West Virginia University
THE RESEARCH REPOSITORY @ WVU

Graduate Theses, Dissertations, and Problem Reports

2016

Determination of Material Parameters of E-GlassEpoxy Laminated Composites in ANSYS

Mehdi Shahbazi

Follow this and additional works at: https://researchrepository.wvu.edu/etd

Recommended Citation

Shahbazi, Mehdi, "Determination of Material Parameters of E-GlassEpoxy Laminated Composites in ANSYS" (2016). *Graduate Theses, Dissertations, and Problem Reports*. 6610.
https://researchrepository.wvu.edu/etd/6610

This Thesis is protected by copyright and/or related rights. It has been brought to you by the The Research Repository @ WVU with permission from the rights-holder(s). You are free to use this Thesis in any way that is permitted by the copyright and related rights legislation that applies to your use. For other uses you must obtain permission from the rights-holder(s) directly, unless additional rights are indicated by a Creative Commons license in the record and/ or on the work itself. This Thesis has been accepted for inclusion in WVU Graduate Theses, Dissertations, and Problem Reports collection by an authorized administrator of The Research Repository @ WVU. For more information, please contact researchrepository@mail.wvu.edu.

Determination of Material Parameters of E-Glass/Epoxy Laminated Composites in ANSYS

Mehdi Shahbazi

Thesis submitted
to the College of Engineering and Mineral Resources at West Virginia University

in Partial Fulfillment of requirements for the degree of

Master of Science in
Mechanical Engineering

Ever J. Barbero, Ph.D., Chair
Konstantinos Sierros, Ph.D.
Marvin Cheng, Ph.D.

Department of Mechanical and Aerospace Engineering (MAE)

Morgantown, West Virginia , 2016

Keywords: ANSYS, Composite, Progressive Damage Analysis, Discrete Damage Mechanics, Damage Behavior, E Glass Epoxy
Copyright 2016 Mehdi Shahbazi

Abstract

Determination of Material Parameters of E Glass/Epoxy Laminated Composites in ANSYS

Mehdi Shahbazi

Prediction of damage initiation and Evolution in composite materials are of particular importance for the design, production, certification, and monitoring of an increasingly large variety of structures. In this study a methodology is presented to calculate the material properties for the progressive damage analysis (PDA) and discrete damage mechanics (DDM) in ANSYS® by using two sets of experimental data for laminates $[0_2/90_4]_S$ and $[0/\pm 40_4/0_{1/2}]_S$. The type of laminates to be used for material property determination are chosen based on a sensitivity study.

This method is based on fi results calculated with PDA and DDM to experimental data by using Design of Experiments and optimization tools in ANSYS Workbench. The method uses experimental modulus-reduction vs. strain data for only two laminates to fi all the parameters of PDA (F_{2t}, F_6, G_c^{tm}) and DDM (G_{Ic}, G_{IIc}) . Fitted parameters are then used to predict and compare with the experimental behavior of other laminates with the same material system. Mesh sensitivity of both PDA and DDM is studied by performing p- and h-mesh refinement. Choice of damage activation function is justified based on goodness of fi with each proposed equation. Comparison between DDM and PDA predictions is shown.

Acknowledgements

I would like to thank all the people who made this possible.

Dedication

This is dedicated to my parents Khodadad Shahbazi and Ghadam-kheir khorshidi.

Table of Contents

List of Tables

List of Figures

Chapter 1

Introduction

A composite material is a combination of two or more materials, whose properties are superior to those of the constituent materials acting independently. Fibre-reinforced polymer composites are usually manufactured by strong fi and less stiff polymeric matrix. The primary role of the fi is to provide strength and stiffness to the composite. Typical reinforcing fi used are glass, carbon and aramid. The most common types of glass fi er used in fi erglass which is considered in this study is E-glass, which is alumino-borosilicate glass with less than 1% w/w alkali oxides, mainly used for glass-reinforced plastics.

Damage initiation and propagation are two important concerns in predicting damage behavior of composite materials, so prediction of damage initiation and propagation are of particular importance for the design, production certification and monitoring of an increasingly large variety of structures. This study focuses on adjusting the material parameters for the discrete damage mechanics (DDM) and progressive damage analysis (PDA) models to precisely predict the damage behavior .

ANSYS Mechanical provides progressive damage analysis (PDA, [2]) starting with release 15. Also, the user can defi DDM model as a user material in ANSYS and use that in an APDL code. In this study a DDM model is defi as a *dll* fi and is used to analyze the damage. Furthermore, ANSYS Workbench allows optimization of any set of variables to any user defi objective defi in a Mechanical APDL (MAPDL) model by importing the APDL script into Workbench and using Design of Experiments (DoE) and Direct Optimization (DO). Since PDA is not implemented in the graphical user interface (GUI), the user must use APDL commands to defi the damage initiation criterion, damage evolution law, and material properties.

Although elastic moduli are available for many composite material systems, the same is not true for the material properties required by PDA and DDM models. However, laminate modulus and Poisson's ratio degradation of laminated composites as a function of applied strain are available for several material systems [24, 25]. This study shows how to use available data to infer the material properties required by PDA. Specifically, the main purpose of this study is to fi in-situ values [7] of *transverse tensile strength* F_{2t}, *in-plane shear strength* F_6, and *energy dissipation per unit area* G_c^{tm} for the material system (composite lamina) that can be used in PDA to predict damage initiation and evolution of laminated composite structures built with the same material system.

The stated objective for PDA is achieved by minimizing the error between PDA predictions and available experimental data. Once the *input* parameters F_{2t}, F_6 and G_c^{tm} are found, the accuracy of PDA predictions is checked by comparing those predictions with experimental data for other laminates that has not been used to fi the input parameters. Also, by minimizing the error between DDM model predictions and available experimental data the *input* parameters G_{Ic} and G_{IIc} are found for DDM model, and the accuracy of DDM predictions is checked by the same method mentioned for PDA model.

In fact, experimental data for only two laminates are required to fi the parameters. Although the input parameters are fi using an specific mesh (one element) and type of element (SHELL 181 for PDA and PLANE 182 for DDM), it is expected that the PDA and DDM constitutive model will be mesh insensitive in order to be useful when mesh refinement and several type of elements are used for the analysis of a complex structure. Mesh sensitivity is thus assessed in this work by performing both p- and h-refi t.

Chapter 2

Progressive Damage Analysis (PDA)

There are lots of failure theories([15], [14], [8], [16]) which are used not only for predicting the initiation of damage but also for progressive failure up to ultimate load. The most popular failure criteria are the those criteria which are easier to use, although it does not mean that they are accurate. The theories such as the maximum stress, the maximum strain, TsaiWu, TsaiHill, and the Hashin failure criteria are still widely used despite their limits because they are simple, easy to understand and implement in analysis ([8],[16]). The maximum stress and maximum strain criteria are typical examples of so-called non-interactive theories which have been shown to produce poor predictions in general [13], these two criteria only predict the damage initiation.

Theories that allow interaction between stress components such as the TsaiWu criterion generally perform better results ([23], [22]). In a review [8], wide variations in prediction by various theories are attributed to diff t methods of modeling the progressive damage process, the nonlinear behavior of matrix-dominated laminates (angle plys), the inclusion or exclusion of curing residual stresses in the analysis, and the defi of ultimate laminate failure (ULF). The latter may be defi in at least three diff t ways: the maximum load attained; the occurrence (or detection) of fi fi er failure (FFF); and the occurrence of last ply failure (LPF). The review also discusses the effects of interactions, with good reported agreement with experiment in the shear-tension quadrant, but less agreement in the shear-compression quadrant. Similar conclusions are reached in another review of failure theories by Icardi et al. ([8], [16]). Recently, the phrase physically based (and mechanism based or similar words) has been used to describe failure theories which have separate predictions of fi er-dominated and matrix-dominated failures ([16], [19], [18] and [17]). Hashins [11] and Pucks [20] criteria are in this category and accounts for their popularity in progressive damage modeling. This study focuses on Hashin [12], since ANSYS uses Hashin criterion for progressive damage analysis (PDA).

2.1 Progressive Damage Analysis in ANSYS

To perform progressive damage analysis of composite materials, the user needs to provide linear elastic orthotropic material properties and three material models: damage initiation, damage evolution law, and material strength limits.

2.1.1 Damage Initiation Criteria

With damage initiation criteria the user can defi how PDA determines the onset (initiation) of material damage. The available initiation criteria in ANSYS are maximum strain, maximum stress, Hashin, Puck, LaRC03, and LaRC04. Besides, the user can defi up to nine additional criteria as user defi initiation criteria, but only the Hashin criterion works with PDA. The remaining only work with instant stiffness reduction (MPDG). The later is similar to ply-drop off but the amount of stiffness drop can be specified in the range 0–100% of the undamaged stiffness. With MPDG, the user can choose failure criteria among those mentioned for each of the damage modes, which are assumed to be uncoupled.

For example, using the Hashin initiation criteria, we have the following four modes of failure: fi er tension, fi er compression, matrix tension, and matrix compression, which are represented by damage initiation indexes F_{ft}, F_{fc}, F_{mt} and F_{mc} that indicate whether a damage initiation criterion in a damage mode has been satisfied or not. Damage initiation occurs when any of the indexes exceeds 1.0. Damage initiation indexes are unfortunately called "failure" indexes in the literature, despite the fact that nothing "fails" but rather a small amount of damage appears.

Fiber tension ($\sigma_{11} \geq 0$)

Fiber tension is a misnomer sometimes used in the literature, since this mode actually represents *longitudinal tension* of the composite lamina, not the fi er. The corresponding damage initiation index is computes as follows

$$F_{ft} = \left(\frac{\sigma_{11}}{F_{1t}}\right)^2 + \alpha\left(\frac{\sigma_{12}}{F_6}\right)^2 \tag{2.1}$$

Fiber Compression ($\sigma_{11} < 0$)

Fiber compression is a misnomer, since this mode actually represents *longitudinal compression* of the composite lamina, not the fi er. The corresponding damage initiation index is computes as follows

$$F_f^c = \left(\frac{\sigma_{11}}{F_{1c}}\right)^2 \tag{2.2}$$

Matrix tension and/or shear ($\sigma_{22} \geq 0$)

This is also misnomer, since this mode actually represents *transverse tension and in-plane shear* of the composite lamina, not the matrix. the confusion is doe to the fact that this is a matrix-dominated mode but still the criteria applied at the meso-scale, that is at the level of a lamina, not at the micro-scale where the fi er and matrix would be analyzed separately. Furthermore, the properties involved (F_{2t}, F_6) are those of a lamina, not of fi er and matrix separately, ans also the resulting index applies to the lamina, not to the matrix.

$$F_m^t = \left(\frac{\sigma_{22}}{F_{2t}}\right)^2 + \left(\frac{\sigma_{12}}{F_6}\right)^2 \tag{2.3}$$

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

Matrix compression is a misnomer, since this mode actually represents *transverse compression and transverse shear* of the composite lamina, not the matrix.

$$F_m^c = \left(\frac{\sigma_{22}}{2F_4}\right)^2 + \left[\left(\frac{F_{2c}}{2F_4}\right)^2 - 1\right]\frac{\sigma_{22}}{F_{2c}} + \left(\frac{\sigma_{12}}{F_6}\right)^2 \tag{2.4}$$

where σ_{ij} are the components of the stress tensor; F_{1t} and F_{1c} are the tensile and compressive strengths of a lamina in the fi er direction; F_{2t} and F_{2c} are the in-situ tensile and compressive strengths in the matrix direction; F_6 and F_4 are the in-situ longitudinal and transverse shear strengths, and α determines the contribution of the shear stress to the fi er tensile criterion. To obtain the model proposed by Hashin and Rotem [12] we set $\alpha = 0$

and $F_4 = 1/2F_{2c}$. Note that in-situ properties should be used for all matrix-dominated modes.

This study uses Hashin initiation criteria for all tensile and compression failures. The command APDL for this purpose is TB, DMGI, as it is shown below:

```
! Damage detection using failure criteria
    TB, DMGI, 1, 1, 4, FCRT
    TBTEMP,0
! 4 is the value for selecting Hashin criteria,
! which is here selected for all four failure modes
    TBDATA,1,4,4,4,4
```

2.1.2 Material Strength Limits

To evaluate the damage initiation criteria, the user defi the maximum stresses or strains that a material can tolerate before damage occurs. Required inputs depend on the chosen criteria in the damage initiation part. For instance, for Hashin criteria the user needs to defi in-situ tensile and compression strength in 1, 2, and 3 lamina orientations (called X, Y, and Z directions in ANSYS), and the shear strength in 12, 13, and 23 lamina planes (called XY, XZ, and YZ in ANSYS).

Since fi er dominated properties are at least one order of magnitude (10X) larger than matrix dominated properties, matrix modes occur much earlier in the life of the structure that fi er modes. Fiber modes (§2.1.1, §2.1.1) do not occur until nearly the end of the life. Furthermore, transverse matrix compression (§2.1.1) does not result in progressive damage but rather leads to sudden failure according to the Mohr-Coulomb criterion [9]. Therefore, this study focuses on matrix tension and shear modes (§2.1.1), which are know to lead to substantial progressive damage [24, 25].

Initial values of in-situ transverse tensile strength F_{2t}, in-situ in-plane shear strength F_6, and intralaminar shear strength F_4 (called XT, XY, and XZ in ANSYS) should be defi
in the APDL script. The command for material strength limit is TB, FCLI, as shown below:

```
! Material Strengths
TB,FCLI,1,1,6
TBTEMP,0
! Failure Stress, Fiber Tension
TBDATA,1, F1t
```

! Failure Stress, Fiber Compression
TBDATA,2,F_{1c}
! Toughness Stress, Matrix Tension
TBDATA,3,F_{2t}
! Failure Stress, Matrix Compression
TBDATA,4,F_{2c}
! Failure Stress, XY Shear
TBDATA,7,F_6
! Failure Stress, YZ Shear
TBDATA,8,F_4

As it was previously stated, the damage initiation part of PDA requires six material properties listed above. Two of those, F_{2t} and F_6, are in-situ values. In-situ values can be calculated using equations that involve the corresponding lamina properties and the corresponding energy dissipation per unit area G_{Ic}, G_{IIc} in modes I and II [9], or the transition thickness of the material [6]. However, these energies and transition thicknesses are not usually available in the literature. Experimental methods to determine in-situ properties are not available either. By focusing on one damage mode, namely matrix damage (§2.1.1), it is show in this work that the material properties required by PDA can be obtained by fi PDA model results to suitable experimental data.

2.1.3 Damage Evolution

After satisfying the selected initiation criteria, further loading will degrade the material. The damage evolution law determines how the material degrades. In ANSYS, there are two options for damage evolution: instant stiffness reduction (MPDG) and continuum damage mechanics (PDA). Since instant stiffness reduction, which is suddenly applied when the criterion is satisfied, does not provide any information about damage evolution, this study uses the PDA method for damage evolution.

PDA requires eight parameters: four values of energy dissipated per unit area (G_c, Figure 2.1) and four viscosity damping coefficients (η) for tension and compression in both fi er and matrix dominated modes. The energy dissipated per unit area is defi as:

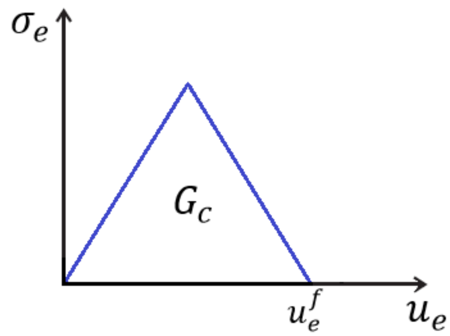


Figure 2.1: Equivalent stress σ_e vs. equivalent displacement u_e.

$$G = \int_0^{u_e^f} \sigma_e du_e \tag{2.5}$$

where:

σ_e = is the equivalent stress. For simple uniaxial stress state, the equivalent stress is the actual stress. For complex stress state, the equivalent stresses and strains are calculated based on Hashin's damage initiation criteria.

u_e = is the equivalent displacement. For simple uniaxial stress state, $u_e = E \times L_c$, and L_c is the length of the element in the stress direction.

u_e^f = is the ultimate equivalent displacement, where total material stiffness is lost for the specific mode.

Viscous damping coefficients η are also specified respectively for each of the damage modes. For a specific damage mode, the damage evolution is regularized as follows:

$$d^i_{t+\Delta t} = \frac{\eta}{\eta + \Delta t} d^i_t + \frac{\Delta t}{\eta + \Delta t} d_{t+\Delta t} \tag{2.6}$$

where:

$d^i_{t+\Delta t}$ = Regularized damage variable at current time. $d_{t+\Delta t}$ is used for material degradation

d_t^i = Regularized damage variable at previous time.

$d_{t+\Delta t}$ = Un-regularized current damage variable

The command for defi damage evolution in APDL is TB,DMGE, as shown below.

! Damage Evolution with CDM Method
TB,DMGE,1,1,8,CDM
TBTEMP,0
! Fracture Toughness, Fiber Tensile
TBDATA,1,G_c^{ft}
! Viscosity Damping Coefficient, Fiber Tensile
TBDATA,2, η^{ft}
! Fracture Toughness, Fiber Compressive
TBDATA,3,G_c^{fc}1E6
! Viscosity Damping Coefficient, Fiber Compressive
TBDATA,4, η^{fc}
! Fracture Toughness, Matrix Tensile
TBDATA,5,G_c^{tm}
! Viscosity Damping Coefficient, Matrix Tensile
TBDATA,6, η^{mt}
! Fracture Toughness, Matrix Compressive
TBDATA,7,G_c^{mc}
! Viscosity Damping Coefficient, Matrix Compressive
TBDATA,8, η^{mc}

As it was previously stated, the damage evolution part of PDA requires four material properties, i.e., four values of G_c (eq. 2.5), that are not available in the literature. Experimental methods to determine these properties are not available either. By focusing on one damage mode, namely matrix/shear damage (§2.1.1), it is show in this work that the material properties in question (F_{2t},F_6,and G_c^{mt}) can be obtained by fi the model results to suitable experimental data.

2.2 PDA Design of Experiments

The next step is to use design of experiments (DoE) and optimization to adjust the values of F_{2t}, F_6, and G_c^{mt} so that the PDA prediction closely approximates the experimental data.

First we use DoE to identify the laminates that are most sensitive to each parameter. The focus at this point is to identify the minimum number of experiments that are needed to adjust the parameters. In this way, additional experiments conducted with diff t laminate stacking sequences (LSS) are not used to adjust parameters but to asses the quality of the predictions.

In principle, the DoE technique is used to fi the location of sampling points in a way that the space of random input parameters $X = \{F_{2t}, F_6, G_c^{tm}\}$ is explored in the most efficient way and that the output function D can be obtained with the minimum number of sampling points. In this study the output function (also called objective function) is the error between the predictions and the experimental data. Given N experimental values of laminate modulus $E(\epsilon_i)$, where ϵ is the strain applied to the laminate, and $i = 1 \ldots N$, the error is defi as

$$D = \sqrt{\frac{1}{N}\sum_{i=1}^{N}\left(\frac{E^{ANSYS}}{E_0}\Big|_{\epsilon=\epsilon_i} - \frac{E^{Experimental}}{E_0}\Big|_{\epsilon=\epsilon_i}\right)^2} \tag{2.7}$$

where E and E_0 are laminate elastic modulus for damaged and undamaged laminate, respectively, and N is the number of experimental data points.

DoE tools can be found in ANSYS Workbench in the Design Explorer (DE) module, which includes also Direct Optimization (DO), Parameters Correlation, Response Surface (RS), Response Surface Optimization, and Six Sigma Analysis.

Let's denote the input parameters by the array $X = \{F_{2t}, F_6, G_c^{tm}\}$. In this case, the output function $D = D(X)$ can be calculated by evaluation of eq. (2.7) through execution of the fi element analysis (FEA) code for N values of strain. Each FEA analysis is controlled by the APDL script, which calls for the evaluation of the non-linear response of the damaging laminate for each value of strain, with parameters X. If the mesh is refined, these evaluations could be computationally intensive.

An alternative to direct evaluation of the output function is to approximate it with a multivariate quadratic polynomial. The approximation is called response surface (RS).

It can be constructed with only few actual evaluations of the output by choosing a small number of sampling points for the input. The sampling points are chosen using Design of Experiments (DoE) theory. The number of evaluations needed to construct the response surface (RS) and for direct optimization (DO) are shown in Table 2.1.

		# of evaluations		
# of inputs	Inputs	RS	DO (Screening)	DO (Adaptive Single-Objective)
1	F_6	5	100	7
2	$F_6, G_c^{\prime\prime\prime\prime}$	9	100	33
3	$F_{2t}, F_6, G_c^{\prime\prime\prime\prime}$	15	100	14

Table 2.1: Number of FEA evaluations used (a) to construct the response surface (RS) and (b) to adjust the input parameters by direct optimization (DO).

The shape of the RS can be inferred by observing the variation of the output D as a function of only one input at a time. This is shown in Figures 2.2, 2.3, and 2.4, for laminate #1 (Table 2.2). The abscissa spans each of the inputs and the ordinate measures the error between predicted and experimental data.

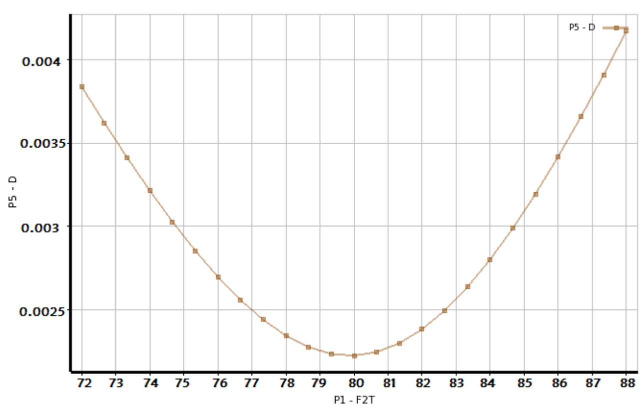


Figure 2.2: Response surface chart. Error (D) vs. F_{2t}.

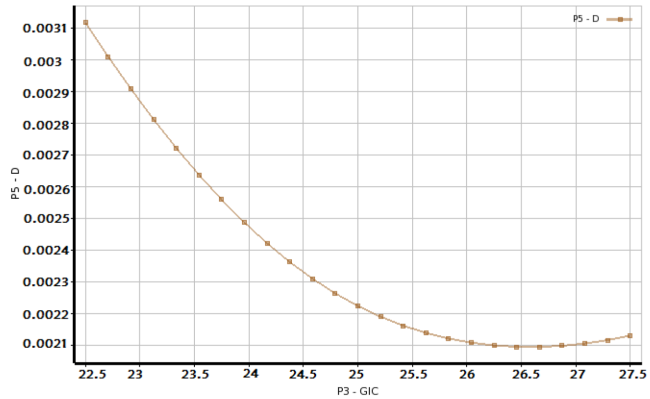


Figure 2.3: Response surface chart. Error (D) vs. G_c^{mt}.

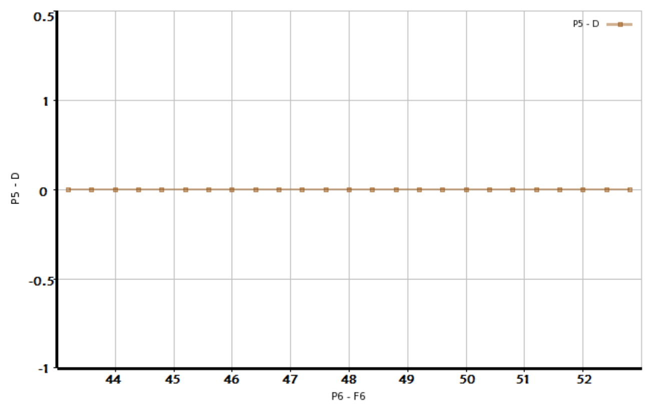


Figure 2.4: Response surface chart. Error (D) vs. F_6.

By observing the RS we can understand which parameters (inputs) have most effect on the error (output). From Figure 2.4 it is clear that the error is not sensitive to F_6 for laminate #1. This is because no lamina in laminate #1 is subject to shear. Therefore, it is decided to use the experimental data of laminate #1 to evaluate only two input parameters (F_{2t} and G_c^{mt}). This makes the optimization algorithm to run faster.

The RS is multivariate quadratic polynomial that approximates the actual output function $D(X)$ as a function of the input variables. In this study, $D_{RS} = f(X_i)$. The sensitivity S of the output to input X in the user specified interval $[X_{min}, X_{max}]$ is calculated as:

Laminate #	LSS
1	$[0_2/90_4]_S$
2	$[\pm 15/90_4]_S$
3	$[\pm 30/90_4]_S$
4	$[\pm 40/90_4]_S$
8	$[0/\pm 40_4/0_{1/2}]_S$
9	$[0/\pm 25_4/0_{1/2}]_S$

Table 2.2: Laminate stacking sequence for all laminates for which experimental data is available.

$$S = \frac{max(D) - min(D)}{average(D)} \tag{2.8}$$

and tabulated in Table 2.3 for laminate # 1. Note that the sensitivity can be calculated from the error evaluated directly from FEA analysis or from the RS. The later is much more expedient than the former, as it can be seen in Table 2.1.

	Input range		Error D			Sensitivity
Input	min(Input)	max(Input)	min(D)	max(D)	ave(D)	S
F_{2t}	72	88	0.0022	0.0041	0.00297	0.64
G_c^{mt}	22.5	27.5	0.0031	0.0020	0.0026	-0.42
F_6	50	88	0.076	0.102	0.093	-0.28

Table 2.3: Sensitivity S of the output (error) to each input (parameter). First two rows refer to laminate #1 and last row to laminate #8.

The charts in Figures 2.2–2.4 are drawn for the input ranges given in Table 2.3. It is convenient to compare all of them in one chart (Figure), where the input range has been normalized to the interval [0–1].

2.3 Methodology

The input parameters can be adjusted with any mesh and any type of elements that represent the gauge section of the specimen, or a single element to represent a single material point of the specimen. For expediency, a single linear element (SHELL 181) is used in this study.

2.3.1 APDL

The APDL script is used to specify the mesh, boundary conditions, and the strain applied to the laminate. The later is specified by imposing a specified displacement. Incrementation of the applied displacement is implemented to mimic the experimental data, which is available for a fi set of values of applied strain.

The APDL script is used also to specify the elastic properties (with MP command, Table 2.4), the laminate stacking sequence (with SECDATA command, Table 2.2), the material strengths (with TB command, Table 2.4). In Table 2.4, the material strengths that are to be adjusted (F_{2t}, F_6) are simply initial (guess) values for the optimization.

Also the APDL script contains values for the eight damage evolution parameters (with TBDATA command, Table 2.5). The strengths of the non-participating modes (FT,FC, and MC) are set to a high value to avoid those modes from interfering with the study of the MT mode, and the values of the corresponding damping coefficients η are thus irrelevant. The energy dissipation per unit area to be adjusted (G_c^{mt}) is set to a guess value, and the corresponding damping coefficient is found by trial and error to obtain a smooth computation of the entire plot of laminate modulus vs. applied strain (Figure 2.5).

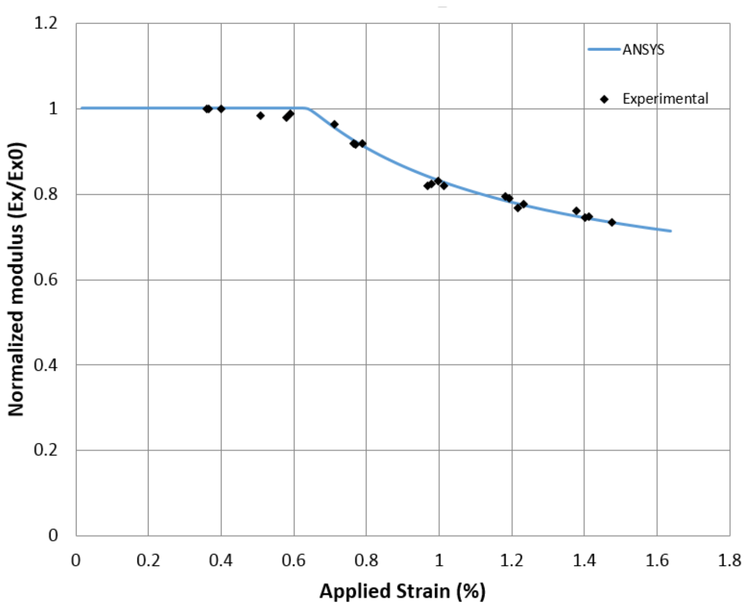


Figure 2.5: Normalized modulus vs. applied strain for laminate #1.

The APDL script also includes a table with experimental data for the laminate being analyzed. Such data consists of a number N of pairs of values representing laminate modulus as a function of applied strain $E_x(E_x)$. Finally, the APDL script calculates the error as per eq. (2.7).

Property	Units	Value	Ref.
E_1	MPa	44700	[24]
E_2	MPa	12700	[24]
G_{12}	MPa	5800	[24]
v_{12}	-	0.297	[24]
v_{23}	-	0.411	[24]
Ply thickness	mm	0.144	[25]
F_{1t}	MPa	1020	[6]
F_{1c}	MPa	-620	[6]
F_{2t}	MPa	80	guess value
F_{2c}	MPa	-140	[6]
F_6	MPa	48	guess value
F_4	MPa	52.7	[6]

Table 2.4: Lamina elastic properties and in-situ strength values.

Property	Units	Value	Ref.
G_f^t	KJ/m^2	1E6	high value
G_f^c	KJ/m^2	1E6	high value
G_m^t	KJ/m^2	25	guess value
G_m^c	KJ/m^2	1E6	high value
η_f^t	-	1E-3	immaterial
η_f^c	-	1E-3	immaterial
η_m^t	-	5E-3	trial and error
η_m^c	-	1E-3	immaterial

Table 2.5: Damage evolution properties of the lamina.

2.3.2 Workbench

First, a *Mechanical APDL* component is added to the Project Schematic by dragging it from the Component Systems menu. The APDL code is then imported into Workbench. See Figure 2.6.

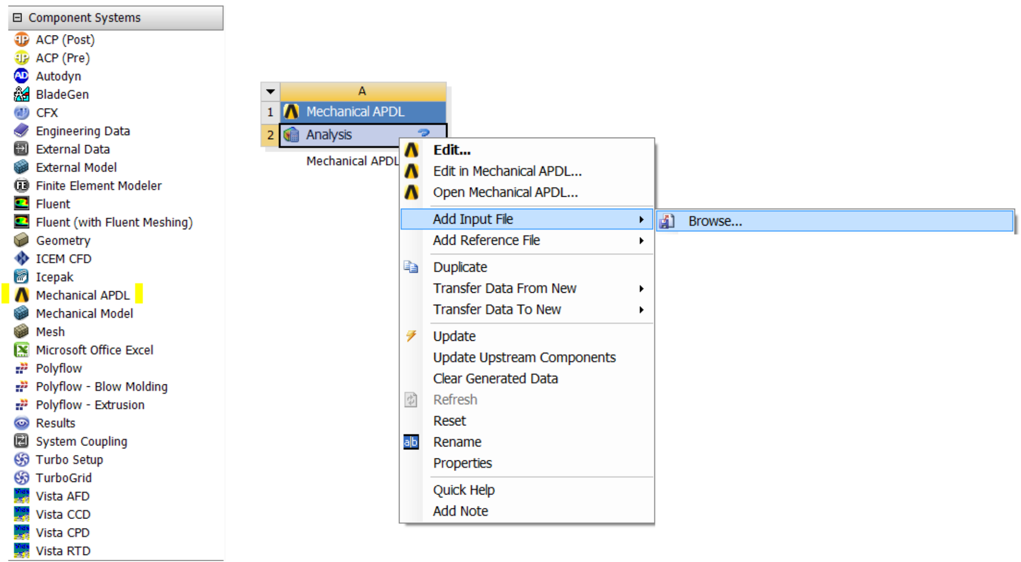


Figure 2.6: Importing the APDL code into Workbench.

Next, from among all the parameters defi in the APDL script, the input parameters ($F2t$, F_6, G_c^{mt}) and output parameters (D) are selected as shown in Figure 2.7.

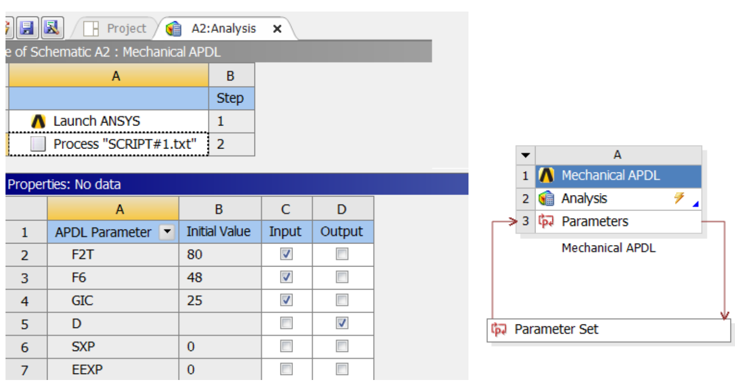


Figure 2.7: Inputs and output parameters are selected.

2.3.3 Optimization

Optimization techniques are used in this study to minimize the error (2.7) by adjusting the input (material) parameters. In these way, the fi values of the parameters represent materials properties for the specific PDA material model used in the underlying FEA.

An *Response Surface Optimization* (RSO) component is now added to the Workbench by dragging it from the Component Systems menu to the Project Schematic (Figure 2.8).

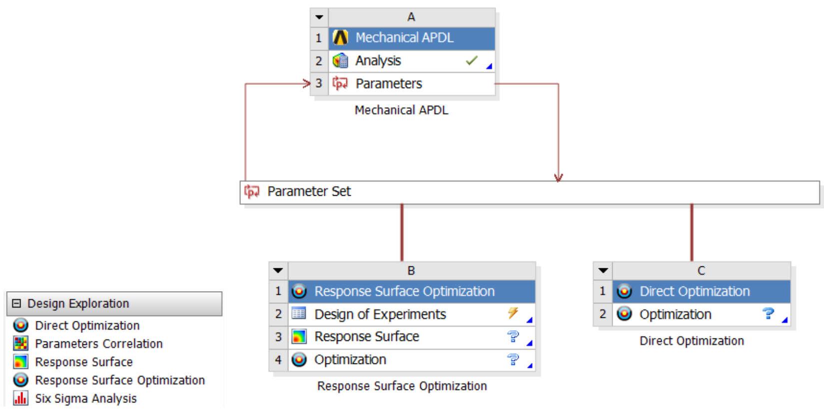


Figure 2.8: Response Surface tools include DoE, RS, and RS Optimization.

Then, DoE is used to adjust a multivariate quadratic polynomial to the actual response (output) of the system as defi   d by the APDL script. In this study the output is the error (2.7). The multivariate are the input parameters, which in this study are three parameters.

Then, the RS is used to plot the response (output) vs. each of the input parameters and to calculate the sensitivities. This allows the user to select, for optimization, only the parameters to which the output (error) is sensitive.

Within RSO, optimization is performed by using the RS rather than actual evaluation of the response via fi element analysis (FEA). This results in significant savings of computer time, as shown in Table 2.1, but the result is approximate because the RS is an approximation to the actual output function.

To get exact optimum parameters (within numerical accuracy) one has to conduct Direct Optimization (DO). It can be seen in Table 2.6 that RS is quite accurate when compared with DO, considering that the number of FEA evaluations (reported in Table 2.1) is much smaller for RS than for DO.

As it is shown in Table 2.7, the accuracy of the parameters is good when RS is used instead of DO. A cost comparison, in terms of number of FEA evaluations, is shown in Table 2.1, where it can be seen that DO is much more expensive.

Laminate	Error D RS	DO	Adjusted values F_{1t}	F_6	G_c^{tm}
#1	0.002046	0.002007	78.32	86.706	26.978
#2	0.003063	0.002682	same	same	same
#3	0.010447	0.007820	same	same	same
#4	0.011243	0.006047	same	same	same
#5	0.015960	0.015278	same	same	same
#6	0.011002	0.010491	same	same	same
#7	0.052601	0.051115	same	same	same
#8	0.076554	0.075605	same	same	same
#9	0.022765	0.020154	same	same	same

Table 2.6: Error and adjusted values of input (material) parameters for all laminates considered.

Parameter	Optimization method RSO	DO (Screening)	DO (Adaptive Single-Objective)
F_{2t}	80.011	78.32	78.55
F_6	87.131	86.705	88.00
G_c^{tm}	26.723	26.978	27.50
Max # of FEM evaluations	15	100	33

Table 2.7: Comparison of adjusted input (material) parameters obtained by using Response Surface Optimization (RSO) and Direct Optimization (DO).

Twenty two experimental data points are available for laminate #1 and nineteen experimental data points are available for laminate #8. Laminate #1 ($[0_2/90_4]_S$) was chosen because this laminates is sensitive to F_{2t} and G_c^{tm}. On the contrary, F_6 does not have any effect on the results of Laminate #1, as shown in Table 2.3 and Fig. 2.9.

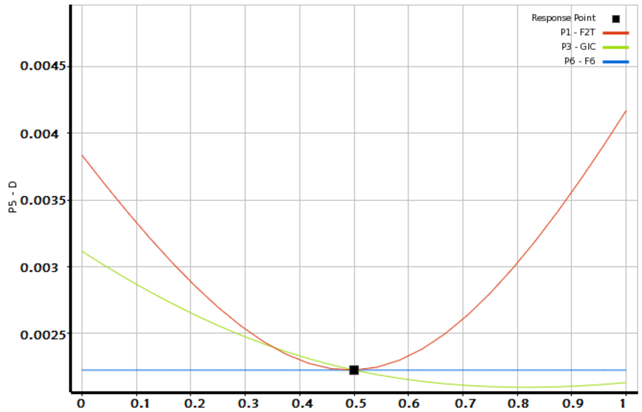


Figure 2.9: Sensitivity of output (error D) to inputs F_{2t}, G_c^{tm}, and F_6.

For minimizing the error D, lower and upper limits must be chosen for each input.

Then with response surface optimization and minimizing the error, three candidate design points are shown in Fig. 2.10. Since the response surface is approximate (performing actual evaluation via FEA for only a few points), direct optimization is used to check for accuracy. Direct optimization performs FEA for all the points explored by the optimization algorithm. Results response surface and direct optimization are compared in Table 2.7.

Table of Schematic C4: Optimization

	A	B	C	D
1	⊟ Optimization Study			
2	Minimize P5	Goal, Minimize P5 (Default importance)		
3	⊟ Optimization Method			
4	Screening	The Screening optimization method uses a simple approach based on sampling and sorting. It supports multiple objectives and constraints as well as all types of input parameters. Usually it is used for preliminary design, which may lead you to apply other methods for more refined optimization results.		
5	Configuration	Generate 1000 samples and find 3 candidates.		
6	Status	Converged after 1000 evaluations.		
7	⊟ Candidate Points			
8		Candidate Point 1	Candidate Point 2	Candidate Point 3
9	P1 - F2T	80.011	79.096	78.136
10	P3 - GIC	26.723	26.829	27.473
11	P5 - D	0.0019932	0.0020194	0.0021065

Figure 2.10: Candidate design points.

Since values of F_{2t} and G_c^{tm} are found with laminate #1, the only parameter that remains to be found is F_6. For this purpose, laminate #8 ($[0/+40/-40/0_{1/2}]_S$) is chosen because it experiences shear stress in the $\pm 40^{\circ}$ laminas. In this way, F_6 has a visible effect on the error (D), as shown in Fig. 2.11.

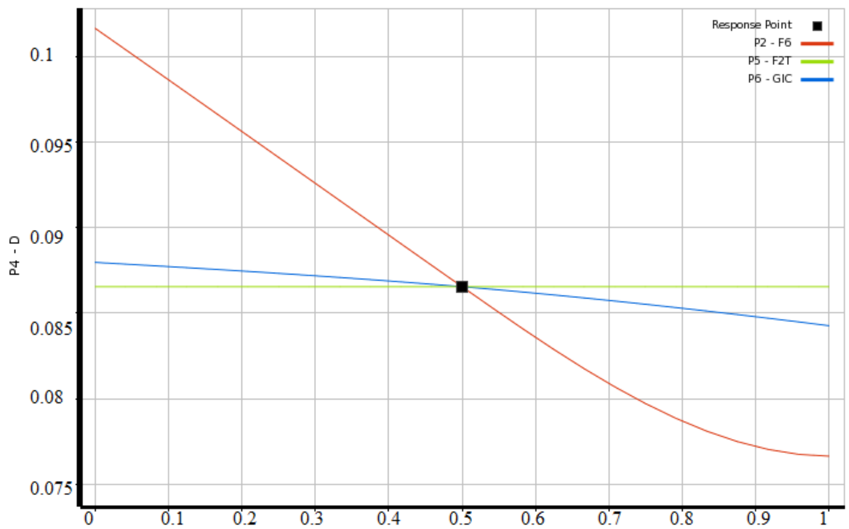


Figure 2.11: Sensitivity curves show how sensitive the output D is to inputs F_{2t}, G^t_m, and F_6 for laminate #8.

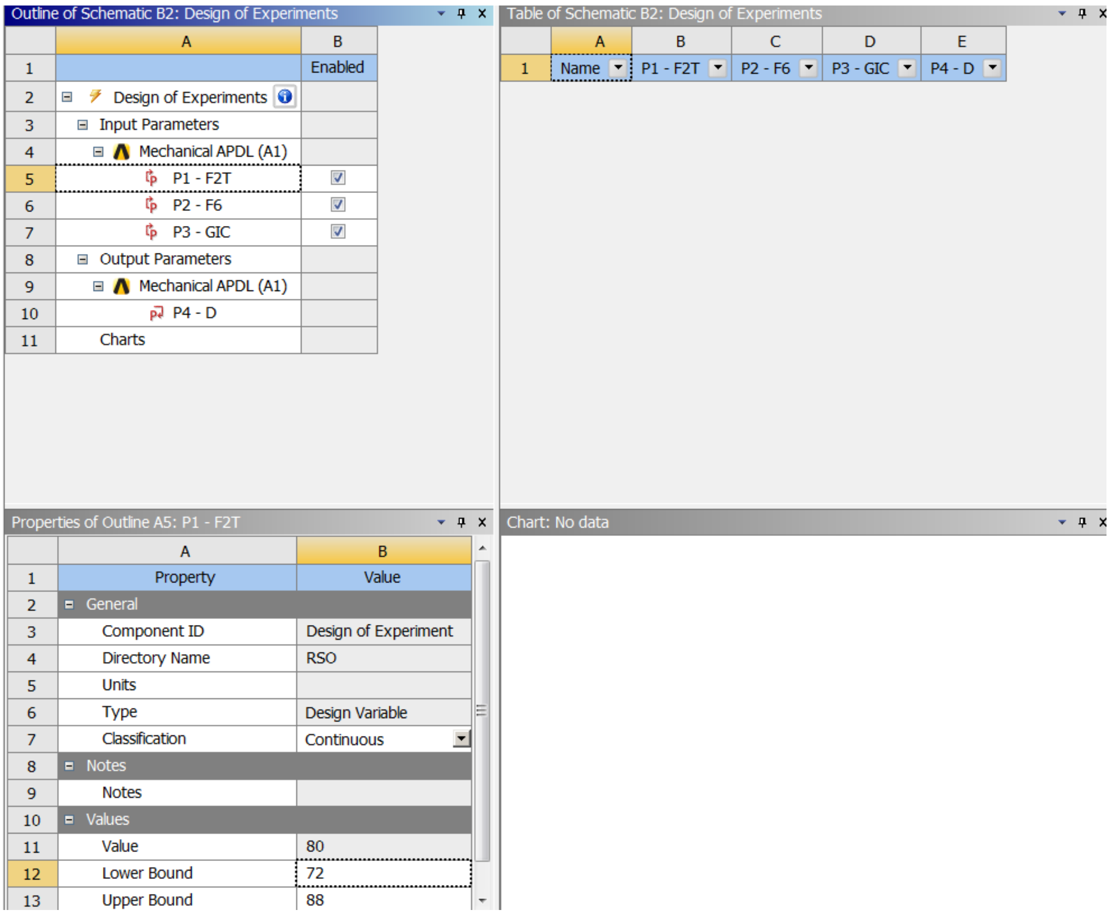


Figure 2.12: Setting the limits (range) for the input parameters.

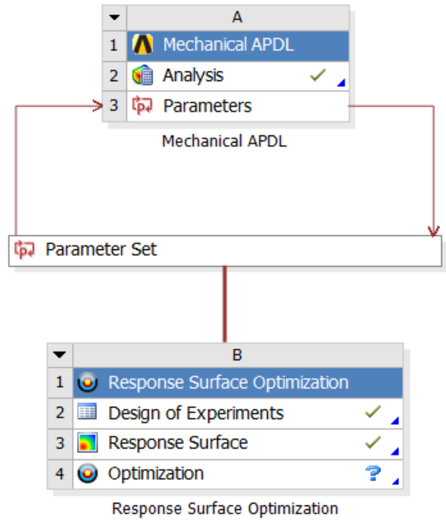
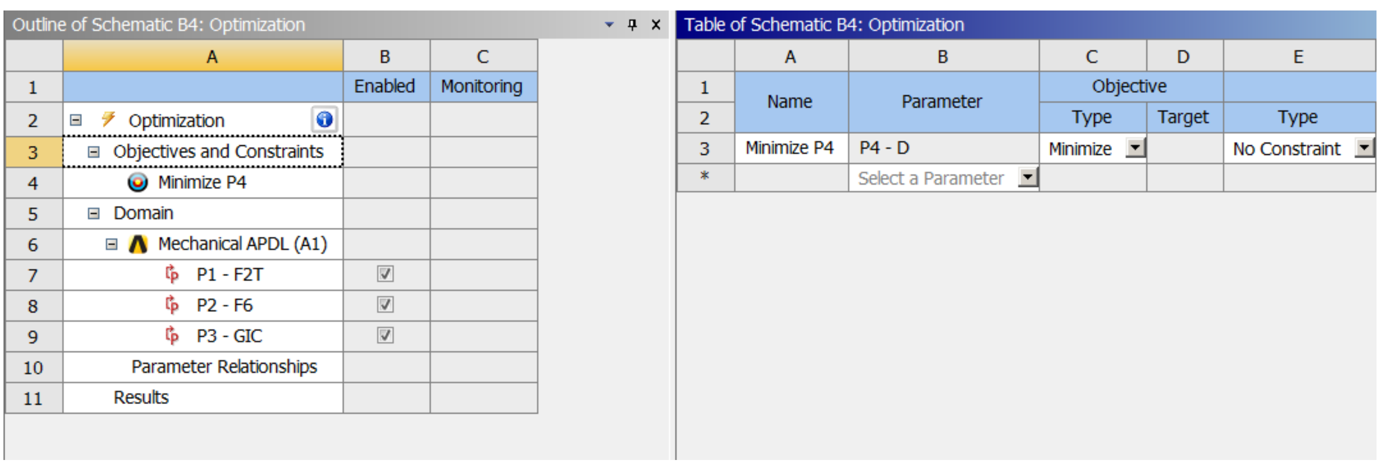


Figure 2.13: Selecting the optimization method.



	A	B	C
1		Enabled	Monitoring
2	Optimization		
3	Objectives and Constraints		
4	Minimize P4		
5	Domain		
6	Mechanical APDL (A1)		
7	P1 - F2T	☑	
8	P2 - F6	☑	
9	P3 - GIC	☑	
10	Parameter Relationships		
11	Results		

	A	B	C	D	E
1	Name	Parameter	Objective		
2			Type	Target	Type
3	Minimize P4	P4 - D	Minimize		No Constraint
*		Select a Parameter			

Figure 2.14: Error (D) is selected to be minimized.

2.4 Comparison with experiments

In this section, predicted laminate modulus $E_x(E_x)$ with parameters listed in Table 2.7 are compared with experimental data for all the laminates. The error for each laminate is reported in Table 2.6.

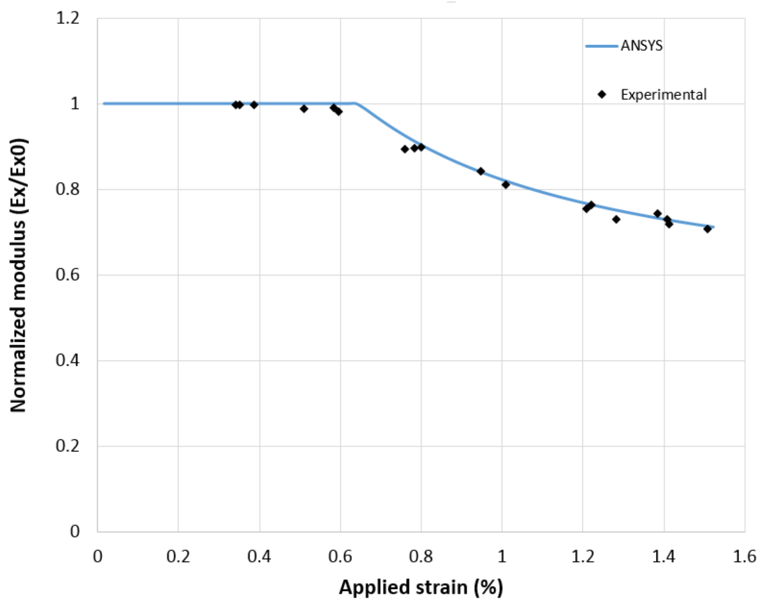


Figure 2.15: Normalized modulus vs. applied strain for laminate #2.

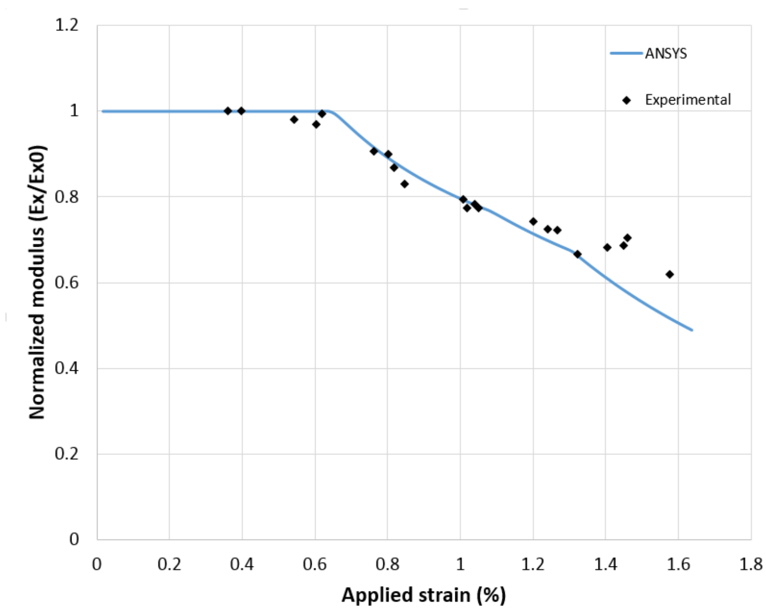


Figure 2.16: Normalized modulus vs. applied strain for laminate #3.

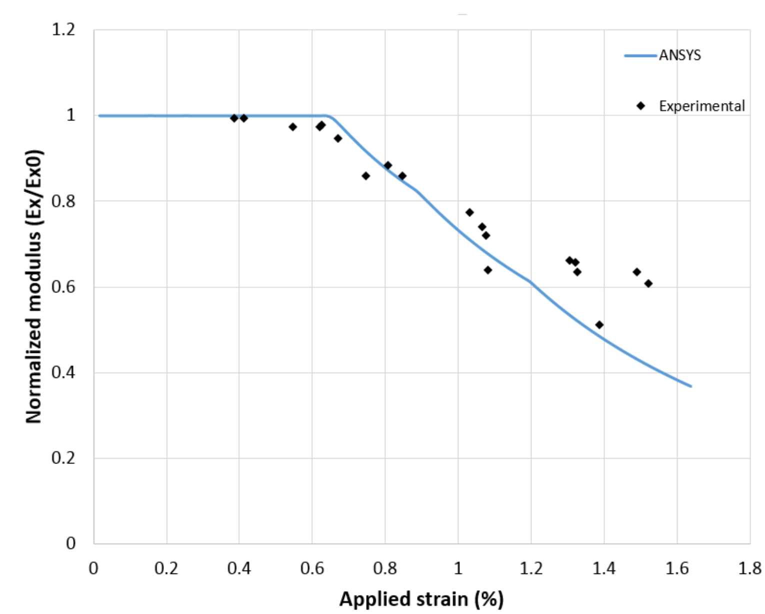


Figure 2.17: Normalized modulus vs. applied strain for laminate #4.

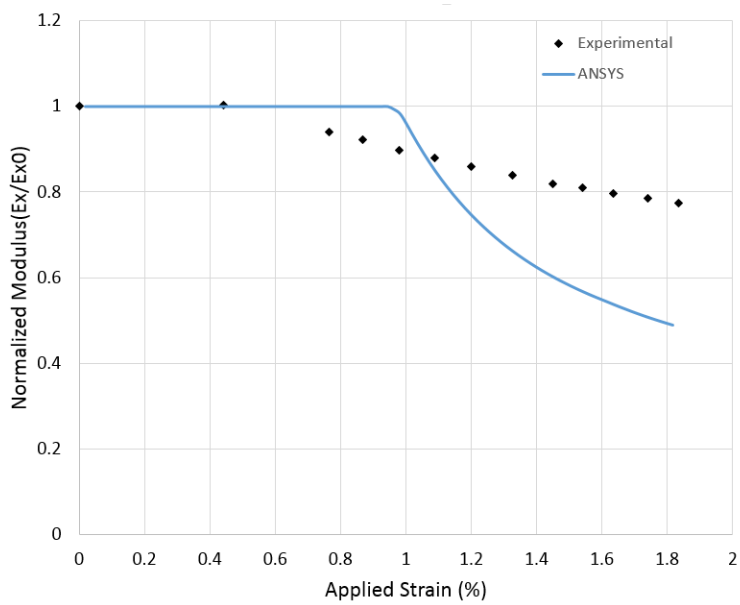


Figure 2.18: Normalized modulus vs. applied strain for laminate #8.

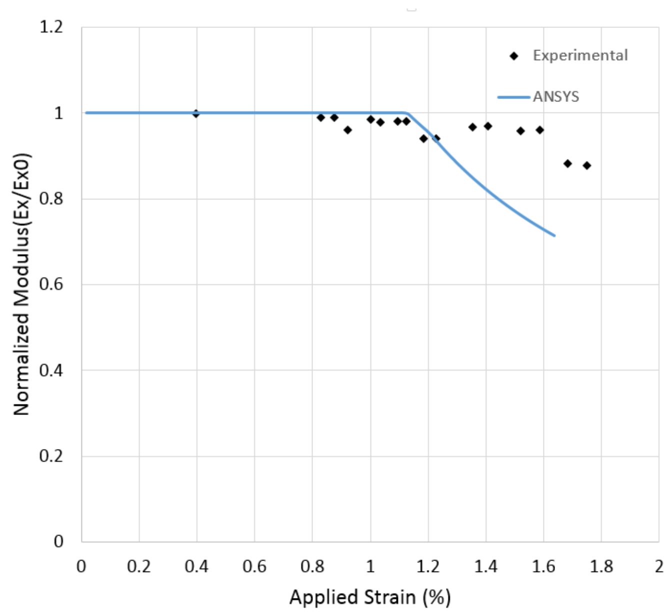


Figure 2.19: Normalized modulus vs. applied strain for laminate #9.

As shown in Fig. 2.18-2.19, ANSYS PDA cannot predict the damage behavior of laminate #8 and laminate #9 as good as damage behavior of laminate #1 to #5, it is because latter laminates do not have to tolerate shear stress due to the angle of fi ers in them (all of them have 90 degree laminas) but laminate #8 and #9 should tolerate shear stress.

2.5 Mesh sensitivity

Mesh sensitivity refers to how much the solution changes with mesh density, number of elements, or number of nodes used to discretize the problem under study. There are two sources of mesh sensitivity. The most obvious is type I sensitivity, where the quality of the solution, particularly stress and strain gradients, depends on mesh density; the fi the mesh, the better the accuracy of the solution. Assuming that the mesh is refined enough to capture stress/strain gradients satisfactorily, type II sensitivity may come from the constitutive model used. When the material response is non-linear, the constitutive model calculates the stress for a given strain and updates one or more state variables to keep track of the history of the material state. Ideally, the response of the constitutive model should be independent of the mesh. To isolate the two sources of mesh dependency, it is customary to test the software with examples for which the strain fi is uniform in the domain regardless of mesh density. The physical tensile test in this study experiences uniform strain everywhere in the rectangular domain representing the gage section of the specimen. Under these conditions, the reaction force calculated by FEA for a given applied strain should be independent of the mesh. There is no type I mesh sensitivity in the calculation of displacement and strain because the strain is uniform in the entire domain. But the reaction force depends on the accuracy of the constitutive model. It can be seen in Fig. 2.20 that PDA is mesh sensitive.

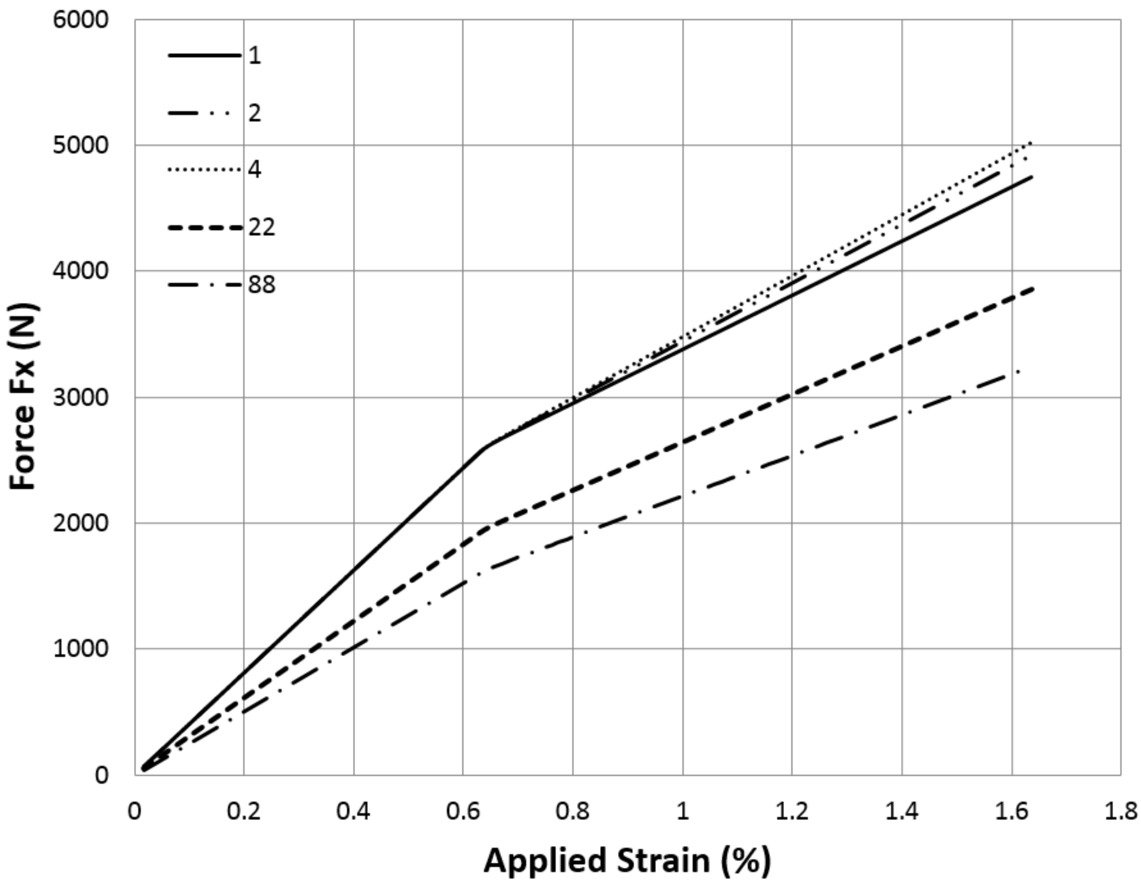


Figure 2.20: Force vs. applied strain for laminate #1 using diff t number of elements.

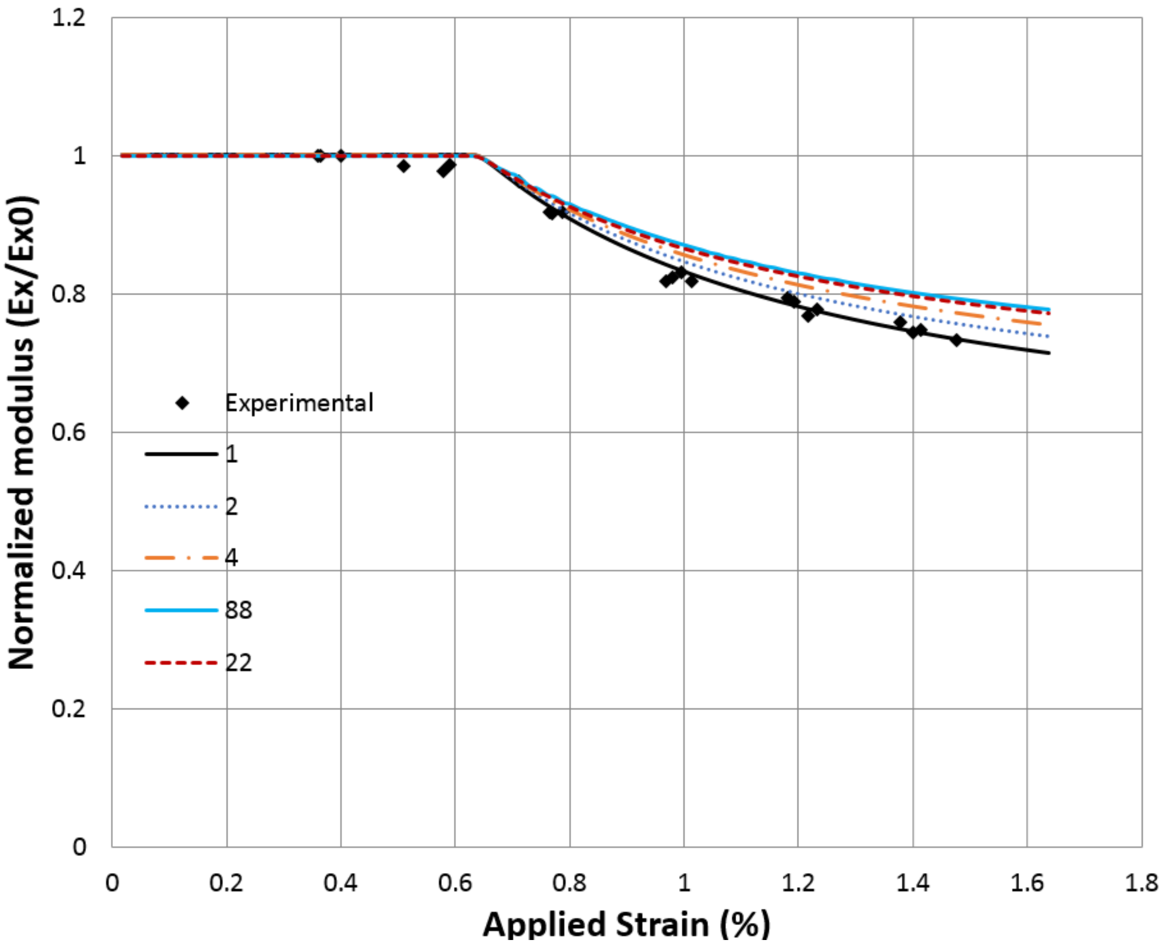


Figure 2.21: Normalized Modulus vs. applied strain for laminate #1 using diff t number of elements.

Chapter 3

Discrete Damage Mechanics (DDM)

DDM [4] is a constitutive modeler that is mesh independent, so DDM does not require the user to choose a characteristic length as in the PDA chapter. Only two material parameters, the fracture toughness in modes I and II, are required to predict both initiation and evolution of transverse and shear damage. Since transverse and shear strengths are not used to predict damage initiation, but rather fracture toughness is used, DDM automatically accounts for in-situ effects. No additional parameters are required to predict damage evolution.

DDM is available to be used in conjunction with commercial FEA environments such as ANSYS/Mechanical [1], in the form USERMAT [5]. Therefore, the objective of this chapter is to propose a methodology to determine values for the material properties required by the DDM model. In this work, the values for the parameters are found using available experimental data and a rational procedure. Once values are found, the DDM model is applied for predicting other, independent results, and conclusions are drawn about the applicability of the model.

An standard test method exist for measuring *interlaminar* fracture toughness in mode I (ASTM D5528) and a proposed method exists for interlaminar mode II [21]. However, no standards exist for *intralaminar* mode I and mode II. *Intralaminar* damage, which is the subject of this thesis, is not the same as *interlaminar* delamination. Therefore, the interlaminar properties cannot be used for predicting intralaminar damage. Instead, the properties can be evaluated as explained in this thesis.

3.1 Discrete Damage Mechanics

By increasing the strain E_x, DDM updates the state variables (crack density λ), and calculate the shell stress resultants N, M, and tangent stiffness matrix A_T, B_T, D_T as functions of crack density. The crack density λ is an array containing the crack density for all laminas at an integration point of the shell element. Since fracture toughness is used to predict damage initiation for DDM model, DDM does not need in-situ correction of strength.

This study shows how to use available data to infer the material properties required by DDM model. Specifically, the main purpose of this study is to find the *critical value of the energy release rate (ERR) in first mode* G_{Ic} and *critical value of ERR in second mode II* G_{IIc} for the material system (composite lamina) that can be used in DDM to predict damage initiation and evolution of laminated composite structures built with the same material system.

3.1.1 Description of DDM Model

In DDM, damage initiation and damage evolution are controlled by an equation which represents the Griffith criterion for an intralaminar crack. Two models have been proposed to represent the undamaged domain. The non-interacting model [6] is

$$g(E,\lambda) = max\left[\frac{G_I(E,\lambda)}{G_{IC}}, \frac{G_{II}(E,\lambda)}{G_{IIC}}\right] - 1 \leq 0 \tag{3.1}$$

where G_I and G_{II} are the strain energy release rates (ERR) in modes I (3.14) and II (3.15), and G_{IC} and G_{IIC} are the invariant material properties representing the critical ERR to create a new crack.

The interacting model [5] is

$$g(E,\lambda) = \left(\frac{G_I(E,\lambda)}{G_{IC}}\right) + \left(\frac{G_{II}(E,\lambda)}{G_{IIC}}\right) - 1 \leq 0 \tag{3.2}$$

G_{IC} and G_{IIC} are of great interest in this study and will be adjusted by minimizing the error between ANSYS data and experimental data using ANSYS Workbench optimiziation tools.

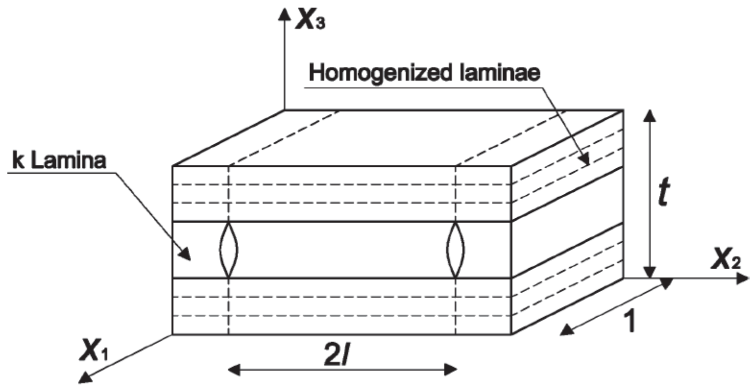


Figure 3.1: Representative volume element (RVE) between two adjacent cracks.

DDM calculates G_I and G_{II} by solving the 3D equilibrium equations in the representative volume element (RVE) between two adjacent cracks (Fig.3.1).

$$v.\sigma - f = 0 \tag{3.3}$$

reduced to 2D by the following method. The u_3 component of displacement is eliminated by assuming a state of plane stress for symmetric laminates under membrane loads,

$$\sigma_3 = 0 \quad ; \quad \frac{\partial u_3}{\partial x_i} \quad \text{with} \quad i = 1, 2 \tag{3.4}$$

Then, (2) is written in terms of the average of the displacements over the thickness of each lamina, defi as

$$\hat{u}_i^{(k)} = \int_{-t_k/2}^{t_k/2} u_i(z)dz \tag{3.5}$$

Where t_k is the thickness of lamina k. The interalaminar shear stress is assumed to be linear in each lamina k, from the interface between laminas $k-1$ and k to the interface between laminas k and $k+1$,

$$\tau_{j3}^{(k)}(x_3) = \tau_{j3}^{k-1,k} + \left(\tau_{j3}^{k,k+1} - \tau_{j3}^{k-1,k}\right)\frac{x_3 - x_3^{k-1,k}}{t_k} \quad ; \quad j = 1, 2 \tag{3.6}$$

Where x_3 is the coordinate indirection of the thickness of laminate, and $x_3^{k-1,k}$ is the coordinate of the interface between laminas $k-1$ and k. Therefore, the 3D equilibrium equations (2) reduce to a 2D second order partial diff tial equations (PDE) as a function of average displacements, with two equations per lamina.

As shown in Fig. 3.1 the crack density is inversely proportional to the length $2l$ of representative volume element (RVE).

$$\lambda = \frac{1}{2l} \tag{3.7}$$

AS seen in (6) the crack density in the DDM model is calculated by the length of the RVE. Since, the RVE is independent of the element size and type, the constitutive model is objective without needing a characteristic length such as element length in ANSYS PDA, so DDM model is mech insensitive that is shown in Fig. 3.30.

The PDE system is complemented by the following boundary conditions. The surface of the cracks in lamina c, located at $x = \pm l$, are free boundaries, and thus subject to zero stress.

$$\int_{-1/2}^{1/2} \hat{\sigma}_j^{(c)}(x_1, l)dx_1 = 0 \quad ; \quad j = 2, 6 \tag{3.8}$$

All laminas except the cracking lamina (c), undergo the same displacement at the boundaries $(-l, l)$ when subjected to a membrane state of strain. Taking an arbitrary lamina $r \neq c$ as a reference, the other displacements are

$$\hat{u}_j^{(m)}(x_1, \pm l) \neq \hat{u}_j^{(r)}(x_1, \pm l) \quad ; \quad m \neq k \quad ; \quad j = 1, 2 \tag{3.9}$$

Finally, the stress resultant from the internal stress equilibrates the applied load. In the parallel direction to the surface of the cracks (fiber direction x_1) the load is supported by all the laminas in the laminate,

$$\frac{1}{2l}\sum_{k=1}^{N} t_k \int_{-l}^{l} \hat{\sigma}_1^{(k)}\left(\frac{1}{2}, x_2\right)dx_2 = N_1 \tag{3.10}$$

but, in the normal direction to the crack surface (x2 direction), only the intact laminas $m \neq c$ carry loads (normal and shear)

$$\sum_{m \neq k} \int_{-1/2}^{1/2} \hat{\sigma}_j^{(m)}(x_1, l)dx_1 = N_j \quad ; \quad j = 2, 6 \tag{3.11}$$

The solution of the PDE system results in fi ding the displacements in all laminas u(k), and by diff tiation, the strains in all laminas. Then, the S matrix of the laminate is calculated by solving three load cases

$$^{a}N/t = \begin{Bmatrix} 1 \\ 0 \\ 0 \end{Bmatrix} ; \quad ^{b}N/t = \begin{Bmatrix} 0 \\ 1 \\ 0 \end{Bmatrix} ; \quad ^{c}N/t = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix} ; \quad \Delta T = 0 \tag{3.12}$$

where t is the thickness of the laminate. Since the three applied stress states are unit values, for each case, a, b, c, the volume average of the strain represents one column in the laminate compliance matrix

$$S = \begin{bmatrix} a_{E_x} & b_{E_x} & c_{E_x} \\ a_{E_y} & b_{E_y} & c_{E_y} \\ a_{\lambda_{xy}} & b_{\lambda_{xy}} & c_{\lambda_{xy}} \end{bmatrix} \tag{3.13}$$

Next, the laminate inplane stiffness $Q = A/t$ in the coordinate system of lamina k is

$$Q = S^{-1} \tag{3.14}$$

The degraded CTE of the laminate $\{\alpha_x, \alpha_y, \alpha_{xy}\}^T$ are given by the values $\{E_x, E_y, \lambda_{xy}\}^T$ obtained for the case with loading $N = \{0, 0, 0\}^T$ and $\Delta T = 1$. Then, the ERR in fracture modes I and II are calculated as follows

$$G_I = -\frac{V_{RVE}}{2\Delta\lambda}(E_2 - \alpha_2 \Delta T)\Delta Q_{2j}(E_j - \alpha_j \Delta T) \quad ; \quad \text{opening mode} \tag{3.15}$$

$$G_{II} = -\frac{V_{RVE}}{2\Delta\lambda}(E_6 - \alpha_6 \Delta T)\Delta Q_{6j}(E_j - \alpha_j \Delta T) \quad ; \quad \text{shearing mode} \tag{3.16}$$

Tearing mode III does not occur because out of plane displacements of the lips of the crack are constrained by the adjacent laminas in the laminate. The crack density is treated as a continuous function, rather than a discrete function. Thus, the crack density is found using a return mapping algorithm (RMA) to satisfy $g = 0$ in (1), as follows

$$\Delta\lambda_k = -\frac{g_k}{\frac{\partial g_k}{\partial \lambda}} \tag{3.17}$$

3.2 DDM Design of Experiments

The next step is to use the design of experiments (DoE) and optimization to adjust the values of G_{Ic} and G_{IIc} so that the DDM prediction closely approximates the experimental data.

First, we use DoE to identify the laminates that are most sensitive to each parameter. The focus at this point is to determine the minimum number of experiments that are needed to adjust the parameters. In this way, additional experiments conducted with diff t laminate stacking sequences (LSS) are not used to adjust parameters but to assess the quality of the predictions.

In principle, the DoE technique is used to fi the location of sampling points in a way that the space of random input parameters $X = \{G_{Ic}, G_{IIc}\}$ is explored in the most efficient way and that the output function D can be obtained with the minimum number of sampling points. In this study, the output function (also called objective function) is the error between the predictions and the experimental data. Given N experimental values of laminate modulus $E(E_i)$, where E is the strain applied to the laminate, and $i = 1 \ldots N$, the error is defi as

$$D = \frac{1}{N}\sqrt{\sum_{i=1}^{N}\left(\frac{E^{ANSYS}}{E_0}\Big|_{\varepsilon=\varepsilon_i} - \frac{E^{Experimental}}{E_0}\Big|_{\varepsilon=\varepsilon_i}\right)^2} \tag{3.18}$$

where E and E_0 are laminate elastic modulus for damaged and undamaged laminate, respectively, and N is the number of experimental data points.

DoE tools can be found in ANSYS Workbench in the Design Explorer (DE) module, which includes also Direct Optimization (DO), Parameters Correlation, Response Surface (RS), Response Surface Optimization, and Six Sigma Analysis.

Let's denote the input parameters by the array $X = \{G_{Ic}, G_{IIc}\}$. In this case, the output function $D = D(X)$ can be calculated by evaluation of eq. (3.18) through execution of the fi element analysis (FEA) code for N values of strain. Each FEA analysis is controlled by the APDL script, which calls for the evaluation of the non-linear response of the damaging laminate for each value of strain, with parameters X. If the mesh is refined, these evaluations could be computationally intensive.

An alternative to direct evaluation of the output function is to approximate it with a multivariate quadratic polynomial. The approximation is called response surface (RS). It can be constructed with only few actual evaluations of the output by choosing a small number of sampling points for the input. The sampling points are chosen using Design of Experiments (DoE) theory. The number of evaluations needed to construct the response surface (RS) and for direct optimization (DO) are shown in Table 3.1.

		# of evaluations	
# of inputs	Inputs	Response Surface	DO (Adaptive Single-Objective)
1	G_{Ic}	5	9
2	G_{Ic}, G_{IIc}	9	21

Table 3.1: Number of FEA evaluations used (a) to construct the response surface (RS) and (b) to adjust the input parameters by direct optimization (DO). Interacting equation (3.2) is used.

The shape of the RS can be inferred by observing the variation of the output D as a function of only one input at a time. This is shown in Figures 3.2 and 3.3 for laminate #1 (Table 3.2). The abscissa spans each of the inputs and the ordinate measures the error between predicted and experimental data.

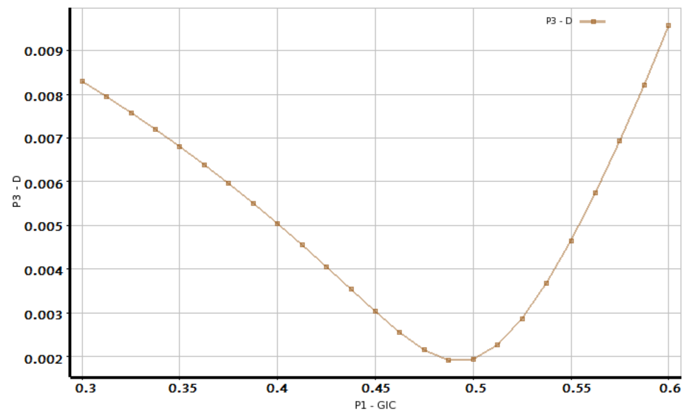


Figure 3.2: Response surface chart. Error (D) vs. G_{IC}.

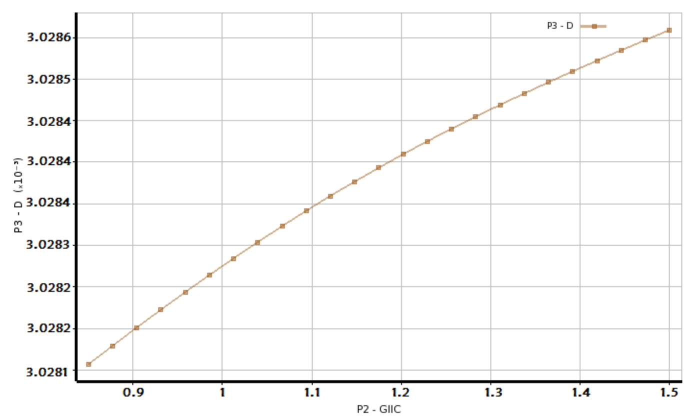


Figure 3.3: Response surface chart. Error (D) vs. G_{IIC}.

By observing the RS we can understand which parameters (inputs) have the most effect on the error (output). Since y axis is in 1e-3 scale in Figure 3.3 it is clear that the error is not sensitive to G_{IIc} for laminate #1. This is because no lamina in laminate #1 is subject to shear. Therefore, it is decided to use the experimental data of laminate #1 to evaluate only (G_{Ic}). This makes the optimization algorithm to run faster.

The RS is multivariate quadratic polynomial that approximates the actual output function $D(X)$ as a function of the input variables. In this study, $D_{RS} = f(X_i)$. The sensitivity S of the output to input X in the user specified interval $[X_{min}, X_{max}]$ is calculated as:

Laminate #	LSS
1	$[0_2/90_4]_S$
2	$[\pm 15/90_4]_S$
3	$[\pm 30/90_4]_S$
4	$[\pm 40/90_4]_S$
5	$[0/90_8/0_{1/2}]_S$
6	$[0/\pm 70_4/0_{1/2}]_S$
7	$[0/\pm 55_4/0_{1/2}]_S$
8	$[0/\pm 40_4/0_{1/2}]_S$
9	$[0/\pm 25_4/0_{1/2}]_S$

Table 3.2: Laminate stacking sequence for all laminates for which experimental data is available.

$$S = \frac{max(D) - min(D)}{average(D)} \tag{3.19}$$

and tabulated in Table 3.3 for laminate # 1. Note that the sensitivity can be calculated from the error evaluated directly from FEA analysis or from the RS. The later is much more expedient than the former, as it can be seen in Table 3.1.

	Input range		Error D			Sensitivity
Input	min(Input)	max(Input)	min(D)	max(D)	ave(D)	S
G_{Ic}	0.3	0.6	0.0019	0.0096	0.00542	1.4136
G_{IIc}	0.9	1.5	0.02524	0.0379	0.03007	0.42101

Table 3.3: Sensitivity S of the output (error) to each input (parameter). First row refers to laminate #1 and last row to laminate #8. Interacting equation (3.2) is used.

The chart in Figure 3.2 is drawn for the input ranges given in Table 3.3. It is convenient to compare all of them in one chart (Figure 3.8), where the input range has been normalized to the interval [0–1].

3.3 Methodology

The input parameters can be adjusted with any mesh and any type of elements that represent the gauge section of the specimen, or a single element to represent a single material point of the specimen. For expediency, a single linear element (PLANE 182) is used in this study.

3.3.1 APDL

The APDL script is used to call the usermaterial (DLL fi specify the mesh, boundary conditions, and the strain applied to the laminate. The later is specified by imposing a specified displacement. Incrementation of the applied displacement is implemented to mimic the experimental data, which is available for a fi set of values of applied strain.

The APDL script is used also to specify the elastic properties (with TB command, Table 3.4), the laminate stacking sequence (with TB command, Table 3.2), the the critical ERRs (with TB command, Table 3.4). In Table 3.4, the critical ERRs that are adjusted (G_{Ic}, G_{IIc}) are simply initial (guess) values for the optimization.

Also the APDL script contains the geometry of the specimen. The dimensions of the specimen are 20mm wide and 110mm free length. All the laminates considered for the study are symmetric and balanced. Therefore a quarter of the specimen was used for the analysis using symmetry boundary conditions and applying a uniform strain with imposed displacements on one end of the specimen. A longitudinal displacement of 1.1mm was applied to reach a strain of 2%.

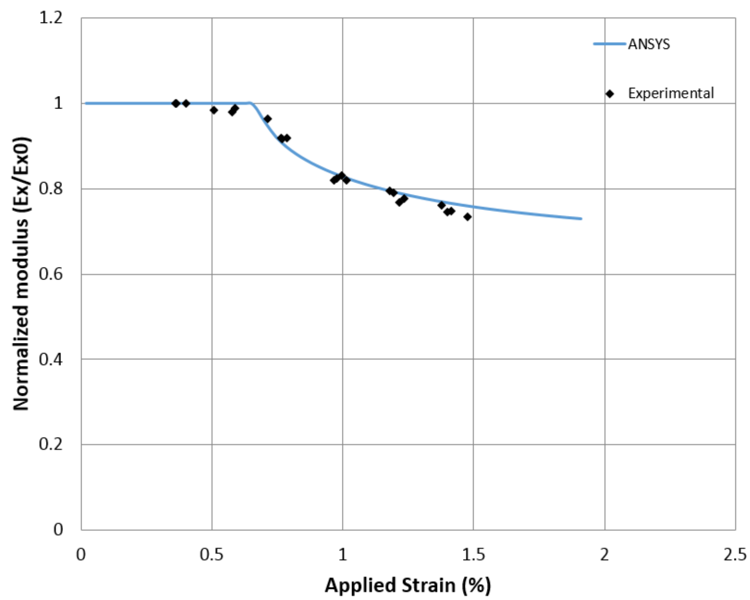


Figure 3.4: Normalized modulus vs. applied strain for laminate #1 (interacting eq.3.2]).

The APDL script also includes a table with experimental data for the laminate being analyzed. Such data consists of a number N of pairs of values representing laminate modulus as a function of applied strain $E_x(E_x)$. Finally, the APDL script calculates the error as per eq. (3.18).

Property	Units	Value	Ref.
E_1	MPa	44700	[24]
E_2	MPa	12700	[24]
G_{12}	MPa	5800	[24]
v_{12}	-	0.297	[24]
v_{23}	-	0.411	[24]
Ply thickness	mm	0.144	[25]
G_{Ic}	KJ/m^2	0.254	guess value
G_{IIc}	KJ/m^2	1.4	guess value
CTE_1	MPa	3.7	[6]
CTE_2	MPa	30	[6]
ΔT	MPa	0	[6]

Table 3.4: Lamina elastic properties and in-situ strength values.

Property	Units	Value	Ref.
G_f^t	KJ/m^2	1E6	high value
G_f^c	KJ/m^2	1E6	high value
G_c^{tm}	KJ/m^2	25	guess value
G_m^c	KJ/m^2	1E6	high value
η_f^t	-	1E-3	immaterial
η_f^c	-	1E-3	immaterial
η_m^t	-	5E-3	trial and error
η_m^c	-	1E-3	immaterial

Table 3.5: Damage evolution properties of the lamina.

3.3.2 Workbench

First, a *Mechanical APDL* component is added to the Project Schematic by dragging it from the Component Systems menu. The APDL code is then imported into Workbench. See Figure 3.5.

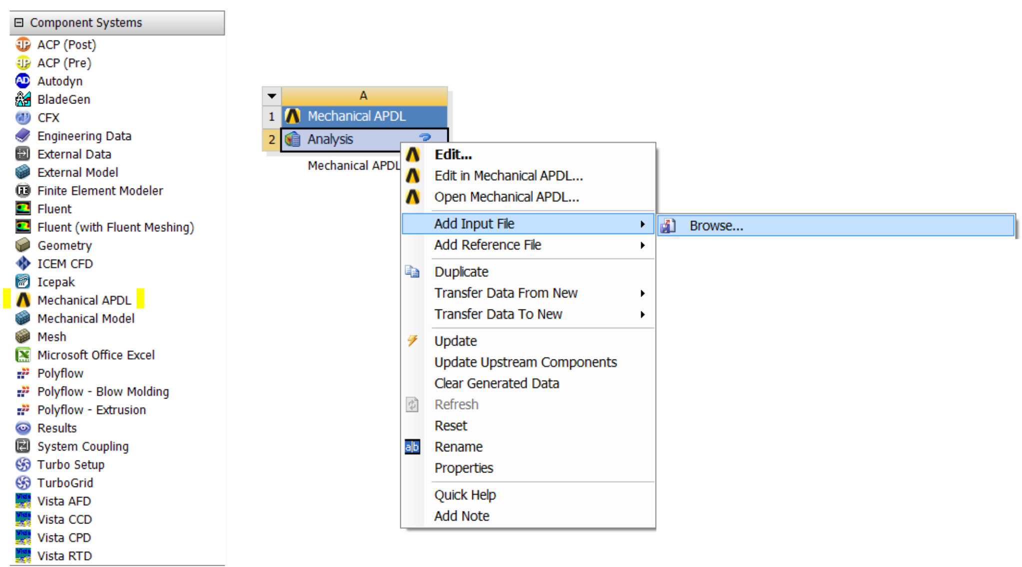
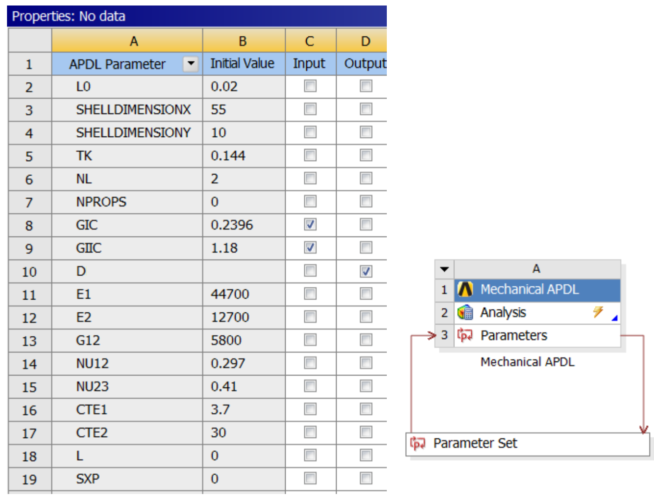


Figure 3.5: Importing the APDL code into Workbench.

Next, from among all the parameters defi in the APDL script, the input parameters ($F2t$, F_6, G_c^{mt}) and output parameters (D) are selected as shown in Figure 3.6.



	A	B	C	D
1	APDL Parameter	Initial Value	Input	Output
2	L0	0.02	☐	☐
3	SHELLDIMENSIONX	55	☐	☐
4	SHELLDIMENSIONY	10	☐	☐
5	TK	0.144	☐	☐
6	NL	2	☐	☐
7	NPROPS	0	☐	☐
8	GIC	0.2396	☑	☐
9	GIIC	1.18	☑	☐
10	D		☐	☑
11	E1	44700	☐	☐
12	E2	12700	☐	☐
13	G12	5800	☐	☐
14	NU12	0.297	☐	☐
15	NU23	0.41	☐	☐
16	CTE1	3.7	☐	☐
17	CTE2	30	☐	☐
18	L	0	☐	☐
19	SXP	0	☐	☐

Figure 3.6: Inputs and output parameters are selected.

3.3.3 Optimization

Optimization techniques are used in this study to minimize the error (3.18) by adjusting the input (material) parameters. In these way, the fi values of the parameters represent materials properties for the specific PDA material model used in the underlying FEA.

An *Response Surface Optimization* (RSO) component is now added to the Workbench by dragging it from the Component Systems menu to the Project Schematic (Figure 3.7).

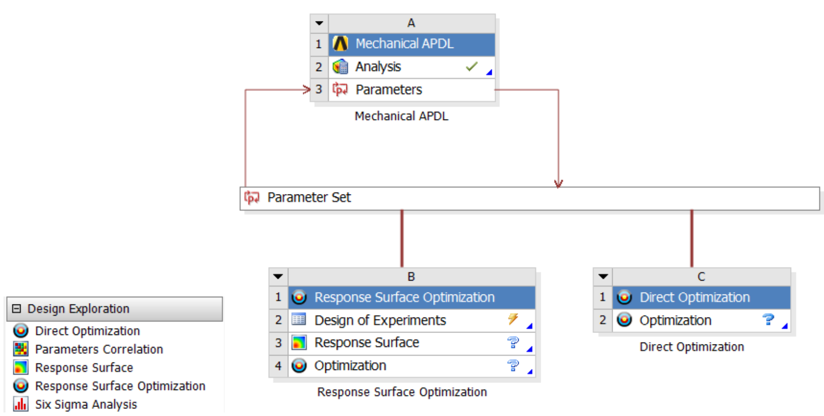


Figure 3.7: Response Surface tools include DoE, RS, and RS Optimization.

Then, DoE is used to adjust a multivariate quadratic polynomial to the actual response (output) of the system as defi by the APDL script. In this study the output is the error (3.18). The multivariate are the input parameters, which in this chapter are two parameters.

Then, the RS is used to plot the response (output) vs. each of the input parameters and to calculate the sensitivities. This allows the user to select, for optimization, only the parameters to which the output (error) is sensitive.

Within RSO, optimization is performed by using the RS rather than actual evaluation of the response via fi element analysis (FEA). This results in significant savings of computer time, as shown in Table 3.1, but the result is approximate because the RS is an approximation to the actual output function.

To get exact optimum parameters (within numerical accuracy) one has to conduct Direct Optimization (DO). It can be seen in Table 3.6 that RS is quite accurate when compared with DO, considering that the number of FEA evaluations (reported in Table 3.1) is smaller for RS than for DO.

	Error D	
Laminate	RS	DO
#1	0.002986	0.002937
#2	0.004918	0.005179
#3	0.005760	0.005891
#4	0.011550	0.011797
#5	0.013845	0.014660
#6	0.035843	0.037435
#7	0.002859	0.004631
#8	0.025511	0.026829
#9	0.011437	0.009954

Table 3.6: Error and adjusted values of input (material) parameters for all laminates considered. Eq. (3.2) is used. Values of GIc and GIIc are given in Table 3.7.

As it is shown in Table 3.7, the accuracy of the parameters is good when RS is used instead of DO. A cost comparison, in terms of number of FEA evaluations, is shown in Table 3.1, where it can be seen that DO is much more expensive.

	Optimization method	
Parameter	RSO (Response Surface)	DO (Adaptive Single-Objective)
G_{IC}	0.4285	0.437
G_{IIC}	0.96597	1.0205
Max # of FEM evaluations	9	21

Table 3.7: Comparison of adjusted input (material) parameters obtained by using Response Surface Optimization (RSO) and Direct Optimization (DO). Eq. (3.2) is used.

Twenty two experimental data points are available for laminate #1 and nineteen experimental data points are available for laminate #8. Laminate #1 ($[0_2/90_4]_S$) was chosen because this laminates is sensitive to G_{IC}. On the contrary, G_{IIC} does not have any effect on the results of Laminate #1, as shown in Table 3.3 and Fig. 3.8.

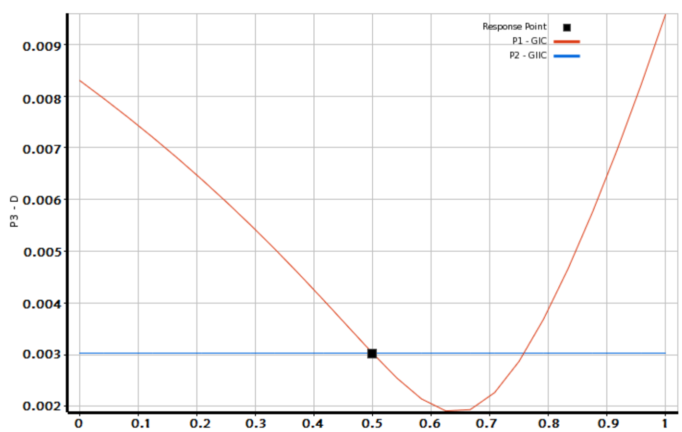


Figure 3.8: Sensitivity of output (error D) to inputs G_{IC} and G_{IIC}.

For minimizing the error D, lower and upper limits must be chosen for each input.

Then with response surface optimization and minimizing the error, three candidate design points are shown in Fig. 3.9. Since the response surface is approximate (performing actual evaluation via FEA for only a few points), direct optimization is used to check for accuracy. Direct optimization performs FEA for all the points explored by the optimization algorithm. Results response surface and direct optimization are compared in Table 3.7.

Table of Schematic B4: Optimization , Candidate Points

	A	B	C	D	E	F
1	Reference	Name	P1 - GIC	P2 - GIIC	P3 - D	
2					Parameter Value	Variation from Reference
3	◉	Candidate Point 1	0.49277	0.85829	0.0018861	0.00 %
4	○	Candidate Point 2	0.49275	1.016	0.0018861	0.00 %
5	○	Candidate Point 3	0.49305	1.341	0.0018862	0.01 %
*		*New Custom Candidate Point*	*0.45*	*1.175*		

Figure 3.9: Candidate design points.

Since values of G_{IC} is found with laminate #1, the only parameter that remains to be found is G_{IIC}. For this purpose, laminate #8 ($[0/+40/-40/0_{1/2}]_S$) is chosen because it experiences shear stress in the $\pm 40^\circ$ laminas. In this way, G_{IIC} has a visible effect on the error (D), as shown in Table 3.3.

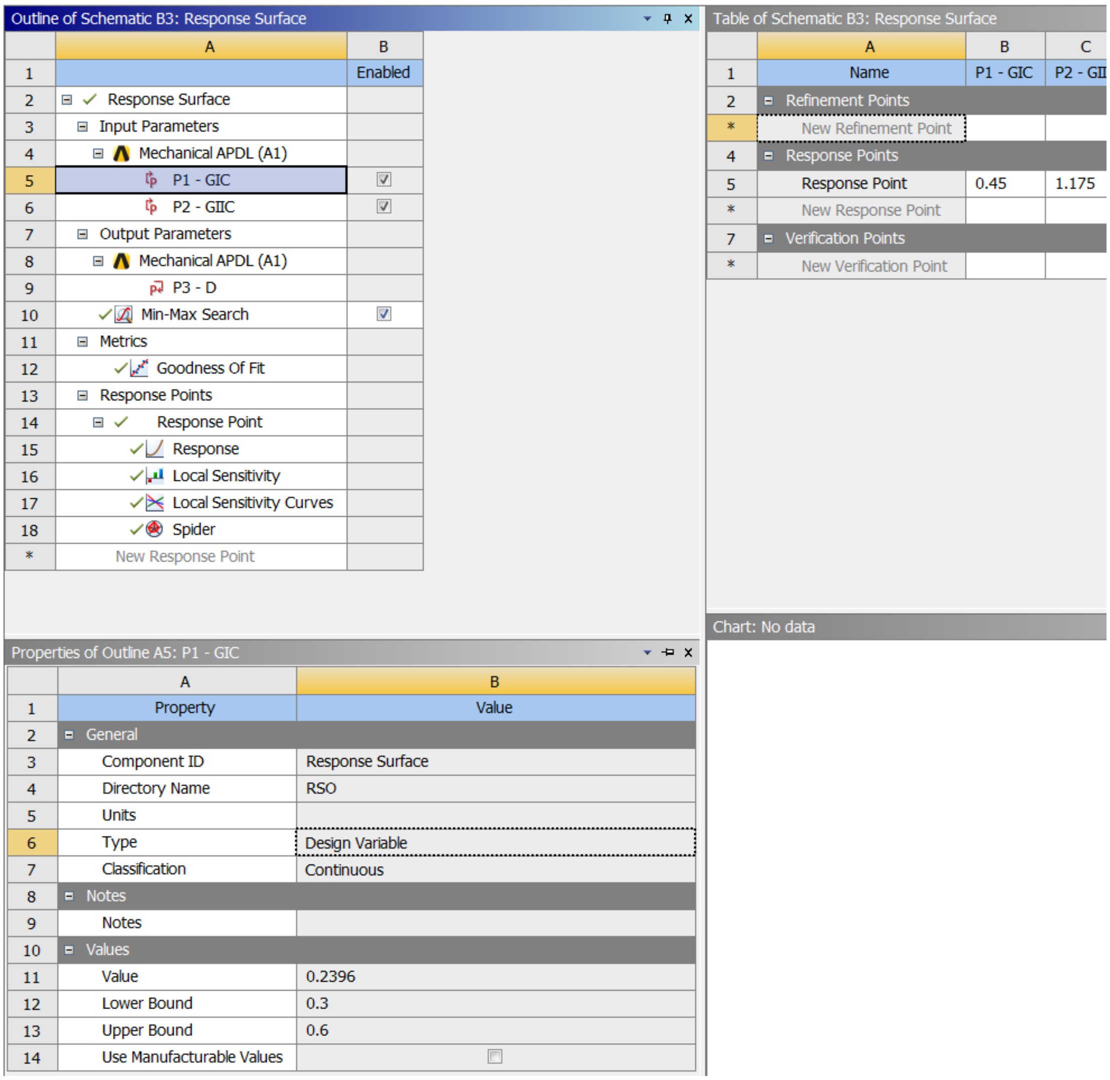


Figure 3.10: Setting the limits (range) for the input parameters.

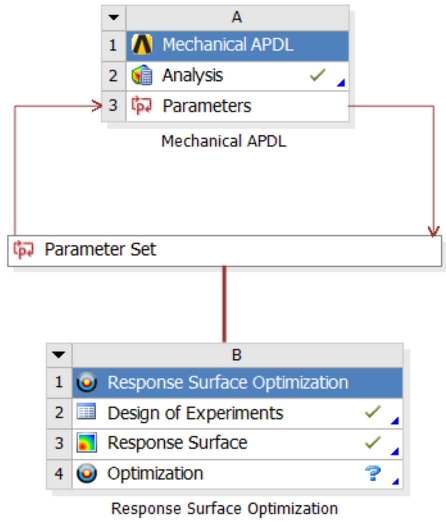


Figure 3.11: Selecting the optimization method.

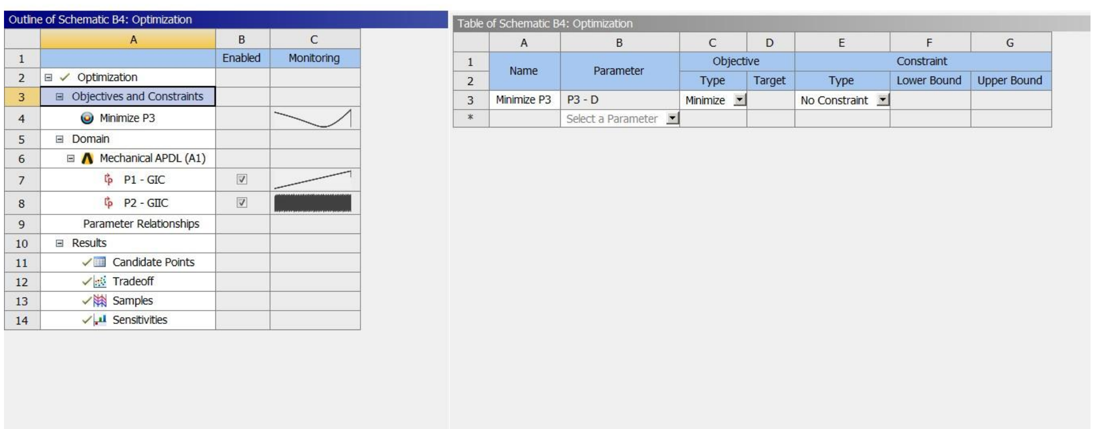


Figure 3.12: Error (D) is selected to be minimized.

3.4 Comparison with experiments

In this section, predicted laminate modulus $E_x(E_x)$ with parameters listed in Table 3.7 are compared with experimental data for all the laminates. The error for each laminate is reported in Table 3.6.

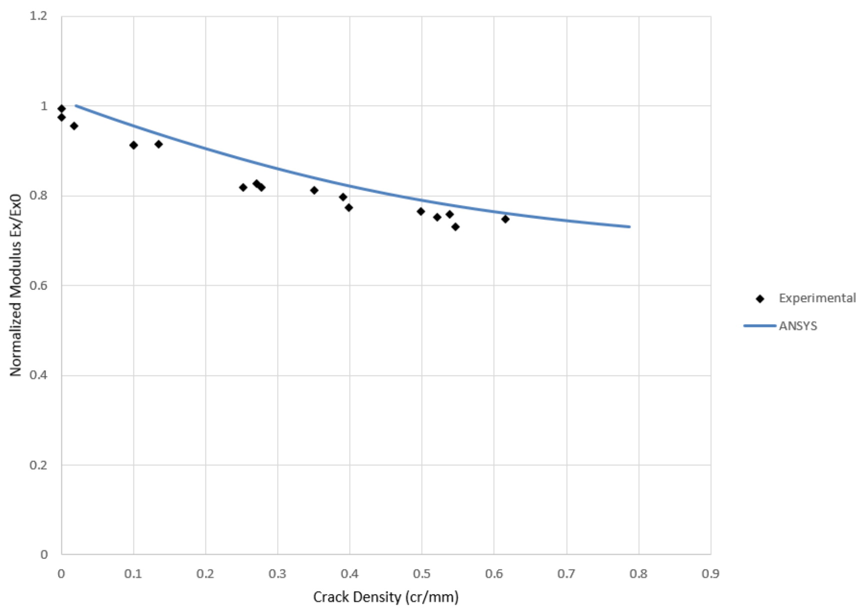


Figure 3.13: ANSYS DDM and experimental data normalized modulus vs. crack density (cr/mm) curves for laminate #1.

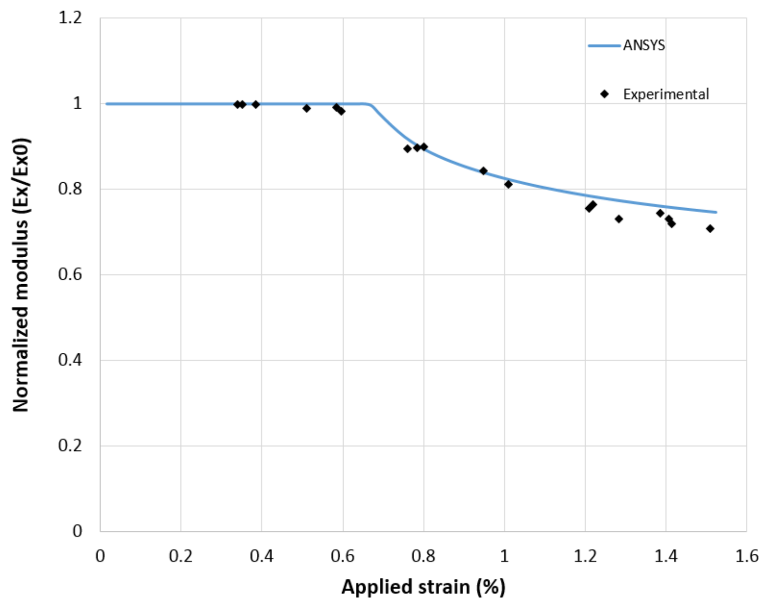


Figure 3.14: Normalized modulus vs. applied strain for laminate #2.

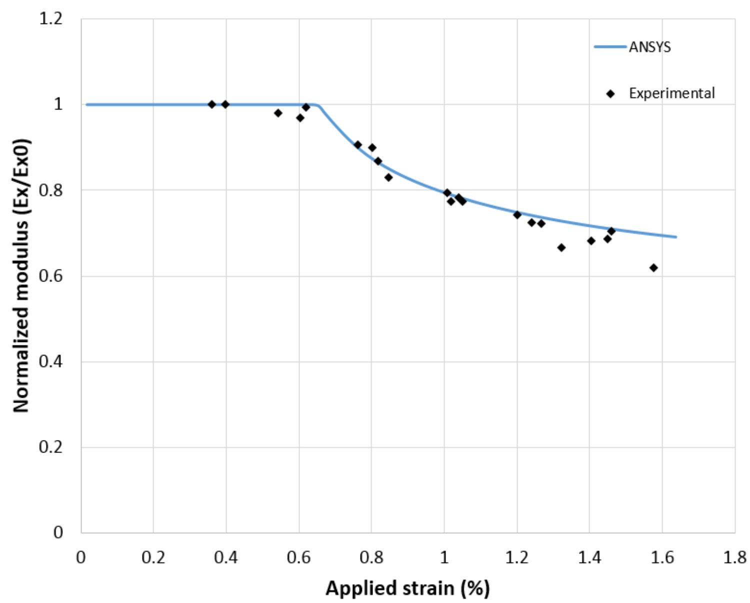


Figure 3.15: Normalized modulus vs. applied strain for laminate #3.

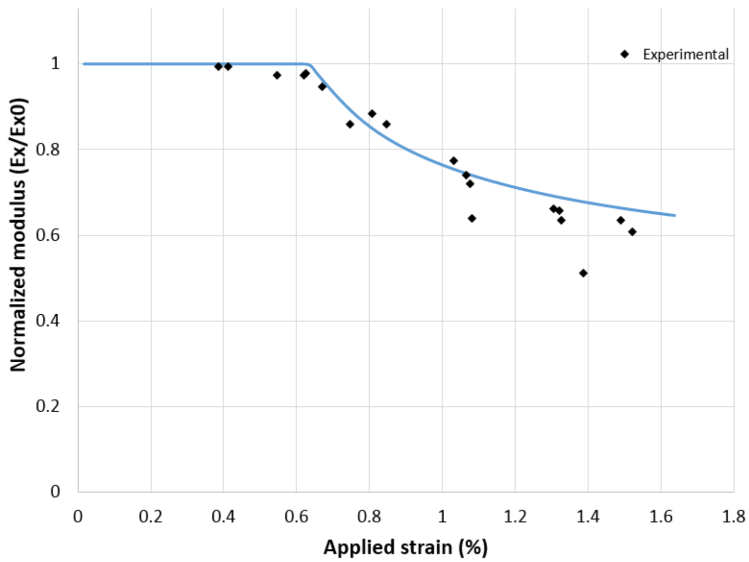


Figure 3.16: Normalized modulus vs. applied strain for laminate #4.

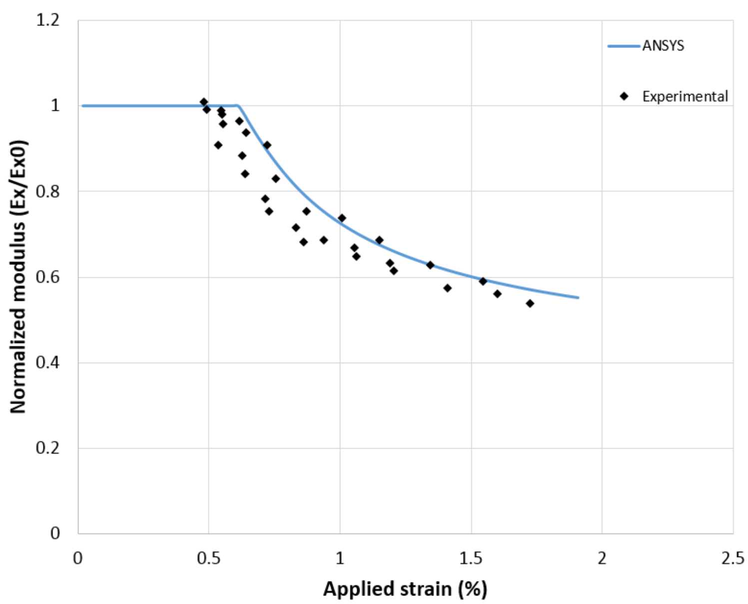


Figure 3.17: Normalized modulus vs. applied strain for laminate #5.

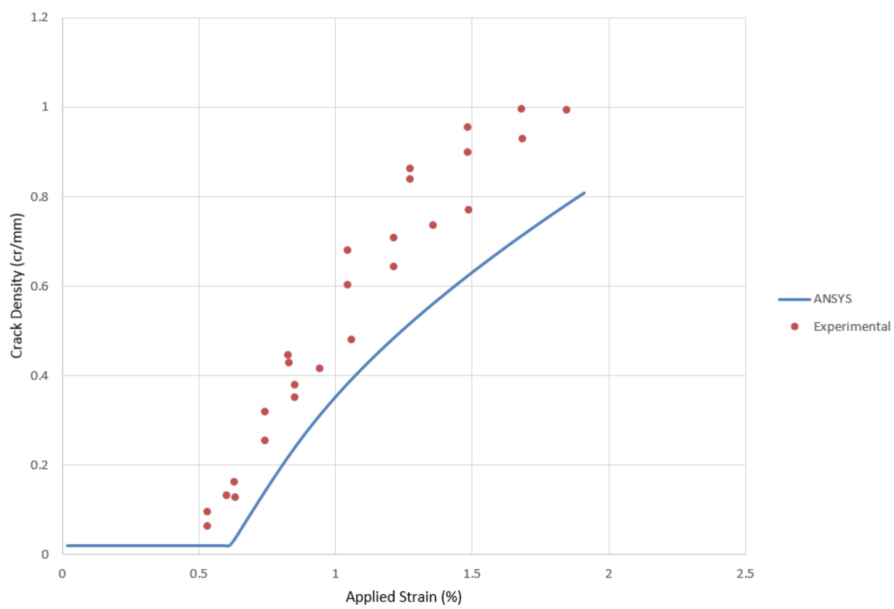


Figure 3.18: ANSYS DDM and experimental data crack density (cr/mm) vs. applied strain curves for laminate #5.

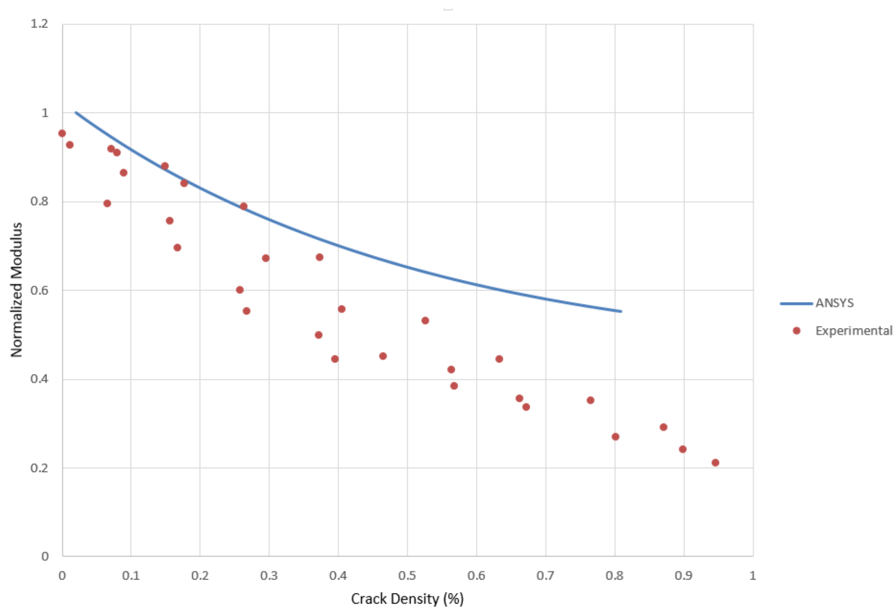


Figure 3.19: ANSYS DDM and experimental data normalized modulus vs. crack density (cr/mm) curves for laminate #5.

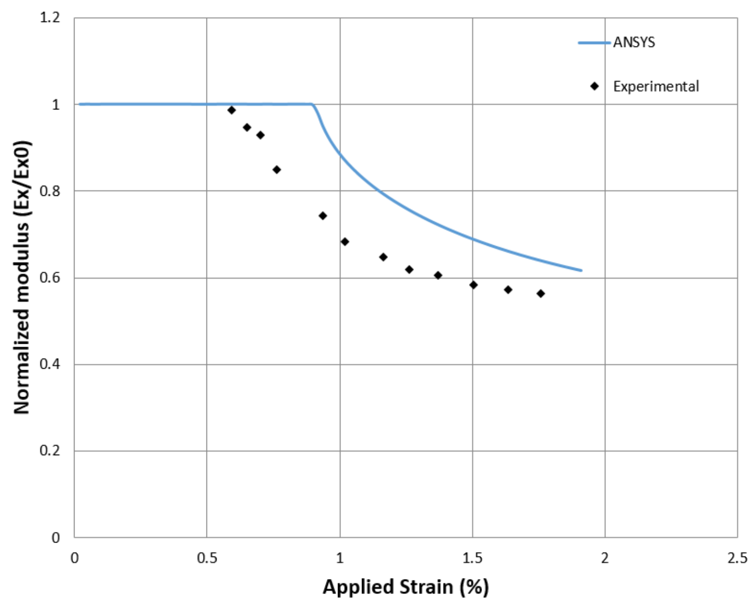


Figure 3.20: Normalized modulus vs. applied strain for laminate #6.

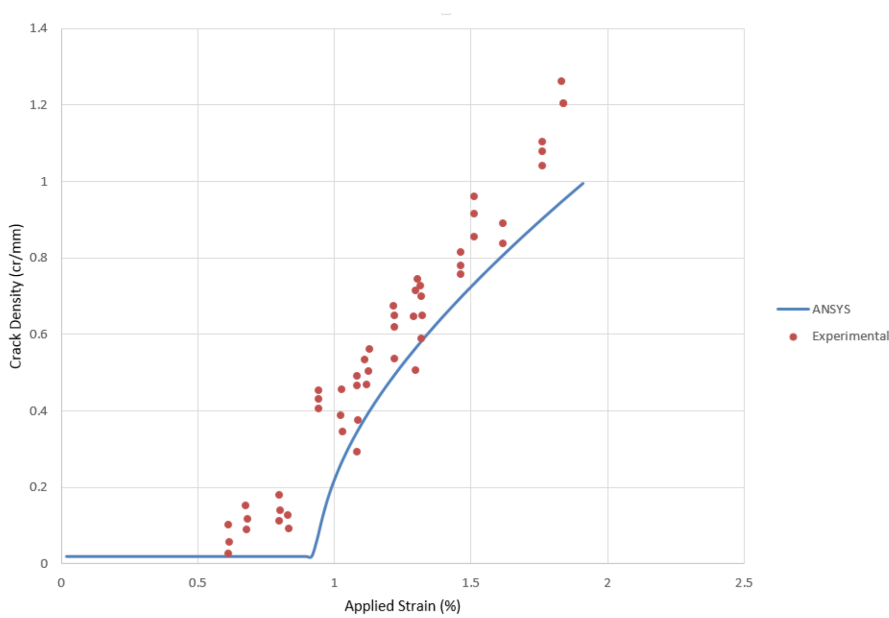


Figure 3.21: ANSYS DDM and experimental data crack density (cr/mm) vs. applied strain curves for laminate #6.

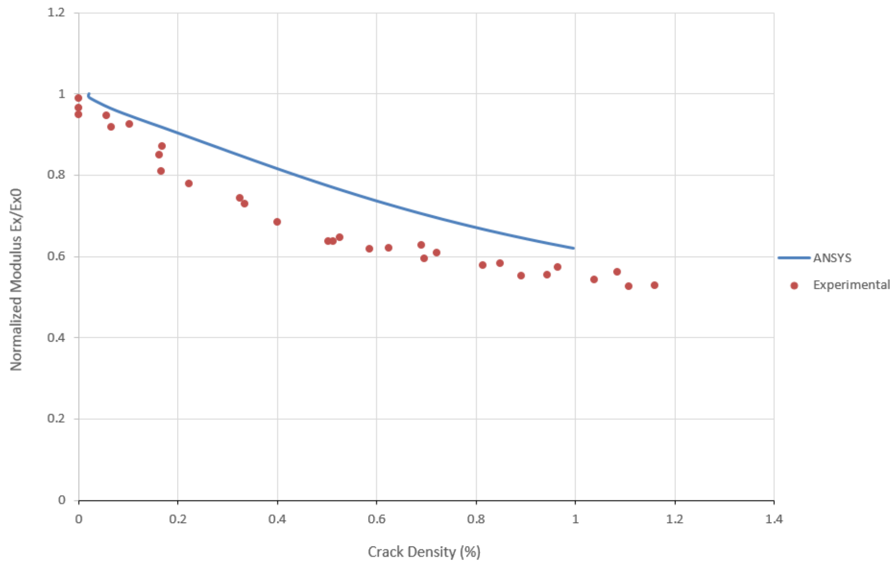


Figure 3.22: ANSYS DDM and experimental data normalized modulus vs. crack density (cr/mm) curves for laminate #6.

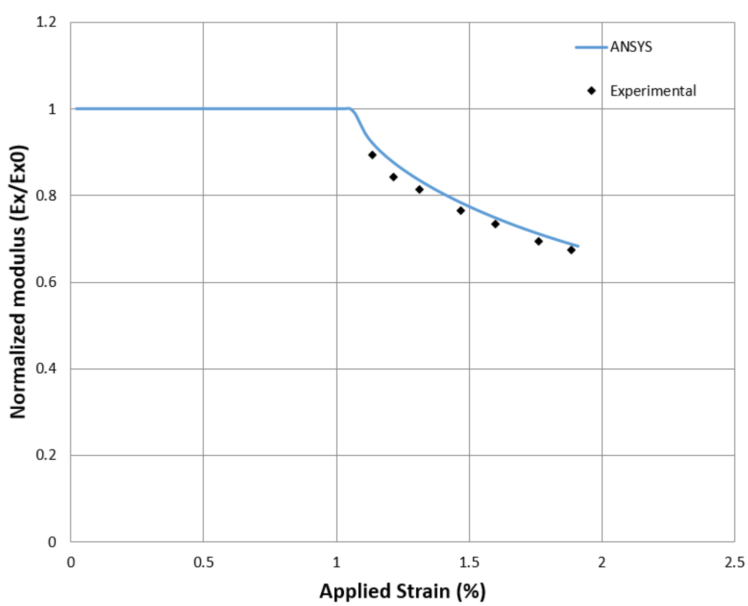


Figure 3.23: Normalized modulus vs. applied strain for laminate #7.

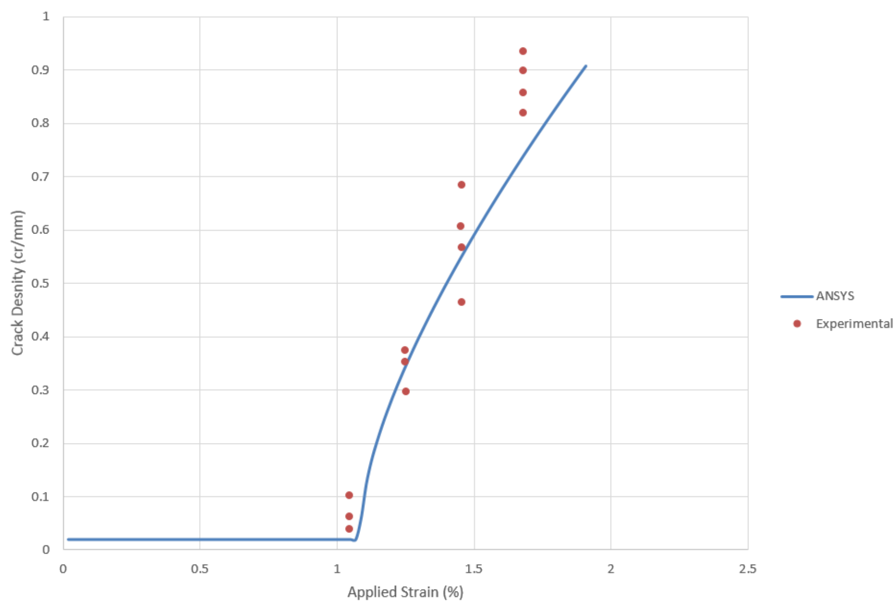


Figure 3.24: ANSYS DDM and experimental data crack density (cr/mm) vs. applied strain curves for laminate #7.

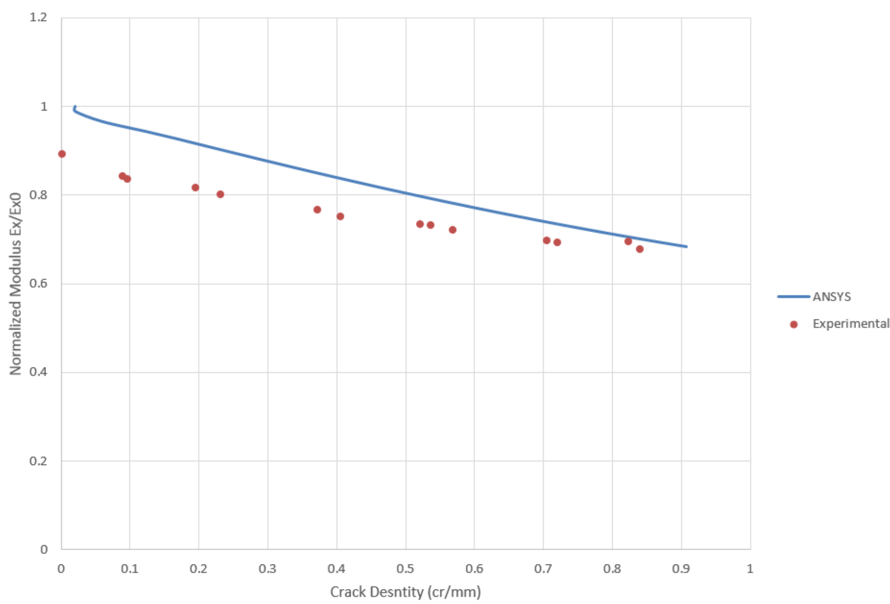


Figure 3.25: ANSYS DDM and experimental data normalized modulus vs. crack density (cr/mm) curves for laminate #7.

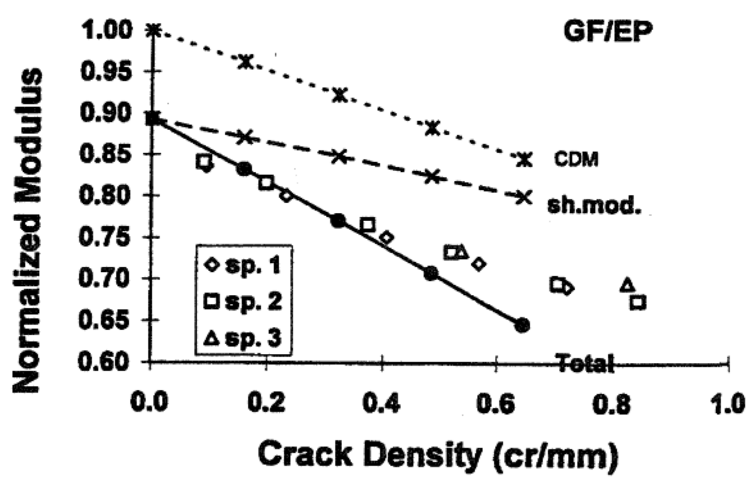


Figure 3.26: Experimental data of Normalized modulus vs. Crack density for laminate #7.

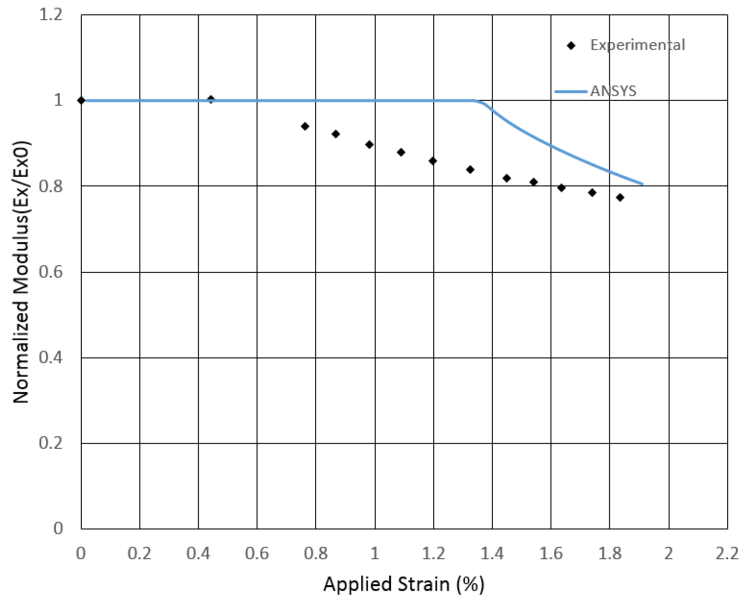


Figure 3.27: Normalized modulus vs. applied strain for laminate #8.

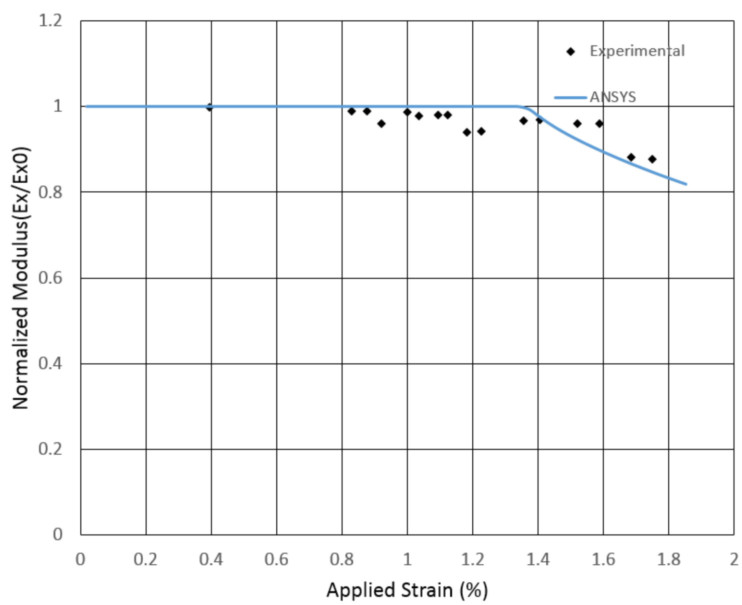


Figure 3.28: Normalized modulus vs. applied strain for laminate #9.

As shown in Figures of this section (Fig.3.13–Fig.3.28) the adjusted values work well for almost all types of laminate except Laminates #6 and #8.

The only way to fi laminate #6's results is to decrease the G_{Ic} to 0.23 instead of current adjusted value which is 0.438, but it is not a good idea because it changes the results for laminate #1 to #5, and since we adjusted the G_{Ic} by laminate #1 it is reasonable to have the best fi for laminate #1.

About Laminate #8, the discrepancies can be eliminated by decreasing the G_{IIc}, but ANSYS crashes for G_{IIc} less than 0.8, so it was not possible to check the values less than 0.8 by ANSYS.

3.5 Mesh sensitivity

Mesh sensitivity refers to how much the solution changes with mesh density, number of elements, or number of nodes used to discretize the problem under study. There are two sources of mesh sensitivity. The most obvious is type I sensitivity, where the quality of the solution, particularly stress and strain gradients, depends on mesh density; the fi the mesh, the better the accuracy of the solution. Assuming that the mesh is refined enough to capture stress/strain gradients satisfactorily, type II sensitivity may come from the constitutive model used. When the material response is non-linear, the constitutive model calculates the stress for a given strain and updates one or more state variables to keep track of the history of the material state. Ideally, the response of the constitutive model should be independent of the mesh. To isolate the two sources of mesh dependency, it is customary to test the software with examples for which the strain fi is uniform in the domain regardless of mesh density. The physical tensile test in this study experiences uniform strain everywhere in the rectangular domain representing the gage section of the specimen. Under these conditions, the reaction force calculated by FEA for a given applied strain should be independent of the mesh. There is no type I mesh sensitivity in the calculation of displacement and strain because the strain is uniform in the entire domain. But the reaction force depends on the accuracy of the constitutive model. It can be seen in Fig. 3.30 that DDM is mesh insensitive.

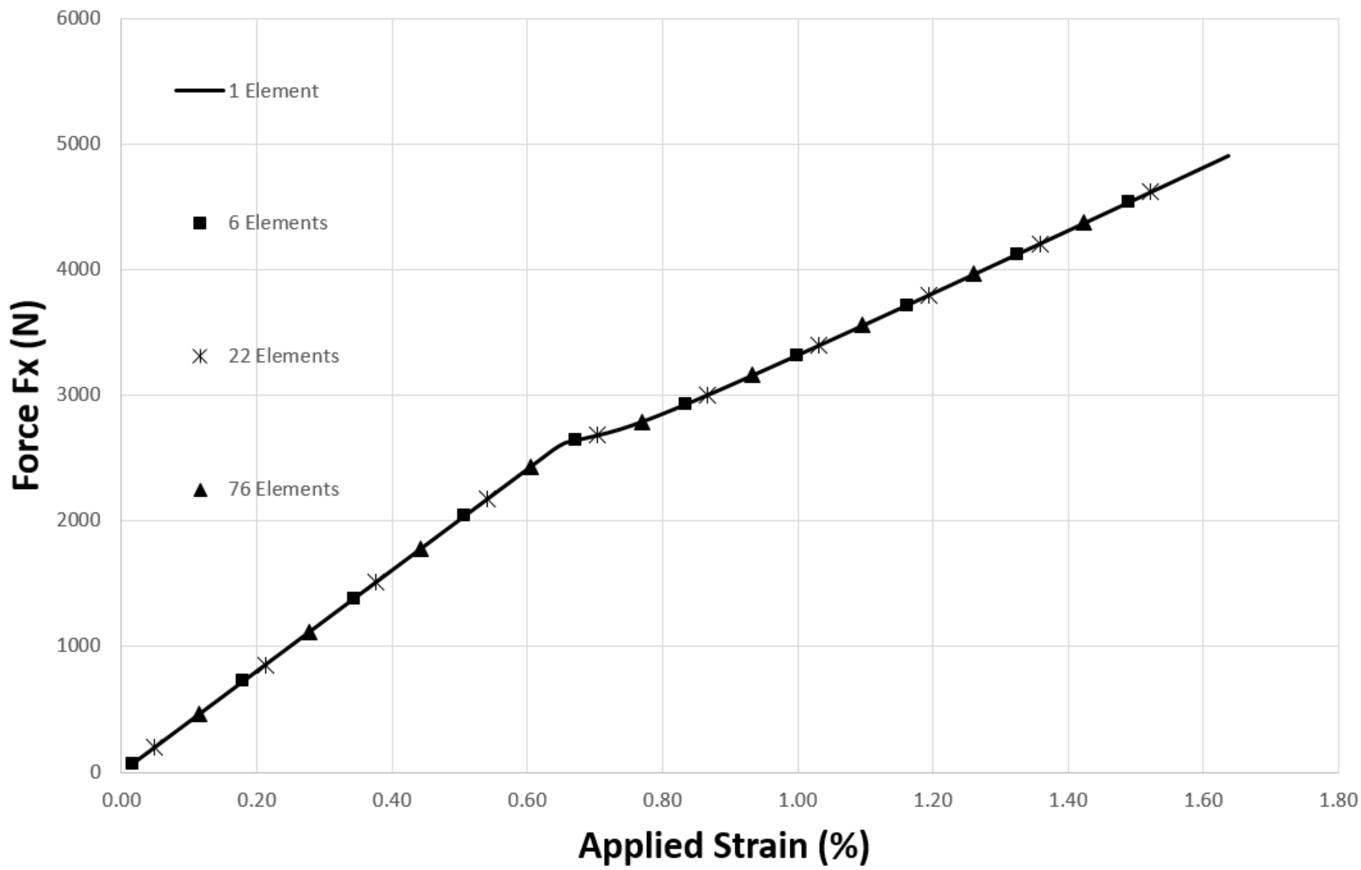


Figure 3.29: Force Fx vs. applied strain for laminate #1 using diff rent number of elements

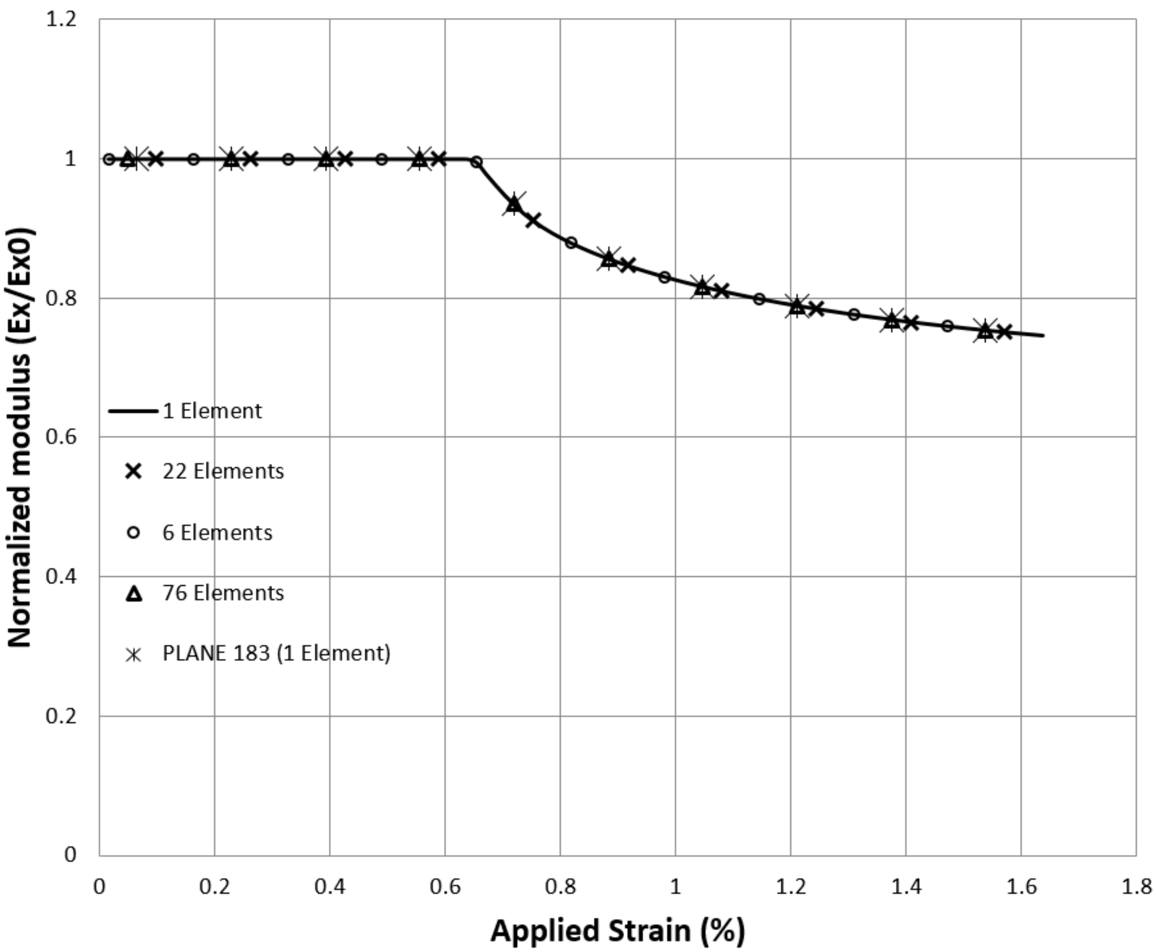


Figure 3.30: Normalized Modulus vs. applied strain for laminate #1 using diff t number of elements for PLANE 182 and one element for PLANE 183.

3.6 Effect of damage activation function

In Fig. 3.31 the results for two DDM models are compared to each other for laminate #7. One of the DDM models uses the non-interacting equation (3.1) and the other one uses interacting equation (3.2). Fig. 3.31 shows that prediction of the damage initiation and damage evolution with DDM model using interacting (3.2) is much better than the prediction of DDM model using (3.1).

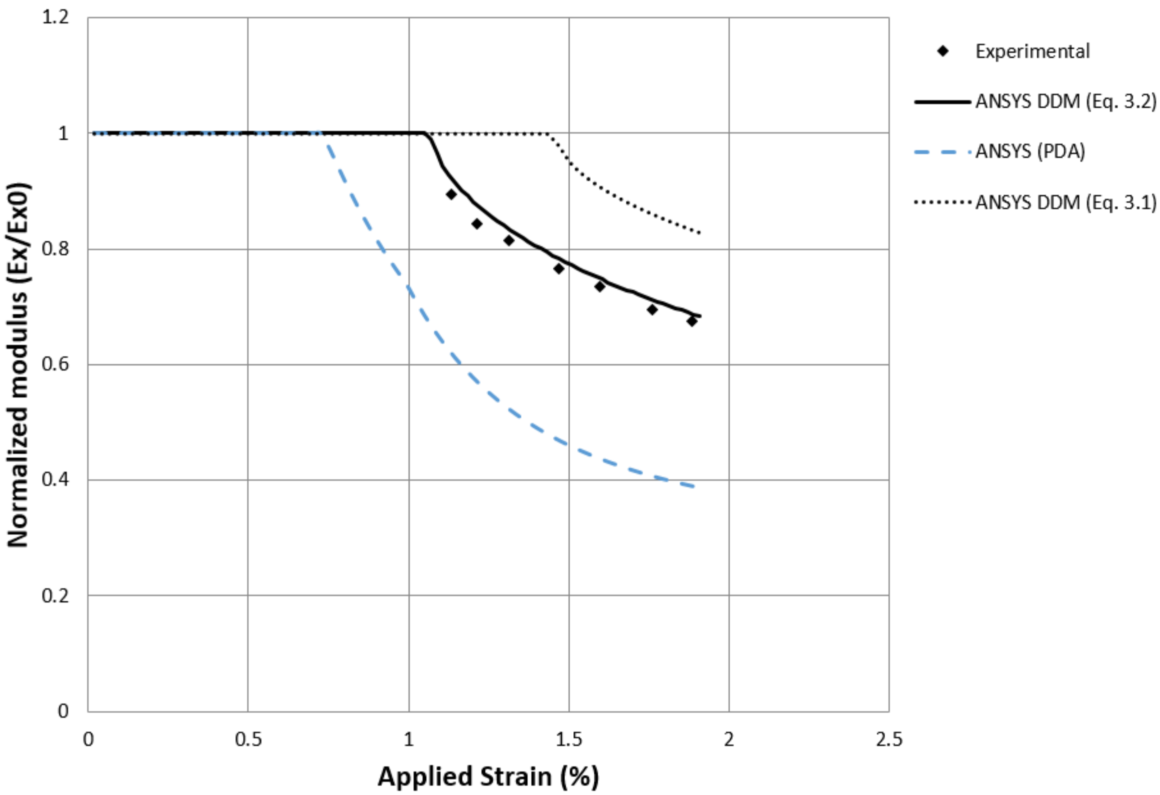


Figure 3.31: Normalized Modulus vs. applied strain for laminate #7 3.1, 3.2, and PDA results.

Chapter 4

Conclusions and Future Work

4.1 Conclusions

This study shows that adjusted transverse and shear strengths (in situ values) predict the damage initiation and evolution for the PDA by comparing the implemented data from ANSYS with available experimental data for nine diff t laminates. Also, the determination of material parameters for DDM give the best prediction damage behavior for E-Glass Epoxy laminated composite. In other words, ANSYS users can use the adjusted values in this thesis to predict damage behavior of any laminated composite they need and get a good prediction for Normalized Modulus Vs. Applied strain with PDA or DDM model (Apendix A.1 or A.2). Also, the user can predict the Crack Density Vs. Applied Strain and Normalized Modulus Vs. Crack Density plots with DDM model and adjusted material parameters in this thesis, as shown in (Fig. 3.25 and 3.24).

As shown in chapters 2 and 3, this study uses the same optimization method to adjust the material parameters for both PDA and DDM models. This methodology can be used for different optimization purposes, and is explained step by step in this thesis that give this opportunity to the ANSYS users to use the strong ANSYS optimization tools instead of writing optimization codes in MATLAB and linking MATLAB with ANSYS. Regarding the mesh sensitivity, ANSYS PDA is dependent to the mesh type and element size, although DDM model does not show any sensitivity to neither p- or h-refi ment. It shows that DDM adjusted values work for all mesh type and element sizes, but for PDA the material parameters can be diff t for small elements or diff t types of element that can be considered as future work.

Also, the DDM model works better with interacting eq.3.2, since Fig.3.31 shows that prediction of the damage initiation and damage evolution with DDM model using interacting eq.3.2 is so much better than the prediction of DDM model using eq.3.1

4.2 Comparison between DDM and PDA

The normalized modulus vs. applied strain curves of PDA and DDM model are plotted and compared for laminate #1 to laminate #9 in this section, as shown in Fig. 4.1 to Fig. 4.9.

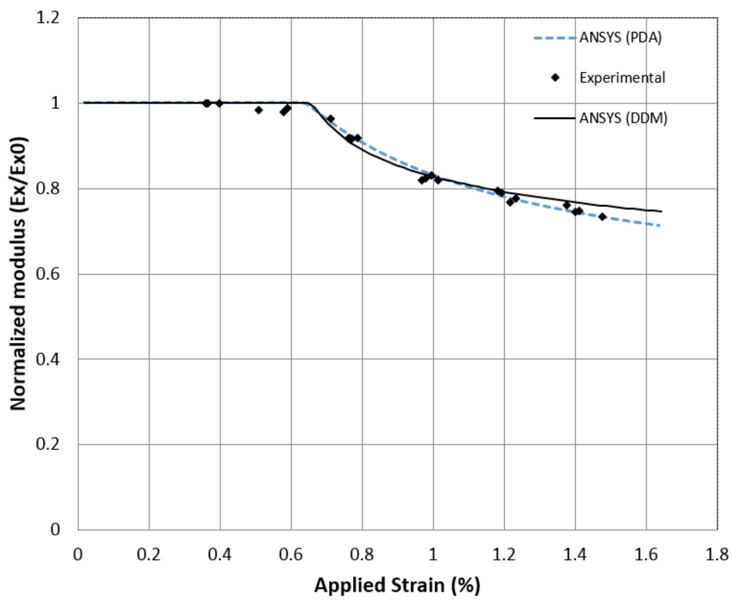


Figure 4.1: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #1.

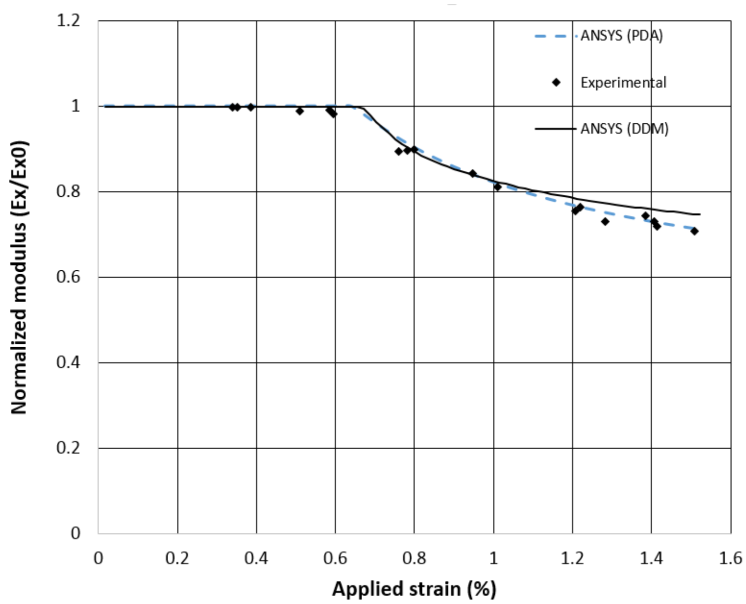


Figure 4.2: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #2.

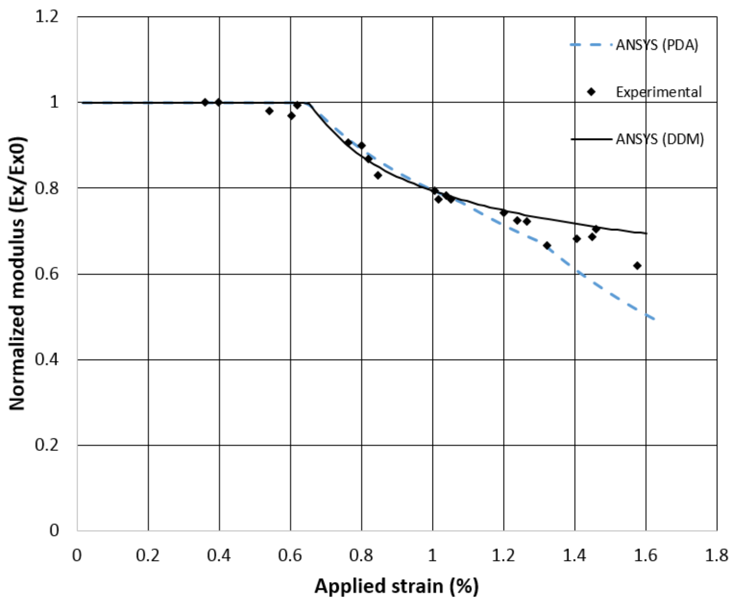


Figure 4.3: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #3.

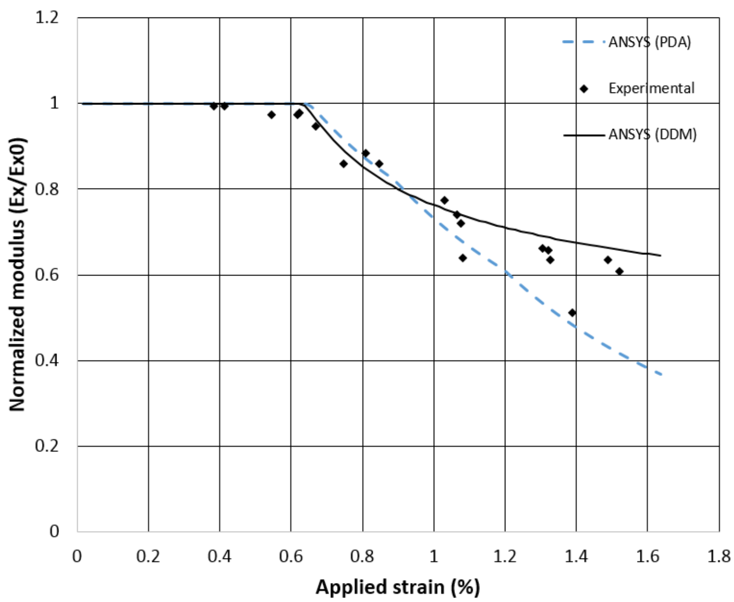


Figure 4.4: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #4.

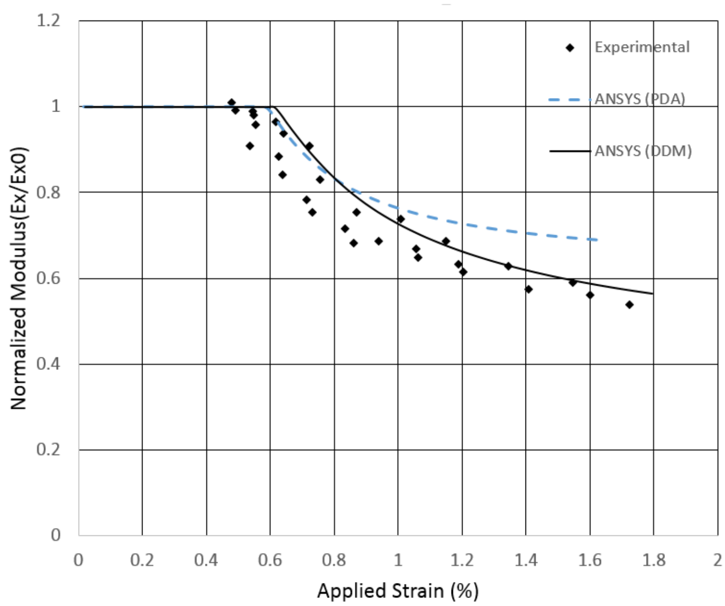


Figure 4.5: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #5.

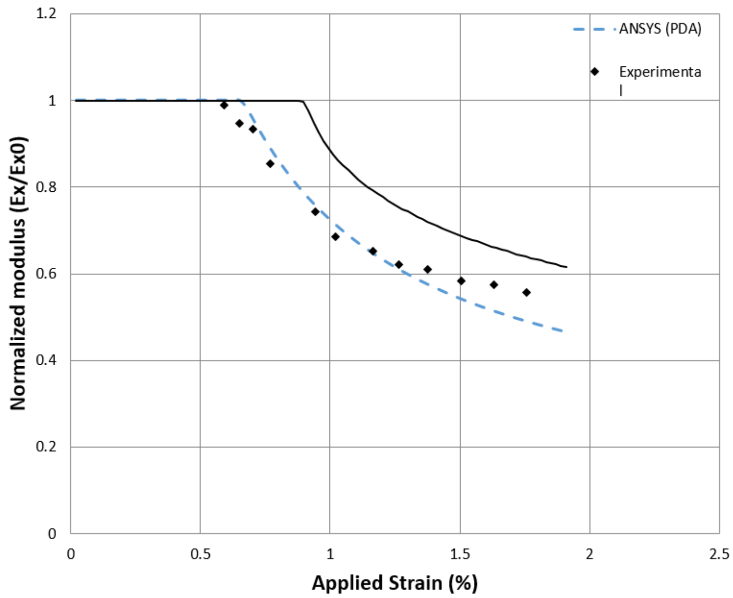


Figure 4.6: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #6.

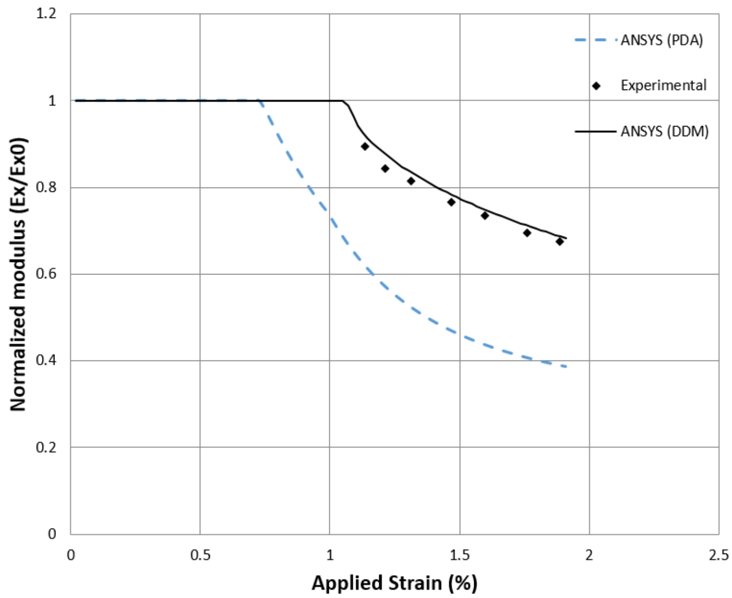


Figure 4.7: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #7.

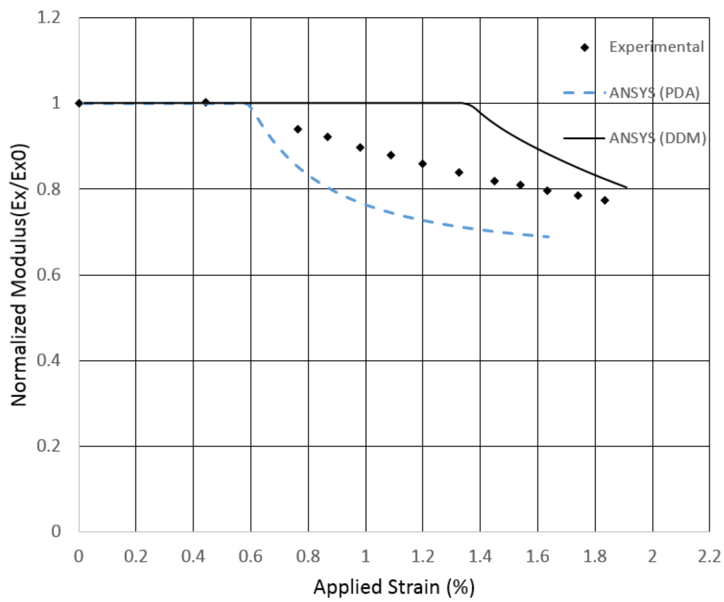


Figure 4.8: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #8.

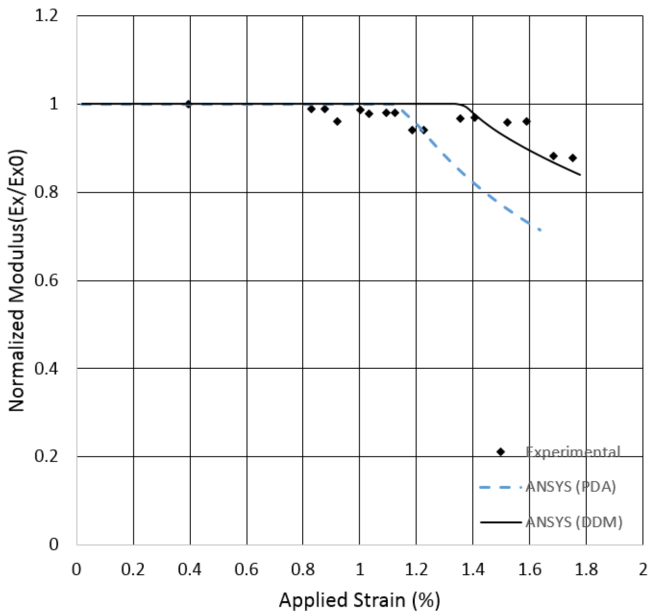


Figure 4.9: ANSYS DDM and PDA, Normalized modulus vs. applied strain for laminate #9.

In the Fig. 4.1 and Fig. 4.2 the results for both PDA and DDM of laminate #1 and #2 fi well with experimental data. For Laminates #3 to #5 (Fig.4.3–Fig.4.5) both PDA and DDM model predict the damage initiation successfully but DDM prediction of the damage evolution is better than PDA's prediction. In the other hand, PDA has a better prediction of damage initiation for laminate #6 than DDM results as shown in Fig. 4.6. However, we cannot conclude that damage initiation can always be predicted better with PDA, since Fig. 4.7 shows that DDM predicts a better damage initiation for laminate #7.

4.3 Future Work

The study of the determination of material parameters for E-glass Epoxy laminated composite in ANSYS can be used to open various pathways to future works by applying the same methodology to other composite materials. With this method, the material parameters of other composites such as Carbon fi er/Epoxy, carbon woven, and so on can be adjusted for ANSYS users to predict the damage behavior of other composites by PDA and DDM model.

The mesh sensitivity of both PDA and DDM are presented in this study. The mesh sensitivity section shows that DDM Model is insensitive to the element type and size, but PDA shows a high dependency to the element size. Adjusting material parameters for the small elements can be considered as a future work.

APPENDICES

Appendix A

ANSYS Mechanical APDL Codes

A.1 Progressive Damage Analysis (PDA) APDL Code

ANSYS APDL Code for Laminate #1

```
/TITLE, Laminate Number 1

 \textit{![0-2/90-4]s}
  \textit{! Units are in mm, MPa, and Newtons}
/UNITS,MPA
\textit{! Pre-Processor Module}

/PREP7
F2t=80
F6=48
GIc=25

\textit{! Layers Properties}

ET,1,SHELL181

\textit{!  SECTYPE,SEID,TYPE,SUBTYPE,NAME}

SECTYPE,1,SHELL,,\#1
```

```
\textit{! (Secdata, Thickness, Number of Layers, Angle of Fibers) }

SECDATA,0.288,1,0
SECDATA,1.152,1,90
SECDATA,0.288,1,0


\textit{! (Orthotropic Material Properties)}

MP,EX,1,44.7E3
MP,EY,1,12.7E3
MP,EZ,1,12.7E3
MP,GXY,1,5.8E3
MP,GYZ,1,4.5E3
MP,GXZ,1,5.8E3
MP,PRXY,1,0.297
MP,PRYZ,1,0.4111
MP,PRXZ,1,0.297


\textit{! (Material Strengths, FCLI)}

TB,FCLI,1,1,6
TBTEMP,0
TBDATA,1,1020          \textit{ ! ([F1t]  Failure STRESS, FIBER TENSION)}
TBDATA,2,-620          \textit{!  ([F1c]  Failure STRESS, FIBER COMPRESSION)}
TBDATA,3,F2t     \textit{!  ([F2t]  Failure Stress, Matrix Tension)}
TBDATA,4,-140          \textit{! ([F2c] Failure STRESS, MATRIX COMPRESSION)}
TBDATA,7,F6      \textit{![F6]      ! (Failure STRESS, XY SHEAR)}
TBDATA,8,52.7    \textit{! [F4]      ! (Failure STRESS, YZ SHEAR)}


\textit{! Initiation Failure criteria, DMGI}

TB,DMGI,1,1,4,FCRT
TBTEMP,0
\textit{! Hashin criteria that can be called by 4 is selected for all tention and compresion}
TBDATA,1,4,4,4,4

\textit{! damage evolution, DMGE, CDM (Continuum Damage Mechanic)}
```

```
TB,DMGE,1,1,8,CDM
TBTEMP,0
TBDATA,1,1E6                    \textit{! (Fracture Toughness, Fiber Tensile)}
TBDATA,2,0.001                  \textit{! (Viscosity Damping Coefficient, Fiber Tens
TBDATA,3,1E6                    \textit{! (Fracture Toughness, Fiber Compressive)}
TBDATA,4, 0.001                 \textit{! (Viscosity Damping Coefficient, Fiber Comp
TBDATA,5,GIc          \textit{! ([ Gc ] Fracture Toughness, Matrix Tensile)}
TBDATA,6, 0.005                 \textit{! (Viscosity Damping Coefficient, Matrix Ten
TBDATA,7,1E6                    \textit{! (Fracture Toughness, Matrix Compressive)}
TBDATA,8, 0.001                 \textit{! (Viscosity Damping Coefficient, Matrix Com

\textit{! Geometry and Mesh}
RECTNG,0,55,0,10          \textit{! (Creates a Rectangle with x=55 m and y=1 m)}
ESIZE,55                  \textit{! (Element Size 100 mm)}

AMESH,all                 \textit{! (Mesh the Area)}
FINISH                    \textit{! (Exit Pre-Processor Module)}

\textit{! Start Solution Module}

/SOLU
ANTYPE,STATIC
OUTRES,ALL,1                  \textit{! (Store Results for Each Substep)}

D,1,all                       \textit{! (Define b.c. on Node 1, Totally Fixed)}

D,2,ROTX,0.00
D,2,ROTZ,0.00
D,2,UY,0.00
D,2,UX,.9                 \textit{! (Define b.c. on Node 2, Uy=0.0)}

D,3,UX,.9                  \textit{! (Define Displacement on Node 3, Ux=0.1)}

D,4,UX,0.00
D,4,ROTY,0.00
D,4,ROTZ,0.0                \textit{! (Define b.c. on Node 4, Ux=0.0)}

NSUBST,100,200,100        \textit{! (100 = Number of Substeps in this Load Step)}
```

```
SOLVE                                   \textit{! (Solve Load Step)}
FINISH                                  \textit{! (Exit Solution Module)}

\textit{! Experimental Data}

*DIM,AA,ARRAY,22,2,1, , ,     !*
*SET,AA(1,1,1) , 0.35909
*SET,AA(1,2,1) , 1
*SET,AA(2,1,1) , 0.364866
*SET,AA(2,2,1) , 1
*SET,AA(3,1,1) , 0.399523
*SET,AA(3,2,1) , 1
*SET,AA(4,1,1) , 0.509413
*SET,AA(4,2,1) , 0.984729
*SET,AA(5,1,1) , 0.578784
*SET,AA(5,2,1) , 0.978818
*SET,AA(6,1,1) , 0.590251
*SET,AA(6,2,1) , 0.987685
*SET,AA(7,1,1) , 0.711778
*SET,AA(7,2,1) , 0.964039
*SET,AA(8,1,1) , 0.764191
*SET,AA(8,2,1) , 0.919704
*SET,AA(9,1,1) , 0.769995
*SET,AA(9,2,1) , 0.916749
*SET,AA(10,1,1) , 0.787295
*SET,AA(10,2,1) , 0.919704
*SET,AA(11,1,1) , 0.967324
*SET,AA(11,2,1) , 0.819212
*SET,AA(12,1,1) , 0.97882
*SET,AA(12,2,1) , 0.825123
*SET,AA(13,1,1) , 0.996091
*SET,AA(13,2,1) , 0.831034
*SET,AA(14,1,1) , 1.01353
*SET,AA(14,2,1) , 0.819212
*SET,AA(15,1,1) , 1.18127
*SET,AA(15,2,1) , 0.795567
*SET,AA(16,1,1) , 1.19288
*SET,AA(16,2,1) , 0.789655
*SET,AA(17,1,1) , 1.21618
```

```
*SET,AA(17,2,1) , 0.768966
*SET,AA(18,1,1) , 1.23343
*SET,AA(18,2,1) , 0.777833
*SET,AA(19,1,1) , 1.378
*SET,AA(19,2,1) , 0.760099
*SET,AA(20,1,1) , 1.40125
*SET,AA(20,2,1) , 0.74532
*SET,AA(21,1,1) , 1.41277
*SET,AA(21,2,1) , 0.748276
*SET,AA(22,1,1) , 1.47645
*SET,AA(22,2,1) , 0.733498
*SET,L,0


/POST26                          \textit{! (Start Time-Historic post-process)}
NSOL,2,3,U,X, UXnode3            \textit{! (Load displacements node 3)}
RFORCE,3,4,F,X,FXnode4           \textit{! (Load reaction force node 4)}
RFORCE,4,1,F,X,FXnode1           \textit{! (Load reaction force node 1)}
LINES,1000
PRVAR,2,3,4                      \textit{! (List Displacements and Reactions)}
VGET,UX3,2,0
VGET,FX4,3,0
VGET,FX1,4,0
L=0
 *DO,I,1,22,1
    *DO,J,1,100,1
   SXP=AA(I,1,1)
   EEXP=AA(I,2,1)
   SCS=1.8181*UX3(J,1)

      FN4=FX4(J,1)
      FN1=FX1(J,1)
  FN2=FX1(2,1)
  SCS2=1.8181*UX3(2,1)
  MRG=ABS((SCS-SXP))             *IF,MRG,LE,0.008145,THEN
        NUM=((FN1+FN4)*(SCS2))
  DEN=2*(FN2)*(SCS)
  EPE=NUM/DEN
  ER1=EPE-EEXP
```

```
   ER2=ER1*ER1
   L=L+ER2
  *ENDIF
 *ENDDO
  *ENDDO
  D=(1/22)*SQRT(L)
FINISH                                   \textit{ ! (Exit Post-Process Module)}
```

A.2 Discrete Damage Mechanics (DDM) Model APDL Code

ANSYS APDL Code for Laminate #1:

```
/TITLE, FEAcomp Ex. 9.01, USERMATLib.DLL
/PREP7                    ! Start pre-processor module

!===  PARAMETERS  ================================================

L0 = 0.02                 ! initial the crack density
ShellDimensionX = 55.    ! model dimensions
ShellDimensionY = 10.    ! mm
tk =.144                  ! ply thickness
NL = 2                    ! number layers half laminate
Nprops = 3+9*NL           ! # material properties

!=== NEXT VALUES GO IN TBDATA ====================================
GIc   =0.43
GIIc  =1.027
deltaT  =  0.0
E1 = 44700                ! MPa
E2 = 12700
G12= 5800
nu12 =.297
nu23 =.410
CTE1 =3.7
CTE2 =30.
!Angle       with TBDATA for each layer
!Thickness   with TBDATA for each layer
```

```
!=== USERMAT DECLARATION SECTION ==================================
TB,USER,1,1,Nprops,         ! DECLARES USAGE OF USERMAT 1, MAT 1,
TBTEMP,0                    ! ref. temperature
TBDATA,,GIc,GIIc,detaT,E1,E2,G12        ! 6 values per TBDATA line
TBDATA,,nu12,nu23,CTE1,CTE2,0,2*tk
TBDATA,,E1,E2,G12,nu12,nu23,CTE1
TBDATA,,CTE2,90,4*tk,
TB,STAT,1,,3*NL         ! NUMBER OF STATE VARIABLES
! INITIALIZE THE STATE VARIABLES
TBDATA,,L0,L0,L0,L0,L0,L0


!=== MESH =========================================================
ET,1,182,,,3                ! PLANE182, plane elements with plane stress
R,1,2*6*tk                 ! Real const. #1, thickness of whole laminate
N,1                         ! Define node 1, coordinates=0,0,0
N,2,ShellDimensionX,0       ! Define node 2,
N,3,ShellDimensionX,ShellDimensionY
N,4,0,ShellDimensionY
E,1,2,3,4                   ! Generate element 1 by node 1 to 4
FINISH                      ! Exit pre-processor module

!
!
!SOLU
!
!
/SOLU                                  ! Start Solution module
ANTYPE,STATIC
OUTRES,ALL,1                           ! Store results for each substep

D,1,all                                ! Define b.c. on node 1, totally fixed


D,2,UY,0.00
D,2,UX,1.05                      ! Define b.c. on node 2, Uy=0.0

D,3,UX,1.05                       ! Define displacement on node 3, Ux=0.1
```

```
D,4,UX,0.00

                    ! Define b.c. on node 4, Ux=0.0

NSUBST,100,200,100          ! 100 = Number of substeps in this load step
SOLVE                                  ! Solve load step
FINISH                                   ! Exit solution module


/POST26                              ! Start time-historic post-process
NSOL,2,3,U,X,  UXnode3               ! Load displacements node 3
RFORCE,3,4,F,X,FXnode4               ! Load reaction force node 4
RFORCE,4,1,F,X,FXnode1               ! Load reaction force node 1
LINES,1000                                 !
PRVAR,2,3,4                               ! list displacements and reactions



!
!
!
!RESULTS
!
!
*DIM,AA,ARRAY,22,2,1, , ,
!*

*SET,AA(1,1,1) , 0.35909
*SET,AA(1,2,1) , 1
*SET,AA(2,1,1) , 0.364866
*SET,AA(2,2,1) , 1
*SET,AA(3,1,1) , 0.399523
*SET,AA(3,2,1) , 1
*SET,AA(4,1,1) , 0.509413
*SET,AA(4,2,1) , 0.984729
*SET,AA(5,1,1) , 0.578784
*SET,AA(5,2,1) , 0.978818
*SET,AA(6,1,1) , 0.590251
*SET,AA(6,2,1) , 0.987685
*SET,AA(7,1,1) , 0.711778
```

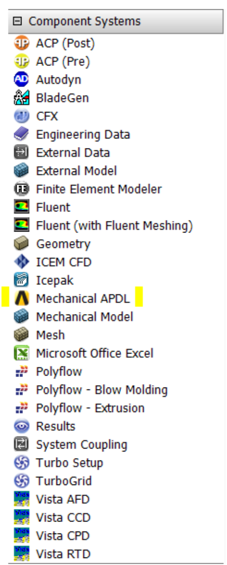
```
*SET,AA(7,2,1)  ,  0.964039
*SET,AA(8,1,1)  ,  0.764191
*SET,AA(8,2,1)  ,  0.919704
*SET,AA(9,1,1)  ,  0.769995
*SET,AA(9,2,1)  ,  0.916749
*SET,AA(10,1,1)  ,  0.787295
*SET,AA(10,2,1)  ,  0.919704
*SET,AA(11,1,1)  ,  0.967324
*SET,AA(11,2,1)  ,  0.819212
*SET,AA(12,1,1)  ,  0.97882
*SET,AA(12,2,1)  ,  0.825123
*SET,AA(13,1,1)  ,  0.996091
*SET,AA(13,2,1)  ,  0.831034
*SET,AA(14,1,1)  ,  1.01353
*SET,AA(14,2,1)  ,  0.819212
*SET,AA(15,1,1)  ,  1.18127
*SET,AA(15,2,1)  ,  0.795567
*SET,AA(16,1,1)  ,  1.19288
*SET,AA(16,2,1)  ,  0.789655
*SET,AA(17,1,1)  ,  1.21618
*SET,AA(17,2,1)  ,  0.768966
*SET,AA(18,1,1)  ,  1.23343
*SET,AA(18,2,1)  ,  0.777833
*SET,AA(19,1,1)  ,  1.378
*SET,AA(19,2,1)  ,  0.760099
*SET,AA(20,1,1)  ,  1.40125
*SET,AA(20,2,1)  ,  0.74532
*SET,AA(21,1,1)  ,  1.41277
*SET,AA(21,2,1)  ,  0.748276
*SET,AA(22,1,1)  ,  1.47645
*SET,AA(22,2,1)  ,  0.733498
*SET,L,0


VGET,UX3,2,0
VGET,FX4,3,0
VGET,FX1,4,0
L=0

 *DO,I,1,22,1
```

```
*DO,J,1,100,1

SXP=AA(I,1,1)
EEXP=AA(I,2,1)

SCS=1.8181*UX3(J,1)


FN4=FX4(J,1)
FN1=FX1(J,1)
FN2=FX1(2,1)
SCS2=1.8181*UX3(2,1)

MRG=ABS((SCS-SXP))
*IF,MRG,LE,0.009545,THEN
NUM=((FN1+FN4)*(SCS2))
DEN=2*(FN2)*(SCS)
EPE=NUM/DEN
ER1=EPE-EEXP
ER2=ER1*ER1
L=L+ER2
*ENDIF
*ENDDO
*ENDDO
D=(1/22)*SQRT(L)

FINISH                                         ! Exit post-process module
```

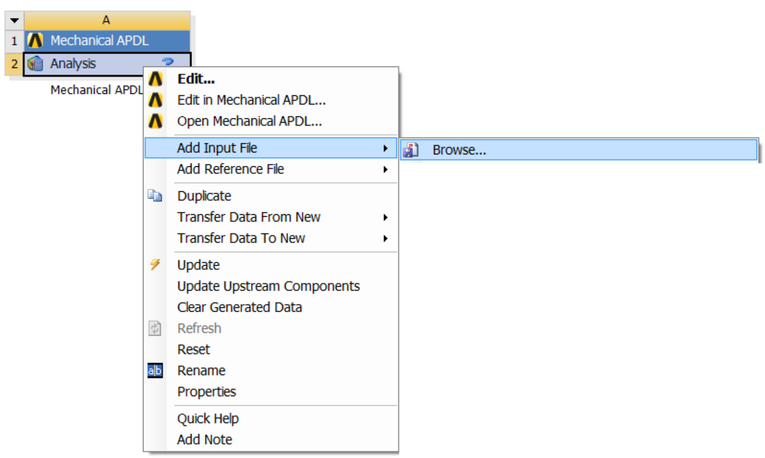
Appendix B

ANSYS Workbench

B.1 Optimization

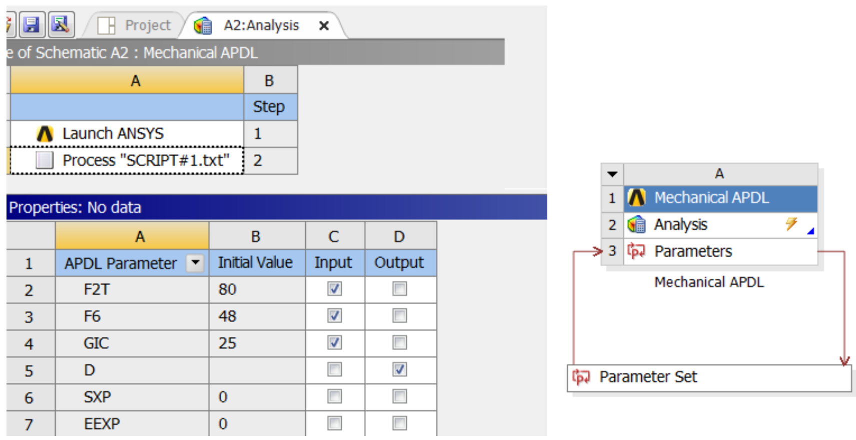


(a) APDL



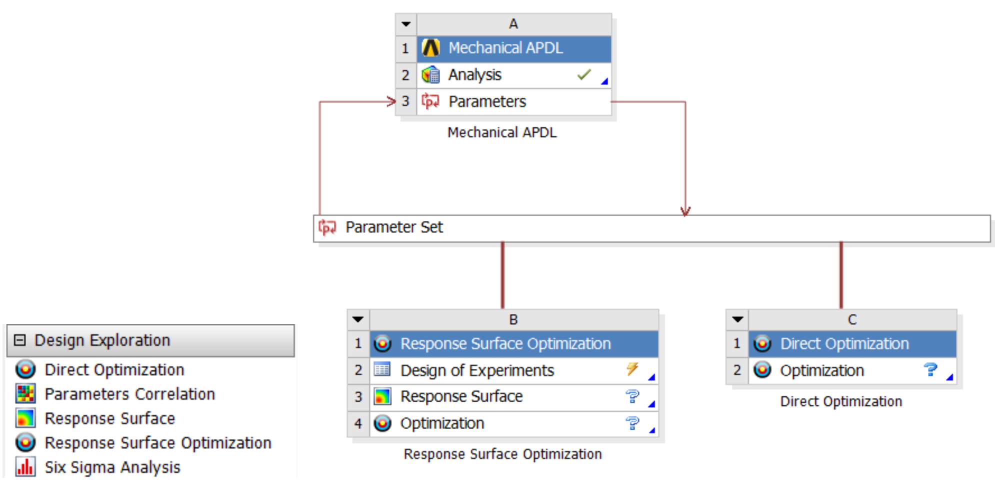
(b) Browse

Figure B.1: Importing the apdl code to Workbench



(a) Inputs and outputs (b) Active parameters

Figure B.2: Inputs and output(error) are selected for optimization purpose



(a) Design Exploration (b) Design Exploration is connected to the parameters

Figure B.3: Design Exploration tools

Afterward, by updating the optimization ANSYS Workbench minimizes the error and shows the desired input parameters.

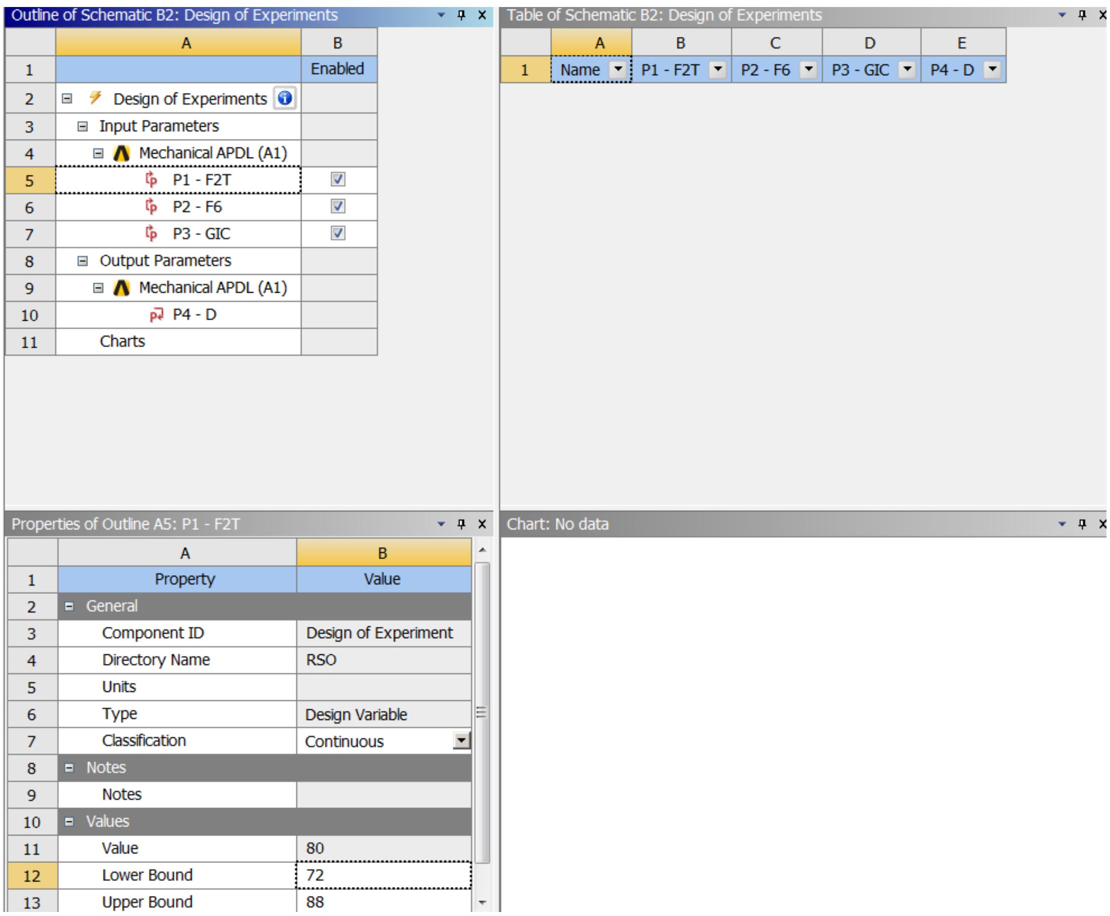


Figure B.4: In Design of Experiment part the user can set the limits for inputs

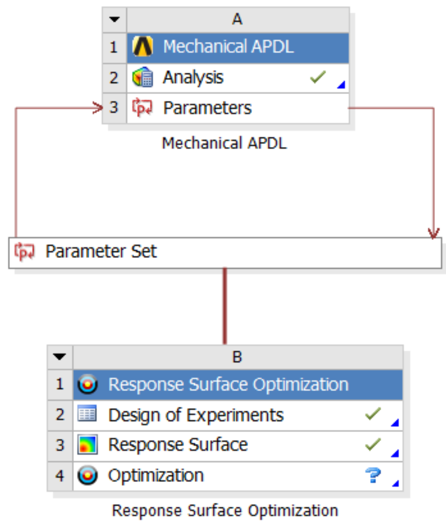


Figure B.5: Updating all the previous steps and selecting the optimization

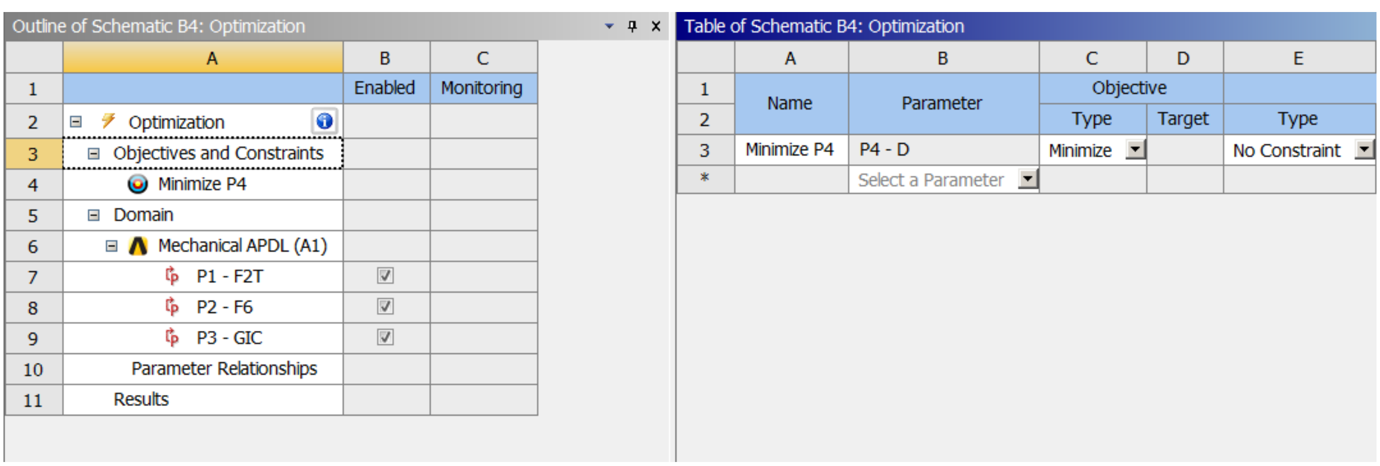


Figure B.6: Error (D) is selected to be minimized

References

[1] *Ansys mechanical apdl programmer's manual, release 16.1*. ANSYS Inc., 2 edition, 2016.

[2] G. Alfano and M.A. Crisfield. Finite element interface models for the delamination analysis of laminated composites: Mechanical and computational issues. *International Journal for Numerical Methods in Engineering*, 50(7):1701–1736, 2001. cited By 468.

[3] E. Barbero and D. Cortes. Constitutive model for elastic damage in fi er-reinforced pmc laminae. *International Journal of Damage Mechanics*, 2001.

[4] E. Barbero and D. Cortes. A mechanistic model for transverse damage initiation, evolution, and stiffness reduction in laminated composites. *Composites Part B*, 2010.

[5] E. J. Barbero. *Finite Element Analysis of Composite Materials using ANSYS*. 2 edition.

[6] E. J. Barbero. *Introduction to Composite Materials Design*. CRC Press, 2 edition, 2011.

[7] P.P. Camanho, C.G. Davila, S.T. Pinho, L. Iannucci, and P. Robinson. Prediction of in situ strengths and matrix cracking in composites under transverse tension and in-plane shear. *Composites Part A: Applied Science and Manufacturing*, 37(2):165–176, 2006. cited By 130.

[8] I.M. Daniel. Failure of composite materials. *Strain*, 2007.

[9] C.G. Davila, P.P. Camanho, and C.A. Rose. Failure criteria for frp laminates. *Journal of Composite Materials*, 39(4):323–345, 2005.

[10] R. Roman E. J. Barbero, F. A. Cosso and T. L. Weadon. Determination of material parameters for abaqus progressive damage analysis of e-glass epoxy lami- nates. *Composites Part B:Engineering*, 2012.

[11] Z. Hashin. Failure criteria for unidirectional fi er composites. *Journal of Applied Mechanics*, 1980.

[12] Z. Hashin and A. Rotem. Fatigue failure criterion for fi er reinforced materials. *Journal of Composite Materials*, 7:448–464, 1973.

[13] C.T. Herakovich. *Mechanics of Fibrous Composite*. John Wiley & Sons, Inc., New York, 1998.

[14] Kaddour A.S. Hinton, M.J. and Soden. A comparison of the predictive capabilities of current failure theories for composite laminates, judged against experimental evidence. *Composites Science and Technology*, 2002.

[15] Soden P.D. Hinton, M.J. and Kaddour. Failure criteria in fi er-reinforced- polymer composites. *Elsevier, Oxford*, 2004.

[16] Locatto S. Student G. Icardi, U. and Longo. Assessment of recent theories for predicting failure of composite laminates. *Applied Mechanics Reviews*, 2007.

[17] Carre‘ re N. Laurin, F. and J.F. Maire. A multiscale progressive failure approach for composite laminates based on thermodynamical viscoelastic and damage mode. *Composites Part A: Applied Science and Manufacturing*, 2007.

[18] Iannucci L. Pinho, S.T. and P. Robinson. Physically based failure models and criteria for laminated fi forced composites with emphasis on fi kinking: Part i: Development. *Composites Part A: Applied Science and Manufacturing*, 2006.

[19] Iannucci L. Pinho, S.T. and P. Robinson. Physically based failure models and criteria for laminated fi composites with emphasis on fi kinking, part ii: Fe implementation. *Composites Part A: Applied Science and Manufacturing*, 2006.

[20] A. Puck and H. Schu rmann. Failure analysis of frp laminates by means of physically based phenomenological models. *Composites Science and Technology*, 1998.

[21] H. Wang A.K. Bledzki A. Korjakin R. Rikards, F.G. Buchholz and H.A. Richard. Investigation of mixed mode i/ii interlaminar fracture toughness of laminated composites by using a cts type specimen. *Engineering Fracture Mechanics*, 1998.

[22] C.T. Sun. Strength analysis of unidirectional composites and laminates. *Comprehensive Composite Materials, Elsevier Science, Ltd., Oxford*, 2000.

[23] S.W. Tsai and E.M. Wu. A general theory of strength for anisotropic materials. *Journal of Composite Materials*, 1971.

[24] J. Varna, R. Joffe, N.V. Akshantala, and R. Talreja. Damage in composite laminates with off-axis plies. *Composites Science and Technology*, 59(14):2139–2147, 1999.

[25] J. Varna, R. Joffe, and R. Talreja. A synergistic damage-mechanics analysis of transverse cracking [/904]s laminates. *Composites Science and Technology*, 61(5):657–665, 2001.