ENERGY RELEASE MANAGEMENT THROUGH MANIPULATED GEOMETRIES OF SURGICAL DEVICES

A Thesis
Presented to
The Graduate Faculty of The University of Akron

In Partial Fulfillment
of the Requirements for the Degree
Master of Science

Jason King
August, 2012
ABSTRACT

Treatment options for spinal trauma, chronic disease and/or age degeneration vary based on the medical prognosis and severity of symptoms, with surgery being one treatment option. One of the many surgical options for stabilization is pedicle screw based fusion, where screws are placed into the spine and metal rods are fixed to these screws using set screws. The tightening torque is critical for these procedures, as under or over torquing leads to slippage, decreased strength, and ultimately poor fusion results. Numerous products in the market use a torque wrench or break-off set screws that ensure the correct tightening torque. The focus of this thesis is to develop techniques (based on geometrical alterations without changing the material) that modify the set screws so that they reduce shock (200-800 g-force) while still ensuring the correct tightening torque.

The hypothesis of this thesis is to show that geometric changes in the set screw tightening structure (i.e. device) can reduce the maximum shock by influencing how the stored energy is released, and yet keep the tightening torque the same. The objective is to increase the time period between yielding and fracture and channel more energy towards plastic deformation. The specific aim is to make geometric changes in the grooved region (designed for break off) to cause the ratio of maximum recoverable strain energy to maximum plastic energy dissipation to decrease. Computer models
based on fracture mechanics are used to demonstrate this. Using a model for ductile metal Al 5083-H116, it was shown that wider, more gradual grooves lead to a 36% decrease in this ratio. A decrease in this ratio indicates that the maximum g-force shock will decrease since a greater percent of the energy will be dissipated during material deformation. In addition, there was a 74% increase in rotation before failure. This behavior is believed to be beneficial since broken bonds between molecules in the shear band are given more time to re-bond with an adjacent molecule, and this new bond consumes more energy when final shear separation occurs. Optimization, experimental validation, and extension of the models to titanium remain as future work.
ACKNOWLEDGEMENTS

My most genuine thanks is extended to all of the people who have helped make my graduate school years and thesis a success. I have learned a tremendous amount from my advisor, Dr. Ajay Mahajan, about how to utilize the expertise available in an academic environment and collaborate to create a tangible product that can meet the unrealized needs of society. His patient coaching on academic, research, and business issues gave me the environment I needed to excel, and I would not be at this point in my life and career without him.

I have the great fortune of also having a fantastic group of committee members: Dr. Wieslaw Binienda, Dr. Atef Saleeb from Civil Engineering Department and Dr. Juay Seng Tan from Biomedical Engineering Department. Through their classes, one-on-one interactions, and thesis meetings, they have given me extensive direction, insight, and encouragement. I would also like to thank Dr. Mary Verstraete for being willing to serve as an emergency committee member.

The following professors also allowed me to consult with their research assistants, which was instrumental in helping me achieve results from the computer models. Abaqus modeling expertise was obtained from Binod Dhakal of Dr. Saleeb’s lab,
Jun Zhou of Dr. Xiaosheng Gao’s lab (Mechanical Engineering Department), and Zhuopei (Vincent) Hu of Dr. Binienda’s lab.

The various stages of my testing were made possible by the gracious help of various other advisors and university faculty, staff, and students. Dr. Sumeer Lal from the Carolina Neurosurgery & Spine Center provided guidance and advice pertaining to the issues in the operating room. Dr. Kye-Shin Lee from Electrical and Computer Engineering Department provided consultation on the sensor setup for future physical validation testing. Mr. Dale Ertley (College of Engineering Sr. Technician) provided countless hours of consultation and machining time for the setup of validation testing fixtures. My colleagues Esra Cipa, Margaret Brass, and Laura VonDeak have provided me testing and research support. In addition, a special thanks to Jackie Lombardi from the Engineering Dean’s Office for aiding in administrative, logistics, and purchasing tasks.

Most of all, I want to thank my amazing wife, Rachelle, and wonderful family for giving me the confidence and support to change my career direction and begin school again.
# TABLE OF CONTENTS

LIST OF TABLES..........................................................................................................................ix

LIST OF FIGURES.........................................................................................................................x

CHAPTER

I. INTRODUCTION.............................................................................................................................1

1.1 Clinical Issues.........................................................................................................................1

1.2 Treatment Options...............................................................................................................8

1.3 Identification/Focusing of the Problem (Problem Background)......................................13

II. HYPOTHESIS AND GOVERNING THEORY.............................................................................17

2.1 Hypothesis..............................................................................................................................17

2.2 Theory.....................................................................................................................................19

2.2.1 Linear Elastic Fracture Mechanics.................................................................................20

2.2.2 Elastic-Plastic Fracture Mechanics...............................................................................24

2.3 Strain Rate and Processing Importance...............................................................................30

2.4 Material Selection.................................................................................................................32

2.5 Proposed Initial Geometries...............................................................................................32

2.6 Testing Outline......................................................................................................................33

III. THEORY TO COMPUTER MODEL..........................................................................................35
3.1 2D-Axisymmetric-With-Twist Setup ................................................................. 38
3.2 Results: Aluminum–Deleted Elements (Triaxial Strain Criteria) .................. 39

IV. SETUP FOR FUTURE WORK ................................................................................. 48
4.1 3D Setup .............................................................................................................. 48
4.2 Suggested Future Complex Geometries ............................................................ 50
4.3 Experimental Setup .......................................................................................... 53
4.4 Confirmation of Modeling Hypothesis: Physical Proof of Concept .............. 62

V. CONCLUSION .......................................................................................................... 64
5.1 Geometry Effect on Plastic Energy and Future Work ..................................... 64
5.3 Future Testing ..................................................................................................... 65
5.4 Key Take Away .................................................................................................. 66

BIBLIOGRAPHY ........................................................................................................... 68

APPENDICES ............................................................................................................... 72

APPENDIX A. SPINE ANATOMY OVERVIEW ......................................................... 73

APPENDIX B. EXPERIMENTAL AND EUATION VALIDATION OF MANUFACTURER
SPECIFIED SET SCREW BREAK OFF (SSBO) TORQUE ........................................ 82

APPENDIX C. INDIVIDUAL OUTPUT GRAPHS FROM ALL TRIALS, WITH NAMING
CONVENTION OF THE LENGTH OF THE GROOVE BOTTOM FOLLOWED BY THE
RADIUS OF THE GROOVE FILLET (IN .1MM UNITS), FOLLOWED BY EXTERNAL
WORK AND ELASTIC ENERGY COMPARISON GRAPHS .......................................... 85

APPENDIX D. AL5083-H116 PARAMETERS USED IN THE MODEL ................. 93

APPENDIX E. ABAQUS (VERSION 6.10-2) STEPS USED TO BUILD THE 2D
AXISYMMETRIC MODEL WITH TWIST ................................................................. 94

APPENDIX F. ABAQUS (VERSION 6.10-2) STEPS USED TO BUILD THE 3D CYCLIC
SYMMETRIC MODEL .............................................................................................. 99
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-1</td>
<td>Testing Order</td>
</tr>
<tr>
<td>3-1</td>
<td>Mesh Convergence Values</td>
</tr>
<tr>
<td>3-2</td>
<td>Comparison of the two geometric extremes L0R2 and L3R8</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-1</td>
<td>The various shape deviations resulting from vertebral fracture and/or degeneration [2]</td>
<td>4</td>
</tr>
<tr>
<td>1-2</td>
<td>Disc herniation severity [2]</td>
<td>4</td>
</tr>
<tr>
<td>1-3</td>
<td>Using an extended spine profile as an example, one can see geometrically how flexion would increase the spinal foramen [2]</td>
<td>5</td>
</tr>
<tr>
<td>1-4</td>
<td>Illustrations of different spine deformities in reference to the general shape [6]</td>
<td>5</td>
</tr>
<tr>
<td>1-5</td>
<td>Effects of position/posture on disc pressure [2]</td>
<td>6</td>
</tr>
<tr>
<td>1-6</td>
<td>Laminectomy and discectomy depiction [2]</td>
<td>10</td>
</tr>
<tr>
<td>1-7</td>
<td>Typical spinal fusion implant options [2]</td>
<td>10</td>
</tr>
<tr>
<td>1-8</td>
<td>Standard spinal fusion implant utilizing break off set screws. The red boxed portion is the focus of this thesis</td>
<td>12</td>
</tr>
<tr>
<td>1-10</td>
<td>Pedicle screw construct (US Patent 6,179,841 B1) with attached fixture and accelerometer [9]</td>
<td>14</td>
</tr>
<tr>
<td>1-11</td>
<td>Two different test runs showing: (A) Oscilloscope output of the initial signal recorded, (B) Excel graph conversion of the .csv data signal to g-force in Excel [9]</td>
<td>15</td>
</tr>
<tr>
<td>2-1</td>
<td>(A) Current g-force and profile curve of set screw break off (B) Desired reduction of maximum g-force</td>
<td>17</td>
</tr>
</tbody>
</table>
2-2 Area under the stress strain curve is the material’s toughness [14]..................................................................................................................20

2-3 The three modes of crack displacement and corresponding relative effects on the stress intensity factor, K [15]........................................................................................................................................21

2-4 Crack growth rate increases with crack size [16].................................................................................................................................23

2-5 Illustration of the competing intrinsic (promoting) and extrinsic (impeding) forces in crack propagation [17]........................................23

2-6 Three dimensional and two dimensional comparisons of Tresca and Von Mises failure envelope [25]..................................................27

2-7 Comparison of multiple failure models in the strain-triaxiality plane [22] ........................................................................................29

2-8 Polycrystalline tantalum (BCC) material structure behavior showing strain increases as strain rate decreases [18].................................................................31

2-9 The axisymmetric cross sections of the two most dissimilar geometries. A (L0R2) has a simple groove and .0002in fillet radius, while B (L3R8) has a .0003in flat at the bottom of the groove with a .0008in fillet radius. In both, the boxed red edge is fixed, while a moment about the central axis is applied to the boxed blue edge. C is what L3R8 looks like when it has been extruded about this central axis by 180 degrees..................................................33

3-1 Mesh sensitivity graph showing how the maximum Mises Stress converges as the mesh is refined. The labeled locations along the x-axis correspond to the stress profile pictures. Mesh size 4e-05 (B) was the best compromise between computational cost and accuracy...............................................................................................................37

3-2 Pivot Graph results of the parametric Abaqus model testing showing (from top to bottom) Abaqus variables ALLPD, ALLSE, ALLWK .................................................................................................40

3-3 Graphs comparing the output of all trials showing that
(from left to right): (A) The maximum moment remains relatively constant while increasing in duration (B) The plastic dissipation of energy and total displacement (strain) increases significantly. The decreasing plastic dissipation energy component is an inaccurate artifact caused by the element deletion that is used to model the fracture.

3-4 Graphical results of the energy and moment values for the plastic dissipation shown in Figure 3-2 (A and B respectively). The thick red line shows the point where first yield occurs, the red circle at the peak shows where the first element is deleted, and the second red circle shows where the final element is deleted.

3-5 L0R2 (A: 3.09E-7 of the rotation cycle) vs. L3R8 (B: 0.0142 of the rotation cycle) at first yield illustrating the locations of first plastic deformation and associated stress field.

3-6 L0R2 (A: 0.2428 of the rotation cycle) vs. L3R8 (B: 0.5542 of the rotation cycle) at maximum moment showing plastic deformation and associated stress field.

3-7 L0R2 (A: 0.4516 of the rotation cycle) vs. L3R8 (B: 0.8624 of the rotation cycle) just before first element deletion illustrating the plastic deformation and associated stress field.

3-8 L0R2 (A: 0.594 of the rotation cycle) vs. L3R8 (B: 0.9399 of the rotation cycle) just before last element deletion illustrating the plastic deformation and associated stress field.

4-1 (A) Shows how the size of elements needed for an entire model to run cause a significant volume to be deleted when the elements are deleted (B) Shows a refined mesh cyclic symmetric model (C) Shows how even deletion of fine 3D elements causes a significant decrease of plastic dissipation energy (ALLPD).

4-2 Embodiment A of the first concept.
4-3 Embodiment B of the first concept .................................................................52

4-4 The two methods of torque measurement that will be used are the direct torque sensor shown in A and B (TQM201-56.5 [30]) and the hand dynamometer applying force at a moment arm (C).................................................................55

4-5 Accelerometer equipment, fixture, and orientation of the sensors .................................................................56

4-6 Data acquisition setup (See text for descriptions).................................................................57

4-7 The testing setup (D) utilizing 2 modified spin index fixtures (A), 2 modified 5C collets (B) [34], and a Trantorque OE Mini keyless bushing (C) [35]..................................................................................59

4-8 Illustration of where NOT to position the accelerometer fixture (A) and how to modify the test piece surface and accelerometer fixture to create a more secure attachment..................................................................................61

4-9 (A) Shows a lathe insert fixture with a triangular insert on the right side (B) Illustrates the insert geometries available for order..................................................................................63
CHAPTER I

INTRODUCTION

1.1 Clinical Issues

Many issues regarding the spine present themselves with the clinical indications of back pain, leg weakness, leg numbness, and/or general shape deviation of the spine. The actual cause of these issues in an individual could vary greatly, even with similar symptoms, and may even be a combination of a few conditions. There are some defects that are due solely to issues with the bone, and others that are due only to the soft tissue, of which disc problems are the most prevalent. Most conditions, however, result in deformation of both the bone and soft tissue due to the complex structure, loading, and motion that occurs within the spine [2]. A general overview of spine anatomy can be found in APPENDIX A.

Spondyloarthropathy is any disease that affects the vertebral joints, while spondylopathy diseases affect the vertebrae bone. For example, Spondylosis is a spondyloarthropathic condition where the disc and or facet joints have degenerated. This reduces the spacing between each vertebra, thereby reducing the intervertebral foramen. If the area becomes small enough, the nerve root may become compressed and result in pain or loss of motion. Spondylolysis is a spondylopathic disease
characterized by a thin, miss-formed, weak, or otherwise defective pars interarticularis (part of the lamina). If an individual participates in activities causing extreme stress at these weak points, the pars interarticularis can completely fracture, allowing the injured vertebrae to slip anterior to the inferior vertebra in a condition called Spondylolytic Spondylolisthesis. However, it is also possible for injury or age to cause this relative anterior-posterior movement between two vertebrae without injury to the pars interarticularis, in which case this defect is simply Spondylolisthesis. A disease can be both spondyloarthropathic and spondylopathic, though, as in the case of Spondylitis [2]. Spondylitis is a swelling of the vertebrae which provides a tendency for surrounding joints to swell as well. Ankylosing spondylitis has so much vertebral expansion that the range of motion is decreased, often to the point of spinal fusion [3].

Usually there are different types and grades of the spinal conditions. There are five classes, which indicate the origin of the condition:

I: Dysplastic (Congenital)

II: Isthmic (early in life due to A:lytic, B:elongated pars, or C:acute pars fracture)

III: Degenerative (later in life usually resulting from worn facets)

IV: Posttraumatic

V: Pathologic (disease related)

VI: Latrogenic (side effect related)

Each of these classes are further categorized into different grades, which indicate severity [2]. For example, a vertebra that has slipped completely off the inferior
vertebrae is categorized as (per the Meyerding grading system) grade 5, which is medically referred to as spondyloptosis[4].

Other spondylopathic defects include osteophytes, stenosis, fractures, and osteoporosis. Osteophytes are calcified protrusions from any normal bone surfaces. Any protrusion into a joint space can inhibit motion or cause pain (osteoarthritis) due to increased wear from the new bone surfaces that do not have protective cartilage. Osteophytes can also narrow the foramen in a condition called stenosis. Lateral (foraminal) stenosis is a narrowing of the intervertebral foramen, and central (spinal) stenosis is a narrowing of the spinal canal. Vertebral fracture and osteoporosis can also change the profile of the spine in a variety of ways, as evident in Figure 1-1. In addition to concavity, compression and wedging, a surface defect called a Schmorl node can also form, which is a small localized pit in the nucleus region [2].

Spondyloarthropathic conditions in the disc may be due to an annulus fibrosis defect or nucleus propulosis defect. Annular injuries (i.e. lesion/fissures) have three main types; rim lesions are tears of the outer annular layers (Sharpey Fibers), concentric tears are separations in the annular layers, and radial tears permeate through all annular layers at a location. These tears often result in creating a weakened pathway for the nucleus, which is pressurized upon loading, to extrude into or completely through the annulus fibers (herniate). Pain resulting from these herniations can be classified in two ways. First, discogenic pain is an intra-discal pain arising from the nucleus contacting the sinu-vertebral nerves located in the posterior third of the annulus layer. Second, disc-herniation-induced sciatica is irritation due to the disc bulge applying
pressure on the nerve (either sciatic rootlets or spinal cord) and/or chemical interaction from the contact [5]. The four stages of disc herniation are illustrated in Figure 1-2.

Herniations that create a bulge narrow the foramen in a condition referred to as stenosis. Disc stenosis can occur independent of, or in conjunction with, the aforementioned osteophytes. In addition to pain, disc herniation radiculopathy (numbness or weakness without primary pain). However, size and severity of the disc herniation is not necessarily correlated with more intense symptoms. Pain is usually only indicated when there is a loss of function, such as in bowel control, muscle atrophy, and/or muscle reflex delays [4]. These herniations can heal themselves, but may take up to 18 months. In addition, the nerves tend to follow the scar tissue and can grow all the way into the disc nucleus, making movement extremely painful whenever the nucleus is put into compression [5]. Nucleus defects, however, are typically due to lowered osmotic pressure which reduces the spacing between two discs or more. This collapsed

<table>
<thead>
<tr>
<th>Figure 1-1: The various shape deviations resulting from vertebral fracture and/or degeneration [2]</th>
<th>Figure 1-2: Disc herniation severity [2]</th>
</tr>
</thead>
<tbody>
<tr>
<td>[Gerant et al., 1993 &amp; Lenchik et al., 2004]</td>
<td>[Eken, 1991]</td>
</tr>
</tbody>
</table>

![Four degrees of disc herniation](image)
disc also causes a protrusion and can cause a critical stenosis of either foramen (Figure 1-3) [2].

Depending on a patient’s genes, problems in a disc may start with a general degeneration rather than a specific condition. Paths of degeneration are illustrated as originating either solely because of the disc or solely from the bone. However, the health of the bone and the disc can vary clinically. If the disc is healthy, but the bone has a relatively low bone mineral density, then fractures, Schmorl nodes and general shape changes may develop under the influence of disc pressure. The now misshapen vertebra can cause disc bulge (independent of nucleus migration) and/or a general
change in the curvature of an individual’s spine (Figure 1-4). This curvature change results in an altered loading pattern, causing further degeneration at the facet joints, annulus fibers, and/or disc nucleus (see how posture affects disc pressure in Figure 1-5), whereas the reduction in foramen area caused by the bulge may lead to the

Figure1-5: Effects of position/posture on disc pressure [2]
previously listed issues of stenosis. The decreased disc height may also lessen the tension in supporting ligaments, thus diminishing spine stability. Furthermore, reduction in range of motion (ROM) due to osteophytes at one level requires that surrounding levels must work in an extended, potentially unhealthy, ROM in order to allow required movements of a person [2]. In the second condition, if the disc has degenerated (usually due to a loss in water absorption capability indicated by a “black disc” in an MRI [4]) and the bone is healthy, this also causes a reduction in disc height and leads to the same problems. The resulting general shape change may be due to two reasons. First, an individual could be trying to alleviate the pain of stenosis by leaning forward. Flexion of the spine causes the disc bulge and inward buckling of the ligamentum flavum to be reduced (Figure 1-3), increasing the foramen area and mitigating the source of pain. This can cause the individual to be permanently bent over due to bone growth responding to the modified loading pattern, in what is formally referred to as kyphosis (also known as “shopping cart syndrome”). Second, the changes of kyphotic and lordotic curves (Figure 1-4) may result from osteoporotic changes to the bone structure, and may not involve pain. Other descriptions of a general deformity to the overall structure include ‘scoliosis’ for abnormal lateral curvature, and ‘flat back’ when the lordotic and/or kyphotic curves are less than a typical spine. For Scoliosis, there are 65 known etiologies (i.e. sources), grouped as idiopathic (“unknown”), neuromuscular, congenital, and syndromic [2].

There are numerous other spinal defects and variations of the mentioned issues, including traumatic injuries. However, the discussion in this section is only meant to
illustrate how degeneration and/or failure in any spinal component can directly or indirectly cause major functional impediments.

1.2 Treatment Options

Due to the complexity of the spine and range of problems that could afflict a person, one could imagine that there would be a number of treatment options available. Most doctors start with conservative and non-invasive treatments that may ultimately lead to surgery. Conservative approaches include anti-inflammatory drugs, physical therapy (including regular stretching and exercising), and attention/tools (ex. “Don’t slouch” /Back pillows) for good posture. They are most appropriate for mild to moderately severe conditions, and although studies have shown surgery has a more immediate benefit (measured at one year from the start of treatment), both treatment options have been shown to perform equally well at the three year mark for these grades of conditions. However, depending on the severity of the spinal condition, surgery may be the only method available to provide successful treatment [2]. Surgical treatment will be the primary focus of the remaining information in this section, since it is in these type of treatments where the proposed medical device will be used.

Surgical approaches are grouped as anterior, posterior, and lateral. The posterior approach often involves involves removal of posterior aspects of the spine to allow room for surgeon tools and minimize disruption of the nerve roots. This usually involves involves partial (-otomy) or complete (-ectomy) removal of the lamina (lamin-) and/or facets (facet-). While lateral and anterior approaches can avoid the need to
modify the lamina and/or facets, other critical anatomy must be moved or carefully avoided (ex. Aorta on the anterior face of the vertebrae). Therefore, a surgeon must carefully coordinate his/her approach and implant based on many factors such as the problem being addressed (taking into account the condition’s severity), other internal issues of the patient (such as position restrictions), and surgeon’s need for visibility [3].

The surgeon’s decision to treat a spinal injury or defect by installing an implant is a complicated decision. Generally, disc bulges and osteophytes can be corrected using a minimally invasive percutaneous (through the skin) laproscopic (using a camera through a tube) surgery without any hardware installation. In this surgery, an attempt is made to keep as much anatomy intact as possible. However, sections of lamina must often be removed in order to remove the extra bone and/or disc material (Figure 1-6). A posterior approach is used due to the primary location of disc herniation being at the lateroposterior circumference of the disc. If too much of the disc or lamina must be removed, then implants must be used to provide the required stability, protection, or position. Also, conditions such as a deformed vertebrae or degenerated disc can sometimes be addressed by injecting a material into the affected component, although long term success of these procedures is still under investigation. As with most other spine defects, a surgeon will generally use an implant if the deformed spinal element is judged to be severe. There are many different options of implants, and these are typically grouped as “non-fusion” or “fusion” devices. Non-fusion devices allow the spine to still move, and they are designed to only augment existing spine functions.

There are a range of technical and regulatory issues surrounding non-fusion devices that
make these devices uncommon in the United States [2]. Thus, only fusion devices will be considered for the purpose of this thesis.

Fusion devices take many different forms because they are currently used to address almost all conditions once the problem has become serious enough in an individual. One or more of four main fusion devices can be used (Figure 1-7). The only device not shown in Figure 1-7 is a facet screw, which is a screw that goes through a facet joint (from the IAP of the superior vertabra to the SAP of the inferior vertebra) and immobilizes it. Particular implants are usually only used at a specific location on the spine. Pedicle screws are used on the posterior elements, interbody fusion cages are put between vertebrae after the disc is removed, and plates are primarily used on the anterior or lateral surfaces of the vertebral body. The specific implant used may differ

| Figure 1-6: Laminectomy and discectomy depiction [2] | Figure 1-7: Typical spinal fusion implant options [2] |
significantly from one surgery to another depending on the surgeon’s preference, even
if the approach, end location, and particular outcome goal is identical. In many severely
degenerated or injured cases, surgeons prefer to combine implant types such that two
of the three spine columns (anterior part of the vertebral body, posterior part of the
vertebral body, posterior elements) are made rigid. For instance, a surgeon would
implant a plate to the anterior vertebral body face and pedicle screws in the posterior
elements, however he/she would not use both pedicle screws and facet screws as these
are both in the posterior element column [3].

In addition, the section of the spine being operated on influences the type of
fusion device used. As of 2010, 33% (182,000) of spine implant surgeries were cervical
fusions and 58% (271,000) were lumbar fusions. Cervical fusions on average involved 2
levels fused, with 74% being a combination of plate and cage, 16% just a plate, 6% just a
cage, and 4% using pedicle screws. Lumbar fusions averaged 2 levels fused, however
their mix was 53% using cage and pedicle screws, 34% pedicle screws only, 7% cages
only, and 6% utilizing other types of constructs. Other constructs can include spinous
process plates and/or use bone material [7]. The device that this paper addresses is only
used in pedicle screw implants, so the following information will focus on this type of
spinal fusion.

There is a large market for pedicle screws and there are numerous designs and
manufacturers of this type of fusion device. The spine market in the U.S. is $6.8 billion,
and 34% of this market (over $2 billion) involves pedicle screw systems [7]. These
systems are usually placed bilaterally and the system on each side is typically composed
of a minimum of one stabilizing rod, a pedicle screw for each vertebra, and a set screw at each pedicle screw to secure the stabilizing rod (Figure: 1-8). Sometimes, the securing feature at the head of the pedicle screw is a separate connector. Each company has a slightly different design of the components illustrated in Figure 1-7, but generally all pedicle screw constructs require a set screw to be tightened to a specific torque which ensures a proper connection between the pedicle screw and stabilizing rod, and thus a rigid fixation. There are three ways in which this torque is obtained. In one method, a surgeon will tighten the set screw by hand until he/she judges that the proper tightness has been achieved. In a second method, a torque wrench is used that either measures the torque and requires the surgeon to read this torque off the instrument or provides an audible sound and undergoes a rotational slip. In the third method, the head that is gripped by the tightening tool shears off the threaded body once the proper torque has

Figure 1-8: Standard spinal fusion implant utilizing break off set screws. The red boxed portion is the focus of this thesis.
been achieved [8]. Regardless of the method, surgeons usually use hand tools similar to Figure 1-9. This thesis specifically addresses the forces that are released on the spine during the set screw tightening operation using the third method, describing their origin and proposing ways of reducing them.

1.3 Identification/Focusing of the Problem (Problem Background)

In past experiments and discussions with surgeons, two behaviors relating to set screw break off were determined. First, surgeons prefer the second and third set screw tightening methods, since each provides confirmational feedback by a physical event that indicates the correct torque has been achieved. This feedback benefits the patient because it reduces the possibility of tightening error, and it benefits the surgeon because it shifts tightening error liability from the surgeon to the company that made the devices in the utilized tightening method. More detail on the importance of tightening torque and a validation study can be seen in APPENDIX B. Second, both methods release approximately the same maximum magnitude of g-force shock (1 g-force = 9.807 m/s²), as confirmed by accelerometers tangentially attached to pedicle screw posts (Figure 1-10).

Discussions with surgeons communicated the problems caused by this shock. The harm to the patient being operated on can include the pedicle screw breaking through the side of the vertebral body, a weaker hold of the bone-pedicle screw interface, and the tools slipping out of the surgeon’s hand. Also, repeated shock to
surgeons during each set screw break off (SSBO) event can cause their hands to fatigue more quickly and may cause surgeons’ hands to develop degenerative changes earlier.

A previous study quantitatively analyzed the magnitude of g-forces acting on an implant during SSBO [9]. SSBO imparts immense shock onto a patient and surgeon due to the energy released during the shear failure when the screw head separates from the threaded portion. There are no known studies that report a specific shock magnitude that, if exceeded, will cause breakdown or damage of tissues. However, there are some ISO standards that give eight hour energy equivalence calculations and limits, and these limits may be refined in the future due to studies indicating the incompleteness of the standards [10]. Since the degree of shock that causes damage is unknown, it is the researchers’ assertion that shock should be minimized in sensitive procedures so that the likelihood of failure is reduced in body-device constructs from reconstructive surgery. Bench top results of the shock were obtained through accelerometers affixed
to a fixture that was attached to pedicle screws just above the surface of a saw bone lumbar model and just below the set screw receptor head (Figure 1-10). The purpose of the fixture was to ensure that the accelerometer was oriented so that the tangential acceleration was always being measured.

The accelerometer readings during each SSBO event recorded g-force shock anywhere from 200-800g (Figure 1-11A and B). The energy released during each event was actually the same since each screw had the same cross section and material at the point of failure. The researchers hypothesized that the variation was due to a multitude of human factors such as how tightly the tool was held during the event and if the tool was held against the spinal structure after failure or allowed to slip. Furthermore, this immense shock at the location of SSBO quickly dissipated and the maximum shock at adjacent pedicle screws was greatly diminished.

Although there are well developed equations for calculating the critical failure torque of an axisymmetric tube, no similar equations can be found that calculates the

Figure 1-11: Two different test runs showing: (A) Oscilloscope output of the initial signal recorded, (B) Excel graph conversion of the .csv data signal to g-force in Excel [9]
released shock upon failure of the same tube. Therefore, computer simulations must be validated with physical tests to determine how to reduce this shock. In the following section, the hypothesis will describe how the researchers planned on mitigating the SSBO shock and how computer simulations were used to indicate a design change’s effect on the released shock.
CHAPTER II

HYPOTHESIS AND GOVERNING THEORY

2.1 Hypothesis

The goal of this thesis was to reduce the maximum g-force felt by all components in the implant installation system by making geometric changes to the shear failure region. As the curves below illustrate, the energy released during SSBO causes reverberations on connected structures, and the researchers’ intent is to use geometry to reduce the maximum g-force (Figure 2-1). The total area under the curve profile is representative of the total kinetic energy released when the set screws are sheared off. This area should remain constant or, preferably, decrease in structures that reduce the maximum g-force. The shock curve profile may be continuous or occur in observable spikes at intermittent times within the time period of shearing failure. Regardless of the shape of

![Figure 2-1](image1.png)

Figure 2-1: (A) Current g-force and profile curve of set screw break off (B) Desired reduction of maximum g-force.
the profile curve or the total area, the end goal is to reduce the maximum g-force. Any design that reduces the maximum shock during shear failure will be considered successful.

**Hypothesis:** Geometric changes in the set screw tightening structure (i.e. device) can influence the energy release behavior of a crack such that the maximum shock released is reduced.

Implied in this hypothesis are the assumptions that:

1. The maximum torque/moment able to be withstood by each device design is the same, and this maximum torque/moment is controllable.
2. Each device design is made of the same material with identical bulk and surface properties.

The work in this thesis provided the groundwork necessary to identify how changes in energy values and structural behavior correlate with a reduction in shock. Specifically, the variable trends initially considered critical in determining favorable relative performance of different geometries are:

1. The ratio of maximum elastic energy to maximum plastic dissipation energy should decrease
2. There should be an increase in the device’s rotational movement before complete shear occurs.
Both of these variables indicate plastic dissipation energy is playing an increasing role in the fracture event. In the researchers’ simplified energy balance, external work performed on the device must primarily be converted into either plastic dissipation energy or recoverable elastic strain energy that manifests itself as shock. Therefore, as the elastic strain energy decreases relative to plastic dissipation energy, the shock should also decrease. So, a ratio of maximum recoverable elastic strain energy to maximum plastic dissipation energy was used to evaluate whether a specific geometry would release more or less shock. The second value demonstrates a physical condition that is advantageous to allowing greater strain. From simple geometry, as the device undergoes increased rotation before failure, then more elements must be experiencing deformation assuming that the plastic strain limit of each element is identical. As more elements experience deformation, then more energy is dissipated plastically.

2.2 Theory

The initial concepts use plastic deformation as presented in fracture mechanics to absorb the energy that creates the shock, but other methods may be discovered and utilized which also serve to reduce the shock. Plastic deformation for most materials is caused when the structure undergoes so much stress that the bonds between individual atoms break and reform to an adjacent atom. Plastic deformation happens in the direction that these atoms move. Essentially, this concept relies on the material’s toughness, or energy absorption potential before failure. One can see a material’s
toughness by observing the area under the engineering stress-strain curve (Figure 2-2) [14].

2.2.1 Linear Elastic Fracture Mechanics

When analyzing the current SSBO method, the sudden release of energy, as illustrated in Figure 2-1A by the high acceleration peak at the start of the signal and the perceptibly instantaneous shear failure when the surgeon achieves the designed torque, indicates crack initiation and propagation is not slow. This SSBO crack speed, together with the set screw geometry, greatly influence the failure behavior of the set screw, and the normally ductile material response of a metal becomes primarily a brittle rupture. The concepts presented in this section will guide geometric suggestions for future set screw geometries in Section 4.2.

Based on the theory of Linear Elastic Fracture Mechanics (LEFM), where the stress at the moving crack tip is considered linear elastic with two-dimensional stress, the crack undergoes a rapid, brittle propagation through the structure’s thickness when it exceeds a “critical stress intensity” [15]:

\[
K = \frac{C\sigma\sqrt{\pi a}}{K_c} \geq K_c
\]
This equation’s dependence on the specific crack separation mode is presented in Figure 2-3.
At this critical stress intensity, the energy release rate \((G=\text{energy per unit length along the crack tip})\) of the separating material (potential energy release of the elastic strain) is greater than the crack resistance. The excess of energy becomes kinetic energy which controls the crack tip speed through the material, with the total kinetic energy equal to \([16]\):

\[
E_{\text{kin}} = (G - R)dA
\]

- \(E_{\text{kin}}\): kinetic energy
- \(G\): energy release rate
- \(R\): crack resistance force

Assuming:

1. Constant stress during crack propagation
2. \(G\) independent of crack speed
3. Constant \(R\)

With the total energy release of all three modes:

\[
G = G_I + G_{II} + G_{III} = \frac{1 - \nu^2}{E} \left( K_I^2 + K_{II}^2 + \frac{K_{III}^2}{1 - \nu^2} \right)
\]

Composed of:

\[
G = \left( \frac{1 - \nu^2}{E} \right) K^2 \quad \text{(Plane Strain)}
\]

\[
G = K^2/E \quad \text{(Plane Stress)}
\]
v: Poisson's Ratio

K: Stress intensity factor

E: Modulus of elasticity

Crack resistance and propagation forces are actually a complex combination of a variety of forces, depending on things such as environment, material, and crack/structure geometry. The preceding equations are only a rough guide since, for example, crack resistance does not remain constant as crack growth rate is controlled by crack size (Figure 2-4) along with other factors. The primary groupings of these forces that influence crack growth are intrinsic and extrinsic as shown in Figure 2-5. Intrinsic forces stimulate crack growth and are dependent on the material properties, while the extrinsic forces hinder propagation and are primarily a function of crack size/geometry. Ductile materials such as metals predominantly toughen intrinsically, whereas brittle materials toughen through extrinsic forces [17]. Material and process variabilities such as strain rate [18], strain hardening, surface irregularities, surface processing (ex. Shot
peening, electro polishing), and grain structure all affect a part’s macroscopic behavior through their influence on the microscopic intrinsic and extrinsic properties.

This thesis does not intend to fully analyze all of the complex crack behaviors and underlying conditions that occur in the failure region. However, it will attempt to reduce the amount of work done that is dissipated as kinetic energy (seen and felt as shock) through varying only geometry where the percent of plastic to elastic energy will be analyzed at failure conditions (see examples in APPENDIX C of energy output graphs). The preceding description was included to give qualitative reasons for the geometry selections of following concepts. Control variables within each concept will be the material and surrounding environment, while the experimental variables will be the part geometry (including all designed geometric irregularities such as holes and bulges). A sensitivity analysis on strain rate will be conducted. However, limits on process and quality variables such as grain structure, allowed inclusions, and surface smoothness are outside the scope of this thesis and will be determined in the production design by a professional engineer specializing in this field.

2.2.2 Elastic-Plastic Fracture Mechanics

Ductile material failure is approximated with varying accuracy by many different models. Engineers have been trying to develop more accurate models for many years which adequately predict macroscopic behavior after large strains occur. These strains are usually due to microscopic material gaps that change shape, expand, and combine as the load grows. Each of these gaps have the extrinsic and intrinsic influence discussed
in Section 2.2.1. The combined effect of these changing voids reduces the strength of the bulk material until a critical point is reached and fracture occurs. Models that attempt to model this ductile behavior can be broken down into two groups. Phenomenological models build equations around the microscopic behavior of material void changes, using things such as “void volume fraction” to predict macroscopic material/structural response. Empirical models fit equations to data obtained from mechanical tests, with the constants used in a particular material’s fit being characteristic of that material. These equations are independent of the microscopic voids, and the characteristic constants for a material are different depending on the empirical model being considered. In this project, an empirical model is used. The choice of model comes down to which model can give adequate results with the least complexity [21].

Significant unrecoverable strain due to stress in a material is called plasticity. The plasticity conditions must be defined correctly in order for the material in a modeled structure to deform and fracture correctly [41]. Plasticity of the model’s material can be defined in numerous ways. The simplest material definition is perfectly plastic, where yield stress does not change based on the plastic strain (i.e. horizontal line on the stress-strain curve). A more accurate definition includes the effects of isotropic hardening/softening, where the stress either increases or decreases upon increased plastic yielding. Further complexity can be added to the definition if strain rates and temperatures are expected to vary significantly enough to affect the yielding stress-strain relationships (such as fast deformations in most metals). In order to account for
strain rate and temperature dependencies, the Johnson-Cook hardening can be used. Rate dependent yielding can be defined in other ways as well [20]. However, the researchers in this experiment intend for their device to be used in room temperature and at strain rates less than 0.1, which is insignificant to the stress strain curve [41]. Also, kinematic hardening definitions are not considered in this research since it is most appropriate for cyclic loading. Therefore, quasi-static stress-strain curve definitions are deemed adequate for current experiments. However, if later validation tests of the device show unexplained variability, additional material definitions such as these may need to be included [20].

In classical metal plasticity models, associated plastic flow is used in conjunction with Mises yield surfaces for isotropic (the same in all directions) yielding and Hill yield surfaces for anisotropic (directionally sensitive) yielding. There are specially derived models that indicate failure conditions of anisotropic materials [40], however the work in this thesis did not attempt to integrate the conditions into a computer simulation, thus the work done assumed isotropic yielding. A significant amount of support has been given to the idea that ductile fracture occurs at the location of maximum shear stress. So assuming shear stress is the central influence, the failure indicator equation takes the form

\[ \tau_{max} = \max\left\{ \frac{\sigma_1 - \sigma_2}{2}, \frac{\sigma_2 - \sigma_3}{2}, \frac{\sigma_1 - \sigma_3}{2} \right\} \]

The \( \sigma_1, \sigma_2, \) and \( \sigma_3 \) are the standard three dimensional principal stress components [22]. This is the form of maximum shear using the Tresca yield envelope which is easiest for
calculating manually. However, computer codes usually utilize the Von Mises yield criteria, which is based on distortional energy. This is because its yield envelope can be described in a single equation, as there are no incongruences [24]. This equation for pure shear yielding is

\[ k = \frac{\sigma_y}{\sqrt{3}} = \frac{1}{\sqrt{6}} \left( (\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2 \right) \]

The results of these equations are close, although the Von Mises yield envelope is usually slightly bigger. A three dimensional comparison is most accurate, but a two dimensional comparison is easiest to visualize (Figure 2-6). For an element, any combination of stresses outside these envelopes will cause yielding and failure [25].

![Figure 2-6: Three dimensional and two dimensional comparisons of Tresca and Von Mises failure envelope [25.]](image)

These yield surface models return a stress based on the local strain as defined by the aforementioned plasticity model, and they assume pressure stress has no influence on the material’s yielding. Although this assumption is accurate for most non-voided metals, it breaks down when there is high triaxial tension such as stress fields at crack
tips when voids are growing (triaxiality will be discussed in a following paragraph).

Associated flow means that material deforms plastically in the direction of the primary stress vector. Although this is accurate in the majority of situations involving metal, necking metal sheets have strains with a normal component to the primary yield surface. However, so long as the localized material behavior is not significant to the study, or failure criteria in terms of a strain limit adequately describes the materials behavior, then this inaccuracy is negligible [20].

Damage and failure modeling requirements are highly dependent on the specific model type that is utilized. However, a damage initiation of some type (usually defined in terms of some form of stress or strain) and a stiffness reduction (defined in terms of a function of energy or element face displacement) must be specified in all of these models [20]. There is much discussion and study on which model is truly adequate for a given situation. As previously shown, the geometry of the proposed device is relatively thin, making the behavior primarily plane stress. Assuming associated flow, the advantage of plane stress conditions is that changes in stress are directly comparable with changes in strain, therefore fracture models can easily be compared regardless of whether they are based on stresses or strains. There are many studies that compare popular ductile failure models, however one of particular interest to the researchers is a study by Wierzbicki et al of seven models in plane stress conditions. All of the models they analyzed will not be described, but as one can see from the Figure 2-7, the models that most closely predict failure loci strain in all triaxial conditions are the Xue-Wierzbicki, CrachFEM (shear), and $\tau_{\text{max}}=\text{constant}$ models. The researchers have utilized
the maximum shear stress model due to the ease of implementation, the availability of
data for different materials, and the relative accuracy of the model throughout the
range of triaxilities [22]. Failure loci and stress triaxiality will be described next so that
the reader can appreciate the graph’s significance.

Figure 2-7: Comparison of multiple failure models in the strain-triaxiality plane [22]

Triaxiality is a non-dimensional value that indicates the influence that a
hydrostatic pressure has on the failure strain in a material. Triaxiality ($\eta$) is the ratio of
the mathematical mean of primary stresses to the equivalent Von Mises stress

$$\eta = \frac{\sigma_m}{\sigma_{eq}} = \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3) \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2]}$$
As stated when discussing classical metal plasticity, if there is high triaxiality in a geometry then it becomes important to consider. However, pure shear is zero triaxiality [23], and although the goal is to have geometry to influence the shear behavior, it is unknown to what extent these geometry changes will affect triaxiality since the device must still undergo primarily shear failure. However, there are studies that indicate, higher triaxiality inhibits void growth and can slow fracture [28]. As shown in Figure 2-7, the maximum shear stress can adequately indicate the failure strain through a range of triaxiality values. The failure locus is the line illustrated by each curve illustrating the strain that will cause failure at a particular value of triaxiality [22].

There is another term which also has been shown to influence material failure behavior called the third-stress invariant, or Lode angle. A significant amount of work has been done in the field of elastic-plastic fracture mechanics trying to easily integrate the concept of Lode angle into models that already consider J2 plasticity and triaxiality [23, 37, 38]. However, this thesis does not consider the influence of Lode angle, as initial research did not reveal a method to easily integrate this into computer models that utilized J2 plasticity and triaxiality.

2.3 Strain Rate and Processing Importance

Strain rate is another important component affecting the plasticity of a structure. Viscoplasticity can be understood through dislocation mechanics, which is highly dependent on the whether the material has a face-centered-cubic (FCC) or body-centered-cubic (BCC) structure. The physical behavior is void nucleation and growth in
shear bands. This is an entirely complicated subject on its own, and will not be discussed in detail in this thesis. However, the significance of the idea can best be summed up by Figure 2-8. In BCC structures such as the beta phase of Ti-6Al-4V, slower strain rates result in higher strains and thereby greater plastic dissipation of energy. Therefore, it is optimal for there to be an increase in time between the start of yielding and final fracture [18]. However, this disadvantage of lower strain rates is that the strength of the material in a structure is highly dependent on bulk material composition and processing conditions such as heat treatments [27]. These additional variables are outside the scope of this thesis, and all simulation and experimental testing were considered quasi-static and independent of these effects.

![Figure 2-8: Polycrystalline tantalum (BCC) material structure behavior showing strain increases as strain rate decreases [18].](image-url)
2.4 Material Selection

Material properties greatly influence the results of testing. The particular material that the set screws are made of is usually Ti-6Al-4V. However, due to relatively high titanium raw material and machining costs, the researchers desired to conduct initial testing on another verified material model of a ductile metal, Al5083-H116 [26] (See APPENDIX D for values). The researchers have made the assumption that geometric influence on the plastic dissipation of one ductile metal will be similar to another ductile metal, and the aluminum test will serve as a proof of concept for the hypothesis and a validation of the computer model. Once the geometry-shock relationship is better understood, then future studies can incorporate a Ti-6Al-4V material model [21, 27, 28].

2.5 Proposed Initial Geometries

In an ideal embodiment, deformable geometry would be designed directly into each set screw to initiate and control crack propagation while reducing shock. By designing the deformable section into each set screw, a surgeon will not have to alter normal surgical procedure. If a deformable element was designed into the tool, then the surgeon would have to replace that deformable tool component after each set screw was tightened. For this reason, set screw tightening method three, set screw break off (SSBO), was chosen for investigation.

The first geometries selected for investigation were simple axisymmetric variations of the groove that currently separates the hex head from the threaded body on a break off set screw. The radius of the groove fillets and the length of the bottom of
the groove are the critical variables, while the minimum and maximum wall thickness remain constant (0.0004in and 0.0008in, respectively). These variables and constants are clearly visible later in Figure 3-2.

2.6 Testing Outline

There are a number of steps required to show that the geometry influences the energy release behavior of shear failure and that this energy release behavior correlates
with the shock released. From this correlation, the particular geometric features that will reduce shear failure shock in the case of the set screw structure can be determined.

This specific thesis completely addressed the 2D axisymmetric modeling, utilizing an element (CGAX3, see Section 3.1) that also allows twist about the axis. It also began the steps that are needed to perform prototype testing and 3D cyclic symmetric modeling. These are only the first few steps upon which all future steps will be dependent in order to create optimal set screw geometry. A 2D-axisymmetric-with-twist parametric computer simulation study using the validated material model of Al5083-H116 will be described next. Then, the setup for future validation testing will be described in detail.

The purpose of the future validation testing will be to determine how the variables identified in the computer model correlate with the shock released upon shear failure. Table 2-1 illustrates all present and future steps required for a successful geometric design of a reduced shock set screw.

<table>
<thead>
<tr>
<th>Table 2-1: Testing Order</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Al5083-H116</td>
</tr>
<tr>
<td>2D-Axisymmetric-with-twist modeling</td>
</tr>
<tr>
<td>2. Ti-6Al-4V</td>
</tr>
<tr>
<td>2D-Axisymmetric-with-twist modeling</td>
</tr>
</tbody>
</table>
CHAPTER III
THEORY TO COMPUTER MODEL

Abaqus 6.10 was used for computer simulations and testing to compare how geometric changes affected the plasticity of the device through monitoring of the previously mentioned critical variables. This program was selected from among others, such as LS-Dyna, based on the advice of university subject matter experts.

The critical variables mentioned in Section 2.1 of this thesis can all be easily tracked in the Abaqus modeling software. The “Energy Balance” section 1.5.5 of the Abaqus Theory Manual[29] explains the different components of energy that were tracked for the whole model. These were “field variables,” with ALLPD referring to the plastically dissipated energy and ALLSE referring to the recoverable elastic strain energy. The third critical variable was rotational strain before failure. The rotation of each test was constant and took exactly one time increment to reach the final rotation location specified (at a constant velocity). The law of conservation of energy states that energy can neither be created nor destroyed. In addition, since basic material laws state that plastic strain is irreversible deformation of a material, then the amount of plastically dissipated energy accumulated can never be decreased. However, since the Abaqus model being utilized deletes elements when they have surpassed the strain limit defined by the material, so the plastically dissipated energy contained in each deleted element is
also deleted. Therefore, since every shear model run showed that the geometry sheared all the way through, then the point at which fracture completed is also the point at which the last element was deleted and the last decrease in ALLPD was observed.

Modeling decisions were made based on recommendations given in “Classical Metal Plasticity” (Abaqus 6.10 Analysis User’s Manual). Progressive damage and failure models in Abaqus are able to model both quasi-static and dynamic situations, and the researchers have made the initial assumption that the manually applied strain rates in the set-screw break off are not high enough to significantly affect the failure stress, thus quasi-static modeling is acceptable. This assumption is due to the Ti 6Al-4V yield strength used in the initial calculations (APPENDIX B), which corresponds to the actual measured and designed for break off torque. This yield strength is the quasi-static yield strength, as higher strain rates result in higher yield strength. The damage model utilized was ductile damage (i.e. failure strain as a function of triaxiality) since other papers judge the validity of material models based on their ability to correctly determine the failure strain throughout all loading conditions (as identified by triaxiality values) [22]. Therefore, since this damage model is defined by maximum equivalent strain at a particular triaxiality, equivalent plastic strain (PEEQ) is used to visualize plastic deformation (see Abaqus/CAE User’s Manual 12.9.3 “Defining Damage” [29]).

A mesh size convergence analysis was performed on the model before running the trials. Unfortunately, due to the model requiring element deletion in order to show crack propagation, the maximum energy values are heavily dependent on the mesh size.
Therefore, the value of the Von Mises Stress at a standard location was used to determine mesh size convergence (Figure 3-1A). Table 3-1 shows the mesh sizes and the associated damage evolution used in this analysis. Due to how small the structure is, the mesh size can be deceiving. Figures 3-1A & C show the largest and smallest element

Figure 3-1: Mesh sensitivity graph showing how the maximum Mises Stress converges as the mesh is refined. The labeled locations along the x-axis correspond to the stress profile pictures. Mesh size 4e-05 (B) was the best compromise between computational cost and accuracy.
sizes used in this study, respectively. Any mesh size smaller caused Abaqus to error out after 12 hours, and any larger mesh size did not provide an adequate number of elements across the failure region. Figure 3-1B shows that the mesh size chosen (4E-005 in) is in the region where the Mises Stress converges.

<table>
<thead>
<tr>
<th>Mesh Size</th>
<th>Linear Damage Evolution</th>
<th>Maximum Stress at constant element (psi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6E-005</td>
<td>1.5E-005</td>
<td>28583</td>
</tr>
<tr>
<td>5E-005</td>
<td>1.25E-005</td>
<td>27879</td>
</tr>
<tr>
<td><strong>4E-005</strong></td>
<td><strong>1.0E-005</strong></td>
<td><strong>27688.2</strong></td>
</tr>
<tr>
<td>3E-005</td>
<td>0.75E-005</td>
<td>27506.4</td>
</tr>
<tr>
<td>2E-005</td>
<td>0.5E-005</td>
<td>(errors out)</td>
</tr>
</tbody>
</table>

**Table 3-1: Mesh Convergence Values**

3.1 2D-Axisymmetric-With-Twist Setup

Detailed information on how the 2D-axisymmetric-with-twist model was created can be found in APPENDIX E. In summary, the general method involves creating a 2D axisymmetric sketch of the part, creating a material with damage conditions, assigning the material to the part, creating a mesh on the part using CGAX3 element that allows twist, assigning node regions to which boundary/rotational conditions were applied (Figure 2-9), and setting the convergence behavior. The element type utilized, CGAX3, was particularly important to the entire model and simulations because it provided the 2D model an additional degree of freedom. Traditional 2D axisymmetric analysis only allows in-plane movement [36]. Per Abaqus Analysis User’s Manual 25.1.6 “Axisymmetric solid element library” [29], the element type CGAX3 also allows elements the freedom to twist about the axis. Movement, moment, and stresses due to torsion
on the modeled structure cannot be obtained without this additional degree of
freedom. The overall length of the modeled section remains the same; however the
fillet radius and width at the bottom of the groove are varied. The fillet radii values are
.0002in, .0004in, and .0008in with the length of the bottom section ranging from 0.0in
(i.e. simple semicircle groove) to .000in in increments of .0001in. Constants are the
minimum and maximum wall thicknesses and total height of the model. A parametric
study was conducted with every combination of these variables (i.e. 12 total models) to
show how these geometric changes affected plasticity. The two extreme cases of these
models are illustrated in Figure 2-9 for clarity. The critical outputs of these two models
and all the models between are discussed in the following section.

3.2 Results: Aluminum–Deleted Elements (Triaxial Strain Criteria)

The general trend of geometry to plastic dissipation can best be seen by the
pivot table graph in Figure 3-2. It also shows the two geometries that have the lowest
and highest values of plastic dissipation. As expected, due to what is known about stress
risers, the smaller groove with the smallest radius has the lowest plastic dissipation and
the widest groove with the largest radius has the greatest plastic dissipation. The
relatively sharp groove in L0R2 has only a small area of influence, while the widest
groove of L3R8 spans, and therefore affects, much more material. Since more material is
influenced by the stress riders of L3R8, there will be more elements experiencing plastic
deformation.
Figure 3-2: Pivot Graph results of the parametric Abaqus model testing showing (from top to bottom) Abaqus variables ALLPD, ALLSE, ALLWK.
Figure 3-3: Graphs comparing the output of all trials showing that (from left to right):

(A) The maximum moment remains relatively constant while increasing in duration

(B) The plastic dissipation of energy and total displacement (strain) increases significantly. The decreasing plastic dissipation energy component is an inaccurate artifact caused by the element deletion that is used to model the fracture.
Despite the change in stress concentrations, the maximum moment required to shear each cross section experiences negligible variation (Figure 3-3A). Therefore, since the minimum cross section did not change, this moment remains primarily a function of the thinnest cross section of the groove as predicted by the equations in APPENDIX B. However, the increased amount of displacement over which the moment must be applied causes the external work done on the structure to increase, since work is force times displacement. Figure 3-3A illustrates the relative shapes of these moment curves. Additionally, whereas Figure 3-2 shows the relative maximums of the plastic dissipation (ALLPD), Figure 3-3B shows the curve profile for every trial (graphs for the comparison of external work and recoverable elastic energy are located in APPENDIX C). The decreasing plastic dissipation energy (Figure 3-3B and ALLPD term in Figure 3-4A & B) is an inaccurate artifact due to element deletion. When the elements are deleted after they have reached the complete failure criteria defined by the material model, the energy terms associated with those elements are also deleted. This also shows that the deformation occurs at a slower rate and over a longer time when comparing the wider L3R8 groove to the narrow L0R2 groove, which is useful because more elongation means more plastic dissipation as discussed in the theory. The displacement at which the structure completely fails for the L0R2 model is 0.54 of the rotation cycle, and the displacement at which the structure completely fails for the L3R8 model is 0.94 of the rotation cycle. This is a 74% increase in rotation before failure. The researchers consider
Figure 3-4: Graphical results of the energy and moment values for the plastic dissipation shown in Figure 3-2 (A and B respectively). The thick red line shows the point where first yield occurs, the red circle at the peak shows where the first element is deleted, and the second red circle shows where the final element is deleted.
this entire increase to be due to plastic strain, since only 3.09E-7 of the rotation cycle is completely elastic for the L0R2 model and .0142 is elastic for L3R8. Using the geometric extremes, the behavior of the structure in relation to the curve profiles is shown in Figure 3-4. It should be noted that complete failure was defined by the researchers to be the moment when the last element was deleted, and this point is not the same moment when the ALLPD curve becomes level.

Although the recoverable elastic energy is slightly higher for L3R8 vs. L0R2 (Figure 3-2), this increase is negligible considering the overall magnitude of the work done as shown by the output graphs of L0R2 and L3R8 in Figure 3-4. As discussed in section 2.1, this is the reason why the ratio of recoverable elastic strain to plastic energy dissipation was used. Numerically, the ALLSE/ALLPD ratio for the smallest groove (L0R2) is 0.0274, and the same ratio for the largest groove (L3R8) is 0.0174. This is over a 36% decrease in the ratio, showing that the geometric structure influences the plastic behavior of a material. Figures 3-5 through 3-8 show the stress and equivalent strain state for each important step in the shear failure cycle for the two extreme geometries. The time that the step occurs for each geometry is in the caption. Although the stress and strain color ranges are the same for each geometry within a step, each step has a different range so as to show the stress or strain profile throughout the structure at that point in time. Furthermore, a numerical summary of results is in Table 3-2.
In summary, the results obtained support the hypothesis. The two trends thought to indicate reduced shock where observed while adhering to the constraints. Geometric changes to the groove profile around the outer circumference of a tube caused plasticity to increase, as indicated by a decrease in the ratio of elastic energy to plastic dissipation energy and an increase in radians revolved before complete shear failure. Through all geometries, the tube was a constant material (Al-5083-H116) and the maximum moment before failure remained approximately constant.

<table>
<thead>
<tr>
<th></th>
<th>Maximum Elastic energy (ALLSE)</th>
<th>Maximum Plastic dissipation energy (ALLPD)</th>
<th>ALLSE/ALLPD</th>
<th>Total work done (ALLWK)</th>
<th>Rotation before complete shear failure</th>
<th>Maximum moment for shear failure</th>
</tr>
</thead>
<tbody>
<tr>
<td>L0R2</td>
<td>2.442E-06</td>
<td>8.902E-05</td>
<td>.0274</td>
<td>7.430E-05</td>
<td>.54*.3 radians = 9.28°</td>
<td>6.307E-04</td>
</tr>
<tr>
<td>L3R8</td>
<td>2.796E-06</td>
<td>16.073E-05</td>
<td>.0174</td>
<td>14.352E-05</td>
<td>.94*.3 radians = 16.16°</td>
<td>6.293E-04</td>
</tr>
</tbody>
</table>

|        | 36% Decrease                  | 74% Decrease                             | Approximately the same |

In summary, the results obtained support the hypothesis. The two trends thought to indicate reduced shock where observed while adhering to the constraints. Geometric changes to the groove profile around the outer circumference of a tube caused plasticity to increase, as indicated by a decrease in the ratio of elastic energy to plastic dissipation energy and an increase in radians revolved before complete shear failure. Through all geometries, the tube was a constant material (Al-5083-H116) and the maximum moment before failure remained approximately constant.

Table 3-2: Comparison of the two geometric extremes L0R2 and L3R8
Figure 3-5: L0R2 (A: 3.09E-7 of the rotation cycle) vs. L3R8 (B: 0.0142 of the rotation cycle) at first yield illustrating the locations of first plastic deformation and associated stress field.

Figure 3-6: L0R2 (A: 0.2428 of the rotation cycle) vs. L3R8 (B: 0.5542 of the rotation cycle) at maximum moment showing plastic deformation and associated stress field.
Figure 3-7: L0R2 (A: 0.4516 of the rotation cycle) vs. L3R8 (B: 0.8624 of the rotation cycle) just before first element deletion illustrating the plastic deformation and associated stress field.

Figure 3-8: L0R2 (A: 0.594 of the rotation cycle) vs. L3R8 (B: 0.9399 of the rotation cycle) just before last element deletion illustrating the plastic deformation and associated stress field.
4.1 3D Setup

An initial 3D model was also created for the two extreme axisymmetric geometries L0R2 and L0R8. The purpose of this was twofold. First, later set screw designs may not be axisymmetric, so 3D models are needed to adequately represent more complex geometry. Second, certain features are available in Abaqus for 3D models that are not available for axisymmetric models, primarily the ability to deactivate element removal.

Non-axisymmetric designs of future geometries may have other geometric irregularities at evenly spaced sections of the tube wall. The entire tube, therefore, cannot be described by a single cross section. From an initial run (Figure 4-1A), modeling the entire tube as a 3D structure with a fine enough mesh to cause convergence has too high of a computational cost to be feasible. The rough mesh in this figure took a day to run, but such a large amount of volume was deleted due to damaged elements that the resulting energy output was unrealistic. However, just as Abaqus can apply correct boundary conditions to a cross section in order to simulate the response of axisymmetric geometry, Abaqus can also simulate the effect on a tube using only a 3D model of the repeating geometric structure. This is referred to as cyclic
symmetry (see Abaqus/CAE User’s Manual 15.13.16 “Defining cyclic symmetry” [29]) and instructions on how to implement this can be found in the APPENDIX F. In summary, the nodes on the visible face in Figure 4-1A (i.e. face perpendicular to the groove feature that looks like an “n”) must be tied to the nodes on the opposing (hidden) face such that corresponding nodes have the exact same translations [36]. Since there will be physical tests of the axisymmetric geometries to validate the axisymmetric computer

![Figure 4-1: (A) Shows how the size of elements needed for an entire model to run cause a significant volume to be deleted when the elements are deleted (B) Shows a refined mesh cyclic symmetric model (C) Shows how even deletion of fine 3D elements causes a significant decrease of plastic dissipation energy (ALLPD)](image-url)
models, 3-D cyclic symmetric models of the L0R2 and L3R8 dimensions were created so that the same physical tests could be used to validate these 3D cyclic symmetric models.

As previously explained, fracture of the geometry was simulated by elements being deleted once the defined failure criteria was met. As shown in the Abaqus graph outputs (Figure 3-4), when the elements are deleted from the model, their values are subtracted from the output, making the decrease in ALLPD an inaccurate artifact. Thus, the plastic energy dissipation variable that was vital to comparison ratio in this experiment is not the true maximum. This difference from the true value was not considered significant enough to cause confusion between which geometries created the most disparate results in the axisymmetric simulations. However, more complex geometries and crack paths may confound the results of future 3D simulations. The advantage of 3D elements, though, is that when defining their mesh element type, they can be manipulated to remain in the model after reaching the failure criteria[29]. In this way, the true maximum ALLPD can be determined since elements and their ALLPD values are no longer deleted from the model. If future simulations are able to return other results such as vibrations, these may also be more accurate when using 3D vs. axisymmetric elements [36].

4.2 Suggested Future Complex Geometries

The following concepts will utilize the LEFM ideas described in Section 2.2.1. Some of the more important components of LEFM judged important for application to SSBO include:
1. Crack propagation rate increases as the crack grows

2. Crack propagation will become brittle when the growth rate is too fast because resistance (R) is smaller than the energy release rate, resulting in an abundance of kinetic energy release (shock)

3. Metals primarily toughen due to intrinsic crack initiation forces

In the first embodiment (Figure 4-2), the grooved portion of the set screw where shearing occurs has constant dimension ridges and constant dimension holes alternating equidistance along the circumference in a cyclic symmetric manner. A study was found showing that tubes with two holes have more ductility before failure than tubes with one hole [39]. The holes are a geometric stress riser that initiates crack propagation. At the intrinsic tip of slow crack propagation, plastic deformation dissipates energy [19]. The multiple holes are intended to increase the amount of energy dissipated through this plastic deformation since each crack initiation site must undergo a certain amount of intrinsic toughening and slow propagation. Therefore, the brittle energy release at the crack tip cannot be the main driver of crack propagation in these geometries, as the critical stress intensity must be reached independently in each section between the holes. The ridges should force additional cracks to require initiation (per crack displacement mode 3) and/or add resistance to reduce crack growth rate, thereby reducing energy release rate. Ideally, this design will cause significant plastic deformation around the entire circumference since the plastic zone will always be
leading the slowly propagating crack, while the process will still seem instantaneous to the surgeon because of the greatly reduced length that each crack must travel.

Although the plastic deformation should absorb most of the potential energy from the initial torque buildup, there still may be a significant kinetic energy release when the resisting cross section provides minimal resistance near the time of complete shear and the crack propagation becomes brittle. However, the summation of resisting cross section area remaining when brittle propagation occurs with the new, complex geometry should be much less than the area of the current SSBO structure that undergoes brittle separation, and thus the energy dissipated as kinetic energy (shock) would be significantly less. In embodiment B (Figure 4-3), constant dimension holes spaced consistently around the circumference would be separated by cross sections of varying thicknesses. No ridges will be used, and the intent of this design is to cause the fractures in each section to occur at different times so as to reduce the initial peak. It is unknown how the placement and geometry of the holes and ridges can affect the shock profile, so various adjustments will be tested as the researchers gain additional insight. Additionally, the design may contain non-constant features if simulated experiments...
indicate that having multiple hole and ridge geometries along the circumference of the screw will allow for more optimal shock reduction behavior.

Ideally the same total cross sectional area would be sheared, which should ensure that the kinetic energy released would be no more than the current SSBO. However, during the design process, it may be determined that the cross sectional area needs to be increased (due to stress risers in the geometry causing earlier failure), which will increase the total energy that must be dissipated and may increase the overall kinetic energy released. Regardless of whether the kinetic energy released stays the same or increases, this energy will be released over a longer time period, resulting in reduced initial shock.

4.3 Experimental Setup

Experimental verification must be conducted to determine how much of an effect the reduction in the elastic energy to plastic energy ratio has on the shock magnitude. The following experimental method is proposed in order to show:

A. Geometric adjustments to a structure can reduce the shock upon fracture, given that the maximum moment, material, and surrounding environment remain constant.

B. To what degree does a reduction in ALLSE/ALLPD energy ratio corresponds to a reduction in the shock.
Results using this experimental setup were not obtained for this thesis, however considerable work did go into the setup so that future experiments could readily be performed.

The set-screw geometry at the point of failure is most like a hollow metal tube, and any proposed geometric adjustments will need to be made to this tube region. The hex head and threaded mode of the screw are considered insignificant due to these features occurring in areas with a much greater wall thickness. Many different complex geometric variations of this tube region must be tested, and the added threaded and hex features are an unnecessary machining step and are not worth the extra cost when machining prototypes. Therefore, a setup was designed that could firmly hold both ends of a simple tube that had the complex geometries machined in the middle of the tube length. In this design setup, one tube end will be fixed, and the other end will be loaded in pure torsion without experiencing any parasitic forces (described later) during the entire loading process. Torque and tangential acceleration of the tube throughout the entire loading process will be measured at a high sample rate and synchronized in order to determine the correlation between shock behavior and structural strength. The sensor setup and data acquisition will be the most important components to the experiment, so a significant amount of time was spent looking for the most optimal data collection equipment.

Both torque and shock need measured in all experiments. The TQM201-56.5 (Figure 4-4A and B) provides the needed range and sensitivity for torque measurements, while giving an analogue output that the researchers could sample at whatever rate is
desired. The sensor measures up to 56.5 N-m with an accuracy of +/- .113 N-m and repeatability of .017 N-m [30]. The particular set screws that will be the control of future experiments are designed to break off at 11 N-m, as shown in APPENDIX B. Therefore,
the accuracy of these future torque measurements will be 1%. To provide an additional torque measurement for verification, a Vernier hand dynamometer (i.e. force sensor) will be used to apply force to a standard spot at the end of the breaker bar (Figure 4-4C). For shock measurements, the same accelerometer setup that was used to measure the g-force shock in the initial experiments [9] will also be used in the new experiments. The accelerometers that are used are Type 8728A500 by Kistler utilizing a power supply/coupler Type 5118B2 (gain set to 1 and high pass filter to .03) that is also by Kistler (Figure 4-5). The voltage inputs and outputs are analogue as one can see from the BNC connectors at the back of the power supply/coupler.

In order to collect the data, it is desirable for one system to collect all of the data so that there is no relative timing error when combining the signals for analysis. The Vernier LabQuest [32] was decided on due to its easy interface with the hand
dynamometer and other inputs, relatively high sample rates for extended intervals of
time, and ease of exporting the data to formats that could be analyzed on a computer.
The hand dynamometer plugs directly into the collection unit, the voltage output from
the accelerometers provides a signal that can be read directly by the Vernier differential
voltage probe sensor (Vernier DVP-BTA, Figure 4-6A), and the Omega torque sensor can
also be read by a DVP-BTA sensor after the signal is powered, shielded, and gained
through the use of a K and H Products “circuit project board” using an inverting op-amp
(3,500 kΩ & 10 kΩ yields 350 gain, CMRR of 130dB). The highest sample rate for a
Vernier LabQuest is only 100kHz versus the 250kHz used in the initial testing [9]. In
order to ensure that this new sampling rate is high enough, two accelerometers are
attached to the fixture using a Kistler supplied bonding wax (Figure 4-5B). Due to the slight differences in conductivity in any two similar systems due to a number of influences [31], the signal collected from both the accelerometers may represent slightly different moments in time. When analyzing the quick tangential accelerations that describe the shock, these time difference may determine whether the shock peak is recorded or not. If, in future tests, the maximum shock is sometimes recorded by one accelerometer and sometimes recorded by the other accelerometer, then missing the peak can be considered a factor in the initial tests’ variability in maximum shock. However, if both accelerometers always read the same or one is always higher than the other (due to inherent calibration differences), then the sample rate is high enough to accurately capture all shock behavior, and the second accelerometer can be removed.

The initial experiments had such variability in the maximum shock peak that it was hypothesized human behavior caused an additional shock resulting in the true signal being overwritten. Also, there was variability in shock magnitudes between surgeons, so set screw break off style may have played a role. These different styles may have led to complex loading conditions which can affect the torque achieved. A study showing the influence of parasitic forces (i.e. additional bending moments and cross forces) indicated that a “usual” deviation ran result in a difference on torque measurements of up to .35% of full scale [33]. Although this appeared relatively small, it was unknown if parasitic forces had a greater influence on sear bands in the set screw
breakoff procedure. Therefore, it was determined that the most beneficial setup would reduce parasitic forces and thus reduce any additional variation due to these influences.
An experimental setup was developed using modified, off-the-shelf components to apply pure torsion to the test pieces while reducing the chance of creating an extra shock event. The completed setup is shown in Figure 4-7D, and when referring to right and left locations in the following description, this is the picture that should be referenced. It is composed of 2 spin index fixtures (Figure 4-7A) that have grooves cut into each base so that bolts can rigidly hold each fixture to the table. Each fixture is further modified by milling a flat on the external slide tube so that the top set screw can adequately resist torque under high loads. This is important for the left fixture which holds the torque sensor and is the fixed end, and it is important for the right fixture during change outs of test piece tubes. The internal tightening tubes are also soldered onto the end handles since this allows the test pieces and torque sensor to be held tighter. In the final modification to these fixtures, a regular wrench socket that fits tightly into the internal tightening tubes is soldered into the end of the right fixture with the square tool end pointing outward. To this end is attached a ½” drive, 25” long breaker-bar that either tightens the test piece when the top fixture set-screw is engaged, or applies torque to the test piece when disengaged. Each fixture is designed to use a 5C collet to hold round shafts (Figure 4-7B). As shown in figure 4-4A, the left collet must be able to hold the ¾” TQM201-56.5 male end. The female end of this sensor is fitted with a ¾” OD Trantorque OE Mini Keyless bushing. The ID of the Trantorque along with the right collet must be able to hold the OD of whatever test piece is being used. The advantage to using the Trantorque instead of 10-32 UNF-2B set screws specified in Figure 4-4B is that the Trantorque grips the entire circumference of
an object and therefore can withstand greater torque loads. For added torque resistance, the IDs of each collet and TQM201-56.5 male end were sandblasted and cleaned so that the coefficient of friction was increased.

The test pieces also play an important role in the testing setup. The distance between the end of the same and the first edge of the groove should be 1.5” so that when the test piece is fully seated in the Trantorque, there will be enough of a length for the accelerometer fixture. It is important for the accelerometer to be on the side of the fixed Trantorque and TQM201-56.5. If instead, the accelerometer is on the rotating side (Figure 4-8A), then accelerations due to the movement of the rotating end may confound results. Figure 4-8B shows that a small groove should be filed into the test piece at the location of the accelerometer. This gives the set screw of the accelerometer fixture a more secure seating into the test piece. Kistler wax on the side of the accelerometer fixture opposing this set screw is used to ensure the fixture does not slip.

![Figure 4-8: Illustration of where NOT to position the accelerometer fixture (A) and how to modify the test piece surface and accelerometer fixture to create a more secure attachment](image)
As a final specification, at least another 1” of virgin test piece length should occur after the groove so that the 5C collet has enough area to grip.

4.4 Confirmation of Modeling Hypothesis: Physical Proof of Concept

Using physical Al5083-H116 geometries similar to L0R2 and L3R8, the maximum torque and maximum g-force shock will be compared using an adequate sample size. Assuming that the maximum torque remains approximately equal, the change in the maximum shock will be compared to the max(ALLSE)/max(ALLPD) ratio generated from the Abaqus simulations.

The axisymmetric physical test pieces may not be able to exactly replicate the Abaqus modeled geometry for a variety of reasons. Issues such as available machining dimensions, variations within machining tolerances, and the presence of physical effects not fully accounted for in the Abaqus simulation model all may need addressed if variations in results are significant. The test pieces will likely need machined on a high precision lathe by an experienced machinist. Due to limitation of available lathe inserts (Figure 4-9A & B), two different insert radii should be selected by the future research team that have a size-scale similar to the 2D and 3D simulations, relative to the test piece size. These simulations must then be re-run for the new dimensions to obtain proper max(ALLSE)/max(ALLPD) values. If variations in torque are shown to be consistently dependent on whether a narrow groove or large groove is being sheared, then new test specimens should be obtained with an adjusted axisymmetric cross section to ensure equal maximum torques are achieved between the two sample types.
Influences due to machining variations can be accounted for using more in depth statistical analysis, but ideally using a high precision machine shop will make these variations negligible.

As the samples are sheared off, both the released shock and maximum torque will be tracked. Ideally, the average reduction in shock between the L0R2 and L3R8 geometries will be compared to the reduction in the recoverable elastic energy to plastic energy dissipation ratio. A correlation will result from this comparison so that the researchers can use the energy ratio in future modeling to estimate what shock a particular geometry will create.

Figure 4-9: (A) Shows a lathe insert fixture with a triangular insert on the right side (B) Illustrates the insert geometries available for order
5.1 Geometry Effect on Plastic Energy and Future Work

Initial computer modeling experiments returned very promising results that geometric manipulations significantly affect the plastic behavior of a structure while still allowing the maximum torque to be separately controlled. By lengthening the groove in a tube section and reducing the stress concentrations from the fillet radii in a 2D axisymmetric model, the percent of work that went into plastic energy dissipation was increased by 36%. In addition, the radians the structure rotated before failure was shown to increase by 74%, along with the area experiencing plastic yielding. All these results are consistent, and show that plasticity of a structure is dependent on the specific geometry surrounding the failure region. However, physical test must be performed utilizing modeled geometry in order to confirm to what degree shock upon failure is affected by the ratio of maximum recoverable elastic energy to maximum plastically dissipated energy.

In all cases the maximum torque required for the structure to experience torsional failure was almost identical. This result indicates that the failure strength of a tube under torsion is primarily a function of the failure region’s mean thickness, which remained constant, and therefore not affected by surrounding geometries. As more
complex geometries affect the mean thickness, more testing would be required to show the effect on maximum torque.

5.3 Future Testing

Physical validation tests and 3D computer simulations both need performed next utilizing the Al5083-H116 material. The physical tests should determine to what degree the elastic/plastic energy ratio and/or rotations to failure are correlated with a reduced shock. Depending on the machining capabilities, the 2D modeling may need to be re-run to match the physical test piece geometries. 3D cyclic symmetric computer models should then be made of these geometries and validated with the same physical tests. The purpose of the validated 3D models is twofold: first, these models should provide increased accuracy of the energy ratio to shock correlation due to the ability to disable element deletion; second, more complex geometries can be built directly from this already validated model. Future validation tests would still need to be run on the more complex geometries to confirm that the results remain consistent. However, assuming consistency, the 3D models can identify the most promising geometries to prototype for validation so that design iterations may be minimized if the assumption is valid. Once all of this is done for the Al5083-H116 material, all tests will need to be re-run using the material used in production for spinal fusion set screws, Ti 6Al-4V.

The opportunities for additional future computer and physical testing to leverage the knowledge gained from the initial tests are endless. A successful 3D model can be used to provide an avenue through which the influence of human and machining
variations can be determined. Initial tolerance testing can be performed to determine machining quality parameters. Furthermore, future studies can analyze the influence that complex moments (due to non-concentric tightening movements by the surgeon) and rate of tightening have on maximum break off torque. Potentially, these variations can be addressed in future set screw designs. Once a reduced shock set screw is prototyped using Ti 6Al-4V, bench top studies can be performed to show analytically how reduced shock can improve patient outcome by creating a more secure pedicle screw/bone interface (as indicated by an increase in the pull-out strength). Finally, incorporating plastic or viscoelastic energy dissipation into tightening tools by way of using disposable inserts or a dashpot mechanism can be explored.

5.4 Key Take Away

Regardless of the outcome of these future tests, the demonstrated ability of using geometry to increased plasticity without affecting maximum torque is optimal for the researchers’ surgical device. It suggests that some sort of geometry can be designed into the failure region of spinal implant set screws which will allow the maximum torque at failure to be maintained while increasing the failure region’s plasticity. This increase in plasticity should reduce the amount of energy available to create g-force shock, thereby reducing the maximum shock magnitude during the set screw break off procedures. That was the main goal of this thesis. Also, the computer model is now ready, and can be used to explore, identify, and generate the right geometric
modification to the set screws, such that the shock is reduced at break off. This is expected to significantly benefit patient and surgeon health and safety.


4. Selvon St. Clair. Abnormal Clinical Biomechanics of the Spine. Cleveland Clinic MD


APPENDICES
APPENDIX A

SPINE ANATOMY OVERVIEW

The main functions of the spine are to carry load, enable motion, and protect sensitive soft tissues. All the other features of a spine are accessories that enable the spine to carry out these functions. Important spinal anatomies include bony vertebrae, and soft tissue groupings of ligaments, muscle, blood vessels, and nerves. There are also general terms that need to be understood in order to describe the anatomic features. These anatomic features and general terms will be referred to later when describing various spinal problems and implant types.

There are three primary reference planes (Figure A-1) from which the spine is described in literature. A sagittal plane divides the body into a right and left side, with the mid-sagittal plane at the point of symmetry. The coronal plane divides the body into front and back sections, and the transverse plane separates the top and the bottom parts of the body. When viewing anatomic pictures, often labels will appear at places in the picture so that the viewer can get an idea of how the particular anatomy is situated in the body. “Medial” means toward the mid-sagittal plane while “lateral” is away. “Posterior”/”dorsal” is toward the rear of the body, and “anterior”/”ventral” is in the direction of the body’s front. “Superior”/”cranial”/”cephalad” refers to the direction of the head and “inferior”/”caudal”/”caudad” is toward the feet. “Proximal” objects are
closer to a source, whether in terms of a geometric or flow source (ex. Blood vessels), and “distal” is further away (ex. The feet are distal to the knees) [2].

The general shape of the spine is categorized in groups defined by the vertebral (bony) sections (Figure A-2). There are four major sections; cervical vertebrae (C1-C7), thoracic (T1-T12), lumbar (L1-L5), and sacrum (S1-S5). Thoracic vertebrae are easily identified as any vertebrae connected to a rib. Anything above this is a cervical vertebra, and anything below until the drastic anatomical change at the sacrum is a lumbar
vertebra. Coccyx sections are traditionally ignored in research and treatment development because they are not considered structurally critical. The general curvatures of the spine normally occur in the sagittal plane (any coronal plane curvature is abnormal scoliosis, discussed later). Cervical and lumbar sections are concave posteriorly, while the thoracic, sacral, and coccyx sections are concave anteriorly [2].

Each vertebra has many sub-components that are identified here for later ease of reference. Vertebrae can look quite different depending on what level is being viewed; however, the particular differences of the various vertebral segments will not be addressed since the proposed medical device primarily addresses lumbar vertebrae. The elements of a lumbar vertebra can be seen in Figure A-3 [1], although various minor elements may not be identifiable (in the spines of some people, ex. AP and MP). Terms defining convenient groupings include “posterior elements” (everything posterior to the vertebral body), the “neural arch” (posterior and lateral boundaries of the spinal foramen that are the pedicles and lamina), and the intervertebral foramen (Figure 1-2, space between the superior and inferior pedicles). Regardless of the shape of any of the bony structures, they are all made of a dense cortical shell with a matrix of trabecular bone making up the cancellous interior [2].

The soft tissues connecting vertebrae sections appear simple; however, they are actually quite complex at a micro-level. An intervertebral disc separates any two vertebral bodies. The places where the inferior and superior articular processes
Figure A-3: Lumbar vertebrae anatomy...P: pedicle; L: lamina; SAF/IAF: superior/inferior articular facet; SAP/IAP: superior/inferior articular process; TP/SP/MP/AP: transverse/spinous/mammillary/accessory process; VF: vertebral (or spinal) foramen; RA: ring apophysis; NA: neural arch [1]
meet are called facet/zygapophysial joints. Collectively, two facets and one disc are called the three joint complex, because all three joints move in order for relative motion to occur between vertebrae. The 23 intervertebral discs will be the primary soft tissue concentration. All discs are made up of three major components consisting of two end plates, a nucleus pulposus, and an annulus fibrosus (Figure A-4). The endplates are cartilaginous, and the nucleus and annulus are made up of varying densities of water, proteoglycans (PGs), and collagen fibers. While not shown by the picture, there is not a definite boundary where the nucleus and annulus meet; rather it is a gradual morphing of one into the other [2].

The nucleus is made up of mostly PGs, as its main purpose is to attract water and resist compression. PGs are long chains of proteins closely spaced and aligned with one another. These proteins are negatively charged which results in strong intra and
intermolecular repulsive force, serving to rigidly straighten each PG macromolecule. Although water is forced out during compression as indicated by the reduction in disc height, the increased density of PGs causes increased nucleus pressure through the increased resistive charge forces [2].

The primary function of the annulus fibrosis is to restrict the pressure of the nucleus. Collagen fibers are the primary construction material of this part of the disc. There are multiple lamina layers that compose the annulus. While the fibers of every layer are 60-70 degrees offset from the cephalad direction of the mid-sagittal plane, each successive layer alternates the side to which this offset occurs. The fibers themselves have strong tensile properties, but almost a non-existent compressive strength. The angular offsets ensure that the fibers undergo tension when the disc is compressed. There is some outward bulging of the annulus when the disc is loaded, but the magnitude of bulge at a given location depends on disk health and direction that the spine is bending [2].

Ligaments are another soft tissue that connect the vertebrae together and stabilize the spine during loading and movement. The tissue is white, fibrous, and usually significantly composed of collagen. Depending on the orientation of attachment, ligaments can restrict motion in certain directions, thereby increasing the joint strength. Although the ligaments do allow motion by lengthening, there is a limit to the fiber strain allowed before failure occurs. Similar to the annulus, these ligaments are strongest when forces of tension are applied co-axial to their fibers, but buckling occurs
Figure A-5: Locations of the spine ligaments [2]
if loaded in compression. The ligaments in the lumbar and thoracic region can be seen in Figure A-5 [2].

Additionally, nerves, muscles, and the blood supply are additional spinal soft tissues. In biomechanical studies, muscles are sometimes simplified as force activation at their point of attachment to the spine, generally a downward vector due to most of the muscles being based at the pelvis. Muscles provide stability to the spinal column during movement and allow a person to stand upright. When considering blood supply, the most significant thing that must be kept in mind is the position of the aorta and major vessels that branch from it (Figure A-6). Nerve locations are important because if abnormal spinal movements or structural changes cause nerve pathways to be impinged upon, then an individual will feel pain and/or loose feeling and control of an associated anatomy. There are 31 nerve pairs that exit the intervertebral foramen along the spine, with each pair affecting a different region of the body, or dermatone. However, the most important nerve feature in the spine is the spinal cord, which is protected inside the spinal foramen and surrounding ligaments. The position of the spinal cord and nerve pairs can be seen in Figure A-7 [2].
The various other nuances of spine anatomy are not important for the purposes of this thesis, since the proposed medical device primarily addresses major functional impairments. However, one can see that there are numerous anatomical features that compose and are adjacent to the spine. The features are intricately connected such that a malfunction in one component can drastically affect the function of an adjacent component.
APPENDIX B

EXPERIMENTAL AND EUATION VALIDATION OF MANUFACTURER SPECIFIED SET SCREW BREAK OFF (SSBO) TORQUE

Surgeons also communicated the importance of proper torque. Not enough torque can allow the stabilizing rod to slip. Over-torque of the set screw can severely deform the screw threads, causing them to lose strength, and possibly allowing the rod to slip when the patient later puts load on the spine. Materials engineering and machine design theories, as well as performing a test using a torque wrench, validated that the set screw does break at the torque specified by the manufacturer [9].

The shear failure during SSBO was explored experimentally and by applying distortion energy theory to thin wall torsion (TWT) and fully plastic torque (FPT) equations. SSBO design torque of 11.00 N-m (as supplied by the manufacturer) was compared to an experimental torque wrench measurement and to the torque required to reach the shear strength calculated by distortion energy theory in both TWT and FPT.

The basic information requirements of TWT and FPT were obtaining the inner diameter (ID) and outer diameter (OD) of the set screw, and calculate the shear strength. First, digital imaging and micrometer readings were used to determine the inner and outer diameters of the set screws at the point of failure (Figure B-1). Second, From the Distortion Energy (Von Mises) principles [11]:

82


\[ \tau_{\text{yield}} = \frac{\sigma_{\text{yield}}}{\sqrt{3}} \]

Using material data of titanium alloy 6-Aluminum 4-Vanadium (Ti-6AL-4V), which is the primary material from which many implants are made [12,13], Quasi-static yield stress for Ti-6Al-4V was determined to be 950 MPa [13]. Therefore, \( \tau_{\text{yield}} \) is about 548 MPa.

<table>
<thead>
<tr>
<th>Inner diameter (ID)</th>
<th>From Digital Image Analysis</th>
<th>Validation from a micrometer</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.005235 m</td>
<td></td>
<td>0.0053 m</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Outer diameter (OD)</th>
<th>From Digital Image Analysis</th>
<th>Validation from a micrometer</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0061111 m</td>
<td></td>
<td>0.0061 m</td>
</tr>
</tbody>
</table>

Figure B-1: A broken set screw with the ID and OD measurements [9]

In thin wall torsion, the stress in the wall of a tube is assumed to be independent of the radius. Therefore, the standard torsional stress equation is combined with the circular tube polar moment of inertia [11]:

\[
\tau = \frac{T \cdot r}{J} \\
J = \frac{\pi \cdot (OD^4 - ID^4)}{32}
\]

Break-off occurs when there is full plasticity at \( \tau_{\text{yield}} \), since the stress is considered constant throughout. The mean radius is used:

\[
T_{\text{break-off}} = \frac{\tau_{\text{yield}} \cdot \pi \cdot (OD^4 - ID^4)}{32 \cdot r_{\text{mean}}}
\]

This break-off torque is 11.24 N-m.
Fully plastic torque of a hollow shaft [11] is calculated by subtracting what the torque carrying capacity would have been of a rod (of the same material) with a diameter equal to the ID from the torque carry capacity of a solid bar with the diameter equal to the OD:

$$T_{	ext{tube}} = T_{\text{solid}} - T_{\text{hollow}}$$

The torque of each section, $T_{\text{solid}}$ and $T_{\text{hollow}}$, is calculated by:

$$T = \left(\frac{\pi}{12}\right) \tau_y D^3$$

Distortion energy is again used for $\tau_y$. The value of $T_{\text{solid}}$ is 32.56 N-m, and $T_{\text{hollow}}$ is 21.36 N-m. Thus, the FPT break-off torque is:

$$T_{\text{tube}} = 32.56 - 21.36 = 11.2 \text{ N-m}$$

Finally, a torque wrench (valid from 3.6 N-m to 29 N-m) validated the maximum torque reached before break-off as approximately 11.3 N-m [9]. A comparison of all the failure torques is in table B-1.

<table>
<thead>
<tr>
<th>Table B-1: Comparison of Torque Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Method</td>
</tr>
<tr>
<td>Design break-off torque (as supplied by Medtronic)</td>
</tr>
<tr>
<td>Thin Wall Torsion (TWT)</td>
</tr>
<tr>
<td>Fully Plastic Torque (FPT)</td>
</tr>
<tr>
<td>Torque wrench</td>
</tr>
</tbody>
</table>
APPENDIX C

INDIVIDUAL OUTPUT GRAPHS FROM ALL TRIALS, WITH NAMING CONVENTION OF THE
LENGTH OF THE GROOVE BOTTOM FOLLOWED BY THE RADIUS OF THE GROOVE FILLET
(IN .1MM UNITS), FOLLOWED BY EXTERNAL WORK AND ELASTIC ENERGY COMPARISON
GRAPHS.
## APPENDIX D

### AL5083-H116 PARAMETERS USED IN THE MODEL

<table>
<thead>
<tr>
<th>Density (lbf s²/in⁴)</th>
<th>Young’s Modulus (psi)</th>
<th>Poisson’s Ratio</th>
<th>Displacement at Failure</th>
<th>Seed size in shear region</th>
</tr>
</thead>
<tbody>
<tr>
<td>.00025</td>
<td>9926807.662</td>
<td>.3</td>
<td>1E-005</td>
<td>4E-005</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Yield Stress (psi)</th>
<th>Plastic Strain</th>
<th>Fracture Strain</th>
<th>Stress Triaxiality</th>
</tr>
</thead>
<tbody>
<tr>
<td>30000</td>
<td>0.3</td>
<td>0.1</td>
<td>1</td>
</tr>
<tr>
<td>31069.9</td>
<td>0.00870102</td>
<td>0.02</td>
<td>0.33</td>
</tr>
<tr>
<td>31948.7</td>
<td>0.00178157</td>
<td>0.04</td>
<td>0.8</td>
</tr>
<tr>
<td>32685.2</td>
<td>0.00270738</td>
<td>0.06</td>
<td>1</td>
</tr>
<tr>
<td>33609.7</td>
<td>0.00411425</td>
<td></td>
<td></td>
</tr>
<tr>
<td>34571.9</td>
<td>0.00591732</td>
<td></td>
<td></td>
</tr>
<tr>
<td>35825.8</td>
<td>0.008891</td>
<td></td>
<td></td>
</tr>
<tr>
<td>37076.5</td>
<td>0.0127137</td>
<td></td>
<td></td>
</tr>
<tr>
<td>38087.2</td>
<td>0.0165607</td>
<td></td>
<td></td>
</tr>
<tr>
<td>38939.1</td>
<td>0.0204236</td>
<td></td>
<td></td>
</tr>
<tr>
<td>39677.6</td>
<td>0.0242979</td>
<td></td>
<td></td>
</tr>
<tr>
<td>40330.9</td>
<td>0.0281809</td>
<td></td>
<td></td>
</tr>
<tr>
<td>40917.5</td>
<td>0.0320705</td>
<td></td>
<td></td>
</tr>
<tr>
<td>41450.6</td>
<td>0.0359655</td>
<td></td>
<td></td>
</tr>
<tr>
<td>41939.6</td>
<td>0.039865</td>
<td></td>
<td></td>
</tr>
<tr>
<td>42529.3</td>
<td>0.0450157</td>
<td></td>
<td></td>
</tr>
<tr>
<td>43402.1</td>
<td>0.0536278</td>
<td></td>
<td></td>
</tr>
<tr>
<td>44434.4</td>
<td>0.0655238</td>
<td></td>
<td></td>
</tr>
<tr>
<td>45526</td>
<td>0.0804138</td>
<td></td>
<td></td>
</tr>
<tr>
<td>46460.3</td>
<td>0.0953197</td>
<td></td>
<td></td>
</tr>
<tr>
<td>47536.8</td>
<td>.115321</td>
<td></td>
<td></td>
</tr>
<tr>
<td>48875.8</td>
<td>.145076</td>
<td></td>
<td></td>
</tr>
<tr>
<td>50665.4</td>
<td>.194896</td>
<td></td>
<td></td>
</tr>
<tr>
<td>53299.4</td>
<td>.294631</td>
<td></td>
<td></td>
</tr>
<tr>
<td>55251</td>
<td>.394434</td>
<td></td>
<td></td>
</tr>
<tr>
<td>56813.8</td>
<td>.494277</td>
<td></td>
<td></td>
</tr>
<tr>
<td>58123.4</td>
<td>.594145</td>
<td></td>
<td></td>
</tr>
<tr>
<td>59254.3</td>
<td>.694031</td>
<td></td>
<td></td>
</tr>
<tr>
<td>60251.6</td>
<td>.79393</td>
<td></td>
<td></td>
</tr>
<tr>
<td>61145.2</td>
<td>.89384</td>
<td></td>
<td></td>
</tr>
<tr>
<td>61955.8</td>
<td>.993759</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
APPENDIX E

ABAQUS (VERSION 6.10-2) STEPS USED TO BUILD THE 2D AXISYMMETRIC MODEL WITH TWIST

Notes:
- 2D axi-symmetric CGAX3 element gives additional twist angle degree of freedom
- Disadvantage of axisymmetric is that one cannot use explicit methods
- Units of values must be kept consistent by the user

PART MODULE: ...(from pull-down window just above view-port...) Select “Module: Part”
1) CREATING THE PART...(from top menu...) “Part”, “Create…”,
   a) name the part “tube” on the text input box, select “Axisymmetric”, select “Deformable”, select “Shell”, check “Include twist” (creates a generalized axisymmetric model that allows rotational movements and force calculations), specify “Approximate size” to at least twice as big as the greatest cross section length or the radius, whichever is greatest (For this thesis: 0.012), click “Continue”
   b) Draw desired cross-section with a defined radius. For L0R2 groove in this thesis:
      i) 0.00265 radius (length from center line to left-most straight edge of tube wall), 0.0058 tube height (continuous vertical dimensioned length on the left side, two non-dimensioned vertical lengths on the right where one extends from the base and one extends from the top, but neither touch one another), 0.0008 regular wall thickness on the upper and lower horizontal boundaries of the tube cross section
      ii) Define the groove by:
         (1) Draw a short straight edge extending from the ends of the nearby right vertical lines toward the left side, constrain these lines as horizontal
         (2) Draw a semi-circle (defined by two points) between the ends of the two new straight lines, defining the semi-circle radius (L0R2 groove is 0.0002), create a “tangent” constraint between the semi-circle and the two new horizontal lines
         (3) Draw an “Isolated Point” midway on the circumference of the semi-circle (location marker will indicate when this point is being selected by turning from a cross to a filled-in circle)
(4) Define the length between this new point and the left edge (For this thesis: 0.0004)

   iii) Confirm the sketch created is a closed structure, otherwise it will be invalid
   
   c) Hit "Esc" on keyboard
   d) Click “Done” just below the view-port

2) PARTITIONING THE PART...(from top menu…) “Tools”, “Partition…”, select “Face”, select “Sketch”
   
   a) (from top menu…) “Add”, “Line”, “Rectangle”
   b) Click the left-top corner of the tube-wall to start the first box and click somewhere just above the groove on the right side of the tube-wall
   c) Click the left-bottom corner of the tube-wall to start the second box and click somewhere just below the groove on the right side of the tube-wall
   d) (from top menu…) “Add”, “Dimension”, define the length slightly away from the start of the groove on both sides such that the region of interest does not extend into either box (For this thesis, the plastic deformation will not extend into either box if both lengths are 0.0002)
   e) Hit “Esc” on keyboard
   f) Click “Done” just below the view-port

PROPERTY MODULE: ...(from pull-down window just above view-port…) Select “Module: Property”

3) DEFINING MATERIAL...(from top menu…) “Material”, “Create…” (see APPENDIX D for AL5083-H116 values)
   a) Name the material as desired (For this thesis: “Al5083”)
   b) General, Density
   c) Mechanical, Elasticity, Elastic (E, Poisson’s ratio)
   d) Mechanical, Plasticity, Plastic (tabular input of “Yield Stress” and “Plastic Strain”)
   e) Mechanical, Damage for Ductile Metals, Ductile Damage
      i) Tabular input of “Fracture Strain” and “Stress Triaxiality” (zero for strain rate)
      ii) Suboptions, Damage Evolution, “Type: Displacement”, “Softening:Linear”, “Degradation:Maximum”, “Displacement at Failure” should be 25% or less of your element seeded length (for this thesis: .25* seeded length of 4E-005 = 1E-005). This value is the amount of elongation an edge will have after the failure criteria is met, and is similar to defining the amount of strain occurring in the necking region of an engineering stress-strain curve

4) CREATE SECTION
   a) (from top menu…) “Section”, “Create…”
   b) Name the section as desired (For this thesis: “tube_sec”), select “Solid”, select “Homogeneous”
   c) Click “Continue”
   d) Select the proper material (For this thesis: “Al5083”)
   e) Check “Plane stress/strain thickness:” and leave value as “1”
   f) Click “OK”
5) ASSIGN SECTION
   a) (from top menu...) “Assign”, “Section”
   b) Select the regions of the part from the viewport that must be associated with the section just created (For this thesis, there are 3 regions)
   c) Hit “Enter” on keyboard
   d) Select the desired section (For this thesis: “tube_sec”)
   e) In the “Thickness” section of the window, leave the “Assignment” selection as “From section”
   f) Click “OK”
   g) Hit “Esc” on keyboard

MESH MODULE: ...(from pull-down window just above view-port...) Select “Module:Mesh”
6) MESHING THE PART
   a) (from top menu...) “Seed”, “Edges...”
      i) HOLD THE “SHIFT” KEY and select all edges bordering the region of interest, including the nearby partition borders (For L0R2 groove, this should be 7 straight edges and one semi-circular edge)
      ii) Hit “Enter” on keyboard
      iii) Keep defaults and define “Approximate element size:” under “Sizing Controls” (For this thesis: 4E-005)
      iv) Click “OK”
      v) HOLD THE “SHIFT” KEY and select the remaining faces (For this thesis, there should be 6 straight edges)
      vi) Hit “Enter” on keyboard
      vii) Keep defaults and define “Approximate element size:” under “Sizing Controls” (For this thesis: 0.0001)
      viii) Click “OK”
      ix) Hit “Esc” on keyboard
   b) (from top menu...) “Mesh”, “Controls...”
      i) HOLD THE “SHIFT” KEY and select all part regions (For this thesis, there are 3 regions)
      ii) Hit “Enter” on keyboard
      iii) Select “Tri”, leave default technique as “Free”
      iv) Click “OK”
   c) (from top menu...) “Mesh”, “Part...”, Hit “Enter” on keyboard
   d) If the mesh does not exactly match up to the seeded edges and this condition is important to the model (ex. Mesh convergence analysis), then repeat all steps in 6-b EXCEPT un-check the “Use mapped meshing where appropriate” under the “Algorithm” section in the pop-up-window.

ASSEMBLY MODULE: ...(from pull-down window just above view-port...) Select “Module:Assembly”
7) CREATE ASSEMBLY
   a) (from top menu...) “Instance”, “Create...”
   b) Select desired part(s) (For this thesis: just “tube”), click “OK” (will automatically assume assigned mesh)

8) CREATE NODE SETS
   a) Line up the viewport to the Y/X-plane
      i) (from top menu...) “View”, “Toolbars”, “Views” must be checked
      ii) Look for the icon in the upper toolbar that has an up-ward and a right-ward facing arrow labeled “Y” and “X”, respectively
      iii) Click this icon
   b) Create the boundary set...
      i) (from top menu...) “Tools”, “Set”, “Create...”
      ii) Name it “BOUNDARY”, select “Node”, click “Continue”
      iii) Click & drag to box/select desired nodes (For this thesis: only the nodes located on the bottom edge of the tube). HOLD THE “SHIFT” KEY and select addition nodes if desired.
      iv) Hit “Enter” on keyboard
   c) Create the loading set...
      i) (from top menu...) “Tools”, “Set”, “Create...”
      ii) Name it “LOAD”, select “Node”, click “Continue”
      iii) Click & drag to box/select desired nodes (For this thesis: only the nodes located on the top edge of the tube). HOLD THE “SHIFT” KEY and select addition nodes if desired.
      iv) Hit “Enter” on keyboard

STEP MODULE: ... (from pull-down window just above view-port...) Select “Module:Step”

9) CREATE LOADING STEP
   a) (from top menu...) “Step”, “Create...”
   b) Leave name as “Step-1”, leave defaults (i.e. “General” & “Static, General”)
   c) Click “Continue”
   d) On the initial “Basic” tab, select “On” for Nlgeom (so large displacements can be considered)
   e) Select the “Incrementation” tab
      i) Leave default “Type” as “Automatic”
      ii) Increase maximum number of increments so this does not limit the calculations (For this thesis: 10,000)
      iii) Leave everything else as default to start with. If later runs produce errors, then “Increment size: ” values may need adjusted. (For this thesis: “Initial was adjusted to 0.001”)
      iv) Click “OK”

10) MODIFY OUTPUT VARIABLES TRACKED
    a) (from top menu...) “Output”, “Field Output Requests”, “Edit”, “F-Output-1”
    b) Leave already selected values alone
c) Expand “Forces/Reactions”, select RM “Reaction moments”

d) Expand “Failure/Fracture”, select DMICRT “Damage initiation criteria”

e) Expand “State/Field/User/Time”, select STATUS “Status (some failure and plasticity models; VUMAT)”

LOAD MODULE: ...(from pull-down window just above view-port…) Select “Module:Load”

11) CREATE FIXED BOUNDARY CONDITIONS
   a) (from top menu…) “BC”, “Create…”
   b) Name as desired, change “Step:” to “Initial”
   c) “Category” is “Mechanical”, select “Displacement/Rotation”
   d) Click “Continue”
   e) Click “Sets…” (at the bottom right of the view-port)
   f) Select the “BOUNDARY” set, click “Continue”
   g) Check U1, U2, UR2, and UR3, click “OK”

12) CREATE ROTATING BOUNDARY CONDITIONS
   a) Since the torsion of the tube top is displacement controlled (as opposed to force controlled), it must be defined as a boundary condition: (from top menu…) “BC”, “Create…”
   b) Name as desired, change “Step:” to “Step-1”
   c) “Category” is “Mechanical”, select “Displacement/Rotation”
   d) Click “Continue”
   e) Click “Sets…” (at the bottom right of the view-port)
   f) Select the “LOAD” set, click “Continue”
   g) Check UR2, set value to radians of rotation for the loaded set (For this thesis: 0.3), click “OK”

JOB MODULE: ...(from pull-down window just above view-port…) Select “Module:Job”

13) CREATE JOB
   a) (from top menu…) “Job”, “Create…”
   b) Name as something OTHER than “Job-1” (this name causes an error for some reason), leave defaults, click “Continue”
   c) Leave defaults, click “OK”

14) RUN JOB: (from top menu…) “Job”, “Submit”, select the desire job

15) VIEW RESULTS: (from top menu…) “Job”, “Results”, select the desire job
APPENDIX F

ABAQUS (VERSION 6.10-2) STEPS USED TO BUILD THE 3D CYCLIC SYMMETRIC MODEL

Notes:
- Units of values must be kept consistent by the user
- To stop element deletion: Section controls are defined when assigning element type in
  the mesh module (mesh, element type, under “Element Controls”, “Element deletion” is
  NO, and “Max Degradation” is Specified as .99)
- When viewing the output, to view the reaction moment on the loaded surface, the
  “Unique Nodal” “Node Sets” used should be “ASSEMBLY_CONSTRAINT-
  1_REFERENCE_POINT”

PART MODULE: ...(from pull-down window just above view-port...) Select “Module: Part”
1) CREATING THE PART...(from top menu...) “Part”, “Create…”,
   a) name the part “tube” on the text input box, select “3D”, select “Deformable”,
      select “Solid”, select “Revolution”, specify “Approximate size” to at least twice as
      big as the greatest cross section length or the radius, whichever is greatest (For
      this thesis: 0.012), click “Continue”
   b) Draw desired cross-section with a defined radius. For 3D-L0R2 groove in this
      thesis:
         i) 0.00265 radius (length from center line to left-most straight edge of tube
            wall), two non-dimensioned vertical lengths on the right where one extends
            from the base and one extends from the top (For this thesis: both are
            0.0002), 0.0008 regular wall thickness on the upper and lower horizontal
            boundaries of the tube cross section
         ii) Define the groove by:
             (1) Draw a short straight edge extending from the ends of the nearby right
                 vertical lines toward the left side, constrain these lines as horizontal
             (2) Draw a semi-circle (defined by two points) between the ends of the two
                 new straight lines, defining the semi-circle radius (L0R2 groove is 0.0002),
                 create a “tangent” constraint between the semi-circle and the two new
                 horizontal lines
             (3) Draw an “Isolated Point” midway on the circumference of the semi-circle
                 (location marker will indicate when this point is being selected by turning
                 from a cross to a filled-in circle)
Define the length between this new point and the left edge (For this thesis: 0.0004)

iii) Confirm the sketch created is a closed structure, otherwise it will be invalid

   c) Hit “Esc” on keyboard
   d) Click “Done” just below the view-port
   e) Specify the desired rotation of part, in degrees (For this thesis: 5 degrees), click “OK”

2) [no cross-section partitioning required, since with the stated size the entire region is important]

PROPERTY MODULE: ...(from pull-down window just above view-port...) Select “Module: Property”

3) DEFINING MATERIAL...(from top menu...) “Material”, “Create...” (see APPENDIX D for AL5083-H116 values)
   a) Name the material as desired (For this thesis: “Al5083”)
   b) General, Density
   c) Mechanical, Elasticity, Elastic (E, Poisson’s ratio)
   d) Mechanical, Plasticity, Plastic (tabular input of “Yield Stress” and “Plastic Strain”)
   e) Mechanical, Damage for Ductile Metals, Ductile Damage
      i) Tabular input of “Fracture Strain” and “Stress Triaxiality” (zero for strain rate)
      ii) Suboptions, Damage Evolution, “Type: Displacement”, “Softening:Linear”, “Degradation:Maximum”, “Displacement at Failure” should be 25% or less of your element seeded length (for this 3D thesis part: 1E-006). This value is the amount of elongation an edge will have after the failure criteria is met, and is similar to defining the amount of strain occurring in the necking region of an engineering stress-strain curve

4) CREATE SECTION
   a) (from top menu...) “Section”, “Create...”
   b) Name the section as desired (For this thesis: “tube_sec”), select “Solid”, select “Homogeneous”
   c) Click “Continue”
   d) Select the proper material (For this thesis: “Al5083”) 
   e) Click “OK”

5) ASSIGN SECTION
   a) (from top menu...) “Assign”, “Section”
   b) Select the desired section (For this thesis: “tube_sec”)
   c) In the “Thickness” section of the window, leave the “Assignment” selection as “From section”
   d) Click “OK”
   e) Hit “Esc” on keyboard

MESH MODULE: ...(from pull-down window just above view-port...) Select “Module:Mesh”
6) MESHING THE PART
   a) (from top menu…) “Seed”, “Part…”
      i) Keep defaults and define “Approximate element size:” under “Sizing Controls” (For this thesis: 2E-005)
      ii) Click “OK”
      iii) Hit “Enter” on keyboard
   b) (from top menu…) “Mesh”, “Controls…”
      i) Select “Hex”, specify “Technique” as “Bottom-up”
      ii) Click “OK”
   c) (from top menu…) “Mesh”, “Create Bottom-Up Mesh…”
   d) Select “Method” as “Sweep”
   e) Click “Select” for the “Parameters, Source side:”, select the closest cross-section face, hit “Enter” on keyboard
   f) Click “Select” for the “Parameters, Connecting sides:”, select all of the faces that connect the source side cross-section to the opposing cross-section
      i) Hold “Shift” on the keyboard while selecting faces
      ii) In order to rotate the part so that other faces can be selected, let go of the “Shift” key, press and hold the “Ctrl” and “Alt” on the keyboard while clicking and dragging to view different faces of the part
      iii) Again, hold “Shift” on the keyboard while selecting more faces
      iv) Hit “Enter” on keyboard
   g) Select the “Target side:” box under “Parameters”, click “Select”, select the cross-section opposing the source side
   h) Click “Mesh”

ASSEMBLY MODULE: ...(from pull-down window just above view-port…) Select “Module:Assembly”
7) CREATE ASSEMBLY
   a) (from top menu…) “Instance”, “Create…”
   b) Select desired part(s) (For this thesis: just “tube”), click “OK” (will automatically assume assigned mesh)

8) CREATE REFERENCE POINTS
   a) (from top menu…) “Tools”, “Reference Point…”
   b) (from the bottom left of view-port…) Give the coordinates of first reference point as the origin (i.e. “0,0,0”), hit “Enter” on the keyboard
   c) (from the bottom left of view-port…) Give the coordinates of the second reference point as the some point on the y-axis within the view-port (For this thesis: “0,0.001,0”)

INTERACTION MODULE: ...(from pull-down window just above view-port…) Select “Module:Interaction”
9) CREATE CYCLIC SYMMETRY
   a) (from top menu…) “Interaction”, “Create…”
b) Name as desired, “Step:” must be “Initial”, select “Cyclic symmetry (Standard)”,
    click “Continue”
c) (from the bottom left of view-port…) “Choose the master region type:” as
    “Surface”, select one of the cross-section faces, hit “Enter” on the keyboard
d) (from the bottom left of view-port…) “Choose the slave type:” as “Surface”,
    select the other cross-section face, hit “Enter” on the keyboard
e) (from the bottom left of view-port…) “Select the first point of the axis of
    symmetry” as the first reference point (created in step 8)
f) (from the bottom left of view-port…) “Select the second point of the axis of
    symmetry” as the second reference point (created in step 8)
g) Uncheck “Adjust slave surface initial position” (Abaqus Users manual 15.13.16)
h) Specify the total number of sections required to make the modeled section
    rotate 360 degrees (For this thesis: 360/5=72 sections)
i) Leave all other defaults as is, click “OK”

10) CREATE CONSTRAINT FOR TORSION SURFACE
    a) (from top menu…) “Constraint”, “Create…”
    b) Name as desired, select “Coupling”, click “Continue”
    c) “Select the constraint control point” as the second reference point
d) (from the bottom left of view-port…) click “Surface”
e) Select the top surface of the tube (i.e. between the two cross sections), hit
    “Enter” on the keyboard
f) Leave all defaults (“Kinematic” and all degrees of freedom selected), click “OK”

STEP MODULE: ...(from pull-down window just above view-port…) Select “Module:Step”
11) CREATE LOADING STEP
    a) (from top menu…) “Step”, “Create…”
    b) Leave name as “Step-1”, leave defaults (i.e. “General” & “Static, General”)
    c) Click “Continue”
    d) On the initial “Basic” tab, select “On” for Nlgeom (so large displacements can be
       considered)
    e) Select the “Incrementation” tab
       i) Leave default “Type” as “Automatic”
       ii) Increase maximum number of increments so this does not limit the
           calculations (For this thesis: 10,000,000)
       iii) To help with convergence, adjust “Increment size: ” so “Initial” is .001,
           “Minimum” is 1E-020, and “Maximum” is .01
       iv) Click “OK”
12) MODIFY OUTPUT VARIABLES TRacked
    a) (from top menu…) “Output”, “Field Output Requests”, “Edit”, “F-Output-1”
    b) Leave already selected values alone
c) Expand “Forces/Reactions”, select RM “Reaction moments”
d) Expand “Failure/Fracture”, select DMICRT “Damage initiation criteria”
103

e) Expand “State/Field/User/Time”, select STATUS “Status (some failure and plasticity models; VUMAT)”

LOAD MODULE: ...(from pull-down window just above view-port...) Select “Module:Load”

13) CREATE FIXED BOUNDARY CONDITIONS
   a) (from top menu...) “BC”, “Create...”
   b) Name as desired, change “Step:” to “Initial”
   c) “Category” is “Mechanical”, select “Displacement/Rotation”
   d) Click “Continue”
   e) (in the view port...) Select the bottom face of the model (i.e. perpendicular face NOT EDGE between the two cross section faces), hit “Enter” on the keyboard
   f) Check U1, U2, U3, UR1, UR2, and UR3, click “OK”

14) CREATE ROTATING BOUNDARY CONDITIONS
   a) Since the torsion of the tube top is displacement controlled (as opposed to force controlled), it must be defined as a boundary condition: (from top menu...) “BC”, “Create...”
   b) Name as desired, change “Step:” to “Step-1”
   c) “Category” is “Mechanical”, select “Displacement/Rotation”
   d) Click “Continue”
   e) (in the view port) Select the second reference point, hit “Enter” on the keyboard
   f) Check UR2, set value to radians of rotation for the loaded set (For this thesis: 0.3), check U1, U3, UR1, and UR3 to set these values to “0”, DO NOT check U2 (this allows the top part to move up or down as forces require while this part is getting sheared off), click “OK”

JOB MODULE: ...(from pull-down window just above view-port...) Select “Module:Job”

15) CREATE JOB
   a) (from top menu...) “Job”, “Create...”
   b) Name as something OTHER than “Job-1” (this name causes an error for some reason), leave defaults, click “Continue”
   c) Leave defaults, click “OK”

16) RUN JOB: (from top menu...) “Job”, “Submit”, select the desire job

17) VIEW RESULTS: (from top menu...) “Job”, “Results”, select the desire job