Development of a Multi material 1D Finite Element Model for Stress Analysis and Failure Prediction

A project report submitted in the fulfillment of the requirement for the course.

ME-685 APPLIED NUMERICAL METHODS

Professor

DR. ABHISHEK SARKAR

Department of Mechanical Engineering

Submitted by

Suresh Ragireddy 231010073

Sirisha Jangam 231010031

First Year Postgraduates

Computational Mechanics

Department of Aerospace Engineering, IIT Kanpur

Table of Contents

1 INTRODUCTION TO MULTI-MATERIAL COMPOSITE ANALYSIS 4

1.1 COMPOSITE MATERIAL PROPERTIES 4
1.1.1 THEORETICAL MODELS FOR EFFECTIVE PROPERTY CALCULATION 4
1.2 FINITE ELEMENT METHOD (FEM) FOR STRESS ANALYSIS 5
1.2.1 DIFFERENTIAL EQUATION TO WEAK FORM 6
1.2.2 MATRIX ASSEMBLY: 6
1.2.3 BOUNDARY CONDITIONS AND SOLVERS: 7
1.3 DAMAGE MECHANISM 7
1.3.1 DAMAGE MODELS 7
1.3.2 ANALYSIS OF FAILURE 7

2 RELEVANCE OF PROBLEM IN ENGINEERING 8

2.1.1 IMPORTANCE IN ENGINEERING 8
2.1.2 CHALLENGES WITH COMPOSITES 8

3 PROJECT OBJECTIVES AND OUTCOMES 8

3.1.1 ANALYZING STRESS 8
3.1.2 ENSURING SAFETY 8

4 APPROACH TO THE PROBLEM. 9

5 MATHEMATICAL MODELLING 10

5.1 FLOWCHART TO DETERMINE MATRIX FRACTURE: 16

6 RESULTS AND DISCUSSION 17

6.1 CASE 1 17
6.2 SUMMARY OF CASE 1 19
6.3 CASE 2 20
6.4 SUMMARY OF CASE 2 22
6.5 CASE -3 23
6.6 SUMMARY OF CASE 3 25
6.7 STANDARD MECHANICS 26

7 MATLAB CODE: 26

8 CONCLUSION OF THE PROJECT REPORT 28

9 SCOPE FOR FUTURE WORK 28

10 REFERENCES 29

1 Introduction to Multi-Material Composite Analysis

Background

Composite materials, particularly those involving high-performance fibers like AS4 carbon fiber embedded in a polymer matrix such as epoxy, are pivotal in several engineering applications due to their advantageous strength-to-weight ratios and customizable properties. The behavior of these materials under various loading conditions is complex due to the heterogeneous and anisotropic nature of their constituents.

Objective

The objective of this project is to develop and implement a finite element model that accurately predicts the behavior and failure mechanisms of a two-material composite system under specified loads. This will involve the integration of material mechanics, finite element methodology, and damage mechanics to provide a holistic view of the stress distribution and potential failure locations within the composite.

1.1 Composite Material Properties

AS4/Epoxy Composite

The choice of AS4 carbon fibers and epoxy resin is driven by their common application in aerospace and automotive industries where high strength and durability are required. The fibers provide high tensile strength and stiffness, while the epoxy matrix distributes stress across fibers and protects them from environmental effects.

1.1.1 Theoretical Models for Effective Property Calculation

To determine the effective mechanical properties of the composite, various theoretical and empirical models can be applied. Each method offers different insights based on the scale of the material structure it considers:

Rule of Mixtures - Provides a first approximation of composite properties based on the volume fraction of each constituent.

Halpin-Tsai Equations - Useful for composites with randomly oriented short fibers, enhancing the rule of mixtures by considering the aspect ratio of fibers.

Micromechanical Models - These models, including the Mori-Tanaka method, delve deeper by considering the interaction between individual fibers and the matrix.

Finite Element Analysis - FEA allows for the detailed modeling of composite microstructures to predict how microscale features influence overall material properties.

Eshelby’s Inclusion Theory - Applies to scenarios where the inclusion shape and properties significantly differ from the matrix, influencing the stress field.

Using the rule of mixtures for initial calculations offers a balance between accuracy and computational efficiency, which is crucial for large-scale simulations:

$$E_{\text{composite}} = V_f E_f + (1 - V_f) E_m$$

$$\nu_{\text{composite}} = V_f \nu_f + (1 - V_f) \nu_m$$

1.2 Finite Element Method (FEM) for Stress Analysis

The Finite Element Method (FEM) is a powerful computational technique used to approximate the solutions to complex problems in engineering and physical sciences. It converts problems from their continuous infinite-dimensional form into a discrete form, solvable by numerical methods. This method is particularly effective for stress analysis in materials where exact solutions are not feasible.

Theoretical Foundation FEM is grounded in the need to solve the partial differential equations (PDEs) that describe the behavior of physical systems. For structural analysis, these equations are typically derived from the momentum balance, relating stresses, strains, and displacements within the material under external loads.

FEM uses Galerkin's method to transform these PDEs into their weak forms. This method involves multiplying the differential equation by a test function and integrating over the domain. Here are the critical steps and equations I'm employing:

1. Weak Formulation: - The primary goal in FEM is to derive the weak form of the governing differential equations. This starts with the strong form, e.g., the equilibrium equation:

$$\nabla \cdot \sigma + f = 0$$

where σ represents the stress tensor, and f the body forces.

2. Galerkin's Weighted Residual Method: - Applying Galerkin's approach, the strong form is integrated against a set of weighting functions w_i, which are typically chosen to be the shape functions N_i that also interpolate the displacement field within the element:

$$\int_{\Omega} w_i \, (\nabla \cdot \sigma + f) \, dV = 0$$

- This leads to the fundamental FEM equation after applying integration by parts and considering boundary conditions:

$$\int_{\Omega} \epsilon \, (w_i) \colon \sigma(u) \, dV = \int_{\Omega} w_i \cdot f \, dV + \int_{\partial\Omega} w_i \cdot t \, dA$$

where $\epsilon(w_i)$ is the strain in the weighting function, u is the displacement field, and t represents tractions on the boundaries.

Alternatively

1.2.1 Differential Equation to Weak Form

addressing the strong form of a differential equation which, for a one-dimensional case, can be represented as:

$$p(x)\frac{d^2u}{dx^2} + q(x)\frac{du}{dx} + r(x)u = f(x)$$

Here, $p(x)$, $q(x)$, and $r(x)$ are coefficients that can vary with position, u is the unknown function (e.g., displacement), and $f(x)$ represents any source terms (e.g., forces). This equation encapsulates the essence of the physical problem, from structural deformation to heat transfer.

Galerkin's Method and Integration by Parts To convert the strong form into its weak form, the Galerkin's Method involves multiplying by a test function ω and integrating over the domain. This approach not only simplifies the problem but also ensures that the solution satisfies the integral form of the equation across the entire domain:

$$\int (\omega p u'' + \omega q u' + \omega r u)dx = \int \omega f dx$$

To fulfill the requirement of the weak form, the highest derivative term is integrated by parts:

$$\int p u'' \omega dx = [p u' \omega] - \int p u' \omega' dx$$

Subtracting and adding $pu'\omega'$ from equation leads to:

$$\int [(-pu'\omega' + pu'\omega' + qu'\omega + ru\omega)dx = \int f\omega dx - [pu'\omega]$$

Formulation of the Problem in Matrix Form FEM requires the representation of the weak form in matrix notation for computational purposes. If we approximate the solution u by a finite series $\tilde{u} = \sum a_i \omega_i$, where ω_i are the basis functions, we can express the system of equations in matrix form that defines the relationship between the nodal displacements and the forces:

$$\int \{\left[-p\sum a_i {\omega_i}' {\omega_j}' - p'\sum a_i {\omega_i}' \omega_j + q\sum a_i {\omega_i}' \omega_j + r\sum a_i \omega_i \omega_j\right]\}dx$$

Application in Composite Materials In the context of composite materials, these equations are further complicated by the anisotropic and heterogeneous nature of the medium. The coefficients $p(x)$, $q(x)$, and $r(x)$ will not only vary spatially but also change depending on the orientation and properties of the fibers and matrix.

1.2.2 Matrix Assembly:

- The global stiffness matrix K and force vector F are assembled from the element matrices and vectors using the connectivity data of the mesh:

$$KU = F$$

- Each element contributes to the global system according to its stiffness properties and the connectivity of the nodes.

1.2.3 Boundary Conditions and Solvers:

- Implementing the correct boundary conditions is critical for accurate simulations. This typically involves specifying displacements or forces at certain nodes in the mesh. - The system of linear equations is then solved using numerical solvers which may be direct (e.g., LU decomposition)

1.3 Damage Mechanism

1.3.1 Damage Models

To predict where and how failure might occur in a composite, various damage models are examined. The selected LaRC04 criterion, which is a physically based damage model, considers the stress states and fracture mechanics to evaluate failure:

$$\sigma_{\max} = \max\left(\sigma_{xx}, \sigma_{yy}, \sigma_{zz}\right), \quad \tau_{\max} = \max\left(\tau_{xy}, \tau_{yz}, \tau_{zx}\right)$$

$$\text{if } \sigma_{\max} > X_T \text{ or } \tau_{\max} > S_T \text{ then failure occurs}$$

1.3.2 Analysis of Failure

The failure analysis involves determining the stress distribution in the composite and identifying the points at which these stresses exceed the material strength. This analysis considers the variability in fiber orientation and loading conditions.

2 Relevance of Problem in Engineering

2.1.1 Importance in Engineering

Structural Analysis: In engineering, checking how structures like beams and bridges react to forces is crucial. This helps ensure they are safe and reliable.

Use of Composites: Composites are materials made from two or more different substances. They are popular in industries like aerospace and automotive because they are strong but lightweight. Understanding how they behave under different conditions is important for safety.

2.1.2 Challenges with Composites

Complex Materials: Composites are complex because they combine different materials, each behaving differently under stress. This makes predicting their behavior challenging.

Need for Advanced Techniques: Traditional methods often can't fully capture how these complex materials behave. This project uses a sophisticated tool called Finite Element Method (FEM) to get a more accurate picture.

3 Project Objectives and Outcomes

3.1.1 Analyzing Stress

Understanding Stress Distribution: The main goal is to see how stress (force per area) spreads across a composite material when forces are applied. This helps identify which parts of the material might fail.

Using FEM: The project uses a versatile FEM code that can analyze any 1D mechanical .This makes it a powerful tool for engineers.

3.1.2 Ensuring Safety

Maximizing Safety: By knowing where the highest stress occurs in the material, engineers can design safer and more reliable structures.

Improving Material Selection: The information from this project can help in choosing the right materials and designs, optimizing both cost and performance.

Predictive Maintenance: Identifying where and how materials might fail helps in planning maintenance to fix problems before they lead to failure.

This project is not just about academic project; it has real-world applications that can make engineering designs safer and more cost-effective. By improving how we analyze materials, especially composites, this project helps in building better, safer products and structures.

4 Approach to the problem.

Section-1

We derive the governing equation for our problem by considering a small segment of the multi-material composite bar and utilizing Taylor series for solution. We incorporate principles of continuum mechanics, compatibility, and equilibrium conditions, while integrating material properties into the equation. By discretizing the problem domain into finite elements, we approximate the displacement field using shape functions and nodal values.

Section-2

we compare the coefficients of the derived governing equation with the generalized differential governing equation. We solve displacements using a Finite Element Method (FEM) solver without relying on inbuilt functions such as integration and differentiation, employing numerical techniques like Lagrange interpolation and Gauss quadrature.

Section-3

The problem performed analyses for all 4 models with different Fibre orientations with each model being considered for 3 different BCs.

Section-4

We take maximum stress value among all nodes and conduct damage analysis.

By systematically addressing these cases, we aim to gain insights into the behavior of the multi-material composite bar under different boundary conditions and loading scenarios, facilitating informed decision-making in structural analysis and design.

5 Mathematical Modelling

Section -1

FEM Solver for solving Any Differential equation.

$$a\frac{d^2u}{dx^2} + b\frac{du}{dx} + c(x)u = f(x)$$

For Solving this, we developed a 1D FEM Solver, which will work for any DE of general form shown above.

Here a and b are constants.

c and f are functions of x

For the problem, we considered multi-material bar or composite.

Fibre = AS4
Matrix = epoxy

Handbook values of these Material

Using effective properties of the composite material As4/epoxy. We took E11 value.

There can be many ways to calculate the effective properties of the Material like

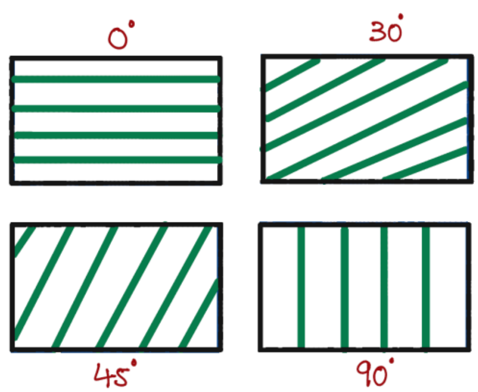
1 Standard Mechanics

2 Mathematical modelling

3 RVE .... etc.

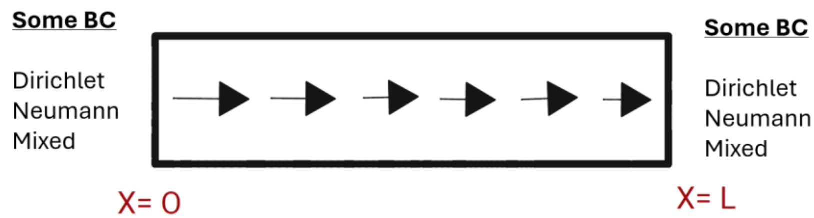
Among them, the simplest and quickest method is standard mechanics approach, the handbook values of AS4 and epoxy were chosen.

The next step, in this analysis is to decide the fibre orientation, we considered 4 models for this analysis.



Fibre orientations: $\theta = 0°, 30°, 45°, 90°$

Next step was deriving the Mathematical model for 1D bar, i.e. Governing equation of the bar.



Here $g(x)$ is the traction term along the bar. Considering a small element of the length Δx, the equilibrium of the element is:

$$\sum F_x = 0$$
$$F(x+\Delta x) - F(x) = g(x)\Delta x$$

Taylor series expansion of $F(x+\Delta x)$

$$F(x+\Delta x) = F(x) + \frac{d}{dx}F(x) \cdot \Delta x + \frac{d^2F(x)}{dx^2}\Delta^2 + \cdots.$$
$$F(x+\Delta x) - F(x) = \frac{d}{dx}F(x)\Delta x + H \cdot \underbrace{O \cdot T(\Delta x^2)}_{\text{Ignored.}}$$
$$\frac{d}{dx}F\Delta x = g(x)\Delta x$$
$$\frac{dF}{dx} = g(x)$$

We know that $F = \sigma A$

$$F_{xx} = \sigma_{xx}A$$
$$= AE\varepsilon_{xx}$$
$$F_{xx} = AE\frac{du}{dx}$$

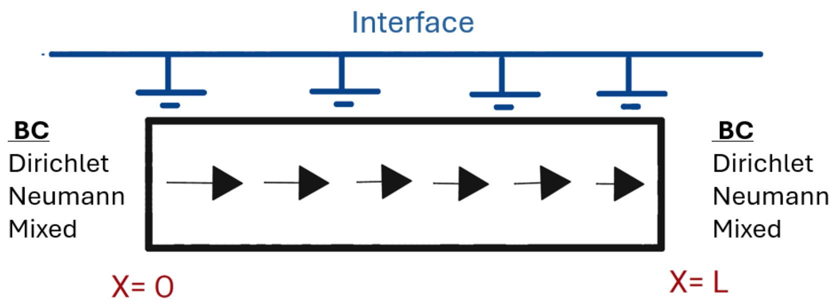
substituting in the above equation, we get:

$$\frac{d}{dx}AE\frac{du}{dx} = g(x)$$

If we consider " AE " to be constant, then we get:

$$AE\frac{d^2u}{dx^2} = g(x)$$

Similarly, if we consider, damping is present we get:



$$AE\frac{d^2u}{dx^2} + c(x)u = g(x)$$

This is the governing equation.

Here,

$g(x)$: Traction

$c(x)$: damper

Section-2

When we compare this equation with the FEM equation which we developed, i.e.

$$\boldsymbol{a\frac{d^2u}{dx^2} + b\frac{du}{dx} + cu = g(x)}$$
$$\boldsymbol{a = AE, b = 0, c = c(x)}$$

A was calculated from Geometry and E_{11} from effective properties.

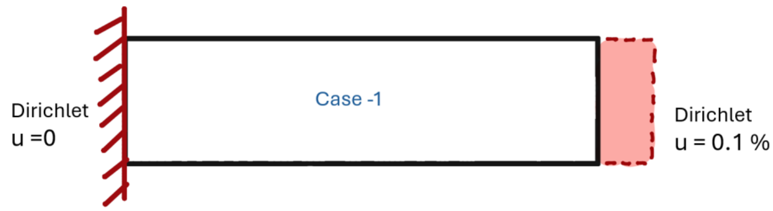
Section-3

The team performed analyses for all 4 models with different Fibre orientations with each model being considered for 3 different BCs,

Therefore, total test cases $= 4 \times 3 = 12$

Case 1:

One end of the bar is fixed (Dirichlet boundary condition), while at other end small deformation of 0.1 % is applied force is applied (Dirichlet boundary condition) at the other end (Neumann boundary condition).

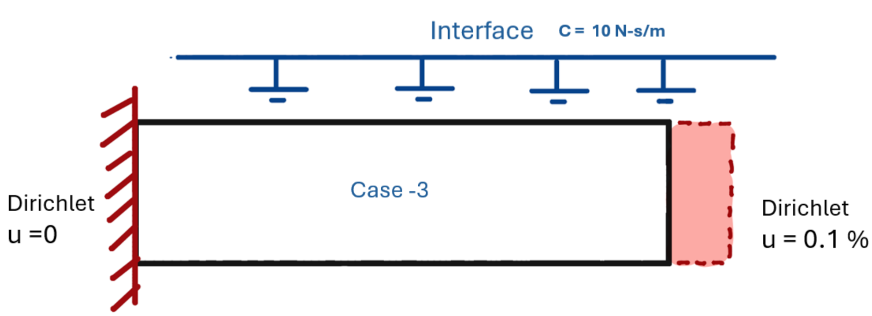


Case 2:

Both ends of the bar are fixed (Dirichlet boundary condition), at the other end (Neumann boundary condition) AE*$\frac{du}{dx}$

Case 3:

One end of the bar is fixed (Dirichlet boundary condition), while at other end small deformation of 0.1 % is applied force is applied (Dirichlet boundary condition) at the other end (Neumann boundary condition).[same as case 1] with addition damping terms, that interface bonding in composite c = 10 N-s/m, here Is the interface value or epoxy resistance, because of this stress will be more



From FEM solver, we got u and $\frac{du}{dx}$ at every node.

To get the solution closer or with less error we used a larger number of elements and higher order interpolation function.

Using $\frac{du}{dx}$ we calculated stresses at every node:

$$\sigma = c : \varepsilon$$
$$\sigma_{xx} = E_{11}\varepsilon_{xx}$$
$$= E_{11}\frac{du}{dx}$$

Section-4

We take maximum stress value among all nodes and conduct damage analysis.

Using damage model, we found the fracture plane ϕ, but for damage analyses we need stresses to be in material co-ordinate system.

$$\sigma_{123} = [T]\sigma_{xyz}$$

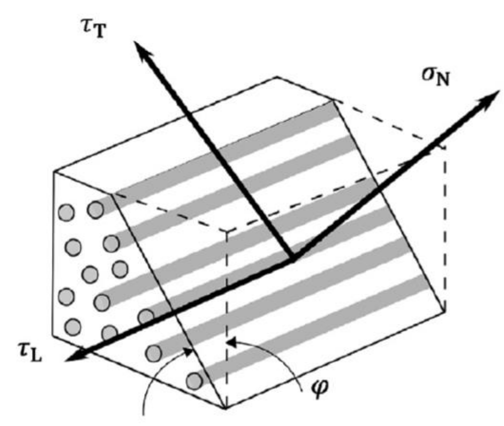
Here,

$[T]$ - Transformation Matrix

123 - Material CS

$$XYZ - Global\ CS$$

The damage model we are developing focuses on transverse matrix cracking. We will implement the LaRC failure criterion to assess matrix fractures and determine whether the AS4/Epoxy material will fail. Additionally, we will identify the potential fracture plane angle orientation with respect to the ply normal.



In 3D, the stress components acting on the fracture plane (oriented at an angle φ to the ply normal) can be obtained from the material frame stresses, by stress transformation (rotation about the 1 axis by angle φ).

$$\sigma_N = \frac{\sigma_2 + \sigma_3}{2} + \frac{\sigma_2 - \sigma_3}{2}\cos 2\varphi + \sigma_4 \sin 2\varphi$$

$$\tau_T = -\frac{\sigma_2 - \sigma_3}{2}\sin 2\varphi + \sigma_4 \cos 2\varphi$$

$$\tau_L = \sigma_6 \cos\varphi + \sigma_5 \sin\varphi$$

The failure criteria under pure transverse compression can therefore be expressed as:

$$\frac{|\tau_T|}{S_T - \eta_T \sigma_n} = 1$$

Notice that the presence of a positive σ_n reduces the effective transverse shear strength (the denominator becomes smaller) and causes failure earlier. The above analysis is done when $\tau_L = 0$ (pure transverse compression case). When $\tau_L \neq 0$, Puck and Sch**u**rmann proposed the following criteria (which is also retained in LaRC04) when stress on the fracture plane is compressive:

$$\left(\frac{\tau_T}{S_T - \eta_T \sigma_n}\right)^2 + \left(\frac{\tau_L}{S_L - \eta_L \sigma_n}\right)^2 = 1 \quad \sigma_n < 0$$

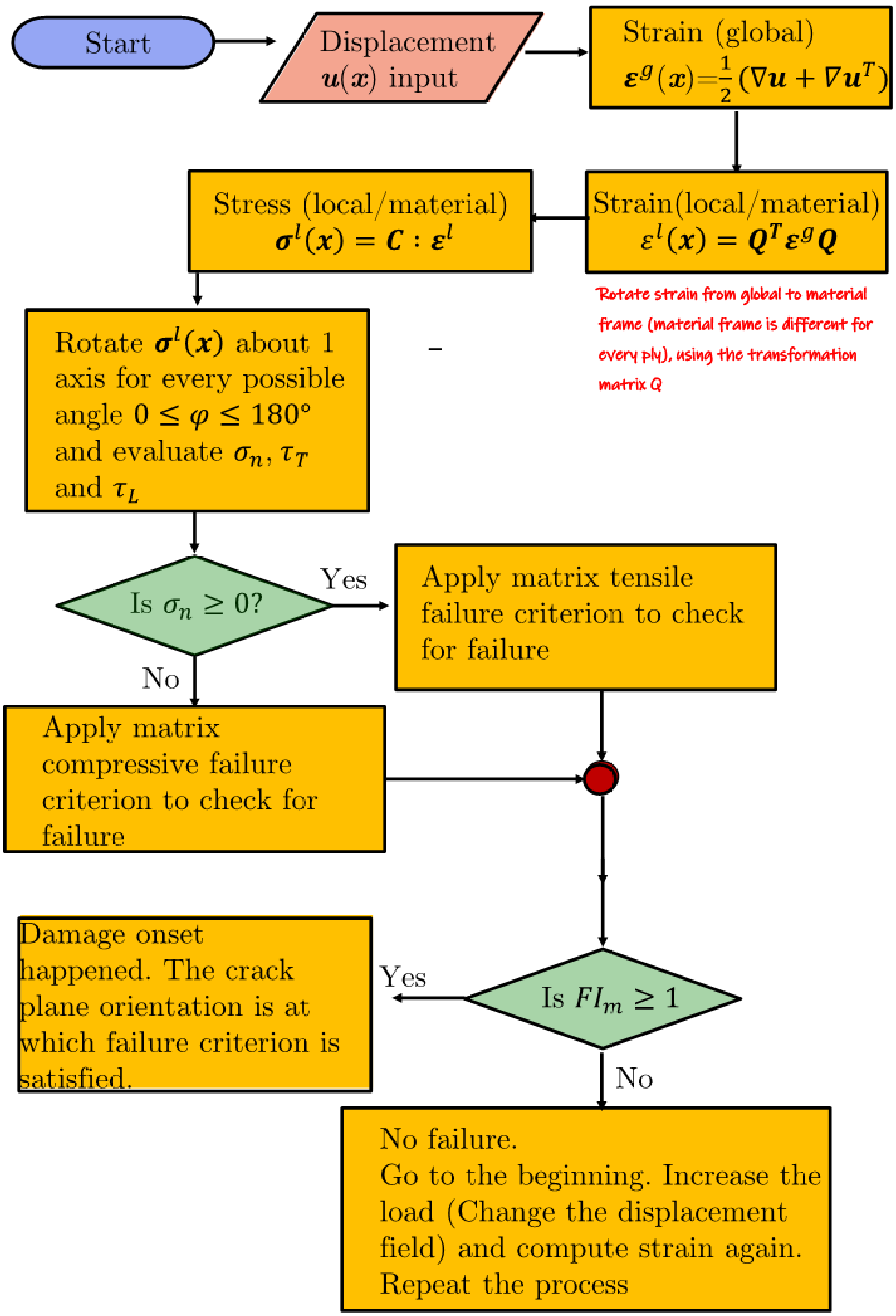
where S_L is the in-plane shear strength (material property), η_L is the friction angle corresponding to longitudinal shear.

Under transverse tension, we use a different criterion. In Transverse tension ($\sigma_2 \geq 0$), the normal stress on the fracture plane σ_n is greater than zero, i.e., it is tensile. ($\sigma_n > 0$) Under such a condition, we can use the following criterion to detect fracture:

$$\left(\frac{\sigma_n}{Y_T}\right)^2 + \left(\frac{\tau_T}{S_T - \eta_T \sigma_n}\right)^2 + \left(\frac{\tau_L}{S_L - \eta_L \sigma_n}\right)^2 = 1 \quad \sigma_n \geq 0$$

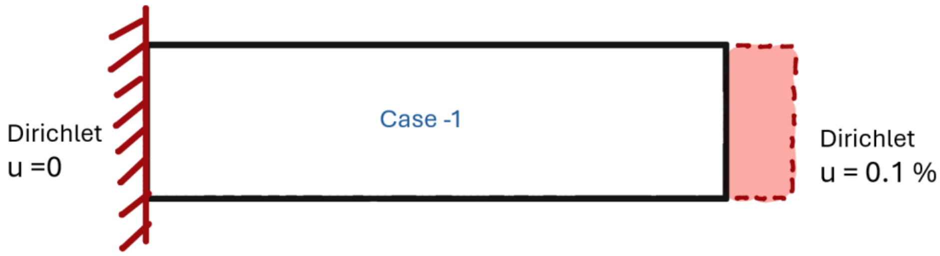
Where Y_T is the transverse tensile strength of the material (Under pure transverse tension, $\sigma_2 > 0$, all other stresses are 0, the fracture angle is exactly 0°. The stress at which this fracture happens is known as 'transverse tensile strength' Y_T)

5.1 Flowchart to determine matrix fracture:



6 Results and discussion

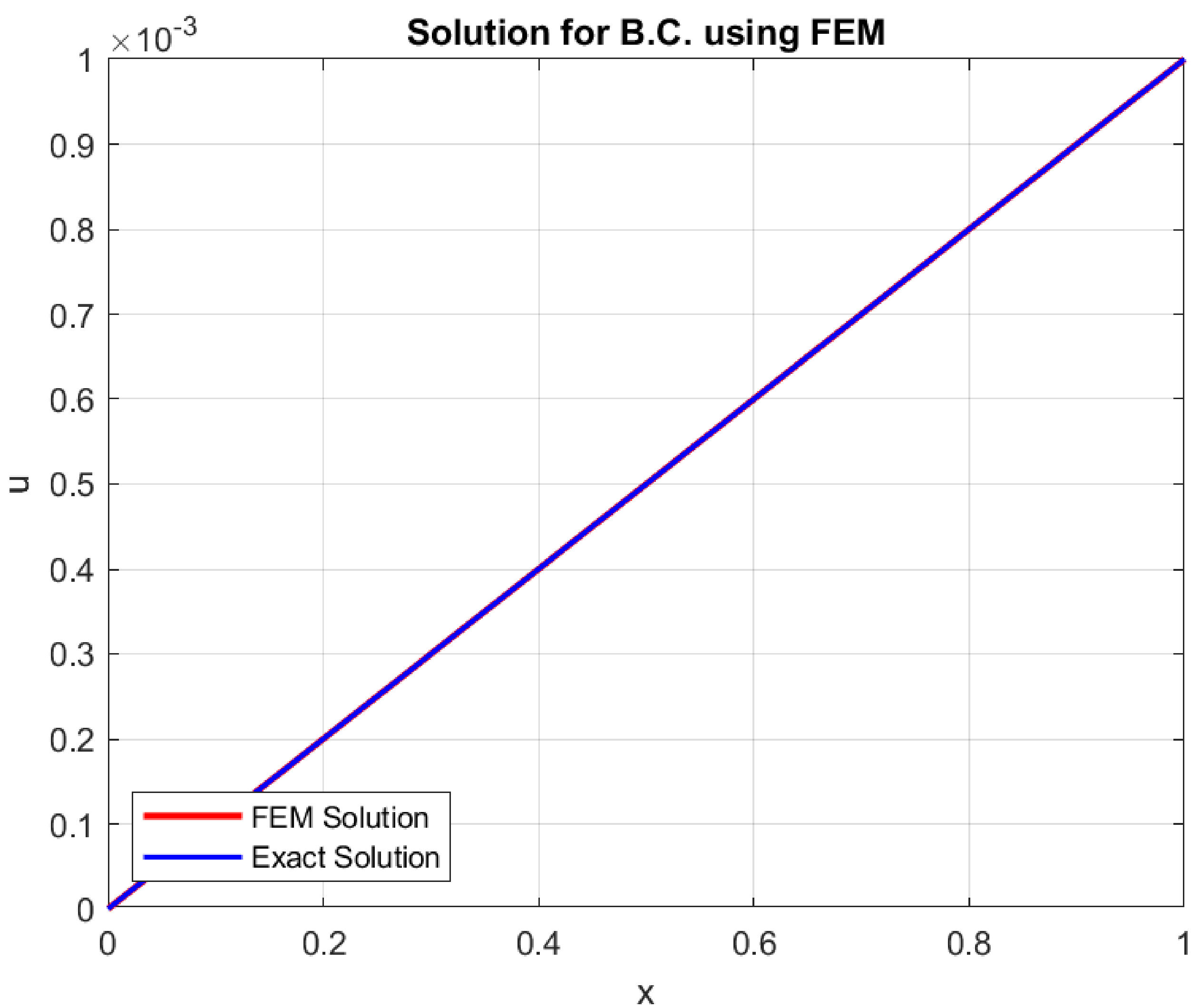
6.1 Case 1



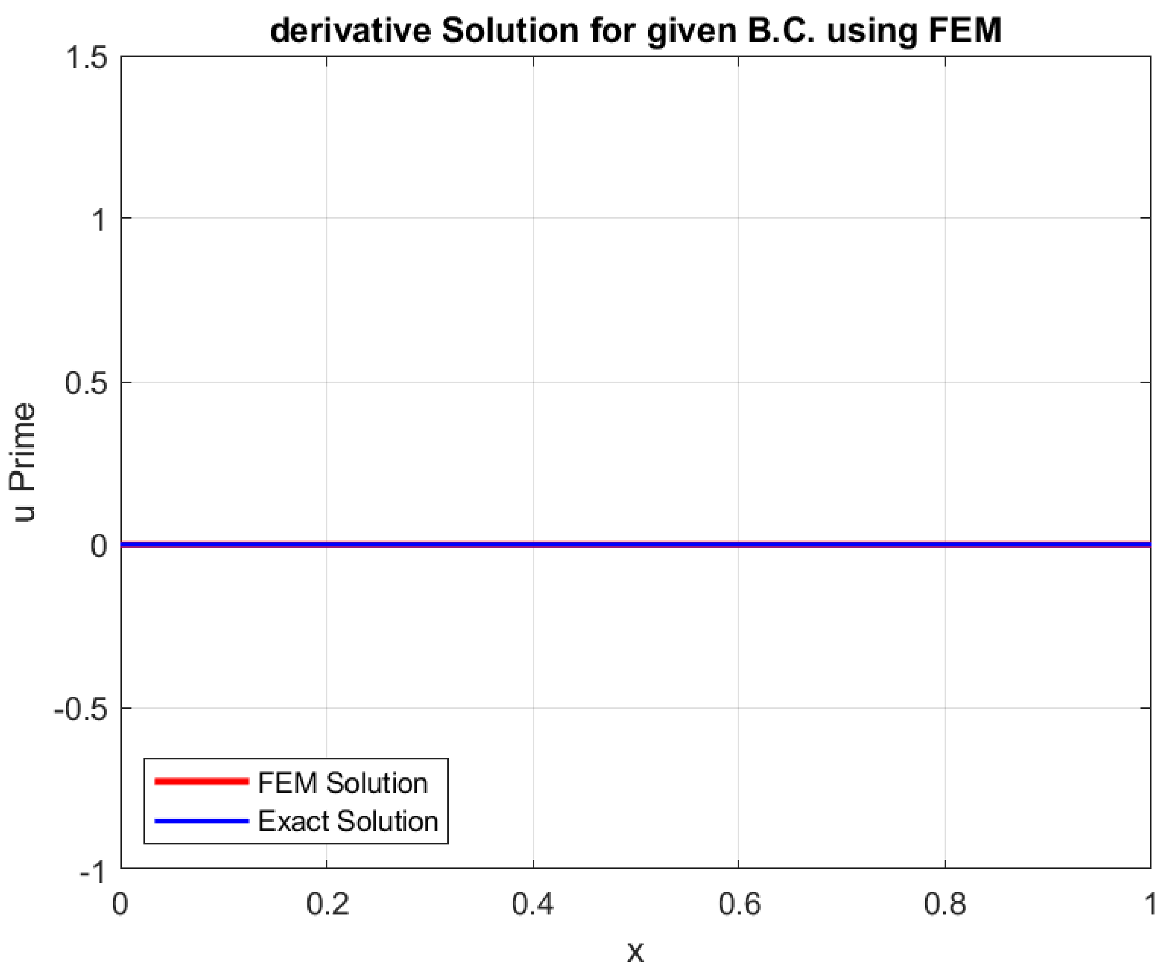
Exact Solution

$$u_{\text{exact}} = x/1000$$

$$\frac{d}{dx}u_{\text{exact}} = 1/1000$$



Nodal Location	**u**	$\frac{du}{dx}$	σ_{xx}
0	0	0.001	142.84
0.1	0.0001	0.001	142.84
0.2	0.0002	0.001	142.84
0.3	0.0003	0.001	142.84
0.4	0.0004	0.001	142.84
0.5	0.0005	0.001	142.84
0.6	0.0006	0.001	142.84
0.7	0.0007	0.001	142.84
0.8	0.0008	0.001	142.84
0.9	0.0009	0.001	142.84
1	0.001	0.001	142.84



Maximum Stress (σ_{XX}) Value: 142.84 MPa

Location of Maximum Stress: 0.mm

1) **Fibre Orientation of:** 0° Degree

- **Damage Assessment**: No damage happened.

2) **Fibre Orientation of**: 30° Degree

- **Damage Assessment**: No damage happened.

3) **Fibre Orientation of:** 45° Degree

- Damage occurred along the fracture planes, which are oriented at an angle ϕ relative to the ply normal, at the following ϕ angles:

$$\text{Angles: } 1^\circ \text{ to } 83^\circ, \quad 97^\circ \text{ to } 180^\circ$$

4) **Fibre Orientation of:** 90° Degree

- Damage occurred along the fracture planes, which are oriented at an angle ϕ relative to the ply normal, at the following ϕ angles:

$$\text{Angles: } 1^\circ \text{ to } 85^\circ, \quad 95^\circ \text{ to } 180^\circ$$

6.2 Summary of Case 1

Fibre Orientation	Boundary Conditions	Max Stress (MPa)	Location of Max Stress	Damage Assessment	Damage Angles
0°	Dirichlet	142.84	0 mm	No damage	N/A
30°	Dirichlet	142.84	0 mm	No damage	N/A
45°	Dirichlet	142.84	0 mm	Damage	1° to 83°, 97° to 180°
90°	Dirichlet	142.84	0 mm	Damage	1° to 85°, 95° to 180°

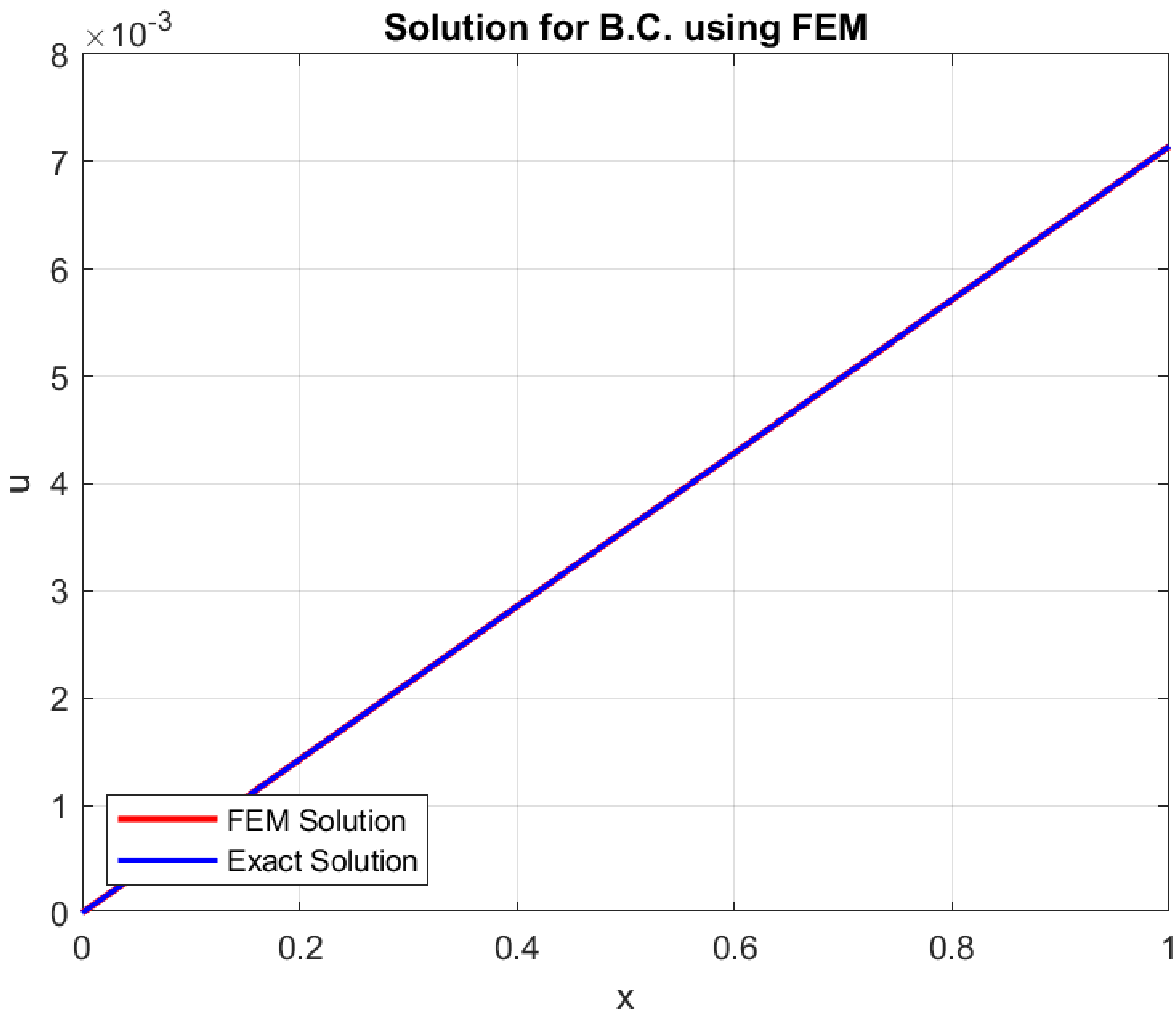
6.3 Case 2

Exact Solution

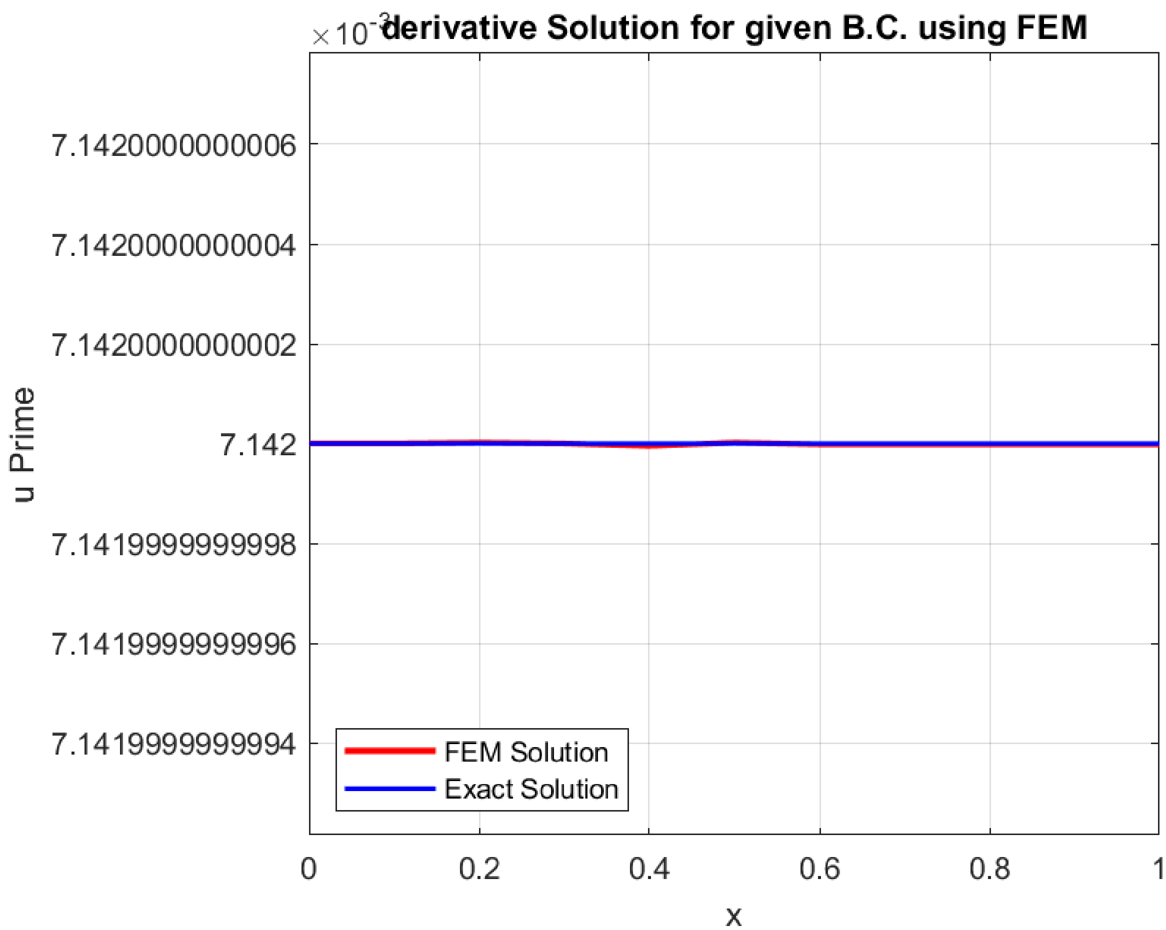
$$u_{\text{exact}} = \frac{8234165385902101 \cdot x}{1152921504606846976}$$

$$u_{\text{exact}} = 0.0071 \cdot x$$

$$\frac{d}{dx} u_{\text{exact}} = 0.0071$$



Nodal Location	**u**	$\frac{du}{dx}$	σ_{xx}
0	0	0.007142	1020.2
0.1	0.000714	0.007142	1020.2
0.2	0.001428	0.007142	1020.2
0.3	0.002143	0.007142	1020.2
0.4	0.002857	0.007142	1020.2
0.5	0.003571	0.007142	1020.2
0.6	0.004285	0.007142	1020.2
0.7	0.004999	0.007142	1020.2
0.8	0.005714	0.007142	1020.2
0.9	0.006428	0.007142	1020.2
1	0.007142	0.007142	1020.2



Maximum Stress (σ_{XX}) Value: 1020.1633 MPa

Location of Maximum Stress: 0.2 mm

1) **Fibre Orientation of**: 0° Degree

- Damage Assessment: No damage happened.

2) **Fibre Orientation of**: 30° Degree

- Damage occurred along the fracture planes, which are oriented at an angle ϕ relative to the ply normal, at the following ϕ angles:

$$\text{Angles: } 1^\circ \text{ to } 81^\circ, \quad 99^\circ \text{ to } 180^\circ$$

3) **Fibre Orientation of**: 45° Degree

- Damage occurred along the fracture planes, which are oriented at an angle ϕ relative to the ply normal, at the following ϕ angles:

$$\text{Angles: } 1^\circ \text{ to } 83^\circ, \quad 97^\circ \text{ to } 180^\circ$$

4) **Fibre Orientation of**: 90° Degree

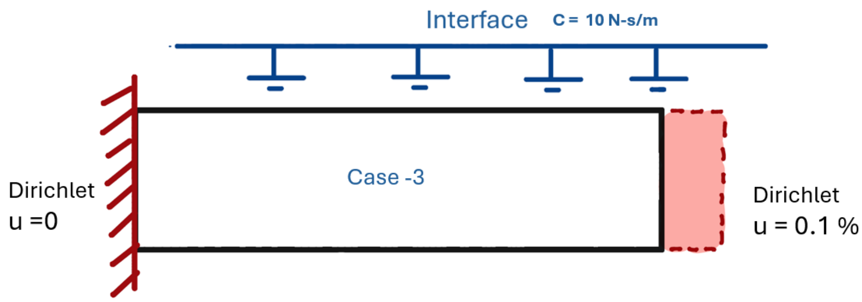
- Damage occurred along the fracture planes, which are oriented at an angle ϕ relative to the ply normal, at the following ϕ angles:

$$\text{Angles: } 1^\circ \text{ to } 85^\circ, \quad 95^\circ \text{ to } 180^\circ$$

6.4 Summary of Case 2

Fibre Orientation	Boundary Conditions	Max Stress (MPa)	Location of Max Stress	Damage Assessment	Damage Angles
0°	Dirichlet and Neumann	1020.1633	0.2 mm	No damage	N/A
30°	Dirichlet and Neumann	1020.1633	0.2 mm	Damage	1° to 81°, 99° to 180°
45°	Dirichlet and Neumann	1020.1633	0.2 mm	Damage	1° to 83°, 97° to 180°
90°	Dirichlet and Neumann	1020.1633	0.2 mm	Damage	1° to 85°, 95° to 180°

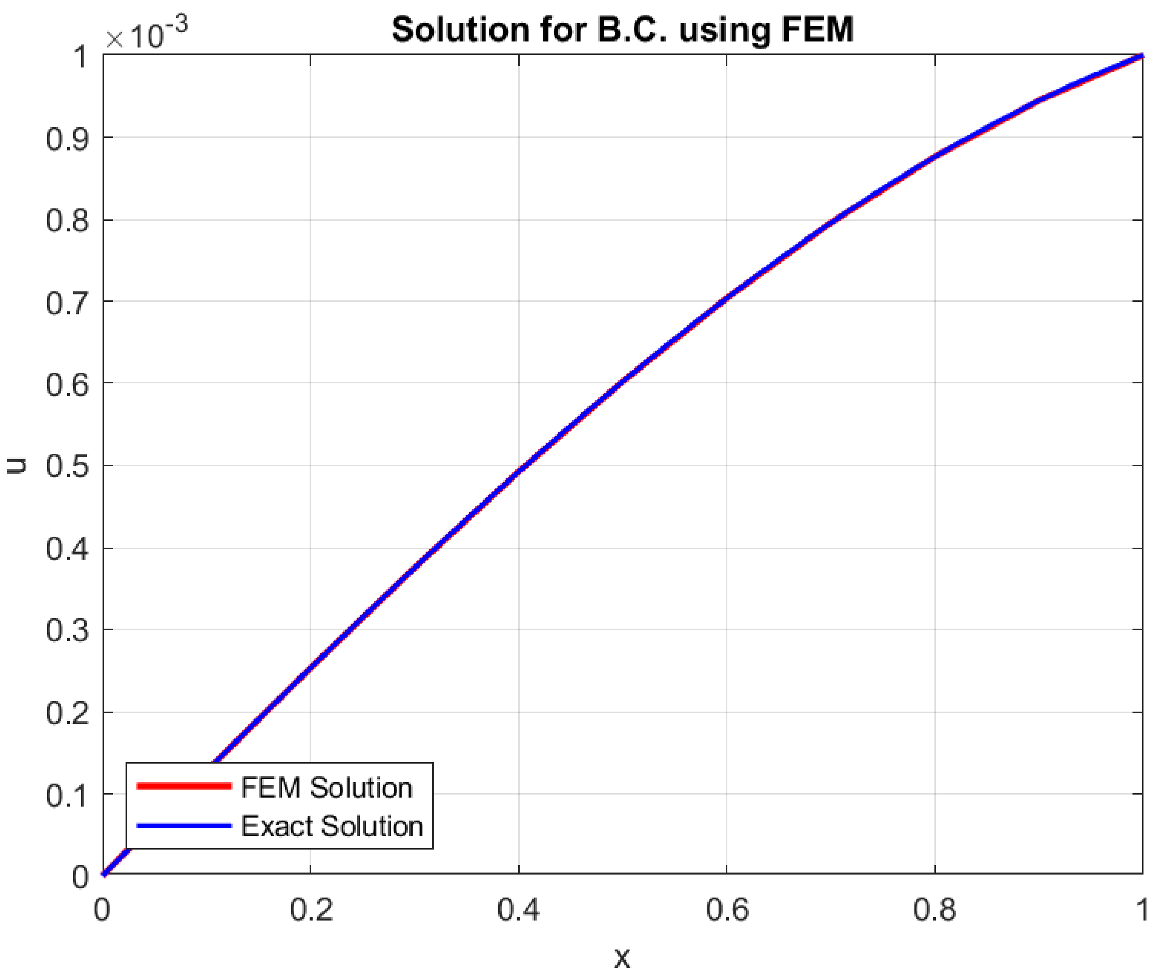
6.5 Case -3



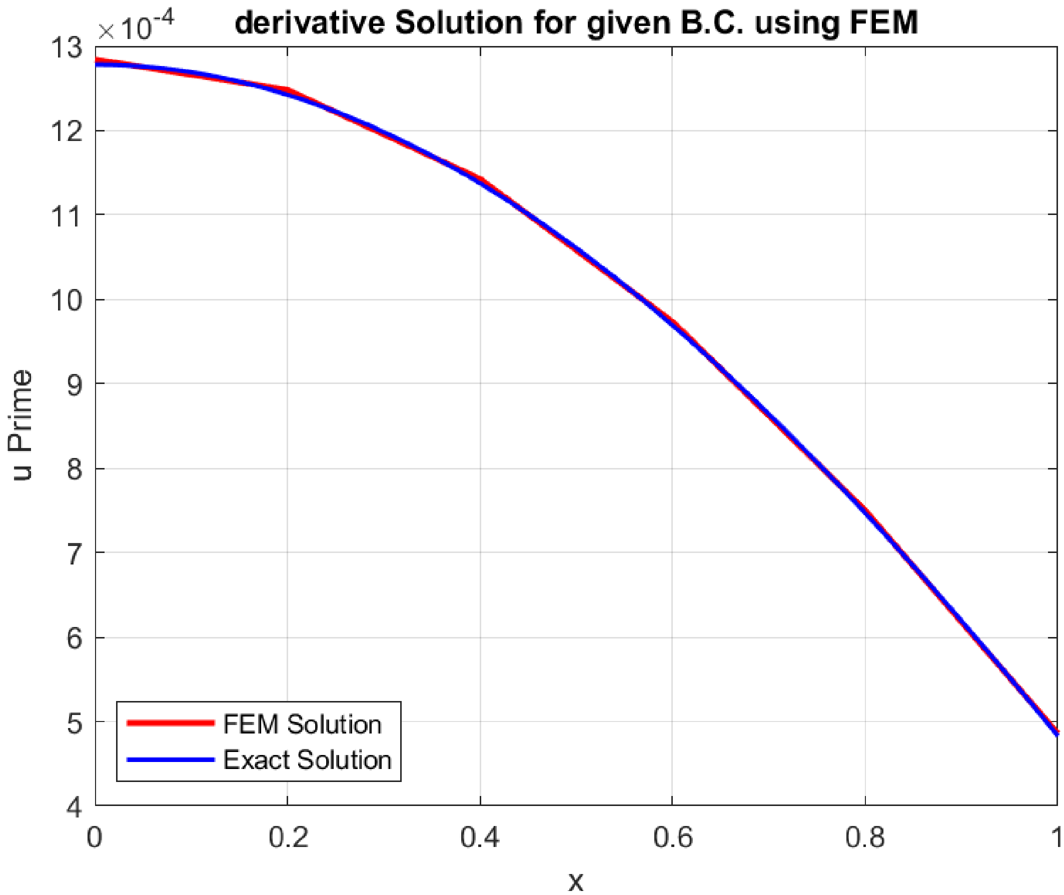
Exact Solution

$$u_{\text{exact}} = \frac{\sin\left(\dfrac{\sqrt{7142}\cdot x}{3571}\right)}{1000\cdot\sin\left(\sqrt{7142}/3571\right)}$$

$$\frac{d}{dx}u_{\text{exact}} = \frac{\sqrt{7142}\cdot\cos\left(\dfrac{50.\sqrt{7142}\cdot x}{3571}\right)}{71420\cdot\sin\left(50.\sqrt{7142}/3571\right)}$$



Nodal Location	**u**	$\frac{du}{dx}$	σ_{xx}
0	0	0.001284	183.4
0.1	0.000128	0.001266	180.86
0.2	0.000253	0.001248	178.27
0.3	0.000375	0.001196	170.78
0.4	0.000492	0.001143	163.2
0.5	0.000602	0.001058	151.17
0.6	0.000704	0.000973	139.04
0.7	0.000796	0.000862	123.14
0.8	0.000876	0.00075	107.12
0.9	0.000945	0.000618	88.25
1	0.001	0.000486	69.38



Maximum Stress (σ_{XX}) Value: 183.4038 MPa

Location of Maximum Stress: 0 mm

1) **Fibre Orientation** of: 0° Degree

- **Damage Assessment:** No damage happened.

2) **Fibre Orientation of**: 30° Degree

- Damage occurred along the fracture planes, which are oriented at an angle ϕ relative to the ply normal, at the following ϕ angles:

$$\text{Angles: } 1^\circ \text{ to } 34^\circ, \quad 146^\circ \text{ to } 180^\circ$$

3) **Fibre Orientation of**: 45° Degree

- Damage occurred along the fracture planes, which are oriented at an angle ϕ relative to the ply normal, at the following ϕ angles:

$$\text{Angles: } 1^\circ \text{ to } 55^\circ, \quad 125^\circ \text{ to } 180^\circ$$

4) **Fibre Orientation of**: 90° Degree

- Damage occurred along the fracture planes, which are oriented at an angle ϕ relative to the ply normal, at the following ϕ angles:

$$\text{Angles: } 11^\circ \text{ to } 66^\circ, \quad 114^\circ \text{ to } 177^\circ$$

6.6 Summary of Case 3

Fibre Orientation	Boundary Conditions	Max Stress (MPa)	Location of Max Stress	Damage Assessment	Damage Angles
0°	Dirichlet with interface	183.4038	0 mm	No damage	N/A
30°	Dirichlet with interface	183.4038	0 mm	Damage	1° to 34°, 146° to 180°
45°	Dirichlet with interface	183.4038	0 mm	Damage	1° to 55°, 125° to 180°
90°	Dirichlet with interface	183.4038	0 mm	Damage	11° to 66°, 114° to 177°

6.7 Standard mechanics

I have calculated the effective properties of As-4/epoxy material and, for comparison, I have included the handbook values as a reference.

Property	Computed Value MPa	Table Value MPa
E11	142840	126000
E22	11172	11000
G12	4121.2	6600
nu12	0.264	0.28

7 MATLAB CODE:

The MATLAB codes for this complete project are provided in the folder named "***ProjectFiles***". I have attached these files in a Zip folder named "***231010079_ME685_Project.zip***". To run this code, I have created several subroutines listed below, which you can find in the same folder.

My project consists of 4 parts, each handled by specific subroutines as detailed below:

Section 1: Calculating Composite Effective Properties

For calculating the effective properties of the composite material, I have used the subroutine:

StdMechanics: This function calculates the stiffness and other mechanical properties of the composite based on the volume fractions and properties of the individual components.

Section 2: Finite Element Method (FEM) Processing

The finite element analysis and setup are handled through various subroutines that manage the assembly and solution of the FEM equations:

adjust_e: Adjusts the element properties based on boundary conditions when point is applied.

choose_e: Selects appropriate elements for processing based on criteria.

FEM_Processor: Main function for processing the finite element model.

find_indices: Identifies the indices for global assembly.

GaussQuadrature: Handles the numerical integration within the finite elements.

GlobalLocation: Determines the global positions of nodes and elements.

lagrangeInterpolation: Used for interpolating values within elements.

SHAPE: Generates shape functions for the elements.

Section 3: Finding Stress

This section involves the calculation of stress at various points within the material:

T: Transforms stress components to different coordinate systems.

uprime: Calculates derivatives of displacement fields necessary for stress computation.

Section 4: Damage Model

The damage model assesses potential failure and damage locations based on the stress analysis:

calculate_stresses: Computes the stress state at each point and checks against failure criteria to predict potential damage.

Running the Code:

To run the simulations, ensure all subroutines are in the same directory as the main scripts. Input parameters specific to the composites and loading conditions must be set within the main scripts before execution.

It is essential to note that the live script has been developed and tested on the latest version of MATLAB. However, any potential discrepancies in functionality may arise if an older version is utilized, as I am currently using the latest version for coding and testing purposes

8 Conclusion of the Project Report

This project has successfully developed and implemented a 1D Finite Element Model for analyzing stress and predicting damage in multi-material composite bars under various boundary conditions. The conclusions drawn from this investigation are summarized below:

- **Stress Analysis:** The Finite Element Model effectively calculated the maximum stress values under different loading and boundary conditions. It was observed that stress distribution varies significantly with changes in boundary conditions and applied forces.
- **Numerical Techniques:** Key numerical techniques such as Lagrange interpolation, Gauss quadrature, numerical differentiation, and the Tri-Diagonal Matrix Algorithm (**TDMA**) for matrix inversion were employed. These methods enhanced the accuracy and efficiency of the Finite Element Method solver used to compute displacements and stresses within the composite. Numerical differentiation was crucial for accurately calculating derivative terms needed for the stiffness matrices, while TDMA provided a fast and reliable method for solving the linear systems of equations that arise in FEM.
- **Fibre Orientation Influence:** The results clearly show that the orientation of the fibers significantly impacts the stress distribution and the onset of damage. Fibers oriented at 0° generally showed no damage under the tested conditions, highlighting their high tensile strength along the fiber direction.
- **Damage Assessment:** For fibers oriented at 45° and 90°, damage was consistently observed, indicating that these orientations are more vulnerable under the same loading conditions. The analysis provided detailed insights into the fractured planes and the critical angles at which damage occurs.
- **Effect of Boundary Conditions:** Different boundary conditions led to varied stress responses in the composite bar. For instance, when both ends of the bar were fixed, the maximum stress experienced was significantly higher than when one end was fixed, and the other was subject to a small deformation.
- **Impact of Interface Bond Between Fiber and Matrix:** Introducing interface bond terms to simulate the connection between fiber and matrix significantly affected the stress distribution, illustrating the importance of considering such factors in composite material design and analysis.

9 Scope for Future Work

Looking ahead, there are several areas where further research could enhance the understanding and capabilities of stress analysis and damage prediction in composite materials:

- **Material Variability**: Exploring a wider range of materials and composite configurations could help generalize the findings and improve the model's applicability to a broader set of real-world conditions.
- **3D Modeling:** Extending the model to three dimensions would provide a more comprehensive understanding of stresses and potential damage in practical applications where 1D assumptions may not hold.
- **Thermal Effects:** Incorporating thermal effects into the model could address additional real-world scenarios, such as those involving temperature variations that significantly impact material behavior.
- **Experimental Validation:** Conducting experimental tests to validate the simulation results would strengthen the model's reliability and help refine its predictive accuracy.
- **Dynamic Loading Conditions:** Analyzing the impact of dynamic loads, such as those encountered in automotive and aerospace applications, could provide insights into the fatigue behavior and longer-term durability of composite materials.

By exploring these areas, future work can expand upon the insights gained in this project, contributing to ongoing efforts in enhancing the design and analysis of composite materials for engineering applications.

10 References

1) Chapra, S. (2010). Numerical Methods for Engineers (6th ed.). New York: McGraw-Hill.

2) Herakovich, C. T. (1997). Mechanics of Fibrous Composites. New York: John Wiley & Sons.

3) Pinho, S. T., Darvizeh, R., Robinson, P., Schuecker, C., & Camanho, P. P. (2012). Material and structural response of polymer-matrix fibre-reinforced composites. Journal of Composite Materials, 46(19), 2313-2341.

4) Ji, S., Li, M., Zhou, Y., & Li, J. (2017). Mechanical properties and electrical conductivity of carbon nanotube reinforced aluminum matrix composites. Applied Composite Materials, 24(3), 473-483.

5) Kelly,The Finite Element Method, https://pkel015.connect.amazon.auckland.ac.nz/SolidMechanicsBooks/FEM/index.html