Finite element analysis of failure of high voltage current joints

Kiran Kumar Pusthay

University of Nevada, Las Vegas

Follow this and additional works at: https://digitalscholarship.unlv.edu/rtds

Repository Citation
https://digitalscholarship.unlv.edu/rtds/1741

This Thesis is brought to you for free and open access by Digital Scholarship@UNLV. It has been accepted for inclusion in UNLV Retrospective Theses & Dissertations by an authorized administrator of Digital Scholarship@UNLV. For more information, please contact digitalscholarship@unlv.edu.
FINITE ELEMENT ANALYSIS OF FAILURE OF HIGH VOLTAGE CURRENT JOINTS

by

Kiran Kumar Pusthay

Bachelor of Engineering
Jawaharlal Nehru Technological University, India
2001

A thesis submitted in partial fulfillment of the requirements for the

Master of Science in Mechanical Engineering
Department of Mechanical Engineering
Howard R. Hughes College of Engineering

Graduate College
University of Nevada, Las Vegas
December 2004
The Thesis prepared by

KIRAN KUMAR PUSTHAY

Entitled

FINITE ELEMENT ANALYSIS OF FAILURE OF HIGH VOLTAGE CURRENT JOINT

is approved in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE, MECHANICAL ENGINEERING

Examination Committee Chair

Dean of the Graduate College

Examination Committee Member

Examination Committee Member

Graduate College Faculty Representative
ABSTRACT

Finite Element Analysis of Failure of High Voltage Current Joints

by

Kiran K Pusthay

Dr. John Wang, Examination Committee Chair
Associate Professor, Mechanical Engineering
University of Nevada, Las Vegas

The Thesis provides the finite element analysis of high voltage current joints for atlas pulsed power machine. The role of preload on maintaining joint integrity for multi-bolt assemblies is discussed. The importance of using correct preload is illustrated, along with the types of failure that result due to improper preload. Various techniques used to model bolted connections for finite element analysis are explained in detail. The high voltage current joints are modeled using Solidworks. The pretension effects of multi-bolt assembly are simulated using Ansys Workbench software. This research successfully simulated the stage-I self-loosening behavior of the bolted joints. The stage-I self-loosening refers to failure of bolted connections due to loss of clamping force. The contact status of the surfaces clamped together by bolts is monitored for damage. Additionally, the fatigue tool from Ansys Workbench is used to monitor damage, safety factor and fatigue sensitivity. Finally this thesis can provide promising platform for new users, working on simulating multibolt assemblies, as there is very little literature available on FE analyses that concentrate on multibolt assemblies.
TABLE OF CONTENTS

ABSTRACT ............................................................................................................................ iii

LIST OF FIGURES .............................................................................................................. vi

LIST OF TABLES ............................................................................................................... vii

ACKNOWLEDGEMENTS ................................................................................................... viii

CHAPTER 1 INTRODUCTION .............................................................................................. 1
  1.1 Introduction .................................................................................................................. 1
  1.2 Organization of thesis ................................................................................................. 5

CHAPTER 2 LOOSENING MECHANISMS OF BOLTED JOINTS: LITERATURE SURVEY ................. 6
  2.1 Preload .......................................................................................................................... 6
  2.2 Joint failure .................................................................................................................. 8
      2.2.1 Mechanical failure of bolts ........................................................................ 9
      2.2.2 Loose bolts & lost bolts .......................................................................... 9
      2.2.3 Bolts too tight ............................................................................................. 9
  2.3 Self-loosening of bolted joints .................................................................................. 10
  2.4 Mechanism of relaxation and self-loosening ......................................................... 11
  2.5 Short-term relaxation of individual bolts ............................................................... 12
      2.6 Sources of short-term relaxation ....................................................................... 12
          2.6.1 Embedment ............................................................................................ 13
          2.6.2 Poor thread engagement ....................................................................... 14
          2.6.3 Conical makeups .................................................................................. 14
  2.7 Vibration loosening ................................................................................................... 16
  2.8 Fatigue Failure ........................................................................................................... 17

CHAPTER 3 FE TECHNIQUES TO MODEL BOLTED JOINTS .................................................. 19
  3.1 Introduction .................................................................................................................. 19
  3.2 Introduction to modeling bolted joint ..................................................................... 20
  3.3 Joint Simulations ....................................................................................................... 25
      3.3.1 No bolt simulation ..................................................................................... 26
      3.3.2 Coupled bolt ............................................................................................... 27
      3.3.3 RBE (rigid body element) bolt .................................................................. 29
      3.3.4 Spider bolt ................................................................................................. 30
      3.3.5 Hybrid bolt ............................................................................................... 31
      3.3.6 Solid bolt ................................................................................................. 33
LIST OF FIGURES

Figure 1.1 Transition assembly of atlas machine ..............................................................3
Figure 1.2 Deformation photograph of the high voltage current joint .............................4
Figure 2.2 Energy delivered to fastener-joint system [4] ...................................................8
Figure 2.3 Paks and valleys on thread and other contact surfaces [4] ..........................13
Figure 2.4 Por thread engagement [4] .............................................................................14
Figure 2.5 Rlaxation mechanism in conical or tapered joints [4] ...................................15
Figure 3.1 Blted joint labels............................................................................................21
Figure 3.2 Clinder section showing coupling action ......................................................21
Figure 3.3 Petension .........................................................................................................22
Figure 3.4 Cntact ...............................................................................................................24
Figure 3.5 N bolt simulation .............................................................................................27
Figure 3.6 Example for coupled bolt simulation .............................................................28
Figure 3.7 RBE simulation example.............................................................................29
Figure 3.8 Spider beam bolt simulation .......................................................................30
Figure 3.9 Example for hybrid bolt simulation .............................................................33
Figure 3.10 Example for solid bolt simulation ...............................................................33
Figure 4.2 Sectional view of the high voltage current joint assembly ...........................37
Figure 4.3 High voltage current joint assembly modeled in solidworks .......................37
Figure 4.4 (1) node to node (2) node to surface (3) surface to surface .........................39
Figure 4.5 Mesh generated using ansys........................................................................40
Figure 4.6 Preload applied to each bolt case-1 (2,800 lbf) .............................................43
Figure 5.1 The deformation contours on the high voltage conductor ............................46
Figure 5.2 Contact status at the interface of two bodies in the assembly ......................47
Figure 5.3 The contact status on the contact surface of high voltage conductor ..........48
Figure 5.4 Resultant bolt load vs. effective load on joint .............................................49
Figure 5.4 Fatigue sensitivity analysis for damage .........................................................53
Figure 5.5 Fatigue sensitivity analysis for safety factor .................................................54
Figure 6.1 The model considering the surface roughness ..............................................58
Figure 6.2 The micro arcing phenomenon illustrated .....................................................59
Figure A2.1 Solid186 geometry ....................................................................................61
Figure A2.2 Solid187 geometry ....................................................................................62
Figure A2.3 Conta174 geometry ...................................................................................63
Figure A2.4 Targe170 geometry ....................................................................................64
Figure A2.5 Surf154 geometry ......................................................................................65
Figure A3.1 Fatigue sensitivity analysis for preload value of 3800 lbf (damage) .........66
Figure A3.2 Fatigue sensitivity analysis for preload value of 3800 lbf (safety factor) ...67
Figure A 3.3 Fatigue sensitivity analysis for preload value of 7600 lbf (damage) .........68
Figure A3.4 Fatigue sensitivity analysis for preload value of 7600 lbf (safety factor)...69
LIST OF TABLES

Table 4.1  Mesh statistics for the model ................................................................. 40
Table 5.1  Stress strain and deformation for each loading condition ................. 45
Table 5.2  Magnitude of damage for three loading conditions ......................... 51
Table 5.3  Minimum and maximum values of safety factor for three scenarios .... 52
Table A.1  Material properties for aluminum alloy (materials library: ansys workbench) ................................................................. 60
Table A.2  Material properties for structural steel (materials library: ansys workbench) ............................................................... 60
ACKNOWLEDGEMENTS

I owe a sincere thank you to the many people who have both directly and indirectly made this work possible. I would like to express my deep sense of gratitude to my advisor Associate Professor Dr. John Wang, for his support and patience, and for his kindness and great enthusiasm for research, which have made it a tremendous pleasure and inspiration to work with him. Dr. John Wang deserves special recognition for personally mentoring me on all of my work so far as a master’s student. His enduring guidance to conceptualize this work and support to make it possible deserve particular appreciation. Dr. Wang’s open door policy and uncanny ability to simplify complex problems led to many stimulating technical discussions that fueled my research efforts and made my stay at this university extremely enriching and enjoyable. He taught me how to effectively communicate and successfully write scientific papers.

I would like to thank Dr. Brendan J. O’Toole, Dr. Ajit Roy and Dr. Yahia Baghzouz for their valuable suggestions. Thank you to Bob Reinovsky, John Frasca and Clark Thompson of Los Alamos National Laboratories who supported the research by providing valuable technical data. Thank you to my parents and siblings who have always encouraged me through their never ending love and support. They were always supportive of my higher studies. Looking back, I realize that they have made a lot of sacrifice for me. I dedicate my thesis to my beloved parents. Last but not the least I would like to thank all my friends and relatives for their support.

viii
CHAPTER 1

INTRODUCTION

1.1 Introduction

Bolt loosening is a major form of failure in machines and structures, according to Bolt Science [1], this fact is further verified by the survey of service managers in U.S automobile industry, which reports that 23% of all service problems are caused by loose fasteners and 12% of the new cars were found to have loose fasteners. The self-loosening of bolted joints is perhaps the most unexplored arena in the field of fastened joints. Zhang et al. [2], describes the self-loosening process as gradual loss of clamping force in the bolted connections under cyclic external loading, especially transverse loading. The self-loosening process is defined by two distinct stages, stage-I and stage-II. The first stage is characterized by significant reduction in clamping force without causing any rotation between nut and the bolt. The second stage is characterized by backing-off of the nut and rapid loosening of the clamping force. There are various theories that have been attached to the process of self-loosening of bolted joints, of them important being improper preload, reduction in clamping force and vibration theories. The self-loosening concept is explained in detail in chapter 2.

In this research, finite element analysis model of high voltage conductor joint assembly of atlas pulsed power machine is used to study the effect of preload on multi-bolt assemblies. Atlas, is the first large scale pulsed power system specifically
designed for executing solid-density liner implosions. In such implosions, the liner housed in the target chamber is imploded due to the magnetic interaction between the flowing electric charges that make up electric current. The amount of energy trapped and dissipated in the target chamber is more than 12 MJ. Atlas is the flagship facility for pulsed power driven hydrodynamics experimentation in the world at this time. With its modular design based on a customized low inductance Marx generator configuration with emphasis in reliability, fault tolerance, and easy access for the experimental chamber, Atlas is known as the highest performance laboratory, microsecond pulsed power system in the world. The complete system stores 24 MJ at rated charging voltage of 240 kV and delivers more than 30-MA currents with rise time from 5–6 μs, depending on the inductance of the load. The current study concentrates on the high voltage conductor assembly, which constitutes of high voltage connector and a high voltage conductor, which are clamped together by six bolts. This component has been reported to fail after certain number of tests. The effect of preload on this assembly and the possible causes of failure of the assembly are studied. Another objective of this simulation is to monitor the effect of varying preload on the high voltage current joints assembly. Figure 1.1 shows the transition assembly of the Atlas pulsed power machine.

1 Atlas Annual report
Preload is the tensile force introduced to the bolt during tightening. This tensile force in the bolt creates a corresponding clamping force in the joint. The clamped parts act like a compression spring; the bolt acts like a tension spring. This clamping force develops frictional force to prevent joint slip or separation. However applying too much preload will damage the fastener or joint member and at the same time insufficient preload may result in other types of failure. According to Esmailzadeh et.al [3], failures resulting from too low preload are found to be more apparent than failures due to too much preload. Bickford [4], points out the following problems are associated with incorrect preload, static failure of fasteners, static failure of joint members, vibration loosening of the nut, fatigue failure of the bolt, stress corrosion cracking, joint separation, joint slip, excessive weight and excessive cost. According to Bickford & Eccles [5], an appropriate level of preload corresponds to a 50%~75% of the nominal yield strength of the bolt. According to S. McGuigan et al., the initial preload applied to the bolt tends to be degraded by each successive application and removal of the external load on the joint. This is due to embedding of the contacting surfaces and flattening of surface roughness, and tends to occur rapidly with the first few load applications, and more slowly.
subsequent applications. The effect is most pronounced with soft materials and poorly finished surfaces, in this case aluminum [6].

There are many reasons which result in joint failure; typically a joint failure occurs when the bolts fail to perform their clamping function as designed. Bickford points out three principle types of failure: self-loosening, fatigue and corrosion. Before going into detail about the types of failure it is essential to have some outlook into different ways in which a joint and/or bolt can fail. The following are the ways in which bolt and/or joint can fail listed by Bickford,

1. Mechanical Failure of bolts
2. Lost Bolts & Loose Bolts
3. Bolts too tight

A detailed description of various failure modes in bolted joints are discussed in chapter 2. Figure 1.2 is the photograph of the deformed portion of the high voltage current joint provided by Los Alamos National laboratory.

![Figure 1.2: Deformation photograph of the high voltage current joint](image-url)
1.2 Organization of thesis

Chapter 1 is the introduction and it discusses the importance of bolted joints and also gives a brief description of the atlas machine and failure of the high voltage current joints. Chapter 2 gives the introduction to failure modes in the bolted joints and the relevant material from literature survey is highlighted here. Chapter 3 is the literature survey on finite element modeling of bolted joints. Various techniques that can be used to model the bolted joints are discussed briefly here. The high voltage current joints are modeled and assembled in Solidworks and then imported to ANSYS work bench to carry out the FE analysis. Chapter 4 will discuss the modeling procedure and the description of the finite element model. The results from the finite element analysis and discussions of the results will be provided in chapter 5. Conclusions and scope for the future work are discussed in chapter 6.
CHAPTER 2

LOOSENING MECHANISMS OF BOLTED JOINTS: LITERATURE SURVEY

2.1 Preload

Preload/pretension is the tensile force introduced to the bolt during tightening. This tension force in the bolt creates a corresponding clamping force in the joint. The clamped parts act like a compression spring; the bolt acts like a tension spring as shown in figure 2.1. This clamping force develops frictional force to prevent joint slip or separation. However, applying too much preload will damage the fastener or joint member and at the same time insufficient preload may result in other types of failure. According to Esmailzadeh et al. [3], too little preload failures are found to be more prevalent than too much preload. Bickford [4], points out the following problems are associated with incorrect preload, static failure of fasteners, static failure of joint members, vibration loosening of the nut, and fatigue failure of the bolt, stress corrosion cracking, joint separation, joint slip, excessive weight and excessive cost.
According to Bickford & Eccles [5], an appropriate level of preload corresponds to a 50%~75% of the nominal yield strength of the bolt. According to S. McGuigan et al. [6], the initial preload applied to the bolt tends to be degraded by each successive application and removal of the external load on the joint. This is due to embedding of the contacting surfaces and flattening of surface roughness, and tends to occur rapidly with the first few load applications, and more slowly with subsequent applications. The effect is most pronounced with soft materials and poorly finished surfaces, in this case aluminum [7]. It is estimated that about 50% of initial torque is lost under the nut and another 40% is lost within the threads, leaving only 10% as potential energy in the bolt. Therefore only 10% ends up as bolt preload or clamping force between the joint members. The following figure 2.2 gives this illustration [4].
2.2 Joint failure

There are many reasons which result in joint failure; typically a joint failure occurs when the bolts fail to perform their clamping function as designed. Bickford points out four principle types of failure: self-loosening, fatigue, corrosion and leakage. Before going into detail about the types of failure it is essential to have some outlook into different ways in which a joint and/or bolt can fail. The following are the ways in which bolt and/or joint can fail listed by Bickford,

1. Mechanical Failure of bolts

2. Lost Bolts & Loose Bolts

3. Bolts too tight
2.2.1 Mechanical failure of bolts

Mechanical failure of bolts occurs due to improper assembly or when the bolt or joint is exposed to elevated temperatures. The assembly errors could be pulling the bolts too hard with the wrench (by the mechanic). According to Sahoo the tensile force during tightening process results in a degree of thread bending between internal and external threads, resulting in reduction in shear area of both internal and external threads. Further he defines the strength ratio as the ratio between the forces necessary to cause the nut thread to strip divided by the force required to cause the bolt thread to strip. Though this type of failures is not very common, there is an increase in number of such failures reported mainly due to the reason that manufacturers of low cost bolts are using boron steel instead of medium carbon steel. Boron steel is found to lose strength more rapidly than carbon steels at elevated temperatures.

2.2.2 Loose bolts & lost bolts

Loose bolts are the most common cause for failure of bolted joints and the lost bolts obviously won't provide the designed clamping force to maintain the joint integrity. The main reason for missing bolts is self loosening phenomenon which is described briefly in the following section of this chapter. Common failure modes resulting from loose and/or lost bolts are joint leakage, joint slip, cramping of machine members, fatigue failure and self-loosening.

2.2.3 Bolts too tight
The joint failure due to too tight bolts is less common, but too tight bolts also contribute to joint failure. Disproportionate bolts loads will cause damage to the joint surfaces or
can squeeze the gaskets or result in stress corrosion cracking and can also reduce the fatigue life.

2.3 Self-loosening of bolted joints

The self-loosening of bolted joints is perhaps the most unexplored arena in the field of fastened joints. Zhang et al. [2] describes the self-loosening process as gradual loss of clamping force in the bolted connections under cyclic external loading, especially transverse loading. The self-loosening process is defined by two distinct stages, stage I and stage II. The first stage is characterized by significant reduction in clamping force without causing any rotation between nut and the bolt. First stage self-loosening is caused by local cyclic plastic deformation, often occurring in the vicinity of the roots of the engaged threads. As a result of local cyclic plasticity cyclic strain ratcheting is observed. Cyclic plastic deformation is also responsible for redistribution of stresses in the bolt and subsequent gradual loss of clamping force with loading cycles [5].

Backing-off of the nut and rapid loosening of the clamping force characterize the second stage. This is a result of gradual rotation of the nut relative to the bolt. Zhang et al. [2] identified two major factors that determine stage two self loosening, they are the variation of contact pressure in the engaged threads with the reversed external load and the micro-slip that occurs between the contact surfaces of the engaged threads. They found that the cyclic transverse load creates a cyclic bending moment on the bolt and as the result of variation of induced bending moment on the bolt there is a variation of the contact pressure between the engaged threads with time.
2.4 Mechanism of relaxation and self-loosening

Bolt Relaxation is the reduction of clamping force over time in a fastener with a fixed distance between surface of head and nut. Bickford [8] provides an extensive description of several important factors that influence joint relaxation. A fastener subjected to vibration will not lose its entire preload instantaneously, this is a gradual process and initially there will be a slow loss of preload caused by some of the relaxation mechanisms. Vibration will amplify the relaxation process through wear and hammering. After sufficient preload is lost, friction forces drop below a critical level and the nut actually starts to back off and shake loose. In this case, the joint will not resemble the ideal boundary conditions but will involve uncertainties. With higher initial preload, longer or more severe vibration is required to reduce preload to the critical point at which back-off occurs. In some circumstances, if the preload is high enough to start with, nut back off will never take place. Usually, safety-wires, coatings and inserts, thread-locking adhesive, and spring-washers are used to prevent loosening. These devices, however, have their limitations and do not necessarily prevent relaxation. In a joint at an elevated temperature, a fastener with a fixed distance between bearing surface of head and nut will produce less and less clamping force over time. This characteristic is called relaxation. It is different from creep because the stress changes without change in elongation. If the relaxation is not compensated for, it will lead to fatigue failure or a loose joint even though it was properly tightened initially. As the temperature environment and the materials of the structure are normally fixed, the design objective is to select a bolt material that will give the desired clamping force at all critical points in the operating range.
There are various theories that have been attached to the process of self-loosening of bolted joints, of them important being improper pre-load, reduction in clamping force and vibration theories. In this research we focus on the effect of pre load variation on self-loosening of the bolted joint assembly.

2.5 Short-term relaxation of individual bolts

According to Bickford [8], there will be initial loss of tension in individual bolts after they are initially tightened and he calls this kind of relaxation in individual bolts as short-term relaxation, to distinguish it from other effects such as stress relaxation that will result in loss of tension over an extended period of time. Short-term relaxation occurs in bolted joint since something has been loaded past its yield point and will creep and flow to get away from under the excessive load, which could be a component such as a soft bolt or a gasket, mostly a small portion of the component like first engaged threads in the nut. Some of the sources for short-term relaxation identified by Bickford are discussed in the following section of this chapter.

2.6 Sources of short-term relaxation

The following section gives the sources and brief explanation for the short-term relaxation.
2.6.1 Embedment

The surfaces of joint members i.e., the nut, the bolt and the faying surfaces are not perfectly flat. Using highly polished surfaces for industrial structures and fasteners is very rare; even if they are highly polished there will certain peaks and valleys if observed under microscope. These peaks and valleys on the surface of joint members will affect the initial contact area between the surfaces in contact, and the parts are in contact through only high spots on the metal surfaces. If a bolt is loaded it will exert high surface pressures on the structural members or on its own threads, but due to the presence of peaks and valleys only small portion of the bolt threads will share these loads, resulting in plastic deformation until enough of the thread surface shares the load without causing further deformation. This process is known as embedment. Embedment is more pronounced in new parts than the reused ones and in critical applications tightening, loosening and retightening the fasteners several times will minimize embedment.

Figure 2.3: Peaks and Valleys on thread and other contact surfaces [4]
2.6.2 Poor thread engagement

Poor thread engagement results from undersized bolts, or oversized nuts. The thread contact area will be less than the planned by the designer and will result in significant plastic deformation. Figure 2.4 gives the graphical representation of poor thread engagement.

![Figure 2.4: Poor thread engagement](image)

2.6.3 Conical makeups

Surface irregularities are not limited to flat surfaces only they can exist on conical joint surfaces as well. In case of embedment on conical surface, effect on the axial tension in the fastener is magnified.
A given amount of relaxation perpendicular to the axis of the fastener possibly will result in significantly greater relaxation in the axial direction as given by the following equation [8].

\[ r = \frac{e}{\sin(\phi)} \quad (1) \]

Where \( e \) = Embedment relaxation perpendicular to the surface of the conical joint member (in., mm)
\( r \) = resulting relaxation parallel to the axis of the fastener (in., mm)
\( \Phi \) = Half angle of the cone (deg)

Other sources for short-term relaxation are

1. Thread engagement too short
2. Soft parts
3. Bending
4. Non-perpendicular Nuts or Bolt Heads

Figure 2.5: Relaxation mechanism in conical or tapered joints [4]
5. Fillets or Undersized holes

6. Oversized holes

After looking at the sources that cause short-term relaxation it is important to focus on the factors that affect short-term relaxation. The amount of relaxation in a given bolt and joint depends on bolt length, belleville washers, number of joint members, tightening speed, simultaneous tightening of many fasteners and bent joint members.

2.7 Vibration loosening

Though bolted joints have the advantage of dismantling, this feature becomes a problem when it occurs unintentionally as a result of operational conditions. Such unintentional loosening is called vibration loosening. According to Sahoo [9], Vibration loosening occurs because of side sliding of the nut or bolt head relative to the joint, resulting in relative motion occurring in the threads. If this does not occur, the bolt will not loosen, even if the joint is subjected to severe vibration. The main causes of relative motion occurring in threads are:

• Bending of parts which results in forces being induced at the friction surface. If slip occurs, the head and threads will slip which can lead to loosening

• Differential thermal effects caused as a result of either differences in temperature or differences in clamped materials

• Applied forces on the joint can lead to shifting of the joint surface leading to bolt loosening.
It is observed that transversely applied alternating forces generate the most severe conditions for self loosening. The key to prevent self loosening of fasteners is to ensure that:

- There is significant clamp force present on the joint interface to prevent relative motion between the bolt head or nut and the joint
- The joint is designed to allow for the effects of embedding and stress relaxation.
- Proven thread locking devices are used (example Loctite)

2.8 Fatigue failure

Cyclic loading can cause fatigue failure of the joint structure and bolts, hence bolt fatigue is an important design criteria. According to Starikov [10], mechanically fastened aluminum joints loaded in fatigue suffer from fretting damage in the faying surfaces of the joined plates. Surface microslip associated with small-scale oscillatory motion of clamped structural members result in contact damage and this process is called fretting. Fretting fatigue damage may start to propagate from number of typical sites. Starikov further discusses three failure origins from the experimental work on the fatigue durability of aluminum bolted joints. First is the fatigue cracks induced by fretting between the bolt shank and the plates in the hole surface at multiple sites. The second failure origin site is associated with surface close to the bolt holes, the damage formation and propagation in this case was the result of fretting of the out-of-plane protrusions on the faying surfaces. Finally, the third and last failure site was located at a certain distance from holes.
Bickford and Nassar define fatigue strength as the ability of a bolt to withstand a given number of cycles of stress. The resulting failure is either high or low cycle fatigue depending on the number of cycles leading to failure. In order to reduce fatigue related problems in an existing joint Bickford and Nassar suggest the following steps:

1. Reduce bending stress by using greater bolt preload or change in design
2. Using bolts with greater tensile or fatigue strength
3. When new design is used check bolt preload at the time of assembly and after operating the equipment.
4. Confirm that bolt and joint materials are compatible.

This chapter discussed the various loosening mechanisms associated with the bolted joints. A brief description of important failure modules from the literature is provided. The following chapter is the literature survey on the various techniques on finite element modeling of the bolted joints. Various techniques that can be used to model the bolted joints are explained.
CHAPTER 3

FE TECHNIQUES TO MODEL BOLTED JOINTS

3.1 Introduction

The Finite Element Method is used to study bolted joints with finite sliding deformable contact where the helical and frictional effects on the load distribution of each thread are included [11]. In order to study and understand the interaction and stresses developed in the threads of a bolted connection two-dimensional, axisymmetric models are used. However there are certain concerns associated with two-dimensional, axisymmetric models, one specific issue with this approach is that the 2D model simulates the threads as separate rings of material, instead of continuous helix. The study of bolted joints is complicated since it has to take into account many factors such as material and geometrical nonlinearities, contact, friction, slippage, bolt-plate interaction and fracture (Mackerle J). Apart from these factors there are certain other parameters which affect the solution obtained from finite element analysis, such as the element type used, discretization, constitutive equations, step size, etc[11].

According to Johnson H.D, et.al [12], two-dimensional modeling cannot simulate the conditions of joint tightening and sliding along the helical thread flanks when the nut is turned. Modeling bolts for three-dimensional finite element analysis have, and still continue to raise questions [13]. Modeling the threads for bolts consumes enormous
computational resources and time. The limitations on model size every so often make modeling of solid bolts highly impossible. Therefore, other techniques to model bolts are developed and used by many analysts. Montgomery of Siemens Westinghouse Power Corporation lists a number of techniques that can be used to model the bolts effectively. Line elements with coupled nodes and line elements with spider beams are a couple of alternative approaches he proposes. Below are the few methods for modeling pretension bolted joints using the finite element method. Pretension is modeled using ANSYS pretension elements, which can be used on solid or line element types. To account for varying contact distribution along flanges, surface-to-surface contact elements are used. To model bolt head and nut behavior coupled nodes, beam elements, rigid body elements (RBE3), or solids are employed. Solid elements, beam elements, pipe elements, or link elements can be applied to model bolt stud. The choice of line elements versus solid elements is determined by the degree of complexity sought. The following sections will discuss the pros and cons of different simulations.

3.2 Introduction to modeling bolted joint

A typical bolted joint is made up of the bolt group comprising head, stud, and nut and the flange (top and bottom), as shown in Figure 3.1. Bolted connections are designed to clamp two or more parts together of an assembly (Figure 3.2). Because of varying loading conditions, especially high loads (depending on the application), and the various causes like reduction in clamping force, poor thread engagement etc discussed in chapter 2, there is a possibility that the bolted connections can separate. In order to minimize this effect, a preload is applied to the bolt (Figure 3.1). This preload is responsible for
maintaining the joint integrity, provided the applied load remains less than the preload. In finite element simulation, the preload characteristics must be accounted for.

Figure 3.1: Bolted Joint Labels

Figure 3.2: Cylinder Section showing coupling action
It is important for the analyst to determine the bolted joint characteristics to be modeled and understand the capability of the finite element program being used before developing a finite element model. This knowledge helps the analyst to determine how closely he could simulate the bolted joint.

Two primary bolted joint characteristics are pretension and mating part contact (Figure 3.3 and Figure 3.4). Figure 3.3 gives the geometric representation of the applied pretension and the resulting contact surfaces of the assembly shown by the arrows. Similarly in figure 3.4 the arrows represent the head contact, nut contact and flange joint contact. Most of the commercial finite element codes available to us do not have the capabilities to simulate preload and mating part contact status. Ansys workbench provides a promising platform to perform bolt preload analysis. Apart from Ansys
workbench only a few commercial software available to us perform the preload analysis, hypermesh is one such program. Montgomery in his paper [13] describes various methods that can be employed to model pretension in bolts for generic applications. Temperature, constraint equations, or initial strains are generally used to model pretension in bolts. In order to generate temperature pretension different temperatures and material properties are assigned to the bolt and the flange. Using preset temperature values can create the thermal shrinkage effect in the bolt. Constraint equation preload is a special case wherein; equations can be applied to direct the behavior of associated nodes instead of coupling nodes to create an initial displacement of the bolt. The pretension element in ANSYS uses the constraint equations approach. This is automated for the user. The user creates the element and applies the pretension load. Initial strain pretension is the more direct approach. In this approach, an initial displacement is applied to the element. Once the solution starts, the initial displacement is considered as a part of the load on the mode [13].
To account for the contact in the bolted joint point-to-point, point-to-surface, or surface-to-surface elements are used. The contact type depends on the model being used. For solid three-dimensional modeling, the surface-to-surface contact is mostly used. Montgomery illustrates the various situations that require the analyst to define contacts. Following are the three different cases described in his paper.

1. Bolt Under Flange Separation

2. Bolt Under Flange Compression

3. Transverse Direction

For bolt under flange separation, there is no need to define contact elements for the contact surface between flange and head/nut of the bolt. Instead these surfaces can be
bonded or glued together. So that, the contact for head can share the same surface as the top flange, and the contact for the nut can share the same surface as the bottom flange. Further, the contact elements are essential at the horizontal joint between the top and the bottom flange. The bolt does not carry any load when a flange is under compression, and the head and nut contacts are modeled such as they separate from the flanges but the horizontal joint contact can share the same surface. Finally, to account for the transverse load, top flange and the bottom flange are coupled to a node from the line element of the stud.

3.3 Joint simulations

Simulating the bolted connections require the analyst to consider both joint separation and compression in the model. The analyst must have thorough knowledge of bolt behavior and should know the type of results desired. The level of detail to be modeled depends on the results desired from the analysis, for example in models for production applications, the bolt's are modeled to simulate the load transfer function from the top flange to the bottom flange, or vice versa, as the joint tries to separate.

Including the bending and shear effects in the model depends on the accuracy desired. The bolt must be sized to hold the joint intact under the flange separation condition. Montgomery suggests the following simulation techniques that can be used to model the bolted connection in Ansys. The following section will include brief discussion of these techniques with the advantages and disadvantages of each technique.

1. No Bolt Simulation
2. Coupled Bolts

3. RBE (Rigid Body Element) Bolt

4. Spider Bolt

5. Hybrid Bolt

6. Solid Bolt

3.3.1 No bolt simulation

The easiest and fastest way to simulate the bolt preload effects in a bolted connection is the no bolt simulation. In this case the preload is applied as a pressure load on the washer surface without including a bolt in the model. Since the model does not contain bolts the number of elements is less and the simulation take less time to give solution. With no bolt simulation, the analyst assumes that the joint will not separate and the bolt stiffness is not required in the simulation. But without bolt stiffness in the model, the bolt load transfer will not be taken into account. The pass/fail criteria will depend on the contact pressure and the gap, but not on the bolt load. Refer to figure 3.5.
Advantages:

1. Modeling of the bolt is not necessary.

2. The number of elements in the model will be less; hence the solution takes the less time.

3. The most simple and easy way to account for the bolt preload.

Disadvantages:

1. Load transfer through the bolt is not possible since bolts are not modeled

2. Since bolt elements are not modeled bolt stiffness cannot be accounted for.

3.3.2 Coupled bolt

Line elements are used to represent the stud and coupled nodes represent the head/nut in the coupled bolt simulation. The head/nut connected similar to the spider bolt except with coupled nodes instead of line elements. This approach reduces the number of
elements significantly. The Coupled Bolt simulation transfers vertical bolt loads without using line or solid elements. The stud is simulated as a Link10 element, which has tension only capabilities, requiring no contact elements at the head/nut flange connection. Refer to figure 3.6.

Figure 3.6: Example for Coupled Bolt Simulation

Advantages:

1. The number of elements in coupled bolt simulation is more than no bolt simulation but fewer as compared to other simulations

2. The stud section is simple and uses line elements

3. Simulation runs easily and easy to obtain results

4. Coupled nodes are used to transfer the tensile loads.

5. In case of flange compression, Link10 with tension only capability will respond as an actual bolt.
Disadvantages:

1. Head/Nut temperature is not accounted for

2. Bending loads are not transferred

3.3.3 RBE (Rigid Body Element) bolt

Line elements are used to represent the stud and RBE elements are used to represent the head/nut, in RBE Bolt simulation. RBE elements are used instead of line elements for head/nut connection. Refer to the figure 3.7. This approach reduces the number of elements significantly. Line or solid elements are not required to transfers all the loads and incorporate the bending effects in RBE Bolt simulation. Contact elements are not used at the head/nut to flange connection hence a section of the stud line elements should be line elements with tension only capability.

Figure 3.7: RBE Simulation Example
Advantages:

1. The number of elements in no bolt simulation is more than the number of elements in RBE bolt simulation, but less when compared to other simulations. The solution runs faster and it is easy to extract results.

2. RBE nodes transfer tensile, bending, and thermal loads.

Disadvantages:

1. The head/nut temperature is not accounted in this case.

2. The threads are not accounted for in the bolt section.

3.3.4 Spider bolt

The spider bolt simulation substitutes line elements for the head, nut, and stud (Refer to figure 3.8). A series of line elements represent the head/nut in a web-like fashion. Thus, the name spider bolts. It is the most logical approach to using line elements and transferring the loads to the stud. The head/nut bending and stiffness must be simulated by the line elements.

Figure 3.8: Spider Beam Bolt Simulation
A portion of the stud, should be line elements with tension only capability, since no contact elements are used at the head/nut to flange connection.

Advantages:

1. The solutions runs faster and it is easy to extract results in spider bolt simulation, the number of elements is more than no bolt, coupled bolt, and RBE bolt simulations, but less than hybrid bolt and solid bolt simulations.

2. The spider elements are used to transfer the tensile, bending, and thermal loads.

Disadvantages

1. Extra effort is required for simulating head/nut stiffness as compared to other simulations.

3.3.5 Hybrid bolt

In the hybrid bolt simulation, the head and the nut are modeled as solid elements and the stud region is modeled as a line element (Refer to figure 3.9). It is recommended that the line element starting point should be located one half diameter from the top flange edge and one half diameter from the bottom flange edge. The line element captures the tensile part of the bolt load. The line element is attached to the solid using coupled nodes. They are coupled in the bolt's axial direction. In the hybrid bolt simulation, the purpose in keeping the head and nut as solid elements is to incorporate the thermal and bending load effects. The contact elements between flange and head/nut are not required if Link10 elements are used as the line elements. This is because the Link10 elements have the tension only option. That is, if the bolt goes into compression, there is no load in the
Link10 element. Link10's reduces the number of contact elements, but it is required to couple the nodes at the top and bottom flange. Transverse effects are as described in the section on transverse direction. If a degree of freedom problem occurs, it is required to restrain the Link10 elements in the transverse direction. If Beam4 elements are used in place of Link10 elements, it will eliminate the transverse coupling requirement, but it will be required to model the contact elements between flange and head/nut to include the zero compression, which is not available in Beam4 elements [13].

Advantages:

1. This is the second best simulation approach for accuracy after solid bolt simulation. Also it is easy to extract results
2. Stud section is modeled using line elements. The line elements are used to transfer tensile, bending, and thermal loads.
3. Complete stress distribution in head and nut can be computed.

Disadvantages:

1. The line elements are required to be coupled to the stud.
2. The threads are not accounted for in the bolt section.
3.3.6 Solid bolt

The solid bolt is the most realistic and accurate simulation technique used for modeling a bolt. It accounts for thermal, bending, and tensile loads. The solid bolt simulation requires that the contact elements be used for the horizontal joint and the contact surface between the flange and the head/nut (Refer to figure 3.10). Pretension element is used to account for the preload.
Advantages:

1. This is considered as the best simulation approach in terms of accuracy.
   Full stress distribution in head, nut and stud can be calculated

2. *Tensile, bending, and thermal loads can be transferred.*

Disadvantages:

1. The computational time required for modeling and run time is more due to number of solid elements.

2. Contact elements should be defined at head and nut to flange.

3. Friction interaction, at the contact surfaces is not accounted for in this case.
CHAPTER 4

FINITE ELEMENT MODELING AND ANALYSIS

The current study concentrates on the high voltage conductor assembly, which constitutes of high voltage connector and a high voltage conductor which are clamped together by six bolts. This component has been reported to fail after certain number of tests. In order to simulate the entire model enormous computational power is required; hence in this study we used the single unit of high voltage joints.

4.1 Background information about the software used

Ansys Workbench, CAE (Computer-Aided Engineering) software program was used in conjunction with Solid works, 3D CAD (Computer-Aided Design) software program to simulate the behavior of mechanical bodies (high voltage conductor assembly) under structural loading condition. Ansys is capable of conducting the finite element analyses considering both geometric and material non-linearity and also includes interface elements and constraint conditions. The structure was modeled and assembled in Solidworks, and it contains two current joints assembled together using six bolts and six washers. Then the model was imported into Ansys workbench. The model was simulated for three different preload values 2800 lbs, 3800 lbs and 7600 lbs. There are number of methods to simulate the bolted joints to account for the given preload values. The solid bolt techniques described in the previous chapter was used to model the bolted...
connections for FE analysis in this case. The solid bolt technique is considered as the most realistic and accurate technique to model bolts. Some FE software gives the opportunity to apply preload as a function of temperature. FE package ABAQUS allows the use of orthotropic thermal expansion material properties. Jiang et.al, explains this process in their work, a thermal expansion coefficient for axial direction was set to non-zero while the thermal expansion coefficient for other two directions was set to zero, so that the material expanded in only one direction thereby simulating the tightening process. The preload value can be easily adjusted by adjusting either the value of expansion coefficient of the material or the applied temperature [14].

4.2 Model description

The high voltage current joint assembly is made of one high voltage conductor, high voltage connector, 6 bolts and 6 washers. The high voltage conductor and high voltage connector are made up of aluminum alloy (AlT6061) and bolts and washers are made of structural steel. The material properties are listed in the table A.1 and A.2 in appendix. The high voltage conductor and high voltage connector was modeled separately using Solidwoks CAD program. The bolts and washers are modeled separately. These parts were joined using the assembly feature in Solidworks. The model was saved in parasolid format, (.X_T extension) for importing into Ansys Workbench software. The following figure 4.2 is the snapshot from Ansys Workbench.
Figure 4.2: Sectional view of the high voltage current joint assembly

Figure 4.3: High voltage current joint assembly modeled in Solidworks
4.4 Assumptions

The assumptions that were used for this finite element analysis study are listed below:

1. Since the actual geometry is complex (fillets and corners) to generate a fine mesh, a simplified model is assumed. The simplified model eliminated the fillets and corners for the high voltage conductor.

2. To simulate the helical threads enormous CPU hours & and advanced workstations are required, hence the bolts are considered without helical threads.

3. The surfaces of the high voltage conductor and high voltage connector are assumed to be in 100 % contact i.e. there is no air gap present in between them.

According to Lehnhoff and Bunyard [15] incorporating the threads in geometry will have significant difference in stiffness of both bolt and member stiffness. Lehnhoff and Bunyard, found that there were significant differences in both bolt and member stiffnesses when the thread geometry was included. When comparing the results of their research to that conducted by Lehnhoff and Wistehuff, they concluded that the bolt stiffness decreased and the member stiffness increased for all models. Further they explain the decrease in the bolt stiffness is result of the increased flexibility of the bolt due to bending of the threads and the decrease in cross-sectional area of the bolt when the threads are included. Increases in member stiffness occur due to the decrease in the initial member deflection when the bolt preload is applied [15].

The element types used in this FE analysis are Solid 186, Solid 187, Conta174, Targe170 and Surf154. The detailed description of each element type and corresponding figures are provided at the end of this chapter.
4.5 Mesh generation

In general, FE software use “common mesh technique” to mesh assemblies, this technique takes assembly geometry and produces one continuous finite element mesh throughout the structure, assigning different material properties to the different parts that comprise the assembly. This technique is suitable for perfectly bonded parts like welded components and is not suitable for simulating the bolted joint assembly. The other meshing techniques available are

1. Node to Node
2. Node to surface
3. Surface to Surface

The following figure 4.3 gives a pictorial representation of each of the above mentioned techniques.

Figure 4.4: (1) Node to Node (2) Node to Surface (3) Surface to Surface

The geometry consists of numerous curves and round surfaces, which are extremely difficult to mesh using any single element type. Hence mixed element types were used to mesh the model. The meshed model consists of quadratic and triangle elements. Figure 4.4 gives the Ansys workbench screen shot of the meshed model.
Figure 4.4: Mesh Generated Using Ansys

The mesh statistics for individual parts are tabulated and are shown in the following table 4.1. The total elements in the model are 50961 and the total number of nodes in the model is 74543.

<table>
<thead>
<tr>
<th>Serial #</th>
<th>Component</th>
<th>Number of nodes</th>
<th>Number of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>High Voltage Conductor</td>
<td>34,260</td>
<td>22,387</td>
</tr>
<tr>
<td>2</td>
<td>High voltage Connector</td>
<td>15,867</td>
<td>16,100</td>
</tr>
<tr>
<td>3</td>
<td>Bolt</td>
<td>22,834</td>
<td>12,288</td>
</tr>
<tr>
<td>4</td>
<td>Washer</td>
<td>1,582</td>
<td>186</td>
</tr>
</tbody>
</table>

4.6 Contacts

The contacts between various components in the assembly are defined automatically. Contact boundaries can be automatically formed where parts meet. The application has the ability to transfer structural loads and heat flows across the contact.
boundaries and to “connect” the various parts. Manual contact definition is extremely
time consuming and prone to error, especially when the number of contacting surfaces
are more. Automatic contact recognition helps to recognize parts that are close together
and then automatically set-up contact between them. In addition to the automatic contact
capability, the ANSYS Workbench environment also provides numerous tools for
manually editing the contact surfaces and also specifying the types of contact conditions
that exist in the model. The use of manual contact options can be justified in the
following cases:

1. If the auto detection does not create the contact regions that are necessary for the
   analysis, then the required contact regions can be added.
2. If the auto-detection creates more contact pairs than necessary the analyst can
   delete the unnecessary contact regions.

In this case the auto detection created the required contact regions for the analysis.

4.7 Boundary conditions

The model is properly constrained and the contacts between various parts of the
assembly are defined. The surface of the assembly (high voltage conductor) that is
attached to the transmission system is fully constrained in all directions. Figure 4.6 shows
the surface of assembly that is constrained. The automatic contact definition function was
used to define the contact between each part. The preload values used in this study are
obtained from the mechanical engineering department at Los Alamos national laboratory.
Initially the preload value of 2,800 lbf was, applied to the six bolts that join high voltage
conductor and high voltage connector. In the initial application a torque of 205 in-lb is
used to create this preload on the screws. The torque was increased to 260 in-lb and 370 in-lb in later applications and the corresponding preload values were 3,800 lbf and 7,600 lbf respectively.

The equation used to calculate the preload value is:

\[0.2 \times \text{diameter (inches)} \times F \text{ (lbs)} + 20 = \text{Torque}\] (2)

Where

a) 0.2 is used as the screw/nut friction coefficient.

b) 20 in-lb constant is the torque found (empirically) in these self-locking (made out-of-round) inserted steel nuts.

Substituting the torque values of torque of 205 in-lb, 260 in-lb and 370 in-lb in the above equation we have the corresponding preload values as 2,800 lbf, 3,800 lbf and 7,600 lbf.

4.8 Loading condition

A pretension bolt load option is added to the existing list of structural loads. Users have the option to apply the load as a preload force or a preadjustment length. The design simulation runs a two load step problem where pretension load are applied in the first step, then they are locked in the second step when other working loads are applied [16]. The six bolts in the model were preloaded using the pre tension option from Ansys Workbench software. In the first case a preload of 2,800 lbs was applied to each of the bolts. The preload values used in subsequent applications were 3,800 lbs and 7,600 lbs respectively. After applying the preload the model is solved using Ansys Workbench.
solver to obtain stress, strain, and deformation patterns. Figure 4.5 shows the preload applied to the model.

It is estimated that 50-90% of structural failure is due to fatigue; hence it is important to take fatigue analysis into account. In general fatigue analysis is time and money consuming, since there are no quality fatigue design tools available. Ansys provides a fatigue tool that helps the analyst to perform fatigue failure analysis. A stress-life approach has been adopted for conducting a fatigue analysis. Several options such as accounting for mean stress and loading conditions are available. The fatigue tool provides
the user to obtain outputs that include fatigue life, damage, factor of safety, stress, fatigue sensitivity etc. which will be discussed in detail in the following chapter along with the results of this study.
CHAPTER 5

RESULTS AND DISCUSSIONS

5.1 Structural results

The model is simulated for three preload values and the resulting maximum and minimum stresses, strains and deformations are tabulated and shown below in the table 5.1. As expected, it can be seen that with the increase in preload value the values of stress, strain and deformation increase correspondingly. In general the magnitude of stress, strain and deformation was high on the bolts and washers when compared to other components of the assembly. Since the preload was applied on the bolts, the stress and strain on the high voltage conductor and high voltage connector was minimum. Similarly the deformation was very less on the high voltage conductor and high voltage connector, and the deformation at the contact surface of these two parts, particularly around the bolt holes was slightly high.

<table>
<thead>
<tr>
<th>Preload (lbf)</th>
<th>Stress (Max) (PSI)</th>
<th>Stress (Min) (PSI)</th>
<th>Strain (Max) (In)</th>
<th>Strain (Min) (In)</th>
<th>Deformation (Max) (In)</th>
<th>Deformation (Min) (In)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2800</td>
<td>28,277.26</td>
<td>1.76</td>
<td>1.27x10-3</td>
<td>1.71x10-7</td>
<td>6.72x10-3</td>
<td>0</td>
</tr>
<tr>
<td>3800</td>
<td>38,375.97</td>
<td>2.39</td>
<td>1.72x10-3</td>
<td>2.32x10-7</td>
<td>9.12x10-3</td>
<td>0</td>
</tr>
<tr>
<td>7600</td>
<td>78,772.39</td>
<td>4.91</td>
<td>3.53x10-3</td>
<td>4.77x10-7</td>
<td>1.87x10-2</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 5.1: Stress Strain and Deformation for Each Loading Condition
Figure 5.1: The deformation contours on the high voltage conductor

Figure 5.1 gives the deformation contours on the high voltage conductor surface. It can be seen that the maximum deformation is on the bolts and around the bolt holes. Similar contours were observed in each case.

5.2 Results for Contact Status

Ansys workbench provides tools to monitor the contact status of the assemblies under consideration. The contact Status codes defined by number 0 through 3 define whether the components of the assembly are in contact or not. Ansys workbench uses the following status codes:

- 0-open and not near contact
- 1-open but near contact
- 2-closed and sliding
- 3-closed and sticking

The labels Far, Near, Sliding, and Sticking are included in the legend for Contact Status. The contact status was defined for all the three loading conditions. The contact status results showed that there was sliding at the interface of two components (High voltage conductor and high voltage connector) of the assembly. Figure 5.1 and figure 5.2 show the contact status. Sliding indicates that there is a loosening of the bolts. The reason for this kind of loosening can be the result of clamping force reduction as described in the chapter 2.

Figure 5.2: Contact Status at the Interface of Two Bodies in the Assembly
Another option, contact sliding distance can be used to estimate the sliding distance between the two bodies in the assembly. It was observed that the sliding distance increased with increase in preload. For initial preload of 2,800 lbf the contact sliding distance was observed to be 0.0172e-4 inches and for the next loading condition of 3,800 lbf the sliding distance increased to 0.233e-4 inches and for the last case of 7,600 lbf preload the sliding distance was found to be 0.467e-4 inches. Even tough the magnitude of the sliding distance is very less; still this can be used to explain the phenomenon of loosening at the contact surface of the two bodies in the assembly.

The explanation for this kind of behavior from the literature is provided in this section. Rudy Alforque, [18] points out that the separation of the clamped parts will occurs when the resultant bolt load (W) equals the effective load (We) on the joint. The 45 degree line from the origin in figure 5.3, illustrates this phenomenon. This point
corresponds to the point when the mating parts no longer share the applied load and the bolt carries the entire load till the point when either the bolt or the joint fails.

![Diagram](image)

**Figure 5.4: Resultant Bolt Load vs. Effective Load on Joint**

In this case since the simulation results points out that separation occurs in each case but there is no bolt failure observed. The contact status shows sliding at the interface of the high voltage conductor and high voltage connector. However at this time there are no tools in this software available to us that measure the magnitude of reduction in the clamping force, but by using the contact sliding distance option we can calculate the magnitude of the sliding distance between the two bodies in the assembly. This type of loosening is similar to the stage -I self loosening which is caused due to localized cyclic plastic deformation that in turn causes the stresses to redistribute in the bolt and the results in gradual loss of clamping force. There are very few tools and techniques that are available to experimentally measure the reduction in clamping force such as using
piezoelectric actuators and sensors and ultrasonic wave propagation. According to Bolt Science [1], the common causes of the relative motion in bolted joint threads are:

1. Component bending that results in forces being induced at the friction surface. If slip occurs, the head and threads will slip, which can lead to loosening.
2. Differential thermal effects caused by either differences in temperature or differences in clamped materials.
3. Applied forces on the joint that can lead to shifting of the joint surfaces and eventually to bolt loosening.

5.3 Fatigue results

It is estimated that 50-90% of structural failures are due to fatigue, thus there is a need for quality fatigue design tools. Apart from the stress, strain and deformation the Ansys workbench has fatigue tool. The fatigue tool option provides the user with the option of damage, safety factor and fatigue sensitivity. The fatigue analysis results are provided in the following section of this chapter.

5.3.1 Damage

Fatigue damage is defined as the design life divided by the available life. The default design life may be set through the Control Panel in the Ansys Workbench. In this case the design life of the machine is 5 years or 60 months. A damage of greater than 1 indicates the part will fail from fatigue before the design life is reached. The design life was set at 60 months and clearly in this case the damage value is less than 1 (table 5.2) for each case and this indicates that the parts will not fail from fatigue before the design life is reached.
Table 5.2: Magnitude of Damage for Three Loading Conditions

<table>
<thead>
<tr>
<th>Name</th>
<th>Scope</th>
<th>Design Life (Months)</th>
<th>Minimum</th>
<th>Maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;Damage&quot; All Bodies In &quot;Model&quot;</td>
<td>60.0</td>
<td>6.0x10^-7</td>
<td>8.42x10^3</td>
<td></td>
</tr>
<tr>
<td>&quot;Damage&quot; All Bodies In &quot;Model&quot;</td>
<td>60.0</td>
<td>6.0x10^-7</td>
<td>2.17x10^2</td>
<td></td>
</tr>
<tr>
<td>&quot;Damage&quot; All Bodies In &quot;Model&quot;</td>
<td>60.0</td>
<td>6.0x10^-7</td>
<td>0.14</td>
<td></td>
</tr>
</tbody>
</table>

5.3.2 Safety factor

Safety factor is the ratio of the yield strength of the material to the corresponding significant stress due to the applied load. Safety factor can also be defined in terms of loads as ratio of design overload to normal overload [19]. This result is a contour plot of the factor of safety (FS) with respect to a fatigue failure at a given design life. The maximum FS reported is 15. The criterion for interpreting Safety Factor is as follows:

1. A value of 1 or lower indicates that material yield will most likely occur.
2. A value of 3 to 4 is ideal.
3. A value greater than 4 indicates an over design.

Table 5.3 gives the observed maximum and minimum safety factor values. Clearly in this case there is a comfortable Safety Factor and an indication that we can experiment with a lighter materials. For each loading condition the factor of safety was found to be greater than 1.
Table 5.3: Minimum and Maximum Values of Safety factor for Three Scenarios

<table>
<thead>
<tr>
<th>Name</th>
<th>Scope</th>
<th>Design Life (Months)</th>
<th>Minimum</th>
<th>Maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;Safety Factor&quot;</td>
<td>All Bodies In &quot;Model&quot;</td>
<td>60.0</td>
<td>6.0</td>
<td>15.0</td>
</tr>
<tr>
<td>&quot;Safety Factor&quot;</td>
<td>All Bodies In &quot;Model&quot;</td>
<td>60.0</td>
<td>4.42</td>
<td>15.0</td>
</tr>
<tr>
<td>&quot;Safety Factor&quot;</td>
<td>All Bodies In &quot;Model&quot;</td>
<td>60.0</td>
<td>2.21</td>
<td>15.0</td>
</tr>
</tbody>
</table>

5.3.3 Fatigue sensitivity

This plot shows how the fatigue results change as a function of the loading at the critical location on the scoped region. Sensitivity may be found for life, damage, or factory of safety. For instance, if the analyst sets the lower and upper fatigue sensitivity limits to 50% and 150% respectively, and the scale factor to 3, this result will plot the data points along a scale ranging from a 1.5 to a 4.5 scale factor. The analyst can specify the number of fill points in the curve, as well as choose from several chart viewing options (such as linear or log-log). See appendix for fatigue sensitivity graphs for other loading conditions. Figure 5.4 & 5.5 gives the fatigue sensitivity analysis of damage and safety factor for the first loading condition, the fatigue sensitivity for damage increases with increase in loading variation whereas the safety factor decreases with increase in loading variation.
Figure 5.4: Fatigue Sensitivity Analysis for Damage
This chapter has provided with the results obtained from the FE analysis of the high voltage current joint assembly. The results obtained in this study give an estimate of how the multi bolt joints behave with varying preload. In order to obtain the exact behavior of the multi body bolted joints enormous computational resources are required, which is out of scope for the current project. For example in order to simulate a single bolted joint with helical threads in ABAQUS it takes about 200 CPU hours for completing one simulation of 32 loading cycles, using Origin 2000 computers in NCSA at the University of Illinois[14]. The conclusions of this study and the scope for future work will be discussed in the next chapter.

Figure 5.5: Fatigue Sensitivity Analysis for Safety Factor
CHAPTER 6

CONCLUSIONS AND SCOPE FOR FUTURE WORK

6.1 Conclusions

In this study, the effect of preload on the multi bolt assembly is studied. The common modes of failures in bolted joints such as reduction in clamping force, self-loosening mechanism, poor thread engagement etc are discussed in detailed. Various techniques that can be used to model the bolted connections for finite element analysis are discussed. The main conclusions and contributions of this study can be summarized as follows:

- The comprehensive literature survey of the importance of preload in the bolted connections is presented. Major failure modes and causes for bolted joint failure such as self-loosening of bolted joints, poor thread engagement etc are explained in detail.

- Very useful techniques to model bolted connections for FE analysis are introduced, such as hybrid bolt technique, spider bolt technique, etc. The pretension effects of multibolt assembly are modeled and simulated using Ansys Workbench software. The Solid bolt technique is used in this study.

- The FE analysis was successful in simulating the stage–I, self-loosening phenomenon observed in the bolted connections. The bolted connection showed that there is loosening at the interface of the high voltage conductor and high
voltage connector assembly. This loosening can be attributed to loss of clamping force, which is common phenomenon in the bolted connections and is widely published in literature. However there are currently no tools available in Ansys to predict the magnitude of the reduction in clamping force. There are few experimental techniques to measure the magnitude of loss of preload in the assembly; using piezoelectric crystals and ultrasonic wave propagation are some of the available techniques.

- The loosening of the bolted connections is further verified by plotting the contact sliding distance for the assembly. It was observed that the contact sliding distance increased in each of the three loading conditions. For the first case the sliding distance was 0.0172e-4 and in the subsequent loading conditions the sliding distance increased to 0.2333e-4 and 0.467e-4 respectively.

- Additionally, the fatigue tool is used to monitor damage, safety factor and fatigue sensitivity. As expected the fatigue sensitivity for safety factor decreased with increase in loading variation whereas the fatigue sensitivity for damage decreased with increase in the loading variation.

6.2 Future work

This study provides a promising platform for starters to analyze the multi-bolt assemblies using FE analysis. While analyzing the effects of preload on the multi-bolt assemblies we made some assumptions to suit the available computational resources to us. For example this study did not account for helical threads for the bolts, but in order to
accurately simulate the loosening behavior of the bolted connections the bolt threads should be considered.

For the current FEA model, the interfaces of the two parts are assumed to be in 100% contact, but in reality the real contact area is less than 100% of the calculated area due to the existence of surface roughness on the two parts. Due to the presence of valleys and peaks (due to surface roughness) and the air gaps resulting try to close there is a possibility of micro arcing. Hence, a modified model with consideration of actual surface roughness values on the interfaces of the two parts should be developed as shown in the figure 6.1. Since the surface roughness values are of the order of micrometers or micro inches it is difficult to model such surfaces. Therefore the model used to study the effects of surface roughness should be scaled. The FEA analysis should focus on the gap size and depth study on the interface and how those gap changes with the preloading and machining quality (Ra). The importance of this study is because those gaps may cause electrical arcing on the interfaces, which is considered as one of the major reasons for inter-surface damage. The arcing may result in deforming the whole surface and may even deform the bolts that join these surfaces. The modified FEA model and its analysis will provide important guidelines to avoid any possible arcing caused surface damages and guidelines for better design, manufacturing, and assembly requirements.
Micro arcing will occur across the air gaps between the peaks when contacts are closing. The arcing phenomenon is graphically illustrated using the figure 6.2.
Finally, improving the mesh size will definitely result in more accurate results. Since the current FE software is the educational version there is a limitation on the number of nodes and elements. The limit on number of nodes for the available software is 75000 nodes and the current model consists of 74543 nodes which correspond to about 99% of the available capacity of the software. In order to simulate complex models such as the model in this study, more elements and nodes are required to get desired results.
## APPENDIX 1. MATERIAL PROPERTIES

### Aluminum Alloy Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Value</th>
<th>Temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus of Elasticity</td>
<td>Temperature-Independent</td>
<td>1.03×10^7 psi</td>
<td></td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>Temperature-Independent</td>
<td>0.33</td>
<td></td>
</tr>
<tr>
<td>Mass Density</td>
<td>Temperature-Independent</td>
<td>0.1 lbm/in³</td>
<td></td>
</tr>
<tr>
<td>Coefficient of Thermal Expansion</td>
<td>Temperature-Independent</td>
<td>9.44×10^-6 1/°F</td>
<td></td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>Temperature-Dependent</td>
<td>1.52×10^-3 BTU/s-in.°F</td>
<td>-148.0 °F</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>Temperature-Dependent</td>
<td>1.93×10^-3 BTU/s-in.°F</td>
<td>32.0 °F</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>Temperature-Dependent</td>
<td>2.21×10^-3 BTU/s-in.°F</td>
<td>212.0 °F</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>Temperature-Dependent</td>
<td>2.34×10^-3 BTU/s-in.°F</td>
<td>392.0 °F</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>Temperature-Independent</td>
<td>0.21 BTU/lbm.°F</td>
<td></td>
</tr>
</tbody>
</table>

Table A.1: Material Properties for aluminum alloy (Materials Library: Ansys workbench)

### Structural Steel Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus of Elasticity</td>
<td>Temperature-Independent</td>
<td>2.9×10^7 psi</td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>Temperature-Independent</td>
<td>0.3</td>
</tr>
<tr>
<td>Mass Density</td>
<td>Temperature-Independent</td>
<td>0.28 lbm/in³</td>
</tr>
<tr>
<td>Coefficient of Thermal Expansion</td>
<td>Temperature-Independent</td>
<td>6.67×10^-6 1/°F</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>Temperature-Independent</td>
<td>8.09×10^-4 BTU/s-in.°F</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>Temperature-Independent</td>
<td>0.1 BTU/lbm.°F</td>
</tr>
</tbody>
</table>

Table A.2: Material properties for structural steel (Materials Library: Ansys workbench)
APPENDIX 2

A 2.1 Element description [17]

SOLID186 element description

SOLID186 is a higher order 3-D 20-node structural solid element. SOLID186 has quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems). 20 nodes having three degrees of freedom per node define the element: translations in the nodal x, y, and z directions. SOLID186 may have any spatial orientation. The element supports plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

Figure A2.1:SOLID186 Geometry
A 2.2: SOLID187 element description

SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

![Figure A2.2:SOLID187 Geometry](image)

A2.3: CONTA174 element description

CONTA174 is used to represent contact and sliding between 3-D "target" surfaces (TARGE170) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analyses. This element is located on
the surfaces of 3-D solid or shell elements with midside nodes (SOLID87, SOLID90, SOLID92, SOLID95, SOLID98, SOLID122, SOLID123, SOLID186, SOLID187, SOLID191, SOLID226, SOLID227, HYPER158, VISCO89, SHELL91, SHELL93, SHELL99, SHELL132, and MATRDC50). It has the same geometric characteristics as the solid or shell element face with which it is connected (see Figure "CONTA174 Geometry" below). Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb and shear stress friction is allowed. Other surface-to-surface contact elements (CONTA171, CONTA172, and CONTA173) are also available.

![CONTA174 Geometry](image)

Figure A2.3: CONTA174 Geometry

A2.4: TARGE170 element description

TARGE170 is used to represent various 3-D “target” surfaces for the associated contact elements (CONTA173, CONTA174, and CONTA175). The contact elements themselves overlay the solid elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE170. This target
surface is discretized by a set of target segment elements (TARGE170) and is paired with its associated contact surface via a shared real constant set. You can impose any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. You can also impose forces and moments on target elements.

To represent 2-D target surfaces, use TARGE169, a 2-D target segment element.

For rigid target surfaces, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid elements describing the boundary of the deformable target body.

![Diagram of TARGE170 Geometry](image)

**Figure A2.4: TARGE170 Geometry**

**A2.5: SURF154 element description**

SURF154 may be used for various load and surface effect applications. It may be overlaid onto an area face of any 3-D element. The element is applicable to 3-D structural analyses. Various loads and surface effects may exist simultaneously.
Figure A2.5: SURF154 Geometry
APPENDIX 3

The following graphs give the fatigue sensitivity for the remaining two loading conditions.

Figure A3.1: Fatigue Sensitivity Analysis for Preload Value of 3800 lbf (Damage)
Figure A3.2 Fatigue Sensitivity Analysis for Preload Value of 3800 lbf (Safety Factor)
Figure A 3.3: Fatigue Sensitivity Analysis for Preload Value of 7600 lbf (Damage)
Figure A3.4: Fatigue Sensitivity Analysis for Preload Value of 7600 lbf (Safety factor)
REFERENCES


7. R.A. Ibrahima,* C.L. Pettith,” Uncertainties and dynamic problems of bolted joints and other fasteners”,2003, journal of sound and vibration,pg1-80


70


17. Ansys 8.0, documentation retrieved from Ansys help.

VITA

Graduate College
University of Nevada, Las Vegas

Kiran K Pusthay

Address:
4236 Chatham Circle Apt#1
Las Vegas, Nevada 89119

Degrees:
Bachelor of Technology, 2001
Jawaharlal Nehru Technological University, India

Publications:


Thesis Title:
Finite Element Analysis of Failure of High Voltage Current Joints

Thesis Examination Committee:
Chairperson, Dr. John Wang, Ph. D.
Committee Member, Dr. Brendan J. O'Toole, Ph. D.
Committee Member, Dr. Ajit Roy, Ph. D.
Graduate Faculty Representative, Dr. Yahia Baghzouz, Ph. D.