Bimodulus-plastic model for pre-failure analysis of fiber reinforced polymer composites

Boon, Yi Di; Joshi, Sunil Chandrakant; Ong, Lin Seng

2019

Boon, Y. D., Joshi, S. C., & Ong, L. S. (2019). Bimodulus-plastic model for pre-failure analysis of fiber reinforced polymer composites. *Mechanics of Materials*, 134, 18-29.

doi:10.1016/j.mechmat.2019.04.003

<https://hdl.handle.net/10356/106055>

<https://doi.org/213063>;

<https://doi.org/10.1016/j.mechmat.2019.04.003>;

<https://doi.org/213063>

© 2019 Elsevier. All rights reserved. This paper was published in *Mechanics of Materials* and is made available with permission of Elsevier.

Downloaded on 26 Apr 2023 17:22:40 SGT

Bimodulus-Plastic Model for Pre-Failure Analysis of Fiber Reinforced Polymer Composites

Yi Di Boon, Sunil Chandrakant Joshi, Lin Seng Ong

School of Mechanical and Aerospace Engineering, Nanyang Technological University,
Singapore

Corresponding author:

Yi Di Boon, School of Mechanical and Aerospace Engineering, Nanyang Technological
University, Singapore 639789

Email: ydboon@ntu.edu.sg

Abstract

Fiber-reinforced polymer composites, such as carbon-epoxy composites, are found to exhibit non-linear behavior when mechanically loaded in the transverse and shear directions. Experimental studies suggest that the non-linear behavior is due to two mechanisms: (i) damage in the matrix in the form of cracks and (ii) yielding of the matrix followed by plastic deformation. In this study, a bimodulus-plastic model that includes these two different damage mechanisms to simulate the non-linearity prior to failure is proposed. The pre-failure, onset of failure and post-failure analysis with the proposed

model is discussed in detail with emphasis put on the transverse and shear components. The process of determining the material properties and parameters required for defining the new model is discussed and demonstrated. The newly developed model is then validated against the experimental results from three-point flexure tests on the composites. The correlation was good showing that the proposed model was able to simulate accurately the non-linear behavior of the composites and thus predict the failure. Finally, the model is applied to a case study on the failure of a composite riser under internal pressure loads.

Keywords

Finite element analysis; non-linear stress-strain; damage mechanism; shear deformation; polymer-matrix composites

1. Introduction

Fiber-reinforced polymer (FRP) composites, in particular carbon-epoxy composites, have been attracting interest from the offshore oil and gas industry because of their high specific strength and good corrosion resistance. Replacing metals with carbon-epoxy composites for the construction of deep water risers can lead to significant weight reduction and cost savings [1, 2]. For a composite riser system, the composite riser needs to be able to withstand various loads including top tension, internal and external pressure, as well as environmental loads such as current and temperature. Many researchers have used finite element (FE) simulations to study the mechanical behavior of composite risers under

various loading conditions [3, 4, 5]. However, many researchers used a linear constitutive model to simulate the mechanical response of the composite materials before failure. FRP composites have been reported to exhibit non-linear behavior in the transverse direction and especially in the shear direction [6, 7, 8]. Therefore, in order to better predict the failure of composite risers, the non-linear behavior needs to be taken into consideration in the FE simulations.

A complete analysis of the mechanical behavior of FRP composites includes three stages: pre-failure, onset of failure and post failure analysis. For pre-failure analysis, many researchers assume anisotropic linear elastic behavior for cases where the fiber reinforcements are the main load bearing components. For the onset of failure, many failure criteria have been proposed for FRP composites. Failure criteria such as Hashin criteria [9], Puck criteria [10, 11] and LarC criteria [12, 13] distinguish the different failure modes in the composite. For post-failure analysis, the mechanical properties of the damaged part are reduced to simulate the effect of damage. The mechanical properties are reduced or degraded instantly [14] or gradually [15, 16, 17, 18].

However, for cases where shear and transverse loads are significant, the non-linear behavior in the pre-failure stage needs to be considered. Van Paepegem et al. studied the stress-strain response of glass fiber-reinforced composites under cyclic tests [19, 20]. They found that the shear modulus decreased when the specimen is unloaded in the non-linear stress-strain region. Permanent shear strain was also observed in the test specimens [19,

20]. Totry et al. studied the damage mechanisms for the non-linear behavior in composites with two different carbon fibers as reinforcement [21]. For the composite with high modulus carbon fiber, matrix cracking was observed in the shear test specimens whereas for composites with high strength carbon fiber, there was no evidence of damage in the matrix before final failure [21]. Researchers have proposed many different models to simulate the non-linear behavior of FRPs under shear loading [20, 22, 23] and have reported simulation results that agree well with experiments. However, the material models can still be improved to better describe the different damage mechanisms related to the non-linear mechanical behavior of FRPs.

In this study, a model for the pre-failure non-linear behavior of fiber-reinforced polymer composites is presented. For the pre-failure stage, the stress-strain response is assumed to be linear in the fiber direction whereas in the transverse and shear directions, the mechanical response is described using a bimodulus-plastic model. The onset of failure is determined using the Hashin criteria. For the post-failure stage, a linear degradation model is used. Carbon-epoxy specimens were cut from filament wound composite pipes and tested under tensile and compressive loadings. The characterization of the carbon-epoxy material using the experimental test results is described, followed by validation of the proposed model using experimental results from three-point flexure tests. The model is then applied to simulate the failure of a composite riser segment under internal pressure loads.

2. Development of Model

2.1 Background

The model used in this study is an extension of the progressive damage model proposed by Lapczyk and Hurtado [18]. The damage compliance matrix H and stiffness matrix C are given by equations (1) and (2):

$$H = \begin{bmatrix} \frac{1}{(1-d_f)E_1} & -\frac{\nu_{21}}{E_2} & 0 \\ -\frac{\nu_{12}}{E_1} & \frac{1}{(1-d_m)E_2} & 0 \\ 0 & 0 & \frac{1}{(1-d_s)G_{12}} \end{bmatrix} \quad (1)$$

$$C = \frac{1}{D} \begin{bmatrix} (1-d_f)E_1 & (1-d_f)(1-d_m)\nu_{21}E_1 & 0 \\ (1-d_f)(1-d_m)\nu_{12}E_2 & (1-d_m)E_2 & 0 \\ 0 & 0 & D(1-d_s)G_{12} \end{bmatrix} \quad (2)$$

where $D = 1 - (1-d_f)(1-d_m)\nu_{12}\nu_{21}$, E_1 , E_2 , G_{12} are the undamaged moduli, ν_{12} , ν_{21} are the undamaged Poisson's ratios, and d_m , d_f are the damage variables for matrix and fiber respectively. The damage variables are different for tension and compression failure modes which are denoted by the subscripts t and c respectively. The damage variable for shear, d_s is dependent on the damage variables for matrix and fiber as shown in equation (3) [18]:

$$d_s = 1 - (1-d_{ft})(1-d_{fc})(1-d_{mt})(1-d_{mc}) \quad (3)$$

The onset of failure is determined using the Hashin criteria [9]. After failure, a linear softening model is used where the stiffness of the material is reduced by changing the damage variables d_m and d_f [18].

The model proposed by Lapczyk and Hurtado can be applied to FRP composites that exhibit linear elastic behavior before failure [18]. For cases where the transverse and shear loads are significant, the pre-failure behavior deviates from being linear elastic (Figure 1). In order to alleviate this problem, in this study, the material behavior in the transverse and shear directions is modified to account for the non-linearity seen in the experimental study such that the predictions are more accurate.

Figure 1. Comparison of simulation using Lapczyk and Hurtado's model [18] and experimental results for tensile test carried out on $[\pm 55^\circ]_3$ carbon-epoxy composite

2.2 Need for Modification

From experimental studies, two damage mechanisms related to the non-linear behavior have been determined. Damage in the matrix in the form of cracks has been observed in FRP laminates when the material is loaded in the shear direction to the non-linear stress-strain region [21]. Here, the damage is termed matrix ductile damage. Permanent deformation in the FRP laminates is also reported when the laminates are unloaded from the non-linear region [19, 20]. Therefore, yielding and plastic deformation should be included in the model for the polymer matrix.

The incorporation of matrix ductile damage and plastic deformation into the model will be discussed in the next section. The material model for the fiber direction remains the same (linear elastic followed by linear softening after failure). As such, only the transverse and shear parts of the model will be discussed.

2.3 New Formulations

2.3.1 Plastic Deformation

Experiments have shown that the nonlinear behavior of carbon fiber-reinforced polymer (CFRP) is more significant in the shear direction compared to the transverse direction [6, 8]. Therefore, in the current model, the plastic deformation of the matrix is assumed to be in the shear direction only. The shear strain is given by equation (4):

$$\varepsilon_{12}^{tot} = \varepsilon_{12}^{el} + \varepsilon_{12}^{pl} \quad (4)$$

where ε_{12}^{tot} is the total shear strain, ε_{12}^{el} is the elastic shear strain and ε_{12}^{pl} is the plastic shear strain.

For the current model, a linear expression is used for the plastic shear strain (equation (5)):

$$\varepsilon_{12}^{pl} = R(\varepsilon_{12}^{tot} - \varepsilon_{12}^y) \quad (5)$$

where ε_{12}^y is the shear strain at yield and R is a factor describing the ratio of plastic to elastic shear strain after yielding. ε_{12}^{pl} has the same direction as ε_{12}^{tot} and its value increases

monotonically until final failure of the matrix. After the point of matrix final failure is reached, the value of ε_{12}^{pl} is kept constant. ε_{12}^y and R can have different values for tension and compression failure modes.

2.3.2 Matrix Ductile Damage

For the matrix ductile damage, a bimodulus model is adopted. The bimodulus model is implemented using the matrix damage variable d_m . The material stiffness in both the transverse and shear directions is thus affected by matrix ductile damage. The different stages of the model are determined using the failure indices calculated from the Hashin criteria for matrix failure F_{mt} and F_{mc} [9]. The criteria for matrix damage are given by equations (6) and (7):

Matrix tension ($\sigma_{22} \geq 0$):

$$F_{mt} = \left(\frac{\sigma_{22}}{Y^T} \right)^2 + \left(\frac{\sigma_{12}}{S^L} \right)^2 \quad (6)$$

Matrix compression ($\sigma_{22} < 0$):

$$F_{mc} = \left(\frac{\sigma_{22}}{2S^T} \right)^2 + \left[\left(\frac{Y^C}{2S^T} \right)^2 - 1 \right] \frac{\sigma_{22}}{Y^C} + \left(\frac{\sigma_{12}}{S^L} \right)^2 \quad (7)$$

where σ_{ij} are the stress tensor components, Y^T and Y^C are the tensile and compressive strengths in the transverse direction, S^L and S^T are the longitudinal and transverse shear strengths.

The equivalent displacement and stress definitions introduced by Lapczyk and Hurtado are also used here [18]. For the matrix failure modes, the equivalent displacements and stresses are given by equations (8), (9), (10) and (11):

Matrix tension ($\sigma_{22} \geq 0$):

$$\delta_{mt,eq} = L_c \sqrt{\langle \varepsilon_{22} \rangle^2 + \varepsilon_{12}^2} \quad (8)$$

$$\sigma_{mt,eq} = \frac{L_c (\langle \sigma_{22} \rangle \langle \varepsilon_{22} \rangle + \sigma_{12} \varepsilon_{12})}{\delta_{mt,eq}} \quad (9)$$

Matrix compression ($\sigma_{22} < 0$):

$$\delta_{mc,eq} = L_c \sqrt{\langle -\varepsilon_{22} \rangle^2 + \varepsilon_{12}^2} \quad (10)$$

$$\sigma_{mc,eq} = \frac{L_c (\langle -\sigma_{22} \rangle \langle -\varepsilon_{22} \rangle + \sigma_{12} \varepsilon_{12})}{\delta_{mc,eq}} \quad (11)$$

where L_c is the characteristic length of the element in FE simulation, δ_{eq} and σ_{eq} are the equivalent displacement and stress respectively, and the subscripts mt and mc denote the matrix tension and compression failure modes.

Figure 2 shows schematic of the stress-strain response of the proposed bimodulus model in terms of the equivalent displacement and stress for failure mode I where $I \in \{mt, mc\}$.

Figure 2. Schematic diagram of the bimodulus model with linear softening after matrix final failure

At the initial stage, the material is undamaged and is linearly elastic with effective modulus E^* . Ductile damage occurs in the matrix when the failure index F_I reaches F_I^d where $0 < F_I^d < 1$. At this stage, the effective modulus of the matrix is reduced by a factor k due to the ductile damage. The arrows in Figure 2 show the unloading and reloading paths after ductile damage takes place. At $F_I = 1$, matrix final failure occurs and the material modulus is degraded through linear softening. The parameters for the bimodulus model can have different values for tension and compression failure modes. The matrix damage variable d_m at the different stages is calculated using equations (12) and (13):

162 For $F_I^d \leq F_I < 1$ (matrix ductile damage),

$$d_I = 1 - \frac{(1 - k)\delta_{I,eq}^d + k_I \delta_{I,eq}}{\delta_{I,eq}} \quad (12)$$

163 For $F_I \geq 1$ (linear softening after matrix final failure),

$$d_I = d_I^0 + (1 - d_I^0) \frac{\delta_{I,eq}^f (\delta_{I,eq} - \delta_{I,eq}^0)}{\delta_{I,eq} (\delta_{I,eq}^f - \delta_{I,eq}^0)} \quad (13)$$

164 The equivalent displacements and stresses at the onset of matrix ductile damage

165 $(\delta_{I,eq}^d, \sigma_{I,eq}^d)$ and matrix final failure $(\delta_{I,eq}^0, \sigma_{I,eq}^0)$ can be determined using a scaling function

166 f_I^{sc} [18]. The scaling functions for the different failure modes are given by equations (14)

167 and (15):

168 Matrix tension ($\sigma_{22} \geq 0$):

$$f_{mt}^{sc} = \frac{\lambda}{\sqrt{F_{mc}}} \quad (14)$$

169 Matrix compression ($\sigma_{22} < 0$):

$$f_{mc}^{sc} = \frac{-\gamma + \sqrt{\gamma^2 + 4\lambda\beta}}{2\beta}$$

$$\text{where } \gamma = \left[\left(\frac{Y^c}{2S^T} \right)^2 - 1 \right] \frac{\sigma_{22}}{Y^c} \text{ and } \beta = \left(\frac{\sigma_{22}}{2S^T} \right)^2 + \left(\frac{\sigma_{12}}{S^L} \right)^2 \quad (15)$$

170 For the onset of matrix ductile damage, $\lambda = F_I^d$ and for the onset of matrix final
 171 failure, $\lambda = 1$. The equivalent displacements and stresses can then be calculated using
 172 equations (16), (17), (18) and (19):

$$\sigma_{I,eq}^d = \sigma_{I,eq} f_I^{sc} \text{ where } \lambda = F_I^d \quad (16)$$

$$\delta_{I,eq}^d = \delta_{I,eq} f_I^{sc} \text{ where } \lambda = F_I^d \quad (17)$$

$$\sigma_{I,eq}^0 = \sigma_{I,eq} f_I^{sc} \text{ where } \lambda = 1 \quad (18)$$

$$\delta_{I,eq}^0 = \delta_{I,eq}^d + \frac{1}{k_I} (\delta_{I,eq} f_I^{sc} - \delta_{I,eq}^d) \text{ where } \lambda = 1 \quad (19)$$

173 The remaining parameters required for the determination of d_I are the damage
 174 variable at the onset of matrix final failure, d_I^0 and the equivalent displacement at the end of
 175 linear softening, $\delta_{I,eq}^f$. They can be computed using equations (20) and (21):

$$d_I^0 = 1 - \left(\frac{\delta_{I,eq}^d}{\sigma_{I,eq}^d} \right) \left(\frac{\sigma_{I,eq}^0}{\delta_{I,eq}^0} \right) \quad (20)$$

$$\delta_{I,eq}^f = \frac{2G_{I,c} + \delta_{I,eq}^d \sigma_{I,eq}^0 - \delta_{I,eq}^0 \sigma_{I,eq}^d}{\sigma_{I,eq}^0} \quad (21)$$

176 where $G_{I,c}$ is the fracture energy for failure in mode I .

177 Viscous regularization for the linear softening of the material after matrix final
 178 failure is used to overcome convergence difficulties in implicit FE simulations. The viscous
 179 damage variable is given by equation (22):

$$\dot{d}_I^v = \frac{1}{\eta_I} (d_I - d_I^v) \quad (22)$$

where η_I is the viscosity coefficient and d_I^v is the regularized damage variable for the failure mode I [18]. This regularization model is also used for the material in the matrix ductile damage stage but a different viscosity coefficient value is used.

In addition to the properties and parameters required for the progressive damage model proposed by Lapczyk and Hurtado, the parameters required for the current model are F_I^d , k_I , ε_{12}^y and R for matrix tension and compression failure modes. These parameters can be determined using experimental results from unidirectional mechanical tests.

3. Procedure for Parameter Determination and Validation

3.1 Materials and Experiments

The FRP composite used in this study consists of the Epolam 5015/5015 epoxy system supplied by Axson Technologies as the matrix and the HexTow® IM2A carbon fiber supplied by Hexcel Corp as the reinforcement. Composite pipes were fabricated using a two axis CNC filament winding machine shown in Figure 3. The composite pipes consist of three helical layers with winding angle of $\pm 55^\circ$ to the axial direction. The pipes were wound on a mandrel with a radius of 38.1 mm. After winding, the composite pipes were cured for 24 hours at room temperature followed by post cure at 80°C for 16 hours. The mandrel was removed after the curing process. The resulting thickness of the composite

pipe is about 1 mm. The undamaged composite material properties are shown in Table 1.

The transverse and shear moduli have different values for tensile and compressive loading denoted by subscripts t and c .

Figure 3. Fabrication of composite pipe using a filament winding machine

Table 1. Material properties of the undamaged carbon fiber-reinforced composites

E_1 (GPa)	E_{2t} (GPa)	E_{2c} (GPa)	ν_{12}	G_{12t} (GPa)	G_{12c} (GPa)
128.5	10.8	8.1	0.3	2.7	2.09

Tensile and compression tests were carried out to determine the parameters required for the proposed model. The tensile test specimens were cut from the composite pipes and have a curved surface parallel to the loading direction. Flat tabs were attached to the ends of the specimens to prevent damage to the specimens from the clamps. The tensile tests were carried out following the ASTM D3039 standard. For the material parameters in compression, axial compression tests were carried out. Cylindrical specimens were loaded in the axial direction until failure. Strain gauges were attached to the middle of the specimens to measure the axial and hoop strains. The ASTM D695 standard was followed closely. Figure 4 shows the tensile and axial compression test specimens before testing.

Figure 4. (a) Tensile test specimens; (b) axial compression test specimen

For the model validation, three-point flexure tests were also carried out. The flexure test specimens were cut from the composite pipes and have a curved surface. The support span for the tests was 60 mm. The tests were carried out following the ASTM D7264 standard. The flexure tests were performed until the mid-span deflection reached about 18.5 mm.

All the mechanical tests were carried out using a Universal Testing Machine. After cutting out of a filament wound pipe, all the test specimens were inspected visually for any processing defects, such as burrs, notches and edge delamination. Only good ones were chosen for testing. The dimensions for the mechanical test specimens are shown in Table 2.

Table 2. Summary of mechanical test specimens

Tensile test specimen	Label	U3-AT-D
	Width (mm)	14
	Thickness (mm)	1
	Overall length (mm)	250
	Tab length (mm)	60
Axial compression test specimen	Label	U6-ACL-D and U7-ACL-D
	Inner diameter (mm)	76.2
	Wall thickness (mm)	1

	Length (mm)	150
Three-point flexure test specimen	Label	U4-F-D
	Width (mm)	13.5
	Thickness (mm)	1
	Overall length (mm)	150

226

227 3.2 Finite Element Simulations

228 The proposed bimodulus-plastic material model for FRP composites is implemented
229 in a user-subroutine UMAT for the ABAQUS/Standard finite element analysis program
230 [24]. The implementation of the proposed model is summarized in Figure 5.

231

232

233 Figure 5. Flow chart of the bimodulus-plastic model for an increment in FE simulation

234 Figure 6 shows the meshes for the tensile, axial compression, and three-point
235 flexure tests in FE simulation. The curved shapes of the tensile and flexural specimens were
236 modeled explicitly such that the curvature effect is accounted for in the simulations.

Figure 6. Finite element mesh for (a) tensile test simulation; (b) axial compression test simulation; and (c) three-point flexure test simulation

For all the simulations, the carbon-epoxy layers were modeled using 8-node hexahedron continuum shell elements.

For the three-point flexure simulation, the three helical layers of the test specimen are modeled separately and then bonded with cohesive contact interaction. The pins are modeled as rigid surfaces. At the area close to the pins, a small mesh size was used to achieve mesh convergence.

For the tensile and axial compression simulations, the inter-laminar interaction did not affect the simulation result. Therefore, the three helical layers were assumed to be perfectly bonded together. For the axial compression simulation, the results obtained when using a full cylinder model were found to be similar to that of a quarter cylinder model. Hence, the quarter cylinder model was used to reduce computation time.

4. Results and Discussions

4.1 Determination of Material Parameters for the Bimodulus-Plastic Model

Figure 7 shows the composite test specimens after tensile and axial compression tests. In both tensile and axial compression tests, the composite specimens failed due to damage in the epoxy matrix.

Figure 7. Composite test specimens after (a) tensile test; and (b) axial compression test

As mentioned in section 3.1, the tensile test specimens have a curved surface. However, from FE simulations of both curved and flat specimens, the effect of the curvature on the tensile test was found to be negligible. This is because the tensile load was applied parallel to the curved surface. Nevertheless, the FE tensile simulation results for the curved specimens were used.

The FE simulations for the tensile and axial compression tests were fitted to the experimental results to determine the parameters required for the bimodulus-plastic model. Figure 8 and Figure 9 show the stress-strain plots for the tensile and axial compression

267 simulations and experiments. The different stages of damage for the tensile and axial
268 compression simulations are also indicated in Figure 8 and Figure 9.

269
270 Figure 8. Tensile test experimental and simulation results

271

Figure 9. Compression test experimental and simulation results: Compressive stress vs axial and hoop strains.

The yielding of the matrix is assumed to occur before ductile damage comes into effect. This assumption is consistent with the findings of Totry et al. [21]. The area under the stress-strain curves is calculated and used to compare the simulations to experiments. The difference between the experimental and simulation results is calculated using equation (23):

$$\% \text{ Difference} = \frac{P_{Exp} - P_{Sim}}{P_{Exp}} \times 100\% \quad (23)$$

where P_{Exp} is the experimental value and P_{Sim} is the simulation value. This difference is minimized to obtain a good fit for the determination of the parameters required for the bimodulus-plastic model.

Table 3 gives a summary of the simulation and experimental results for the tensile and axial compression tests. The parameters required for the bimodulus-plastic model for the tension and compression failure modes for matrix were determined through experimental data fitting and are shown in Table 4.

Table 3. Comparison of simulation and experimental results for the tensile and axial compression tests

		Experiment	Simulation	% Difference
Tensile test	Maximum tensile stress (MPa)	56.6 ± 1.5	55.4	-2.1
	Strain at maximum stress ($\mu\epsilon$)	13838 ± 1090	14023	+1.3
	Area under graph ($\times 10^5$ Pa)	5.32 ± 0.66	5.36	+0.8
Axial compression	Maximum compressive stress (MPa)	77.8 ± 7.5	81.4	+4.6

test	Axial strain at maximum stress ($\mu\epsilon$)	18196 $\pm$ 916	17882	-1.7
	Area under axial graph ($\times 10^5$ Pa)	9.01 $\pm$ 0.76	8.92	-1.0

Table 4. Parameters for the bimodulus-plastic model for tension and compression failure modes

	Tension	Compression
F_I^d	0.50	0.75
k	0.28	0.25
R	0.50	0.25
ϵ_{12}^y	0.0035	0.0090

4.2 Model validation

The model parameters determined using the tensile and axial compression tests were used in the simulation of the three-point flexure test. The three-point flexure test simulation is compared to experiments for validation of the proposed model.

For the three point flexural tests, the stresses and strains were calculated using equations for flat specimens, and as such, are only approximates. The stresses (σ_{flat}) and strains (ϵ_{flat}) were calculated using equations (24) and (25):

$$\sigma_{flat} = \frac{3PL}{4bh^2} \quad (24)$$

$$\epsilon_{flat} = \frac{6\delta h}{L^2} \quad (25)$$

where P is the applied load, δ is the mid-span deflection, b is the specimen width, h is the specimen height and L is the support span. The width and height used in the calculations are shown in Figure 10. From the experiments, it was observed that the curvature of the flexural test specimens remains almost the same throughout the experiment. Therefore, using the width and height as shown allows the effect of the specimen's curvature to be included in the calculation.

Figure 10. Width and height used in calculation of stresses and strains for flexural tests.

The simulation and experimental results for the three-point flexure test are summarized in Table 5.

Table 5. Comparison of simulation and experimental results for the three-point flexure test

	Experiment	Simulation	% Difference
Maximum stress (MPa)	51.7 $\pm$ 2.0	47.6	-7.9
Permanent mid-span deflection (mm)	2.14 $\pm$ 0.11	2.00	-6.5

The maximum stress calculated from the FE simulation is slightly lower than the average maximum stress obtained from experiments. Plastic deformation was observed in the flexure test specimens after unloading. The permanent mid-span deflection after unloading is measured to compare the plastic deformations in the experiments and simulation. The permanent deflection calculated from the simulation was also slightly smaller than that of the experiments.

The stress-strain response of the FE simulation compared to experiments for the flexure test is shown in Figure 11. The stress-strain response calculated from the FE simulation using the bimodulus-plastic model was in good agreement with the experimental results.

Figure 11. Three-point flexure test experimental and simulation results

Figure 12 shows the deformed shape of the flexure test specimen after unloading.

The shape of the specimen in the FE simulation agrees well with the experiment.

326

327 Figure 12. Permanent mid-span deflection after unloading in the three-point flexure test: (a)
 328 experiment; (b) FE simulation

329 Bending or flexural loads are relevant to the analysis of composite risers as the
 330 risers are subjected to loads due to waves and water currents. Due to the $\pm 55^\circ$ angle of the
 331 FRP composite, the shear stress is a significant component in the three-point flexure test.
 332 Therefore, the shear stress calculated from the FE simulation using the current model is
 333 examined further. Figure 13 shows the shear stress at the top and bottom surfaces of the
 334 flexure test specimen from the simulation at mid-span deflection of 12.5 mm. This is when
 335 the maximum stress of 47.6 MPa was reached.

Figure 13. Contour plot of the shear stress σ_{12} calculated from FE simulation at (a) the bottom surface and (b) the top surface of the specimen at mid-span deflection of 12.5 mm

At the bottom surface, the shear stress is concentrated at the two ends across the width. For the top surface, the shear stress is concentrated near the middle. This is due to the curved shape of the flexure test specimen. The convex side was placed upwards (Figure 6) causing the two ends across the width to experience larger deformation. The stress distribution is also not completely symmetrical along the specimen length due to the angle $\pm 55^\circ$ of the FRP laminate.

For the application of the current model, it is important to ensure that the mechanical tests were carried out properly so that the parameters determined are accurate. In the model formulation, the Hashin criteria were used to determine failure initiation. Failure initiation in the through thickness direction is not modeled explicitly. Therefore, the application of the current model should be limited to cases where the through thickness stresses are small relative to in-plane stresses (for example, thin FRP laminates). Also, a simple expression was chosen to describe the plastic deformation (equation (5)) to lessen the number of parameters required. For FRP composites which exhibit highly nonlinear plasticity, a more complexed expression is required.

5. Application of model: Burst simulation of composite riser

The bimodulus-plastic model was applied to the analysis of a composite riser under internal pressure loads. The dimensions for the composite riser analyzed are shown in Table 6.

Table 6. Composite riser dimensions

Aluminum inner liner	Inner radius	123.4 mm
	Thickness	6.4 mm
Carbon/epoxy composites	Thickness	24.7 mm

Overall	Thickness	31.1 mm
	Segment length	1900 mm

359

360 The riser dimensions are the same as the riser studied by Kim [2]. Martins et al.
361 determined that a length to diameter ratio (L/D) of 12 gives the length of a riser segment
362 that is sufficient to represent an infinite riser [3]. Using this ratio, the length of the
363 representative composite riser segment was determined to be 1900 mm. For the
364 carbon/epoxy composite component of the riser, the layup configuration of $[\pm 55^\circ]_{23}$ was
365 used. This configuration was chosen because experimental data for the $\pm 55^\circ$ configuration
366 are available in the literature for comparison. However, for different layup configurations,
367 the same analysis approach can still be applied. For the inner liner, aluminum was used but
368 the analysis method is also applicable for inner liner made of different materials such as
369 titanium and steel.

370 For the aluminum liner, the plastic deformation of the liner was modeled using a
371 power law with an exponent N . The composite-liner interface was modeled using surface
372 based cohesive behavior. The material properties of aluminum and interface properties
373 reported in an earlier study were used [25]. They are shown in Table 7 and Table 8.

Table 7. Material properties of the aluminum liner [25]

	E (GPa)	ν	Initial yield stress, σ_Y (MPa)	Exponent N
Aluminum	69	0.33	201	0.08

Table 8. Composite-liner interface properties used in simulation [25]

Interface stiffness, K (Pa/m)	4×10^{11}
Traction at onset of failure, τ^o (MPa)	6
Critical energy release rate, G_{IC} (Pa·m)	60
Critical energy release rate, G_{IIC} and G_{IIIC} (Pa·m)	110

Figure 14 shows the finite element mesh used in the simulations. The model represents 1/8 of the full composite riser segment and was found to give similar results to the full segment model. The composite body is shown in green while the aluminum liner is colored blue.

Figure 14. Finite element mesh for the composite riser simulation

For the burst simulations, due to the model size and various failure modes considered in the model, the explicit solver was used. The loading rate was increased to enable the simulations to be completed in a practical number of increments. The total work and internal energies for the simulations were checked to ensure that quasi-static conditions were maintained.

For the boundary conditions, the riser segment is constrained in the axial direction in the middle such that displacement $u_z = 0$. At the end of the riser segment, the displacement in the radial direction was constrained. The composite body and the aluminum liner were also tied at the end so that they have the same axial displacement. The boundary conditions are also shown in Figure 14.

For the burst simulations, inner pressure and axial tensile forces were applied to the riser such that the hoop stress to axial stress ratio, $S = 2H:1A$. This stress ratio is chosen because experimental data from past studies are available and can be used for the verification of the simulation results. Final failure of the composite riser segment is taken as the point when fiber failure occurs. The proposed bimodulus-plastic model was used for the carbon/epoxy composite to account for the nonlinear pre-failure behavior. For comparison, a burst simulation assuming linear-elastic pre-failure behavior for the composite was also carried out. The simulation results are summarized in Table 9.

Table 9. Results for burst simulations performed using different models (experimental data included for comparison)

Failure mode	Inner pressure (MPa)		
	Linear-elastic model for pre-failure behavior	Bimodulus-plastic model for pre-failure behavior	Experiment (from literature)
Liner yielding	40.5	40.5	
Matrix failure through riser wall and possible fluid leakage	114	91.5 (matrix ductile damage) 228 (matrix final failure)	87 (from Soden et al. [26])
Liner-composite	121.5	195	

debonding

Final failure

251.6

246

404

405

From the simulations, the different stages of failure can be determined: liner

406

yielding, matrix tensile failure, followed by final failure due to fiber tensile failure. For

407

matrix failure, the failure started at the inner most composite layer. The damage at the inner

408

most composite layer caused the other layers to carry additional load and eventually fail as

409

well. As a result, the matrix damage progressed outward. Damage in the matrix through the

410

composite riser wall corresponds to the damage when fluid leakage is observed in burst

411

experiments. For the simulation using linear-elastic model for pre-failure behavior, fluid

412

leakage was predicted to occur at the inner pressure of 114 MPa, whereas for the simulation

413

carried out with the bimodulus-plastic model, fluid leakage due to matrix ductile damage

414

was predicted at 91.5 MPa. From past experimental studies, for carbon/epoxy composite,

415

matrix damage was observed when the hoop stress reaches about 410 MPa [26]. For the

416

riser dimensions used, this corresponds to an inner pressure of about 87 MPa. The

417

bimodulus-plastic model gave a better prediction compared to the linear-elastic model. For

418

the liner-composite interface, the debonding at the interface occurred following matrix

419

(final) failure and was not the main factor which determined the failure of the riser. Final

420

failure was predicted at similar inner pressures for the two models (251.6 and 246 MPa)

421

because the material behavior in the fiber direction is modeled the same way in both cases.

The burst simulation carried out using the bimodulus-plastic model was studied further. Figure 15 shows a plot of the hoop stress against axial and hoop strains.

Figure 15. Plots of hoop stress vs hoop and axial strains from burst simulation using the bimodulus-plastic model

The stresses and strains in Figure 15 were taken at the outer surface of the composite riser segment model. The plot of the axial strain shows that the riser segment was expanding in the axial direction with increased inner pressure until hoop stress reached about 170 MPa. This corresponds to the point when the aluminum liner started yielding. The riser segment model then contracted in the axial direction until final failure was reached. This behavior was also observed in experimental studies [27]. On the other hand, the hoop strain increased almost linearly with hoop stress despite matrix failure taking

place. This shows that the deformation in the hoop direction is mainly determined by the carbon fiber.

6. Conclusions

A bimodulus-plastic model to simulate the pre-failure non-linear behavior in FRP composites is proposed. The model includes two damage mechanisms that were observed in experimental studies: (i) ductile damage and (ii) plastic deformation in the polymer matrix. The material properties and parameters required for the proposed model were determined by fitting the model to experimental data.

For the validation of the current model, the model was used in a finite element simulation of a three-point flexure test. The simulation result was found to agree well with experimental results. The model was able to determine the matrix ductile damage in the transverse and shear directions of the FRP composite. The plastic deformation in the flexure test specimen was also reproduced in the simulation using the proposed model.

The bimodulus-plastic model was applied to the study of the failure of a composite riser segment subjected to internal pressure loads. The simulation carried out using the new model was able to predict failure more accurately compared to the simulation assuming linear-elastic pre-failure behavior.

Acknowledgement

Y.D. Boon is grateful for the financial support by Nanyang Technological University, Singapore through the PhD scholarship award.

References

- [1] O. O. Ochoa, "Composite riser experience and design guidance," Offshore Technology Research Center, Texas A&M University, 2006.
- [2] W. K. Kim, "Composite Production Riser Assessment," Thesis, Texas A&M University, 2007.
- [3] L. A. L. Martins, F. L. Bastian and T. A. Netto, "Reviewing Some Design Issues for Filament Wound Composite Tubes," *Materials and Design*, vol. 55, pp. 242-249, 2014.
- [4] L. A. L. Martins, F. L. Bastian and T. A. Netto, "Structural and Functional Failure Pressure of Filament Wound Composite Tubes," *Materials and Design*, vol. 36, pp. 779-787, 2012.
- [5] M. L. Ribeiro, V. Tita and D. Vandepitte, "A New Damage Model for Composite Laminates," *Composite Structures*, vol. 94, pp. 635-642, 2012.
- [6] H. T. Hahn, "Nonlinear Behavior of Laminated Composites," *Journal of Composite*

- Materials*, vol. 7, pp. 257-271, 1973.
- [7] H. T. Hahn and S. W. Tsai, "Nonlinear Elastic Behavior of Unidirectional Composite Laminae," *Journal of Composite Materials*, vol. 7, pp. 102-118, 1973.
- [8] H. Koerber, J. Xavier and P. P. Camanho, "High Strain Rate Characterization of Unidirectional Carbon-Epoxy IM7-8552 in Transverse Compression and In-Plane Shear Using Digital Image Correlation," *Mechanics of Materials*, vol. 42, pp. 1004-1019, 2010.
- [9] Z. Hashin, "Failure criteria for unidirectional fibre composites," *Journal of Applied Mechanics*, vol. 47, pp. 329-334, 1980.
- [10] A. Puck and H. Schürmann, "Failure Analysis of FRP Laminates by means of Physically Based Phenomenological Models," *Composites Science and Technology*, vol. 28, pp. 1045-1067, 1998.
- [11] A. Puck, J. Kopp and M. Knops, "Guidlines for the Determination of the Parameters in Puck's Action Plane Strength Criterion," *Composites Science and Technology*, vol. 62, pp. 371-378, 2002.
- [12] C. G. Dávila and P. P. Camanho, "Failure Criteria for FRP Laminates in Plane Stress," NASA/TM-2003-212663, National Aeronautics and Space Administration, U.S.A., 2003.
- [13] S. T. Pinho, C. G. Dávila, P. P. Camanho, L. Iannucci and P. Robinson, "Failure Models and Criteria for FRP under In-Plane or Three-Dimensional Stress States

- including Shear Non-Linearity," NASA/TM-2005-213530, National Aeronautics and Space Administration, U.S.A., 2005.
- [14] S. C. Tan and R. J. Nuismer, "A Theory for Progressive Matrix Cracking in Composite Laminates," *Journal of Composite Materials*, vol. 23, pp. 1029-1047, 1989.
- [15] F. K. Chang and K. Y. Chang, "Post-Failure Analysis of Bolted Composite Joints in Tension or Shear-Out Mode Failure," *Journal of Composite Materials*, vol. 21, pp. 809-833, 1987.
- [16] P. Maimí, P. P. Camanho, J. A. Mayugo and C. G. Dávila, "A Continuum Damage Model for Composite Laminates: Part I - Constitutive Model," *Mechanics of Materials*, vol. 39, pp. 897-908, 2007.
- [17] P. Maimí, P. P. Camanho, J. A. Mayugo and C. G. Dávila, "A Continuum Damage Model for Composite Laminates: Part II - Computational Implementation and Validation," *Mechanics of Materials*, vol. 39, pp. 909-919, 2007.
- [18] I. Lapczyk and J. A. Hurtado, "Progressive Damage Modeling in Fiber-Reinforced Materials," *Composites: Part A*, vol. 38, pp. 2333-2341, 2007.
- [19] W. Van Paepegem, I. De Baere and J. Degrieck, "Modelling the Nonlinear Shear Stress-Strain Response of Glass Fibre-Reinforced Composites. Part I: Experimental Results," *Composite Science and Technology*, vol. 66, pp. 1455-1464, 2006.
- [20] W. Van Paepegem, I. De Baere and J. Degrieck, "Modelling the Nonlinear Shear Stress-Strain Response of Glass-Fibre Reinforced Composites. Part II: Model

- Development," *Composite Science and Technology*, vol. 66, pp. 1465-1478, 2006.
- [21] E. Totry, J. M. Molina-Aldareguía, C. González and J. Llorca, "Effect of Fiber, Matrix and Interface Properties on the In-Plane Shear Deformation of Carbon-Fiber Reinforced Composites," *Composite Science and Technology*, vol. 70, pp. 970-980, 2010.
- [22] W.-P. Lin and H.-T. Hu, "Nonlinear Analysis of Fiber-Reinforced Composite Laminates Subjected to Uniaxial Tensile Load," *Journal of Composite Materials*, vol. 36, pp. 1429-1450, 2001.
- [23] A. Boutaous, B. Peseux, L. Gornet and A. Bélaidi, "A new modeling of plasticity coupled with the damage and identification for carbon fibre composite laminates," *Composite Structures*, vol. 74, pp. 1-9, 2006.
- [24] *ABAQUS version 6.14 User's Manual*, Dassault Systemes, 2014.
- [25] Y. D. Boon, S. C. Joshi and L. S. Ong, "Interfacial bonding between CFRP and mechanically-treated aluminum liner surfaces for risers," *Composites Structures*, vol. 188, pp. 374-386, 2018.
- [26] P. D. Soden, M. J. Hinton and A. S. Kaddour, "Biaxial Test Results for Strength and Deformation of a Range of E-Glass and Carbon Fibre Reinforced Composite Laminates: Failure Exercise Benchmark Data," *Composite Science and Technology*, vol. 62, pp. 1489-1514, 2002.

[27] B. Spencer and D. Hull, "Effect of Winding Angle on the Failure of Filament Wound Pipe," *Composites*, vol. 9, pp. 263-271, 1978.

456

457