ANALYSIS OF AN ICE CONE IMPACT WITH A HULL STRUCTURE THROUGH THE USE OF LARGE PENDULUM IMPACTS AND FINITE ELEMENT ANALYSES

by

© Spencer J. Holloway, B.Eng.

A thesis submitted to the School of Graduate Studies

in partial fulfillment of the requirements for the degree of

Master of Engineering

Faculty of Engineering and Applied Science

Memorial University of Newfoundland

October 21st, 2021

St. John's Newfoundland and Labrador Canada

Abstract

When considering hull structure impacts with large ice pieces, there is still much left to understand. As well, there is a lack of information available when considering the development of finite element models of these impact types. This thesis analyzes a grillage from the ex-HMCS IROQUOIS Royal Canadian Navy destroyer, which had a service life of nearly 45 years. Through a large double-pendulum apparatus, the grillage was impacted by an ice cone indenter. The study then continues by replicating the grillage as accurately as possible through the use of newly procured steel. This copy grillage was then also impacted by a large ice cone indenter. Both experiments were then replicated through finite element analysis. The study focuses on the capabilities of hull structures through comparing aged and new steel, as well as comparing the experimental data to respective finite element analyses. The capabilities of the original and copy grillages were shown and the overall study was developed as a benchmarking tool when considering the development of finite element models with respect to real-world ice impact experiments.

Acknowledgements

A person's success is not their own, it is also the success of those around them. The work I put into this thesis was only a fraction of what those around did for me and, through the support of multiple people and organizations, I was able to achieve the research that has been presented.

I would like to start by thanking my supervisor, Dr. Bruce Quinton. Not only for this incredible opportunity, but also for all the guidance and advice he has provided. I cannot thank him enough for his patience throughout this unprecedented time, allowing me to complete my research without worry.

I would like to thank Dr. Ahmed Elruby and Mr. Evan Hipditch, for all the help throughout my research, aiding with the large pendulum experiments, and the wonderful conversations.

I would like to thank Mr. Sthefano Andrade, not only for being incredibly helpful and friendly during all the experiments and conversations, but also for going out of his way for me when I was going through the data acquisition stage of my research.

I would like to thank the project partners at the Defence Research and Development Canada (DRDC), the American Bureau of Shipping (ABS), and Vard Marine Inc., for the chance to be involved in such an incredible project and the consistent guidance.

I would like to thank my fellow peer Mr. Jordan Norman, for going through the program with a constant positivity, which helped me stay optimistic in my work.

I would like to thank Mr. Matt Curtis, for his never-ending knowledge and eagerness to help. I know the experiments would be nowhere near as smooth nor completed with such a high amount of attention to detail without him.

I would like to thank Memorial University of Newfoundland's Technical Services team, for the incredibly skilled work they completed throughout this research.

Lastly, and most importantly, I would like to thank my parents, Carl and Barbara Holloway. Without them I would not be where I am today. Through their constant support and undying belief in me, they have paved the way for me to achieve anything I have ever wanted. I can never thank them enough for how fortunate I am to have them.

Table of Contents

Abstract ii

Acknowledgements iii

Table of Contents iv

List of Tables ix

Chapter 1 Introduction 1

1.1 Research Goal and Purpose 3

1.2 Background 4

1.3 Covid-19 Impact 6

Chapter 2 Literature Review 7

2.1 Introduction 7

2.2 Impact Load Testing 7

2.3 Mechanics of Ship Collisions 9

2.4 Finite Element Analysis 10

2.4.1 Step 1: Pre-Processing 12

2.4.2 Step 2: Solving the Model 15

2.4.3 Step 3: Post-Processing 15

2.5 Previous Works 16

Chapter 3 Large Pendulum Apparatus Experiments 19
3.1 Large Pendulum Apparatus....................19
3.1.1 LPA Grillage Holding Carriage....................20
3.1.2 LPA Impacting Carriage....................23
3.1.3 LPA Capabilities....................25
3.2 Instrumentation....................26
3.3 Development of the Ex-HMCS IROQUOIS Grillage....................28
3.4 Development of the Copy Grillage....................31
3.4.1 Copy Panel Plating....................32
3.4.2 Copy Panel Stiffeners....................32
3.4.3 Copy Panel Welding Specification....................34
Chapter 4 Numerical Model of Experiments....................36
4.1 Problem Type....................36
4.2 Model Description....................36
4.2.1 Simplifications of the Model....................38
4.2.2 Model Dimensions and Design....................40
4.2.3 Ice Cone Model Design....................43
4.3 Material Models....................44
4.3.1 Plastic Kinematic Material Model....................45

4.3.2 Piecewise Linear Plasticity Material Model 46
4.3.3 Rigid Material Model...49
4.3.4 Crushable Foam Material Model 50
4.4 Boundary Conditions 51
4.5 Contact 53
4.6 Solution Controls 55
4.7 Finite Element Model Simplifications 56
4.8 Mesh...57
4.8.1 Meshing the Model 57
4.8.2 Mesh Convergence Analysis..58
4.8.3 Mesh Quality Check 61
Chapter 5 Results and Discussions 64
5.1 Real-World Experiment Results 64
5.1.1 Original Ex-HMCS IROQUOIS Grillage 66
5.1.1.1 Maximum Displacement Results 67
5.1.1.2 Maximum Force Results 70
5.1.2 Copy Ex-HMCS IROQUOIS Grillage..71
5.1.2.1 Maximum Displacement Results 71
5.1.2.2 Maximum Force Results 74

5.2 Finite Element Analysis Results 75

5.2.1 Original Ex-HMCS IROQUOIS Grillage FEA 77

5.2.1.1 Maximum Displacement Results 77

5.2.1.2 Maximum Force Results 79

5.2.1.3 Von Mises Stress Results 79

5.2.2 Copy Ex-HMCS IROQUOIS Grillage FEA 82

5.2.2.1 Maximum Displacement Results 82

5.2.2.2 Maximum Force Results 83

5.2.2.3 Von Mises Stress Results 84

5.3 Data Comparisons 87

5.3.1 Original Grillage Experimental Data vs FEA 87

5.3.2 Copy Grillage Experimental Data vs FEA......................................87

5.3.3 Original Grillage vs Copy Grillage 88

5.4 Discussion 89

5.5 Sources of Error 89

Chapter 6 Conclusions and Future Recommendations 91

6.1 Notable Insights 92

6.2 Future Work 92

Bibliography 94

Appendices..97

List of Tables

Table 1 – Copy Grillage Framing Plastic Kinematic Material Model Data 46

Table 2 - Original Grillage Plating Piecewise Linear Plasticity Material Model - Elastic Data 46

Table 3 - Original Grillage Plating Piecewise Linear Plasticity Material Model - Plastic Data 47

Table 4 - Original Grillage Framing Piecewise Linear Plasticity Material Model - Elastic Data 47

Table 5 - Original Grillage Framing Piecewise Linear Plasticity Material Model - Plastic Data 48

Table 6 - Copy Grillage Plating Piecewise Linear Plasticity Material Model - Elastic Data 48

Table 7 - Copy Grillage Plating Piecewise Linear Plasticity Material Model - Plastic Data 48

Table 8 - Ice Cone Indenter Steel Backing Plate Rigid Material Model 50

Table 9 - Ice Cone Indenter Crushable Foam Material Model 50

Table 10 - Original Grillage Impact Speeds 54

Table 11 - Copy Panel Impact Speeds 55

Table 12 - Original Grillage FEM Check 62

Table 13 - Copy Grillage FEM Check 63

Table 14 - Original Ex-HMCS IROQUOIS Panel Displacement Measurement Array.....67

Table 15 - Original Ex-HMCS IROQUOIS Panel Total Displacements...........................68

Table 16 – Copy Ex-HMCS IROQUOIS Panel Displacement Measurement Array.........71

Table 17 - Copy Ex-HMCS IROQUOIS Panel Total Displacements 71

Table 18 - Original Ex-HMCS IROQUOIS Grillage Data Comparison 87

Table 19 - Copy Ex-HMCS IROQUOIS Grillage Data Comparison 88

Chapter 1 Introduction

The North Atlantic Ocean has always contained icebergs of varying sizes that are an issue for ships travelling in those waters. Even with today's ship design standards, there is still uncertainty about how ice reacts during an impact with a vessel's hull. Therefore, any information that could provide a better understanding of what to expect from an ice impact is beneficial to the shipbuilding industry. However, this knowledge is not easily attainable, as recreating these impacts through an experiment can be difficult, due to the scale and accuracy needed to be met of those methods. Therefore, experiments of ice impacts that are comparable to their real-world counterparts are unique and difficult to undertake.

Currently, a significant amount of the effort in regards to determining the effects of an impact is done through risk assessments. This, although a very reasonable method of determining the overall safety of a ship, does not provide data as precise as experiments recreating impact scenarios. Similarly, vessel impacts can be predicted through analytical, empirical, or numerical methods, however, these are usually based on assuming general impact geometries and forces, or taking best guesses from similar vessels that have already dealt with impacts at a similar scale. However, due to technology changing, it is difficult to predict how each vessel will compare, even with these other methods, due to every ship being designed with newer materials, different structural design types, and by different groups and companies. Therefore, if one had access to pieces of a vessel's hull that had operated for a significant time, and those pieces were in relatively good condition, this would create an uncommon situation where the grillages could be analyzed versus new copies of them. By creating as similar copies as possible, this would offer a link between the new and older grillages, ideally providing some insight as to how long ship hulls can last based on their material and design properties, instead of guessing at general corrosion and fatigue possibilities or using empirical data.

This research attempts to achieve this uniqueness. Through having a decommissioned naval destroyer hull grillage, a complete copy of said grillage, and a full large double-pendulum apparatus, experiments were completed that involved high load impacts on both grillage types. This large pendulum apparatus was initially developed to compare ship-to-ice and ship-to-ship interactions (Gagnon, et. al 2015; Alam, et. al 2012; Gagnon, et. al 2020), making it ideal for this type of experiment. Through creating large ice cones and impacting them against a piece of a vessel's hull, ice impacts have been recreated that can be used to better understand and predict real-world impacts.

The intention of this research is to complete a large pendulum impact on the ex-HMCS IROQUOIS grillage, as well as completing an impact on its respective copy grillage; made with new steel and stiffeners. The research then recreated these impacts through finite element analysis and compared each set of data to one another.

As useful as this experiment data is, it is limited by the scale of the piece of the hull being impacted. Therefore, to further explore this information, a numerical investigation has been completed to provide more information of this type of ice impact. From this, the numerical data can be expanded upon and utilized in an impact scenario involving a full ship model.

This thesis is divided into six main chapters. Starting with the introductory chapter; this will describe the problem at hand, as well as any background information involving the overall project. The second chapter will describe the literature review, outlining any important previous works or information leading up to this research. The third chapter will describe the process involving the real-world experiments, which were completed at Memorial University of Newfoundland's Structures Laboratory. The fourth chapter outlines the steps involved in the numerical modelling process. The fifth chapter will analyze the results from both experiment types; the real-world experiments and the numerical modelling, including a comparison of the data. The final chapter of this thesis will present any conclusions found during this research, as well as future recommendations.

1.1 Research Goal and Purpose

Ship collisions are essentially dynamic problems (Paik and Pedersen 1996). Ship collisions are a concern for vessels, especially considering that they are capable of causing both local and global deformation or failure. One of the first attempts at understanding ship collision responses involved an empirical formula, which represented a linear relationship between the kinetic energy involved in the collision and the volume of the perceived damage found on the vessel (Minorsky 1959). Even though this work has been improved upon many times over the years, it is still dependent on the available ship collision data (Paik and Pedersen 1996).

As well, there is still significant information unknown about ice impacts, which includes benchmarking against real-world experiments. This thesis aims to help improve this lack of information, through the use of benchmarking numerical models against experimental impacts. By comparing the large pendulum apparatus experiment data to a similar numerical model, this will provide a benchmarked numerical analysis that could be scaled up to a full ship model in future works. Therefore, one could use the information found from this research as a benchmarking mechanism when analyzing ice impacts numerically during a ship's design process. This research provides a reasonable stepping stone from comparing ice impacts during vessel operation to numerical models of similar events. It will also help provide more information on the differences between aged and new steel structures.

1.2 Background

Memorial University of Newfoundland was sent six pieces of a Canadian ex-naval destroyer that had been decommissioned in 2015. As the Royal Canadian Navy had recently begun developing new vessels to replenish their fleet, this involved decommissioning certain long-running vessels, including the ex-HMCS IROQUOIS naval destroyer, which can be seen in the following Figure.

Figure 1 – HMCS IROQUOIS (Dirtsc, 2013)

After roughly 45 years of operation, this vessel was taken out of service and six pieces of its hull were cut out and sent to the University for structural analyses. Originally, each piece was roughly 7 feet by 7 feet in area and one of these panels can be seen in the following Figure.

Figure 2 – One of the Six HMCS IROQUOIS Grillages, Intact

1.3 Covid-19 Impact

Originally, all six pieces of the ex-HMCS IROQUOIS were intended to be reported on in this thesis, however, significant delays occurred due to the COVID-19 pandemic. Prior to March 2020, slight delays had already occurred to the project. Due to the pendulum apparatus needing certain upgrades prior to the high impacts that would be involved in the ex-HMCS IROQUOIS experiments, none of the six original or copy panels had been impacted prior to the beginning of the COVID-19 pandemic. Once the pandemic started, this caused delays to the real-world experiments, which in turn delayed the numerical analyses. Therefore, the first impacts did not occur until December of 2020. Due to this delay, the research conducted in this thesis had to be cut down and now only includes data from impacts on a single ex-HMCS IROQUOIS grillage and its respective copy, as well as numerical analyses of these two experiments.

Chapter 2 Literature Review

2.1 Introduction

This literature review will provide a description of the various relevant topics that are available and related to the completed work.

2.2 Impact Load Testing

An uncommon method of analyzing hull structure includes doing large impact testing on a portion of the side hull of a vessel. These methods are considered uncommon due to the scale and financial backing needed to provide comparable data to real-world impacts. As dynamic strength behaviour of ice is often different from static or quasi-static behaviour, and most research has been put into determining the strength properties of ice, global ice loads and pressure distributions during ice contacts, dynamic strength behaviour analysis has had very few studies carried out on it (Alam et. al. 2012). A possible method of completing a dynamic strength behaviour analysis is through recreating the side hull of a current vessel or cutting out a portion of a decommissioned vessel and instigating a large impact on it. The impact load testing completed throughout this project will be done on both hull types; a cut-out piece of a decommissioned vessel and a respective copy developed through new steel. This will ideally provide a link between the new and aged steel and how structural properties can change over time. Few experiments have been completed through a large pendulum swing, with most of these types of experiments being completed at the Memorial University of Newfoundland. A large double-pendulum swing was initially developed at Memorial University in 2012 by Alam, et. al. This design was originally chosen as it was capable of completing direct collisions between ice and ships, it allowed for high impact energies, it could continue these impacts in a controlled and repeatable way, and it was a self-reacting system, which did not transfer loads to its surroundings (Alam, et. al. 2012). This double-pendulum apparatus was found to meet all the requirements necessary for the research, with certain design changes made to meet the impact requirements. It has also been mentioned that the double pendulum was a benefit because it could provide reasonably high impact loads while also minimizing the space requirements needed at the university (Gagnon, et. al. 2015). Other experiments to study dynamic impacts have been completed through different methods, such as with a stationary grillage and some form of press recreating an ice impact, or doing drop tests on an impact module (Sopper, et. al. 2015), however, the double-pendulum swing provides data that is a representation of a similar dynamic impact that could occur during a vessel's regular operation, except at a smaller energy level. The overall dimensions of the large pendulum apparatus include a width of 4 metres, a height of 5 metres, and a length of 8 metres. As well, both of the carriages have 2 metre long swing arms. A full description of the large double-pendulum apparatus can be conveniently found in the paper written by Andrade et al. (2021).

This double pendulum apparatus has been used for multiple experiments by Memorial University, including, but not limited to Alam et al. (2012), Gagnon et al. (2015), Sopper et al. (2015), Gagnon et al. (2020), and Robbins, M. L. (2020).

This research will require fitting this large pendulum apparatus with an ice cone to replicate ice impacts. This equipment has been used in the past with the ice cone impacting an instrumented panel (Sopper, et. al. 2015; Gagnon, et. al 2020, Robbins, M. L. 2020). The purpose of the Sopper, et. al (2015) research was to better understand variations in ice impact pressures, where an impact module was used to record these variations. Instead of an impact module, various instrumentation, which will be described later, was placed along both swinging carriages and the large pendulum apparatus to record data in real-time as it was being impacted by the ice cone.

2.3 Mechanics of Ship Collisions

A ship collision is a complicated process, involving multi-physical and highly coupled processes (Liu and Amdahl 2010). However, these types of collisions can be broken down into two separate processes, known as external and internal mechanics. Primarily, external processes will include a vessel's rigid body motions and allow for the understanding of a vessel's energy to be devolved into strain energy (Liu and Amdahl 2010). The internal mechanics, however, covers how the strain energy developed through the two objects as the impact occurred, including the evaluation of the structural resistance during the impact (Liu and Amdahl 2010). Along with this, the location of an ice impact, as well as the geometry of the impact location, will have a significant effect on the outcome. This includes the fact that the impact may produce significant motion in the vessel, making the six degrees of freedom important when considering both the object striking and the object being struck. As well, the shape of the location may have a direct effect in developing the contact force direction and energy involved (Liu and Amdahl 2010).

As stated previously, initially ship collision mechanics were determined and improved upon through the use of empirical relationships; comparing an impact's kinetic energy to its respective damage volume (Paik and Pedersen 1996). However, it was found that these formulae were not consistent, nor relevant, to certain hull types, including double-hulled vessels. Therefore, models had to be developed to incorporate the coupling effects between the local and global failure modes (Paik and Pedersen 1996). To properly analyze ship collisions, one must take significant considerations into account. These considerations include, but are not limited to (Paik and Pedersen 1996):

- Yielding due to membrane tension of the hull plating.
- Ductile failure of strength members in tension.
- Crushing of the compressed plate intersections between local and global failure modes.
- The effect of stiffeners on stiffness and strength.
- The influence of material strain-rate sensitivity.
- Rupture.

These are not only a consideration when analyzing the real-world experiment data, but are highly influential when developing a realistic numerical model.

2.4 Finite Element Analysis

Finite element analysis (FEA) is a numerical modelling method that is well established in the ship structures industry, acting as a validation tool when comparing structural models. As well, in recent years, there has been an increase in the amount of finite element modelling (FEM) generated to replicate ship collisions.

Finite element analysis is a numerical tool, used as a method for approximating the solution to a complex problem (Quinton 2019a). It is a method of simplifying a problem and its relative domain into smaller pieces, known as elements, which can then be manipulated to define the problem at hand. Commonly, elements are used to define the geometry of various objects involved. Through mathematical formulae, an element is capable of approximating its respective response, based on the various inputs describing the environment and how it is being manipulated. By defining a geometry, and a respective loading pattern through this method, it creates a way of predicting the response of a structure. Commonly, this method is implemented when a problem must be solved, but it is too complicated to be done with current analytical or empirical methods (Quinton 2019a).

Finite element programs are capable of analyzing static, quasi-static and dynamic problems. However, due to the complexity of quasi-static and dynamic problems, finite element analysis is usually applied to these problems. This leads to two types of finite element analyses; explicit and implicit FEA. Implicit FEA is optimal for static, quasi-static, or low-to-medium frequency steady-state dynamic problems, while explicit FEA is more commonly used for analyses that involve a transient and dynamic or medium-to-high frequency problem (Quinton 2019b). Each FEA type has its own benefits and drawbacks. Commonly, implicit FEA is used in stable, time-independent analyses, while explicit FEA is used in dynamic, time-dependent analyses.

As the research in this thesis involves a large load impacting a grillage at a high velocity and the possibility of elastic and plastic strain occurring at high speeds, this is considered a transient and dynamic problem. For the purposes of this finite element analysis, LS-

DYNA was used, as it was determined to meet all the goals sought out for this finite element analysis.

The finite element method consists of three processes; the pre-processing portion, the solution portion, and the post-processing analysis.

2.4.1 Step 1: Pre-Processing

The pre-process involves the development of the finite element model. This includes setting up the computer-aided drafted models, inputting the various parameters and material models, applying the appropriate boundary and loading conditions, and specifying the type of results that are desired for the problem type (Quinton 2019a).

For proper analysis in a finite element program, many assumptions must be made regarding the real-life problem that is being analyzed. These assumptions, which may include assumptions involving boundary conditions, force impacts, or even model geometries, will determine the accuracy of the results of the structural response. Therefore, the process of how the analysis has been developed is equally as important as the results themselves.

Often, the main contributor to how a finite element analysis is completed is through the use of a predetermined failure point. The failure point is decided upon during the pre-processing stage, whether that be fracture or just meeting a relevant yield threshold. However, the FEA used in this thesis will be used as a benchmarking comparison of the laboratory experimental results. Therefore, no specific failure point will need to be met throughout the duration of the finite element model.

Initially, the geometry was developed using the computer-aided drafting software Rhinoceros 5. Once the three-dimensional (3-D) model had been developed, the meshing process had to be completed. This included turning the 3-D Rhinoceros 5 model into a series of smaller elements, allowing for it to be properly analyzed in finite element software. This meshing process was done through the software HyperMesh. HyperMesh is capable of importing a Rhinoceros 5 IGES file type, meshing it, and then exporting the mesh as a Keyword file, which is a file type that is compatible with LS-DYNA analyses.

Once a file is imported into LS-DYNA, there are many options when considering the type of element formulation that will be involved with the simulation. The LS-DYNA default element option is Belythschko-Tsay shell elements, which include stiffness and warping considerations. Through previous research, it has been found that this element type is acceptable when considering a finite element analysis of ship impacts (Quinton 2015). Once the model meshes have been finalized, a series of checks must be completed to ensure element quality. LS-DYNA is not capable of properly analyzing elements of a certain shape and, if not properly fixed, certain irregular elements may cause shear locking, volumetric locking, and hourglassing. These can be analyzed through the use of aspect ratio checks, Jacobian checks, and ensuring each element has appropriate amounts of warpage and skew.

As material data was available for most hull specimens, the material data cards were completed through a multi-linear elastic-plastic material model. For specimens with material data that did not provide enough info for a multi-linear elastic-plastic material model, a bi-linear elastic-plastic model was chosen, as it has been found to still be capable of representing nonlinear behaviour in ship hulls made of steel (Quinton, et al. 2016).

The numerical ice cone indenter mesh was originally developed by Dr. Bruce Quinton (used here with permission). The material model for the ice is taken from Gagnon, R. E. et al (2006). The same ice cone was implemented during each finite element analysis completed throughout this research.

Boundary conditions (BCs) are used to define the field of interest in a given problem. These conditions set the degrees of freedom available to certain parts of the geometry of a finite element model. As the finite element analysis completed through this research is developed based on a real-world experiment, boundary conditions in the FEM should be as similar as possible to their real-world counterparts. During the development of the large pendulum apparatus, the real-world experiment boundary conditions were developed with the foreknowledge that numerical analyses of these experiments would eventually occur. Therefore, the finite element models were able to be developed with similar boundary conditions.

There are multiple methods to apply a load in LS-DYNA, including, but not limited to; adding pressure to a specific portion of a geometry, adding point loads to single or multiple elements, or by having two geometries impacting one another in the finite element model. As the intention of this finite element analysis is to stay as accurate as possible to the respective real-world experiments, loading conditions were completed through the contact card option. Therefore, the only data necessary in the FEA included the weight of both carriages and their respective speeds found at the point of impact.

Solution controls are used in a finite element model to describe how the model will be checked and how the data will be output. Commonly, a timestep is involved, which ensures data is output each time this time step is met. Therefore, if the timestep was 0.1 seconds, then the model will be checked and data will be output every 0.1 seconds that the simulation runs. Therefore, this information will be available at roughly the peak of the impact, as well as including multiple points as the impact is increasing and decreasing to its respective apex. The research involved in this thesis includes information as the contact is occurring, once the contact has reached its maximum elastic and plastic deformation, and continued to provide information as the two models were moving away from one another post-impact.

2.4.2 Step 2: Solving the Model

Once the finite element model had been developed, the file was run through a relevant finite element software; in this case being LS-DYNA. Even though nothing can be done during this phase, things can be implemented to help improve and speed up this process. This includes, but is not limited to, completing a mesh convergence analysis, determining an appropriate time-step, not having an overcomplicated load, or having the pre-process section developed correctly (Quinton 2019a). These methods will be better described later in Chapter 4. Each method has a way of increasing the time it takes for a finite element model to solve and ensuring that these are as optimal and accurate as possible can help mitigate the duration of a finite element analysis.

2.4.3 Step 3: Post-Processing

Once the model has been solved, various results and data will be available as requested during the pre-processing stage. With this data, the model results can be analyzed to find useful information about a structures response and benchmarked to determine the data’s accuracy. As well, the information can be used to ensure your model is calibrated correctly, using it directly in a mesh convergence analysis (Quinton 2019a).

Once the overall simulation is completed, this can be used as a direct comparison tool to real-world experiments that are similarly designed, as is the intention of the finite element analysis completed in this research.

FEA has been used to benchmark many types of ship structure impacts, including the research completed at the Memorial University of Newfoundland. A separate notable study on collision simulations was completed by the MARSTRUCT Virtual University (Ringsberg, et al. 2018). This study involved fifteen different research groups from various locations around the world. This research found a generally good agreement between the finite element analysis data as compared to the experimental data (Ringsberg et al. 2018), providing an optimistic view when considering the use of FEA.

2.5 Previous Works

There are many different works discussed throughout this literature review that provide confidence in this experiment. Most notably, this includes all experiments completed with Memorial University of Newfoundland's double pendulum apparatus, as well as the MARSTRUCT Virtual University study(Ringsberg, et al. 2018). However, one study was found to have noticeable similarities with the research completed throughout this thesis. That research was completed by Kim, H-M, Daley, C.G., & Kim, H. (2018).

Similar to this thesis, the experiment completed by Kim, H-M, Daley, C.G., & Kim, H. (2018) involved the comparison of structural grillages being impacted by an ice load through both experimental and numerical methods. As well, this research involved quasi-static loads well past the elastic limit of the material (Kim, H-M, Daley, C.G., & Kim, H. 2018). This research showed many comparisons to the research completed throughout this thesis, however, it had two main differences. This included the grillage type, which was that of an Ice Class PC6 vessel, and a quasi-static loading rate (Kim, H-M, Daley, C.G., & Kim, H. 2018). Although this research involved a non-polar class ice vessel and a dynamic loading state, it still has many similarities to the research completed by Kim, H-M, Daley, C.G., & Kim, H. 2018.

Throughout Kim, Daley, and Kim's (2018) research, two grillage experiments were completed, which then led to a numerical analysis of both respective impacts. Even though many simplifications were made throughout the finite element model, once completed, the finite element analysis and experimental results showed an excellent agreement with one another (Kim, H-M, Daley, C.G., & Kim, H. 2018). Agreement was also found in the cross-sectional views of both the experiment and numerical analyses. The experiments also found that local deformations did not necessarily impair the large grillage's overall strength, it increased its stiffness and load-carrying capacity (Kim, H-M, Daley, C.G., & Kim, H. 2018).

Although there are some differences between the work done by Kim, H-M, Daley, C.G., & Kim, H. 2018 and the research completed throughout this thesis, this work provides confidence that the analysis of a dynamic ice impact on a grillage and its respective numerical analysis can be completed. Ideally, this type of research can aid in the design of more efficient structures and support a higher level of safety for vessels that have to consider ice impacts (Kim, H-M, Daley, C.G., & Kim, H. 2018).

Chapter 3 Large Pendulum Apparatus Experiments

To analyze the capabilities of the ex-HMCS IROQUOIS grillage, large impact testing was completed on the original grillage, as well as its respective copy. This included the use of the large double-pendulum apparatus already available through the Memorial University of Newfoundland, along with a new panel carriage. The new carriage is capable of holding the ex-HMCS IROQUOIS grillages and copies as the previous carriage was not designed for this purpose. This carriage was designed by Dr. Bruce Quinton and built by the Memorial University of Newfoundland's in-house Technical Services group.

3.1 Large Pendulum Apparatus

The large pendulum apparatus (LPA) consists of two swinging carriages; an "impact carriage" and a "grillage holding carriage". Both carriages are capable of being swung from an angle of up to fifty degrees, allowing for the possibility of significant impacting forces. Both carriages are aligned so that when they are released they will impact directly at the center of the grillage and respective impactor. The impacting carriage is capable of utilizing either a rigid indenter or large ice cone indenter. For the purposes of this experiment, ice cones were developed and used in the impacting of the grillages, as ice impacts are commonly outside the design requirements of a Royal Canadian Naval Destroyer, but understanding bergy bit impact reactions with warships are becoming a growing need.

Figure 3 - Large Double-Pendulum Apparatus

3.1.1 LPA Grillage Holding Carriage

The grillage holding carriage had been developed in such a way to ensure ideal “clamped” boundary conditions. This included having the entirety of the outer edges of the grillage bound in place while it was in the holder.

Figure 4 - LPA Grillage Holding Carriage

Therefore, a separate piece of steel had to be welded on to ensure that the boundary conditions could be met. These boundary conditions were not only beneficial to ensuring accurate data in the real-world experiments, but helped with the ease of development of the numerical models. Pieces of steel with bolt holes were fabricated and welded along the outer edges of the plating, as well as the ends of the stiffeners. This allowed for the grillage to be bolted in place along its outer perimeter, developing the boundary conditions as intended. Schematics of the fabricated boundary conditions can be found in Appendix A. Below, Figure 5 shows the grillage plating boundary connectors, which were welded onto the outer edges of the grillage plating and Figure 6 shows the stiffener boundary connectors, which were weld to both ends of all three stiffeners.

Figure 5 - Grillage Plating Boundary Connectors

Figure 6 - Grillage Stiffener Boundary Connectors

3.1.2 LPA Impacting Carriage

As stated previously, the impact carriage is capable of using different types of impactors, whether that is an ice impactor or a steel impactor. This research only used an ice impactor, which was held using a V-shaped holder, allowing for a simple method of installing the ice cone. It is important for this to be a quick process, as once the ice has been shaped and taken out of its respective cooler, the experiments must be completed quickly, as to ensure no significant change to the ice due to temperature. This V-shaped holder is shown in Figure 7.

Figure 7 - Ice Cone V-Holder

The ice cone is produced at Memorial University's in-house freezer rooms. Ice is ground up into small pieces and then mixed into a larger cylindrical form with a mix of ice chips and water. This ratio of chips to water in the ice is kept the same throughout, ensuring the ice is as similar as possible throughout its entirety, mitigating the possibility of certain areas mixing differently and having different material characteristics. The ice cylinder is frozen as is, with a diameter of 1 metre. Once the ice has been frozen for roughly two weeks, it is taken to a shaper, which cuts the upper half of the cylinder down into a conical shape. The ice cone is frozen into its respective LPA holder to make the transition from the freezer room to the laboratory as simple as possible.

Figure 8 - Installed Ice Cone

The impact carriage is also capable of being loaded with more weight, through the use of a holder connected to the back-side of the carriage. This holder is commonly filled with multiple rectangular steel plates, roughly two feet long by two feet wide. Multiple plates can be placed in the rectangular holder without affecting the location of the impact point between the two carriages. This holder, with plates installed, is shown in Figure 9 below.

Figure 9 - Ice Indenter Carriage Additional Weight

3.1.3 LPA Capabilities

Due to the possibility of any swing angle up to fifty degrees and the ability to change the weight and type of impactor, this large impact apparatus provides a significant amount of versatility when considering a large grillage impact experiment.

Grillages can be impacted multiple times and at multiple angles, therefore, if rupture does not occur during the first impact, then more impacts may be completed until rupture occurs. The number of impacts will be determined by the required final outcome of the grillage. This can also be replicated in the computer simulations, as the data and shape of a grillage from an initial impact can be exported and re-impacted by the software, or multiple impacts can be included in a single finite element analysis. Even though both the original and copy grillage were impacted multiple times, only one impact will be compared throughout this research, the initial impact. This is due to the fact that after the first impact, the ice cone would have been crushed and nearly impossible to remodel through numerical methods. This means that only one impact included an idealized ice cone, which in turn allows for a direct comparison between the numerical analyses and the real-world experiments.

3.2 Instrumentation

To properly record the data involved in this experiment a multi-sensor array had to be developed, with their respective locations being optimally placed around the two impacting pendulum carriages and the LPA. The locations of the instrumentation prioritized either regions where they would not get damaged or areas that expected high loads that needed to be recorded. Each individual instrumentation placement was dependent on the respective instrumentation type and the requirements that it needed to meet.

The placement of the strain gauges were not in locations of the maximum strains. They were located on the side and top boundary condition structure of the holding panel, as well as a third one being placed on one of the swingarms. Their purpose was to see if they were taking any strain throughout the impacts. Two accelerometers were included in the experiments, with one on the direct centre of the non-impacted side of the panel and the other on the back of the ice carriage. Their purpose was to measure acceleration in the panel at the point of impact and rigid body motions respectively.

Behind the indenter included three 1.2 MN Kistler 9091B piezoelectric load washers, which are used to determine the impact force throughout the experiment. These are located between the indenter carriage and indenter mounting plate. They were placed in a triangular fashion, with one load washer being located directly above the ice cone and the other two being located at the bottom, all equidistant from one another. These load washers are capable of recording loads from transient dynamic impact forces and have the following specifications:

- A rigidity of 65kN/μm.
- A natural frequency greater than 11kHz.
- A small hysteresis of <0.2 %FSO.

Along with this, standard strain gauges were placed in strategic locations along both pendulum carriages and in the panel carriage, with one being placed directly at the expected point of impact. As well, acceleration of the panel in the X-direction was found through the use of a Kistler 5000g accelerometer, located on the stiffener side of the plating of the impacted grillage. This accelerometer was placed directly in the centre of the impact.

Instrumentation outside of the LPA included a series of high-speed cameras, using digital image correlation (DIC). DIC was used to capture the carriage kinematics, providing velocity and acceleration information on both swinging pendulum carriages. On top of this, impacts were recorded with multiple GoPro cameras, located at different angles to provide various views of the same impact. Therefore, this gave a multi-recording apparatus, where high-speed and thermal cameras focused on the impact area, while DSLR cameras captured the entire event.

Along with this, 3-D Laser scans of the panel plating were completed before and after the impacts. This was completed to give an exact recording of the panels pre- and post-impact.

3.3 Development of the Ex-HMCS IROQUOIS Grillage

When the six grillages were originally sent to Memorial University, they were roughly 7 feet wide by 7 feet long. Due to the holding carriage only having enough room for a 80 inch long by 53.54 inch wide grillage, the grillages had to be cut down to fit. Due to the curvature found in the grillage plating, the panel had to be cut in an optimal way so that it would fit into the holder while also meeting the boundary condition requirements. This positioning also included the locations of the stiffeners, which had to be as centred as possible. To do this, three-dimensional (3-D) models were developed in the computer-aided drafting software Rhinoceros 5. These models were then compared to a 3-D version of the grillage holder so that the optimal cut locations could be found.

To develop these models, the panel was measured by hand, using calipers, tape measures, and an array of laser measurement devices. Originally, all six ex-HMCS IROQUOIS grillages were modelled using this method, which led to them being optimally cut for the holding carriage. However, as stated previously, only one of the six grillages ended up being impacted during the time frame of this research.

Due to the age of the grillages, a significant amount of curvature could be found in their respective plating. This curvature was found to be due to three main reasons:

1. The original curvature of the hull.
2. Dents or bumps due to impacts in the hull during the vessel's service life.
3. Dents or bumps due to the decommissioning process of the vessel, such as when cutting out the grillages from the hull.

A laser distancing array was developed in such a way to capture the curvature in the plating within a reasonable degree of accuracy. This array provided measurements that allowed for the development of a 3-Dimensional model of the plating that would provide accurate finite element analysis results that could be comparable to the physical experiments. Seven laser distance measuring devices were used across this array, all separated by roughly a foot from one another.

The array was moved six inches down the panel at a time and measurements were recorded for all seven laser displacement measuring devices. As well, extra points were taken in locations that featured significant changes in the curvature of the plating. This was done for the entirety of the plating until a plot of points was measured. This plot of points was then developed into a point cloud, which was imported to the computer program Rhinoceros 5 and turned into a flat panel.

The stiffeners were measured by hand through the use of measuring tapes, rulers, and calipers. These measurements were then transferred into Rhinoceros 5. As most of the stiffeners did not show much curvature, as compared to the actual plating, only five points were measured across each stiffener, with extra measurement points occurring in the few areas where significant changes in the stiffener geometry were observed. These stiffeners were then translated to the three-dimensional model to represent the actual stiffeners within a reasonable amount of accuracy.

The grillage chosen for this experiment was the one that showed the least amount of curvature in the plating. This ensured the process would be easiest for those cutting the panel and placing it into the grillage holding carriage. As this was a new experience for everyone involved, this was a benefit for both the researchers and Memorial University's Technical Services group. Once the optimal cutting locations had been decided for the grillage, the grillage was cut and the boundary pieces were welded onto its exterior. The grillage was then bolted into the holding carriage frame and instrumentation was placed in the appropriate areas along the grillage and holding carriage.

Figure 10 - Grillage in Holding Carriage

3.4 Development of the Copy Grillage

Once the original grillage had been cut and finalized, its respective copy was developed. The copy panel had to be as similar as possible to the original, however, simplifications had to be made. This included procuring steel of slightly different material properties, stiffeners that were made through a different method and of a different size, and welding practices that were much more advanced than when the original ex-HMCS IROQUOIS vessel had been built.

Design schematics of how the copy panel was built can be seen in Appendix A.

3.4.1 Copy Panel Plating

To get an identical copy of the original panels plating, there are obvious indentations and curvature that would be nearly impossible to replicate. This is due to the fact that it is unknown how much impact was involved in creating each amount of curvature, as well as what type of impacts caused it. Due to this uncertainty, it was decided that the copy would still provide an adequate comparison between aged and new steel if the plating is kept unbent and flat, as none of the curvature in the original plating was significant and to replicate it would cause internal stresses which may be inaccurate to the original. Therefore, priority was set on procuring steel and stiffeners that would have their material properties and dimensions as similar as possible to the original grillages.

With knowledge of the original grillage's material properties, through the use of tensile tests (which are explained below), the steel that represented the copy panel plating was CSA G40.21 44W/ASTM A36 steel. This was found to have as similar characteristics as could be found to that of the original ex-HMCS IROQUOIS grillages. The stiffeners were chosen as W10x17 beams, with a specification of ASTM A6-17. These were then cut along their respective web at a point that met the height requirements necessary of the copy panel stiffeners.

3.4.2 Copy Panel Stiffeners

A method of designing stiffeners is to have them be pre-built, with the web and flange already connected to one another. This minimizes the amount of welding needed as the ship is being built and can help with the ease of construction. This was the case with the ex-

HMCS IROQUOIS. As well, the stiffeners used in the ex-HMCS IROQUOIS were made with the flanges getting thinner the further they reached from the web. Due to the necessity of the copy panel needing to be as accurate as possible to the original, and these types of stiffeners no longer being available, a creative solution was found. Instead of finding T-stiffeners of this shape, W10x17 beams were found that had similar dimensional and material properties. This W-beam was found to have been designed in a similar way to the original panels T-stiffeners. Therefore, the W-beams were cut at a point along its web so that they would be the same height as the original grillage stiffeners. These stiffeners are not identical to the originals, however, this simplification had to be made as it was the best possible option considering how shipbuilding methods have changed over the past 50 years. A second simplification that was made in regards to the stiffeners was the difference in the original panel’s web heights and flange widths. Each stiffener is within reasonable sizing of each other, but from a lifetime of operation and how they were welded, they do have some amount of individuality. Due to this, all three copy stiffeners were cut at 7 inches high, as the difference of one or two millimetres was deemed unlikely to have a significant effect on the data. The following Figure shows the original panel stiffener versus the copy panel stiffener.

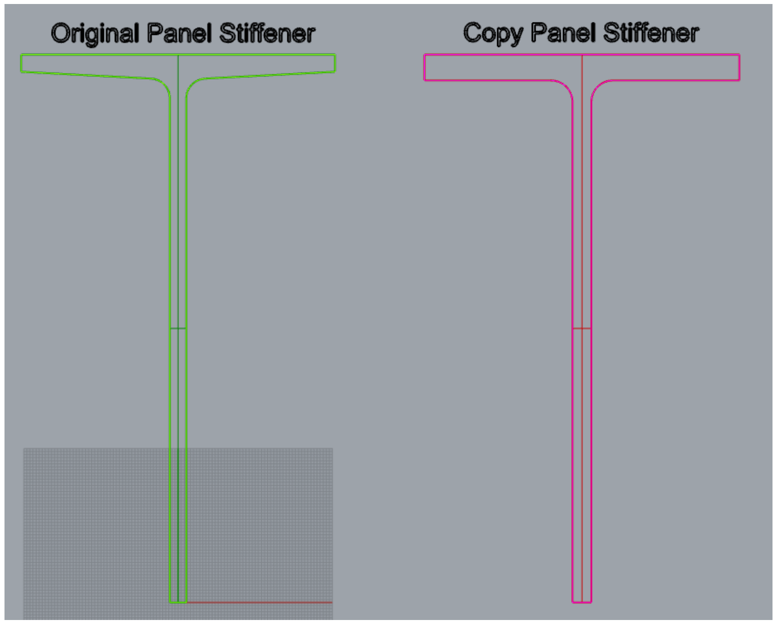


Figure 11 - Stiffener Cross-Sections

As can be seen, the original and copy panel stiffeners are not identical to each other. However, the copy panel stiffeners meet the requirement of being pre-built and having its web thickness increase as it meets with the flange. Both stiffener cross-sections have their own respective amount of individuality.

3.4.3 Copy Panel Welding Specification

When determining how the stiffeners would be welded onto the copy panel plating, significant consideration was given to making the welding practices as similar as possible to the original ex-HMCS IROQUOIS. To do this, the 1963 version of the Royal Canadian Navy Specification for Standard Methods of Welding was analyzed. Various welding techniques were considered throughout the analysis, as well, the Royal Canadian Navy (RCN) document provided the specifics for many types. Due to the age of the ex-HMCS IROQUOIS, there were no documents available describing the process as to which the welding had been completed on the vessel itself and, after a discussion with the University's technical services, efforts were prioritized on making the weld as similar as possible to its respective original and a method was determined which was thought to be representative of the processes originally used.

It was determined that shielded metal arc welding (SMAW) would be appropriate, as it was referenced in the RCN Specification for Standard Methods of Welding and the University's technical services department was capable of this type of welding. As well, emphasis was put on including any similarities that could be seen. This included:

- Any deviations in the weld.
- Any discontinuities in the weld.
- Any small holes located along the connection between the stiffener web and the plating.

Through technical services expertise, the welds on the copy grillage were able to be completed similar to that of those on the respective original ex-HMCS IROQUOIS grillage.

Chapter 4 Numerical Model of Experiments

4.1 Problem Type

In assessing the problem, several key areas were found that determined how to approach this analysis using the finite element method. Fundamentally, the type of collision being modelled between the ex-HMCS IROQUOIS grillage, the respective copy panel, and the ice-cone impactor is of relatively short duration. Therefore, this could be considered a time-domain problem, with the entirety of the impact occurring in under one second.

Along with this, several possible nonlinearities were identified in the problem. Specifically, there were significant geometric nonlinearities in the grillages as they deformed. Both elastic and plastic deformation should be expected. Another nonlinearity considered was the material model, however, this will be discussed in the material section of the analysis.

Due to the expected nonlinearities and the impact type, the analysis was treated as a transient dynamic problem. Therefore, a nonlinear explicit FEA solver was selected. LS-Dyna was chosen and utilized throughout the FEA simulations.

4.2 Model Description

As described previously, the ex-HMCS IROQUOIS hull grillage and its respective copy panel were impacted using a double-pendulum apparatus. This includes two sides, one holding the grillage and the other holding an ice-cone indenter. The grillages were cut to 80 inches long by 53.54 inches wide and positioned in the large-pendulum apparatus to be hit. Each grillage consists of plating and three longitudinal stiffeners. The panels were originally cut so that the longitudinal stiffeners would match three respective holders in the large-pendulum carriage. Therefore, this keeps a clamped boundary condition along the panels exterior free edges as it is sitting in the apparatus, allowing for zero degrees of freedom at the panel’s edges once installed in the carriage.

As described previously, the 3-D model of the original ex-HMCS IROQUOIS grillage was initially developed using the hand measurements and a laser-displacement array. Seven laser-displacement measuring devices were held above the grillage and data measurements were acquired at multiple points along the grillages plating, providing an array for different distance measurements. This array was then developed into a model in Rhinoceros 5, where the various plastic curvature in the plating, that had occurred before Memorial University procured the grillage, was captured within a reasonable degree of accuracy. The following shows this point cloud and the respective plating it was developed into.

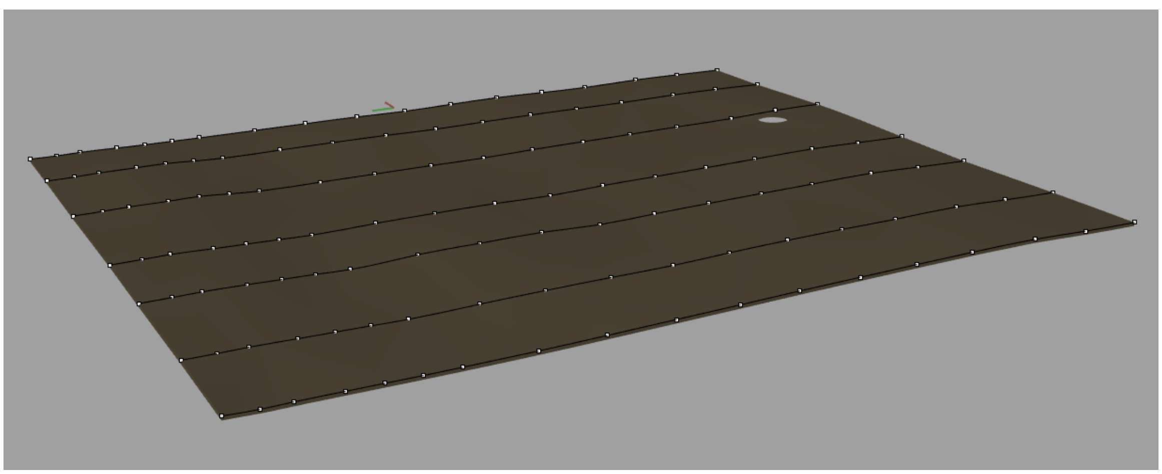


Figure 12 – Original Panel Point Cloud and Plating

As can be seen in Figure 12, there was no significant curvature in this grillages plating.

Measurements of the longitudinal stiffeners and transverse frames were then completed by hand, using a measuring tape and calipers. Once this had been completed, an average of these measurements was taken, as there were no significant differences between the values taken across the three stiffeners. Less detail was recorded for the two transverse frames, as it had already been known that the real-world panel would have these cut off prior to being placed in the LPA carriage for experimentation. Each stiffener was then developed through these height and width averages to provide the following full ex-HMCS IROQUOIS Grillage 3-D model.

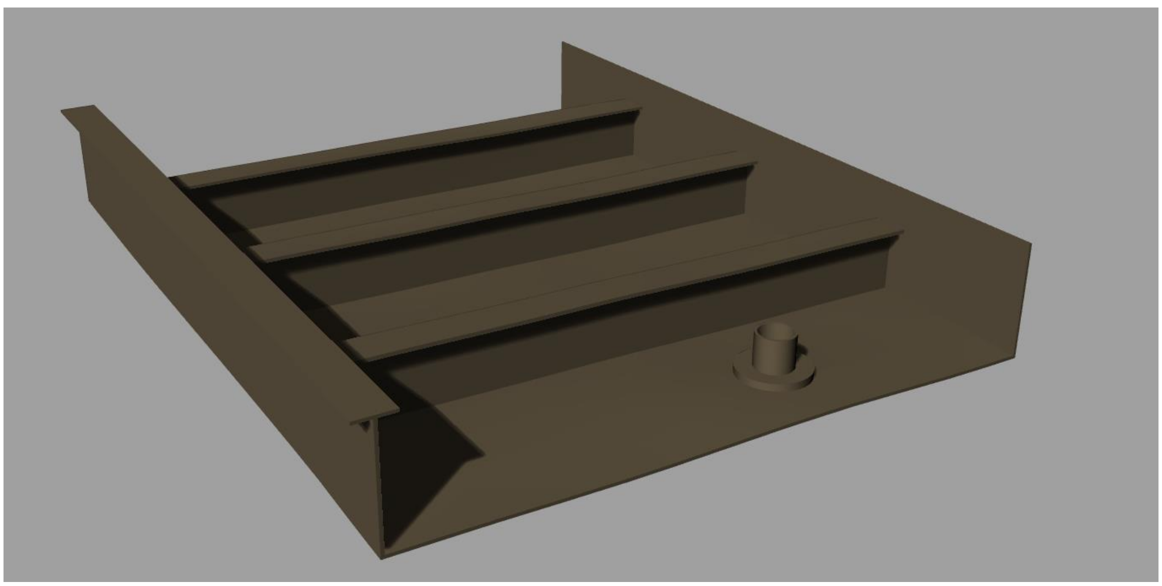


Figure 13 - Full HMCS IRQUOIS Original Grillage

4.2.1 Simplifications of the Model

Through developing the final 3-D model of the ex-HMCS IROQUOIS grillage to be used in the FEA simulations, certain simplifications were made. Over time, the ex-HMCS IROQUOIS had taken enough low-to-medium impacts around its hull to have permanent minor plastic deformation. Along with this, the vessel was not intended to have flat plating along the portion of the hull the six panels were cut from, adding a natural curvature to the plating of the panel. Therefore, this grillage had locked-in stresses throughout that would most likely affect the shape and curvatures of the plating once it had been cut into a size that could fit into the large pendulum apparatus. As well, the grillage plating curvature would change again due to the method as to which it was installed in the large pendulum apparatus. As the original grillage was 3-D modelled prior to being cut and before fitting into the large pendulum apparatus, the plating of the original grillage was simplified to a flat panel, as this was a controlled source of error, as opposed to using the measured curvature in the plating, since that had most likely changed and would create an uncontrolled source of error. Out of the six ex-HMCS IROQUOIS panels, the panel chosen for this research showed the least amount of curvature prior to being cut for carriage installation, however, with its imperfections, there were still differences between the original panel and its respective 3-D model. This is due to the fact that the original was not modelled by a 3-D scanner and the point cloud of the plating would not be able to capture every detail. As well, when the measurements of the original ex-HMCS IROQUOIS grillage longitudinal stiffeners had been made and incorporated into the original 3-D models, these dimensions were simplified to the average measured heights and widths of the respective webs and flanges. Both these simplifications were assumed to not have a significant impact on the final FEA model.

As described previously, the copy grillage was also simplified to aid not only in its 3-D model design, but also for ease of construction. As the original grillage had very similar dimensions across its three longitudinal stiffeners, a single longitudinal stiffener design was chosen for all three stiffeners across the copy panel. As well, the plating was kept flat, similar to the original ex-HMCS IROQUOIS grillage 3-D model. It would also be impractical to replicate the curvature of the original grillage in a 3-D model or in the physical copy grillage itself, and it was therefore decided that it would be appropriate to keep the copy panel flat in both instances. The dimensions of the copy grillages 3-D model are identical to its real-world counterpart.

Along with these two grillage models, the FEA model included an ice cone model, representing the ice cone indenter. The ice cone mesh was originally modelled by Dr. Bruce Quinton and the crushable foam material model was originally developed by Dr. Robert Gagnon (Gagnon, R. E. et al. 2006). Through this model, an ice indenter was able to impact the grillages, compressing and reacting as the two models continued to exert force onto one another. However, this model does have its limitations. It is unable to shatter and crack, as ice would normally do in an actual real-world impact. As well, it would most likely provide much higher impact loads in the finite element analysis. This ice material model would normally best match an ice cone with a spherical or truncated end, but due to the ice having a conical shape, this will create a source of error and different impact loads than that of the real-world experiment ice cone.

4.2.2 Model Dimensions and Design

The original model was initially developed through the hand measurement method described previously and the respective simplifications. Once this panel was imported and fully created in the 3-D computer-aided drafting program Rhinoceros 5, another issue was considered for the finite element analysis. This issue involved the development of the curvature in the connection between the web and flange of the stiffeners. Due to the time period when the vessel was designed and built, the stiffeners were pre-rolled to have a unique design. This included curvature at the connection between the web and flange, as well as the flange continuously getting thinner the further it reached from its middle section. This led to difficulty in its finite element analysis development, due to its uniqueness. To account for this, the flange was split up into seven sections, where each section was given its own thickness. There were four different thicknesses in total; one for the outer sections, one for the second-most outer sections, one for the inner-most sections, and a final one for the exact middle of the flange. As well, to account for the web-to-flange connection, this area was given its own thickness, to replicate this curvature. The following Figure shows how these thicknesses were represented in LS-Dyna.

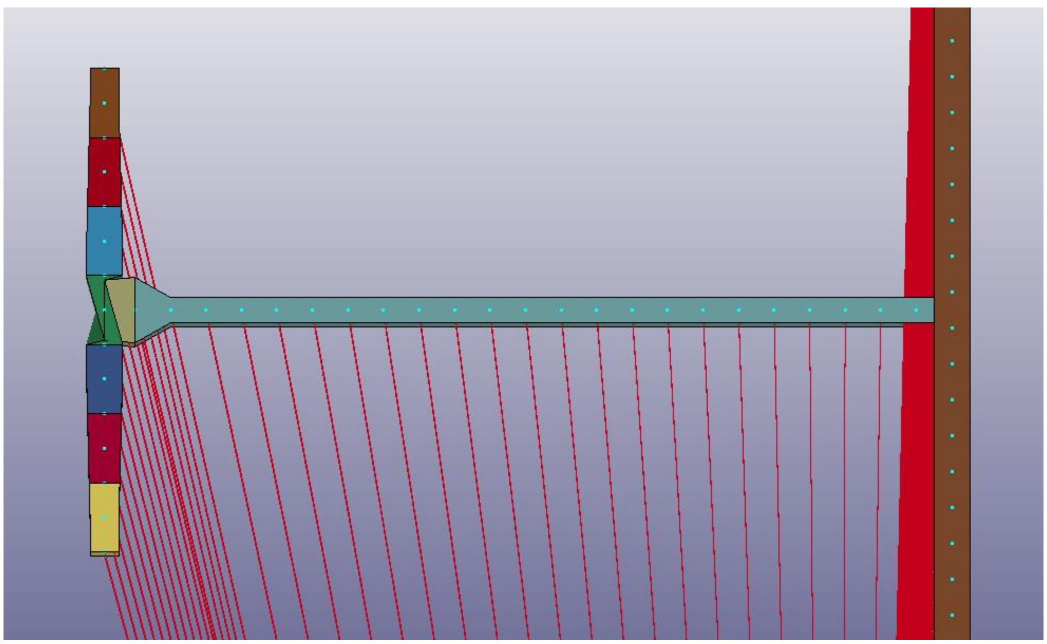


Figure 14 - Original Grillage Stiffener FEA Thicknesses

It should also be noted that the thickness of the green middle section of the flange is actually modelled correctly and that the issue in the Figure above is non-existent. This is a common issue for the version of LS-Dyna that was used throughout this analysis and will have zero effect on the finite element analysis.

Due to the copy panel also incorporating a unique stiffener, the thicknesses across the stiffener had to be changed to better match its real-world counterpart. However, the thickness of the flange of the copy panel was consistent throughout. Therefore, the only change involved the connection between the web and the flange. Figure 15 shows how the stiffeners in the copy panel were represented in the finite element analysis.

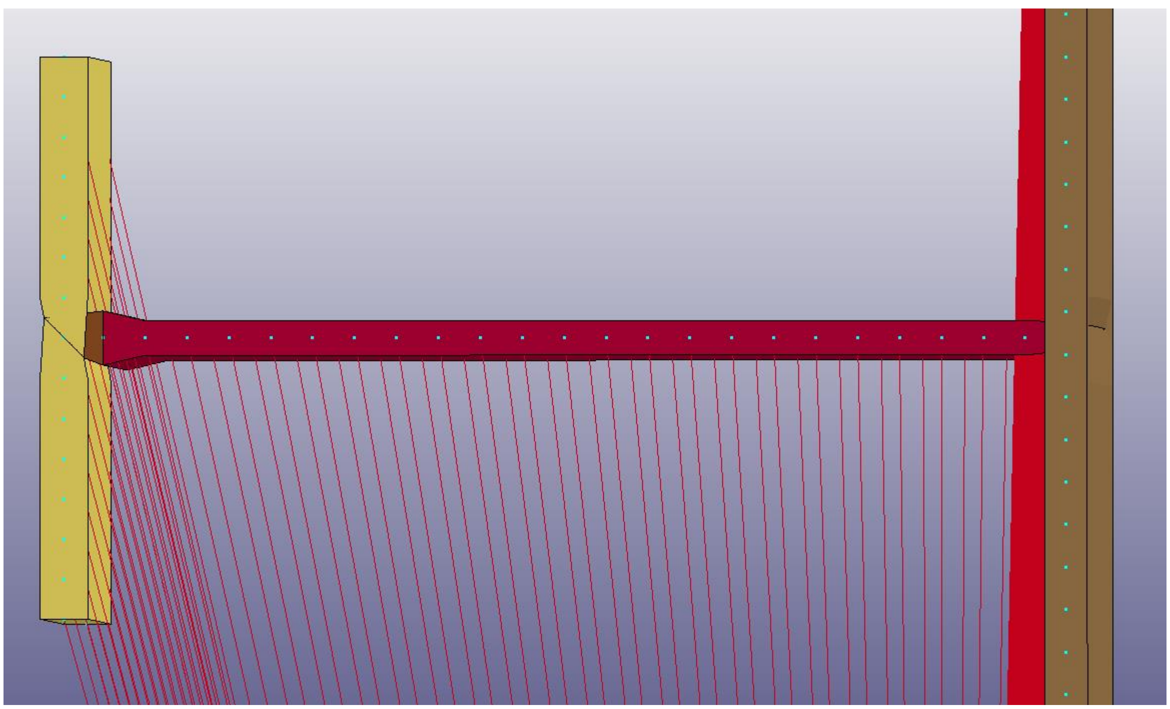


Figure 15 - Copy Panel Stiffener FEA Thicknesses

Similar to before, the same LS-Dyna issue can be seen in the connection between the flange and the web, however, this was again not how the panel was inevitably treated throughout the finite element analysis.

When compared to the stiffener cross-sections shown in Section 3.4.2, the final finite element models are within a reasonable degree of accuracy to their real-world counterparts, being as similar as possible given the technology involved with the project.

4.2.3 Ice Cone Model Design

The ice cone model (Quinton, B.W.T, et al. 2012) was designed as similar as possible to its real-world counterpart. The model consisted of two specific parts, the actual ice cone and its respective backing plate. The backing plate was treated as a rigid body and was modelled using solid elements. The ice cone backing plate was constrained in such a way that the only way it could move was translations in the direction normal to the panels plating.

The ice cone itself had two main parts to it, the cylindrical portion, which took up the majority of its model, and the conical point, which was the part that would impact the panel. The cylindrical section was developed with rigid boundary conditions around its out surface. This represents the holder in which the ice cone is shaped and held during the physical experiments, as the ice would be unable to deform outwards due to this barrier. The cone, however, is left to deform in all directions, including compressing into itself. This is accurate to the boundary conditions of the real-world experiment ice cone. The ice cone is also developed with solid elements throughout.

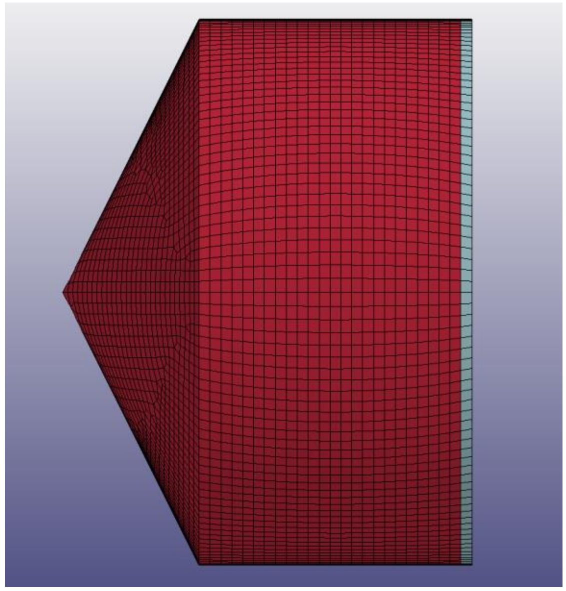


Figure 16 - Crushable Ice Cone Model

4.3 Material Models

Material models for the grillages were developed through two methods. The first was by using its respective foundry certification that was included when the steel was purchased. However, this was only applicable to the copy panel T-stiffeners that were purchased as W-beams, as described previously. The second method was material tests completed at the University. A series of tensile tests were completed through the use of an INSTRON machine located in the University's Mechanical Laboratory. INSTRON machines are capable of applying tensile stress onto a material specimen and measure its respective response to said stress (Illinois Tool Works Inc., 2021). Through measuring the material while a tension force is being applied to it, its complete tensile properties can be obtained. Therefore, this was deemed an accurate method of determining the material data needed for the material card of the finite element analysis. Tensile tests were completed on the plating and each of the three frames of the original ex-HMCS IROQUOIS grillage. This was done by analyzing the excess steel leftover from when the grillage had been cut into a size that could properly fit into the large double-pendulum apparatus. As well, the bulk steel utilized for the plating of the copy panel was also tested through the same method. This provided the highest level of certainty when considering the material models of the finite element analyses.

The two material cards chosen for the finite element analysis grillages were piecewise linear plasticity material and plastic kinematic material. For the ice cone model, a rigid material model and crushable foam material model were used.

4.3.1 Plastic Kinematic Material Model

The plastic kinematic material model is a fairly common material model used in finite element analysis, however, it is often a much simpler version of the piecewise linear plasticity material model. The only information provided in a nonlinear kinematic material model includes the density, the Young's Modulus, the Poisson's Ratio, the yield stress and the tangent modulus. The plastic deformation portion of the model is determined by a linear tangent modulus, which is a simplified comparison between the yield and ultimate strengths of a material. This is a straight-line comparison between the two stresses, providing a basic understanding of how the material model would react. Due to the only available material data of the copy panel stiffeners being the foundry's original certifications, this material model was used for that part of the finite element model. The data used for the copy grillage stiffeners through this material card is shown below.

Table 1 – Copy Grillage Framing Plastic Kinematic Material Model Data

Plastic Kinematic Material Model Data	
Density (kg/m^3)	7850
Young's Modulus (Pa)	2.00E+11
Poisson's Ratio	0.26
Yield Stress (Pa)	4.05E+08
Tangent Modulus	1.50E+09

4.3.2 Piecewise Linear Plasticity Material Model

This material card was used for the original ex-HMCS IROQUOIS grillage plating and stiffeners, and the plating of the copy panel. Similar to the plastic kinematic material model, the density, Young's Modulus, Poisson's Ratio, Yield Stress, and Tangent Modulus are given for the elastic portion of the impact. However, for the plastic portion of the impact, an increasing comparison of the stress and strain values are inputted. The data input for these geometries is shown below.

Table 2 - Original Grillage Plating Piecewise Linear Plasticity Material Model - Elastic Data

Original Grillage Plating Piecewise Linear Plasticity Material Model - Elastic Data	
Density (kg/m^3)	7850
Young's Modulus (Pa)	1.86E+11
Poisson's Ratio	0.3
Yield Stress (Pa)	3.68E+08
Tangent Modulus	8.04E+08

Table 3 - Original Grillage Plating Piecewise Linear Plasticity Material Model - Plastic Data

Original Grillage Plating Piecewise Linear Plasticity Material Model - Plastic Data		
Data Point	Strain Value	Stress Value
1	0.0000	3.68E+08
2	0.0377	4.37E+08
3	0.0705	5.12E+08
4	0.1065	5.58E+08
5	0.1452	5.90E+08
6	0.1688	6.05E+08
7	0.2295	6.34E+08
8	0.2907	5.81E+11

Table 4 - Original Grillage Framing Piecewise Linear Plasticity Material Model - Elastic Data

Original Grillage Framing Piecewise Linear Plasticity Material Model - Elastic Data	
Density (kg/m^3)	7850
Young's Modulus (Pa)	1.88E+11
Poisson's Ratio	0.3
Yield Stress (Pa)	3.64E+08
Tangent Modulus	1.60E+09

Table 5 - Original Grillage Framing Piecewise Linear Plasticity Material Model - Plastic Data

Original Grillage Framing Piecewise Linear Plasticity Material Model - Plastic Data		
Data Point	Strain Value	Stress Value
1	0.0000	3.64E+08
2	0.0385	4.45E+08
3	0.0728	5.04E+08
4	0.1113	5.41E+08
5	0.1535	5.69E+08
6	0.1902	5.85E+08
7	0.2256	5.85E+08
8	0.2599	5.20E+08

Table 6 - Copy Grillage Plating Piecewise Linear Plasticity Material Model - Elastic Data

Copy Grillage Plating Piecewise Linear Plasticity Material Model - Elastic Data	
Density (kg/m^3)	7850
Young's Modulus (Pa)	2.40E+11
Poisson's Ratio	0.3
Yield Stress (Pa)	3.55E+08
Tangent Modulus	1.96E+09

Table 7 - Copy Grillage Plating Piecewise Linear Plasticity Material Model - Plastic Data

Copy Grillage Plating Piecewise Linear Plasticity Material Model - Plastic Data		
Data Point	Strain Value	Stress Value
1	0.0000	3.55E+08
2	0.0451	4.67E+08
3	0.0861	5.71E+08
4	0.1285	6.49E+08
5	0.1719	7.17E+08
6	0.2153	7.75E+08
7	0.2480	7.57E+08
8	0.2775	5.90E+08

The piecewise linear plasticity model was chosen because the tensile tests provided accurate data points for each material past the point where plastic deformation began to occur. Piecewise linear plasticity provides a much more accurate description of the material model from the point where elastic deformation starts to become plastic deformation, up until the ultimate strength of a material and provides a much more accurate description of how the model will react post-yield stress, as a materials plastic strain will change as the stress increases. This will create a strain relationship that will curve leading up to the ultimate strength of the material, as opposed to a straight line. The elastic deformation portion of this material model will stay the same as other material models. As well, since the tensile test only provided accurate data up until the ultimate strength, a method must be used to extrapolate the entire true stress versus true strain curve. There are multiple methods that could be used, but the true stress was found through multiplying the Engineering Stress by the tared tensile strain plus one, and the true strain was found by calculating the natural logarithm of the tared tensile strain plus one.

4.3.3 Rigid Material Model

This material model was only used for one portion of the ice cone, which was the steel backing plate connected to the rear of the ice cone. This did not make contact with any part of the grillage, however, it aided in the development of appropriate boundary conditions of the ice cone. The rigid material model only had a density, Young's Modulus, and Poisson's Ratio provided for it, as rigid material models are unable to deform in a finite element analysis. Values inputted for this material card are shown below.

Table 8 - Ice Cone Indenter Steel Backing Plate Rigid Material Model

Rigid Material Model Data	
Density (kg/m^3)	7850
Young's Modulus (Pa)	2.07E+11
Poisson's Ratio	0.3

4.3.4 Crushable Foam Material Model

The ice model was developed under the crushable foam material card model in LS-DYNA. This model allows for reasonable deformation in the ice cone as it makes contact, with the deformation being based on a load curve and a tensile stress cut-off value. Values inputted for this material card are shown below.

Table 9 - Ice Cone Indenter Crushable Foam Material Model

Crushable Foam Material Model Data	
Density (kg/m^3)	900
Young's Modulus (Pa)	9.00E+09
Poisson's Ratio	0.003
Tensile Stress Cutoff (Pa)	8.00E+08

To define the stress-strain data for the crushable foam material model, a load curve was developed. The following Figure shows this load curve.

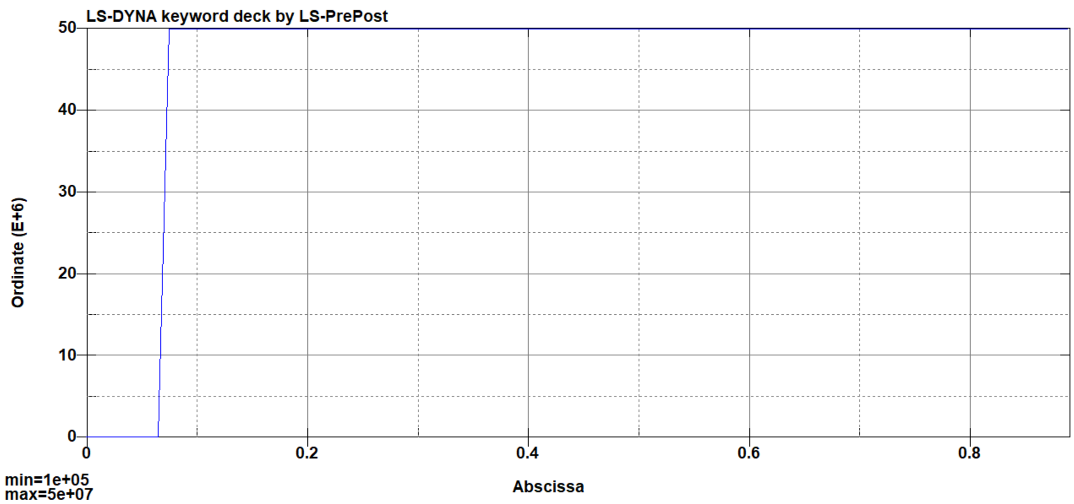


Figure 17 - Crushable Foam Material Model Stress-Strain Load Curve

4.4 Boundary Conditions

When designing a finite element model, boundary conditions must be included to appropriately analyze the field of interest. The ex-HMCS IROQUOIS grillage and its respective copy were clamped to the large-pendulum apparatus' carriage along the outer edge of the plate and stiffeners. Therefore, for an appropriate comparison, the boundary conditions of the finite element model were developed in the same way. These boundary conditions will occur at the outer nodes of the model, bounding them from both translations and rotations in all three axes. The boundary condition setup is shown in the figure below.

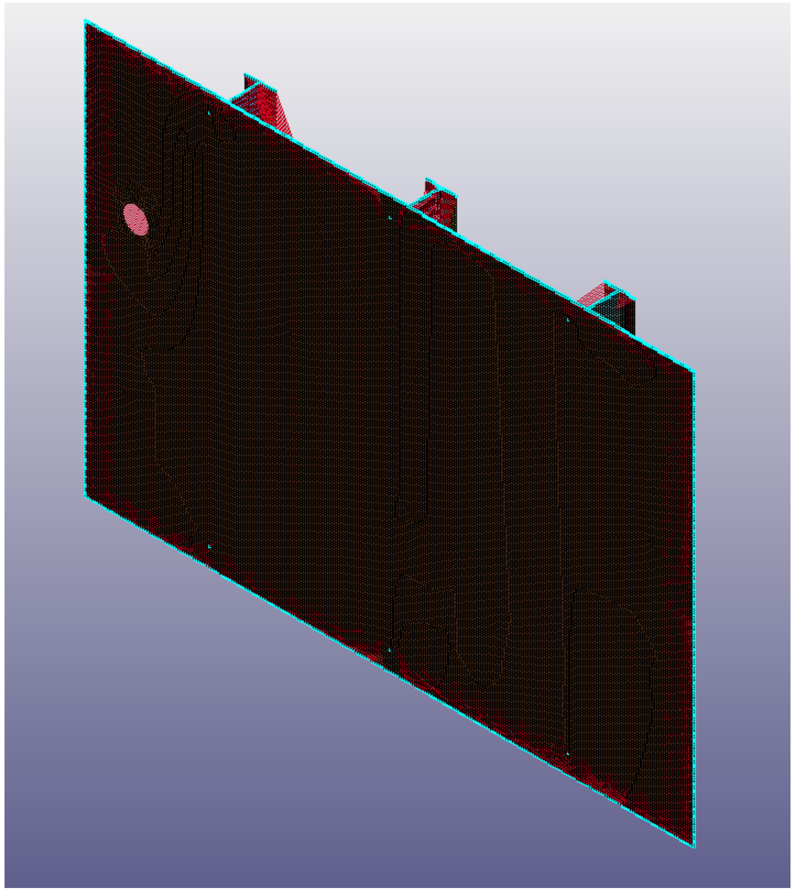


Figure 18 – Grillage Boundary Conditions

As can be seen, the boundary conditions (outlined in light blue) were placed along the outer edge nodes of both the plating and the stiffeners. These were developed as constrained nodal rigid boundaries (CNRBs), as this meets the criteria of nodes being unable to translate and rotate in all directions once impacted, while also allowing for the grillage to move forwards in the X-direction as a whole. CNRBs are also able to have weight data added to them and the weight of the overall carriage was distributed across these nodes, providing an even distribution of said weight data and allowing for the accuracy of the weight involved in the impact without having to manipulate the grillage itself.

4.5 Contact

There are multiple ways to apply a load to a model in finite element analysis. It not only depends on the type of problem you are facing, but also the program you inevitably choose to complete the simulations. For this experiment, LS-DYNA was chosen, which is capable of using multiple types of loading conditions. It was determined that to provide the highest accuracy to the model as compared to the real-world experiments, a two-body surface to surface type contact should be chosen. Therefore, the AUTOMATIC_SURFACE_TO_SURFACE card was used for this experiment. LS-DYNA's automatic surface to surface card meets all the requirements needed to keep the finite element simulation as similar as possible to the real-world experiments.

Two contact definitions were used during each simulation. The first contact definition involved the ice cone contacting the entirety of the grillage. This allowed for both full models to interact completely with one another. However, an issue arose where the grillage plating was passing through the middle stiffener web as the ice cone was deforming the grillage, showing that there was undetected contact between the plating and the stiffener web of the panel. This was fixed by including a second contact card, between just the grillage plating and the middle stiffener. The following figure shows the two model setup at the beginning of the simulation.

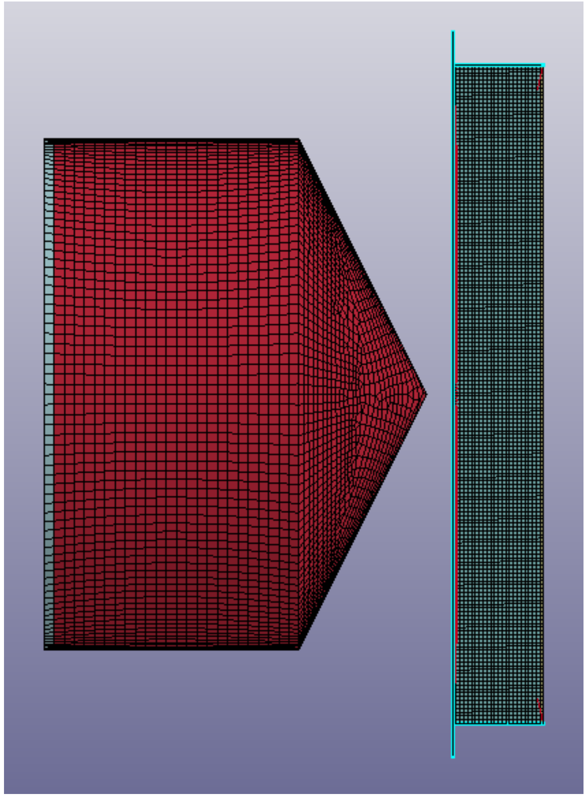


Figure 11 - Full FEA Model Setup

To complete the contact, the initial velocity card was used. This allows for individual parts or sets of parts to be given a velocity. By giving both the grillage and the ice indenter a velocity in opposing directions and having them placed so that contact will occur exactly where it did in the physical experiments, the experiment could be completely recreated using the real-world data that was available. The following tables show the speeds used for the original and copy grillage impacts.

Table 10 - Original Grillage Impact Speeds

Original Ex-HMCS IROQUOIS Grillage Impact Speeds		
Carriage	Starting Angle (Deg)	Respective Speed at Impact (m/s)
Ice	40.5	3.06
Panel	40.6	3.07

Table 11 - Copy Panel Impact Speeds

Copy Ex-HMCS IROQUOIS Grillage Impact Speeds		
Carriage	Starting Angle (Deg)	Respective Speed at Impact (m/s)
Ice	50.35	3.77
Panel	50.7	3.79

4.6 Solution Controls

The solution controls GLSTAT and MATSUM were used to check the energy of the model. GLSTAT represents the total energy reported in a system, including (DYNAmore GmbH, 2021):

- Internal Energy
- Kinetic energy
- Contact Energy
- Hourglass Energy
- System Damping Energy

MATSUM, however, is used to write these energies in a part-by-part basis (DYNAmore GmbH, 2021). These allow for the energy balance to be checked, providing a direct comparison of the total energy to the initial energy.

Along with this, the RCFORC option was used to gather the force data, as this is the force solution control card that is affiliated with contact simulations in LS-DYNA, and D3PLOT was used to show the von Mises stresses and displacements. The TERMINATION card was used to end the simulation after 0.07 seconds for the original grillage and 0.05 seconds for the copy grillage. As well, due to using explicit FEA methods, this meant that an automatic time step had to be chosen, which specified the time step in which all data would be recorded. This value was chosen to be 0.001 seconds.

4.7 Finite Element Model Simplifications

Several simplifications were made to the model, including both geometric and material simplifications already described. However, several other simplifications were made as necessary. These included:

- The boundary conditions were created to be fixed in all degrees of freedom along the outer shell and stiffeners. This is a result of the simplified structure that was used throughout the real-world experiments and finite element analyses.
- The load application was simplified to a crushable foam indenter.
- The material models, where necessary, were simplified to an average of the material tensile tests that were completed.

Overall, these simplifications will have some impact on the results of the finite element analysis, however, these have been justified due to the nature of the experiment and so that the project could compare finite element model impacts to the real-world experiment data.

4.8 Mesh

4.8.1 Meshing the Model

Once the initial model had been developed in Rhinoceros 5, the model was meshed using HyperMesh. HyperMesh is a meshing software developed by Altair Hyperworks that can import Rhinoceros 5 IGES files, mesh them, and then export them into LS-DYNA keyword files. Due to the ease and capabilities of this software, it was deemed an appropriate option when considering the large and complex meshes that were necessary for these finite element analyses.

The grillage was meshed entirely out of 3-D shell elements. These shell elements were designed with 5 through-thickness integration points, meaning that 5 points along the cross-section were used to analyze its behavior. This allowed for the thickness to be simulated, without the need of using solid elements. This also increases the efficiency of the model without losing accuracy, as shell elements are much more computationally efficient than solid elements. The shell elements utilized reduced integration, as this not only alleviated the risk of volumetric locking, which is likely in this scenario as the shell plating will be transitioning from elastic to plastic failure, but it also alleviated the risk of shell elements being too thin and being subjected to bending, which is known as shear locking. As well, reduced integration elements are computationally efficient.

Each element was meshed the same size throughout the model at 7 millimetres, as determined through a mesh convergence analysis, and this element sizing was used for both the original and copy grillage analyses. In total the original ex-HMCS IROQUOIS panel model had 71,285 elements and the respective copy panel had 71,586 shell elements.

4.8.2 Mesh Convergence Analysis

When deciding on a mesh size for a finite element model, the element sizes must be small enough to show accurate results based on the analysis. However, if a mesh is too small, then this can take a significant amount of computational effort to analyze such a model. Therefore, a mesh convergence analysis (MCA) is used to find an appropriate mesh size. In this case, the maximum displacement of the grillage plating was used to complete the mesh convergence analysis. If stress were to be used as the basis for your mesh convergence analysis, then geometric discontinuities may cause incorrect and theoretically infinite peaks in the stress at certain points in the model, therefore making the maximum stress incorrect. This shows that using displacement for an MCA would be more appropriate. The displacement was taken in the x-direction, as that is the direction of the initial velocity the grillage will be moving in. It should be noted that as the grillage was moving throughout the simulation, this value is not a direct representation of the actual maximum displacement of the panel due to deformations alone, however, this will be further explained and explored in the finite element analysis results section throughout Chapter 5.

For this study, an MCA was completed on the original ex-HMCS IROQUOIS finite element model using five different shell element sizes. Each of the five element sizes were ran within the model under the same conditions. The following element sizes were considered:

- Shell Element Size – 20 mm, Number of Elements – 10,702
- Shell Element Size – 15 mm, Number of Elements – 18,218
- Shell Element Size – 10 mm, Number of Elements – 41,602
- Shell Element Size – 7 mm, Number of Elements – 71,285
- Shell Element Size – 5 mm, Number of Elements – 162,212

Through noting the maximum x-displacements of these five finite element models, the following graph was developed.

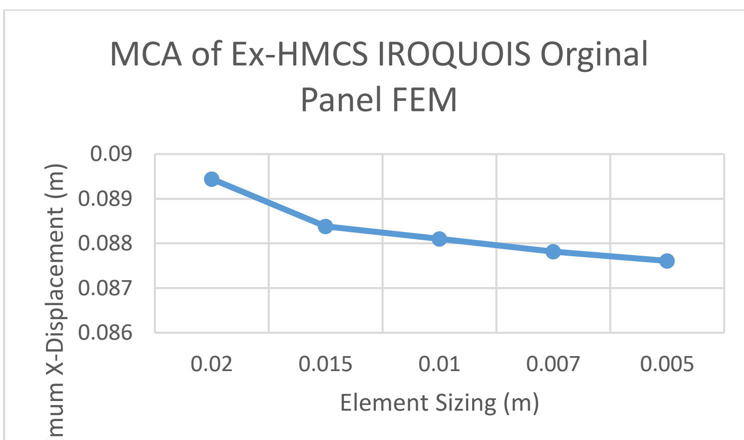


Figure 20 - Mesh Convergence Analysis of Original Grillage

Figure 20 shows the model has leveled out and is showing a resultant x-displacement of roughly 8.77 mm. Therefore, it can be assumed that this final displacement value is reasonably correct and that the fourth element size option is appropriate. Choosing the fifth sizing would also provide similar and appropriate results, however, it would result in an unreasonably long processing time.

Along with this, a second smaller MCA was completed on the copy panel. It was initially assumed that the element size chosen for the original panel finite element model would be appropriate for the copy panel model, however, a three element size MCA was completed to ensure this was accurate. The following element sizes were considered:

- Shell Element Size – 20 mm, Number of Elements – 10,300
- Shell Element Size – 10 mm, Number of Elements – 40,095
- Shell Element Size – 7 mm, Number of Elements – 71,586

Through noting the maximum x-displacements of these three finite element models, the following graph was developed.

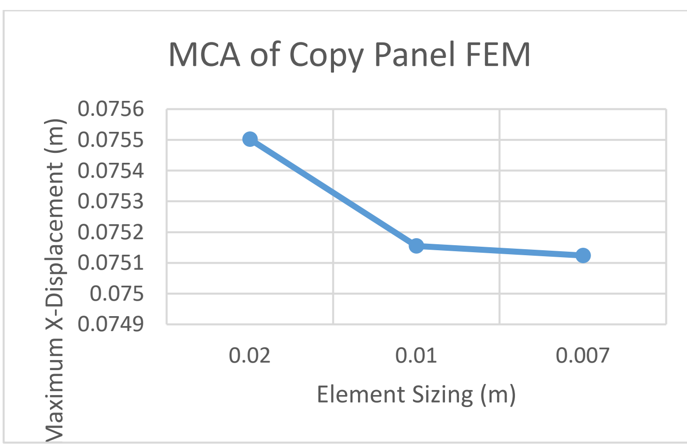


Figure 21 - Mesh Convergence Analysis of Copy Grillage

As can be seen, the MCA holds true for the copy panel as well, and the 7 mm element size chosen is reasonable for both finite element models, providing accurate results without unnecessary computational effort.

4.8.3 Mesh Quality Check

When checking the quality of the elements, shell elements have different qualities that must be analyzed. Specifically, four main aspects of the mesh must be reviewed to determine if the mesh was of adequate quality. These four checks include the following:

- Warpage: The amount by which an element deviates from being planar, or "warped."
- Aspect Ratio: The ratio between the longest edge and the shortest edge of an element.
- Skew: The minimum angle between two lines made from joining opposite mid-sides of an element.
- Jacobian: A measure of the deviation of an element from its "ideal shape."

Controls were chosen and the model was then scanned for any elements that violated these controls. Table 12 details the limits as well as the number and percentage of elements that violated these limits for the ex-HMCS IROQUOIS original panel finite element model.

Table 12 - Original Grillage FEM Check

Original Ex-HMCS IROQUOIS Panel FEM Check			
Aspect Ratio Results			
Min. Value	Max. Value	Allowable Value	# Violated
1	1.54	5	0 (0%)
Warpage Check Results			
Min. Value	Max. Value	Allowable Value	# Violated
0	0.0198	10	0 (0%)
Skew Check Results			
Min. Value	Max. Value	Allowable Value	# Violated
0	21.2	45	0 (0%)
Jacobian Ratio Check Results			
Min. Value	Max. Value	Allowable Value	# Violated
0.833	1	0.6	0 (0%)

As can be seen from Table 12, the mesh quality for the original panel FEM shell elements showed zero violations throughout the four quality checks. The copy panel was then checked under the same conditions, which gave the following information.

Table 13 - Copy Grillage FEM Check

Copy Panel FEM Check			
Aspect Ratio Results			
Min. Value	Max. Value	Allowable Value	# Violated
1	1.49	5	0 (0%)
Warpage Check Results			
Min. Value	Max. Value	Allowable Value	# Violated
0	0.0198	10	0 (0%)
Skew Check Results			
Min. Value	Max. Value	Allowable Value	# Violated
0	18.5	45	0 (0%)
Jacobian Ratio Check Results			
Min. Value	Max. Value	Allowable Value	# Violated
0.873	1	0.6	0 (0%)

The shell elements for the copy panel FEM also showed zero violations across the four quality checks.

As can be seen from the mesh quality checks, both panel types were shown to be meshed appropriately, therefore showing that the elements would not cause any issues or inaccurate results once both finite element analyses were completed.

Chapter 5 Results and Discussions

5.1 Real-World Experiment Results

Once the two grillages had been impacted, the data had to be analyzed. Even though strain gauges, accelerometers, and load cells were used to record the data of the experiment, only the load cell data was used in this research. This is due to the fact that the strain and accelerometer data did not provide useful information at this point of the research project, however, the data will be kept for future works if a benefit is found. The force values found by each load cell were tared individually and then summed to get the total impact force. Once this had been completed, this allowed for a graph showing the force value as it changed throughout the impact. Along with the force data, the indentations in the panel were measured. This was done through hand measurements before and after each impact. Therefore, the maximum residual displacement of the real-world grillage plating was found within a reasonable degree of accuracy. These measurements were taken at the vertical center of the panel, in line with the ice cone's initial contact location. Nine data points were found across the horizontal length of the panel plating, as well, an additional point was taken at the location of maximum displacement, in the case where one of the nine data points did not line up directly with this location. These nine data points were kept consistent across every experiment and the following diagram shows where each data point was located across the panel.

Figure 22 - Experiment Displacement Data Point Locations

Each data point lined up with the right side of each piece of green tape, shown in Figure 22, and this straight edge was given only this purpose for each experiment. Each data point was 7.2 inches from one another, with the middle point being located directly in the center of the panel.

Determining the results of the real-world experiments involves the analysis of these two sets of data. Once the data had been received, both sets of data were translated to a Microsoft Excel file.

For the maximum displacement data, there were only two sets of data; the values of the measurements found before and those found after the experimental impact occurred, from a respective datum to the grillage. Therefore, this data was tared from one another to determine the displacement measurements across the panel plating.

The load cell data were also analyzed in a Microsoft Excel file, with significantly more data points to choose from. Data recorded at times prior to and after the impact were deleted, as these occurred due to the need to manually start and stop the data recording process in the lab, leading to significant amounts of data being recorded outside of the impact range. This impact range consists of the moments just prior to and after the impact, meaning any additional data was not useful to this research. Once the data had been selected, the information from each individual load cell was tared. This was done by taking a constant portion of an individual load cell's data, prior to or after the impact, and an average taken. Once the average of the non-impact portion had been taken, the entire load cell data was tared by this value. This process was then repeated for both remaining load cells. The total load was then developed through addition of the three sets of load cell data.

5.1.1 Original Ex-HMCS IROQUOIS Grillage

Two sets of data were measured; the values of the panel plating distance from a respective datum before and after the impact. However, the real-world experiments for this panel were completed differently. Instead of using a perfect cone during the first impact, the cone was accidentally truncated while it was being loaded into the ice cone holder on the LPA. Once this impact was completed, the ice cone was completely destroyed and the panel was impacted by a newly developed ice cone. This ice cone was not truncated and, therefore, kept its conical shape. To ensure the force data was as accurate to the finite element model as possible, this led to the experimental force data chosen to be that of the second experiment on the same ex-HMCS IROQUOIS original panel. Therefore, this may show some differences when comparing the two, as the panel had a small indent prior to this second impact. This difference will also show changes in the displacement values, as the initial plating was not flat. Therefore, the displacement values due to both the first and second impacts were considered once the data was compared against the respective finite element model.

5.1.1.1 Maximum Displacement Results

The following measurements were found prior to any impact, after the first impact, and after the second impact. All measurements were in centimetres and in reference to the respective datum.

Table 14 - Original Ex-HMCS IROQUOIS Panel Displacement Measurement Array

Impact	Panel Displacement Measurement Array								
	1	2	3	4	5	6	7	8	9
Pre-Impact	7.0	6.7	6.6	6.5	6.5	7.0	7.1	7.1	7.1
After First Impact	7.0	6.6	6.6	6.7	6.5	7.0	7.1	7.0	7.0
After Second Impact	7	6.6	6.7	7	6.7	7.2	7.1	6.9	6.9

The first impact involved a truncated ice indenter and the second impact involved a conical ice indenter. As can be seen, there was a minimal change after the first impact, only showing no more than 0.2 cm of change across any of the points.

Therefore, the maximum displacement results were taken from the change before and after the second impact. The following displacement changes were found due to this impact.

Table 15 - Original Ex-HMCS IROQUOIS Panel Total Displacements

Original Ex-HMCS IROQUOIS Panel Total Displacement									
Measurement Location	1	2	3	4	5	6	7	8	9
Total Disp. Change (cm)	0.0	0.0	0.1	0.3	0.2	0.2	0.0	-0.1	-0.1

Table 15 shows there was minimal plastic damage to the panel due to the ice cone impact. This was found in both the truncated cone and the perfect cone ice indenter impacts. The maximum displacement found from the impact was 0.3 centimetres and was located just to the left of the middle stiffener. The following figure shows the panel after the second impact, showing the lack of residual plastic deformation post-impact.

Figure 23 - Original Ex-HMCS IROQUOIS Grillage Post-Impact

5.1.1.2 Maximum Force Results

The force data was recorded by three load cells located on the ice holder carriage, behind the ice indenter. Each individual set of load data was analyzed, along with the total of the three. This provided the maximum force experienced throughout the experiment. The following figure shows the force data throughout the original ex-HMCS-IROQUOIS panel physical experiment from all three load cells, as well as their summed total.

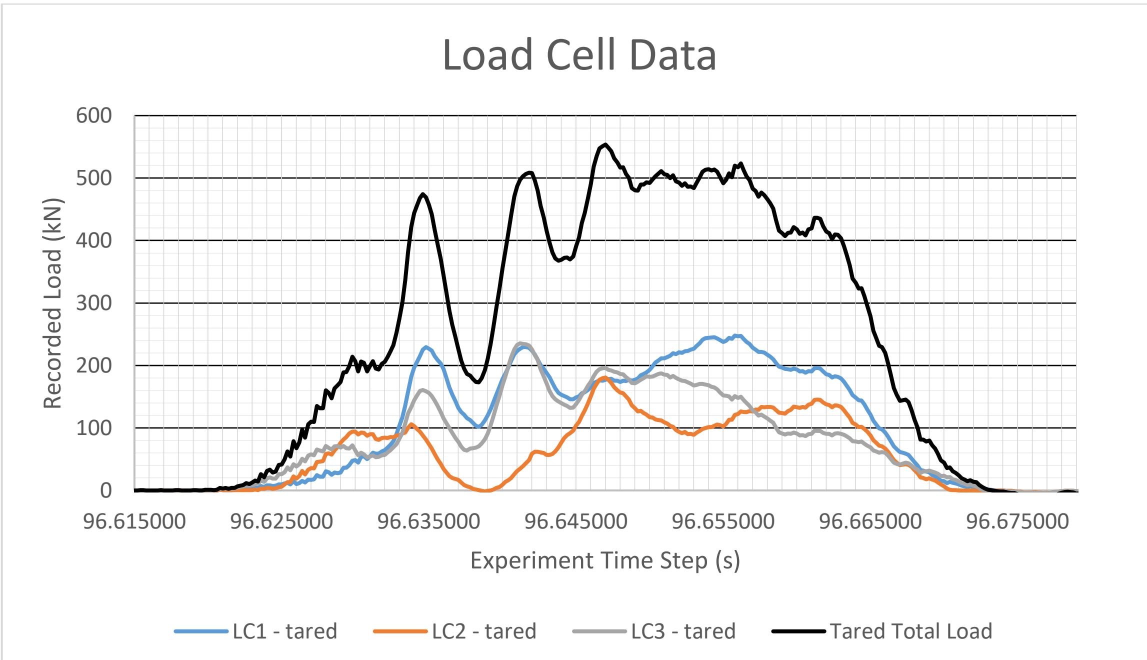


Figure 24 - Original Ex-HMCS IROQUOIS Panel Load Cell Data (Figure was generated by S. Andrade, and is used with permission)

Figure 24 the impact started at 96.625 seconds into the experiment and completed at 96.675 seconds. This gave the total impact time as approximately 0.05 seconds. There are many peaks in the data, which was due to the ice indenter breaking apart. The changes of high- and low-pressure zones in the ice are what provided these peaks and troughs in the load cell data.

Overall, the maximum force that was experienced by the grillage due to the impact was approximately 548 kiloNewtons (kN).

5.1.2 Copy Ex-HMCS IROQUOIS Grillage

Although the original grillage had to utilize the data from its second impact experiment, this was not the case for the copy grillage. The cone used in the first impact of the copy panel was not truncated accidentally and, therefore, the data from the first impact experiment was used throughout this research.

5.1.2.1 Maximum Displacement Results

Table 16 shows the measurements found prior to any impact and after the first impact. All measurements were in centimetres and in reference to the respective datum.

Table 16 – Copy Ex-HMCS IROQUOIS Panel Displacement Measurement Array

Impact	Panel Displacement Measurement Array								
	1	2	3	4	5	6	7	8	9
Pre-Impact	7.0	6.8	7.1	7.2	6.9	7.2	7.2	6.9	7.0
After First Impact	6.9	6.9	7.7	8.7	9.5	9.4	7.9	7.0	6.9

There were much higher plating displacements due to this impact. That is because the copy panel was impacted with the carriages starting at an angle approximately 10 degrees larger than they did for the original panel impact. Therefore, this yielded higher indentations in the panel plating. Table 17 shows these plastic deformation totals across the copy panel.

Table 17 - Copy Ex-HMCS IROQUOIS Panel Total Displacements

Copy Ex-HMCS IROQUOIS Panel Total Displacement									
Measurement Location	1	2	3	4	5	6	7	8	9
Total Disp. Change (cm)	-0.1	0.1	0.6	1.6	2.6	2.2	0.7	0.1	-0.1

As can be seen, the largest residual deformation found along the copy panel was right in the middle, with a measurement of 2.6 centimetres. However, this was not the actual point of highest deformation, this was only the highest deformation found along the nine predetermined data recording locations. Therefore, a separate measurement was taken at the point of the largest plastic deformation with respect to the datum. This pre-tared measurement was found to be 10.8 centimetres. This value was found to be higher due to the majority of the impact occurring at this point, just to the right of the middle stiffener.

As the measurement of the panel at the location of the highest deformation was not recorded prior to the impact, an estimate was made. The deformation was found to occur roughly in the middle of points 5 and 6 along the panel, therefore, these points were averaged to give an approximate pre-impact measurement of 7.05 centimetres. When taking the measurement of maximum deformation and subtracting the pre-impact estimate, the value for maximum deformation in the panel plating is equal to 3.75 centimetres.

The following Figure shows the copy panel after impact with the ice cone indenter.

Figure 25 - Copy Ex-HMCS IROQUOIS Grillage Post-Impact

5.1.2.2 Maximum Force Results

The force data of the copy panel was also recorded by three load cells located on the ice holder carriage. The data for each individual load cell, along with the total combined load data, was analyzed. The following figure shows the forces that were experienced throughout the impact.

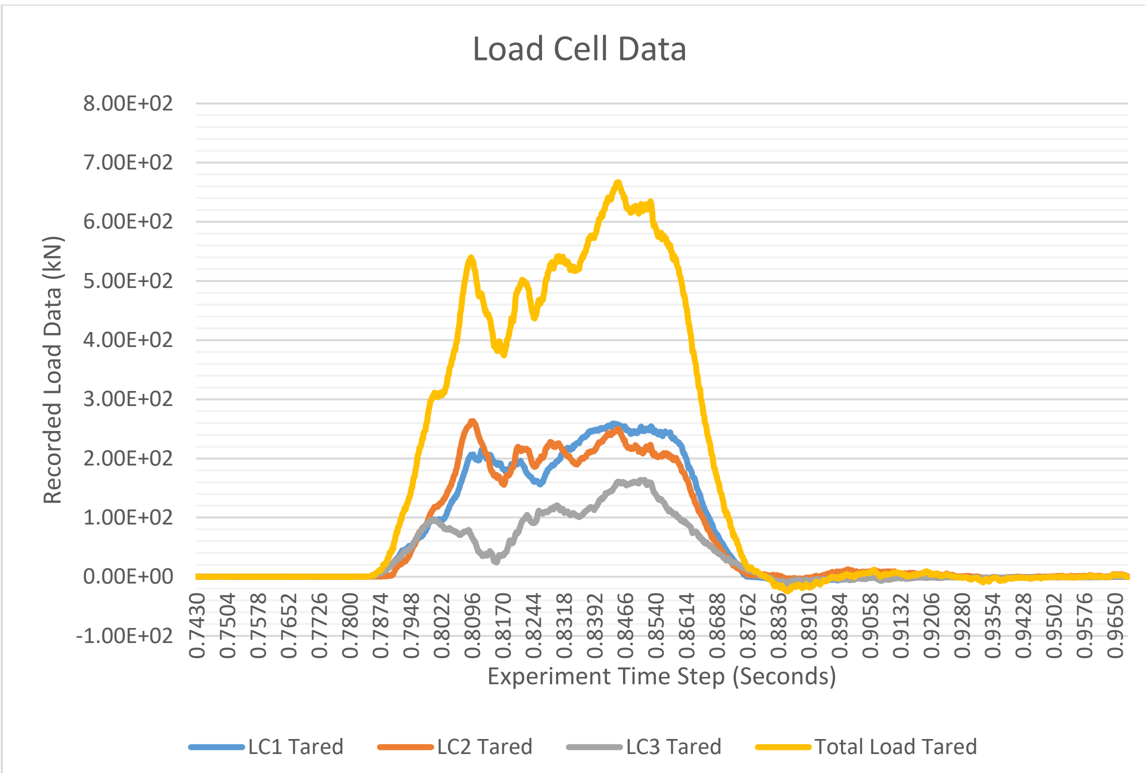


Figure 26 - Copy Ex-HMCS IROQUOIS Panel Load Cell Data

As can be seen, the impact started at roughly 0.7874 seconds into the experiment and completed at roughly 0.8762 seconds. This gave the total impact time as approximately 0.1 seconds. Similar to the original panel impact, there are many peaks in the data, which were due to the ice indenter breaking apart. The changes of high- and low-pressure zones in the ice are what provided these peaks and troughs in the load cell data.

Overall, the maximum force that was experienced by the grillage due to the impact was approximately 667 kiloNewtons (kN).

5.2 Finite Element Analysis Results

Similar to the real-world experimental results, the maximum force and maximum grillage plating displacement were found for both the original and copy grillage finite element models. This was found through LS-DYNA's post-processing software. Once this had been found, the von Mises stress data was analyzed to expand upon the finite element models themselves and to provide a better understanding of how they reacted as compared to their real-world counterparts. As described previously, the original grillage experiment data used throughout this research was from its second impact and the copy grillage experimental data used throughout this research was from its first impact. This was reflected in the finite element models and analyses. However, as described in Chapter 4, the finite element model of the original panel was developed flat. This was not only a simplification made as compared to the original grillage prior to experimental impacts, but this added another source of error when considering the comparison of the data analysis of the flat finite element model to the already indented original ex-HMCS IROQUOIS grillage. Although significant indentation did not occur from the first real-world experiment impact, this source of error should be remembered throughout this portion of the thesis.

When considering the analysis of the maximum displacement data in the two finite element analyses, there are two different types of maximum values that are available, as opposed to having only one recorded value from the real-world experiments. This is due to the fact that the FEA is able to record data during the entire impact, while the experiments only provided displacement data that was recorded after the impacts. Therefore, the value that must be taken from the FEM that can be reasonably compared to the experimental data is that which was found after the impact occurred. The finite element model would have a larger peak in its maximum displacement data due to elastic deformations, but once the impact had finished the only residual displacements would be due to the plastic deformations. Therefore, the value taken at the end of the simulation would involve only the residual deformations, making it comparable to the real-world experiments. To meet this need, two nodes were compared to one another. As the grillage was given an initial velocity and the deformations were changing throughout, just the maximum value of the node at the centre of the impact was not able to provide an accurate value. Therefore, this value was compared to a node at the panel plating boundary. As the nodes at the outer perimeter of the grillage were also moving with the initial velocity and not affected by the impact, this allowed for an accurate representation of the maximum displacement value. By taking the x-location of the outermost node and subtracting it from the x-location of the node which undertook the highest change due to plastic deformation, the maximum displacement of the grillage plating could be determined. This method was used for both the copy and original grillages.

The force data was able to be directly compared to the experimental data, as both provided a maximum value and a history throughout the impact. Along with this, the von Mises stress, or effective stress, was also analyzed. This data is unable to be compared to the experimental data, however, it was still included in this research due to its availability.

5.2.1 Original Ex-HMCS IROQUOIS Grillage FEA

5.2.1.1 Maximum Displacement Results

Through the method described previously, the maximum displacement value of the plating due to plastic deformation was found. The following graph shows the two elements as compared to one another.

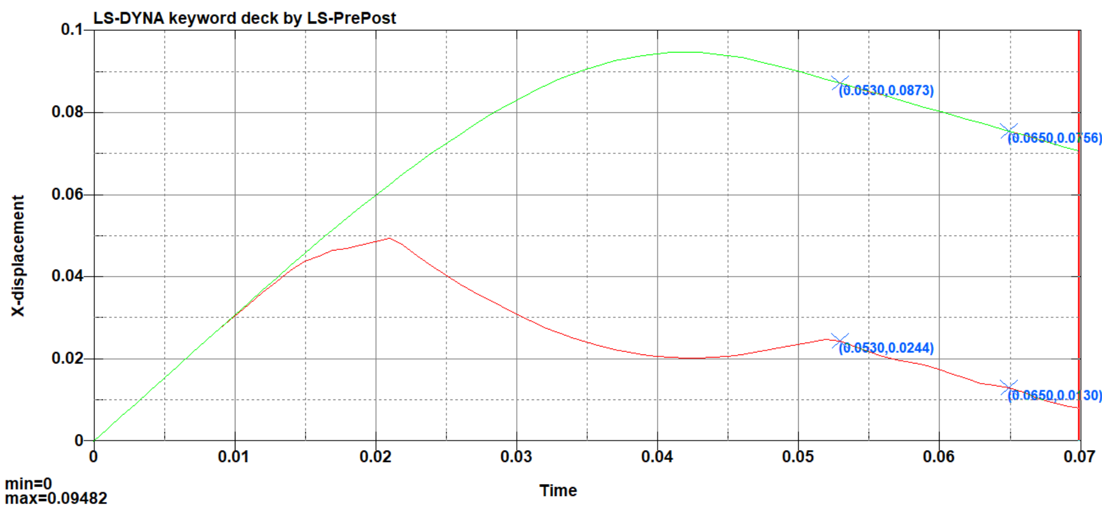


Figure 27 - Original Ex-HMCS IROQUOIS Panel FEA Maximum Displacement

As can be seen, there are two lines, each one representing the history of a specific element in the original grillage finite element model. The green line represents the element at the perimeter of the plating, which was unaffected by the impact, while the red line represents the element most affected by the impact, at the centre of the impact. By subtracting the two x-displacement values at the same point in the simulation where the full effects of the impact have been completed, the maximum displacement of the grillage due to deformation can be found. It should also be noted that the curvature of the graph is due to the contact type, with neither the grillage nor the ice indenter being bounded in one specific location and they impacted each other through both being given an initial velocity. This leads to the grillage as a whole moving, which in turn causes the curvature in the graph above.

From the graph, the effects of the impact are fully complete at approximately 0.053 seconds into the simulation, where the graph hits a second peak. Therefore, the maximum x-displacement for both nodes levels out and shows the magnitude of the plastic deformation. By checking multiple points along this line, the maximum displacement experienced by the grillage plating was found to be approximately 0.063 metres, or 6.3 centimetres. The following Figure shows the change in dent depth versus time throughout the finite element simulation.

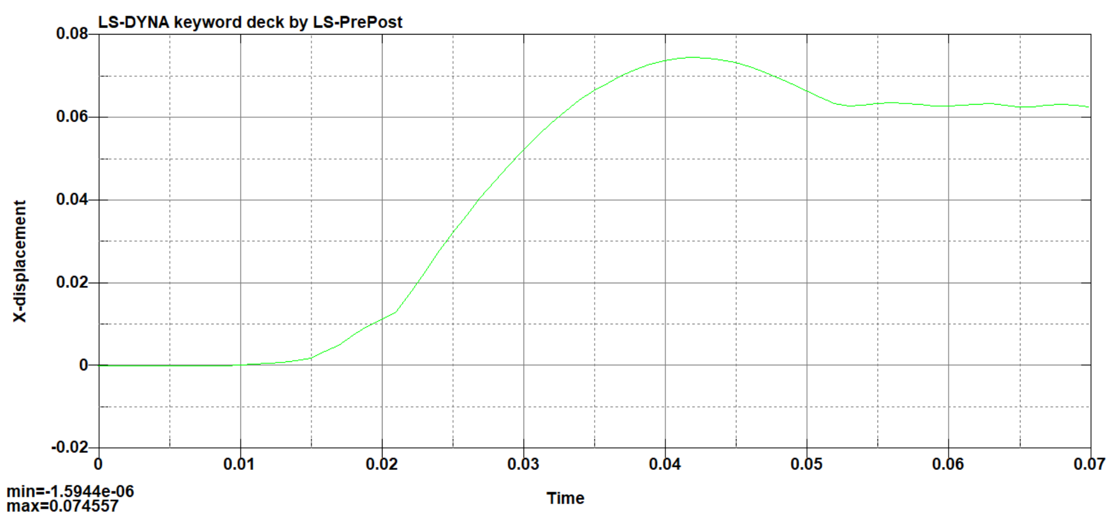


Figure 28 - Dent Depth versus Time of Original Grillage

As can be seen, the dent depth increased to an apex and then decreased, due to elastic deformation, finalizing with the residual dent depth of the impact from plastic deformation.

5.2.1.2 Maximum Force Results

Once the maximum displacement had been found, the maximum force was determined. This was done through LS-DYNA's post-processing software. The following graph shows the maximum x-force experienced throughout the finite element model.

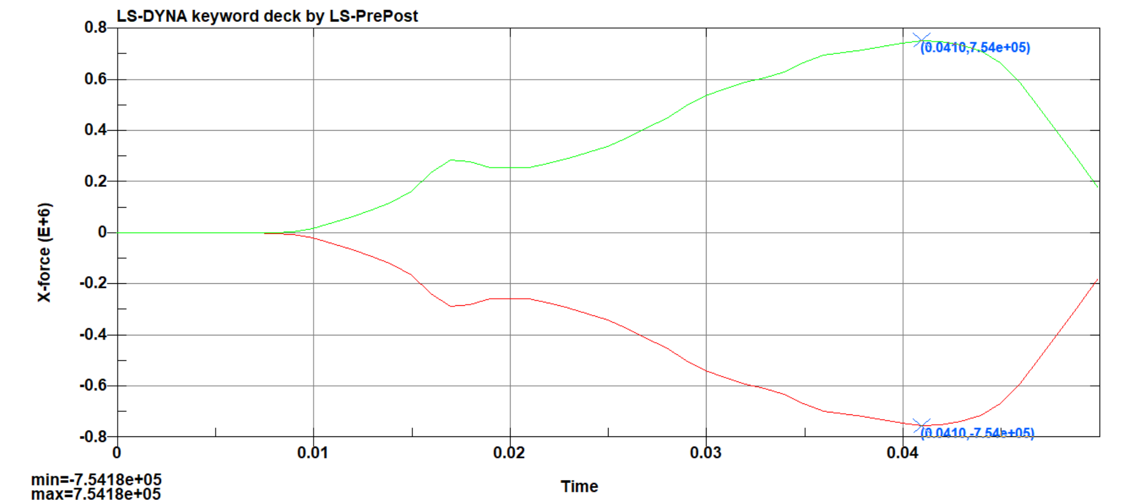


Figure 29 - Original Ex-HMCS IROQUOIS Panel FEA Maximum Force

Each line represents each respective entity involved in the impact; the ice indenter and the grillage. The simulation shows an equivalent and opposite force being enacted on each geometry.

As can be seen, for both the ice indenter and the grillage, the maximum force peaked at approximately 754,180 Newtons.

5.2.1.3 Von Mises Stress Results

As described previously, the effective stress was included in this research to provide another way of analyzing the finite element models. The following Figures show the results of this data.

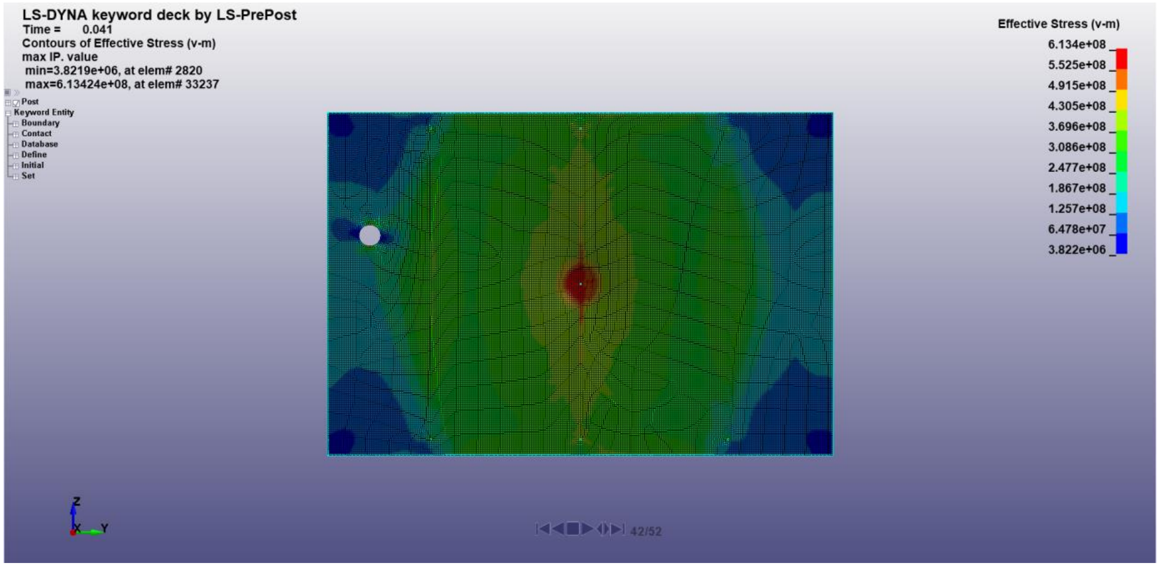


Figure 30 - Original Ex-HMCS IROQUOIS Panel Von Mises Stress - Front

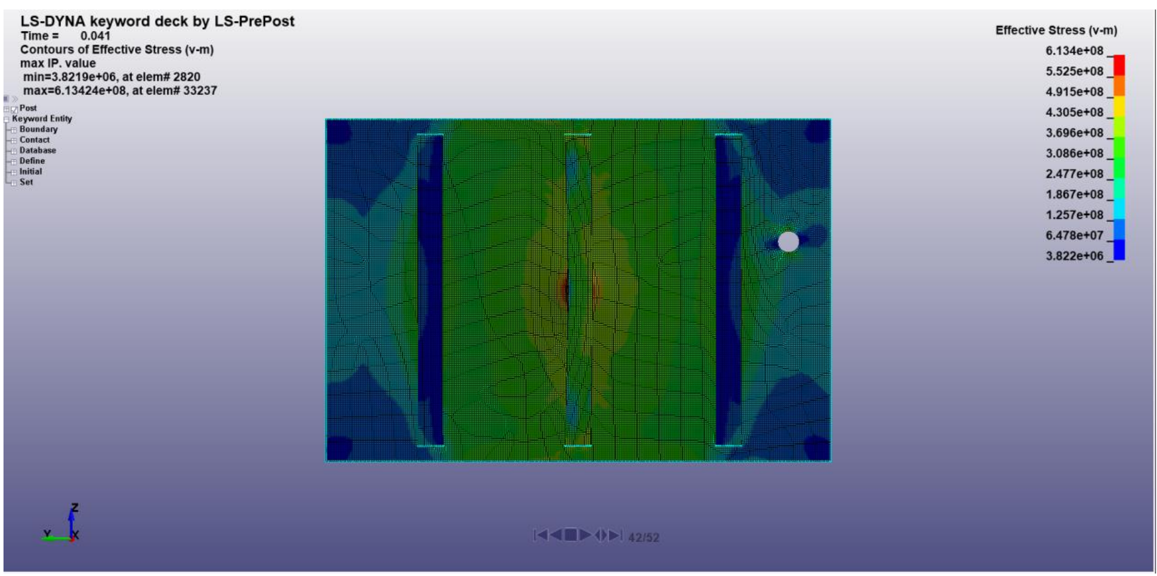


Figure 31 - Original Ex-HMCS IROQUOIS Panel Von Mises Stress - Back

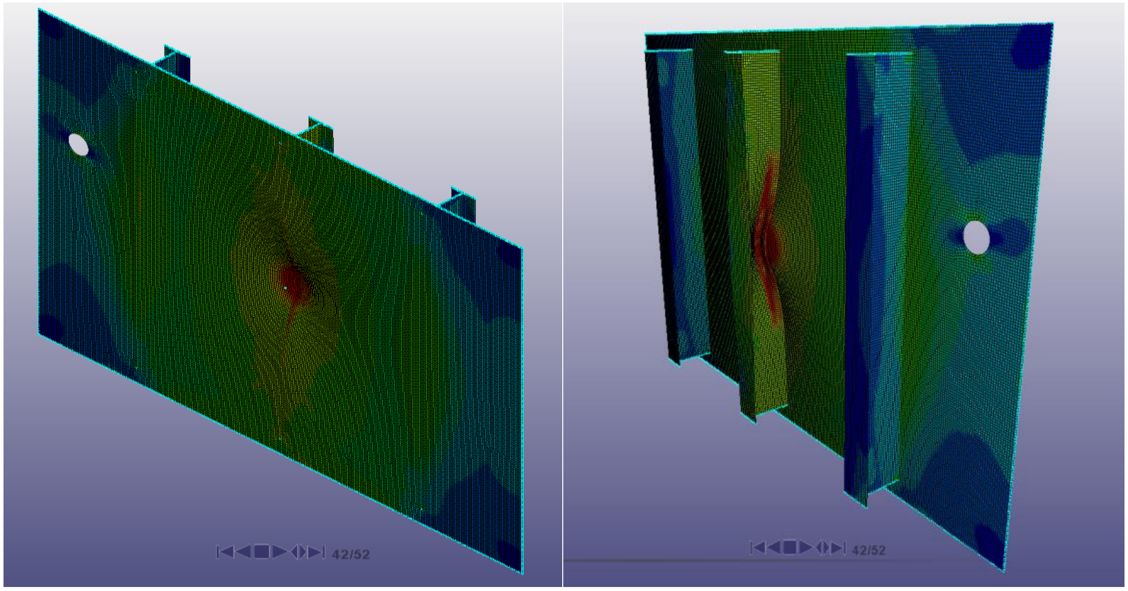


Figure 32 - Original Ex-HMCS IROQUOIS Panel Von Mises Stress - Isometric Front and Back

The maximum effective stress in the finite element analysis was found to be approximately 613.4 MPa.

5.2.2 Copy Ex-HMCS IROQUOIS Grillage FEA

5.2.2.1 Maximum Displacement Results

Through the method described previously, the maximum displacement value of the plating due to plastic deformation was found. The following graph shows the two elements as compared to one another.

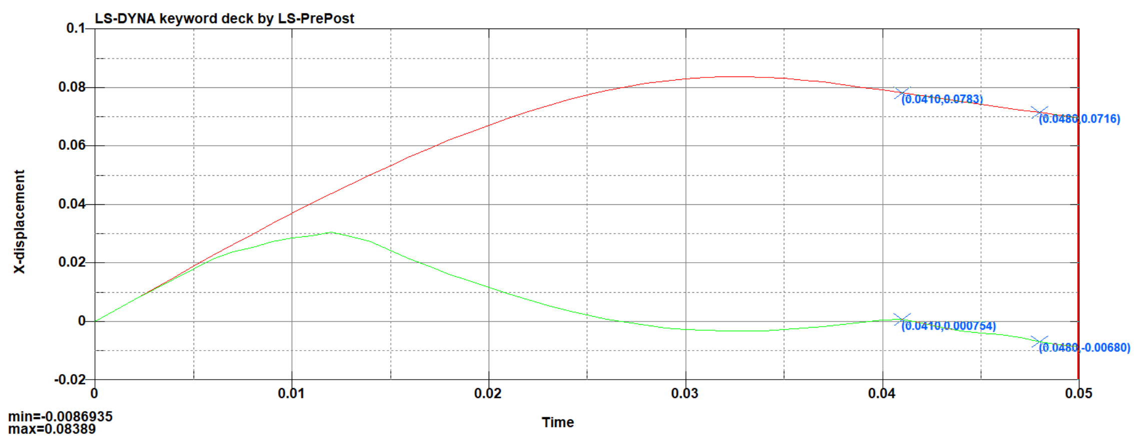


Figure 33 - Original Ex-HMCS IROQUOIS Panel FEA Maximum Displacement

In this case, the red line represents the element along the perimeter of the panel plating while the green line represents the element most affected by the impact. Therefore, through the same method as before, the maximum displacement value can be found by subtracting the x-displacement values of the two lines once the simulation has completed the impact. As can be seen, the impact ends at the second peak, which was approximately at 0.041 seconds into the simulation. Therefore, by checking multiple points at and after this simulation time, the maximum displacement of the panel plating can be found. This maximum displacement due to plastic deformation was approximately 0.078 metres, or 7.8

centimetres. The following Figure shows the change in dent depth versus time throughout the finite element simulation.

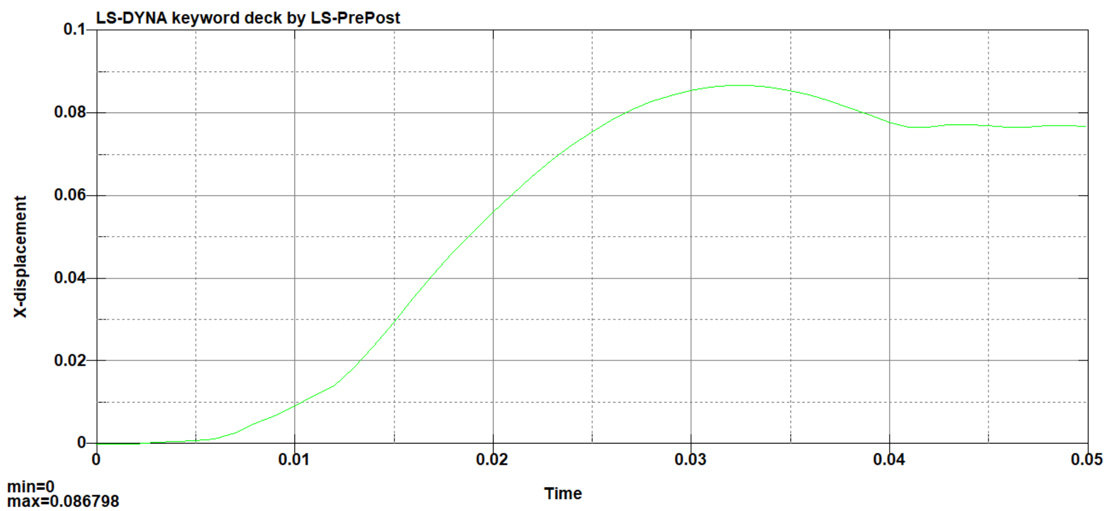


Figure 34 - Dent Depth versus Time of Copy Grillage

Similar to the original grillage finite element model, the dent depth increased to an apex and then decreased due to elastic deformation, finalizing with the residual dent depth of the impact from plastic deformation.

5.2.2.2 Maximum Force Results

Similar to the original grillage, the maximum force value was found through LS-DYNA's post-processing software. The following graph shows the maximum x-force experienced throughout the finite element model.

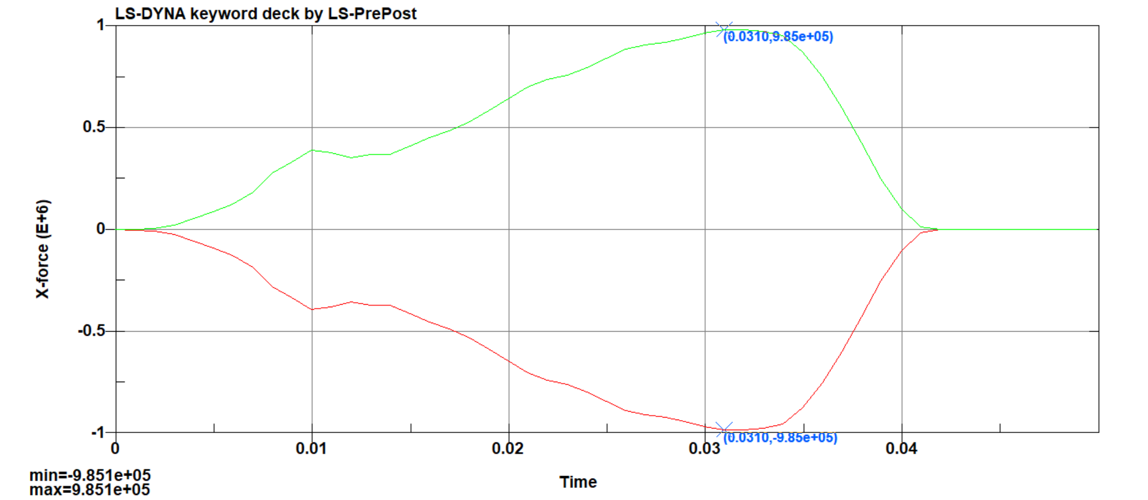


Figure 35 – Copy Ex-HMCS IROQUOIS Panel FEA Maximum Force

Again, each line represents each respective entity involved in the impact; the ice indenter and the grillage, showing an equivalent and opposite force enacting on both geometries.

As can be seen, for both the ice indenter and the grillage, the maximum force peaked at approximately 985,100 Newtons.

5.2.2.3 Von Mises Stress Results

The effective stress of the copy panel was found. The following Figures show the results of this data.

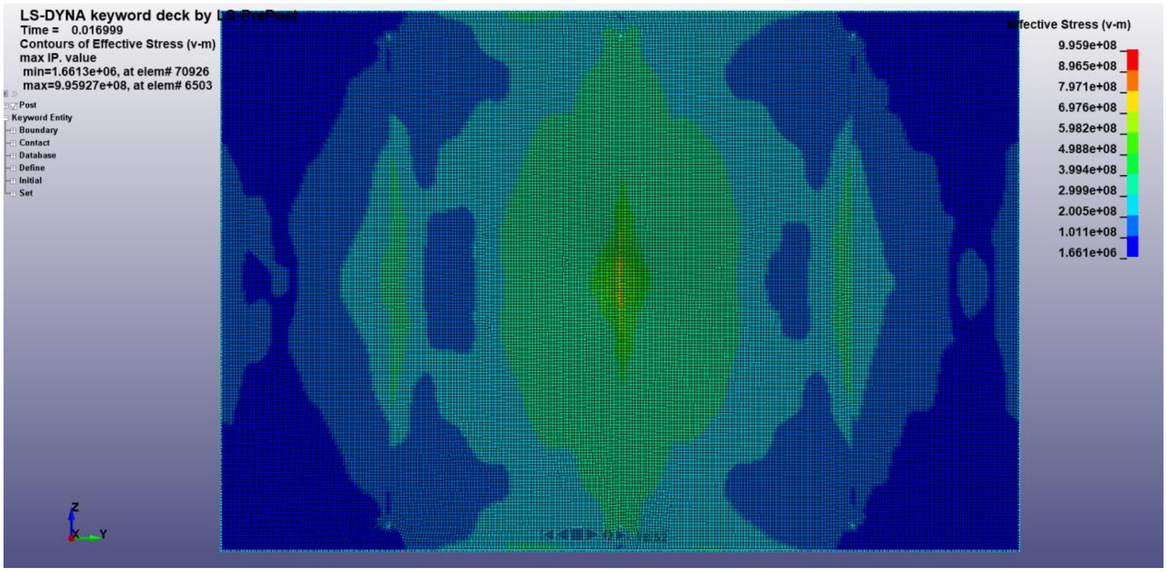


Figure 36 - Copy Ex-HMCS IROQUOIS Panel Von Mises Stress - Front

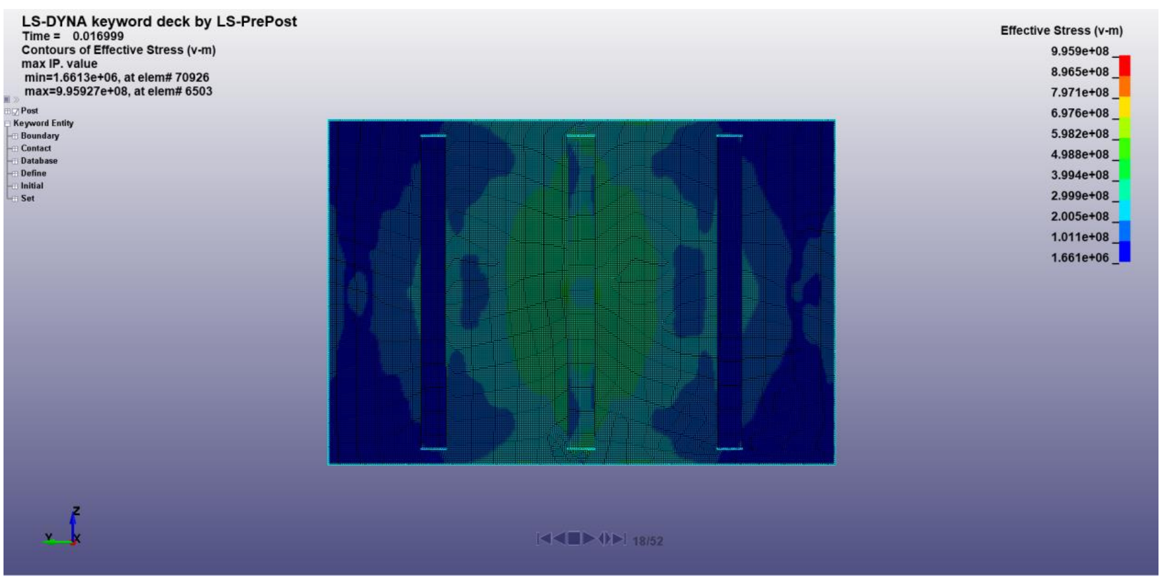


Figure 37 - Copy Ex-HMCS IROQUOIS Panel Von Mises Stress – Back

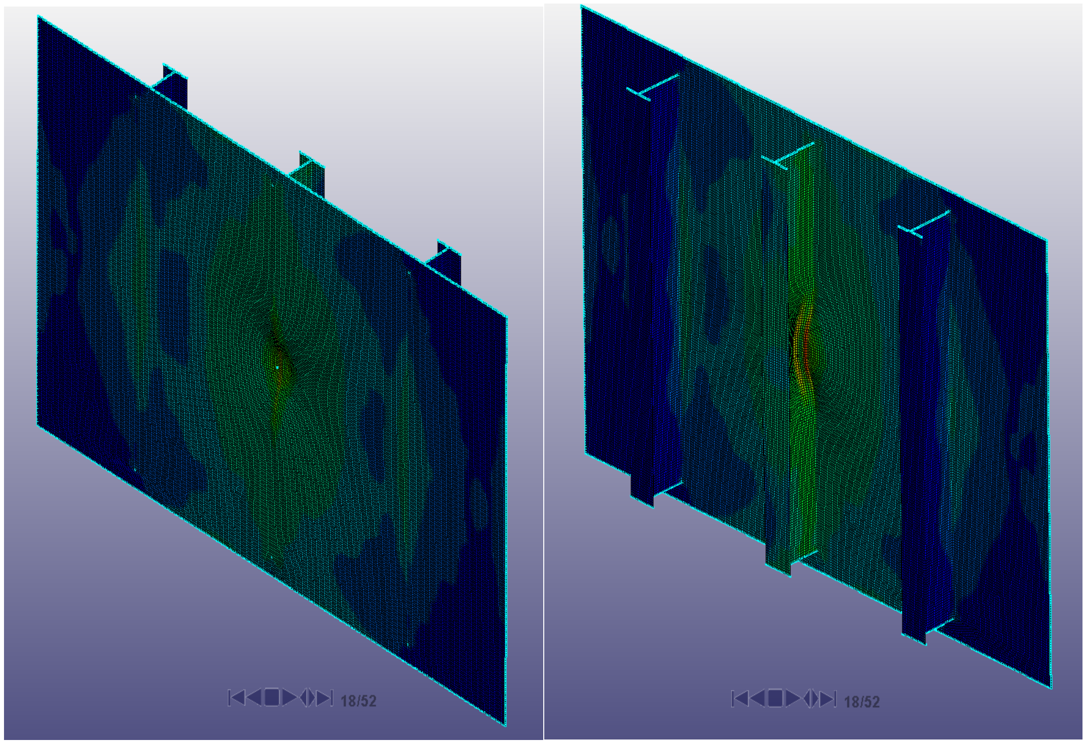


Figure 38 - Copy Ex-HMCS IROQUOIS Panel Von Mises Stress - Isometric Front and Back

The maximum effective stress in the finite element analysis was found to be approximately 995.9 MPa. This value is relatively high, as compared to what would be expected from the respective real-world experimental equivalent. This value was found in the stiffener web, at the connection between the web and plating near the location of the ice impact.

5.3 Data Comparisons

To give a better understanding as to how similar the finite element models were to their respective real-world counterparts, the displacement and force data sets were compared.

5.3.1 Original Grillage Experimental Data vs FEA

The comparable data between the original grillages experiment and FEA are shown in the table below.

Table 18 - Original Ex-HMCS IROQUOIS Grillage Data Comparison

Original Ex-HMCS IROQUOIS Grillage		
Data Measurement	Experiment Data	FEA Data
Max. Displacement (cm)	0.3	6.3
Max. Force (kN)	548	754

As can be seen, the FEA data were found to be much larger than that of the experimental data. The indentation in the panel was noticeably larger, with the finite element model having over ten times the amount of deformation as the real-world experiment. The maximum experienced force was not significantly different, however, the FEA force was still much higher.

5.3.2 Copy Grillage Experimental Data vs FEA

The comparable data between the original grillages experiment and FEA are shown in the table below.

Table 19 - Copy Ex-HMCS IROQUOIS Grillage Data Comparison

Copy Ex-HMCS IROQUOIS Grillage		
Data Measurement	Experiment Data	FEA Data
Max. Displacement (cm)	3.75	7.8
Max. Force (kN)	667	985

Similar to the original grillage, the copy grillage FEA data was much larger than the experimental data. In this case, the finite element model displacement was approximately two times the amount of the experimental grillage. However, the difference between the two was similar to the difference found for the original panel. The maximum experienced force was again higher for the FEA when compared to the experimental data, with the difference being at a similar scale to that of the original panel.

5.3.3 Original Grillage vs Copy Grillage

Both the original grillage and copy grillage had reasonably similar results. Due to the copy grillage being impacted at an angle 10 degrees higher than that of the original grillage impact, it is expected that the data values for both the experimental and FEA data would be noticeably larger. This remained true throughout the results analysis, with both the displacement and the maximum experienced force increasing at similar amounts across both panels when comparing the experimental and FEA data.

5.4 Discussion

In each case, the finite element model impacts involved much higher forces when compared to those of the experimental impacts, which led to a higher amount of deformation in the finite element models. This was mainly due to the differences in the ice cone finite element model when compared to the real-world ice indenter. Having a much stiffer indenter, one which does not break upon impact, will provide higher force values within the finite element analyses. In this case, the fact that the finite element model of the ice indenter did not spall once impacted aided in the increase in the finite element analysis results. This led to the higher plastic deformations found in the plating.

When comparing the differences between the original and copy grillages, the main comparison is the age of the steel and its respective material properties. Through this experiment, it was seen that the old steel could still perform well when compared to similar impacts on new steel. This experiment allowed for a direct comparison between the two types, which as stated previously, is a unique opportunity in the shipbuilding industry.

5.5 Sources of Error

As stated throughout this thesis, there were many sources of error. When considering the comparison of the panels against one another, this was difficult to complete as they were impacted at different speeds, due to the difference in starting impact angles. When considering the comparison of the finite element models to their respective real-world counterparts, the main sources of error included the ice cone model and the simplifications of the finite element models. The ice cone indenter has a higher impact on the results, as its stiffness and lack of spalling added significant changes to the analyses when compared to the real-world experiments. The simplifications in the finite element models also had an impact on the data, however, these impacts were not as significant as the other sources of error. The simplification of the copy panel would also cause a source of error, where simplifications were made for ease of construction and the lack of a way to properly recreate the original panel curvature. Another source of error included certain parts of the data acquisition. This was mainly due to certain data being found through hand measurements; the measuring of the original grillage and the measuring of the grillage plating deformation. A final source of error involves the real-world original grillages. Due to the lack of information, it is unknown if portions of the six ex-HMCS IROQUOIS grillages had been retrofitted with new steel throughout the vessel's operating life. This would cause concerns when considering the comparison of new and aged steel and should be strongly considered when comparing the data. All of these sources of error will affect on the final outcome of the results and should be strongly considered throughout this thesis.

Chapter 6 Conclusions and Future Recommendations

Two experiments were completed, along with their respective numerical analyses, with the intention of having a benchmarked comparison between a real-world ice impact and its respective numerical analysis. Along with this, a direct comparison was completed between aged and new steel, with respect to a vessel's hull. The study yielded positive results, showing that finite element analysis is capable of providing an understanding of real-world impacts. The experiment also provided a better understanding of aged ship steel and its capabilities, even after 40 years of operation. Both sets of grillages performed well under their respective analyses, yielding positive and expected impact and ship collision mechanics.

The original ex-HMCS IROQUOIS grillage experiment showed little deformation due to the ice impact, however, it should be noted that the applied load was less than that of the copy grillage. Its plastic deformation was much less than initially anticipated, where it was originally expected to not handle the high loads of the ice impact. This is even more notable as this was the second impact on the original grillage.

The copy panel experiment showed a higher amount of deformation from the ice impact, which was mainly due to the higher applied load. This was closer to the initial hypotheses, where prior to starting these experiments, it was expected that high amounts of deformations would occur due to the ice impacts.

The original grillage finite element analysis provided positive results. The differences between the FEA results as compared to the experimental results were analogous to one another, with the differences being mainly attributed to the development of the ice cone model, which inevitably led to the original grillage finite element model showing a higher amount of deformation and maximum experienced force.

Similar to the original grillage FEA, the copy grillage also provided positive results. As well, the difference seen between the copy panel experimental results and FEA results were again mainly attributed to the ice cone model.

6.1 Notable Insights

Multiple insights were found throughout the research and analyses completed throughout this thesis. Firstly, the finite element method chosen to analyze these grillages was found to be reasonable, with the differences between the experimental data and FEA data being a positive stepping stone in understanding and performing numerical ice impacts. Along with this, it should be noted that both the original and copy real-world grillages performed much better than initially anticipated in the experimental ice impact, being within reasonable levels of one another.

The original ex-HMCS IROQUOIS grillage chosen was considered the one with the least amount of curvature, as compared to the other five grillages. Therefore, it should be noted that this may have had an effect on the performance of the grillage. However, this did make comparisons to the copy grillage and numerical models much more reasonable.

6.2 Future Work

Firstly, the most important work that must be completed in the future includes the double-pendulum impacts on the other five ex-HMCS IROQUOIS grillages. Once the six impact experiments have been completed, this will provide a much better idea of how the ex-HMCS IROQUOIS vessel hull would react to impacts of this type, as well as, avoiding any outliers that may exist within the data.

A notable difference between the original and copy grillage experiments involves the fact that both experiments were started at different angles. With the difference between the angles being 10 degrees, this means there was a noticeable difference between the impact speeds of the original grillage experiment and the copy grillage experiment. Therefore, all future ex-HMCS IROQUOIS experiments should be completed at the same starting angle. With five grillages left to be experimented on, this problem should be a non-issue for the overall project moving forward as long as this is taken into account.

The final future work that must be strongly considered includes the finite element models. Many simplifications were made to the models as certain parts of the model were difficult to develop. Most notably, the ice cone finite element model was different than a real-world ice cone. This difference was the main contributor to the differences in the experimental and FEA results and should be the main focus for any future finite element work moving forward. Other smaller differences could be made to the model, including using a scanning device to record the exact shape of the panel plating and using that information to develop a more accurate 3-D model, but when considering this method it should be noted that this would significantly increase the computing power. Therefore, priority should be set on the ice model first to ensure that the result values are more similar while minimizing the increase of computational power.

Bibliography

Alam, M. S., Daley, C., Colbourne, B., Hermanski, G., Gagnon, R., Bruneau, S., . . . Quinton, B. (2012). Double Pendulum dynamic impact test set-up for ice-grillage collision. *International Conference and Exhibition on Performance of Ships and Structures in Ice 2012*. ICETECH 2012.

Andrade, S. L., Elruby, A. Y., Hipditch, E., Daley, C. G., & Quinton, B. W. (2021). Full-scale ship-structure ice impact laboratory experiments: experimental apparatus and initial results. *International Coneference on Ships and Offshore Structures 2021*. Hamburg, Germany: International Coneference on Ships and Offshore Structures.

Dirtsc. (2013, May 12). *HMCS Iroquois (DDG 280) at Port of Hamburg near Oevelgönne*. Retrieved from Wikimedia Commons: https://commons.wikimedia.org/wiki/File:HMCS_Iroquois_(DDG_280)_at_Port_of_Hamburg_near_Oevelg%C3%B6nne.jpg

DYNAmore GmbH. (2021, 05 03). *LS-DYNA Support*. Retrieved from https://www.dynasupport.com/

Gagnon, R. (2008). Analysis of data from bergy bit impacts using a novel hull-mounted external Impact Panel. *Cold Regions Science and Technology*, (52): 50-66.

Gagnon, R. E., & Derradii-Aouat, A. (2006). First results of numerical simulations of bergy bit collisions with the CCGS terry fox icebreaker. *Proceedings of the 18th IAHR Internation Symposium on Ice.* Sapporo, Japan.

Gagnon, R., Andrade, S. L., Quinton, B., Daley, C., & Colbourne, B. (2020). Pressure distribution data from large double-pendulum ice impact tests. *Cold Regions Science and Technology, 175*, 103033.

Gagnon, R., Daley, C., & Colbourne, B. (2015). A large double-pendulum device to study load, pressure distribution and structure damage during ice impact tests in the lab. *Proceedings of the 23rd International Conference on Port and Ocean Engineering under Arctic Conditions.* Trondheim, Norway: National Research Council of Canada.

Illinois Tool Works Inc. (2021, 05 10). *Instron Tensile Testing and Tensile Testing Machines*. Retrieved from Instron.com: https://www.instron.us/en-us/our-company/library/test-types/tensile-test

Kim, H., Daley, C., & Kim, H. (2018). Evaluation of large structural grillages subjected to ice loads in experimental and numerical analysis. *Marine Structures, 61*, 467-502.

Liu, Z., & Amdahl, J. (2010). A new formulation of the impact mechanics of ship collisions and its application to a ship–iceberg collision. *Marine Structures, 23 (3)*, 360-384.

Minorsky, V. U. (1958). *AN ANALYSIS OF SHIP COLLISIONS WITH REFERENCE TO PROTECTION OF NUCLEAR POWER PLANTS.*

Paik, J. K., & Pedersen, P. T. (1996). Modelling of the internal mechanics in ship collisions. *Ocean Engineering, 23 (2)*, 107-142.

Quinton, B. (2015). *Experimental and numerical investigation of moving loads on hull structures.* Memorial University of Newfoundland.

Quinton, B. (2019a). FEA Review and History. In *Advanced Numerical Structural Analysis - Course Notes* (pp. 3-7). St. John's, NL, Canada: Memorial University of Newfoundland.

Quinton, B. (2019b). Structural Dynamics in FEA - Part 2. In *Advanced Numerical Structural Analysis - Course Notes* (pp. 1-11). St. John's, NL, Canada: Memorial University of Newfoundland.

Quinton, B. W. (2008). *Progressive damage to a ship's structure due to ice loading.* Memorial University of Newfoundland.

Quinton, B. W., Daley, C. G., & Gagnon, R. E. (2012). Realistic moving ice loads and ship structural response. *Proceedings of the Twenty-second International Offshore and Polar Engineering Conference.* Rhodes, Greece.

Quinton, B. W., Daley, C. G., Gagnon, R. E., & Colbourne, D. (2016). Guidelines for the nonlinear finite element analysis of hull response to moving loads on ships and offshore structures. *Ships and Offshore Structures, 12*, S109-S114.

Ringsberg, J. W., Amdahl, J., Chen, B. Q., Cho, S.-R., Ehlers, S., Hu, Z., . . . Zhang, S. (2018). MARSTRUCT benchmark study on nonlinear FE simulation of an experiment of an indenter impact with a ship side-shell structure. *Marine Structures, 59*, 142-157.

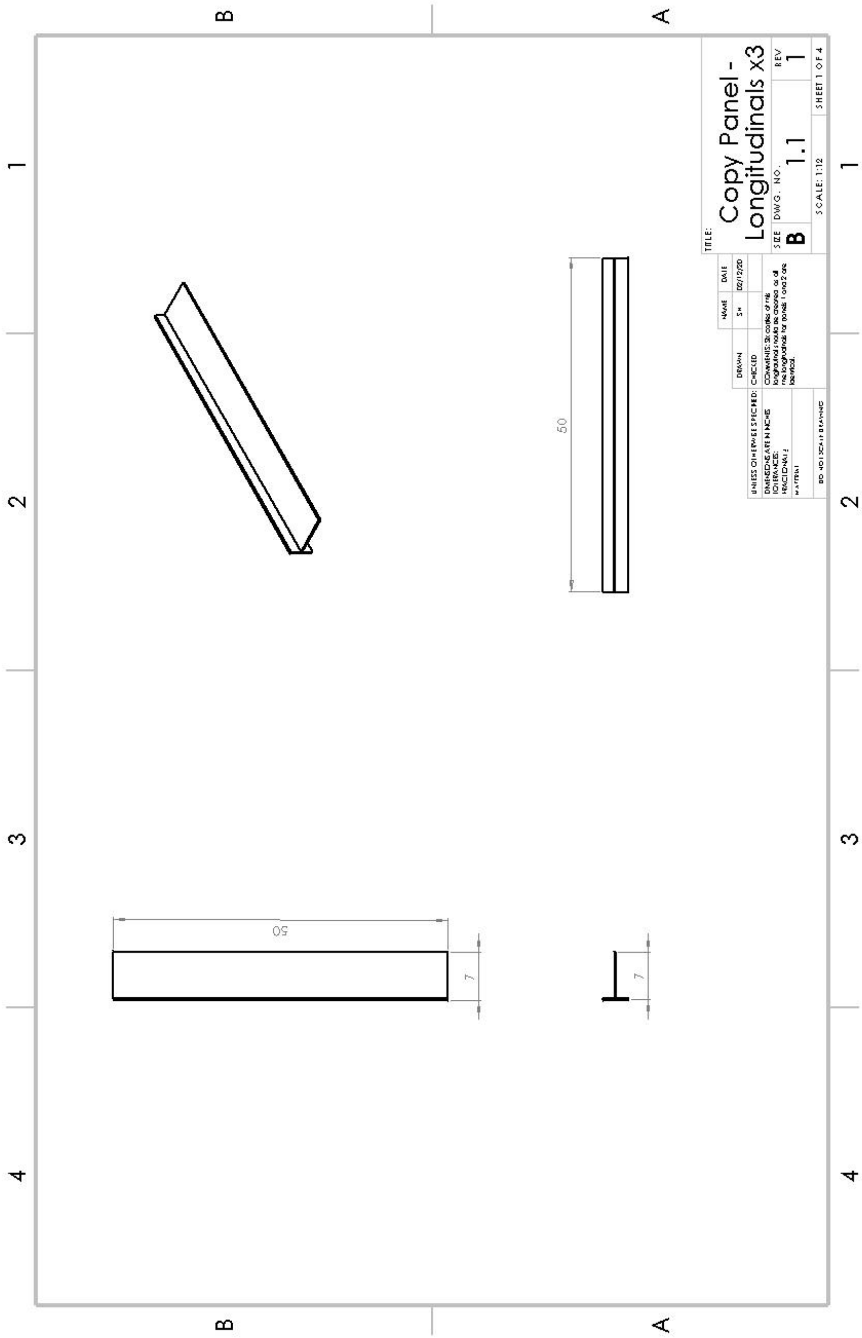
Robbins, M. L. (2020). *Characterization of post-yield behavior of a warship grillage subject to repeated impacts.* Memorial University of Newfoundland.

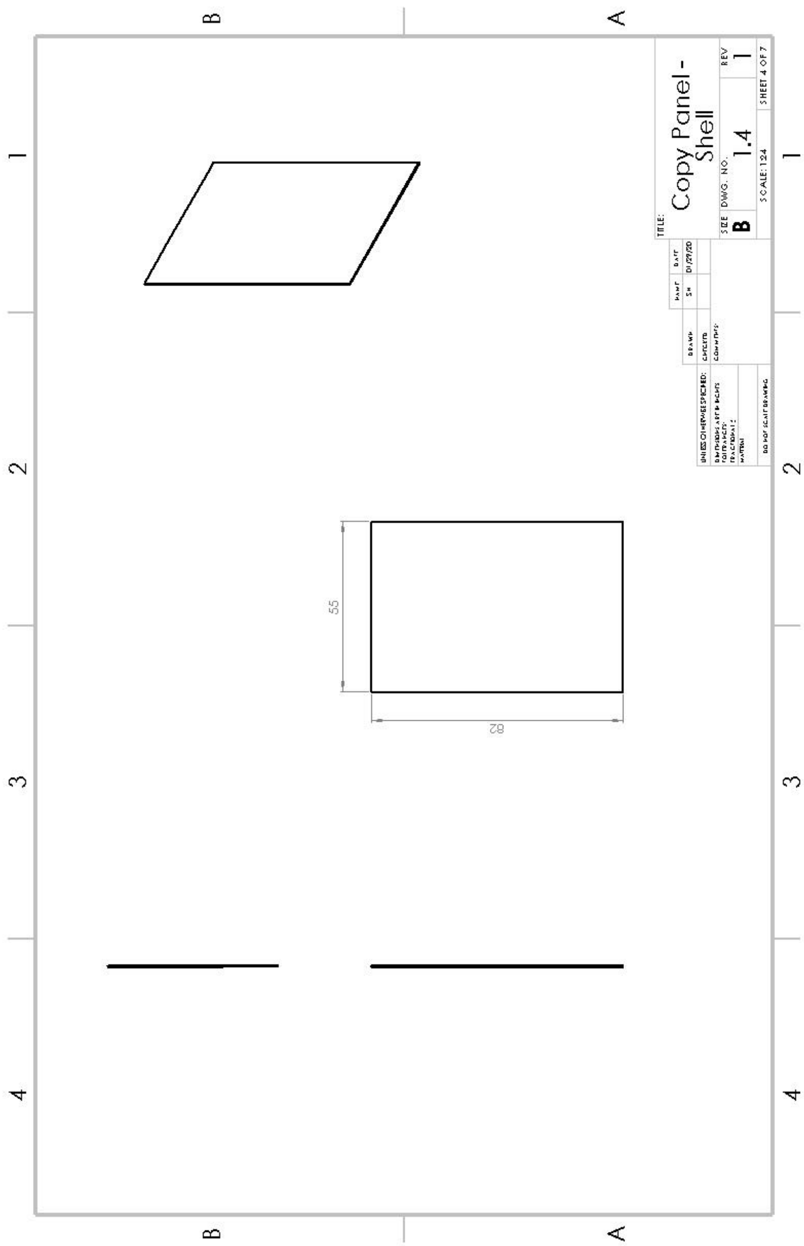
Sopper, R., Gagnon, R., Daley, C., & Colbourne, D. (2015). Measurements of spatial and temporal variations in ice impact pressures. *International Conference on Port and Ocean Engineering.* Trondheim, Norway:: Port and Ocean Engineering under Arctic Conditions.

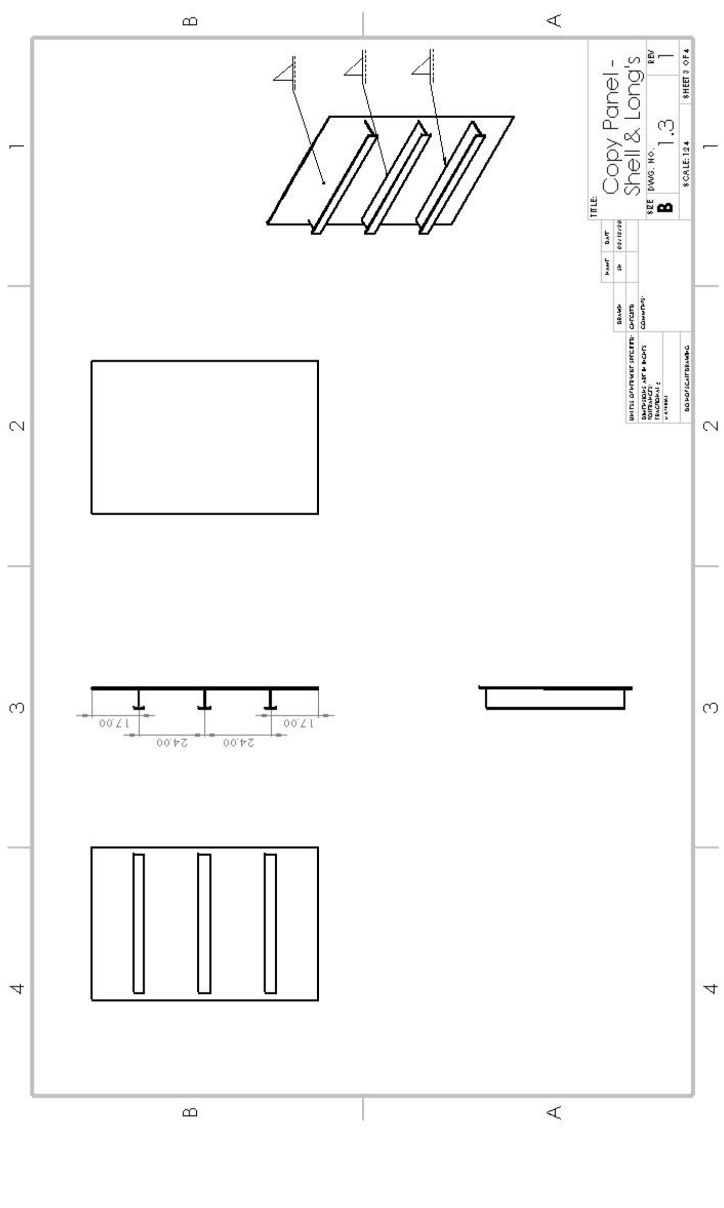
Appendices

Appendix A – **Fabrication Drawings**

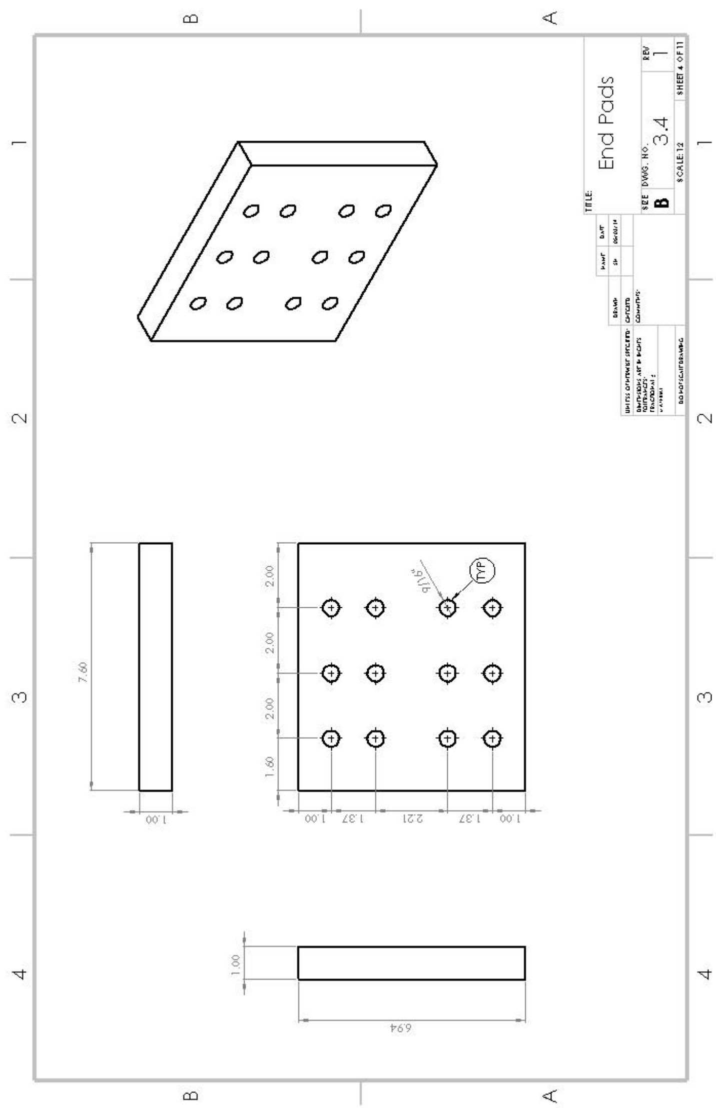
Appendix A1.1 – Copy Panel Fabrication Drawings


TITLE:
Copy Panel - Longitudinals x3
SIZE
B
DWG. NO.
1.1
REV
1
SCALE: 1:12
SHEET 1 OF 4
NAME
DATE
DRAWN
SH
02/12/20
CHECKED
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
DO NOT SCALE DRAWING
50
7


Copy Panel - Shell
TITLE:
SIZE
B
DWG. NO.
1.4
REV
1
SCALE: 1:24
SHEET 4 OF 7
55
82

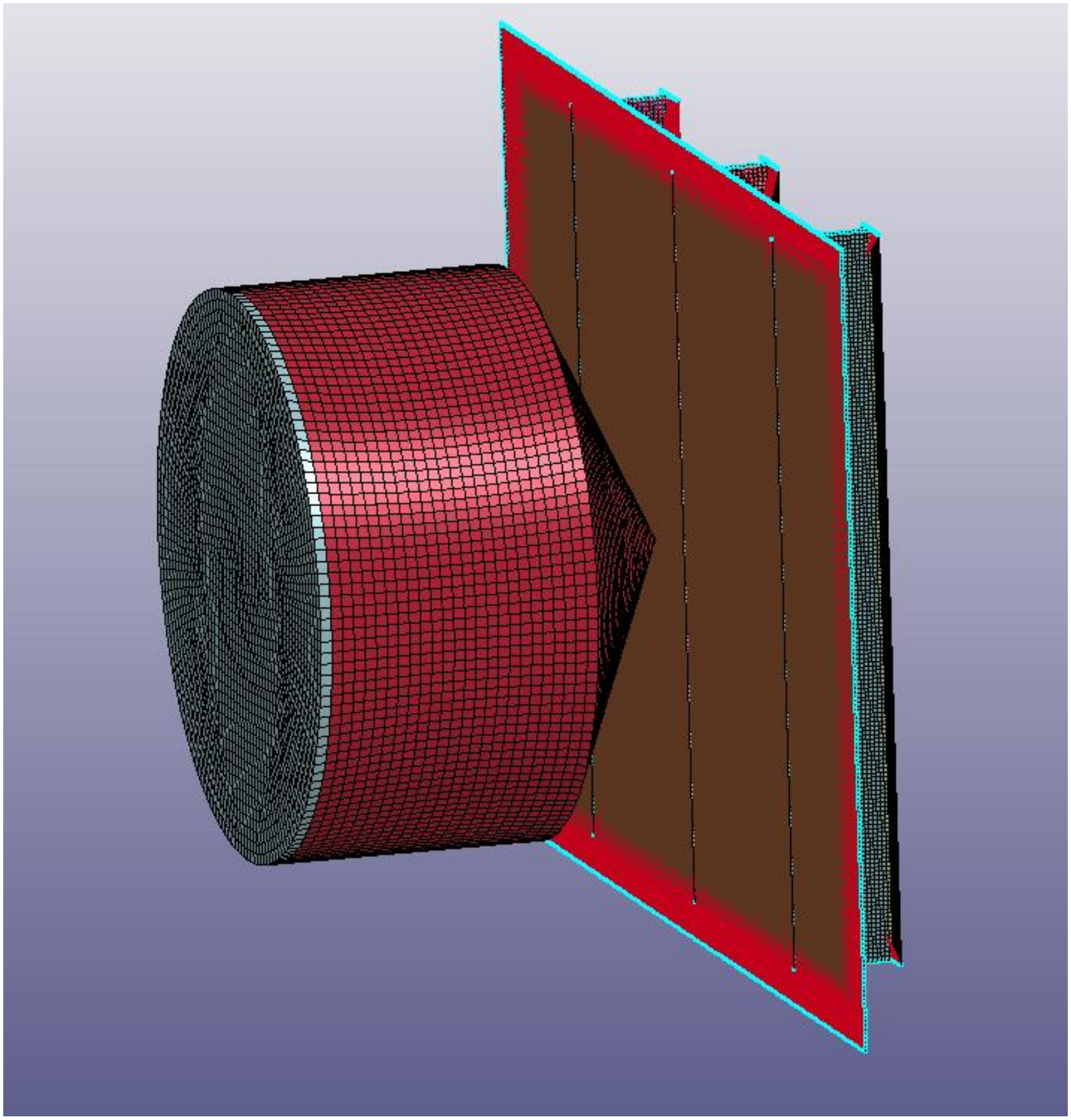

TITLE:
Copy Panel - Shell & Long's
SIZE
B
DWG. NO.
1.3
REV
1
SCALE: 1:24
SHEET 2 OF 4
17.00
24.00
24.00
17.00

Appendix A1.2 – End Pads Fabrication Drawings

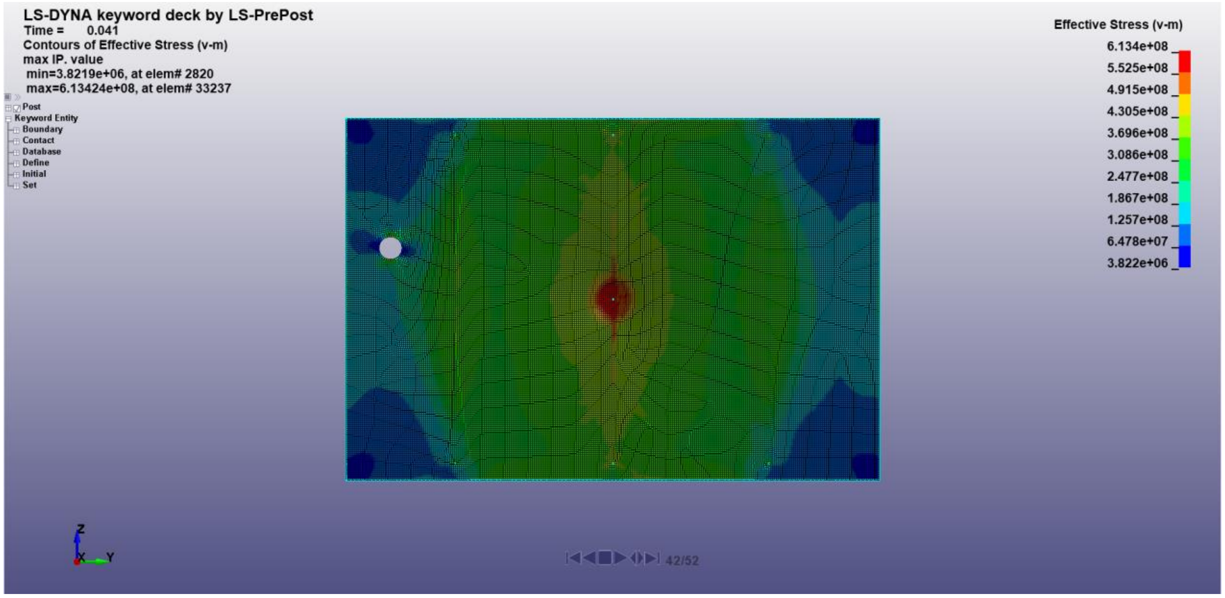

TITLE:
End Pads
SIZE
B
DWG. NO.
3.4
REV
1
SCALE:1:2
SHEET 4 OF 11
TYP
9/16"
7.60
2.00
2.00
2.00
1.60
1.00
1.37
2.21
1.37
1.00
1.00
1.00
6.94

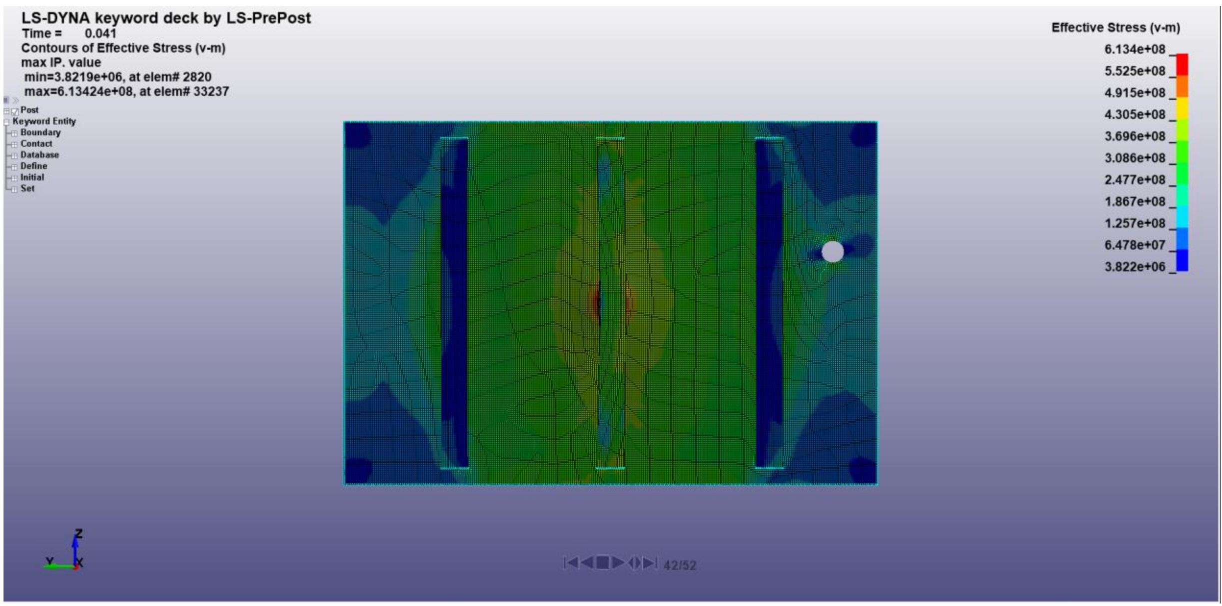
Appendix B **– Original Grillage FEA**

Appendix B1.1 – Geometry Pre-Impact

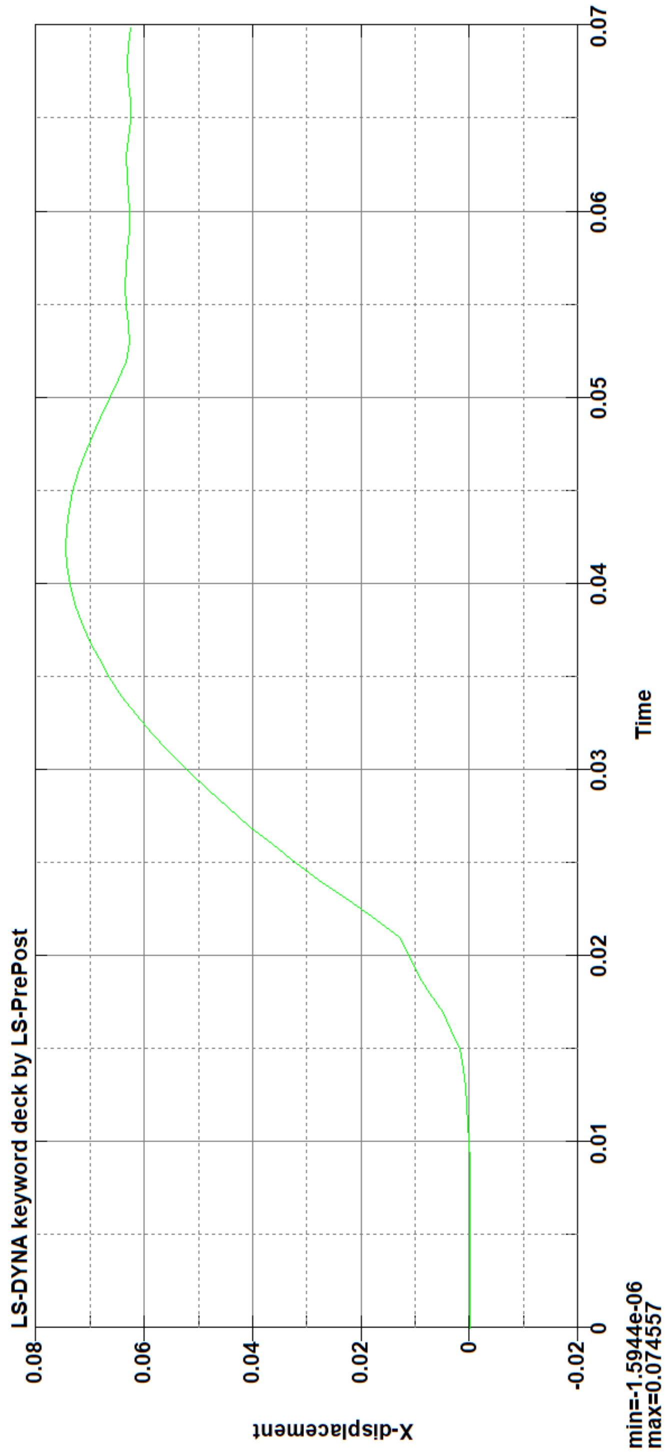


Appendix B1.2 – Von Mises Maximum Stress and Maximum Displacement Geometry Point



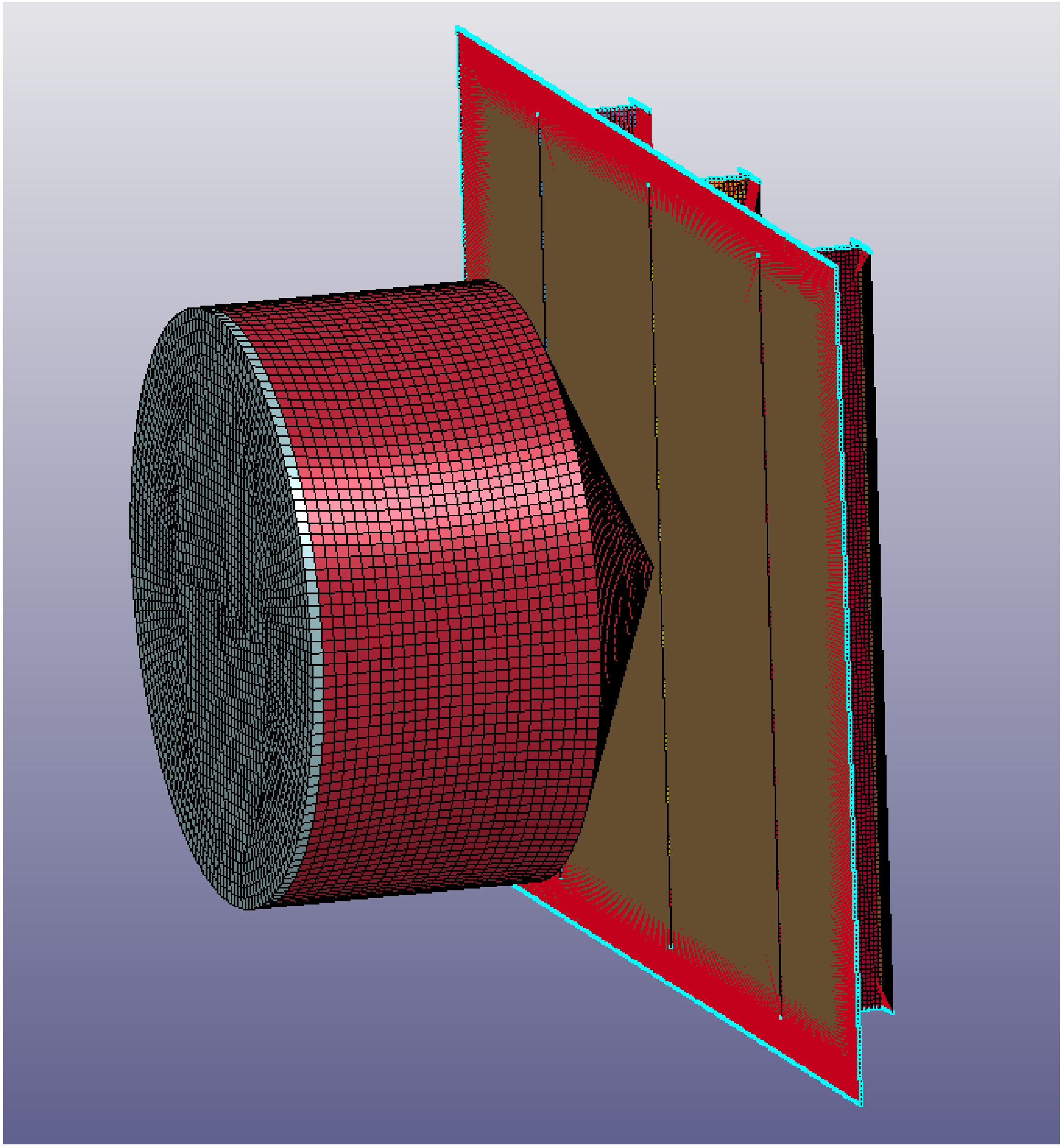


Appendix B1.3 – Dent Depth vs Time Graph

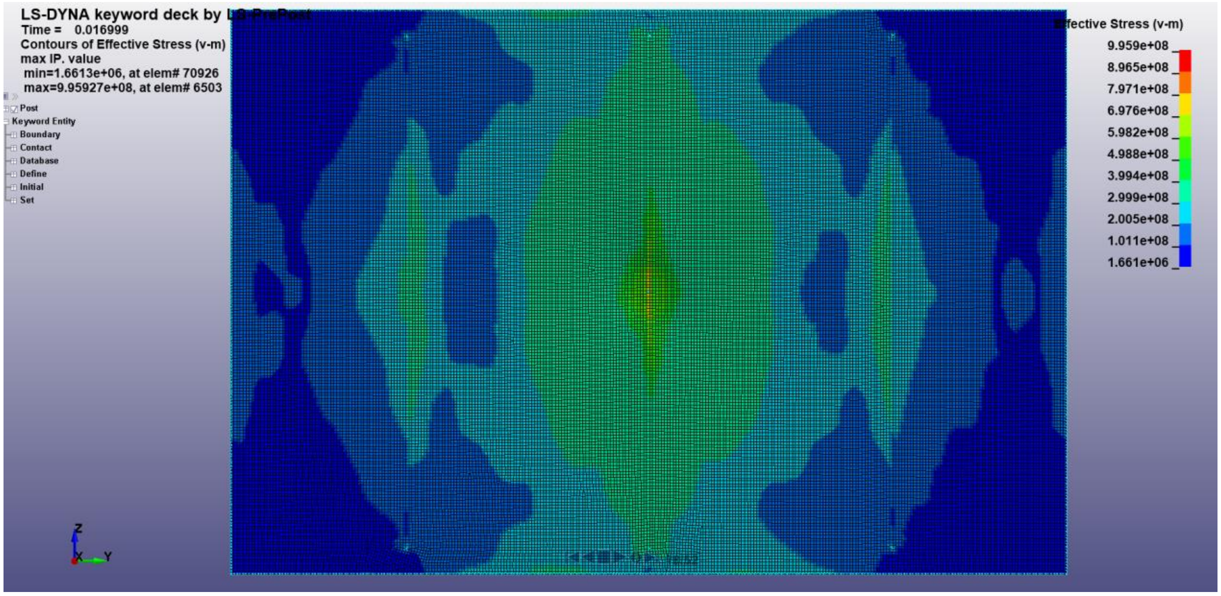


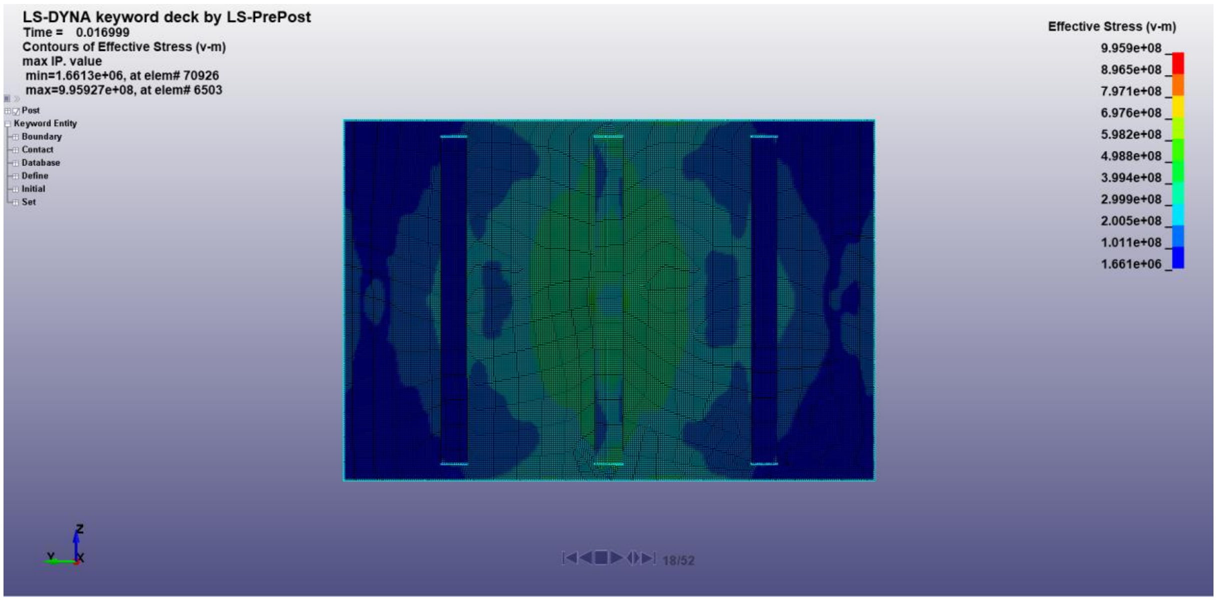
Appendix C **– Copy Grillage FEA**

Appendix C1.1 – Geometry Pre-Impact



Appendix C1.2 – Von Mises Maximum Stress and Maximum Displacement Geometry Point





Appendix C1.3 – Dent Depth vs Time Graph