

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

Mesh Generation for Solving Complex Structural Problems using Finite Element Methods

Balakrishnan Devarajan
Engineering Science and Mechanics
Virginia Tech
 Blacksburg, USA
 dbalak9@vt.edu

Abstract—Developing detailed Finite Element Models related to aerospace, automobile, civil, material science etc could be a very challenging task, especially generating a mesh that could produce accurate results. This review article provides an overview of different types of structural mechanics problems which are of interest to researchers and recent developments in meshing methodology related to each type of problems. The article also describes an automatic 2D mesh generation framework leveraging the capabilities of commercial FEA software MSC.PATRAN.

Key words: Structural Mechanics; Finite Element Analysis; Plates, Microstructures; Cracks; XFEM; Mesh generation; MSC.PATRAN/NASTRAN; Isogeometric Analysis

I. INTRODUCTION

Design of structural components are constrained optimization problems that requires analysis and constraint determination related to strength and buckling. This often requires the development of complex three-dimensional mathematical models. One of the most popular numerical techniques for solving such problems is the Finite Element Method [?], [1]. Broadly speaking, the Finite Element Method is a technique to solve ordinary and partial differential equations for which no close-form solutions exist. In this method, the continuum domain is divided into a finite number of cells which are called elements. Within each of them, the solution is approximated using a class of functions. The following sections of this article describe the method and science behind mesh generation to solve complicated structural problems and recent developments in this field.

II. LAMINATED PLATES MODELING

One of the capabilities currently available in commercial software packages like ABAQUS [2] includes 3D modeling of laminated composite structures. Knight et al. modeled a laminate composite plate with a hole at the centroid is modeled in ABAQUS [3]. One of the possible approaches for modeling a composite laminate involves representing each ply of material with an individual layer of 3D finite elements. The example in Figure 1 shows the meshing capabilities of ABAQUS for a

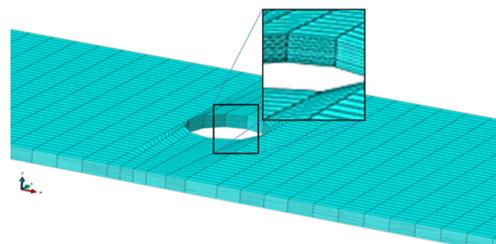


Fig. 1. Laminated Plate with Hole Model; each Ply is Modeled by one Layer of C3D8 Brick Elements

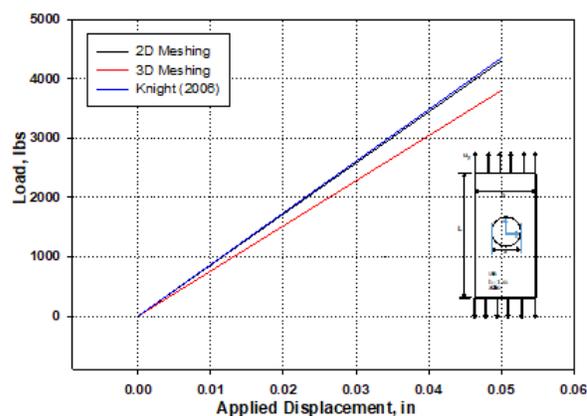


Fig. 2. Displacement vs Load Behavior of a Laminated Plate with a Hole; Different Modeling Approach Comparison

laminated plate with a hole and characterized by eight plies. Each ply of the laminate is meshed using one layer of 8-nodes C3D8 brick elements. The built-in partitioning technique in ABAQUS is used to automatically generate the mesh around the hole in the structure. It is necessary to ensure that a sufficient number of elements are generated around the hole in order for the analysis to achieve converged results.

III. DELAMINATION PROCESS MODELING

The modeling of the delamination onset and propagation process requires the use of 3D solid elements for a proper

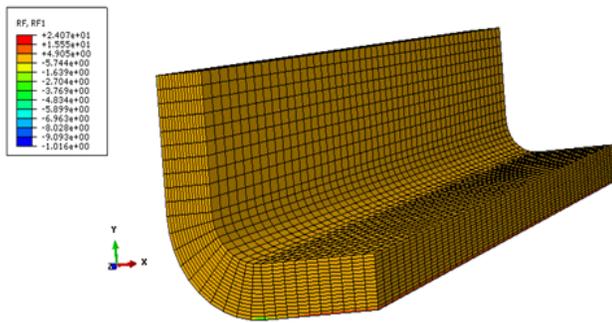


Fig. 3. Mesh of an L-Shaped Laminated Composite Structure Generated for Delamination Analysis

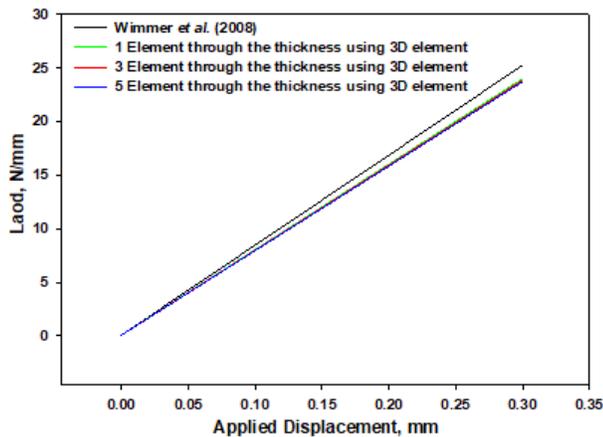


Fig. 4. Displacement vs Load Behavior of the L-shaped Composite Structure Modeled by Wimmer

analysis. The example presented in Figure 3 shows an L-shaped composite structure used by Wimmer et al. [4] for the purpose of studying the delamination process in these types of structures. The plies of this laminated structure were also modelled independently using the same techniques as were mentioned earlier. The load-displacement behavior for the L-shaped structure is shown in Figure 4.

IV. MODELING OF 3D MICROSTRUCTURES

The Voronoi Cell Finite Element method [5] is a technique for micro-mechanical modeling of materials characterized by the presence of non-uniform heterogeneous microstructures, as seen in real micrographs. The Voronoi cell FE model evolves by tessellation of the microstructure to result in morphology based network of multi-sided Voronoi polygons or polyhedrons. The Voronoi diagrams can be generated in Mathematica using the add-on package called “Computational geometry” as in Zhao et al. [6]. The geometry of each Voronoi polygon is described in terms of its vertices. These geometrical data are transferred via an interfacing computer code to a selected mesh generator, as MSC.PATRAN or CadFIX, to form points (vertices), lines (polygon boundaries, grain boundaries)

and surfaces (polygons, grains). This process is detailed by Weyer et al. [7]. Ghosh [8] developed a framework, called 3D-VCFEM, for analyzing microstructural stresses and strains in linear elastic domains containing ellipsoidal inclusions or voids.

The meshing software Neper has been used in generating Voronoi tessellations as discussed in Quey et al [9]. Neper is built around three modules of which Module -M is dedicated to microstructure meshing. The input data is a tessellation file (.tess). The mode of generation of such input files are detailed in [reference manual]. Once the input data is generated, Module-M offers capabilities like free unstructured meshing (tetrahedrons) and mapped meshing (hexahedrons) of microstructures. The output mesh can be written in several formats, including the Abaqus '.inp', the Zset '.geof', and the Gmsh '.msh' and hence can be interfaced with other commercial FEM software.

V. MODELING OF CRACKS/HOLES

Several numerical methods have been successfully employed to solve problems involving cracks and/or holes in 3D structures. These methods are the dual element boundary method [10], the natural element method [11], and the NURBS based iso-geometric element method [12].

Composites have been analyzed using numerical and experimental methods for a long time. Zhang et al. [13] presented a comprehensive literature review of developments in finite element analysis of laminated composite plates from 1990-2008. Delamination or interfacial cracking between composite layers is unarguably one of the predominant modes of failure in laminated composite and has been studied for almost the past 30 years. A. C. Garg [14] discussed at length about delamination as a damage mode in composite touching topics like causes, effects and ways to suppress delamination. V.V. Bolotin [15] presented research in the mechanics of delaminations and related crack-like defects in laminate and fiber composites considering both internal and near-surface delaminations. Liu et al. [16] presented a review on the recent developments for the methodologies in progressive failure analysis of composite laminates.

When material non-linearities are small, methods based on Linear Elastic Fracture Mechanics (LEFM) can be effective in predicting delamination growth. But LEFM requires that an initial crack be present. Some of the earlier works involved methods combining a stress analysis with a characteristic distance to predict the initiation of delamination [17], [18]. LEFM can be used to predict delamination growth after the onset of crack [19]–[21]. Techniques such as virtual crack closure technique (VCCT) [22]–[24], Virtual crack extension [25] and stiffness derivative [26] have often been used to predict delamination growth. These methods can be used to calculate

the components of the energy release rate. Cohesive zone modeling using de-cohesion elements have been presented in the work of Shahwan et al. and Goyal et al. [27], [28]. Cohesive interface elements have been applied to problems where the crack paths are not known a priori. Makhecha [29] used cohesive zone modeling method in modeling adhesive failure in bonded joint. However all the methods discussed above have their drawbacks which is that the interface fracture or the cohesive finite elements need to be placed between the plies where the delamination would be predicted to occur. In VCCT method, the calculation of fracture parameters requires nodal variables and topological information of nodes ahead and behind the crack front. Such calculations may require re-meshing as crack advances. Most of the time the location of delamination is unknown and hence there is a necessity to place such elements between all layers which significantly increases computational cost.

To resolve the issue of having a lot of cohesive zone elements at the interface, several enriched finite element formulations have been proposed, as per example by Moës et al. [30] and Remmers et al. [31]. The Extended Finite Element Method (XFEM) which allows for mesh independent representations of discontinuities (e.g. cracks and delaminations) by introducing kinematic enrichment locally in the vicinity of the propagating crack was also used to simulate delamination [32]–[35]. The major advantage of this method is that it simplifies preprocessing by lifting the requirement for the mesh to conform to the geometry of the growing delamination. The interaction between the delamination plane and the mesh is resolved during the solution step by using enrichment functions.

However, implementing the XFEM requires special integration techniques [36], computational geometry algorithms to perform the mesh-geometry interaction, pre-conditioners [37]. Its implementation [38] is therefore more involved than most standard methods.

Several researchers have worked on developing newer element formulations to simulate delamination behavior in the recent past. Eijo et al. [39] presented a numerical method based on the Refined Zigzag Theory (RZT) to model delamination in composite laminated plate/shell structures. It was an extension of the beam delamination model to plate/shell structures using the QLRZ element. Abdullah et al. [40] developed a new element which could be transformed into two physically independent 4-node shell elements. This separation into two shells is governed by a delamination criterion. Iannucci [41] proposed an interface modeling technique for explicit FE codes. The formulation is based on damage mechanics and uses only two constants for each delamination mode; firstly, a stress threshold for damage to commence, and secondly, the critical energy release rate for the particular delamination mode. The model was implemented into the commercial FE software LS-DYNA3D. Pietropaoli et al. [42] enhanced the

VCCT technique by a front-tracing algorithm and suitable expressions for the evaluation of the Strain Energy Release Rate when dealing with non-smoothed delamination fronts. Larsson et al. [43] focused on the consistent incorporation of a dis-continuum with a regularized strong discontinuity in the context of shell kinematics. Nguyen et al. [44] developed isogeometric cohesive elements for two- and three-dimensional delamination by exploiting the knot insertion algorithm directly from CAD data to generate cohesive elements along delamination. Mi et al. [45] developed a procedure, which could be incorporated within the non-linear finite element method, based on the use of interface elements in conjunction with softening relationships between the stresses and the relative displacements.

VI. MODELING SHELL-TO-SOLID COUPLING

Shell-to-solid coupling- minimizes the discontinuity displacement and generalized force field between two components, one of which is meshed using solid elements while the other meshed with shell elements. This method is widely used in global-local analysis and optimization of assemblies containing parts that require to be meshed with solid elements to capture the details and thin components for which shell elements might be good enough. The major challenges in Solid-to-shell coupling are:

- 1) The solid and the shell parts are usually meshed with elements of different element size with results in the discontinuity between nodal positions
- 2) The rotational degrees of freedom of the shell elements are needed to be transformed to the translational degrees of freedom of the connected solid elements
- 3) For each type of shell elements, the ‘directors’ (thickness direction assigned at the nodes of shell elements) are formulated differently, and the coupling is dependent on this formulation
- 4) For problems involving large rotations, the “current configuration” (after rotation) must be taken into account.

SIMULIA/Abaqus including Abaqus/Standard, Abaqus/Explicit, and Abaqus/CAE have the capability has the capability to handle solid-to-shell coupling and has the following characteristics:

- 1) The motion of the nodes located on the edge of a shell model is coupled with the motion nodes on the solid surface using an internally defined set of constraints.
- 2) It defines the region of influence on the solid surface and automatically

- 3) It automatically selects the coupling nodes located on the solid surface within a defined region of influence
- 4) Compatible for 3D stress computation
- 5) It can handle problems with misalignment of nodes of solid and shell elements
- 6) It can be used for both geometrically linear and nonlinear analysis.

For each shell node involved in the coupling, a distinct internal constraint is created with the shell node acting as the reference node and the associated solid nodes acting as the coupling nodes. The internal constraint distributes the forces and moments acting at its shell node as forces acting on the related set of coupling surface nodes in a self-equilibrating manner. ABAQUS allows the user to define the parameters: 'position tolerance' and 'influence distance' which determines the nodes on the shell-edge and nodes the solid surface that will be coupled. If the user do not define these parameters, ABAQUS takes default values for these parameters.

Position Tolerance: The parameter 'position tolerance' controls the selection of which shell nodes are involved in the shell-to-solid coupling according to how far they are from the solid surface. The solid surface can be specified as 'element-based' and 'node-based'. For 'element-based' surface the distance tolerance is defined as the projected distance along a line from the shell node to the closest point on the solid surface. On the other hand, for 'node-based' surface definition, the distance tolerance is the distance of a shell node to the closest node on the solid surface.

Influence Distance: This parameter is also known as geometric tolerance and used to control the node on the solid mesh. For a given node or element facet on the solid surface to be considered in the coupling constraint, its perpendicular distance from at least one edge facet must be less than or equal to the 'influence distance.'

VII. 2D AUTOMATIC MESH FRAMEWORK EXAMPLE

The Unitized Structure Group at Virginia Tech aerospace and ocean engineering department developed a framework for the automatic meshing of curved surfaces defined in the tri-dimensional space. In particular, this framework is applied for the rapid mesh generation of aircraft wing and fuselage, including the internal structural elements as spars, ribs, curvilinear SpaRibs, frames and stringers. In addition, the same framework is used for generating the mesh of stiffened and unstiffened quadrilateral panels characterized by generic shape.

This framework leverages the geometric and meshing capabilities of MSC.PATRAN through the generation of so-called



Fig. 5. Mesh Generation Process for surfaces in tridimensional space Capabilities

session files. A Python script generates an MSC.PATRAN session file containing all the instructions needed for generating the aircraft or panel geometry and mesh, given a set of design variables. The session file is then executed leveraging Windows or Linux shell commands and files containing the geometry, the finite element mesh and the finite element model are generated. The framework is set up to create a NASTRAN input file ready for the analysis. [46]–[53] Figure 5 shows the process briefly described above.

The capabilities of the 2D meshing framework are briefly outlined in the following:

Geometry generation: The automatic mesh generation tool developed by the unitized structure group is capable of generating the complete geometry of a generic quadrilateral panel with straight or curvilinear stiffeners. Different kind of stiffeners can be modeled, including blade stiffeners, L-shaped and T-shaped stiffeners. In addition, grid stiffened panel can be modeled and analyzed [54]. The geometry and mesh of a complex aircraft can be created along with the internal structural members. Spars and ribs can be straight or curvilinear [55], [56].

Mesh generation: This tool is able to mesh panels and wing geometry with CQUAD4 and CTRIA3 elements type. Stiffeners, spars and ribs caps and fuselage frames and stringers can be modeled using CBEAM elements. Connection between control surfaces and the main structure can be modeled with RBE2 and RBE3 elements. Concentrated masses (for engine modeling or fuel loading simulation) are modeled using CONM1 elements.

Finite element attributes: It assigns material properties, boundary conditions and load conditions can be set automatically.

Analysis supported: The tool can automatically generate MSC.NASTRAN ready to use input files for the following solution sequences: SOL101 (linear static analysis), SOL103 (modal analysis), SOL105 (buckling analysis), SOL144 (static aeroelastic analysis) and SOL145 (flutter/aeroservoelastic analysis).

Post-processing: MSC.NASTRAN analysis results can be retrieved for further computations and or visualization.

Optimization: This tool can be used as a standalone application for running one analysis or in a chain of tools in a multidisciplinary optimization framework [55], [57]. Show some of the finite element models created using the mesh generation tool.

VIII. ISOGEOMETRIC METHOD OF ANALYSIS

Even though the Finite Element Method is the most popular method of analysis, data transmission back and forth between Finite Element Analysis and Computer-aided Design is relatively expensive. Isogeometric Analysis [58] has been successful in merging these two fields in the recent past. Devarajan et al. developed Isogeometric finite element approach (IGA) in combination with the first-order deformation plate theory (FSDT) for thermal buckling analysis of laminated composite plates [59]–[66]. The IGA utilizes non-uniform rational B-spline (NURBS) as basis functions, resulting in both exact geometric representation and high order approximations. It enables to achieve quickly the smoothness with arbitrary continuous order. The analyses have been performed using Bezier extraction and conventional IGA. In traditional isogeometric analysis, the basis functions are not confined to one single element but span a global domain, whereas the Bézier extraction operator decomposes a set of linear combinations of Bernstein polynomials. The work involves B-splines, NURBS, and the concept of Bézier decomposition of these spline functions. Bézier extraction eased the implementation into an already existing finite element code.

With the advent of More Electric Aircrafts [67], solving thermal and structural problems are of utmost importance in the aerospace industry. A static thermal, structural validation problem has been developed for both constant and linear thermal temperature variation along the thickness. The influences of length-to-thickness ratio, aspect ratio, boundary conditions, stacking sequence, and material property on the critical buckling temperature have also been studied. Devarajan et al. also explain the procedures implemented for stress recovery and computing the geometric stiffness matrix. Numerical results of circular and elliptical plates will be provided to validate the effectiveness of the proposed method as compared to traditional FEA.

REFERENCES

- [1] R. K. Kapania, M. Jrad, and S. De, "Manual of ebf3glwingopt."
- [2] Abaqus, "Analysis user's guide v6. 13," 2013.
- [3] N. F. Knight Jr and J. R. Reeder, "User-defined material model for progressive failure analysis," Tech. Rep., 2006.
- [4] G. Wimmer, C. Schuecker, and H. E. Pettermann, *Numerical simulation of delamination onset and growth in laminated composites*. na, 2006.
- [5] S. Ghosh and R. Mallett, "Voronoi cell finite elements," *Computers & Structures*, vol. 50, no. 1, pp. 33–46, 1994.
- [6] J.-H. Zhao, P. Su, M. Ding, S. Chopin, and P. S. Ho, "Microstructure-based stress modeling of tin whisker growth," *IEEE transactions on electronics packaging manufacturing*, vol. 29, no. 4, pp. 265–273, 2006.
- [7] S. Weyer, A. Fröhlich, H. Riesch-Oppermann, L. Cizelj, and M. Kovac, "Automatic finite element meshing of planar voronoi tessellations," *Engineering fracture mechanics*, vol. 69, no. 8, pp. 945–958, 2002.
- [8] S. Ghosh, *Micro Mechanical Analysis and Multi-Scale Modeling*. CRC Press, Boca Raton, 2011.
- [9] R. Quey, P. Dawson, and F. Barbe, "Large-scale 3d random polycrystals for the finite element method: Generation, meshing and remeshing," *Computer Methods in Applied Mechanics and Engineering*, vol. 200, no. 17–20, pp. 1729–1745, 2011.
- [10] Y. Mi and M. Aliabadi, "Dual boundary element method for three-dimensional fracture mechanics analysis," *Engineering analysis with boundary elements*, vol. 10, no. 2, pp. 161–171, 1992.
- [11] T. J. Hughes, J. A. Cottrell, and Y. Bazilevs, "Isogeometric analysis: Cad, finite elements, nurbs, exact geometry and mesh refinement," *Computer methods in applied mechanics and engineering*, vol. 194, no. 39–41, pp. 4135–4195, 2005.
- [12] N. Sukumar, B. Moran, and T. Belytschko, "The natural element method in solid mechanics," *International journal for numerical methods in engineering*, vol. 43, no. 5, pp. 839–887, 1998.
- [13] Y. Zhang and C. Yang, "Recent developments in finite element analysis for laminated composite plates," *Composite structures*, vol. 88, no. 1, pp. 147–157, 2009.
- [14] A. C. Garg, "Delamination—a damage mode in composite structures," *Engineering Fracture Mechanics*, vol. 29, no. 5, pp. 557–584, 1988.
- [15] V. V. Bolotin, "Delaminations in composite structures: its origin, buckling, growth and stability," *Composites Part B: Engineering*, vol. 27, no. 2, pp. 129–145, 1996.
- [16] P. Liu and J. Zheng, "Recent developments on damage modeling and finite element analysis for composite laminates: A review," *Materials & Design*, vol. 31, no. 8, pp. 3825–3834, 2010.
- [17] P. Camanho and F. Matthews, "Delamination onset prediction in mechanically fastened joints in composite laminates," *Journal of Composite Materials*, vol. 33, no. 10, pp. 906–927, 1999.
- [18] C. G. Dávila and E. R. Johnson, "Analysis of delamination initiation in postbuckled dropped-ply laminates," *AIAA journal*, vol. 31, no. 4, pp. 721–727, 1993.
- [19] P. Liu and M. Islam, "A nonlinear cohesive model for mixed-mode delamination of composite laminates," *Composite Structures*, vol. 106, pp. 47–56, 2013.
- [20] Z. Zou, S. Reid, P. Soden, and S. Li, "Mode separation of energy release rate for delamination in composite laminates using sublaminates," *International Journal of Solids and Structures*, vol. 38, no. 15, pp. 2597–2613, 2001.
- [21] Z. Zou, S. Reid, S. Li, and P. Soden, "Modelling interlaminar and intralaminar damage in filament-wound pipes under quasi-static indentation," *Journal of composite materials*, vol. 36, no. 4, pp. 477–499, 2002.
- [22] E. F. Rybicki and M. F. Kanninen, "A finite element calculation of stress intensity factors by a modified crack closure integral," *Engineering fracture mechanics*, vol. 9, no. 4, pp. 931–938, 1977.
- [23] I. Raju, "Calculation of strain-energy release rates with higher order and singular finite elements," *Engineering Fracture Mechanics*, vol. 28, no. 3, pp. 251–274, 1987.
- [24] R. Krueger, "Virtual crack closure technique: History, approach, and applications," *Appl. Mech. Rev.*, vol. 57, no. 2, pp. 109–143, 2004.
- [25] T. Hellen, "On the method of virtual crack extensions," *International Journal for numerical methods in engineering*, vol. 9, no. 1, pp. 187–207, 1975.
- [26] D. M. Parks, "A stiffness derivative finite element technique for determination of crack tip stress intensity factors," *International Journal of fracture*, vol. 10, no. 4, pp. 487–502, 1974.
- [27] K. W. Shahwan and A. M. Waas, "Non-self-similar decohesion along a finite interface of unilaterally constrained delaminations," *Proceedings of the Royal Society of London. Series A: Mathematical, Physical and Engineering Sciences*, vol. 453, no. 1958, pp. 515–550, 1997.
- [28] V. Goyal, E. Johnson, C. Davila, and N. Jaunky, "An irreversible constitutive law for modeling the delamination process using interface elements," in *43rd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, 2002, p. 1576.
- [29] D. P. Makhecha, "Dynamic fracture of adhesively bonded composite structures using cohesive zone models," Ph.D. dissertation, Virginia Polytechnic Institute and State University, 2005.

- [30] N. Moës, J. Dolbow, and T. Belytschko, "A finite element method for crack growth without remeshing," *International journal for numerical methods in engineering*, vol. 46, no. 1, pp. 131–150, 1999.
- [31] J. J. Remmers, G. N. Wells, and R. d. Borst, "A solid-like shell element allowing for arbitrary delaminations," *International Journal for Numerical Methods in Engineering*, vol. 58, no. 13, pp. 2013–2040, 2003.
- [32] T. Nagashima and H. Suemasu, "Stress analyses of composite laminate with delamination using x-fem," *International Journal of Computational Methods*, vol. 3, no. 04, pp. 521–543, 2006.
- [33] J. C. Sosa and N. Karapurath, "Delamination modelling of glare using the extended finite element method," *Composites Science and Technology*, vol. 72, no. 7, pp. 788–791, 2012.
- [34] F. Van der Meer and L. Sluys, "Mesh-independent modeling of both distributed and discrete matrix cracking in interaction with delamination in composites," *Engineering Fracture Mechanics*, vol. 77, no. 4, pp. 719–735, 2010.
- [35] S. Natarajan, D. R. Mahapatra, and S. P. Bordas, "Integrating strong and weak discontinuities without integration subcells and example applications in an xfem/gfem framework," *International journal for numerical methods in engineering*, vol. 83, no. 3, pp. 269–294, 2010.
- [36] N. Sukumar and J.-H. Prévost, "Modeling quasi-static crack growth with the extended finite element method part i: Computer implementation," *International journal of solids and structures*, vol. 40, no. 26, pp. 7513–7537, 2003.
- [37] A. Menk and S. P. Bordas, "A robust preconditioning technique for the extended finite element method," *International Journal for Numerical Methods in Engineering*, vol. 85, no. 13, pp. 1609–1632, 2011.
- [38] S. Bordas, P. V. Nguyen, C. Dunant, A. Guidoum, and H. Nguyen-Dang, "An extended finite element library," *International Journal for Numerical Methods in Engineering*, vol. 71, no. 6, pp. 703–732, 2007.
- [39] A. Eijo, E. Oñate, and S. Oller, "A numerical model of delamination in composite laminated beams using the lrz beam element based on the refined zigzag theory," *Composite Structures*, vol. 104, pp. 270–280, 2013.
- [40] E. Abdullah, J.-F. Ferrero, J.-J. Barrau, and J.-B. Mouillet, "Development of a new finite element for composite delamination analysis," *Composites Science and Technology*, vol. 67, no. 10, pp. 2208–2218, 2007.
- [41] L. Iannucci, "Dynamic delamination modelling using interface elements," *Computers & Structures*, vol. 84, no. 15-16, pp. 1029–1048, 2006.
- [42] E. Pietropaoli and A. Riccio, "Formulation and assessment of an enhanced finite element procedure for the analysis of delamination growth phenomena in composite structures," *Composites Science and Technology*, vol. 71, no. 6, pp. 836–846, 2011.
- [43] R. Larsson, "A discontinuous shell-interface element for delamination analysis of laminated composite structures," *Computer Methods in Applied Mechanics and Engineering*, vol. 193, no. 30-32, pp. 3173–3194, 2004.
- [44] V. P. Nguyen, P. Kerfriden, and S. P. Bordas, "Two-and three-dimensional isogeometric cohesive elements for composite delamination analysis," *Composites Part B: Engineering*, vol. 60, pp. 193–212, 2014.
- [45] Y. Mi, M. Crisfield, G. Davies, and H. Hellweg, "Progressive delamination using interface elements," *Journal of composite materials*, vol. 32, no. 14, pp. 1246–1272, 1998.
- [46] S. De, K. Singh, J. Seo, R. K. Kapania, E. Ostergaard, N. Angelini, and R. Aguero, "Structural design and optimization of commercial vehicles chassis under multiple load cases and constraints," in *AIAA Scitech 2019 Forum*, 2019, p. 0705.
- [47] M. Jrad, S. De, and R. K. Kapania, "Global-local aeroelastic optimization of internal structure of transport aircraft wing," in *18th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference*, 2017, p. 4321.
- [48] S. De, K. Singh, J. Seo, R. K. Kapania, E. Ostergaard, N. Angelini, and R. Aguero, "Lightweight chassis design of hybrid trucks considering multiple road conditions and constraints," *World Electric Vehicle Journal*, vol. 12, no. 1, p. 3, 2021.
- [49] S. De, M. Jrad, D. Locatelli, R. K. Kapania, and M. Baker, "Sparibs geometry parameterization for wings with multiple sections using single design space," in *58th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, 2017, p. 0570.
- [50] S. De, K. Singh, J. Seo, R. Kapania, R. Aguero, E. Ostergaard, and N. Angelini, "Unconventional truck chassis design with multi-functional cross members," SAE Technical Paper, Tech. Rep., 2019.
- [51] S. De, "Structural modeling and optimization of aircraft wings having curvilinear spars and ribs (sparibs)," Ph.D. dissertation, Virginia Tech, 2017.
- [52] S. De, K. Singh, B. Alanbay, R. K. Kapania, and R. Aguero, "Structural optimization of truck front-frame under multiple load cases," in *ASME International Mechanical Engineering Congress and Exposition*, vol. 52187. American Society of Mechanical Engineers, 2018, p. V013T05A039.
- [53] S. De and R. K. Kapania, "Algorithms for 2d mesh decomposition in distributed design optimization," *arXiv preprint arXiv:2002.00525*, 2020.
- [54] S. Mulani, D. Locatelli, and R. Kapania, "Grid-stiffened panel optimization using curvilinear stiffeners," in *52nd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference 19th AIAA/ASME/AHS Adaptive Structures Conference 13t*, 2011, p. 1895.
- [55] D. Locatelli, A. Yeilaghi Tamijani, S. B. Mulani, Q. Liu, and R. K. Kapania, "Multidisciplinary optimization of supersonic wing structures using curvilinear spars and ribs (sparibs)," in *54th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, 2013, p. 1931.
- [56] D. Locatelli, S. Mulani, and R. Kapania, "Parameterization of curvilinear spars and ribs (sparibs) for optimum wing structural design," in *53rd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference 20th AIAA/ASME/AHS Adaptive Structures Conference 14th AIAA*, 2012, p. 1359.
- [57] D. Locatelli, S. Mulani, R. Kapania, P. Chen, and D. Sarhaddi, "A multidisciplinary analysis optimization (mdao) environment for wings having sparibs," in *53rd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference 20th AIAA/ASME/AHS Adaptive Structures Conference 14th AIAA*, 2012, p. 1676.
- [58] T. J. Hughes, J. A. Cottrell, and Y. Bazilevs, "Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement," *Computer Methods in Applied Mechanics and Engineering*, vol. 194, no. 39-41, pp. 4135–4195, 2005.
- [59] B. Devarajan, "Free vibration analysis of curvilinearly stiffened composite plates with an arbitrarily shaped cutout using isogeometric analysis," *arXiv preprint arXiv:2104.12856*, 2021.
- [60] B. Devarajan and R. K. Kapania, "Thermal buckling of curvilinearly stiffened laminated composite plates with cutouts using isogeometric analysis," *Composite Structures*, vol. 238, p. 111881, 2020.
- [61] J. Miglani, B. Devarajan, and R. K. Kapania, "Thermal buckling analysis of periodically supported composite beams using isogeometric analysis," in *2018 AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, 2018, p. 1224.
- [62] B. Devarajan and R. K. Kapania, "Analyzing thermal buckling in curvilinearly stiffened composite plates with arbitrary shaped cutouts using isogeometric level set method," *Aerospace Science and Technology*, p. 107350, 2022.
- [63] B. Devarajan, "Thermomechanical and vibration analysis of stiffened unitized structures and threaded fasteners," Ph.D. dissertation, Virginia Tech, 2019.
- [64] B. Devarajan, D. Locatelli, R. K. Kapania, and R. J. Meritt, "Thermo-mechanical analysis and design of threaded fasteners," in *57th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, 2016, p. 0579.
- [65] J. Miglani, B. Devarajan, and R. K. Kapania, "Isogeometric thermal buckling and sensitivity analysis of periodically supported laminated composite beams," *AIAA Journal*, pp. 1–10, 2021.
- [66] A. Karakoti, V. R. Kar, and B. Devarajan, "Hygrothermoelastic responses of sinusoidally corrugated fiber-reinforced laminated composite structures," in *Advanced Composite Materials and Structures*. CRC Press, pp. 201–216.
- [67] Q. Li, B. Devarajan, X. Zhang, R. Burgos, D. Boroyevich, and P. Raj, "Conceptual design and weight optimization of aircraft power systems with high-peak pulsed power loads," SAE Technical Paper, Tech. Rep., 2016.