be extended to account for their intrinsic heterogeneity. Composite structures often exhibit complex stress-strain relationships due to fiber orientation, matrix properties, and interface interactions. The introduction of advanced interpolation methods, layer-wise modeling, and multi-scale approaches would allow for an even more accurate representation of these materials, further improving predictive capabilities.

A particularly relevant research direction is the investigation of fluid-structure interaction (FSI) phenomena, focusing on the flutter and structural response of aerospace components under varying flow conditions. Flutter, a dynamic instability caused by the interaction between aerodynamic forces and structural elasticity, poses significant challenges in the design of high-performance aircraft and spacecraft structures. Future developments will involve coupling CUF-based structural models with aerodynamic solvers to capture these interactions more accurately.

References

- [1] Thomas Henry Gordon Megson. *Structural and stress analysis*. Butterworth-Heinemann, 2019.
- [2] DL Raffle. Analysis and design of flight vehicle structures. professor ef bruhn. tri-state offset co., cincinnati, ohio, usa. 1965. approx 500 pp. diagrams. tables. \$18.75. *The Aeronautical Journal*, 69(654):419–420, 1965.
- [3] Alexander Tessler, H Ronald Riggs, Colin E Freese, and Geoffrey M Cook. An improved variational method for finite element stress recovery and a posteriori error estimation. *Computer methods in applied mechanics and engineering*, 155(1-2):15–30, 1998.
- [4] Alexander Tessler, HR Riggs, and M Dambach. A novel four-node quadrilateral smoothing element for stress enhancement and error estimation. *International journal for numerical methods in engineering*, 44(10):1527–1543, 1999.
- [5] Alexander Tessler and Jan L Spangler. A least-squares variational method for full-field reconstruction of elastic deformations in shear-deformable plates and shells. *Computer methods in applied mechanics and engineering*, 194(2-5):327–339, 2005.
- [6] Cuong C Quach and Sixto L Vazquez. Comparison of fiber optic strain demodulation implementations. Technical report, 2005.
- [7] Wiesław M Ostachowicz, Marek Krawczuk, Karen M Holford, and Janice M Dulieu-Barton. Damage assessment of structures vi. 2005.
- [8] Klaus-Jürgen Bathe. *Finite element procedures*. Klaus-Jurgen Bathe, 2006.
- [9] M Beckes. *Zeitschrift für Angewandte Mathematik und Mechanik*. De Gruyter, 1981.

- [10] Richard Courant et al. Variational methods for the solution of problems of equilibrium and vibrations. *Lecture notes in pure and applied mathematics*, pages 1–1, 1994.
- [11] Alexander Hrennikoff. Solution of problems of elasticity by the framework method. 1941.
- [12] Douglas McHenry. A lattice analogy for the solution of stress problems. *Journal of the institution of civil engineers*, 21(2):59–82, 1943.
- [13] M Jon Turner, Ray W Clough, Harold C Martin, and LJ Topp. Stiffness and deflection analysis of complex structures. *journal of the Aeronautical Sciences*, 23(9):805–823, 1956.
- [14] RW Clough. The finite element in plane stress analysis “proc. In *ASCE Conf. on electronic computation, Pitusburg, Pa*, 1960.
- [15] Singiresu S Rao. *The finite element method in engineering*. Elsevier, 2010.
- [16] Singiresu S Rao and James P Sawyer. Fuzzy finite element approach for analysis of imprecisely defined systems. *AIAA journal*, 33(12):2364–2370, 1995.
- [17] Lewis Fry Richardson. Ix. the approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a masonry dam. *Philosophical Transactions of the Royal Society of London. Series A, containing papers of a mathematical or physical character*, 210(459-470):307–357, 1911.
- [18] Tamara Dembo, Kurt Beck, Harold Kelley, Albert Petione, Alex Bavelas, Morton Deutsch, Leon Festinger, Stanley Schachter, and Dorwin Cartwright. Lewin, kurt lewin, kurt dr. phil. dr. phil. dr. phil. prof., gestaltpsychologe prof., gestaltpsychologe prof., gestaltpsychologe.
- [19] OC Zienkiewicz. Taylorr. l.: The finite element method, 5-th edition, 2000.
- [20] Darrell W Pepper and Juan C Heinrich. *The finite element method: basic concepts and applications*. Taylor & Francis, 2005.
- [21] M Van Dyke. Perturbation methods in fluid mechanics academic press. *New York*, page 110, 1964.
- [22] B Ray, G Biswas, and A Sharma. Generation of secondary droplets in coalescence of a drop at a liquid–liquid interface. *Journal of Fluid Mechanics*, 655:72–104, 2010.

- [23] Olek C Zienkiewicz and Robert L Taylor. *The finite element method for solid and structural mechanics*. Elsevier, 2005.
- [24] Thomas JR Hughes. *The finite element method: linear static and dynamic finite element analysis*. Courier Corporation, 2003.
- [25] Alson E Hatheway. Review of finite element analysis techniques: capabilities and limitations. *Optomechanical Design: A Critical Review*, 10265:335–365, 1992.
- [26] Arturo O Cifuentes, Timothy L Bock, and Robert N Coppolino. A finite element technique for tape-head interaction problems in high-speed recording: The steady-state case. *Applied mathematical modelling*, 14(12):661–665, 1990.
- [27] Victor Genberg. Optical pathlength calculations via finite elements. In *Structural Mechanics of Optical Systems II*, volume 748, pages 81–88. SPIE, 1987.
- [28] Alson E Hatheway. Finite element methods for evaluating optical system performance. In *Optical Systems Engineering IV*, volume 518, pages 145–149. SPIE, 1985.
- [29] Alson E Hatheway. Evaluating stresses in adhesive bond lines. In *Proc. MSC/NASTRAN Users’ Conference, Universal City*, 1989.
- [30] Ernest B Paxson Jr. Simulation of room temperature control system (rtcs) with msc/nastran. In *The 1988MSC World Users’ Conference Proceedings, The MacNeal-Schwendler Corporation, Los Angeles, CA*, 1988.
- [31] E Carrera, A Pagani, and D Scano. Legendre-based node-dependent kinematics shell models for the global–local analysis of homogeneous and layered structures. *International Journal of Solids and Structures*, 289:112630, 2024.
- [32] Erasmo Carrera, Daniele Scano, and Enrico Zappino. Plate finite elements with arbitrary displacement fields along the thickness. *Finite Elements in Analysis and Design*, 244:104296, 2025.
- [33] Daniele Scano. Analysis of composite beams, plates, and shells using jacobi polynomials and ndk models. *Materials Research Proceedings*, 33, 2023.
- [34] Alfonso Pagani, E Carrera, R Augello, and D Scano. Use of lagrange polynomials to build refined theories for laminated beams, plates and shells. *Composite Structures*, 276:114505, 2021.

- [35] Ivo Babuška and Werner C Rheinboldt. A-posteriori error estimates for the finite element method. *International journal for numerical methods in engineering*, 12(10):1597–1615, 1978.
- [36] Thomas Grätsch and Klaus-Jürgen Bathe. A posteriori error estimation techniques in practical finite element analysis. *Computers & structures*, 83(4-5):235–265, 2005.
- [37] Catarina Patoilo Teixeira, Cláudia Oliveira Fernandes, and Jack Ahern. Adaptive planning design and management framework for urban climate change adaptation and mitigation. *Urban Forestry & Urban Greening*, 70:127548, 2022.
- [38] Erasmo Carrera, Maria Cinefra, Marco Petrolo, and Enrico Zappino. *Finite element analysis of structures through unified formulation*. John Wiley & Sons, 2014.
- [39] Maria Cinefra, Alberto Garcia de Miguel, Matteo Filippi, Caroline Houriet, Alfonso Pagani, and Erasmo Carrera. Homogenization and free-vibration analysis of elastic metamaterial plates by carrera unified formulation finite elements. *Mechanics of Advanced Materials and Structures*, 28(5):476–485, 2021.
- [40] M Cinefra, G D’Amico, AG De Miguel, M Filippi, A Pagani, and E Carrera. Efficient numerical evaluation of transmission loss in homogenized acoustic metamaterials for aeronautical application. *Applied Acoustics*, 164:107253, 2020.
- [41] Leonhard Euler. Additamentum i: De curvas elasticis. *Methodus inveniendi lineas curvas maximi minimive proprietate gaudentes [...]*, pages 245–310, 1744.
- [42] Stephan P Timoshenko. X. on the transverse vibrations of bars of uniform cross-section. *The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science*, 43(253):125–131, 1922.
- [43] Gustav Kirchhoff. Über das gleichgewicht und die bewegung einer elastischen scheibe. *Journal für die reine und angewandte Mathematik (Crelles Journal)*, 1850(40):51–88, 1850.
- [44] RD0044 Mindlin. Influence of rotatory inertia and shear on flexural motions of isotropic, elastic plates. 1951.

- [45] Eric Reissner. The effect of transverse shear deformation on the bending of elastic plates. 1945.
- [46] Junuthula N Reddy. A simple higher-order theory for laminated composite plates. 1984.
- [47] Junuthula Narasimha Reddy. *Mechanics of laminated composite plates and shells: theory and analysis*. CRC press, 2003.
- [48] Rakesh K Kapania and Stefano Raciti. Recent advances in analysis of laminated beams and plates. part i-sheareffects and buckling. *AIAA journal*, 27(7):923–935, 1989.
- [49] E19178181062 Carrera. Theories and finite elements for multilayered, anisotropic, composite plates and shells. *Archives of computational methods in engineering*, 9:87–140, 2002.
- [50] Erasmo Carrera. Theories and finite elements for multilayered anisotropic, composite plates and shells: a unified compact formulation with numerical assessment and benchmarking. *Archives of Computational Methods in Engineering*, 10(3):216–296, 2003.
- [51] Maria Cinefra, Stefano Valvano, and Erasmo Carrera. A layer-wise mitc9 finite element for the free-vibration analysis of plates with piezo-patches. *International Journal of Smart and Nano Materials*, 6(2):85–104, 2015.
- [52] Alfonso Pagani, Stefano Valvano, and Erasmo Carrera. Analysis of laminated composites and sandwich structures by variable-kinematic mitc9 plate elements. *Journal of Sandwich Structures & Materials*, 20(1):4–41, 2018.
- [53] Lorenzo Dozio. Natural frequencies of sandwich plates with fgm core via variable-kinematic 2-d ritz models. *Composite Structures*, 96:561–568, 2013.
- [54] Michele D’Ottavio, Philippe Vidal, Emmanuel Valot, and Olivier Polit. Assessment of plate theories for free-edge effects. *Composites Part B: Engineering*, 48:111–121, 2013.
- [55] Erasmo Carrera and Gaetano Giunta. Refined beam theories based on a unified formulation. *International Journal of Applied Mechanics*, 2(01):117–143, 2010.
- [56] Erasmo Carrera, Marco Petrolo, and Enrico Zappino. Performance of cuf approach to analyze the structural behavior of slender bodies. *Journal of Structural Engineering*, 138(2):285–297, 2012.

- [57] Erasmo Carrera, Alfonso Pagani, and Marco Petrolo. Use of lagrange multipliers to combine 1d variable kinematic finite elements. *Computers & Structures*, 129:194–206, 2013.
- [58] Erasmo Carrera, EM Filippi, and Enrico Zappino. Analysis of rotor dynamic by one-dimensional variable kinematic theories. *Journal of engineering for gas turbines and power*, 135(9), 2013.
- [59] Erasmo Carrera and Alfonso Pagani. Evaluation of the accuracy of classical beam fe models via locking-free hierarchically refined elements. *International Journal of Mechanical Sciences*, 100:169–179, 2015.
- [60] Erasmo Carrera and Marco Petrolo. Refined beam elements with only displacement variables and plate/shell capabilities. *Meccanica*, 47:537–556, 2012.
- [61] Erasmo Carrera, Maria Maiaru, and Marco Petrolo. Component-wise analysis of laminated anisotropic composites. *International Journal of Solids and Structures*, 49(13):1839–1851, 2012.
- [62] E Carrera, A Pagani, and M Petrolo. Classical, refined and component-wise theories for static analysis of reinforced-shell wing structures. 2012. *To be Submitted*.
- [63] Erasmo Carrera, Alfonso Pagani, and Marco Petrolo. Refined 1d finite elements for the analysis of secondary, primary, and complete civil engineering structures. *Journal of Structural Engineering*, 141(4):04014123, 2015.
- [64] Erasmo Carrera and Enrico Zappino. One-dimensional finite element formulation with node-dependent kinematics. *Computers & Structures*, 192:114–125, 2017.
- [65] Erasmo Carrera, Alfonso Pagani, and Stefano Valvano. Multilayered plate elements accounting for refined theories and node-dependent kinematics. *Composites Part B: Engineering*, 114:189–210, 2017.
- [66] EZGLE Carrera, Enrico Zappino, and Guohong Li. Finite element models with node-dependent kinematics for the analysis of composite beam structures. *Composites Part B: Engineering*, 132:35–48, 2018.
- [67] Enrico Zappino, Guohong Li, Alfonso Pagani, and Erasmo Carrera. Global-local analysis of laminated plates by node-dependent kinematic finite elements with variable esl/lw capabilities. *Composite Structures*, 172:1–14, 2017.

- [68] M Cinefra. Non-conventional 1d and 2d finite elements based on cuf for the analysis of non-orthogonal geometries. *European Journal of Mechanics-A/Solids*, 88:104273, 2021.
- [69] M. Cinefra, G. D’Amico, A. De Miguel Garcia, M. Filippi, A. Pagani, and E. Carrera. Efficient numerical evaluation of transmission loss in homogenized acoustic metamaterials for aeronautical application. *Applied Acoustics*, 164:107253, 2020.
- [70] John Bell, Marsha Berger, Jeff Saltzman, and Mike Welcome. Three-dimensional adaptive mesh refinement for hyperbolic conservation laws. *SIAM Journal on Scientific Computing*, 15(1):127–138, 1994.
- [71] Tommaso Sorgente, Daniele Prada, Daniela Cabiddu, Silvia Biasotti, Giuseppe Patané, Micol Pennacchio, Silvia Bertoluzza, Gianmarco Manzini, and Michela Spagnuolo. Vem and the mesh. *arXiv preprint arXiv:2103.01614*, 2021.
- [72] Arun L Gain, Cameron Talischi, and Glaucio H Paulino. On the virtual element method for three-dimensional linear elasticity problems on arbitrary polyhedral meshes. *Computer Methods in Applied Mechanics and Engineering*, 282:132–160, 2014.
- [73] Stefan A Funken and Anja Schmidt. Adaptive mesh refinement in 2d—an efficient implementation in matlab. *Computational methods in applied mathematics*, 20(3):459–479, 2020.
- [74] Randolph E Bank and Andrew H Sherman. An adaptive, multi-level method for elliptic boundary value problems. *Computing*, 26(2):91–105, 1981.
- [75] Edward Granville Sewell. *Automatic generation of triangulations for piecewise polynomial approximation*. Purdue University, 1972.
- [76] Carsten Carstensen, Michael Feischl, Marcus Page, and Dirk Praetorius. Axioms of adaptivity. *Computers & Mathematics with Applications*, 67(6):1195–1253, 2014.
- [77] Rüdiger Verfürth. A review of a posteriori error estimation and adaptive mesh-refinement techniques. *(No Title)*, 1996.
- [78] Leif Kobbelt. Interpolatory subdivision on open quadrilateral nets with arbitrary topology. 15(3):409–420, 1996.

- [79] Long Chen and Chensong Zhang. A coarsening algorithm on adaptive grids by newest vertex bisection and its applications. *Journal of Computational Mathematics*, pages 767–789, 2010.
- [80] M Ortiz and JJ Quigley Iv. Adaptive mesh refinement in strain localization problems. *Computer Methods in Applied Mechanics and Engineering*, 90(1-3):781–804, 1991.
- [81] SW Sloan. A fast algorithm for constructing delaunay triangulations in the plane. *Advances in Engineering Software (1978)*, 9(1):34–55, 1987.
- [82] Viggo Tvergaard, A. Needleman, and Koon Kai Lo. Flow localization in the plane strain tensile test. *Journal of the Mechanics and Physics of Solids*, 29(2):115–142, 1981.
- [83] Miguel A Salazar de Troya and Daniel A Tortorelli. Adaptive mesh refinement in stress-constrained topology optimization. *Structural and Multidisciplinary Optimization*, 58(6):2369–2386, 2018.
- [84] Geng Dong Cheng and Xiao Guo. ε -relaxed approach in structural topology optimization. *Structural optimization*, 13:258–266, 1997.
- [85] Matteo Bruggi. On an alternative approach to stress constraints relaxation in topology optimization. *Structural and multidisciplinary optimization*, 36:125–141, 2008.
- [86] Youn Kyu Park. *Extensions of optimal layout design using the homogenization method*. University of Michigan, 1995.
- [87] Alexandre Nana, Jean-Christophe Cuillière, and Vincent Francois. Towards adaptive topology optimization. *Advances in Engineering Software*, 100:290–307, 2016.
- [88] Kristian Ejlebjerg Jensen. Solving stress and compliance constrained volume minimization using anisotropic mesh adaptation, the method of moving asymptotes and a global p-norm. *Structural and Multidisciplinary Optimization*, 54:831–841, 2016.
- [89] Vien Minh Nguyen-Thanh, Xiaoying Zhuang, Hung Nguyen-Xuan, Timon Rabczuk, and Peter Wriggers. A virtual element method for 2d linear elastic fracture analysis. *Computer Methods in Applied Mechanics and Engineering*, 340:366–395, 2018.

- [90] Daniel van Huyssteen and BD4103807 Reddy. A virtual element method for isotropic hyperelasticity. *Computer Methods in Applied Mechanics and Engineering*, 367:113134, 2020.
- [91] Louie L Yaw. Introduction to the virtual element method for 2d elasticity. *arXiv preprint arXiv:2301.11928*, 2022.
- [92] Heng Chi, L Beirão Da Veiga, and Glaucio H Paulino. Some basic formulations of the virtual element method (vem) for finite deformations. *Computer Methods in Applied Mechanics and Engineering*, 318:148–192, 2017.
- [93] L Beirão da Veiga, Franco Brezzi, Luisa Donatella Marini, and Alessandro Russo. The hitchhiker’s guide to the virtual element method. *Mathematical models and methods in applied sciences*, 24(08):1541–1573, 2014.
- [94] M. Cinefra, A.G. de Miguel, M. Filippi, C. Houriet, A. Pagani, and E. Carrera. Homogenization and free-vibration analysis of elastic metamaterial plates by cuf finite elements. *Mechanics of Advanced Materials and Structures*, 1-10, 2019.
- [95] L. Euler. *De curvis elasticis*. Lausanne and Geneva: Bousquet, 1744.
- [96] S. P. Timoshenko. On the transverse vibrations of bars of uniform cross section. *Philosophical Magazine*, 43:125–131, 1922.
- [97] G. Kirchhoff. Uber das gleichgewicht und die bewegung einer elastischen scheibe. *Crelles Journal*, 40:51–88, 1850.
- [98] R.D. Mindlin. Influence of rotatory inertia and shear in flexural motions of isotropic elastic plates. *Journal of Applied Mechanics*, 18:28–31, 1951.
- [99] E. Reissner. The effect of transverse shear deformation on the bending of elastic plates. *Journal of Applied Mechanics*, 12:69–76, 1945.
- [100] J.N. Reddy. *Mechanics of laminated composite plates and shells: theory and analysis*. CRC Press, 2004.
- [101] J.N. Reddy. A simple higher-order theory for laminated composite plates. *Journal of Applied Mechanics*, 51(4):745–752, 1984.
- [102] K. Kapania and S. Raciti. Recent advances in analysis of laminated beams and plates, part I: Shear effects and buckling. *AIAA Journal*, 27(7):923–935, 1989.

- [103] K. Kapania and S. Raciti. Recent advances in analysis of laminated beams and plates, part II: Vibrations and wave propagation. *AIAA Journal*, 27(7):935–946, 1989.
- [104] E. Carrera. Theories and finite elements for multilayered, anisotropic, composite plates and shells. *Archives of Computational Methods in Engineering*, 9(2):87–140, 2002.
- [105] E. Carrera. Theories and finite elements for multilayered plates and shells: a unified compact formulation with numerical assessment and benchmarking. *Archives of Computational Methods in Engineering*, 10(3):216–296, 2003.
- [106] M. Cinefra, S. Valvano, and E. Carrera. A layer-wise MITC9 finite element for the free-vibration analysis of plates with piezo-patches. *International Journal of Smart and Nano Materials*, 6(2):85–104, 2015.
- [107] A. Pagani, S. Valvano, and E. Carrera. Analysis of laminated composites and sandwich structures by variable-kinematic MITC9 plate elements. *Journal of Sandwich Structures and Materials*, 2016. In Press.
- [108] L. Dozio. Natural frequencies of sandwich plates with FGM core via variable-kinematic 2-d ritz models. *Composite Structures*, 96:561–568, 2013.
- [109] M. D’Ottavio, P. Vidal, E. Valot, and O. Polit. Assessment of plate theories for free-edge effects. *Composites Part B: Engineering*, 48:111–121, 2013.
- [110] E. Carrera and G. Giunta. Refined beam theories based on carrera’s unified formulation. *International Journal of Applied Mechanics*, 2(1):117–143, 2010.
- [111] M. Petrolo E. Carrera, G. Giunta. *Beam structures: classical and advanced theories*. John Wiley and Sons, 2011.
- [112] E. Carrera, M. Petrolo, and E. Zappino. Performance of cuf approach to analyze the structural behavior of slender bodies. *Journal of Structural Engineering*, 138(2):285–297, 2012.
- [113] E. Carrera, A. Pagani, and M. Petrolo. Use of Lagrange multipliers to combine 1D variable kinematic finite elements. *Computers and Structures*, 129:194–206, 2013.
- [114] E. Carrera and A. Pagani. Analysis of reinforced and thin-walled structures by multi-line refined 1D/beam models. *International Journal of Mechanical Sciences*, 75:278–287, 2013.

- [115] E. Carrera and A. Pagani. Evaluation of the accuracy of classical beam FE models via locking-free hierarchically refined elements. *International Journal of Mechanical Sciences*, 100:169–179, 2015.
- [116] E. Carrera, A. Pagani, M. Petrolo, and E. Zappino. Recent developments on refined theories for beams with applications. *Mechanical Engineering Reviews*, 2(2):14–00298, 2015.
- [117] E. Carrera and M. Petrolo. Refined beam elements with only displacement variables and plate/shell capabilities. *Meccanica*, 47(3):537–556, 2012.
- [118] E. Carrera, M. Maiarú, and M. Petrolo. Component-wise analysis of laminated anisotropic composites. *International Journal of Solids and Structures*, 49:1839–1851, 2012.
- [119] E. Carrera, A. Pagani, and M. Petrolo. Classical, refined and component-wise theories for static analysis of reinforced-shell wing structures. *AIAA Journal*, 51(5):1255–1268, 2013.
- [120] E. Carrera, A. Pagani, and M. Petrolo. Refined 1D finite elements for the analysis of secondary, primary, and complete civil engineering structures. *Journal of Structural Engineering*, 141(4):art. no. 04014123, 2014.
- [121] E. Carrera and E. Zappino. One-dimensional finite element formulation with node-dependent kinematics. *Computers & Structures*, 192:114–125, 2017.
- [122] E. Carrera, A. Pagani, and S. Valavano. Multilayered plate elements accounting for refined theories and node-dependent kinematics. *Composites Part B: Engineering*, 114:189–210, 2017.
- [123] E. Carrera, E. Zappino, and G. Li. Finite element models with node-dependent kinematics for the analysis of composite beam structures. *Composites Part B: Engineering*, 132(Supplement C):35–48, 2018.
- [124] G. Li, E. Carrera, M. Cinefra, A.G. de Miguel, A. Pagani, and E. Zappino. An adaptable refinement approach for shell finite element models based on node-dependent kinematics. *Composites Part B: Engineering*, 210:1–19, 2019.
- [125] F. Moleiro, E. Carrera, E. Zappino, G. Li, and M. Cinefra. Layerwise mixed elements with node-dependent kinematics for global–local stress analysis of multilayered plates using high-order legendre expansions. *Computer Methods in Applied Mechanics and Engineering*, 359:112764, 2020.

- [126] E. Zappino, G. Li, A. Pagani, and E. Carrera. Global-local analysis of laminated plates by node-dependent kinematic finite elements with variable esl/lw capabilities. *Composite Structures*, 172:1–14, 2017.
- [127] M. Cinefra, S. Valvano, and E. Carrera. A layer-wise mitc9 finite element for the free-vibration analysis of plates with piezo-patches. *International Journal of Smart and Nano Materials*, 6(2):85–104, 2015.
- [128] E. Carrera, E. Zappino, and G. Li. Analysis of beams with piezo-patches by node-dependent kinematic finite element method models. *Journal of Intelligent Material Systems and Structures*, 29(7):1379–1393, 2018.
- [129] K. Washizu. *Variational Methods in Elasticity and Plasticity*. Elsevier Science & Technology, 1974.
- [130] E. Carrera, M. Cinefra, M. Petrolo, and E. Zappino. *Finite Element Analysis of Structures through Unified Formulation*. John Wiley & Sons, 2014.
- [131] E. Carrera, A. Pagani, and M. Petrolo. Component-wise method applied to vibration of wing structures. *Journal of Applied Mechanics*, 80(4):art. no. 041012 1–15, 2013.
- [132] E. Carrera and A. Pagani. Free vibration analysis of civil engineering structures by component-wise models. *Journal of Sound and Vibration*, 333(19):4597–4620, 2014.
- [133] G.M. Kulikov, A.A. Mamontov, S.V. Plotnikova, and S.A. Mamontov. Exact geometry solid-shell element based on a sampling surfaces technique for 3d stress analysis of doubly-curved composite shells. *Curved and Layered Structures*, 3(1), 2016.
- [134] G.M. Kulikov and S.V. Plotnikova. A method of solving three-dimensional problems of elasticity for laminated composite plates. *Mechanics of Composite Materials*, 48(1):15–26, 2012.
- [135] Olgierd Cecil Zienkiewicz and Robert Leroy Taylor. *The finite element method: solid mechanics*, volume 2. Butterworth-heinemann, 2000.
- [136] Maria Cinefra, M D’ottavio, O Polit, and E Carrera. Assessment of mitc plate elements based on cuf with respect to distorted meshes. *Composite structures*, 238:111962, 2020.
- [137] S. Timoshenko and S. Woinowsky-Krieger. *Theory of Plates and Shells*. McGraw-Hill, 1959.

- [138] A.W. Leissa. *Vibration of Plates*. NASA SP-160, 1969.
- [139] J.N. Reddy. *Theory and Analysis of Elastic Plates and Shells*. CRC Press, 2006.
- [140] K.F. Graff. *Wave Motion in Elastic Solids*. Dover, 1975.
- [141] E Carrera and M Filippi. A refined one-dimensional rotordynamics model with three-dimensional capabilities. *Journal of Sound and Vibration*, 366:343–356, 2016.
- [142] Erasmo Carrera, Matteo Filippi, and Enrico Zappino. Free vibration analysis of rotating structures by one-dimensional, variable kinematic theories. In *ASME International Mechanical Engineering Congress and Exposition*, volume 56383, page V009T10A021. American Society of Mechanical Engineers, 2013.
- [143] SS Manson. Determination of elastic stresses in gas-turbine disks. *NTRS - NASA Technical Reports Server*, 1947.
- [144] Ayoob Entezari, Mohammad Ali Kouchakzadeh, Erasmo Carrera, and Matteo Filippi. A refined finite element method for stress analysis of rotors and rotating disks with variable thickness. *Acta Mechanica*, 228:575–594, 2017.
- [145] M Cinefra and A Rubino. Adaptive mesh using non-conventional 1d and 2d finite elements based on cuf. *Mechanics of Advanced Materials and Structures*, pages 1–11, 2022.