5-2011

Finite Element Analysis Prediction of Stresses in H.L. Hunley Submarine by Global-to-Local Model Coordination

Aditya sai nag Choragudi
Clemson University, achora@clemson.edu

FINITE ELEMENT ANALYSIS PREDICTION OF STRESSES IN H.L. HUNLEY SUBMARINE BY GLOBAL-TO-LOCAL MODEL COORDINATION

A Thesis
Presented to
the Graduate School of
Clemson University

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

by
Aditya Sai Nag Choragudi
May 2011

Accepted by:
Dr. Vincent Blouin, Committee Chair
Dr. Gang Li
Dr. Richard Miller
Dr. Lonny Thompson

ABSTRACT

H.L Hunley was a submarine of the Confederate States of America that participated in the American Civil War. On February 17, 1864, H.L.Hunley created history by becoming the first submarine to sink a enemy ship after its attack on USS Houstanic. After Hunley never returned to the shore and the details of its wreck were unknown. On August 8, 2000, H. L Hunley was brought to the surface after 136 years of its wreckage. The submarine is currently at the Warren Lasch Conservation Center located in Charleston.

This study focuses on the structural analysis of the H.L Hunley submarine to predict stresses and potential structural failure. Modeling the structure is challenging because of (1) the lack of symmetry due to its current position, (2) non-uniformity due to high corrosion, and (3) the riveted connections with more than 4000 rivets. Although connections between plates in ships are generally considered stronger and stiffer than the rest of the structure, this assumption is assumed to be invalid in the case of the Hunley because of the high and non-uniform corrosion. Since modeling the entire submarine and its 4000 rivet is impossible, the purpose of this study is to create a coordination procedure between the global model of the submarine with simplified connections and the local model of a riveted connection to affectively predict the stresses. The Global model is the whole submarine modeled using shell elements to decrease complexity. The local model consists of one of the riveted connections in the submarine. The validation of the procedure is discussed.

DEDICATION

To my parents Mr. Ch. Subrahmanyeswara Rao and Ch. Vijaya Nirmala.

ACKNOWLEDGMENTS

I would like to acknowledge my advisor, Dr. Blouin for his continuous support, encouragement and patience.

I would like to acknowledge all the Committee members, Dr. Gang Li, Dr. Richard Miller, Dr. Lonny Thompson, for providing their valuable input and actively participating in decision making for recommendations of this study.

Additionally I would like to acknowledge Warren Lasch Conservation Center for its financial support.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

Figure Page

LIST OF FIGURES(CONTINUED)

Figure Page

LIST OF FIGURES (CONTINUED)

Figure Page

CHAPTER 1

INTRODUCTION

1.1 Background

The H.L. Hunley is a submarine from the American Civil War era that sank off the coast of Charleston, South Carolina, in 1864. The submarine was discovered in 1995 and recovered from the ocean in 2000.

The submarine was under water for over 130 years and hence it is highly corroded. Due to the high amount of corrosion the submarine is highly unstable and if the submarine is exposed to the atmospheric air, irreversible damage could take place. In order to be able to handle and treat the submarine to stabilize its corrosion, the structural integrity of the hull must be studied. Therefore stress analyses on H.L. Hunley are carried out using FEA.

Figure 1.1. Recovery of H.L. Hunley, August 2000
(Photo courtesy of the Friends of the Hunley)

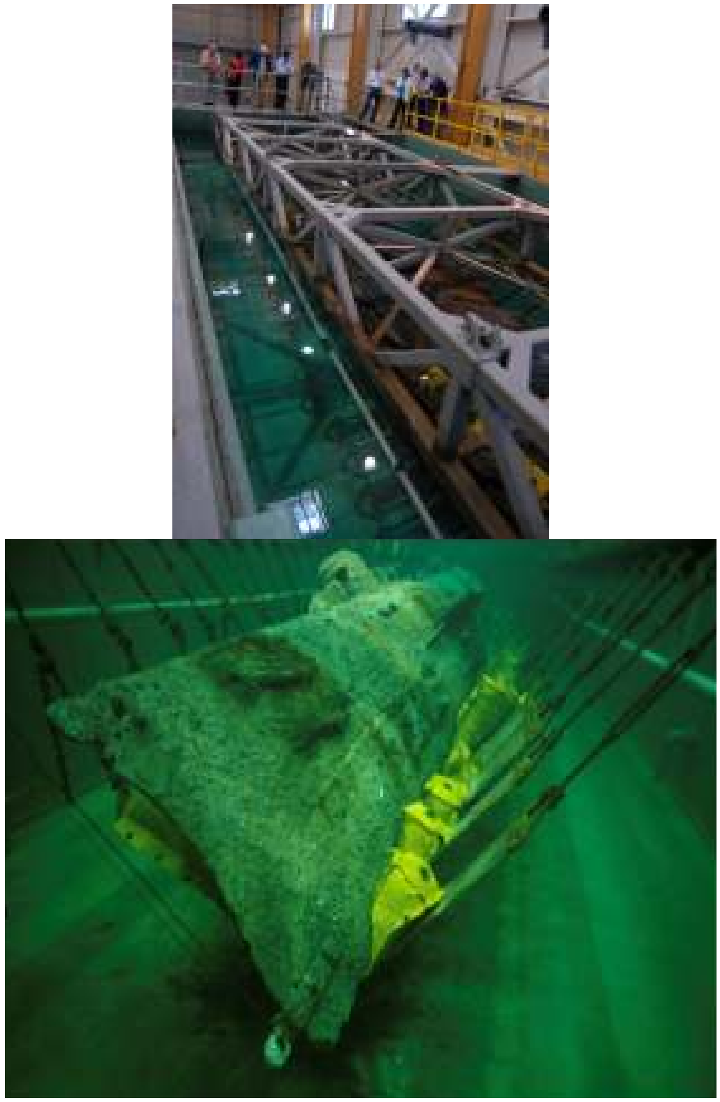


Figure 1.2. H.L. Hunley, cushions and sling system in conservation tank
(Photo courtesy of the Friends of the Hunley)

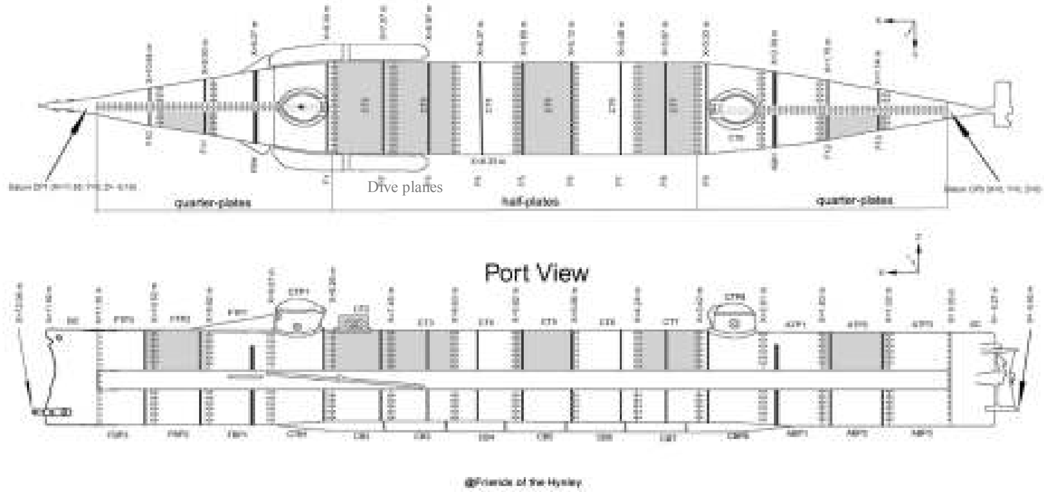


Figure 1.3. The submarine is composed of about 70 plates and backing plates riveted together. Shaded areas show removed plates for excavation

As shown in Figure 1.1, the submarine was raised from the ocean floor in its original position using a truss and sling system composed of 32 slings and expanded polyurethane foam cushions. The submarine was then placed in a fresh water tank (Figure 1.2) at the Clemson University Conservation Center in Charleston SC and is being treated for long term conservation

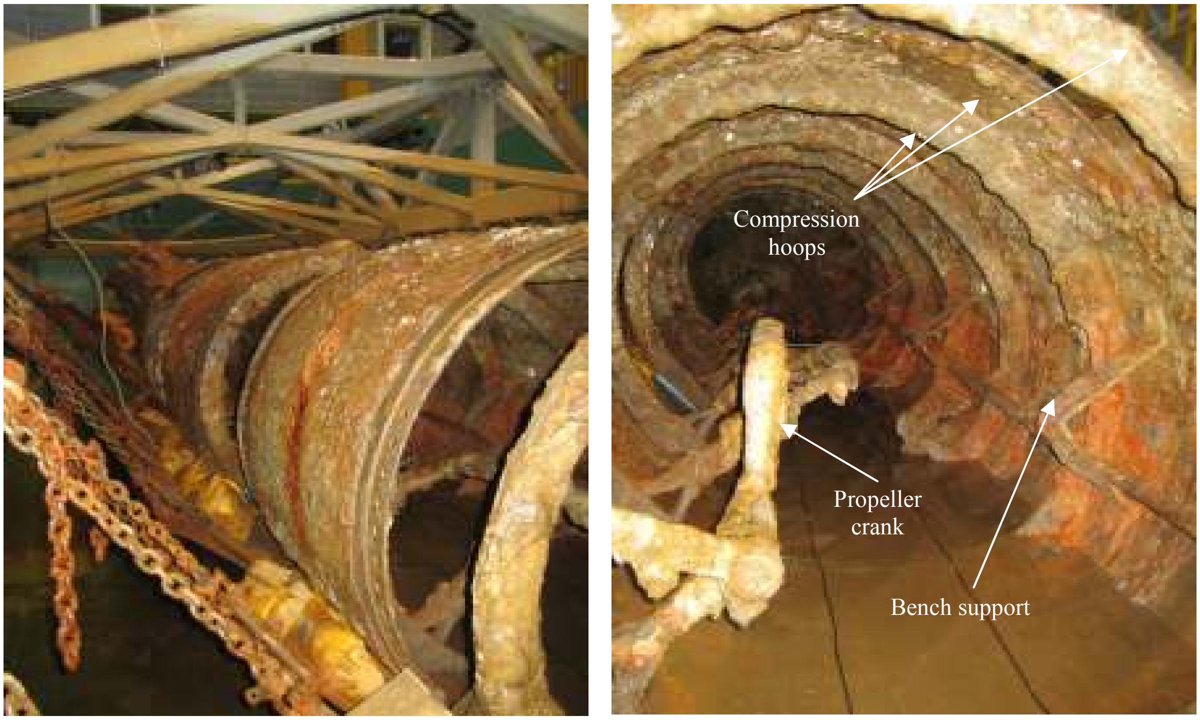


Figure 1.4. The submarine is now open and emptied. The irregular concretion layer can be seen on all surfaces, compression hoops and appendages (bench support, propeller crank) (Photo courtesy of the Friends of the Hunley)

The submarine was originally built using cast iron and wrought iron. Since it was amongst the first submarines to be built, the riveting techniques of 1850's were used.

1.2 Research Goals

The main goal of the research is to use Finite Element Analysis to evaluate the structural integrity of the H.L Hunley submarine. Modeling the structure is challenging because of the high level of corrosion, the relatively large number of parts of the structure, the lack of symmetry, and the fact that the submarine is currently supported by a flexible catenary system (i.e., slings) with surface-to-surface contacts between the hull and the slings. Several of these difficulties could be the subject of a research thesis.

Given the geometric and mechanical complexity of the system, representing the submarine using a single model is not sufficient. The whole representation of the submarine and its supports is done using a global model. The global model assembly is discussed in detail in chapter 2. The hull is made of thin plates riveted together. Therefore, the global model uses three-dimensional shell elements. The rivets are very critical components of the structure and representing them in the shell model is impossible. However, neglecting the localized stresses developed in the riveted connections can prove to be dangerous while predicting the structural integrity of the submarine.

Hence a global-local coordination procedure is developed to effectively calculate the stresses generated at the riveted connections in the submarine. The global model consists of the entire submarine modeled with shell elements, and the local model represents one of the riveted connections in the submarine using 3D solid elements.

This research focuses on the modeling aspects related to the evaluation of stresses in the riveted connections. In particular, the research focuses on how to effectively model the simplified riveted connections without losing accuracy. The questions raised in this research can be illustrated as shown in Figure 1.5. If a rivet maintains two plates connected, depending on the geometry and stiffness of the rivet, the connection has a certain mechanical behavior. Typical questions include:

- If the riveted connection (i.e., zone highlighted in red in Figure 1.5) is simplified and modeled as a plate, what should be its thickness and material stiffness?

These properties would be pseudo properties that should provide the same mechanical behavior as the actual complex geometry.

- Do these pseudo properties depend on the friction of the surfaces in contact and the pre-tension of the rivet?
- Should these pseudo properties be altered in the case of a corroded rivet?

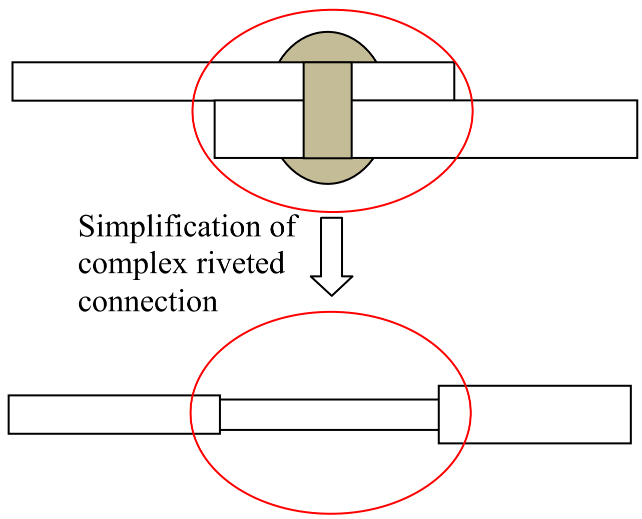


Figure 1.5. Simplification of complex geometry

The rationale for this research lies in the fact that since the hull includes many rivets, each connection seam line must be simplified and modeled using pseudo geometric and material properties. These properties, which are used in a global model of the entire submarine, must be selected non-arbitrarily.

Therefore, the main research goals can be listed as follows:

1. Develop a coordination procedure to analyze the complex structure using a global model of the entire submarine and a local model of a riveted connection.
2. Find the appropriate modeling properties and their implications on the structural behavior of the hull.

3. Estimate the static stresses in the hull under gravitational loads and various sets of boundary conditions.

4. Study the effects of corrosion on the behavior of riveted connections and the overall structure.

1.3 History of H.L. Hunley

H.L. Hunley was a submarine of the Confederate States of America that participated in the American Civil War. As per the findings of Sally M. Walker in his book "*Secrets of a Civil War submarine: solving the mysteries of the H.L. Hunley",* the idea of a submarine was not new during the American civil war. In the early sixteenth century Cornelis Grebbel had built a submarine that resembled a rowboat enclosed with a leather cover [1].

David Bushnell built the first American submarine named as the Turtle during the American Revolution. Although Turtle could navigate underwater, it was not successful in attacking an enemy warship. In fact, no submarine had ever sunk an enemy ship until then. [1]

H.L. Hunley was built in Mobile, Alabama in the spring of 1862 at Park & Lyons Machine Shop with the help of machinists, businessmen engineers James McClintock, lawyer Horace L. Hunley and four members from a manufacturing organization called Singer Submarine Corps [2]. Although H.L. Hunley was technically remarkable for its time, it possessed both advantages and dangers in attacking an enemy ship [3]. Due to the size of the submarine, it had to navigate very low in the water and an unexpected wave could wash into an open hatch, sinking the submarine [1].

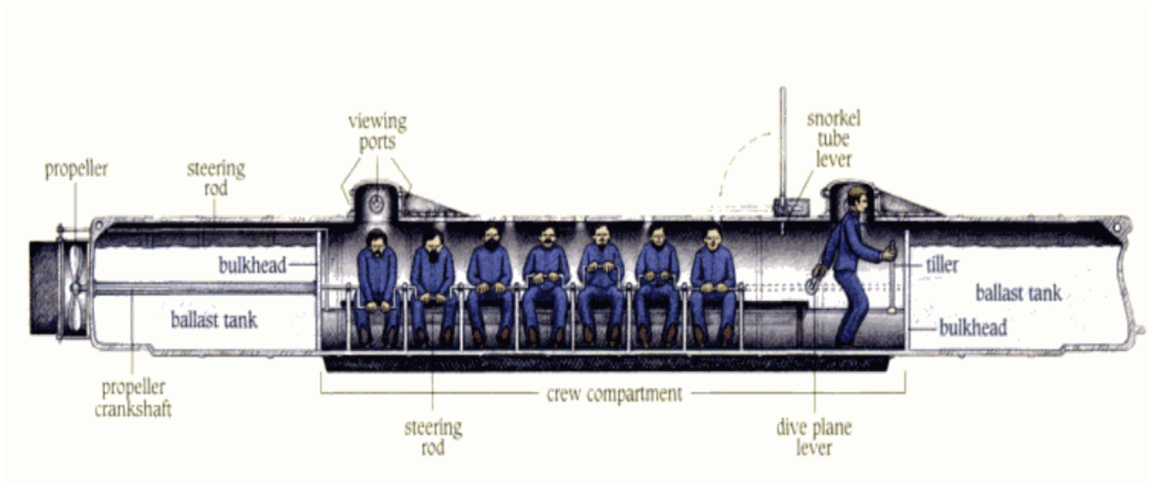


Figure 1.6. H.L Hunley with its crew [1]

In fact during a test run in Charleston, the submarine sank due to an error of a crew member killing five of its sailors on board. A few months later, it sank for the second time, killing all of its crew members. Despite this H.L. Hunley became the first submarine to successfully attack an enemy ship [4]. On February 17, 1864, H.L. Hunley created history by becoming the first submarine to sink an enemy ship after its attack on USS Houstanic. After the successful attack, H.L. Hunley never returned to the shore and the details of its wreck were unknown.

1.3.1 Recovery of H.L. Hunley

After its wreck in 1864, the whereabouts of Hunley were unknown. It is due to the efforts of Clive Cussler, a Novelist and adventurer and his divers, Hunley was found in 1995 [5]. On August 8, 2000, H.L. Hunley was brought to the surface after 136 years. After resolving the major issues of locating the submarine and recovering it, the most important and difficult part was to conserve the remains of the structure. The submarine is currently at the Warren Lasch Conservation Center located in Charleston, South Carolina.

1.3.2 Conservation of H.L. Hunley

The Hunley submarine could be considered as one of the most complex composite structures to be ever recovered by a team of conservators and archaeologists. Any step taken in the conservation is carefully analyzed so that the structural integrity is not comprimised. After its recovery H.L. Hunley was placed in a water tank consisting of chilled water at 10 degrees Celsius to minimize the impact of potential enzymatic reactions on organic remains and also to reduce the impact of corrosion [6]. The uncontrolled exposure to air of H.L. Hunley could lead to damage and the disastrous loss of archaeological data. Because the Hunley was mainly built using wrought and cast iron, it makes it even more susceptible to oxygen. Since it was underwater for over 136 years, chlorides from the seawater penetrated the iron at the molecular level. These chlorides would destroy the submarine if exposed to oxygen rich environment due to a sudden change in equilibrium. Hence it is very important for the conservators to gain sufficient knowledge regarding the handling, storage, stabilization, and study the recovered artifacts [7].

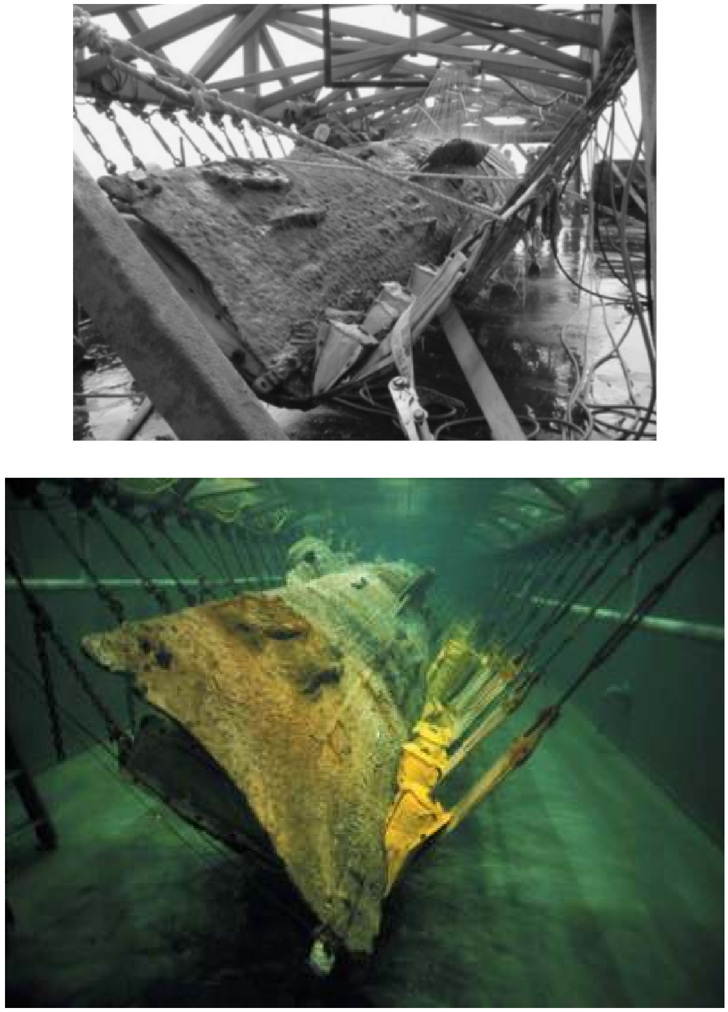


Figure 1.7. H.L Hunley during the recovery and in Conservation tank

Also one of the problems faced during recovery of the submarine was the position of the submarine while it was underwater for 136 years. Filled with sediment, Hunley rested at an angle of 45 degrees on its starboard side. So, tilting the submarine by even a few degrees can cause the rivets on the submarine to fail. Detailed analysis needs to be done on the structure before it could be moved.

1.4 Literature review

There have been quite a few research studies on corroded marine structures. Finite Element Analysis (FEA) has been essential in designing new structures and analyzing existing corroded marine structures.

Russell *et al.* [8] developed a method to measure the corrosion rate of steel-hulled shipwrecks in seawater. Measuring the corrosion rate is very difficult due to the many factors that need to be considered. This research was applied to the wreck of the USS Arizona, a battleship which was sunk during the attack on Pearl Harbor on December 7-th, 1941, in order to predict the degradation in structural integrity of the wreck during the next century. Researchers were able to estimate the decrease in thickness of the ship plates over time and implement this information in a FEA model of a section of the ship and predict its collapse. Concerning the connections between plates, the researchers assumed that they were not the weakest points of the hull and that failure would occur within the plates as opposed to at the connections.

Slater *et al.* [9] used FEA to model the bucking behavior of plates on corroded ships. Although the actual corrosion is never uniform, corrosion was modeled as a uniform loss of metal in the corroded regions in order to better understand its effects on the overall structure. The corrosion area was modeled in five different geometric patterns. In their analysis they found that the buckling strength of the plates was decreased due to corrosion and its location on the plates. They argued that the plates lose most of their buckling strength when corrosion reaches 20% of the plate thickness, at which point they recommended replacing the entire plate.

Most studies confirm that the presence of corrosion is equivalent to a reduction in amount of material from the exterior surface as exemplified by the work of Dunbar [10] who investigated the effect of localized corrosion on ship plates and stiffened panels using FEA. In this study, it was found that the ultimate load of the plates decreased when there was corrosion and this effect increased when the location of the corrosion was closer to the center of the plate. The local corrosion was applied to the model by reducing the thickness of the plate at the corroded area.

The initial work in finite element analysis of the three dimensional connections was done by Krishnamurthy [11]. The eight-node parametric brick elements were used to model the behavior of a bolted end plate connection. There were several other models built later on but there were few issues regarding the modeling of connections [12, 13]. Buris *et al.* discussed the issues of modeling a bolted connection and how numerical simulations depend heavily upon the step size used in the analysis, kinematic descriptions, element types, and mesh size [14]. When there is a large assembly, the bolt/riveted connections connecting the assemblies are generally modeled as a beam-spider assembly [15]. Research studies have shown that solid connection including the actual connections is preferable [16, 17].

In recent years the importance of studying localized stresses when evaluating the structural integrity of a large structure has increased. Imam and Righiniotis [18] studied the fatigue evolution of riveted railway bridges using a global and local analysis. The Global Model consisted of a typical railway bridge with no riveted connections. This model was analyzed to find out the critical areas in the model at which the structure experiences higher stresses. After evaluating the critical areas a global-local model was created in which only the critical region was modeled with a detailed geometric description of the riveted connection. The single most critical connection was modeled with a shell-to-solid transition. Using the global-local model local stresses were found and also potential crack initiation was evaluated in the detailed part of the model. As explained in the following chapters, the approach presented in this thesis differs from the global-local model of [18] since three distinct models are used in a coordination procedure as opposed to a single global-local model.

1.5 Thesis Outline

This dissertation is divided into six chapters. Chapter 2 describes the global model assembly and analysis. The modeling of the global model, material properties, and interaction properties are discussed in this chapter. Also the stress analysis results of the global assembly in different loading conditions are presented. Chapter 3 describes the local model which consists of the 3D riveted model and the local shell model. The modeling technique is discussed in detail. The results of the stress analysis of both the 3D local model and the shell local model are presented. Chapter 4 elucidates on the global-local model coordination. Also a simplified global model is introduced in this chapter. In Chapter 5, the corroded rivet model is presented. The effect of corrosion on the stresses developed in the local model is presented. Chapter 6 concludes the thesis with a section on recommendations and future work.

CHAPTER 2

GLOBAL MODEL

2.1 Global Model

The first three-dimensional mathematical model of the Hunley was developed by engineers from Oceaneering, Inc., before the recovery (before 2000) of the submarine based on measurements from James R. McClintock's sketches [19]. Since the material properties of the hull were unknown, they were assumed to be the same as that of a conventional wrought iron.

After recover, several FEA models were developed based on more accurate dimensions. Since the whole assembly contains a large number of components, modeling the entire hull structure is challenging. Also, finding the appropriate material properties to be assigned to the submarine is an issue. Since the submarine is a protected historic artifact, limited engineering studies can be performed on the structure. As a result, most mechanical properties and plate thicknesses are still unknown.

The submarine is made of iron, specifically ductile wrought iron for most of the structure and brittle cast iron for the bow, stern, coning towers and keel ballast blocks. The iron is covered with a layer of concretion which is assumed to add weight without adding much structural stiffness and strength. In essence, the iron plates are sandwiched between two concretion layers of up to one inch in thickness. The thickness of iron, the thickness of the concretion layer as well as the bond between iron and concretion are unknown and heterogeneous. Attempts to measure thicknesses using ultrasonic techniques failed due to the excessive presence of corrosion. Instead, the hull thickness, approximately 0.375” (9.5 mm), was measured from a rivet hole in which the rivet had completely disintegrated [20].

The goal of the FEA is to enable project engineers and conservators to identify the conditions in which the submarine will be handled (during rotation and treatment) such that stresses are minimized. Also, the stresses computed by FEA are used to estimate a factor of safety with respect to the yield strength of today’s iron.

In order to produce a meaningful finite element model, four pieces of information are needed:

- a numerical model of the geometry of the structure,
- the mechanical properties of all materials,
- the interaction properties of parts in contact, and
- the loads applied on the structure including boundary conditions.

If accurate information is defined in the model, FEA can provide valuable data, such as the distribution of stresses and location and value of areas of highest stress. In the case of the Hunley submarine, obtaining these four pieces of information presents significant challenges due to the lack of available data. Therefore, important assumptions were made and various parametric studies are being conducted to evaluate the hypothetical effect of several parameters and account for their inherent uncertainty.

The FEA models were created and analyzed using the commercial software ABAQUS® version 6.8 [21]. Abaqus is one of the leading FEA software for structural analysis. Version 6.8 offers a recently improved contact model, which is particularly important in the analysis of the Hunley submarine. Also, Abaqus provides two different solvers: the implicit solver (Abaqus/Standard) which is computationally efficient but requires well-defined boundary conditions and contact interactions, and the explicit solver (Abaqus/Explicit) which is significantly slower but is able to solve singular problems with loose boundary conditions and offers a general contact tool which greatly simplifies model development.

The global model is a very essential part of the analysis. It comprises of the closest real time simulation of the Hunley submarine. Since the Hunley submarine is a large complex structure, it is impossible to completely incorporate all the components in one single model. Therefore the global model, shown in Figure 2.1, includes the major components of the Hunley.

Figure 2.1. The complete assembly of the global model

As discussed earlier, the submarine is currently resting on its starboard side at a roll angle of 45 degrees in a water tank in Charleston, SC. The submarine is currently supported by about 30 slings, which are belts with polyurethane foam cushions. The FEA global model shown in Figure 2.1, however, includes only half of the current slings, which is the configuration that will be used for the rotation process of the submarine from its current 45-degree roll angle to the upright vertical position.

2.2 Plate thickness and mass density

The effects of corrosion are a major concern for modeling and analyzing the Hunley. Although corrosion occurred in a non-uniform manner across the submarine, the thickness of shell components (i.e., plates, backing plates, riveted connections and coning towers) is assumed to be uniform throughout each component. Since the actual thickness of plates cannot be exactly known, parametric studies were conducted by varying the thickness parameters.

As mentioned earlier, the iron plates are sandwiched between two layers of concretion. However, in order to simplify the global model, all plates are assumed homogeneous with a predefined thickness and density, as shown in Figure 2.2. For the thickness, we use the thickness, t_i, of the iron plate and a pseudo material density, $\tilde{\rho}$, to account for the presence of the concretion layers. The pseudo density can be calculated using conservation of mass and is defined by:

$$\tilde{\rho} = \frac{t_i \rho_i + t_c \rho_c}{t_i} \tag{1}$$

Where t_i, t_c, ρ_i and ρ_c are the thickness and density of iron and concretion, respectively. As a result, the added weight from to concretion is accounted for without altering stiffness. For instance, assuming t_i = 10 mm, t_c = 25.4 mm, ρ_i = 7000 kg/m^3 and ρ_c = 2500 kg/m^3, then $\tilde{\rho}$=13,350kg/m^3.

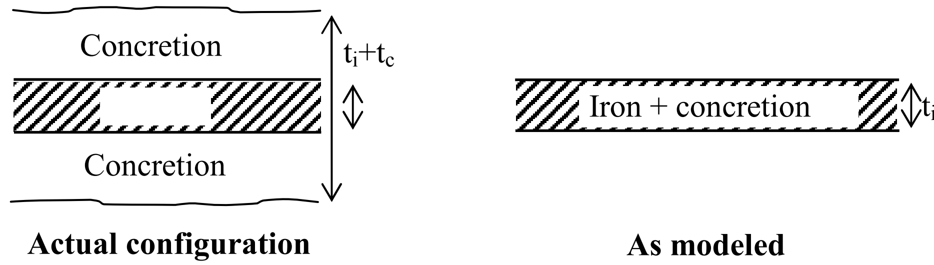


Figure 2.2. Iron plates are sandwiched between two concretion layers

The effect of buoyancy, which was estimated to be equivalent to at most 20 percent reduction in mass density of the iron, is neglected as a conservative measure since a possible worst-case scenario is likely to occur when the tank is emptied.

2.3 Mechanical properties

2.3.1 Stiffness

Stiffness of the metal is defined by the modulus of elasticity, E, and Poisson's ratio, ν. Metallurgical studies of cast and wrought iron components from the Hunley were conducted at the National Institute of Standards and Technology (NIST) to determine the chemical constituents and microstructure of the Hunley's iron components [20]. Due to sampling constraints, testing of the cast iron was inconclusive, but testing of the wrought iron samples demonstrated that a metal core was still present and that the alloy was of 'good' quality by contemporary standards. However, the diversity of the corrosion noted throughout the submarine's structure does not permit assessment of a single stiffness value, as would be possible with freshly cast or wrought iron. Therefore, given the uncertainty of the analysis, elastic properties are estimated to vary within a range of values described in Table 2.2.

2.3.2Strength

The strength of the metal, which is necessary to predict the risk of failure, represents the upper stress limit that the metal can withstand before permanent deformation will occur. Metallurgical studies from the Hunley carried out by NIST, showed a relatively high level of silicate slag in wrought iron components, which generally provides added strength [22]. Since then, micro-hardness tests were performed on wrought iron rivets of the Hunley. By direct comparison with known metals, the strength of the iron was found to be comparable to today's wrought iron [22].

2.3.3 Interaction Properties

The hull of the Hunley is constructed from nearly one hundred different parts riveted together. Several parts, namely the compression hoops and the keel ballast blocks are in surface-to-surface contact with the hull. More specifically, the compression hoops are maintained in place by friction and localized brackets. Each keel ballast block is attached to the bottom of the hull either by three large bolts or a single key originally intended to be used for releasing the block from the inside of the sub. When properly modeled as surface-to-surface contacts, the computational time becomes prohibitively high. Therefore, these contacts are modeled as rigid connections, which is assumed to be a satisfactory assumption given the high friction between parts.

The submarine is supported by a set of movable slings (i.e., belts and cushions) that are in surface-to-surface contact with the hull. These interactions represent an important aspect for the reliability of the numerical model. The contact friction coefficient is set to a relatively large baseline value of 0.95.

Table 2.1. Physical and mechanical properties used in the global model

Property	**Unit**	**Baseline**	**Minimum**	**Maximum**
Density of iron (ρ_i)	kg/m^3	7,430	-	-
Density of concretion (ρ_c)	kg/m^3	2,160	-	-
Density of iron + concretion ($\tilde{\rho}$)	Depends on plate thickness (Eq. (1))			
Thickness of iron plates (t_i)	mm	8.0	2.5	9.5
Thickness of concretion (both sides) (t_c)	mm	20.0	40.0	0.0
Thickness of riveted connections (t_r)	mm	14.5	5.0	16.0
Modulus of elasticity of wrought iron (E_{wi})	GPa	200	160	220
Modulus of elasticity of cast iron (E_{ci})	GPa	150	100	200
Modulus of elasticity of riveted connection (E_r)	GPa	200	160	220
Modulus of elasticity of foam cushions (E_f)	GPa	0.001	0.0002	0.01
Strength of wrought iron (S_{wi})	MPa	180	-	-
Strength of cast iron (S_{ci})	MPa	120	-	-
Friction coeff. between hull and cushions	-	0.95	-	-
Friction coeff. between cushions and belts	-	0.95	-	-

2.4 Global model assembly:

The global model assembly consists of many parts. Each part and its pertaining assumptions are discussed in this section.

2.4.1 Slings

The slings are the supports of the submarine. As shown in Figure 2.3, the global model includes 14 slings modeled as isotropic deformable shells. In reality, the slings are belts made of a flexible polymeric fabric with high longitudinal stiffness and low bending stiffness. Although Abaqus has the ability to model membranes (i.e., no bending stiffness), modeling the slings as membranes has not been successful due to convergence issues. Instead, the slings are modeled as shells, which induce unrealistic added stiffness in the system. This is stiff valid as long as the belts do not bend significantly as the load is applied. This was prevented by making sure that the initial unloaded position of the submarine is close to the static equilibrium position.

The initial length of each sling is based on the measured length of the real system. The tensioning of each sling, which is arbitrary, corresponds to translating the ends of each sling vertically. As a sling is translated upward, which is equivalent to tensioning it, it applies more load on the bottom of the submarine and reduces the load on the adjacent slings. Defining the appropriate position of each sling is a challenge that has not been completely resolved. At this point, the shape and position of each sling are defined based on an arbitrary undeformed thickness of the foams and the shape of the hull directly above the sling.

The slings used in the analysis are given the following section properties:

Type	:	Shell/ Continuum Shell
Shell Thickness	:	10 mm
Density	:	1000 kg/m^3
Young's Modulus	:	.01 GPa
Poisson's Ratio	:	0.3

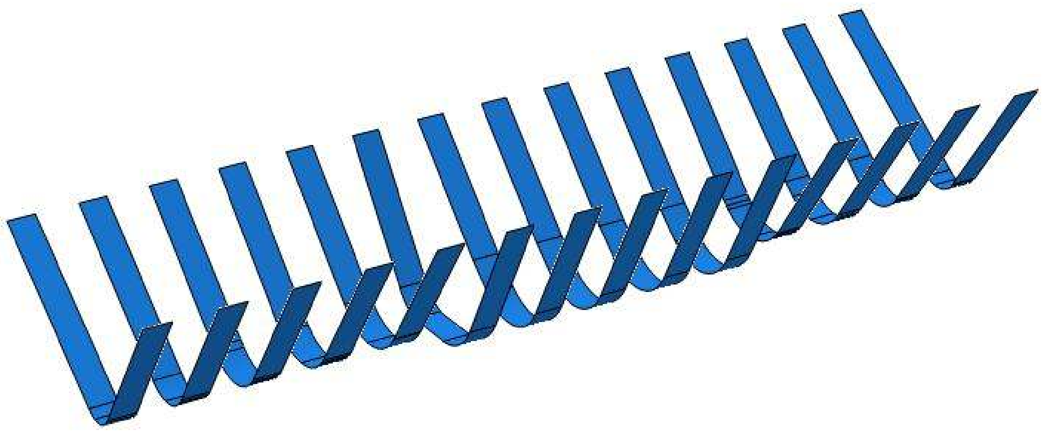


Figure 2.3. Slings used in the analysis model

2.4.2 Foam cushions

Foam cushions are used to block the submarine in the slings and distribute any concentrated load thereby decreasing the amount of pressure on the hull. The actual foam cushions are the original cushions made of expanded polyurethane that were installed during the recovery of the Hunley. Since the polyurethane was expanded in cylindrical bags under the hull, they are naturally pre-shaped to match the curvature of the hull. The foam cushions of the global model are also shaped to perfectly match the curvature of the hull.

Type	:	Solid, Homogenous
Density	:	200 kg/m^3

Young’s Modulus	:	200 MPa
Poisson’s Ratio	:	0.2

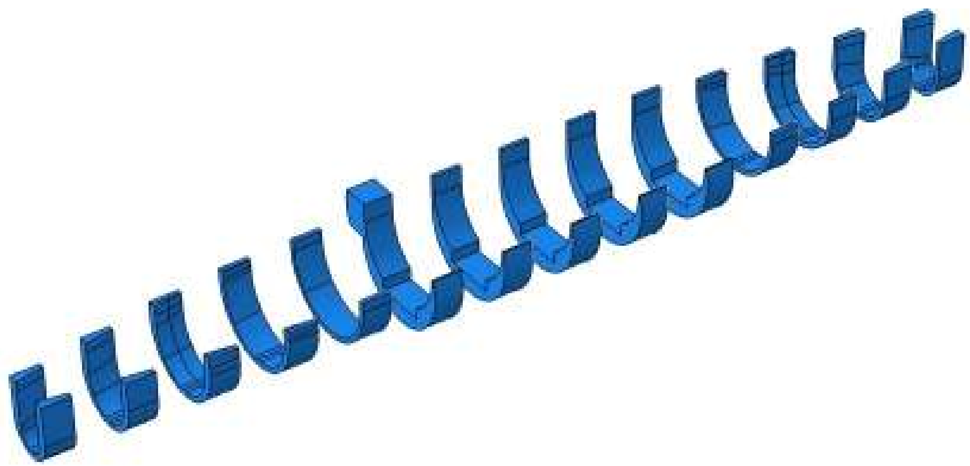


Figure 2.4. The design of the foams used in the analysis

2.4.3 Hull

The hull is an assembly of plates and back-plates riveted together to form a cylindrical shape with tapered regions at both extremities. All plates and back-plates are modeled as shells. The bow and stern, which are the extremities of the submarine, are solid parts made of cast iron modeled with solid elements. The submarine except the coning towers, bow and the stern are modeled using the following properties:

Type	:	Shell/ Continuum Shell, Homogenous
Thickness	:	8 mm
Material	:	Wrought Iron
Density	:	12,000 kg/m^3 to account for concretion
Young’s Modulus	:	210 GPa
Poisson’s Ratio	:	0.3

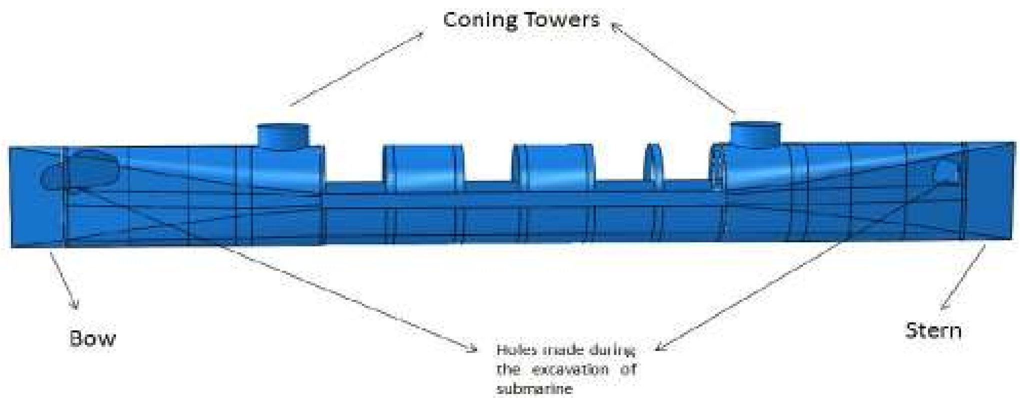


Figure 2.5. Global model of the hull

2.4.4 Compression hoops

The compression hoops, shown in Figure 2.6, are circular rings placed inside the submarine initially intended to resist the underwater hydrostatic pressure.

Type	:	Solid, Homogenous
Material	:	Cast Iron
Density	:	7000 kg/m^3
Young’s Modulus	:	210 GPa
Poisson’s Ratio	:	0.3

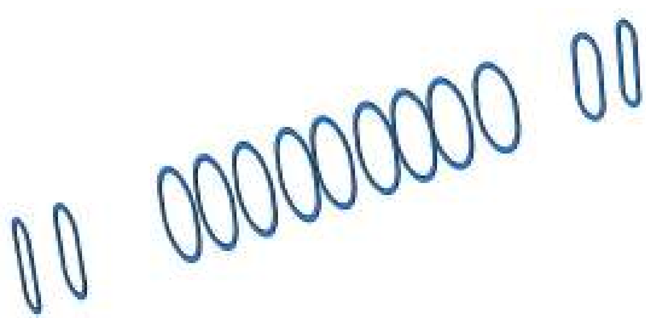


Figure 2.6. Compression hoops used in the analysis

The structural stability of the submarine is increased by the presence of these hoops even though they are not rigidly connected to the hull. Two brackets and friction maintain them in place.

Figure 2.7 shows the all parts of the submarine modeled in the upright position. Three other models were also developed using the same parts: One model of the submarine rotated 45 degrees on starboard in slings (Figures 2.1 and 2.9), one model rotated 20 degrees on starboard in slings, and one model of the submarine in the upright position supported by fixed keel blocks (Figure 2.8).

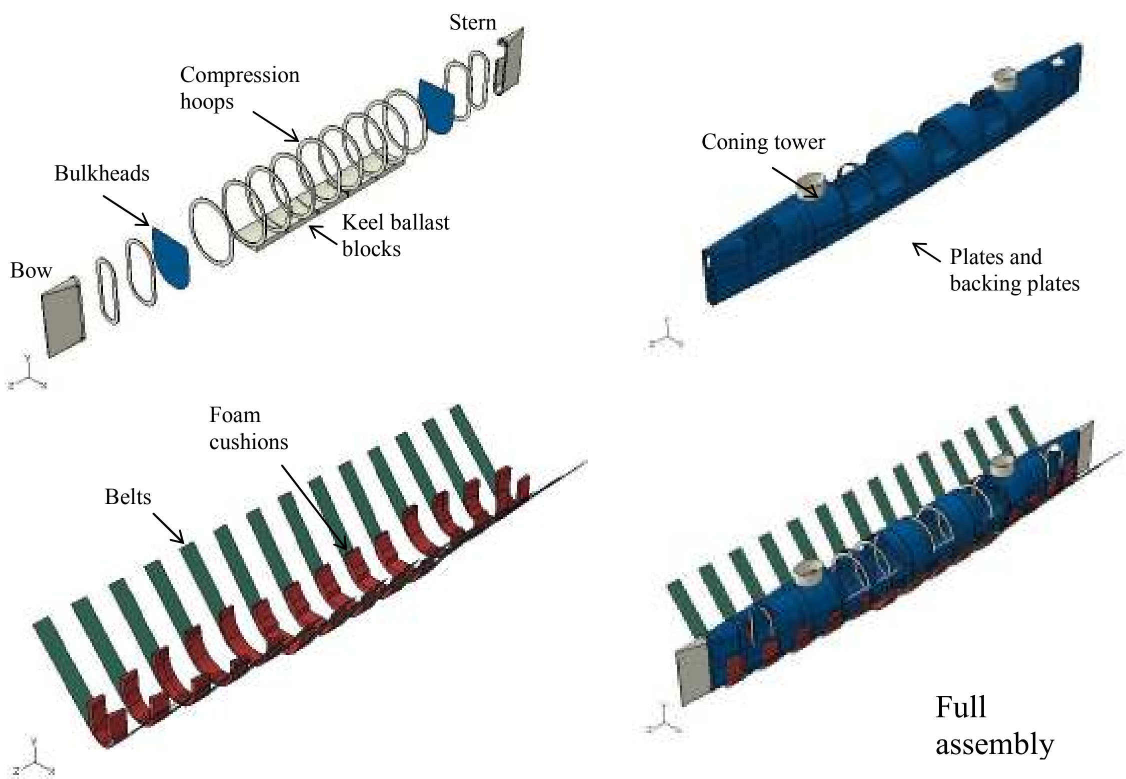


Figure 2.7. Model components: bow, stern, compression hoops, keel ballast blocks and foam cushions are continuum (solid) elements, bulkheads, plates, backing plates, coning towers and belts are shell elements. All components of the sub are rigidly

connected using tie constraints. Interactions between belts, foam cushions and the sub are modeled as surface-to-surface contacts.

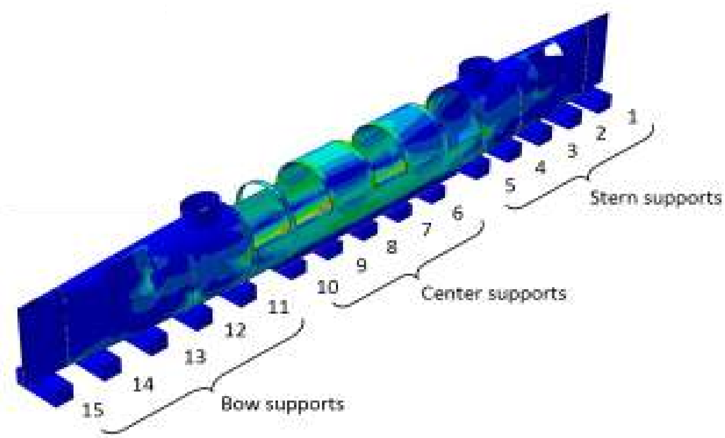


Figure 2.8. Model of the submarine in upright position on fixed keel blocks

2.5 Boundary Conditions and Loads

The goal of the analysis is to determine the static stresses due to gravitational loads under specific boundary conditions. The boundary conditions are specified at the end points of the slings as controlled displacements, as shown on Figure 2.9.

Since the submarine is resting on the cushions without any fixed points, its static equilibrium is achieved by the surface-to-surface contacts between the hull and the cushions and between the cushions and the slings. This type of analysis is numerically challenging for Abaqus since the submarine and the cushions do not have any fixed boundary conditions. In order to allow convergence, the analysis includes three steps. Initially, a small gap of about 1 mm is defined between the hull and the cushions and between the cushions and the slings. In the first step, an artificial boundary condition is applied on the hull and the cushions to maintain them in a fixed position while the slings are moved upward in order to close the first gap and compress the cushions. In this step, the surface-to-surface contacts between the slings and cushions are activated. In the second step, the cushions are releases and the slings continue to move upwards to close the second gap. In this step, the surface-to-surface contacts between the cushions and the hull are activated. In the third step, the submarine is released, gravity is applied and the ends of the slings are maintained in place. Using these three specific steps, Abaqus is generally able to converge and find the final static equilibrium position of the submarine and determine the stress distribution.

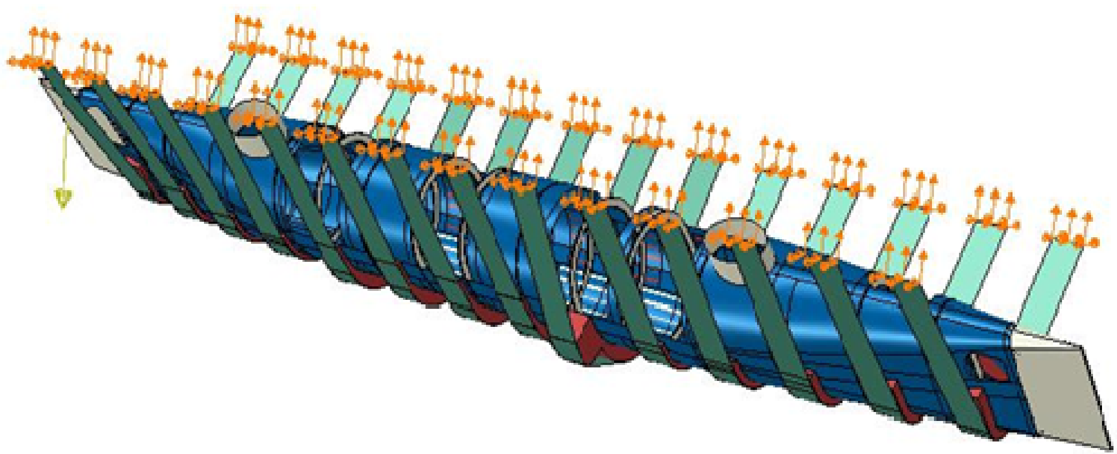


Figure 2.9. The Boundary conditions and loads on the global model

2.6 FEA mesh

The geometrical model is then discretized into more than 300,000 finite elements of 2 cm in average size creating a 3D mesh of the submarine and support system as shown in Figure 2.10. A mesh size convergence check was performed to determine the coarsest mesh that would provide the most accurate results. The computational time of the current models vary between 1 hour and 12 hours depending on the type of analysis using 4 cores of a 512-node computer cluster (8 cores per node, 2.33 GHz CPU, 12 GB RAM per node). The computational time is large because of the numerous surface-to-surface contact interactions and also because the submarine is not rigidly fixed to the ground, instead it is "floating" on the sling system.

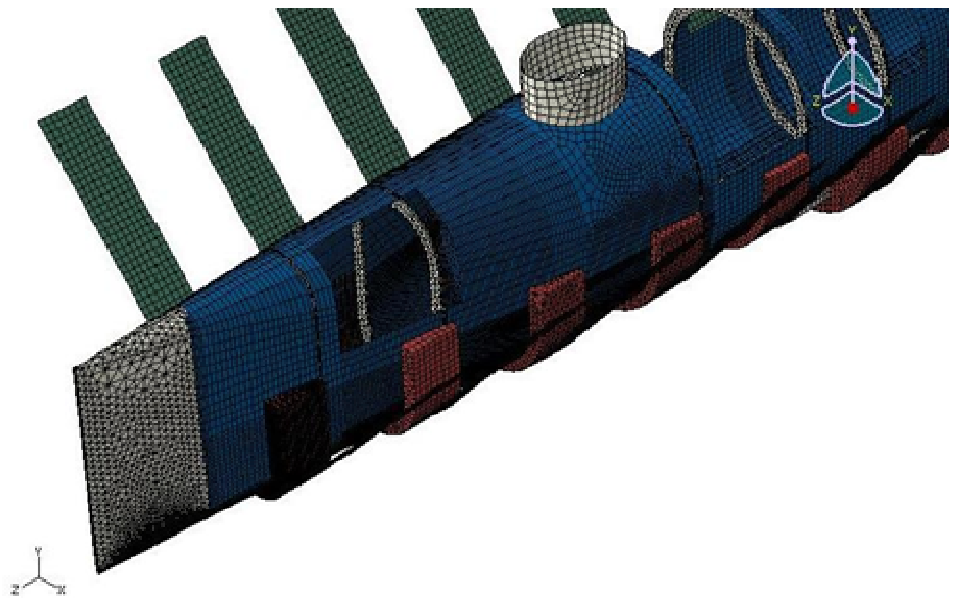


Figure 2.10. Finite element mesh

2.7 Results:

The stresses were evaluated under three conditions:

- 45-degree roll angle in slings,
- upright position in slings, and
- upright position on fixed keel blocks.

The Von Mises stress distribution is shown in Figure 2.11 to 2.14. The maximum stress varies between 5.6 and 17.3 MPa. The maximum stress generally occurs are the corners where plates are missing due to stress concentration. The bottom of the hull is also highly stressed in locations of contact with the fixed keel blocks when in the upright position. Compared to the estimated strength of wrought iron (i.e., 180 MPa), however, these stresses translate to a factor of safety between 10 and 32, which is fairly large [11].

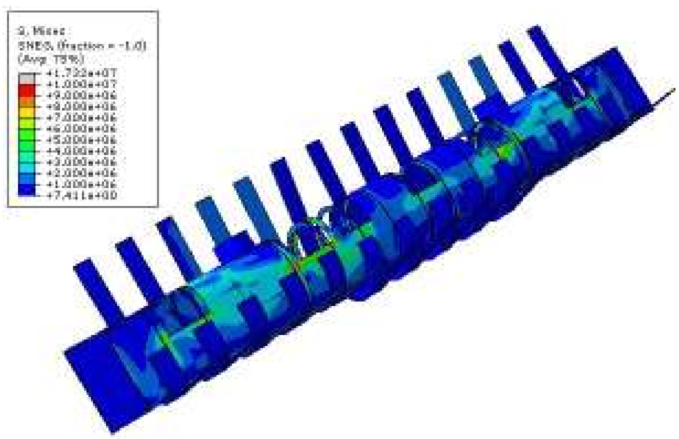


Figure 2.11. Submarine in slings at 45-degree roll angle (max stress = 17.3 MPa)

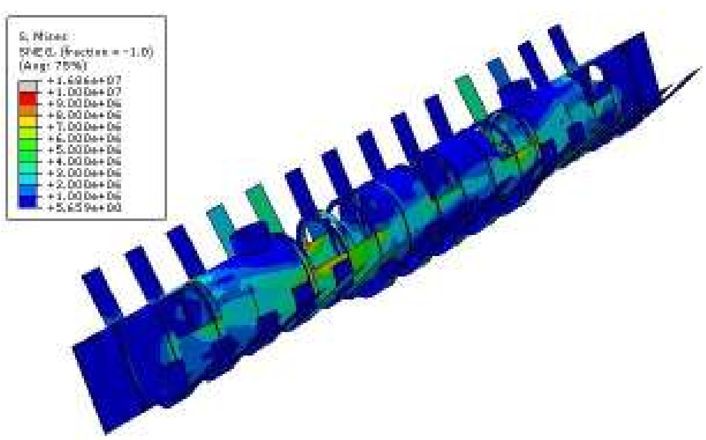


Figure 2.12. Submarine in slings at 20-degree roll angle (max stress = 16.9 MPa)

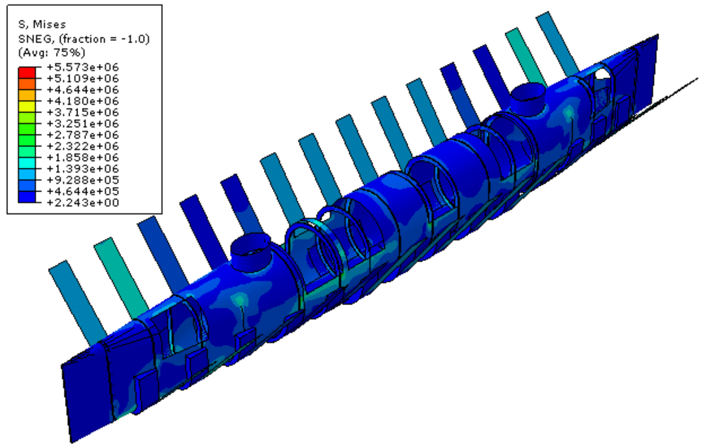


Figure 2.13. Submarine in slings in upright position (max stress = 5.6 MPa)

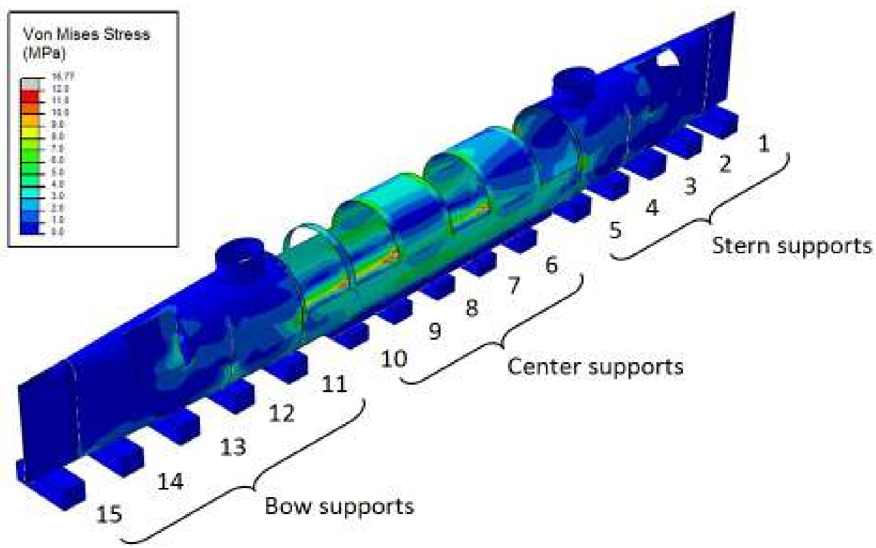


Figure 2.14. Submarine in upright position on fixed keel blocks (max stress = 16.8 MPa)

The above results do not consider the effects of the riveted connection. Hence the stress values found do not correctly represent the actual stresses in the submarine. The localized stress values at the riveted connections are found through local-global coordination.

2.8. Parametric study

The initial thickness of all plates is 8 mm. However, to find out the effects of corrosion, the plates were assumed be decrease in thickness. A parametric study was performed using the model at 45-roll angle in slings. The thickness of the plates was set to 8 mm, 6 mm, 4 mm, and 2 mm and the maximum stress was recorded.

Although this method assumes that the plate thickness is uniform throughout the submarine, which is probably not the case, it provides valuable information on the effect of plate thickness on the maximum stress.

The maximum stress does not occur at the same position. Therefore two locations were selected and the effect of thickness on stress at those particular locations is summarized in Table 2.1.We can observe that, with the decrease in the thickness of the submarine, the stress experienced by the submarine is increased significantly. Hence, the effects of corrosion cannot be ignored.

Table 2.2 Parametric study of effect of corrosion on stress in global model

Thickness	Max Stress (Location 1)	Max Stress (Location 2)	Overall Max Stress
8mm	17.32MPa	10MPa	17.32MPa
6mm	21.2MPa	12.5MPa	21.2MPa
4mm	20MPa	25MPa	30MPa
2mm	60MPa	79MPa	79MPa

CHAPTER 3

LOCAL MODEL

3.1 Riveted connections

The Hunley submarine includes many riveted seam connections between plates (Figure 3.1) and a large number of rivets (more than 4000). Since it is virtually impossible to model the complete structure with a detailed description of each rivet, a coordination procedure between the global model and a local model of a riveted connection will be used to estimate the stresses in riveted connections and in the hull. The overall idea is to apply the stresses of the global model to the local model in the form of an equivalent force applied on the plates and induce a transverse shear in the rivet. The procedure is explained in more detail in Chapter 4.

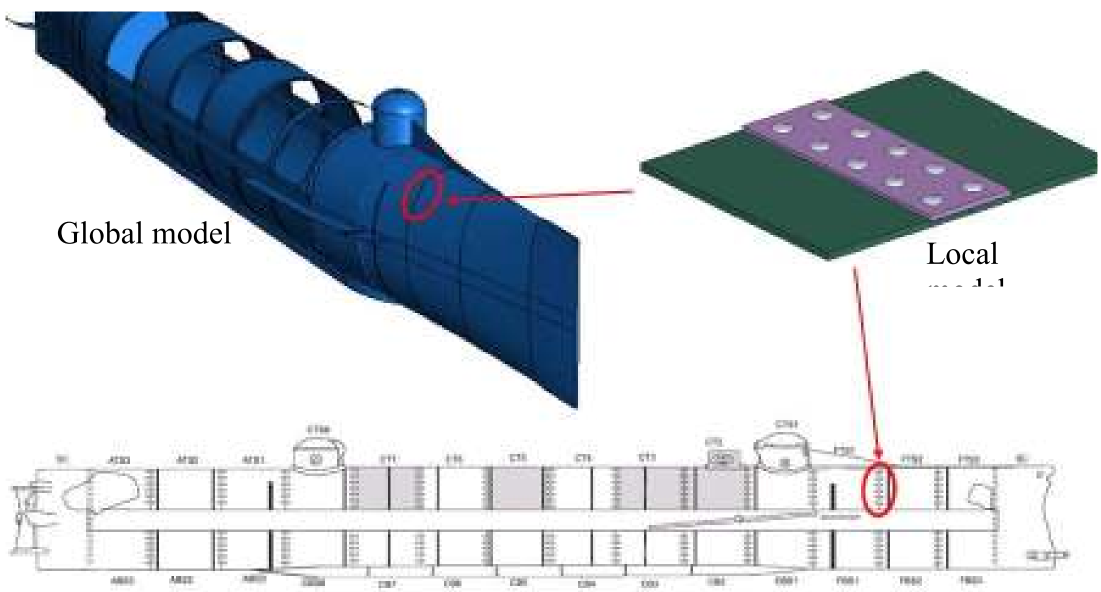


Figure 3.1. Riveted connections are modeled as uniform plates in the global model

Three different models are used to employ the local-global coordination. To reduce the complexity of modeling and run time, a simplified global model is used instead of the global model introduced in the previous chapter. The detailed assembly of the Simplified global model of the submarine is presented in the Chapter 4. Three Finite Element models are used in this coordination procedure as shown in Figure 3.2:

a) Simplified global model of the submarine,

b) A three-dimensional local model of a riveted connection, and

c) A three-dimensional local model of a riveted connection using the same simplification as in the global model, namely, using shells.

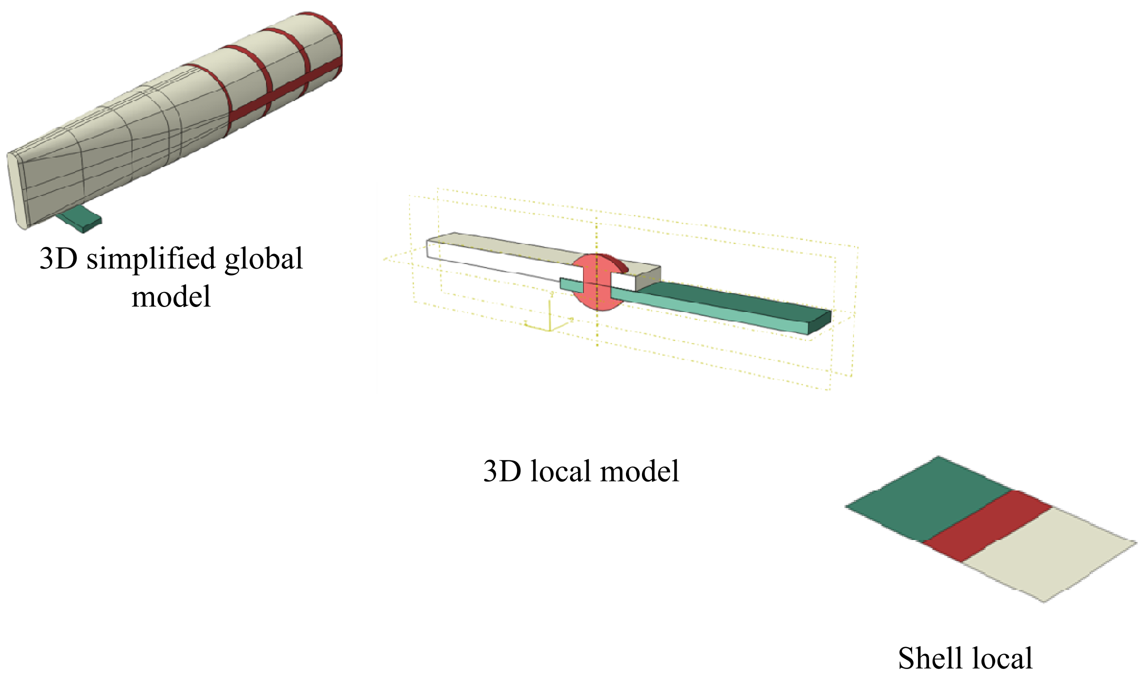


Figure 3.2. The three models used in the coordination procedure

Chapter 3 focuses on the description and results of the three-dimensional local model of riveted connections. The actual shape of the rivets of the Hunley is slightly different from the shape of the local model as shown in Figure 3.3. This was done to simplify the numerical simulation for the current research. Future work should apply the procedure to a more realistic shape of the rivets.

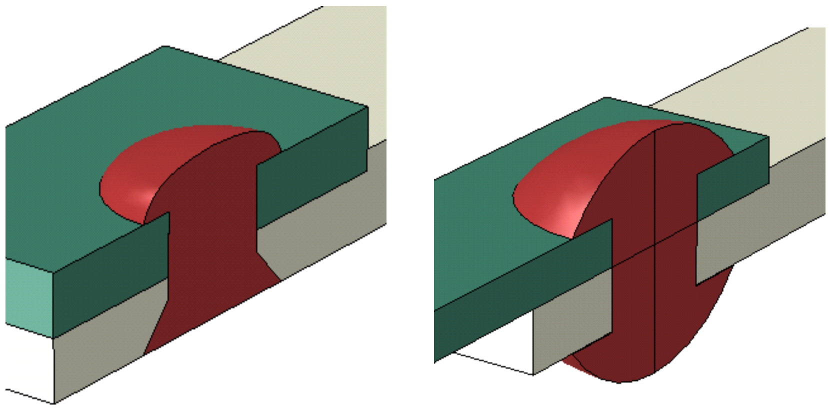


Figure 3.3. Actual shape of rivets of the Hunley (left) and shape used in this research (right). The exterior end of the actual rivets is flush with the exterior plate.

3.2 3D local model

The 3D local model consists of top plate and the bottom plate connected using a rivet. It represents one of the riveted connections in the H.L Hunley submarine.

3.2.1Top Plate

The top plate is plate with 150mmx 25mm dimensions. The detailed Top view and the front view of the plate can be seen in the picture below. The riveted hole has a diameter of 18mm.

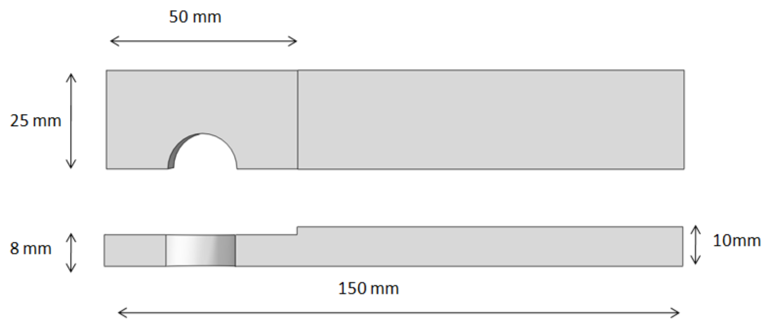


Figure 3.4. Front view and the Top view of the Top Plate.

3.2.2 Bottom Plate

The bottom plate is also a square plate with 150mmx25mmx6mm dimensions. The isometric view of the bottom plate can be seen the figure below.

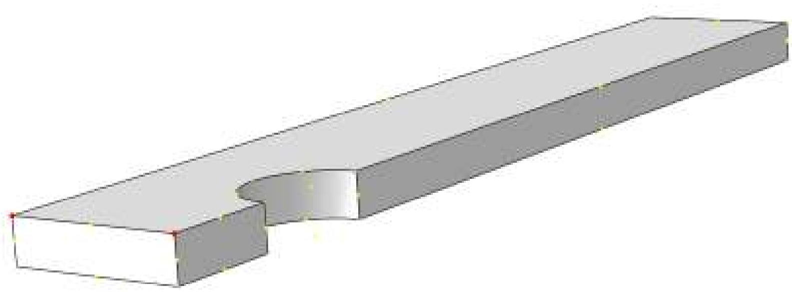


Figure 3.5 Three dimensional model of the Bottom plate

3.2.3. Rivet

The length of the rivet is 14.1mm. A 100 microns allowance is created in rivet to accommodate for the bolt load. The 3D view of a rivet can be seen the following figure.

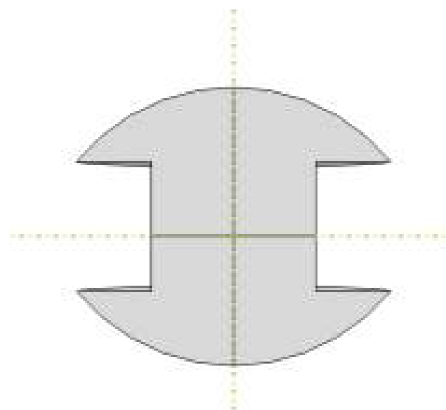


Figure 3.6 Three dimensional model of the rivet

3.3 Assembly and Material Properties:

The top plate and the bottom plate are connected using the three rivets. The bottom plate, top plate and rivet are made of wrought iron. The materials properties are the same as those defined in the global model. The assembly of the three dimensional local riveted model is shown in the figure below.

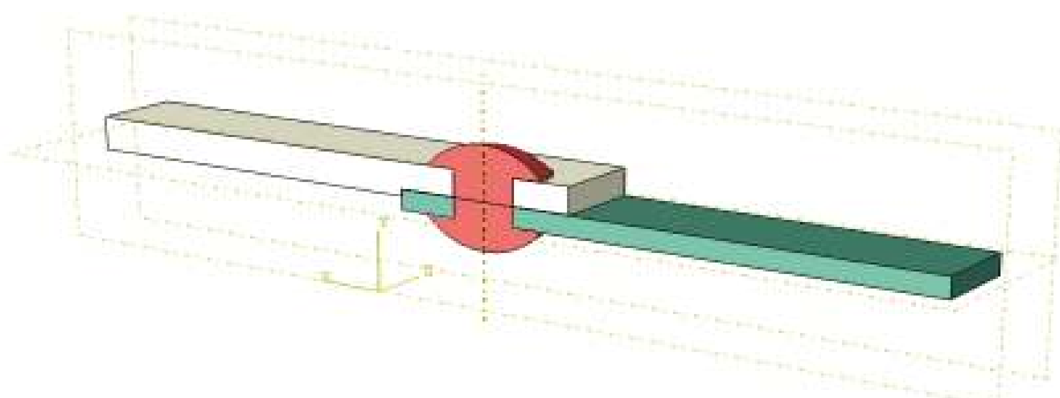


Figure 3.7 Assembly of the 3D local model

3.4 Interaction properties:

The contact properties applied in the 3D local model are critical since the entire coordination between the 3D local model and the local shell model is based on the deformation. ABAQUS 6.8 allows us to use the surface-to-surface contact interaction property when two surfaces are in contact with each other. One of the surfaces is assigned to be the master surface and the other surface is assigned to be the slave surface. It is recommended to use a finer mesh size for the slave surface than for the master in order to reduce numerical error [23].

There are three contacts in the 3D local model. All the contacts are modeled as surface-to-surface contacts.

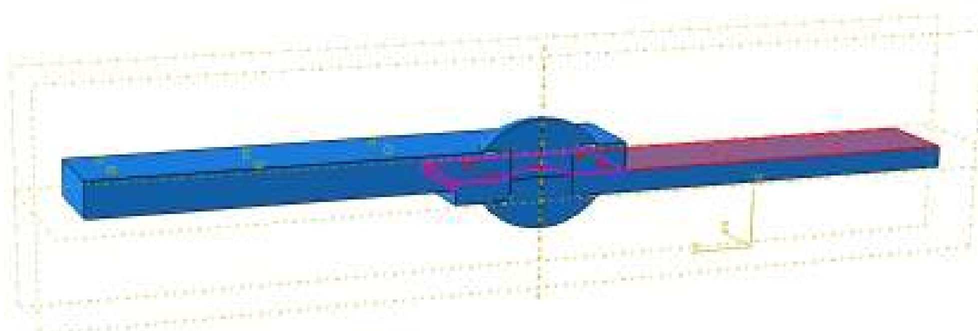


Figure 3.8. Contact interaction between the top plate and the bottom plate

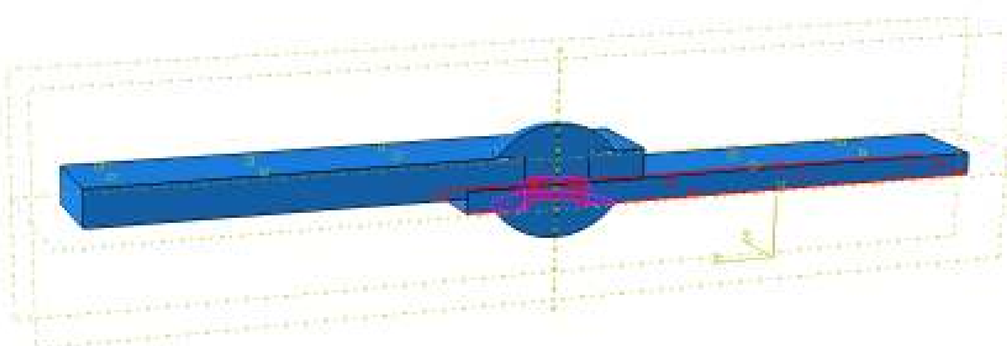


Figure 3.9. Contact interaction between the bottom plate and the rivet

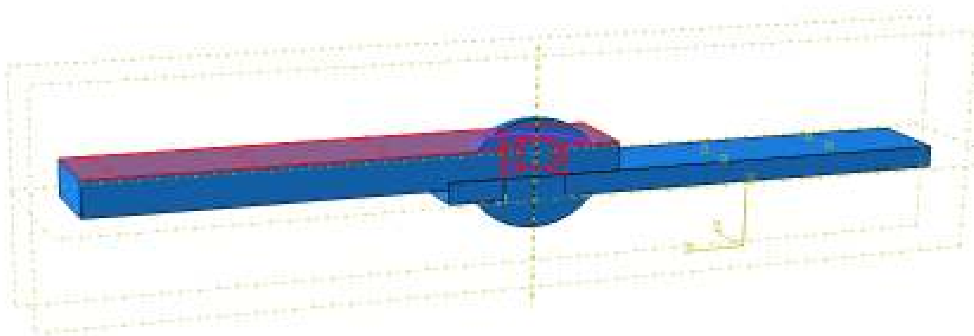


Figure 3.10. Contact interaction between the top plate and the rivet

A surface smoothing technique is enabled to alleviate potential numerical issues in calculating the contact stresses. Also since the 3D local model analysis is expected to have lower deformation, the surface smoothing technique is expected to have a higher significance [24].

The friction coefficient between all surfaces is assumed to be 0.8 as a reference. Since the friction coefficient is unknown, a parametric study, presented in Section 3.7, was performed to estimate its effect on the response of the riveted connection.

3.5 Boundary Conditions

Since the rivet is not rigidly connected to any fixed point, its static equilibrium can be numerically found only when the contacts are properly activated. Therefore the analysis is decomposed into at least three consecutive steps. In the initial configuration, a small gap is defined between the two plates and both heads of the rivet by making the rivet slightly longer than expected (by 0.1 mm). In the first step, one end of the rivet is fixed and a “bolt load” (i.e., artificial shrinkage) is applied to the rivet to close the gap and activate all contacts. In the second step, a longitudinal force is applied on the top plate to represent the stress of the global model. In the third step, the rivet is releases, which provides the static equilibrium of the connection due to the force applied in step 2. A fourth step can then be defined to release the bolt load, which would be equivalent to the creep behavior of the rivet over time. Three boundary conditions are applied and removed at different steps of the analysis

- Boundary condition 1: One end of the rivet is fixed in the first step. This boundary condition becomes inactive in step 3 after the bolt load is applied on the rivet.

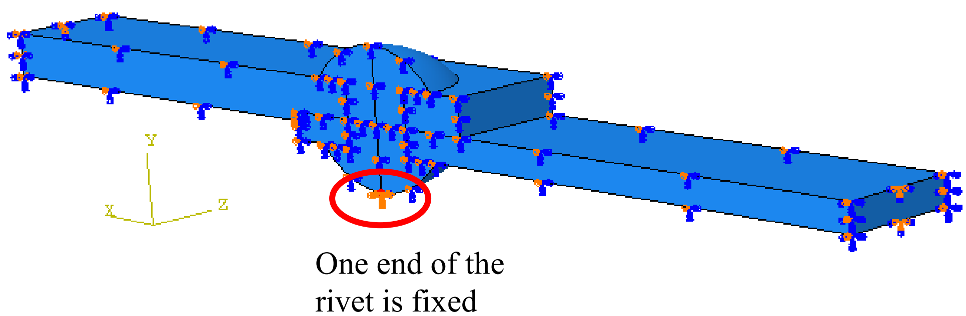


Figure 3.11. One end of the rivet in fixed in steps 1 and 2

- Boundary condition 2: To apply the bolt load, the bottom plate is fixed. In step 3 pressure is applied on the face of the bottom plate and hence this boundary condition becomes inactive once the bolt load on the Rivet and the pressure on the bottom plate are applied.

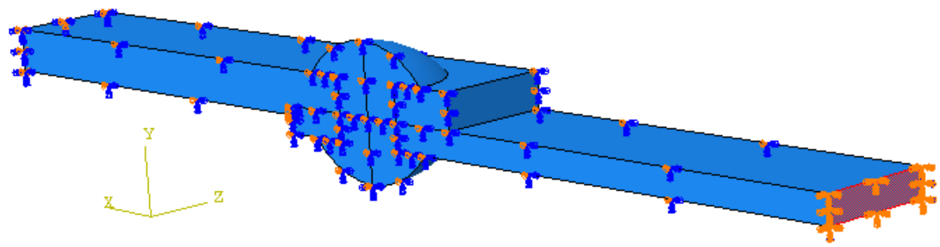


Figure 3.12. The end of the bottom plate is fixed in the steps 1 and 2

- Boundary condition 3: Since we are assuming symmetry and repetition for each rivet along the seam line, the sides of the plates and the rivet are constrained in rotation about the x- and y-axes and translation in the z-direction. This boundary condition is active for the whole analysis.

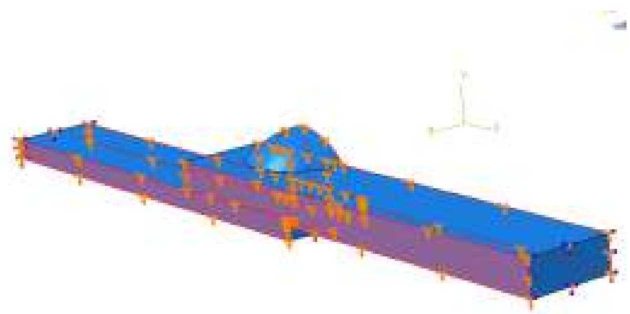


Figure 3.13. Boundary conditions applied for symmetry and repetition of the riveted connection along the seam line

- Boundary Condition 4: The top plate is fixed at one end for the entire analysis.

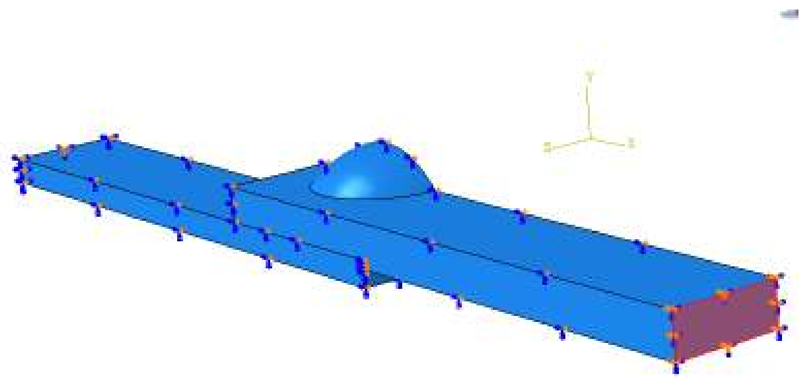


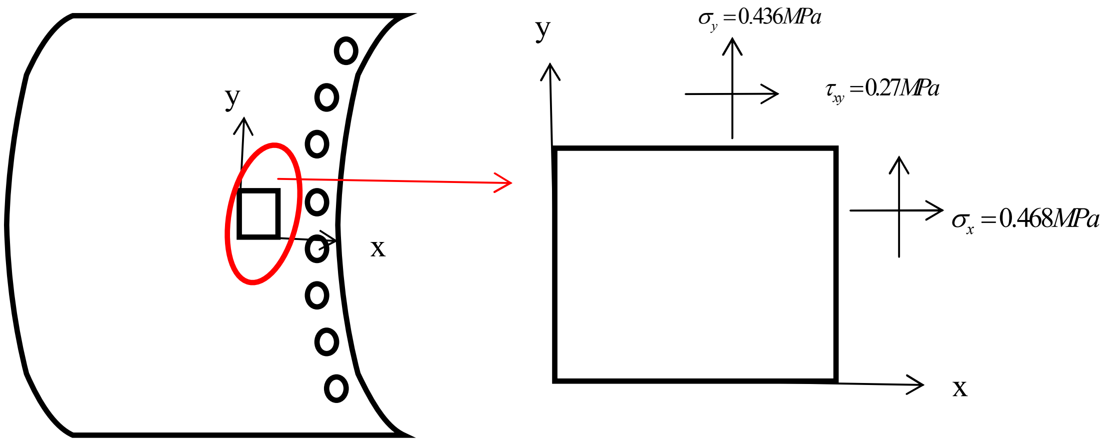
Figure 3.14. The end of the top plate is fixed for all steps

3.6 Loads

The loads applied on the local model are supposed to be representative of the state of stress in the riveted connections of the global model. In theory, this state of stress includes three normal stresses and three shear stresses. Since the global model considers the connections as thin shells, the only significant stresses are in the normal stresses σ_x and σ_y and the shear stress τ_{xy} shown in Figure 3.12. The other three components are assumed to be negligible. Since the normal stress σ_y is applied in the direction of the seam line, it is transferred through both plates in the same direction and therefore does not induce significant stress in the rivets.

Comparatively, the normal stress σ_x is applied in the perpendicular direction of the seam line and therefore has a tendency to pull the plates away from each other (if in tension) and as a tendency to shear the rivet transversally.

The shear stress τ_{xy} has a tendency to move the two plates away from each other in the y-direction, which induces a transverse shear stress in the rivet. However, this thesis focuses exclusively on the effect of σ_x.



Global model region with riveted connections

Stress calculations in x, y, directions, from global model analysis for an element in the riveted connection region

Figure 3.15. State of stress from the global model

Based on the above explanation, the load applied on the local model corresponds to the normal stress σ_x of the global model. The load is defined as a pressure applied on the end surface of the top plate as shown in Figure 3.13. Arbitrary amplitude of 1 MPa is used for the pressure load.

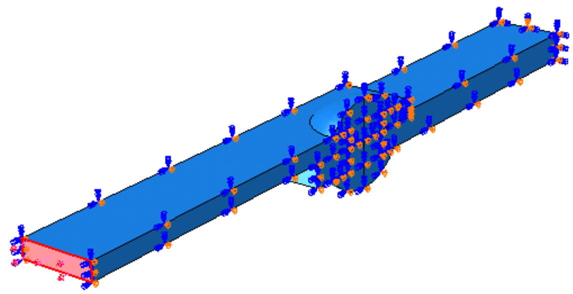


Figure 3.16. Pressure load applied on the top plate

The second load applied on the local model is the bolt load used to close the initial gap of 0.1 mm and activate all contacts. The bolt load corresponds to shrinkage of the rivet of 0.11 mm, which induces a compression of 0.01 mm in the two plates. This value is also arbitrary since the actual pre-tension in the rivets is unknown. However, the effect of this pre-tension is studied in the next section.

3.7 Results

The maximum stress found in the model is 10.76 MPa (Figure 3.14). These results are for fixed values of the friction coefficient and pre-tension of the rivet.

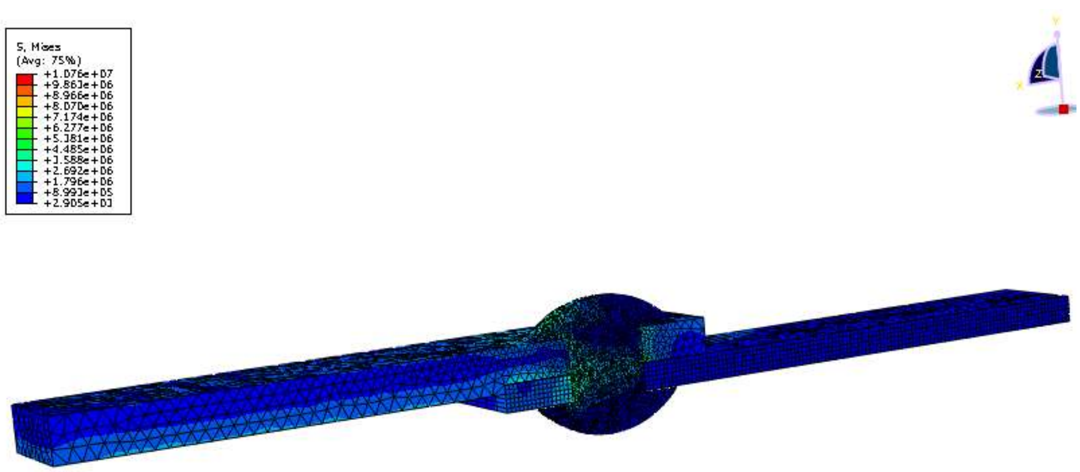


Figure 3.17. Stress calculations in the local rivet model

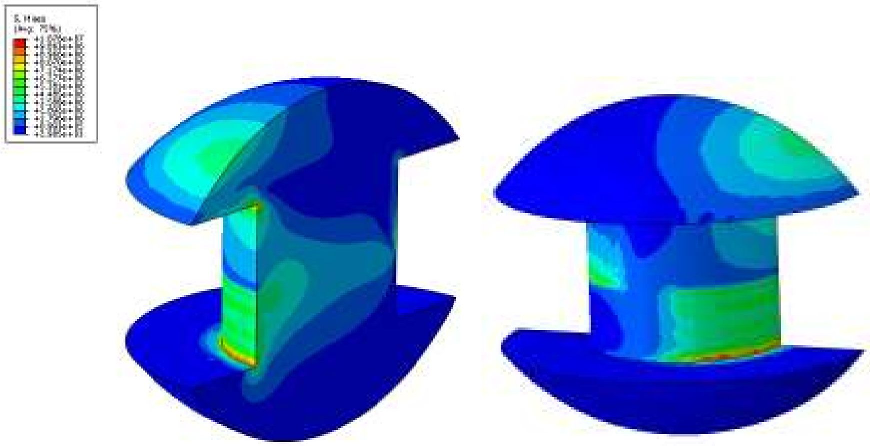


Figure 3.18 Von Mises stress distribution in rivet

The effect of the friction coefficient and pre-tension is studied by varying the values of the friction coefficient from 0.4 to 1.0 with an increment of 0.2. The stress values are evaluated at a single location on the rivet. From the parametric study it was found that the friction coefficient does not affect the stress in the rivet significantly.

Table 3.1 Parametric study on the effect of Frictional coefficient on Maximum stress

Friction coefficient	Maximum stress
1	9.05 MPa
0.8	9.19 MPa
0.6	9.21 MPa
0.4	9.69 MPa

Studying the effect of pre-tension on the stress generated in the riveted connection is crucial. Since the Hunley was under water for many years, it is hard to predict if there is still any pretension in the rivets. It was found that pretension has a negligible effect on the amount of stress generated in the riveted connection.

Table 3.2 Parametric study on the effect of pretension on Maximum stress

Pretension as a length adjustment (mm)	Maximum stress(MPa)
0.10	9.19
0.20	9.21
0.30	9.22
0.40	9.19

3.8 Shell local model

The second local model of the riveted connection is modeled using shell elements. The Shell model has three regions with three different thicknesses in it. The central region is supposed to mimic the behavior of the actual rivet. In this section, the thickness of the central region is set to 14 mm and the other regions are the same as in the 3D local model, i.e., 10 mm and 6 mm. The width is 25 mm. The material properties assigned are similar to the local 3D model.

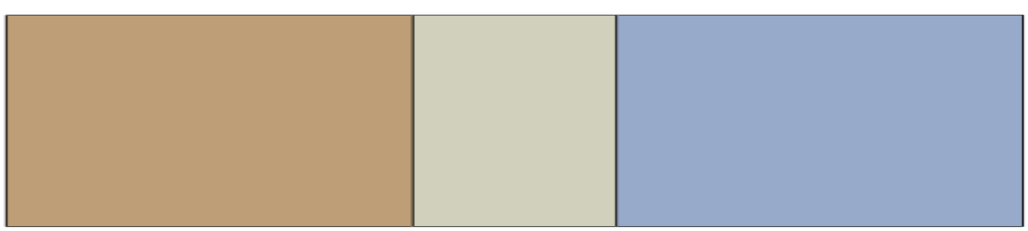


Figure. 3.19. Shell local model

3.8.1 Loads and Boundary conditions

The shell edge with 10mm thick is equivalent to the top plate in the 3D local model, therefore the edge is fixed. An edge force of 6000 N/m is applied at the other end, which, given the width of 25 mm and thickness of 6 mm, is equivalent to the stress of 1 MPa defined in the 3D local model.

3.8.2 Results

The stresses generated in the shell model, which represents the 3D-local model, are determined. The values of stress vary between the 3D local model and the shell local model. Since the shell local model do not have the imperfections of the 3D local model such as the rivets and the rivet holes, the stresses and the displacement do not match. The Maximum stress was found to be 0.89 MPa. The displacement is also less compared to the 3D riveted model. These are matched using a method explained in the next chapter.

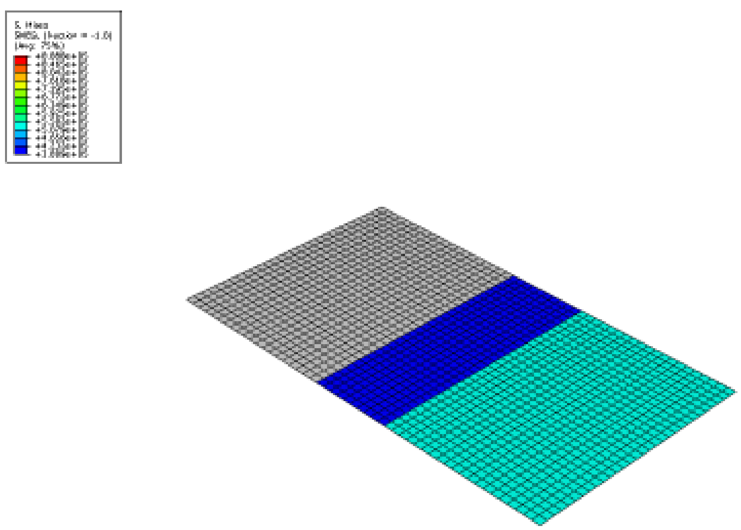


Figure 3.20 Stress analysis of local shell model

CHAPTER 4

FINDING ACTUAL STRESSES IN RIVETED CONNECTION

4.1 Matching 3D local model to the shell local model

The 3D local model has a riveted connection which is replaced by a plate with higher thickness in the shell local model. Hence there are bound to be differences in the results between the models. Now a method is employed to match both models.

Method adopted:

1. The displacement of the 3-D local model is calculated

2. The displacement of the shell local model is calculated

3. The thickness and young's modulus of the riveted region in the shell local model is manipulated such that both the displacements in 3-D local model and shell model are equivalent

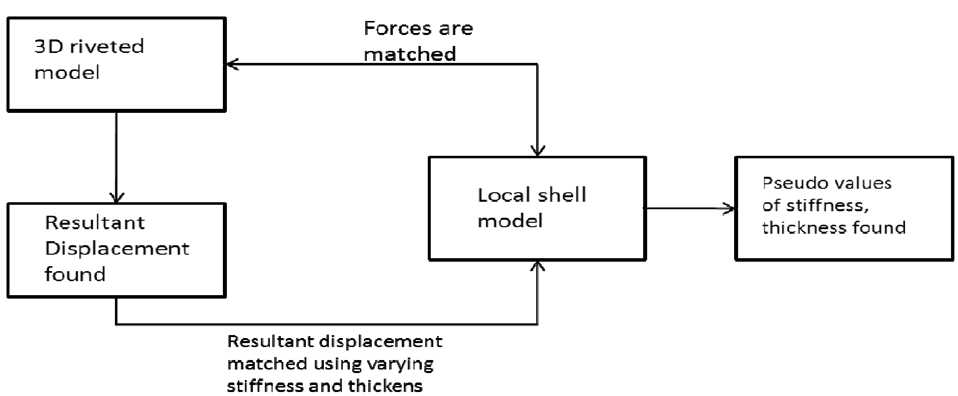


Figure 4.1. Block diagram of the procedure used to find the pseudo values of stiffness, thickness of riveted region

From the results of the 3D local model, it was found that the bottom plate has a displacement of 3.36 microns. Also the displacement of the shell plate was found to be 0.78 microns. Since we need to match the displacements of both the model, a parametric study is conducted to find out the values of Young's modulus and thickness in the riveted connection of the shell model. These values will increase the displacement of the shell model to 3.36 microns.

The initial thickness of the riveted connection in the shell model is 14 mm. A parametric study was conducted by decreasing the thickness from 14 mm to 1 mm. For this entire model the Young's modulus was 210 GPa. However, even with a thickness value of 1 mm the displacement value was less than that of the 3D model. Now at the decreased thickness, the young's modulus of the riveted region was also decreased. The Young's modulus was decreased from 210 GPa to 100 GPa to find the appropriate combination of Young's modulus and thickness of the riveted region.

After conducting the parametric study of the thickness and the stiffness values, few combinations of these values were found. All these values when given to the riveted region of the shell model will yield in a displacement value equal to the 3D model.

Table 4.1 Pseudo values of stiffness, thickness of riveted region after matching displacement

Stiffness(GPa)	Thickness(mm)
204	0.5
102	1
51	2

25.5	4
12.75	8

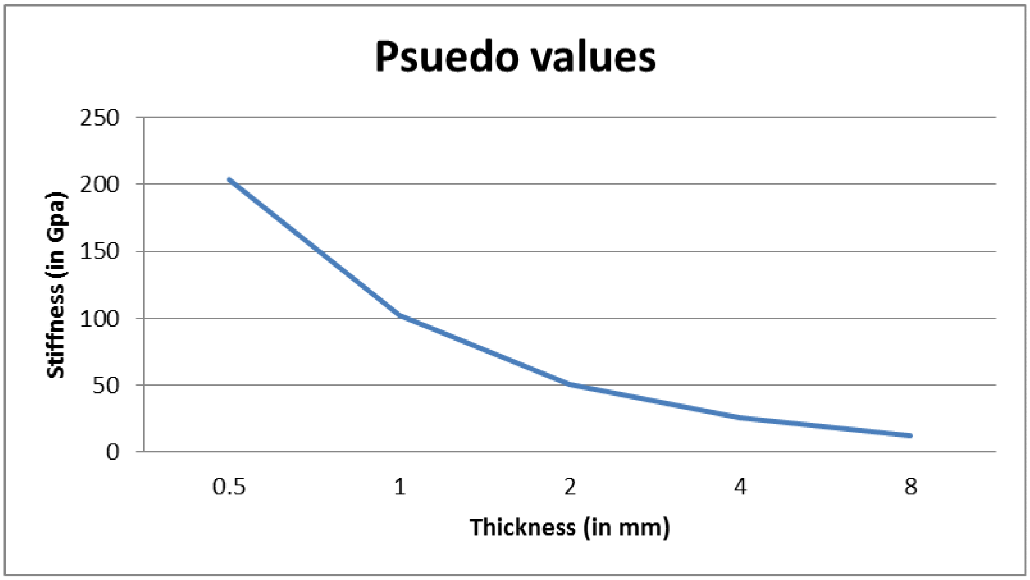


Figure 4.2 Graph demonstrating the pseudo values of the stiffness and thickness of the riveted region

4.2 Simplified global model

The pseudo values of stiffness and thickness of the riveted connection which corresponds to the riveted region in a 3D local model were found. These pseudo values must now be substituted in the riveted regions of the global model.

Since the global model is a large model with a high number of parts and interactions, the effect of the riveted regions are studied using a simplified global model.

4.2.1 Simplified submarine

The simplified submarine, shown in Figure 4.3, is a shell model which does not consist of the different parts of the main assembly such as the bow, stern, keel block, bulk heads, hoops etc. It is a simple shell model which is representative of the submarine. For reducing the analysis time, only half of the model was considered.

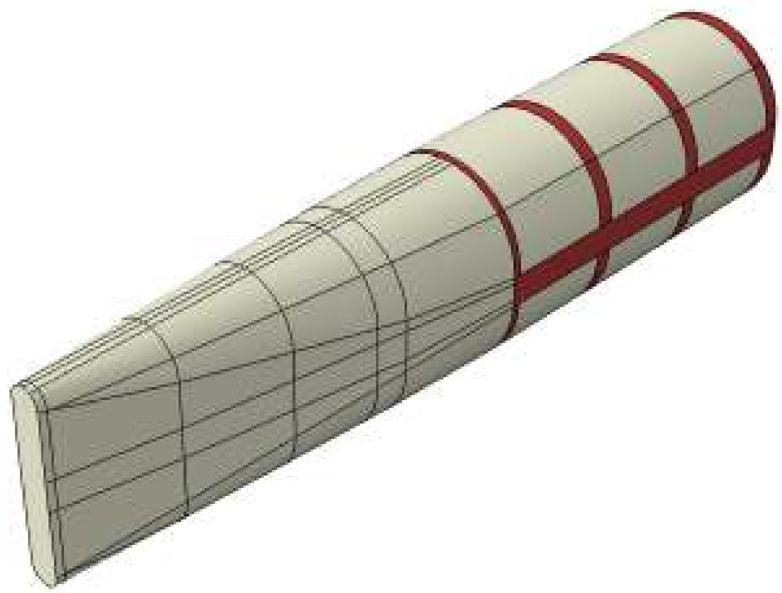


Figure 4.3. Simplified 3D global model

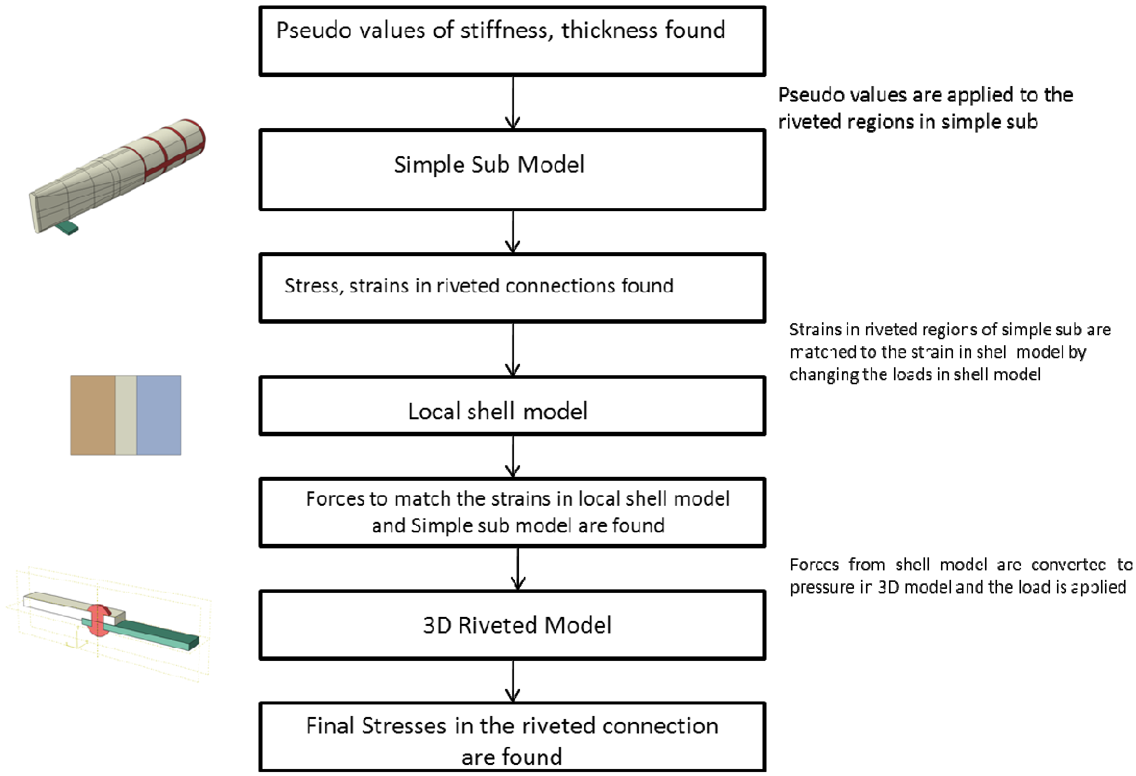


Figure 4.4. Flow diagram of the procedure used to find the final stress values in the riveted connection

4.2.2 Material properties

The simplified sub has two regions, which are a main submarine region and the riveted connections regions. The whole submarine except the riveted connections is assigned the following material properties:

Region	Thickness	Young's Modulus	Density
Submarine (without riveted connections)	8mm	210 GPa	7800
Riveted connection region	14mm	210 GPa	7800

4.2.3 Finite Element Analysis of the simplified submarine

The effect of the riveted connections in the submarine is studied using the simplified submarine. Since we just need to study the effect of the riveted connections, the submarine is assumed to be supported on a single cushion. The simplified submarine is assumed to be cantilevered at one end. The cushion is placed at distance of 12 microns below the submarine before the start of the simulation.

The stresses developed in the submarine and the cushions are studied when the support is raised by a distance of 15 microns towards the submarine. This can be alternatively understood as the stresses developed when the submarine is lifted by 3 microns.

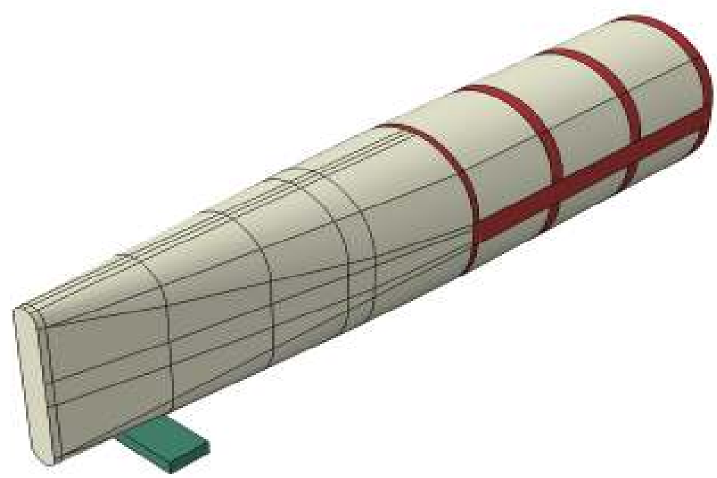


Figure 4.5. The 3D assembly of the simplified submarine model

An interaction between the cushion and the submarine was simulated using a surface-to-surface contact in Abaqus and a frictional coefficient of 0.95 was assigned.

4.2.4 Results

The maximum stresses occurred in the submarine were at the contact of the cushion and the submarine. The results obtained from the analysis were as below.

Table 4.2 The results of the Simple global model analysis

Max Stress in Sub (MPa)	Max Displacement in Sub (μm)	Max Stress in cushion (MPa)	Reaction Force in cushion (KN)
83.46	1.29	1.28	59.7

We have obtained the results of the submarine when the thickness of the riveted region was not accounted for the local to global model analysis. Now we substitute the pseudo values that were obtained from our analysis to the riveted regions and compare the results. The riveted regions are assigned the flowing properties.

Table 4.3 The material properties assigned to riveted region in the Simple global model

Thickness	Young's modulus	Density
4 mm	25.5 GPa	7800 kg/m^3

The resultant assembly is less stiff. The overall stress resulted in the submarine was found to be much smaller when the pseudo values were assigned to the riveted regions. Since the regions were less stiff the submarine was lifted easily. This resulted in the lower reaction forces in the cushion and lower stresses in the submarine. The displacement was also found to be higher when the stiffness was low. Detailed comparison between the values of both the scenarios is given below.

Table 4.4 The difference in the results of Simplified before and after adding pseudo values

Pseudo thickness and stiffness (riveted region)	Max Stress in Sub (MPa)	Max Displacement in Sub (μm)	Max Stress in cushion (MPa)	Reaction Force in Cushion (KN)
14mm, 210 Gpa	83.46	1.29	1.28	59.7
4mm, 25.5 Gpa	54.88	1.36	0.97	42.94

4.3 Validating the thickness and stiffness values

From the procedure discussed in the section we have found a set of thickness and stiffness values. Now these values were assigned to the riveted regions in the submarine and check whether the values yield similar results. The values are in the table below.

Table 4.5 Validation of the pseudo values

Pseudo thickness and stiffness (riveted region)	Max Stress in Sub (MPa)	Max Displacement in Sub (μm)	Max Stress in cushion (MPa)	Reaction Force in Cushion (KN)
0.5 mm, 204 GPa	54.34	1.36	0.97	42.97
1 mm, 102 GPa	54.37	1.36	0.97	42.95
2 mm, 51GPa	54.25	1.36	0.965	42.89
4 mm, 25.5GPa	54.88	1.36	0.97	42.94

From the above table we can see that the results for pseudo values of the thickness and stiffness of the riveted region are comparable.

4.4 Matching the strain

We now match the strain of the riveted region in the simple sub model to the strain in the local shell model. The strains are matched by changing the load that we applied in the shell model.

The Maximum strain in the riveted connections in the simple sub was found to be 4.18×10^{-4}.Now this strain is matched to the strain in the local shell model. The local shell model has the strain of 5.59×10^{-5}.

The strain in the local shell model has to be matched to the strain in the simplified global model. This is done by increasing the load on the local shell model. The local shell model currently has a shell edge load of 6000 N. The load applied is increased until the resultant strain is increased from 5.59×10^{-5} to 4.18×10^{-4}.

The resultant load applied was found to be 45000 N by trial and error method.

4.5 Finding the final stresses in the riveted connection

Now we have the shell edge load that matches the strain values of the local shell model and the simplified global model. The shell edge load is converted into an equivalent pressure to be applied onto the 3D local model. This will provide the final stresses in the riveted connection.

The pressure to be applied on the face of the 3D local model was found to be 7.5 MPa. This pressure is applied onto the final model and the resultant maximum stress is 92.11 MPa.

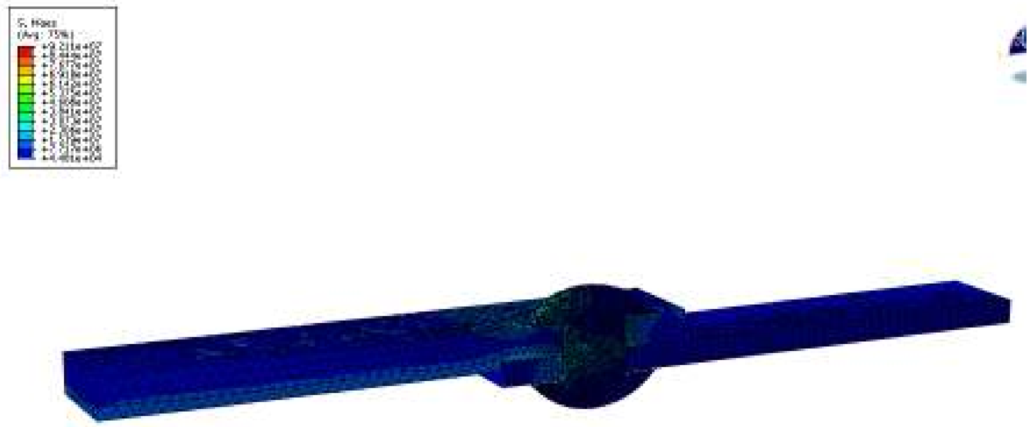


Figure 4.6 Stress calculations in the 3d rivet model after increased pressure

The maximum stress form the simplified global model was found to be 54.88 MPa. After applying the local-global model coordination it was found that the localized stresses in the riveted connection are 92.11 MPa. This shows that modeling the whole submarine as shell elements would neglect the stresses in the riveted connection which are comparatively larger than the stresses found from the global model analysis.

Verification of the model:

The final model with increased pressure is verified with the local shell model. This is done by verifying whether the displacements of the local shell model and the displacement of the 3D riveted model with increased pressure are equal.

The displacement of the 3D riveted model was found to be 0.246 microns and the displacement of the shell model was found to be 0.252 microns, which is a good correlation.

CHAPTER 5

CORRODED RIVETED MODEL

5.1 Introduction to Marine corrosion

When a metal is dropped into sea water, the surface of the metal reacts with the sea water and this reactive action is known as corrosion. Types of corrosion can be broadly categorized in to the following:

- General Corrosion
- Localized Corrosion
- Erosion/Corrosion
- Galvanic corrosion
- Stress corrosion cracking
- Corrosion Fatigue[25]

As discussed earlier, the H.L. Hunley is heavily corroded. There is a lot of uneven corrosion in the hull, which is difficult to model using Finite Element Analysis.

5.2 Modeling corrosion

During the conservation of H.L. Hunley, it was found that the rivet heads in the submarine were heavily corroded. The remaining part of the rivet is found to be healthy. It is important to analyze the effect of the corroded rivet on the 3D riveted model. Earlier parametric study was conducted to study the effects of corrosion on the stresses in the global model. Corrosion was incorporated in the global model by assuming the corrosion to be loss of metal. Therefore the shell thickness was reduced to incorporate the effect of corrosion. Several studies on corrosion were done using this method [8, 9, 10].

Figure 5.1. The corrosion in the H.L Hunley submarine (Photo courtesy of the Friends of the Hunley)

Since the corrosion is significant in the rivet heads, corrosion was incorporated as the loss of metal in the rivet head region of the 3D local model. Initially a part of rivet head was modeled as the corroded region. The material properties were assumed to have smaller stiffness and density in the corroded regions. However, this approach lead to numerical difficulties and the FEA analysis did not converge. . Therefore the corrosion was incorporated as a void equivalent to the loss of metal as shown in Figure 5.2.

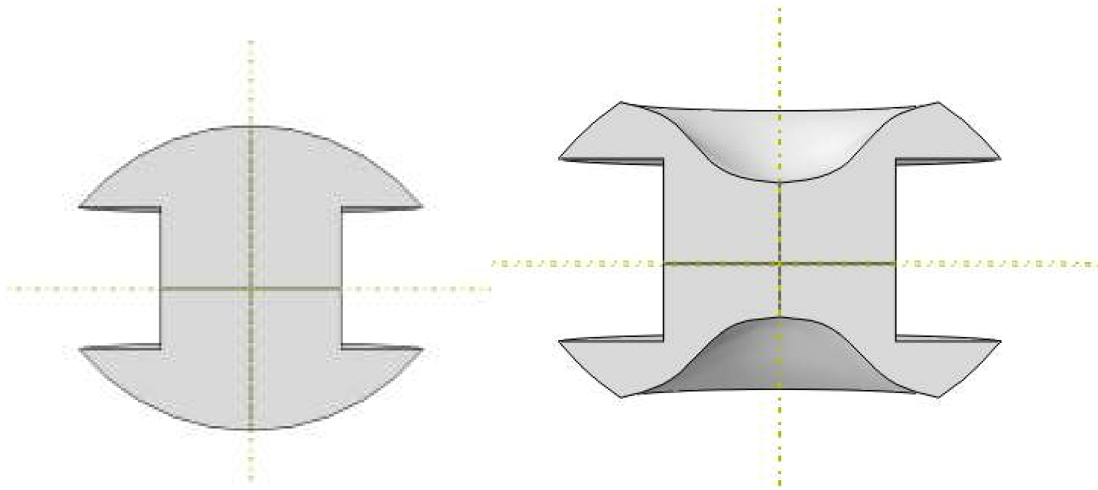


Figure 5.2. Rivet model before and after the application of corrosion

5.3 Finite Element Analysis

The 3D riveted model is analyzed as discussed in chapter 3. The rivet used earlier is substituted by the corroded rivet. All the boundary conditions and the loads applied to the model are same as earlier. The 3D corroded rivet assembly is shown in the figure below. The interaction properties are also assumed to be the same as the 3D riveted model analysis.

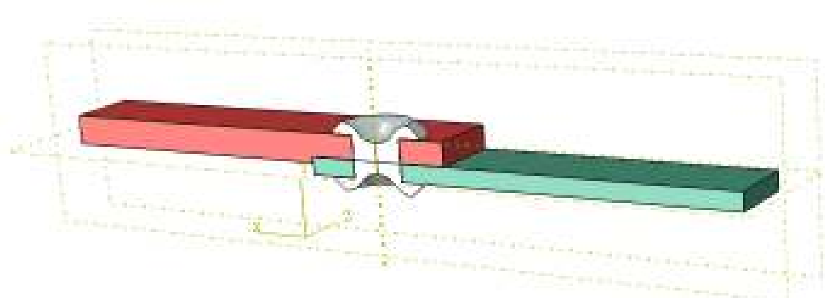


Figure 5.3. Three-dimensional local model of a riveted connection

5.4 Results

The effect of corrosion of the rivets on stresses induced in the 3D local model is studied. The stresses are found to be critical in the riveted connections. The 3D local model has a load of 1MPa applied on the bottom plate. Since only the top and bottom of the riveted part are assumed to be affected by corrosion, it was found that the corrosion does not have a large effect on the stresses induced in the connection.

The riveted connection has one critical node with high stresses. The Von-Misses stresses at that node in the rivet are determined. The critical node in the rivet is shown in the figure below.

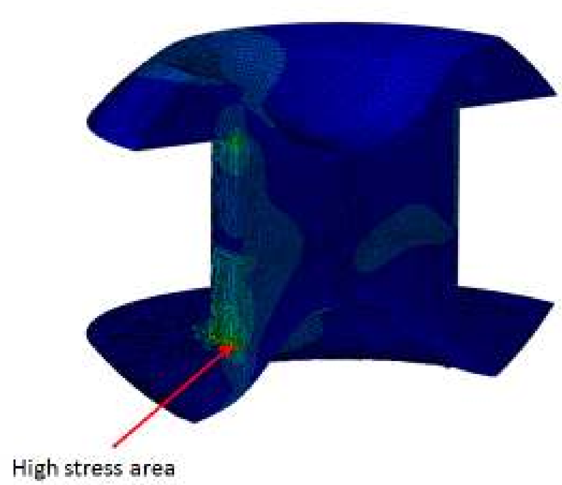


Figure 5.4 Von Mises stress in the corroded rivet

The Principal stresses in the high stress region are found out from the corroded rivet analysis. They are as follows:

$\sigma_1 = 1.3$ MPa	$\sigma_2 = 12.7$ MPa	$\sigma_3 = 3.85$ MPa
$\sigma_{12} = 4.45$ MPa	$\sigma_{13} = 0.14$ MPa	$\sigma_{23} = 0.25$ MPa

The Von Misses stress is found by using the following equation:

$$\sigma_{mises} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 \, (\sigma_3 - \sigma_1)^2 + 6(\sigma_{12}^2 + {}_{13}^2 + \sigma_{32}^2)}{2}}$$

Substituting the principal stress values in the above equation, the Von Misses stress in the corroded rivet was found be 12.92 MPa. The non-corroded rivet had a maximum stress of 10.73MPa.

Although there is slight increase in the stresses in the rivet after corrosion, the stress values suggest that corrosion to the head and bottom of the rivet do not compromise the strength of the riveted connection.

Since most of the rivets in the H.L. Hunley are corroded in the similar way, this study suggests that the corrosion to the rivets is not very critical.

CHAPTER 6

CONCLUSION AND FUTURE WORK

6.1 Conclusion

The intent of this thesis is to provide the foundation and the initial ground work for further study on stress analysis of H.L. Hunley submarine using a global-local model coordination procedure. The research presented in this thesis includes the following main contributions:

1. Partial development of several global models of the H.L. Hunley submarine to analysis the structural integrity of the hull.
2. Created a 3D riveted model, shell model to represent the riveted connections in the H.L. Hunley. Carried out the Finite element analysis using ABAQUS 6.8 and determined the stresses in the riveted connection.
3. Created a Simplified Submarine model, corresponding to H.L. Hunley and carried out the stress analysis using ABAQUS 6.8. Developed a global- global model coordination procedure to more accurately find the localized stresses in the riveted connection.
4. Demonstrated the difference between the stresses found using the shell elements in the global model and the stresses in the riveted connection using global- global model coordination.
5. Created a corroded rivet model and studied the effects of corrosion in the rivets in the H.L. Hunley submarine.

6.2 Future Work

Based on the experience gathered during this research project, the following recommendations for future work include:

1. Validation of the global-local coordination procedure using experimental analysis to support the results from the Finite Element Analysis.
2. Substituting the simplified global model with the actual global model. Finding the difference between the localized stresses and the stresses induced in the global model.
3. Analysis of crack propagation around riveted connections using ABAQUS 6.10 should be carried out. Methods such as finding the stress intensity factor, predicting crack growth and propagation should be carried out.

REFERENCES

[1] Walker, S. M. (2005). In S. M. Walker, Secrets of a Civil war submarine: solving the mysteries of the H.L Hunley. Minneapolis: Carolthoda Books Inc.

[2] Naval Historical Center. (2000, June 27). Navy Credits Clive Cussler with Locating Hunley. Retrieved September 22, 2010, from National Underwater and marine agency: http://www.numa.net/articles/navy_credits_clive_cussler.html

[3] Young, M. (2007, October 24). 1838-1874: Sectionalism, the Civil War, and Reconstruction: H.L. Hunley. Retrieved September 22, 2010, from Encyclopedia of Alabama: http://www.encyclopediaofalabama.org/face/Article.jsp?id=h-1376

[4] Curry, A. (2007, June 24). Time Capsule From the Sea: Artifacts from the South's Submarine are Turning Fable into Fact. Retrieved September 07, 2010, from US News & World report: http://www.usnews.com/usnews/news/articles/070624/2hunley_2.htm

[5] Historic Naval Ships Association. (1997). H.L. HUNLEY. Retrieved September 22, 2010, from Historic Naval Ships Association: http://www.hnsa.org/ships/hunley.htm

[6] Mardikian, P. (2004). Conservation and Management Strategies Applied to Post-Recovery and Analysis of the American civil war submarine. The International Journal of Nautical Archeology , 137-148

[7] CONSERVATION. (2010). Retrieved September 22, 2010, from Friends of the Hunley: http://www.hunley.org/main_index.asp?CONTENT=CONSERVATION

[8] Boulent M. Imam, T. D. (2010). Fatigue evaluation of riveted railway bridges through global and local analysis. Journal of Constructional Steel Research, 1411-1421

[9] Matthew A. Russell, D. L. (2006). A Minimum-Impact Method for Measuring Corrosion Rate of Steel-Hulled Shipwrecks in Seawater. The International Journal of Nautical Archaeology, 310-318.

[10] P.A. Slater, M. P. (2000). Finite element analysis of buckling of corroded ship plates. Can. J. Civ. Eng. Vol. 27, 463-474.

[11] Krishnamurthy N, Graddy D. Correlation between 2 and 3-dimensional Finite element analysis of steel bolted end-plate connections. Comput Struct 1976;6:381-389

[12] Kukreti AR, Murray TM, Abolmaali A. End plate con-nection moment±rotation relationship. J Constr Steel Res 1987;8:137-57.

[13] Kukreti AR, Murray TM, Ghassemieh M. Finite element modeling of large capacity stiffened steel tee-hanger connections. Comput Struct 1989;32(2):409-22

[14] O.S. Bursi, J. J. (1998). Basic issues in the Finite element simulation of extended end plate connections. Computers and Structures *69* , 361-382

[15] Ashok L.Ramteke, P. B. (2009). Improving analysis accuracy by modeling rivet/bolts as solids in sheet metal structure. 7th European LS-DYNA conference. DYNAmore GmbH

[16] Montgomery Jerome. Methods of modeling bolts in the bolted joints, ANSYS World conference, 2006

[17] John D. Reid and Nicholas R. Hiser. Detailed modeling of bolted joints with slippage, Finite elements in analysis and design, Volume 41, Issue 6, 2005, Pages:547-562

[18] Timothy E. Dunbar, N. P. (2004). A computational investigation of the effects of localized corrosion on plates and stiffened panels. Marine Structures, 385-402.

[19] Simulia product: ABAQUS, http://www.simulia.com/products/abaqus_fea.html, accessed May 15, 2010.

[20] Mardikian, P., M. Drews, N. Gonzalez, P. deVivies, "H.L. Hunley conservation plan", ed. J. Hunter III., Submitted to the US Naval Historical Center for approval after peer review, Charleston, 2006.

[21] Oceaneering International, Inc., H.L. Hunley General Recovery Procedures, Internal document, Clemson Conservation Center, North Charleston, SC, January 4, 2000.

[22] Vincent Y. Blouin, Paul Mardikian, Christopher Watters (2010). Finite element analysis of the H.L Hunley submarine: A turning point in the project's history. Metal Conference

[23] ABAQUS. (n.d.). Contact formulations in Abaqus/Standard. Retrieved 04 14, 2011, from Abaqus analysis user manual: http://129.25.22.58:2180/v6.8/books/usb/default.htm

[24] ABAQUS. (n.d.). Smoothing contact surfaces in Abaqus/Standard. Retrieved 04 14, 2011, from Abaqus User manual: http://129.25.22.58:2180/v6.8/books/usb/default.htm?startat=pt09ch30s02aus121.html#usb-cni-asmoothsurfaces

[25] Morton, A. G. MARINE CORROSION. Annapolis, Maryland 2140: David Taylor Naval Ship Research and Development.