FRONT-END WHEELCHAIR ATTACHMENT ANALYSIS

By

Karanjit Khatkar Set: B

Artem Gridnev Set: B

A report presented to the

British Columbia Institute of Technology

In partial fulfilment of the requirement for the degree of

Bachelor of Engineering (Mechanical)

Project Sponsor: Dr. Jamie Borisoff, PhD., P.Eng

Faculty Advisor: Dr. Johan Fourie PhD, PEng

Program Head: Dr. Mehrzad Tabatabian PhD, PEng

Burnaby, British Columbia, Canada, 2018

Author's Declaration

We hereby declare that we are the sole authors of this report.

Artem Gridnev

Karanjit Khatkar

We further authorize the British Columbia Institute of Technology to distribute digitally, or printed paper, copies of this report, in total or in part, at the request of other institutions or individuals for the purpose of scholarly activity.

Artem Gridnev

Karanjit Khatkar

Abstract

Wheelchairs are devices used to assist a user in mobility, wheelchairs can be a permanent or a temporary solution depending on the severity of the reasons behind the lack of mobility. Due to the trust in these devices and their responsibility of ensuring the safety of the user, correct testing procedures are required to ensure the wheelchair holds up to standards. In order to ensure the quality and reliability of the wheelchair numerous tests are conducted prior to production to validate the safety of the product. Following the purchase of the base model, users often add front-end attachments to assist in providing smooth motion over different terrains, and often to make the mobility process easier.

The objective of this project is to develop a SolidWorks simulation model which reflects the same geometry as a physical model given by the sponsor, furthermore has the capability to conduct accurate static loading simulations. As well an objective is to conduct a static simulation stress study on the foot-peg of the wheelchair to simulate and determine the stresses that are formed by a 200lb load (mimicking a person) on the wheelchair while the front-end attachment is secured onto the foot-peg. To ensure the results gathered from simulations are correct, physical validation conducted using a strain gauge test and an analytical calculation were used to determine the stresses at a specified point. Using the simulation and two validation methods, an acceptable result for the stress at a specified location on the frame can be verified and analyzed. The purpose being to determine if the SolidWorks simulation model results at the same point can be validated by the physical and analytical.

The majority of the specifications of this project were given to the team by the sponsor. Some of the specifications included are:

- 200lb simulated load
- Strain Gauge Test for physical validation
- Specific boundary conditions for the behavior of the simulation model

The specifications given by the sponsor are further outlined in the request for proposal, which can be referenced in Appendix H. The deliverables for the project include a SolidWorks simulation model, physical strain gauge validation results, the physical set-up which includes the jig, electrical circuit, and wiring. Also, analytical calculations and an analysis of failure on the stress point the tests were conducted.

Acknowledgements

Special thanks to:

Taco Niet BEng (ME), MASc, PEng

For advising and helping out with the physical validation by providing information on strain gauges and testing processes.

Vahid Askari PhD, Peng

For advising with the analysis of geometry and the distribution of forces within the complex model, also validating the assumptions made during the stress analysis of the simulation.

Cyrus Raoufi PhD, Peng

For advising with the simulation model.

Neil Munro BASc, Peng

For advising with the simulation model.

Table of Contents

Author's Declaration II
Abstract III
Acknowledgements V
1. Introduction 1
2. Detailed Description of the Current Status 3
3. Theoretical Background 5
4. Description of the Project Activity and Equipment 6
  4.1 Virtual Environment 6
    4.1.1 Model Configuration 6
    4.1.2 Model Simplification 6
    4.1.3 Model Meshing 6
    4.1.4 Model Boundary Conditions 8
    4.1.5 Simulation 8
  4.2 Physical Environment 10
    4.2.1 Physical Strain Gauge Calculations 10
    4.2.2 Strain Gauge Application 10
    4.2.3 Testing Set-Up Summary 11
    4.2.4 Data Collection 12
    4.2.5 Jig Manufacturing 13
  4.3 Risk Plan 16
5. Discussion of Results 17
  5.1 Calculation of stresses – Initial validation 17
    5.1.1 Results 17
    5.1.2 Analysis 18
    5.1.3 Conclusion 18
  5.2 Physical Model: Strain Gauge Testing Results and Analysis 19
    5.2.1 Results 21
    5.2.2 Analysis 23
    5.2.3 Conclusion 23
  5.3 Virtual Model Simulation Initial Results and Analysis 24
    5.3.1 Phase 1 24

5.3.2 Phase 2 27
5.3.3 Phase 3: Final Results and Analysis 30
5.4 Analysis of Failure 37
6. Conclusion 40
Bibliography ..... **Error! Bookmark not defined.**
APPENDICES 42
APPENDIX A: Iterative Solver Stopped 44
APPENDIX B: Soft Spring Stabilization Settings 45
APPENDIX C: Thickness undefined 46
APPENDIX D: Material Properties undefined 47
APPENDIX E: 219 Interferences 48
APPENDIX F: Interference Error 49
APPENDIX G: Design Review Package 50
APPENDIX H: Request for Proposal for Wheelchair Front End Attachment Stress Analysis 61
APPENDIX I: Responsibility Assignment Matrix 64
APPENDIX J: GANTT CHART 65
APPENDIX K: Initial Electrical Wiring Diagram for Strain Gauge 66
APPENDIX L: Initial Strain Gauge Measurement To Force / Stress Conversion 67
APPENDIX M: Initial Definition of Variables 68
APPENDIX N: Testing Code 69
APPENDIX O: Work Breakdown Structure 70
APPENDIX P: Further (Iterative) Results from Strain Gauges for Confirmation 71

Table of Figures

Figure 1.1: FreeWheel Front-End Attachment 1

Figure 4 . 1: Section of Electrical Circuit 11
Figure 4 . 2: Front-End Manufactured Attachment 13
Figure 4 . 3: Roller Bearing on Front-End Attachment 14
Figure 4 . 4: Rear Wheel Rig 15

Figure 5 . 1: Iterative Solver Stopped 25
Figure 5 . 2: Strain Results retrieved through piece-by-piece approach 28
Figure 5 . 3: Foot-peg Simulation Set-up 30
Figure 5 . 4: Simulation Set-up for Entire Model 31
Figure 5 . 5: Results of Foot-peg Simulation 33
Figure 5 . 6: FBD of wheel chair 17
Figure 5 . 7: Section Cut of Frame 17
Figure 5 . 8: Test set up 19
Figure 5 . 9: Side view of the set up 20
Figure 5 . 10: Initial - 0 load reference run 21
Figure 5 . 11: Stress Results from Strain Gauge vs. Time 22

1. Introduction

Wheelchairs are used worldwide for people who do not have the ability to walk, or it is difficult to do so due to illness, injury, disability or age. Wheelchairs come in multiple different forms and can be customized to specifications. Specifications can include seating adaptions, specific controls, specific actions, or it may be customized by using an attachment. The first recorded use of wheelchairs by a disabled person is dated back to the 17th century; during this time they were also used by the wealthy as a means of transportation. It wasn't until the later 18th century where these devices were used specifically for surgical and medical procedures and conditions [3].

The present wheelchairs are made of many different alloys, and use different structures, the most common structure and material being tubular steel. This provides stability and strength for the regular use of wheelchairs. There are many terrains that are difficult to conquer on a wheelchair, therefore the use of assistive devices is necessary to restore the ease of mobility. One group of these assistive devices are pre-installed onto your wheelchair and do not need to be added upon your required use. The other group of assistive devices are attachments, which can include motor-attachments, or specifically designed wheel attachments such as one seen below:

Figure 1.1: FreeWheel Front-End Attachment [1]

The problem occurs when these external attachments are added to wheelchair frames attaching to parts not designed for the added forces and stresses on the frame. These stresses can cause frame failures and result in injuries occurring to the user, so to prevent these failures, accurate testing and validation of the stress resulting from attached accessories needs to be done. For these purposes, the objectives of our project are to generate an accurate testing model in a 3D environment for the wheelchair frame provided to the team by the project sponsor, also to run a static simulation on the model according to the request for proposal, as seen in Appendix H. To confirm the precision of the results from this simulation we are to validate the numbers physically through a strain gauge test and analytical calculations.

2. Detailed Description of the Current Status

Currently, our sponsor Dr. Jamie Borisoff PhD., P.Eng., who is the Canada Research Chair in "Rehabilitation Engineering Design" and head of operations at the REDLab requires a 3D model of the wheelchair he provided the team to be set-up for simulation testing in SolidWorks. The model should be accurate enough to run different types of simulations and produce results that reflect the wheelchair frames behavior in real life. The purpose for having such an accurate model is to test different front-end attachments on the wheelchair and identify what types of stresses or failures may occur in the frame due to these attachments. The objective of this project is to determine stresses at a specific point in the frame caused by the provided front-end attachment followed by a physical validation of these results using analytical calculations and a strain gauge test. A specific location on the foot-peg was chosen to ensure results indicate comparable values.

The initial SolidWorks model provided to the project team by the sponsor had issues with material properties, correct constraints, part interferences, and more. The team was tasked with re-modeling of the initial 3-d model described above into one that matched the properties and geometries of the physical one given by the sponsor. Once the re-modelling was complete, a static simulation reflecting the weight of a 200lb person and an addition of a front-end attachment device was to be tested in virtual environment to analyze the resulting stress and strain on to the frame. These simulation results will need to be further validated for precision and accuracy. The purpose of validation is to ensure the resulting stress and strain in the frame reflect the same behavior if these tests were to be reproduced physically on the provided wheelchair frame. In other words, the SolidWorks model should be able to accurately enough reflect how a real-life model would react and behave when certain conditions and forces are applied to it. If the two results from the physical and virtual testing match up, the virtual model can be used for further testing and would accurately and precisely reflect the physical frames behavior.

After gathering and comparing results from the physical strain gauge validation and the analytical calculations, it was seen they were almost identical. Using the value of 8045psi as a reference to the stress at the point the strain gauge was applied, numerous simulation studies were conducted on the model. Each of these simulations included further defining the model to retrieve results similar to the physical and analytical results. After researching into the nature of the equation used for the von Mises stress and principal stresses in SolidWorks it was deemed the results were not directly comparable between a complex model to the analytical calculations or uni-directional physical strain gauge results. Although, when the model was simplified to focus on the foot-peg with a calculated reaction force at the end of the front-end attachment the results matched the physical and analytical within 15%.

Finally, it was determined that with a 200lb force applied on the frame, which mimicked a person of such weight sitting, using physical validation via strain gauges, an analytical calculation, and a simplified SolidWorks simulation, the stresses on the foot-peg from the front-end attachment will not cause static failure on the foot-peg.

3. Theoretical Background

An assumption was made that the wheelchair frame would have minimal flex at all other points except the foot-peg, to which the front-end attachment is secured. To minimize interference of the stress measurements, a rig was designed out of rigid materials that would not deflect under the testing conditions. Therefore, an analytical solution was derived assuming the wheelchair frame and the rig as rigid components with no deflection.

The stress data gathered focused around the area of the foot-peg insertion into the main frame. This area was assumed to experience highest stresses from any attachments to the foot-peg due to the bending moment it would experience. The insertion point applies a clamping force on the foot-peg to secure it and creates an area that produces a reaction moment and forces from any forces applied to the front. The strain gauge was placed near the area to extract stress information close to the assumed point of highest stress.

The main support wheels in the back of the wheelchair are designed to allow the frame to settle naturally under weight by providing a flex by allowing a change in camber. This flex needed to be represented in the simulation model and physical tests. The jigs for physical testing were outfitted with transfer bearing whilst in the simulation instead of using the circular wheels that reflected the initial model, simplified extrusions were designed to provide identical behaviour with the aid of contact sets and connections within the simulation environment.

All other theoretical background used to make decisions during results and duration of this project are discussed in their respective sections within the report.

4. Description of the Project Activity and Equipment

4.1 Virtual Environment

4.1.1 Model Configuration

The SolidWorks model given by the sponsor was not configured to match the size of the physical model given to be tested. Therefore, a new configuration had to be created and any differences in measurements and sizes needed to be reflected in the new configuration. Measurements were taken of the physical model and the SolidWorks model was configured to match.

4.1.2 Model Simplification

From the analysis of the model's components the team found that numerous bolts, washers and other fasteners were causing a high number of interferences, which is referenced in Appendix E. It was possible to suppress these components completely from the simulation environment as they were going to be reflected by the models contact sets. Contact sets represent how certain components assemble and allow the user to set a relationship between components. Following the suppression of all the fasteners, the total number of interferences in the model decreased significantly.

To accurately reflect camber changes of the rear wheels due to the frame flexing in the simulation environment the existing rigid wheels needed to be replaced. The designing of these parts (the jig) needed to be identical in both the physical and simulation environment to ensure the results of the testing would remain comparable and not be skewed due to material or geometry properties.

4.1.3 Model Meshing

Before conducting a static simulation in SolidWorks, the model is required to be meshed. This creates pre-determined nodes of all the components that are to be analyzed by the software after. The complexity of the components in the assembly dictates the required mesh size, in the case of high complexity smaller mesh size is required, and for low complexity larger mesh sizes can be used. The configured wheelchair required very fine mesh due to the complexity of the connection handled and tube designs used. After an iterative process of finding appropriate mesh size it was concluded that 2mm would be the element size for mesh. If we used the suggested mesh size determined by the following equation the simulation would constantly crash:

$$Mesh = \frac{thickness}{3}$$

The crash is due to the mesh required with the use of this equation is 0.6mm which creates over 50million nodes in the model, and our system was incapable of handling such precision with the computing power that was provided.

4.1.4 Model Boundary Conditions

To simulate the physical conditions in SolidWorks correctly, boundary conditions needed to be applied that included the connection of the components, loads and fixtures. The conditions applied were gravity, a force to represent a 200lb load, one fixed fixture and two roller fixtures. The fixed fixture represented the contact between the front-end attachment and the ground, meanwhile the two roller fixtures were applied to the main (back) wheels of the wheelchair allowing for free translation.

The boundary conditions proved to be difficult in terms of the connections within the simulation environment. The SolidWorks environment did not translate the constraints applied in the main model, it rather used self-detection methods to recognize components in contact. Further SolidWorks sets a global contact for the model which includes all these contacts. The issues arose because if there were any gaps or spaces between components, they would be considered as "free-floaters" or not attached by the simulation, and result in errors during the testing. The contact sets of this model proved to be a challenge during the course of this project, which is further discussed in the virtual simulation of section of the report.

4.1.5 Simulation

SolidWorks Simulation is a software that has an industry reputation for being accurate and precise in the results it provides. One of the reasons it can be so accurate is due to the requirements within the software. It requires the user to input boundary conditions, forces, and fixtures that accurately represent the physical environment, which can sometimes prove to be difficult due to the complexity and geometry of certain parts. The simulation software requires the model to be "clean" prior to entry into the simulation environment, "clean" meaning free of any spaces or gaps or interferences, because if they exist in the model the software will either crash and not conduct any tests, or the results will not be accurate.

SolidWorks Simulation software is used to virtually represent the stress and strain effects of loading a wheelchair of identical geometry in real life. This allows experimenting with the virtual model with specific loading or excessive forces and gives you the ability to push testing to its absolute limit, whereas doing this physically would have consequences to the physical model such as failure, fatigue, testing rig error and multiple tests would be found costly.

4.2 Physical Environment

4.2.1 Physical Strain Gauge Calculations

$$\epsilon = \frac{\sigma}{E}$$

$$\therefore \sigma = E\epsilon$$

4.2.2 Strain Gauge Application

A strain gauge test is a complex yet accurate method of strain data collection at the applied spot. Although accurate results are produced by strain gauges, the procedure of application must be done with care to make sure a good level of accuracy is achieved. The surface on the wheelchair where the strain gauges were to be installed required to be cleaned, sanded free of any finish, and buffed to a perfectly smooth surface prior to gauge application. Following the preparation for application of the strain gauge a specific procedure needed to be followed to correctly apply the gauge without causing any damage that could affect the sensitivity.

4.2.3 Testing Set-Up Summary

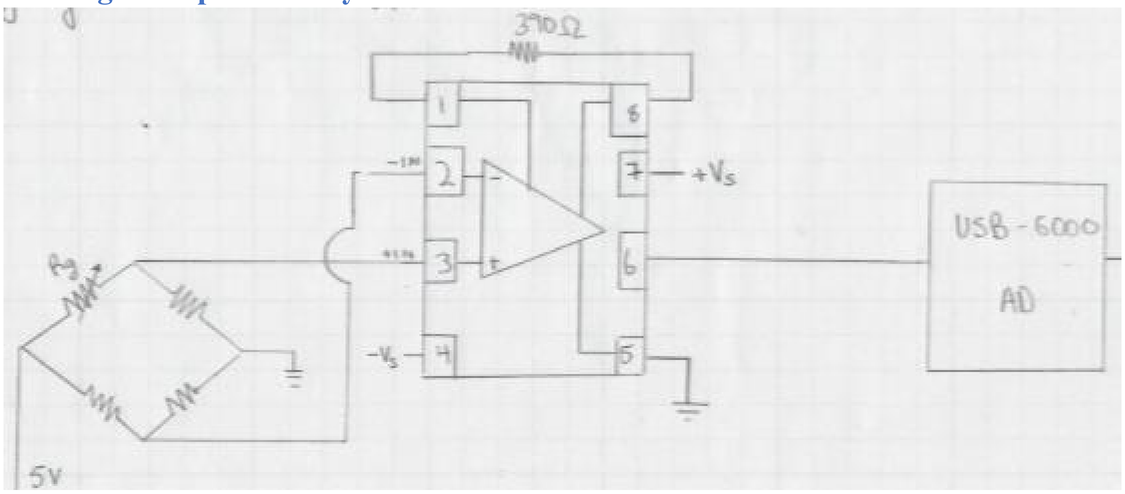


Figure 4 . 1: Section of Electrical Circuit

Figure 4.1 above, is a section of the electrical circuit which was implemented to extract the strain data from the test. This circuit uses a quarter-bridge strain gauge to measure the changes of compressive and tensile strain. A quarter-bridge was chosen due to the sensitivity of the circuit, if the sensitivity of the circuit to load changes is required to be higher, such as when using dynamic loading, you would choose a more adaptive circuit such a half or a full-bridge.

The test will be conducted by placing a static 200lb weight on the two main frame bars that would normally hold the seating for the user. The weight will be added and untouched through the length of the test.

To conduct a successful test, a calibration of the system had to be performed. The use of zeroing methods should allow the system to be set to the correct equilibrium prior to conducting a loaded the test. This was done by running the circuit multiple times with no load on the frame and making the necessary adjustments to the code.

4.2.4 Data Collection

In order to successfully collect data and be able to convert it into applicable information MatLAB was required, as well as the equipment connecting the circuit to the computer. Using the practices from prior experiences with strain gauges, it was decided a NI-USB6000 would be used to collect data from the circuit and add it into MatLAB which then converts the data into the required information (stress in psi for this experiment).

Due to the voltage required in the circuit an OP-AMP was used as an amplification device. The circuitry used can be fully referenced in Appendix K, the electrical circuit generated for the strain gauge test being conducted was referenced from a previous, successful strain gauge uses. To successfully conduct a strain gauge test, the strain gauge needed to be placed in a quarter bridge circuit. This was determined because the type of Wheat-Stone bridge applied determines the sensitivity of the circuit, and under these circumstances the test is static therefore there is no need for higher change sensitivity in the measurements.

The plan to collect data was decided to be for a fairly short period, as the load does not change following the initial application. This was done just long enough to see if the circuit is behaving in a constant manner and to make sure there is no noise or flux error within the circuit. The sampling rate can be relatively small because the load applied is static, and the data can be plotted after full collection rather than a live plot.

4.2.5 Jig Manufacturing

A stiff jig was manufactured to obtain accurate strain gauge results of the foot support reaction on the frame. If actual wheelchair wheels where to be used with the actual front-end attachment, it would prove to be difficult to represent in the FEA simulation. The jig directly represents the SoildWorks parts designed for the FEA model and the type as well as the thickness of the steel allows for a rigid representation in FEA environment as the assumption is made that the aluminum frame will flex before steel jig would.

4.2.5.1 Front – End Attachment

Figure 4 . 2: Front-End Manufactured Attachment

Figure 4.2 above is the front-end attachment which was manufacturing for testing purposes. The design takes into consideration the important distances that would result in a moment on the frame. The current front-end attachment it very costly and the design is not viable for testing due to the spokes and tire flex that result from an applied load. This design above has a transfer bearing at the front to allow for the correct flex behaviour of the frame as load is applied. The jigs attachment mechanism mimics that of the front-end that is being tested: pin style clamp on the front most foot-peg bar while supporting L-shaped bracket resting on the back loop. The transfer bearing can be seen below in Figure 4.3:

Figure 4 . 3: Roller Bearing on Front-End Attachment

4.2.5.2 Rear Wheels

The jig replacing the rear wheels was designed and manufactured from 2”x 2”x 0.125” square tubing. This was deemed not only stiff enough to be considered rigid under the applied load but also readily available in the supply room. A pair of ½” bolts were modified to act as axles with wingnuts for quick release and adjustment. ½”- ID tubes were welded through the tubing at the top of the triangles for axial bolt support. A flat base plate with transfer bearings used on the bottom to allow for motion while keeping the frame stable. The rig can be observed in Figure 4.4 below.

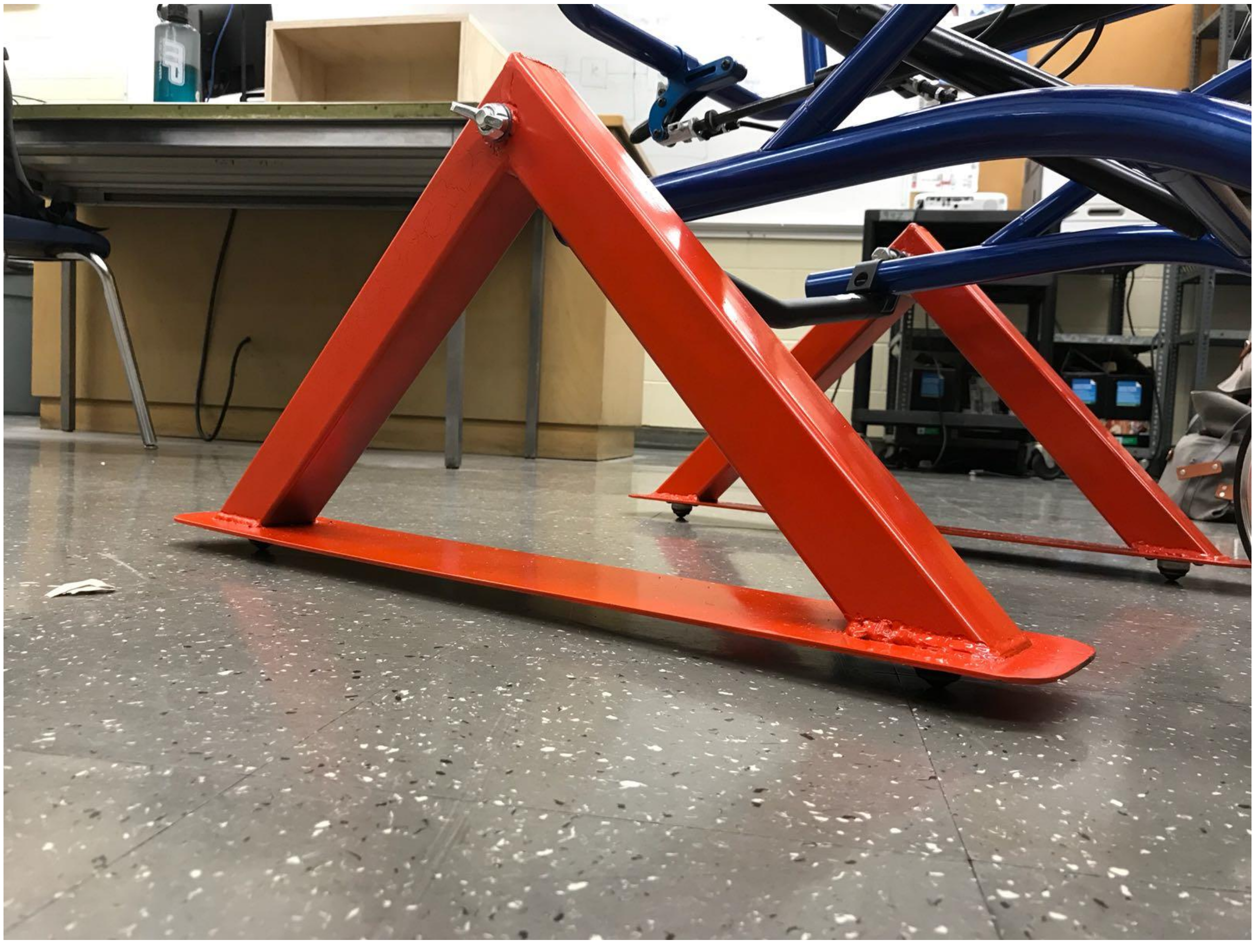


Figure 4 . 4: Rear Wheel Rig

4.3 Risk Plan

The purpose of the risk plan is to predict any possible risks or issue that may occur throughout the duration of this project, as well as during the product use later. Along with the possible risks, possible causes, probability and severity rating, risk factor and actions to minimize risks are included in the risk plan. The risks are coordinated into specific areas of focus such as the design, assessment, and management phases of the project. The risk factor indicates the importance of the risk and the danger they bring to the project. Risks with higher risk factor should be addressed predominantly.

	Risk	Cause	Probability (1-5)	Severity (1-5)	Risk Factor (P*S)	Action to Minimize Risk
SolidWorks Simulation Risks	Meshing will fail	Complex parts, material, shell thickness	3	2	6	Exclude Shells if possible, simplify parts, and select materials
	Interferences Detected	Cosmetic Parts, too many unnecessary components	3	3	9	Simplify the model, suppress cosmetics
	Failure due to boundary conditions	Poorly applied fixtures, inadequate loading	2	2	4	Adjust fixtures according to correct geometry, apply adequate loading
	Contact Set Failure	Design intent used for components lacks correct practices	3	4	12	Delete global contact and assign contact sets manually component by component
Physical Testing Risks	Failure of Strain Gauges	Inadequate application procedure	3	2	6	Follow professional application manual
	Failure of Data Collection	Incorrect code, incorrect circuitry	2	3	6	Calibrate code, and have expert check both code and circuit
	Fried Circuitry	Incorrect electrical circuit design, poor components	1	5	5	Ensure voltage across each part of the circuit as it should be

5. Discussion of Results

5.1 Calculation of stresses – Initial validation

To validate initial results from the simulation environment a calculation of stresses on the foot peg were done analytically. A 200lbs force was used on the frame at the centre of the seat. The frame was excluded from this analysis and considered rigid. SolidWorks FEA simulation calculates von Mises stress, while the hand calculations were simplified to find only bending stress in one direction. An assumption was made that a significant majority of the stresses caused by bending around the Z axis (in to the page) and therefore it was the only sigma considered.

5.1.1 Results

$$\sum F = 0 => (-200lbs)Cos(6.56°) + Fr1 + Fr2$$

$$\sum M_B = 0 => (-200lbs)Cos(6.56°)(25.98") - Fr1(30.82")$$

$$\boldsymbol{Fr1 = 167.488lbs, Fr2 = 31.202lbs}$$

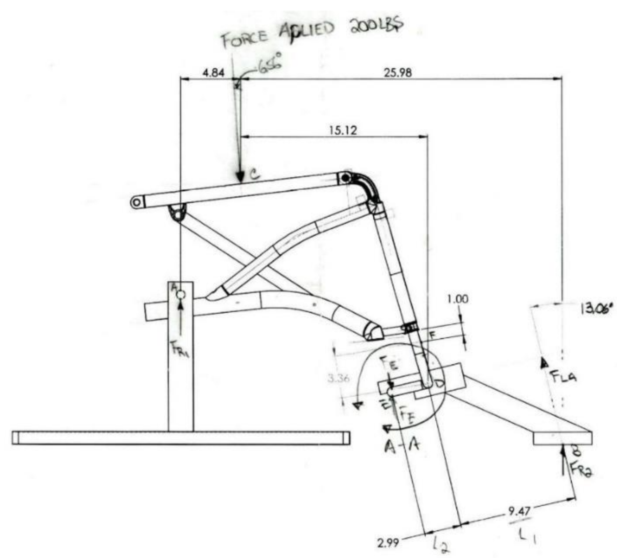


Figure 5 . 1: FBD of wheel chair

$$M_D = Fr2Cos(13.06°) * (9.47")$$

$$\boldsymbol{M_D = 287.84lbs \cdot in}$$

$$\sigma_{max} = \frac{M_F C}{I_Z} = \frac{287.84(lbs \cdot in) * 0.375"}{0.0066in^4} = 16207psi\ (both\ posts)$$

$$\boldsymbol{\sigma_{BEND}} = \frac{16207.207psi}{2posts} = \boldsymbol{8103.6psi\ each\ post}$$

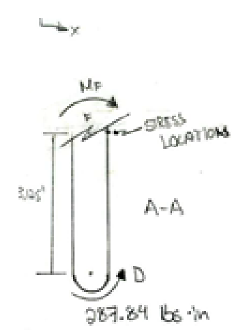


Figure 5 . 2: Section Cut of Frame

5.1.2 Analysis

The resulting bending stress on each vertical post area from the simplified calculation above yielded 8103.6psi. A further simplification assumption was made that this stress is the main benefactor and other stresses were ignored. Using above assumptions, the numbers are as follows:

$$\sigma_1 = 8103.6psi \qquad \sigma_2 = 0 \qquad \therefore Von\ Mises\ Stress = \frac{8103.6 - 0}{2} = 4051.8psi$$

This calculation is based on full rigidity of the frame and exact placement of the force. There were no assumptions made that the physical or SolidWorks simulation results will be exact due to several factors discussed in each section.

5.1.3 Conclusion

Above calculations were thoroughly checked and assumed to be adequate. The results are a good representation of the posts reacting to the 200lbs load applied causing the bending stress along the Z-axis. From this point on we moved onto the strain gauge calculations to further validate these results.

5.2 Physical Model: Strain Gauge Testing Results and Analysis

To validate the virtual model and hand calculations, a physical strain gauge test was set up as seen in the images below. Micro-Measurements unidirectional MMF006836 strain gauge was used along with a 7805c voltage regulator, an 8-pin Op-amp, and120Ω and 390Ω resistors with 0.10% error. The strain gauge used is very sensitive therefore the circuit was built from good quality components to lessen the noise error in the results. The weights were placed 5.5" from the ends of the seat bars, which is 0.6" closer to the front than that of the hand calculations. This was done to counteract the change in the angle due to the gas shocks compressing under the weight on the real model.

Figure 5 . 3: Test set up

Figure 5 . 4: Side view of the set up

5.2.1 Results

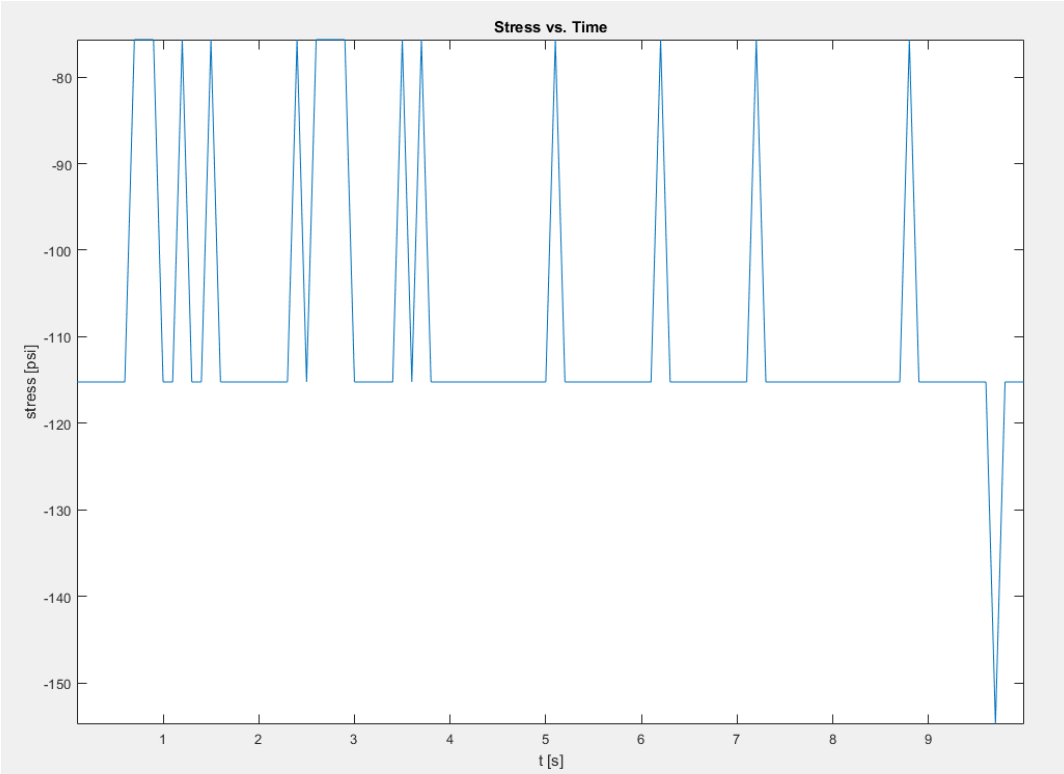


Figure 5 . 5: Initial - 0 load reference run

Figure 5.10 above shows the initial strain calculation with no load applied. This sets a reference point for the system so that the difference of values can be calculated.

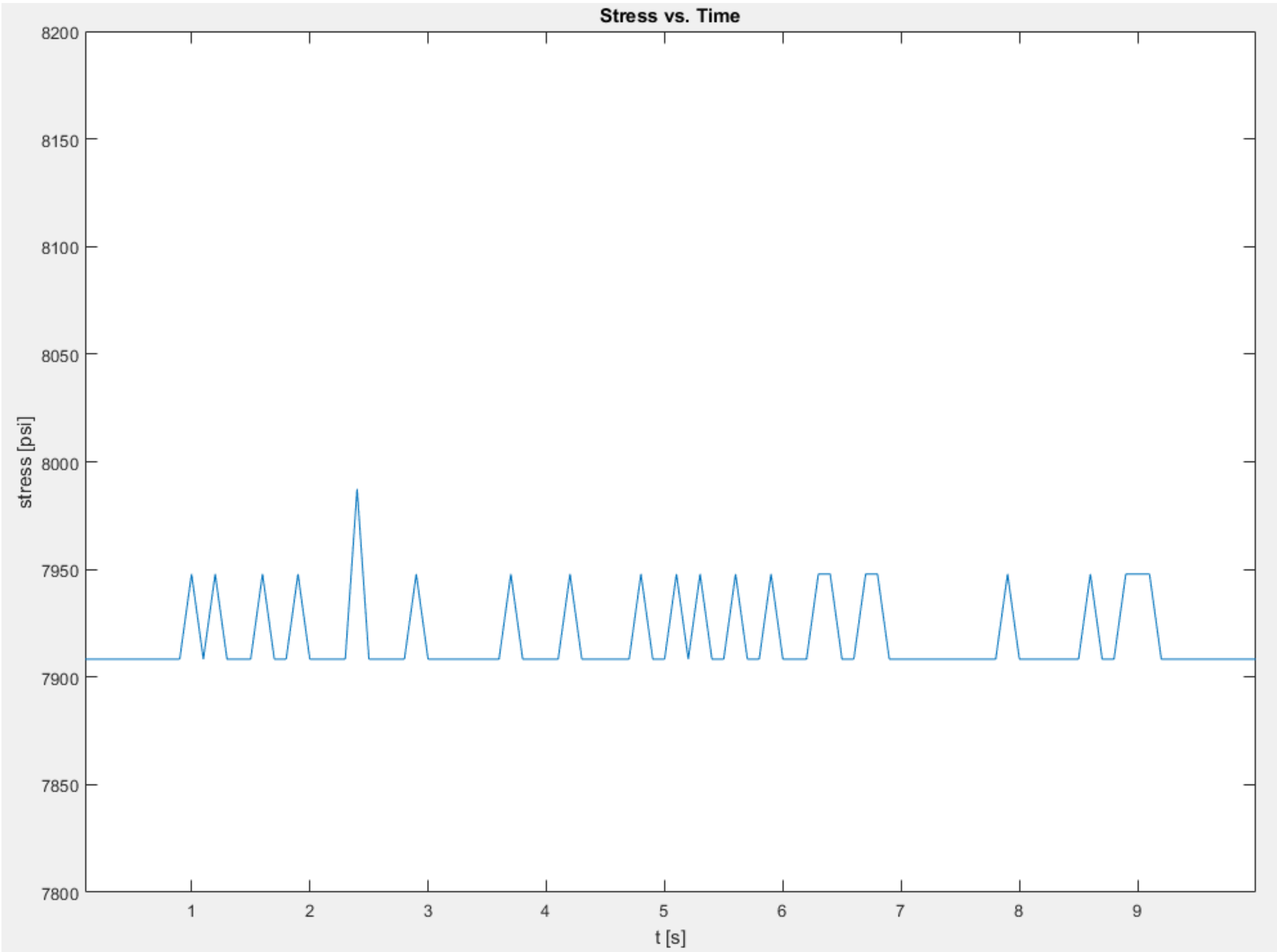


Figure 5 . 6: Stress Results from Strain Gauge vs. Time

5.2.2 Analysis

Figure 5.11 above contains the graph that was produced from measuring the strain and then converting it further to stress using the following equation:

$$\sigma = \varepsilon E$$

- σ = Stress
- ε = Strain
- E = Elastic Modulus

The results which can be analyzed in Figure 5.11 show slight spikes in the stress graph even though the force being applied was static, these spikes were a result of noise existing in the electrical circuit. The constant stress is seen at a value of 7910 psi with the largest spike seen as ~85 psi, after taking into account the reference measurement of -115 psi seen in Figure 5.10, the constant stress lines equate to **8025psi.** This measurement of stress only considers a unidirectional surface stress along the Y-axis. Refer to Figure 5.8 and 5.9 above for the testing set up.

5.2.3 Conclusion

Strain gauge analysis results are very close to the hand calculations and confirm the stress result of the system along the Y-axis surface which is the direct result of the bending stress around the Z-axis. Strain measurements can vary due to noise in the system, as well as the changing angle of the rear shocks, a 10% error is acceptable for this verification. The results obtained from the strain gauge test were more accurate than expected, therefore a confirmation was required, and Appendix P can be referenced for two more test results that confirmed the results seen above in Figure 5.11.

5.3 Virtual Model Simulation Initial Results and Analysis

5.3.1 Phase 1

When the initial model was brought into the simulation environment an attempt to run a simulation was conducted and failed. The initial problem that occurred was "One or more Shell thicknesses are not defined." By analyzing the shells that were causing this error it was determined they had no relationship to the assembly or geometry of the model but were rather used for mating, or references, so the shells were suppressed.

Once the shells were suppressed, the next error that presented itself was a lack of assigned material properties. Due to material properties influencing the simulation, it was important correct materials were selected and applied. All the necessary materials were selected by consulting the sponsor.

A problem that caused the largest difficulty was interferences in the model. There were over 200 to be specific. By analyzing the model, many cosmetic components were noticed, including bolts, screws, nuts and washers. Many of these cosmetic components were the reason for interferences, and their absence would have no influence on the simulation. To further simplify this model, these cosmetics were excluded from the analysis. This left a remainder of around 40 interferences. The remainder of interferences had to do directly with component design, therefore this required a re-design of several pieces by tiny amounts to create part clearances. After going through an iterative process of correcting each design, the interferences were no longer present in the model.

Due to the complexity of some parts the standard mesh size was not applying to the model, therefore, it needed to be set manually. The mesh was set to 2mm. and was applied to the model. Due to the complexity of several components, an adaptive meshing was required to ease the simulation process.

5.3.1.1 Results

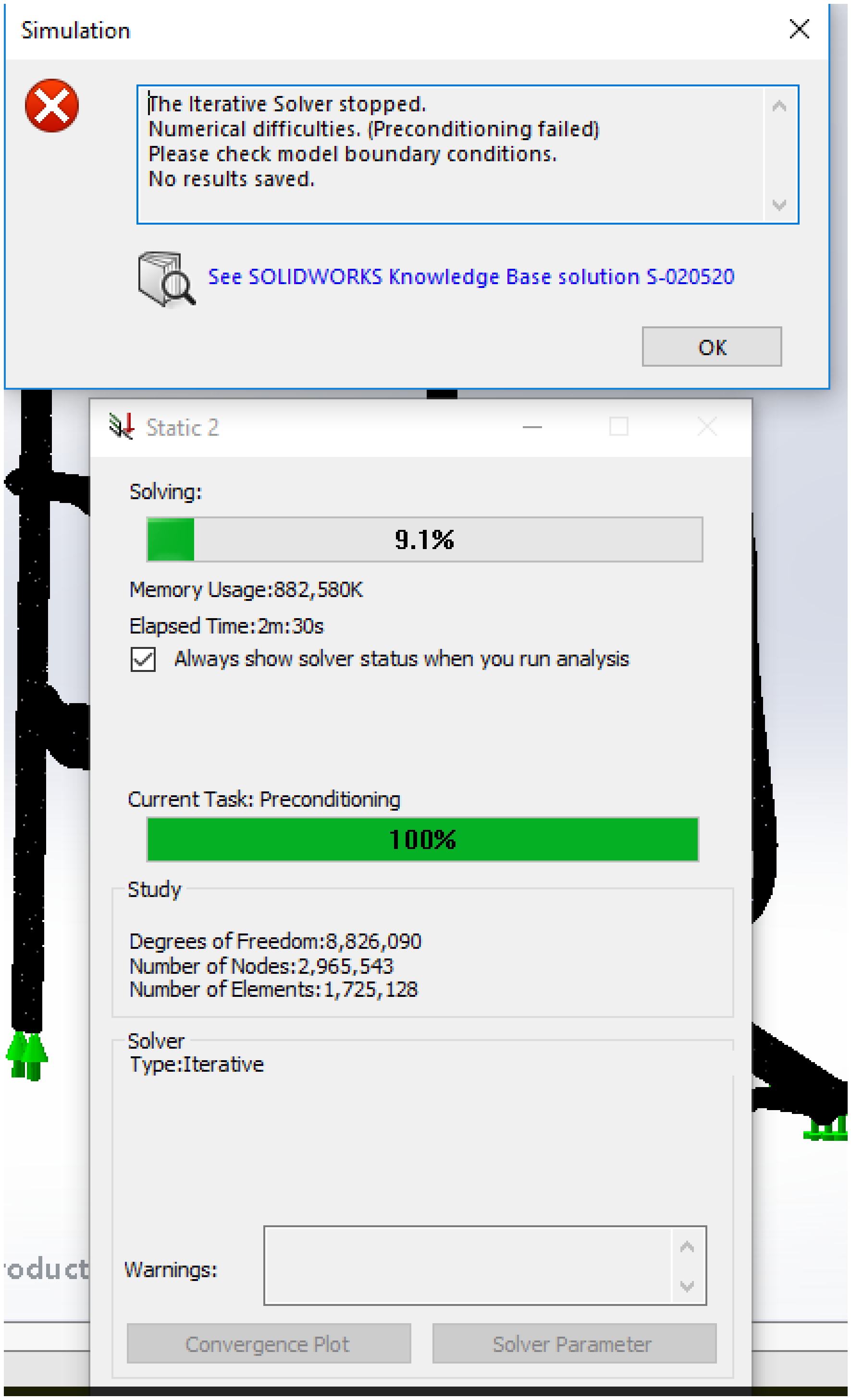


Figure 5 . 7: Iterative Solver Stopped

5.3.1.2 Analysis

After conducting research on the type of failure during simulation pictured above in Figure 5.1, we realized it had to do with the contacts within the simulation environment. In the simulation environment the model sets a global contact which is dependent on the physical contact between two components. If there is any empty distance between two parts, it fails to create a contact the parts become “free-floating”. If a component is set to “free-float” the distribution of forces within the model is represented inaccurately and often unable to compute, resulting in a critical error during the simulation.

5.3.1.3 Conclusion

To solve this problem, a manual re-assignment of all contact sets was required. To see which components were behaving incorrectly and get initial results the model simulation was simplified to a “Soft Spring Stabilization”. Once the results were available by using the visual display of stresses it was clear which components did not have contacts, therefore they could be assigned accordingly.

5.3.2 Phase 2

Following Phase 1, the model was correctly assigned manual contact sets and component contacts as required. This was achieved with an iterative process of going through "pseudo" simulations and noticing which aspect of the model was not behaving correctly to the forces applied. After running numerous simulations and assigning contact sets correctly the simulation was still resulting in the same error seen below.

Due to the simulation still resulting in an error, meetings were conducted with two faculty experts of SolidWorks simulations. *Cyrus Raoufi*, who was the first expert, suggested that an attempt be made at re-designing components so that they all have physical contact with their adjacent components. At this point in our model this was already represented by the contacts that were manually created. A second suggestion that was made was due to the thickness of the tubing and the complexity of the geometry, we should use a beam analysis approach to the tubing. The last suggestion suggested was to develop a completely new model from scratch, but that would result in large inaccuracies as the teams experience was insufficient for those types of material replacements.

After speaking with a second expert, *Neil Munro*, a similar approach was suggested to the team. Suggestion was to treat the tubes as beams while conduction the simulation experiment, but he also insisted the error in the results of such simplification would be significant. The final suggestion made was by the project sponsor, Dr. Jaimie Borisoff, who suggested we approach the model by starting from two components and then add components singularly while making the repairs and testing results to be successful following each change.

The approach that was decided was to begin with the single front piece and run simulations iteratively while adding more of the model's components until the complete model could be tested upon. Adding components within the assembly singularly proved to be successful by targeting the component sets and required design adjustments with each component addition.

5.3.2.1 Results

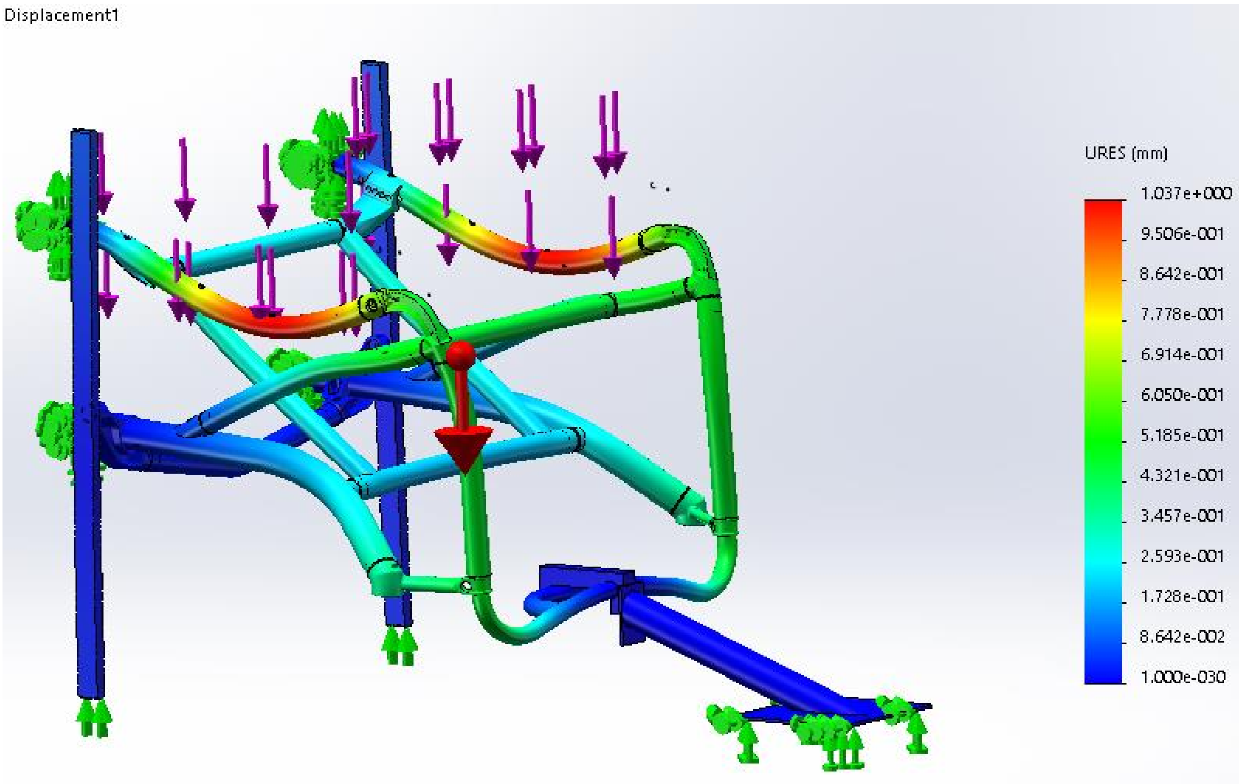


Figure 5 . 8: Strain Results retrieved through piece-by-piece approach

5.3.2.2 Analysis

The results shown in the displacement of the model above, under the conditions presented, represents how the model would behave in real life if the same conditions were bestowed upon it. Unfortunately, these results did produce the stress results of the physical validation that were necessary for validation. Therefore, the rear wheels and front attachment had to be adjusted according to the measurements of the physical model, as well as applying the correct constraints to reflect the flex and behaviour of both the front attachment and rear wheels.

5.3.2.3 Conclusion

After analyzing the results in Figure 5.2 above, the plan was to keep the added contact sets throughout the model, and only adjust the required components and constraints. Following a redesign of the components and reassignment of constraints, another simulation was conducted. This experience taught us that approaching an unknown problem in the simulation environment can be solved by approaching the complex model component by component. This allows a step by step inspection of the errors propagated at each of the nodes and directs the focus onto that specific area, rather than attempting to analyze the entire model without the knowledge of where to focus first.

5.3.3 Phase 3: Final Results and Analysis

The results were successfully collected from the SolidWorks simulation by taking into account the required adjustments that were found by conducting the previous simulations. The redesigns took into consideration the transfer of forces through the geometry, to ensure the forces acting on the foot-peg were reflective of the real-life environment. After applying these updates to the model, the results retrieved from the simulation were much more reflective to the numbers obtained during the physical validation. These results considered a rigid transfer of forces through the model, therefore only the foot-peg and the front-end attachment were used in the simulation: this set-up can be seen in Figure 5.3 below.

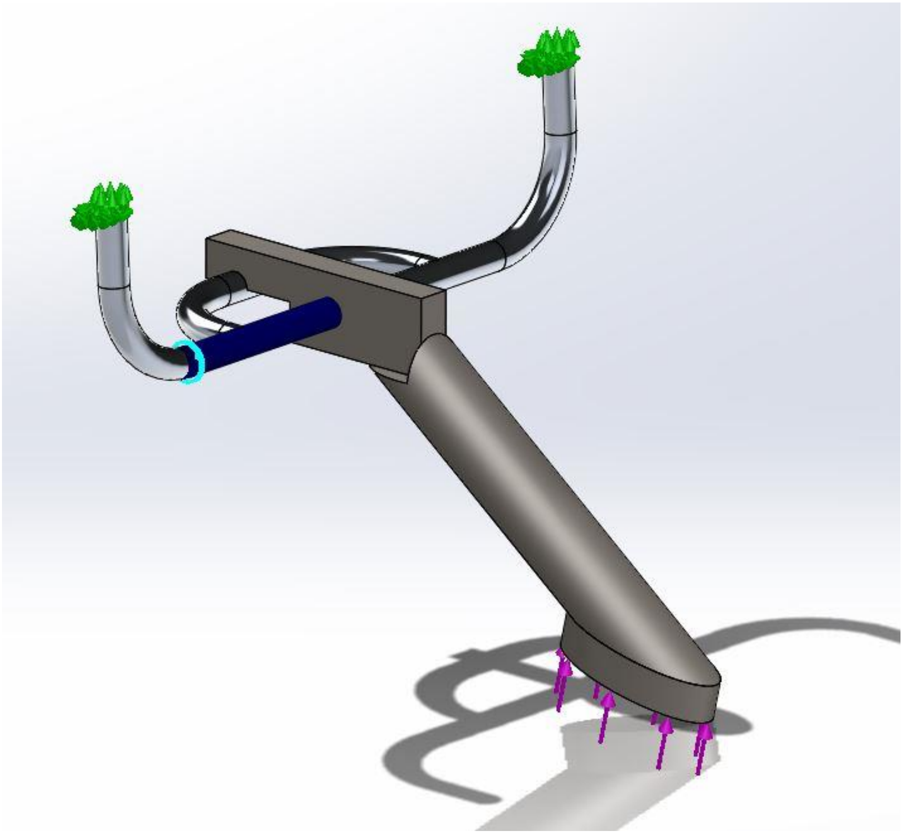


Figure 5 . 9: Foot-peg Simulation Set-up

An adjustment to the rear wheel was made such that it would allow flex at the point where it attaches to the main frame. The front wheel, which required to act as a pinned connection around the foot-peg was redesigned to reflect such a constraint. The set-up can be seen below in Figure 5.4.

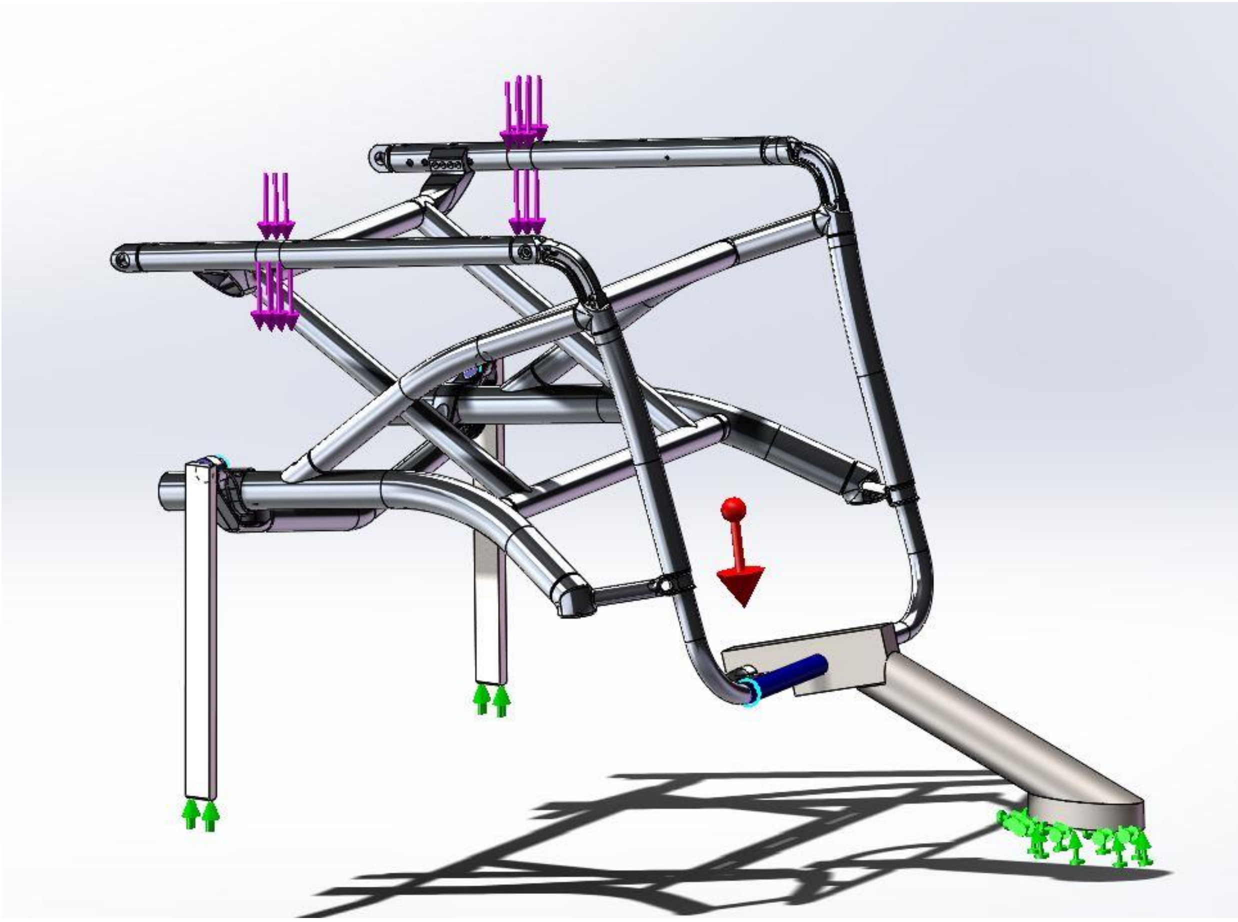


Figure 5 . 10: Simulation Set-up for Entire Model

After making all the changes and to the team's best knowledge correctly contacting all components within the simulation environment, the results were still inadequate. The deflection results can be seen below in Figure 5.5, which represents the expected deflections of the model, and Figure 5.6 shows the deformation further in a more exaggerated form. The problem existing was assumed to be within the transfer of stresses throughout the frame. After analyzing and iteratively running simulations on the complete model, it was determined simulating on the complete model as a whole required a lot more experience and knowledge of simulation software to obtain accurate results. Another consideration was made that the results seen in the stress probes of the simulations are based on a von Mises calculation of stress, significance of which is discussed further in the analysis.

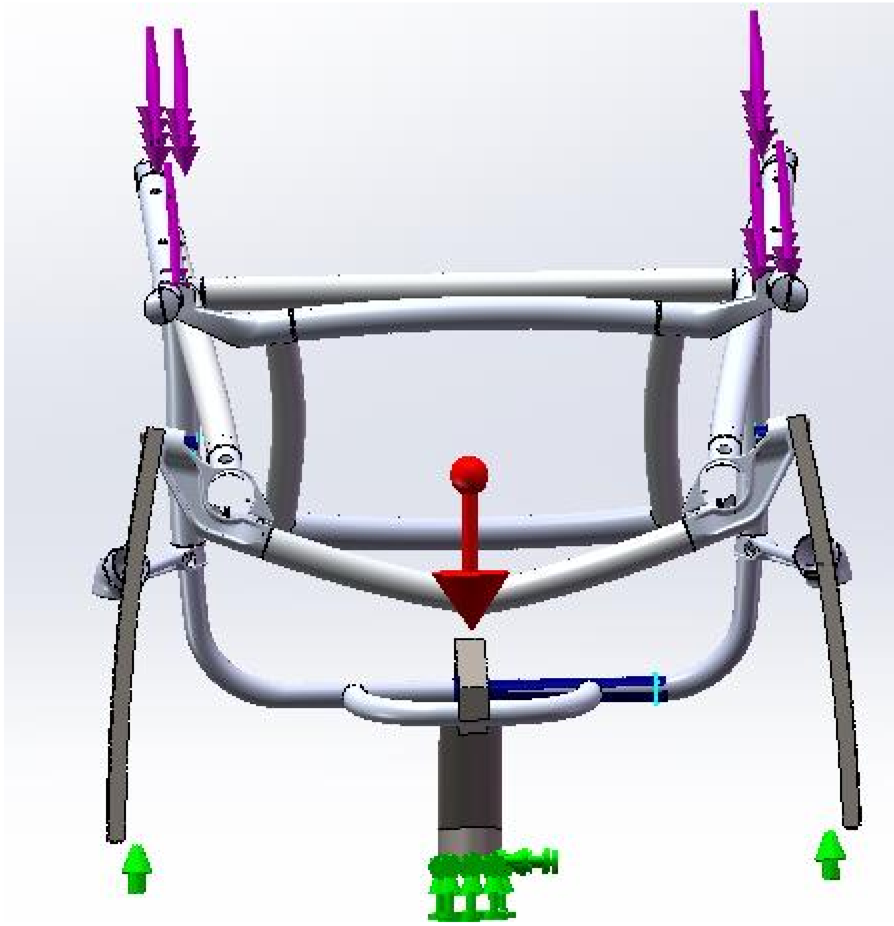


Figure 5 . 11: Displacement Shown on Frame

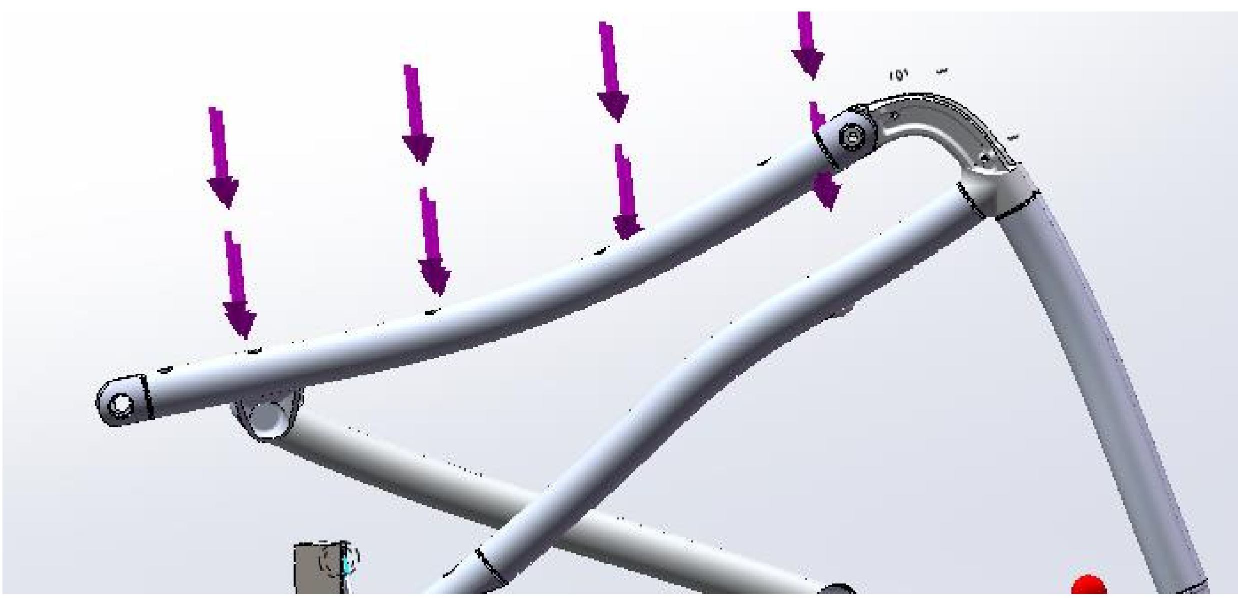


Figure 5 . 12: Close exaggerated deformation of loaded bars

5.3.3.1 Results

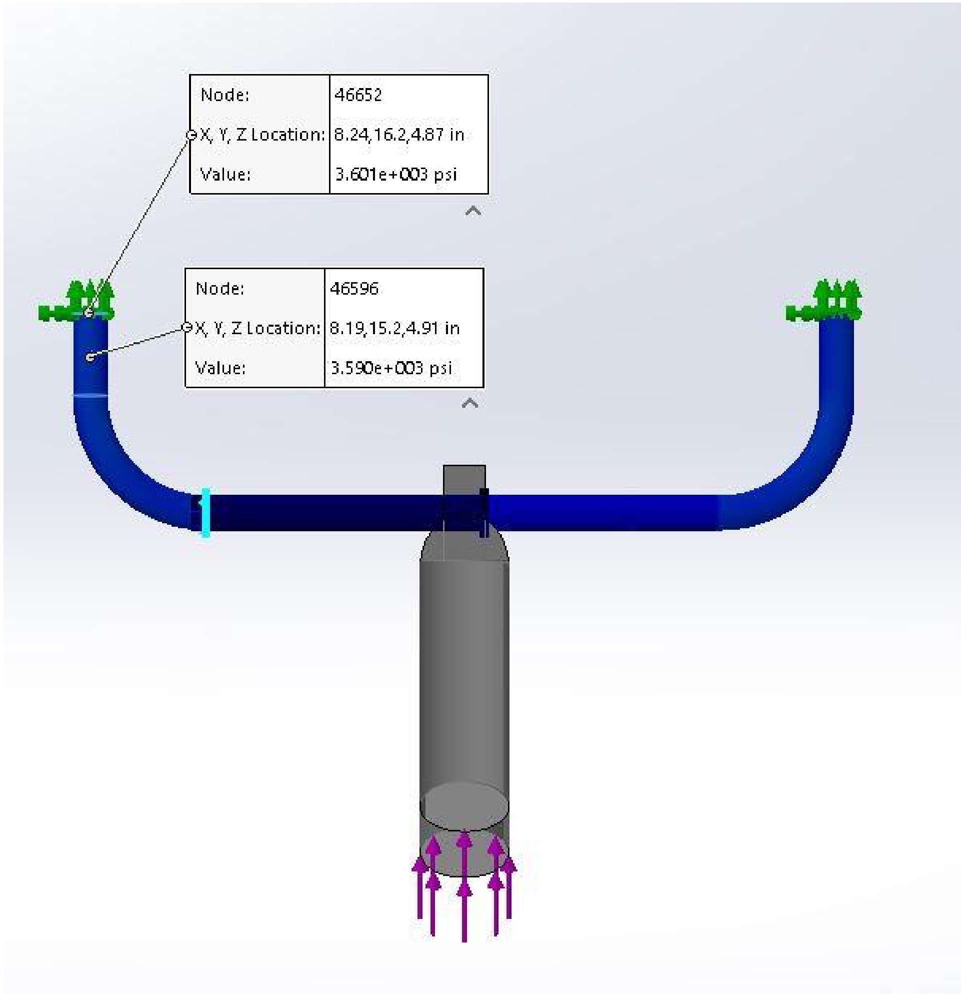


Figure 5 . 13: Results of Foot-peg Simulation

5.3.3.2 Analysis

The results seen in the analysis in Figure 5.7 are those of the Von Mises Stress, therefore they cannot be directly compared to the unidirectional stress which was determined by the analytical and physical validations. The Von Mises stress is solved using the following equation [2]:

$$\sigma_{von\ mises} = \sqrt{\left\{\frac{[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2]}{2}\right\}}$$

The principle stress calculations also contain other stresses within the principal direction, the equation for two of these stresses are governed by the following:

$$\sigma_1 = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x + \sigma_y}{2}\right)^2 + \tau_{xy}^2}$$

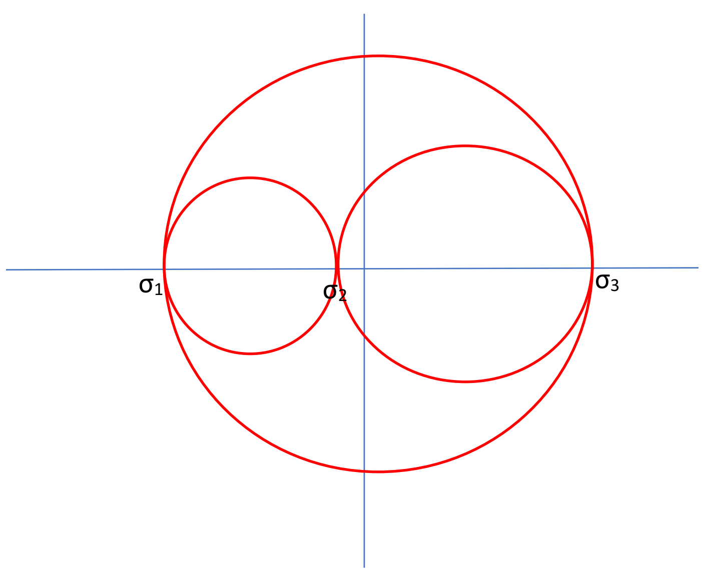
$$\sigma_2 = \frac{\sigma_x + \sigma_y}{2} - \sqrt{\left(\frac{\sigma_x + \sigma_y}{2}\right)^2 + \tau_{xy}^2}$$

The equations above cannot be directly compared to the analytical results or strain gauge measurements due to both of those validation methods yield in only the σ_y. After simplification and further research into the stress results produced by the simulation environment, it was determined that the von Mises for a simple structure within the environment with a static unidirectional stress could be approximated to σ_y by the following equation:

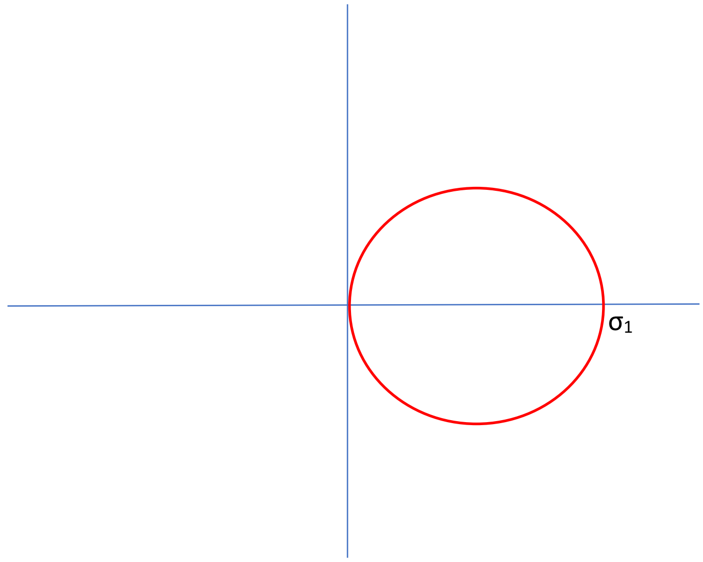
Equation 5.1: Unidirectional Stress Equation

$$\sigma_y = (2)(\sigma_{von\ mises})$$

The calculation of the Von Mises stress using the three principal stresses behaved in way that can be described and visualized using a Mohr's Circle:



This represents that there are three principal stresses at each node, therefore the calculation of average, max, and minimum stresses differ from those done by physical validation or analytical calculations. Each of these three stresses contains three more stresses that must be considered, so an equation with 9 variables is considered when solving for von Mises stress. The one used by the team after simplification of the model would be represented by the following Mohr's Circle:



The Mohr's Circle above indicates the simplification considered when simulating on the foot-peg section only with the assumption that the dominant bending stress is the only stress. The stress is singular and can be analyzed easily, mainly due to the absence of the other two principal stresses. This assumption allows for a comparison with the validations computed earlier due to the simplicity of the loading on the system and the static environment.

5.3.3.3 Conclusion

Using Equation 1, it was determined the foot-peg simulation showed stresses similar to the validations. The foot-peg showed a stress of 3590psi at the location of the strain gauge, which is equivalent to a unidirectional y-stress of **7180psi** which is approximately within 12% accurate when compared to the results of analytical and physical tests.

The results from SolidWorks for the entire model were deemed incomparable to those from strain gauge analysis or analytical calculations. This was determined mainly due to the nature of the equations governing the simulation stress calculations at each node of the entire model. These calculations could skew the results from node to node exponentially if there are any stresses present in 1 of the 9 directions that were not considered during the validations, as discussed during the analysis of this section. Also, the contacts within the model may not have the capability to represent the exact behavior of the model in real-life.

5.4 Analysis of Failure

To ensure that no failure occurs on the front foot-peg it is required to validate the safety factor and maximum stress that can occur at the specified location of measurement. The following calculations were conducted to analyze the maximum static and dynamic loads the frame could handle with the front-end attachment.

$$\sigma_{yield} = 35{,}000\ psi\ (6061\ Aluminum\ '\mathrm{min}\ \ value\ used')$$

$$\sigma_y = \frac{MC}{I_z}$$

$$35{,}000 = \frac{M * 0.375"}{0.0066 in^4}$$

$$\therefore M = 616\ lb * in$$

$$616 = Fr2Cos(13.06°) * (9.47")$$

$$\therefore\ Fr2 = 66.77 lbs$$

$$\therefore Maximum\ Static\ Load = 836.8 lbs$$

$$\therefore Static\ Safety\ Factor = 4.3$$

$$\therefore Maximum\ Dynamic\ Load = \sim 415 lbs$$

$$\therefore Dynamic\ Safety\ Factor = \sim 2.0$$

An assumption is made that the static load of 8025psi and dynamic load of 16050psi will be cycles to -8025psi and -16050psi. Although this is not a rotating beam, so these cycles will be uncommon as the would only occur when the user bumps into rigid objects. A fatigue load is usually calculated for 1 million cycles at which point if it passes that threshold it is considered failure will not occur.

$$\Delta\sigma = \sigma_{MAX} - \sigma_{MIN}$$

$$\sigma_M = \frac{\sigma_{MAX} + \sigma_{MIN}}{2}$$

$$\sigma_A = \sigma_{MAX} - \sigma_M$$

$$S_{f'} = S_{UT}(0.40)$$

$$S_f = S_{f'}(C_L C_G C_S C_T C_R)$$

$C_L = Load\ Factor = 1.0$

$C_G = Gradient\ Factor = 0.9$ (0.4<Dia<2.0)

$C_S = Surface\ Factor = 0.8\ (off\ the\ chart, safe\ approximation)$

$C_T = Temperature\ Factor = 1.0\ (Room\ Temperature)$

$C_R = Reliability\ Factor = 0.897\ (approximation)$

These are the formulas used to calculate the endurance limit, sf, and the cyclic stresses of our system. The calculation performed involved approximations and should not be considered as a definite answer.

Static:

$\Delta\sigma = 16050psi, \quad \sigma_M = 0psi, \quad \sigma_A = 8025psi$

Dynamic:

$\Delta\sigma = 32100psi, \quad \sigma_M = 0psi, \quad \sigma_A = 16050psi$

For both loads the endurance limit, which was based on the 6061 S_{UT} value of 42,000psi is as follows:

$S_f = 0.4(42{,}000)(1.0)(0.9)(0.8)(1.0)(0.897) = 10851psi$

The endurance limit was compared to the amplitude stress of the expected cyclic loads (σ_A) and it was concluded that under static loads there will be no fatigue failure, whilst under dynamic load in the worst-case scenario there will be over 500,000 cycles before any sign of possible fatigue in the foot-peg. Taking into account each cycle is an impact against a large obstacle, it can be concluded this failure should not occur during the life cycle of a wheelchair frame.

6. Conclusion

The sponsor of this project requested a SolidWorks simulation model of a wheelchair which represented the properties and behaviour of a physical model provided to us. Also, to determine the stresses applied to the foot-peg by a specified front-end attachment. The results from the simulation were to be validated using a physical strain gauge test and analytical calculations.

The initial problems were caused by the model within the simulation environment included the model configuration, simplification, meshing, boundary conditions, and interferences which are all further discussed in section 4.1 of the report. Each of these problems were solved periodically until any initial results were obtained, from which point refinement of the simulation environment was done iteratively to find suitable solutions. The refinement and adjustment processes of the SolidWorks simulation model are further explained in section 5.3.

After analyzing the results produced from conducting a static simulation on the entire frame, it was concluded that the frame was producing results that were unacceptable. These results were unacceptable because of the equations governing the different stress calculations within the simulation environment, which are analyzed in depth in Section 5.3.3.2. The results from the simulation using the governing von Mises and principle stress equations could not be compared with the analytical or physical results obtained by the team. The results of the validation methods are unidirectional, single stress calculations, and cannot be compared to the 9-stress calculation of the von Mises (refer Section 5.3.3.2). A note should be made that the results of displacement showed expected behaviour of flex and movement, so it is possible that the results of stress may be correct but are in need of a more intricate validation.

The stress calculations on a complex model cannot be solved accurately to determine a single unidirectional stress, but if the model is simplified enough to consider two principal stresses as negligible at each node, then a comparison between von Mises stress in the simulation with the physical and analytical validations can be conducted.

The physical validation was done by using a strain gauge placed at a point of interest, slightly below the location of the foot-peg attachment onto the main frame. This physical set-up is discussed more thoroughly in Section 4.2. The results gained from this method closely agreed with the analytical calculations. The results and analysis of the strain gauge test stresses can be read further in Section 5.2. The analytical calculation assumed the frame to be rigid and found a bending stress of 8103.6 psi, refer to Section 5.1, which matched closely with the 8045psi determined from the strain gauge results.

Simulation results were gathered by simplifying the model down to the foot-peg and front-end attachment only. The fixtures were placed at the locations to match the physical model, and loads applied represented the calculated reaction force that would be produced from the 200lb load. Using Equation 1, seen in Section 5.3.3.2, a comparable stress of 7180psi was found.

The simulation and two validation methods provided results of stress at the specific location and was further used to determine whether failure would occur under the given boundary conditions. Using the analysis seen in Section 5.4 a conclusion was made that the wheelchair will not statically fail, but rather has a safety factor of over four on a static load. The dynamic loading has been approximated to have a safety factor of two, but is suggested to be further analyzed.

Bibliography

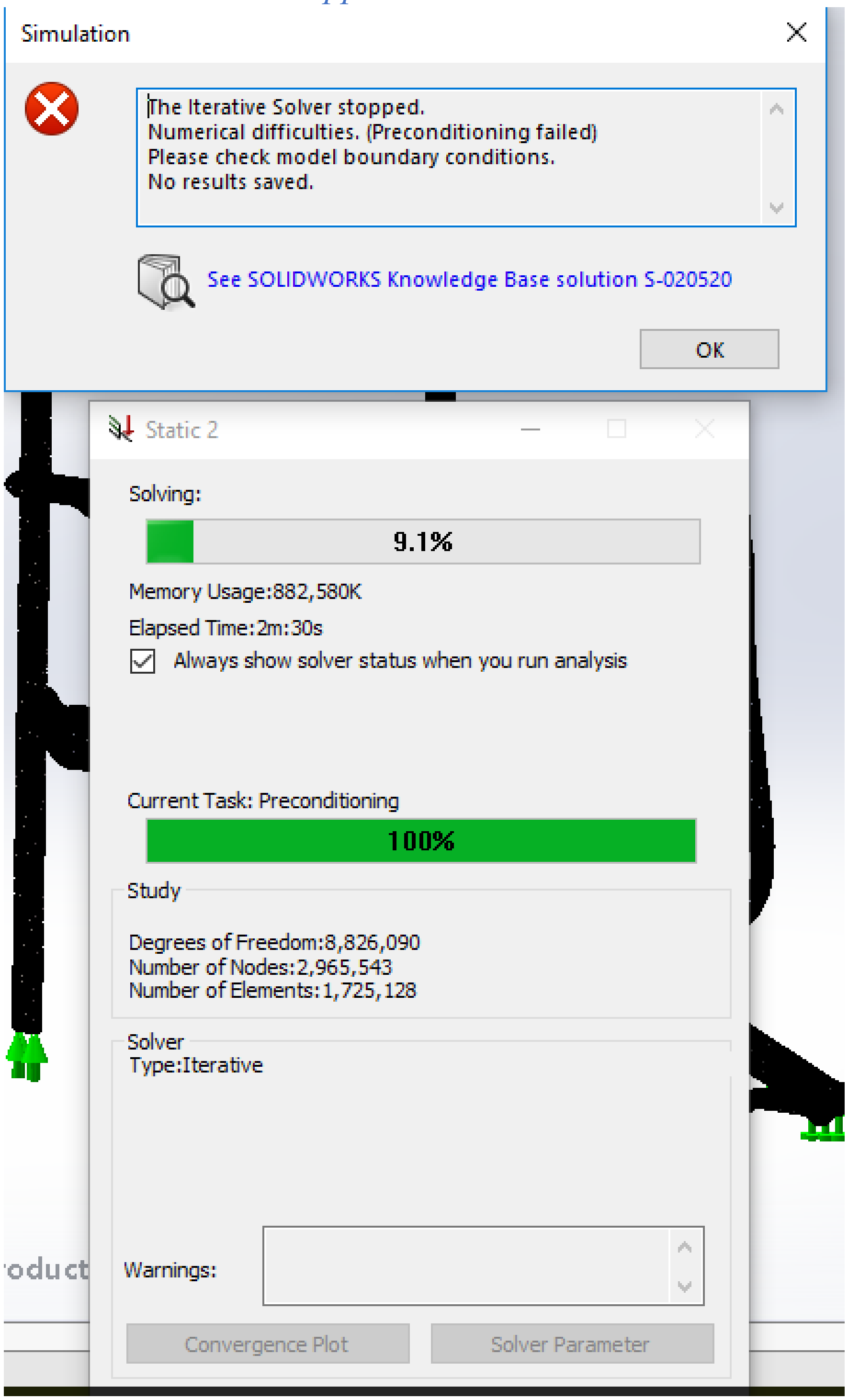
[1] Astris Lifecare, "FreeWheel Attachment," 2016. [Online]. Available: http://www.ablerehab.com.au/shop/item/free-wheel. [Accessed January 2018].

[2] Dassault Systemes, "Maximum Von Mises Stress Criterion," 2016. [Online]. Available: http://help.solidworks.com/2016/english/solidworks/cworks/r_maximum_von_mises_stress_criterion.htm. [Accessed April 2018].

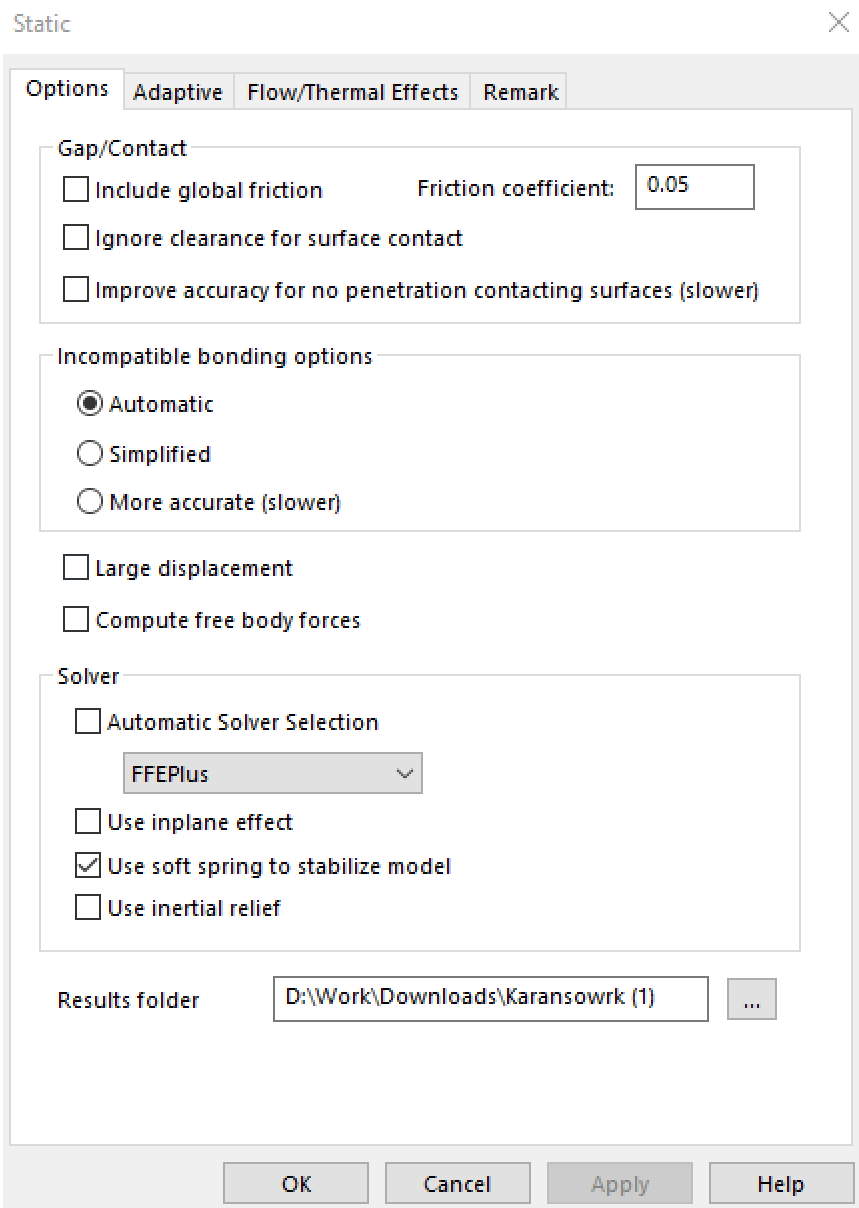
[3] N. W. Brian Woods, "History of the Wheelchair," [Online]. Available: https://www.britannica.com/topic/history-of-the-wheelchair-1971423. [Accessed January 2018].

APPENDICES

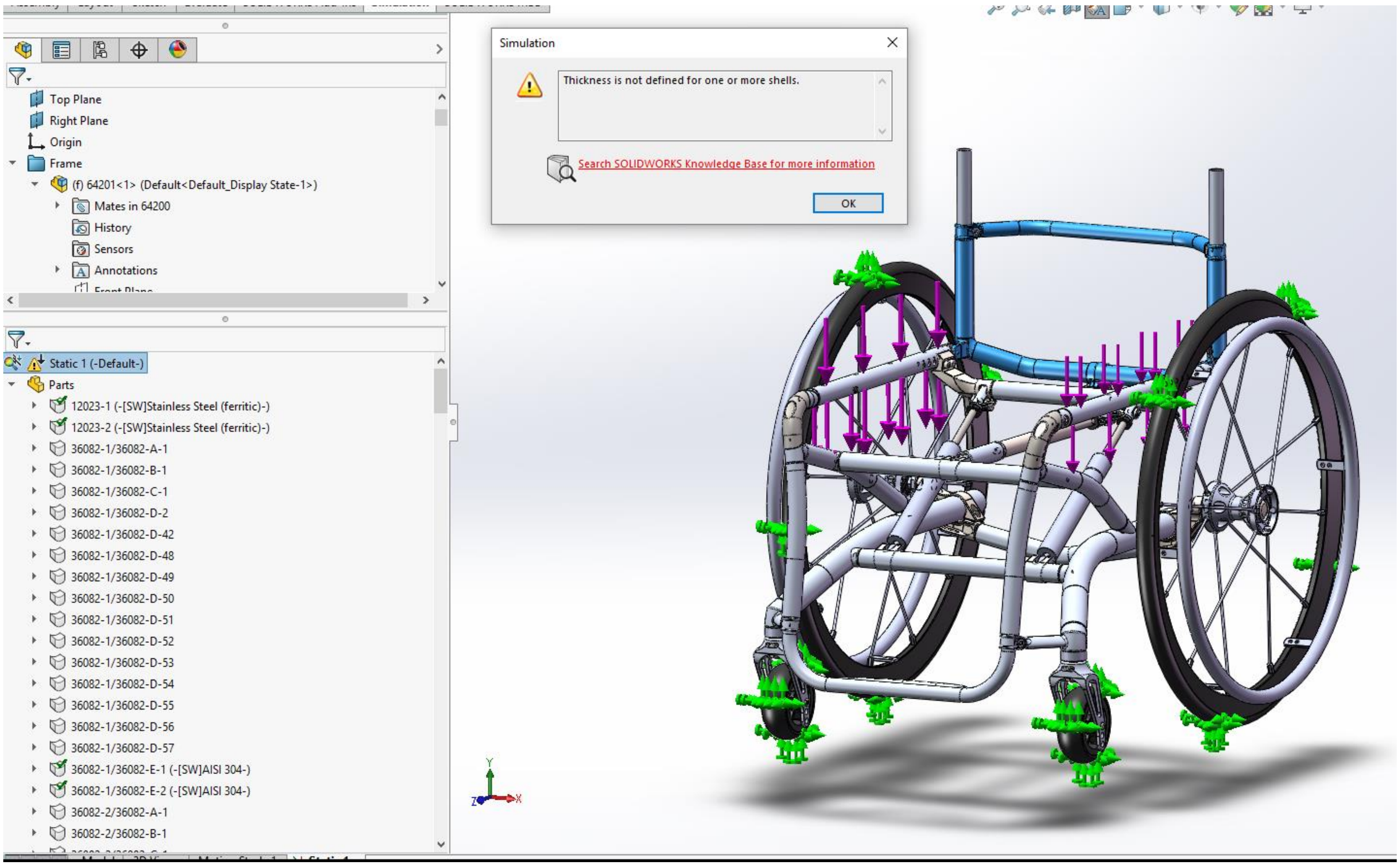
APPENDIX A: Iterative Solver Stopped



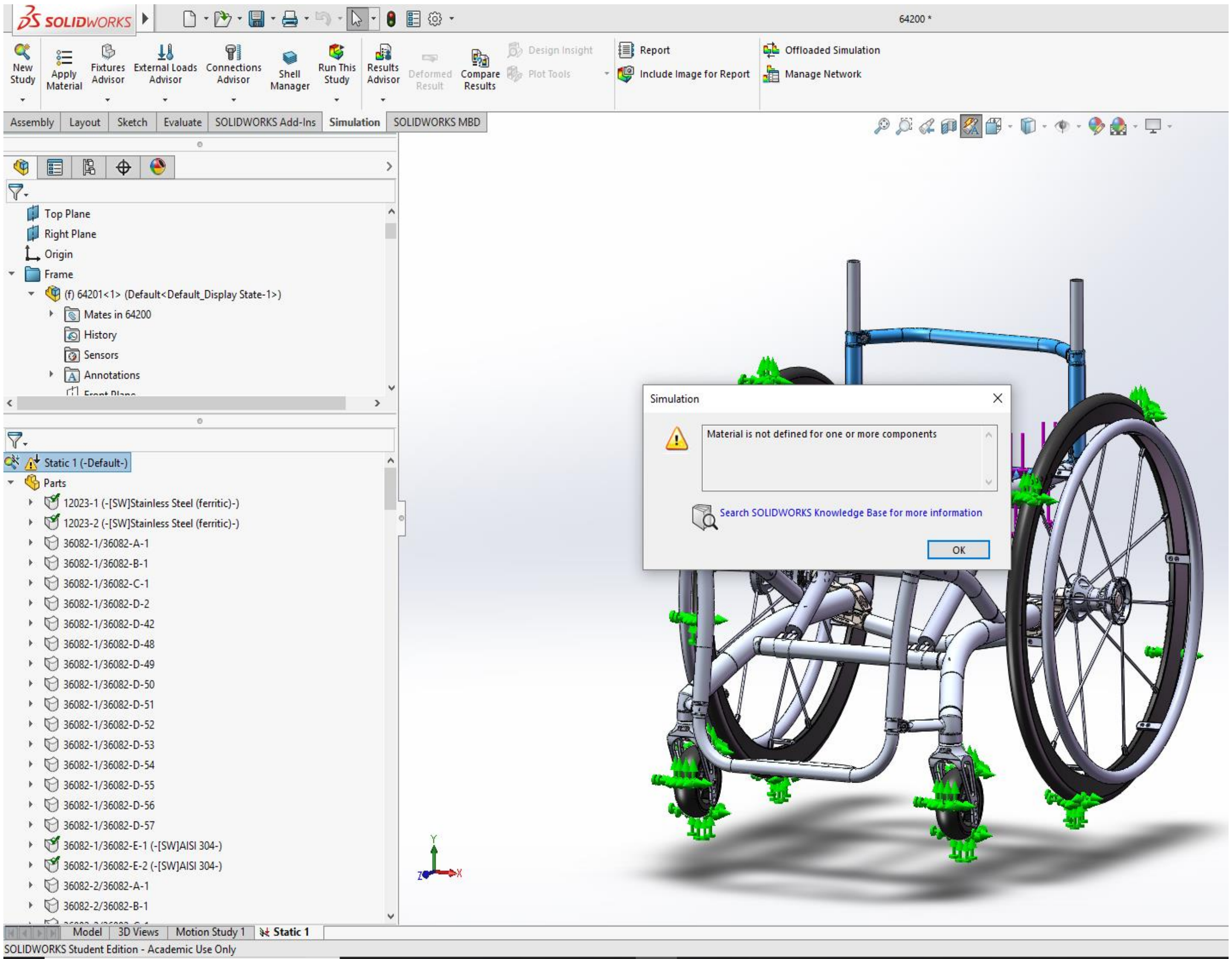
APPENDIX B: Soft Spring Stabilization Settings



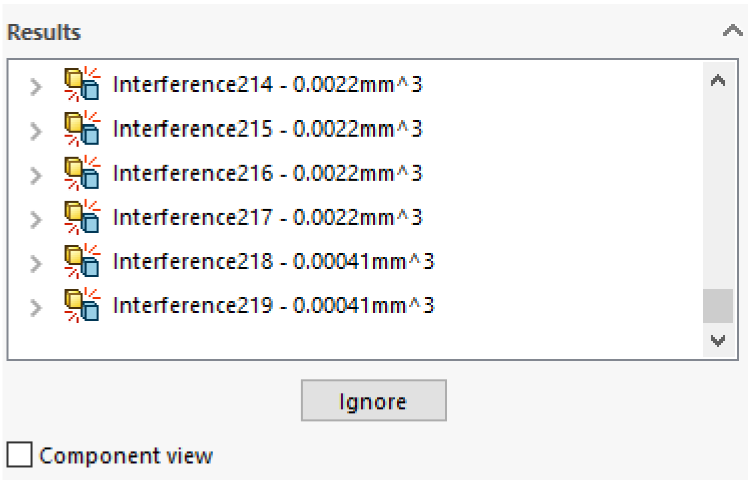
APPENDIX C: Thickness undefined



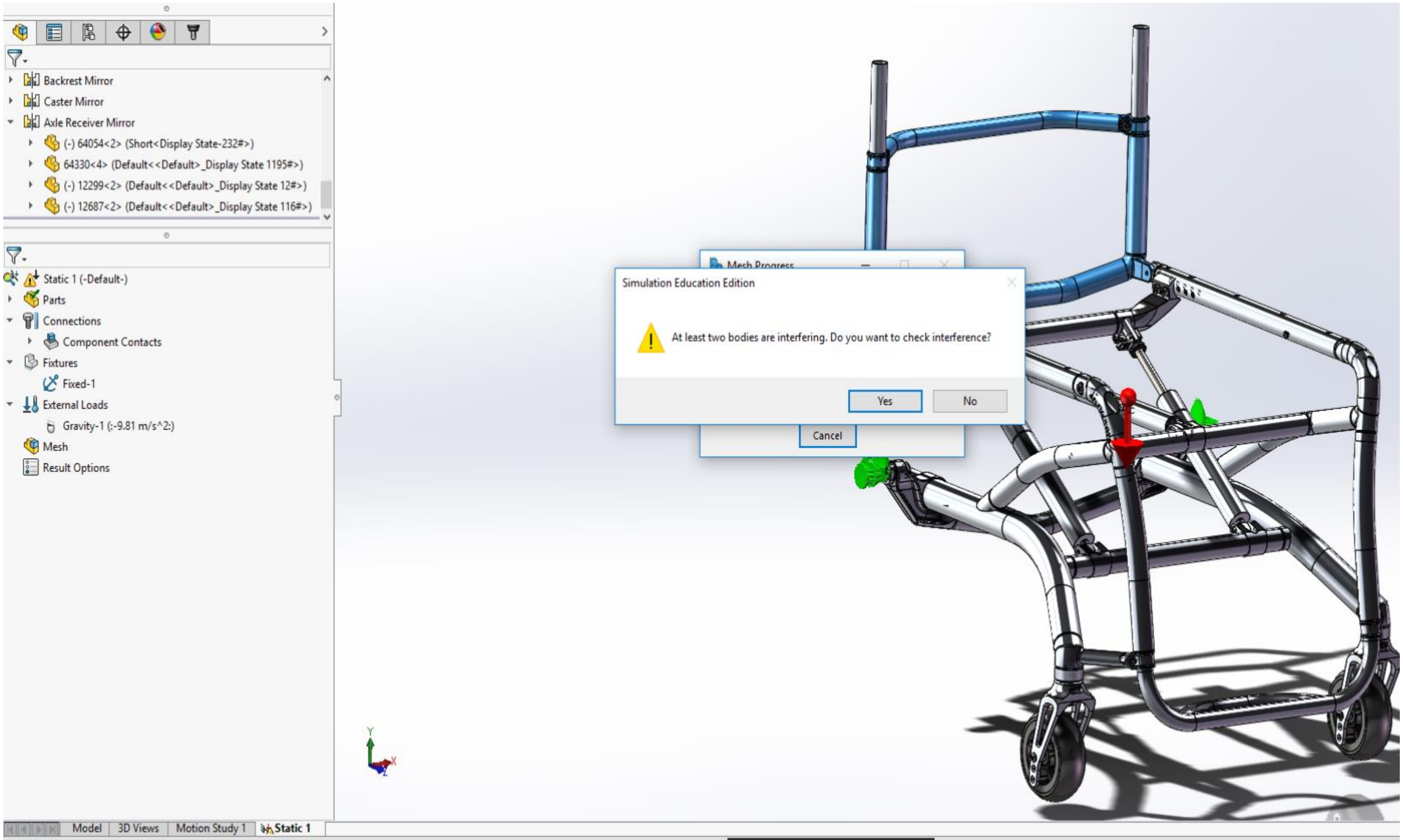
APPENDIX D: Material Properties undefined



APPENDIX E: 219 Interferences



APPENDIX F: Interference Error



APPENDIX G: Design Review Package

MECH 8290

Capstone Project

Design Review Package

Requirements Review

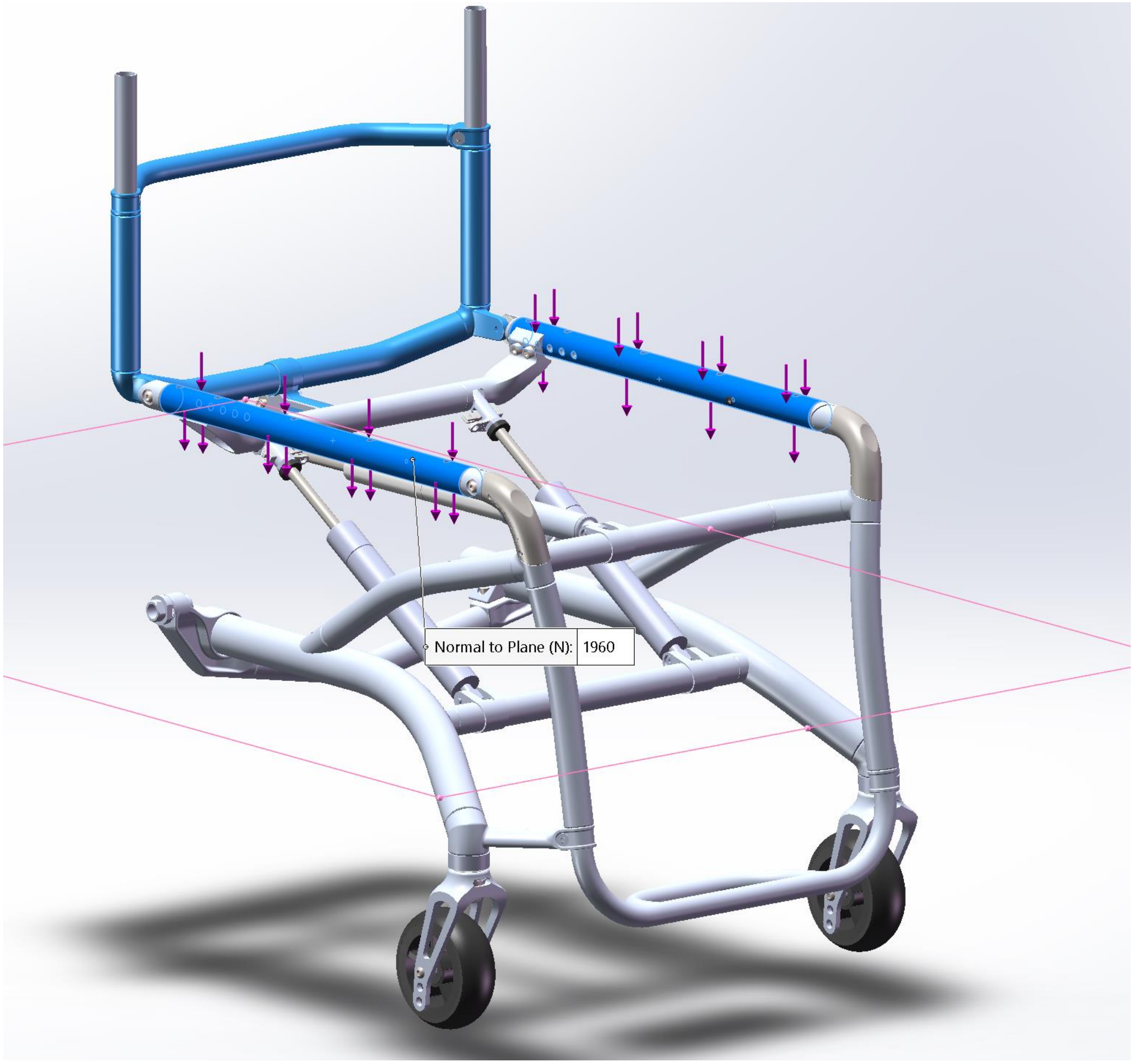
Prepared by:

Karanjit Khatkar, Set: B

Artem Gridnev Set: B

Table of Figures

Figure 1.1: FreeWheel Front-End Attachment....................1
Figure 3: Forces Applied.................... **Error! Bookmark not defined.**
Figure 4: Fixtures.................... **Error! Bookmark not defined.**
Figure 5: Wheelchair Wheels Flex.................... **Error! Bookmark not defined.**
Figure 6: Strain Gauge Locations .... **Error! Bookmark not defined.**
Figure 7: Strain Gauge Location Zoomed.................... **Error! Bookmark not defined.**
Figure 8: Adjustable Part.................... **Error! Bookmark not defined.**
Figure 9: Configurations.................... **Error! Bookmark not defined.**
Figure 10: Test Jig Round Side and Front View.................... **Error! Bookmark not defined.**
Figure 11: Test Jig Flat Side and Front View .... **Error! Bookmark not defined.**
Figure 12: Front-end Attachment Jig .... **Error! Bookmark not defined.**
Figure 13: Transfer Bearing.................... **Error! Bookmark not defined.**



In Figure 1 above, the force being applied onto the bars on each side represent a person with a weight 200kg sitting on the wheelchair. The reason this was distributed onto the side bars is to avoid needing to display the aesthetics of the seat that would be hanging in the middle of the chair. This is done under the assumption that the real force will transfer from the cloth seat to the frame as represent in the model above.

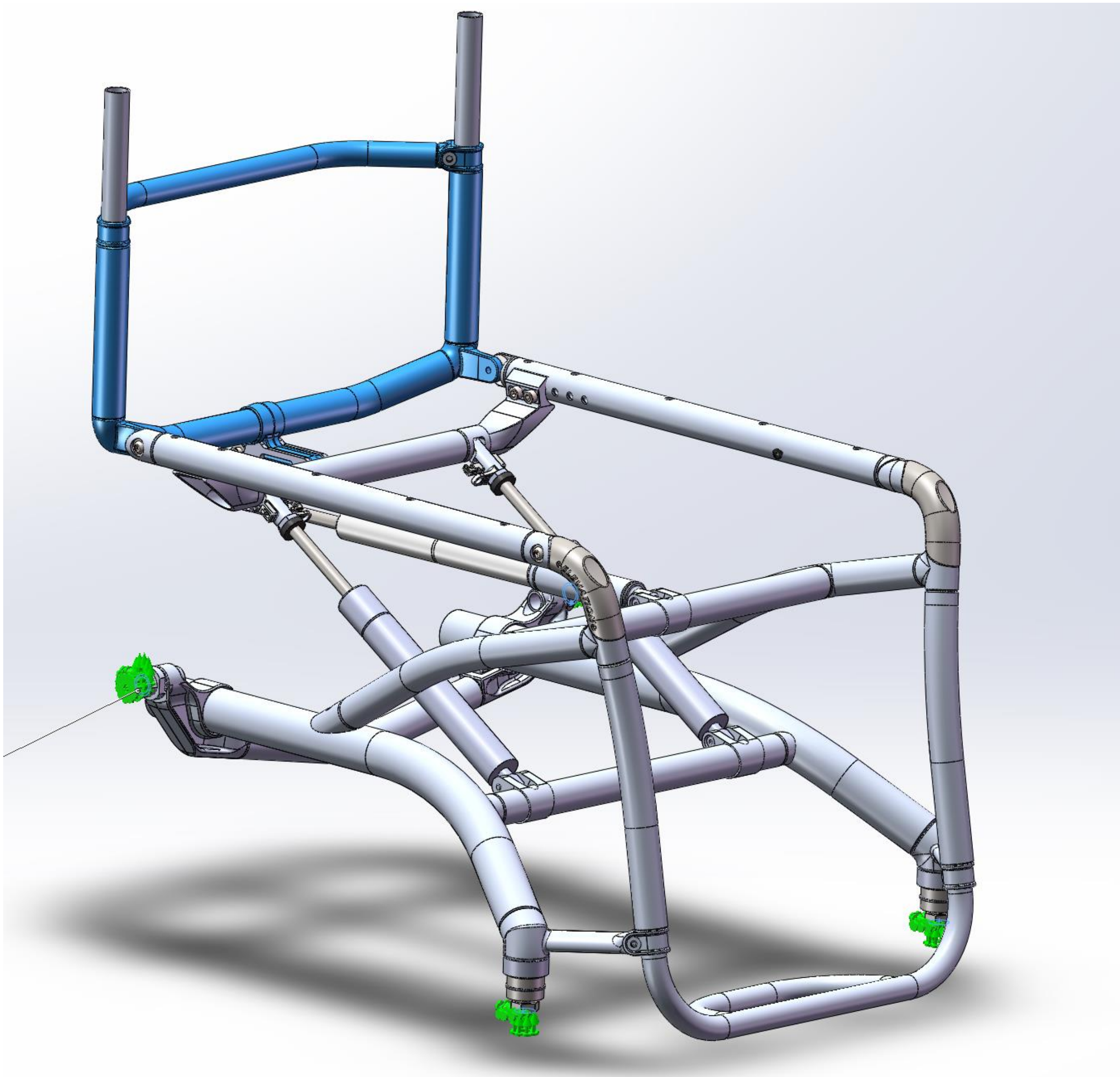
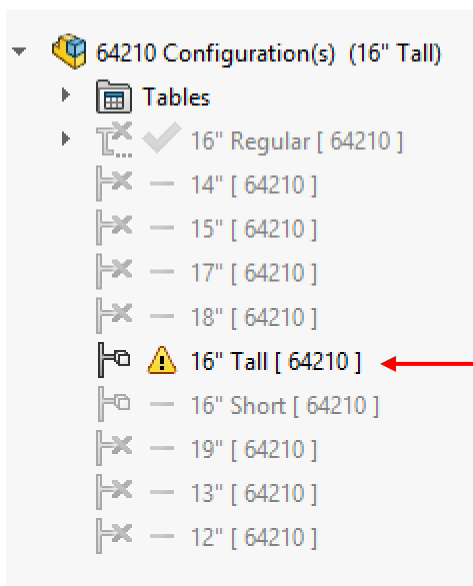


Figure 2 above displays the locations of fixtures that will be used in the SolidWorks simulation model. The fixtures are going to be assigned at the locations of all 4 wheels. Due to the way the wheelchair is expected to flex, an external “pseudo” part will need to be designed to allow the flex to occur, the way the wheelchair is expected to flex can be seen below in Figure 3.

Expected flex of the wheel axles is to occur in such a direction represented by the arrows above. Therefore the “fixed” fixtures in the simulation cannot be applied, nor will the rollers represent this well, so as stated earlier a “pseudo part” that does not affect the simulation will need to be designed in order to allow this flex to occur.

Figure 4 above shows where we plan to apply the strain gauges during the physical validation. We are planning to apply them on both sides of the expected location of highest stress/strain. Which is where the footrest piece is inserted into the frame and clamped. This can be seen closer in Figure 5 below.

Figure 6 above represents the feature of each wheelchair model that is selected for users according to their physical size. The configuration is highlighted in blue above in Figure 6. Below you can see the configuration we chose because this matches the physical wheelchair we have been given to do the physical validation for. Although our physical chair frame is not 16" Tall so 3" will be required to be added to widen the required tubes.



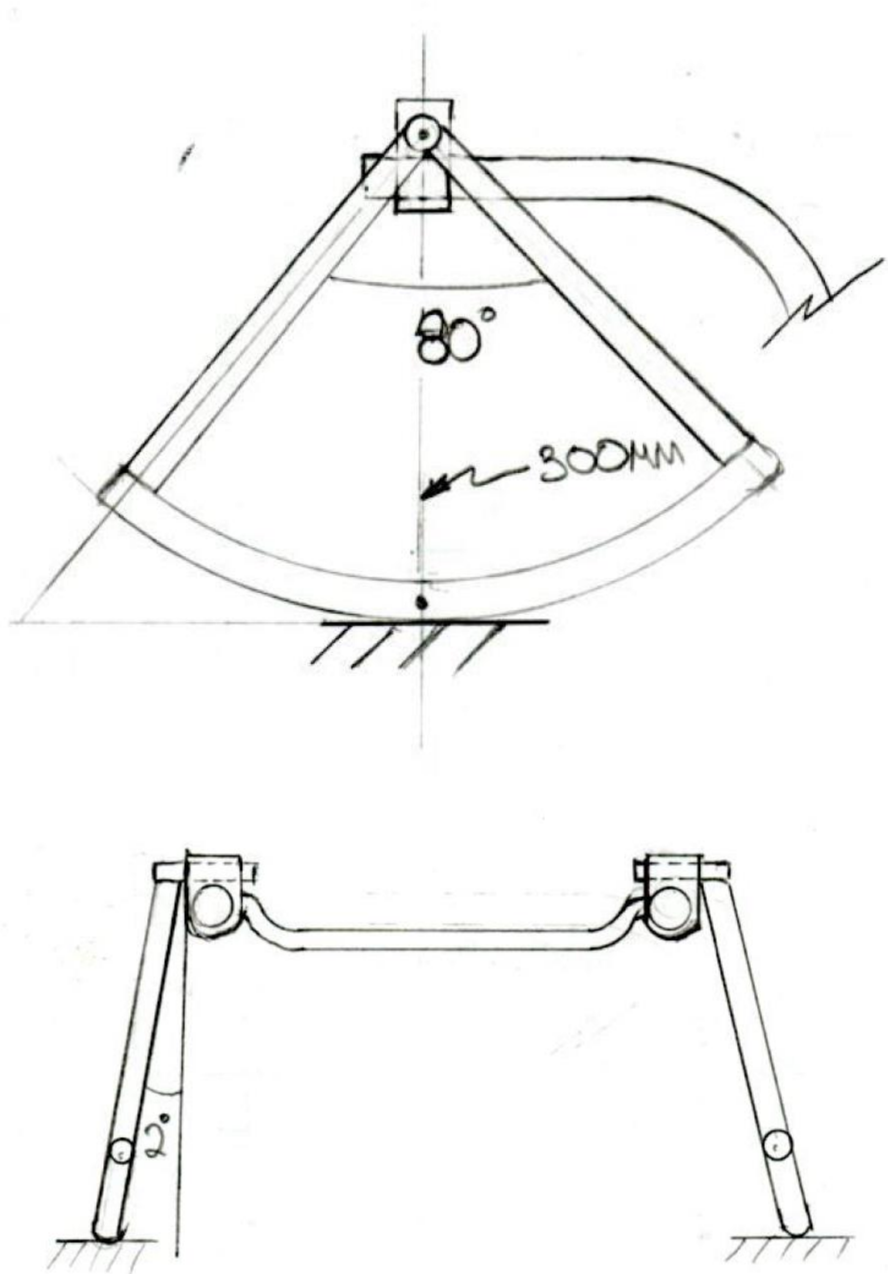


Figure 8 above represents one of the two ideas for the jig device to be manufactured and utilized in physical testing. This design is made from round tubing and is meant to mimic real wheels but remain fully rigid. Round tubing allows camber flex from the frame when a weight is applied as well as the freedom of some front and back motion. This design will allow for a rigid fixture permitting a more accurate inspection of the stresses on the frame.

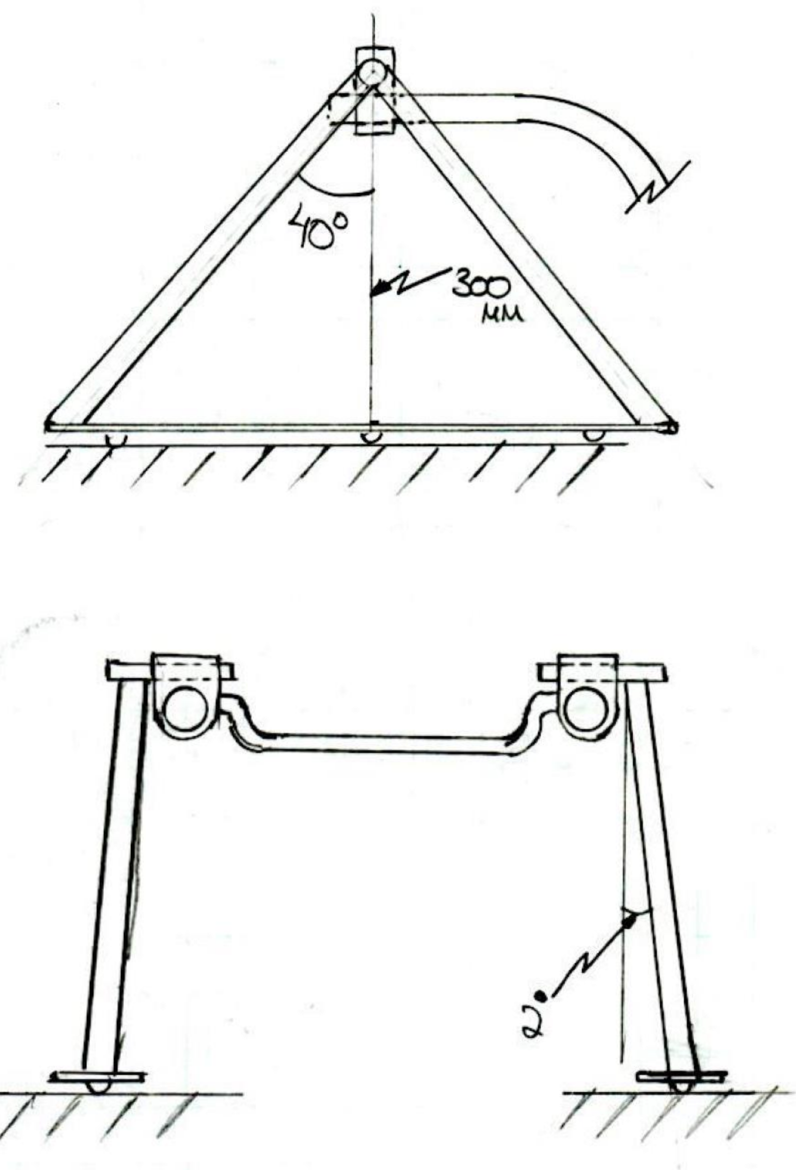
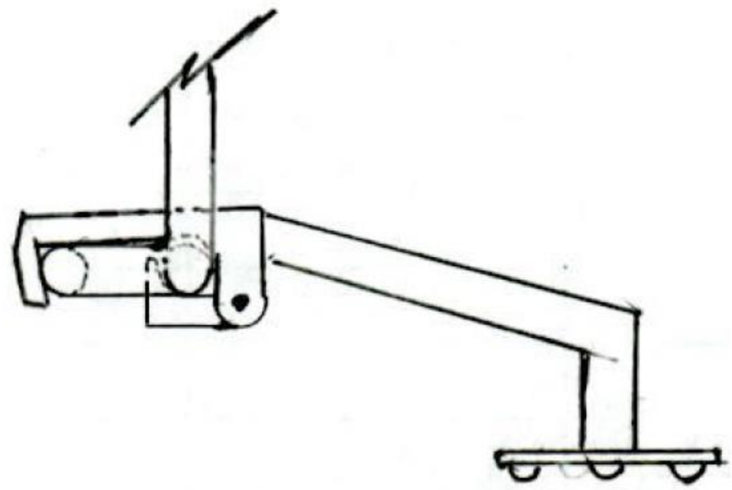


Figure 9 above is the second design option for the frame jig. The characteristics of this design is similar to the design mentioned above in Figure 8: this will provide a rigid fixture allowing for more accurate testing of the stresses on the wheelchair frame. This design differs by using square tubing for legs and a flat, triangle-like bottom providing more stability. Initial angle to mimic the 2 degree camber will be in the design, but roller bearings will also be used for flexibility, they are pictured in Figure 11 below.



Above in Figure 10 is the design of the front-end attachment jig. The clamping mechanism and its dimensions will be manufactured to represent the actual attachment in order to preserve the angles of forces and moments acting on the chair frame. Transfer bearings will be used in the front plate to allow motion between the fixture and the ground as the real wheel would have, resulting in a reproduction of the stresses caused by the actual front-end attachment but under a controlled test environment.

APPENDIX H: Request for Proposal for Wheelchair Front End Attachment Stress Analysis

Request for Proposal for Wheelchair Front End Attachment Stress Analysis

October 24th, 2017
Dr. Jaimie Borisoff
Rehabilitation Engineering Design Lab
4355 Mathissi Pl, Burnaby, BC V5G 4S8

Dear Sir or Madam:

Our organization is accepting proposals from Engineering firms to provide a detailed stress analysis for front-end attachments of wheelchairs. We invite your firm and associates to submit a proposal to us by November 7th, 2017, for consideration. A description of our organization, the services needed, and other pertinent information follows:

Background of Rehabilitation Engineering Design (RED) Lab

The Rehabilitation Engineering Design (RED) Lab is a portion of BCIT aims to develop novel solutions that improve and expand activities of daily living and community participation for people with disabilities. As much as possible, we engage users in our design process in order to create solutions that meet user needs. The REDLab works with a diverse, inter-disciplinary network of researchers (e.g. Rehabilitation Engineering, Occupational Therapy, Mechanical Engineering, Human Kinetics an others) as well as industry and community partners. The research program focuses on studying and developing technologies that positively impact a person's ability to interact with the built and natural environments, and/or promote health and community participation for those with disabilities. This research entails 1) studying the issues, barriers, and gaps faced by people with spinal cord injury and other mobility impairments; and 2) developing new technologies to help mitigate these issues.

Services to Be Performed

1. Develop a finite element model of a wheelchair or handbike frame when fitted with a front-end attachment device.
2. The model will be used to investigate static and dynamic loads that these front-end attachments place on wheelchair frames.
3. A validation of the model with physical experiments using strain gauges and/or load cells to study real-life stresses on wheelchair frames.

Key Personnel

Following are key contacts for information you may seek in preparing your proposal:

Dr. Jaimie Borisoff   Research Chair   Jaimie_Borisoff@bcit.ca

Requests for additional information, visits to our site, and/or appointments with any members of the REDLab should be coordinated through Dr. Jaimie Borisoff. You may reach him at the e-mail listed above. Please return the completed proposal to his e-mail provided above as well.

Deliverables of Project

- Problem analysis, including consideration of different wheelchairs and devices
- FEA modeling of stress analysis with loading
- Physical validation of FEA results

Your Response to This Request for Proposal

In responding to this request, we request the following information:

1. Detail your experience in designing finite element models, as well as your previous experience with running Finite Element Analysis for static and dynamic loads.
2. Explain your relative experience with hands-on use of equipment used to measure stress and strain in real life situations.
3. Provide information on whether you provide services to any related companies or groups.
4. Describe how you will approach the services we require and if there will be any interaction with external companies.
5. Explain if you will need external/internal funding for non-research related activities.
6. Identify the people who will be assigned to our job if you are successful in your proposal, and provide biographies. Indicate any complaints against them that have been leveled other associates, if any. Indicate any corrective actions that have been taken by the firm with respect to these people.
7. Set forth your fee proposal for the requested deliverables with whatever guarantees can be given regarding increases throughout the life of the project.
8. Provide the names and contact information for other, similarly sized, previous clients for reference purposes.
9. Describe how and why your firm is different from other firms being considered, and why our selection of your firm is the best decision we could make.

Evaluation of Proposals

REDLab will evaluate proposals on a qualitative basis. This includes our review of the firm's related materials, interviews with senior engagement personnel to be assigned to our organization, results of discussions with other clients, and the firm's completeness and timeliness in its response to us.

Please submit your response to this request for proposal by November 7th, 2017. We would also appreciate a response if you decline to submit a proposal.

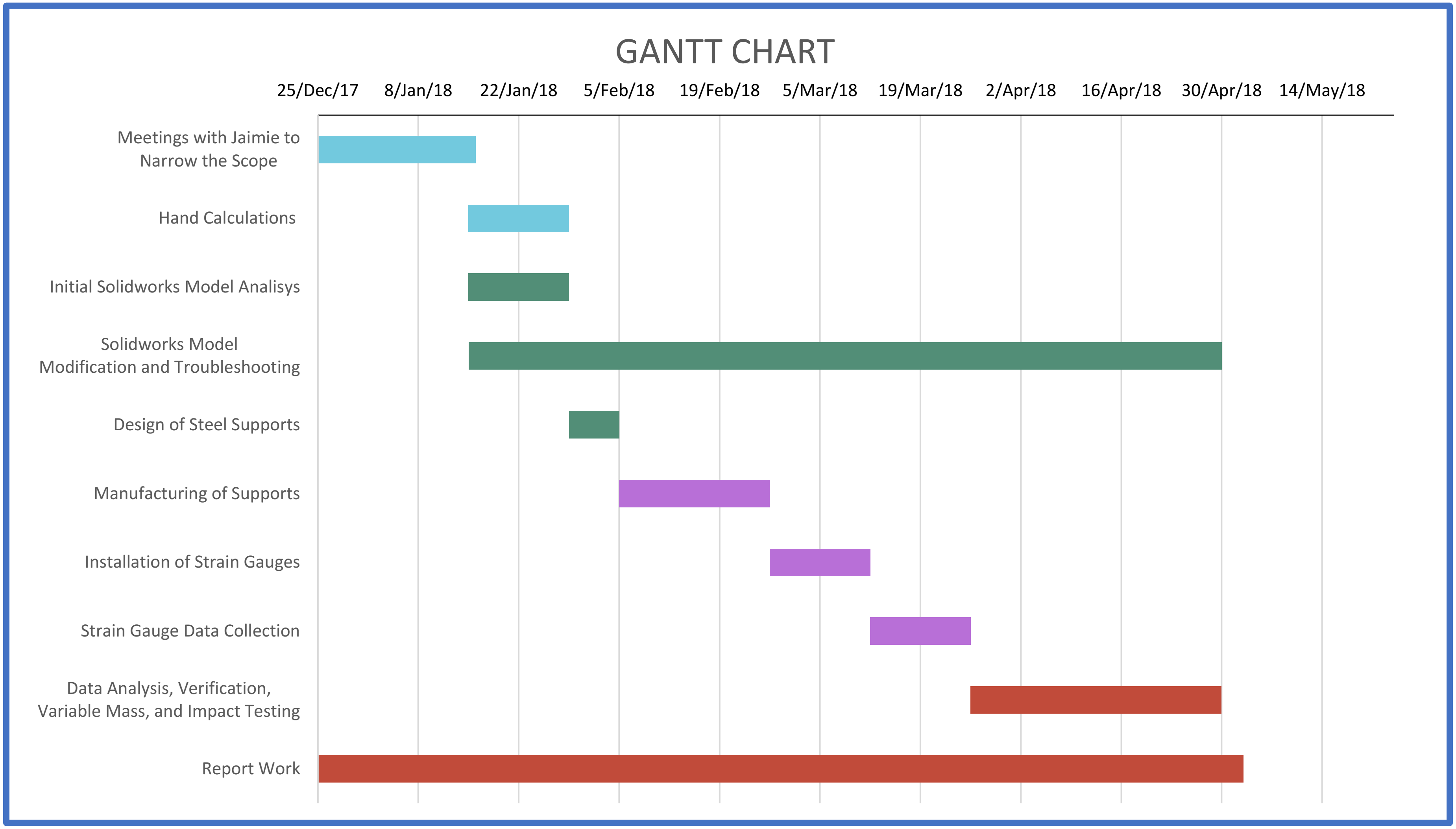
Sincerely,

Dr. Jaimie Borisoff
Research Chair of REDLab

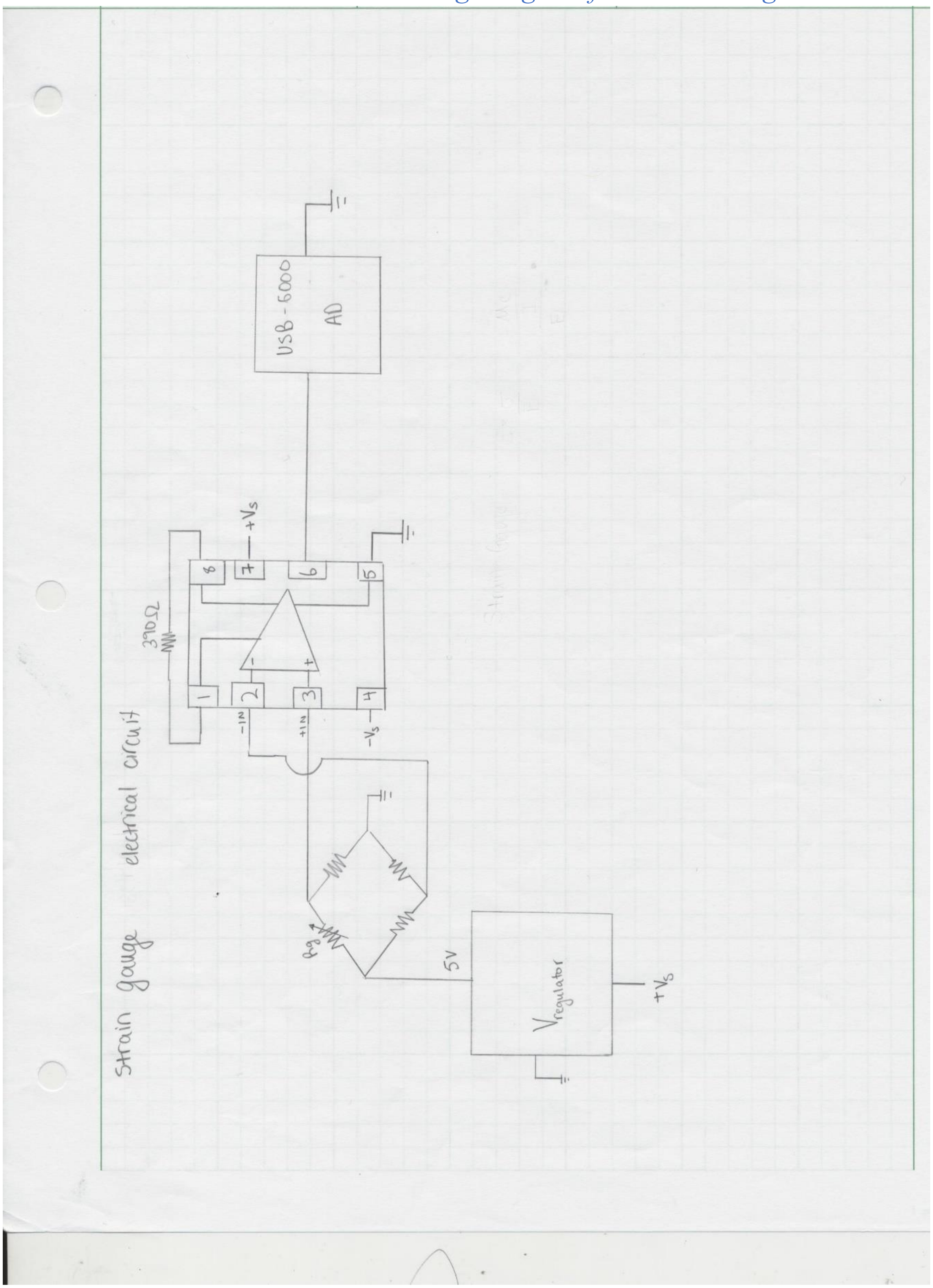
APPENDIX I: Responsibility Assignment Matrix

RCI Chart	**Person**		
Activity	**Karan**	**Artem**	**Dr.Jamie Borisoff**
Wheel Chair Modelling	Responsibility	Consult	Consult
Fatigue Analysis	Consult	Responsibility	Inform
Stress Analysis	Consult	Responsibility	Inform
Strain Analysis	Responsibility	Consult	Inform
Physical Strain Test	Responsibility	Responsibility	Inform
Device Modelling	Responsibility	Responsibility	Inform
Design and Manufacturing of Stand	Consult	Responsibility	Inform

APPENDIX J: GANTT CHART

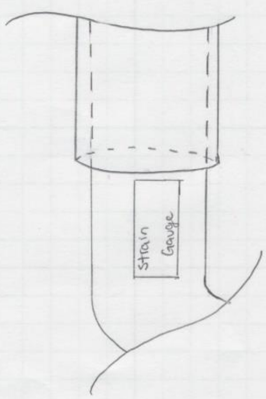


APPENDIX K: Initial Electrical Wiring Diagram for Strain Gauge



APPENDIX L: Initial Strain Gauge Measurement To Force / Stress Conversion

$$\varepsilon = \frac{\sigma}{E} = -\frac{\frac{Mc}{I}}{E}$$



$$I_{\text{hollow cylinder}} = \frac{\pi}{4}(r_2^4 - r_1^4)$$

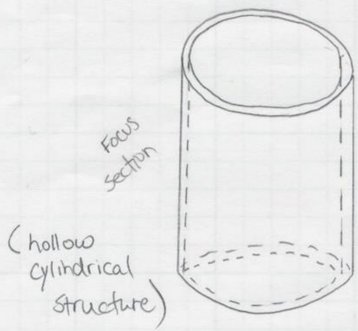
$$= \frac{\pi}{4}(R^4 - r^4)$$

inner radius = r

outer radius = R

$c = R$

$$\therefore \varepsilon = \frac{-4MR}{\pi E(R^4 - r^4)}$$



$$\varepsilon = \frac{-4MR}{\pi E(R^4 - r^4)}$$

$$M = F \cdot L = F(x - L)$$

refer to next page for x and L

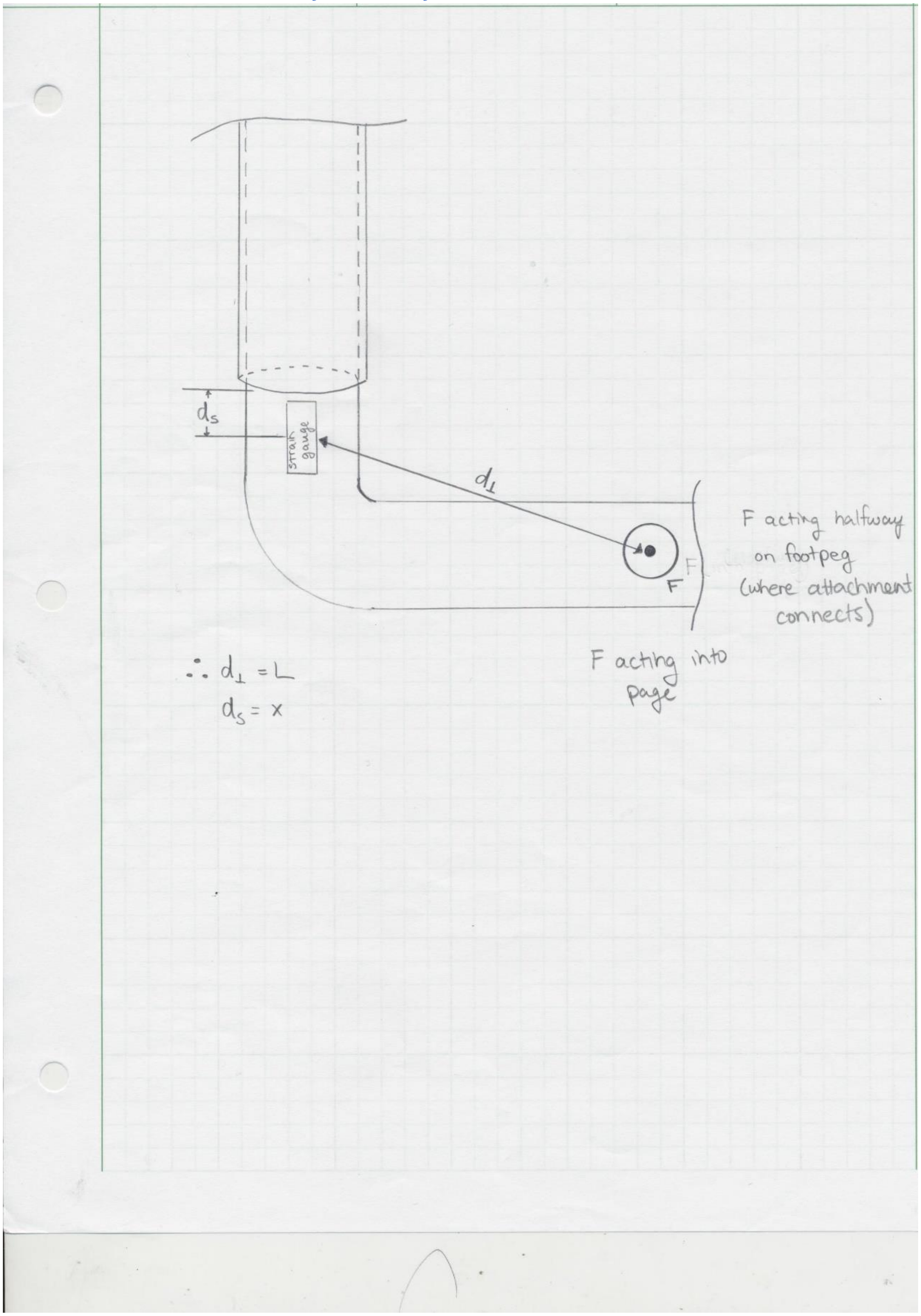
measured by S.G. → $\varepsilon = \dfrac{-4F(x-L)R}{\pi E(R^4 - r^4)}$

$$\therefore F = \frac{\varepsilon \pi E(R^4 - r^4)}{-4(x-L)R}$$

$$\sigma = -\frac{Mc}{I} = \frac{-4F(x-L)R}{\pi(R^4 - r^4)}$$ = bending stress at location

compare to σ_y and find S.F.

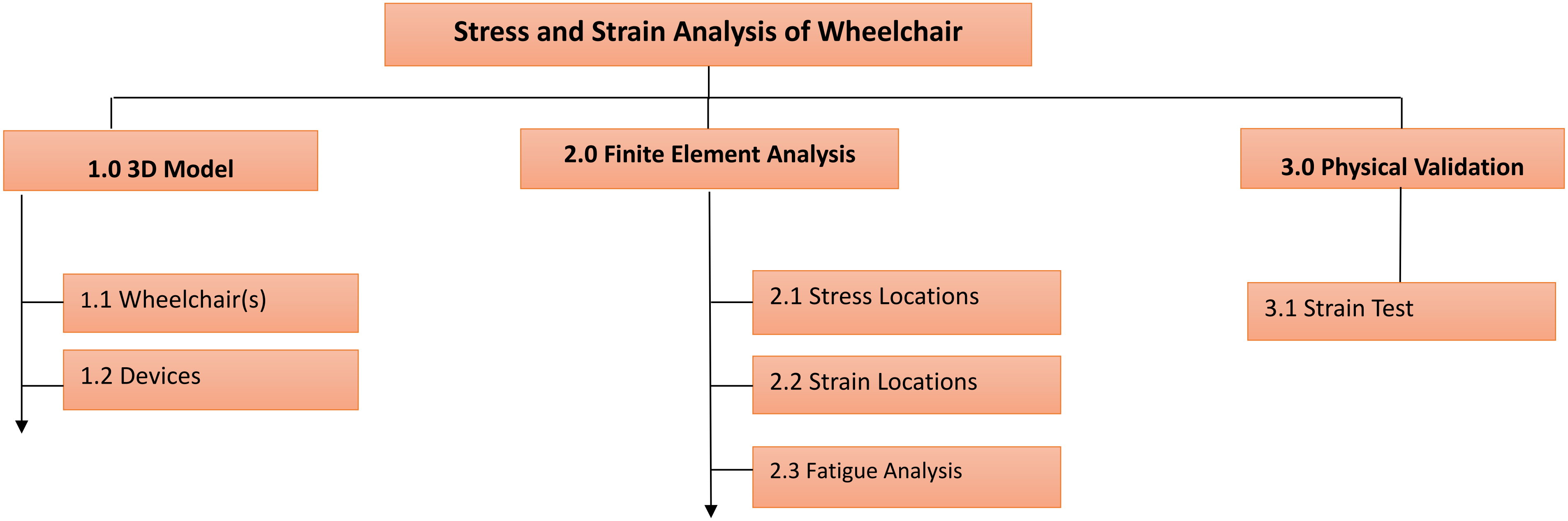
APPENDIX M: Initial Definition of Variables



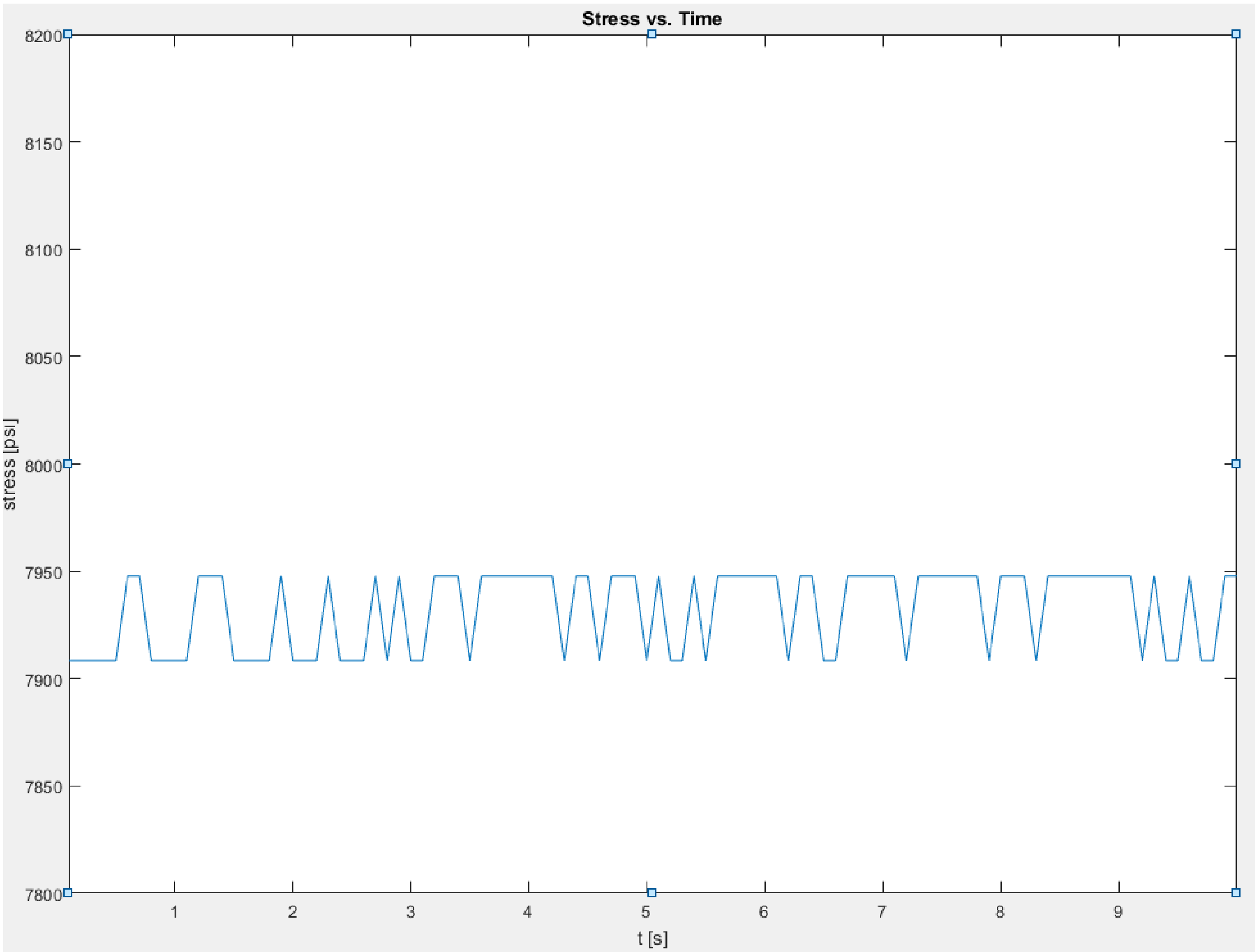
APPENDIX N: Testing Code

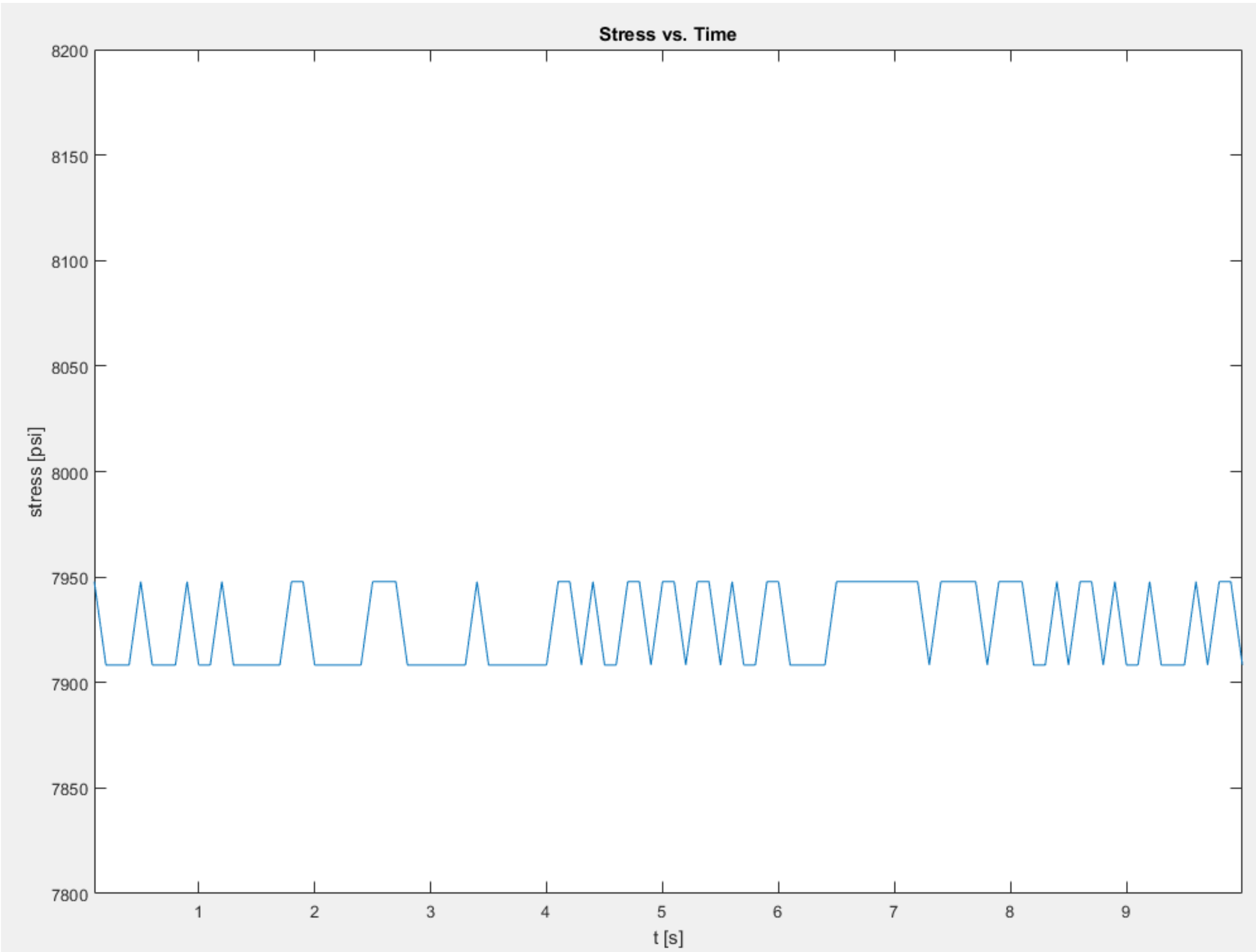
```
clear
clc
Availabledevices=daq.getDevices;
% Lists the available DAQ devices on the system
s=daq.createSession('ni');
Rate=10;
s.Rate=Rate;
%acquires # of samples per second
s.DurationInSeconds=10;
%How long the DAQ will collect data for
% s.IsContinuous=true;
addAnalogInputChannel(s, 'Dev1' ,2, 'Voltage' )
s.addlistener('DataAvailable', @(src,event) plot(event.TimeStamps,
event.Data));
data = startForeground(s);
s.wait();
sr = 10;
dt = 1/sr;

E = 10007603; %psi
zero = 0;

vIn = 5; %V
G = 517.8205128; %Gauge Factor
sg = 2.06; %Strain Gauge conversion (from spec sheet)
n = size(data,1);
time = zeros(n,1);
strain = zeros(n,1);
final = zeros(n,1);
time(1,1) = dt;
for i = 2:n
time(i,1) = time(i-1,1) + dt;
end
for i = 1:n
data(i,1) = data(i,1);
end
for i = 1:n
strain(i,1) = (4*(data(i,1)))/(G*sg*vIn) - 3.84e-4;
end
for i = 1:n
final(i,1) = E*strain(i,1) - zero;
end

final(:,2)=(final(:,1));
plot(time(:,1),final(:,2))
title('Stress vs. Time');
axis tight
xlabel('t [s]');
ylabel('stress [psi]');
```

APPENDIX O: Work Breakdown Structure



APPENDIX P: Further (Iterative) Results from Strain Gauges for Confirmation




Stress vs. Time
stress [psi]
8200
8150
8100
8050
8000
7950
7900
7850
7800
1
2
3
4
5
6
7
8
9
t [s]