



**TIME-DEPENDENT VALIDATION OF
FINITE ELEMENT STRAIN DISTRIBUTION
OF A PLASTICALLY-DEFORMED PLATE
VIA DIGITAL IMAGE CORRELATION**

THESIS

Kevin R. Knapp, First Lieutenant, USAF
AFIT-ENY-MS-17-M-269

**DEPARTMENT OF THE AIR FORCE
AIR UNIVERSITY**

AIR FORCE INSTITUTE OF TECHNOLOGY

Wright-Patterson Air Force Base, Ohio

DISTRIBUTION STATEMENT A.
APPROVED FOR PUBLIC RELEASE; DISTRIBUTION UNLIMITED.

The views expressed in this document are those of the author and do not reflect the official policy or position of the United States Air Force, the United States Department of Defense, or the United States Government. This material is declared a work of the U.S. Government and is not subject to copyright protection in the United States.

AFIT-ENY-MS-17-M-269

TIME-DEPENDENT VALIDATION OF FINITE ELEMENT STRAIN
DISTRIBUTION OF A PLASTICALLY-DEFORMED PLATE VIA DIGITAL
IMAGE CORRELATION

THESIS

Presented to the Faculty
Department of Aeronautical Engineering
Graduate School of Engineering and Management
Air Force Institute of Technology
Air University
Air Education and Training Command
in Partial Fulfillment of the Requirements for the
Degree of Master of Science in Aeronautical Engineering

Kevin R. Knapp, B.S. Ch.E.

First Lieutenant, USAF

March 3, 2017

DISTRIBUTION STATEMENT A.
APPROVED FOR PUBLIC RELEASE; DISTRIBUTION UNLIMITED.

TIME-DEPENDENT VALIDATION OF FINITE ELEMENT STRAIN
DISTRIBUTION OF A PLASTICALLY-DEFORMED PLATE VIA DIGITAL
IMAGE CORRELATION

Kevin R. Knapp, B.S. Ch.E.
First Lieutenant, USAF

Committee Membership:

Anthony Palazotto, Ph.D.
Chair

Casey Holycross, Ph.D.
Member

Onome Scott-Emuakpor, Ph.D.
Member

Abstract

Generation of residual stress is an essential step in the generation of Goodman data via Air Force Research Laboratory's vibration-based fatigue test. Conventional Goodman data is constructed through uniaxial fatigue testing at a rate of 40 Hz, while the vibration-based testing excites stresses at 1,600 Hz in a stress state similar to those of gas turbine engine airfoils in service. A pre-strain procedure is conducted to form residual tensile stress, which serves as a steady stress when the specimen is subjected to fully-reversed vibration-based fatigue loads. This steady tensile stress is desired at the fatigue zone of the test article, but is the result of an adjacent compressive region in equilibrium, and as such, a finite element model (FEM) is necessary to determine the stress distribution throughout the entire sample. The goal of this work is to improve the FEM analysis associated with the pre-strain method for better accuracy of steady stress generation for Goodman data fatigue assessment. Improvements were made to the FEM by more effectively incorporating empirical tensile stress-strain behavior, in addition to more accurately representing the pressures and forces acting on the specimen during monotonic loading. Validations of improvements to the pre-strain steady stress generation procedure will be demonstrated on Aluminum 6061-T6 by comparing strain field results from digital image correlation to FEM analysis. The converged quasi-static FEM solution had a standard deviation in ε_{yy} of 2,557 microstrain and predicted a residual σ_{yy} of 10.72 ksi, while the optimized time-dependent solution had a standard deviation in ε_{yy} of 308.9 microstrain (less than the experimental variation of 376.1 microstrain) and predicted a residual σ_{yy} of 4.93 ksi. The increased accuracy of these models altered residual stresses on a Goodman line by as much as 26% compared with past results.

Acknowledgments

I would like to express my gratitude to my research advisor, Dr. Anthony Palazotto for his guidance and support throughout the course of this thesis project. His insight and experience was invaluable and will not be forgotten. I would also like to thank Dr. Onome Scott-Emuakpor for his unwavering support of not only this research effort, but my development as an engineer and experimentalist. The last three years have been pivotal in developing my capabilities and, more importantly, observing the value of his work passed on through the many students he has mentored. Additionally, I would like to thank Dr. Casey Holycross for his assistance with the experimental setup, FEM, coding, and countless discussions on constitutive behavior. His thorough review of my work has been both insightful and entertaining.

I would also like to thank Dr. Tommy George and everyone in the TEFF for their continued support my professional advancement as an officer, engineer, and future educator. My experience (and often lack thereof) at AFRL will be treasured for years to come, and I could not have had a better group of people to help jump start my Air Force career. Finally, I would like to extend my sincere appreciation to my family for all of their support throughout my time at AFIT.

Kevin R. Knapp

Table of Contents

	Page
Abstract	iv
Acknowledgments	v
List of Figures	ix
List of Tables	xiii
List of Acronyms	xiv
List of Symbols	xv
I. Introduction	1
1.1 Motivation for Research	1
1.2 Research Scope	3
1.3 Problem Statement	3
1.4 Research Objectives	4
1.5 Assumptions	4
1.6 Chapter Outline	5
II. Theory	7
2.1 High Cycle Fatigue	8
2.1.1 Goodman Diagram	9
2.1.2 Engine Structural Integrity Program	10
2.1.3 ASTM Test Standards	10
2.2 Vibration-based Fatigue Test	11
2.3 Pre-strain Residual Stress Generation Method	13
2.3.1 Elastic, Perfectly Plastic Modeling	15
2.3.2 Multilinear Hardening Modeling	16
2.3.3 Past Personal Research on Pre-strain Method	17
2.4 Three Dimensional Material Constitutive Equations	18
2.4.1 Von Mises Isotropic Hardening	18
2.4.2 Prandtl-Reuss Relations	19
2.4.3 Quasi-static vs. Rate-dependent Plasticity	21
2.5 Finite Element Modeling	25
2.5.1 Mesh Generation	26
2.5.2 Displacement Boundary Conditions	27
2.5.3 Load Boundary Conditions	27
2.5.4 Time-dependent vs. Quasi-static FEM Solutions	28
2.6 Digital Image Correlation	29
2.6.1 Correlation Function	30

	Page
2.6.2 Raw Strain Calculation.....	31
2.6.3 Image Data Filtering.....	34
2.6.4 Application of DIC	35
2.7 Root Mean Squared Error and Deviation	36
2.8 Chapter Summary	37
III. Research Methodology	38
3.1 Sample Preparation.....	39
3.1.1 Speckle Pattern Application and Coordinate Transformation.....	39
3.2 Pre-strain Method	42
3.2.1 Specimen Alignment	43
3.2.2 Multi-Purpose Testware Procedure.....	45
3.2.3 Compressive Forces	47
3.3 Axial Dogbone Testing	48
3.4 Digital Image Correlation.....	48
3.5 Vibration-based Fatigue Test Method	50
3.5.1 Failure Determination and Approximation	52
3.6 Chapter Summary	53
IV. Results	54
4.1 Specimen Features.....	54
4.1.1 Dogbones	54
4.1.2 Plates.....	57
4.1.3 FEM Geometric Modeling	58
4.1.4 FEM Load Boundary Conditions	61
4.1.5 Plate Data Processing.....	65
4.1.6 DIC Data Processing	66
4.2 Comparison of Experimental and Analytical Results	66
4.3 Finite Element Model Results	68
4.3.1 Quasi-static Analyses	68
4.3.2 Time-dependent Analyses	75
4.4 Digital Image Correlation Results.....	78
4.4.1 Optimized Loading due to Strain Field Matching	91
4.5 Relationship of Strain Error and Stress Predictions	97
4.5.1 Goodman Line Reassessment.....	100
4.6 Chapter Summary	101
V. Conclusions and Recommendations	102
5.1 Review of Research Objectives	102
5.2 Recommendations for Future Work	104
5.2.1 Experimental Stress Measurement	104

	Page
5.2.2 Representative Gas Turbine Engine Materials	104
5.2.3 Lens Dust Reduction.....	105
5.2.4 Further Improved Boundary Conditions	105
5.3 Discussion of Results.....	106
5.3.1 DIC Experimental Noise.....	106
5.3.2 Time-dependent vs. Quasi-static FEM Models.....	106
5.3.3 Optimized Strain Matching of FEM Models	107
5.3.4 Improved Residual Stress Impacts on Goodman Data.....	108
Appendix A. ANSYS Script	109
Appendix B. MATLAB Scripts.....	119
B.1 Data Extraction.....	119
B.1.1 Plate Data Extractor	119
B.1.2 Grip Data Extractor	124
B.1.3 Dogbone Data Extractor and Script Generation	126
B.2 Grip Force Script Generation	130
B.3 DIC vs FEM Comparison Figures	137
Appendix C. Plate Mesh Script	142
Appendix D. Correlation Function Example.....	154
Bibliography	158

List of Figures

Figure	Page
1.1 Goodman Diagram	2
1.2 Y-strain Hole Comparison, 17,000 lbf	5
2.1 Cyclic Loading Profiles	8
2.2 Fatigue Life Approximation Methods	9
2.3 Specimens with Tangentially Blending Fillets	11
2.4 Vibration Based Fatigue Test Specimen	13
2.5 Residual Stress-Strain	14
2.6 Pre-strain Method	15
2.7 Load Step Curve	16
2.8 Al 6061-T6 Vibration-based Fatigue 10^6 Failure Cycles Goodman Line	17
2.9 Isotropic Yielding	19
2.10 Aluminum 6061-T6 Experimental Data with Newton-Raphson Iterations	20
2.11 Change from Yield Surface to Dynamic Load Surface due to Strain and Strain Rate Hardening	22
2.12 Viscoplastic Phenomena	24
2.13 Basic Element Types	25
2.14 3D Finite Element Model	25
2.15 Mesh Generation Procedure	26
2.16 Boundary Conditions	27
2.17 Point Load Boundary Conditions	28
2.18 Time-dependent vs. Quasi-static	29
2.19 Digital Image Correlation Capture	30

Figure		Page
2.20	Digital Image Correlation Overview	31
2.21	DIC Displacement Vectors	32
2.22	Strain Calculation Overview	32
2.23	DIC Strain Vectors	33
2.24	Gaussian Distribution	34
2.25	Strain Data Filtering	35
3.1	Chronological Test Procedure	39
3.2	Specimen Geometry (in)	40
3.3	Speckle Pattern	40
3.4	Coordinate System Alignment	42
3.5	Pre-strain Experimental Setup.....	43
3.6	Plate/Grip Placement Tooling	44
3.7	Initial Plate/Grip Alignment	44
3.8	Final Plate/Grip Spacing	45
3.9	MTS Control Inputs	46
3.10	MTS Test Procedure	46
3.11	Grip System	47
3.12	Digital Image Correlation Examples	49
3.13	DIC Focusing Procedure	49
3.14	DIC Error Estimate	50
3.15	DIC Spatial Calibration	50
3.16	Vibration Based Fatigue Test.....	51
3.17	Fatigue Test Setup	51
4.1	Dogbone Stress vs. Strain.....	56

Figure		Page
4.2	Dogbone Data Processing	57
4.3	Convergence Study Results	60
4.4	Gripped Plate Model	62
4.5	Single Grip Model	63
4.6	Tensile Force Application	64
4.7	σ_{yy} Equilibrium Along Plate Centerline	67
4.8	Residual Stress Flowchart	69
4.9	Quasi-static FEM σ - ε Results, Loaded State	70
4.10	Effective Stress-Strain	71
4.11	FEM σ - ε Results (Reprinted from Figures 4.9 and 4.12 for Ease of Reading)	72
4.12	Quasi-static FEM σ - ε Results, Unloaded State	74
4.13	Time-dependent FEM σ - ε Results, Loaded State	76
4.14	Time-dependent FEM σ - ε Results, Unloaded State	77
4.15	Full Plate FEM ε_{yy}	78
4.16	Error Between DIC Measurements During/After 15,000 lb _f Load	81
4.17	Quasi-static Strain Results, Loaded State	82
4.18	DIC vs. FEM Strain Results, Loaded State	83
4.19	Quasi-static DIC vs. FEM Strain Error, Loaded State	84
4.20	Time-dependent DIC vs. FEM Strain Error, Loaded State	85
4.21	ε_{yy} RMS Error vs. Time	86
4.22	DIC vs. FEM Strain Results, Unloaded State	88
4.23	Quasi-static DIC vs. FEM Strain Error, Unloaded State	89

Figure		Page
4.24	Time-dependent DIC vs. FEM Strain Error, Unloaded State.....	90
4.25	Optimized DIC Field vs. FEM Strain Error, Loaded State.....	92
4.26	Optimized Error Along Plate Centerline, Loaded State.....	93
4.27	Optimized ε_{yy} RMS Error vs. Time	94
4.28	Optimized DIC Field vs. FEM Strain Error, Unloaded State.....	95
4.29	Error Along Plate Centerline, Unoaded State	96
4.30	Optimized Residual σ Results, Unloaded State.....	99
4.31	Updated Vibration-based Fatigue 10^6 failure cycles Goodman line	100
D.1	Correlation Function	154
D.2	Deformation of a Perfect Contrast Speckle	155
D.3	Poor Correlation.....	156
D.4	Perfect Correlation.....	156

List of Tables

Table		Page
4.1	Dogbone Measurements	55
4.2	Plate Measurements.....	57
4.3	Lengthwise Convergence of ε_{yy}	59
4.4	Widthwise Convergence of ε_{yy}	59
4.5	Convergence of ε_{yy} both Length and Width	60
4.6	RMS Error and Deviation between DIC Measurements.....	80
4.7	Quasi-static ε_{yy} RMSE & RMSD vs. Maximum Load	92
4.8	Error Assessment of FEM Analysis Types	97
4.9	Deviation Assessment of FEM Analysis Types	97
4.10	Residual Stress Results at HCF Failure Location	99
4.11	Goodman Residual Stress Results at HCF Failure Location.....	101

List of Acronyms

AFRL	Air Force Research Laboratory
Al	aluminum
ASTM	American Society for Testing and Materials
DIC	digital image correlation
DoD	Department of Defense
ENSIP	Engine Structural Integrity Program
FEM	finite element model
GTE	gas turbine engine
HCF	high cycle fatigue
LVDT	linear variable displacement transducer
MPT	Multi-purpose Testware
MTS	Mechanical Test Systems
Ni	nickel
RMSD	root mean squared deviation
RMSE	root mean squared error
SS	stainless steel
Ti	titanium
USAF	United States Air Force

List of Symbols

Basic Stress and Strain Symbols

σ	Stress
σ_m	Mean Stress
σ_T	True Stress = $\sigma (1 + \varepsilon)$
σ_e	Effective Stress (von Mises)
σ_0	Proportional Limit (Yield Stress)
s_{ij}	Deviatoric Stresses
R	Stress Ratio
ε	Strain
ε_T	True Strain = $\ln (1 + \varepsilon)$

Grip Symbols

F_c	Clamping Force of Wedge on Specimen
$f_{c,w}$	Frictional Force, Grip Chamber to Wedge
$f_{w,s}$	Frictional Force, Wedge to Specimen
$\mu_{c,w}$	Friction Coefficient, Grip Chamber to Wedge
$\mu_{w,s}$	Friction coefficient, Wedge to Specimen
$N_{c,w}$	Normal Force, Grip Chamber to Wedge
$N_{w,s}$	Normal Force, Wedge to Specimen
P	Grip Supply Pressure
L	Axial Load Applied to Specimen
t_s	Thickness of Flat Specimen
W_s	Width of Flat Specimen

Plasticity Symbols

$\dot{\varepsilon}_e^p$	Incremental Effective Plastic Strain
$\dot{\lambda}$	Plastic Multiplier
$F(\sigma, \varepsilon^p)$	Load Function
$f(\sigma, \varepsilon^p)$	Dynamic Load Surface Fxn
$\kappa(W_p)$	Work-Hardening Parameter
W_p	Plastic Work
m	Perzyna $\dot{\varepsilon}$ Hardening Parameter
γ	Perzyna Viscosity Parameter

Other Symbols

AR	Aspect Ratio
F	Applied Force
W	Width
T	Thickness
D	Diameter
$\hat{y}_i$	Predicted Data Point
y_i	Measured Data Point
N_i	Shape Function
$[\mathbf{B}]$	Spatial Derivative Matrix
σ_a	Stress Amplitude
σ_{pr}	Stress, Previous Step
σ_{fail}	Failure Stress
N_{fail}	Failure Cycles, Current Step
N_{step}	Step Cycles
C_π	Cosine of π Radians
S_π	Sine of π Radians

TIME-DEPENDENT VALIDATION OF FINITE ELEMENT STRAIN DISTRIBUTION OF A PLASTICALLY-DEFORMED PLATE VIA DIGITAL IMAGE CORRELATION

I. Introduction

1.1 Motivation for Research

Failure of gas turbine engine (GTE) components in high cycle fatigue (HCF) is an unavoidable reality for United States Air Force (USAF) aircraft and engine designers, and as such each component must be assessed and deemed acceptable for desired flight envelopes [1]. The United States Department of Defense (DoD) offers instructions for proper procedures to assess life approximations for military engines via a document called Engine Structural Integrity Program (ENSIP). One recommendation from ENSIP states that all component materials should be empirically characterized for HCF failure up to 10^9 cycles, assuming no endurance limit is present [2]. However, experimentally accumulating 10^9 fatigue cycles with conventional methods can be challenging due to time constraints and equipment limitations.

Typically, this type of material data is gathered using servohydraulic load frames (to be described further in Chapter III), which come with two major short falls: 290 days of non-stop testing is required to accumulate 10^9 cycles [3], and axial data that does not adequately represent the multi-axial HCF failure mechanisms of vibrating turbine engine components [1, 3, 4]. Realistic failure of engine airfoils occurs under higher order vibrational fatigue and often in combined bending-torsion modes, which is what led to the development of a vibration-based bending fatigue test method

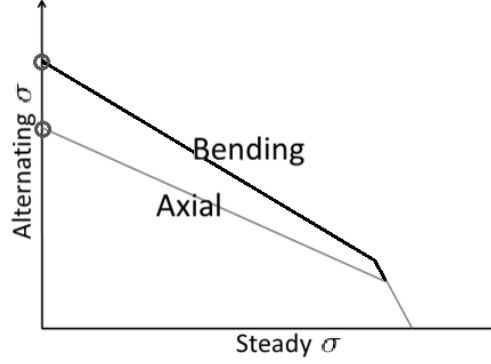


Figure 1.1. Goodman Diagram

[4, 5]. The excitation of this vibration test is supplied at the resonant frequency of the test specimen (typically 1200–1600 Hz), which is not only more representative of engine airfoil failure mechanisms, but also accumulates cycles 40 times faster than conventional testing.

The assessment of component life approximations, extracted from experimentally driven failure under varying steady and alternating loads is typically represented via a constant life diagram such as a Haigh or Goodman Diagram (as seen in Figure 1.1). The utilization of cantilever vibration testing to gather material data typically comes with a limitation of only applying fully reversed loading (shown by blue circles), while material stresses in operation are subject to steady (mean) stresses. In a novel approach to alleviate this drawback, Air Force Research Laboratory (AFRL) researchers have used localized yielding (pre-strain) methods to generate a residual stress in the fatigue zone of the Vibration-based specimen prior to fatigue testing [6]. The plastically deformed region will experience compressive residual stresses after unloading, which in turn generates an equilibrium-driven constant tensile stress in the adjacent regions, and the residual stress in the fatigue zone is plotted as the steady stress on a Goodman Diagram. The magnitude of residual stress that exists as a result of this process is difficult to measure, so it is instead approximated through finite element model (FEM) analysis of the monotonic pre-strain procedure.

The relationship of specimen tensile loading and stress generation was first analyzed by Dr. Tommy George with an elastic-plastic analysis showing the experimental procedure was valid and could be reasonably represented through FEM [6]. Errors between and empirical results were high enough to warrant further study by Dr. Onome Scott-Emuakpor, in which empirical strain hardening relationships were gathered from American Society for Testing and Materials (ASTM) standard coupon (dog-bone) specimens and uniformly distributed compressive forces due to grip pressures were added to the model [7, 8]. The contributions of this research will further this work by generating higher fidelity models to ensure converged results, and utilizing digital image correlation (DIC) to validate FEM-generated strain profiles (as opposed to strain points gathered from gages), in both quasi-static and time-dependent FEM loading scenarios [9].

1.2 Research Scope

The purpose of this research is to supplement empirical and analytical results of pre-straining with the more advanced strain measurement technique of DIC. This technique, along with FEM and AFRL's Vibration-based Fatigue Test will be carried out experimentally and analytically, and consistency between each method will be assessed. The intended use of these results is to support design practices already in place for GTE components, but can be proven more cost effectively with aluminum (Al) 6061-T6, with possible extension to specimens consisting of titanium (Ti) and other materials if favorable results are found.

1.3 Problem Statement

The Air Force's engine designers are required to have empirically characterized the fatigue life of engine component materials for HCF failure up to 10^9 cycles, which is

typically time consuming and overly conservative for many applications. Through the application of residual stress profiles via plastic deformation, the AFRL's Vibration-based Fatigue Test will be improved to gather empirical data and validate FEMs to deliver faster and more representative results to engine designers.

1.4 Research Objectives

The goal of this research is to determine the fidelity of mean stress approximations from specimen yielding and subsequent generation of Goodman data after fatigue testing, which will increase accuracy of that data in applications for turbine engine fatigue life estimation and generate data more rapidly. In support of these assessments, this research will focus on the following objectives:

- Establish more representative boundary conditions of FEM load distributions to better approximate experimental specimen yielding
- Utilize DIC strain profiles to assess FEM accuracy throughout entire measurable strain field [10]
- Create a time-dependent FEM to more accurately represent development of plastic deformation, and thus generate more accurate predictions of residual stress and strain

1.5 Assumptions

Many simplifying assumptions exist in order to model the test article both effectively and efficiently. The most obvious of these assumptions is the removal of the holes in the region that will later be clamped to the electrodynamic shaker, as seen in Figure 1.2. These holes will certainly contribute to the equilibrium stress-strain state both before and after plastic deformation, but will be filled by the clamp fixture

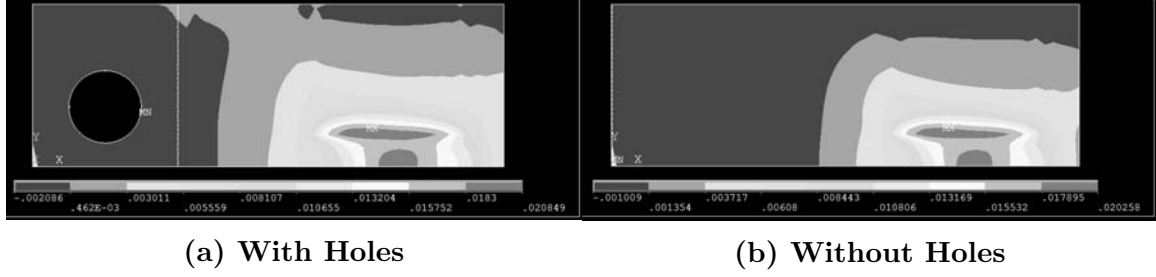


Figure 1.2. Y-strain Hole Comparison, 17,000 lb_f

during vibration testing. However, due to their distance from the applied load and the two-dimensional stress relaxation as a direct result of Poisson’s effect, removing them from the FEM is both acceptable and essential to have a model that converges in a reasonable amount of time. In addition to this, the FEM assumes the test article is cut exactly to the requested geometry when, in fact, small deviations were measured once the articles were received. Global deviations in specimen geometry were on the order of ≈ 0.001 in (as shown in Chapter IV), and appropriate adjustments were made to the FEM geometric constraints. Consistent, but not necessarily constant (due to plastic deformation) material properties throughout the entire plate geometry were also assumed, which implies that stresses imposed on the samples in the machining process could be neglected. The final assumption is that the global vibrational response of the part is unchanged after the residual stresses have been applied via the pre-straining process.

1.6 Chapter Outline

Research is summarized in Chapter II, containing a review of literature pertaining to HCF testing, FEM, plastic deformation, and DIC. The testing methodology, data collection devices, and relationships between the various test procedures will be given in Chapter III. Chapter IV covers the results of all processed data, and their relationships. Final conclusions and recommendations for future work are given in

Chapter V.

II. Theory

The purpose of Chapter II is to provide a comprehensive review of the theoretical and experimental background supporting the work completed as part of this thesis. The following section will offer an explanation for high cycle fatigue (HCF), as well as operational limitations to Department of Defense (DoD) engines in regards to HCF failure and experimental standards for life approximations, at which point the background of the importance of this research should be better appreciated. The Vibration-based Fatigue Test Method developed by Air Force Research Laboratory (AFRL) is then introduced, and furthermore the necessity of the Pre-strain Method in its application of characterizing engine airfoil failure criteria for all applicable loading conditions. Approaches to achieve higher fidelity in this test method will be briefly explained, followed by more complex constitutive relationships to accurately model plastic deformation.

The utilization of constitutive relationships in a computational model is then realized through a finite element model (FEM), requiring discretization of the specimen geometry and boundary conditions similar to experimental conditions. Finally, theoretical and experimental background will be given for digital image correlation (DIC), followed by relationships of these various measurement and modeling techniques to the Pre-strain Method. The concluding section will tie the extensive material covered in this chapter together by utilizing root mean squared error (RMSE) and root mean squared deviation (RMSD) to directly quantify the work to be performed as well as the interpretation of relationships behind the various measured phenomena.

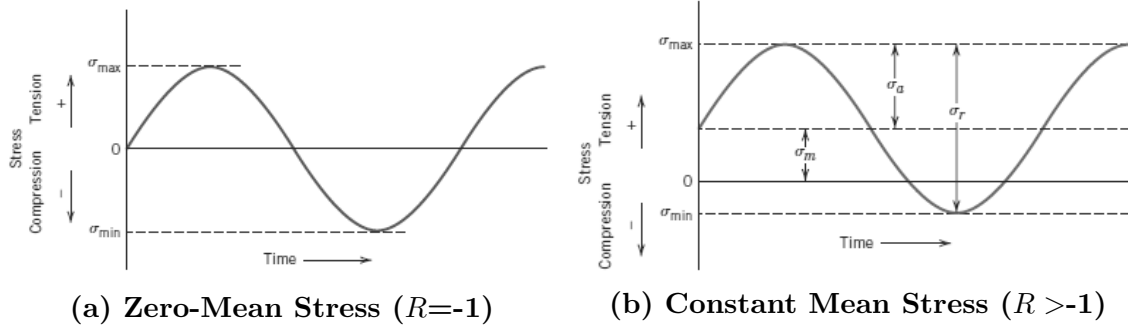


Figure 2.1. Cyclic Loading Profiles [11]

2.1 High Cycle Fatigue

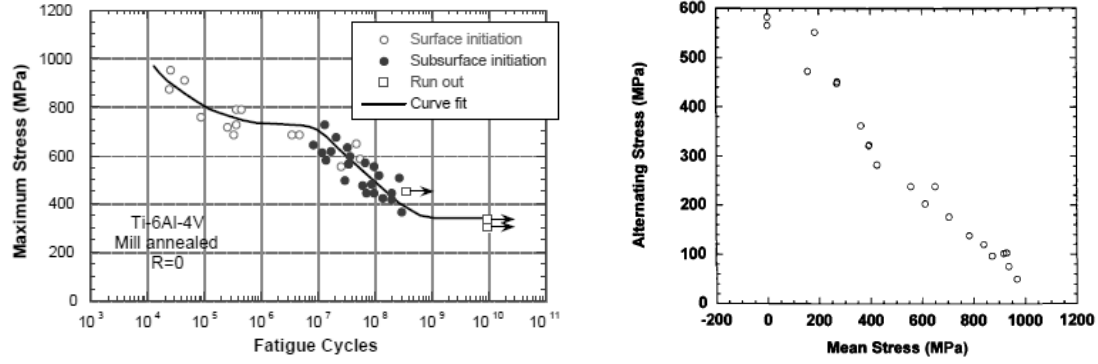
Fatigue is a type of loading that a structure has over time when subjected to fluctuating forces, and accounts for the vast majority of mechanical failures [1]. This deterioration of material strength is named “fatigue” because breakdown often occurs well below the yield stress (σ_0), where structures normally would not fail, but long durations of repeated loading result in failure nonetheless. Oscillatory loading of a structure can occur where the maximum and minimum stresses are of equal magnitude, known as fully reversed loading as seen in Figure 2.1a, or with a constant non-zero average stress, σ_m , as seen in Figure 2.1b and Equation 2.1.

$$\sigma_m = \frac{\sigma_{max} + \sigma_{min}}{2} \quad (2.1)$$

Additionally, Equation 2.2 can be used to quantify the deviations between the mean stress and the stress amplitude (σ_a) by a stress ratio (R), which is equal to negative one when the loading is fully reversed, as seen in Figure 2.1a.

$$R = \frac{\sigma_m - \sigma_a}{\sigma_m + \sigma_a} \quad (2.2)$$

The number of cycles a specimen can withstand under cyclic loading is known as the fatigue life, and is primarily dependent on the stress amplitude and mean stress, with



(a) Ti-6Al-4V SN Curve [12] (b) Ti-6Al-4V Haigh Diagram² (10^7) [13]

Figure 2.2. Fatigue Life Approximation Methods

ancillary dependence on temperature and other environmental parameters.

2.1.1 Goodman Diagram

The relationship between the stress amplitude, mean stress, and their combined effect on fatigue life is most commonly characterized through an SN curve or a Goodman¹ diagram. The SN curve (Figure 2.2a) is useful to display the lower number of failure cycles with respect to rising maximum stresses, but is limited in that it cannot effectively attribute changes in R (and thus σ_m). The particular SN curve shown highlights that as the maximum stress declines, specimen failure transitions from surface cracks to cracks below the surface, and below 400 MPa, the fatigue life is found to grow substantially (defined as an endurance limit). The Goodman diagram is widely used for addressing the relationship of all of these parameters for a given number of cycles (10^7 cycles in the case of Figure 2.2b), and shows clearly that an increased σ_m decreases the magnitude of alternating stress that a material

¹“The attribution of a diagram to Goodman or Haigh is historically inaccurate. The use of a constant fatigue life diagram of some type using maximum stress, minimum stress, stress ratio, alternating stress, or mean stress in some combination predates either of these individuals. The alternating stress amplitude, the shape of the curve, the equation describing the curve, or the specific axes used can be attributed to various individuals including, but not limited to (in chronological order), Rankine (1864), Mueller (1873), Launhardt (1873), Gerber (1874), Schaffer (1874), Weyruch (1877), Smith (1880), Unwin (1888), and Johnson (1897).”[13, 14]

can withstand. The number of acceptable cycles to failure is typically defined by design guidance or an industry standard for a given application, and the DoD has various programs to define these standards. The Engine Structural Integrity Program (ENSIP) is the document most pertinent to HCF performance of gas turbine engines (GTEs) [2].

2.1.2 Engine Structural Integrity Program

ENSIP adopted many practices from the similarly named and composed Aircraft Structural Integrity Program, and “Establishes the structural performance, design development, and verification guidance for turbine engines” [2]. The term “guidance” is used suggestively, as it is not truly a requirement that binds contractors, but is still strongly adhered to in a majority of cases. The driving statement that generates the importance of this research is critical engine components should have HCF life characterization to 10^9 cycles. This is driven by the fact that most metallic components appear to exhibit no limit to fatigue life (endurance limit) at stresses below that which would drive failure in 10^9 cycles, as can be seen in Figure 2.2a. If components can be proven to survive for 10^9 cycles, then it is likely their life at lower stresses is so long that it is beyond service life. While the geometry of most engine components is very complex, controlled testing of simplified components can be carried out to 10^9 cycles, the results of which can be extrapolated to more complex geometry for preliminary life analysis. This saves the DoD substantial time and money in meeting the challenging recommendations of ENSIP through material and component testing.

2.1.3 ASTM Test Standards

American Society for Testing and Materials (ASTM) has developed a standard test procedure (designation E 466-07) that explains the proper test procedure for axial

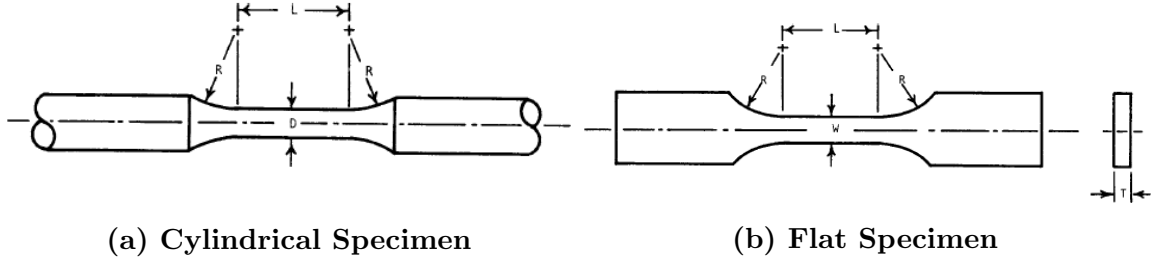


Figure 2.3. Specimens with Tangentially Blending Fillets [3]

force controlled fatigue tests in the elastic region of the material [3]. The specimen geometry has a fillet (see Figure 2.3a) to ensure that stresses due to the mechanical grips loading the specimen do not cause a failure outside the test region. A derivative of this geometry (seen in Figure 2.3b) is used for flat specimens, which was used for the purpose of this research due to specimens being extracted from 24 in by 24 in flat sheets of aluminum (Al) 6061-T6. In this case, the test section width, W , as well as the fillet radius should be at least eight times the thickness and $\sigma = \frac{F}{W \times T}$. The final requirement is that the frequency be between 10^{-2} Hz and 10^2 Hz for this test method, but typical load frames can only cycle around 40 Hz, meaning the test would take more than 290 days to reach 10^9 cycles. In addition to this long testing time, GTE components are also much more complex than this “dogbone” geometry, and as such, the stresses are much harder to calculate because analytical models do not exist to directly calculate material behavior. Advanced numerical methods such as FEM can be used to more accurately approximate the stresses of these structures. ASTM E 8M is very similar to ASTM E 466, except that loading is static tension testing instead of oscillatory loading [7].

2.2 Vibration-based Fatigue Test

While a significant amount of fidelity can be obtained using uniaxial fatigue data and extrapolating to three dimensional geometries via FEM approximations (further explained in Section 2.5), in reality, GTE airfoils typically fail in a biaxial bending

modes due to vibrational fatigue [4]. The vibrational characteristics of rectangular plates has been studied extensively to determine resonant frequencies and their associated mode shapes for given aspect ratios [13]. Intuition follows that a GTE airfoil of moderate camber will have similar mode shapes and resonant frequencies to that of a rectangular plate with the same aspect ratio (AR in Equation 2.3), which is easier to manufacture and measure experimentally.

$$AR = \frac{\text{Span Length}}{\text{Chord Length}} \quad (2.3)$$

The rectangular plate will fail in a similar state of stress, and the empirical results can be used to develop a more representative FEM of the airfoil geometry, and thus a better prediction of survivable load cycles before failure.

Based on this notion, AFRL, in conjunction with The Ohio State University, developed the Vibration-based Fatigue Test Method [15, 16]. The test methodology will be discussed further in Chapter III, but the basic premise is the specimen geometry seen in Figure 2.4b, with an unclamped aspect ratio of 1. The stress contours predicted by FEM show a maximum stress location at the free end (see Figure 2.4c), which is predicted based on experimental strain gage measurements over that location and a known elastic modulus (E). The value of this test method is that the stress state and frequency are more representative of GTE airfoils in service, and the expected test time to gather empirical data is reduced by 95% when compared to the conventional uniaxial fatigue methods. The main detriment to gathering data this way is that vibrational fatigue testing is typically restricted to fully reversed loading ($R=-1$), but research is progressing past this limitation as well [5, 6].

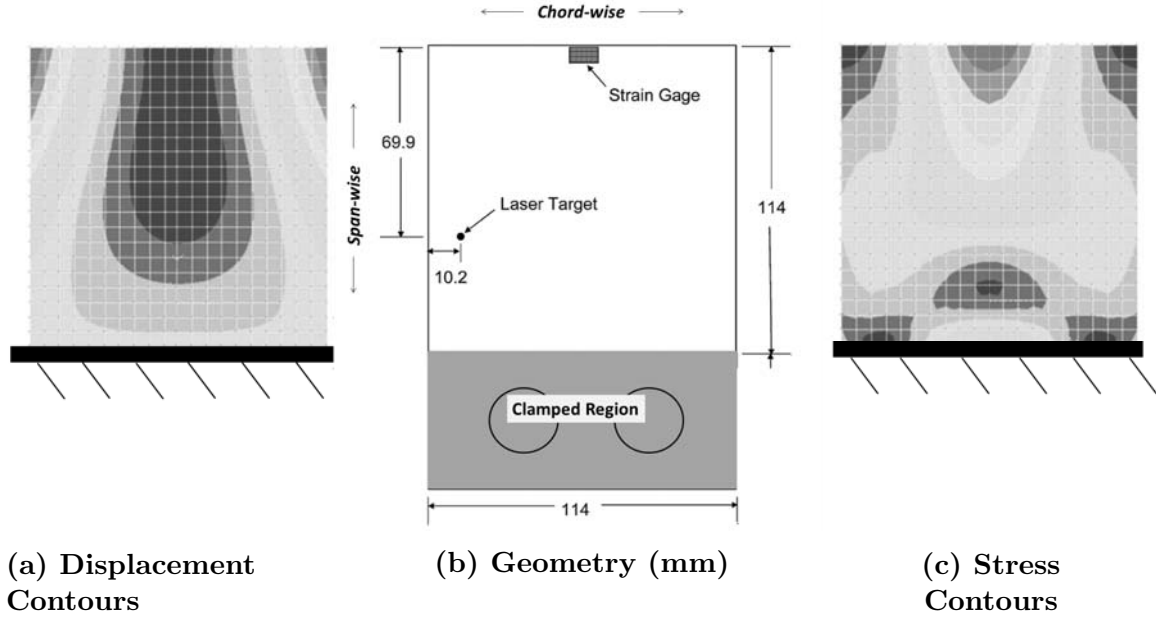


Figure 2.4. Vibration Based Fatigue Test Specimen [4]

2.3 Pre-strain Residual Stress Generation Method

Static loading can generate plastic deformation which is permanent after unloading. In the case of an axially loaded dogbone specimen, this may cause necking, but for significantly large plate geometry, the surrounding material restricts necking from occurring. Instead, a permanent residual state of strain exists in the region that has been plastically deformed, and after unloading, a residual state of stress exists in the surrounding regions to satisfy force equilibrium. A stress-strain diagram for the maximum principle strain direction can be seen in Figure 2.5, where nonlinear strain hardening occurs after the elastic limit is exceeded, and upon unloading a residual tensile state of stress and strain exists. This residual stress (σ^{res}) is a product of the elastic recovery from the total strains developed in the prior loading configuration ($\varepsilon_{tot}^{peak} - \varepsilon_{tot}^{res}$), which can be used to directly calculate stress in the residual state based

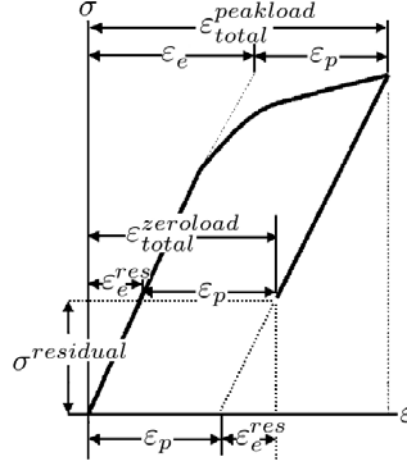


Figure 2.5. Residual Stress-Strain [6]

on generalized Hooke's Law, seen in Equation 2.4.

$$\sigma_{ij}^{res} = \sigma_{ij}^{peak} - \frac{E}{1 + \nu} \left[\left\{ \varepsilon_{ij}^{peak} - \varepsilon_{ij}^{res} \right\} + \frac{\nu}{1 - 2\nu} \left\{ \varepsilon_{kk}^{peak} - \varepsilon_{kk}^{res} \right\} \delta_{ij} \right] \quad (2.4)$$

This preliminary loading step, called the Pre-strain Method, was developed to generate a residual steady stress in the region of expected failure for a Vibration Based Fatigue Test specimen [6]. The presence of this stress, when combined with the alternating stress imparted by resonance vibration, creates a state of stress with $R > -1$ in a different method than classical Goodman data. The so-called “Modified” Goodman diagram accounts for the uniaxial bending fatigue stress, which is not only more representative of GTE airfoils, but also predicts a slightly higher level of alternating stress for a given steady stress on a constant life diagram, as seen in Figure 2.6b. It follows from this deviation that life approximations for an operational failure due to bending fatigue would have been overly conservative if empirical data was collected from a uniaxial Goodman diagram, which is typically the case. This testing step requires proper use of experimentally determined constitutive relationships and computationally derived residual stresses using FEM. It has been iteratively improved by removing certain assumptions of boundary conditions and material behavior, as

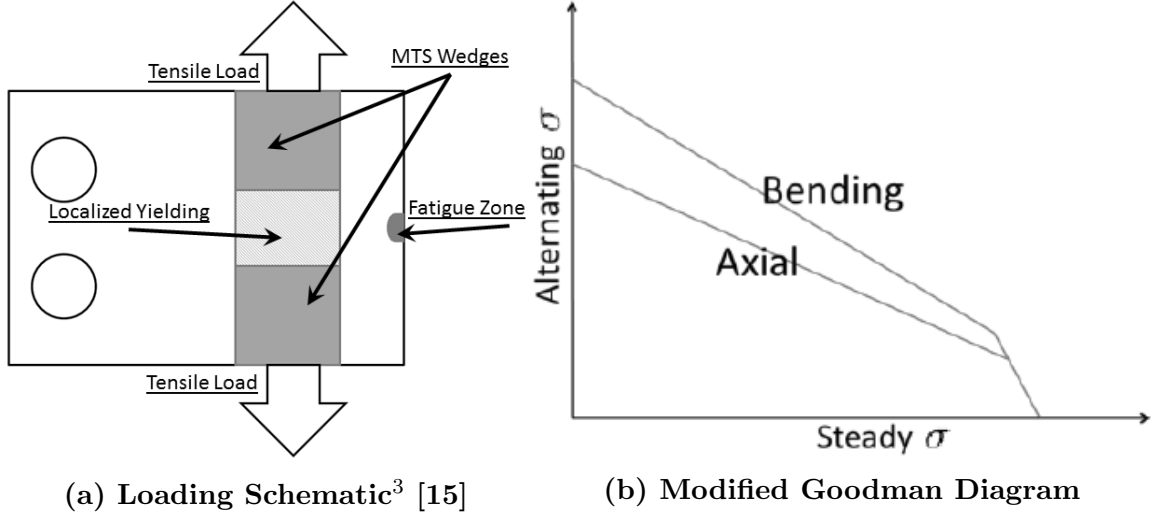


Figure 2.6. Pre-strain Method

described in the following sections.

2.3.1 Elastic, Perfectly Plastic Modeling

The initial modeling of the Pre-strain Method was carried out with Al 6061-T6 and the material properties that are known for that specific alloy, namely a Young's Modulus of 10^7 psi and a proportional limit of 39 ksi were accounted for in FEM analysis[6]. The model created by George approximated the plate specimen with eight-noded brick elements (explained further in Section 2.5) with the tensile force imparted by the grips modeled as a concentrated shear force at each node that would have been in contact with a grip surface [15]. The material model used was an elastic, perfectly plastic model which, while inconsistent with known Al behavior, is still a reasonable model because most stresses in the plate are less than the proportional limit. This research proved that the Pre-strain Method was possible experimentally, that plastic deformation could be accurately resolved using FEM (with residual ε_{yy} errors of 68%), and that empirical Goodman data matched expected values (with lim-

³As modified by Kevin Knapp

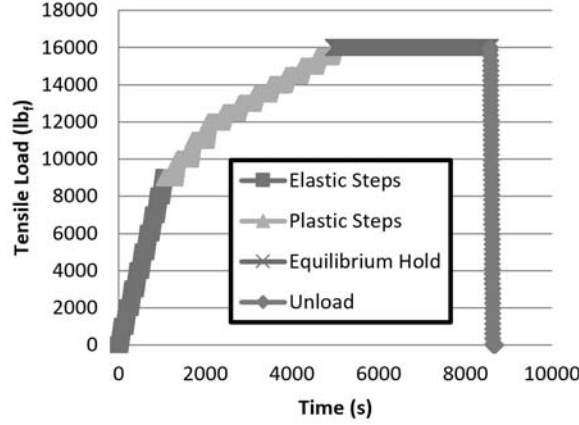


Figure 2.7. Load Step Curve

ited precision) for residual stresses up to 14.5 ksi. One of the main drivers restricting precision was the assumption of elastic, perfectly plastic material behavior. This precision can be improved with knowledge of material behavior beyond the proportional limit, as well as more accurate boundary conditions.

2.3.2 Multilinear Hardening Modeling

Scott-Emuakpor readdressed the work of George, successfully applying more complex constitutive models in addition to the pressures imparted on the specimen by the grips [8]. Grip pressures, similar to the tensile shear forces, were added as concentrated loads at all nodes in contact with the grips, with an even distribution across all nodes. The constitutive relationship between stress and strain beyond the proportional limit was determined from flat dogbone specimen tests, as discussed in ASTM E 8M, and input into an Abaqus FEM solution procedure [7]. In this procedure, the load was incrementally increased by 1,000 lb_f steps every 30 seconds when the specimen is in the elastic region, with steps as small as 250 lb_f once the specimen exceeds yield, as seen in Figure 2.7. These results, as expected, produced Goodman data that was even more in alignment with the expected results (with residual ε_{yy} errors of 38%) based on the FEM-approximated residual stress at the free end of the

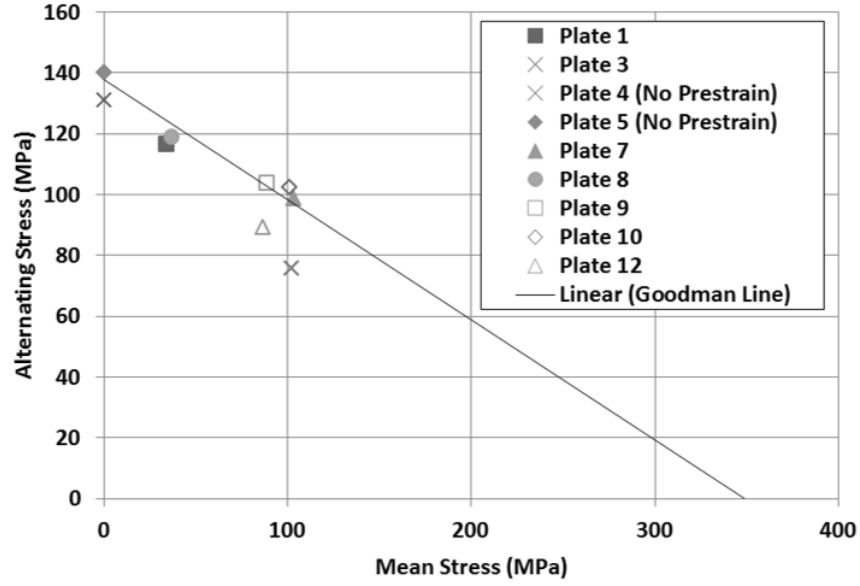


Figure 2.8. Al 6061-T6 Vibration-based Fatigue 10^6 Failure Cycles Goodman Line [17]

test specimen.

2.3.3 Past Personal Research on Pre-strain Method

Preliminary work was performed prior to this research, mirroring of the work performed by Scott-Emuakpor [8], modeling in ANSYS rather than Abaqus. Both FEM solvers are well-developed, and produced similar results, but strain data was gathered predominantly with strain gages or point strain measurements from DIC (as opposed strain field data presented in this research). Goodman data acquired with the Pre-strain and Vibration-based Fatigue Methods are plotted in Figure 2.8. It can be seen that while some experimental variation is present, FEM predictions of residual stress as a result of the pre-straining procedure follow Goodman line predictions very well. The aim of this research is to gain a better understanding of the mean stresses as a result of plastic deformation by means of DIC strain measurements, and to develop a more accurate FEM.

2.4 Three Dimensional Material Constitutive Equations

Basic one dimensional constitutive equations are useful for the purpose of describing axisymmetric loading conditions, but more robust models are often necessary for the multiaxial loading in three-dimensional FEM analyses. In the case of modeling plastic deformation, axially derived constitutive relations must at the very least be transformed to three-dimensional relations, and assumptions of plastic behavior must also be made.

2.4.1 Von Mises Isotropic Hardening

The modeling of elastic-plastic material response curves in FEM requires the utilization of a yield criterion, a flow rule, and a hardening rule. The yield criterion in this three-dimensional stress analysis is based on an axially determined yield stress, expanded to three dimensions by an effective stress based on the von Mises yield criterion. Plastic deformation is manifested via deviatoric stresses (s_{ij} , Equation 2.5) only, so the effective stress, (σ_e , Equation 2.6), is calculated by subtracting the hydrostatic stress (Δ) from the stress state in order to calculate von Mises stress, which is the square root of the sum of all squared stresses.

$$s_{ij} = \sigma_{ij} - \frac{1}{3}\sigma_{kk}\delta_{ij} = \sigma_{ij} - \frac{\sigma_{xx} + \sigma_{yy} + \sigma_{zz}}{3} = \sigma_{ij} - \Delta \quad (2.5)$$

$$\sigma_e = \sqrt{\frac{3}{2} [s_{xx}^2 + s_{yy}^2 + s_{zz}^2 + 2(s_{xy}^2 + s_{yz}^2 + s_{zx}^2)]} \quad (2.6)$$

All plastic deformation occurs after the effective stress has reached the axially determined yield stress, σ_0 , which in three dimensions can be interpreted as being at a state of stress coincident with a yield surface, as seen in Figure 2.9a [19]. At this yield surface, the effective stress is related to an effective strain via Hooke's law for

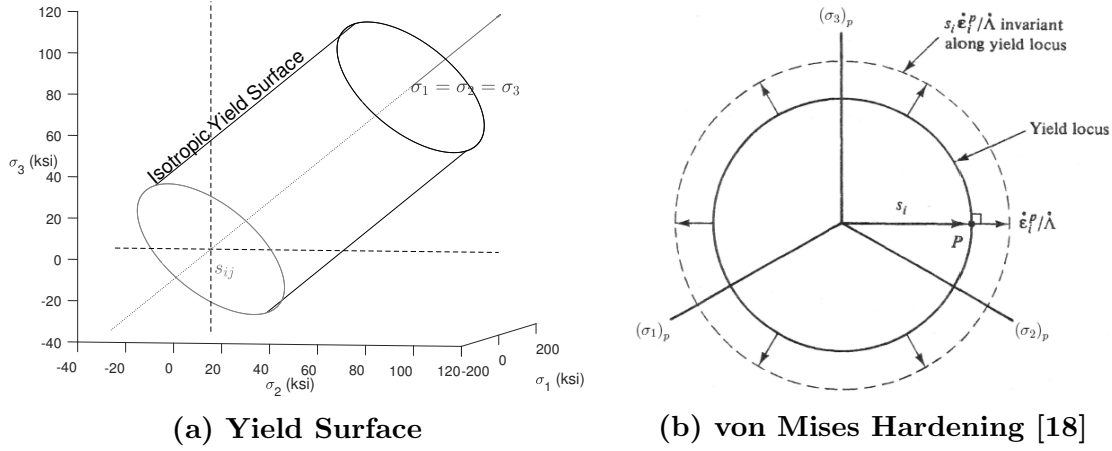


Figure 2.9. Isotropic Yielding

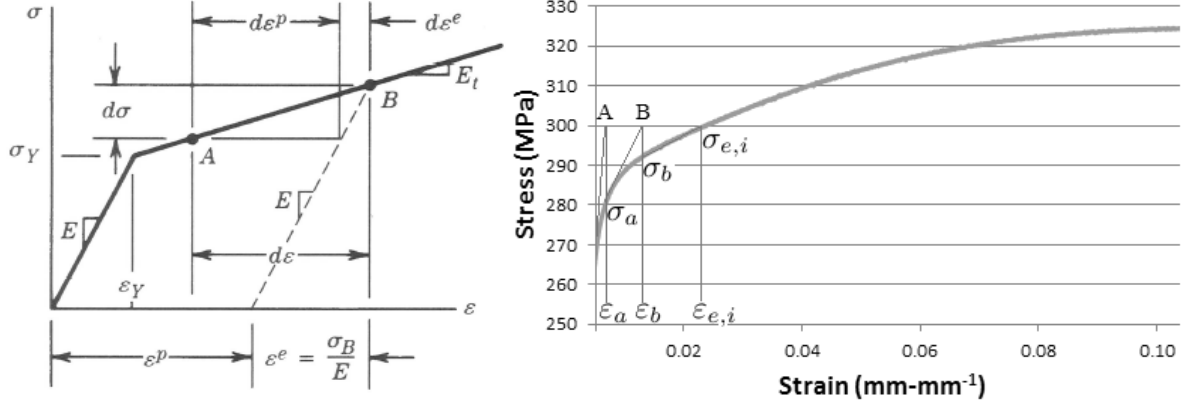
an isotropic material, in which the material maintains the same elastic modulus in all principle directions, while accounting for the material's respective Poisson's ratio (ν) [19].

2.4.2 Prandtl-Reuss Relations

The development of plastic flow is calculated as an incremental effective plastic strain (shown as ε_e^p in Equation 2.7), directly correlated to an increment of effective stress above the proportional limit [19].

$$\dot{\lambda} = \dot{\varepsilon}_e^p = \sqrt{\frac{2}{3} \left[(\dot{\varepsilon}_{xx}^p)^2 + (\dot{\varepsilon}_{yy}^p)^2 + (\dot{\varepsilon}_{zz}^p)^2 + 2 \left\langle (\dot{\varepsilon}_{xy}^p)^2 + (\dot{\varepsilon}_{yz}^p)^2 + (\dot{\varepsilon}_{zx}^p)^2 \right\rangle \right]} \quad (2.7)$$

These increments of plastic flow are incompressible ($\nu = 0.5$), and by the same evaluation of three-dimensional effective stress/strain being correlated to an axially determined result, the incremental effective stress/strain will still follow the axial response curve. The response curve itself is manifested by the hardening rule, which is known to be isotropic for Al and thus the isotropic character is maintained both during and after plastic deformation. This can be described as a yield that surface moves further from the origin in all principle directions under plastic deformation, but maintains



(a) Plastic Strain Development [19]

(b) Iterative σ_e vs. ϵ_e^p

Figure 2.10. Aluminum 6061-T6 Experimental Data with Newton-Raphson Iterations

the same shape as seen in Figure 2.9b. The combination of the yield criterion, flow rule, and hardening rule results in incremental stress-strain behavior known as the Prandtl-Reuss relations, shown in Equation 2.8 [19].

$$\frac{\dot{\epsilon}_{xx}^p}{s_{xx}} = \frac{\dot{\epsilon}_{yy}^p}{s_{yy}} = \frac{\dot{\epsilon}_{zz}^p}{s_{zz}} = \frac{\dot{\epsilon}_{xy}^p}{s_{xy}} = \frac{\dot{\epsilon}_{yz}^p}{s_{yz}} = \frac{\dot{\epsilon}_{zx}^p}{s_{zx}} = \frac{3}{2} \frac{\dot{\lambda}}{\sigma_0} \quad (2.8)$$

The preceding explanation was developed to explain how to iteratively solve for plastic strains at a given loading condition, but in general, the plastic multiplier ($\dot{\lambda}$) is an indeterminate function of s_{ij} and $\dot{\epsilon}_{ij}^p$, and must be iteratively converged to a material response curve using the Newton-Raphson Method [19]. An input effective stress ($\sigma_{e,i}$) is supplied from the specimen loading, and an elastic assumption of resultant strains is made (with Young's Modulus of E as in Figure 2.10a), seen at point A for an input stress of 300 MPa in Figure 2.10b. The strain, according to the axially generated constitutive model is then used to check if the calculated stress at A exceeds the yield stress, thus entering a state of overstress that results in plastic deformation. If an additional plastic portion of strain exists, a subsequent iteration is carried out from the tangent modulus (E_T) at σ_a along the material response curve, again checking for sufficient stresses to reach the equilibrium strain response ($\epsilon_{e,i}$)

[20].

$$E_T = \frac{\sigma_{(i)} - \sigma_{(i-1)}}{\varepsilon_{(i)} - \varepsilon_{(i-1)}} \quad (2.9)$$

The deviatoric stresses (s_{ij}) in excess of the original proportional limit (σ_0) are then calculated and Prandtl-Reuss relations are used for calculating the incremental effective plastic strains ($\dot{\varepsilon}_{ij}^p = \frac{3}{2} \dot{\lambda} \frac{s_{ij}}{\sigma_0}$) and the isotropic expansion of the yield surface due to plastic flow. Following these calculations, the updated yield stress ($\sigma_{0_{new}}$) is established as the new *current* yield stress, according to the previously mentioned isotropic hardening law. This means repetition to the previous stress would produce no additional plastic strains, and the total strain is now the summation of all elastic strain and incremental plastic strain ($\varepsilon^{tot} = \varepsilon^{el} + \varepsilon^p$). This same procedure is iteratively performed until the stresses input from structural loading of the test specimen are fully accounted for and global force equilibrium has been satisfied[21].

2.4.3 Quasi-static vs. Rate-dependent Plasticity

Until introducing $\dot{\varepsilon}_e^p$ and the Prandtl-Reuss relations, approximations of stresses and resultant strains have been described as being applied instantaneously with respect to time. In fact, increments of plastic strain could also be modeled as *instantaneous* by adjusting the strain rates to differential changes in strain ($\dot{\varepsilon}_{ij}^p \Rightarrow d\varepsilon_{ij}^p$), but this would be more accurately described as a quasi-static solution. Plastic deformation in many materials exhibits viscoplastic behavior, in which time dependence in strain development will change the material properties beyond the proportional limit (σ_0), as seen in Figure 2.12b. The quasi-static strain rate can be thought of as the equilibrium rate of plastic deformation, in which case loading at a lower strain rate will cause no change in results, but higher rates will cause a stress state that is in excess of the equilibrium state (also known as overstress). In the case of Al 6061 T6, this rate is higher than the strain rate of the monotonic dogbone tests, so the

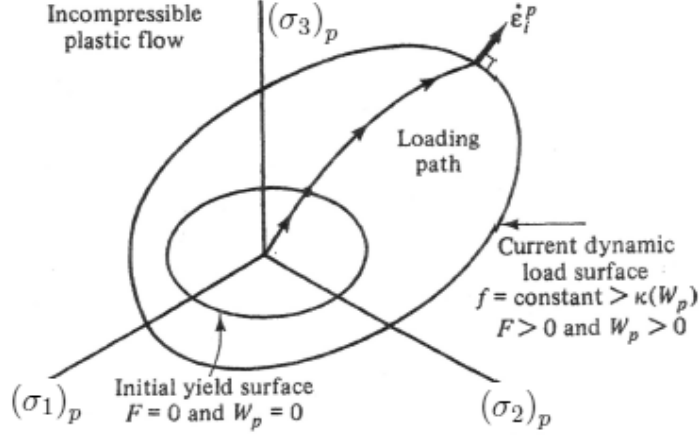


Figure 2.11. Change from Yield Surface to Dynamic Load Surface due to Strain and Strain Rate Hardening [18]

input material response curve, converted to true stress-strain, is also the quasi-static viscoplastic curve [22].

The experimental loading of the plate, however cannot be assumed to follow the quasi-static response curve throughout the entire specimen. Regions of very localized high stresses will exceed this rate, and the “instantaneous” FEM solution (dependent on Boltzmann’s Superposition Principle and the summation of plastic and elastic strains) will generate results inconsistent with the experimental results. A more robust approximation can be made when accounting for the strain rate as a function of overstress (deviation from quasi-static solution) by using Perzyna’s Model, as shown in Equation 2.10.

$$F(\sigma, \varepsilon^p) = \frac{f(\sigma, \varepsilon^p)}{\kappa(W_p)} - 1 \quad (2.10)$$

In this model, the load function ($F(\sigma, \varepsilon^p)$) is assumed to be developed from a dynamic load surface function ($f(\sigma, \varepsilon^p)$) and a work hardening parameter ($\kappa(W_p)$), to the extent that the load function is negligible until the material has sufficient stress to exceed the yield surface, as seen in Figure 2.11. Defining yielding in this manner allows for a variety of yielding criteria in the form of the dynamic load function as well as many work hardening models, but the chosen application utilizes isotropic hardening

and the von Mises yield criterion, while accounting for viscoplasticity. The strain rate is a function of a strain rate hardening parameter (m), a viscosity parameter (γ), and the stress that exists beyond the proportional limit [23]. The proportional limit at any point in time is a function of the isotropic strain hardening that has taken place in the history of the material loading, which is taken directly from the quasi-static solution.

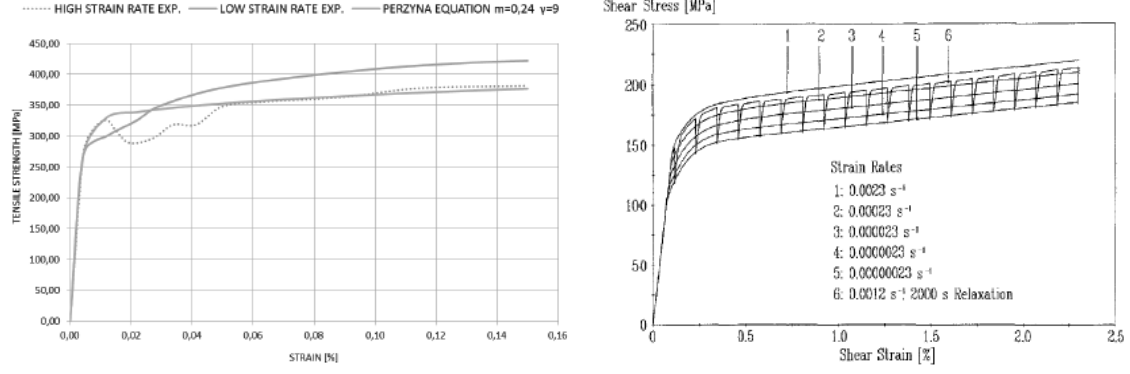
For the purpose of this research, the experimental loading of the plate and the accuracy of the FEM approximation are of primary concern. The parameters for the Perzyna strain rate model can be approximated experimentally via stress-strain behavior at various strain rates by solving Equation 2.11a for stress, as shown in Equation 2.11b, and matching the model to the experimental data.

$$\dot{\varepsilon}^p = \gamma \left(\frac{\sigma}{\sigma_0} - 1 \right)^{\frac{1}{m}} \quad (2.11a)$$

$$\sigma = \left[1 + \left(\frac{\dot{\varepsilon}^p}{\gamma} \right)^m \right] \sigma_0(\varepsilon^p) \quad (2.11b)$$

This procedure was carried out by Andrzej Mróz based on experimental data gathered by Toshiji Mukai for varying strain rates on Al 6060 alloys at differing strain rates, as seen in Figure 2.12a [23, 25]. The research performed by Mukai was performed using a Split Hopkinson bar for the high strain rate ($2 \times 10^3 \text{ s}^{-1}$, seen as dotted blue line) and the low strain rate tests ($1 \times 10^{-3} \text{ s}^{-1}$, seen as solid blue line) were gathered from monotonic testing. The extracted parameters were $\gamma = 9$ and $m = 0.24$, gathered from a curve fit of Equation 2.11b to the high strain rate data (seen as orange line by Mróz) which were used throughout the time-dependent analyses.

A more complete method of developing the model framework for this material would require a series of tests for the full range of expected strain rates to ensure



(a) Al 6060 Perzyna Approximation [23] (b) XCrNi18.9 SS Rate Dependence [24]

Figure 2.12. Viscoplastic Phenomena

analysis of quasi-static behavior. Such tests were performed by Haupt under both strain and stress control to characterize viscoplastic behavior of XCrNi18.9 stainless steel, as seen in Figure 2.12b [24]. True characterization of quasi-static behavior was shown via a series of creep (defined as strain developed at a constant controlled stress) and relaxation (defined as stress reduction at a constant controlled strain) tests to highlight equilibrium behavior under a variety of strain histories [24]. The behavior shown by stainless steel (and consequently Al [25]) is shown to exhibit states of overstress when loading rate exceeds the quasi-static strain rate ($2.3 \times 10^{-7} \text{ s}^{-1}$ for stainless steel (SS) in Figure 2.12b) beyond the proportional limit. For stainless steel, Haupt showed that loading at a rate of $1.2 \times 10^{-3} \text{ s}^{-1}$ exhibits overstress behavior, but holding at a constant strain caused the overstress to approach zero after 2000 seconds [24]. Such a material characterization scheme was not feasible within the time constraints of the research, so monotonic dogbone tests were used as a quasi-static response with the Perzyna parameters approximated by Mróz [23].

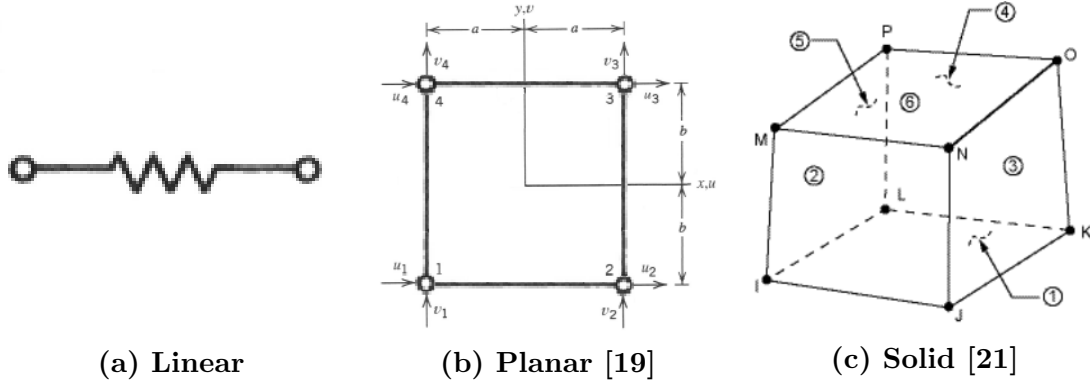


Figure 2.13. Basic Element Types

2.5 Finite Element Modeling

The method of finite elements is a numerical technique that evaluates a field problem as the accumulation of smaller field problems of a finite size, as opposed to the infinitesimal scale utilized in differential calculus [19]. The power of FEM is that complex problems can be simplified to a series of more manageable problems with the use of computational tools, with little deviation from the real application that may have otherwise been impossible to solve. Conversely, the basic dogbone test region described thus far could be solved with a single linear elastic element, as seen in Figure 2.13a, which is no different than simply solving for stress and strain analytically. The axial data obtained from a dogbone test can be extended to a planar or solid element (Figures 2.13b and 2.13c, respectively), and a series of those elements

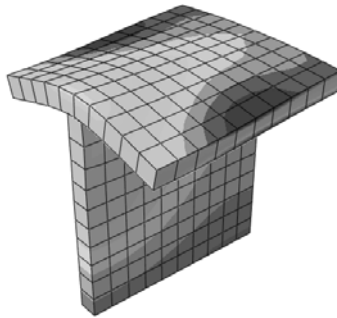


Figure 2.14. 3D Finite Element Model

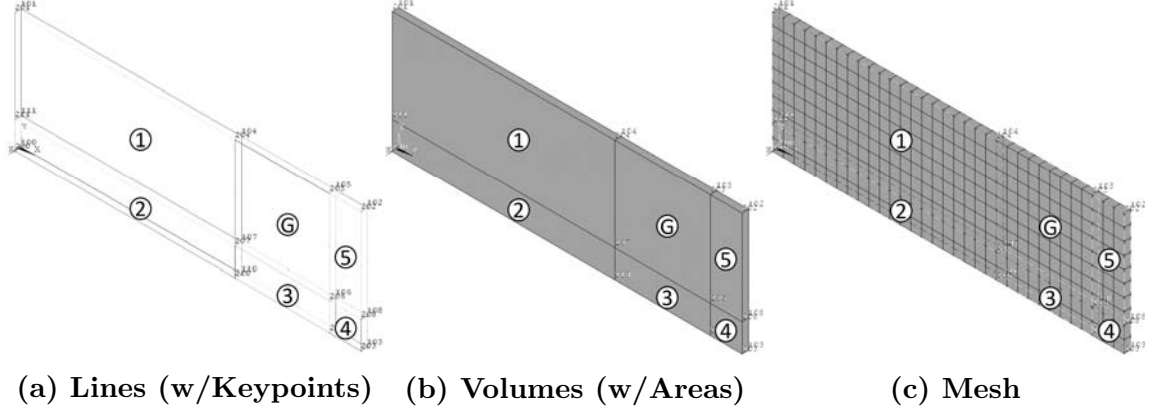


Figure 2.15. Mesh Generation Procedure

can be iteratively solved for more complex structures such as Figure 2.14. While there are multiple FEM solvers commercially available, the research presented in this work was completed using ANSYS, and the models were generated as three-dimensional models using eight-noded brick elements (Solid-185, as seen in Figure 2.13c).

2.5.1 Mesh Generation

A full ANSYS input script is provided in Appendix A to run a time-dependent analysis with external inputs of material properties, grip reaction forces, and mesh generation parameters. The latter two inputs are generated in MATLAB with scripts in Appendix B, but the mesh generation is left out due to the extensive length and marginal utility. In this input, the geometry is generated through keypoints, which are then utilized to generate subsequent lines, areas, and volumes in that respective order. The main discovery in mesh generation was that if generation of any geometric feature was carried out in the order of points, lines, areas, volumes, mesh, and each respective step started with the regions associated with grip contact (seen as circled G in Figure 2.15), then the numbering of nodes generated would start from one and proceed to a final number of nodes on the grip interaction surface. While this may seem trivial, it aids substantially in the ability to transpose force distributions as the

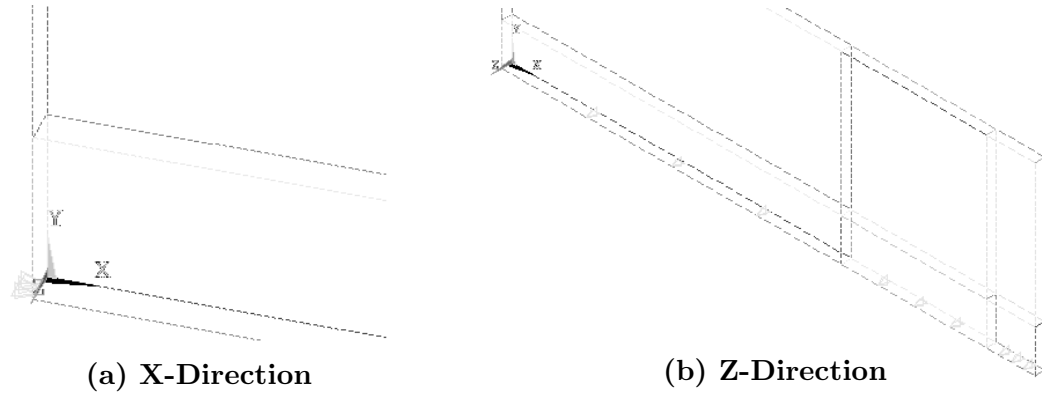


Figure 2.16. Boundary Conditions

FEM boundary conditions.

2.5.2 Displacement Boundary Conditions

Boundary conditions must be placed on the plate in order to ensure that it does not travel in free space. A symmetric boundary condition was placed along the length of the plate at the centerline of the width in order to ensure the previously mentioned half-plane symmetry. Half-plane symmetry simplifies the model to aid in faster convergence of solutions with the same resultant load response. Along the same area, rigid boundary conditions were placed in the Y-direction to create the appropriate response to tensile loads imparted by the grips. Displacements in the Z-Direction were limited along the plate centerline, as seen in Figure 2.16b, to simply hold the plate in free space. A similar boundary condition was placed in the X-Direction as seen in Figure 2.16a.

2.5.3 Load Boundary Conditions

Loads are applied to the half-plate geometry as point loads at each node in contact with the grip surface, as seen in Figure 2.17. Point loads must be broken into vertical components (tensile shear) and normal components (compressive force), which are overwritten (rather than added) if a new load component is specified in the same

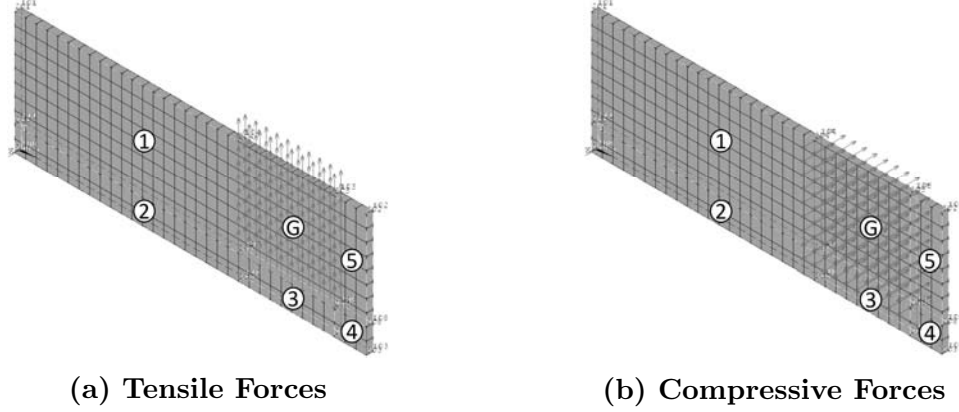


Figure 2.17. Point Load Boundary Conditions

direction. For the purpose of this research, the compressive force distribution does not change throughout the FEM analysis, but the tensile force distribution changes for each loadstep (described further in Chapter IV), and is overwritten for each subsequent loadstep. It is for this reason that proper node numbering along the grip contact surface is of such importance, as explained in the mesh generation section above.

2.5.4 Time-dependent vs. Quasi-static FEM Solutions

Two separate solutions methods were used in FEM to predict plastic deformation and residual stress of the specimen. The quasi-static solution (sometimes referred to as instantaneous) is achieved in only two loadsteps, as seen in Figure 2.18a, while the time-dependent solution considers multiple loadsteps and their associated ramp time in specimen loading. The quasi-static solution applies loads incrementally, with many intermediate equilibrium iterations to adhere to the quasi-static σ - ϵ response curve (Figure 2.18b) using the Newton-Raphson solution procedure. The load increments in this method are differential increments, with no consideration of time. Conversely, the time-dependent solution procedure considers not only the time associated with its load ramp increments, but also the rate dependence of the σ - ϵ response curve

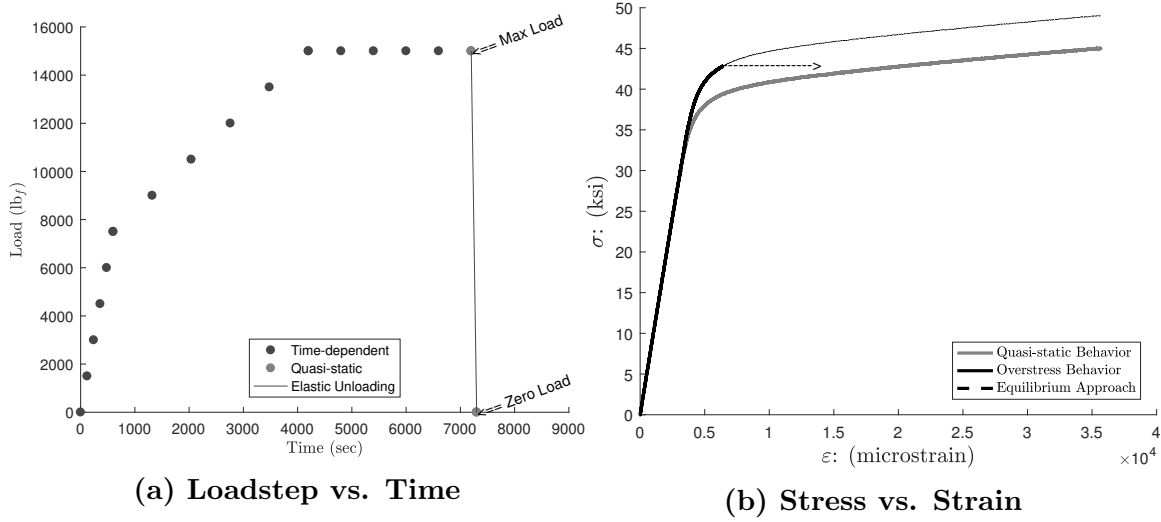


Figure 2.18. Time-dependent vs. Quasi-static

when loaded at rates above the quasi-static strain rate, according to Perzyna's Model (described in Section 2.4.3). Loading at a rate above the quasi-static response is often referred to as an overstress state due to strain-rate hardening, which approaches the equilibrium response when held at a constant load as seen in Figure 2.18b. The application and results of these two solution methods will be discussed extensively in Chapter IV.

2.6 Digital Image Correlation

Most common experimental measurements of strain are taken using strain gages and the theory behind their use is well understood. However, a newer and more advanced method of strain measurement exists, in which strain can be measured in a two-dimensional state (rather than the one-dimensional measurements of strain gages), namely DIC. While the mathematical algorithms behind data processing for DIC are computationally challenging to carry out (as introduced in 2.6.1 and further explained in Appendix D), the theory is quite simple. An image with a random distribution of high and low contrast regions (referred to as a speckle pattern in

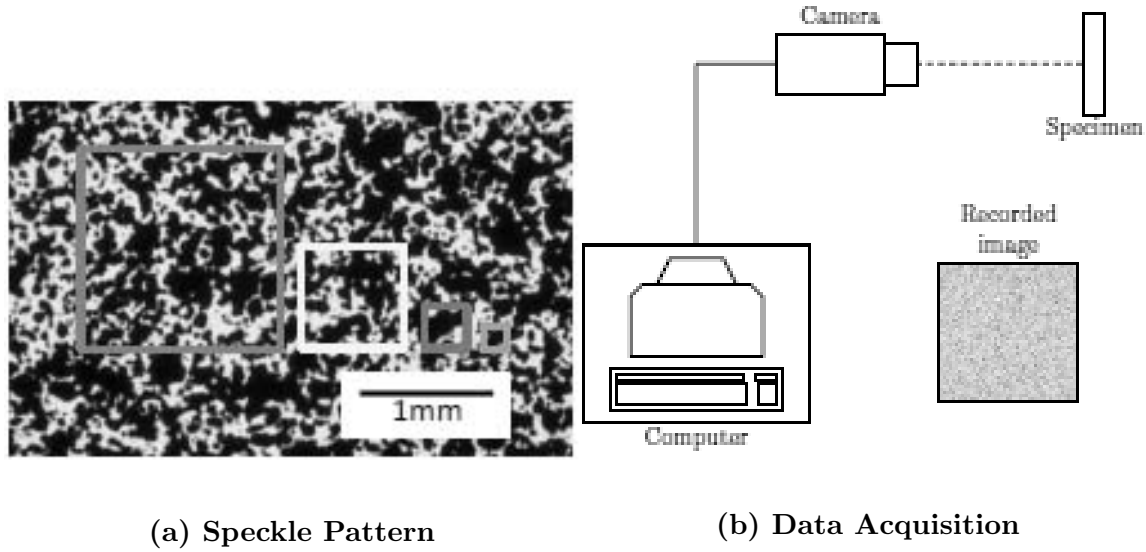


Figure 2.19. Digital Image Correlation Capture [10]

Figure 2.19a) is captured in a reference state and a deformed state, as seen in Figure 2.20a. The images are stored digitally and discretized into overlapping subsets, each of which is offset by a finite step size as seen in Figure 2.20b. Deformation of each subset is not calculated directly, but rather assessed nondeterministically by scoring the gray value (Appendix D) of all pixels in the subset by means of a correlation function.

2.6.1 Correlation Function

The correlation function (C , Equation 2.12) is of paramount importance in understanding how speckle pattern deformations measured by DIC are correlated into fields of displacements [26].

$$C(x, y, u, v) = \sum_{i, j=-n/2}^{n/2} [I(x + i, y + j) - I^*(x + u + i, y + v + j)]^2 \quad (2.12)$$

Individual pixels are not tracked throughout each subsequent image (denoted as I for reference and I^* for deformed), as this would be computationally inefficient. Instead

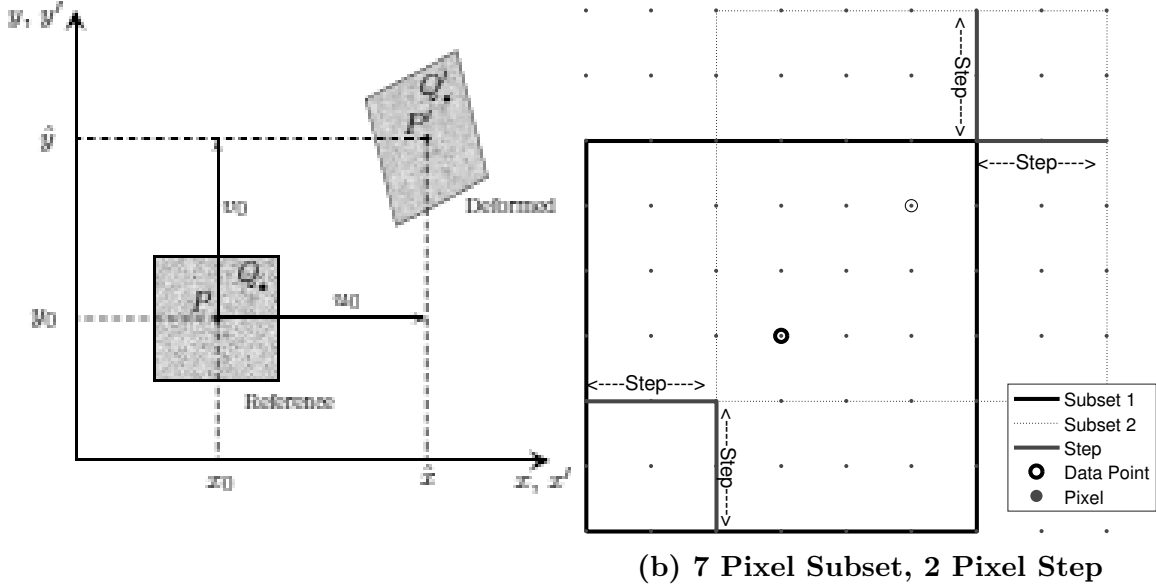


Figure 2.20. Digital Image Correlation Overview

the correlation function is used for each subset of the region of interest (distinguished by n), and a statistical *best match* is calculated via the lowest value of the correlation function throughout adjacent regions. This is analogous to minimization of squared deviations between two regions, followed by deformation (u, v) attributed to each pixel within the subset. This algorithm must be carried out for each subset within the region of interest (usually there are many) for each image that is captured, so computational time of this calculation increases. Further explanation will be given on the application of this tool in Section 2.6, with an example in Appendix D.

2.6.2 Raw Strain Calculation

DIC strains are calculated from the previously correlated displacements, in a method very similar to strain calculation from nodal displacements in FEM. Calculated displacements are reported for each subset, separated by the step size specified before the correlation. The displacement of each point in the grid of Figure 2.21a is



Figure 2.21. DIC Displacement Vectors [27]

assessed separately according to a local grid seen in Figure 2.21b. This region is then divided into four constant strain triangles (for the typical structured grid generated by subset and step definition) and strain in each triangular region is calculated from the deformation of the triangle using Equation 2.13 as seen in Figure 2.22.

$$\begin{aligned}
 \varepsilon = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} &= \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} \\
 &= [\mathbf{B}] \{\mathbf{d}\} \quad \text{where} \quad N_i = \frac{(x_j y_k - x_k y_j) + (y_j - y_k)x + (x_k - x_j)y}{2A}
 \end{aligned} \tag{2.13}$$

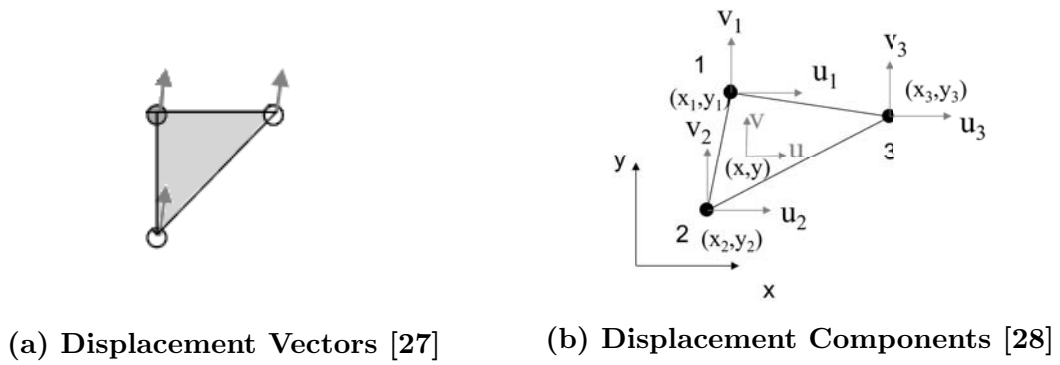


Figure 2.22. Strain Calculation Overview

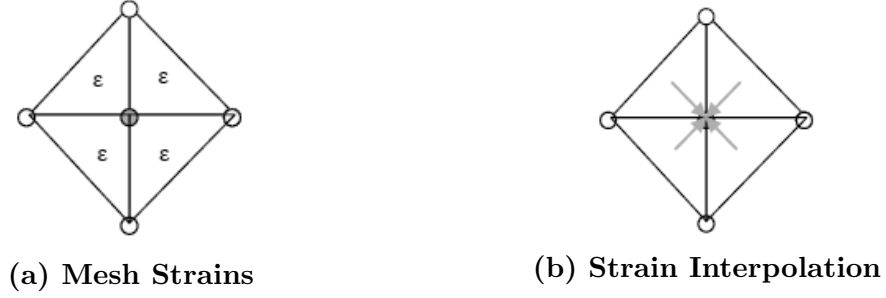


Figure 2.23. DIC Strain Vectors [27]

The displacements of the three vertices for each triangular region are used to calculate strain through the shape functions (N_i) evaluated at the coordinates of each point (x_i, y_i , reported in pixels). Each shape function is defined to allow only linear deformations (the lines all remain straight after deformation), and the spatial derivative matrix ($[B]$) calculates strain directly from displacements [19].

Once the strain of each triangular region is calculated, the raw strain reported at each point is interpolated from the neighboring triangular regions as seen in Figure 2.23 and resolved to the center point through a weight function based on the area of each of the adjacent triangles. The raw strains are rarely reported for scientific use, as proper use of the DIC generates very small mesh regions for strain calculation. A more useful grid of strain results can be made by filtering of the strain data from adjacent regions of raw strain to remove computational noise from the results of the correlation function. Strain is typically filtered over a region much smaller than the effective area of a strain gage. Data from a strain gages are reported point-wise, but are actually effected by the strain in the entire area of the gage dimensions, with higher weight to the strain measured in the center of the gage. A Gaussian filter is computationally similar to a gage's behavior.

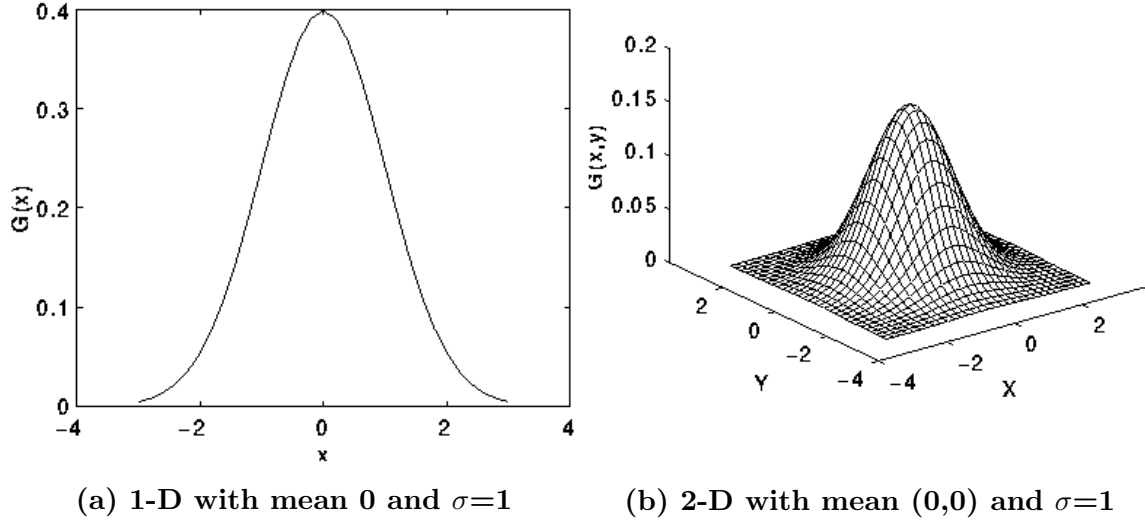


Figure 2.24. Gaussian Distribution [29]

2.6.3 Image Data Filtering

A Gaussian filter (G) is a numerical convolution operator based upon a normal distribution of data within the filter. For 1-D data sets, this is represented as a weight function (Equation 2.14) and is seen in Figure 2.24a.

$$G(x) = \frac{1}{\sqrt{2\pi}\sigma} e^{-\frac{x^2}{2\sigma^2}} \quad (2.14)$$

Data measured about the mean location ($x=0$) is given the highest weight, while data beyond three standard deviations (3σ) is essentially given no weight at all. Similarly, for a measurement made within a 2-D field, such as that of DIC, a Gaussian filter is an effective tool for reducing numerical noise that actually increases the accuracy of the data measured at the center of the filter. The 2-D Gaussian filter is based on Equation 2.15, where data is measured along a grid of coordinates in the X-Y plane, with a standard deviation of σ and a normal distribution as seen in Figure 2.24b.

$$G(x, y) = \frac{1}{2\pi\sigma^2} e^{-\frac{x^2+y^2}{2\sigma^2}} \quad (2.15)$$

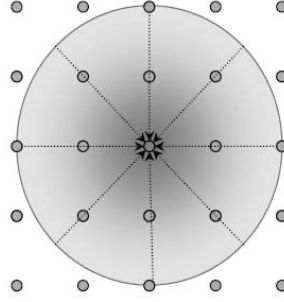


Figure 2.25. Strain Data Filtering [27]

Note that σ is standard deviation only in this section, otherwise σ refers to stress.

The utilization of a filter is such a crucial step in DIC strain approximations that it is required for correlation of any images. Other filter types are available, but the Gaussian filter is used for the purpose of this research. The total diameter of the filter is simply the product of the filter size and the step size defined before the correlation takes place (Figure 2.20b), and the separation of data points is the separation of adjacent subsets, as seen in Figure 2.25 [27]. This filter diameter is reported in units of pixels, but can be converted to geometric units if a scale is designated for the images.

2.6.4 Application of DIC

Holycross et al. was able to utilize DIC to measure strain energy in the regions near to notch roots for tension tests [9]. Fracture mechanics almost exclusively focuses on the energy of an existing crack, whereas mechanical fatigue is carried out up to (but not including) the point of failure, which is often defined as crack initiation. The work of Holycross analyzed the observed energy in the region known to fail due to a crack using DIC, and was able to predict the point in which energy was sufficient to form a crack, marrying two very well understood but disparate bodies of knowledge in structural analysis. While this is somewhat of a tangent to the work in this thesis, the same principles will be applied to the pre-straining of the Vibration-based Fatigue

Test specimens. The DIC measurements were shown to be representative of strain gage measurements on the opposite side of the plate, and the strain contours will be used to iteratively determine the most appropriate FEM solution in Chapter IV.

2.7 Root Mean Squared Error and Deviation

One mathematical tool used frequently in the evaluation of differences between measurement types is the RMSE, as shown in Equation 2.16 [30].

$$RMSE = \sqrt{\frac{\sum_{i=1}^n \left(\frac{\hat{y}_i - y_i}{\hat{y}_i} \right)^2}{(n - 1)}} \quad (2.16)$$

This equation is derived as the square root of the sum of all squared errors, normalized to the magnitude of the data being measured at that point. As opposed to strain gages which give a single-point measurement, DIC errors from FEM predictions cannot simply be added throughout the full field, and thus RMSE is the recommended figure of merit. For the purpose of this research, the predicted data point ($\hat{y}_i$) is typically strain (as predicted by FEM) and error is assessed as a deviation from that same data when measured experimentally (y_i), which would be from DIC. The benefit of this method is that the error of the entire strain field is represented with a single scalar value, but this also assumes even weight to all measured errors, which may be undesirable for certain applications. The RMSD (Equation 2.17), which is commonly known as the standard deviation, could also be calculated by neglecting the $\hat{y}_i$ in the denominator of the summation, and would report the error in the units of measurement of the inputs.

$$RMSD = \sqrt{\frac{\sum_{i=1}^n (\hat{y}_i - y_i)^2}{(n - 1)}} \quad (2.17)$$

2.8 Chapter Summary

This chapter has offered a comprehensive review of the theoretical background necessary to understand the work to be completed. A description of fatigue testing was explained for uniaxial states of stress, and correlated to the requirements set by ENSIP, which led into the utility of the Vibration-based Fatigue Test Method. The Pre-strain Method is used to generate constant residual stress fields, allowing capture of Goodman data for GTE component life approximations. Three-dimensional constitutive relationships were introduced to improve the ability of FEM to model plastic behavior of the specimen, followed by DIC and the ability to measure experimental strain fields. The RMSE and RMSD were the adopted equations to directly assess strain matching between experimental and computational results. Chapter III builds on everything that has been described in this chapter, and experimental methods will be covered in much more detail to ensure repeatability of the results shown in Chapter IV.

III. Research Methodology

This chapter builds on the background and theory discussed in Chapter II, discussing methods used to conduct research for this thesis. The predominance of this material can be broken into two categories: experimental methodologies and analysis procedures. In practice, these two distinct areas are utilized in an iterative fashion that builds upon prior results, but it is more intuitive to describe them separately in order of precedence and briefly describe how their results are compared and employed.

The chronological order of testing is to gather material properties, strain hardening parameters, and strain rate parameters (in this case γ and m in the Perzyna Model), to be used in the initial finite element model (FEM). This estimated response to a simulated loading is then carried out experimentally through the Pre-strain procedure to generate plastic deformation, with the resulting strains being measured via strain gages and digital image correlation (DIC). This data is then used to update the FEM to match the experimental material response, generating a reasonable approximation of residual stress. The specimen is then run through the Vibration Based Fatigue Test to generate failure under a given oscillatory load, combined with the previously determined residual stress to generate Goodman data. The purpose of this chapter is to outline the methods of the test procedure described above (graphically represented in Figure 3.1) by explaining all of the experimental and computational requirements necessary for generation of Goodman data via this test procedure.

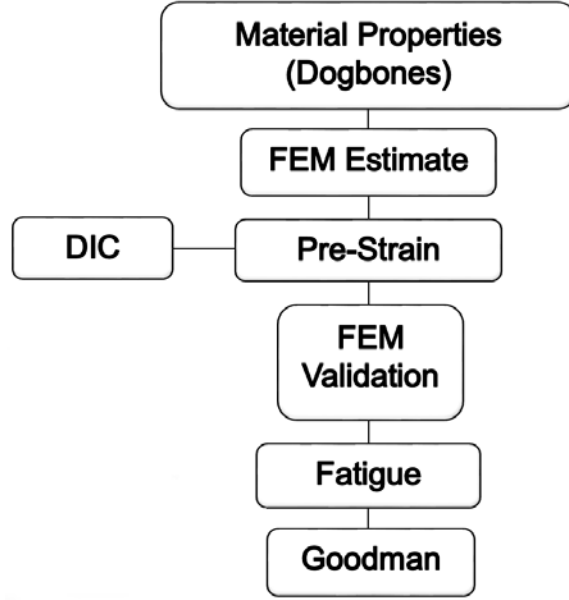


Figure 3.1. Chronological Test Procedure

3.1 Sample Preparation

It is well known that material properties can vary at different locations in stock material, so test specimens must be extracted from 24 in by 24 in stock aluminum (Al) 6061-T6 such that the test section of both specimen types are located as close as possible. Dogbones are extracted adjacent to the free end of the vibration plates and aligned such that failure directions of both specimen types are consistent, as seen in Figure 3.2, minimizing variation in material properties. Additionally, careful polishing of specimens is necessary, especially in the region of expected failure, to ensure that failure is not prematurely initiated by surface defects.

3.1.1 Speckle Pattern Application and Coordinate Transformation

The speckle pattern required for DIC is only necessary in the region of the plate that will have line of sight with the DIC cameras, as seen in Figure 3.3. It is important that the speckle pattern be random, with distinct regions of high and low contrast, with sparsity that is fully resolvable by the camera resolution. There are several

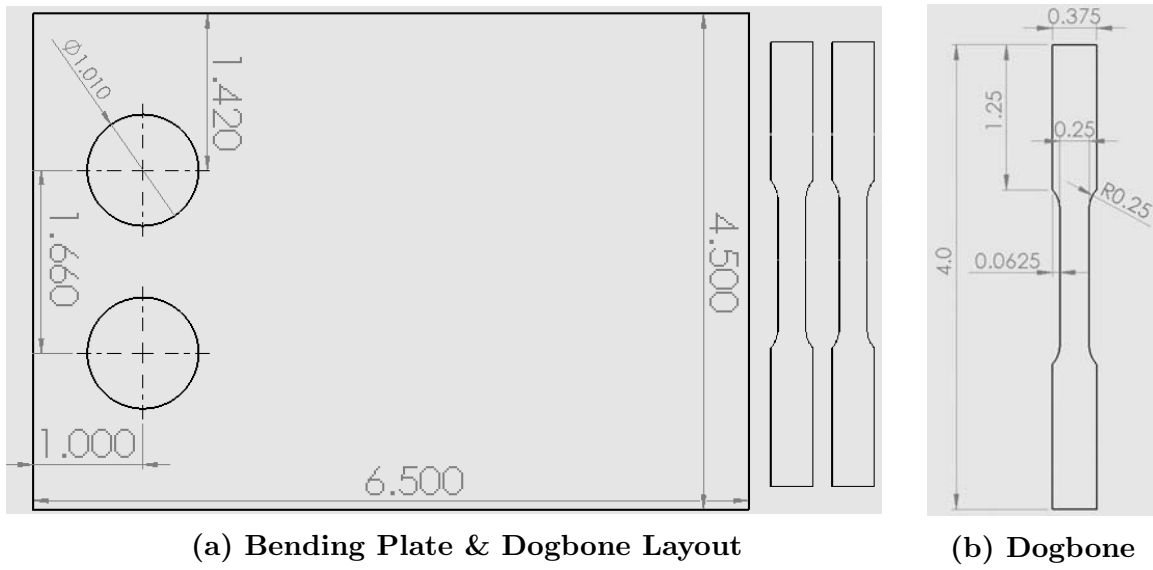


Figure 3.2. Specimen Geometry (in)

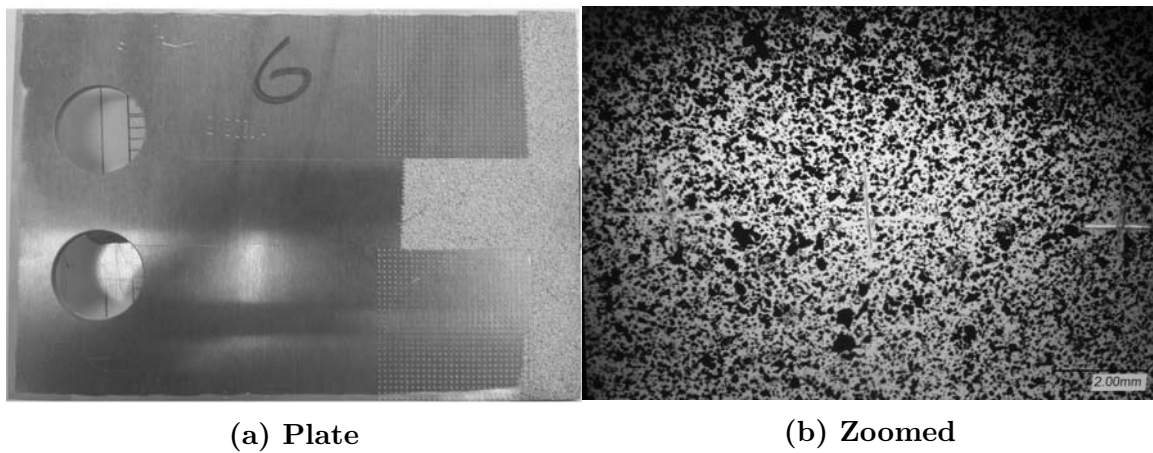


Figure 3.3. Speckle Pattern

methods of speckle application, ranging from highly repeatable processes such as laser printing or material etching, to rudimentary processes such as angled application of aerosol-propelled paints or “flicking” of a saturated paint brush. Iterative application of these methods for the purpose of this research has proven that careful use of aerosol-propelled paints produces a sufficient quality speckle with minimal gradients in speckle density or large speckles with respect to the rest of the field. There is no surface preparation for this type of speckle application. It is unknown if excessive paint along the grip surface will contribute to slipping under tensile load, so these regions were masked from speckle application.

Prior to testing, alignment features must be placed within the region of interest for DIC measurement to ensure measurements are taken in the proper coordinate system for orientation and scale. Small crosses are placed along the centerline of the plate, slightly offset from the free end as well as 2 in from the free end. These locations have been determined to be near to the edges of the measurable DIC field, and as such they will produce minimal error in strain measurement during testing, while still serving as measurable features for post-processing. The locations of these features are then measured with a Keyence Macroscope with respect to the free edge and lower edge, as seen in Figure 3.4b. The offset in these locations between the Keyence coordinate frame and the DIC coordinate frame is used as a translation for the DIC strain field in post processing, described further in Section 4.2.

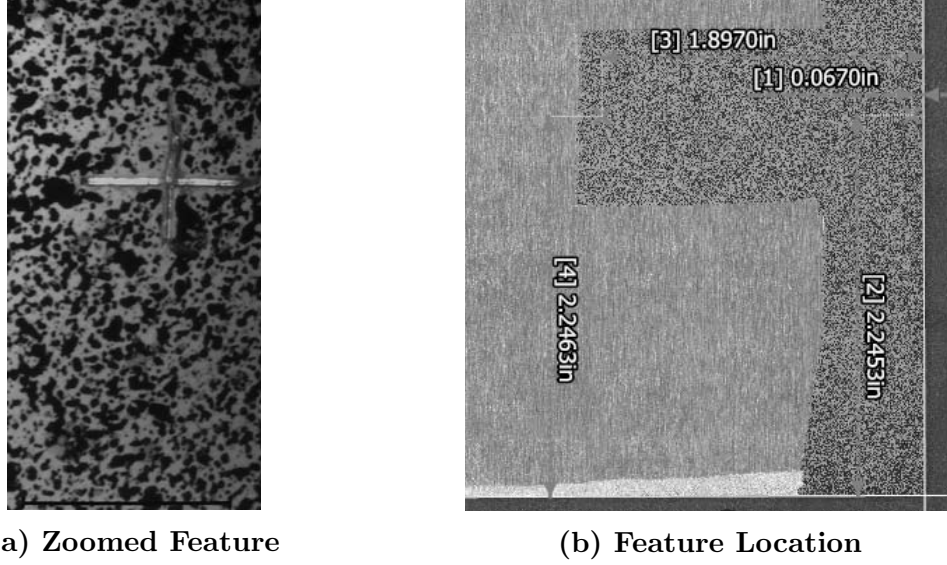


Figure 3.4. Coordinate System Alignment

3.2 Pre-strain Method

The Pre-strain Method, and especially the improved application and simulation of its results, is the focus of this research. Localized yielding of the vibration plate specimen is utilized to reach a desired level of plastic deformation. Upon unloading, the plastically deformed region is placed in a residual state of compression, driving residual tensile stresses in the regions adjacent to the yielded material. This new state of equilibrium is controlled by the preceding loading conditions (tensile load, grip locations, and loading area) during monotonic testing, and can effectively generate residual stresses in the fatigue zone of the specimen to enable vibration-based fatigue testing at various R [6].

A Mechanical Test Systems (MTS) 312.21 (22,500 lb_f load cell capacity) uniaxial load frame is used to apply a monotonic tensile load to the vibration-based fatigue specimen as seen in Figure 3.5. The machine is equipped with hydraulic grips with diamond-faced wedges (1.75 in by 1.75 in effective gripping area), supplying compressive forces up to 8 ksi. The hydraulic grip pressure is set to 1,600 psi to eliminate

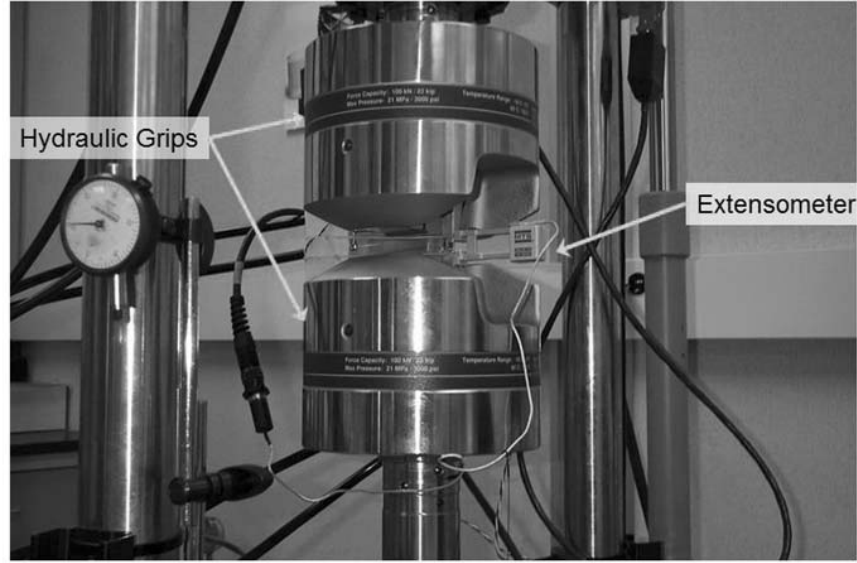


Figure 3.5. Prestrain Experimental Setup [8]

the possibility of specimen slippage, and the load is increased according to the load control curve seen in Figure 3.9b using Multi-purpose Testware (MPT). It is not until the anticipated yield loading has been exceeded that the plastic deformation of the Pre-strain Method can be controlled, at which time the ramp rate is decreased as the local maximum load approaches the ultimate tensile stress. A desired level of strain is predicted using FEM, and that strain (measured by strain gages or DIC) drives the limits of testing, after which the specimen is unloaded and prepared for vibration testing [6, 8, 17].

3.2.1 Specimen Alignment

Symmetric placement of the specimen in the load frame is of significant importance in the Pre-strain Method. Most American Society for Testing and Materials (ASTM) test standards focused on monotonic loading, especially for application of plastic deformation, would be performed with specimens constructed such that the region to be exposed to the highest stresses would be sufficiently far from the location of grip application, as seen in Figure 2.3b. However, the Pre-strain Method is tied to



(a) Plate Alignment Tool

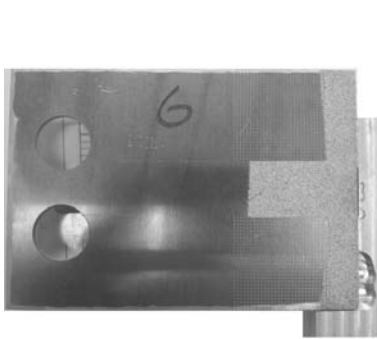


(b) Grip Spacing Tool

Figure 3.6. Plate/Grip Placement Tooling

a geometry optimized for the Vibration-based Fatigue test (See Figure 3.16a), which increases the importance of the grips being applied symmetrically along the plate centerline. For this reason, the tooling seen in Figure 3.6 was developed to align the plate to the grip as initial grip pressure is applied. The plate alignment tool has 90° notches machined to keep the 1.75 in by 1.75 in grip region properly spaced from the plate edges when the top grip is applied. The procedure for aligning the plate to the top grip with the alignment tool is seen in Figure 3.7.

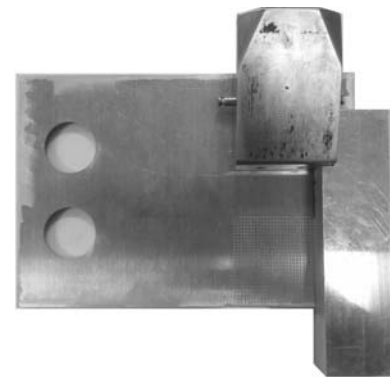
Due to grip surface geometry and fillet sizes, a final spacing between the two grips of 0.730 in is necessary to ensure the 1 in ungripped portion of the plate is centered along the geometric centerline of the plate. Once the top grip is properly aligned and initial grip pressure is applied, the lower grip is raised until a small compressive force is placed on the spacing tool, as seen in Figure 3.8. It is important to note that



(a) Plate Notch



(b) Grip Notch



(c) Aligned Grip/Plate

Figure 3.7. Initial Plate/Grip Alignment

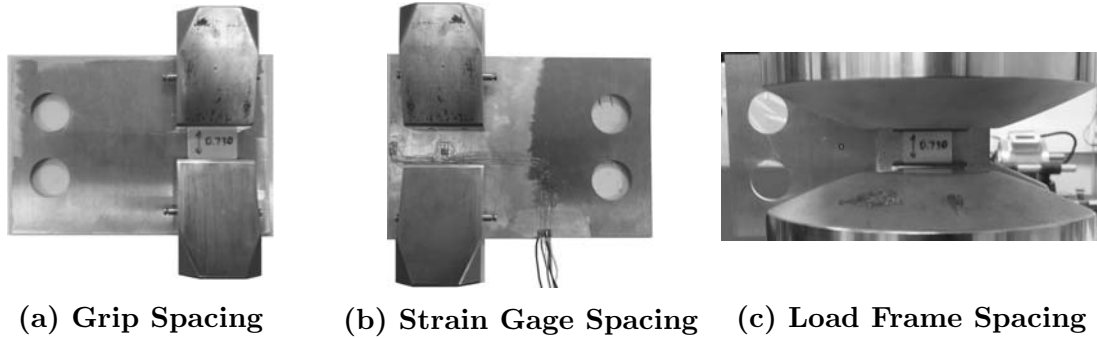


Figure 3.8. Final Plate/Grip Spacing

placement of strain gages, as well as terminals and connecting wires, must be made such that the grips will not interfere, as seen in Figure 3.8b. For this reason, it is best to measure the projected grip region prior to application of strain gages and DIC speckle pattern.

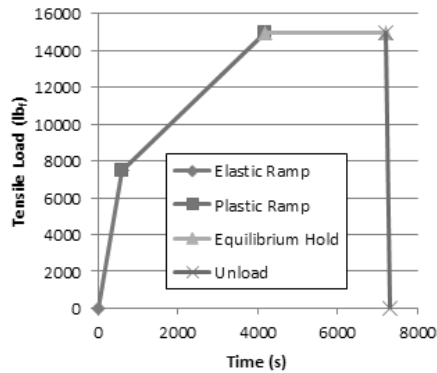
3.2.2 Multi-Purpose Testware Procedure

The experimental control for the MTS load from is programmed using MPT. Timed acquisition of load (100 kN transducer, model 661.20E-03), displacement (linear variable displacement transducer (LVDT)), and strain (two Micro-Measurements CEA-06-062UW-350 uniaxial strain gages) is recorded to a data file every second. For the entire duration of the test, interlocks are in place to limit the load to 18,000 lb_f of tension or 500 lb_f of compression, as seen in Figure 3.9a, beyond which the test procedure is canceled. Similarly, displacements greater than 0.5 in will end the test procedure to protect the testing equipment.

The specimen should be under no load prior to the start of the test procedure, however a ramp to 0 lb_f takes place over a ten sec period, with a subsequent hold at 0 lb_f for twenty seconds. Ramp 1 (so named in Figure 3.10) approaches 7,500 lb_f over the course of 600 seconds. This portion of loading has been experimentally and computationally verified to exhibit no plastic deformation (proven later in Chapter



(a) Experimental Limits



(b) Load Control Curve

Figure 3.9. MTS Control Inputs

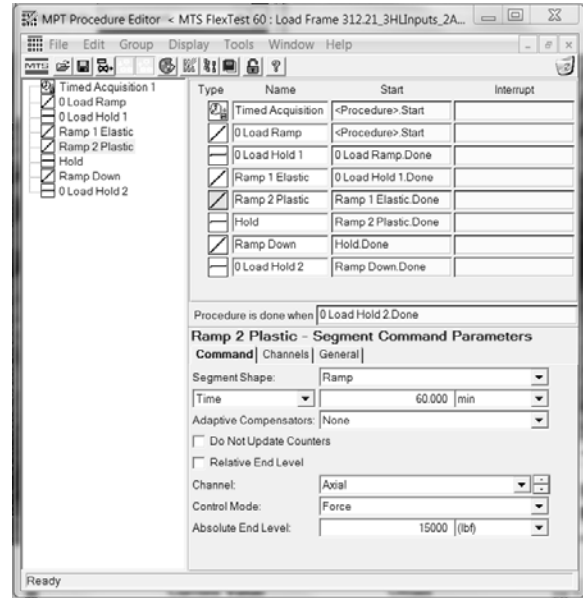


Figure 3.10. MTS Test Procedure

IV, Figure 4.21), so the rate of loading is high. Ramp 2 provides the bulk of plastic deformation of the specimen, approaching 15,000 lb_f in a one-hour period, which is $\frac{1}{6}$ th the rate of loading in the elastic region.

The specimen has been proven experimentally and computationally (see Chapter IV, Figures 4.21 and 4.27) to be in a transient state of plastic deformation immediately following the second ramp. A hold step at maximum load is programmed for a duration of 50 minutes, for a total loaded duration of two hours, allowing the specimen to achieve equilibrium. The unloading takes place rapidly over a period of 100 seconds because the material response is fully elastic (thus no rate-dependent contribution to residual stress), and ends with a subsequent 100 sec hold step at 0 lb_f. The entire load control curve is seen in Figure 3.9b. Programming the loading in this way allows for the experimental gathering of strain data via DIC using a repeatable procedure, and also allows modeling in FEM with data reported at any point along the ramp or

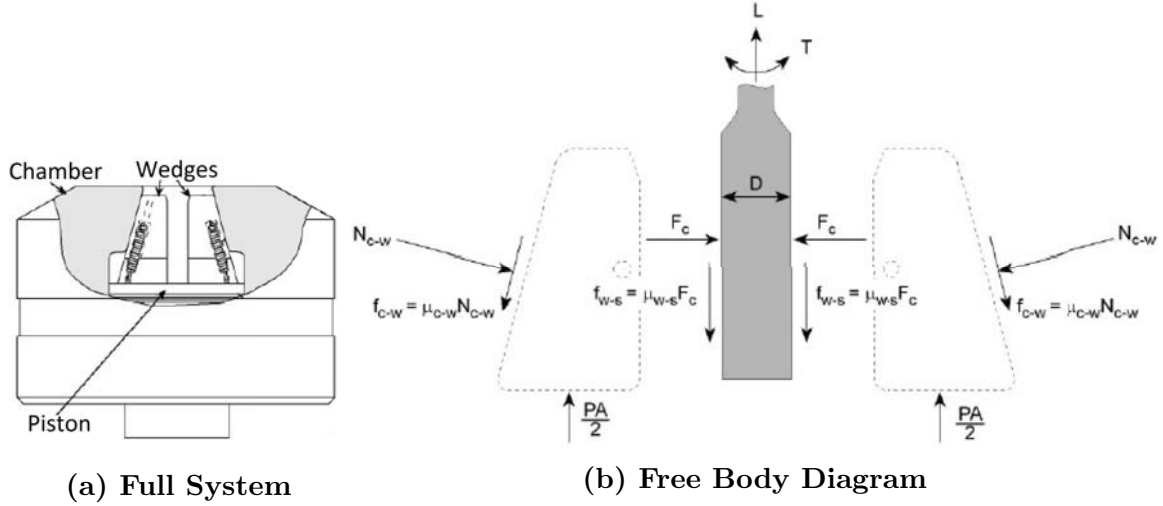


Figure 3.11. Grip System [31]

hold phases.

3.2.3 Compressive Forces

The compressive load on the specimen is derived from the equilibrium of all forces placed on the grip wedges due to the grip chamber, friction, and specimen, as seen in Figure 3.11a and Equation 3.1.

$$F_c = \frac{PA}{2} \left(\frac{\cos 15^\circ - \mu_{c,w} \sin 15^\circ}{\sin 15^\circ + \mu_{c,w} \cos 15^\circ} \right) \approx \frac{PA}{2} (3) \quad (3.1)$$

Hydraulic pressure is supplied to the grip housings, which is transferred to the wedges via a grip piston with an area of 10.30 in^2 [31]. The surface of the grip that is in contact with the grip chamber is at an angle of 15° from the clamping surface, such that the normal force between the grip chamber and the wedge ($N_{c,w}$) keeps the wedge in static equilibrium with the force supplied by the grip piston ($\frac{PA}{2}$) and the reaction force of the specimen (F_c), as seen in Figure 3.11b. The manufacturer of the MTS 647.10A hydraulic grips recommends a coefficient of static friction ($\mu_{c,w}$) of 0.06 for lubricated steel on steel, resulting in a final compressive force of $F_c = 1.5PA$ [31].

3.3 Axial Dogbone Testing

An important, additional step to the original Pre-strain process [4] is the gathering of empirical hardening data [8]. Strain hardening material properties are determined by pulling dogbone specimens monotonically until separation. Like the Pre-strain Method, monotonic tensile testing of dogbones is conducted with the MTS axial load frame [32]. Along with the 22,500 lb_f load cell mentioned earlier, the MTS load frame is also equipped with an LVDT for displacement control and an extensometer for strain measurement [32]. The displacement rate and the specimen geometry for monotonic testing of dogbones is based on recommendations of ASTM standard E 8 [7]. The displacement rate for each dogbone tensile test is 0.2 inches per hour ($5.6 \times 10^{-5} \text{ s}^{-1}$) for a 1.25 inch long test section specimen.

3.4 Digital Image Correlation

Validation of the FEM models in this research is done so by DIC. While the underlying calculations behind the post-processing of DIC data (Explained in Chapter II) are very different from traditional strain gages, the resulting data is nearly identical. Both DIC and strain gages measure total mechanical strain (regardless of elastic or plastic strains), but the region measured by DIC can be much larger, offering strain field validations as opposed to single-point measurements made by strain gages. The region measured for the purpose of this research is limited to the region between the grips, and based on the focal capabilities of the 11 megapixel Prosilica GE4000 camera [33], this allows for measurement of about 2 inch from the free end of the plate, as seen in Figure 3.12a. This field allows for assessment of strain 1.3 inch from the free end (consistent with the Gage 2 location of maximum ε_{yy} in Figure 3.16a), and the comparison of results seen in Figure 3.12b shows favorable agreement.

DIC is an optical measurement technique, and as such required properly adjusted

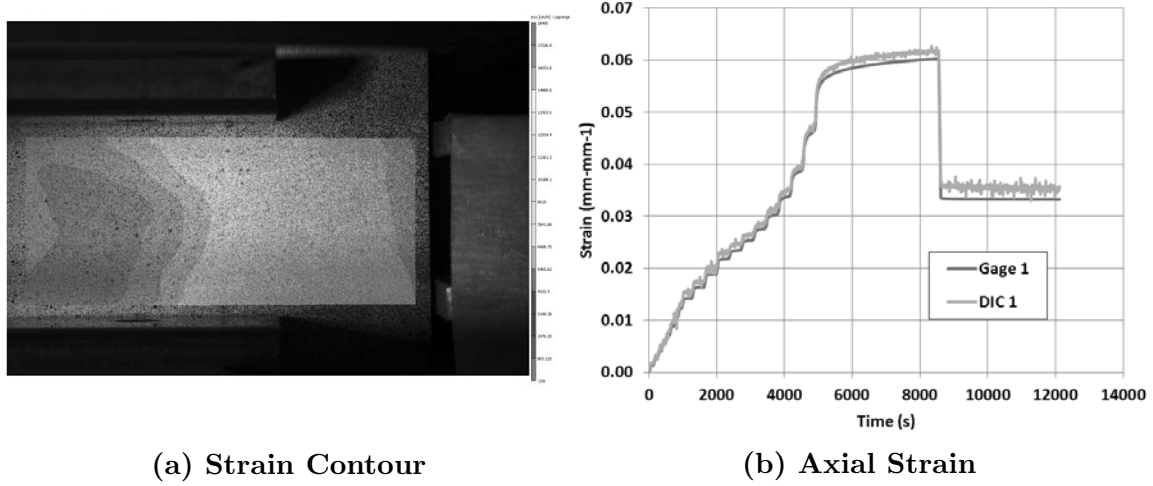


Figure 3.12. Digital Image Correlation Examples

cameras, lenses, and light sources. To ensure the maximum area of the specimen is within the focal plane of the camera, the focal length must be set with the aperture fully open (and thus minimum lighting), which minimizes the depth of field as seen in Figure 3.13b. Once optimal focus is obtained, the aperture is minimized to expand the depth of field, requiring a high powered light source to resolve the specimen's speckle pattern. For this test setup, two high intensity LED light fixtures (with a three-light array on each fixture) are necessary to ensure even lighting of the region of interest.

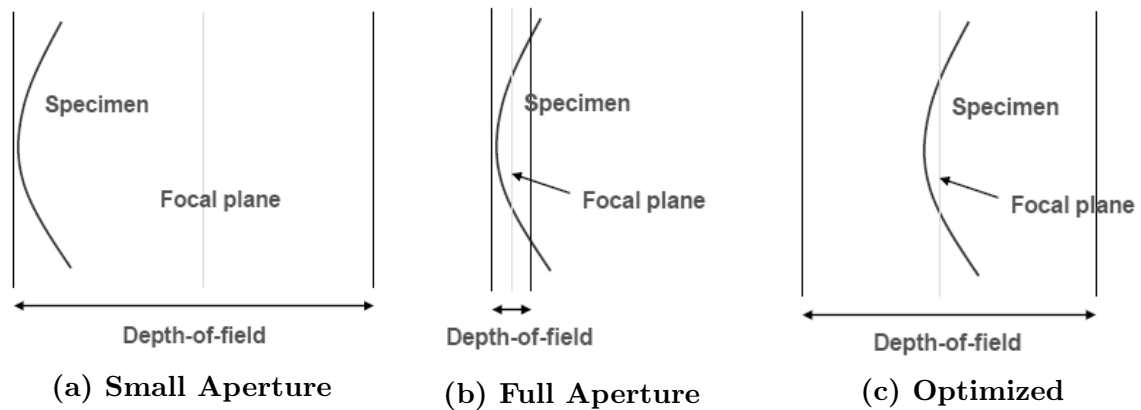


Figure 3.13. DIC Focusing Procedure [34]

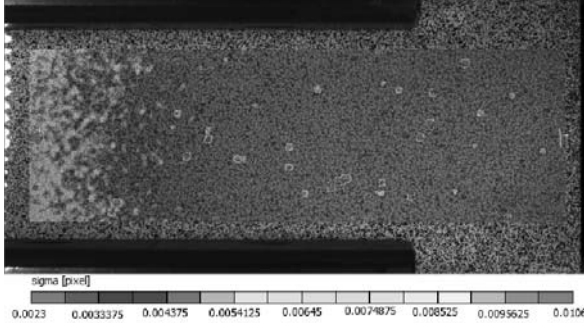


Figure 3.14. DIC Error Estimate

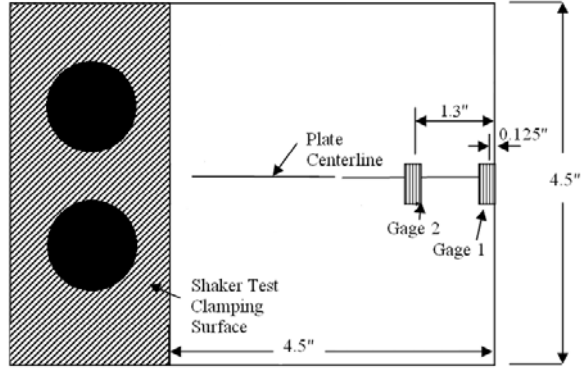


Figure 3.15. DIC Spatial Calibration

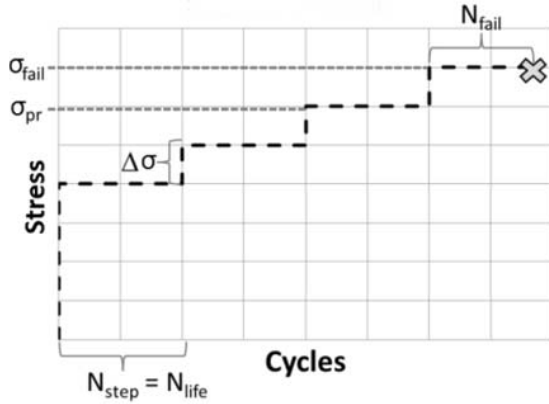
The error estimate is a good indicator of the lighting quality, and should be kept below 0.03 pixels (<0.01 , as seen in Figure 3.14). The Vic-Snap software by Correlated Solutions is used for image capture, and affords the capability of recording analog data with each image captured. The load, displacement, and strains are read directly from the MPT computer. Prior to testing, several reference images are captured for future correlation, as well as offering a calibration scale for coordinates and displacements, as seen in Figure 3.15. During testing, the software is set to capture an image every 100 lb_f, or every 60 seconds, which is reset at every load increment. Based on the load control ramps and hold step, this guarantees 150 images for the load ramps and 50 images for the hold steps. Each of these images will have an associated time, load, displacement, and strain (as measured from gages) to characterize the state of the specimen.

3.5 Vibration-based Fatigue Test Method

Fatigue testing of pre-strained plates is conducted according to the Vibration-based Fatigue Test [4, 5, 6], involving a sinusoidal excitation of a cantilevered specimen by an electrodynamic shaker. A resonant mode is excited such that a uniaxial bending load exists within the specimen, which varies based upon the material composition or specimen geometry [35, 36]. The particular geometry being studied has a square



(a) Fatigue Plate Instrumentation [5]



(b) Step Test Diagram [17]

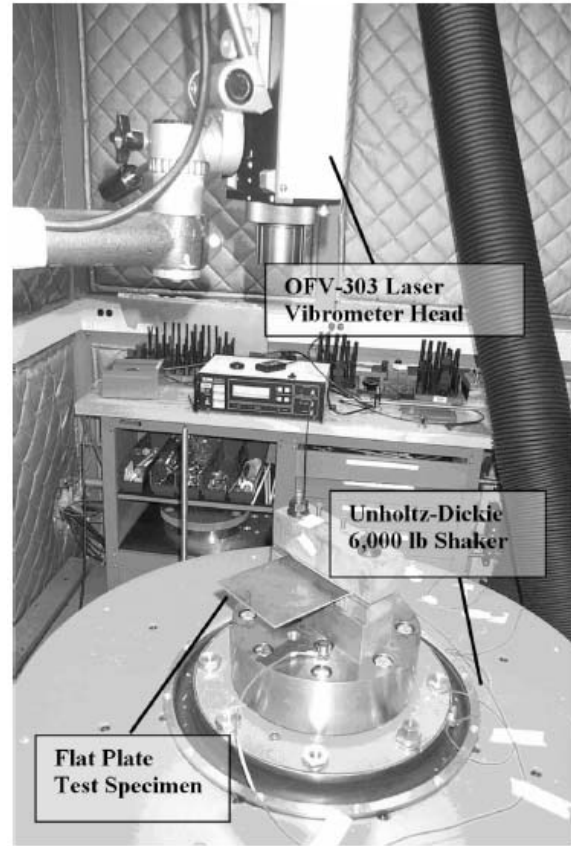


Figure 3.17. Fatigue Test Setup [5]

Figure 3.16. Vibration Based Fatigue Test

unclamped region with length of 4.5 inch and a thickness of 0.125 inches, as seen in Figure 3.16a, composed of Al 6061 T-6. A chord-wise bending (or two-stripe) mode is present at frequencies near 1600 Hz, and is the optimal mode for maximizing the stress in the free end of that specimen.

The measurement of strain (direct correlation to stress via Hookean elasticity) is performed by means of a strain gage, placed at the region of highest predicted stress during fatigue testing (gage 1 in Figure 3.16a). The acceleration of the 6000 lb_f Unholtz-Dickie electrodynamic shaker head (thus the acceleration imparted on the specimen) is controlled by an Endevco 2271A accelerometer, and the velocity of the specimen (approximated strain control explained in referenced literature [32]) is

measured by a non-contact Polytec OFV 303 single-point laser vibrometer positioned near a nodal line of the vibrating specimen to reduce the likelihood of measurement saturation [4].

3.5.1 Failure Determination and Approximation

Failure in operation of gas turbine engines (GTEs) may be catastrophic due to high loading conditions, and thus is easy to determine. This is not the case for the Vibration-based Fatigue Test, so a more distinct and measurable procedure was developed. By imparting a crack in the specimen, the geometry changes (equivalent to stiffness in a dynamic loading scenario), driving changes in both the resonance frequency of a given mode and the damping of the dynamic system. The test is controlled to follow the two-stripe bending mode at a constant amplitude of strain, and driving frequency changes of more than 0.1% have been correlated to crack lengths as small as 2-3 mm [6, 37]. This change in frequency, accompanied by an increase in the required force (product of damping change) has been established as the specimen failure criteria.

Fatigue testing to failure can be conducted two ways: sinusoidal amplitude dwell or the step test procedure. Sinusoidal amplitude dwell, referred to as dwell in this research, is conducted by controlling the strain amplitude, via the velocity amplitude, at a constant value throughout the duration of the test until failure occurs [38]. The step test procedure, illustrated in Figure 3.16b, is a series of dwell steps for a specified number of cycles (N_{step}); each step has consecutively increasing sinusoidal dwell amplitudes until failure occurs. The failure stress amplitude is calculated by interpolating between the failure and previous steps [1, 13, 38]. The interpolation is

represented by Equation 3.2, where

$$\sigma_a = \sigma_{pr} + \frac{N_{fail}}{N_{step}} (\sigma_{fail} - \sigma_{pr}) \quad (3.2)$$

σ_a is the failure stress amplitude, σ_{pr} is the stress amplitude during the previous un-failed step, σ_{fail} is the stress amplitude during the failed step, and N_{fail} is the accumulated cycles to failure during the last step [38]. Therefore, σ_a represents the failure stress amplitude of N_{step} cycles to failure.

3.6 Chapter Summary

In this chapter, an extensive description was given for the various experimental and computational methods used in this research. The methods were first briefly explained chronologically, and later in full depth based upon the order of precedence for the proper utilization of the test method. The preparation of samples for the Pre-strain Method and DIC was first described, followed by the experimental procedure for carrying out the Pre-strain Method in a repeatable and properly aligned test. The gathering of material data via dogbones was introduced to supplement the Pre-strain Method, followed by the use of DIC to gather field data to correlate between FEM and experimental results. Finally, the Vibration-based Fatigue Test Method was described, as well as the designated failure criteria for Goodman data acquisition. The following chapter will now explain the results gathered using all of these methods.

IV. Results

This chapter puts to use the theoretical background of Chapter II via the experimental methodology in Chapter III. The geometry of the dogbone specimens is reported, followed by a description of the MATLAB script developed to convert raw monotonic load and strain data to a direct input script of material properties in ANSYS. Similiar geometric information is reported on plates, with a description of how the finite element model (FEM) mesh and boundary conditions are developed through experimental syncing of the Multi-purpose Testware (MPT) setup and digital image correlation (DIC) acquisition, as well as converging the model to more consistent strain predictions. Following this, extensive reporting of stress and strain data is explained in both experimental and computational results, with qualitative and quantitative comparisons throughout the entire load history of the specimen. Finally, the residual stresses predicted at the region of high cycle fatigue (HCF) failure are reported for application as steady stress on a Goodman Diagram.

4.1 Specimen Features

4.1.1 Dogbones

Dogbones were cut with a waterjet from the stock material in locations adjacent to the failure location of the plate specimens, as show in Figure 3.2. The total length and grip section width are important in ensuring failure occurs in the gage section, but are of no utility for post processing. The width and thickness of the gage section is used for calculations of stress from the monotonic testing. The cross section thickness and width are reported in Table 4.1, as measured by a micrometer and calipers, respectively. The specimen numbering convention identifies the associated plate with the first number, and the dogbone specimens are separated by the second

Table 4.1. Dogbone Measurements

Specimen	X-Section Width	X-Section Thickness
1-1	0.251 in	0.122 in
1-2	0.255 in	0.123 in
2-1	0.251 in	0.122 in
2-2	0.252 in	0.122 in
3-1	0.249 in	0.122 in
3-2	0.252 in	0.122 in

number (see Figure 3.2). These measurements were used in the MATLAB script `Data_Extractor_Dogbone.m` (See Appendix B.1.3) to input material properties into the FEM.

4.1.1.1 Dogbone Data Processing

`Data_Extractor_Dogbone.m` (See Appendix B.1.3), was developed to automate the use of experimental stress-strain data in ANSYS. The output data file (`name.dat`) from the Mechanical Test Systems (MTS) testing reports strains from the extensometer and forces from the load cell, which are converted to engineering stresses based on the measurement of dogbone cross-sectional areas prior to testing. Proper assessment of plastic response requires conversion of this engineering data to the true stress-strain curve. This does not account for data gathered after the onset of necking, but the plate specimen will not experience such loading conditions, so the final data near the point of failure can be neglected. Experimental noise must also be removed so that the FEM solver of ANSYS can properly predict material response. In this portion of the code, the data set is decimated to only 5000 points of true stress vs. true strain, which allows the Gaussian smoothing algorithm (see Equation 2.14 and Figure 2.24a) to solve faster. Once the smoothing algorithm is complete, the data set is further decimated to 100 points of smoothed true stress and true strain, as this is the maximum number of stress-strain data points that can be utilized for multilinear

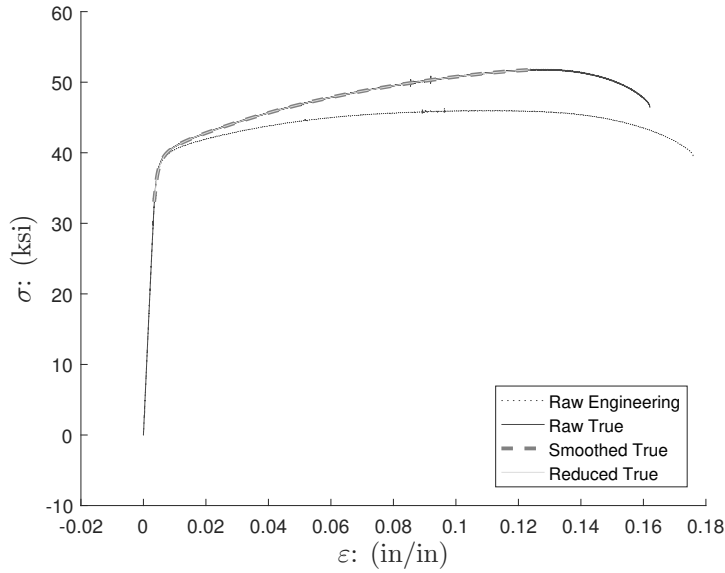


Figure 4.1. Dogbone Stress vs. Strain

isotropic hardening material properties in ANSYS. The output of these results is seen in Figure 4.1.

In addition to a smoothed data set of no more than 100 data points, ANSYS also has requirements on the initial elastic modulus and tangent modulus in order to properly converge results in FEM analyses. Even after the Gaussian smoothing algorithm has completed, small magnitudes of noise in strain measured by the extensometer causes very large shifts in tangent modulus, and as such, much of the elastic response is ignored. Instead, data is utilized just before the material response deviates from a linear response, in which case the quickly decreasing tangent modulus is sufficient to accommodate for this noise in subsequent data points. The very first point of stress and strain in this curve must then be used to ensure the elastic modulus is consistent with the material response curve. Finally, the last data point utilized must be slightly less than the ultimate tensile stress (σ_{UTS}), and the entire portion of utilized true stress-strain data must have a monotonically-decreasing tangent modulus in order to be utilized in ANSYS. Once all conditions are ensured (as seen in Figure

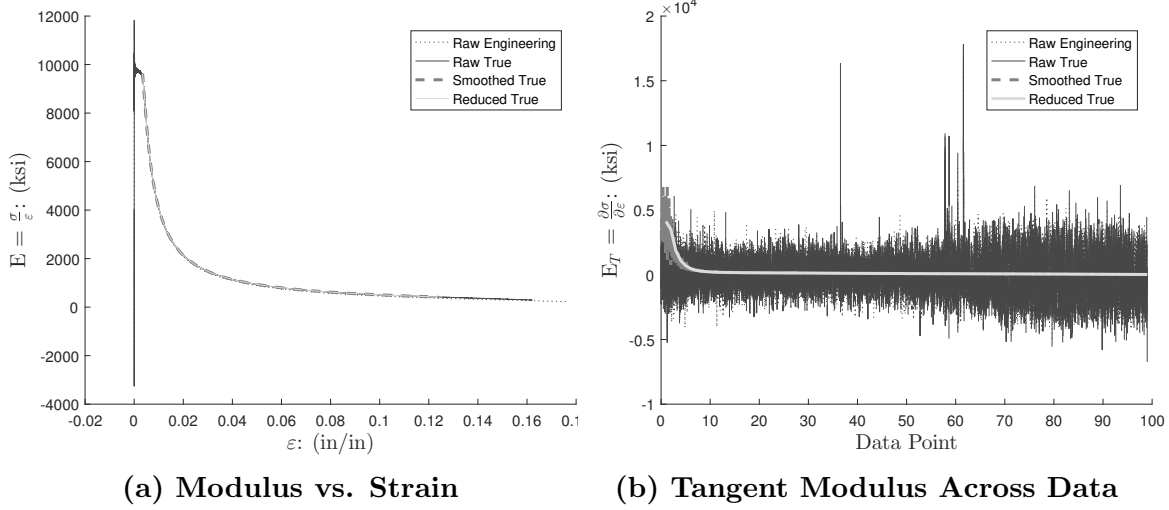


Figure 4.2. Dogbone Data Processing

4.2a), the 100 point data set is output to a text file (`Script_name.txt`) that can be used as a direct input to ANSYS.

4.1.2 Plates

The plate specimens were also cut with a waterjet from the stock material. The thickness was measured with a micrometer, the width was measured with calipers, and the length was measured with a linear scale, as reported in Table 4.2. The alignment features placed on the specimens were measured with a Keyence Macroscope, as seen in Figure 3.4b. The plate geometry was used in the ANSYS script in Appendix A to generate a geometrically accurate FEM model, while the alignment features were used in MATLAB scripts similar to `DIC_Max.m` to align the DIC data to the FEM geometry.

Table 4.2. Plate Measurements

Specimen (Results: in)	Geometry			DIC Feature 1		DIC Feature 2	
	Length	Width	Thickness	X ₁	Y ₁	X ₂	Y ₂
1	6.49	4.501	0.122	0.0687	2.2454	1.9154	2.2363
2	6.50	4.495	0.122	0.0681	2.2278	1.9093	2.2472
3	6.50	4.501	0.122	0.0670	2.2453	1.8970	2.2463

4.1.3 FEM Geometric Modeling

The test article was initially modeled using half-plane symmetry and meshed with 55X35X4, eight-noded brick elements (SOLID-185, as seen in Figure 2.13c): 55 elements across the 6.5 in span; 35 elements across the 2.25 in half-chord; and four elements through the 0.125 in thickness. The volume between the grips was meshed with the same element geometry, with an area of 1.75 in by 1.75 in at a distance of 0.59 in from the free end. Once the convergence study was performed that increased mesh densities, an integer multiple of any of the 55X35X4 elemental dimensions was specified.

4.1.3.1 FEM Convergence Study

A convergence study was performed by iteratively increasing mesh density along each dimension while analyzing maximum strain along the plate center line (gage 2 in Figure 3.16a). Using this method, it is expected that a point of minimal change (<1%) in FEM strain response to loading will be met, thus proving that the model has a sufficient density to converge to a real solution. It can be seen in Table 4.3 that increasing mesh density in the direction of the specimen length causes the strain to rise at a constant load. This shows not only that the model was not properly converged with the 55x35x4 elemental divisions, but that the original course mesh was stiffer than the converged model, which is common in FEM analysis. The change between a length multiplier of 5 and 6 (i.e. 275 elements along the length versus 330) showed a change of only 0.58% in strain, so higher densities would only increase computation time with little added fidelity.

In a similar analysis with the model returned to 55 elemental divisions along the length, and increasing density along the direction of the plate width. It can be seen in Table 4.4 that changes in mesh density in this manner produced negligible changes

Table 4.3. Lengthwise Convergence of ε_{yy}

Mult	Node	Strain 2 ($\mu\varepsilon$)	Deviation from Cvg
1	8,707	17,669	-11.91%
2	17,233	18,942	-5.56%
3	25,759	19,425	-3.15%
4	34,285	19,770	-1.43%
5	42,811	19,940	-0.58%
6	51,337	20,057	0.00%

in predicted strains. On the most basic level, this means that increasing mesh density in the direction of the plate width beyond 35 elemental divisions would only increase computation time, but would add little fidelity to the results obtained. The same could be shown for changes in elemental divisions along the plate thickness. A final study, shown in Table 4.5 was carried out to verify that there were no paired effects between increasing density along the length and width. The converged model must at least have elemental dimensions of 330x35x4 along the length, width, and thickness, respectively. A plot of all results is given in Figure 4.3.

If this study could be carried out at multiple locations to analyze strains in multiple directions throughout the entire specimen, it is likely the 330X35X4 model would still not be fully converged in some locations. This is a result of elemental geometries that have shapes deviating very far from a cubic structure. Increased stiffness predictions in a given direction can be overestimated due to this type of elemental geometry,

Table 4.4. Widthwise Convergence of ε_{yy}

Mult	Node	Strain 2 ($\mu\varepsilon$)	Deviation from Cvg
1	8,707	17,668.884	-0.05%
2	17,152	17,669.547	-0.04%
3	25,597	17,675.674	-0.01%
4	34,042	17,676.134	-0.01%
5	42,487	17,675.607	-0.01%
6	50,932	17,677.255	0.00%

Table 4.5. Convergence of ε_{yy} both Length and Width

Mult	Node	Strain 2 ($\mu\varepsilon$)	Deviation from Cvg
1	8,707	17,669	-12.17%
2	33,948	18,941	-5.85%
3	75,729	19,438	-3.38%
4	134,050	19,813	-1.52%
5	208,911	19,995	-0.61%
6	300,312	20,118	0.00%

and should be avoided. As such, the accepted model for the remainder of this study had 330X105X6 elemental divisions between the length, width, and thickness, for an elemental geometry of 0.019 in by 0.0214 in by 0.0203 in. The full plate model then consists of 207,900 8-noded brick elements, with a total of 245,602 nodes.

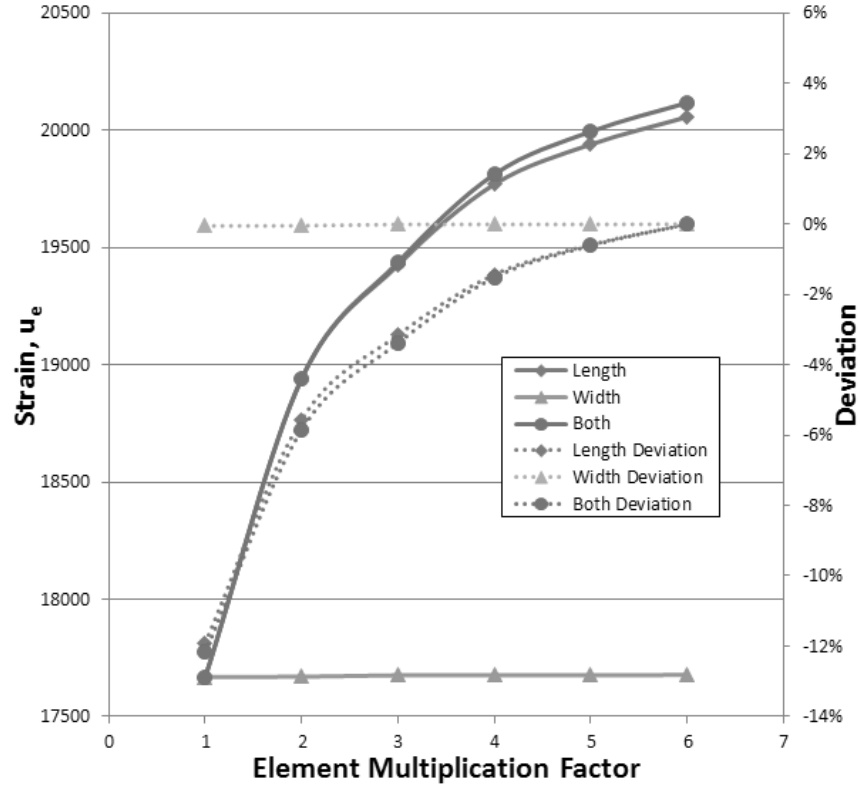


Figure 4.3. Convergence Study Results

4.1.4 FEM Load Boundary Conditions

4.1.4.1 Compressive Loads

The application of compressive forces for the FEM account for the total compressive force (F_c) explained in Section 3.2.3, which can be carried out numerous ways with varying degrees of accuracy with respect to the experimental loading conditions. The simplest of these methods is to assume an even pressure distribution along the specimen surface directly in contact with the grips, which has shown favorable results when comparing with experimental results [6, 8, 17]. An alternative method, which should produce the most realistic pressure distributions, would be to model the entire system, consisting of the aluminum (Al) specimen, the steel grip wedges, and the steel chamber. Modeling the system in this manner requires assessment of contact surfaces in the FEM and localized specimen deformation due to localized forces upon impact of each of the diamond features on the wedge surface, which in turn requires three dimensional scanning of all four wedges. A multitude of assumptions are necessary with contact element thicknesses and displacement tolerances that are very challenging to experimentally validate, which was not feasible within the scope of this research. As an intermediate step between these two methodologies, the grip wedges were modeled with the specimen without the diamond surfaces.

Initial modeling of the grips in contact with the specimen produced unfavorable results, as seen in Figure 4.4b. The FEM model seen in Figure 4.4a imparts loads correctly, however the discontinuous change in geometry and material stiffness along the grip edges are modeled as large changes in load over an infinitesimally small region, driving the localized stresses. At just one tenth the desired compressive loading, the plate was already predicted to be plastically deforming in these regions, which is physically impossible.

To alleviate these regions of localized compressive stress, the grip wedge was in-

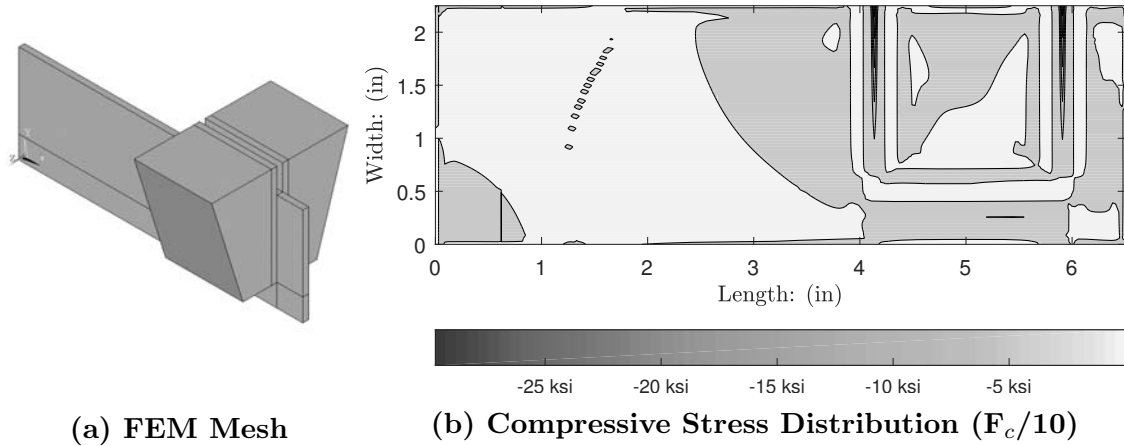


Figure 4.4. Gripped Plate Model

stead modeled separate from the plate, with a zero-displacement boundary condition placed along the surface that would later be in contact with the specimen. The results of the FEM analysis were written to text files and later read into MATLAB via `Data_Extractor_Grip.m` (Appendix B.1.2), and the nodal reaction forces were then applied to the plate FEM model through an input script generated with `Forces_Compression.m` (Appendix B.2). In order for this method to work, the FEM mesh of the grip must correspond perfectly to the mesh of the plate, and the node numbering of both mesh systems must be such that the final plate model is still in a state of equilibrium.

This method extracts reaction forces from 7,462 nodes in the grip model and applies them to 14,924 nodes in the plate model, with opposite magnitude for half of the nodes to account for both applied grips in the experimental setup. Previously, the even pressure distribution accounted for the hydraulic supply pressure of 1,600 psi for a total compressive force of 24,720 lb_f (Equation 3.1), for a total of 3.313 lb_f applied to each node. The improved model produced a much more realistic pressure distribution, with declining magnitudes toward the lower edge of the grip, with a root mean squared error (RMSE) of 19% when compared to the even force distribution.

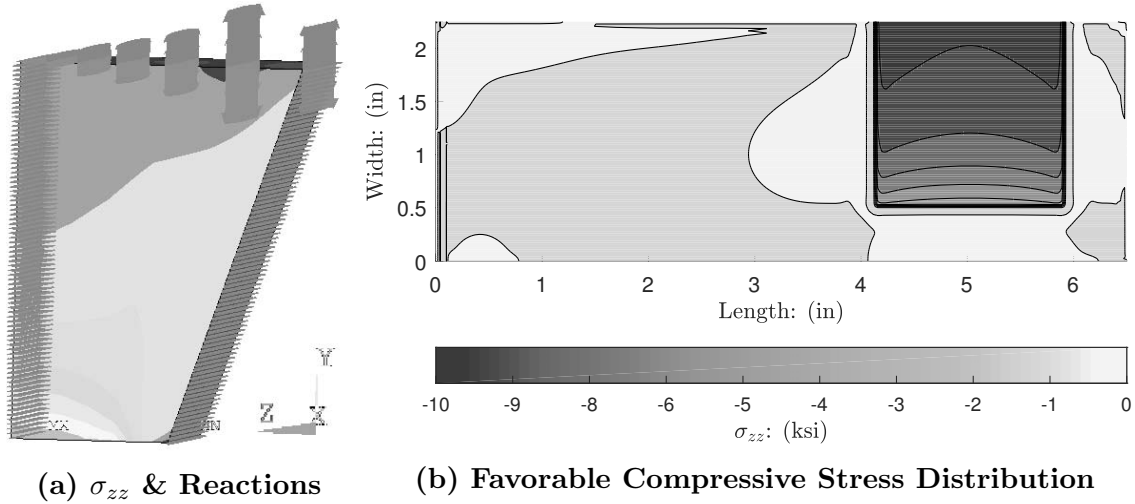


Figure 4.5. Single Grip Model

The region along the lower edge of the grip is predicted to have lower compressive stress, as seen in Figure 4.5b, which will more accurately predict the amount of plastic deformation in that region once tensile loading has been applied.

4.1.4.2 Tensile Loads

The tensile loading applied to the plate is modeled as shear forces which are distributed evenly along the surface in contact with the grips, as seen in Figure 4.6. Tensile load varies throughout the Pre-strain Method, and as such must be varied for different portions of the test, while maintaining this distribution throughout the grip surface (1.75 in by 1.75 in, 7,462 nodes). The ANSYS input script provided in Appendix A is constructed such that this distribution is generated for each loadstep, overwriting the tensile load data from the previous loadstep, and results are written to text files with the total tensile force reported in the file name.

4.1.4.3 ANSYS Scripted Loading/Unloading Procedure

In order to properly model residual stresses in a way that allows direct comparison with DIC experimental results, a multi-step process was scripted into ANSYS. The

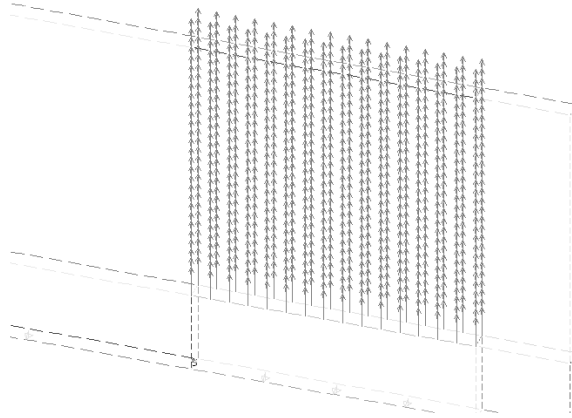


Figure 4.6. Tensile Force Application

initial loadstep consists only of the application of the compressive forces on the plate from the grip reaction forces. Following this, a loop is constructed to add tensile forces incrementally along the elastic ramp, for a total tensile force of 7500 lb_f, reporting data every 100 lb_f in the same way as the DIC data that is captured. The second ramp approaches the desired maximum load (15,000 lb_f for this research), still reporting data at every 100 lb_f, but with a longer duration of time for each loadstep (48 seconds vs. 8 seconds in elastic regime) to allow plastic deformation that is closer to the quasi-static state. The sum total of these ramps is 150 loadsteps. Upon completion of the ramps, a third loop is constructed to hold a constant force and report results every 60 seconds for 50 minutes, which is again consistent with the reporting of DIC results.

The unloading of the specimen, while of equal importance experimentally, is much simpler to model in ANSYS. All load data is deleted from the FEM model, and the compressive forces are reapplied, to predict results consistent with the plate during the duration of the unloading ramp. This unloading process takes place over the course of 100 seconds, and as such it is modeled as a single loadstep. Should transient unloading data be desired similar to the 150 loadsteps during loading, a loop could easily be constructed. However, because the unloading process is perfectly elastic, it is unlikely this data will be of any utility. The final loadstep is then modeled as the removal of all

point loads, resulting the final state of equilibrium with all residual stresses present. Therefore, the final loadstep predicting the residual stress-strain state is 203, which can be directly compared with DIC data at the unloaded condition.

4.1.5 Plate Data Processing

The script `Data_Extractor_Plate.m` (Appendix B.1.1), was developed to extract the useful results of the FEM analysis. The main data of interest are the coordinates and the six components of stress and strain for each node, especially the surface nodes for comparison to experimental results. This data is output from ANSYS in text files with a file name of the form `load_loadstep_property.txt`, where the load is given in lb_f of tensile load and the properties are either coordinates, normal strains, or normal stresses. The script is written to extract all three of these designators from the file name before processing of their contained data. Constructing the script in this way allows for combination of data of all three property types at equivalent loading conditions into a table for the sake of consistency, and all tables are then saved to a structure of the form `Plate.Lload.LSloadstep`, to aid in further use of data in comparison with experimental results under equivalent load histories.

While this succinct description of data processing of FEM results appears simple, it is very time intensive for large data sets where all information may not be reported for each node, and thus node numbering is not consistent in the text files generated by ANSYS. The combination of this data for the 203 loadsteps output to match time-dependent FEM results with DIC generates file sizes in excess of 1 GB when only saving data for surface nodes. Nevertheless, the efficiency and effectiveness of this comparison is the cornerstone of this research. A final step added after the extraction of all FEM data is generation of a matrix of index values, which is consistent for all loading conditions that are analyzed. As such, this index file (saved as I_{ijk}) greatly

aids in efficiency of processing results.

4.1.6 DIC Data Processing

Once all desired images have been captured for a given test, the images must be correlated to the reference image, and the outputs must be reported in the desired format. The region of interest must be defined such that strain can be resolved throughout (i.e. avoid geometric edges), and the subset size and step size must be defined based on speckle pattern details. It is always desirable to minimize the subset size, but it must be large enough to encompass the largest speckle in the region of interest. In this study, the subset size was typically 51 pixels, with a step size of 7 pixels. For the purpose of this research, the images are correlated only to the reference image (no incremental correlation), with a filter size of 35, as explained in Section 2.6.3. This means that strain reported at a point is a product of a Gaussian filter placed over a diameter of 105 pixels. The strains reported are Lagrange strains, and the data is output to a .mat file, which allows direct point-wise comparison with FEM results once the testing and modeling is complete.

4.2 Comparison of Experimental and Analytical Results

The final MATLAB script in Appendix B.3 is `DIC_Max.m`, which affords the capability of combining measurements made by DIC with predictions made with FEM. The measurement of geometric alignment features must first be used to ensure that comparisons are made in the appropriate geometric locations by translating the calibrated coordinates (x_c and y_c). Stress-strain data from FEM is stored as vectors, and must be converted into matrices via the index matrix I_{ijk} to be compared with DIC. The script first shows contours of FEM strain with the measurable DIC region defined, and can also plot stress components or von Mises stresses. In order to make

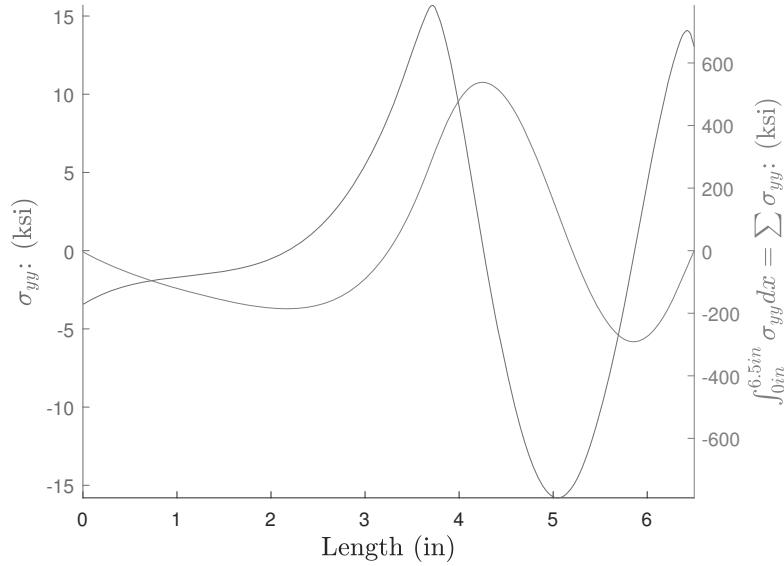


Figure 4.7. σ_{yy} Equilibrium Along Plate Centerline

qualitative comparisons between FEM and DIC, the FEM grid must match that of the DIC field (Typically DIC data is in a more dense grid than FEM). This is done via two-dimensional linear interpolation with the *griddata* command in MATLAB by default, but cubic and nearest neighbor interpolations are also possible.

Once the two data sets have been aligned to matching grids, calculations between them become simple. The contour shapes of the strains reported by both techniques can be overlayed on one another, or the deviation between the two fields can be calculated with simple subtraction. This script also calculates the RMSE of the entire field to report the match between the two fields as a scalar value, and plots error along the plate center line. Looping of the provided script with multiple DIC data sets and FEM results from the `Plate_Data.mat` is possible as well. The results of these methodologies will be further described later in Sections 4.4 and 4.5.

4.3 Finite Element Model Results

The results of the various FEM analyses are reported in this section. For each of the solutions presented, equilibrium of all stresses and strains was satisfied throughout the entire specimen. One such verification is seen in Figure 4.7, where the stresses in the Y-direction are plotted along the plate centerline. The integration of this stresses along the length is shown to be equal to zero at the free end (6.5 in from the origin) to ensure equilibrium of forces in the Y-direction. Within the ANSYS solver, equilibrium must be satisfied at every load condition (203 loadsteps in time-dependent model) as well as each intermittent time step (5-6 in the load ramps and hold) and the iterative solutions of the Newton-Raphson method (often 4-5 equilibrium iterations) in order to converge to a final result, for a total of roughly 5000 equilibrium iterations in the loading procedure. During this procedure, the equilibrium solution must also check for plastic deformation within each of the 207,900 elements, and those regions the generate plastic deformation must follow the appropriate constitutive relations for further loading and unloading, as seen in Figure 4.8.

4.3.1 Quasi-static Analyses

The results of the quasi-static loading case is presented first. In this analysis, the final load is applied “instantaneously,” or rather in a single loadstep that has converged to the quasi-static stress-strain curve. It should be noted that because this loading will generate plastic deformation, the final result is the product of many incremental additions of stress and plastic strains for all loading beyond the proportional limit (σ_0).

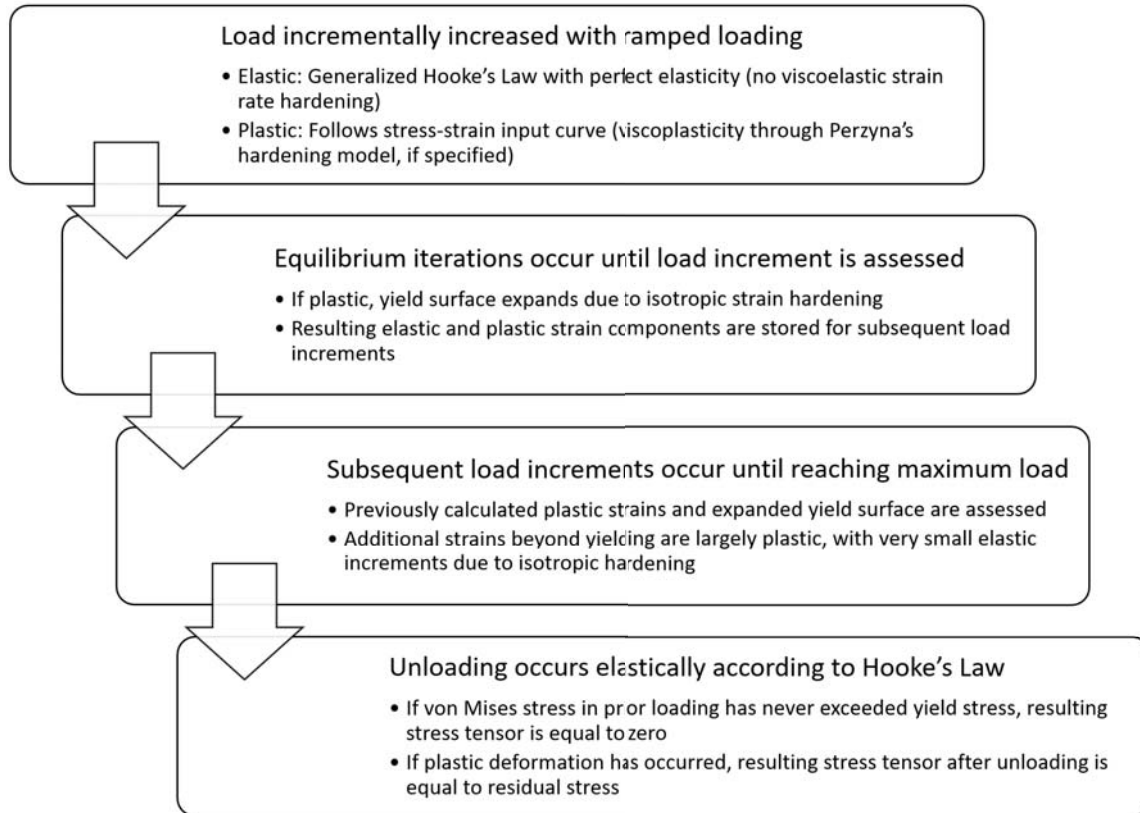


Figure 4.8. Residual Stress Flowchart

4.3.1.1 Maximum Load

The FEM results of the specimens at maximum load are seen in Figure 4.9, with strains on the left and stresses on the right. Qualitatively, one can associate a strain distribution that is inconsistent with the stress distribution as a probable location for plastic deformation. Quantitatively, plastic deformation is the product of an effective stress (Figure 4.9g) that has exceeded the proportional limit of approximately 39 ksi for Al 6061-T6. Beyond this von Mises stress, the effective plastic strains are developed in the direction of maximum deviatoric stress according to the Prandtl-Reuss relations (Equation 2.8).

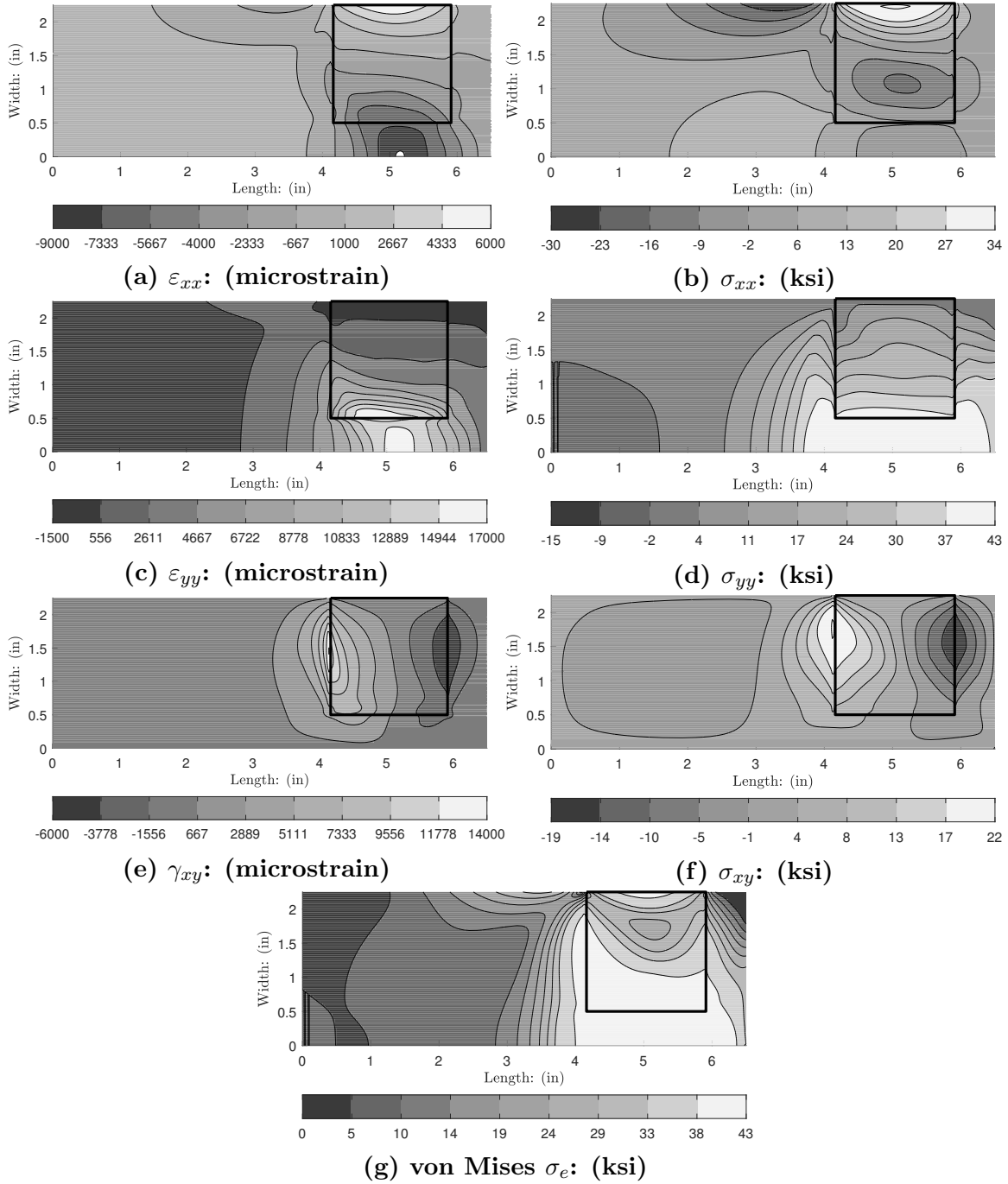


Figure 4.9. Quasi-static FEM σ - ε Results, Loaded State

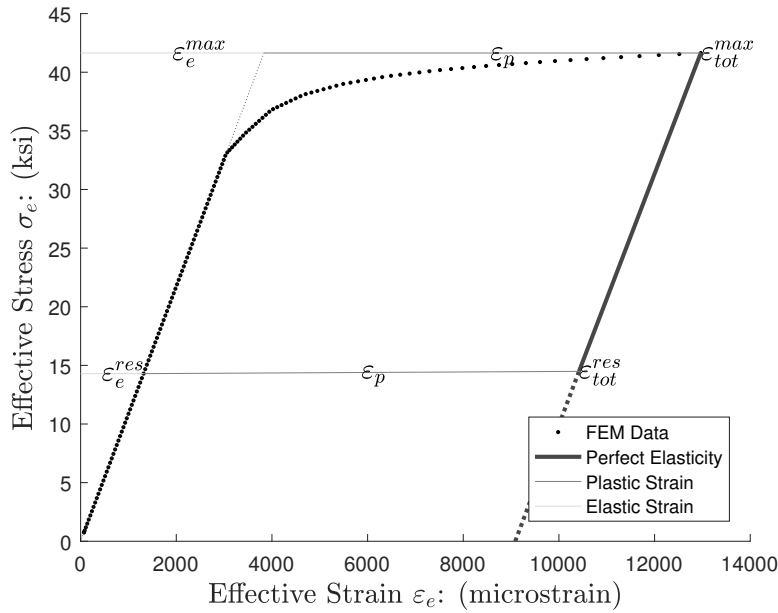


Figure 4.10. Effective Stress-Strain

4.3.1.2 Zero Load

The unloading procedure is perfectly elastic, meaning that generalized Hooke's Law can be used to calculate the change in stress from the loaded state to the residual state according to Equation 2.4. Plastic strain, by definition, is present in a state of zero stress, as seen in Figure 4.10 at a value of 8,802 microstrain.

The resulting residual stress can be directly calculated from the elastic offset of strain from the previously generated plastic strains, which can be observed by comparison of stresses and strains in Figure 4.11. Analysis of three dimensional stress states from uniaxial data is more easily described by von Mises stresses, as seen in Figure 4.10 (reported at a location of 5.2 in from the origin, along the plate centerline) with a maximum stress of 41.64 ksi and a residual stress of 14.49 ksi. The maximum stress is the product of the equilibrium solution of the loaded specimen, according the input stress-strain curve, and the elastic residual strain ($\epsilon_e^{res} = 1,315$

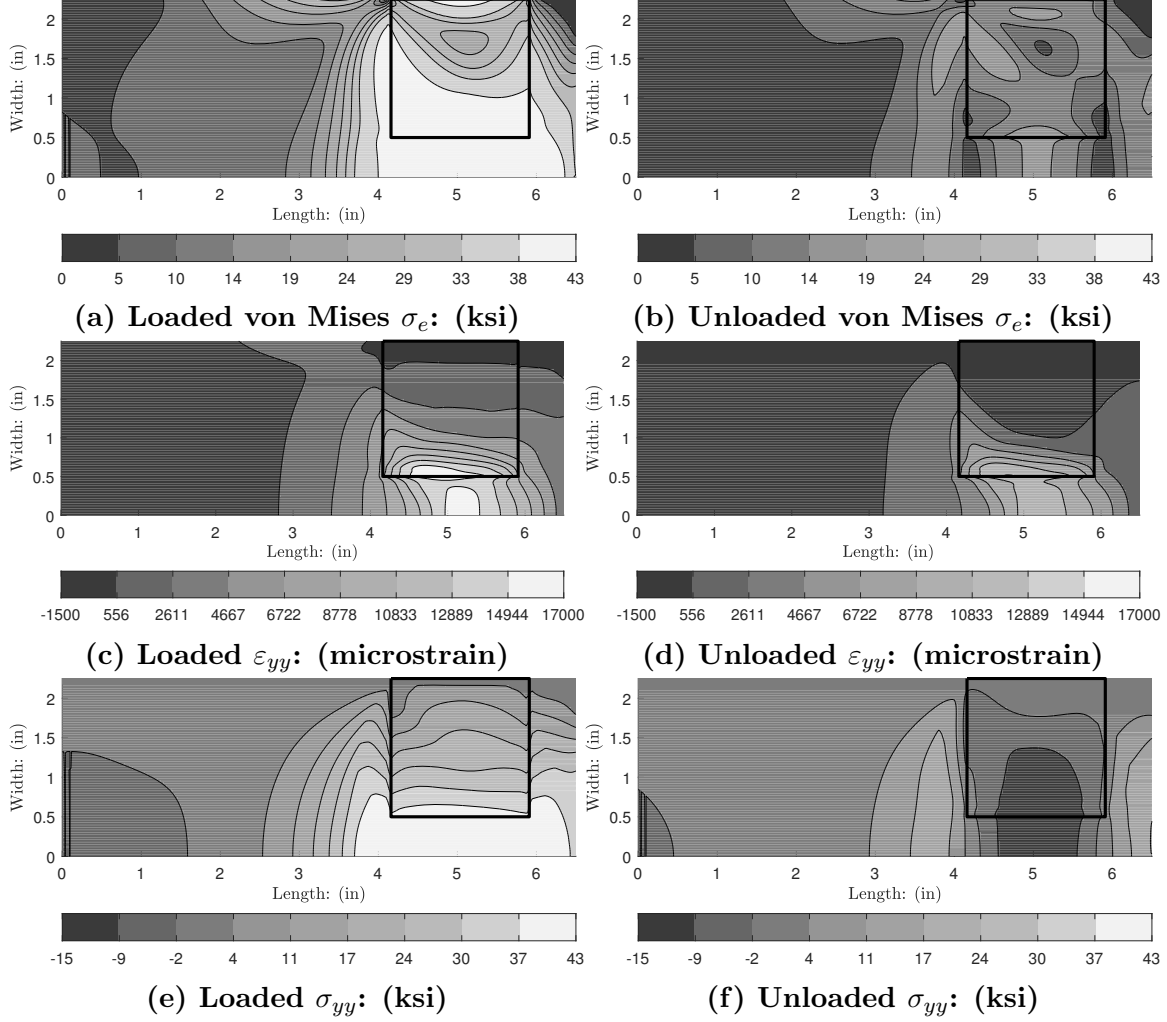


Figure 4.11. FEM σ - ε Results (Reprinted from Figures 4.9 and 4.12 for Ease of Reading)

microstrain) can be used to calculate the residual stress according to Hooke's Law (experimental Young's Modulus for this specimen was 10.868 Mpsi). An identical solution for residual stress could be made by subtracting the total recovered stress ($E[\varepsilon_{tot}^{max} - \varepsilon_{tot}^{res}]$) from the maximum von Mises stress of 41.64 ksi, because the unloading is perfectly elastic.

While stress-strain relations in a von Mises form are easier to relate to axial data than the six components in a full three dimensional analysis, the results can be misleading. Because all values of stress and strain are squared to analyze distortional energy, there is no indication of whether stresses and strains are tensile or compressive.

In this particular case, the maximum deviatoric stresses are in the vertical direction, and as such, the increments of plastic strain ($d\varepsilon_e^p$) are also in the vertical direction. Since most of the distortional energy is in the vertical direction at max load, the region between the grips is put into a state of residual compression of 14.15 ksi (See Figure 4.12d), which accounts for a majority of the von Mises stress that is present.

Further inspection of Figures 4.9 and 4.12 will show that the scale is held constant between the loaded and unloaded state for each component of the stress and strain tensors. While the results of σ_{yy} and σ_e show significant changes in contour shape, the predicted ε_{xx} , ε_{yy} , and σ_{xx} profiles are relatively the same between the unloaded state, with lower magnitudes in the unloaded state. This relationship shows that the residual stress/strain predictions are predominantly a result of equilibrium with the large plastic strains that exist in the vertical direction. In terms of shear, the predictions along the plate centerline show negligibly small magnitudes of both stress and strain in the loaded and unloaded states, but appreciable contributions to plastic deformation and resulting plastic shear strains can be seen along the sides of the grip in Figures 4.9e and 4.12e, respectively.

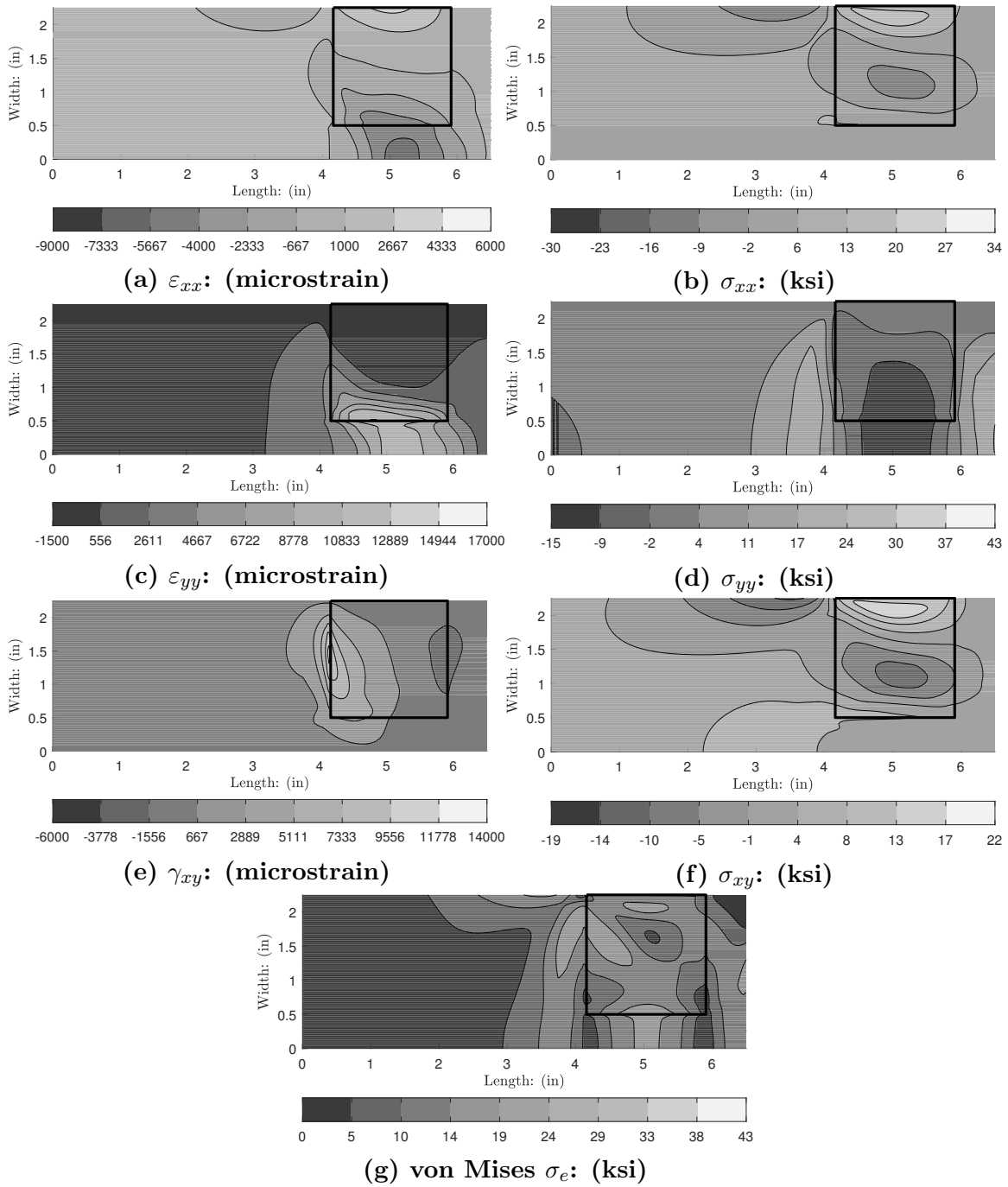


Figure 4.12. Quasi-static FEM σ - ε Results, Unloaded State

4.3.2 Time-dependent Analyses

The time-dependent FEM model predicts results that are qualitatively very similar to those of the quasi-static solution. In this analysis, the final load is approached incrementally via a series of load ramps and a hold period as defined in Figure 3.9b. Computationally, the loading procedure is accomplished in 150 loadsteps, with load ramps between, followed by 50 steps of constant load at 15,000 lb_f. Computationally this method is much more challenging, but it affords the ability to resolve time-dependent viscoplastic responses, and thus should model the experimental loading more accurately. Additionally, the FEM results are output along the same load interval as image capture via DIC to aid comparison. The unloading procedure is perfectly elastic, as mentioned above, and while it is modeled as a single loadstep, a much smaller timestep is required to iteratively maintain equilibrium.

The results of the time-dependent FEM analysis at the maximum load are seen in Figure 4.13, again with strain results on the left and stress results on the right. The qualitative changes between the time-dependent response and the quasi-static response (Figure 4.9) are difficult to discern, but quantitatively they are not identical. At the location of maximum ε_{yy} along the centerline (still 5.2 in), the strain is now 15,100 microstrain, as opposed to the 15,860 microstrain predicted in the instantaneous response. It should make sense that the model assessing viscoplasticity would predict a lower level of strain than the model that assumes perfect adhesion to the equilibrium stress-strain curve, as seen in Figure 2.18b. The 50 minute hold at the end of loading should minimize the deviations between these two models, however regions of highly localized stress will cause highly localized strains that may not be properly predicted in the quasi-static solution.

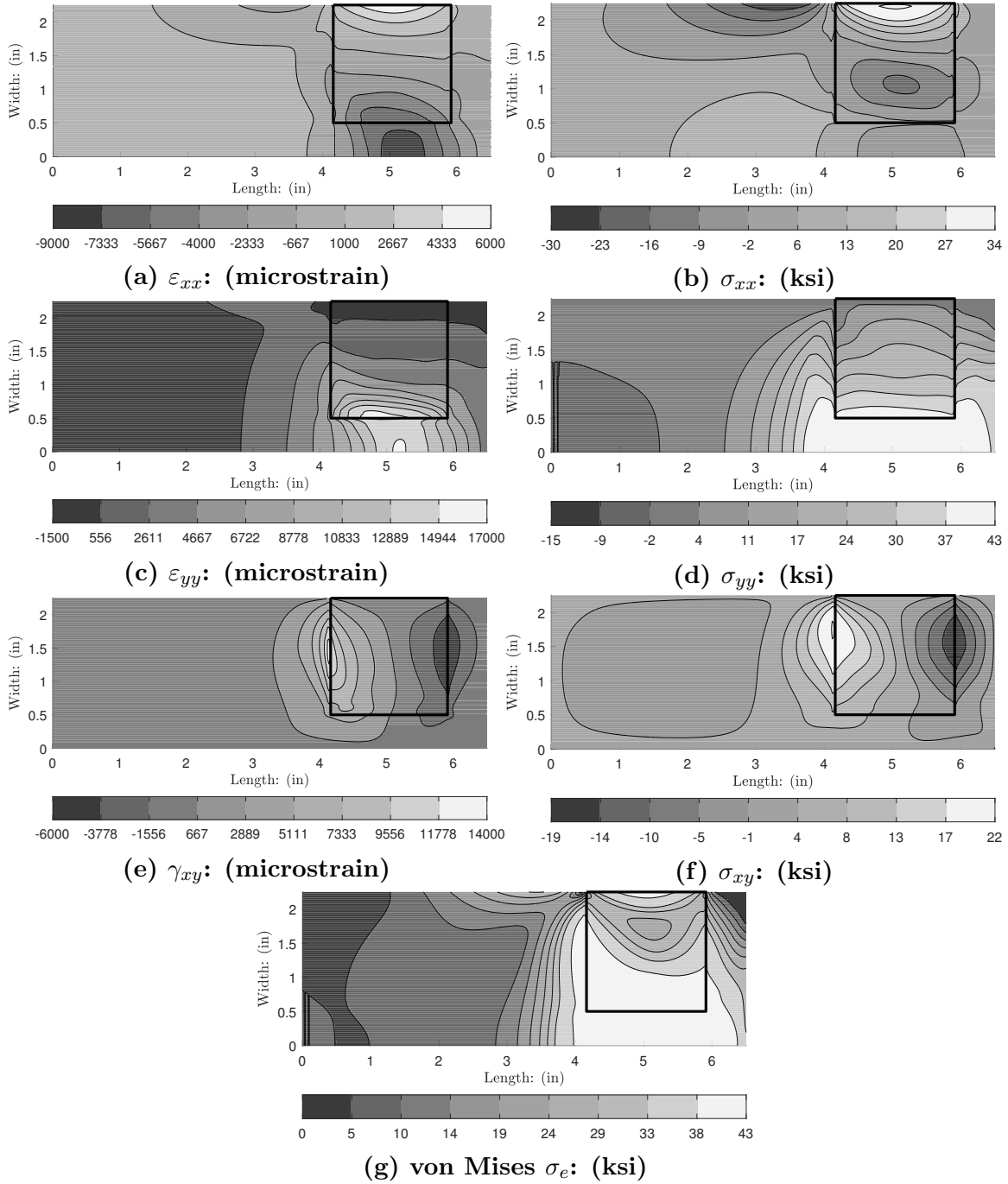


Figure 4.13. Time-dependent FEM σ - ε Results, Loaded State

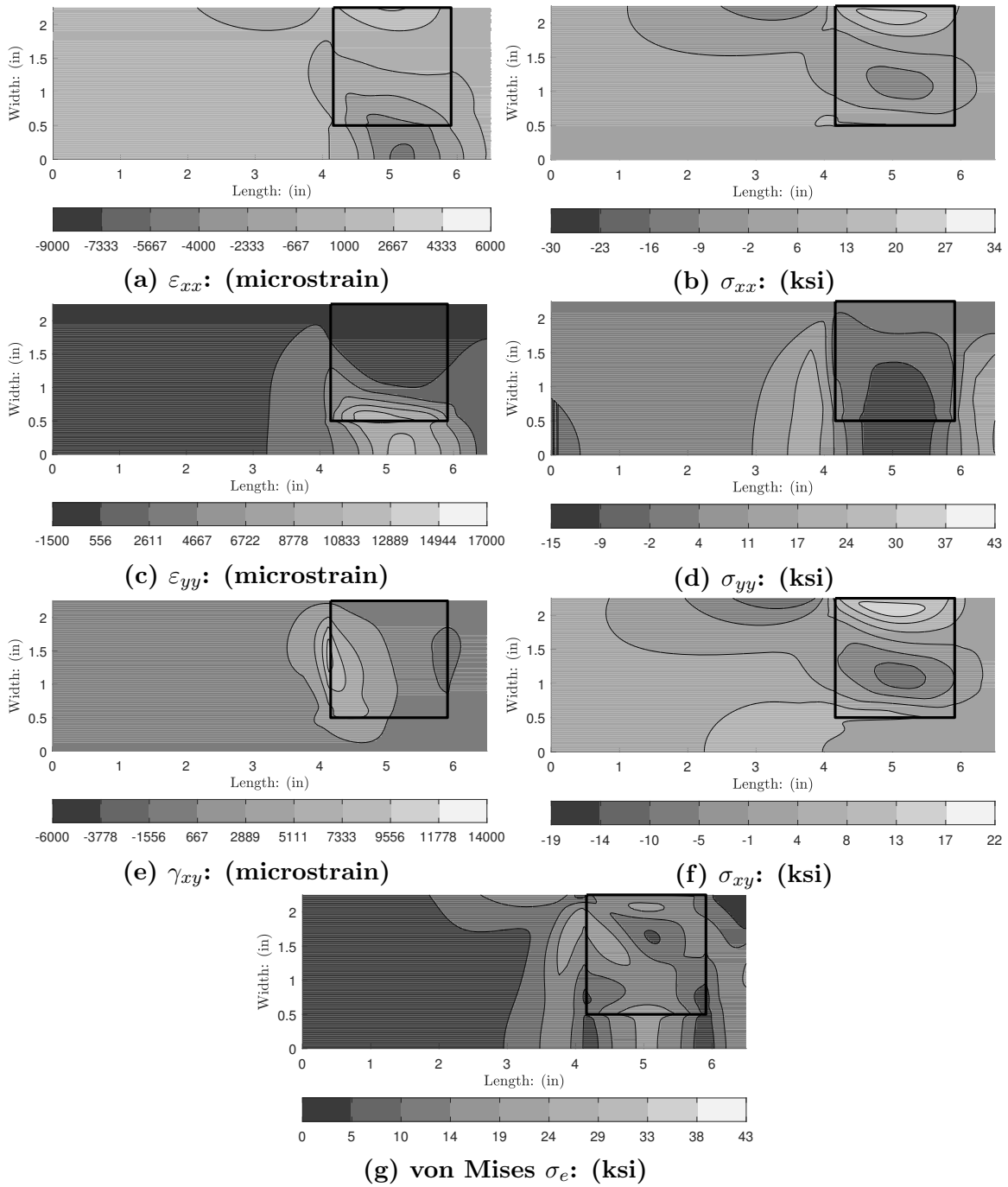


Figure 4.14. Time-dependent FEM σ - ε Results, Unloaded State

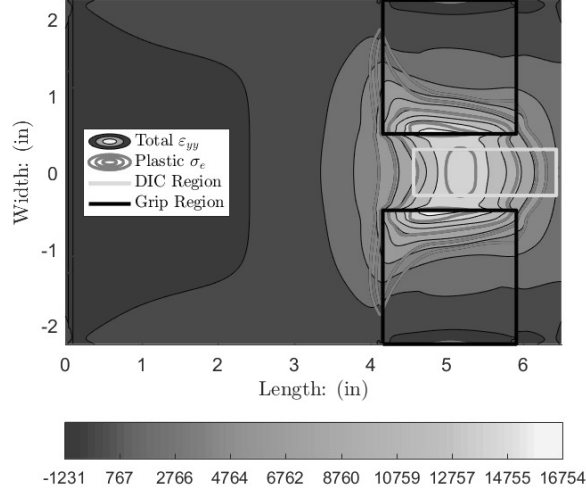


Figure 4.15. Full Plate FEM ε_{yy} : (microstrain)

4.4 Digital Image Correlation Results

The previous sections showed that both the quasi-static and time-dependent FEM solutions in the loaded and unloaded state follow the expected material response, with minor changes between the approximations of the residual state of stress-strain. To truly validate results, the FEM solutions should be consistent with the experimental deformation of the specimen as measured by DIC. The half-plane symmetric FEM model is rotated about the horizontal axis using two rotational transformation matrices according to Equation 4.1, and the coordinates of the DIC alignment features in Table 4.2 are matched to DIC-calibrated coordinates. The regions of Figure 4.15 outlined in black are covered by the grips, and the effective area measured by DIC is outlined in green.

$$\begin{aligned}
 \begin{bmatrix} \varepsilon_{xx} & \gamma_{xy} & 0 \\ \gamma_{yx} & \varepsilon_{yy} & 0 \\ 0 & 0 & 0 \end{bmatrix}^{bot} &= \begin{bmatrix} 1 & 0 & 0 \\ 0 & C_\pi & S_\pi \\ 0 & -S_\pi & C_\pi \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} & \gamma_{xy} & 0 \\ \varepsilon_{yx} & \varepsilon_{yy} & 0 \\ 0 & 0 & 0 \end{bmatrix}^{top} \begin{bmatrix} 1 & 0 & 0 \\ 0 & C_\pi & -S_\pi \\ 0 & S_\pi & C_\pi \end{bmatrix} \\
 &= \begin{bmatrix} \varepsilon_{xx}^{top} & -\gamma_{xy}^{top} & 0 \\ -\gamma_{yx}^{top} & \varepsilon_{yy}^{top} & 0 \\ 0 & 0 & 0 \end{bmatrix}
 \end{aligned} \tag{4.1}$$

The red contours are consistent with the levels of von Mises stress (see Figure 4.9g) that are in excess of the yield stress of 39 ksi, with subsequent contours every 1 ksi beyond yielding. As von Mises stresses increase beyond yield, plastic deformation increases in the direction of maximum deviatoric stress according to Equation 2.8, which is shown to be consistent with the contours of ε_{yy} . This shows that plastic deformation is being predicted properly in the FEM, but also that the regions of maximum plastic deformation are localized to the inner edge of the grip wedges, which is outside of the field of view of the DIC. The discernible region of interest of DIC then accounts for total strains (elastic and plastic components) in between the grips, which are also a product of the nearby regions of highly-localized plastic deformation.

The error between three experiments measured by DIC (all loaded to 15,000 lb_f using the same MPT test setup) are seen in Figure 4.16. Error along the centerline of the specimen shows very good agreement between tests for ε_{yy} , with slightly more variation in ε_{xx} and substantially more experimental variation in γ_{xy} . This comparison is slightly misleading, as the magnitude of strain is much lower for γ_{xy} than the normal strains, so the experimental noise is due to the inability to resolve strains on the order of 50 microstrain by DIC.

A better assessment of error between tests would be the RMSE of the entire measured strain field from the average strain field of all tests. This calculation was performed after matching the locations of reported strain in MATLAB using the *griddata* function, accounting for error in all 76,145 points of the DIC strain field, and the results are reported above each of the plots. The RMSE in the γ_{xy} fields is large, as expected, but the root mean squared deviation (RMSD) accounts for the very low magnitude of shear strains present. Conversely, the largest RMSD in the

Table 4.6. RMS Error and Deviation between DIC Measurements

RMS Type	Loaded			Unloaded		
	ε_{xx}	ε_{yy}	γ_{xy}	ε_{xx}	ε_{yy}	γ_{xy}
RMSE: (%)	7.09	4.46	10,200	13	10.1	1,540
RMSD: (microstrain)	240.6	377.1	57.87	250.6	376.1	52.68

loaded state was measured in ε_{yy} , but the RMSE is still the smallest due to the large amount of Y-direction strains in the region of interest. The RMSE and RMSD are reported in Table 4.6 to assess variation between test samples, and will later be used in Tables 4.8 and 4.9 to compare variation between FEM solutions against experimental variation.

Another important discovery is that the calculated RMSEs are larger in the unloaded state, while the RMSDs remain relatively unchanged (with the exception of noisy γ_{xy} data). The deviation can thus be assumed as a reasonable estimate of the standard deviation between tests, accounting for both experimental variation and noise in the DIC results. The latter of these two sources of error reaches a lower limit at very low strains, and is likely due to DIC's inability to correlate a shear deformation for many of the subsets.

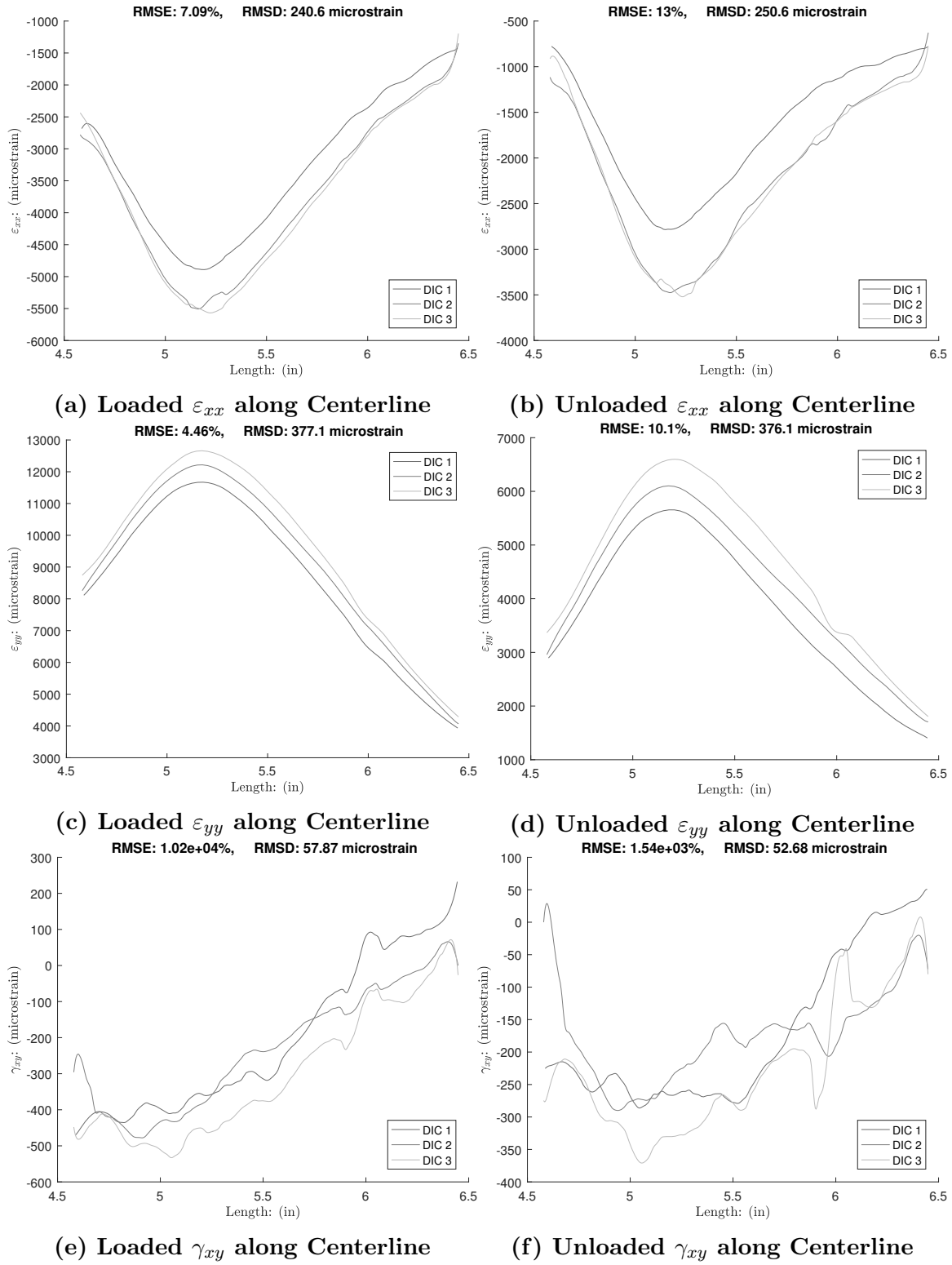


Figure 4.16. Error Between DIC Measurements During/After 15,000 lb_f Load

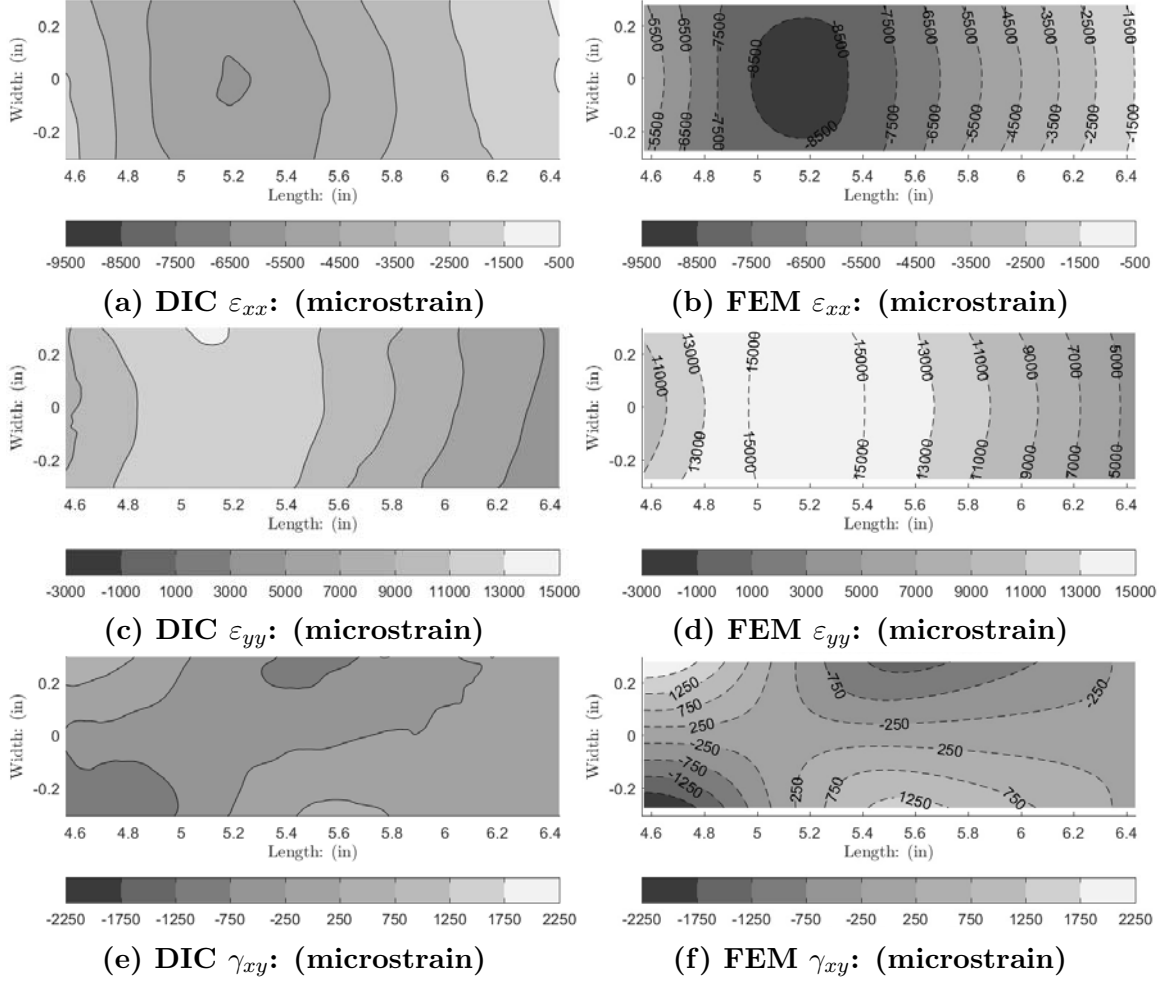


Figure 4.17. Quasi-static Strain Results, Loaded State

4.4.0.1 DIC vs. FEM, Maximum Load

By aligning the FEM and DIC coordinates, relationships can now be directly assessed. Figure 4.17 shows both experimental strains (measured by DIC) and predicted strains (quasi-static FEM) at a load of 15,000 lb_f to show a qualitative match of strain profiles in all three types of measurable strain. In all cases, the total strain is of a larger magnitude in the FEM predictions than the experimental DIC results. The increased noise in γ_{xy} is also observed, while the FEM solution has clean contours.

In comparing the validity of the two different FEM loading analyses, it is easier to

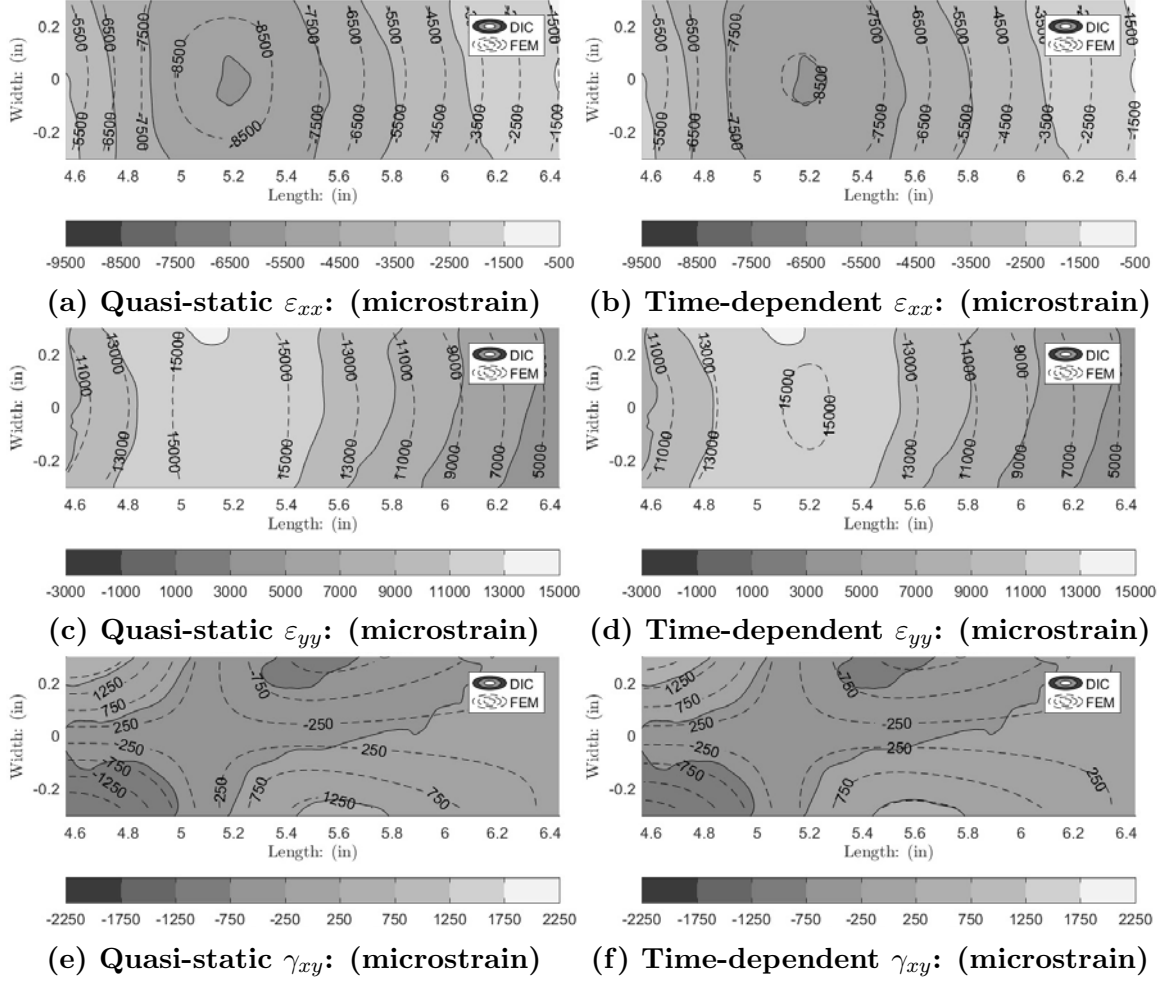
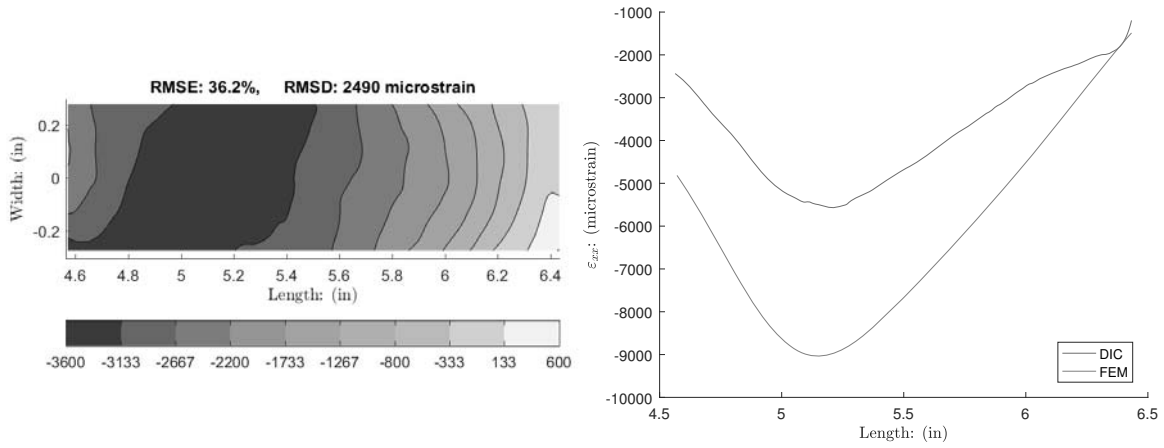
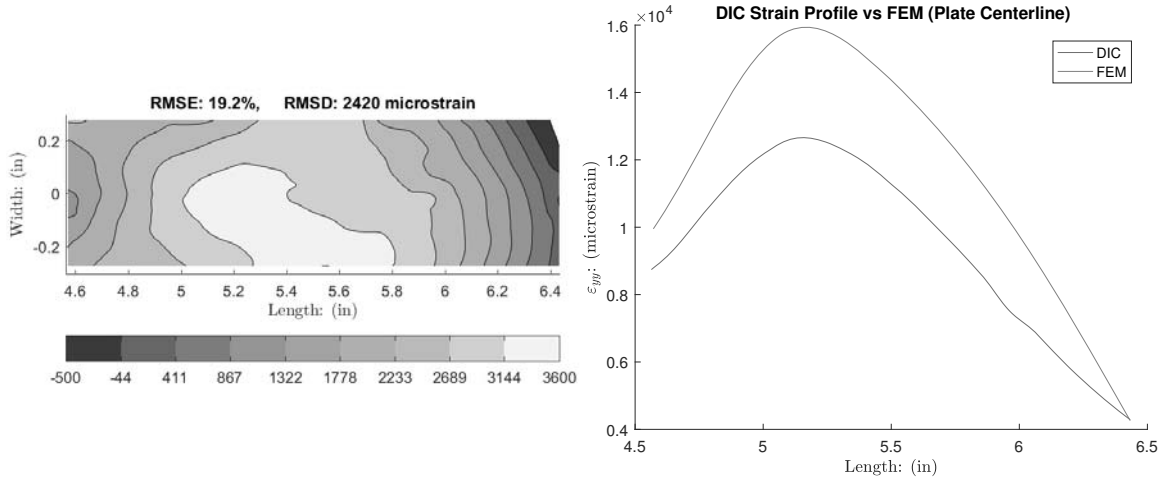


Figure 4.18. DIC vs. FEM Strain Results, Loaded State

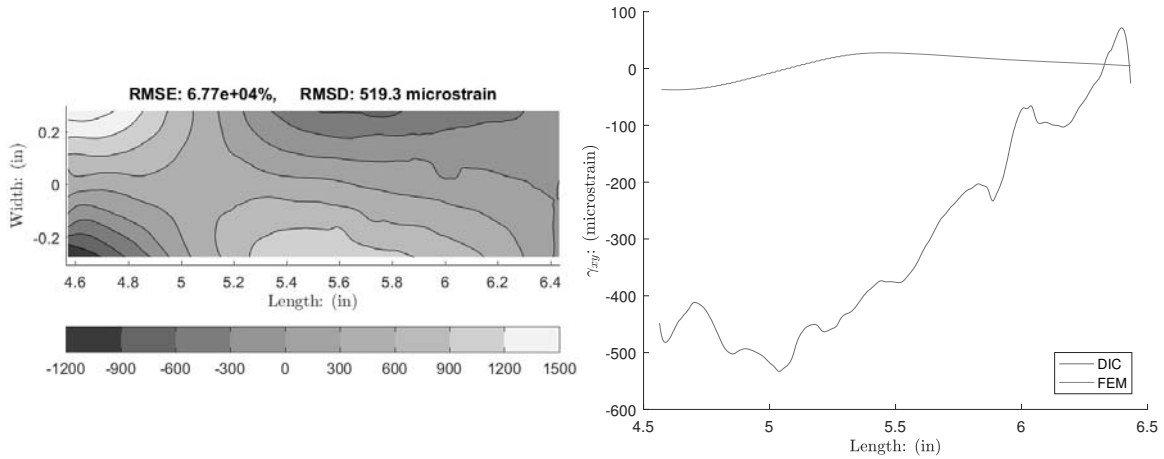
visualize the contours of predicted strains overlayed on top of the measured contours of DIC predicted strains, as seen in Figure 4.18. Similar to the quasi-static results, the time-dependent strain results are also predicted to be of a higher magnitude than the experimentally generated results. Contours of these errors are seen in Figures 4.19 and 4.20, with regions of ε_{yy} offset by over 2,000 microstrain along the plate centerline, and even larger errors along the lower edge. The RMSEs and RMSDs are again reported at the top of the error plots, and later presented in Tables 4.8 and 4.9.



(a) DIC Field ϵ_{xx} Error: (microstrain) (b) ϵ_{xx} Error Along Plate Centerline

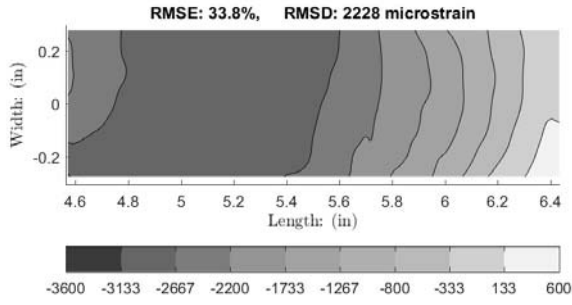


(c) DIC Field ϵ_{yy} Error: (microstrain) (d) ϵ_{yy} Error Along Plate Centerline

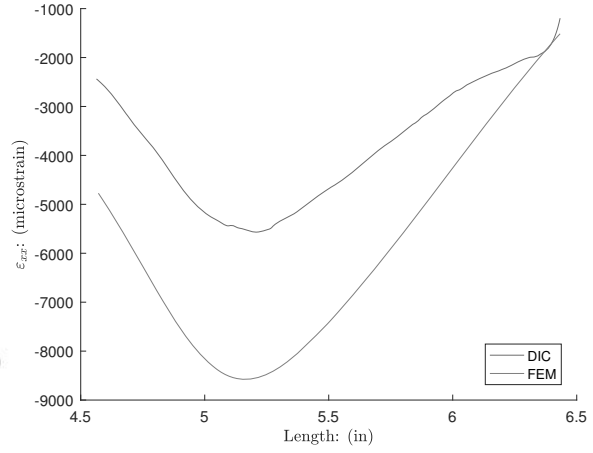


(e) DIC Field γ_{xy} Error: (microstrain) (f) γ_{xy} Error Along Plate Centerline

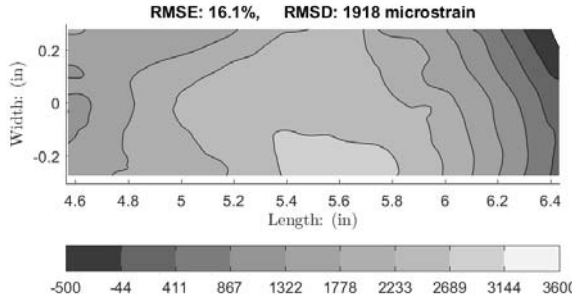
Figure 4.19. Quasi-static DIC vs. FEM Strain Error, Loaded State



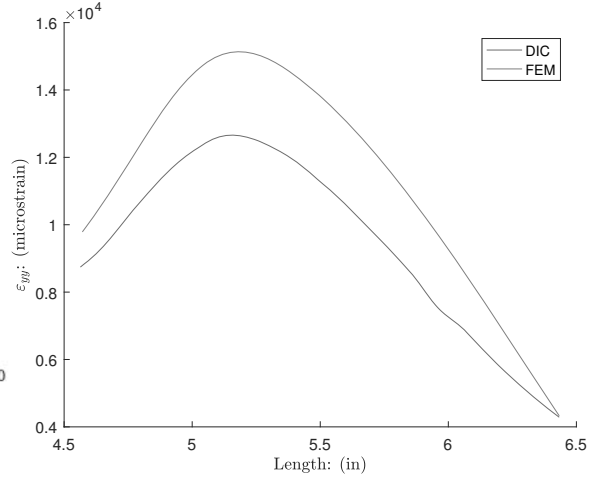
(a) DIC Field ε_{xx} Error: (microstrain)



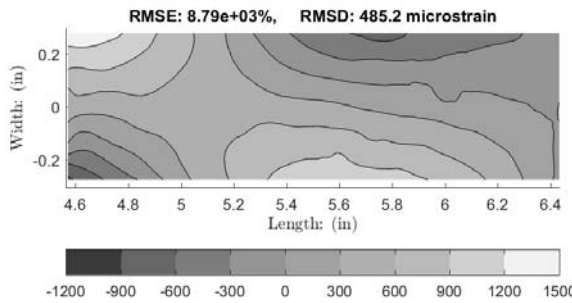
(b) ε_{xx} Error Along Plate Centerline



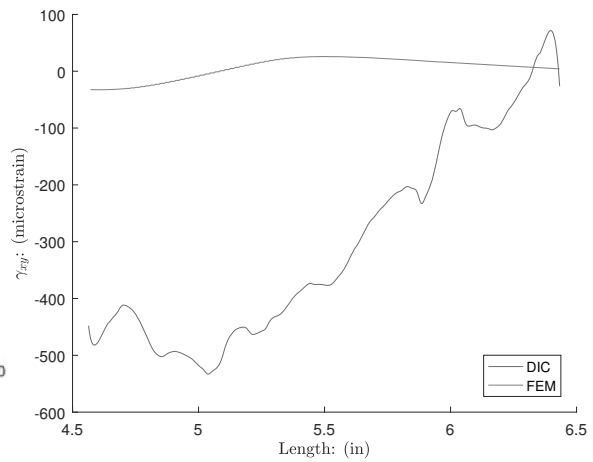
(c) DIC Field ε_{yy} Error: (microstrain)



(d) ε_{yy} Error Along Plate Centerline



(e) DIC Field γ_{xy} Error: (microstrain)



(f) γ_{xy} Error Along Plate Centerline

Figure 4.20. Time-dependent DIC vs. FEM Strain Error, Loaded State

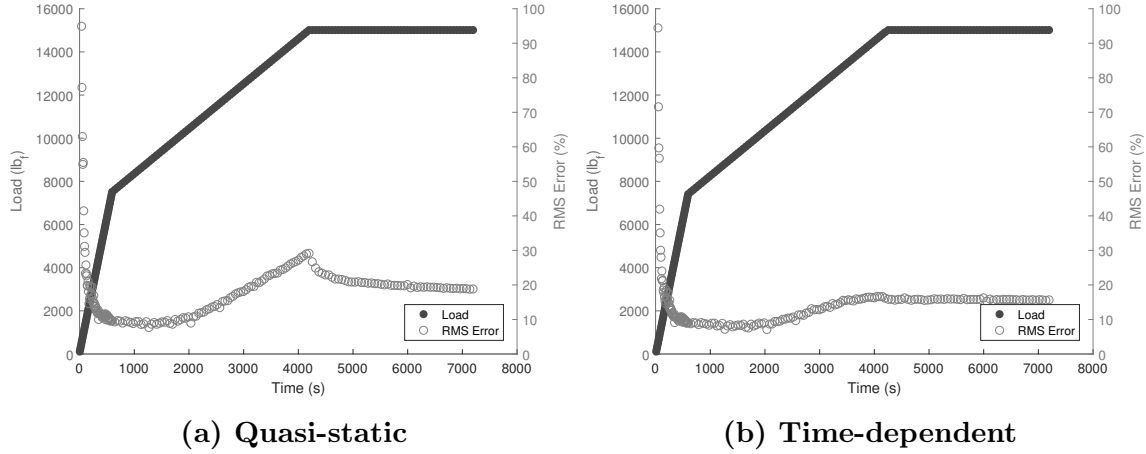


Figure 4.21. ε_{yy} RMS Error vs. Time

The time-dependent FEM model may appear as a better representation of the DIC strain measurements than the quasi-static model, but the difference between the two FEM solutions is not large. This observation is misleading, because the experimental results are reported after a 50 minute hold at the maximum load. The justification for the hold is to avoid a highly transient state of overstress (as mentioned in Chapter III) until the measured strain rate is sufficiently small, which by definition is approaching the quasi-static stress-strain curve (see Figure 2.18b). However similar, arriving at a converged solution with an “instantaneous” loading is still not the same as assessing the full time-dependent model, as regions of concentrated strains may not be represented properly.

The difference between the two models is better represented when assessed throughout the entire loading history. The time-dependent FEM solution was defined to report results at every 100 lb_f under ramp loading, as was DIC, so the quasi-static FEM solution was also assessed at every 100 lb_f to offer direct comparison of results. The RMSE of the experimental DIC strains (ε_{yy}) from the two FEM models are plotted in Figure 4.21, showing that immediately after the end of the second ramp the plastic strains are much better represented by the time-dependent model than that of the

quasi-static.

In both cases, there is a very large error (often outside the scale of the plot) at very low levels of load due to small strain magnitudes compared to the noise of the measurement (similar to γ_{xy} errors). After loading is greater than 2000 lb_f, there is a minimum error in the elastic response of the specimen until roughly 2000 seconds, regardless of the decreased load rate which is indicative of rate-independent perfectly elastic behavior. The error in both specimens grows after 2000 seconds, which is due to poorly represented strains beyond the proportional limit, either as a result of the viscoplastic model or the FEM boundary conditions being inconsistent with the experimental results. The error in the quasi-static response exceeds time-dependent error, and quickly declines during the hold step (during this period, the quasi-static FEM solution does not change, but the DIC strains do) until the final RMSE of 19.2% is met. In the time-dependent response, the final error is not only less (16.1%) but the error of the entire strain field remains low throughout the transient high strain rate response. This is proof that the strain predicted by overstress in the Perzyna model is predicting rate-dependent plastic behavior quite well.

4.4.0.2 DIC vs. FEM, Zero Load

A similar set of analyses was performed on the unloaded state, which is of higher importance because it is paired with the FEM approximation of stress in the residual state. In both the quasi-static and the time-dependent model, the magnitude of strain was over-approximated on all three specimens, as seen in Figure 4.22. However, careful comparison between the loaded and unloaded results show that while the strain error from DIC measurements to FEM approximations is higher in the unloaded state, the deviation (in units of microstrain) has remained relatively constant (best seen if comparing Figures 4.23 vs. 4.19 or 4.24 vs. 4.20). While the magnitude

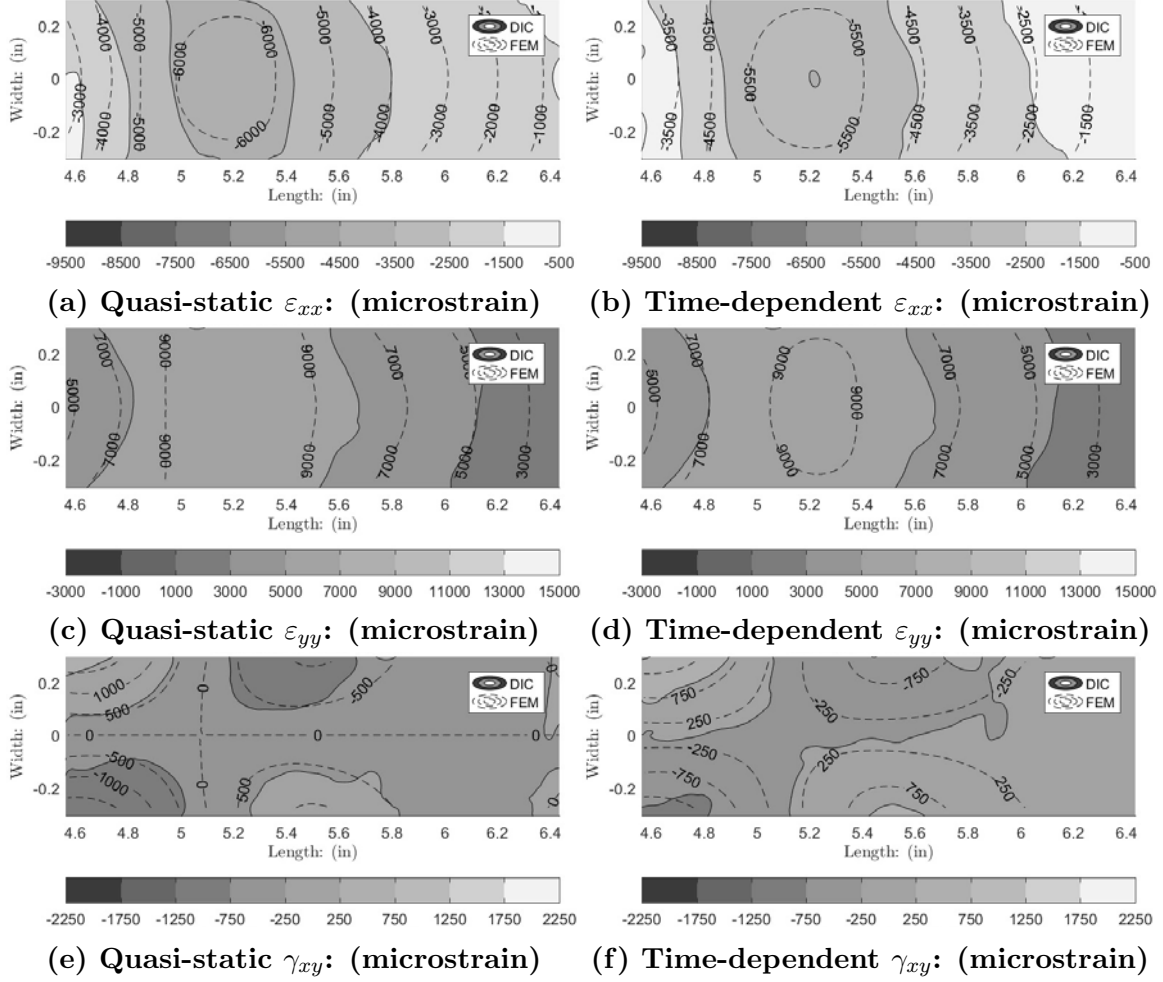
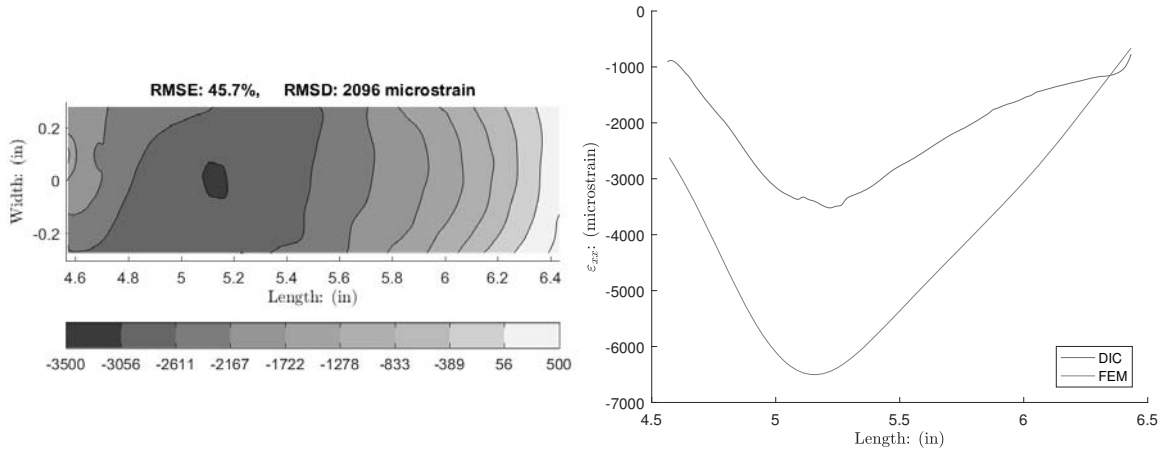
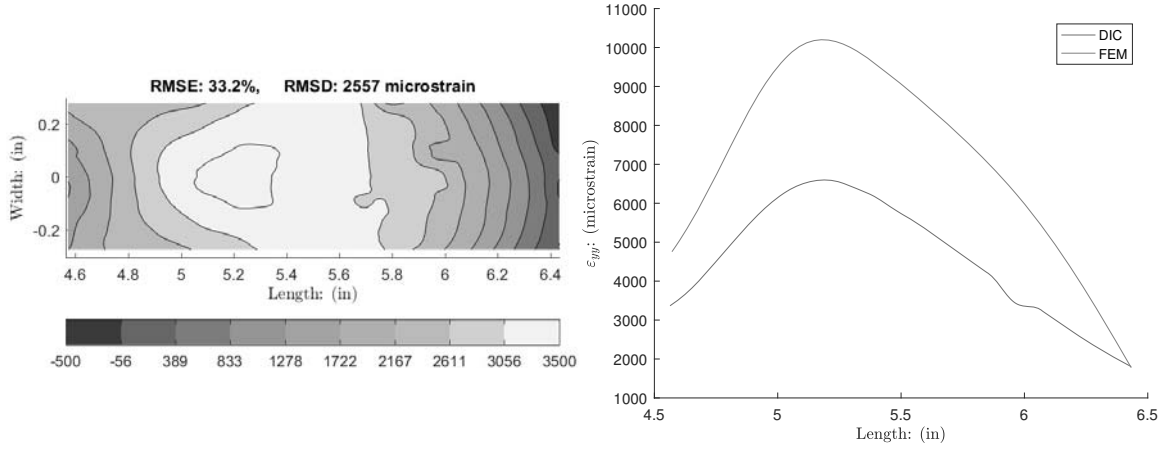


Figure 4.22. DIC vs. FEM Strain Results, Unloaded State

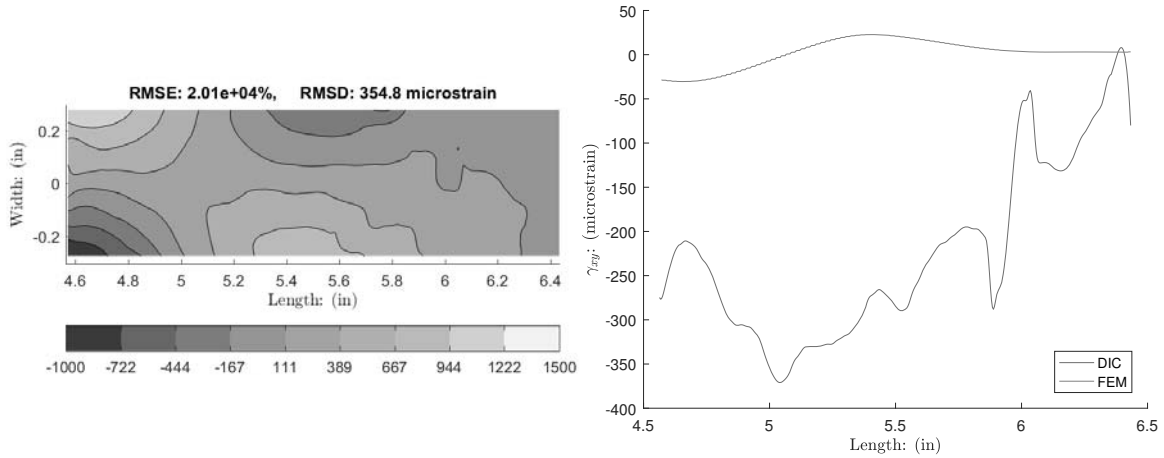
of actual strain (as measured by DIC) has reduced, the offset in the FEM solution has remained the same between the loaded and unloaded states. While DIC only has the ability to measure total strain (the summation of elastic and plastic strains) the RMSD between experimental and computational results is constant in the loaded and unloaded state, and thus the deviation is in the plastic portion of the strain. As was mentioned previously, it is challenging to definitively claim the source of these deviations without extensive analysis of the results, but it is clear that they are permanent after the loading has taken place.



(a) DIC Field ε_{xx} Error: (microstrain) (b) ε_{xx} Error Along Plate Centerline

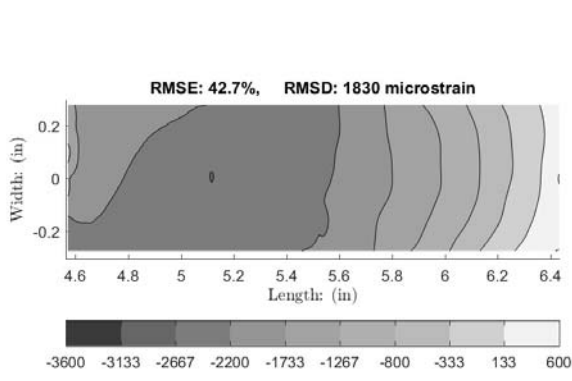


(c) DIC Field ε_{yy} Error: (microstrain) (d) ε_{yy} Error Along Plate Centerline

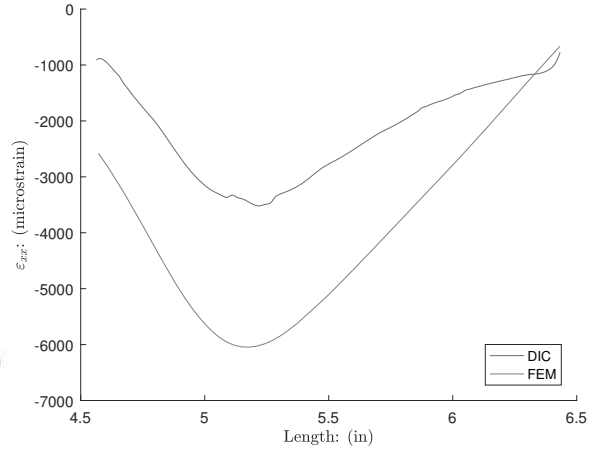


(e) DIC Field γ_{xy} Error: (microstrain) (f) γ_{xy} Error Along Plate Centerline

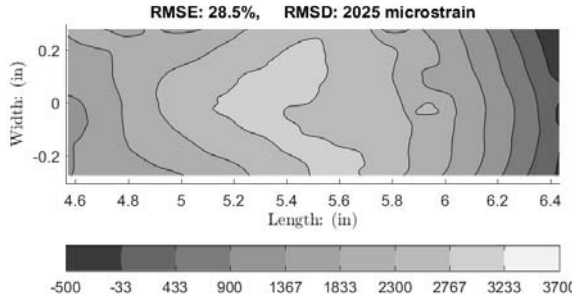
Figure 4.23. Quasi-static DIC vs. FEM Strain Error, Unloaded State



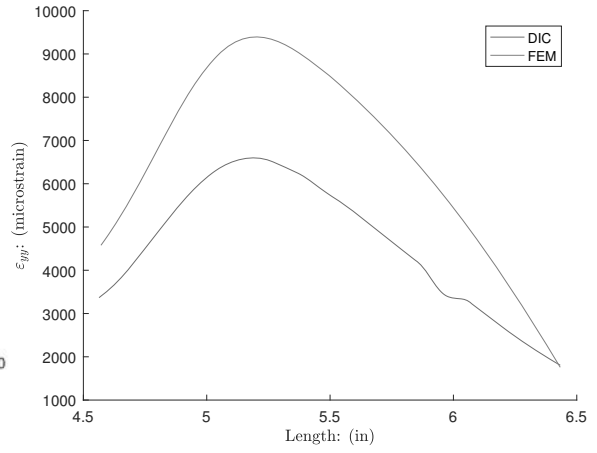
(a) DIC Field ε_{xx} Error: (microstrain)



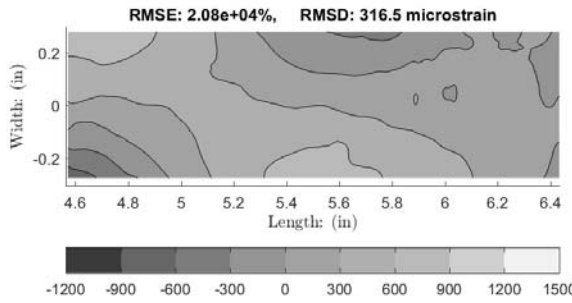
(b) ε_{xx} Error Along Plate Centerline



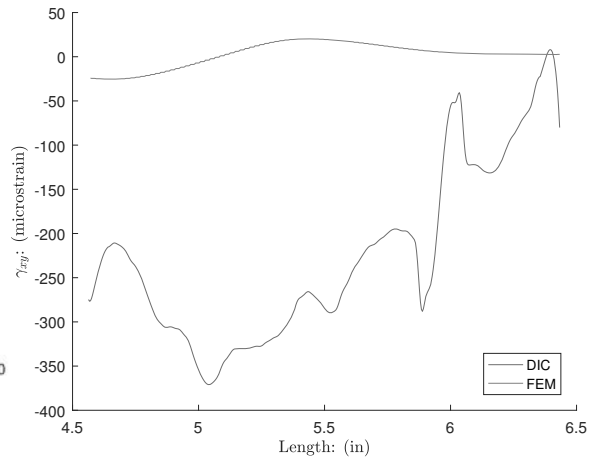
(c) DIC Field ε_{yy} Error: (microstrain)



(d) ε_{yy} Error Along Plate Centerline



(e) DIC Field γ_{xy} Error: (microstrain)



(f) γ_{xy} Error Along Plate Centerline

Figure 4.24. Time-dependent DIC vs. FEM Strain Error, Unloaded State

4.4.1 Optimized Loading due to Strain Field Matching

After assessing the relationships between DIC measurements and computational models in both the loaded and unloaded state, it is clear that there is a permanent bias in computational results that is consistently over-approximating strain development. While rigor was taken to ensure that material property variations were accounted for (in both static and dynamic behavior) and experimental variation was reduced (in specimen geometry and alignment), this offset continues to persist. However, Table 4.6 displayed RMSEs between “identical” tests of 4.5-7% in normal strains in the loaded state, which doubled in the unloaded state, corresponding to just 250-350 microstrain. The quantification of this experimental variation can be used to allow minor offsets in FEM tensile loading in search of solutions that match better with experimental strains.

4.4.1.1 Optimized Loading, Maximum Load

In the quasi-static FEM solution, the offset in ε_{yy} from DIC can be reduced by decreasing the maximum tensile load, which would reduce the plastic strains *vis-à-vis* the lowered stresses imparted on the specimen. It is appropriate to focus on minimizing the deviation of ε_{yy} , as opposed to ε_{xx} or γ_{xy} , because the maximum distortional energy is in the Y-direction under maximum load, so deviations in ε_{yy} correlate most directly to the permanent plastic deformation that generates residual stress upon unloading. In this manner, reducing the maximum load from 15,000 lb_f to 14,400 lb_f (4% reduction) minimizes deviation in ε_{yy} from 2420 to 359.1 microstrain, while further reductions in tensile loading would predict strain fields that were lower than experimental, as shown in Table 4.7.

This adjustment was then made for both the quasi-static and time-dependent FEM solutions, and the match of strain was significantly improved, as seen in Figures 4.25

and 4.26. The RMS error in ε_{yy} is just 7.23% in that of the quasi-static solution and

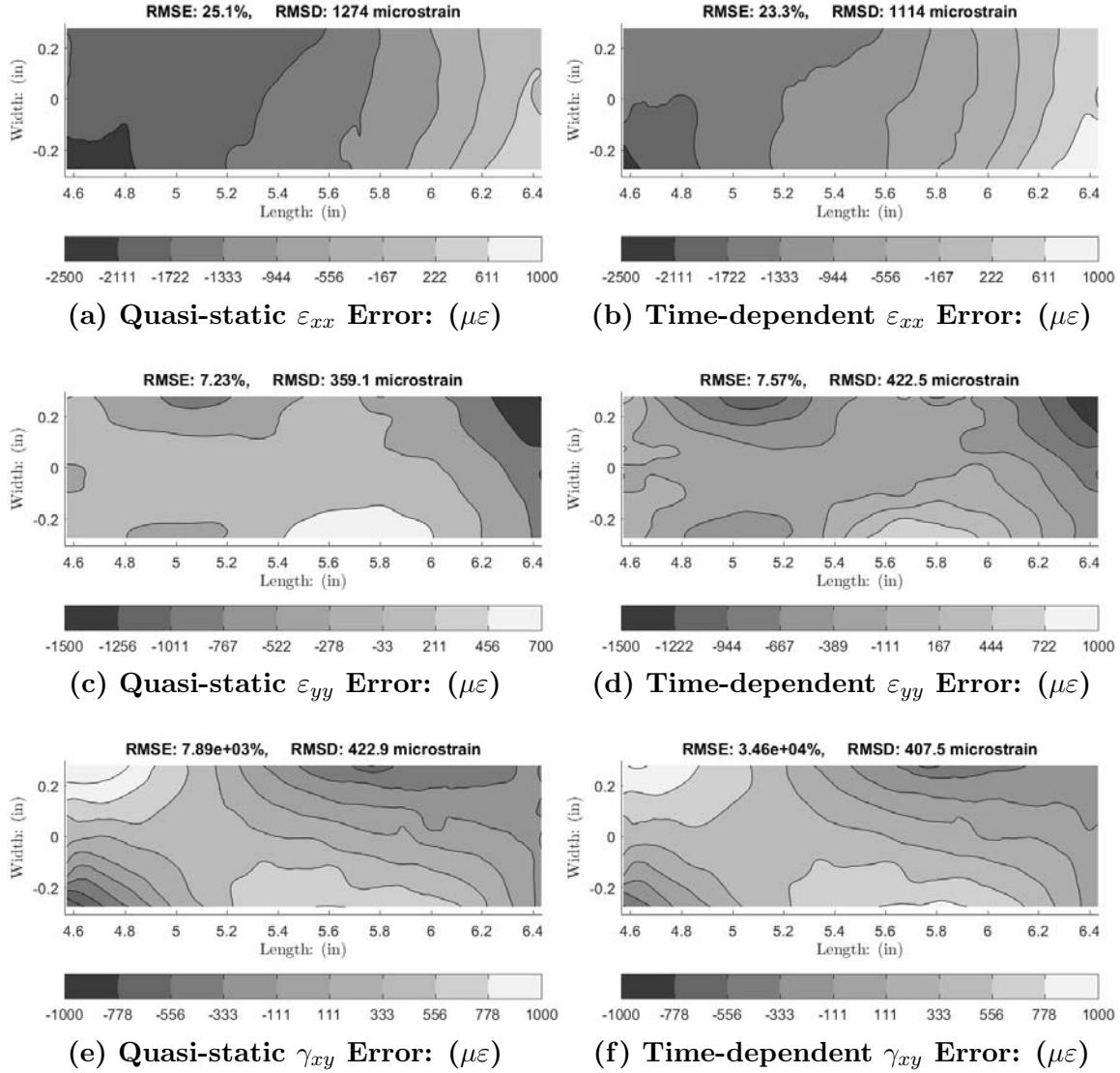


Figure 4.25. Optimized DIC Field vs. FEM Strain Error, Loaded State

Table 4.7. Quasi-static ε_{yy} RMSE & RMSD vs. Maximum Load

Load: ($\times 10^3$) lb _f	15.0	14.9	14.8	14.7	14.6	14.5	14.4	14.3	14.2
ε_{yy} RMSE: %	19.2	16.2	13.2	10.6	8.38	7.06	7.23	8.84	11.3
ε_{yy} RMSD: $\mu\varepsilon$	2420	1967	1548	1162	813.2	519.3	359.1	446.9	670

7.57% in the time-dependent, which is not much greater than the variation between experiments. The errors in the compressive ε_{xx} are smaller than the un-optimized

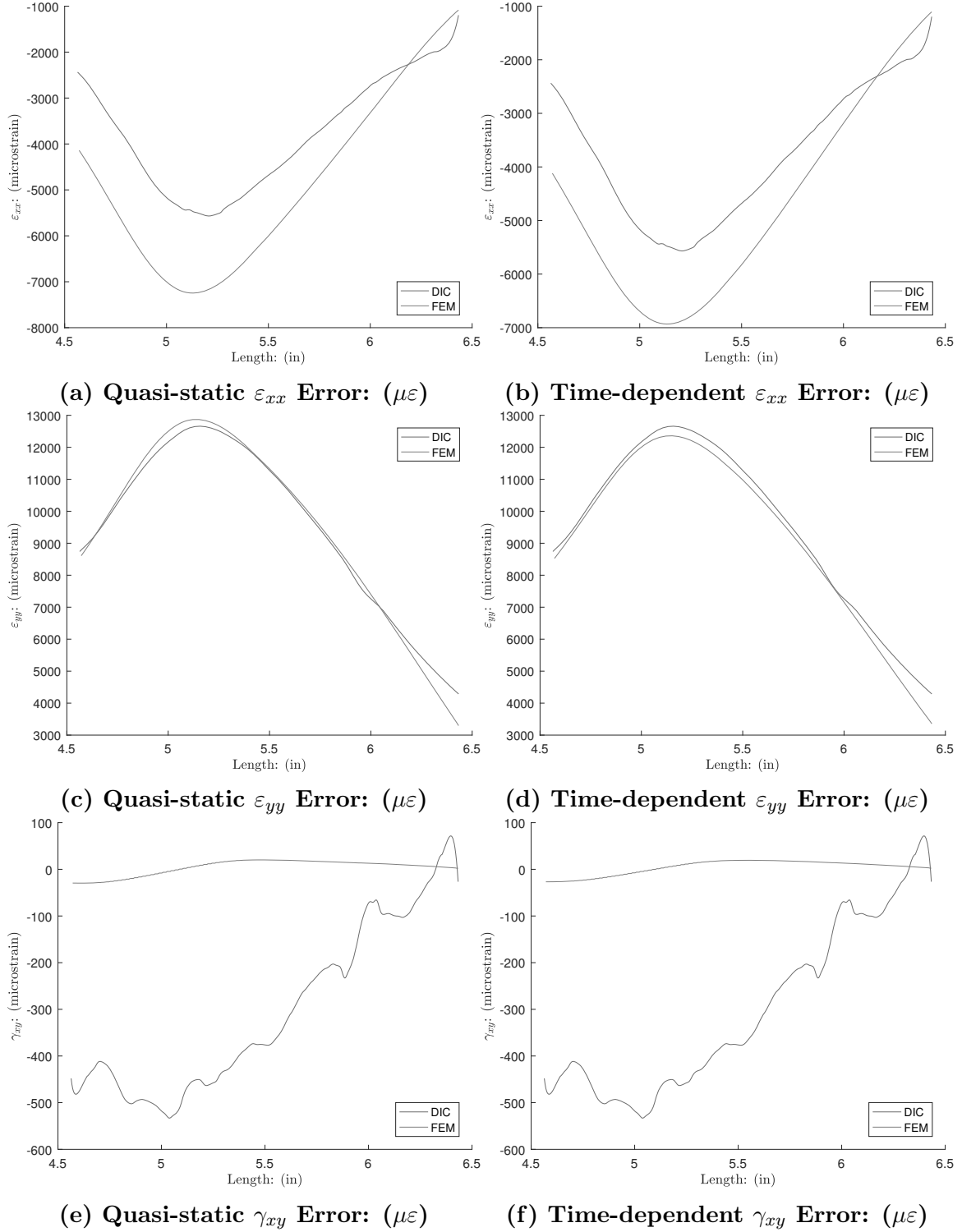


Figure 4.26. Optimized Error Along Plate Centerline, Loaded State

FEM, but still much larger than that of ε_{yy} . Errors in shear strain have grown, but it

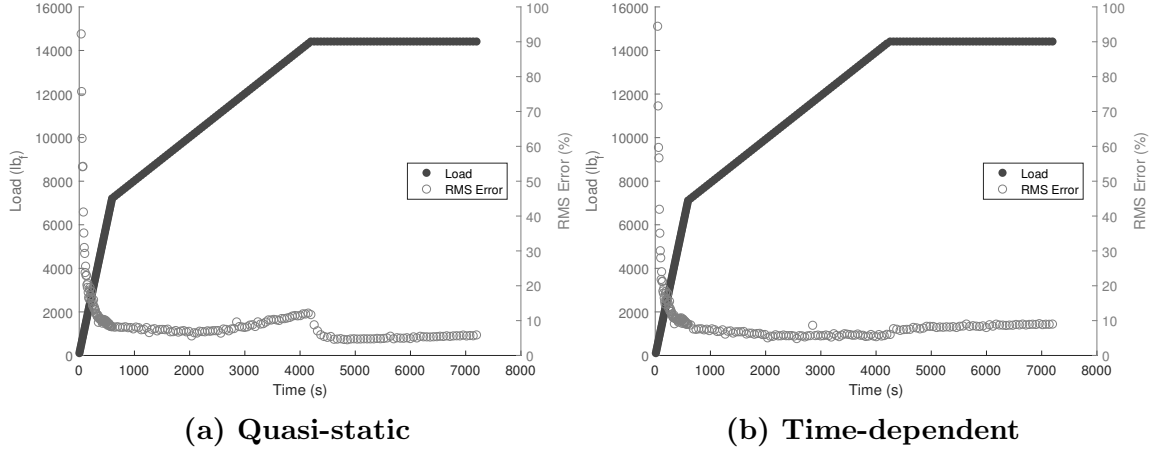


Figure 4.27. Optimized ε_{yy} RMS Error vs. Time

should be clear that shear strains are so small that they cannot be properly resolved in DIC and they make minimal contribution to the plastic deformation.

The error in ε_{yy} as a function of the optimized controlled loading can also be assessed by proportionally reducing tensile load by 4% in each FEM model. Results are instead reported every 96 lb_f of loading across the ramps, but compared against DIC data reported every 100 lb_f to assess optimized correlation of results throughout the entire load history. It is important to mention that RMSE reports all errors as a positive magnitude, so if the FEM specimen loading produced results that were below experimental, the RMSE curve would still climb. However, in Figure 4.27 it is clear that the error rises in the quasi-static solution at the end of the second load ramp.

Similar behavior can be observed in Figures 4.21 and 4.27 with respect to specimen loading and plastic deformation. The initial error still declines until elastic strains are measurable, however the plastic strain error (beyond 2000 seconds) only appears to rise in the quasi-static solution. Even at lower tensile loading, it is expected that the quasi-static solution would over-predict plastic strains. However there is an increasing trend in strain error during the hold step for the time-dependent model, which is due to under-approximated strains at the end of the second ramp. Under-approximated strains will translate to an under-approximated strain rate, which explains this

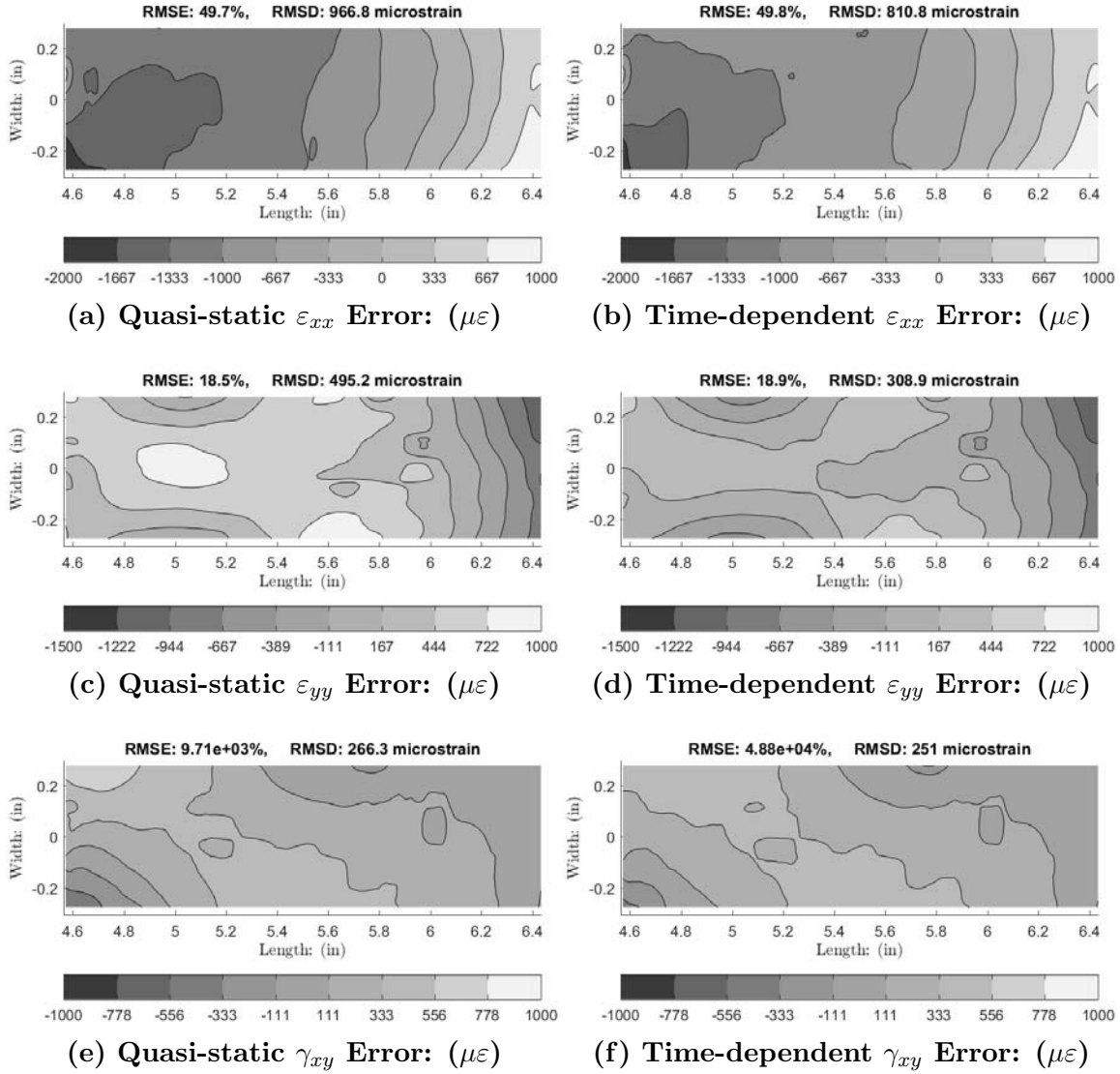


Figure 4.28. Optimized DIC Field vs. FEM Strain Error, Unloaded State

upward trend in RMSE. Nevertheless, the final errors are both around 7.5%, as described above, which shows good agreement between experimental and computational strain fields. The errors in ϵ_{xx} are still fairly large, but it will be explained at the end of this chapter (and more directly in Chapter V) why that is less of a concern.

4.4.1.2 Optimized Loading, Zero Load

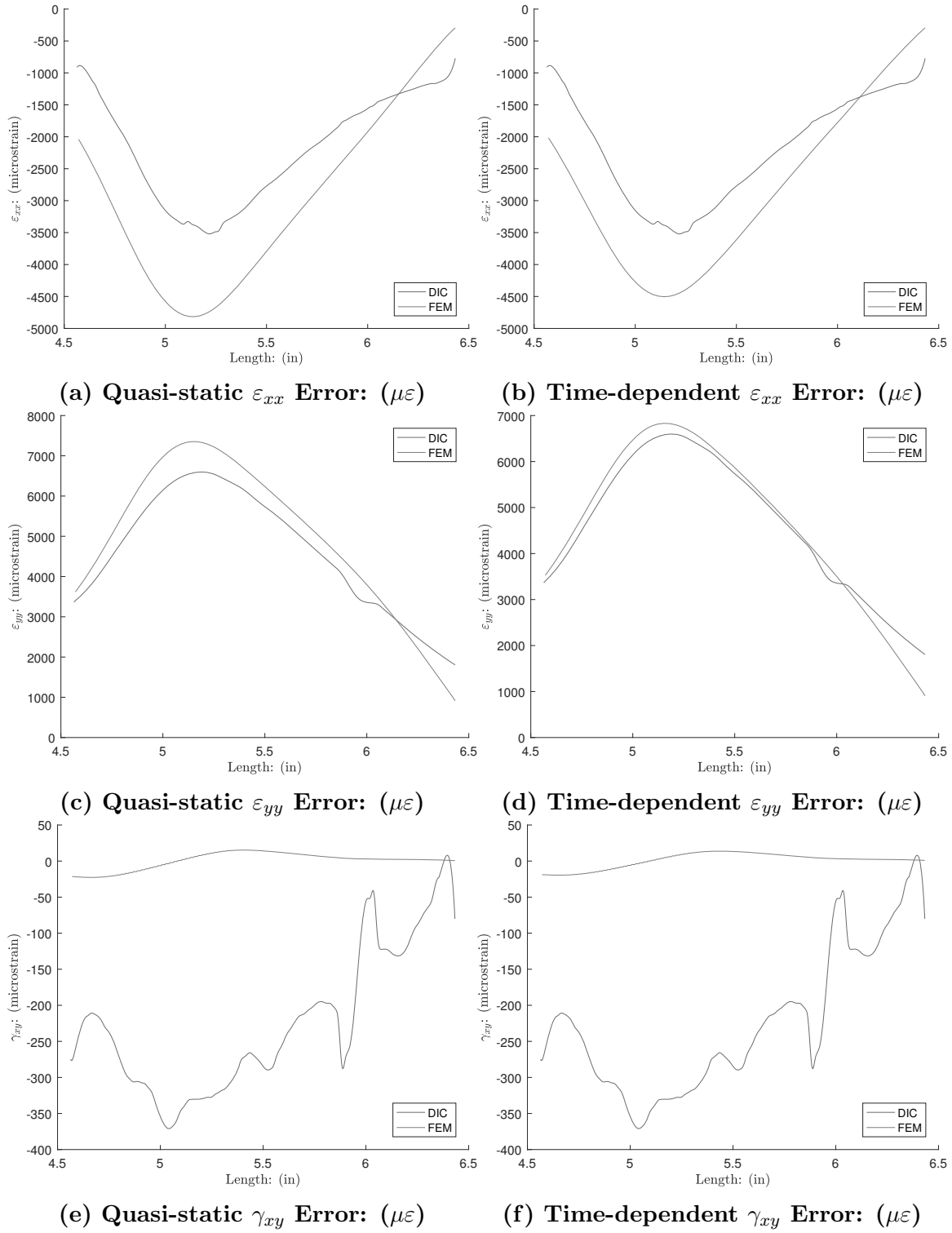


Figure 4.29. Error Along Plate Centerline, Unloaded State

Table 4.8. Error Assessment of FEM Analysis Types

Analysis Type (Results: %)	Loaded RMSE			Unloaded RMSE		
	ε_{xx}	ε_{yy}	γ_{xy}	ε_{xx}	ε_{yy}	γ_{xy}
Quasi-static	36.2	19.2	67,700	45.7	33.2	20,100
Time-dependent	33.8	16.1	8,790	42.7	28.5	20,800
Opt Quasi-static	25.1	7.23	7,890	49.7	18.5	9,710
Opt Time-dependent	23.3	7.57	34,600	49.8	18.9	48,800
DIC RMSE	7.09	4.46	10,200	13	10.1	1,540

4.5 Relationship of Strain Error and Stress Predictions

This section summarizes the preceding comparisons of DIC measurements to the various FEM solutions into condensed tables of RMSE and RMSD to make more generalized statements regarding to FEM accuracy in strain, which in turn raises confidence in predicted residual stress due to experimentally verified constitutive relationships. Table 4.8 assesses RMSE, and shows that the original time-dependent model has a better prediction of strain than the quasi-static model in all directions (note that errors in γ_{xy} are large due to the very low magnitude of strain in both prediction and experimentation, and as such they will be neglected from most discussion). The RMSE grows in the unloaded state, however this error is normalized to the magnitude of FEM strain, and Table 4.9 shows that the offset in units of microstrain (therefore no normalization according to Equation 2.16) shows that the offset in strain is relatively constant between the loaded and unloaded states.

Table 4.9. Deviation Assessment of FEM Analysis Types

Analysis Type (Results: microstrain)	Loaded RMSD			Unloaded RMSD		
	ε_{xx}	ε_{yy}	γ_{xy}	ε_{xx}	ε_{yy}	γ_{xy}
Quasi-static	2490	2420	519.3	2096	2557	354.8
Time-dependent	2228	1918	485.2	1830	2025	316.5
Opt Quasi-static	1274	359.1	422.9	966.8	495.2	266.3
Opt Time-dependent	1114	422.5	407.5	810.8	308.9	251
DIC RMSD	240.6	377.1	57.87	250.6	376.1	52.68

Because the strain offset is constant, the error in strain is predominantly an error in plastic strains. These plastic strains were reduced by diminishing the tensile loading in the FEM solutions by 4% to better match the total strain measured by DIC for the “optimized” cases. It is clear from both tables that both the RMSE and RMSD has been reduced in the optimized solutions, and the ε_{yy} deviation is nearly within the experimental variation exhibited in DIC measurements between specimens. This deviation is, once again, relatively constant between loaded and unloaded states, but less than $\frac{1}{4}$ that predicted by the original models, from a decrease in tensile load of only 4%. The focus of the results has been almost exclusively on strain, due to the availability of *in situ* experimental results through DIC, but the ultimate goal is more accurate predictions of residual stress.

The predicted contours of residual stress for the original quasi-static and time-dependent models were seen earlier (Figures 4.9d & 4.9g and 4.13d & 4.13g, respectively), but the error in stress is difficult to measure experimentally. The validity of stress predictions is instead applied as a direct result of matching strain distributions between experimental and FEM solutions. Thus, the most accurate predictions of residual stress are made from the optimized solution methods, with results seen in Figure 4.30. While these solutions account for stresses in all directions, it is only the stresses at the fatigue failure location that are of major importance. At this location (6.5 in from the origin along the plate centerline) the only stresses that exist are in the vertical direction, so only σ_{yy} and von Mises stress are shown. The regions of residual compression between the grips are due to the elastic unloading and previously generated plastic deformation, and the regions of residual tension are a result of equilibrium within the specimen. Ideally, the tensile region would be consistent with the free end of the plate, but lowering the input tensile force by 4% has driven this region slightly to the left (as compared with Figures 4.9d and 4.13d).

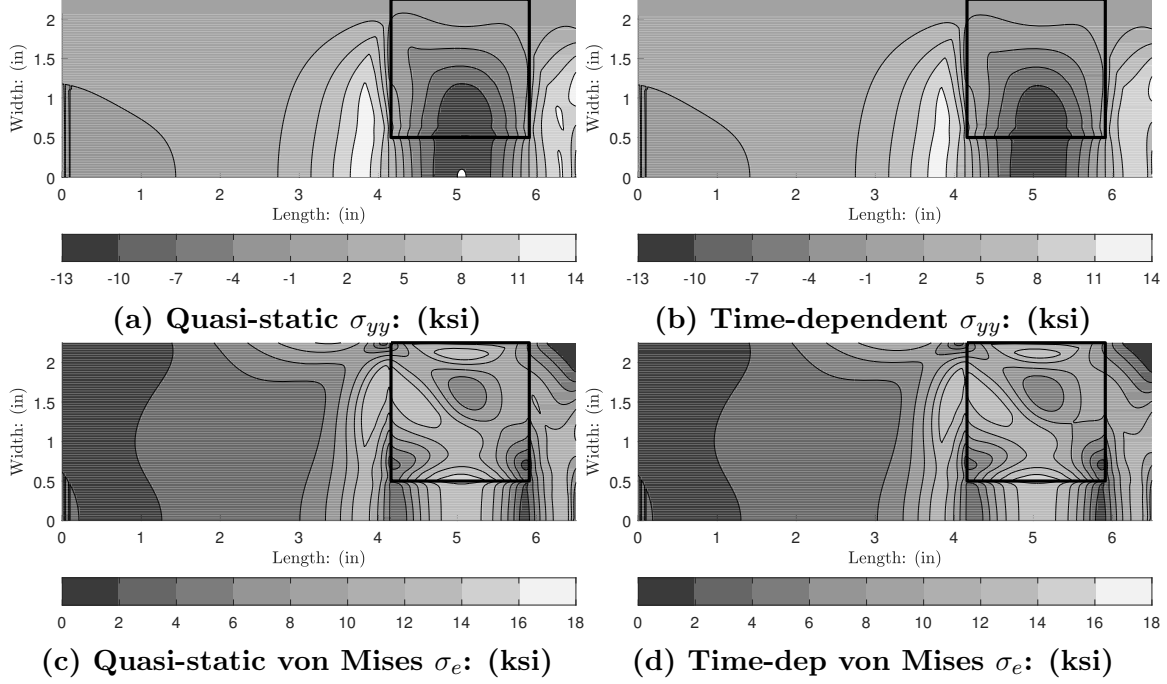


Figure 4.30. Optimized Residual σ Results, Unloaded State

The final results of residual stress, and the associated strain field errors and deviations are shown in Table 4.10, with stresses reported at the failure location under HCF loading. It is clear that the original analyses predicted a larger residual stress (10.72 ksi or 10.89 ksi) due to the high tensile region being closer to the free edge, but the strains are offset by over 2000 microstrain, with RMSEs of roughly 30%. The more favorable FEM analyses to use would be those optimized to match the strain field of the experiment, which predict a reduction of residual stress by over 50%. The utility in these residual stress approximations is large when combined with fatigue

Table 4.10. Residual Stress Results at HCF Failure Location

Analysis Type (residual)	RMSE (%)	RMSD (microstrain)	Stress σ_{yy} : (ksi)
Quasi-static	33.2	2557	10.72
Time-dependent	28.5	2025	10.89
Opt Quasi-static	18.5	495.2	4.5
Opt Time-dependent	18.9	308.9	4.93
DIC	10.1	376.1	—

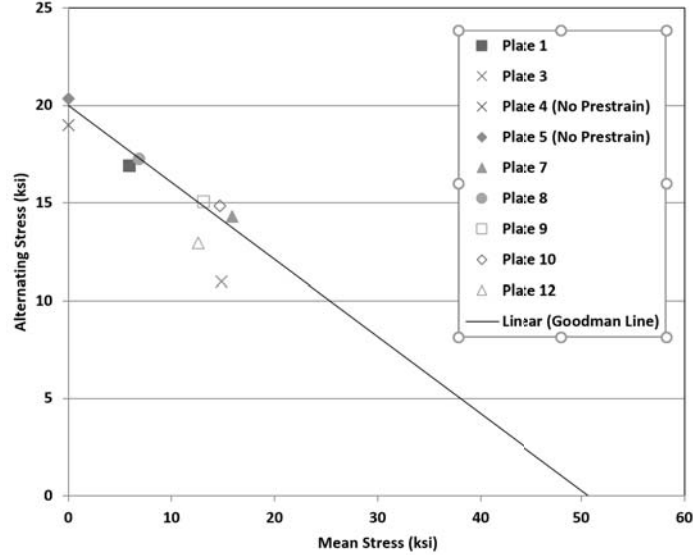


Figure 4.31. Updated Vibration-based Fatigue 10^6 failure cycles Goodman line

failure to assess component life under identical loading scenarios for more complex geometries such as airfoils in gas turbine engines (GTEs).

4.5.1 Goodman Line Reassessment

For the sake of consistency, the work presented in this research has always been in reference to specimens before and after 15,000 lb_f of loading, allowing repeatable assessment of experimental variation and the agreement of various FEM solutions with empirical strain results. However, the product of the improved FEM solutions is improved approximation of residual stress for the Vibration-based Fatigue Test at varying levels of alternating stress. Because the experimental alternating stress is already known for a subset of samples from previous work, the prediction of residual stress can be reassessed from that data set [17].

Table 4.11 shows several of the reported residual stress points from Figure 4.31, as well as the updated approximations when run through the quasi-static FEM procedure developed in this research. Plate 3 is neglected because it was an improper test (and thus marked as X), while plates 4 and 5 have no residual stress, and plate 12 was run

Table 4.11. Goodman Residual Stress Results at HCF Failure Location

Specimen	Old σ_{yy}^{res} (ksi)	Updated σ_{yy}^{res} (ksi)	Offset (%)
Plate 1	4.957	5.946	19.95
Plate 7	14.981	15.874	5.96
Plate 8	5.404	6.827	26.33
Plate 9	12.877	13.096	1.70
Plate 10	14.637	14.716	0.54

with a different grip location than the rest of the results. The results show a change in as much as 26% in reported residual stress when run through the updated FEM solution procedure, which is of a much higher mesh density (207,900 elements vs. 6,600) and was previously shown to converge better to strain results. This improved estimate validates the ability of using the Vibration-based Fatigue Test with existing steady stress, thus generating a modified Goodman line for more representative life approximations of critical GTE components, as introduced in Chapter I.

4.6 Chapter Summary

This chapter put to use the theoretical background of Chapter II via the experimental methodology in Chapter III to generate the results desired in this research effort. Specimen geometry was reported, as well as several scripts between MATLAB and ANSYS to properly model the experiment and make direct comparisons between empirical and computational results. Extensive results of stress and strain were reported in both experimental and computational results, with qualitative and quantitative comparisons throughout the entire load history. Finally, the residual stresses predicted at the region of HCF failure were reported for application as steady stress on a Goodman Diagram. The following chapter will summarize the conclusions derived from this chapter and tie them to the original scope of this work, as well as propose recommendations for future work and points of major impact to the Pre-strain Method.

V. Conclusions and Recommendations

5.1 Review of Research Objectives

The goal of this research was to determine the fidelity of mean stress approximations from specimen yielding and subsequent generation of Goodman data after fatigue testing. The results increased the dependability of that data for load cases of turbine engine fatigue life estimation, while gathering results 40X faster than conventional testing via the Vibration-based Fatigue Test Method. In support of these assessments, this research focused on the following objectives:

- Establish more representative boundary conditions of finite element model (FEM) load distributions to better approximate experimental specimen yielding
- Utilize digital image correlation (DIC) strain measurements to assess FEM validity throughout entire measurable strain field [10]
- Create a time-dependent FEM to more accurately represent development of plastic deformation, and thus generate more accurate predictions of residual stress and strain

While multiple samples were run to verify optimum wedge type and supply pressure in the Mechanical Test Systems (MTS) grips, the research focused on the results of three plates of aluminum (Al) 6061-T6 and six dogbones extracted from the same stock material.

The first objective was met through the application of the grip reaction forces from a preliminary FEM solution to the subsequent plate FEM model. Initially the plate-grip assembly was modeled as a single model with grip volumes meshed with material properties of hardened steel and the plate consisting of the experimentally

determined material properties from dogbones. This produced regions of unrealistic stresses along the edges of the grip due to discontinuous geometry and material properties, and resulted in localized plastic deformation that would not manifest experimentally. Instead, the reaction forces from the grip alone produced a force distribution with a root mean squared error (RMSE) of 19% compared to the previously assumed even compressive force distribution along the grip surface. These forces were read into MATLAB using `Data_Extractor_Grip.m` and written to an ANSYS input script using `Forces_Compression.m`.

The second objective was accomplished through the majority of Chapter IV to determine experimental variation between DIC results, as well as rank which FEM procedures were more representative of experimental loading. Initially, comparisons between FEM and DIC results were qualitative to show that the predicted strain profiles were similar to experimental results, using contours of three planar strain components and plots along the plate centerline. Due to similarity between FEM solutions, RMSE and root mean squared deviation (RMSD) were the most useful figures of merit to quantify which strain fields were more consistent with experiment, the latter of which was assessed for ε_{yy} with respect to controlled loading, reported every 100 lb_f of tensile load. In order to properly align the predicted and experimental strain fields, alignment features were created and located with a high power microscope and FEM data from the half plate simulation had to be rotated about the horizontal axis.

The third objective was achieved by means of an experimentally derived quasi-static stress-strain curve and the Perzyna Hardening model. In the event that the specimen had no regions of loading that exceeded the quasi-static rate ($5.6 \times 10^{-5} \text{ s}^{-1}$) the results of the time-dependent model would be identical to the quasi-static solution. However, this is not the case at the end of the load ramp generating plastic

deformation, and as such, the quasi-static solution RMSE rises in this regime. Nevertheless, after holding the specimen at a constant load of 15000 lb_f for 50 minutes, the two solution procedures generate results that are of negligible difference. The rate-dependent model negates the need for the 50 minute hold step for accurate prediction of residual stress, which could reduce the Pre-strain test time by half if desired.

5.2 Recommendations for Future Work

5.2.1 Experimental Stress Measurement

This research was largely in support of more accurate approximations of residual stress by means of FEM solutions that matched experimental strains with appropriate material properties. Predicting residual stress in this method allows immediate transition to the Vibration-based Fatigue Test, but there is still no experimental validation of the stress existing in the specimen. Non-destructive stress measurement is possible via X-Ray diffraction or neutron diffraction, which is of no detriment to the subsequent vibration test, and destructive methods exist which measure elastic relaxation when holes or channels are cut from a material with existing residual stress. Any of these methods would provide further insight to the accuracy of the FEM stress approximations, either verifying that the FEM results of this research are sufficient or indicating that further study is necessary.

5.2.2 Representative Gas Turbine Engine Materials

The Vibration-based Fatigue Test was developed to gather fatigue data on specimen geometries that are more representative of gas turbine engine (GTE) airfoils, but more affordable to manufacture than actual components. Gathering of Goodman data with existing states of steady stress requires the Pre-strain Method, but thus far it has only been demonstrated for Al components as a proof of concept. The test

method is now sufficiently developed that it should be run on more representative materials such as titanium (Ti) or nickel (Ni) alloys to assess how the Goodman predictions would change life predictions of real hardware. Due to the higher yield stress of these materials, the tensile load required to generate sufficient plastic deformation is greater, and as such would require a larger load frame than that utilized in this study.

5.2.3 Lens Dust Reduction

After extensive analysis of the DIC results, several particles of dust were discovered in the region of interest that were skewing the calculated strains. The dust was originally believed to have been on the surface of the camera sensor, but later found to have been in the lens. These particles remain stationary against a deforming speckle pattern on the specimen, which is correlated to fictitious regions of concentrated strain. The filter size of 35 pixels was chosen because it was sufficient to dissipate these fictitious strains, but at the loss of spatial resolution. It is recommended that the Pre-strain Method be run with a lens setup without visible dust, affording the capability of smaller filter sizes and higher spatial resolution. The large-scale strain profile changes with respect to the specimen geometry will be predominantly the same as what was found in this research, but should be proven nonetheless.

5.2.4 Further Improved Boundary Conditions

While improvements were made to the boundary conditions by extracting reaction forces from the grip FEM model, this is still not a truly accurate representation of the experimental loading. The contours of von Mises stress greater than yield stress were extracted from Figure 4.9g and overlayed on Figure 4.15 to show regions of plastic deformation. It is clear that this prediction leads to plastic deformation under almost

$\frac{1}{3}$ of the grip surface, when in reality much of this material would not be free to move due to contact with the grip wedges. A more robust set of boundary conditions would take into account the diamond-faced wedge surface geometry and localized indentation on the plate specimen due to both compressive and tensile loading. This model would in turn produce plastic strain distributions that match the experimental strains calculated by DIC, and would negate the need to lower the total tensile force by 4% in order to “optimize” the FEM solution.

5.3 Discussion of Results

5.3.1 DIC Experimental Noise

Three tests were run under the exact same load control procedure in Multi-purpose Testware (MPT) to a peak load of 15,000 lb_f, with a 50 minute hold period before unloading. All specimens were of the same geometry (± 0.005 in) and speckle patterns were applied in the same manner. Between specimens, there was an offset in RMSD of 250 microstrain in ε_{xx} , 377 microstrain in ε_{yy} , and 58 microstrain in γ_{xy} that was relatively constant in the loaded and unloaded states. This gives a reasonable assessment of the standard deviation of the three planar strain components between tests, which accounts for both experimental inconsistencies and noise in the DIC results.

5.3.2 Time-dependent vs. Quasi-static FEM Models

A time-dependent FEM model was developed in conjunction with the two-ramp load control setup in MPT to assess the material response based on overstress from the equilibrium stress-strain curve. The behavior in overstress (non-equilibrium) was modeled based on the Perzyna Hardening Model, as approximated by Andrzej Mróz based on experimental data gathered by Toshiji Mukai for varying strain rates on Al

6060 alloys. The time-dependent solution consisted of 203 loadsteps, with a RMSD in ε_{yy} of 1918 microstrain at the maximum load after a 50 minute hold, while the two-step quasi-static solution resulted in a deviation of 2420 microstrain.

The utility of the time-dependent model is more clear when the RMSE is assessed across the entire load history, where the error climbs for the quasi-static solution due to dynamic strain behavior, but remains constant for the time-dependent Perzyna Hardening solution. If shorter test times were required for the Pre-strain Method, the time-dependent model negates the need for the 50 minute hold, and would also model plastic deformation closely if the load control ramps were faster as well. Ultimately, the time-dependent FEM solution had an RMSE of 28.5% in the residual state while the quasi-static solution had an error of 33.2%, but the RMSD remained relatively constant between loaded and unloaded states. This implies that the overprediction of strain predicted by FEM analyses is largely plastic, and as such are more closely related to driving higher error in residual stress. The reported residual stress at the free end of the specimen along the plate centerline was 10.72 ksi and 10.89 ksi for the quasi-static and time-dependent FEM models, respectively.

5.3.3 Optimized Strain Matching of FEM Models

Both the quasi-static and time-dependent FEM solutions were reassessed after decreasing the maximum load to 14,400 lb_f, to reduce the amount of plastic strain in the specimen to more closely match the experimental strains from DIC. The load was proportionally lowered by 4% for the entire load history and assessed against the true experimental load curve, which indicated lower values of RMSE in ε_{yy} for the entire test for both models. This indicates that the strains were more closely represented even before the onset of plastic deformation, when the response is fully elastic, supporting the previous recommendation of false boundary conditions driving

errors between FEM and DIC. The final RMSDs in ε_{yy} were 495.2 microstrain and 308.9 microstrain for the quasi-static model and time-dependent model, respectively, which was only 15% and 19% of the RMSD values reported at 15,000 lb_f. The reported residual stress for these models were 4.50 ksi (quasi-static) and 4.93 ksi (time-dependent), which is a more accurate residual stress, validated by RMSDs that are roughly the same as the 376.1 microstrain experimental deviation between specimens with identical loading.

5.3.4 Improved Residual Stress Impacts on Goodman Data

In an effort to be more consistent in comparisons between FEM and experimental DIC measurements, all specimens were loaded to 15,000 lb_f experimentally, as well as computationally (except in the case of the optimized conditions with tensile forces reduced by 4%). However, the product of improved FEM solutions modeling the Pre-strain procedure is a more accurate approximation of residual stress. This stress is present as a constant stress during the Vibration-based Fatigue Test at varying levels of alternating stress, and can be used to generate Goodman data. While performing the experimental vibration tests was not the focus of this research, known values of alternating stress with pre-existing levels of residual stress were reassessed with the updated FEM solutions. The largest offset in predicted residual stress at the high cycle fatigue (HCF) failure location was 26.44% for a sample with 6.827 ksi of residual stress when compared to the old model, but other samples saw changes of less than 1% in residual stress.

Appendix A. ANSYS Script

```
finish
/clear,start

! Ensure Data is Saved to Desired Working Directory
/CWD,'C:\Users\1379789559A\Desktop\Time Dependence\Auto Step'
/FILNAME,Plate_Project,0
/TITLE,Plate_Project

!!! User inputs

! 6061 T6 Aluminum Material Properties
Youngs_mod    = 10.0e6 ! psi          (10e6) Input from Dogbone Data
Poissons_ratio = 0.33    !            (0.33)
rho            = 0.098   ! lbf/in^3 (0.16)

! Define Geometry and Elemental Divisions (Mesh Density Multipliers Given)
length         = 6.5      ! in        (6.5)
length_elem    = 330      !        (55) times 6
width          = 2.25     ! in        (2.25)
width_elem     = 105      !        (35) times 3
thickness      = 0.121    ! in        (0.125)
thickness_elem = 6        !        (4) times 1.5
gap_elem       = 30       !        (5) times 6
gap            = length*gap_elem/length_elem
grip_len_elem  = 90       !        (15) times 6
grip_len       = length*grip_len_elem/length_elem
grip_wid_elem  = 81       !        (27) times 3
grip_wid       = width*grip_wid_elem/width_elem

! Script is written such that Nodes 1 through grip_nodes belong to grip surfaces
grip_nodes     = (grip_len_elem+1)*(grip_wid_elem+1)

! Total Force Applied
r1_load = 7500          ! lbf          (7500)
r2_load = 15000         ! lbf          (15000)
!pressure_force = 24720 ! lbf          (24720)

! Total Time Applied
r1_time = 600           ! sec         (600)
r2_time = 4200          ! sec         (4200)
h_time  = 7200          ! sec         (7200)
dt = 1                  ! sec Time Step

! Number of Ramp Steps
```

```

r1_n   = 75           !           (75)
r2_n   = 75           !           (75)
h_n    = 50           !           (50)

```

```

!!! Preprocessing
/PREP7

```

```

! Define element type
ET,1,SOLID185

```

```

! Input Material Properties
/INPUT,'Script_Dogbone_1-2','txt','.'

```

```

! Generate Plate Mesh
/INPUT,'Plate_Mesh','txt','.'

```

```

!!!! Load Steps
FINISH
/SOL

```

```

! Transient Analysis
ANTYPE,4
!*
TRNOPT,FULL
LUMPM,0
!*

```

```

!!! Boundary Conditions
! No Displacement Along Bottom Areas in Y
FLST,2,3,5,ORDE,3
FITEM,2,15
FITEM,2,19
FITEM,2,23
!*
/GO
DA,P51X,UY,0

```

```

! Symmetric BC on Bottom Areas
FLST,2,3,5,ORDE,3
FITEM,2,15
FITEM,2,19
FITEM,2,23
DA,P51X,SYMM

```

```

! No Z displacement on lower bottom lines
FLST,2,3,4,ORDE,2
FITEM,2,25
FITEM,2,-27
!*
/GO
DL,P51X, ,UZ,0

! No X displacement along bottom leading edge
FLST,2,1,4,ORDE,1
FITEM,2,38
!*
/GO
DL,P51X, ,UX,0

!!! Initial Compressive Loads (Use ! to Toggle Comp Force Dist)
/INPUT,'Compressive_Force','txt','.'

! Top Grip Force
FLST,2,grip_nodes,1,ORDE,2
FITEM,2,1
FITEM,2,-grip_nodes
!*
/GO
!F,P51X,FZ,-pressure_force/grip_nodes

! Bottom Grip Force
FLST,2,grip_nodes,1,ORDE,2
FITEM,2,grip_nodes+1
FITEM,2,-2*grip_nodes
!*
/GO
!F,P51X,FZ,pressure_force/grip_nodes

! Time Step Settings
NLGEOM,1
DELTIM,10,dt,10
AUTOTS,1
KBC,0.0
RATE,1
TIME,1
TIMINT,1
!MIDTOL,1,0,0

! Save Load Step
LSWRITE,1,

!!! Ramp 1 Loads (Looped for r1_n Steps)

```

```

*do,ii,1,r1_n,1

tensile_force = r1_load*ii/r1_n
t = r1_time*ii/r1_n

! Tensile Load
FLST,2,2*grip_nodes,1,ORDE,2
FITEM,2,1
FITEM,2,-2*grip_nodes
!*
/GO
F,P51X,FY,tensile_force/(2*grip_nodes)

! Time Step Settings
NLGEOM,1
DELTIM,10,dt,10
AUTOTS,1
KBC,0.0
RATE,1
TIME,t
TIMINT,1

! Save Load Step
LSWRITE,1+ii,

*enddo

!!! Ramp 2 Loads (Looped for r2_n Steps)
*do,ii,r1_n+1,r1_n+r2_n,1

tensile_force = r2_load*ii/(r1_n+r2_n)
t = (r2_time-r1_time)*(ii-r1_n)/(r2_n)+r1_time

! Tensile Load
FLST,2,2*grip_nodes,1,ORDE,2
FITEM,2,1
FITEM,2,-2*grip_nodes
!*
/GO
F,P51X,FY,tensile_force/(2*grip_nodes)

! Time Step Settings
NLGEOM,1
DELTIM,10,dt,10
AUTOTS,1
KBC,0.0
RATE,1
TIME,t
TIMINT,1

! Save Load Step

```

```

LSWRITE,1+ii,

*enddo

!!! Hold Loads (Looped for h_n Steps)
*do,ii,r1_n+r2_n+1,r1_n+r2_n+h_n,1

t = (h_time-r2_time)*(ii-r1_n-r2_n)/(h_n)+r2_time

! Time Step Settings
NLGEOM,1
DELTIM,10,dt,10
AUTOTS,1
KBC,0.0
RATE,1
TIME,t
TIMINT,1

! Save Load Step
LSWRITE,1+ii,

*enddo

! Final Compressive Loads (Use ! to Toggle Comp Force Dist)
FDELE,ALL,ALL
/INPUT,'Compressive_Force','txt','.'

! Top Grip Force
FLST,2,grip_nodes,1,ORDE,2
FITEM,2,1
FITEM,2,-grip_nodes
!*
/GO
!F,P51X,FZ,-pressure_force/grip_nodes

! Bottom Grip Force
FLST,2,grip_nodes,1,ORDE,2
FITEM,2,grip_nodes+1
FITEM,2,-2*grip_nodes
!*
/GO
!F,P51X,FZ,pressure_force/grip_nodes

! Time Step Settings
NLGEOM,1
DELTIM,10,dt/100,10
AUTOTS,1
KBC,0.0
RATE,1
TIME,t+100

```

```

TIMINT,1

! Save Load Step
LSWRITE,2+ii,

! Final Residual State
FDELE,ALL,ALL

! Time Step Settings
NLGEOM,1
DELTIM,10,dt/10,10
AUTOTS,1
KBC,0.0
RATE,1
TIME,t+111
TIMINT,1

! Save Load Step
LSWRITE,3+ii,

!!! Solve Load Steps
LSSOLVE,1,3+ii,1,

!!!!Data Output to Text File
/post1
SAVE
/graph,power      ! permit midside node
/efacet,2         ! results if available

! Extend all numerical output so there are no concatenations
/FORMAT,,17,9

! Delete header
/HEADER,ON,ON,ON,OFF,ON,OFF

! Suppress repeating header in output files
/PAGE,999999999,,999999999

! Select all nodes
NSEL,ALL

! Set Grip Step (1)
tensile_force = 0
SET,,,,,1

! Output nodal coordinates
/OUTPUT,%tensile_force%_LS1_COORDINATES.txt
NLIST,ALL, , ,XYZ,NODE,NODE,NODE
/OUTPUT,TERM

```

```

! Output stress to file
/OUTPUT,%tensile_force%_LS1_NORMAL_STRESS,txt
PRNSOL,S,COMP
/OUTPUT,TERM

! Output strain to file
/OUTPUT,%tensile_force%_LS1_EL_NORM_STRAIN,txt
PRNSOL,EPTO,COMP
/OUTPUT,TERM

! Set Load steps (1+ii)
*do,ii,1,r1_n+r2_n,1

tensile_force = r2_load*ii/(r1_n+r2_n)

SET,,,,,,1+ii

! Output stress to file
/OUTPUT,%tensile_force%_LS%1+ii%_NORMAL_STRESS,txt
PRNSOL,S,COMP
/OUTPUT,TERM

! Output strain to file
/OUTPUT,%tensile_force%_LS%1+ii%_EL_NORM_STRAIN,txt
PRNSOL,EPTO,COMP
/OUTPUT,TERM

! Plot Results in png
/DSCALE,ALL,1
PLNSOL, S,Y, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,max_stress_%tensile_force%,png
/delete,Plate_Project000,png
PLNSOL, S,EQV, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,max_mises_stress_%tensile_force%,png
/delete,Plate_Project000,png
/EFACET,1
PLNSOL, EPTO,Y, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,max_strain_%tensile_force%,png
/delete,Plate_Project000,png
! Plot Y-Strain Along Bottom Edge
FLST,2,2,1
FITEM,2,172775
FITEM,2,223343
!*

```

```

PATH,Bottom,2,30,20,
PPATH,P51X,1
PATH,STAT
!*
AVPRIN,0, ,
!*
PDEF, ,EPT0,Y,AVG
/PBC,PATH, ,0
!*
PLPATH,EPT0Y
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,Mstrain_profile_%tensile_force%,png
/delete,Plate_Project000,png

*enddo

! Set Hold Steps (1+ii)
*do,ii,r1_n+r2_n+1,r1_n+r2_n+h_n,1
SET,,,,,,,,1+ii

! Output stress to file
/OUTPUT,%tensile_force%_LS%1+ii%_NORMAL_STRESS,txt
PRNSOL,S,COMP
/OUTPUT,TERM

! Output strain to file
/OUTPUT,%tensile_force%_LS%1+ii%_EL_NORM_STRAIN,txt
PRNSOL,EPT0,COMP
/OUTPUT,TERM

! Plot Results in png
/DSCALE,ALL,1
PLNSOL, S,Y, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,max_stress_%tensile_force%,png
/delete,Plate_Project000,png
PLNSOL, S,EQV, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,max_mises_stress_%tensile_force%,png
/delete,Plate_Project000,png
/EFACET,1
PLNSOL, EPT0,Y, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,max_strain_%tensile_force%,png
/delete,Plate_Project000,png
! Plot Y-Strain Along Bottom Edge
FLST,2,2,1

```

```

FITEM,2,172775
FITEM,2,223343
!*
PATH,Bottom,2,30,20,
PPATH,P51X,1
PATH,STAT
!*
AVPRIN,0, ,
!*
PDEF, ,EPT0,Y,AVG
/PBC,PATH, ,0
!*
PLPATH,EPT0Y
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,Mstrain_profile_%tensile_force%,png
/delete,Plate_Project000,png

*enddo

! Set Grip Unload Step (2+ii)
SET,,,,,,2+ii

! Plot Results in png
/DSCALE,ALL,1
PLNSOL, S,Y, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,resid_stress_%tensile_force%,png
/delete,Plate_Project000,png
PLNSOL, S,EQV, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,resid_mises_stress_%tensile_force%,png
/delete,Plate_Project000,png
/EFACET,1
PLNSOL, EPT0,Y, 0,1.0
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,resid_strain_%tensile_force%,png
/delete,Plate_Project000,png
! Plot Y-Strain Along Bottom Edge
FLST,2,2,1
FITEM,2,172775
FITEM,2,223343
!*
PATH,Bottom,2,30,20,
PPATH,P51X,1
PATH,STAT
!*
AVPRIN,0, ,

```

```

!*
PDEF, ,EPTO,Y,AVG
/PBC,PATH, ,0
!*
PLPATH,EPTOY
/ui,raise
/ui,copy,save,png,,,portrait,yes
/copy,Plate_Project000,png,,Rstrain_profile_%tensile_force%,png
/delete,Plate_Project000,png

! Output stress to file
/OUTPUT,%tensile_force%_LS%2+ii%_NORMAL_STRESS,txt
PRNSOL,S,COMP
/OUTPUT,TERM

! Output strain to file
/OUTPUT,%tensile_force%_LS%2+ii%_EL_NORM_STRAIN,txt
PRNSOL,EPTO,COMP
/OUTPUT,TERM

! Set Final Unload Step (3+ii)
SET,,,,,,3+ii

! Output stress to file
/OUTPUT,%tensile_force%_LS%3+ii%_NORMAL_STRESS,txt
PRNSOL,S,COMP
/OUTPUT,TERM

! Output strain to file
/OUTPUT,%tensile_force%_LS%3+ii%_EL_NORM_STRAIN,txt
PRNSOL,EPTO,COMP
/OUTPUT,TERM

```

Appendix B. MATLAB Scripts

B.1 Data Extraction

B.1.1 Plate Data Extractor

```
1 %%
2 clear all; close all; clc;
3 %%
4
5 % Establish Pressure and Load Index Matrices
6 P_index = [];
7 T_index = [];
8
9 % Select .txt Files to be Read
10 [file,path] = uigetfile({'*.txt'}, 'MultiSelect', 'on');
11
12 %Ensures single file still formatted as 1x1 cell
13 if iscell(file),
14     file = file;
15 else
16     file = cellstr(file);
17 end
18
19 % Determine Number of Files
20 num_files = numel(file);
21
22
23 % Determine Number of Lines in First File
24 txt=fileread(strjoin(file(1,1)));
25 n_line = sum(txt==10)+1;
26
27
28
29 %%
30 for jj = 1:num_files
31
32     % Open file(jj)
33     fid_mat=fopen(char(file(jj)),'r');
34
35     % Reads load from filename
36     name = char(file{jj});
37     for ii = 1:7 % Determine number of digits in load
38         textscan(name(ii),'%s');
39         if strcmp(strjoin(ans{1}),'_')==1
40             n_l = ii-1;
41             break
42         else
```

```

43         end
44     end
45     load = textscan(name(1:n_l), '%s');
46     load = char(load{1}); %char
47
48     % Reads loadstep from filename
49     for ii = n_l+1:n_l+7 % Determine number of digits in loadstep
50         textscan(name(ii), '%s');
51         if strcmp(strjoin(ans{1}), '_')==1
52             n_ls = ii-1;
53         else
54             end
55     end
56     loadstep = textscan(name(n_l+2:n_ls), '%s');
57     loadstep = char(loadstep{1}); %char
58
59     % Populates Load Index
60     if strcmp(loadstep, ('LS0'))==1
61         L_type = 'P';
62         P_index = [P_index; str2num(load)];
63     elseif strcmp(loadstep, ('LS1'))==1
64         L_type = 'T';
65         T_index = [T_index; str2num(load)];
66     else
67         L_type = 'T';
68         T_index = [T_index; str2num(load)];
69     end
70
71     % Reads properties from filename
72     endname = strfind(file{jj}, '.txt')-1;
73     property = textscan(name(n_ls+2:endname), '%s');
74     property = char(property{1}); %char
75
76     % Identifies type of data and format
77     if strcmp(property, ('EL_NORM_STRAIN'))==1;
78         dat_type = 'STRAIN';
79         FormatSpecS = '%s %s %s %s %s %s %s';
80         FormatSpecF = '%f %f %f %f %f %f %f';
81         junk_headers = 8;
82     elseif strcmp(property, ('NORMAL_STRESS'))==1;
83         dat_type = 'STRESS';
84         FormatSpecS = '%s %s %s %s %s %s %s';
85         FormatSpecF = '%f %f %f %f %f %f %f';
86         junk_headers = 8;
87     elseif strcmp(property, ('COORDINATES'))==1;
88         dat_type = 'COORDINATES';
89         FormatSpecS = '%s %s %s %s';
90         FormatSpecF = '%f %f %f %f';
91         junk_headers = 4;
92     else
93         dat_type = 'UNKNOWN';
94     end

```

```

95
96
97 %Skips junk headers
98 for ii = 1:junk_headers
99     out=fgetl(fid_mat);
100     if isequal(out,' Memory resident data base increased from ...
        1024 MB to      2048 MB.') == 1;
101         for kk = 1:junk_headers-2
102             out=fgetl(fid_mat);
103         end
104         break
105     elseif isequal(out,' Memory resident data base increased ...
        from      2048 MB to      4096 MB.') == 1;
106         for kk = 1:junk_headers-2
107             out=fgetl(fid_mat);
108         end
109         break
110     elseif isequal(out,' Memory resident data base increased ...
        from      4096 MB to      8192 MB.') == 1;
111         for kk = 1:junk_headers-2
112             out=fgetl(fid_mat);
113         end
114         break
115     elseif isequal(out,' PRINT EPTO NODAL SOLUTION PER NODE') == 1;
116         out=fgetl(fid_mat);
117         out=fgetl(fid_mat);
118     elseif isequal(out,' PRINT S      NODAL SOLUTION PER NODE') == 1;
119         out=fgetl(fid_mat);
120         out=fgetl(fid_mat);
121     end
122 end
123
124 %Reads headers into cell
125 out=fgetl(fid_mat);
126 headers = textscan(out,FormatSpecS);
127 num_head = numel(headers);
128 for ii = 1:num_head
129     headers{ii} = char(headers{ii});
130 end
131
132 Nodes = n_line;
133
134 %Reads file and outputs data matrix
135 data = cell(1,num_head);
136 if (exist('Tc') == 1) && (strcmp(dat_type,'COORDINATES'))
137     % Ignore subsequent coordinate files to save time
138 else
139     for ii = 1:Nodes+junk_headers
140         out = fgetl(fid_mat);
141         if out == -1;
142             break

```

```

143         elseif isequal(out,' NODAL RESULTS ARE FOR MATERIAL 2 ...
144                                     ') == 1;
145         break
146         elseif isequal(out,'') == 1;
147             % Ignores repeating headers
148         else
149             data(ii,:) = textscan(out,FormatSpecF);
150         end
151     end
152
153     if strcmp(dat_type,'UNKNOWN')
154         % Ignore unknown data types
155
156
157         % Process Stress or Strain Data
158     elseif strcmp(dat_type,'COORDINATES')-1
159         %Put data into table and store table in structure
160         data = cell2mat(data);
161         T = array2table(data,'VariableNames',headers);
162         if exist('Tc')==1
163             T = array2table([zeros(length(T.NODE),3) ...
164                             data],'VariableNames',['X' 'Y' 'Z' headers]);
165             for ii = 1:length(T.NODE)
166                 T.X(ii) = Tc.X(T.NODE(ii));
167                 T.Y(ii) = Tc.Y(T.NODE(ii));
168                 T.Z(ii) = Tc.Z(T.NODE(ii));
169             end
170         else
171             % If coordinate table doesnt exist, move on
172             end
173             % If Desired Structure Exists, Append Stress/Strain to Table
174             eval(['y=1;', ['PLATE.',L_type,load, '.',loadstep, ';' ]], 'y=0;');
175             if y == 1
176                 T = join(eval(['PLATE.',L_type,load, '.',loadstep]),T);
177             else
178                 % If plate data doesnt exist, move on
179                 end
180                 % Save Table to Structure
181                 eval(['PLATE.',L_type,load, '.',loadstep] ' =T;');
182
183         % Process Coordinate Data
184     else
185         if exist('Tc') == 1
186             % Ignore subsequent coordinate files to save time
187         else
188             %Put data into table and store table in structure
189             data = cell2mat(data);
190             Tc = array2table(data,'VariableNames',headers);
191         end
192

```

```

193     end
194     clc
195     sprintf('Percent Complete: %3.2f \n\n',100*jj/num_files)
196     fclose(fid.mat);
197 end
198
199 %%
200 fclose('all')
201
202 % Remove repeating index values
203 P_index = unique(P_index);
204 T_index = unique(T_index);
205
206
207 %% Populate Index Matrix for Plate
208 X = unique(T.X);
209 Y = unique(T.Y);
210 Z = unique(T.Z);
211
212 % Rank by X-Location
213 T.Xi = zeros(length(T.X),1);
214 for ii = 1:length(X)
215     T.Xi = T.Xi + eq(T.X(:),X(ii))*ii;
216 end
217
218 % Rank by Y-Location
219 T.Yi = zeros(length(T.Y),1);
220 for ii = 1:length(Y)
221     T.Yi = T.Yi + eq(T.Y(:),Y(ii))*ii;
222 end
223
224 % Rank by Z-Location
225 T.Zi = zeros(length(T.Z),1);
226 for ii = 1:length(Z)
227     T.Zi = T.Zi + eq(T.Z(:),Z(ii))*ii;
228 end
229
230 % Generate Matrix of Indices to Quickly Access Data
231 I = zeros(length(Y),length(X),length(Z));
232 for ii = 1:length(T.X)
233     I(T.Yi(ii),T.Xi(ii),T.Zi(ii)) = ii;
234 end
235
236
237 %% Populate Index Matrix for Gripped Area
238 [X,Y,Z] = meshgrid(X,Y,Z);
239
240 for ii = 1:size(I,1)
241     for jj = 1:size(I,2)
242         for kk = 1:size(I,3)
243             if Y(ii,1,1) == 0.514285714286
244                 Ymin_i = ii;

```

```

245         end
246         if X(1,jj,1) == 4.13636363636
247             Xmin_i = jj;
248         end
249         if X(1,jj,1) == 5.90909090909
250             Xmax_i = jj;
251         end
252     end
253 end
254 end
255
256 % Generate Matrix of Indices to Quickly Access Data
257 Ig = I(Ymin_i:size(I,1),Xmin_i:Xmax_i,:);
258
259 % Directly Specify Nodes in Contact with Grips
260 gNodes_P = zeros(size(Ig,1),size(Ig,2),size(Ig,3));
261 for ii = 1:size(Ig,1)
262     for jj = 1:size(Ig,2)
263         for kk = 1:size(Ig,3)
264             if Ig(ii,jj,kk) == 0
265                 else
266                     gNodes_P(ii,jj,kk) = T.NODE(Ig(ii,jj,kk));
267                 end
268             end
269         end
270     end
271
272
273 %% Save Data to .mat File (Append if .mat Exists)
274 if exist('Plate_Data.mat') == 2
275     save('Plate_Data.mat', 'PLATE', 'P_index', 'T_index', 'Tc', 'X', ...
276         'Y', 'Z', 'gNodes_P','I', 'Ig', '-append','-v7.3')
277 else
278     save('Plate_Data.mat', 'PLATE', 'P_index', 'T_index', 'Tc', 'X', ...
279         'Y', 'Z', 'gNodes_P','I', 'Ig', '-v7.3')
280 end

```

B.1.2 Grip Data Extractor

The premise of the code to extract data from the FEM grip is identical to that of the above code for the plate. However, when extracting reaction forces from the grip there will be many nodal force components that are zero, which are read as empty data in MATLAB. Use of the `fixed_width_import` function from MathWorks can mitigate this.

```

1 % Read force data properly
2 if strcmp(dat_type,'FORCES')
3     [Fi] = ...
        fixed_width_import(file{jj},junk_headers+2,size(data,1),[10 ...

```

```

        15 15 15]);
4      data = mat2cell(Fi,[ones(1,size(Fi,1))],[ones(1,size(Fi,2))]);
5 end

```

The following segment of code is then used to combine reaction force data with the subsequent tables and structures calculated from the data already being processed using coordinates, stresses, and strains.

```

1 % Process Force Data (Empty Data set to Zero)
2 elseif strcmp(dat_type,'FORCES') == 1
3     emptyIndex = cellfun(@isnan,data);
4     data(emptyIndex) = {0};
5     data = cell2mat(data);
6     % Create Force Table
7     Tf = array2table(data,'VariableNames',headers);
8     % Align Force Data to Coordinate Table
9     if exist('Tc')==1
10         Tc.FX = zeros(length(Tc.X),1);
11         Tc.FY = zeros(length(Tc.Y),1);
12         Tc.FZ = zeros(length(Tc.Z),1);
13         for ii = 1:length(Tf.NODE)
14             Tc.FX(Tf.NODE(ii)) = Tf.FX(ii);
15             Tc.FY(Tf.NODE(ii)) = Tf.FY(ii);
16             Tc.FZ(Tf.NODE(ii)) = Tf.FZ(ii);
17         end
18     else
19         % If coordinate table doesn't exist, you're screwed
20     end
21     % If Desired Structure Exists, Append Forces to Table
22     eval(['y=1;', ['GRIP.',L-type,load, '.',loadstep, ';']], 'y=0;');
23     if y == 1
24         T.FX = zeros(length(T.X),1);
25         T.FY = zeros(length(T.Y),1);
26         T.FZ = zeros(length(T.Z),1);
27         for ii = 1:length(T.NODE)
28             T.FX(ii) = Tc.FX(T.NODE(ii));
29             T.FY(ii) = Tc.FY(T.NODE(ii));
30             T.FZ(ii) = Tc.FZ(T.NODE(ii));
31         end
32     else
33         % If plate data doesnt exist, move on
34     end
35     % Save Table to Structure
36     eval(['['GRIP.',L-type,load, '.',loadstep] '=T;']);

```

B.1.3 Dogbone Data Extractor and Script Generation

```
1 %%
2 clear all; close all; clc;
3 %% Open Raw Data Output from MTS
4
5 [file,path] = uigetfile({'*.dat'}, 'MultiSelect', 'on');
6
7 % Input Cross-Section Dimensions
8 thickness = 0.123;
9 width = 0.255;
10
11
12 psi2MPa = 6.8948/1000;
13
14 %Ensures single file still formatted as 1x1 cell
15 if iscell(file),
16     file = file;
17 else
18     file = cellstr(file);
19 end
20
21 %Opens all files (limited to ~508?)
22 num_files = numel(file);
23 fid_mat = zeros(1,num_files);
24 for ii = 1:num_files
25     fid_mat(1,ii)=fopen(char(file(ii)),'r');
26 end
27
28 % Determine number of lines in first file
29 txt=fileread(strjoin(file(1,1)));
30 n_line = sum(txt==10)+1;
31
32
33
34 %% Input Dogbone Data
35
36 %Reads name from filename
37 name = char(file{1});
38 for ii = 1:length(name)
39     textscan(name(ii),'%s');
40     if strcmp(strjoin(ans{1}),'.')==1
41         n_l = ii-1;
42     else
43         end
44 end
45 name = textscan(name(1:n_l),'%s');
46 name = char(name{1}); %char
47
48
```

```

49 %Identifies type of data and format
50 FormatSpecF = '%f %f %f %f %f %f';
51 junk_headers = 5;
52
53 %Skips junk headers
54 for ii = 1:junk_headers
55     out=fgetl(fid_mat(1));
56 end
57
58 %Reads headers into cell
59 out=fgetl(fid_mat(1));
60 headers = textscan('Time Load Disp Strain Time2 Date','%s %s %s %s ...
    %s %s');
61 num_head = numel(headers);
62 for ii = 1:num_head
63     headers{ii} = char(headers{ii});
64 end
65
66
67 %Reads file and outputs data matrix
68 data = cell(1,num_head);
69 for ii = 1:n_line+junk_headers
70     out = fgetl(fid_mat(1));
71     if out == -1;
72         break
73     elseif isequal(out,'') == 1;
74         % Ignores repeating headers
75     else
76         data(ii,:) = textscan(out,FormatSpecF);
77     end
78     clc
79     sprintf('Percent Complete: %3.2f\n\n',100*ii/(n_line+junk_headers))
80 end
81 data = cell2mat(data);
82 % Save data Matrix as Table
83 T = array2table(data,'VariableNames',headers);
84
85
86 %% Process Stress/Strain
87
88 % Calculate Stress from Load
89 Stress = [];
90 for ii = 1:length(T.Load)
91     Stress(ii) = T.Load(ii)/(thickness*width);
92 end
93
94 % Calculate True Stress/Strain/Modulus
95 T.Stress = Stress';
96 T.Modulus = T.Stress./T.Strain;
97 T.Stress_T = T.Stress.*(1+T.Strain);
98 T.Strain_T = log(1+T.Strain);
99 T.Modulus_T = T.Stress_T./T.Strain_T;

```

```

100
101 % Sample to 5000 pts
102 Indices1 = ...
        round(linspace(0.06*length(T.Stress-T),0.8*length(T.Stress-T),5000));
103 % Save Data After First Reduction
104 Stress_R1 = T.Stress-T(Indices1);
105 Strain_R1 = T.Strain-T(Indices1);
106 Modulus_R1 = Stress_R1./Strain_R1;
107
108 % Smooth Data
109 Stress_S = smooth(Strain_R1,Stress_R1,150);
110 Modulus_S = Stress_S./Strain_R1;
111
112 % Gather data starting from Max Modulus
113 Data_Offset = find(Modulus_S==max(Modulus_S));
114
115 % Sample to 100 pts
116 Indices2 = round(linspace(Data_Offset,length(Stress_S),100));
117 % Save Data After Second Reduction
118 Stress_R2 = Stress_S(Indices2);
119 Strain_R2 = Strain_R1(Indices2);
120 Modulus_R2 = Stress_R2./Strain_R2;
121
122
123 % Plot all Stress vs Strain Curves
124 figure(1)
125 hold on
126 plot(T.Strain,T.Stress.*psi2MPa,':b')
127 plot(T.Strain-T,T.Stress-T.*psi2MPa,'b')
128 plot(Strain_R1,Stress_S.*psi2MPa,'--r','LineWidth',2)
129 plot(Strain_R2,Stress_R2.*psi2MPa,'g')
130 legend('Raw Engineering', 'Raw True', 'Smoothed True', 'Reduced ...
        True', 'Location', 'southeast')
131 title('Stress vs Strain')
132 xlabel('$\varepsilon$: (mm/mm)', 'Interpreter', 'LaTeX', 'FontSize', 14)
133 ylabel('$\sigma$: (MPa)', 'Interpreter', 'LaTeX', 'FontSize', 14)
134
135
136 % Plot Local Modulus vs Strain Curves
137 figure(2)
138 hold on
139 plot(T.Strain,T.Modulus.*psi2MPa,':b')
140 plot(T.Strain-T,T.Modulus-T.*psi2MPa,'b')
141 plot(Strain_R1,Modulus_S.*psi2MPa,'--r','LineWidth',2)
142 plot(Strain_R2,Modulus_R2.*psi2MPa,'g')
143 legend('Raw Engineering', 'Raw True', 'Smoothed True', 'Reduced True')
144 title('Modulus vs Strain')
145 xlabel('$\varepsilon$: (mm/mm)', 'Interpreter', 'LaTeX', 'FontSize', 14)
146 ylabel('E = $\frac{\sigma}{\varepsilon}$: ...
        (MPa)', 'Interpreter', 'LaTeX', 'FontSize', 14)
147
148

```

```

149 for ii = 1:length(T.Modulus)-1
150     tMod(ii) = ...
        (round(T.Stress(ii+1),5)-round(T.Stress(ii),5))/(round(T.Strain(ii+1),5)-round(T
151 end
152
153 for ii = 1:length(T.Modulus_T)-1
154     tMod_T(ii) = ...
        (round(T.Stress_T(ii+1),5)-round(T.Stress_T(ii),5))/(round(T.Strain_T(ii+1),5)-r
155 end
156
157 for ii = 1:length(Modulus_S)-1
158     tMod_S(ii) = ...
        (round(Stress_S(ii+1),5)-round(Stress_S(ii),5))/(round(Strain_R1(ii+1),5)-round(
159 end
160
161 for ii = 1:length(Modulus_R2)-1
162     tMod_R2(ii) = ...
        (round(Stress_R2(ii+1),5)-round(Stress_R2(ii),5))/(round(Strain_R2(ii+1),5)-roun
163 end
164 figure(3)
165 hold on
166 plot(linspace(1,length(tMod),length(tMod)).*(length(tMod_R2)/length(tMod)),tMod.*psi2MPa
167 plot(linspace(1,length(tMod_T),length(tMod_T)).*(length(tMod_R2)/length(tMod_T)),tMod_T
168 plot(linspace(1,length(tMod_S),length(tMod_S)).*(length(tMod_R2)/length(tMod_S)),tMod_S
169 plot(linspace(1,length(tMod_R2),length(tMod_R2)),tMod_R2.*psi2MPa,'g','LineWidth',2)
170 legend('Raw Engineering', 'Raw True', 'Smoothed True', 'Reduced True')
171 title('Tangent Modulus vs Strain')
172 xlabel('Data Point','Interpreter','LaTeX','FontSize',14)
173 ylabel('E$-T$ = $\frac{\partial \sigma}{\partial \varepsilon}$: ...
        (MPa)','Interpreter','LaTeX','FontSize',14)
174
175 %% Write Processed data to Input Script
176
177 % Open .txt File for Writing Data
178 fileID = eval(['fopen('Script_',name,'.txt','w')']);
179
180 % Input Lines for Elastic Material Properties
181 fprintf(fileID,'MPTEMP,,,,,,,,\r\n');
182 fprintf(fileID,'MPTEMP,1,0\r\n');
183 fprintf(fileID,'MPDATA,EX,1,,%5.5f\r\n',round(Stress_R2(1),5)/round(Strain_R2(1),5));
184 fprintf(fileID,'MPDATA,PRXY,1,,Poissons_ratio\r\n');
185
186 % Input Lines for Isotropic Hardening Stress vs Strain
187 fprintf(fileID,'TB,MISO,1,1,%3.0f,0\r\n',length(Strain_R2));
188 fprintf(fileID,'TBTEMP,0\r\n');
189 for ii = 1:length(Strain_R2)
190     fprintf(fileID,'TBPT,,%5.5f,%5.5f\r\n',[round(Strain_R2(ii),5) ...
        round(Stress_R2(ii),5)]);
191 end
192
193 % Input Lines for Material Density
194 fprintf(fileID,'MPTEMP,,,,,,,,\r\n');

```

```

195 fprintf(fileID, 'MPTEMP,1,0\r\n');
196 fprintf(fileID, 'MPDATA,DENS,1,,rho/386\r\n');
197
198
199
200 %%
201 fclose('all')
202
203 eval(['save(''Data-',name,'.mat'', 'T'', 'Strain_R1'', ...
      'Strain_R2'', 'Stress_R2'', 'Stress_S'', 'Modulus_R2'', ...
      'Modulus_S'', 'tMod'')']);

```

B.2 Grip Force Script Generation

```

1 clear all; close all; clc;
2
3 load('Plate_Data.mat')
4
5 load('Grip_Data.mat')
6
7 %% Generate Grid of Indices on Z=0 Plane
8
9 Tp = PLATE.T15000.LS1;
10
11 for ii = 1:size(Ig,1)
12     for jj = 1:size(Ig,2)
13         if Ig(ii,jj,1) == 0
14
15             else
16                 X(ii,jj) = Tp.X(Ig(ii,jj,1));
17                 Y(ii,jj) = Tp.Y(Ig(ii,jj,1));
18                 Z(ii,jj) = Tp.Z(Ig(ii,jj,1));
19             end
20         end
21     end
22
23
24 %% Process Grip Nodal Forces at Every Value of P_index
25 for ll = 1:length(P_index);
26
27
28     % Open Grip Data
29     T = GRIP.(['P' num2str(P_index(ll))]).LS0;
30
31     % Establish Coordinates and Forces from Grip Data
32     Xg = [];
33     Yg = [];
34     Zg = [];
35     FXg = [];

```

```

36     FYg = [];
37     FZg = [];
38     Ng = [];
39     for ii = 1:length(T.Z)
40         if abs(T.Z(ii)) <= 0
41             if abs(T.Y(ii)) <= 2.25
42                 Xg = [Xg;T.X(ii)];
43                 Yg = [Yg;T.Y(ii)];
44                 Zg = [Zg;T.Z(ii)];
45                 FXg = [FXg;T.FX(ii)];
46                 FYg = [FYg;T.FY(ii)];
47                 FZg = [FZg;T.FZ(ii)];
48                 Ng = [Ng;T.NODE(ii)];
49             end
50         end
51     end
52
53     % Combine Grip Data to Matrix
54     data = [Xg, Yg, Zg, FXg, FYg, FZg, Ng];
55     headers = {'X', 'Y', 'Z', 'FX', 'FY', 'FZ', 'NODE'};
56     % Convert data to Table
57     Tg = array2table(data,'VariableNames',headers);
58
59     % Populate Index Matrix for Plate
60     X = unique(Tg.X);
61     Y = unique(Tg.Y);
62
63     % Rank by X-Location
64     Tg.Xi = zeros(length(Tg.X),1);
65     for ii = 1:length(X)
66         Tg.Xi = Tg.Xi + eq(Tg.X(:),X(ii))*ii;
67     end
68
69     % Rank by Y-Location
70     Tg.Yi = zeros(length(Tg.Y),1);
71     for ii = 1:length(Y)
72         Tg.Yi = Tg.Yi + eq(Tg.Y(:),Y(ii))*ii;
73     end
74
75     % Generate Matrix of Indices
76     I = zeros(length(Y),length(X));
77     for ii = 1:length(Tg.X)
78         I(Tg.Yi(ii),Tg.Xi(ii)) = ii;
79     end
80
81     % Create Coordinate Matrices for Contour Plot
82     [X,Y] = meshgrid(X,Y);
83
84     % Create Force Matrices for Contour Plot
85     FZ = [];
86     for ii = 1:size(Y,1)
87         for jj = 1:size(X,2)

```

```

88         if I(ii,jj) == 0
89
90         else
91             FX(ii,jj) = Tg.FX(I(ii,jj));
92             FY(ii,jj) = Tg.FY(I(ii,jj));
93             FZ(ii,jj) = Tg.FZ(I(ii,jj));
94             gNodes_G(ii,jj) = Tg.NODE(I(ii,jj));
95         end
96     end
97 end
98
99 % Plot Force Distribution
100 contour3(X,Y,FY,20)
101 %view(-78, 22)
102 %M(11) = getframe;
103
104
105 % Open .txt File for Writing Data
106 fileID = ...
107     eval(['fopen(''Pressure-',num2str(P_index(11)),'-C.txt',''w'')']);
108
109 % Write Reaction X-Force Input to Plate Top
110 for ii = 1:size(Y,1)
111     for jj = 1:size(X,2)
112         fprintf(fileID,'FLST,2,1,1,ORDE,1\r\n');
113         fprintf(fileID,'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,1));
114         fprintf(fileID,'!* \r\n');
115         fprintf(fileID,'/GO\r\n');
116         fprintf(fileID,'F,P51X,FX,%f\r\n',-FX(ii,jj));
117         fprintf(fileID,'\r\n');
118     end
119 end
120 % Write Reaction X-Force Input to Plate Bottom
121 for ii = 1:size(Y,1)
122     for jj = 1:size(X,2)
123         fprintf(fileID,'FLST,2,1,1,ORDE,1\r\n');
124         fprintf(fileID,'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,size(gNodes_P,3)));
125         fprintf(fileID,'!* \r\n');
126         fprintf(fileID,'/GO\r\n');
127         fprintf(fileID,'F,P51X,FX,%f\r\n',FX(ii,jj));
128         fprintf(fileID,'\r\n');
129     end
130 end
131
132 % Write Reaction Y-Force Input to Plate Top
133 for ii = 1:size(Y,1)
134     for jj = 1:size(X,2)
135         fprintf(fileID,'FLST,2,1,1,ORDE,1\r\n');
136         fprintf(fileID,'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,1));
137         fprintf(fileID,'!* \r\n');
138         fprintf(fileID,'/GO\r\n');

```

```

139         fprintf(fileID, 'F,P51X,FY,%f\r\n',-FY(ii,jj));
140         fprintf(fileID, '\r\n');
141     end
142 end
143 % Write Reaction Y-Force Input to Plate Bottom
144 for ii = 1:size(Y,1)
145     for jj = 1:size(X,2)
146         fprintf(fileID, 'FLST,2,1,1,ORDE,1\r\n');
147         fprintf(fileID, 'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,size(gNodes_P,3)));
148         fprintf(fileID, '!*\r\n');
149         fprintf(fileID, '/GO\r\n');
150         fprintf(fileID, 'F,P51X,FY,%f\r\n',-FY(ii,jj));
151         fprintf(fileID, '\r\n');
152     end
153 end
154
155
156 % Write Reaction Z-Force Input to Plate Top
157 for ii = 1:size(Y,1)
158     for jj = 1:size(X,2)
159         fprintf(fileID, 'FLST,2,1,1,ORDE,1\r\n');
160         fprintf(fileID, 'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,1));
161         fprintf(fileID, '!*\r\n');
162         fprintf(fileID, '/GO\r\n');
163         fprintf(fileID, 'F,P51X,FZ,%f\r\n',-FZ(ii,jj));
164         fprintf(fileID, '\r\n');
165     end
166 end
167
168 % Write Reaction Z-Force Input to Plate Bottom
169 for ii = 1:size(Y,1)
170     for jj = 1:size(X,2)
171         fprintf(fileID, 'FLST,2,1,1,ORDE,1\r\n');
172         fprintf(fileID, 'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,size(gNodes_P,3)));
173         fprintf(fileID, '!*\r\n');
174         fprintf(fileID, '/GO\r\n');
175         fprintf(fileID, 'F,P51X,FZ,%f\r\n',FZ(ii,jj));
176         fprintf(fileID, '\r\n');
177     end
178 end
179
180
181 % Close .txt File
182 fclose(fileID);
183
184 clc
185 sprintf('Pressure Percent Complete: %3.2f ...
        \n\n',100*ll/length(P_index))
186 end
187
188 %figure(jj+1)
189 %movie(M,10,3)

```

```

190
191
192 %% Process Grip Nodal Forces at Every Value of T_index
193 for kk = 1:length(T_index);
194
195     % Open Grip Data
196     T = GRIP.(['T' num2str(T_index(kk))]).LS1;
197
198     % Establish Coordinates and Forces from Grip Data
199     Xg = [];
200     Yg = [];
201     Zg = [];
202     FXg = [];
203     FYg = [];
204     FZg = [];
205     Ng = [];
206     for ii = 1:length(T.Z)
207         if abs(T.Z(ii)) <= 0
208             if abs(T.Y(ii)) <= 2.25
209                 Xg = [Xg;T.X(ii)];
210                 Yg = [Yg;T.Y(ii)];
211                 Zg = [Zg;T.Z(ii)];
212                 FXg = [FXg;T.FX(ii)];
213                 FYg = [FYg;T.FY(ii)];
214                 FZg = [FZg;T.FZ(ii)];
215                 Ng = [Ng;T.NODE(ii)];
216             end
217         end
218     end
219
220     % Combine Grip Data to Matrix
221     data = [Xg, Yg, Zg, FXg, FYg, FZg, Ng];
222     headers = {'X', 'Y', 'Z', 'FX', 'FY', 'FZ', 'NODE'};
223     % Convert data to Table
224     Tg = array2table(data,'VariableNames',headers);
225
226     % Populate Index Matrix for Plate
227     X = unique(Tg.X);
228     Y = unique(Tg.Y);
229
230     % Rank by X-Location
231     Tg.Xi = zeros(length(Tg.X),1);
232     for ii = 1:length(X)
233         Tg.Xi = Tg.Xi + eq(Tg.X(:),X(ii))*ii;
234     end
235
236     % Rank by Y-Location
237     Tg.Yi = zeros(length(Tg.Y),1);
238     for ii = 1:length(Y)
239         Tg.Yi = Tg.Yi + eq(Tg.Y(:),Y(ii))*ii;
240     end
241

```

```

242 % Generate Matrix of Indices
243 I = zeros(length(Y),length(X));
244 for ii = 1:length(Tg.X)
245     I(Tg.Yi(ii),Tg.Xi(ii)) = ii;
246 end
247
248 % Create Coordinate Matrices for Contour Plot
249 [X,Y] = meshgrid(X,Y);
250
251 % Create Force Matrices for Contour Plot
252 FZ = [];
253 for ii = 1:size(Y,1)
254     for jj = 1:size(X,2)
255         if I(ii,jj) == 0
256
257             else
258                 FX(ii,jj) = Tg.FX(I(ii,jj));
259                 FY(ii,jj) = Tg.FY(I(ii,jj));
260                 FZ(ii,jj) = Tg.FZ(I(ii,jj));
261                 gNodes-G(ii,jj) = Tg.NODE(I(ii,jj));
262             end
263         end
264     end
265
266 % Plot Force Distribution
267 contour3(X,Y,FZ)
268 xlabel('Grip Length (in)')
269 ylabel('Grip Width (in)')
270 %view(-78, 22)
271 %M2(kk) = getframe;
272
273 % Open .txt File for Writing Data
274 fileID = ...
275     eval(['fopen(''Tension-',num2str(T_index(kk)),'-C.txt','w')']);
276
277 % Write Reaction X-Force Input to Plate Top
278 for ii = 1:size(Y,1)
279     for jj = 1:size(X,2)
280         fprintf(fileID,'FLST,2,1,1,ORDE,1\r\n');
281         fprintf(fileID,'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,1));
282         fprintf(fileID,'!* \r\n');
283         fprintf(fileID,'/GO\r\n');
284         fprintf(fileID,'F,P51X,FX,%f\r\n',-FX(ii,jj));
285         fprintf(fileID,' \r\n');
286     end
287
288 % Write Reaction X-Force Input to Plate Bottom
289 for ii = 1:size(Y,1)
290     for jj = 1:size(X,2)
291         fprintf(fileID,'FLST,2,1,1,ORDE,1\r\n');
292         fprintf(fileID,'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,size(gNodes_P,3)));
293         fprintf(fileID,'!* \r\n');

```

```

293         fprintf(fileID, '/GO\r\n');
294         fprintf(fileID, 'F,P51X,FX,%f\r\n',FX(ii,jj));
295         fprintf(fileID, '\r\n');
296     end
297 end
298
299 % Write Reaction Y-Force Input to Plate Top
300 for ii = 1:size(Y,1)
301     for jj = 1:size(X,2)
302         fprintf(fileID, 'FLST,2,1,1,ORDE,1\r\n');
303         fprintf(fileID, 'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,1));
304         fprintf(fileID, '!*\r\n');
305         fprintf(fileID, '/GO\r\n');
306         fprintf(fileID, 'F,P51X,FY,%f\r\n',-FY(ii,jj));
307         fprintf(fileID, '\r\n');
308     end
309 end
310 % Write Reaction Y-Force Input to Plate Bottom
311 for ii = 1:size(Y,1)
312     for jj = 1:size(X,2)
313         fprintf(fileID, 'FLST,2,1,1,ORDE,1\r\n');
314         fprintf(fileID, 'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,size(gNodes_P,3)));
315         fprintf(fileID, '!*\r\n');
316         fprintf(fileID, '/GO\r\n');
317         fprintf(fileID, 'F,P51X,FY,%f\r\n',-FY(ii,jj));
318         fprintf(fileID, '\r\n');
319     end
320 end
321
322 % Write Reaction Z-Force Input to Plate Top
323 for ii = 1:size(Y,1)
324     for jj = 1:size(X,2)
325         fprintf(fileID, 'FLST,2,1,1,ORDE,1\r\n');
326         fprintf(fileID, 'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,1));
327         fprintf(fileID, '!*\r\n');
328         fprintf(fileID, '/GO\r\n');
329         fprintf(fileID, 'F,P51X,FZ,%f\r\n',-FZ(ii,jj));
330         fprintf(fileID, '\r\n');
331     end
332 end
333 % Write Reaction Z-Force Input to Plate Top
334 for ii = 1:size(Y,1)
335     for jj = 1:size(X,2)
336         fprintf(fileID, 'FLST,2,1,1,ORDE,1\r\n');
337         fprintf(fileID, 'FITEM,2,%6.0f\r\n',gNodes_P(ii,jj,size(gNodes_P,3)));
338         fprintf(fileID, '!*\r\n');
339         fprintf(fileID, '/GO\r\n');
340         fprintf(fileID, 'F,P51X,FZ,%f\r\n',FZ(ii,jj));
341         fprintf(fileID, '\r\n');
342     end
343 end
344

```

```

345     % Close .txt File
346     fclose(fileID);
347
348     clc
349     sprintf('Tension Percent Complete: %3.2f ...
            \n\n',100*kk/length(T_index))
350 end
351
352 %figure(kk+1)
353 %movie(M2,10,3)

```

B.3 DIC vs FEM Comparison Figures

```

1 clear all; close all; clc
2
3 %% Read Data
4
5 load('plate_9-1700_0.mat')
6 load('Plate_Data')
7 T = PLATE.T15495.LS1;
8
9 % Unit Conversions
10 in2mm = 25.4;
11 psi2MPa = 6.8948/1000;
12
13 %% Translate DIC Coordinates
14
15 % Feature Location
16 Loc_x = 6.4; % in
17 Loc_y = 0; % in
18
19 % Translation to Global Coordinates
20 x_t = Loc_x - x_c(round(size(y_c,1)/2),size(x_c,2));
21 y_t = Loc_y - y_c(round(size(y_c,1)/2),size(x_c,2));
22
23 x_c = x_c + x_t;
24 y_c = y_c + y_t;
25
26 %% Total FEM Data
27
28 % Create FEM Stress/Strain Matrices from Index Matrix
29 EX = [];
30 EY = [];
31 EXY = [];
32 SY = [];
33 Seff = [];
34 Svm = [];
35 for ii = 1:size(I,1)
36     for jj = 1:size(I,2)

```

```

37         for kk = 1:size(I,3)
38             if I(ii,jj,kk) == 0;
39
40                 else
41                     EX(ii,jj,kk) = T.EPTOX(I(ii,jj,kk));
42                     EY(ii,jj,kk) = T.EPTOY(I(ii,jj,kk));
43                     EXY(ii,jj,kk) = T.EPTOXY(I(ii,jj,kk));
44                     SY(ii,jj,kk) = T.SY(I(ii,jj,kk));
45                     Seff(ii,jj,kk) = ...
                        (sqrt(2)/2)*sqrt((T.SX(I(ii,jj,kk))-T.SY(I(ii,jj,kk)))^2+(T.SY(I(ii,
46                 end
47             end
48         end
49     end
50
51 % Plot FEM Strain Contours
52 hfig = figure(1);
53 hold on
54 a = contourf(X(:, :, 1)*in2mm, Y(:, :, 1)*in2mm, 10^6*EY(:, :, 1));
55 [a,b] = contourf(X(:, :, 1)*in2mm, -Y(:, :, 1)*in2mm, 10^6*EY(:, :, 1));
56 contour(X(:, :, 1)*in2mm, Y(:, :, 1)*in2mm, Seff(:, :, 1)*psi2MPa, [0 39000 ...
    41000 43000].*psi2MPa, 'r');
57 [c,d] = ...
    contour(X(:, :, 1)*in2mm, -Y(:, :, 1)*in2mm, Seff(:, :, 1)*psi2MPa, [0 ...
    39000 41000 43000].*psi2MPa, 'r');
58 set(get(get(b, 'Annotation'), 'LegendInformation'), 'IconDisplayStyle', 'off'); ...
    % Exclude line from legend
59 set(get(get(d, 'Annotation'), 'LegendInformation'), 'IconDisplayStyle', 'off'); ...
    % Exclude line from legend
60 xlabel('Length: (mm)', 'Interpreter', 'LaTeX');
61 ylabel('Width: (mm)', 'Interpreter', 'LaTeX');
62 daspect([1 1 1]);
63 cb = colorbar('southoutside');
64 yt=get(cb, 'XTick'); tick = ...
    linspace(min(min(min(10^6.*EY))), max(max(max(10^6.*EY))), 10); ...
    tickl = num2cell(round(tick, 0));
65 set(cb, 'Ticks', tick); set(cb, 'Limits', [tick(1) tick(end)]); ...
    set(cb, 'TickLabels', tickl);
66 ylabel(cb, 'Maximum Load $\varepsilon_{yy}$: ...
    (microstrain)', 'Interpreter', 'LaTeX');
67 cb.Label.FontSize = 11;
68 % Plot DIC Limits
69 plot([min(min(x_c)) min(min(x_c)) max(max(x_c)) max(max(x_c)) ...
    min(min(x_c))].*in2mm, [min(min(y_c)) max(max(y_c)) ...
    max(max(y_c)) min(min(y_c)) min(min(y_c))].*in2mm, 'g', 'LineWidth', 2)
70 plot([4.16 5.91 5.91 4.16 4.16].*in2mm, [-2.25 -2.25 -0.5 -0.5 ...
    -2.25].*in2mm, 'k', 'LineWidth', 2)
71 plot([4.16 5.91 5.91 4.16 4.16].*in2mm, [2.25 2.25 0.5 0.5 ...
    2.25].*in2mm, 'k', 'LineWidth', 2)
72 L = legend('Total $\varepsilon_{yy}$', 'Plastic $\sigma_e$', 'DIC ...
    Region', 'Grip Region', 'Location', 'west');
73 set(L, 'Interpreter', 'LaTeX')

```

```

74 title('FEM Strain Profile')
75 movegui('northwest')
76
77
78 % Plot FEM Stress Contours
79 figure(2)
80 hold on
81 c = contourf(X(:, :, 1)*in2mm, Y(:, :, 1)*in2mm, Seff(:, :, 1)*psi2MPa);
82 xlabel('Length: (mm)', 'Interpreter', 'LaTeX');
83 ylabel('Width: (mm)', 'Interpreter', 'LaTeX');
84 daspect([1 1 1]);
85 cb = colorbar('southoutside');
86 yt=get(cb, 'XTick'); tick = ...
    linspace(min(min(min(Seff*psi2MPa))), max(max(max(Seff*psi2MPa))), 10); ...
    tickl = num2cell(round(tick, 0))';
87 set(cb, 'Ticks', tick); set(cb, 'Limits', [tick(1) tick(end)]); ...
    set(cb, 'TickLabels', tickl);
88 ylabel(cb, 'Maximum Load  $\sigma$ : (MPa)', 'Interpreter', 'LaTeX');
89 cb.Label.FontSize = 11;
90 plot([4.16 5.91 5.91 4.16 4.16].*in2mm, [2.25 2.25 0.5 0.5 ...
    2.25].*in2mm, 'k', 'LineWidth', 2)
91 title('FEM Von Mises Stress Profile')
92 movegui('north')
93
94
95
96 %% DIC Data
97
98 % Plot DIC Strain
99 figure(3)
100 hold on
101 [c, h] = contourf(x_c.*in2mm, y_c.*in2mm, 10^6*eyy, linspace(5000, 17000, 7));
102 axis('equal')
103 xlabel('Length: (mm)', 'Interpreter', 'LaTeX');
104 ylabel('Width: (mm)', 'Interpreter', 'LaTeX');
105 daspect([1 1 1]);
106 cb = colorbar('southoutside');
107 yt=get(cb, 'XTick'); tick = linspace(5000, 17000, 7); tickl = ...
    num2cell(round(tick, 0))';
108 set(cb, 'Ticks', tick); set(cb, 'Limits', [tick(1) tick(end)]); ...
    set(cb, 'TickLabels', tickl);
109 ylabel(cb, 'Maximum Load  $\epsilon$ : ...
    (microstrain)', 'Interpreter', 'LaTeX');
110 cb.Label.FontSize = 11;
111
112
113 %% Match Reduced FEM Grid to DIC
114
115 % Create Reduced FEM Strain Vectors from Index Matrix and DIC Limits
116 EY = [];
117 X = [];
118 Y = [];

```

```

119 for ii = 1:size(I,1)
120     for jj = 1:size(I,2)
121         for kk = 1:size(I,3)
122             if I(ii,jj,kk) == 0;
123                 elseif (T.X(I(ii,jj,kk)) >= min(min(x_c))) & ...
                     (T.Y(I(ii,jj,kk)) <= max(max(y_c))) ;
124                     X = [X;T.X(I(ii,jj,kk))];
125                     Y = [Y;T.Y(I(ii,jj,kk))];
126                     EY = [EY;T.EPTOY(I(ii,jj,kk))];
127             end
128         end
129     end
130 end
131
132 % Mirror FEM Strain about X-Axis
133 X = [X;X];
134 Y = [Y;-Y];
135 EY = [EY;EY];
136
137 % Match FEM Strain Vector to DIC Grid
138 EY_c = griddata(X,Y,EY,x_c,y_c);
139
140 % Plot FEM Strain Contours on DIC (Contour mirrored about x-axis)
141 contour(x_c*in2mm,y_c*in2mm,10^6*EY_c,h.LevelList,'LineStyle','--', ...
        'LineColor', 'k','ShowText', 'on');
142 xlabel('Length: (mm)','Interpreter','LaTeX');
143 ylabel('Width: (mm)','Interpreter','LaTeX');
144 xlim([min(min(x_c)) max(max(x_c))].*in2mm)
145 ylim([min(min(y_c)) max(max(y_c))].*in2mm)
146 daspect([1 1 1]);
147 legend('DIC', 'FEM', 'Location', 'northeast')
148 title('DIC Strain Profile vs FEM')
149 movegui('northeast')
150
151 % Plot FEM Strain Contours
152 hfig = figure(4);
153 hold on
154 contourf(x_c*in2mm,y_c*in2mm,10^6*EY_c,h.LevelList,'LineStyle','--','ShowText', ...
        'on');
155 xlabel('Length: (mm)','Interpreter','LaTeX');
156 ylabel('Width: (mm)','Interpreter','LaTeX');
157 xlim([min(min(x_c)) max(max(x_c))].*in2mm)
158 ylim([min(min(y_c)) max(max(y_c))].*in2mm)
159 daspect([1 1 1]);
160 cb = colorbar('southoutside');
161 yt=get(cb,'XTick'); tick = h.LevelList; tickl = ...
        num2cell(round(tick,0))';
162 set(cb,'Ticks',tick); set(cb,'Limits',[tick(1) tick(end)]); ...
        set(cb,'TickLabels',tickl);
163 ylabel(cb,'Maximum Load $\varepsilon_{yy}$: ...
        (microstrain)','Interpreter','LaTeX');
164 cb.Label.FontSize = 11;

```

```

165 title('FEM Reduced Strain Profile')
166 movegui('southwest')
167
168
169 %% Plot Error From FEM to DIC
170
171 EY_err = EY_c - eyy;
172
173 figure(5)
174 hold on
175 [c,h] = contourf(x_c*in2mm,y_c*in2mm,10^6*EY_err);
176 axis('equal')
177 xlabel('Length: (mm)','Interpreter','LaTeX');
178 ylabel('Width: (mm)','Interpreter','LaTeX');
179 daspect([1 1 1]);
180 cb = colorbar('southoutside');
181 yt=get(cb,'XTick'); tick = linspace(yt(1),yt(end),6); tickl = ...
    num2cell(round(tick,0))';
182 set(cb,'Ticks',tick); set(cb,'Limits',[tick(1) tick(end)]); ...
    set(cb,'TickLabels',tickl);
183 ylabel(cb,'Maximum Load $\varepsilon_{yy}$: ...
    (microstrain)','Interpreter','LaTeX');
184 cb.Label.FontSize = 11;
185 movegui('southeast')
186
187 % Compute Root Mean Squared Error
188 err2 = 0;
189 for ii = 1:size(eyy,1)*size(eyy,2)
190     if isnan(EY_err(ii)) == 1
191     else
192         err2 = err2 + (EY_err(ii)/EY_c(ii))^2;
193     end
194 end
195 RMS = sqrt((1/(ii-1))*err2)*100;
196 title(['Error from FEM to DIC (RMS: ', num2str(RMS,3), '%)'])
197
198
199
200 %% DIC vs FEM Line
201
202 figure(6)
203 hold on
204 plot(x_c(round(size(x_c,1)/2),:)*in2mm,eyy(round(size(eyy,1)/2),:)*10^6)
205 plot(x_c(round(size(x_c,1)/2),:)*in2mm,EY_c(round(size(EY_c,1)/2),:)*10^6)
206 xlabel('Length: (mm)','Interpreter','LaTeX');
207 ylabel('$\varepsilon_{yy}$: (microstrain)','Interpreter','LaTeX');
208 legend('DIC', 'FEM', 'Location', 'northeast')
209 title('DIC Strain Profile vs FEM (Plate Centerline)')
210 text(5.2*in2mm,EY_c(round(size(EY_c,1)/2),85)*10^6,'X <== Gage 2')
211 text(6.08*in2mm,EY_c(round(size(EY_c,1)/2),428)*10^6,'Gage 1 ==> X')
212 movegui('south')

```

Appendix C. Plate Mesh Script

! Define keypoints

```
K,100,0,0,0,
K,101,0,width,0,
K,102,length,width,0,
K,103,length,0,0,
K,104,length-gap-grip_len,width,0,
K,105,length-gap,width,0,
K,106,length-gap,width-grip_wid,0,
K,107,length-gap-grip_len,width-grip_wid,0,
K,108,length,width-grip_wid,0,
K,109,length-gap,0,0,
K,110,length-gap-grip_len,0,0,
K,111,0,width-grip_wid,0,
```

```
K,200,0,0,thickness,
K,201,0,width,thickness,
K,202,length,width,thickness,
K,203,length,0,thickness,
K,204,length-gap-grip_len,width,thickness,
K,205,length-gap,width,thickness,
K,206,length-gap,width-grip_wid,thickness,
K,207,length-gap-grip_len,width-grip_wid,thickness,
K,208,length,width-grip_wid,thickness,
K,209,length-gap,0,thickness,
K,210,length-gap-grip_len,0,thickness,
K,211,0,width-grip_wid,thickness,
```

! Define lines

```
L,100,111
L,111,101
L,101,104
L,104,105
L,105,102
L,102,108
L,108,103
L,103,109
L,109,110
L,110,100
L,104,105
L,105,106
L,106,107
L,107,104
L,107,111
L,107,110
L,106,109
L,106,108
```

L,	200,	211
L,	211,	201
L,	201,	204
L,	204,	205
L,	205,	202
L,	202,	208
L,	208,	203
L,	203,	209
L,	209,	210
L,	210,	200
L,	204,	205
L,	205,	206
L,	206,	207
L,	207,	204
L,	207,	211
L,	207,	210
L,	206,	209
L,	206,	208

L,	100,	200
L,	101,	201
L,	102,	202
L,	103,	203
L,	104,	204
L,	105,	205
L,	106,	206
L,	107,	207
L,	108,	208
L,	109,	209
L,	110,	210
L,	111,	211

! Define areas

! Grip Top

FLST,2,4,4

FITEM,2,30

FITEM,2,21

FITEM,2,28

FITEM,2,29

AL,P51X

! Grip Bottom

FLST,2,4,4

FITEM,2,4

FITEM,2,11

FITEM,2,12

FITEM,2,13

AL,P51X

! Grip Edges

```

FLST,2,4,4
FITEM,2,42
FITEM,2,30
FITEM,2,13
FITEM,2,39
AL,P51X
FLST,2,4,4
FITEM,2,4
FITEM,2,39
FITEM,2,21
FITEM,2,40
AL,P51X
FLST,2,4,4
FITEM,2,11
FITEM,2,28
FITEM,2,40
FITEM,2,41
AL,P51X
FLST,2,4,4
FITEM,2,29
FITEM,2,12
FITEM,2,41
FITEM,2,42
AL,P51X

```

```

! Region 1
FLST,2,4,4
FITEM,2,30
FITEM,2,20
FITEM,2,19
FITEM,2,31
AL,P51X
FLST,2,4,4
FITEM,2,2
FITEM,2,3
FITEM,2,13
FITEM,2,14
AL,P51X
FLST,2,4,4
FITEM,2,46
FITEM,2,2
FITEM,2,19
FITEM,2,36
AL,P51X
FLST,2,4,4
FITEM,2,20
FITEM,2,36
FITEM,2,3
FITEM,2,39
AL,P51X
FLST,2,4,4
FITEM,2,31

```

FITEM,2,42
FITEM,2,14
FITEM,2,46
AL,P51X

! Region 2
FLST,2,4,4
FITEM,2,18
FITEM,2,27
FITEM,2,31
FITEM,2,32
AL,P51X
FLST,2,4,4
FITEM,2,1
FITEM,2,14
FITEM,2,10
FITEM,2,15
AL,P51X
FLST,2,4,4
FITEM,2,18
FITEM,2,1
FITEM,2,46
FITEM,2,35
AL,P51X
FLST,2,4,4
FITEM,2,27
FITEM,2,35
FITEM,2,10
FITEM,2,45
AL,P51X
FLST,2,4,4
FITEM,2,45
FITEM,2,15
FITEM,2,32
FITEM,2,42
AL,P51X

! Region 3
FLST,2,4,4
FITEM,2,32
FITEM,2,29
FITEM,2,26
FITEM,2,33
AL,P51X
FLST,2,4,4
FITEM,2,12
FITEM,2,15
FITEM,2,9
FITEM,2,16
AL,P51X
FLST,2,4,4
FITEM,2,9

FITEM,2,45
FITEM,2,26
FITEM,2,44
AL,P51X
FLST,2,4,4
FITEM,2,16
FITEM,2,33
FITEM,2,41
FITEM,2,44
AL,P51X

! Region 4
FLST,2,4,4
FITEM,2,25
FITEM,2,33
FITEM,2,34
FITEM,2,24
AL,P51X
FLST,2,4,4
FITEM,2,17
FITEM,2,16
FITEM,2,8
FITEM,2,7
AL,P51X
FLST,2,4,4
FITEM,2,44
FITEM,2,25
FITEM,2,8
FITEM,2,38
AL,P51X
FLST,2,4,4
FITEM,2,38
FITEM,2,24
FITEM,2,7
FITEM,2,43
AL,P51X
FLST,2,4,4
FITEM,2,34
FITEM,2,41
FITEM,2,17
FITEM,2,43
AL,P51X

! Region 5
FLST,2,4,4
FITEM,2,28
FITEM,2,22
FITEM,2,23
FITEM,2,34
AL,P51X
FLST,2,4,4
FITEM,2,5

FITEM,2,11
FITEM,2,6
FITEM,2,17
AL,P51X
FLST,2,4,4
FITEM,2,43
FITEM,2,23
FITEM,2,6
FITEM,2,37
AL,P51X
FLST,2,4,4
FITEM,2,22
FITEM,2,37
FITEM,2,5
FITEM,2,40
AL,P51X

! Define volumes

! Grip Volume
FLST,2,6,5,ORDE,2
FITEM,2,1
FITEM,2,-6
VA,P51X

! Region 1
FLST,2,6,5,ORDE,3
FITEM,2,3
FITEM,2,7
FITEM,2,-11
VA,P51X

! Region 2
FLST,2,6,5,ORDE,2
FITEM,2,11
FITEM,2,-16
VA,P51X

! Region 3
FLST,2,6,5,ORDE,3
FITEM,2,6
FITEM,2,16
FITEM,2,-20
VA,P51X

! Region 4
FLST,2,6,5,ORDE,2
FITEM,2,20
FITEM,2,-25
VA,P51X

```

! Region 5
FLST,2,6,5,ORDE,3
FITEM,2,5
FITEM,2,25
FITEM,2,-29
VA,P51X

!!! Meshing

!! Mesh controls

! Grip Region
FLST,5,4,4,ORDE,2
FITEM,5,39
FITEM,5,-42
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL, ,_Y
!*
LESIZE,_Y1, , ,thickness_elem, , , ,1
!*
FLST,5,4,4,ORDE,4
FITEM,5,4
FITEM,5,12
FITEM,5,21
FITEM,5,29
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL, ,_Y
!*
LESIZE,_Y1, , ,grip_len_elem, , , ,1
!*
FLST,5,4,4,ORDE,4
FITEM,5,11
FITEM,5,13
FITEM,5,28
FITEM,5,30
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL, ,_Y
!*
LESIZE,_Y1, , ,grip_wid_elem, , , ,1
!*

!Region 1
FLST,5,4,4,ORDE,4
FITEM,5,3
FITEM,5,14

```

```

FITEM,5,20
FITEM,5,31
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,length_elem-gap_elem-grip_len_elem, , , ,1
!*
FLST,5,2,4,ORDE,2
FITEM,5,2
FITEM,5,19
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,grip_wid_elem, , , ,1
!*
FLST,5,2,4,ORDE,2
FITEM,5,36
FITEM,5,46
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,thickness_elem, , , ,1
!*

!Region 2
FLST,5,4,4,ORDE,4
FITEM,5,1
FITEM,5,15
FITEM,5,18
FITEM,5,32
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,width_elem-grip_wid_elem, , , ,1
!*
FLST,5,2,4,ORDE,2
FITEM,5,10
FITEM,5,27
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,length_elem-gap_elem-grip_len_elem, , , ,1

```

```

!*
FLST,5,2,4,ORDE,2
FITEM,5,35
FITEM,5,45
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,thickness_elem, , , ,1
!*

!Region 3
FLST,5,2,4,ORDE,2
FITEM,5,9
FITEM,5,26
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,grip_len_elem, , , ,1
!*
FLST,5,2,4,ORDE,2
FITEM,5,16
FITEM,5,33
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,width_elem-grip_wid_elem, , , ,1
!*
FLST,5,1,4,ORDE,1
FITEM,5,44
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,-Y
!*
LESIZE,_Y1, , ,thickness_elem, , , ,1
!*

!Region 4
FLST,5,4,4,ORDE,4
FITEM,5,8
FITEM,5,17
FITEM,5,25
FITEM,5,34
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE

```

```

CMSEL,,_Y
!*
LESIZE,_Y1, , ,gap_elem, , , ,1
!*
FLST,5,2,4,ORDE,2
FITEM,5,7
FITEM,5,24
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,_Y
!*
LESIZE,_Y1, , ,width_elem-grip_wid_elem, , , ,1
!*
FLST,5,2,4,ORDE,2
FITEM,5,38
FITEM,5,43
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,_Y
!*
LESIZE,_Y1, , ,thickness_elem, , , ,1
!*

!Region 5
FLST,5,2,4,ORDE,2
FITEM,5,6
FITEM,5,23
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,_Y
!*
LESIZE,_Y1, , ,grip_wid_elem, , , ,1
!*
FLST,5,2,4,ORDE,2
FITEM,5,5
FITEM,5,22
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,_Y
!*
LESIZE,_Y1, , ,gap_elem, , , ,1
!*
FLST,5,1,4,ORDE,1
FITEM,5,37
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,_Y

```

```
!*
LESIZE,_Y1, , ,thickness_elem, , , ,1
!*
```

```
!! Mesh Volumes
```

```
! Grip Region
```

```
MSHAPE,0,3D
```

```
MSHKEY,1
```

```
!*
```

```
CM,_Y,VOLU
```

```
VSEL, , , , 1
```

```
CM,_Y1,VOLU
```

```
CHKMSH,'VOLU'
```

```
CMSEL,S,_Y
```

```
!*
```

```
VMESH,_Y1
```

```
!*
```

```
CMDELE,_Y
```

```
CMDELE,_Y1
```

```
CMDELE,_Y2
```

```
!*
```

```
!Region 1
```

```
CM,_Y,VOLU
```

```
VSEL, , , , 2
```

```
CM,_Y1,VOLU
```

```
CHKMSH,'VOLU'
```

```
CMSEL,S,_Y
```

```
!*
```

```
VMESH,_Y1
```

```
!*
```

```
CMDELE,_Y
```

```
CMDELE,_Y1
```

```
CMDELE,_Y2
```

```
!*
```

```
!Region 2
```

```
CM,_Y,VOLU
```

```
VSEL, , , , 3
```

```
CM,_Y1,VOLU
```

```
CHKMSH,'VOLU'
```

```
CMSEL,S,_Y
```

```
!*
```

```
VMESH,_Y1
```

```
!*
```

```
CMDELE,_Y
```

```
CMDELE,_Y1
```

```
CMDELE,_Y2
```

```
!*
```

```

!Region 3
CM,_Y,VOLU
VSEL, , , ,      4
CM,_Y1,VOLU
CHKMSH,'VOLU'
CMSEL,S,_Y
!*
VMESH,_Y1
!*
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2
!*

!Region 4
CM,_Y,VOLU
VSEL, , , ,      5
CM,_Y1,VOLU
CHKMSH,'VOLU'
CMSEL,S,_Y
!*
VMESH,_Y1
!*
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2
!*

!Region 5
CM,_Y,VOLU
VSEL, , , ,      6
CM,_Y1,VOLU
CHKMSH,'VOLU'
CMSEL,S,_Y
!*
VMESH,_Y1
!*
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2
!*

```

Appendix D. Correlation Function Example

Tracking of specimen deformation and resulting strains using digital image correlation (DIC) is non-deterministic, as opposed to deterministic measurements of strain via gages. The underlying algorithm for DIC is the correlation function, introduced in Section 2.6.1 and seen in Figure D.1, which is actually a statistical best fit field of deformation throughout the entire region of interest. It is for this reason that a random speckle pattern of high contrast is necessary to discern deformation of each subset, as repeating patterns may associate deformation as a product of two confused locations and poor gradients in contrast may drive errors in the correlation function to support unrealistic deformations. The following example shows how two possible deformations are weighted to probabilistically support the real deformation.

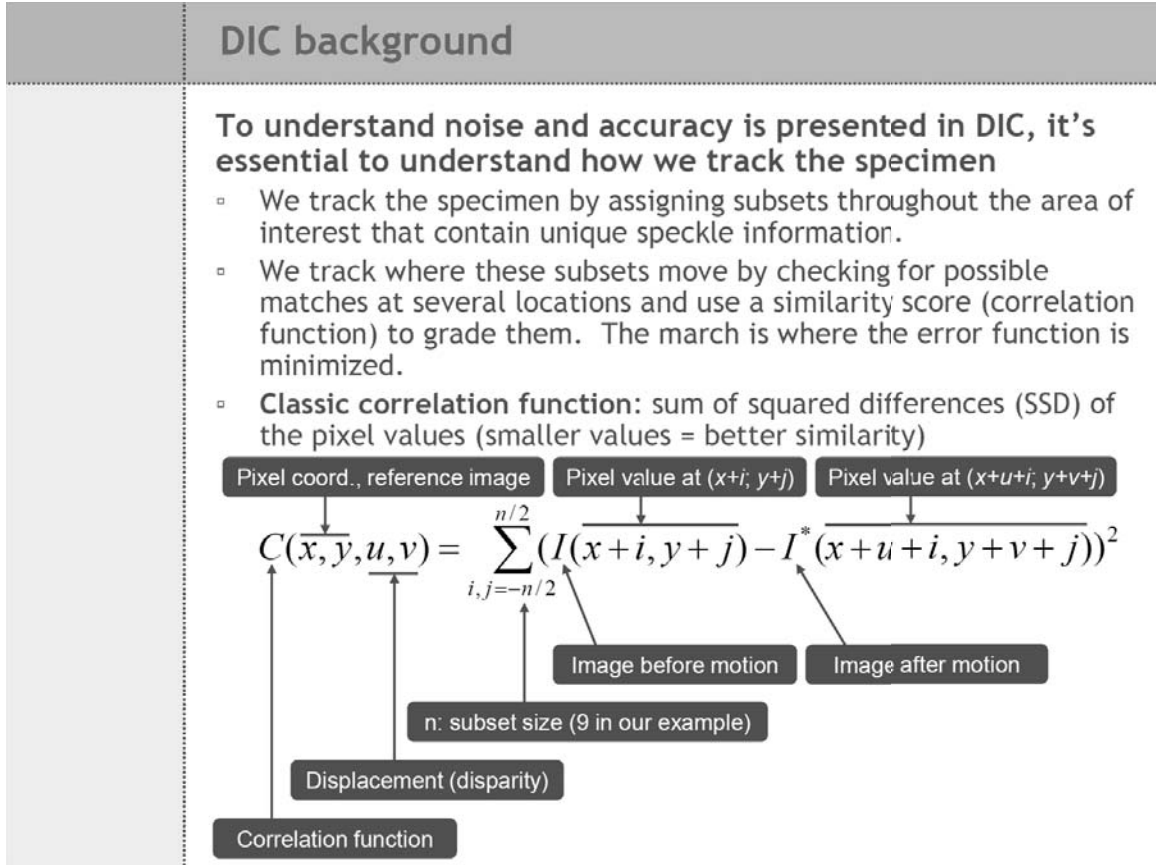


Figure D.1. Correlation Function [26]

The data contained in the reference image seen in Figure D.2a is shown as both the mathematically stored image and the visual representation. Contrast is a perfect disparity between black and white, with gray levels of 0 and 100, respectively. The speckle in this case is perfectly uniform, and as such the deformation of this small region (seen in Figure D.2b) is easy to resolve. The imposed deformation is 1 pixel to the right and once pixel up ($u=1$, $v=1$), which again is very easy to resolve because of the high change in contrast.

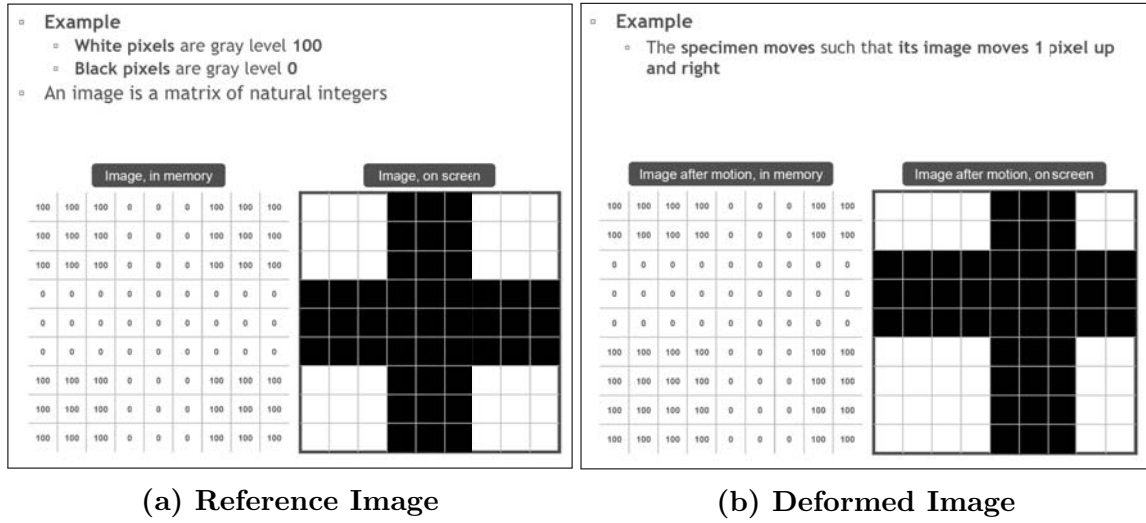


Figure D.2. Deformation of a Perfect Contrast Speckle [26]

Recall that while in this example the true deformation is known, most experimental measurements are made because deformations are unknown, and must be resolved via DIC. The software supplied by Correlated Solutions will assess the score of the correlation function for small potential deformations in multiple directions, with one such case seen in Figure D.3. The 5x5 subset is compared between the reference image and the potential deformation of that subset ($u=-2$, $v=-2$) in the deformed image, and a correlation scoring is shown assessing squared deviations of every pixel within the subset. In this case the correlation is poor, and as such the sum of squared deviations (SSD) is very large at 18,000.

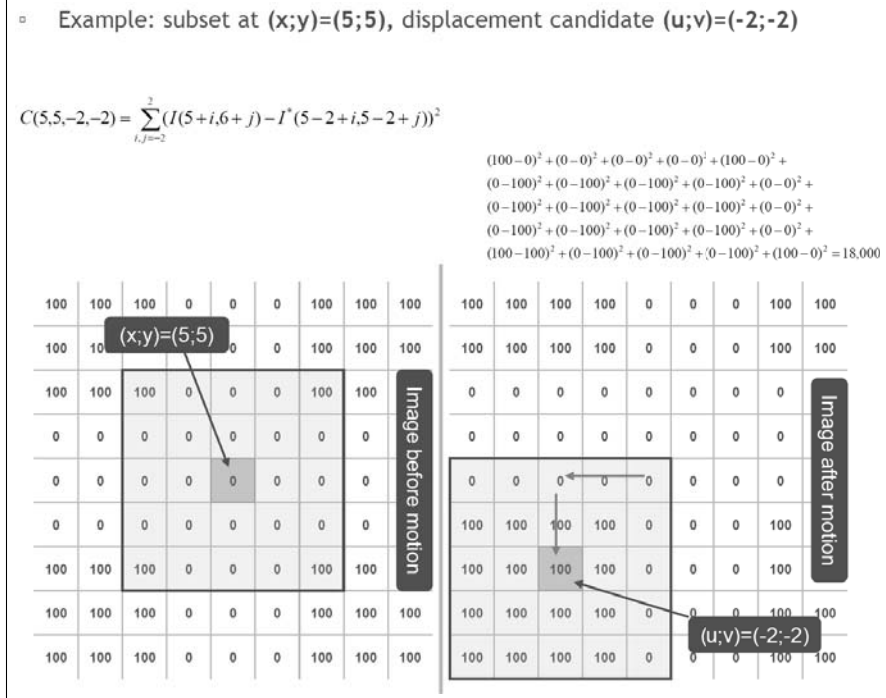


Figure D.3. Poor Correlation [26]

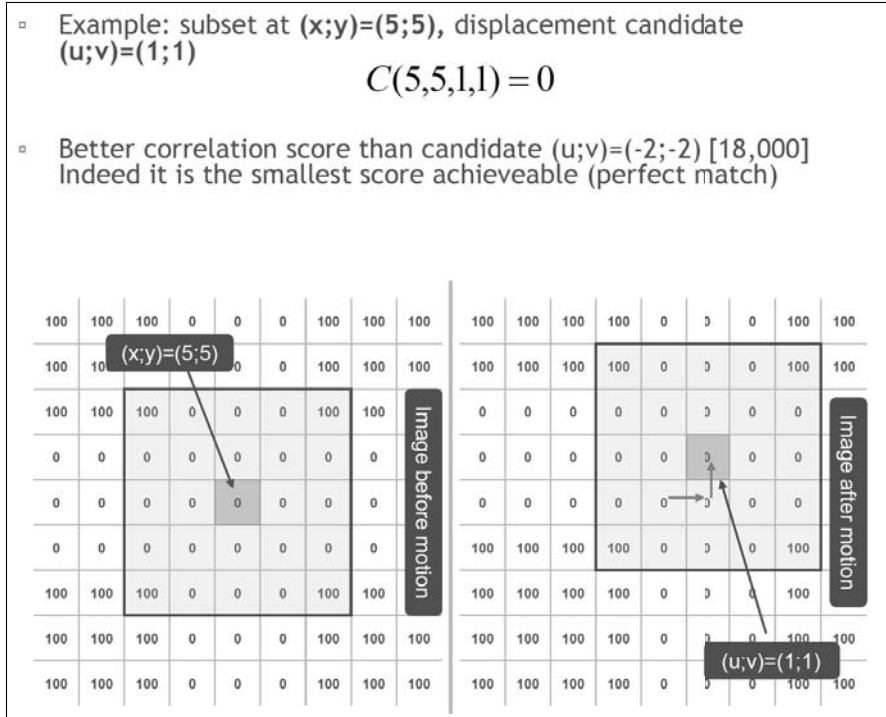


Figure D.4. Perfect Correlation [26]

Conversely, when the potential deformation is consistent with what was generated originally ($u=-2$, $v=-2$), the data within the subset is identical between the reference and deformed images. The correlation between them is perfect, with a SSD of zero, and thus the DIC software would report this as the real deformation and calculate resulting strains as shown in Chapter II. For the purpose of this research, roughly 76,000 subsets are assessed in each image to correlate a mesh of two-dimensional deformations, the results of which are passed through a Gaussian filter (explained in Section 2.6.3) before outputting resultant strains.

It is important to note that “real” data will not have perfect contrast between white and black colors in the speckle, which makes the visual representation of the correlation difficult to interpret, and may increase error in reported deformations due to less disparate correlation scores. Additionally, it should be clear that a uniformly repeated speckle pattern will dramatically increase errors unless the deformations are nonuniform throughout the neighboring regions of each subset. For these reasons, it is important to generate a random speckle pattern with high changes in contrast, as well as minimize the subset size to alleviate potential errors.

Bibliography

- [1] T. Nicholas, *High Cycle Fatigue: A Mechanics of Materials Perspective*. Oxford, UK: Elsevier, 2006.
- [2] *Engine Structural Integrity Program (ENSIP)*, MIL-HDBK-1783B (USAF), 15 Feb 2002.
- [3] ASTM, *E 466: Standard Practice for Conducting Force Controlled Constant Amplitude Axial Fatigue Tests of Metallic Materials*, ASTM International, 2015.
- [4] T. George *et al.*, “Development of a novel vibration-based fatigue testing methodology,” *International Journal of Fatigue*, vol. 26, no. 5, pp. 477–486, 2004.
- [5] T. George *et al.*, “A new multiaxial fatigue testing method for variable amplitude loading and stress ratio,” *Journal of Engineering for Gas Turbines and Power, ASME*, vol. 128, pp. 857–864, 2006.
- [6] T. George *et al.*, “Goodman diagram via vibration-based fatigue testing,” *ASME Journal of Engineering Materials and Technology*, vol. 127, no. 1, pp. 58–64, 2005.
- [7] ASTM, *8M - 09: Standard Test Methods for Tension Testing of Metallic Materials*, ASTM International, 2009.
- [8] O. Scott-Emuakpor, “Development of an improved energy-based criterion for fatigue life assessment,” Master’s thesis, Dept. of Mechanical Engineering, The Ohio State University, Columbus, OH, 2004.
- [9] C. Holycross *et al.*, “In situ study of static and dynamic strain energy density at notch roots and fatigue cracks using digital image correlation,” *57th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, pp. AIAA 2016–0734, 2016.

- [10] C. Holycross *et al.*, “In situ study of strain energy density at notch roots using digital image correlation,” *56th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, pp. AIAA 2015–1576, 2015.
- [11] W. D. Callister and D. G. Rethwisch, *Materials Science and Engineering: An Introduction*, 8th ed. Hoboken, NJ: John Wiley & Sons, 2010.
- [12] A. Atrens *et al.*, “Subsurface crack initiation in high cycle fatigue in Ti6Al4V and in a typical martensitic stainless steel,” *Scripta Met*, pp. 601–606, 1983.
- [13] D. Maxwell and T. Nicholas, “A rapid method for generation of a haigh diagram for high cycle fatigue,” *Journal of Fatigue and Fracture Mechanics*, vol. 29, pp. 626–641, 1998.
- [14] G. Sendeckyj, “History of constant life diagrams,” *High Cycle Fatigue of Structural Materials*, pp. 95–107, 1998.
- [15] T. George, “Development of methodologies for ensuring structural safety of gas turbines and launch vehicles,” Ph.D. dissertation, Dept. of Mechanical Engineering, The Ohio State University, Columbus, OH, 2002.
- [16] J. Seidt, “Development of a novel vibration based high cycle fatigue test method,” Master’s thesis, Dept. of Mechanical Engineering, The Ohio State University, Columbus, OH, 2001.
- [17] K. Knapp *et al.*, “Improved pre-strain method for generating goodman data with vibration-based fatigue testing,” *57th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, pp. AIAA 2016–0925, 4-8 January 2016.
- [18] I. H. Shames and F. A. Cozzarelli, *Elastic and Inelastic Stress Analysis*. Philadelphia, PA: Taylor & Francis Group, 1997.

- [19] R. D. Cook *et al.*, *Concepts and Applications of Finite Element Analysis*, 4th ed. New York, NY: John Wiley & Sons, 2002.
- [20] N.-H. Kim. Chapter 4: FEA for elastoplastic problems. [Online]. Available: <http://slideplayer.com/slide/3431936/>
- [21] ANSYS, *ANSYS Mechanical APDL Material Reference*, ANSYS, Inc., November 2013.
- [22] ANSYS, *ANSYS Mechanical APDL Theory Reference*, ANSYS, Inc., November 2013.
- [23] A. Mróz, “The viscoplastic effect in the heat-treated, thin-walled al-6060 alloy profiles subjected to compressive axial impact,” *Mechanics and Mechanical Engineering*, pp. 53–71, 2014.
- [24] P. Haupt and A. Lion, “Experimental identification and mathematical modeling of viscoplastic material behavior,” *Continuum Mechanics and Thermodynamics*, pp. 73–96, 1995.
- [25] T. Mukai *et al.*, “Influence of strain rate on the mechanical properties in fine grained aluminum alloys,” *Materials Science and Engineering*, pp. 12–18, 1995.
- [26] M. Simonsen, *Minimizing Noise and Bias in 3D DIC*, Correlated Solutions, 2016.
- [27] M. Simonsen, *Strain Calculation in VIC-3D*, Correlated Solutions, 2016.
- [28] S. De, “Constant strain triangle (CST),” *Introduction to Finite Elements*.
- [29] R. Fisher *et al.* (2003) Image processing learning resources. [Online]. Available: <http://homepages.inf.ed.ac.uk/rbf/HIPR2/gsmooth.htm>
- [30] S. Holmes. (2000) RMS error. [Online]. Available: <http://statweb.stanford.edu/~susan/courses/s60/split/node60.html>

- [31] *MTS 647 Hydraulic Wedge Grips Reference Manual: Axial Grips, Axial-Torsional, All-Temperature Grips*, MTS Systems Corporation, 2013.
- [32] O. Scott-Emuakpor *et al.*, “Bending fatigue life comparison between DMLS and cold-rolled nickel alloy 718,” *Machinery Failure Prevention Technology (MFPT) Society Annual Conference*, 20-22 May 2014.
- [33] G. Vision, *Prosilica GE 4000*, Allied Vision.
- [34] U. of South Carolina, *Digital Image Correlation: Overview of Principles and Software*, Correlated Solutions, 2016.
- [35] D. Simons and A. Leissa, “Vibrations of rectangular cantilever plates to in-plane acceleration loads,” *Journal of Sound and Vibration*, vol. 17, no. 3, pp. 407–422, 1971.
- [36] L. Mierovitch, *Principles and Techniques of Vibrations*. New Jersey, USA: Prentice Hall, Inc., 1997.
- [37] J. Bruns, “Fatigue crack growth behavior of structures subject to vibratory stresses,” *Society of Experimental Mechanics Annual Conference*, 2-6 June 2014.
- [38] O. Scott-Emuakpor *et al.*, “Regression study to standardize piezoelectric axial fatigue testing,” *AIAA Science and Technology Forum and Exposition*, pp. AIAA 2015–0891, 5-9 January 2015.

REPORT DOCUMENTATION PAGE					Form Approved OMB No. 0704-0188	
The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.						
1. REPORT DATE (DD-MM-YYYY) 23 March 2017		2. REPORT TYPE Master's Thesis			3. DATES COVERED (From — To) Sept 2016 — 23 March 2017	
4. TITLE AND SUBTITLE TIME-DEPENDENT VALIDATION OF FINITE ELEMENT STRAIN DISTRIBUTION OF A PLASTICALLY-DEFORMED PLATE VIA DIGITAL IMAGE CORRELATION					5a. CONTRACT NUMBER	
					5b. GRANT NUMBER	
					5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S) Knapp, Kevin R., 1st Lieutenant					5d. PROJECT NUMBER	
					5e. TASK NUMBER	
					5f. WORK UNIT NUMBER	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Air Force Institute of Technology Graduate School of Engineering and Management (AFIT/EN) 2950 Hobson Way, Building 640 WPAFB OH 45433-7765					8. PERFORMING ORGANIZATION REPORT NUMBER AFIT-ENY-MS-17-M-269	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) Turbine Engine Fatigue Facility Att: Dr. Tommy George 1950 5th St B20018D RD136 (937)255-7299 Email: Tommy.George@us.af.mil					10. SPONSOR/MONITOR'S ACRONYM(S) AFRL/RQ	
					11. SPONSOR/MONITOR'S REPORT NUMBER(S)	
12. DISTRIBUTION / AVAILABILITY STATEMENT DISTRIBUTION STATEMENT A. APPROVED FOR PUBLIC RELEASE; DISTRIBUTION UNLIMITED.						
13. SUPPLEMENTARY NOTES						
14. ABSTRACT Generation of residual stress is an essential step in the generation of Goodman data via Air Force Research Laboratory's vibration-based fatigue test. Conventional Goodman data is constructed through uniaxial fatigue testing at a rate of 40 Hz, while the vibration-based testing excites stresses at 1,600 Hz in a stress state that is similar to those seen in gas turbine engine airfoils. A pre-strain procedure is conducted to form residual tensile stress, which serves as a steady stress when the specimen is subjected to fully-reversed vibration-based fatigue loads. This steady tensile stress is desired at the fatigue zone of the test article, but is the result of an adjacent compressive region in equilibrium, and as such, a FEM is necessary to determine the stress distribution throughout the entire sample. The goal of this work is to improve the FEM analysis associated with the pre-strain method for better accuracy of steady stress generation for Goodman data fatigue assessment. Improvements were made to the FEM by more effectively incorporating empirical tensile stress-strain behavior, in addition to more accurately representing the pressures and forces acting on the specimen during monotonic loading. Validations of improvements to the pre-strain steady stress generation procedure will be demonstrated on Aluminum 6061-T6 by comparing strain field results from digital image correlation to FEM analysis. The converged quasi-static FEM solution had a standard deviation in ϵ_{yy} of 2,557 microstrain and predicted a residual σ_{yy} of 10.72 ksi, while the optimized time-dependent solution had a standard deviation in ϵ_{yy} of 308.9 microstrain (less than the experimental variation of 376.1 microstrain) and predicted a residual σ_{yy} of 4.93 ksi. The increased accuracy of these models altered residual stresses on a Goodman line by as much as 26% compared with past results.						
15. SUBJECT TERMS Plastic Deformation, Digital Image Correlation, Finite Element Modeling, High Cycle Fatigue						
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON	
a. REPORT	b. ABSTRACT	c. THIS PAGE			Dr. A. Palazotto, AFIT/ENY	
U	U	U	U	178	19b. TELEPHONE NUMBER (include area code) (937) 255-3636, x4599	