Development of Accurate Finite Element Models and Testing Procedures for Bolted Joints in Large Caliber Gun Weapon Systems

Michael Koehler[1], Gregory Fish[1], Ryan Braughler[1]

[1]U.S. Navy, Naval Surface Warfare Center – Dahlgren Division

1 Abstract

Large caliber weapon systems, such as those mounted on tracked vehicles or in turrets on ships, typically develop large forces when fired, which are transmitted to the structure that houses them. In the case of modular weapon systems, often the platform for the weapon system is already developed and the weapon system is integrated at a later stage. In this scenario, it becomes imperative to fully quantify the loads experienced in the bolted connection between the weapon system and the platform in order to ensure a satisfactory fatigue life of the platform. Typically, this is accomplished through the use of finite element analysis in the early stages of development and verified through testing during later stages of development.

This work investigates the development of finite element models in LS-DYNA that accurately model bolted joints and compares the tradeoff between efficient models with a coarse mesh and more accurate models with an expensive, fine mesh. Furthermore, these models are validated against an instrumented test specimen in order to determine the accuracy of the finite element models. Throughout this comparison, various sensors are used in multiple locations in order to demonstrate that various conflicting results may be obtained from the same load applied on the joint depending on the type of sensor used and the sensor's location in the joint.

2 Introduction

Within the defense industry, it has become increasingly common to develop high fidelity finite element models of large weapon systems in order determine the response of the system to a variety of scenarios. Typically, the finite element model then provides a verification of the design and helps to establish the necessary confidence to build and test a prototype system. This prototype system is then used to confirm the behavior of the system through incremental testing. During the testing of the prototype system, verification of the finite element model occurs through close collaboration between test engineers and analysts. This ideally occurs through well thought out placement of instrumentation on the test article based on key reaction points on the system or areas of concern, with the data collected from the testing used to verify the finite element model.

On large platforms such as ships and aircraft, the design of the overall system is divided among multiple groups, often belonging to various organizations that are contracted for portions of the design. In order to ensure that the integration of weapon system does not result in an adverse impact to the platform, controls are often on the interface between the platform and the weapon system. This typically occurs through an interface control document that, among other things, establishes the mounting configuration for the component (i.e. bolt pattern and sizing) and the load limits at selected locations, typically areas where the weapon system is bolted to the platform.
While analyzing the weapon system, in this case a large caliber gun weapon system, the analyst must evaluate the interface between the weapon system and the platform to ensure that the design loads of the interface are not exceeded during a variety of operations within the system. Often, these load scenarios will vary from the dynamic loading imparted from the gunfire to unique scenarios tailored to the platform. Examples include a shock wave imparted to a ship due to a near miss explosion or a crash landing of an aircraft. This paper discusses various ways that these critical bolted joints may be analyzed in the finite element analysis software LS-DYNA. The focus of this discussion is developing efficient representation of bolted joints at the interface that minimizes computational cost without sacrificing accuracy, investigating common testing procedures, and correlating the data with a finite element model.

3 Behavior of Bolted Joints

A common analytical approach to analyzing bolted joints, specifically determining the load imparted on the bolt and the load imparted into the joint, is to use Shigley's Frustum of a Hollow Cone [1] approach. This method models the stress field imparted into a joint as the frustum of a hollow cone and the stiffness of each layer is determined. Through this method, each layer of the joint, represented by a frustum of a hollow cone, can be modeled as a spring with the entire joint becoming a series of springs in series. Furthermore, the bolt itself behaves as a spring as well, resulting in the bolted joint being analyzed as two springs in parallel, the first being the bolt and the second being the series of springs that represents the joint. As seen in Figure 1, once the spring coefficients are determined for each layer of the joint and for the bolt, the distribution of force, F, imparted into the bolt and the joint can be determined based on the ratio of spring coefficients, k.

$$\frac{F_{bolt}}{F_{joint}} = \frac{k_{bolt}}{k_{joint}} \tag{1}$$

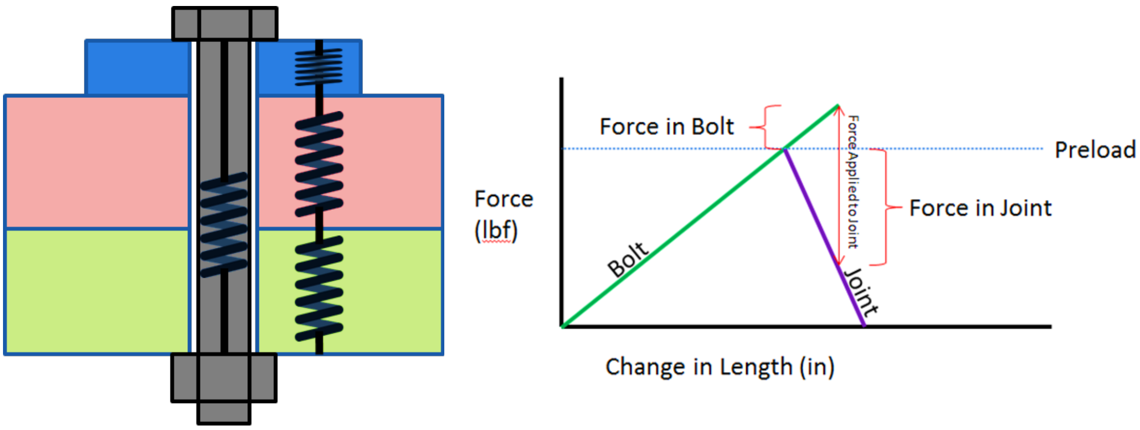

Fig.1: Bolted joint modeled as a series of springs using Shigley's frustum of a cone approach

Although Shigley's method has been used for more than half a century, a variety of issues exist with the method that may limit its practical use [2]. Some of these issues include determining the proper frustum angle and the initial diameter of the frustum, while other limitations exist regarding the size of the joint, locations where the frustum would fall outside of the joint, or the use of dissimilar metals in the joint. Even if Shigley's method for analyzing a bolted joint may not be suitable for complex joints, it provides a fundamental understanding of the behavior of bolted joints and how an applied load will be transmitted through the joint.

4 Bolted Joint Test Setup

In order to determine the optimal modeling approach for a finite element model, a test setup was developed that would allow for the bolt load or joint load to be measured, given a known excitation. In order to accomplish this, an MTS Systems Corporation 810 Material Test System was used to provide a tensile loading. The joint to be analyzed consisted of a top plate composed of 6061-T6 aluminum and a bottom plate, with a threaded hole, consisting of ASTM A36 steel. The two plates were bolted together with a NAS screw and washer. In order to measure loads in the bolted joint, a Dytran1210C2 load cell was used; that particular load cell provides sufficient response to dynamic loading conditions. The test setup was arranged in the two configurations seen in Figure 2. The first test configuration, Test Setup A, placed the load cell between the aluminum and steel plates, while the second configuration, Test Setup B, placed the load cell between the head of the bolt and the aluminum plate. For each test, the bolt was preloaded to 6,800 lbf of axial preload.

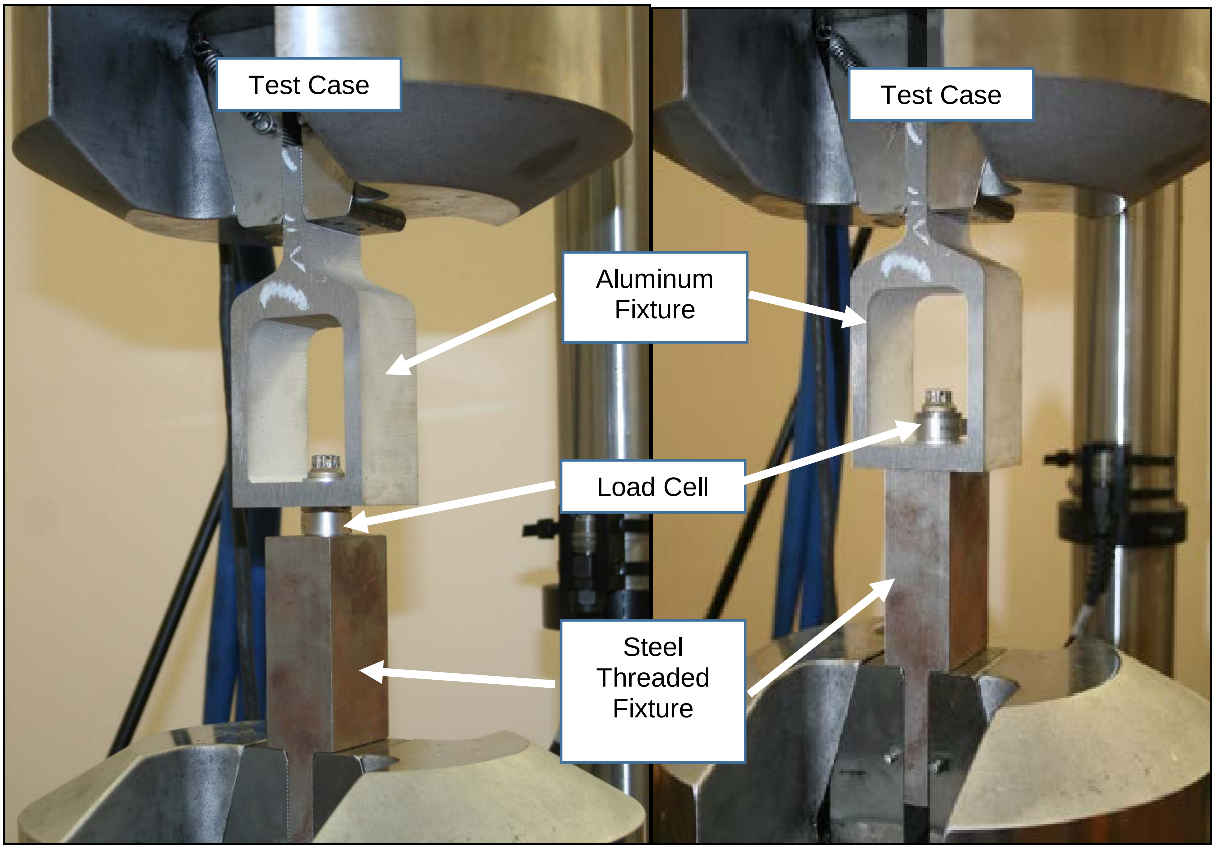

Fig.2: Test Setup A, depicted on the left, with the force transducer placed between the steel and aluminum plates and Test Setup B, depicted on the right, with the force transducer placed above the aluminum plate.

In order to test the joint, a dynamic load was applied to the specimen that would mimic the dynamic response that might be seen in a bolted joint in the interface between a platform and a weapon system resulting from the firing of a large caliber gun weapon system. The load profile consisted of the specimen at rest, followed by a tensile force ramping to 5,000 lbf over a period of 15ms, followed by a dwell period at the 5,000 lbf. A graph of the load applied to the test specimen can be seen in Figure 3.

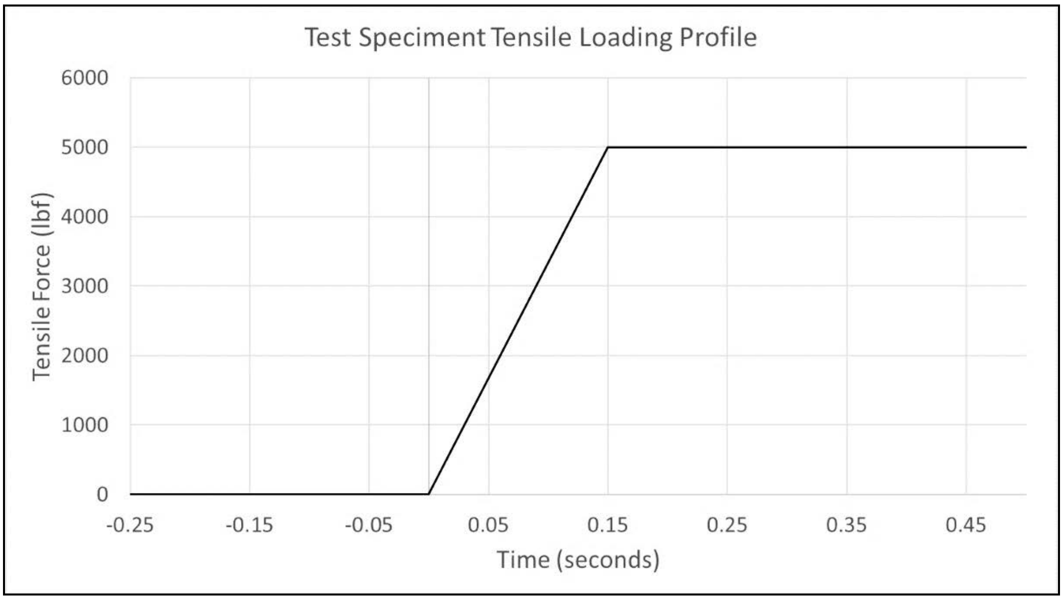

Fig.3: Tensile load profile of test specimen.

Data was collected from both the load cell on the MTS tensile tester, in order to confirm the 5000 lbf load was applied, and from the Dytran loadcell, in order to determine the load in the bolted joint.

5 Test Results

The results from Test Case A can be seen in Figure 4, which depicts both the load applied and the load experienced in the bolted joint between the aluminum and steel plates. The positive value for the force transducer, due to the setup of the transducer, is indicative of a loss of preload on the transducer. In essence, the 6,800 lbf preload decreased by 3,830 lbf as the load was applied. This indicates that the force transducer, located in this configuration, is measuring the load in the joint and, as the joint is being pulled apart, the load on the force transducer is decreasing.

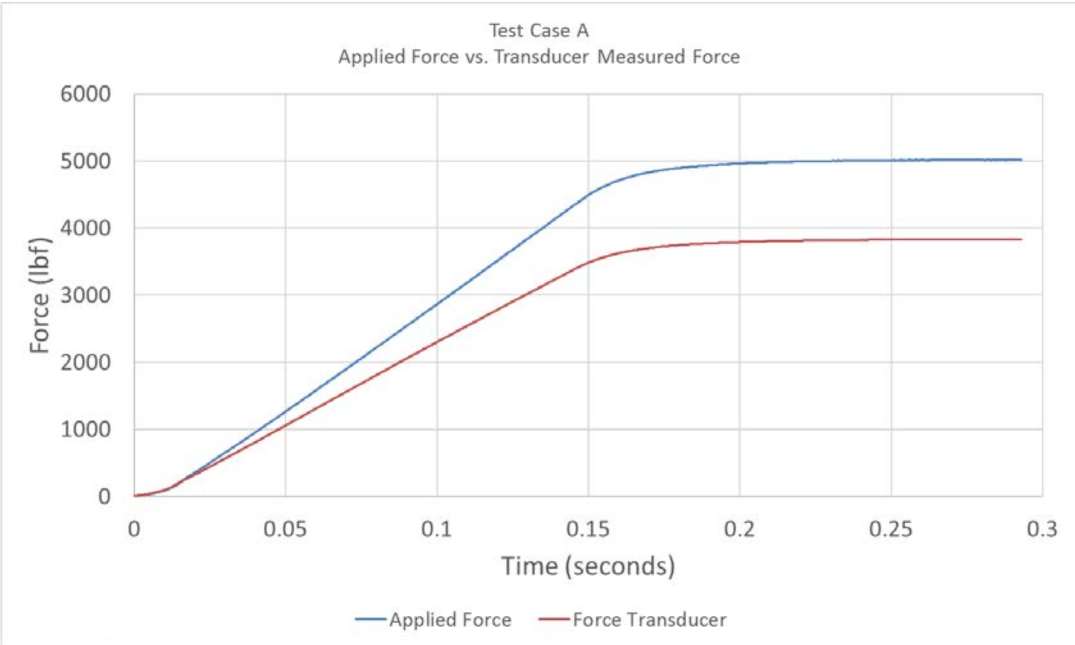

Fig.4: Graph of the applied force to the entire bolted joint and the load measured in the force transducer for Test Case A.

The results for Test Case B can be observed in Figure 5, with the graph showing the load measured by the load cell on the MTS and the load measured by the transducer located between the bolt head and the aluminum plate. The negative value for the transducer indicates a compressive force. Figure 6 shows the graph of the transducer data scaled so it can be easily viewed, with the load in the transducer leveling at 313 lbf. Due to the load being compressive and relatively small, it can be surmised that the load being measured in this scenario is the load being experienced within the bolt.

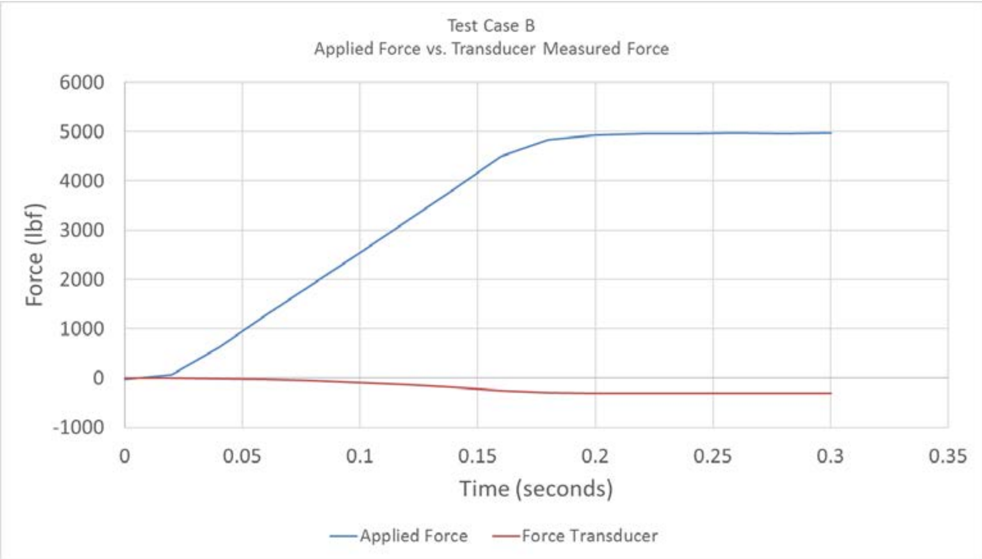

Fig.5: Graph of the applied force to the entire bolted joint and the load measured in the force transducer for Test Case B.

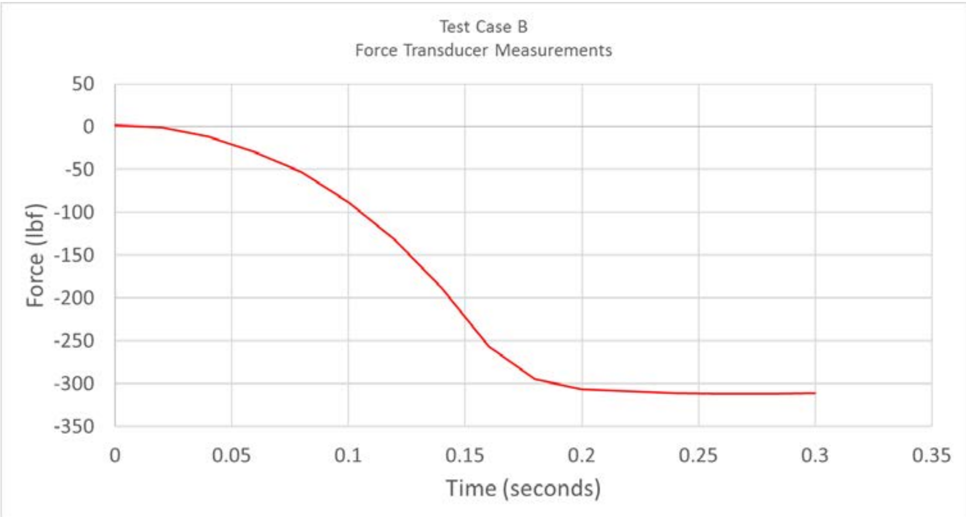

Fig.6: Load measured in the force transducer for Test Case B.

As seen from Test Case A and Test Case B, what might be assumed as minor adjustments to the location of the load cell can have somewhat drastic effects on what is actually being measured. Furthermore, as we know from the behavior of a bolted joint, if a 5,000 lbf tensile load is placed on the joint, a portion of that load will be experienced in the bolt while the remainder is passed through the joint. An interesting point worth noting is that with these two configurations, Test Case A experienced a load of 3,830 lbf as measured in the joint, while Test Case B experienced 313 lbf as measured in the bolt. As it can be seen, these two values clearly do not add to 5,000 lbf. This is a classic example of the observer effect. In essence, the very act of adding a force transducer to the joint has changed the results and this effect is further intensified by changing the location of the force transducer between the two tests. Furthermore, as it might be expected, both scenarios do not provide the total reaction load at the joint, but only a portion of the load. As an analyst works to verify a complex model with test data, it is clear that they must have a clear understanding of the following: a) what is actually being measured by the instrumentation; and b) the effect that the instrumentation may have on data being collected.

6 Finite Element Model

A finite element model of bolted joint fixture was created for both Test Case A and Test Case B, as seen in Figure 7. The lower steel portion of the joint was secured with Single Point Constraints bound in all 6 degrees of freedom in the area that would be secured by the grip on the MTS machine. Since the MTS machine was commanded by a force curve during the experimental test, this was replicated in LS-DYNA using the card ***LOAD_NODE_FORCE** on the nodal locations on the upper aluminum portion in the location that the upper grip would clamp the fixture. A load curve was developed using the keyword ***DEFINE_CURVE** that matched the load curve used in the experimental setup [3].

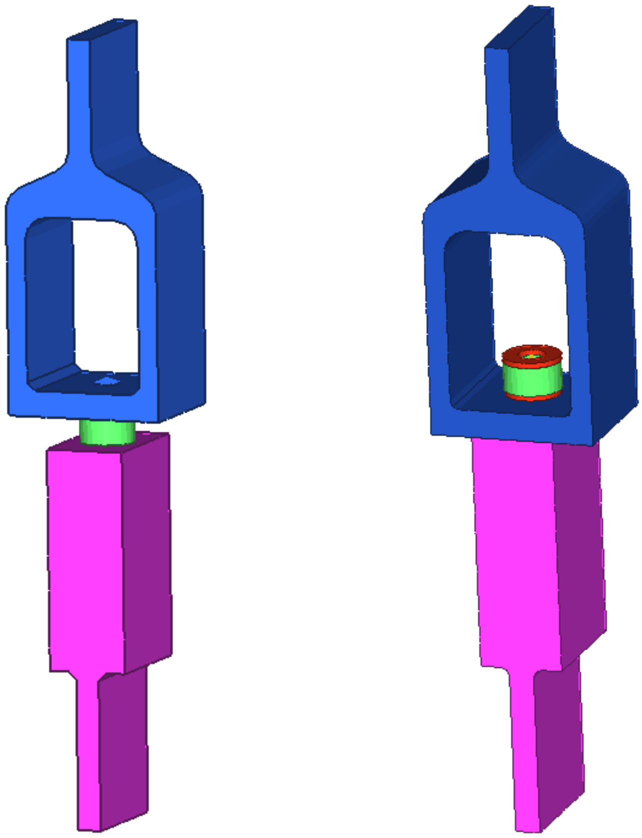

Fig.7: Finite Element Model, Test Case A depicted on the left and Test Case B depicted on the right.

All components were modeled using hexahedral elements with the exception of the bolt, which was modeled using a 1D beam and ***CONSTRAINED_NODAL_RIGID_BODY** for the bolt head and bolt threads [4,5]. Furthermore, the bolt preload of 6,800 lbf was applied during dynamic relaxation using the card ***INITIAL_AXIAL_FORCE_BEAM** [4,5]. Finally, contact between the components was modeled using ***CONTACT_AUTOMATIC_SURFACE_TO_SURACE_MORTAR** between each component with defined segment sets [6], which allowed for the easy collection of contact forces through the RCFORC option in the ***DATABASE_OPTION** card. The model was setup to use the LS-DYNA implicit solver and the recommendations provided from Dynamore Nordic AB [7] for the ***CONTROL** and ***DATABSE** cards and the recommendations from the LS-DYNA Aerospace Working Group [8] were implemented to the greatest extent possible.

7 Finite Element Model Refinement Methodology

In order to determine the optimal finite element model that would produce results within an acceptable variance (~10%) from our test setup, the following parameters were altered in an effort to create an accurate model with the least computational expense. The parameters that were varied were the following:

1. Mesh density for all components.

2. Nodal locations 1-D beam element (i.e. location of the node for the 1-D beam element in the threaded section of the fixture).
3. ***SECTION_SOLID ELFORM** (-2, -1, 1, 2).
4. Presence of the washer between the bolt head and lower component.

8 Comparison of Results

Over 30 different finite element models were developed for each test case in order to determine the various effects of modifying the parameters established in the model refinement methodology. Of these models, five models representative of trends experienced will be discussed in detail. Four models are depicted in Figure 8. Model 1 depicts a very fine mesh that was used as the baseline model, Model 2 (not depicted) used a coarser mesh with at least 3 elements through the thickness, Model 3 utilized the same mesh as Model 2 but with only 2 elements through the thickness of the washer, Model 4 used an even coarser mesh, and Model 5 had an extremely coarse mesh.

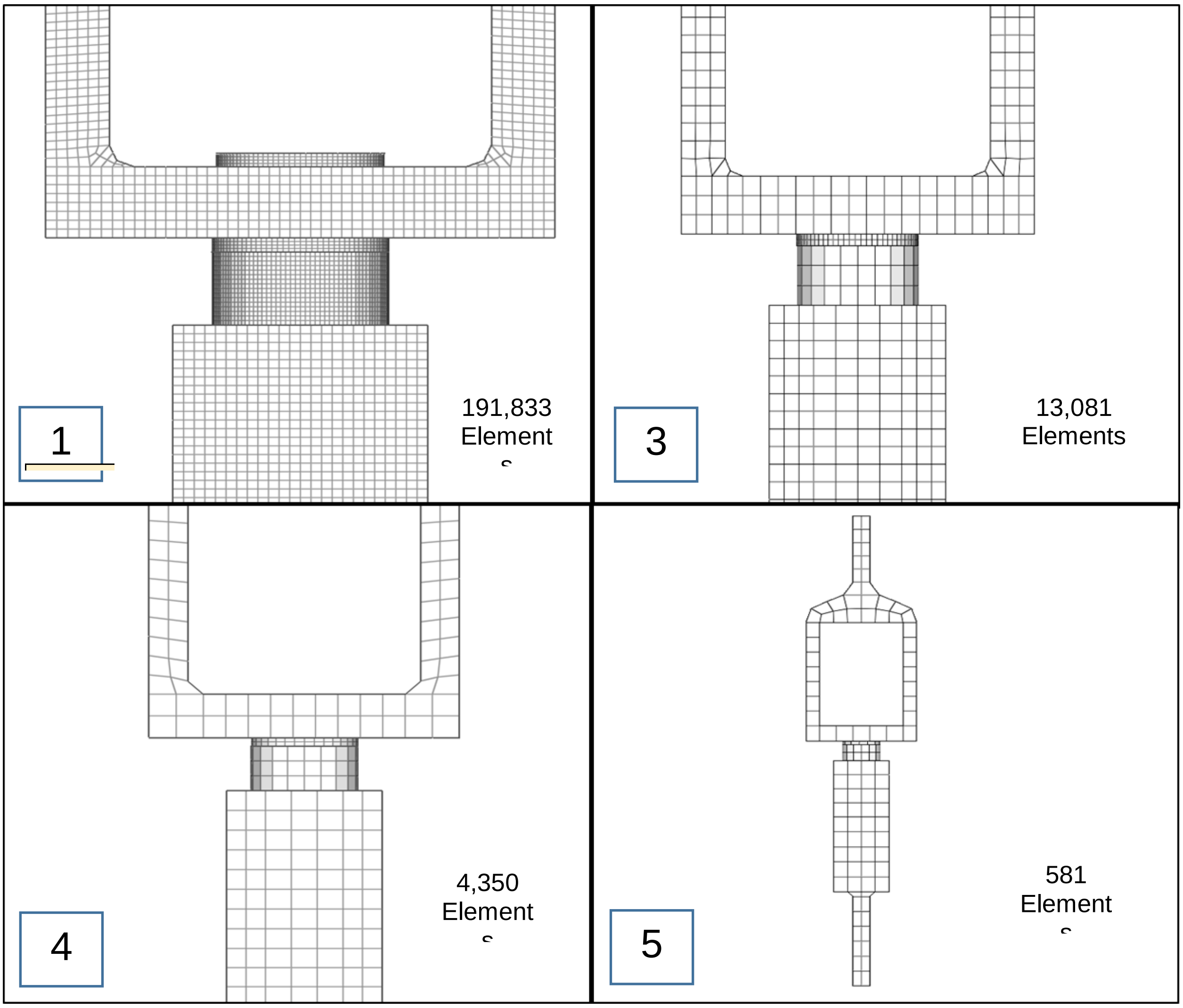

Fig.8: Depiction of various iterations in the mesh density of the test fixture in Test Case A. All of these iterations, with the exception of the bottom right (4), provided reasonably accurate results.

As seen in Figure 9 and Table 1, for both Test Case A and Test Case B the finite element analysis results correlated reasonably well with the experimental data. Only the model with a very coarse mesh (Model 5) experienced a significant deviation from the test results, which was expected due to the washer and top fixture plate only having one element through the thickness. Furthermore, many elements were distorted in this mesh due to the bolt hole in the components being roughly equivalent to the mesh size. It can also be seen in Table 1 that the analytical approach using Shigley's frustum of a cone method provided less than ideal results, which can be attributed to the complexity of the joint due to the various materials and multiple layers, the thickness of the joint, and the likelihood of the frustum falling outside of the joint in Test Case B [2].

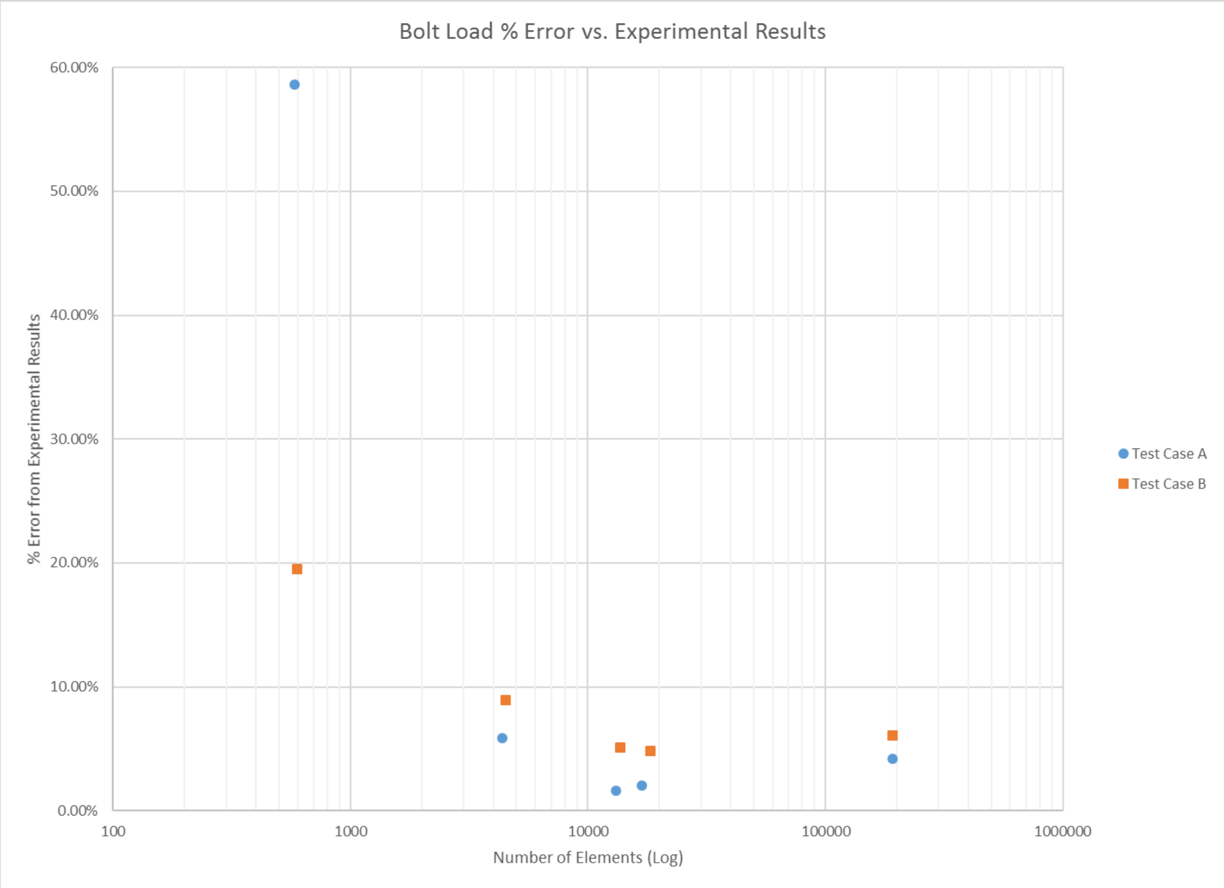

Fig.9: Percent difference in bolt load between experimental and finite element data for various mesh densities.

Table 1: Table of the loads in the bolt and joint for the experimental, analytical, and finite element analysis for Test Case A and B. Percent Error is calculated against the experimental data.

Test Case A	Experimental Test	Shigley	Finite Element Model 1	Finite Element Model 2	Finite Element Model 3	Finite Element Model 4	Finite Element Model 5
Load in Bolt	1160	947	1209	1136	1141	1228	1840
Load in Joint	3840	4053	3791	3864	3859	3772	3160
Total Load	5000	5000	5000	5000	5000	5000	5000
% Error in Bolt Load		18.36%	4.22%	2.07%	1.64%	5.86%	58.62%
% Error in Joint Load		5.55%	1.28%	0.63%	0.49%	1.77%	17.71%
Number of Elements			191833	16806	13081	4350	581
Test Case B	**Experimental Test**	**Shigley**	**Finite Element Model 1**	**Finite Element Model 2**	**Finite Element Model 3**	**Finite Element Model 4**	**Finite Element Model 5**
Load in Bolt	313	791	332	328	329	285	374
Load in Joint	4687	4209	4668	4672	4671	4715	4726
Total Load	5000	5000	5000	5000	5000	5000	5100
% Error in Bolt Load		152.72%	6.07%	4.79%	5.11%	8.95%	19.49%
% Error in Joint Load		10.20%	0.41%	0.32%	0.34%	0.60%	0.83%
Number of Elements			191833	18735	13743	4523	595

Throughout the iteration process, beginning with the baseline model and continuing through the various refined models, the following observations were made regarding best practices for producing accurate bolt loads:

- Washers between the bolt head and joint can be replaced with rigid elements without significant impact to the joint behavior in tensile loading.

- The nodal locations of 1-D beams representing bolt elements are critical for proper joint behavior.
  - If washers are excluded, the node of the 1-D beam should be placed at the location of the interface between the bolt head and washer.
  - If washers are modeled, the node of the beam should be coincident with the plane at the top of the washer.
  - For a threaded hole, optimal results were obtained when the 1-D beam node was placed at ½ diameter down into the hole. This distance matched Shigley's recommendation when analyzing a bolted joint using the frustum of a cone approach [1].
- For a fine mesh, constant stress solid elements provided accurate results **(*SECTION_SOLID**, ELFORM 1)
- For a coarse mesh (≤ 2 elements through the thickness), selective reduced integrated solid elements results **(*SECTION_SOLID**, ELFORM 2, -1, -2) provided superior results. For this test, no difference was noted in all models between ELFORMS 2, -1, or -2.

9 Summary

This test has shown that the accurate measuring of not only the total load of a bolted joint, but the individual bolt and joint loads, can be a difficult endeavor, both through testing and finite element analysis. When validating a finite element model through testing, this becomes an even more difficult endeavor as the sources of error in both the test specimen and finite element model must be recognized and reconciled in order to validate the results. Often, this is complicated further since the finite element model consists of the "as designed" component, where the tested component may vary significantly due to test conditions and instrumentation installed on the equipment. From this, the following conclusions can be made.

- When testing a bolted connection for the reaction load, it is imperative that the instrumentation used will capture both the bolt load and the joint load. Simply measuring one or the other will only give a portion of the total load in the joint. If only the bolt or joint load is measured, the addition of instrumentation will likely change the behavior of the joint, which must be accounted for when trying to determine the full reaction load.
- The total reaction load at a bolted joint can be accurately measured in LS-DYNA through measuring the loads within the bolts (1-D beams) and through outputting joint loads through contact forces between members of the joint. Summing these two components will provide an accurate total bolted joint load, but depending on the refinement of the model, the individual bolt and joint loads may be inaccurate (i.e. an overly stiff joint due to a low mesh density will result in increased joint load and decreased bolt load).
- Accurate individual bolt loads and joint loads in the finite element model depend significantly on the density of the mesh and accuracy of the joint representation. At a minimum, at least 2 elements should be used through the thickness of all components and the nodal locations of 1-D beams representing bolts should be accurately modeled to reflect reality.

10 Literature

[1] J. E. Shigley, C. R. Mischke and R. G. Budynas, "Mechanical Engineering Design," McGraw-Hill Book Company, New York, 2004.

[2] K. H. Brown, C. Morrow, S. Durbin and A. Baca, "Guidelines for Bolted Joint Design and Analysis: Version 1.0," Sandia National Laboratories, Albuquerque, 2008.

[3] LS-DYNA Keyword User's Manual, Livermore Software Technology Corporation, 2015.

[4] U. Sonnenschein, "Modelling of bolts under dynamic loads," in *LS-DYNA Anwenderfroum*, Bamberg, 2008.

[5] S. Narkhede, N. Lokhande, B. Gangani and G. Gadekar, "Bolted Joint Representation in LS-DYNA to Model Bolt Pre-Stress and Bolt Failure Characteristics in Crash Simulations," in *11th International LS-DYNA Users Conference*, Detroit, 2010.

[6] T. Borvall, "A Guide to Solving Implicit Mortar Contact Problems in LS-DYNA," DYNAmore Nordic AB, Linkoping, 2015.

[7] A. Jonsson and M. Lilja, "Some guidelines for implicit analyses using LS-DYNA," DYNAmore Nordic AB, Linkoping, 2016.

[8] "LS-DYNA Aerospace Working Group Modeling Guidelines Document," LS-DYNA Aerospace Working Group, 2016.