University of Cincinnati

Date: 1/17/2014

I, Keith A Adkins, hereby submit this original work as part of the requirements for the degree of Master of Science in Civil Engineering.

It is entitled:
A Model for Prediction of Fracture Initiation in Finite Element Analyses of Welded Steel Connections

Student's name: Keith A Adkins

This work and its defense approved by:

Committee chair: James Swanson, Ph.D.

Committee member: Kenneth Wurzelbacher, M.S.

Committee member: Gian Rassati, Ph.D.
A Model for Prediction of Fracture Initiation in Finite Element Analyses of Welded Steel Connections

A thesis submitted to the

School of Advanced Structures

In partial fulfillment of the requirements for the degree of

Master of Science in Civil Engineering

College of Engineering and Applied Science
University of Cincinnati

by

Keith Alan Adkins
Bachelor of Science Civil Engineering, 2012
From the University of Cincinnati, College of Engineering and Applied Science

Committee Chair: James A. Swanson, Ph.D.

Committee Members:
Gian Andrea Rassati, Ph.D.
Kenneth P. Wurzelbacher, MSCE
Abstract

This paper investigates the implementation of damage mechanics into finite element models of fillet-welded structural steel assemblies in order to predict connection fracture at realistic displacements. Utilizing a previously developed IDS (instability, ductile, and shear) material failure envelop for aluminum extrusions, the failure loci of the aluminum material were scaled to meet the point of fracture in welded steel lap splice assemblies, loaded in-plane at various angles, with a fillet weld leg size of 5 mm. The weld electrode under investigation was CSA E480xx, which is equivalent to AWS E70xx electrodes. A linear relationship for damage data scaling factors, as they relate to the loading angle of the fillet weld, was established. Three similar assemblies with fillet leg sizes of 9 mm were modeled employing this damage scaling relationship and deformations at fracture were found to coincide with experimental results and established relationships. This modeling methodology was also investigated for use in a T-stub connection tension scenario.
Acknowledgements

A number of individuals have assisted me both directly and indirectly throughout the development of thesis, allowing its completion to become a reality.

I would first like to thank my committee members, Dr. James Swanson and Dr. Gian Andrea Rassati, for their instruction both within the classroom and in the world of scientific conquest. Their advising methods both provided the necessary guidance to complete the tasks at hand and allowed for personal growth in the realm of individual research. Additionally, I’d like to thank Kenneth Wurzelbacher for taking time out of his foray into the world of structural engineering practice to advise my efforts, for which his research laid the groundwork.

I’d like thank my parents for their love and support throughout my entire life, providing the guidance and financial support to achieve my academic and life goals. Without their care taken to ensure satisfactory progress in school throughout my academic career, I would not be where I am today.

I’d like to thank my colleagues Kyle Ficker, Kyle Kungle, Cory Larkin, Mathias Kozma-Thomas and Utsab Dhungana. Their camaraderie has been a nearly constant source of lively banter provided a much needed source of levity to my research endeavors.

A special thanks is owed to my girlfriend Jacki O’Brien, whose constant support, admiration, and cooking has been essential to the successful completion of my research goals. I cannot imagine a better individual with whom to spend my time.
Table of Contents

Abstract..................................................................................................................................................... i
Acknowledgements ....................................................................................................................................... iii
Chapter 1: Introduction .................................................................................................................................. 1
  1.1 Objective .............................................................................................................................................. 2
  1.2 Outline ................................................................................................................................................ 2
Chapter 2: Background ............................................................................................................................... 3
  2.1 Fillet Weld Design Specification ........................................................................................................ 3
  2.2 Damage Mechanics in Abaqus .......................................................................................................... 7
  2.3 Applications of Damage Mechanics in Finite Element Modeling ..................................................... 10
Chapter 3: Procedural Outline ................................................................................................................... 12
  3.1 Procedure .......................................................................................................................................... 12
  3.2 Tested Lap Splice Assemblies .......................................................................................................... 14
  3.3 Tested T-Stub Assemblies ............................................................................................................... 19
Chapter 4: Modeling Methodology .......................................................................................................... 22
  4.1 Material Property Definition ......................................................................................................... 22
  4.2 Choice of Modeling Methodology ................................................................................................. 23
  4.3 Nodal Connectivity and Contact ................................................................................................... 27
  4.4 Element Selection and Convergence Study ..................................................................................... 30
  4.5 Summary of Methodology ............................................................................................................. 34
Chapter 5: Finite Element Models ........................................................................................................... 36
  5.1 Lap Splice Assemblies ..................................................................................................................... 36
  5.2 T-Stub Assemblies .......................................................................................................................... 42
Chapter 6: Analysis of Results ................................................................................................................... 46
  6.1 Displacements at Fracture and Ultimate Load ................................................................................ 46
  6.2 Collection of Results (Lap Splice Assemblies) ............................................................................... 47
  6.3 Results of 5 mm Weld Assemblies .................................................................................................. 49
  6.4 Establishment of Damage Scaling Factor Relationship ................................................................. 56
  6.5 Results of 9-mm Weld Assemblies .................................................................................................. 57
  6.6 Collection of Results (T-Stub Assemblies) ..................................................................................... 62
  6.7 Results for T-Stub Assemblies ....................................................................................................... 63
List of Tables

Table 5-1: Representative Plasticity Data for 300W Steel .............................................................. 38
Table 5-2: Representative Plasticity Data for E48014 Weld Metal ..................................................... 38
Table 5-3: Representative Plasticity Data for A572-Gr. 50 Steel .......................................................... 43
Table 5-4: Representative Plasticity Data for E70T-7 Weld Metal ....................................................... 44
Table 6-1: Locations of Displacement Measurement from Edge of Test Assembly ............................. 48
Table 6-2: 90 Deg-5 mm Experimental/Predicted Ultimate Loads and Deformations ....................... 49
Table 6-3: 0 Deg-5 mm Experimental/Predicted Ultimate Loads and Deformations ........................... 51
Table 6-4: 60 Deg-5 mm Experimental/Predicted Ultimate Loads and Deformations ........................ 54
Table 6-5: 30 Deg-5 mm Experimental/Predicted Ultimate Loads and Deformations ........................... 55
Table 6-6: Tabulated Results of Aluminum Damage Scaling Factors .................................................. 56
Table 6-7: 90 Deg-9 mm Experimental/Predicted Ultimate Loads and Deformations ....................... 58
Table 6-8: 0 Deg-9 mm Experimental/Predicted Ultimate Loads and Deformations ........................... 59
Table 6-9: 45 Deg-9 mm Experimental/Predicted Ultimate Loads and Deformations .......................... 60
Table A-1: Experimentally Determined Ductile Failure Parameters ................................................... 74
Table A-2: Experimentally Determined Shear Failure Parameters ...................................................... 74
Table C-1: Aluminum Damage Data (Abaqus Example Problems Manual, 2012) ............................... 76

List of Figures

Figure 2-1: Typical Stress Distribution - Longitudinal Lap Joint (Adapted: Salmon et al. 2009) ........ 3
Figure 2-2: Typical Stress Distribution - Transverse Lap Joint (Adapted: Salmon et al. 2009) .......... 4
Figure 2-3: Effective Throat for Equal Leg Fillet Welds (Adapted: Salmon et al. 2009) .................... 4
Figure 2-4: Load-Deformation Relationship for Fillet Welds (AISC 2010) ........................................ 6
Figure 2-5: Typical Uniaxial Stress-strain Response of a Metal Specimen (Abaqus 2012) .............. 7
Figure 2-6: Hydrostatic Pressure and Growth of Microvoids (Engelen et al. 2011) ......................... 9
Figure 2-7: Shear Band Causing Growth of Microvoids Illustration (Engelen et al. 2011) ............. 9
Figure 2-8: Experimental/Analytical Models of T-section Post Failure (Wurzelmacher 2012) ....... 11
Figure 3-1: Ductile Failure Criterion Envelope for EN AW-7108 T6 Aluminum ........................... 13
Figure 3-2: Shear Failure Criterion Envelope for EN AW-7108 T6 Aluminum .............................. 13
Figure 3-3: Representative Test Specimen Schematic (Miazga and Kennedy, 1889) ............... 15
Figure 3-4: Selected Specimens for 5-mm Fillet Welds (Adapted: Miazga and Kennedy 1989) 16
Figure 3-5: LVDT Frame for Continuous Weld Specimens (Miazga and Kennedy 1986) .... 18
Figure 3-6: LVDT Frame for Discontinuous Weld Specimens (Miazga and Kennedy 1986) .... 18
Figure 3-7: T-Stub Tension Test Schematic (Gomez et al. 2008) ........................................ 20
Figure 3-8: Representative Test Photo (Gomez et al. 2008) .................................................. 20
Figure 3-9: Schematic of Potentiometer Mounting Cart (Gomez et al. 2008) ......................... 21
Figure 4-1: 2-D FE Mesh of T-stub Assembly (Gomez et al. 2008) ....................................... 24
Figure 4-2: Isometric View of Initial Lap Splice Model ......................................................... 25
Figure 4-3: Normalized Load-Displacement for Lap Splice Models with Eqn. 4-3 ............... 27
Figure 4-4: Isometric View of Fully Meshed Input Model ..................................................... 29
Figure 4-5: Force-Displacement Results of Nodal Connectivity Study ................................. 29
Figure 4-6: Weld Mesh Cross Section with Initially Deformed Element Marked ................. 31
Figure 4-7: Investigated Partitioning Schemes (top) and Corresponding Mesh (bottom) ...... 32
Figure 4-8: Mesh Sensitivity Analysis for the "2x" Partitioning Scheme ................................. 34
Figure 5-1: Comparison of Full Model vs. Half-Symmetric Model .................................... 37
Figure 5-2: Representative Views of 90 Degree-5 mm Model ............................................. 39
Figure 5-3: Representative Views of 60 Degree-5 mm Model ............................................. 40
Figure 5-4: Representative Views of 30 Degree-5 mm Model ............................................. 40
Figure 5-5: Representative Views of 0 Degree-5 mm Model ............................................. 41
Figure 5-6: Representative Views of 45 Degree - 9 mm Assembly ................................... 42
Figure 5-7: Isometric View of T-stub Model ................................................................. 45
Figure 6-1: General Illustration Corresponding to Table 6-1 ........................................... 48
Figure 6-2: Results of 90 Degree - 5 mm Weld Assembly Model ......................................... 50
Figure 6-3: Results of 0 Degree - 5 mm Weld Assembly Model ........................................ 51
Figure 6-4: Illustration of Full-Fracture Initiation ............................................................ 53
Figure 6-5: Results of 60 Degree - 5 mm Weld Assembly Model .......................................... 54
Figure 6-6: Results of 30 Degree - 5 mm Weld Assembly Model .......................................... 55
Figure 6-7: Comparison of Normalized 5-mm Results to Eqn. 4-3 at Varying Weld Angles .... 56
Figure 6-8: Results of Damage Data Calibration with Linear Relationship Plotted ............ 57
Figure 6-9: Results of 90 Degree - 9 mm Weld Assembly Model .......................................... 58
Figure 6-10: Results of 0 Deg - 9 mm Weld Assembly Model ................................................................. 59

Figure 6-11: Results of 45 Deg - 9 mm Weld Assembly Model ............................................................... 61

Figure 6-12: Comparison of Normalized 9-mm Results to Eqn. 4-3 at Varying Weld Angles...................... 62

Figure 6-13: View of T-Stub Assembly with Reference Points Shown ....................................................... 63

Figure 6-14: Results of T-Stub Test ........................................................................................................... 64

Figure 6-15: Progression of Damage in T-Stub Assembly (Von Mises Contours Plotted) ......................... 65

Figure D-1: E48014 Weld Metal Engineering Stress vs. Strain (Miazga and Kennedy 1986) ............... 79

Figure D-2: E48014 Weld Metal True vs. Engineering Stress-Strain ...................................................... 79

Figure D-3: 300W Steel Engineering Stress vs. Strain (Miazga and Kennedy 1986) ............................ 80

Figure D-4: 300W Steel True Stress vs. Logarithmic Plastic Strain ......................................................... 80

Figure D-5: E70T-7 Weld Metal Engineering Stress vs. Strain (Gomez et al. 2008) ............................. 81

Figure D-6: E70T-7 Weld Metal Engineering vs. True Stress-Strain ...................................................... 81

Figure D-7: A572 Gr. 50 Steel Engineering Stress vs. Strain (Gomez et al. 2008) ................................. 82

Figure D-8: A572 Gr. 50 Steel Engineering vs. True Stress-Strain ......................................................... 82

Figure D-9: E70T-7 & E48014 Weld Metal True Stress-Strain Comparison ............................................ 83

Figure F-1: Mesh Sensitivity Analysis for the "Null" Partitioning Scheme ............................................ 85

Figure F-2: Mesh Sensitivity Analysis for the "1x" Partitioning Scheme ................................................ 86

Figure F-3: Mesh Sensitivity Analysis for the "3x" Partitioning Scheme ................................................ 86

Figure G-1: 5 mm Fillet Weld Assemblies .................................................................................................. 87

Figure G-2: 9 mm Fillet Weld Assemblies .................................................................................................. 88

Figure H-1: 0-Deg Assembly Experimental Results (Lesik and Kennedy 1990) ..................................... 89

Figure H-2: 30-Deg Assembly Experimental Results (Lesik and Kennedy 1990) ................................... 89

Figure H-3: 90-Deg Assembly Experimental Results (Lesik and Kennedy 1990) .................................. 90

Figure H-4: 0-Deg Experimental vs. Analytical Comparison ................................................................. 90

Figure H-5: 30-Deg Experimental vs. Analytical Comparison ............................................................... 91

Figure H-6: 90-Deg Experimental vs. Analytical Comparison ............................................................... 91
Chapter 1: Introduction

Welded steel connections are commonly used in steel construction, but the strength of the weld itself is rarely optimized and typically overdesigned due to high variance in the strength and ductility of welds. This is represented by the resistance factor of 0.75 used in the American Institute of Steel Construction Specifications for Structural Steel Buildings, which governs the design and construction of steel structures. Recently, the proliferation of shop welding and the utilization of automated machinery to perform welding procedures has led to a higher consistency of weld quality. These innovations have, in turn, increased the viability of simulations to accurately predict the behavior of a welded joint. The prediction of fracture initiation in fillet welded connections will be the focus of this study.

Welded connections have been the subject of finite element studies in the past, however an assembly in which the weld is the critical element in the connection is rarely the subject of such studies. Studies have investigated the stress distribution of the weld and some detailed simulations have included the effects of residual stresses in and around the weld nugget (Popov et al. 1998; Kanvinde et al. 2013), but the technology of finite element software has not allowed for a true fracture analysis until recently. The ability to reduce the cost of new connection research by utilizing finite element simulation for pre- and post-design of assemblies has the potential to increase the pace of innovation in structural engineering dramatically.

New research into continuum damage mechanics has led to further applications of finite element simulations in the realm of structural connections (Hooputra et al. 2004; Wurzelbacher 2012). A newly developed strategy for scaling a fully characterized fracture locus of aluminum extrusions to match fracture of structural steel and bolts has been presented in recent work at the University of Cincinnati. This study will extend these efforts to structural fillet welds.
1.1 Objective

The objective of this thesis is to formulate a method for utilizing finite element analysis to predict the initiation of fracture of fillet welded structural connections. The commercial finite element simulation software Abaqus Ver. 6.12 was utilized for this study. A previously developed material failure envelope for aluminum extrusions based on IDS (instability, ductile, and shear) damage initiation criteria was used in conjunction with failure locus scaling methodologies from Kenneth P. Wurzelbacher’s research regarding fracture prediction of bolted steel connections (Wurzelbacher 2012). Damage initiation criteria was calibrated for the finite element analysis by utilizing experimental data from Miazga and Kennedy’s 1986 study of fillet weld behavior and scaling the aluminum damage data to the point of fracture in welded assembly experimental tests (Miazga and Kennedy 1986).

1.2 Outline

This current chapter introduces the objectives and goals of the research contained herein. Chapter 2 provides background into the current design methodologies of structural welds, the application of damage mechanics into finite element simulations, and the previous work conducted on this topic. Chapter 3 describes the procedure used to complete the goals of this thesis and describes the assemblies upon which the finite element models were created. Chapters 4 and 5 detail the general methodology utilized in creating the finite element models and the particular modeling methods employed, specific to the individual models. Analyses of the results obtained in the study are provided in Chapter 6. Chapter 7 provides conclusions based upon the work performed, and the final chapter provides recommendations for future analytical and experimental studies. Appendices are provided to supplement the main body text.
Chapter 2: Background

2.1 Fillet Weld Design Specification

The AISC Specification for Structural Steel Buildings (AISC 2010) provides provisions for the various types of welds commonly used in steel construction. The strength of a weld per unit length is given as a function of its effective area and the strength of the weld electrode used, limited by the strength of the joined base metal (AISC 2010). Certain weld types, such as groove welds, are considered to exhibit identical strength as the parts being joined so long as a “matching” weld metal is utilized, and will transfer loads similarly (Salmon et al. 2009).

The primary focus of this study will be on fillet welds, which have a more complex stress distribution and a variable strength-ductility relationship based on the angle of loading (AISC 2010). Representative stress distributions for longitudinal and transverse loading orientations are shown in Figures 2-1 and 2-2, respectively (Adapted from Salmon et al. 2009).

Figure 2-1: Typical Stress Distribution - Longitudinal Lap Joint (Adapted: Salmon et al. 2009)
The effective area of a fillet weld is determined as the effective throat multiplied by the effective length of the weld. The effective throat is defined by AISC 2010 as the shortest distance from the root to the face of the diagrammatic weld, as shown in Figure 2-3. For an equal leg fillet weld, the effective throat can be simply determined as \( t_e = 0.707a \), where 'a' is the length of a single leg.

Figure 2-3: Effective Throat for Equal Leg Fillet Welds (Adapted: Salmon et al. 2009)
The strength calculations for fillet welds assume that failure is governed by shear through the effective throat, regardless of the angle of loading relative to the weld axis. This assumption is made for simplicity, despite the fact that a fillet loaded perpendicular to the weld axis (transversely) has a greater strength than that of a weld loaded longitudinally (Salmon et al. 2009). AISC 2010 presents the governing strength calculations for a fillet weld in the following equation:

$$R_n = F_{nw}A_{we} \quad \text{(Eqn. 2-1)}$$

where:

- $F_{nw} = 0.60 F_{EXX}$
- $A_{we} = \text{Effective area of weld}$

As observed in the preceding equation, the nominal tensile strength of the weld metal is multiplied by a factor of 0.60, which is the typical multiplication factor used in AISC 2010 to relate tensile strength to shear strength. The distribution of stresses in fillet welded connections is non-uniform and highly dependent on complex factors regarding the respective stiffness of the weld metal in relation to that of the connected members and the thicknesses of the connected material (Salmon et al. 2009; AISC 2010). However, for longitudinally end-loaded fillet weld groups, in which fillet welds parallel to the line of force are used to transmit axial load to the end of a member, experience has shown that in spite of this non-uniform distribution, a weld with a length up to approximately 100 times the weld size can be assumed to resist force equally at any point along the length (AISC 2010).

Unlike bolt groups, the force-displacement relationship of fillet weld groups is a function of the angle by which the fillet weld group is loaded. This relationship was originally postulated by Butler et al. in 1971 and incorporated into the AISC 1993 LRFD Specification (AISC 1993) following research by Lesik and Kennedy in 1990 (Salmon et al. 2009). The normalized graphic representation of the force-deformation relationship, championed by Lesik and Kennedy, is shown in Figure 2-4, reproduced from AISC 2010.
As alluded to previously, the transversely loaded fillet weld (90°) generally provides a 50% increase in strength over the longitudinal orientation (0°), but provides significantly less ductility. The original empirically derived formula by Lesik and Kennedy has been modified for conservatism to its current form in the *Specification*, as shown in Eqn. 2-2. Taking the pre-existing shear factor of 0.60 in place of Lesik and Kennedy’s reduction factor, the *Specification* provides “a reasonable margin for any variation in welding techniques and procedures,” (AISC 2010). The maximum allowable deformation of weld elements is also limited to 0.17w (where w represents the weld size) to reduce the risk of computational difficulties. This limit is represented by the dashed line in Figure 2-4.

\[
R_n = 0.60 F_{EXX} \left(1.0 + 1.5 \sin^{1.5} \theta\right) A_{we} \tag{Eqn. 2-2}
\]
An accurate replication of the load-deformation relationship, presented above, in the finite element simulations is a paramount goal of this study. The preliminary studies conducted by the author, presented herein, have exhibited a correlation to the load-deformation relationship postulated by Lesik and Kennedy, but are unsuccessful in replicating an ultimate strength without the implementation of damage mechanics.

2.2 Damage Mechanics in Abaqus

The Abaqus finite element analysis software offers a number of options for the inclusion of damage mechanics into FE simulations. To introduce the concept of damage in the context of finite element analysis, the Abaqus 6.12 User’s Manual (Abaqus 2012) offers the graphic reproduced as Figure 2-5, representing a typical uniaxial stress-strain response of a metal specimen.

Figure 2-5: Typical Uniaxial Stress-strain Response of a Metal Specimen (Abaqus 2012)

In Fig. 2-5, line a-b represents the initial loading phase where the specimen is linearly elastic; b-c shows the strain hardening phase, typical of metals; and c-d represents unloading and ultimately, the failure of the specimen. Diverging at point c, the dashed line leading to d' illustrates the typical response of a finite element simulation that does not include damage mechanics. This
is due to the fact that once Abaqus has iterated an element through the user-defined plastic hardening portion of the plasticity definition, the element begins behaving in a perfectly plastic deformation mode. A properly defined damage model consists of the implementation of both damage initiation criteria and damage evolution to achieve a realistic representation of failure.

For ductile metals, Abaqus offers three general modes of including damage initiation. These modes are based on the work of H. Hooputra in his 2004 publication, “A Comprehensive Failure Model for Crashworthiness Simulation of Aluminium (sic) Extrusions” (Hooputra et al. 2004). Hooputra’s damage criteria, referred to as IDS (Instability, Ductile, and Shear) criteria were published in the International Journal of Crashworthiness in conjunction with the BMW automotive group. Intended for use in commercial crashworthiness finite element simulations, the material upon which these criteria were developed is EN AW-7108 T6 automobile grade aluminum. Please note that in the confines of this thesis, failure by instability will not be investigated as it applies primarily to low-thickness sheet metal and, further, did not present itself as the dominating failure mode in the experiments performed by Hooputra (Hooputra et al. 2004).

The ductile failure mode postulated by Hooputra is a function of the stress triaxiality, which is a ratio of the hydrostatic stress to the von Mises stress. Ductile failure (Figure 2-6) occurs with the “initiation, growth, and coalescence of micro-voids” in the ductile metal (Hooputra et al. 2004). As these micro-voids continue to grow in area of high strain, they will ultimately join together to cause material failure. Hooputra also notes that the strain rate upon which a material is deformed has a significant impact on the strain at damage initiation. To account for this, parameters were established at both quasi-static (0.001 strain rate) and dynamic (250 strain rate) loading rates. The tests required for determination of these parameters are listed in Appendix B.
Abaqus employs damage initiation criterion by utilizing the following function. If the condition in the proceeding expression for the state variable $\omega_D$ is met, damage will initiate:

$$\omega_D = \int \frac{d\varepsilon^{pl}}{\varepsilon^{pl}(\eta, \varepsilon^{pl})} = 1 \quad \text{(Eqn. 2-3)}$$

In addition to dependence on stress triaxiality, Abaqus Ver. 6.12 offers additional functionality to define ductile fracture based upon the third invariant of deviatoric stress, related to the Lode angle, from the work of Bai and Wierzbicki in the 2008 publication “A New Model of Metal Plasticity and Fracture with Pressure and Lode Dependence.” This definition of ductile damage initiation is discussed in Chapter 8.

Shear fracture (Figure 2-7) occurs due to the localization of shear bands in a ductile metal, leading to void growth and material failure (Hooputra et al. 2004). Shear fracture is dependent upon both the stress triaxiality and the ratio of the maximum shear stress to the von Mises stress. These two parameters are related by a material parameter, $k_s$, which for metal can be taken generally as 0.3 (Wurzelbacher 2012).
Based on the criteria defined for damage initiation, Abaqus employs damage evolution criteria to control element deletion. Damage evolution can be defined in Abaqus using tabular, linear or exponential functions for the progression of fracture through a material (Abaqus 2012). Based on experimental observations, Wurzelbacher suggests that damage evolution need not be utilized in finite element simulations of structural steel due to the relatively instantaneous rate of crack propagation through the material. Additionally, any delay in crack propagation will be accounted for by the damage initiation criteria, as it is defined in terms of equivalent plastic strain at failure (Wurzelbacher 2012).

2.3 Applications of Damage Mechanics in Finite Element Modeling

The initial investigation of damage initiation implementation into structural steel finite element simulations was performed at the University of Cincinnati by Kenneth Wurzelbacher and is presented in the 2012 thesis, “A Model for Prediction of Fracture Initiation in Finite Element Analysis of Bolted Steel Connection.” Wurzelbacher’s work focused on utilizing the ductile and shear damage initiation criteria parameters presented by Hooputra for aluminum, in lieu of conducting material testing to establish the full failure surface, to predict failure in bolted connections. Welded connections were not investigated in Wurzelbacher’s study. Hooputra’s data from aluminum specimens was scaled to match the initial failure from previously tested structural steel and bolt specimens. The result was a model for shifting the failure locus of EN AW-7108 T6 aluminum accordingly to predict initial fracture in typical steel and bolt designations.

From Wurzelbacher’s investigation of A36 bar specimens, the established scale factor to predict the initial onset of fracture in tension was established at 0.8. For A992 T-sections, a scale factor of 2.0 was selected. A325 and A490 bolts in both tension and shear used unmodified AW-7108 aluminum data. Using these scale factors calibrated from the separate component testing, this
data was incorporated into two full scale simulations of A572-50 T-stub beam-to-column connections for verification and predicted the onset of fracture with a substantial degree of success. A side-by-side comparison of an experimental image and the finite element model of a T-section in uniaxial tension after failure is shown in Figure 2-8. It is the goal of this study to formulate a similar fracture prediction module for welded structural connections.

Figure 2-8: Experimental/Analytical Models of T-section Post Failure (Wurzelbacher 2012)
Chapter 3: Procedural Outline

The steps that were taken to complete the objectives of this thesis are described in this chapter. Included are the preliminary studies that were completed to ensure proper finite element modeling of the welded assemblies. The previously conducted experimental tests that were used for modeling and the establishment of damage relationships are also discussed in this chapter.

3.1 Procedure

The procedure used to develop a damage initiation relationship for fillet welds, described herein, generally follows the procedure presented in Wurzelbacher 2012. Similarly, weld metal specimens were not subjected to the litany of tests required to establish a full material failure locus. Instead, the failure surface for EN AW-7108 aluminum developed by Hooputra, was scaled to match the point of failure of assemblies from Miazga and Kennedy’s 1986 experimental testing program that was used to develop the load-deformation relationship that appears in the AISC *Specification for Structural Steel Buildings* (AISC 2010). The tested assemblies utilized two weld sizes and a number of loading angles. Scaling the equivalent plastic strain axis while leaving the constituents of the failure surface unmodified allowed the deformation of the element at failure to change but leaves the general characteristics of the surface intact. Therefore, only the point of weld failure truly represents the fracture locus of the weld metal, while the general shape of the surface still is representative of aluminum. Illustrations of the ductile and shear failure loci are show in Figure 3-1 and Figure 3-2, respectively. Values outside of the range determined by Hooputra are held constant.

As introduced previously, the stress triaxiality and shear stress ratio are measures of the state of stress of a material. Since fillet welds loaded at differing angles exhibit varied responses, the damage scaling factor for fillet welds must be established as a function of the loading angle.
This is due to the fact that the critical elements in a fillet weld model will experience a unique state of stress at failure, based on its loading condition.

**Figure 3-1**: Ductile Failure Criterion Envelope for EN AW-7108 T6 Aluminum

**Figure 3-2**: Shear Failure Criterion Envelope for EN AW-7108 T6 Aluminum
Once the damage scaling factor relationship was established and verified for weld size independence, the damage values were implemented into a model of a T-stub tension test of a fillet welded connection, via 2008 research by Ivan Gomez. This modeling investigates use of the aluminum data scaling relationship in a different test set up and allows implementation of Wurzelbacher's base metal damage values verifying compatibility with the weld damage data.

3.2 Tested Lap Splice Assemblies

In 1989 G.S. Miazga and D.J.L. Kennedy published the results of their 1986 study entitled “Behavior of Fillet Welds as a Function of the Angle of Loading” in the Canadian Journal of Civil Engineering. As a part of the study, 42 fillet-welded lap splice specimens were tested in tension to examine the behavior of fillet welds subjected to a varying loading angle. These tests later became the basis of D.F. Lesik and D.J.L. Kennedy’s load-deformation relationship, currently accepted by the AISC Specification, documented in the 1990 publication “Ultimate Strength of Fillet Welded Connections Loaded in Plane.” These tested assemblies were chosen based on the availability of modeling and result data, the investigation of varying loading angles and weld sizes, and the use of an E70xx equivalent weld metal.

The test specimens were prepared and tested at the I.F. Morrison Structural Laboratory of the University of Alberta. Each test specimen was designed in accordance with 1984 Canadian Standards Association (CSA) Specifications. The assemblies consisted of three plates connected via equal leg fillet welds. Two fillet weld leg sizes of 5 mm and 9 mm were investigated. The middle plate was thicker than the two connecting plates, with an 18 mm thick middle plate connecting to 9 mm thick plates for the 5 mm fillet-welded assemblies (35 mm and 18 mm for 9 mm fillet welds, respectively). The bottom end of the test assembly was placed directly into hydraulic grips while the top end was pinned in the testing machine. A representative schematic is
provided in Figure 3-3. Refer to “Structural Engineering Report No. 133”, Miazga and Kennedy 1986, for further details regarding the assembly design process.

Figure 3-3: Representative Test Specimen Schematic (Miazga and Kennedy, 1889)
All of the assemblies were tested in nominally uniaxial tension. To mimic the effect of loading the assemblies in different orientation, the fillets were deposited at angles measured from the plane of uniaxial tension – 0 degrees, representing a longitudinal loading condition, 90 degrees representing a transverse loading condition and the intermediate angles of 15, 30, 45, 60 and 75 degrees. Figure 3-4 provides schematics of the selected angle orientation assemblies for the 5-mm fillet weld specimens. The inclusion of a plate connected on the bottom side of the test specimen was used to create a symmetric condition about the neutral axis of the middle plate. This reduces the effect of eccentric loading causing bending moments.

Figure 3-4: Selected Specimens for 5-mm Fillet Welds (Adapted: Miazga and Kennedy 1989)
For certain weld orientations, it was determined in design that a continuous weld bead would have resulted in general plate yielding prior to weld rupture. This situation arose in the 45, 30 and 15 degree orientations for the 5-mm fillet weld specimens and in the 30 and 15 degree orientations for the 9-mm fillet welds. To combat this, the smaller connecting plates were milled at angles accommodating two lesser-length welds and which were deposited accordingly. Run-off tabs were used for the placement of all fillet welds, connected with tack welds placed in dead zones for continuous weld beads and on the plate faces for specimens utilizing discontinuous weld beads. All fillet welds were machined parallel to the direction of loading on both ends of the test length.

Data was collected from the test via a data acquisition system connected to the MTS testing machine, various electronic resistance strain gauges mounted on the test specimen, and linear variable differential transformers mounted on the test specimen via rigid frames. The tests were conducted in quasi-static load control in the 1300 kN load range. Strain gages were provided to monitor strain of the plates to ensure the plates remained elastic throughout the extent of testing. Gages were also mounted adjacent to the welds to monitor local plate yielding.

The LVDTs were utilized to monitor the deformations experienced in the fillet welds, data of primary interest in the context of this thesis. A rigid frame mounted on the middle plate provided support for 2 LVDTs on each face of the test specimen. The spindle of the LVDT made contact with the connection plates a short distance above the weld, allowing for relative displacement between the plates adjacent to the welds. Schematics for the LVDT frames used for continuous bead specimens and discontinuous bead specimens are shown in Figures 3-5 and 3-6, respectively.
In addition to the fillet welded assemblies, a number of ancillary tests were performed to determine the material properties of the base and weld metals. The material used for the plates was grade CSA Standard 300W steel. Via CSA Standard W59 (1982), E48014 electrodes were selected.
as the matching electrode for 300W steel. E480xx electrodes are accepted as an equivalent designation to American Welding Society (AWS) E70xx electrodes (480 MPa ≈ 70 ksi), which are commonly used in structural connections. Seven tension coupons of 300W steel were produced from each of the three plate thicknesses used in the testing, and three all-weld metal tension coupons of E48014 filler material were produced, all in accordance with current standards at the time of material testing. The results of these ancillary tests were used by the author for representative material plasticity data in the finite element simulations.

For establishment of a scale factor relationship as it relates to the fillet weld loading angle, the 0, 30, 60 and 90 degree orientation of the 5-mm fillet welds were selected for FE modeling. Schematics for these for assemblies were shown in Figure 3-4. These four assemblies were selected as they provided three even increments of loading angle to establish scaling factors and allowed for study of both the continuous and discontinuous fillet weld depositions. Once a scale factor relationship was developed from these orientations, the 9-mm fillet weld assemblies for the 0, 45 and 90 degree orientations were used to verify independence from weld size and also investigated a loading angle (45°) different from which the factors were established.

3.3 Tested T-Stub Assemblies

In 2008, Ivan Gomez and Amit Kanvinde (University of California, Davis) in conjunction with Yu Kay Kwan and Gilbert Grondin (University of Alberta) published the results of their study “Strength and Ductility of Welded Joints Subjected to Out-of-Plane Bending” as a report to the American Institute of Steel Construction (AISC). As a part of this study, the effect of the root notch length on a fillet weld group’s strength and ductility was investigated via T-stub tension tests. The full scale out-of-plane bending tests were not investigated in the confines of this thesis but are suggested as future work in Chapter 8.
In the study, two nominal root notch lengths of 1.25” and 2.5” were investigated by varying the thickness of the connected plate between the fillet weld lengths. In addition to varying the root notch length, two weld metal classifications were used deposited as 5/16” and 1/2” equal leg fillet weld sizes. The electrode classifications used were AWS E70T-7 (non-toughness rated) and E70T7-K2 (toughness rated). Three specimens for each combination were tested, resulting in 24 total tests. All base metal used was ASTM A572 Gr. 50 steel. A general test set-up illustration and photo are shown in Figure 3-7 and Figure 3-8, respectively.

Figure 3-7: T-Stub Tension Test Schematic (Gomez et al. 2008)

Figure 3-8: Representative Test Photo (Gomez et al. 2008)
Deformation of the fillet weld shear leg due to tension was measured using linear potentiometers as shown in Figure 3-9. Results of peak loadings and ultimate shear leg deformations were provided in the publication. These results were used for comparison in the finite element models.

As with Miazga and Kennedy’s study, ancillary tests of tension coupons for the weld electrodes used and A572-Gr. 50 base metal were conducted with results provided. These results were again used for representative plasticity data in the finite element simulations.

Figure 3-9: Schematic of Potentiometer Mounting Cart (Gomez et al. 2008)
Chapter 4: Modeling Methodology

A discussion of the finite element modeling procedure of fillet welded connections is provided in this chapter. The tests prior to the implementation of damage into the FE models are described and results are provided. All finite element simulations were conducted using Abaqus Ver. 6.12 analysis software under the licensing of the University of Cincinnati. Preliminary tests used the Abaqus Standard solver method, while Abaqus Explicit was utilized as the primary solver in this study for its ability to handle the dynamic shifts experienced during damage evolution.

4.1 Material Property Definition

The primary materials in this study are structural steel and weld electrode metal. Typically accepted values of Young’s Modulus and Poisson’s Ratio for structural steels are 29,000 ksi (200,000 MPa) and 0.3, respectively. Analysis of all-weld metal tension tests from Miazga and Kennedy (1986) and Gomez et al. (2008) suggest that Young’s Modulus for weld metal can generally be taken equal to that of steel. Research of Lee, J.Y. et al. (2010) regarding the Poisson’s Ratio of deposited V-groove welds suggest that 0.3 is also representative of weld material. Therefore, consistent values of Young’s Modulus and Poisson’s Ratio were used for all materials in the finite element models.

When using the Abaqus Explicit solver, the primary solver used in this study, density must be defined for all models. Due to the stable increment relationship used in Abaqus Explicit’s central difference method computation, use of the true density of steel will result in a smaller time increment, thus longer computational time. The number of increments required can be estimated via the following equation (Abaqus 2012):

\[ n \approx T \max \left( \frac{1}{L_e} \sqrt{\frac{\lambda + 2\mu}{\rho}} \right) \]  

(Eqn. 4-1)
In the previous equation, $T$ represents the time period of the event being simulated, $L_e$ is the characteristic length of the critical element, and the variables $\hat{\lambda}$ and $\hat{\mu}$ are the effective Lamé’s constants defined by the material properties of the element. By observation, a smaller density ($\rho$) will increase the number of increments required for convergence. Since all assemblies modeled in this study were tested at quasi-static loading levels, the inertial effects of the components will be largely negligible prior to fracture. Therefore, a mass scaling strategy may be used to decrease computational cost.

A density sensitivity analysis performed in Wurzelbacher (2012) concluded that for quasi-static load cases, a density of 0.1 kip/in$^3$, which is roughly 380 times the true density of steel ($\sim$0.000284 kip/in$^3$), was the highest density that would preserve the integrity of the model when compared to a true density analysis. For all models, a density of 0.1 kip/in$^3$ (SI $\approx$ 0.3 kg/mm$^3$) is used to improve computational efficiency.

To characterize the isotropic hardening properties of the various metals used, the engineering stress vs. engineering strain plots provided in Miazga and Kennedy (1986) and Gomez et al. (2008) from the representative ancillary tension tests were digitized into numerical data. Abaqus requires the isotropic hardening data to be entered in terms of “true” stress and logarithmic plastic strain. The equations that were employed to perform this conversions are shown in Appendix D, alongside graphical comparisons of the digitized data and true stress-strain data.

4.2 Choice of Modeling Methodology

The creation of a representative finite element model for fillet welds is challenging for a number of reasons. Fillet welds have an irregular profile that is variable along the length, typically deviating from the idealized isosceles triangle used when specifying fillet weld dimensions. Previous studies have formulated detailed FE models of fillet welds in both two and three
dimensions. Gomez et al. (2008) presented a two-dimensional FE simulation of the T-stub tension assembly experiment in their study, shown in Figure 4-1. In this model, the inner portion of the weld nugget was modeled with elements extending radially out from the root notch tip. Additionally the mean weld profile was modeled as measured from the tested assemblies.

![2D FE Mesh of T-stub Assembly](image)

**Figure 4-1: 2-D FE Mesh of T-stub Assembly (Gomez et al. 2008)**

Kanvinde et al. (2013) extended the FE modeling of Gomez’s 2008 tests into a three dimensional model. Similarly, the measured mean weld profile was used for the entire length of the three dimensional weld. This required the use of varied finite element shapes and higher order elements which result in a greater computational cost as compared to the size of the model. Both of these models utilized Abaqus as the finite element software of choice and produced adequate results in through the strain hardening portion of the load history. However, these models were unable to model failure of the specimens.
It was the goal of the author to formulate an easily-deployable method for representing the fracture of fillet welded connections using the Hooputra damage data scaling method introduced in Wurzelbacher (2012). Since the exact profile of a fillet weld is not known prior to deposition, the idealized isosceles triangle was chosen for representation of the fillet in the FE models. To investigate the pre-failure validity of this method, a simple lap splice model was created consisting of two plates (PL 1/2” x 6”, PL 1/2” x 3”) connected via two 3/8” equal leg fillet welds placed longitudinally to the direction of loading as shown in Figure 4-2. A symmetric boundary was used on the bottom face of the assembly to eliminate the effects of eccentricity on the load-displacement results. The welds were meshed using 10-node quadratic tetrahedral (Abaqus Element Definition: C3D10), using surface-to-surface tie constraints to connect the welds to the plates. The weld material definition used E70xx electrode hardening properties via Gomez et al. (2008) as described in Section 4.1. The plates were modeled as elastic perfectly-plastic with a fictitious yield strength of 200 ksi to encourage weld failure. A similar model utilizing a single transversely loaded fillet was also created. As stated previously, all pre-damage tests were conducted using the Abaqus Standard solver.

Figure 4-2: Isometric View of Initial Lap Splice Model
As a means of determining the adequacy of this model to reflect the realistic load-deformation relationship of fillet welds, relationships published in Lesik and Kennedy (1990) were utilized. The research of Miazga and Kennedy (1989) proposed a load-deformation relationship for ultimate strength, based off of Tresca’s (maximum shear stress) failure theory of the form:

\[
\frac{P_\theta}{P_0} = \frac{1 + 0.141 \sin \theta}{\sin(45+\alpha) \sqrt{(\sin \theta \cos \alpha - \sin \theta \sin \alpha)^2 + \cos^2 \theta}}
\]  
(Eqn. 4-2)

In the previous equation, \(P_\theta\) represents the ultimate load of a fillet weld loaded at any angle, normalized by \(P_0\), the ultimate strength a longitudinally loaded fillet weld of the same length and size. The loading angle is represented by \(\theta\), the angle of fracture is given by \(\alpha\), and the variable \((\alpha)\) is given as 0.345, the mean experimentally determined value for welds loaded in tension induced shear (Miazga and Kennedy 1989). A relationship for the angle of fracture was also proposed. Lesik and Kennedy (1990) simplified the preceding equation using an empirical approximation and multiplied in a polynomial function \(f(\rho)\) to describe the entire load deformation relationship for any loading angle, shown below.

\[
P = P_0 (1.00 + 0.50 \sin^{1.5} \theta) f(\rho)
\]  
(Eqn. 4-3)

Using the above equation, the quasi-static load deformation results for both the longitudinal and transverse lap slice models were normalized using the process described in Lesik and Kennedy (1990). The calculations for this normalization are shown in Appendix E. As a means of comparison, Eqn. 4-3 for the longitudinal and transverse loading conditions were plotted alongside of the FE results, as shown in Figure 4-3.
It is observed that the general curve and the relative yield strengths of the models exhibited good agreement with the empirically approximated Eqn. 4-3. The idealized fillet weld shape model was thus used for all models investigated herein. It should be noted that since damage was not implemented in this model, no drop in load occurred in the above FEA results.

4.3 Nodal Connectivity and Contact

During the simulations performed for the initial ductility study, excessive deformation of elements at the root node of the weld/plate interface at high displacements prompted further investigation into the discretization techniques used by Abaqus for surface-to-surface tie constraints. Tie constraints are powerful and useful tools in Abaqus, but must be used wisely due to the elimination of degrees of freedom on the slave nodes. For a fillet weld model constructed in Abaqus CAE (Complete Abaqus Environment), the graphical user interface for creating models
and interpreting results from Abaqus simulations, the use of tie constraints cannot be avoided in order to preserve a sliding contact relationship between the plates.

To investigate the use of tie constraints in the fillet weld models, an input file was manually composed without the assistance of CAE. This method offers increased user control over a CAE formulation, and was used to create an “ideal” situation where nodes at the weld/plate interfaces are shared, as opposed to using coincident nodes occupying the same space and being connected via tie constraints. The lap splice model previously investigated was altered slightly in the input file model for ease of composition. The fully-meshed representative model is shown in Figure 4-4. Two models were created – one in which the three coincident nodes at the weld root were not tied, and another where these nodes were tied using multi-point tie constraints. All other nodes at the weld/plate interfaces were shared between elements. These models were then replicated in Abaqus CAE using tie constraints for comparison. Figure 4-5 shows the load-deformation results of the four models plotted simultaneously. The results from the simulations of the input file models are plotted as lines and the CAE-created model results are plotted as points to illustrate that when coincident nodal surfaces are connected using tie constraints, the simulations bear a nearly identical solution to that of the “ideal” situation where nodes at shared between interface elements. Based on the results of this study, it was decided to use coincident nodal surfaces connected with tie constraints in all following models.

Additionally, both normal and tangential contact controls were enforced in the FE models. For contact normal to the plane of exterior faces, a “hard” contact property was enforced such that elements would not protrude through each other. A static coefficient of sliding friction of 0.3 was enforced for tangential contact, per the research of Ruffley (2011).
Figure 4-4: Isometric View of Fully Meshed Input Model

Figure 4-5: Force-Displacement Results of Nodal Connectivity Study
4.4 Element Selection and Convergence Study

Due to the findings in the nodal connectivity study, it was clear that the use of coincident nodes connected at the plate-weld interfaces would be necessary to provide the most consistent and accurate results. This is difficult to achieve with the geometry of tetrahedral elements that the initial models utilized for the weld mesh. In this case, use of quadratic tetrahedral elements (C3D10) for the entire model would be necessary to line up the nodes properly. These elements have two severe disadvantages in terms of computational efficiency – not only does quadratic interpolation greatly increase computational cost, but reduced integration is not also not compatible with tetrahedral elements. The use of linearly interpolated tetrahedral elements, which would offer a lower computational cost than their quadratically interpolated counterparts, is strongly discouraged by the Abaqus 6.12 User’s Manual due to their inherent over-stiffness and inability to accurately represent complex stress distributions. Linearly interpolated hexahedral elements with reduced integration (C3D8Rs) are suggested by the User’s Manual for accurate stress field representation in high strain situations, and would eliminate the computational disadvantages of C3D10 elements.

To investigate differing element types, a convergence study was performed using four different mesh densities (not to be confused with material densities) for five different element types: linear brick (hexahedral) with and without reduced integration, quadratic brick with reduced integration, and both linear and quadratic tetrahedral elements. The results of this study, performed using Abaqus Standard solver, showed the linear brick elements to perform best in terms of convergence. However, since Abaqus Explicit was used as the primary solver in this research, it was decided to study further the convergence between linear brick elements with and without reduced integration using the Explicit solver.
A significant discovery of the Abaqus Standard convergence study was that an initially ill-shaped brick element with reduced integration, such as in the case of C3D8R elements, was prone to falling victim to a particular zero-energy mode referred to as “hour-glassing”. This failure mode occurs in high strain situations when a reduced integration method is used. In C3D8R elements, a single Gauss point is used for integration, resulting in a constant strain scenario. In a high strain situations, the Gauss point may calculate a strain value equal to zero, in which case the element will provide zero resistance and essentially collapses on itself (Cook 1995). Reduced integration elements in Abaqus are equipped with “hourglass controls” which uses an artificial stiffness increase to combat the zero energy mode (Abaqus 2012). Despite these measures, element failure was experienced in certain situations in the convergence study when C3D8Rs were used. This was due to the automatic mesh generator’s inability to mesh the geometry of the isosceles triangle shape using rectangular prisms. The representative weld model that experienced the zero-energy mode is shown in Figure 4-6 with the particular initially deformed elements marked.

![Element Prone to Hourglassing](image-url)

**Figure 4-6: Weld Mesh Cross Section with Initially Deformed Element Marked**
To reduce the risk of this potential failure mode, coupled with the fact that hexahedral elements are inherently sensitive to initial element shape (Abaqus 2012), a number of partitioning schemes were investigated along with three different mesh densities for both fully-integrated and reduced integration brick elements. The partitioning schemes (referred to as “1x”, “2x” and “3x”) were simulated alongside of a control, (referred to as “Null”) in which CAE’s automatic mesh generator is used solely, to demonstrate the effect of improved initial element shape. Figure 4-7 illustrates the partitioning schemes investigated and the corresponding mesh patterns.

The welds used were 1/2” equal-leg fillet welds of 1 inch length. The seed sizes investigated were 1/8”, 1/16” and 1/32”, chosen as functions of the weld size. Kinematic coupling restraints were utilized on each weld leg to fully restrain the horizontal leg and apply a displacement to the vertical leg. This set-up represents a longitudinal loading condition without the connected plates being explicitly modeled. It was observed during the modeling process that over-partitioning on the 1/8” and 1/16” weld sizes resulted in elements with a larger dimension along the length than the width and height. It was predicted that this non-symmetry would lead to issues with convergence - a theory supported by the test results.
After initial runs of the convergence study, it was observed that the use of fully integrated brick elements exhibited a substantial increase in computation cost of the use of reduced integration, and further provided fairly similar results. Due to the inherent over-stiffness of a finite element solution, the accuracy of an FE solution actually decreases as the level of accuracy in integrating the stiffness matrix increases. Hence, reduced integration also offers the additional benefit of combatting this over-stiffness (Cook 1995). Additionally, the Abaqus User’s Manual suggests the use of C3D8R elements for problems involving high strain. For these reasons, C3D8R element were selected for use in the extents of this study.

Results of the study concluded that the “2x” partitioning scheme exhibited the most satisfactory overall performance. Further results can be found in Appendix F. As mentioned previously, the “3x” partitioning scheme disrupted the aspect ratio of the elements when used in conjunction with 1/8” and 1/16” seed sizes. Figure 4-8 provides the load-deformation results of the convergence study for the three seed sizes using the “2x” partitioning scheme. The figure shows values for deformations up to 3/10”, which is roughly twice the deformation at fracture for a 1/2” fillet weld as predicted by equations in Lesik and Kennedy (1990). It was observed that there is very little difference in the load-deformation response between the three seed sizes. In order to have a satisfactory number of elements in the weld segment for the implementation of damage, but to avoid a large computation cost, the 1/16” seed size was selected for a 1/2” equal leg fillet weld. Therefore, the mesh densities of weld segments were established with a primary seed size of 1/8 of the weld leg size.

A convergence study was not conducted for the plate sections. Since it was determined that all nodes at the weld/plate interface must be coincident for an accurate analysis, the mesh density of the plates will be governed by the weld seeding. A coarser mesh was used for areas a substantial
distance away from the connection, with the size limited by retaining an element aspect ratio less than 5 in the coarse areas. This aspect ratio limit was conservative when compared to the suggested maximum element aspect ratio of 10 (Abaqus 2012). In all circumstances, the aspect ratio limit yielded more elements along the width of the plates than the recommended minimum of two.

4.5 Summary of Methodology

The following list summarizes the guidelines established in this chapter governing the modeling methodology used for generating the finite element models in this study.

1) Abaqus Explicit is used as the solver of choice

2) All models utilize representative plasticity data for the particular material being modeled and a density 380 times the true density of steel
3) Fillet welds are modeled in the idealized isosceles triangle shape, utilizing coincident nodes at weld/plate interfaces and surface-to-surface tie constraints to connect the parts.

4) Hexahedral brick elements with reduced integration and hourglass control (C3D8R) are used for all models.

5) Partitioning scheme “2x”, as defined previously, is utilized for weld sections.

6) Weld element seed sizes are chosen as 1/8 the weld leg size.

These rules were employed throughout the FE modeling process. Additional details regarding the modeling of the tested assemblies are presented in the next chapter.
Chapter 5: Finite Element Models

This chapter describes the details of the modeling process used for creating the finite element models that were used for establishing and validating the damage data used to predict fracture in fillet welds segments. These models include four assemblies featuring 5 mm fillet welds and three assemblies featuring 9 mm fillet welds from the work of Miazga and Kennedy (1989) and one variation of the T-stub tension test via Gomez et al. (2008).

5.1 Lap Splice Assemblies

The first assemblies modeled for damage data calibration were the aforementioned 5-mm fillet weld assemblies from Miazga and Kennedy. In lieu of modeling all 7 assemblies tested in the study, loading orientations of equal intervals that featured an equal number of continuous and discontinuous weld beads were chosen: 0, 30, 60 and 90 degrees. The schematics in the Structural Engineering Report (Miazga and Kennedy 1986) supplemental to their 1989 researched served as the basis for the FE modeling. These schematics are provided in Appendix G.

Two primary considerations were made to preserve computational efficiency. The first was to reduce the overall length of the test assembly. Modeling the full assembly as it is shown in Figure 3-3 would increase computational time dramatically without providing any additional accuracy. The LVDTs used in the experimental testing program to measure displacements were mounted in such a way that only the deformations of the welds were measured. Therefore, only the distance needed to equalize the stress distribution in the plates was necessary on either side of the welded joint. The other size-reducing modeling consideration made was the use of a symmetric boundary condition at mid-thickness of the test assembly. With this boundary condition in place, only the assembly about the mid-thickness needed to be modeled, with Z-direction displacements and X/Y rotations constrained to zero, as shown in Figure 5-1. The same boundary condition was
also applied to the bottom of the connecting plate to simulate the effect of spacers that were used between the two smaller thickness plates in experimental testing (Miazga and Kennedy 1986).

The plate thicknesses used in all of 5-mm fillet weld assemblies were 18 mm for the primary plate and 9 mm for the connected plates. Note that the models utilized a thickness of 9 mm for the primary plate due to the symmetric boundary condition. The plate strain hardening properties of the CSA 300W steel were characterized from the results of the ancillary tension coupon tests performed experimentally. The results of three tension coupons cut from three plate sizes as represented graphically in Miazga and Kennedy (1986) were digitized and converted to true stress and logarithmic plastic strain as stated in Section 4.1. Table 5-1 shows the average stress-strain profile for 300W steel that was used in the FE models.

The strain hardening properties of the weld segments were defined similarly. A representative stress-strain curve was provided via the results of the ancillary tests performed by Miazga and Kennedy. The digitized engineering stress/strain values along with the appropriately converted “true” values are show in Table 5-2. Further details of the material plastic data are shown in Appendix D.
Table 5-1: Representative Plasticity Data for 300W Steel

<table>
<thead>
<tr>
<th>Strain Hardening</th>
<th>Logarithmic Plastic Strain (mm/mm)</th>
<th>True Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_y$</td>
<td>0.000</td>
<td>355</td>
</tr>
<tr>
<td></td>
<td>0.010</td>
<td>360</td>
</tr>
<tr>
<td></td>
<td>0.025</td>
<td>414</td>
</tr>
<tr>
<td></td>
<td>0.050</td>
<td>475</td>
</tr>
<tr>
<td></td>
<td>0.075</td>
<td>524</td>
</tr>
<tr>
<td></td>
<td>0.100</td>
<td>560</td>
</tr>
<tr>
<td></td>
<td>0.125</td>
<td>585</td>
</tr>
<tr>
<td></td>
<td>0.150</td>
<td>597</td>
</tr>
<tr>
<td>$f_u$</td>
<td>0.175</td>
<td>598</td>
</tr>
</tbody>
</table>

Table 5-2: Representative Plasticity Data for E48014 Weld Metal

<table>
<thead>
<tr>
<th>Strain Hardening</th>
<th>Engineering Strain (mm/mm)</th>
<th>Engineering Stress (MPa)</th>
<th>Logarithmic Plastic Strain (mm/mm)</th>
<th>True Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_y$</td>
<td>0.002</td>
<td>460</td>
<td>0.000</td>
<td>461</td>
</tr>
<tr>
<td></td>
<td>0.035</td>
<td>460</td>
<td>0.032</td>
<td>476</td>
</tr>
<tr>
<td></td>
<td>0.056</td>
<td>490</td>
<td>0.052</td>
<td>517</td>
</tr>
<tr>
<td></td>
<td>0.080</td>
<td>512</td>
<td>0.074</td>
<td>552</td>
</tr>
<tr>
<td></td>
<td>0.120</td>
<td>528</td>
<td>0.110</td>
<td>591</td>
</tr>
<tr>
<td></td>
<td>0.160</td>
<td>528</td>
<td>0.145</td>
<td>613</td>
</tr>
<tr>
<td>$f_u$</td>
<td>0.200</td>
<td>519</td>
<td>0.179</td>
<td>623</td>
</tr>
</tbody>
</table>

It should be noted that the yield strength of the E48014 weld metal via the experimental testing is less than 480 MPa. The minimum requirements at the time of testing (1986) via CSA W48.1-M1980 specified the minimum yield stress of E48014 as 410 MPa, with a minimum ultimate stress of 500 MPa. In this regard, the tests met the minimum requirements, however this designation differs from the American Welding Society (AWS) which designates filler materials by their minimum required tensile strength.
The 5-mm fillet welds were modeled using the “2x” partitioning scheme, as described in Chapter 4. The seed size was chosen as 0.625 mm, one-eighth of the fillet weld leg size, determined to be sufficient in the convergence study discussed in Chapter 4. This mesh density controlled for the plates as well, to ensure proper nodal connectivity at the weld/plate interfaces. The coarser mesh density used for the plates a sufficient distance away from the connection was chosen to avoid exceeding a global aspect ratio of 5, retaining well-shaped elements for the entire model.

The modeling of the 90 degree assembly was straightforward, featuring a width of 100 mm for both plates and a 25 mm overlap. The fillet extended across the entire width of the plates. The weld terminations of tested specimens were milled parallel to the direction of loading after the removal of the run-off tabs (Miazga and Kennedy 1986), and were modeled as such. Representative views of the 90 degree FE model are shown in Figure 5-2.

![Figure 5-2: Representative Views of 90 Degree-5 mm Model](image)

The 60 and 30 degree models required alterations to the geometry and meshing of the plates both to maintain well-shaped elements and to achieve the desired connectivity. The 60 degree specimen featured a continuous weld bead placed at 60 degree to the loading axis, and the plates
were milled accordingly. The plate overlap was also 25 mm to the axis of the weld. The FE model in show in Figure 5-3.

![Representative Views of 60 Degree-5 mm Model](image)

**Figure 5-3: Representative Views of 60 Degree-5 mm Model**

The 30 degree specimen required the use of two discontinuous weld beads to avoid plate yielding to control over weld rupture (Miazga and Kennedy 1986). To accommodate this weld design, the plate was milled at the weld angle with a 25 mm space perpendicular to the axis of loading between the weld terminations. Again, a 25 mm lap was utilized at the end of the connecting plate cut. Representative views of the FE model are shown in Fig. 5-4. It is worth noting that an element face angle of 30 degrees, exhibited by the elements at the connection, is below the typical 45 degree limit for minimum face angle of well-shaped elements.

![Representative Views of 30 Degree-5 mm Model](image)

**Figure 5-4: Representative Views of 30 Degree-5 mm Model**
To properly accommodate two longitudinally loaded fillet welds, the 0 degree model utilized a larger primary plate width of 130 mm. The typical connector plate width of 100 mm was retained and a plate overlap of 80 mm was used to provide a total weld length of 160 mm. Views of the 0 degree model are shown in Figure 5-5.

![Figure 5-5: Representative Views of 0 Degree-5 mm Model](image)

The three lap splice assemblies chosen for modeling that featured 9 mm fillet welds were those with weld angles of 90, 45 and 0 degrees. Of these, the 90° and 0° assemblies were identical to the corresponding 5 mm weld assemblies aside from larger weld sizes and plate thicknesses. A larger weld size corresponded to the larger controlling seed size: 1.125 mm, again one-eighth of the fillet weld leg size. Plate thicknesses of 35 mm for the primary plate and 18 mm for the connected plate were used in the experimental testing and the FE models again utilized a half-thickness primary plate model to employ the symmetry boundary condition appropriately. The plate widths, plate overlaps and weld lengths remained consistent with the 5 mm lap splice assemblies, commensurate with the experimental testing. Small changes were made in plate partitioning to ensure node-to-node contact as the weld/plate interfaces and the course meshing was selected to maintain a well-shaped elements throughout the models.
The set-up of the 45 degree-9 mm specimen is similar to the previously introduced 60 degree assembly. It features a continuous weld bead at an angle of 45 degree to the loading direction with both plates cut accordingly. Consistent with the other assemblies, a 25 mm lap in the direction of the weld axis was utilized. Again, it’s noted that the desired nodal connectivity between the parts was achieved, but the 45 degree element face angle is the upper-limit for well-shaped hexahedral elements. This appeared to have an effect on the angle of fracture through the fillet weld, but the load displacement results did not exhibit such ill effects. Representative views are shown in Figure 5-6.

![Representative Views of 45 degree - 9 mm Assembly](image)

**Figure 5-6: Representative Views of 45 degree - 9 mm Assembly**

### 5.2 T-Stub Assemblies

After the damage data calibration process was completed, a T-stub assembly was modeled to investigate the validity of using the modeling methodology with a connection other than a lap splice. As described in Chapter 3, Gomez et al. (2008) utilized 8 combinations of fillet weld size, weld electrode type and root notch length. For the purpose of this thesis, one combination was chosen for investigation. The model utilized standard E70T-7 (non-toughness rated) material characterization and a connecting plate thickness of 1.25”, representing the root notch length, was chosen since the effect of the root notch length was shown to be negligible in Gomez’s research. A fillet weld leg size of 5/16” was used in this model.
The experimental assembly, shown in Figure 3-7, was reduced in the FE model to only model the middle plate, the test welds, and a length of the connected plate satisfactory for equalization of stresses. This was done for computational efficiency. A symmetric boundary constraint through the thickness of the connected plate was considered to further reduce the size of the model, but was decided against to ensure accurate results.

The material properties for both the plates and welds were defined similar to the method used for the lap splice models. The authors of the study conducted ancillary tension coupon tests to characterize the stress-strain response of the E70T-7 weld material and A572-Gr. 50 steel, which was used for all plates in the testing program of Gomez. Representative engineering stress vs. engineering strain curves were selected from the research, digitized and converted to true stress/logarithmic plastic strain values per the procedure outlined in Chapter 4. The strain hardening data used in the FE models for the plates and weld segments are show in Table 5-3 and 5-4, respectively. Additional details are provided in Appendix D.

Table 5-3: Representative Plasticity Data for A572-Gr. 50 Steel

<table>
<thead>
<tr>
<th>Strain Hardening</th>
<th>Engineering Strain (in/in)</th>
<th>Engineering Stress (ksi)</th>
<th>Logarithmic Plastic Strain (in/in)</th>
<th>True Stress (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>fy</td>
<td>0.002</td>
<td>52.6</td>
<td>0.000</td>
<td>52.8</td>
</tr>
<tr>
<td></td>
<td>0.019</td>
<td>53.1</td>
<td>0.017</td>
<td>54.1</td>
</tr>
<tr>
<td></td>
<td>0.033</td>
<td>59.6</td>
<td>0.031</td>
<td>61.6</td>
</tr>
<tr>
<td></td>
<td>0.068</td>
<td>66.6</td>
<td>0.063</td>
<td>71.1</td>
</tr>
<tr>
<td></td>
<td>0.087</td>
<td>68.9</td>
<td>0.081</td>
<td>74.9</td>
</tr>
<tr>
<td></td>
<td>0.109</td>
<td>70.1</td>
<td>0.101</td>
<td>77.7</td>
</tr>
<tr>
<td></td>
<td>0.133</td>
<td>70.8</td>
<td>0.122</td>
<td>80.2</td>
</tr>
<tr>
<td></td>
<td>0.188</td>
<td>71.6</td>
<td>0.169</td>
<td>85.0</td>
</tr>
<tr>
<td>fu</td>
<td>0.220</td>
<td>71.4</td>
<td>0.196</td>
<td>87.1</td>
</tr>
</tbody>
</table>
Table 5-4: Representative Plasticity Data for E70T-7 Weld Metal

<table>
<thead>
<tr>
<th>Strain Hardening</th>
<th>Engineering Strain (in/in)</th>
<th>Engineering Stress (ksi)</th>
<th>Logarithmic Plastic Strain (in/in)</th>
<th>True Stress (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_y$</td>
<td>0.002</td>
<td>75.0</td>
<td>0.000</td>
<td>75.1</td>
</tr>
<tr>
<td></td>
<td>0.003</td>
<td>75.7</td>
<td>0.001</td>
<td>76.0</td>
</tr>
<tr>
<td></td>
<td>0.008</td>
<td>80.2</td>
<td>0.005</td>
<td>80.8</td>
</tr>
<tr>
<td></td>
<td>0.013</td>
<td>85.6</td>
<td>0.010</td>
<td>86.7</td>
</tr>
<tr>
<td></td>
<td>0.024</td>
<td>91.5</td>
<td>0.020</td>
<td>93.7</td>
</tr>
<tr>
<td></td>
<td>0.038</td>
<td>95.3</td>
<td>0.034</td>
<td>98.9</td>
</tr>
<tr>
<td></td>
<td>0.053</td>
<td>96.9</td>
<td>0.048</td>
<td>102.1</td>
</tr>
<tr>
<td></td>
<td>0.088</td>
<td>97.6</td>
<td>0.081</td>
<td>106.3</td>
</tr>
<tr>
<td></td>
<td>0.105</td>
<td>96.9</td>
<td>0.096</td>
<td>107.1</td>
</tr>
<tr>
<td>$f_u$</td>
<td>0.121</td>
<td>96.2</td>
<td>0.111</td>
<td>107.9</td>
</tr>
</tbody>
</table>

The width and thickness of the middle was not explicitly stated in Gomez’s research. To ensure sufficient size, a large preliminary model was simulated. The dimensions of the middle plate were then reduced to a 1” thickness and width of 4.5”, determined to provide sufficient stress equalization. The length of the connecting plate was reduced using the same method.

The welds were modeled according to the experimental assembly with a length of 12” on each side of the connecting plate. The 13” length shown in Figure 3-7 included 1/2” run-off tabs on each side of the plates that were removed prior to testing (Gomez et al. 2008). The partitioning strategy and seed sizes correspond to the methods used throughout this study. Controlling seed sizes of 5/128” was used for the 5/16” fillet welds. The critical areas of the plates were modeled with this seed size, and the coarser region sizes were chosen to not violate an element aspect ratio of 5. An isometric view of the T-stub model is shown in Figure 5-7.
Figure 5-7: Isometric View of T-stub Model
Chapter 6: Analysis of Results

The output data from analyses conducted in this study are presented and analyzed in this chapter. Presented first are the equations via Lesik and Kennedy (1990) that were used as controlling variables to calibrate the damage data scaling factors in conjunction with selected results from Miazga and Kennedy. The first results analyzed were the 5-mm fillet weld assemblies, upon which the damage data was calibrated. A relationship between the damage scaling factor and the weld loading angle was developed upon these results and examined for validity using the larger 9 mm fillet weld assemblies. The T-stub model was then used to investigate validity of this scale factor relationship with a T-stub style tension connection.

6.1 Displacements at Fracture and Ultimate Load

The work of Lesik and Kennedy in their 1990 publication “Ultimate Strength of Fillet Welded Connections Loaded in Plane” served as an extension to the experimental testing program of Miazga and Kennedy. Provided was additional analysis of the lap splice connection test results and a code-adaptable version of the load-deformation relationship for fillet welds loaded at varying angles, as introduced in Section 4.2, Eqn. 4-3. Lesik and Kennedy would go on to suggest this relationship was viable for use in analyzing eccentrically loaded fillet welded connections in conjunction with the instantaneous center of rotation method (Lesik and Kennedy 1990). This method was accepted and is featured in the AISC Specification for Structural Steel Buildings.

The most important relationship developed in this publication, as it relates to this thesis, is the empirically derived equation used to predict deformations at the point of failure. This equation is presented as Eqn. 6-1.

$$\frac{\Delta f}{w} = 1.087 (\theta + 6)^{-0.65}$$  \hspace{1cm} (Eqn. 6-1)
In the above equation, \( w \) represents the fillet weld leg sizes and \( \Theta \) represents the weld loading angle. For Miazga and Kennedy’s fillet weld tests, this equation provided a mean test-to-predicted ratio of 1.000, with a coefficient of variation of 0.194. In this study, the value of Eqn. 6-1 for a particular assembly was used as the primary variable in determining the damage data scale factor.

Also utilized for comparison was Lesik and Kennedy’s equation describing the ultimate load of a fillet weld group as a function of the loading angle. This equation is an empirical approximation of Eqn. 4-2, as proposed by Miazga and Kennedy, and it presented below as Eqn. 6-2.

\[
P = P_0 (1.00 + 0.50 \sin^{1.5} \theta)
\]  \hspace{1cm} (Eqn. 6-2)

In the above equation, \( P_0 \) is taken as 290.7 N/mm/mm, which represents the average experimental ultimate load for the longitudinal weld assemblies, given as Newton per millimeter of weld segment length per millimeter of the fillet weld leg size. This equation provided a mean test-to-predicted ratio of 1.010 with a standard deviation and coefficient of variation of 0.090 and 0.089, respectively. This equation was used only as a means of comparison, along with the ultimate strengths reported from the experimental testing, but was not explicitly considered in the establishment of the damage data scaling factors.

### 6.2 Collection of Results (Lap Splice Assemblies)

All models were tested in uniaxial tension in the long direction of the assembly, as per the experimental procedure. The measurement of deformation of the fillet welds in the FE models exhibited significant dependence on the points from where the displacement was measured for particular models. The location of the LVDT’s along the width of the test assemblies was not explicitly stated in Miazga and Kennedy’s research. The research did indicate however, that
displacements were measured in the direction of loading, with the LVDT frames set directly adjacent to the weld segments so that no plate deformations were recorded. Referring to Figures 3-5 and 3-6, the locations of the LVDTs in relation to the frames were estimated, and the locations in relation to the tests assembly were approximated. Table 6-1 and corresponding Figure 6-1 provides the distance from the edge of the test assemblies that the displacements were measured from, in the FE modes. For all cases, the weld deformations were measured from each side of the plate width, as indicated in Figure 6-1 and the results were averaged.

Table 6-1: Locations of Displacement Measurement from Edge of Test Assembly

<table>
<thead>
<tr>
<th>Assembly</th>
<th>x (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 Deg</td>
<td>20</td>
</tr>
<tr>
<td>30 Deg</td>
<td>12.5</td>
</tr>
<tr>
<td>45 Deg</td>
<td>30</td>
</tr>
<tr>
<td>60 Deg</td>
<td>22.5</td>
</tr>
<tr>
<td>90 Deg</td>
<td>15</td>
</tr>
</tbody>
</table>

Figure 6-1: General Illustration Corresponding to Table 6-1

The applied load in each case was simply derived from the total reaction force experienced at the restrained end of the test assembly. Note that the reaction force was multiplied by two for proper comparison to the experimental results, due to the use of half-symmetry FE models. It was not necessary to apply this amplification to the weld displacements.
6.3 Results of 5 mm Weld Assemblies

For all models, Hooputra’s unmodified EN AW-7108 T6 aluminum ductile and shear damage initiation criteria was applied to the weld segments prior to damage data scaling. No damage properties were applied to the plates, considering all the models tested by Miazga and Kennedy exhibited failure only in the welds (Miazga and Kennedy 1986).

The first 5-mm fillet weld assembly simulation run was of the 90 degree (transverse) case. Element deletion in this case commenced at the weld root and propagated almost immediately. A numerical method for estimating a proper damage scaling factor was developed by querying the first element deleted using unmodified aluminum damage data and manually calculating the damage variable (Eqn. 2-1) in Excel, using hypothetical damage scaling factors. In all cases, the shear criterion controlled element deletion, but the failure strain was scaled for both criteria equivalently. Table 6-2 presents details of the 90° assembly including the ultimate load and corresponding deformation from the experimental results, the total weld segment length and the values of deformation at fracture and ultimate load via Eqn. 6-1 and Eqn. 6-2, respectively.

Table 6-2: 90 Deg-5 mm Experimental/Predicted Ultimate Loads and Deformations

<table>
<thead>
<tr>
<th>Test</th>
<th>Experimental $P_u$ (kN)</th>
<th>Experimental $\Delta_u$ (mm)</th>
<th>Total Weld Length (mm)</th>
<th>$\Delta_f$ Via Eqn. 6-1 (mm)</th>
<th>$P_u$ Via Eqn. 6-2 (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>90.1</td>
<td>421</td>
<td>0.312</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>90.2</td>
<td>431</td>
<td>0.203</td>
<td></td>
<td>0.280</td>
<td>436</td>
</tr>
<tr>
<td>90.3</td>
<td>407</td>
<td>0.353</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 6-2 shows the results of applying scaled variations of Hooputra’s damage data, along with the experimental and predicted loads and deformations tabulated above. For all results shown in this chapter, “SF” refers to the scale factor multiplied by Hooputra’s aluminum damage data, i.e. “SF =1” indicates unmodified aluminum damage data. It can be observed that when
Hooputra’s data is multiplied by a scale factor of 0.4, the deformation at fracture occurs at the predicted value of Eqn. 6-1 and exhibits good agreement with the experimental results. The FE model under-predicted the ultimate strength given by Eqn. 6-2 by 13.17%, in which Eqn. 6-2 agreed well with the experimental results. While this difference is significant, the model is limited by the material characteristics provided via Miazga and Kennedy’s representative stress-strain curve for E48014 electrode material. For the purposes of this study, a realistic prediction of deformation at fracture is preferred over a perfect prediction of ultimate strength.

![Graph](image)

**Figure 6-2: Results of 90 Deg - 5 mm Weld Assembly Model**

The next assembly modeled was that of the longitudinal (Θ = 0°) case, assuming that if a scale factor relationship did indeed exist, the longitudinal and transverse cases would serve as the bounds of the relationship. It was discovered that the use of unmodified aluminum damage data predicted weld fracture consistent with the calculated value of Eqn. 6-1, which also agreed with
the experimental fracture deformation. Experimental and predicted values are shown in Table 6-3 and model results are shown in Figure 6-3. The FE model exhibited an ultimate strength 9.65% greater than what was predicted in Eqn. 6-2, which also slightly under-predicted the average ultimate strength of the experimental tests.

Table 6-3: 0 Deg-5 mm Experimental/Predicted Ultimate Loads and Deformations

<table>
<thead>
<tr>
<th>Test</th>
<th>Experimental $P_u$ (kN)</th>
<th>Experimental $\Delta_u$ (mm)</th>
<th>Total Weld Length (mm)</th>
<th>$\Delta_f$ Via Eqn. 6-1 (mm)</th>
<th>$P_u$ Via Eqn. 6-2 (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>513</td>
<td>0.937</td>
<td>320</td>
<td>1.696</td>
<td>465</td>
</tr>
<tr>
<td>0.2</td>
<td>487</td>
<td>1.018</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.3</td>
<td>483</td>
<td>0.982</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 6-3: Results of 0 Deg - 5 mm Weld Assembly Model

In this model, an important discovery regarding the nature of the FE model was made that altered the scale factor determination for the upcoming $60^\circ$ and $30^\circ$ models. Previous research by
Wurzelbacher indicated that first element deletion typically indicated fracture. In the longitudinal model, however, first element deletion did not correspond to an immediate load drop. This disparity could be explained in a number of ways. Firstly, the number of field output requests from the analysis has an effect on the interpretation of results. Recall that Abaqus Explicit uses the Central Difference Method for computing results, which is notorious for the large number of increments required for convergence. An Abaqus output file could produce results, for example, in 300 frames when the actual number of increments performed in the analysis is greater than one million. Thus, it could be perceived that initial element deletion coincided with an immediate load drop when this did not truly occur in the analysis. However, since the number of field output requests is typically definite for equal time steps, this inaccurate perception was not suspected.

Rather, it appeared the tie constraints used at weld/plate interfaces in the lap splice models was contributing to reduction of the early onset weld strain away from the root of the fillet weld. The nodes at the root of the fillet weld are tied to nodes on both the primary and connecting plate, with the root node serving as the “master” node in both situations. Thus, the root node and their corresponding root elements were over-constrained, resulting in zero differential motion between the plates along this row of nodes. The load drop for the longitudinal model occurred when an element away from the weld leg reached its failure strain and consequently was deleted from the analysis. This situation is illustrated in Figure 6-4, indicating the prior location of the deleted elements which, upon deletion, was closely followed by full fracture of the weld. This same sequence of events was exhibited by the 60° and 30° assemblies. It is believed that this failure sequence was not noted in the transverse model due to the extremely quick rate of failure when compared to the time step of the field output requests.
The next 5-mm model to be investigated was the 60° weld angle assembly. The computational method for approximating the scale factor was modified to consider the most critical element of the unmodified model located away from the leg of the fillet weld, instead of the first element deleted. Figure 6-5 provides results of the FE simulation and the experimental/predicted results for ultimate load and displacement are shown in Table 6-4. It’s observed that a scale factor of 0.6 applied to Hooputra’s damage data closely matches the fracture deformation given by Eqn. 6-1.

The FE model under-predicted the ultimate load given by Eqn. 6-2 by a similar error percentage to the transverse model of 12.39%. In this case, the ultimate load of the experimental assemblies significantly exceeded the prediction of Eqn. 6-2. Again, the model is limited by the representative stress-strain curve provided in Miazga and Kennedy’s research and accurate prediction of deformation at fracture was the primary goal of this study.
Table 6-4: 60 Deg-5 mm Experimental/Predicted Ultimate Loads and Deformations

<table>
<thead>
<tr>
<th>Test</th>
<th>Experimental $P_u$ (kN)</th>
<th>Experimental $\Delta_u$ (mm)</th>
<th>Total Weld Length (mm)</th>
<th>$\Delta_f$ Via Eqn. 6-1 (mm)</th>
<th>$P_u$ Via Eqn. 6-2 (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>60.1</td>
<td>568</td>
<td>0.333</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>60.2</td>
<td>566</td>
<td>0.321</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>60.3</td>
<td>559</td>
<td>0.342</td>
<td>230</td>
<td>0.357</td>
<td>469</td>
</tr>
</tbody>
</table>

Figure 6-5: Results of 60 Deg - 5 mm Weld Assembly Model

The final assembly investigated in the 5-mm fillet weld test set was the 30° model. Similarly, Table 6-5 provides the experimental/predict ultimate strengths and deformations and Figure 6-6 illustrates the results. A scale factor of 0.8 applied to Hooputra’s damage data provides a fracture deformation akin the value provided by Eqn. 6-1. Again, the predicted ultimate load via Eqn. 6-2 under-predicts the experimental values, but the FE model is only 6.04% lower than the Eqn. 6-2 value.
Table 6-5: 30 Deg-5 mm Experimental/Predicted Ultimate Loads and Deformations

<table>
<thead>
<tr>
<th>Test</th>
<th>Experimental $P_u$ (kN)</th>
<th>Experimental $\Delta_u$ (mm)</th>
<th>Total Weld Length (mm)</th>
<th>$\Delta_f$ Via Eqn. 6-1 (mm)</th>
<th>$P_u$ Via Eqn. 6-2 (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>30.1</td>
<td>614</td>
<td>0.429</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30.2</td>
<td>626</td>
<td>0.382</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30.3</td>
<td>610</td>
<td>0.405</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 6-6: Results of 30 Deg - 5 mm Weld Assembly Model

Figure 6-7 provides a comparison to the idealized load-deformation response of fillet weld groups loaded as varying angles, as represented by Eqn. 4-3. The identical normalization process as used in Section 4.2 was applied to the analytical results of the 5-mm fillet weld assemblies. The curves representing Eqn. 4-3 for each weld orientation terminate at the proper normalized deformation as given by Eqn. 6-1 for each orientation. As can be observed, the analytical curves terminate at similar normalized deformations.
6.4 Establishment of Damage Scaling Factor Relationship

Having established damage scaling factors for fillet welds loaded at 0, 30, 60 and 90 degrees via the 5 mm fillet weld assemblies, a linear aluminum damage scaling factor relationship for predicting fracture in equal leg fillet welds loaded at any angle is proposed in Eqn. 6-3, with tabulated results and a representative plot shown in Table 6-6 and Figure 6-8:

$$\text{Scale Factor} = 1 - \frac{1}{150} \theta \quad \text{(Eqn. 6-3)}$$

<table>
<thead>
<tr>
<th>Loading Angle (deg)</th>
<th>Scale Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>30</td>
<td>0.8</td>
</tr>
<tr>
<td>60</td>
<td>0.6</td>
</tr>
<tr>
<td>90</td>
<td>0.4</td>
</tr>
</tbody>
</table>
6.5 Results of 9-mm Weld Assemblies

Having established a relationship for scaling Hooputra’s EN AW-7108 T6 aluminum ductile/shear damage data to match the point of fracture in fillet welds loaded at any angle, three 9 mm fillet weld assemblies from Miazga and Kennedy’s research were modeled to examine validity of the method with a larger weld size. The transversely and longitudinally loaded assemblies were modeled along with the 45° assembly which provided an avenue to examine an orientation that was not considered with 5-mm fillet welds. In each case, the simulations were performed using only the damage data scaling factor obtained from Eqn. 6-3.

Table 6-7 provides the experimental/predicted ultimate strengths and deformations, and Figure 6-9 provides the results of the simulation for the transverse 9-mm assembly. The simulation using a damage scale factor of 0.4 exhibits fracture at a deformation reasonably close to the value
predicted in Eqn. 6-1. The ultimate load of the simulation under-predicts the value given by Eqn. 6-2 by nearly the exact percentage of the corresponding 5-mm assembly model, 13.22%.

Table 6-7: 90 Deg-9 mm Experimental/Predicted Ultimate Loads and Deformations

<table>
<thead>
<tr>
<th>Test</th>
<th>Experimental $P_u$ (kN)</th>
<th>Experimental $\Delta_u$ (mm)</th>
<th>Total Weld Length (mm)</th>
<th>$\Delta_f$ Via Eqn. 6-1 (mm)</th>
<th>$P_u$ Via Eqn. 6-2 (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>90.11</td>
<td>789</td>
<td>0.448</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>90.12</td>
<td>807</td>
<td>0.436</td>
<td>200</td>
<td>0.504</td>
<td>785</td>
</tr>
<tr>
<td>90.13</td>
<td>791</td>
<td>0.380</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 6-9: Results of 90 Deg - 9 mm Weld Assembly Model

The next assembly investigated was the longitudinal 9-mm specimen. Experimental and predicted values for comparison are represented in Table 6-8 and Fig. 6-10 illustrates the results. As with the 5-mm assembly, unmodified aluminum damage data was utilized. Fracture in this model was decidedly more ductile than any previous model, and it is believed that the larger
element sizes used in the weld segments had an effect on this behavior. Despite the ductility, a significant load drop commensurate with fracture occurs at a deformation that corresponds to the value given in Eqn. 6-1. The ultimate load of the model exceeds the value given in Eqn. 6-2 by a similar percentage that was observed in the corresponding 5-mm assembly of 7.92%. The deformation at ultimate load occurs later than what was observed in the experimental models, which is also believed to be an effect of a larger element size used in the weld segment.

Table 6-8: 0 Deg-9 mm Experimental/Predicted Ultimate Loads and Deformations

<table>
<thead>
<tr>
<th>Test</th>
<th>Experimental $P_u$ (kN)</th>
<th>Experimental $\Delta_u$ (mm)</th>
<th>Total Weld Length (mm)</th>
<th>$\Delta_f$ Via Eqn. 6-1 (mm)</th>
<th>$P_u$ Via Eqn. 6-2 (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.11</td>
<td>752</td>
<td>1.347</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.12</td>
<td>825</td>
<td>0.983</td>
<td>320</td>
<td>3.053</td>
<td>837</td>
</tr>
<tr>
<td>0.13</td>
<td>787</td>
<td>1.544</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 6-10: Results of 0 Deg - 9 mm Weld Assembly Model
The last of the 9-mm assemblies investigated featured a weld angle of 45°. The corresponding 5-mm assembly was not modeled in the development of the damage data scale factor relationship. Thus, a scale factor of 0.7 was calculated via Eqn. 6-3 and applied to the weld material in the model. Table 6-9 provides experimental and predicted values for comparison and Figure 6-11 illustrates the results of the simulation. It is observed that fracture in the model occurs at a larger deformation than the value given in Eqn. 6-1, a result not experienced in previous models. To examine the validity of Eqn. 6-1 for the 45° weld orientation, the experimentally observed fracture deformations were added to Table 6-9 and the average deformation at fracture is represented in the corresponding figure. It is clear that Eqn. 6-1 significantly under-predicts the deformation at which the researchers observed fracture in the experimental study. In either case, fracture occurs in the finite element model within this range of deformations and is accepted as successful demonstration of the validity of the scale factor relationship for the 45° weld angle.

In terms of ultimate load, the model falls below the value given by Eqn. 6-2 by 15.08%. In this instance, Eqn. 6-2 over-predicts the average ultimate load of the experimental assemblies by roughly the same percentage.

<table>
<thead>
<tr>
<th>Test</th>
<th>Experimental $P_u$ (kN)</th>
<th>Experimental $\Delta_u$ (mm)</th>
<th>Experimental $\Delta_f$ (mm)</th>
<th>Total Weld Length (mm)</th>
<th>$\Delta_f$ Via Eqn. 6-1 (mm)</th>
<th>$P_u$ Via Eqn. 6-2 (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>45.11</td>
<td>842</td>
<td>0.613</td>
<td>1.119</td>
<td>280</td>
<td>0.760</td>
<td>950</td>
</tr>
<tr>
<td>45.12</td>
<td>858</td>
<td>0.506</td>
<td>1.093</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>45.13</td>
<td>861</td>
<td>0.739</td>
<td>1.179</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 6-9: 45 Deg-9 mm Experimental/Predicted Ultimate Loads and Deformations**

Average: 1.130
From the preceding results, the use of the aluminum damage data scaling relationship represented by Eqn. 6-3 was demonstrated as adequate for use in lap splice assemblies featuring fillet welds up to a 9-mm (~ 3/8”) leg size. This was also demonstrated for a loading angle not considered in the formulation of the scale factor relationship. Figure 6-12 illustrates a comparison similar to that shown in Fig. 6-7. The analytical load-deformation data for the 9-mm assembly simulations is normalized by the process described in Appendix E, and compared to Lesik and Kennedy’s idealized load-deformation response represented by Eqn. 4-3. Again, the curves of Eqn. 4-3 are terminated at a normalized deformation corresponding to the value given in Eqn. 6-1. As can be observed, the normalized analytical curves terminate at a similar deformation. Additional plots for loading angles of 0, 30 and 90 degrees comparing analytical to experimental results are shown in Appendix H.
The T-stub assemblies described in “Strength and Ductility of Welded Joints Subjected to Out-of-Plane Bending” by Gomez et al. (2008) were tested in uniaxial tension applied to the connected plate. Figure 3-9 provided a schematic that detailed the assembly used for recording displacements. Figure 6-13 illustrates the points where displacements were recorded from the T-stub model in Abaqus. Reference points (RP-X) 3/4 and 5/6 were used in pairs for determining the deformations in the shear leg of the weld. Each pair is located 3” from the short edge of the primary plate. Reference points 4 and 6 are located 4 element lengths (5/32”) in the negative Z direction from the toe of the weld. Differential displacements between the reference points in each pair were calculated for each frame of the simulation result. This was also done on the other side of the assembly and the differential displacements values were averaged.

Figure 6-12: Comparison of Normalized 9-mm Results to Eqn. 4-3 at Varying Weld Angles
Figure 6-13: View of T-Stub Assembly with Reference Points Shown

Displacement in the model was imparted quasi-statically to Reference Point #2 in the negative Z direction, utilizing a kinematic coupling restraint to apply this displacement uniformly to the surface upon which it lies. As in the lap splice assembly, the applied load was taken as the total reaction force on the restrained (positive Z) surface of the center plate.

6.7 Results for T-Stub Assemblies

Gomez’s T-stubs were examined in this study as a brief investigation into the validity of this modeling methodology for a loading condition other than a lap splice. The weld leg size modeled was 5/16” which lies within the range of weld sizes (5-9 mm) previously validated for this method. The loading angle in this connection can be described as 90° since displacement is
being applied perpendicular to the longitudinal weld axis. For that reason, Hooputra’s damage data was scaled by 0.4, per the relationship given in Eqn. 6-3, and applied to the weld material definition in the model. For this initial test, no damage characteristics were applied to the plates.

As a means of comparison to the simulation results, the load displacement curve given for Gomez’s Test #4, representative for each of the three tests conducted of this particular assembly, was digitized and plotted along with the analytical data. Values for Eqn. 6-1 and 6-2 were calculated as 0.0175 inches and 237 kips, respectively. Results are presented in Figure 6-14.

![Graph](image)

**Figure 6-14: Results of T-Stub Test**

It is observed that fracture of the weld for both the experimental and analytical models was predicted accurately by Eqn. 6-1. The ultimate load of the analytical model was higher than observed in the experimental testing, and approached the value given by Eqn. 6-2. There is a great
deal of numerical noise post-fracture initiation in the results of the analytical model. This appears to have stemmed from the failure mode of the fillet welds in the FE model.

It was noted in Gomez’s research that fracture initiated at the root of the fillet and propagated outward toward the toe of the shear leg. The opposite sequence occurred in the FE model, as evidenced in Fig. 6-15. It is believed that this was caused by over-constraint of the root nodes, previously noted in Section 6.3. This over-constraint again did not allow for relative motion between the two plates to occur at the root of the fillets, and thus the strain of the fillet was absorbed at the toe of the shear leg. Since the damage initiation criteria used is a function of equivalent plastic strain, the elements at the toe of the shear leg, which exhibited similar values of stress triaxiality and shear stress ratio to the root elements, reached their failure strain earlier. Potential methods for avoiding this failure mode are discussed in Chapter 8.

Figure 6-15: Progression of Damage in T-Stub Assembly (Von Mises Contours Plotted)

A second model was run to verify the use of the fillet weld modeling and damage scaling methodologies for fillet welds in conjunction with Wurzelbacher’s damage data for structural steel. Hooputra’s aluminum damage data multiplied by 2, determined by Wurzelbacher for use with A572-Gr. 50 steel, was applied to the plates. No element deletion occurred in the plates, as desired, and thus the load deformation results were identical.
Chapter 7: Conclusions

The objective of this study was to develop a model for the prediction of the onset of fracture in welded steel connections. To achieve this end, the previously developed ductile and shear failure loci for EN AW-7108 T6 aluminum was scaled to exhibit fracture in the finite element models corresponding to the results of previously tested lap splice connections with welds deposited at varying angles. Dependence on variable experimental results was reduced by using established relationships for fracture deformations in the determination of the scale factors for each loading orientation. A damage scaling relationship was developed and was verified for weld size independence by utilizing models of experimental assemblies featuring larger weld sizes.

This chapter summarizes the recommended modeling methodology, followed by the results of the damage data calibration process. Conclusions and limitations of the model are presented then discussed.

7.1 Modeling Recommendations

The studies conducted prior to the implementation of damage mechanics provided guidelines on modeling fillet welded connections in finite element software. Paramount is the use of properly aligned nodes at weld/connected component interfaces. While the discretization techniques for surface-to-surface tie constraints in modern finite element software packages are quite robust, this is not desirable in critical areas of FE models. When nodes are properly aligned between connected surfaces, it was shown that the use of tie constraints is admissible, exhibiting a response identical to the use of common nodes, even at high strains.

Suggested is the use of the idealized isosceles triangle shape for representation of fillet welds in finite element models. This methodology was shown to replicate the established load-deformation response adequately, and allows for ease of modeling.
Critical to the use of the damage scaling relationship for weld segments is the use of a mesh density proportional to the weld size being modeled. This study used a controlling seed size of 1/8 the weld leg size. This was shown adequate for convergence and allowed for a suitably dense mesh for proper execution of element deletion when the failure strain of weld elements is reached. The use of a different constant of proportionality for the weld mesh density will result in a redistribution in the state of stress, upon which the ductile and shear damage criterion is based, and may yield errant results. Note that this study used the specified weld size for modeling. If a model is being formulated for an assembly of which the true weld profile is known, explicit modeling of the average weld profile may yield more accurate results, so long as the mesh density is proportioned accordingly.

Additionally, the use of partitioning scheme “2x” is suggested for obtaining initially well-shaped elements. Refer to Section 4.5 for a complete summary of the model methodology used in this study.

7.2 Lap Splice Assemblies

From the modeling of Miazga and Kennedy’s 5-mm fillet welded lap-slice assemblies loaded at angles of 0, 30, 60 and 90 degrees, a relationship for scaling Hooputra’s ductile and shear loci for EN AW-7108 T6 aluminum was developed into the following equation, in which $\Theta$ is the loading angle (in degrees) of the weld group:

$$ Scale\ Factor = 1 - \frac{1}{150} \theta $$

(Eqn. 6-3)

Utilizing this scaling methodology, fracture deformations in the finite element models coincided with Lesik and Kennedy’s empirically derived relationship for fracture deformation in fillet welds loaded at varying angles, to reduce the variability in using raw experimental results for
the establishment of the scaling relationship. These relationships were developed using the
material properties of CSA E480xx weld metal, which is an accepted equivalent to AWS E70xx
filler material. Additional details of the material property definitions are provided in Chapter 5.

The damage data scaling relationship was then examined for weld size dependence by
modeling Miazga and Kennedy’s 9-mm fillet weld assemblies with loading angles of 0, 45 and 90
degrees. The load-deformation results of the assemblies indicated that the modeling methodology
and damage scaling relationship was adequate for use in larger weld sizes.

7.3 T-Stub Assemblies

To investigate the use of this modeling methodology in a connection scenario other than a
lap splice, Gomez’s T-stub tension test was simulated. An aluminum damage data scaling factor
corresponding to the transverse loading scenario was utilized. The load-deformation results
exhibited fracture at a similar deformation to what was observed in the experimental study.
However, the sequence of fracture did not follow what was observed in the experimental testing,
and resulted in a great deal of numerical noise past the initial fracture point of the fillet welds.
Suggestions to combat this failure mode are provided in the next chapter.

A secondary goal was achieved when Wurzelbacher’s damage scaling was applied to the
base metal in the T-stub model to investigate compatibility between the two damage data scaling
methodologies. No element deletion occurred in the plates when Wurzelbacher’s data was applied,
which would have been indicative of incompatibility, considering fracture occurred only in the
welds via the results of the experimental study.
Chapter 8: Future Research

This study served to expand scope of the aluminum damage data scaling methodologies, first introduced by Wurzelbacher for bolted steel connections, by formulating a similar methodology for structural welds. These results unlock a number of pathways for both advancing and improving upon the suggested methodologies, which are discussed in the next paragraphs.

The most obvious route for advancing the research conducted is to perform the litany of material characterization tests for structural steel, bolt and weld material as prescribed by Hooputra for defining failure loci for the ductile and shear criteria. Fully developed failure envelopes, in theory, will exhibit element deletions stemming from satisfaction of both the ductile and shear criteria. Recall that scaling both failure envelopes equivalently in this study resulted in shear damage to initiate element deletions in all the considered models. However, the docket of experiments to achieve this end is expansive and would be both time and resource intensive to complete. A less resource intensive option that was introduced in Chapter 2 is from the work of Bai and Wierzbicki, in the 2008 publication “A New Model of Metal Plasticity and Fracture with Pressure and Lode Dependence.” This particular fracture model does not explicitly differentiate between ductile and shear damage, but rather introduces the additional parameter of the third invariant of the deviatoric stress, which is based on the Lode angle, in conjunction with the stress triaxiality. The characterization of this failure locus requires only 8 identical butterfly specimens, loaded in various conditions, which is a substantial reduction from the number of tests required to characterize IDS failure criteria. Additionally, Abaqus finite element analysis software has incorporated this definition of damage initiation into its latest version, 6.12.

An analytical approach for furthering the research conducted in this study would be to investigate the use of this methodology for use in modeling an eccentrically loaded connection,
for both in- and out-of-plane eccentricities. Current AISC specifications use the load-deformation relationship, championed by Lesik and Kennedy and referenced in this research, in conjunction with the instantaneous center of rotation analysis method. The author expects a method similar to this may reproduce adequate results when modeling fracture in eccentrically-loaded fillet welded connections.

Further investigation into the use of this scaling methodology for connections such as Gomez’s T-stub is recommended. Over-constraint of the root elements of the fillet weld exists somewhat as a “necessary evil” in this current formulation, despite its effectiveness in modeling fracture in lap splice assemblies. But how will the modeling methodologies presented in Chapter 4 perform when fully developed fracture loci are applied to the models? Are there methods to alleviate this over-constraint? A recommendation would be to develop a method in which tie constraints are eliminated entirely from the FE models, but with the retention of a sliding contact condition between lapped surfaces. This may be achieved through post-processing methods prior to analysis. Will the use of higher order elements, such as 20-noded hexahedral brick elements with or without reduced integration perform better in this arrangement? To what extent will this alter the scaling factors established in this study? These are all questions that must be answered moving forward to advance this research.

As noted in Wurzelbacher (2012), Hooputra’s instability criterion was ignored within the context of this thesis. This stems from the consideration of the weld segments as dimensionally stable, such that instability failures were of no concern. However, certain connected components that may be future topics of study, such as special plate shear walls, may be prone to failure by instability. Additionally, loading scenarios involving compression were not investigated in this study, a scenario likely to arise in the modeling of eccentrically loaded connections.
Chapter 9: References


Appendix A: Ductile and Shear Criteria and Constants

Equations for damage initiation (Hooputra et al. 2004; Abaqus 2012):

Ductile Damage

Damage initiates at:

\[ \omega_D = \int \frac{d\varepsilon^{-pl}}{\varepsilon_D^{-pl}(\eta, \varepsilon^{+pl})} = 1 \]

The equivalent plastic strain at the onset of damage:

\[ \varepsilon_D^{-pl}(\eta, \varepsilon^{+pl}) = \frac{\varepsilon_T^{+} \sinh[k_0(\eta^+ - \eta)] + \varepsilon_T^{-} \sinh[k_0(\eta - \eta^+)]}{\sinh[k_0(\eta^- - \eta^+)之外]}} \]

The stress triaxiality:

\[ \eta = \frac{p}{q} = \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3) \]

\[ q = \sqrt{\frac{1}{2}[((\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2]} \]

Where: p = equivalent mean stress
q = von Mises equivalent stress

Shear Damage

Damage initiates at:

\[ \omega_S = \int \frac{d\varepsilon^{-pl}}{\varepsilon_S^{-pl}(\theta_s, \varepsilon^{+pl})} = 1 \]

The equivalent plastic strain at the onset of damage:

\[ \varepsilon_S^{-pl}(\theta_s, \varepsilon^{+pl}) = \frac{\varepsilon_T^{+} \sinh[f(\theta_s - \theta_s^-)] + \varepsilon_T^{-} \sinh[f(\theta_s^+ - \theta_s^-)]}{\sinh[f(\theta_s^+ - \theta_s^-)]} \]
The shear stress ratio:

$$\theta_s = \frac{(1 - k_s \eta)}{\phi}$$

Where: $\phi = \frac{\tau_{\text{max}}}{\sigma_{eq}}$

Constants for Damage Initiation

Constants used in the above equations for ductile and shear initiation criteria at both quasi-static and dynamic strain rates: (Hooputra et al. 2004 via Abaqus Example Problems Manual 2012)

### Table A-1: Experimentally Determined Ductile Failure Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Quasi-Static</th>
<th>Dynamic (strain rate = 260 s⁻¹)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon_\tau^+$</td>
<td>0.26</td>
<td>0.44</td>
</tr>
<tr>
<td>$\varepsilon_{\tau^-}$</td>
<td>193.0</td>
<td>1494.0</td>
</tr>
<tr>
<td>$k_0$</td>
<td>5.277</td>
<td>8.6304</td>
</tr>
</tbody>
</table>

### Table A-2: Experimentally Determined Shear Failure Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Quasi-Static</th>
<th>Dynamic (strain rate = 260 s⁻¹)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon_\tau^+$</td>
<td>0.26</td>
<td>0.44</td>
</tr>
<tr>
<td>$\varepsilon_{\tau^-}$</td>
<td>193.0</td>
<td>1494.0</td>
</tr>
<tr>
<td>$f$</td>
<td>5.277</td>
<td>8.6304</td>
</tr>
<tr>
<td>$k_s$</td>
<td>0.3</td>
<td>0.3</td>
</tr>
</tbody>
</table>
Appendix B: Required Tests for Derivation of Ductile and Shear Constants

The experimental tests performed to determine the material constants for EN AW-7108 T6 Aluminum (Hooputra et al. 2004 via Wurzelbacher 2012):

- Erichsen Tests (equibiaxial stress with $\eta = 2$):
- Three Point Bending of Sheet Coupons (width/thickness > 4 with plane strain tension and $\eta = \sqrt{3}$)
- Wasted Tensile Coupons, fracture at the notch root (uniaxial tension $\eta = 1$)
- Tensile specimen (rectangular cross-section) with a groove to half-depth 45° to loading ($\theta_s = 1.469$)
- Tensile specimen (rectangular cross-section) with a groove parallel to loading ($\theta_s = 1.732$)
- Erichsen (biaxial tension with $\theta_s = 1.6$)
### Appendix C: Hooputra EN AW-7108 T6 Aluminum Damage Data

#### Table C-1: Aluminum Damage Data (Abaqus Example Problems Manual, 2012)

<p>| Fracture Strain | Ductile Criteria | | Shear Criteria |
|-----------------|------------------|------------------|
|                 | Fracture Stress Triaxiality | Strain Rate | Fracture Strain | Shear Stress Ratio | Strain Rate |
| 33.2380         | -3.3333          | 0.0001          | 0.2761          | -10.0000          | 0.0001 |
| 33.2380         | -3.3333          | 0.0001          | 0.2761          | 1.4236            | 0.0001 |
| 23.3810         | -0.2667          | 0.0001          | 0.2613          | 1.4625            | 0.0001 |
| 16.4470         | -0.2000          | 0.0001          | 0.2530          | 1.5013            | 0.0001 |
| 11.5700         | -0.1333          | 0.0001          | 0.2510          | 1.5401            | 0.0001 |
| 8.1934          | -0.0667          | 0.0001          | 0.2551          | 1.5789            | 0.0001 |
| 5.7268          | 0.0000           | 0.0001          | 0.2656          | 1.6177            | 0.0001 |
| 4.0303          | 0.0667           | 0.0001          | 0.2825          | 1.6566            | 0.0001 |
| 2.8377          | 0.1333           | 0.0001          | 0.3065          | 1.6954            | 0.0001 |
| 2.0000          | 0.2000           | 0.0001          | 0.3379          | 1.7342            | 0.0001 |
| 1.4124          | 0.2667           | 0.0001          | 0.3778          | 1.7730            | 0.0001 |
| 1.0013          | 0.3333           | 0.0001          | 0.4269          | 1.8118            | 0.0001 |
| 0.7155          | 0.4000           | 0.0001          | 0.4865          | 1.8506            | 0.0001 |
| 0.5192          | 0.4667           | 0.0001          | 0.5581          | 1.8895            | 0.0001 |
| 0.3878          | 0.5333           | 0.0001          | 0.6435          | 1.9283            | 0.0001 |
| 0.3048          | 0.6000           | 0.0001          | 0.7448          | 1.9671            | 0.0001 |
| 0.2600          | 0.6667           | 0.0001          | 0.8644          | 2.0059            | 0.0001 |
| 0.2476          | 0.7301           | 0.0001          | 1.0053          | 2.0447            | 0.0001 |
| 0.3028          | 0.8510           | 0.0001          | 1.1710          | 2.0835            | 0.0001 |
| 0.6195          | 1.0237           | 0.0001          | 1.3655          | 2.1224            | 0.0001 |
| 1.9018          | 1.2435           | 0.0001          | 1.5937          | 2.1612            | 0.0001 |
| 7.5608          | 1.5058           | 0.0001          | 1.8611          | 2.2000            | 0.0001 |
| 7.5608          | 3.3333           | 0.0001          | 1.8611          | 10.0000           | 0.0001 |
| 33.2380         | -3.3333          | 0.0010          | 0.2761          | -10.0000          | 0.0010 |
| 33.2380         | -0.3333          | 0.0010          | 0.2761          | 1.4236            | 0.0010 |
| 23.3810         | -0.2667          | 0.0010          | 0.2613          | 1.4625            | 0.0010 |
| 16.4470         | -0.2000          | 0.0010          | 0.2530          | 1.5013            | 0.0010 |
| 11.5700         | -0.1333          | 0.0010          | 0.2510          | 1.5401            | 0.0010 |
| 8.1934          | -0.0667          | 0.0010          | 0.2551          | 1.5789            | 0.0010 |
| 5.7268          | 0.0000           | 0.0010          | 0.2656          | 1.6177            | 0.0010 |
| 4.0303          | 0.0667           | 0.0010          | 0.2825          | 1.6566            | 0.0010 |
| 2.8377          | 0.1333           | 0.0010          | 0.3065          | 1.6954            | 0.0010 |
| 2.0000          | 0.2000           | 0.0010          | 0.3379          | 1.7342            | 0.0010 |
| 1.4124 | 0.2667 | 0.0010 | 0.3778 | 1.7730 | 0.0010 |
| 1.0013 | 0.3333 | 0.0010 | 0.4269 | 1.8118 | 0.0010 |
| 0.7155 | 0.4000 | 0.0010 | 0.4865 | 1.8506 | 0.0010 |
| 0.5192 | 0.4667 | 0.0010 | 0.5581 | 1.8895 | 0.0010 |
| 0.3878 | 0.5333 | 0.0010 | 0.6435 | 1.9283 | 0.0010 |
| 0.3048 | 0.6000 | 0.0010 | 0.7448 | 1.9671 | 0.0010 |
| 0.2600 | 0.6667 | 0.0010 | 0.8644 | 2.0059 | 0.0010 |
| 0.2476 | 0.7301 | 0.0010 | 1.0053 | 2.0447 | 0.0010 |
| 0.3028 | 0.8510 | 0.0010 | 1.1710 | 2.0835 | 0.0010 |
| 0.6195 | 1.0237 | 0.0010 | 1.3655 | 2.1224 | 0.0010 |
| 1.9018 | 1.2435 | 0.0010 | 1.5937 | 2.1612 | 0.0010 |
| 7.5608 | 1.5058 | 0.0010 | 1.8611 | 2.2000 | 0.0010 |
| 7.5608 | 3.3333 | 0.0010 | 1.8611 | 10.0000 | 0.0010 |
| 84.1320 | -3.3333 | 250.0000 | 0.3338 | -10.0000 | 250.0000 |
| 84.1320 | -0.3333 | 250.0000 | 0.3338 | 1.4236 | 250.0000 |
| 47.3270 | -0.2667 | 250.0000 | 0.3336 | 1.4625 | 250.0000 |
| 26.6210 | -0.2000 | 250.0000 | 0.3355 | 1.5013 | 250.0000 |
| 14.9740 | -0.1333 | 250.0000 | 0.3396 | 1.5401 | 250.0000 |
| 8.4238 | -0.0667 | 250.0000 | 0.3457 | 1.5789 | 250.0000 |
| 4.7393 | 0.0000 | 250.0000 | 0.3541 | 1.6177 | 250.0000 |
| 2.6675 | 0.0667 | 250.0000 | 0.3647 | 1.6566 | 250.0000 |
| 1.5034 | 0.1333 | 250.0000 | 0.3777 | 1.6954 | 250.0000 |
| 0.8508 | 0.2000 | 250.0000 | 0.3930 | 1.7342 | 250.0000 |
| 0.4878 | 0.2667 | 250.0000 | 0.4108 | 1.7730 | 250.0000 |
| 0.2908 | 0.3333 | 250.0000 | 0.4312 | 1.8118 | 250.0000 |
| 0.1926 | 0.4000 | 250.0000 | 0.4543 | 1.8506 | 250.0000 |
| 0.1601 | 0.4667 | 250.0000 | 0.4804 | 1.8895 | 250.0000 |
| 0.1820 | 0.5333 | 250.0000 | 0.5094 | 1.9283 | 250.0000 |
| 0.2658 | 0.6000 | 250.0000 | 0.5417 | 1.9671 | 250.0000 |
| 0.4400 | 0.6667 | 250.0000 | 0.5774 | 2.0059 | 250.0000 |
| 0.7434 | 0.7301 | 250.0000 | 0.6168 | 2.0447 | 250.0000 |
| 2.0883 | 0.8510 | 250.0000 | 0.6601 | 2.0835 | 250.0000 |
| 9.2599 | 1.0237 | 250.0000 | 0.7076 | 2.1224 | 250.0000 |
| 61.7180 | 1.2435 | 250.0000 | 0.7596 | 2.1612 | 250.0000 |
| 593.6700 | 1.5058 | 250.0000 | 0.8163 | 2.2000 | 250.0000 |
| 593.6700 | 3.3333 | 250.0000 | 0.8163 | 10.0000 | 250.0000 |
| 84.1320 | -3.3333 | 1000.0000 | 0.3338 | -10.0000 | 1000.0000 |
| 84.1320 | -0.3333 | 1000.0000 | 0.3338 | 1.4236 | 1000.0000 |
| 47.3270 | -0.2667 | 1000.0000 | 0.3336 | 1.4625 | 1000.0000 |
| 26.6210 | -0.2000 | 1000.0000 | 0.3355 | 1.5013 | 1000.0000 |
| 14.9740 | -0.1333 | 1000.0000 | 0.3396 | 1.5401 | 1000.0000 |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>8.4238</td>
<td>-0.0667</td>
<td>1000.0000</td>
<td>0.3457</td>
<td>1.5789</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>4.7393</td>
<td>0.0000</td>
<td>1000.0000</td>
<td>0.3541</td>
<td>1.6177</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>2.6675</td>
<td>0.0667</td>
<td>1000.0000</td>
<td>0.3647</td>
<td>1.6566</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>1.5034</td>
<td>0.1333</td>
<td>1000.0000</td>
<td>0.3777</td>
<td>1.6954</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.8508</td>
<td>0.2000</td>
<td>1000.0000</td>
<td>0.3930</td>
<td>1.7342</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.4878</td>
<td>0.2667</td>
<td>1000.0000</td>
<td>0.4108</td>
<td>1.7730</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.2908</td>
<td>0.3333</td>
<td>1000.0000</td>
<td>0.4312</td>
<td>1.8118</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.1926</td>
<td>0.4000</td>
<td>1000.0000</td>
<td>0.4543</td>
<td>1.8506</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.1601</td>
<td>0.4667</td>
<td>1000.0000</td>
<td>0.4804</td>
<td>1.8895</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.1820</td>
<td>0.5333</td>
<td>1000.0000</td>
<td>0.5094</td>
<td>1.9283</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.2658</td>
<td>0.6000</td>
<td>1000.0000</td>
<td>0.5417</td>
<td>1.9671</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.4400</td>
<td>0.6667</td>
<td>1000.0000</td>
<td>0.5774</td>
<td>2.0059</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>0.7434</td>
<td>0.7301</td>
<td>1000.0000</td>
<td>0.6168</td>
<td>2.0447</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>2.0883</td>
<td>0.8510</td>
<td>1000.0000</td>
<td>0.6601</td>
<td>2.0835</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>9.2599</td>
<td>1.0237</td>
<td>1000.0000</td>
<td>0.7076</td>
<td>2.1224</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>61.7180</td>
<td>1.2435</td>
<td>1000.0000</td>
<td>0.7596</td>
<td>2.1612</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>593.6700</td>
<td>1.5058</td>
<td>1000.0000</td>
<td>0.8163</td>
<td>2.2000</td>
<td>1000.0000</td>
<td></td>
</tr>
<tr>
<td>593.6700</td>
<td>3.3333</td>
<td>1000.0000</td>
<td>0.8163</td>
<td>10.0000</td>
<td>1000.0000</td>
<td></td>
</tr>
</tbody>
</table>
Appendix D: Material Plasticity Comparisons

Figure D-1: E48014 Weld Metal Engineering Stress vs. Strain (Miazga and Kennedy 1986)

Figure D-2: E48014 Weld Metal True vs. Engineering Stress-Strain
Figure D-3: 300W Steel Engineering Stress vs. Strain (Miazga and Kennedy 1986)

Figure D-4: 300W Steel True Stress vs. Logarithmic Plastic Strain
Figure D-5: E70T-7 Weld Metal Engineering Stress vs. Strain (Gomez et al. 2008)

Figure D-6: E70T-7 Weld Metal Engineering vs. True Stress-Strain
Figure D-7: A572 Gr. 50 Steel Engineering Stress vs. Strain (Gomez et al. 2008)

Figure D-8: A572 Gr. 50 Steel Engineering vs. True Stress-Strain
The following equations were utilized to convert engineering stress and strain from the respective ancillary material characterization tests performed by Miazga and Kennedy (1986) and Gomez et al. (2008) to true stress and logarithmic plastic strain for input into Abaqus for finite element analysis. The equations are reproduced from the Abaqus User’s Manual (2012).

\[
\text{True Stress} = \text{Eng. Stress} \times (1 + \text{Eng. Strain})
\]

\[
\text{Log. Plastic Strain} = \ln (1 + \text{Eng. Strain}) - \frac{\text{True Stress}}{E}
\]

In the preceding equation, \( E \) represents Young’s Modulus, which results in the first data point representing the initial point of yield, i.e. a plastic strain value equal to zero at the true yield stress.
Appendix E: Eqn. 4-3 Normalization Process

This Appendix details the steps taken to normalize the results of the various analytical models for comparison with Eqn. 4-3. These steps are derived from Lesik and Kennedy’s 1990 publication “Ultimate Strength of Fillet Welded Connections Loaded in Plane.”

\[ P = P_0(1.00 + 0.50 \sin^{1.5} \theta)f(\rho) \]

Reproduced is Eqn. 4-3, representing the load resisted by a fillet weld group at any deformation. \( P_0 \) is taken at 290.7 N/mm/mm, which is the average normalized ultimate strength of the longitudinal assemblies from Miazga and Kennedy’s experimental testing. The loading angle of the fillet weld is given in degrees, represented by \( \theta \). The function \( f(\rho) \) is a polynomial given in the form:

\[
\begin{align*}
    f(\rho) &= 8.234\rho; \quad 0 < \rho \leq 0.0325 \\
    f(\rho) &= -13.29\rho + 457.32\rho^2 - 3385.9\rho^3 + 9054.29\rho^4 - 9952.13\rho^5 + 3840.71\rho^6; \\
    & \quad \rho > 0.0325
\end{align*}
\]

The variable \( \rho \) is then given as a function of the deformation \( (\Delta) \) and weld size \( (w) \):

\[
\rho = \frac{\Delta}{0.209(\theta + 2)^{-0.32}w}
\]

To normalize the analytical data, the applied loads are divided by the weld size and the length of the weld. The deformations are divided by the weld size. To normalize by the ultimate strength of longitudinal welds, a value of unity is used for \( P_0 \) in Eqn. 4-3, and the applied loads from the analytical data are divided by 290.7 N/mm/mm. Conversion to customary units can be performed with relative ease.
Appendix F: Results of Convergence Study

This Appendix provides graphical comparisons of the seed sizes and partitioning schemes investigated in the convergence study performed in the development of the modeling methodology. All models were analyzed using the Abaqus Explicit solver and utilized C8D8R hexahedral parametric elements with reduced integration. The partitioning schemes and seed sizes investigated are introduced and discussed in Section 4.4. The results for the “2x” partitioning scheme are presented in Section 4.4 and are not reproduced in this Appendix.

![Graph showing mesh sensitivity analysis](image)

Figure F-1: Mesh Sensitivity Analysis for the "Null" Partitioning Scheme
Figure F-2: Mesh Sensitivity Analysis for the "1x" Partitioning Scheme

Figure F-3: Mesh Sensitivity Analysis for the "3x" Partitioning Scheme
Appendix G: Lap Splice Assemblies (Miazga and Kennedy 1986)

Figure G-1: 5 mm Fillet Weld Assemblies
Figure G-2: 9 mm Fillet Weld Assemblies
Appendix H: Lap Splice Experimental vs. Analytical Comparisons

This appendix provides graphical force-displacement comparisons between the lap splice assemblies tested by Miazga and Kennedy (1986) and the relationships presented in Lesik and Kennedy’s research (1990). The charts representing the experimental data that are presented below were digitized and plotted alongside analytical results for comparison purposes.

Figure H-1: 0-Deg Assembly Experimental Results (Lesik and Kennedy 1990)

Figure H-2: 30-Deg Assembly Experimental Results (Lesik and Kennedy 1990)
Figure H-3: 90-Deg Assembly Experimental Results (Lesik and Kennedy 1990)

Figure H-4: 0-Deg Experimental vs. Analytical Comparison
Figure H-5: 30-Deg Experimental vs. Analytical Comparison

Figure H-6: 90-Deg Experimental vs. Analytical Comparison