

Nonlinear Elastic Finite Element Analysis of Frictionless Orthogonal Cutting of Composite Materials Using a Bucket Sort Contact Detection Method

Faouzi Rahmouni

rahmounifaouzi01@gmail.com

LMSS, University of Djillali Liabes, Sidi Bel Abbas

Mohamed Elajrami

LMSS, University of Djillali Liabes, Sidi Bel Abbas

Ziad Elajrami

LMSS, University of Djillali Liabes, Sidi Bel Abbas

Madani Kouider

LMSS, University of Djillali Liabes, Sidi Bel Abbas

Mohammed Benyettou

LMSS, University of Djillali Liabes, Sidi Bel Abbas

Research Article

Keywords: nonlinear FEM analysis, bucket sort detection method, orthogonal cutting, composite materials

Posted Date: September 3rd, 2026

DOI: <https://doi.org/10.21203/rs.3.rs-10797468/v1>

License: © ⓘ This work is licensed under a Creative Commons Attribution 4.0 International License.

[Read Full License](#)

Additional Declarations: No competing interests reported.

Nonlinear Elastic Finite-Element Analysis of Frictionless Orthogonal Cutting of Composite Materials Using a Bucket Sort Contact Detection Method

Faouzi Rahmouni¹, Mohamed Elajrami¹, Ziad Elajrami¹, Madani Kouider¹, Mohammed Benyettou*¹

¹Mechanical Engineering Department, LMSS, University of Djillali Liabes, Sidi Bel Abbes, Algeria.

Faouzi Rahmouni

rahmounifaouzi01@gmail.com

Mohamed Elajrami

eladjrami_mohamed@yahoo.fr

Ziad Elajrami

zelajrami@gmail.com

Kouider Madani

koumad10@yahoo.fr

Mohammed Benyettou *corresponding author

mohammed.benyettou97@gmail.com

Abstract:

Orthogonal cutting simulations rely critically on an efficient and transparent computational framework capable of detecting contact, enforcing interaction constraints, and accurately computing the resulting normal contact forces. This study develops such a framework by integrating an adapted bucket sort detection method with a nonlinear finite element formulation solved through the Newton-Raphson algorithm and penalty-based node-to-segment contact enforcement. Together, these components form a unified computational package designed to efficiently identify contact and compute normal forces under controlled frictionless and elastic conditions. The methodology is applied to two composite material examples to assess its robustness and numerical performance. Validation against Abaqus/Standard demonstrates excellent agreement, with discrepancies in normal contact forces remaining below 1% across all benchmark cases. These results clarify the numerical behavior of frictionless contact within a nonlinear elastic regime and establish a reproducible baseline for future extensions involving frictional interactions, progressive damage modeling, and experimental calibration. All MATLAB code and benchmark examples are made publicly available to support reproducibility (GitHub website).

Keywords: nonlinear FEM analysis, bucket sort detection method, orthogonal cutting, composite materials.

1 Introduction:

In modern manufacturing, processes such as milling, drilling, and grinding are essential for shaping components across aerospace, automotive, and biomedical industries [1]. Orthogonal cutting is a simplified two-dimensional (2D) representation of these operations and is widely

employed to study chip formation and tool-workpiece interaction under controlled conditions, such as cutting speed, feed rate, tool geometry, and material properties. Despite its apparent simplicity, orthogonal cutting involves strongly nonlinear mechanics due to large deformations, incremental contact evolution, and the inherent heterogeneity of composite materials. Among the different physical quantities governing the process, the normal contact force plays a central role because it dictates the penetration of the tool into the workpiece, influences subsurface stresses, and directly affects chip formation, fibre-matrix separation, and damage initiation. Its accurate prediction is therefore essential for reliable simulation of composite machining. Experimental investigations have provided valuable insights into these mechanisms, but their use is limited by high cost and measurement difficulty, which highlights the importance of numerical simulation as a critical tool for understanding and optimizing cutting processes [2, 3].

In composite material research, the finite element method has been the main numerical approach for simulating machining processes, enabling detailed representation of constitutive behavior, thermal effects, and contact phenomena. Classical Lagrangian finite element formulations are widely employed, yet the presence of large deformations causes severe mesh distortion, which makes remeshing procedures necessary and introduces significant computational cost and numerical instabilities [4]. Mesh-free formulations such as the Element Free Galerkin method have been applied to mitigate mesh distortion and provide improved flexibility, although they require more complex implementation and higher computational effort [5]. More recently, multiscale and micromechanical finite element frameworks have been developed to explicitly represent fibre-matrix interactions and damage mechanisms, yielding improved predictive accuracy relative to experimental measurements [6, 7]. However, the efficiency of such formulations remains

constrained by the complexity of the contact problem, which must be treated accurately and robustly to ensure algorithmic stability.

Contact mechanics remains a central challenge in machining simulations because the correct detection and enforcement of contact conditions govern the interaction between the tool and the workpiece. Numerous strategies have been developed, including the pinball algorithm [8], QuickTrace [9], mesh-free search procedures [10], shortest link approaches [11], and bounding-box-based parallel algorithms [12, 13]. Although these approaches improve robustness, they often involve considerable implementation complexity or become computationally expensive when applied to refined meshes, dynamic remeshing operations, and coupled multi-physics analyses [14]. Spatial partitioning techniques offer an attractive alternative for contact detection, and recent work has shown that bucket sort-based approaches can efficiently restrict the search to local neighborhoods by subdividing the simulation domain into smaller regions [15]. The first phase of this research program confirmed that the adapted bucket sort method provides substantial reductions in computational cost compared with classical all-to-all strategies, particularly in orthogonal cutting simulations where the contact zone remains highly localized [16].

The present work represents the second phase of this research program and applies the adapted bucket sort detection method within a nonlinear elastic finite element framework [17] to compute the normal contact force with high accuracy under orthogonal cutting conditions. The formulation employs the Newton-Raphson method with penalty-based enforcement of frictionless node-to-segment contact constraints, while the bucket sort algorithm accelerates the identification of potential contact interactions during each equilibrium iteration. A MATLAB implementation is validated against Abaqus/Standard to assess its capability in predicting normal contact forces,

displacement fields, and stress distributions in composite laminates, with additional analysis of the influence of tool advancement, cutting depth, mesh density, and penalty parameter selection on the contact response. The novelty of this work lies in assembling an integrated and lightweight computational package that combines bucket sort contact detection, nonlinear finite element analysis, Newton-Raphson iteration, and penalty-based node-to-segment enforcement into a single framework tailored for orthogonal cutting. This combined strategy provides fast contact search, stable iterative updates, and highly accurate elastic normal force prediction within 1% of a commercial solver. Furthermore, the study establishes a reproducible and computationally efficient baseline for future developments that will incorporate frictional contact, progressive damage mechanisms, and experimental calibration, thereby positioning this phase as a foundational step toward more realistic and physics-rich composite machining simulations.

2 Physical modeling and limitations

To validate the contact detection and enforcement strategy in a controlled setting, the physical model is intentionally simplified. These assumptions isolate the numerical behavior of the bucket sort detection method and penalty-based contact formulation, allowing their performance to be evaluated without interference from more complex machining physics:

- **Frictionless contact:** Only normal contact forces are considered to isolate the performance of the detection algorithm, without the added complexity of frictional behavior.
- **Rigid cutting tool:** The tool is assumed perfectly rigid, which removes tool-deformation effects and focuses the analysis on contact accuracy.

- **Two-dimensional orthogonal cutting:** A 2D plane stress configuration is used to represent the cross-section of the cutting process and reduce computational effort while assessing the node-to-segment contact strategy.
- **Nonlinear elastic workpiece:** The composite is modeled as an orthotropic elastic material in plane stress, allowing evaluation of the contact formulation without involving plasticity or progressive damage.
- **No thermal or inelastic effects:** Thermal coupling, plasticity, delamination, chip formation, and other damage mechanisms are excluded because they are beyond the scope of this validation-focused study.

3 Mathematical formulation

In the present study, the normal contact forces between a master tool and a slave workpiece are evaluated in the elastic regime. Since the contact constraints are nonlinear, the problem cannot be treated using a linear formulation. The developed approach captures nonlinear behavior and accurately represents contact conditions while preserving equilibrium during the elastic phase of the analysis.

3.1 Contact problem

This study employs a node-to-segment contact discretization in which the deformable workpiece is modeled as the slave surface and the rigid tool as the master surface (see **Fig. 1**). Each slave node is checked against master segments to detect and enforce contact constraints [15, 16]. The detailed implementation of the contact algorithm is summarized in the stepwise procedure presented in **Algorithm 1**.

3.1.1 Geometric definitions

Let the current position of a slave node be denoted by $\mathbf{x}_s \in \mathbb{R}^2$. A master segment is defined by its two endpoints $\mathbf{x}_{m1}, \mathbf{x}_{m2} \in \mathbb{R}^2$ as shown in **Fig. 1b**. The tangent vector and its normalized counterpart are given by:

$$\mathbf{t} = \mathbf{x}_{m2} - \mathbf{x}_{m1}, \quad \mathbf{e}_t = \frac{\mathbf{t}}{\|\mathbf{t}\|}$$

Where $\|\cdot\|$ denotes the Euclidean norm. The outward unit normal $\mathbf{e}_n$ is obtained by a left-hand rotation of the tangent:

$$\mathbf{e}_n = \begin{bmatrix} -\mathbf{e}_{t_y} \\ \mathbf{e}_{t_x} \end{bmatrix}, \quad \|\mathbf{e}_n\| = 1$$

With $\mathbf{e}_{t_x}$ and $\mathbf{e}_{t_y}$ the Cartesian components of $\mathbf{e}_t$.

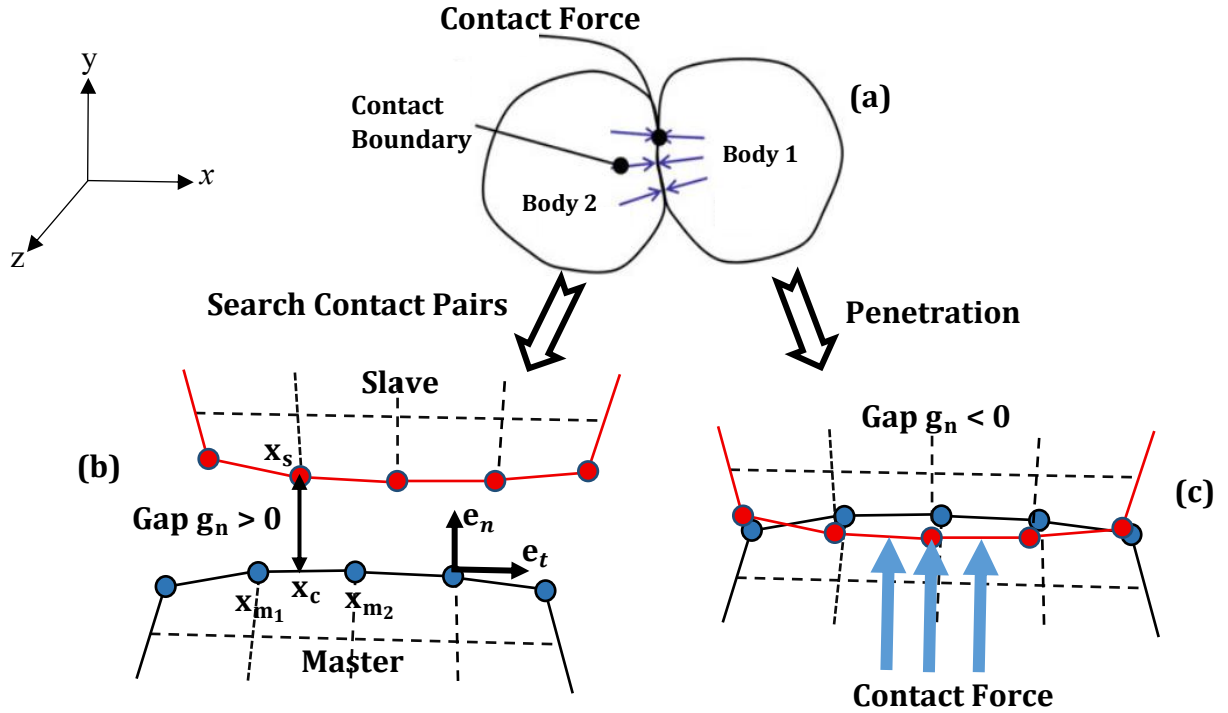


Figure 1. Node-to-segment contact formulation: (a) Contact interaction, (b) Gap definition, (c) Penalty enforcement.

3.1.2 Projection and normal gap

As illustrated in **Fig. 1b**, $\mathbf{x}_c$ is the projection of $\mathbf{x}_s$ onto the segment $[\mathbf{x}_{m_1}, \mathbf{x}_{m_2}]$, expressed in (x, y) coordinates as:

$$\mathbf{s} = \text{clamp}\left((\mathbf{x}_s - \mathbf{x}_{m_1}) \cdot \mathbf{e}_t, 0, 1\right), \quad \mathbf{x}_c = \mathbf{x}_{m_1} + \mathbf{s} (\mathbf{x}_{m_2} - \mathbf{x}_{m_1})$$

Where $\cdot$ denotes the scalar (dot) product and the operator $\text{clamp}(\cdot, 0, 1)$ restricts the projection to lie on the segment.

The scalar normal gap is then given by:

$$\mathbf{g}_n = \mathbf{e}_n^T (\mathbf{x}_s - \mathbf{x}_c)$$

- If $\mathbf{g}_n > 0$, the slave node is distant from the master segment, indicating no contact.
- If $\mathbf{g}_n \leq 0$ and $0 \leq \mathbf{s} \leq 1$, the node penetrates the master segment and the contact pair becomes active.

Here, $\mathbf{g}_n \in \mathbb{R}$ has units of length and represents the distance between the node and the segment in the normal direction.

3.1.3 Penalty enforcement

To enforce the non-penetration condition ($\mathbf{g}_n > 0$), a penalty regularization is employed. Small penetrations ($\mathbf{g}_n \leq 0$) are allowed but penalized through a penalty stiffness parameter $\omega_n \in \mathbb{R}^+$.

The nodal contact force (see **Fig. 1c**) is then expressed as [17]:

$$\mathbf{F}_c = \begin{cases} -\omega_n \mathbf{g}_n \mathbf{e}_n & \mathbf{g}_n < 0 \\ \mathbf{0} & \mathbf{g}_n \geq 0 \end{cases}$$

Where $\mathbf{F}_c \in \mathbb{R}^2$ is the vector of contact forces acting on the slave node. The associated consistent tangent (contact stiffness) operator is given by:

$$\mathbf{K}_c = \omega_n \mathbf{e}_n \mathbf{e}_n^T$$

With $\mathbf{K}_c \in \mathbb{R}^2$ a rank-one, positive semi-definite tensor.

The contributions $\mathbf{F}_c$ and $\mathbf{K}_c$ are summed over all active slave nodes and incorporated into the global residual vector and stiffness matrix. This penalty approach is computationally efficient and avoids the introduction of additional Lagrange multipliers [18]. However, the choice of ω_n is critical. If it is too small, excessive penetration occurs, whereas overly large values can ill-condition the global stiffness matrix and cause the solution to diverge [19, 20].

3.2 Bucket sort

The bucket sort method accelerates contact detection by partitioning the space into a grid of buckets (spatial cells) and focusing only on regions where the tool and workpiece may overlap. In this approach, the maximum expected tool displacement $\mathbf{D}_{\max}$ is first computed, and axis-aligned bounding boxes around the tool and the workpiece are expanded by $\mathbf{D}_{\max}$. Only the overlapping volume of these bounding boxes is considered a potential contact region (see **Fig. 2**), so nodes outside this overlap are immediately ignored. This coarse filtering ensures that only slave nodes near the tool are checked for contact, greatly reducing the number of comparisons. For further details on the adapted bucket sort implementation employed in this study, the reader is referred to Rahmouni et al. [16].

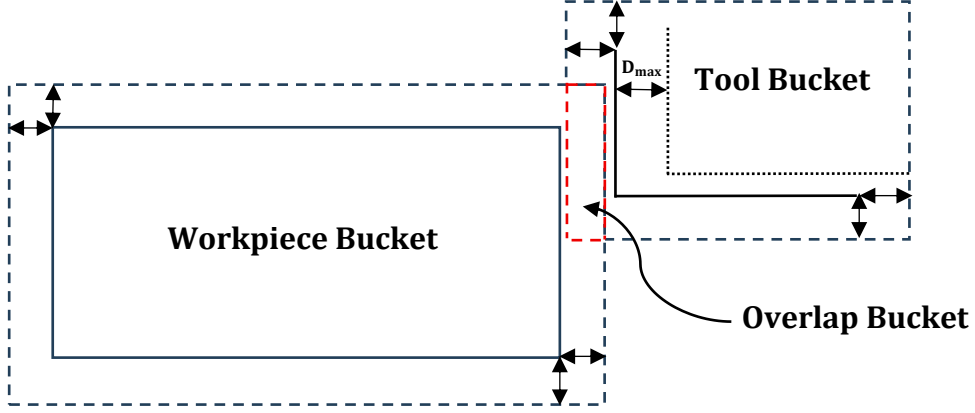


Figure 2. Illustration of the bucket sort contact detection strategy.

Algorithm 1. Calculation of nodal contact force and contact stiffness

1. Perform contact search:

- Partition domains into buckets.
- Create contact pairs only for slave nodes corresponding to master segments within the overlap bucket, ignoring all nodes outside this region.

2. For each contact pair (slave node, master segment):

- Compute tangent vector $\mathbf{t}$ and unit tangent $\mathbf{e}_t$.
- Compute outward unit normal $\mathbf{e}_n$.
- **Project slave node onto segment:**

$$\mathbf{s} = \text{clamp}((\mathbf{x}_s - \mathbf{x}_{m1}) \cdot \mathbf{e}_t, 0, 1)$$

$$\mathbf{x}_c = \mathbf{x}_{m1} + \mathbf{s}(\mathbf{x}_{m2} - \mathbf{x}_{m1})$$

- **Compute normal gap:**
- **Check contact condition:**

$$\mathbf{g}_n = \mathbf{e}_n^T (\mathbf{x}_s - \mathbf{x}_c)$$

If $\mathbf{g}_n > 0 \rightarrow$ no contact

Continue.

If $\mathbf{g}_n \leq 0$ and $0 \leq \mathbf{s} \leq 1 \rightarrow$ active contact:

- **Compute nodal contact force** $\mathbf{F}_c = -\omega_n \mathbf{g}_n \mathbf{e}_n$
 - **Compute contact stiffness:** $\mathbf{K}_c = \omega_n \mathbf{e}_n \mathbf{e}_n^T$
-

3.3 Finite element method formulation

3.3.1 Governing equilibrium equations

For a deformable elastic body $\Omega \subset \mathbb{R}^2$ with boundary decomposition $\Gamma = \Gamma_g \cup \Gamma_s \cup \Gamma_c$. The strong form of equilibrium (**Fig. 3**):

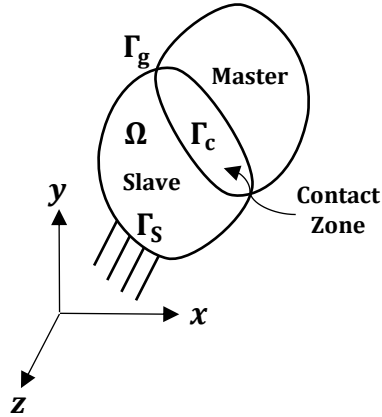


Figure 3. Boundary conditions for elastic contact analysis.

$$\sigma_{ij,j} + f_i^B = 0 \quad \text{in } \Omega$$

Subject to essential and natural boundary conditions:

$$\mathbf{u}_i = \bar{\mathbf{u}}_i \quad \text{on } \Gamma_g \qquad \sigma_{ij} \mathbf{n}_j = \mathbf{f}_i^S \quad \text{on } \Gamma_s$$

On the potential contact boundary Γ_s , unilateral Signorini-type constraints enforce impenetrability and compressive-only contact stresses:

$$\mathbf{u}^T \mathbf{e}_n + \mathbf{g}_n \leq 0 \qquad \sigma_n \leq 0 \qquad \sigma_n (\mathbf{u}^T \mathbf{e}_n + \mathbf{g}_n) = 0$$

Where $\mathbf{u}^T \mathbf{e}_n$ is the normal displacement, and $\boldsymbol{\sigma}_n = \sigma_{ij} \mathbf{n}_i \mathbf{n}_j$ is the normal contact stress. These contact conditions ensure that the slave nodes cannot penetrate the rigid master tool, while allowing separation without tensile contact.

3.3.2 Weak form of equilibrium

Starting from the governing equilibrium equations with unilateral contact constraints described above, the weak form is obtained by multiplying the strong form by an admissible virtual displacement field $\delta \mathbf{u} \in \Omega$ and integrating over the domain. Application of Gauss's theorem gives:

$$\int_{\Omega} \sigma_{ij} \delta \varepsilon_{ij} d\Omega = \int_{\Omega} \mathbf{f}_i^B \delta \mathbf{u}_i d\Omega + \int_{\Gamma_s} \mathbf{f}_i^S \delta \mathbf{u}_i d\Gamma - \int_{\Gamma_c} \mathbf{f}_i^c \delta \mathbf{u}_i d\Gamma$$

Where $\delta \varepsilon_{ij} = \frac{1}{2}(\delta \mathbf{u}_{i,j} + \delta \mathbf{u}_{j,i})$ is the virtual strain, $\mathbf{f}_i^B$ denotes body forces, $\mathbf{f}_i^S$ the prescribed surface tractions, and $\mathbf{f}_i^c$ represents the contact tractions obtained from the penalty formulation.

Approximating the displacement field by finite element interpolation:

$$\mathbf{u}(\mathbf{x}) \approx \mathbf{N}(\mathbf{x}) \mathbf{d} \quad \boldsymbol{\varepsilon} = \mathbf{B} \mathbf{d}$$

Where $\mathbf{N}(\mathbf{x})$ is the shape functions matrix, $\mathbf{d}$ is the vector of nodal displacement unknowns, and $\mathbf{B}$ is the strain-displacement matrix derived from the derivatives of the shape functions. The discretized equilibrium system takes the form:

$$(\mathbf{K} + \mathbf{K}_c) \mathbf{d} = \mathbf{F}_{\text{ext}} - \mathbf{F}_c$$

With the elastic stiffness matrix $\mathbf{K}$ and the external force vector $\mathbf{F}_{\text{ext}}$ defined as:

$$\mathbf{K} = \int_{\Omega} \mathbf{B}^T \cdot \mathbf{D} \cdot \mathbf{B} \, d\Omega$$

$$\mathbf{F}_{\text{ext}} = \int_{\Omega} \mathbf{N}^T \cdot \mathbf{f}^B \, d\Omega + \int_{\Gamma_S} \mathbf{N}^T \cdot \mathbf{f}^S \, d\Gamma$$

Where $\mathbf{D}$ is the constitutive (elasticity) matrix of the material (for details on how to compute the $\mathbf{D}$ matrix, see [21, 22]), relating stresses to strains equation. The additional term $\mathbf{F}_c$ and $\mathbf{K}_c$ denotes the assembled nodal contact forces vector and the contact stiffness matrix, which accounts for the contributions of all active slave nodes, as detailed in section **2.1 Contact Problem**.

Since both $\mathbf{F}_c$ and $\mathbf{K}_c$ depend on the current displacement field through the contact gap function, the equilibrium system is nonlinear. A Newton–Raphson scheme is therefore employed, in which the tangent problem at iteration k :

$$(\mathbf{K} + \mathbf{K}_c^k) \Delta \mathbf{d}^k = \mathbf{R}(\mathbf{d}^k)$$

With residual

$$\mathbf{R}(\mathbf{d}^k) = \mathbf{F}_{\text{ext}} - \mathbf{F}_c^k - (\mathbf{K} + \mathbf{K}_c^k) \Delta \mathbf{d}^k$$

The displacement field is updated incrementally $\mathbf{d}^{k+1} = \mathbf{d}^k + \Delta \mathbf{d}^k$ until convergence is achieved when the residual norm satisfies the tolerance criterion:

$$\|\mathbf{R}(\mathbf{d}^{k+1})\| \leq \textit{eps}$$

Where *eps* is a prescribed convergence tolerance, typically chosen as a small positive number on the order of 10^{-6} to 10^{-8} . This ensures that the residual is sufficiently small, practically approaching zero, thereby indicating that equilibrium has been attained.

4 Description of the problem

Normal contact forces are the forces acting perpendicular to the interface between the tool and the workpiece. They control how the load is transferred and how stresses develop in the material. In composites, these forces closely follow the material's deformation and play a key role in predicting damage, such as delamination or fracture. Accurately calculating them is therefore essential for understanding the mechanics of cutting, both in the elastic and failure stages. This study investigates frictionless contact between a rigid tool and a rectangular composite plate under the plane-stress approximation, using a bucket-sort strategy for efficient contact detection. The principal objective is to extract the normal contact response caused by a prescribed tool advance. Composites generally respond elastically over most of the applied load and then fail suddenly, in a manner consistent with Linear Elastic Fracture Mechanics (LEFM). The plastic zone can be considered negligible, and an elastic-only model provides a valid and efficient simplification [23]. Within this framework, the analysis examines how variations in material properties, tool depth and advancement, mesh density, and penalty parameter affect the evolution of the normal contact forces.

The numerical setup consists of a rectangular elastic plate representing the workpiece (slave surface) and a rigid two-segment line representing the tool (master surface), as illustrated in **Fig. 4**. The plate dimensions are $L = 100 \text{ mm}$ in length and $W = 100 \text{ mm}$ in width, discretized into quadrilateral finite elements with nodes along the right edge acting as potential slave nodes within the defined cutting depth. The tool geometry is constructed from two connected master segments of equal length (100 mm), oriented by angles $\alpha = \gamma = 0$ and advanced horizontally to cause contact with the plate. Two composite material systems are considered, namely Carbon/Epoxy and

Glass/Epoxy, and their elastic properties are summarized in **Table 1**, with fiber orientations fixed at $\theta = 15^\circ$. The contact interaction is enforced using a penalty formulation with a normal penalty stiffness ω_n varied incrementally between 10^1 and 10^9 , with 10^7 adopted in the proposed model, enabling assessment of the influence of penalty magnitude on penetration control, numerical stability, and convergence.

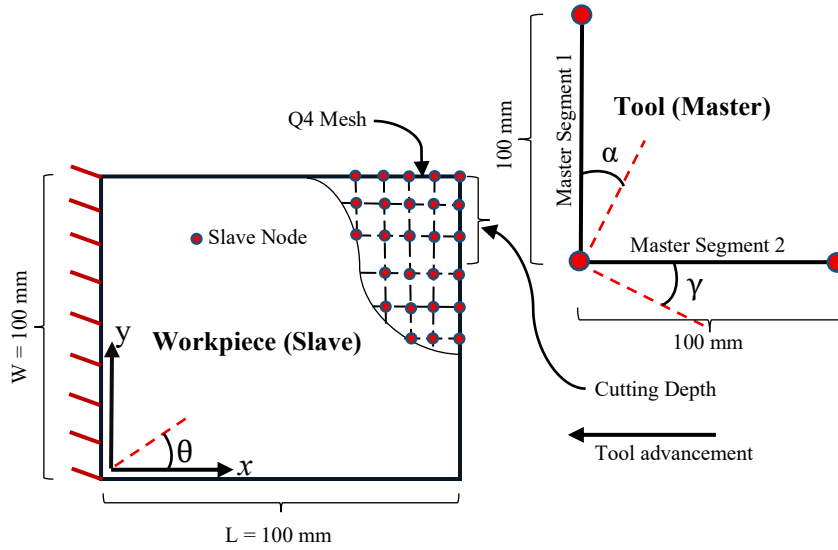


Figure 4. Simulation setup of the orthogonal cutting process

Validation is performed by running tests on Carbon/Epoxy and Glass/Epoxy composite plates using both Abaqus and the MATLAB implementation, and comparing their contact responses. The same geometry, boundary conditions, and loading are applied in both platforms, and the comparison of structural deformation, stress distributions, and normal contact forces provides a reference for evaluating the accuracy of the developed model. Following this validation, a series of tests are conducted to evaluate robustness and sensitivity. The first test monitors the evolution of the normal contact force with tool advancement. The second investigates the effect of cutting depth, the third examines mesh refinement from coarse to fine discretization to demonstrate convergence, and the fourth evaluates the influence of the penalty parameter. Computational

timings are also recorded to assess the efficiency of the bucket-sort contact search compared to a full search and relative to Abaqus performance. Collectively, these tests examine how material properties, tool advancement, cutting depth, mesh resolution, and penalty parameters affect the evolution of normal contact forces.

Table 1 Mechanical properties of composites [22].

Material	E_1 (MPa)	E_2 (MPa)	G_{12} (MPa)	ν_{12}
Carbon/Epoxy	121000	8600	4700	0.27
Glass/Epoxy	54900	18300	9140	0.25

5 Results and discussion

5.1 Validation of the proposed model

Validation of the proposed model is carried out by comparing its predictions with Abaqus reference simulations. **Fig. 5a** and **5b** present the results for Carbon/Epoxy with a small tool advancement of 3 *mm*, ensuring the response remains within the elastic range. In both cases, the displacement field shows clear deformation localized near the tool–workpiece interaction, while the rest of the plate remains largely undeformed. The stress contours further confirm a clear concentration in the contact region, consistent with the mechanics of elastic contact. The proposed model successfully predicts both the overall deformation and the stress distribution near the contact area, demonstrating its ability to accurately represent contact behavior. Computationally, the MATLAB implementation completes the simulation in an average of 19.14 s, compared to an average of 24.02 s for Abaqus under identical hardware and software conditions, highlighting the efficiency of the developed approach. This agreement, both in observed patterns and numerical values, provides confidence in the predictive capability of the formulation.

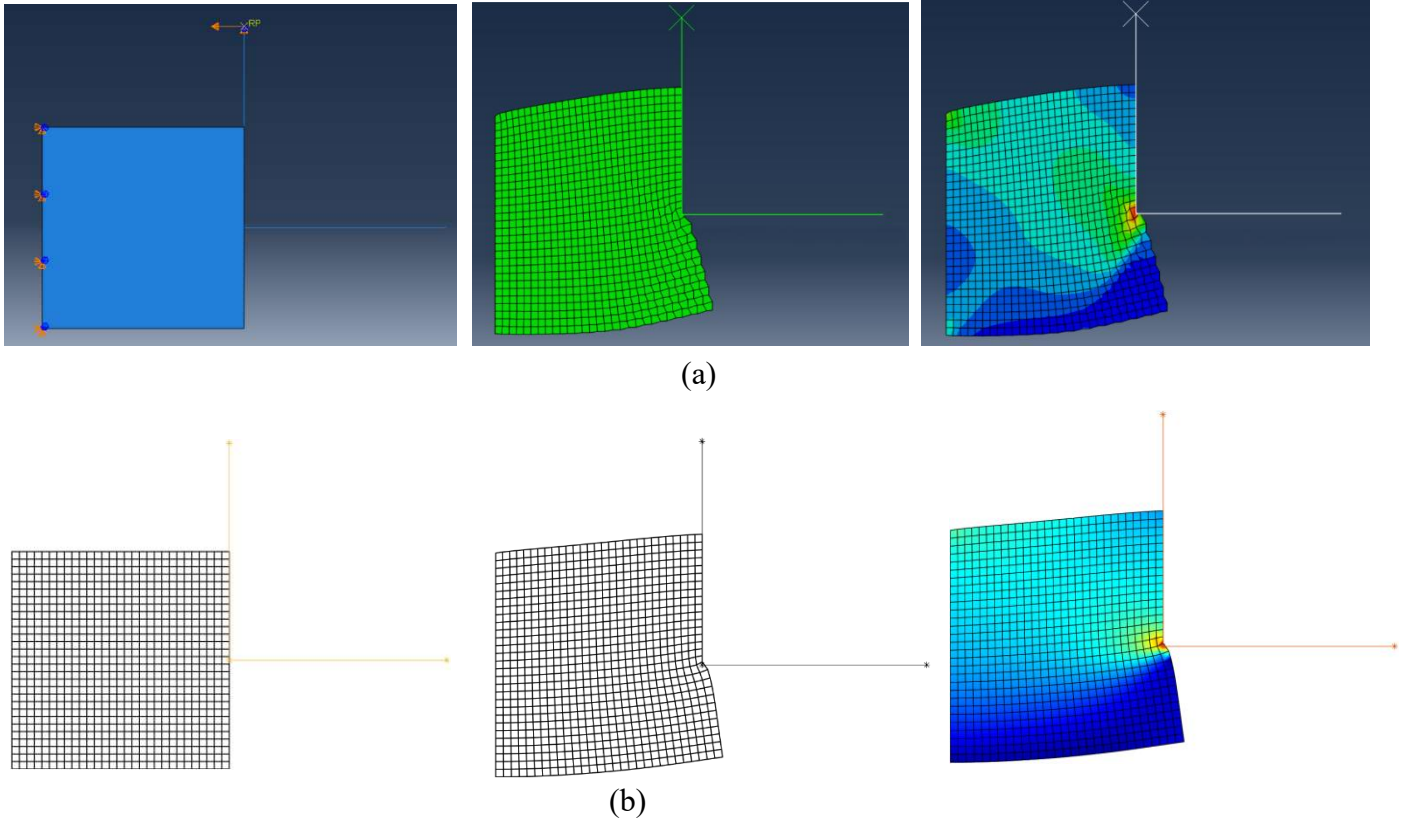


Figure 5. Simulation results for Carbon/Epoxy with 3 *mm* tool advancement: **(a)** Abaqus, **(b)** Proposed model (Matlab).

5.2 Effect of tool advancement

The numerical validation is presented in **Tables 2** and **3**, which compare the predicted normal contact forces with Abaqus results for Carbon/Epoxy and Glass/Epoxy laminates, respectively. The agreement between the two approaches is excellent at all levels of tool advancement, with errors remaining below **1%**. This correspondence is further reinforced by **Fig. 6**, which plots the data from **Tables 2** and **3** and shows that the values obtained from both models coincide closely. The results show that Carbon/Epoxy experiences higher normal contact forces than Glass/Epoxy under the same tool advancement, reflecting the role of material stiffness in governing the contact response. Together, the observed patterns (*section 4.1*) and numerical validations confirm that the proposed implementation provides reliable predictions of both displacement and force behavior

and can be confidently employed for further parametric investigations. Furthermore, the normal contact forces predicted in this study are consistent with previous experimental and numerical results published in literature, which document thrust forces of approximately 450-1,300 N [24,25], reinforcing the realism of the results and supports the accuracy of the implemented model.

Table 2. Validation of normal contact forces predictions: Abaqus vs proposed model for Carbon/Epoxy.

Tool advancement	Normal contact force (N)	
	Abaqus	Proposed model
0.1	23.46	23.49
0.5	116.11	116.32
1.5	342.02	342.33
3	659.01	659.80

Table 3. Validation of normal contact forces predictions: Abaqus vs proposed model for Glass/Epoxy.

Tool advancement	Normal contact force (N)	
	Abaqus	Proposed model
0.1	14.22	14.31
0.5	71.12	71.55
1.5	215.48	215.81
3	436.03	436.53

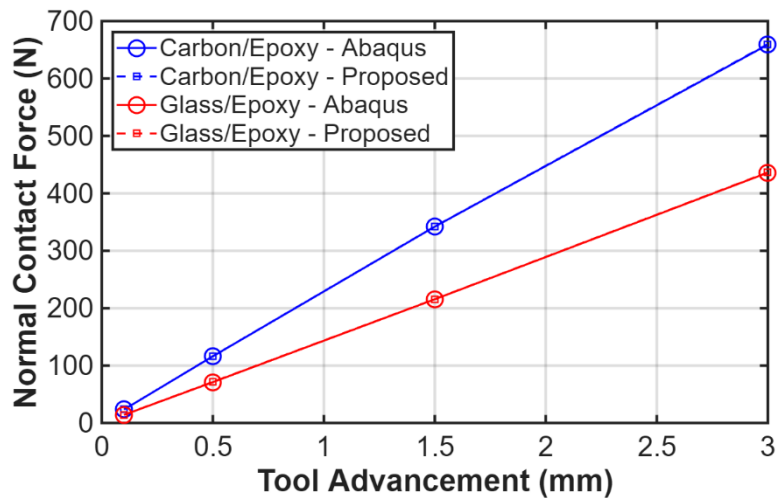


Figure 6. Validation of normal contact forces predictions.

5.3 Effect of cutting depth

It is important to emphasize that, within the elastic regime considered in this study, the cutting depth is not expected to significantly affect the normal contact force. The reason lies in the fact that, under the purely elastic phase, the composite plate behaves as an isotropic and homogeneous

material. In other words, the normal contact response depends primarily on the material stiffness and tool advancement rather than on the engaged depth. This unchanging behavior is clearly observed in the Glass/Epoxy results, where the forces remain nearly constant across all cutting depths, as shown in **Table 5** and **Fig. 7**. For Carbon/Epoxy, a slight variation of the predicted forces with increasing depth is observed in **Table 4** and **Fig. 7**, however, this difference is due to numerical effects, such as mesh discretization, local stress distribution, or penalty parameter sensitivity, rather than actual physical behavior. Thus, for both materials, the correct interpretation is that cutting depth does not alter the normal contact force in the elastic regime, consistent with theoretical expectations.

Table 4. Effect of cutting depth on normal contact forces for Carbon/Epoxy.

Cutting depth (<i>mm</i>)	Normal contact force (N)
15	982.38
30	812.82
50	659.80

Table 4. Effect of cutting depth on normal contact forces for Glass/Epoxy.

Cutting depth (<i>mm</i>)	Normal contact force (N)
15	413.92
30	400.26
50	436.53

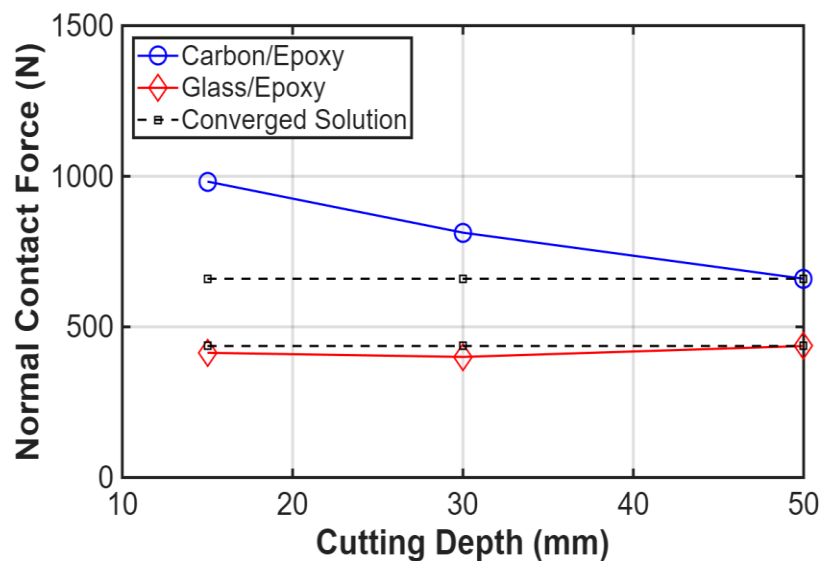


Figure 7. Effect of cutting depth on normal contact forces.

Furthermore, **Fig. 8** provides additional validation of this conclusion, as the maximum stresses are consistently concentrated in the vicinity of the tool-workpiece contact region and gradually decrease away from the cutting zone, with no significant change in the overall distribution as cutting depth increases. The persistence of this pattern across all cases indicates that the approach consistently captures elastic behavior and demonstrates the robustness of the implementation.

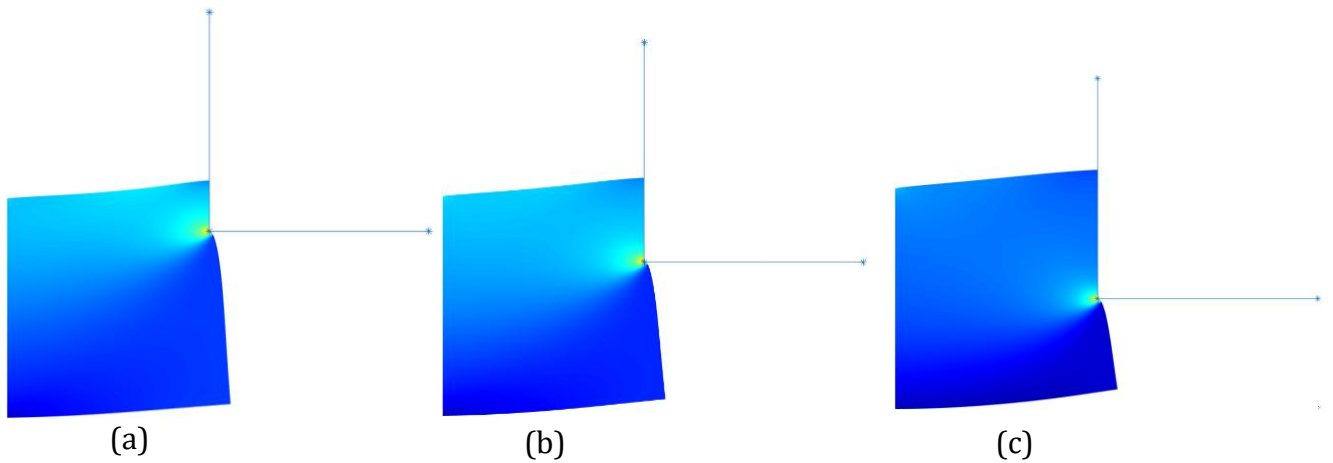


Figure 8. Stress distribution at different cutting depths: **(a)** 15 mm, **(b)** 30 mm, **(c)** 50 mm.

5.4 Effect of mesh density

The influence of mesh density on the predicted normal contact forces is reported in **Tables 6** and **7** and illustrated in **Fig. 9**. The results show that as the mesh is refined, the normal contact forces gradually reach a stable, converged value. Where coarse discretization with 100 Q4 elements results in much higher values than the converged solution, and a considerable variation is observed between 100 and 900 elements, while refinements with more than 4,600 elements and close to 10,000 elements lead to stable values that indicate numerical convergence. This behavior is consistent for both Carbon/Epoxy and Glass/Epoxy composites, with Carbon/Epoxy showing higher forces across all mesh levels. At the finest resolution of 10,000 elements, the contact forces converge to 659.8 N for Carbon/Epoxy and 436.5 N for Glass/Epoxy, which can be considered

mesh-independent. These results demonstrate the essential role of mesh refinement in contact simulations and confirm that sufficiently fine discretization is required to ensure reliable predictions of contact forces, which demonstrates the robustness of the proposed approach.

Table 6. Effect of mesh density on normal contact forces for Carbon/Epoxy.

Mesh density (Q4 element)	Normal contact force (N)
100	7282.15
900	2269.15
3600	1116.12
6400	680.20
10000	659.80

Table 7. Effect of mesh density on normal contact forces for Glass/Epoxy.

Mesh density (Q4 element)	Normal contact force (N)
100	5310.68
900	1594.30
3600	753.25
6400	445.32
10000	436.53

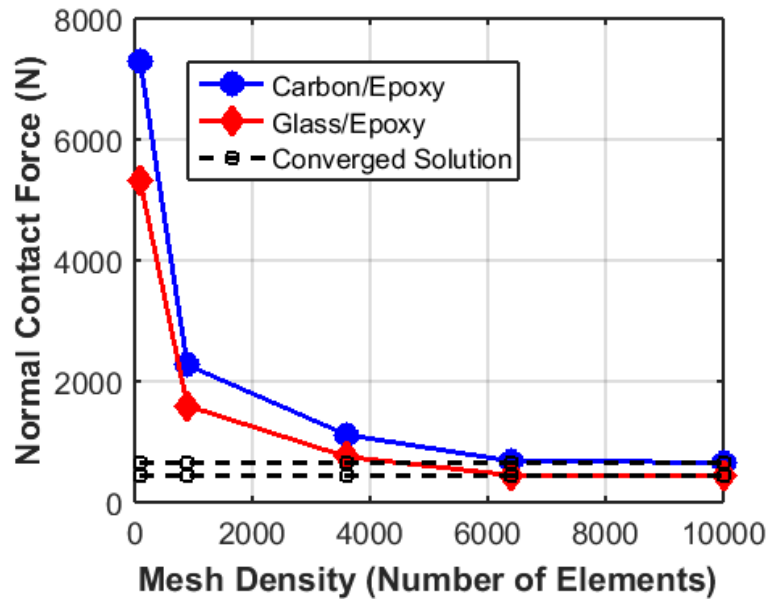


Figure 9. Effect of mesh density on normal contact forces.

5.5 Effect of penalty parameter

The effect of the penalty parameter on the predicted normal contact forces is summarized in **Table 8** and illustrated in **Fig. 10**. The results show that for very small values of the penalty parameter

(10^1 - 10^2), the computed forces are significantly underpredicted due to the excessive penetration permitted by the weak enforcement of the contact constraint. As the penalty parameter increases to intermediate values (10^3 - 10^5), the normal contact forces approach the converged solution. For values greater than 10^5 and close to 10^9 , the results stabilize, with Carbon/Epoxy converging to approximately 659.7 N and Glass/Epoxy to about 436.5 N, which can be considered penalty-independent solutions. However, excessively large penalty values (10^9 and above) introduce numerical ill-conditioning and lead to residual divergence, indicating a loss of stability in the iterative solution. These findings highlight the dual role of the penalty parameter: it must be sufficiently large to limit penetration while avoiding excessive stiffness that compromises numerical convergence. An appropriate choice in the range of 10^6 - 10^8 provides accurate and stable results for both composite systems.

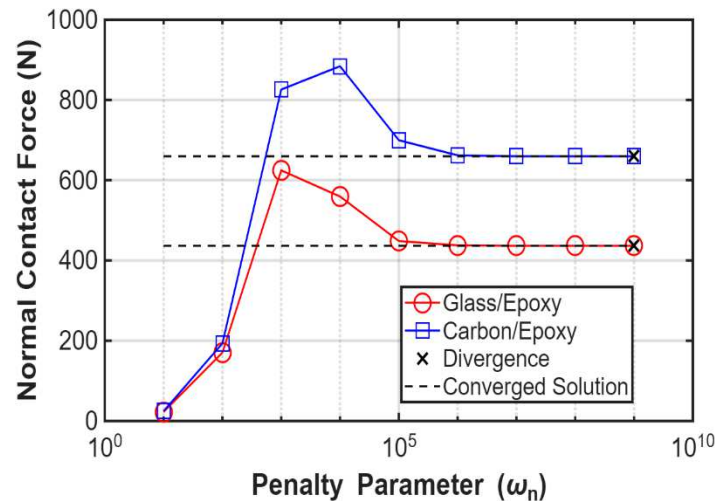


Figure 10. Effect of penalty parameter on normal contact force.

Table 8. Effect of penalty parameter on normal contact forces for Carbon/Epoxy.

Penalty parameter Normal contact force (N)	10^1	10^2	10^3	10^4	10^5	10^6	10^7	10^8	10^9
Carbon/Epoxy	24.58	193.24	826.13	884.22	699.75	662.08	659.80	659.71	Residual divergence
Glass/Epoxy	22.67	170.31	624.62	559.50	448.51	437.68	436.53	436.51	Residual divergence

Conclusion

This study presented a finite element formulation for the analysis of elastic frictionless contact between a rigid tool and composite plates using an adapted bucket-sort detection method under the plane-stress approximation. The formulation was implemented in MATLAB and validated against Abaqus solutions for both Carbon/Epoxy and Glass/Epoxy composite plates. The comparison of structural deformation, stress distributions, and normal contact force evolution demonstrated excellent agreement, with relative errors consistently below 1%. In addition to providing accurate predictions, the MATLAB implementation achieved shorter computation times compared with Abaqus, thereby confirming the efficiency of the proposed approach. The key findings of this work can be summarized as follows:

- Tool advancement is the main driver of normal contact force, with Carbon/Epoxy showing higher forces than Glass/Epoxy.
- Cutting depth has no significant effect on elastic normal contact force.
- Mesh refinement is essential, with convergence achieved only for sufficiently fine discretization.
- Penalty parameters between 10^6 and 10^8 provide accurate and stable results.

Overall, the results demonstrate that the combined use of nonlinear elasticity, penalty enforcement, and bucket-sort spatial partitioning provides a reliable and computationally efficient framework for evaluating normal contact forces in orthogonal cutting. The validated performance of the method, both numerically and computationally, establishes this work as a baseline phase within a broader research program. The framework provides a foundation for future extensions that will incorporate frictional contact, large deformation effects, progressive damage and failure, and

experimental calibration to achieve a fully predictive composite machining simulation environment.

Author Contributions

Faouzi Rahmouni: Software, Validation, Conceptualization, Modelling, Writing -original draft.

Mohamed Eladjrami: Investigation, Writing -original draft, Supervision, **Ziad Elajrami:**

Software, Simulation support, Results analysis. **Kouider Madani:** Investigation, Supervision,

Writing - review & editing. **Mohammed Benyettou:** Resources, Software, Visualization,

Writing - review & editing

Conflict of Interest

The authors declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Data Availability Statement

Data sharing not applicable to this article as no datasets were generated or analyzed during the current study.

Funding

The authors received no financial support for the research, authorship, and/or publication of this article.

References

- [1] Arola, D. and Ramulu, M., 1997. Orthogonal cutting of fiber-reinforced composites: a finite element analysis. *International journal of mechanical sciences*, 39(5), pp.597-613.
- [2] Usui, S., Wadell, J. and Marusich, T., 2014. Finite element modeling of carbon fiber composite orthogonal cutting and drilling. *Procedia Cirp*, 14, pp.211-216.
- [3] An, Q., Zhang, J., Xiao, G., Xu, C., Yi, M., Chen, Z., Chen, H., Zheng, C. and Li, G., 2025. Fiber Orientation Effects in CFRP Milling: Multiscale Characterization of Cutting Dynamics, Surface Integrity, and Damage Mechanisms. *Journal of Composites Science*, 9(7), p.342.
- [4] Movahhedy, M., Gadala, M.S. and Altintas, Y., 2000. Simulation of the orthogonal metal cutting process using an arbitrary Lagrangian–Eulerian finite-element method. *Journal of materials processing technology*, 103(2), pp.267-275.
- [5] Kahwash, F., Shyha, I. and Maheri, A., 2017. Meshfree formulation for modelling of orthogonal cutting of composites. *Composite Structures*, 166, pp.193-201.
- [6] Abena, A. and Essa, K., 2019. 3D micro-mechanical modelling of orthogonal cutting of UD-CFRP using smoothed particle hydrodynamics and finite element methods. *Composite Structures*, 218, pp.174-192.
- [7] Hassouna, A., Mzali, S., Zemzemi, F. and Mezlini, S., 2020. Orthogonal cutting of UD-CFRP using multiscale analysis: Finite element modeling. *Journal of Composite Materials*, 54(18), pp.2505-2518.
- [8] Belytschko, T. and Neal, M.O., 1991. Contact-impact by the pinball algorithm with penalty and Lagrangian methods. *International journal for numerical methods in engineering*, 31(3), pp.547-572.

- [9] Bruneel, H.C. and De Rycke, I., 2002. QuickTrace: a fast algorithm to detect contact. International journal for numerical methods in engineering, 54(2), pp.299-316.
- [10] Li, S., Qian, D., Liu, W.K. and Belytschko, T., 2001. A meshfree contact-detection algorithm. Computer methods in applied mechanics and engineering, 190(24-25), pp.3271-3292.
- [11] G. Nezami, E., MA Hashash, Y., Zhao, D. and Ghaboussi, J., 2006. Shortest link method for contact detection in discrete element method. International Journal for Numerical and Analytical Methods in Geomechanics, 30(8), pp.783-801.
- [12] Malone, J.G. and Johnson, N.L., 1994. A parallel finite element contact/impact algorithm for non-linear explicit transient analysis: Part I—The search algorithm and contact mechanics. International journal for numerical methods in engineering, 37(4), pp.559-590.
- [13] Attaway, S.W., Hendrickson, B., Plimpton, S.J., Gardner, D.R., Vaughan, C.T., Brown, K.H. and Heinstein, M.W., 1998. A parallel contact detection algorithm for transient solid dynamics simulations using PRONTO3D. Computational Mechanics, 22(2), pp.143-159.
- [14] Laursen, T.A. ed., 2006. Computational contact mechanics (Vol. 2, p. 25). Berlin: Springer.
- [15] Yastrebov, V.A., 2013. Numerical methods in contact mechanics. John Wiley & Sons.
- [16] RAHMOUNI, F., ELAJRAMI, M. and MADANI, K., Numerical Comparison of Contact Detection Algorithm Adapting for Orthogonal Cutting Process.
- [17] Kim, N. H. (2015). Introduction to Nonlinear Finite Element Analysis (2nd ed.). Springer.
- [18] Yu, C., Jing, D., Fu, C. and Yang, Y., 2021. A Kind of FM-BEM Penalty Function Method for a 3D Elastic Frictional Contact Nonlinear System. Journal of Mathematics, 2021(1), p.6626647.

- [19] Kang, S.H., Lee, S.M. and Shin, S., 2023. Improved area regularization technique for penalty-method-based node-to-segment contact analysis. *Computational Mechanics*, 71(4), pp.801-825.
- [20] Layton, W. and Xu, S., 2022. Conditioning of linear systems arising from penalty methods. *arXiv preprint arXiv:2206.06971*.
- [21] Rahmouni, F., Elajrami, M., Madani, K. and Campilho, R.D., 2023. Isogeometric analysis of stress concentrations and failure strength in composite plates with circular holes using RHT-splines. *European Journal of Mechanics-A/Solids*, 99, p.104904.
- [22] Rahmouni, F., Elajrami, M., Madani, K. and Campilho, R.D.S.G., 2022. Isogeometric analysis based on non-uniform rational B-splines technology of stress and failure strength in inter-ply hybrid laminated composite. *Journal of Composite Materials*, 56(18), pp.2921-2932.
- [23] Talreja, R., 2016. On failure theories for composite materials. In *Advanced Methods of Continuum Mechanics for Materials and Structures* (pp. 379-388). Singapore: Springer Singapore.
- [24] Jagadeesh, P., Rangappa, S.M., Suyambulingam, I., Siengchin, S., Puttegowda, M., Binoj, J.S., Gorbatyuk, S., Khan, A., Doddamani, M., Fiore, V. and Cuadrado, M.M.M., 2023. Drilling characteristics and properties analysis of fiber reinforced polymer composites: a comprehensive review. *Heliyon*, 9(3).
- [25] Pecat, O. and Brinksmeier, E., 2014. Low damage drilling of CFRP/titanium compound materials for fastening. *Procedia Cirp*, 13, pp.1-7.