Finite Element Analysis of Contra Bass Bridge

by

Nurul Fatin Syazwani Binti Azman

11513

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

MAY 2012

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Finite Element Analysis of Contra Bass Bridge

by

Nurul Fatin Syazwani Binti Azman

11513

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

Approved:

____

AP DR BAMBANG ARIWAHJOEDI

Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

MAY 2012

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

(Nurul Fatin Syazwani Binti Azman)

ABSTRACT

Double bass is a string type instrument that has been evolved each generation due to the increased demand for better quality sound. A lot of amplifiers and transducers were used to transferring and amplifying the original sound of the contra bass but the most important is to maintain its original acoustic timbre of wood-bass characteristics. The main objective of this project is to determine stress concentration area on the bridge of contra bass in the proportion to propose the best mounting position of the sound transducer. Based on author's research and background study in FYP I, the acoustic timbre characteristic would be improved by embedded the sensor at the location on the bridge with maximum stress concentration. To determine the location, the author has chosen Finite Element Analysis using ANSYS among some available methods due to its high feasibility. CATIA: Sketch Tracer also being used in conjunction with ANSYS in the development of the bridge solid modeling. Stress concentration area on the bridge was successfully determined at the end of the project and all related analysis, results and discussion had been compiled in the thesis.

ACKNOWLEDGEMENT

First and foremost, thanks to Allah S.W.T for letting me live my life, now at least for 22 years. With His blessing, I am able to complete my Final Year Project with full strength and great health. It is my pleasures to thank all individuals that have directly and indirectly made this project a success.

My next utmost gratitude goes to my supervisor, AP Dr Bambang Ariwahjoedi for his continuous guidance and support throughout this project. I am very thankful to him for his words of motivation and encouragement from the beginning to the final level of this project. He assisted me, gave suggestions, solutions and recommendation for me to accomplish this project. Although I was sometimes, lost my focus, my supervisor will be the one who will bring me back to the right path. I am totally grateful and proud to have him as my supervisor.

I would also like express my greatest appreciation to Mr Munir Lubis and Mr Mohaimin Mokhlis for their willingness to share their knowledge throughout the completion of the project. They have been really helpful in giving advices and guidance for the software used in this project.

My appreciation also goes to my beloved parents and family who always been there for me, supporting me. Thanks for their encouragement, love and emotional support that they had given to me. Without them, my project would not have finished on time.

Also, I would like to thank my course mate, Khairunnisa Abu Bakar for helping me whenever I have problem with the project. She always reminds me to focus and work on the project whenever she realized I was sitting back for too long. Having a friend with similar background of project provides advantages of team cooperation and we were able to sit together discussed the same issues.

Last but not least, I want to thank the university, Universiti Teknologi PETRONAS for providing me the facilities throughout the project completion. Besides, having Final Year Project had given me chances to do analysis, researches and apply software learnt in class.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

UTP	Universiti Teknologi PETRONAS
FEA	Finite Element Analysis
FYP	Final Year Project

CHAPTER 1

INTRODUCTION

1.1 Background Study

The revolution of musical instrument has been started thousand years back as old as the history of man civilization (Neville, 2000). Music can be considered as one way communication to express feelings and to convey messages. As time goes by, the demand on the music field also been increasing parallel with the transformation of new era. In this project, it focussed on the need of sound amplification of contra bass. There are increasing in number of spectators and fans which introduced function of sound amplification. Many types of amplifier such as microphone, magnetic transducer and many more were made. Apart from that, there also some come up with new instrument with built-in amplifier. This new modern type of instrument may be richer in sound quality and superior in responsiveness, but if it does not really maintain the original sound then it is considered not a better instrument (Neville, 2000). In the production of acoustic timbre of wood contra-bass characteristics, the good understanding on the mechanism, acoustic characteristics and deep view from physics side of music instrument would provide details of sound production of an instrument.

Among the available method in engineering field nowadays, the author has chosen Finite Element Analysis in order to determine the best mounting position of the sound transducer on contra bass's bridge. The positioning of sound transducer on the highest stress concentration area may be able to produce superior sound which still maintains the timbre characteristics of contra bass. Proper procedures should be followed in order to achieve the objectives.

1.2 Problem Statement

The need of amplification has been very important in music industrial nowadays. A lot of amplifiers, pickups and transducers were come into the role of transferring and amplifying the original sound of the contra bass. The important part to be deal with is to maintain the acoustic timbre of wood-bass characteristics. Miller & McIntire, 1987 say that "Acoustic emission is the class of phenomena where transient elastic waves are generated by the rapid release of energy from localized sources within a material, or the transient waves so generated" (Miller, 1987).

Bridge is the part of these instruments which will be further explained in Chapter 2. The significant of the project is to maintain the timbre characteristics of the sound produced by contra bass and propose the best mounting position of sound transducer which are cost effective and having high efficiency of amplification.

Nowadays, there were many sound transducers introduced. They are producing superior sound than contra bass without sound transducer. The problem arises when improper positioning of the sound transducer may cause sound instability as for example, the used of microphone may include external sound which are not originated from contra bass. It called external disturbance. Apart from that, improper positioning may cause discomfort to the player when playing the contra bass. As for example, some sound transducer is located at the body of the contra bass as the wiring will cause inconvenient playing mode to the player.

The significant of the project is to find the area with the highest stress concentration which the author believes will be the perfect location to embed the sound transducer and able to reproduce the acoustic timbre of wood-bass sound characteristic. The project is cost effective as it only deals with the analysis and simulation of stress distribution on the bridge. Old century bass player will not have to buy new design of contra bass as they only have to own a bridge embedded with sound transducer which located at the highest stress concentration area.

1.3 Objectives and Scope of Study

There are several objectives that need to be achieved at the end of the project which are:

- To perform Finite Element Analysis in the market to analyse complex shape of the bridge.
- To determine the best mounting position of sound transducer.

In order to analyse the 3D solid modelling in ANSYS, first the author need to translate the complex shape of the bridge into digital model. The scope of the project is to determine the area with the highest stress concentration on the bridge. Upon the determination of the stress location, software such as CATIA and ANSYS will be applied.

The project is relevance to be done as music is a part of life. The music lovers are everywhere and this proves that the project would have a positive feedback from them. The project applied the knowledge in using CATIA and ANSYS. Basic knowledge in design is needed. Apart from that, main method used to determine the best mounting position of sound transducer is using Finite Element Analysis which is the most popular used method in engineering field which it provides various types of analysis such as Von Misses Stress Analysis, Thermal Analysis and many more.

The author has been given at about eight (8) months for the project accomplishment. It is believed that the author will manage to finalize the project within given timeline as the author will be only focussing on the analysis and simulation part of the project which is determining the best mounting position of sound transducer using Finite Element Analysis.

CHAPTER 2

LITERATURE REVIEW

In order to understand and to achieve the desired objectives of the project, thorough reading regarding contra bass and also factors that manipulate the characteristics of sound production should be done. Hence, comprehensive and depth research for resources are needed for the accomplishment of the project. Literature review is prepared to provide the author better understanding on essential parts of the project.

2.1 Contra Bass

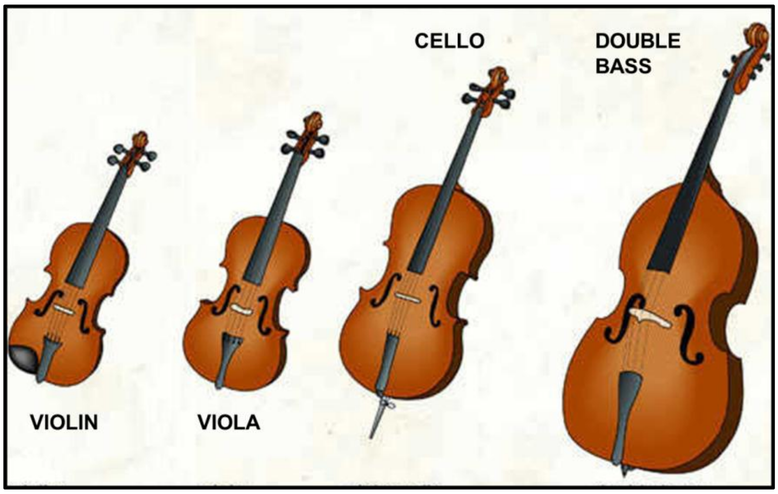


Figure 1: Size difference of Viols Family Instruments

Contra Bass also known as upright bass, string bass, double bass and many more is regarded as the largest instrument in viols family instrument as can be referred as Figure 1. It has four strings and each tuned at E_1 (41.2 Hz), A_1 (55.0 Hz), D_2 (73.4 Hz) and G_2 (98.0 Hz). While the cello, which also have four strings but each tuned at C_2 (65.4 Hz), G_2 (98.0 Hz), D_3 (147.0 Hz) and A_3 (220.0 Hz). Cello is the second largest instrument in string family which is about twice of a violin (Neville, 2000) while violin is the smallest in the family. Double bass produce the low notes due to its largest body compared to others. The violin produces the highest pitch.

The major parts of this instrument can be referred in APPENDIX A which are Scroll, Peg Box, Nut, Fingerboard, String Roller Hole, Top part, Strings, the Bridge, F-Holes, Sound post and Tailpiece. Each part play important role toward the body vibrational frequencies. Another important part which is not included in these instruments body is the bow. The rate of quality of the bow can be measured according to its playing properties and its tonal qualities (Neville, 2000).

Other than that, the timbre or tone colors of stringed instruments are much dependent on various factors such as instrument stimulus spectrum, waveforms, sound pressure/loudness, pitch of tone, temporal characteristics and sustain duration (Pickering, 1997).

The motion of the top plate of the body which is the source of most of the bass sound is the results of a complex interaction between the driving force from the string through the bridge and the various resonances of the instrument's body. More specifically, the vibrational characteristics of bass wooden plate is determined by various factors such as size, shape and design, thickness, arching, density, mechanical properties (Haines, 2000) such as stiffness and internal damping which more or less relate its structure (Segerman, 2001) and inorganic content (Bucur, 2000), moisture content (Hutchins, 2000) (Thompson, 2000), relative humidity (Thompson, 2000), type of varnish and temperature. These are factors which might have been motivated ones to develop synthetic composite materials instead of wood musical instruments (Besnainou, 2000).

Vibrational resonances modes (Woodhouse, 2002) of stringed instrument body have become visually significant. It has been proven that tonal quality as well as timbre of an acoustic instrument such as the double bass will be very much dependent upon pitch (Rodgers, 2001), phase of each high frequency harmonic components (Plomp, 1969) and their variation in transient pattern with time (Fletcher, 1996).

Apart from the sole presence of body enclosed air (Bissinger, 2001) there is also influence of air-body coupling (Hutchins, 2000) which in effect subsequently alter the directivity patterns (Wang, 1999) of sound radiation (Weinreich, 2002) emanating from the instrument. Finally, the effect of long term cyclic deformation due to long term playing (Hutchins, 2000) or the age of the instrument is worth mentioning.

Focally for bowed stringed instrument; Double Bass, the acoustic characteristics is particularly affected by the bridge and mid portion of the top plate between f-holes (Jansson, 1997). Currently in this project, the author is going to focus toward the bridge, by assuming that it has the role as the major characteristic determining part of the whole vibrating system of the instrument. The details on the relation of the bridge in the production of acoustical characteristic will be explained later in the next sub-section.

2.2 Acoustical Characteristics and the Bridge

The definition of acoustic is the study of sound which generally refers to transmission of sound through solid and fluid media and any other phenomenon engendered by its propagation through media (Daniel, 2000). Daniel (2000) says that an acoustic signal can arise from a number of sources such as turbulence of air or any other gas, the passage of a body through a fluid, or the impact of a solid against another solid. The most common unit of frequency used in acoustic is Hertz. Significantly, acoustic is a matter of communication as the message is tried to be conveyed effectively by minimizing effect of noise (Daniel, 2000).

Figure 2: The Bridge of Contra Bass

Apart from that, the bridge (as per Figure 2) is then functioning as transforming the motion of vibrating strings into periodic driving forces applied by its two feet to the top plate of the instrument. Many studies had been done on the effect of the bridge to the sound and frequency production. Savart (1937) says that shaping the bridge is a convenient and effective way to alter the acoustic characteristic of the instrument (Savart, 1937). Bridge has a large number of vibration modes such as longitudinal, flexural and torsional vibrations which can be determined using Holographic interferometry (Neville, 2000).

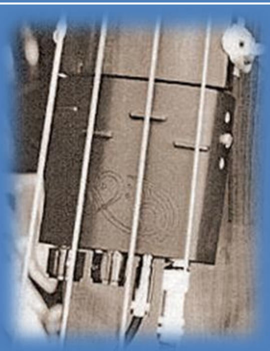
In another study, Bladier (1960) concluded that the bridge acts as an acoustical lever or amplifier with a gain of about 2 (6 dB) between 66 and 660 Hz, decreasing to 1 or below above 600 Hz (Bladier, 1960). The bridge is generally carved quite symmetrically due to aesthetic reasons (Neville, 2000). Referring to other findings, the misaligned of the bridge also may affect the sound produced (Joseph, 2012). Cutting of some part of the bridge may also contribute to tonal consequences. In conclusion, the bridge can be considered as an essential part in the role of producing the timbre wood-bass characteristics.

2.3 Sound Transducers

Sound transducer, similarly functions as sensor, amplifier or transducer is device used to capture mechanical vibrations and convert them into electrical signal that then can be amplified and recorded as required. There are basically two types of pickup which passive; which not requires any external source of power while active; as they need preamplifier and external source before directly transmitted to Main Power Amplifier.

Over centuries, amplifications technology has developed vastly. There are many types of transducers have been introduced to the world of music. Some of them can be referred in Table 1.

Table 1: Variation of Sound Transducers (Jonas, 2012)

TYPE OF TRANSDUCER	DESCRIPTION
 **Microphone**	Usually, this type of pickups will be mounted between bridge and fingerboard which will ensure the audio reinforcement is achieved when performing live. Disadvantage: Problem dealing with external noise.
 **Magnetic**	This type of pickup consists of permanent magnet. It is often mounted on the body of the instrument but also can be attached to the bridge. Disadvantage: Magnetic sensor causes the effect of magnetizing the sound and produce higher output sound.
 **Pickup Mounting Clamp**	This pickup is mounted to the body. Made up of rubber-like-plastic to prevent the clamp from buzzing. Disadvantage: Not feasible and does not consider aesthetic value.
 **Piezoelectric Ceramics**	Has been used widely for string family instruments due to its ability of not picking up any other magnetic fields.

2.4 Method to Determine Stress Concentration Area

The author has done thorough analysis for each related component whereby deep reading has been done in order to get the understanding on the project. Assisted by Supervisor, AP Dr Bambang Ariwah Joedi, the author has identified some available methods that can be used to obtain stress concentration on bridge surface. The method can be viewed as per Figure 3.

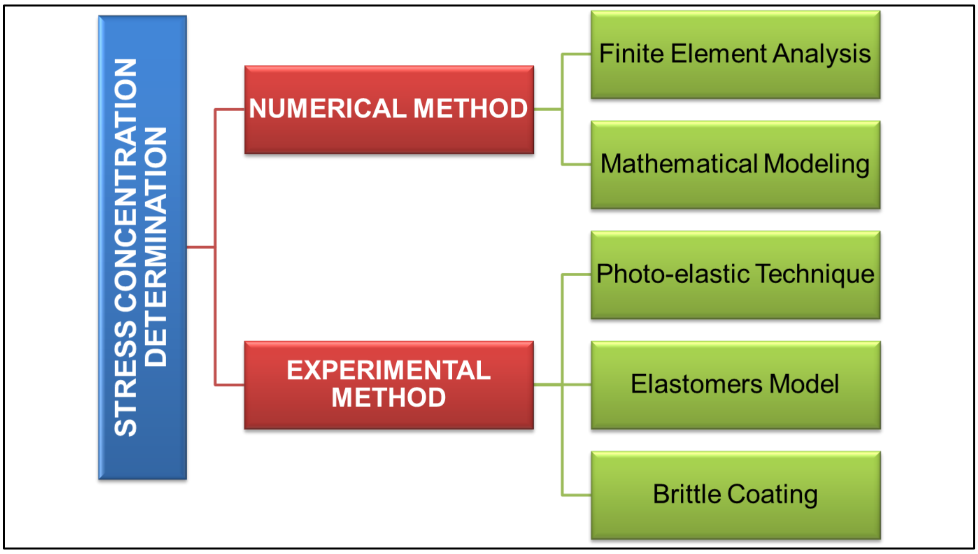


Figure 3: Possible Approaches in Determining Stress Concentration

Basically, there are two types of method that can be used in determining stress concentration which are analytical or experimental method. As if the experimental method is chosen, there are two ways in determining the stress concentration which are either use Photo-elastic Technique or Bridge Replication (Labossiere, 2007).

Photo-elastic Technique is one of the most powerful techniques which are able to identify stress concentration easily. It manages to gives overall picture of stress field showing region of stress intensification. Furthermore, the direction of principal stresses is also easily determined. This method requires a model of the shape analysed (the bridge) which is made from a suitable transparent material. The changes in properties when loaded in similar manner to the real part will be measured by polariscope. The phenomenon of synthetic birefraction is the basis of photo-elastic techniques (Labossiere, 2007).

Other option is using Elastomers Model. This method is usually chosen when the elastic behaviour is the main concern. Metal specimens are bad in demonstrating elastic behaviour and stress concentration effects due to their high stiffness. On the surface of the specimen, a square grid of lines is printed. While the specimen is in the loading frame, the shape of the square grid will be carefully observed (Labossiere, 2007).

The third option when choosing experimental method is using Brittle Coating. Labossiere (2007) said that this is the most straight-forward methods of experimental stress analysis. Much more sensitive brittle coating known as Spray Stresscoat is available nowadays with the ability to crack at strain levels as low as 400 $\mu m/m$. The Stresscoat is simply sprayed on to an average film thickness and dried for several hours. There is guideline provided to read the series of crack when the specimen is hold in special loading fixture (Labossiere, 2007).

Considering Analytical Method, the author would have two options which are using Finite Element Analysis or Mathematical Modelling. It is a tedious work to come up with formula and solve it manually. Even though software as Mat Lab is used, still it requires more effort in finding the stress concentration. In contrast, Finite Element Analysis provides much easier ways compared to the Mathematical Modelling. Finite Element Analysis (FEA) can be done using ANSYS software which will further explain in next subunit. All in all, FEA has been chosen as a way to determine the stress concentration point as it provides the easiest approach among others. Experimental Method may be implemented as recommended future work.

2.4.1 Finite Element Analysis

Finite Element Analysis (FEA) is dividing the complex objects into much simpler and manageable elements (Liu, 2003). It can be applied in computer simulation such as ANSYS. 3D Elasticity Theory in Mathematical Modelling method is able to analyse stress concentration area of a 3D object as shown in Figure 4.

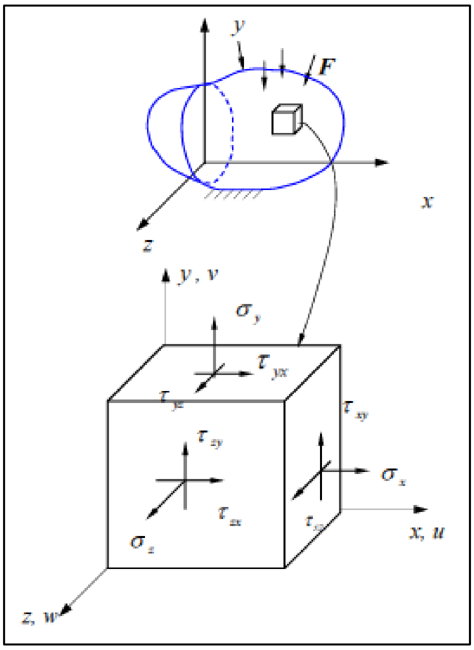


Figure 4: Stress State for 3D Problem (Liu, 2003)

Stress:

$$\boldsymbol{\sigma} = \{\boldsymbol{\sigma}\} = \begin{Bmatrix} \boldsymbol{\sigma_x} \\ \boldsymbol{\sigma_y} \\ \boldsymbol{\sigma_z} \\ \boldsymbol{\tau_x} \\ \boldsymbol{\tau_y} \\ \boldsymbol{\tau_z} \end{Bmatrix}, \text{ or } [\boldsymbol{\sigma_{ij}}]$$

Strains:

$$\boldsymbol{\varepsilon} = \{\boldsymbol{\varepsilon}\} = \begin{Bmatrix} \boldsymbol{\varepsilon_x} \\ \boldsymbol{\varepsilon_y} \\ \boldsymbol{\varepsilon_z} \\ \boldsymbol{\gamma_x} \\ \boldsymbol{\gamma_y} \\ \boldsymbol{\gamma_z} \end{Bmatrix}, \text{ or } [\boldsymbol{\varepsilon_{ij}}]$$

Stress-Strain Relation:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_x \\ \tau_y \\ \tau_z \end{Bmatrix} = \frac{E}{(1+v)(1-2v)} \begin{bmatrix} 1-v & v & v & 0 & 0 & 0 \\ v & 1-v & v & 0 & 0 & 0 \\ v & v & 1-v & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2v}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2v}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2v}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_x \\ \gamma_y \\ \gamma_z \end{Bmatrix}$$

$$\sigma = E\varepsilon \; (Eq. 1)$$

Solving equation 1 with strain displacement relation and element equilibrium equations under boundary conditions will lead to stress, strain and displacement analysis but note that analytical solutions are difficult to solve. In contrast, using Finite Element Formulation, stress will be evaluated based on selected points including nodes on each element (Liu, 2003). Available FEA software are ANSYS, ABAQUS, HyperMesh, COSMOS and many more. Among them all, ANSYS is one of the software which is available in Universiti Teknologi PETRONAS and it is feasible to use it. Figure 5 shows the steps in the ANSYS FEA Procedure.

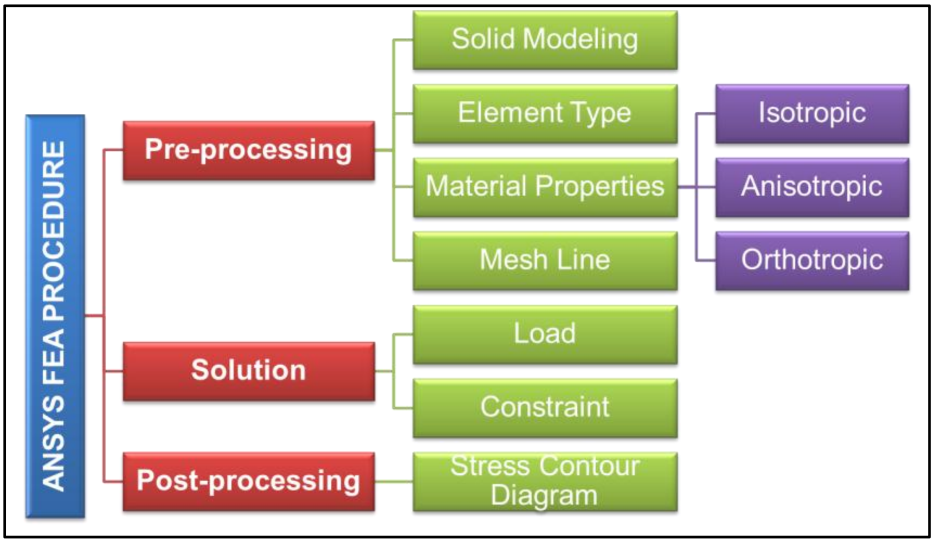


Figure 5: FEA Computer Implementation

At the end of Final Year Project, it is expected that the author will be able to generate stress concentration field on the surface of the bridge.

According to Figure 5, there are three (3) main stages:

1. **Pre-processing**

   i. **Solid Modelling**

   Referring to ANSYS terminology, solid modelling or often called model generation means the process of defining the geometric configuration of the model's nodes and elements. In ANSYS, it provides some approaches in order to produce 3D modelling which are:

   - Creating a solid model within ANSYS.
   - Using direct generation.
   - Importing a model created in a computer-aided-design (CAD) system.

   The author has chosen step 3 to draw 3D modelling of the bridge. Detailed procedures in generating the model will be discussed in Chapter 3: Methodology.

   ii. **Element Type**

   The author has to specify the element type that is going to be analysed. In the Element Library, it includes a summary table of element input. Element type is identified by a name which consists of a group label and unique identifying number. In this project, the most suitable element is **SOLID285**. Usually, SOLID285 is applicable for 3D modelling of general solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y and z directions. The element has the function of plasticity, hyper elasticity, stress stiffening, creep, large deflection and large strain capabilities. The effects of pressure load stiffness are automatically included in this element.

iii. **Material Properties**

In the function of defining Material Properties, the author needs to define material properties, material models and model combinations. The bridge is made of **Maple, sugar (Acer saccharum)** which is having Young Modulus of $\mathbf{1.26 \times 10^{10}}\ \boldsymbol{N/m^2}$.

iv. **Mesh Line**

Meshing is the next step after defining element attributes. ANSYS offers a large number of mesh controls though as for the project the author chose free mesh for the complex shape of the bridge. Free mesh has no restriction in terms of element shapes and has no specified pattern applied.

2. Solution

i. **Load**

Load can be applied on the solid model whether on key points, lines or areas. The solver will treat all loads to be in terms of the finite element model. Any specified loads on the model will automatically transferred to nodes and elements at the beginning of solution. In the project, there will be three (3) different cases which are:

CASE 1

Static bridge: the force is coming from the tension of the string.

CASE 2

CASE 2(a): Only string G is bowed to the right.

CASE 2(b): Only string D is bowed to the right.

CASE 2(c): Only string A is bowed to the right.

CASE 2(d): Only string E is bowed to the right

CASE 3

CASE 3(a): Only string G is bowed to the left.

CASE 3(b): Only string D is bowed to the left.

CASE 3(c): Only string A is bowed to the left.

CASE 3(d): Only string E is bowed to the left.

String tensions are applicable in each case. The only difference is the addition of the force acting due to bowing action to the left and to the right. Additional forces from bowing action to the left and right are vary at about **1.5 N** (Neville, 2000).

Figure 6: String Tension Holds the Bridge from Slip

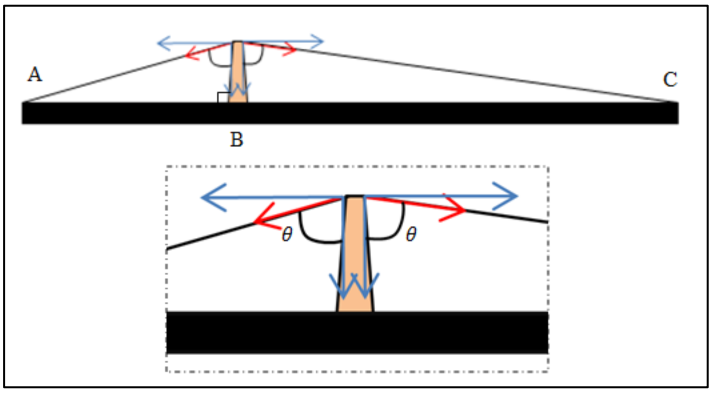


Figure 7: Free Body Diagram of Unbowed Bridge

Figure 6 shows the position of the bridge on the bass body and Figure 7 shows free body diagram when the bridge in static position (only string tensions are acting on the bridge). Downward force string tension will be determined from the average value of string tension as per Table 2.

Table 2: String Tensions (BAEN, 2007)

String Type	G	D	A	E
Pirastro Eudoxa	295.04	291.03	284.36	289.25
Pirastro Flexocore	299.04	295.04	321.74	291.92
Pirastro Obligato Orchestra	291.92	288.36	290.14	285.25
Pirastro Oliv	281.24	270.56	262.55	266.56
Pirastro Permanent	295.93	290.14	286.14	280.35
Thomstik-Infield Precision Orchestra 127	313.73	313.73	343.10	343.10
Thomstik-Infield Spirocore Soft (Weich) S42	264.78	284.36	294.15	303.94
Thomstik-Infield Spirocore Medium (Mittel) S42	294.15	313.73	333.75	333.75
Average	**291.98**	**293.37**	**301.99**	**299.26**

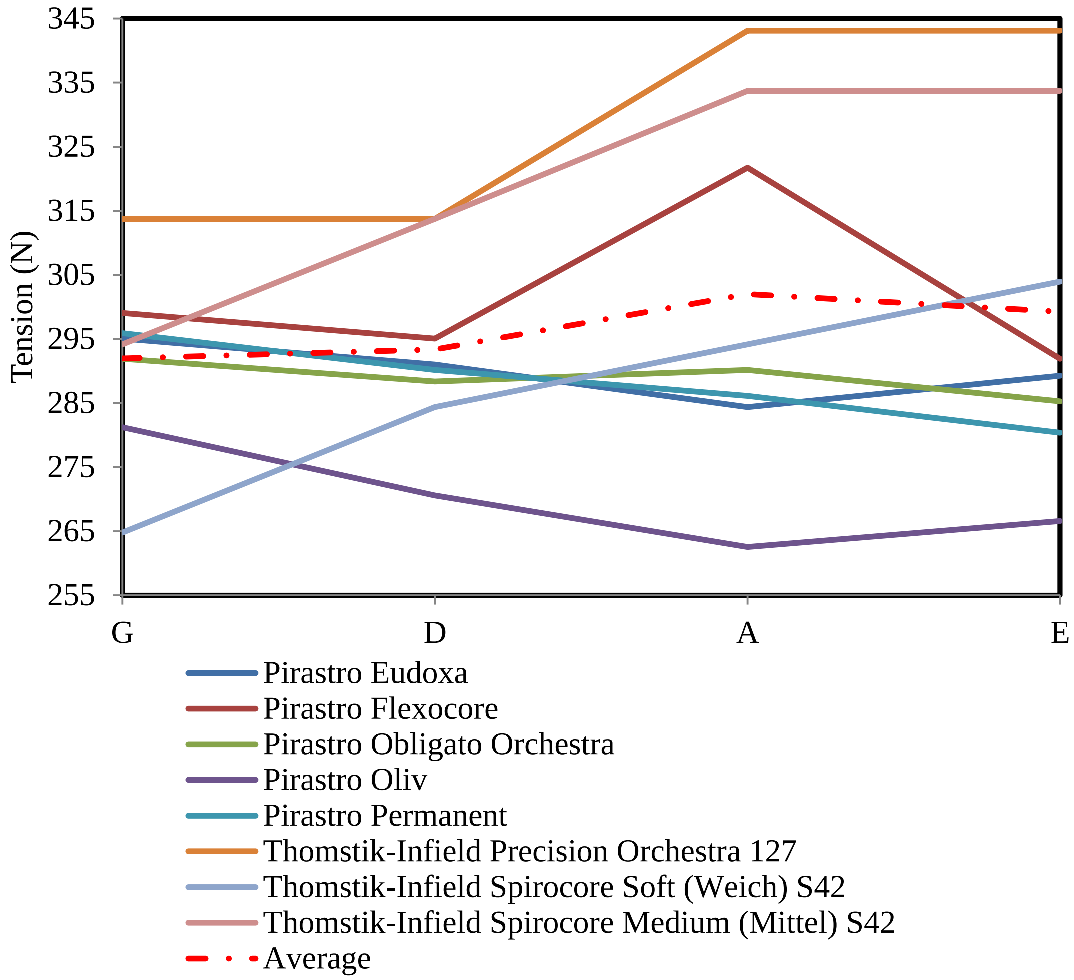


Figure 8: String Tensions Chart

The author will apply the average string tension on the bridge model in ANSYS.

ii. Constraint

At the bottom of the bridge, there will be a fix displacement force acting which avoid the bridge from slipping.

3. Post-processing

Based on the project, one of the objectives is to determine the best mounting position of the sound transducer which believes on the highest stress concentration area. Due to that, the author interest is to obtain stress analysis through stress contour diagram. Detailed methodology and results of the analysis will be further explained in the next chapter.

CHAPTER 3

METHODOLOGY

3.1 Research Methodology

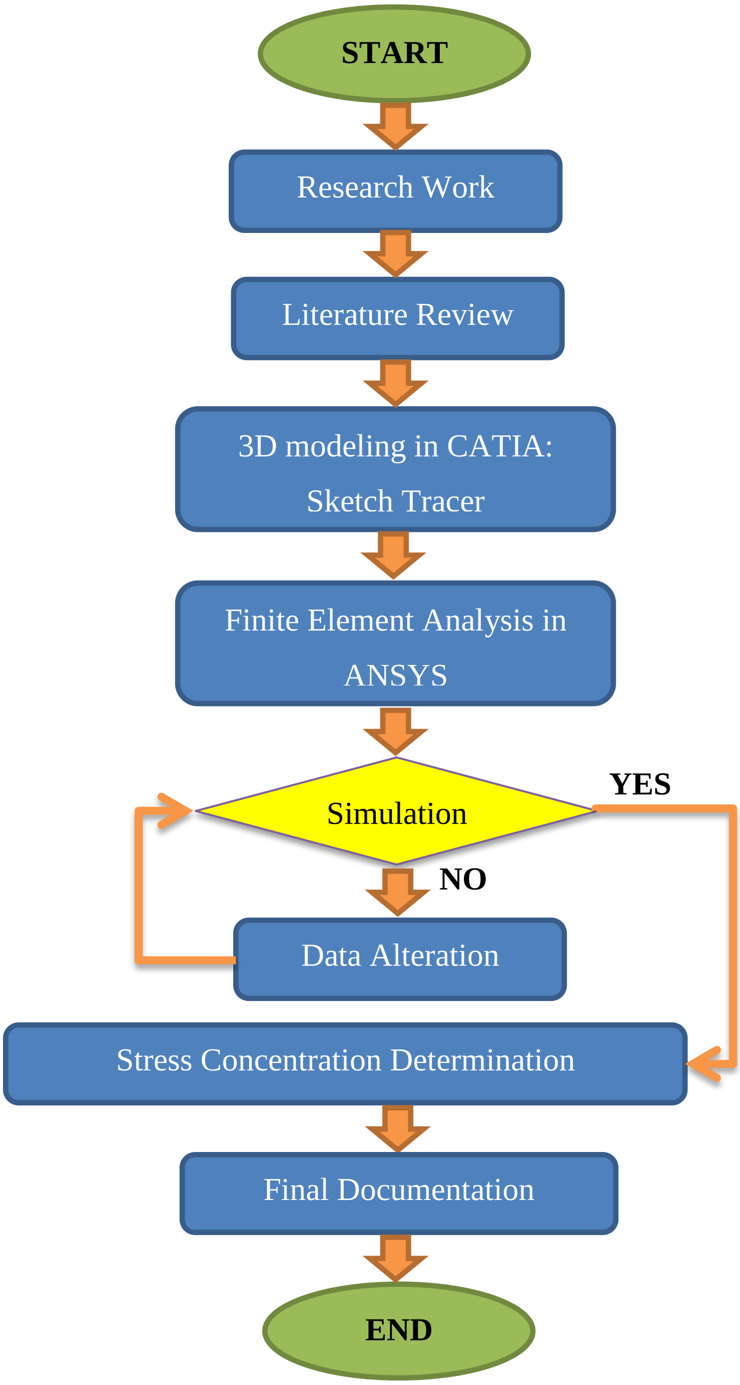


Figure 9: Project Process Flow

Figure 9 shows the planned flow chart of the project. Based on the flowchart, the steps and direction of the project can be seen clearly. It has been designed to fully utilise the time frame given to complete the project. The project will be divided into five main stages which are first, doing the preliminary research work. Second is to prepare the literature review to build better understanding of the project. The third stage is the development of 3D drawing of the bridge. Next is to determine stress concentration by simulating the 3D drawing using Finite Element Method. The last stage will be the final documentation that compiles all the research works and presents the outcome of the project.

3.2 Project Activities

Each stage is then discussed in detail as per Table 3. The project activities give the basic overview of what is going to be done throughout the completion of the project.

Table 3: Project Work Description

METHODOLOGY	DESCRIPTION
Research Work	Perform comprehensive study on the characteristic of double bass also other essential parts involved in the project such as the bridge.
Literature Review	Set out clear objectives and prepare the outline (scope of the project) using outcome of the research as a guideline. Understand deeply each essential part.
3D modelling in CATIA: Sketch Tracer	3D drawing of the bridge is drawn using CATIA Sketch Tracer.
Stress Concentration Determination	Simulate the drawing by assigning profile to the bridge (i.e., acting force). Determine stress concentration on the bridge using Finite Element Analysis in ANSYS.
Final Documentation	Outcomes of the project from the beginning of the project will be documented for future use.

Research work can be considered as an essential part of the project as it's the starting point to get to know into deep with the project. Through research work, some previous work and references have been referred to. However, there are no specific previous researches that were focussing on the effect of locating the sensor on the different area on the bridge. Based on research work and literature review, the author has come up with the idea on the development of 3D drawing of the bridge and on how to determine stress concentration location.

3.2.1 Development of 3D Drawing of the Bridge

At the beginning of the project, the author decided to apply Reverse Engineering knowledge in order to develop 3D drawing of the bridge. Computer Aided Reverse Engineering Technology Machine or also known as Digitizer is available in Building 16, Universiti Teknologi PETRONAS which it able to scan a 3D object and convert it into 3D drawing. Unluckily the machine is currently under maintenance and takes at about three months to repair. Due to that, the author has chosen to use CATIA: Sketch Tracer function. In order to use Sketch Tracer Function, the author needs to sketch the bridge manually using pencil on the paper as per Figure 10. The sketch was then being scanned using normal scanner. After transferred as image, the drawing is ready to being traced in CATIA. Details procedures can be referred in APPENDIX C.

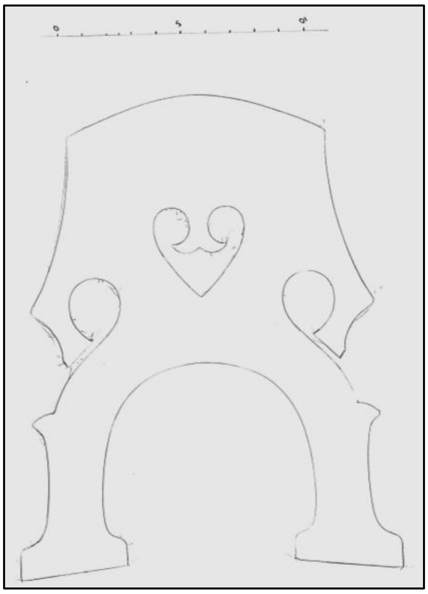


Figure 10: Sketch of the Bridge

After undergone drawing steps in CATIA, the 3D solid modelling is successfully built. Figure 10, 11, 12 and 13 each show the view from each side of the bridge which have been captured from CATIA.

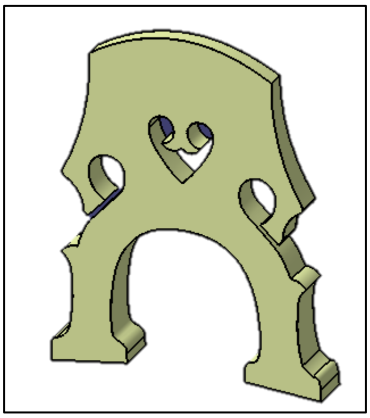


Figure 11: Isometric View of the Bridge

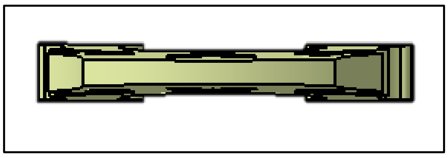


Figure 12: Top View of the Bridge

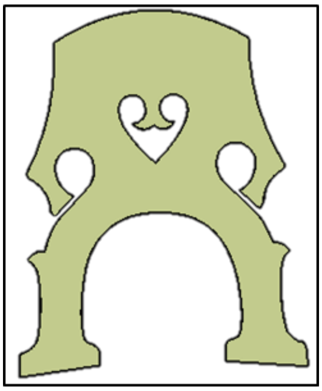


Figure 13: Front View of the Bridge

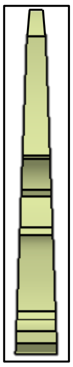


Figure 14: Side View of the Bridge

3.2.2 Stress Concentration Determination in ANSYS

Previously, it has been stated that the author will chose Finite Element Analysis in order to determine the highest stress concentration area on the bridge which believes will be the best mounting position of sound transducer. Finite Element Analysis will be applied using ANSYS software. As can be seen in Figure 15, the 3D solid modelling produced by CATIA can be imported successfully into ANSYS for analysis. Detail procedures can be referred in APPENDIX D. The results of each case will be discussed in Chapter 4: Results and Discussions.

Figure 15: 3D Solid Modeling of the Bridge in ANSYS

3.3 Gantt Chart with Key Milestone

The Gantt chart is built as a guideline for the project timeline. It can be changed from time to time depending on certain circumstances. The key milestone will be functioned as monitoring the work progress and to accurately determine whether or not the project is on schedule.

Table 4: Project Scheduling of FYP 1

No	Descriptions/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Topic Selection and Confirmation	■	■★												
2	Preliminary Research Work		■	■	■★										
3	Preparation of Extended Proposal				■	■	■★								
4	Transformation of Bridge into 3D Drawing						■	■	■	■★					
5	Proposal Defence								■	■					
6	Study and Familiarize with Simulation Software									■	■				
7	Run Stress Point Simulation										■	■	■★		
8	Preparation of Interim Draft Report												■	■★	
9	Improvement of Interim Draft Report													■	■★

■ Process ★ Suggested Milestone

Table 5: Project Scheduling of FYP 2

No	Descriptions/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Stress Point Determination using ANSYS	■	■	■	■	■	■	★								
2	Submission of Progress Report							■	★							
3	Stress Point Confirmation								■	■	★					
4	Pre-SEDEX										★					
5	Submission of Draft Report											■	★			
6	Submission of Dissertation (soft bound)												■	★		
7	Submission of Technical Paper												■	★		
8	Oral Presentation													■	★	
9	Submission of Project Dissertation (hard bound)														■	★

■ Process ★ Suggested Milestone

3.4 Tools/Software Selection

Throughout the project, the main software used are CATIA P3 V5R17 and ANSYS 14.0. CATIA is used in order to translate the bridge into digitize solid modelling while ANSYS is used to determine the stress concentration area which is proposed to be the best mounting position of the sound transducer. Other than that, the author also used Microsoft Office 2010, for the final report preparation and compilation. All software needed are available in Universiti Teknologi PETRONAS laboratory.

CHAPTER 4

RESULTS AND DISCUSSIONS

This chapter compiles the results and discussions of the projects. It contains the results retrieved from Finite Element Analysis (FEA) done in ANSYS. 3D solid modelling of the bridge was successfully developed in CATIA and being imported to be analysed in ANSYS. The detail on force acting in each case was previously explained in Chapter 2.

4.1 CASE 1: When The Strings Are Not Bowed

The model was first being analysed during its static condition when the strings were not bowed. Only string tensions were acting along the top of the bridge. As per Figure 16, it can be seen that, the highest stress concentration area was observed at curvy part of the bridge under string G. This shows that although during unbowed condition, the string tensions itself have given some stress to the bridge.

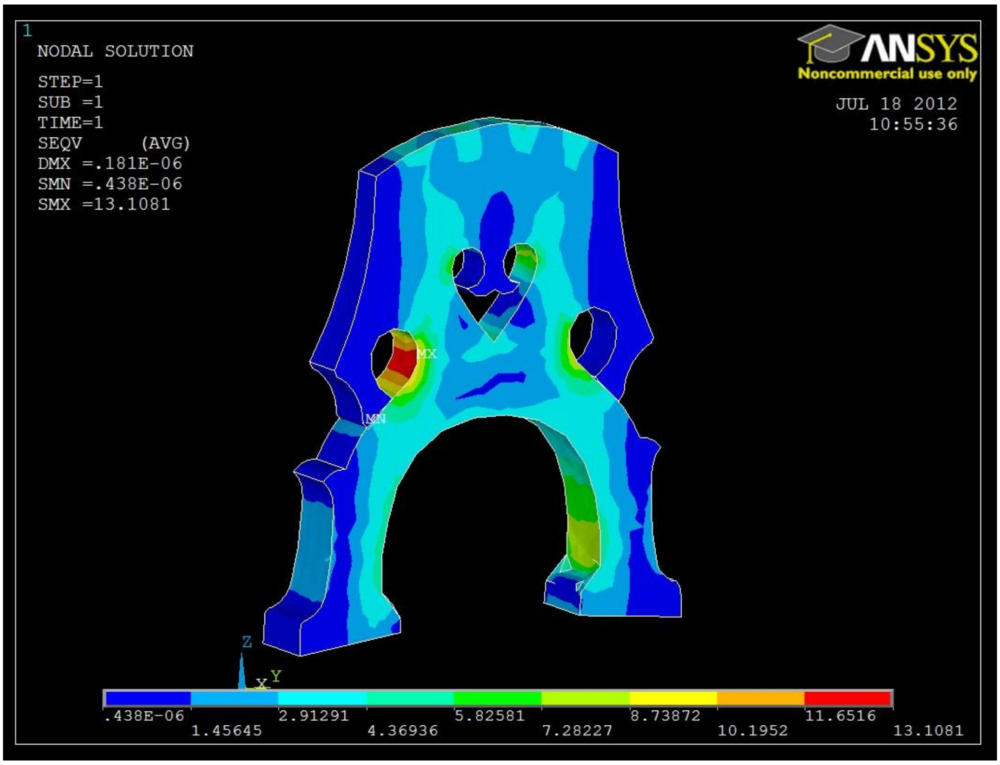


Figure 16: Stress Contour Diagram of CASE 1

4.2 CASE 2(a): Only String G Is Bowed To The Right

The analysis was then proceeded to the bowed condition. All together, there are four (4) strings and there will be four possible conditions when the string is bowed to the right. The first case is when only string G is bowed to the right as per Figure 17.

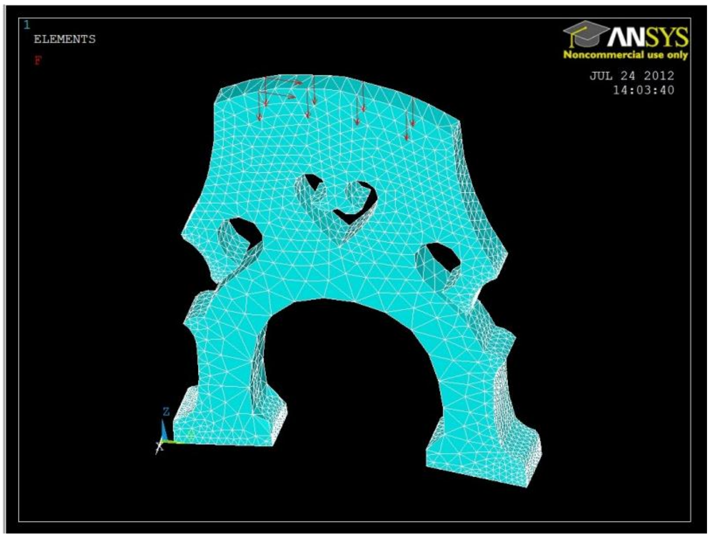


Figure 17: Forces Acting on Top of the Bridge in CASE 2 (a)

As the result, the highest stress concentration area is observed located in the same position as previous case.

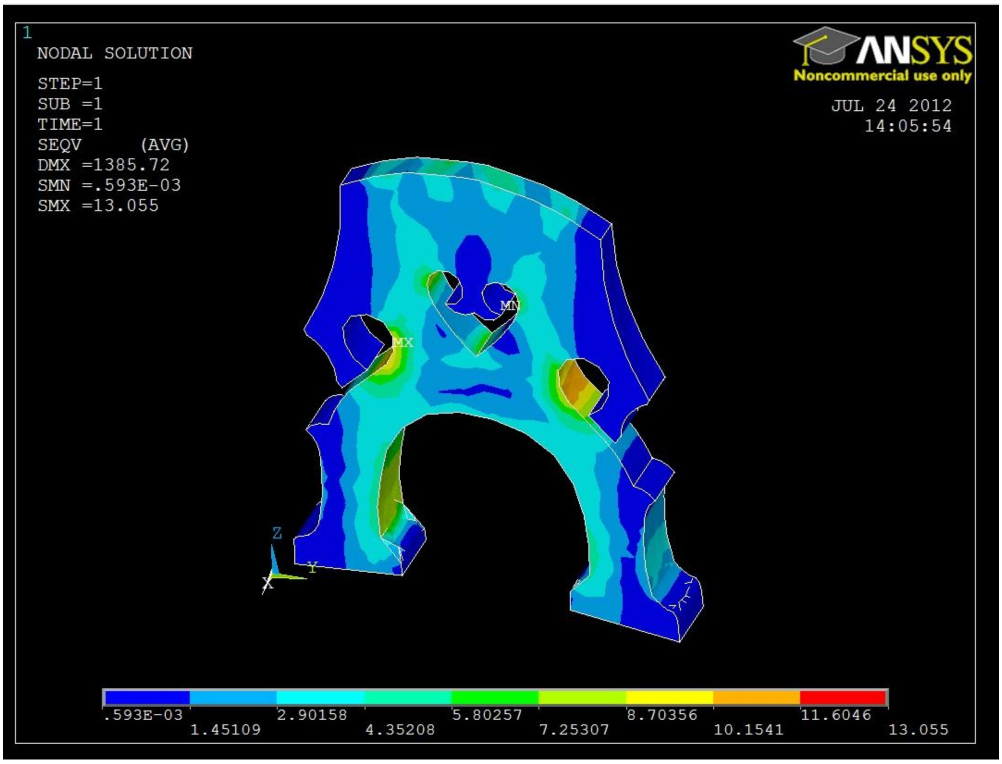


Figure 18: Stress Contour Diagram of CASE 2 (a)

4.3 CASE 2(b): Only String D Is Bowed To The Right

Next is when only string D is bowed to the right as shown in Figure 19. As a result as per Figure 20, the stress concentration area seems to be at the same position which is near the curvy area under G string location.

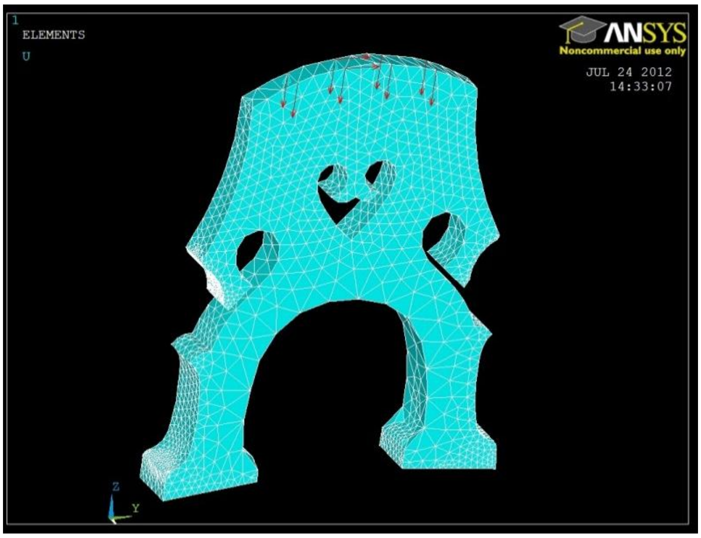


Figure 19: Forces Acting on Top of the Bridge in CASE 2 (b)

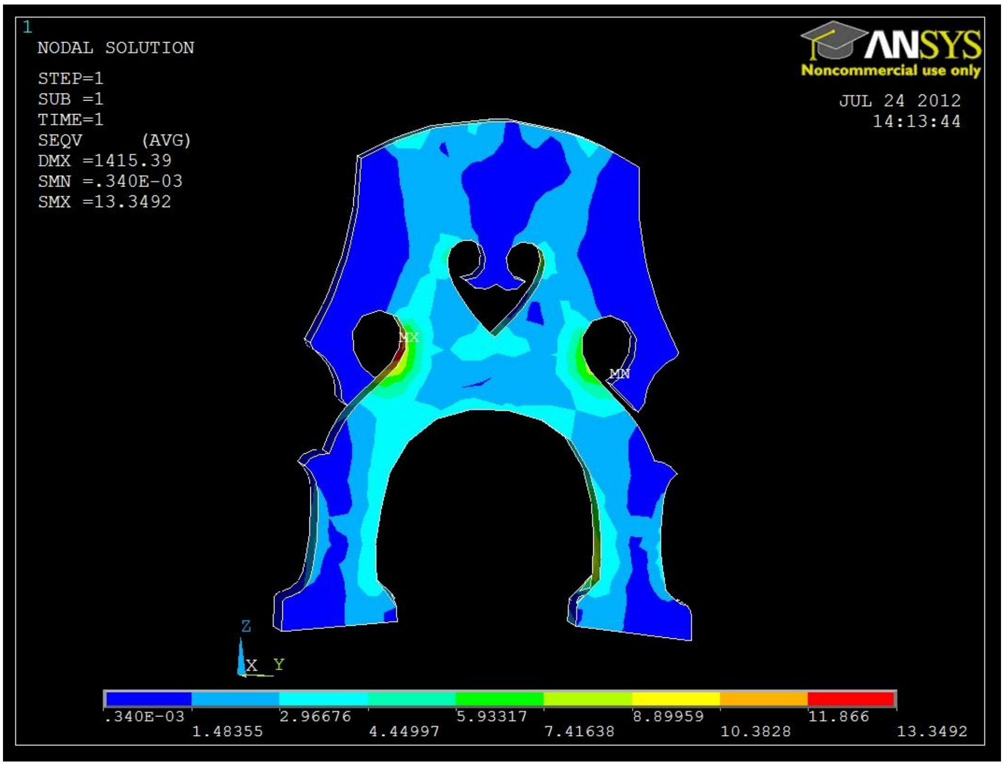


Figure 20: Stress Contour Diagram of CASE 2 (b)

4.4 CASE 2(c): Only String A Is Bowed To The Right

When only string A is bowed to the right as per Figure 21, the result was observed to show the same position of highest concentration area.

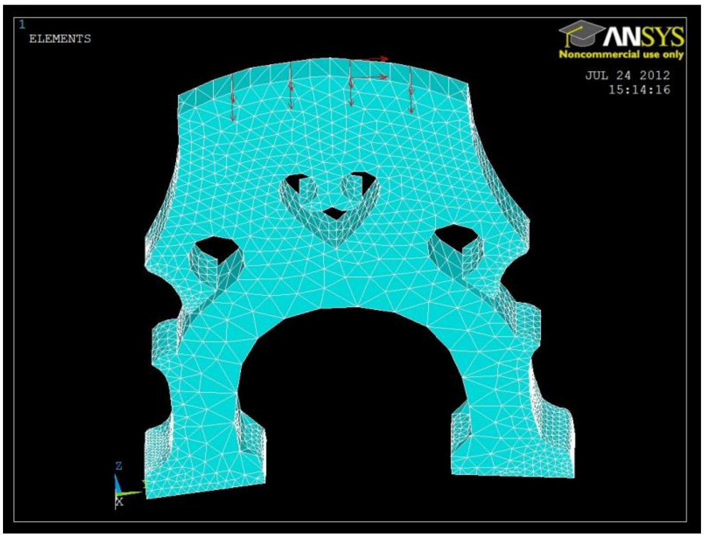


Figure 21: Forces Acting on Top of the Bridge in CASE 2 (c)

Both areas at the waist having highest stress acting on them. Between the two, the highest is the one near the G string location.

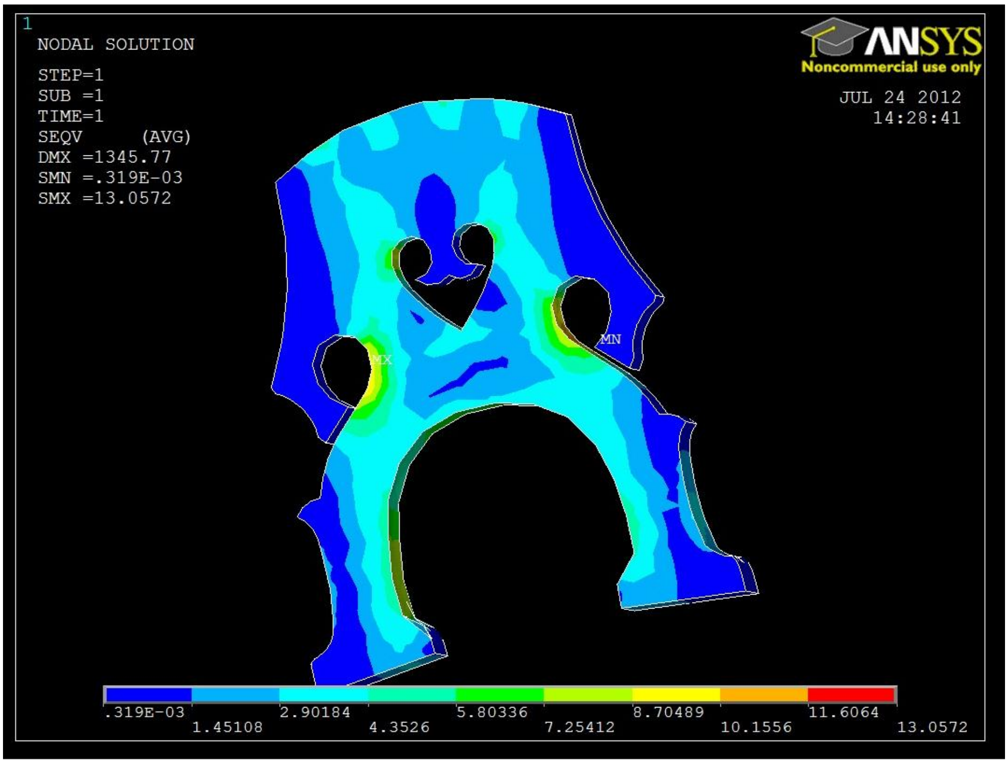


Figure 22: Stress Contour Diagram of CASE 2 (c)

4.5 CASE 2(d): Only String E Is Bowed To The Right

Figure 23 shows the forces acting on the top of the bridge when only string E is bowed to the right.

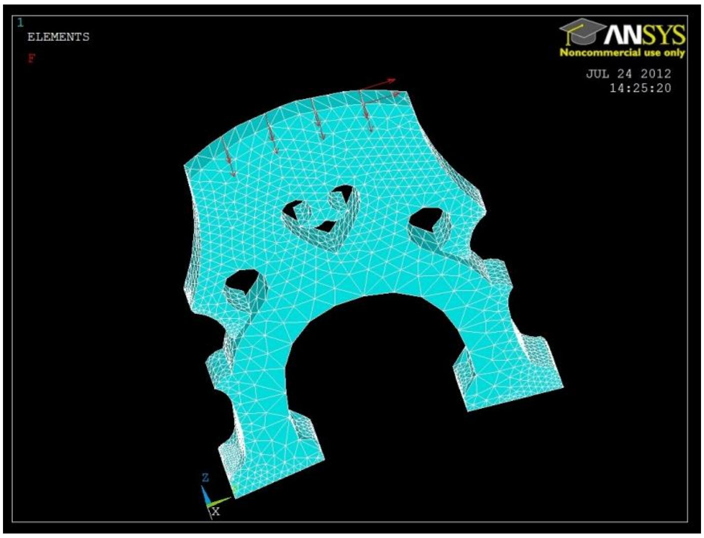


Figure 23: Forces Acting on Top of the Bridge in CASE 2 (d)

The result shows that the highest stress concentration area is located at the waist near the G string (curvy area).

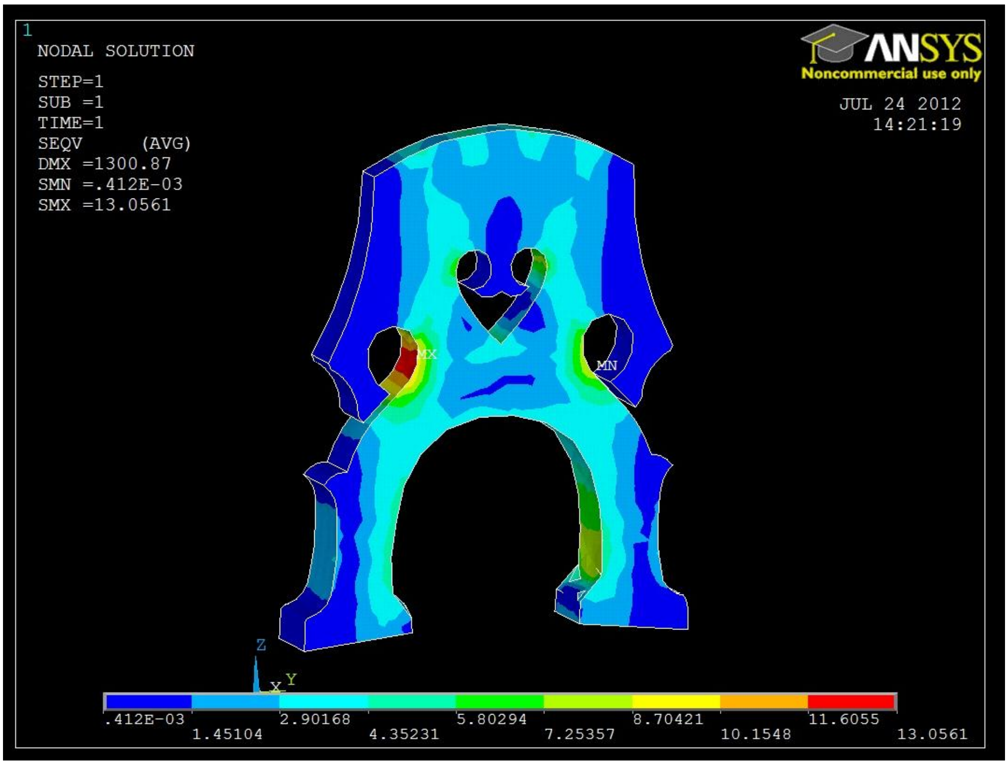


Figure 24: Stress Contour Diagram of CASE 2 (d)

4.6 CASE 3(a): Only String G Is Bowed To The Left

The analysis was then proceeding to the next case which is when the string is bowed to the left. There will be four (4) possible cases. Firstly, as per Figure 25, case when only string G is bowed to the left.

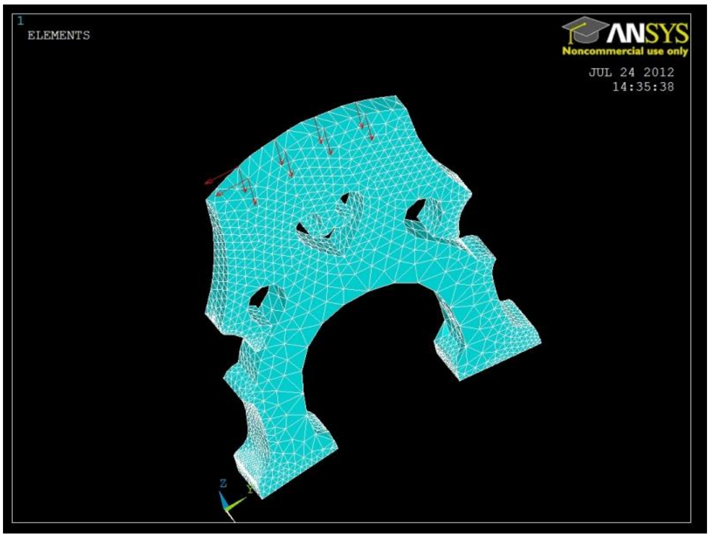


Figure 25: Forces Acting on Top of the Bridge in CASE 3 (a)

Results in ANSYS shows that the highest stress concentration area was observed to be at the both waist of the bridge. The highest is at curvy location near G string.

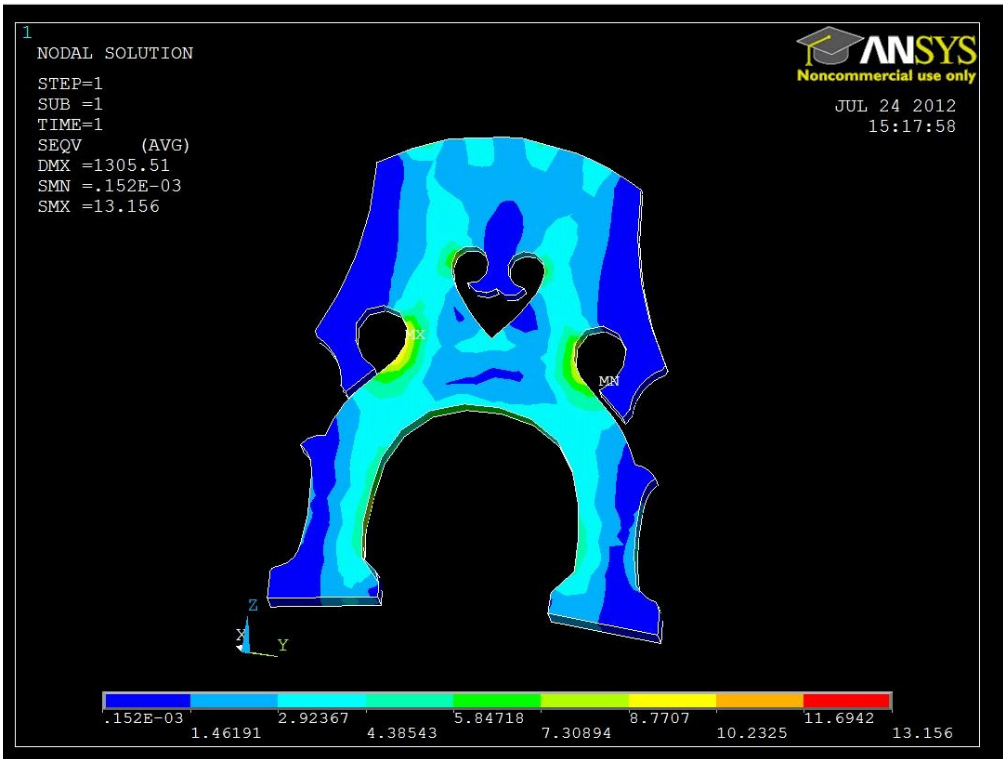


Figure 26: Stress Contour Diagram of CASE 3 (a)

4.7 CASE 3(b): Only String D Is Bowed To The Left

Next is the case when only string D is bowed to the left. The result is shown as per Figure 28. The highest stress concentration area was observed to be similar as previous cases.

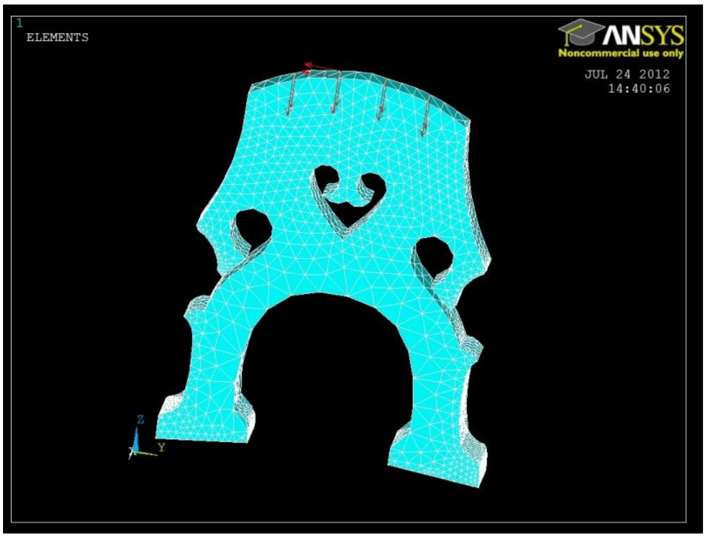


Figure 27: Forces Acting on Top of the Bridge in CASE 3 (b)

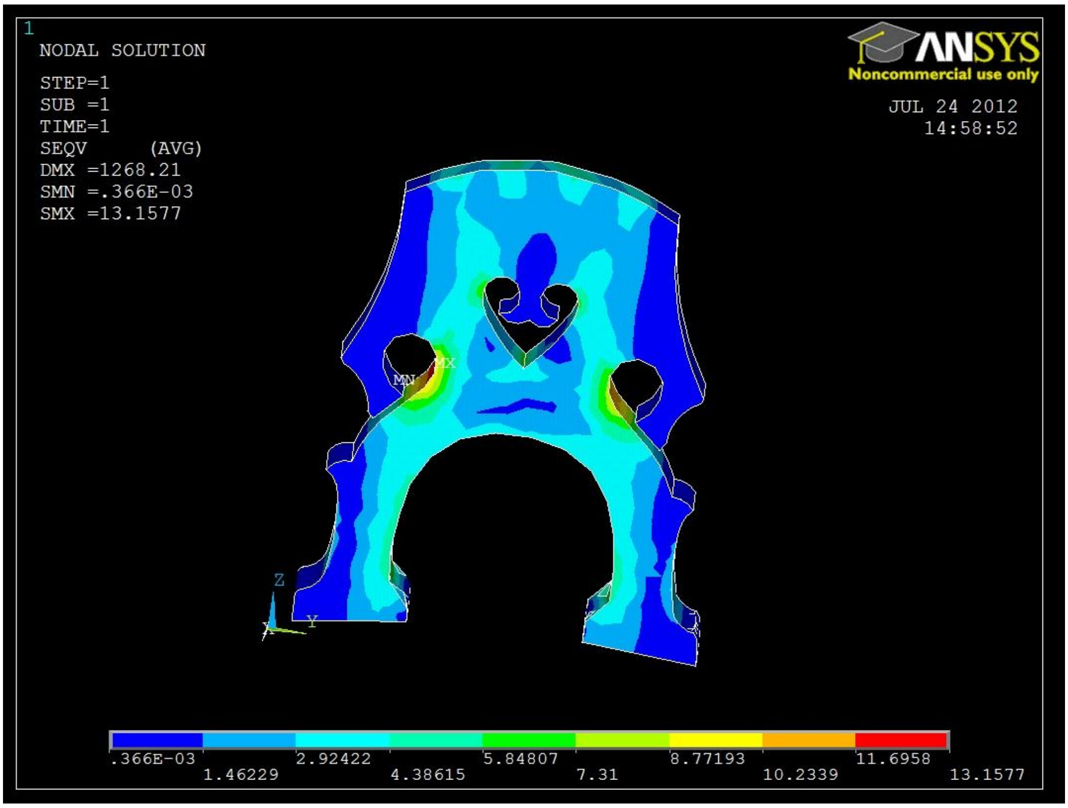


Figure 28: Stress Contour Diagram of CASE 3 (b)

4.8 CASE 3(c): Only String A Is Bowed To The Left

Figure 29 shows the forces acting on top of the bridge when only string A is bowed to the left. The force acting is having the same magnitude as CASE 2 but different in direction.

Figure 29: Forces Acting on Top of the Bridge in CASE 3 (c)

The result is shown in Figure 30. The highest stress concentration area is located at the waist near G string location.

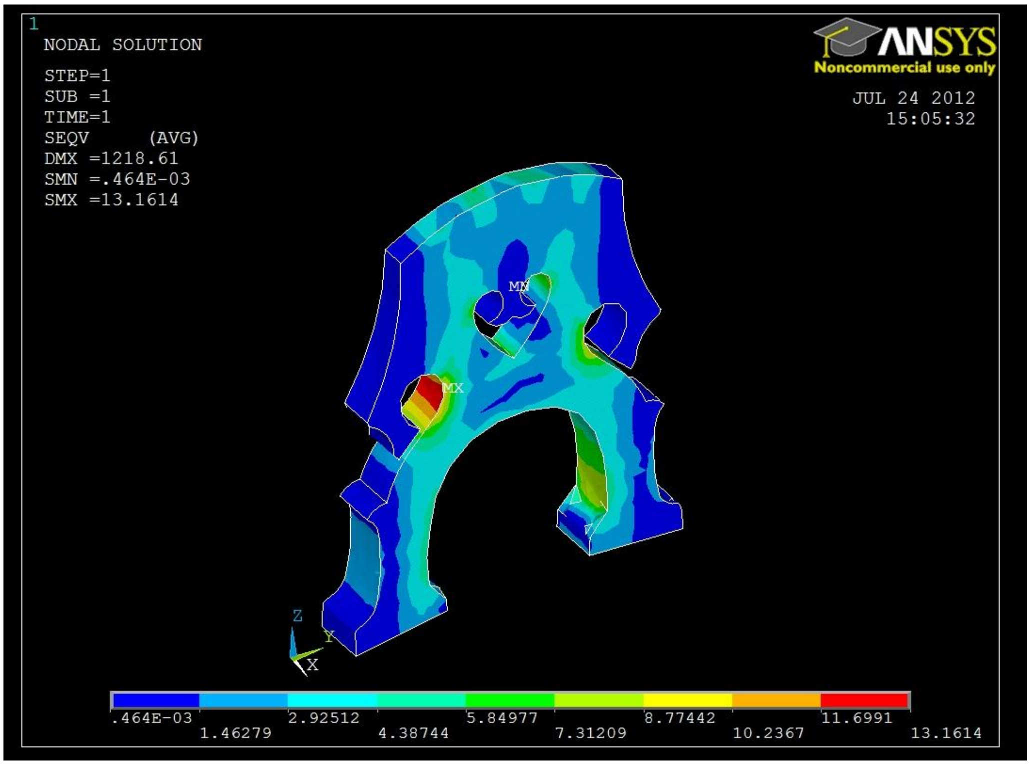


Figure 30: Stress Contour Diagram of CASE 3 (c)

4.9 CASE 3(d): Only String E Is Bowed To The Left

Finally, the last possible case is when only string G is bowed to the left. The forces acting during this case is shown in Figure 31.

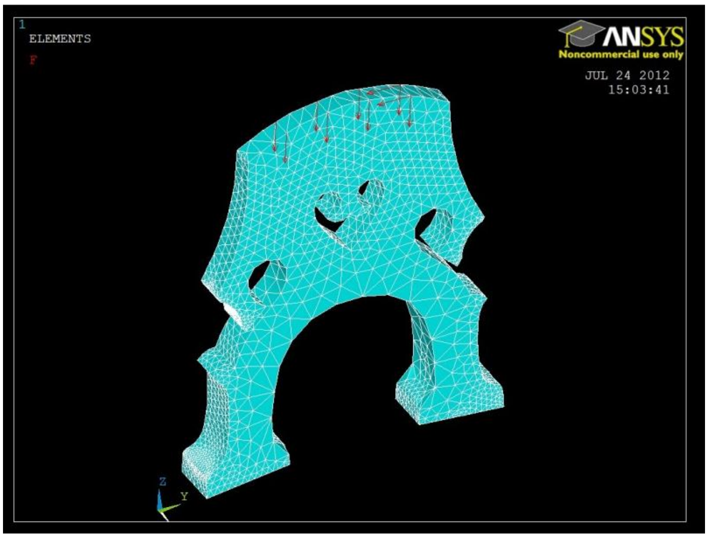


Figure 31: Forces Acting on Top of the Bridge in CASE 3 (d)

As the result, the location of the highest stress concentration is determined to be at the curvy area near G string location.

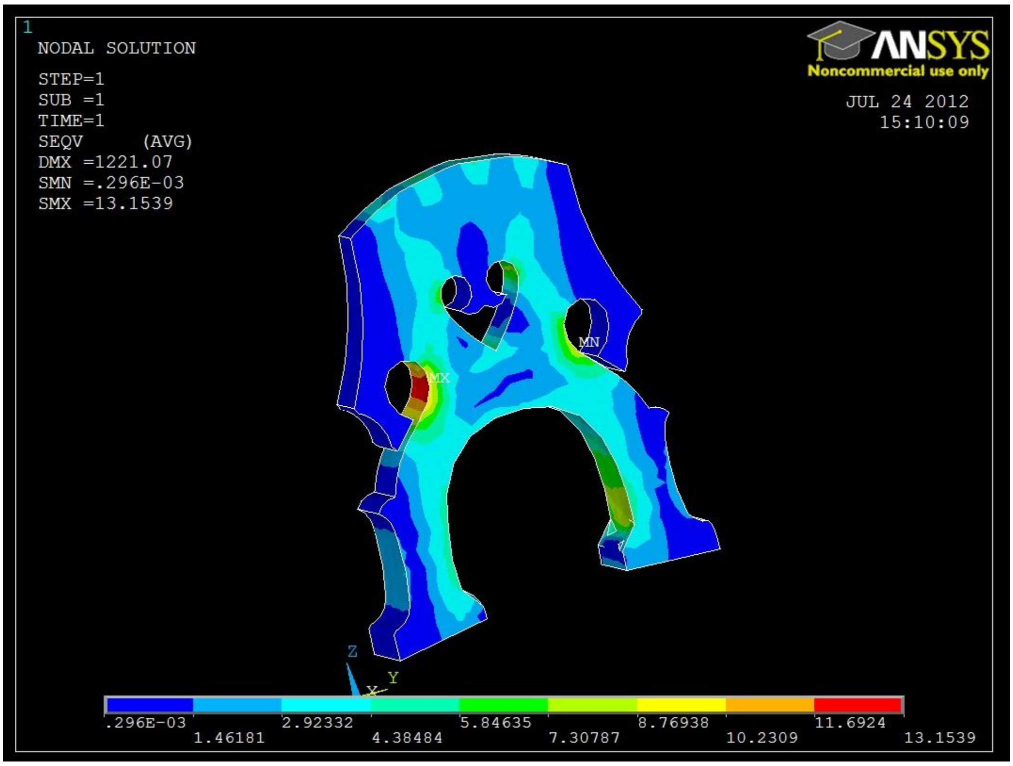


Figure 32: Stress Contour Diagram of CASE 3 (d)

4.10 Discussions

At the end of the analysis, the author observed that all the cases had shown the same location which having the highest stress concentration. The location with the highest stress concentration is determined to be at the middle of the waist near the curvy side of the bridge under G string location. The results obtained are similar in any case may be due to the shape complexity which cause the curvy area to be the critical area and cause the stress to be concentrated at the area.

Apart from that, the analysis had shown that this new position could be proposed as the best mounting position of sound transducer in order to reproduce wooden timbre characteristics of contra bass.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Conclusions

In conclusions, the author was successfully completed the project. The objectives set at the beginning of the project were all achieved. The bridge was translated into digitized 3D solid modelling using CATIA: Sketch Tracer instead of applying Reverse Engineering technology. The model was then being analysed using Finite Element Analysis (FEA) and the best mounting position was successfully determined as the author proposed the area with the highest stress concentration is the most suitable mounting position. The position was then observed to be at the curvy part of the bridge. At all cases, they gave the similar location of concentration area as the position at the curvy part at the middle of the waist is giving the highest stress concentration in any case and could be proposed as the mounting position of sound transducer.

5.2 Recommendations

For continuation of the project, the author proposes to implement the sound transducer at the position determined from the Finite Element Analysis. The implementation of the sound transducer can be followed by having experiment in order to compare the sound quality produced by the bridge with sound transducer located at the current position in the market and the sound quality produced by the bridge with the sound transducer mounted at the new proposed position determined from the project. The result from the experiment will assist the analytical analysis made.

REFERENCES

BAEN. (2007, January 26). *Bass String Diameters and Tensions*. Retrieved July 1, 2012, from http://baen.tamu.edu/users/rel/personal/BassStrings.html

Besnainou, C. (2000). *Introduction to the Use of Composite Materials in Musical Instruments.* Catgut Acoust. Soc. J.

Bissinger, G. a. (2001). *Air-Plate-Neck-Fingerboard Coupling and the Feel of a Good Violin.* Catgut Acoust. Soc. J.

Bladier, B. (1960). *Sur Le Chevalet du Violencelle.* in Musical Acoustic, Part I. Violin Famiy lComponent. Dowden, Hutchinson, and Ross, Stroudsburg, Pennsylvania: Translated by R. B. Lindsay.

Bucur, V. C. (2000). *Relationships between theInorganic Component of the Cellular Wall and the Acoustic Properties of Wood for Violin.* Catgut Acoust. Soc. J.

Craig, W. (2000). The Composite Recurve Bow. *Analysis Using ANSYS*.

Daniel, R. R. (2000). *The Science and Applications of Acoustics.* New York: Springer-Verlag.

Fletcher, H. B. (1996). *Quality of Violin, Viola, Cello & Bass Viols Tones.* J. Acoust. Soc. Am.

French, M. (1996). *Piezo-Film Sensors for Luthiers.* Catgut Acoust. Soc. J.

Gautschi, G. (2002). *Piezoelectric Sensorics: Force, Strain, Pressure, Acceleration and Acoustic Emission Sensors, Materials and Amplifiers.* New York: Springer-Verlag.

Haines, D. W. (2000). *The Essential Mechanical Properties of Wood Prepared for Musical Instruments.* Catgut Acoust. Soc. J.

Hutchins, C. M. (2000). *Problem of Moisture Change when Tuning Violin Plates.* Catgut Acoust. Soc. J.

Infeld, T. (n.d.). Retrieved 7 11, 2012, from Thomastik Infeld: www.thomastik-infeld.com

Jansson, E. V. (1997). *On Body Resonance C3 and Violin Construction.* Catgut Acoust. Soc. J.

Jonas, L. (2012). *Double Bass Guide*. Retrieved July 27, 2012, from http://doublebassguide.com/?page_id=21

Joseph, C. (2012, April). Challenging Assumptions & Reinventing the Bridge. *All Things Strings*, pp. 45-46.

Kirkness, J. (2012). *Double Bass String Tensions*. Retrieved July 11, 2012, from http://jordankirkness.tripod.com/dbstringtension.html

Labossiere, P. E. (2007). *UW Courses Web Server*. Retrieved April 21, 2012, from ME354 Mechanics of Materials Laboratory: http://courses.washington.edu/me354a/chap6.pdf

Liu, Y. (2003, May 21). *Introduction to the Finite Element Analysis*. Retrieved March 19, 2012, from http://urbana.mie.uc.edu/yliu/FEM-525/FEM-525.htm

Miller, R. K. (1987). *Non-destructive Testing Handbook, Second Ed., Vol 5: Acoustic Emission Testing.* American Society for Non-destructive Testing.

Neville, H. F. (2000). *The Physics of Musical Instruments.* New York: Springer-Verlag.

Pickering, N. C. (1997). *Strings and Metallurgy*. Catgut Acoust Soc. J., Ser. II.

Plomp, R. a. (1969). *Effect of Phase on the Timbre of Complex Tone.* J. Acoust. Soc. Am.

Rodgers, O. (2001). *Finite Element Analysis of a Violin Corpus.* Catgut Acoust. Soc. J.

Savart, F. (1937). *Des Instrument de Musique.* Hutchins: Translated by D. A. Fletcher.

Segerman, E. (2001). *Some Aspects of Wood Structure and Function.* Catgut Acoust. Soc. J.

Thompson, R. P. (2000). *The Effect of Variations in Relative Humidity on the Frequency Response in Free Violin Plates.* Catgut Acoust. Soc. J.

Wang, L. M. (1999). *Directivity Patterns of Acoustic Radiation form Bowed Violins.* Catgut Acoust. Soc. J.

Weinreich, G. (2002). *Sound Radiation from the Violin-As We Know It Today*. Catgut Acoust. Soc. J.

Woodhouse, J. (2002). *Body Vibration of the Violin-What Can a Maker Expect to Control.* Catgut Acoust. Soc. J.

APPENDIX A

Parts of Double Bass/Cello

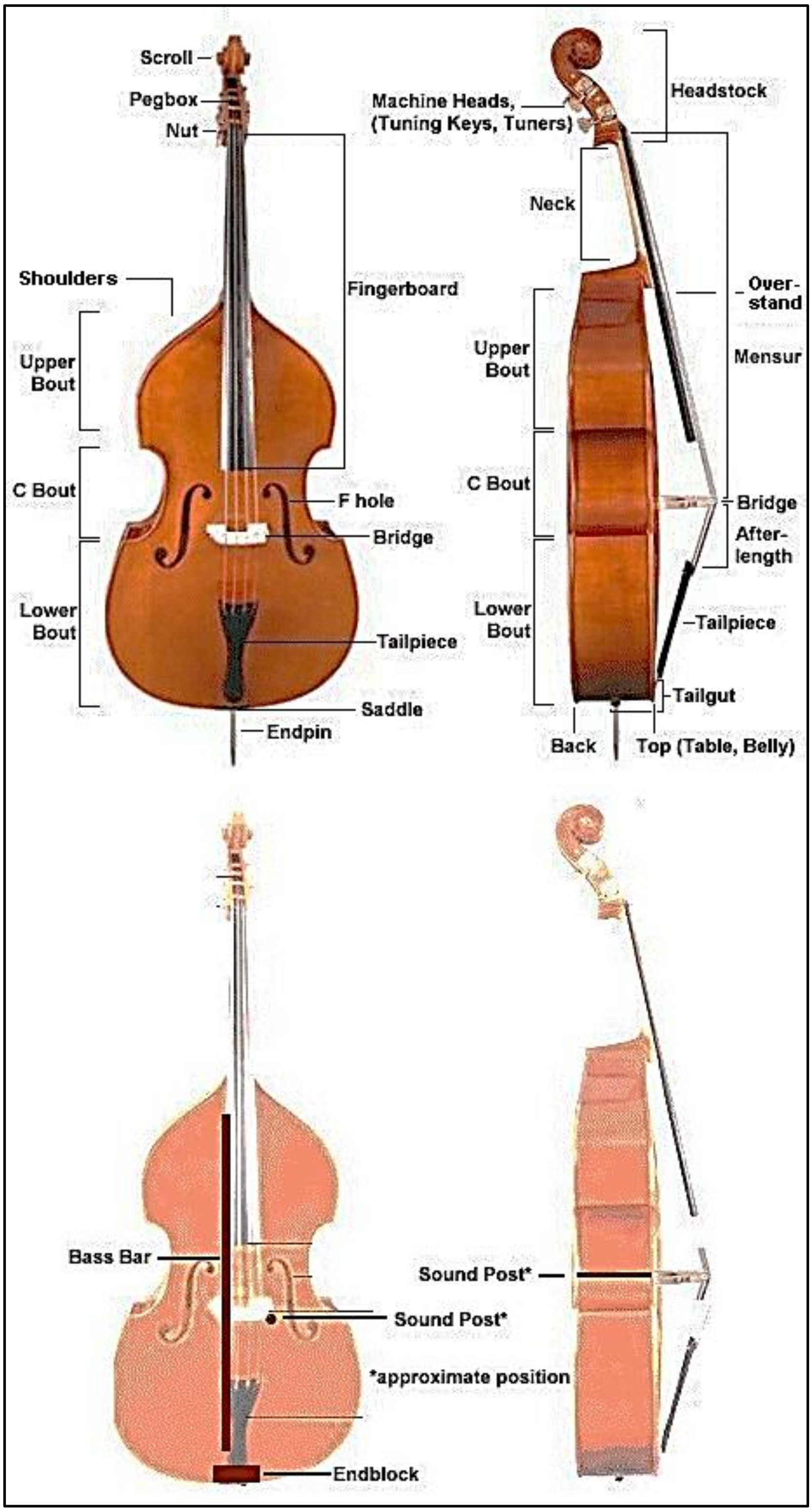


Figure 33: Parts of Double Bass/Cello

APPENDIX B

Multi View of 3D Drawing of the Bridge

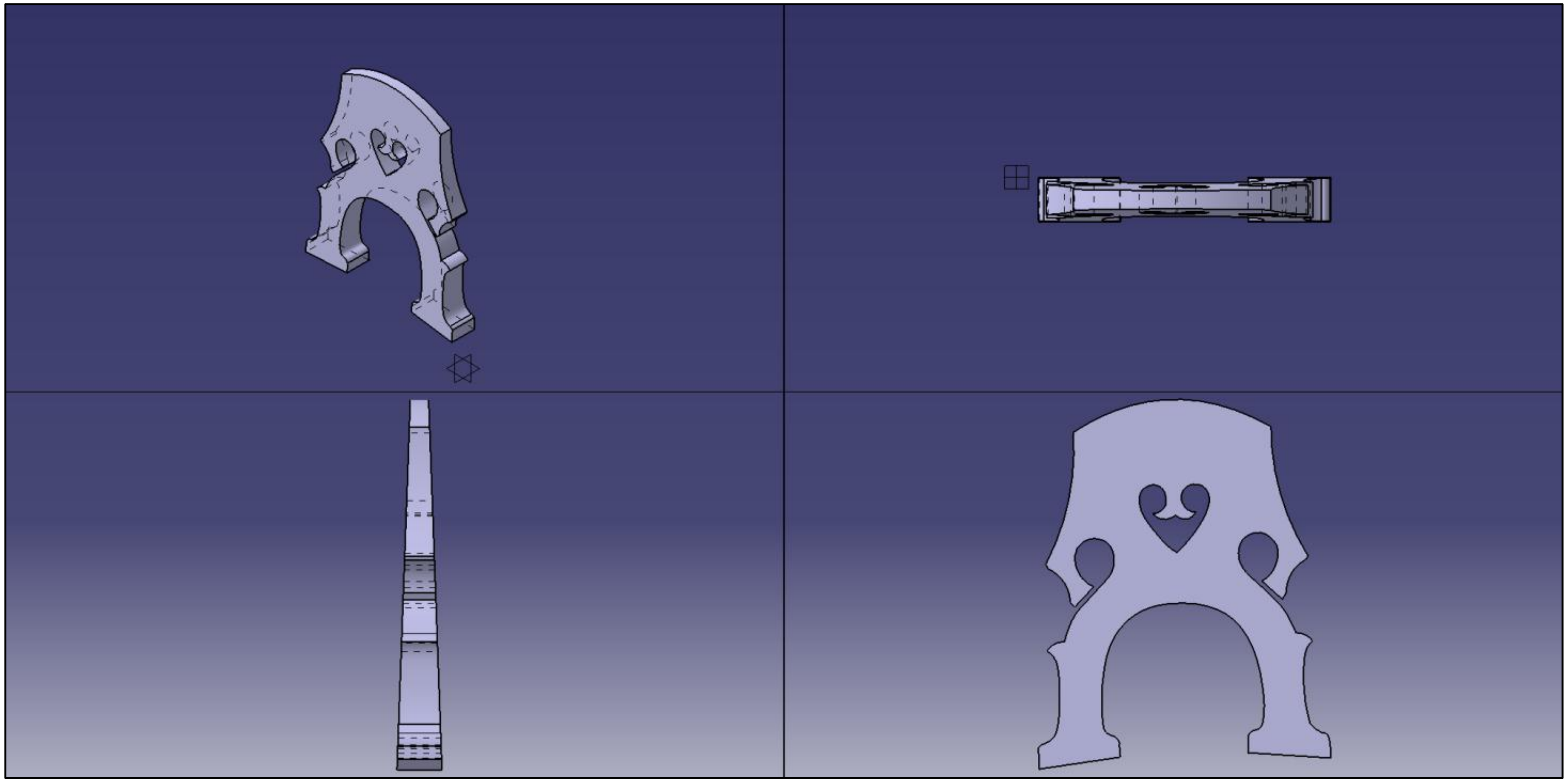


Figure 34: Multi View of 3D Drawing of the Bridge

APPENDIX C

Procedures of converting the bridge into 3D solid modelling using CATIA P3 V5R17:

1. CATIA V5 was loaded. Sketch Tracer function was selected from the Start Menu.

Start > Shape > Sketch Tracer

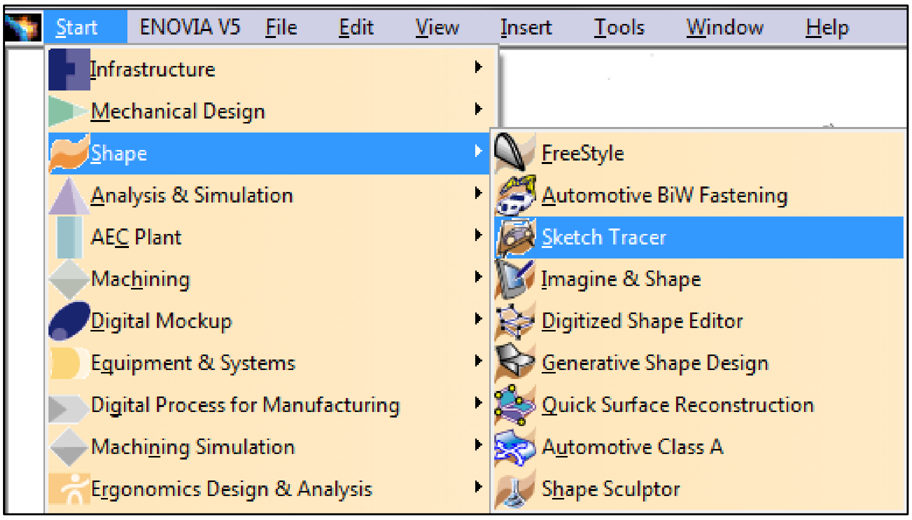
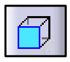
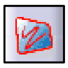


Figure 35: Start Bar Menu

2. The view was changed to front view by clicking  icon.

3. Shading with material function was chosen by clicking on  icon.

4. Immersive sketch was created by clicking on  icon.

5. File containing the bridge sketch image was loaded.

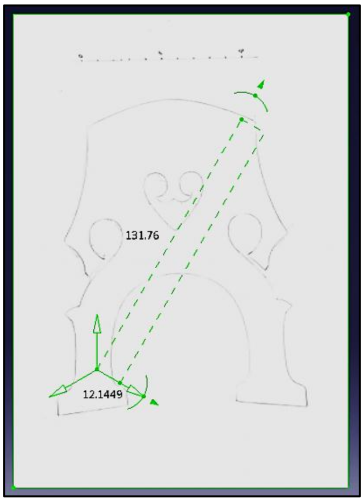
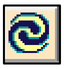


Figure 36: Scanned Sketch of the Bridge

6. Cube view in Sketch Parameters toolbar was chosen.

Sketch Parameters > Use a cube >  **Reload Image > OK**

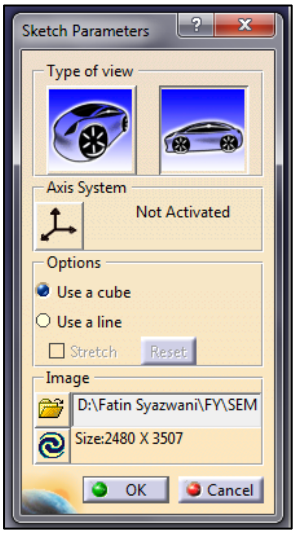


Figure 37: Sketch Parameters

7. In the same file, new part design was opened.

Product (right click) > Components > New Part

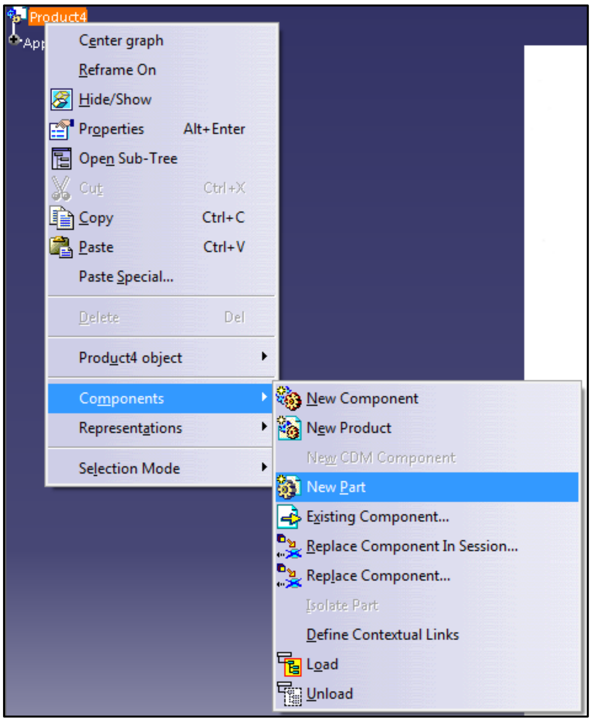
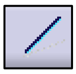
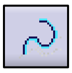
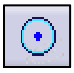


Figure 38: New Part Command

8. Double click on the part body to start new part drawing. At this point forward, the drawing was made referring to the image uploaded.

Product > Part1 > yz plane > Click 

Use any icon to obtain the best fitted line of the image. Icon available such as , ,  and many more.

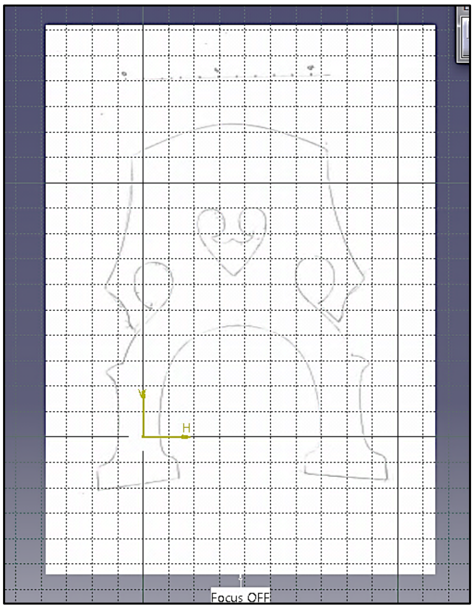


Figure 39: The Sketch acts as Reference

9. After completed the drawing, proceed for 3D modelling.

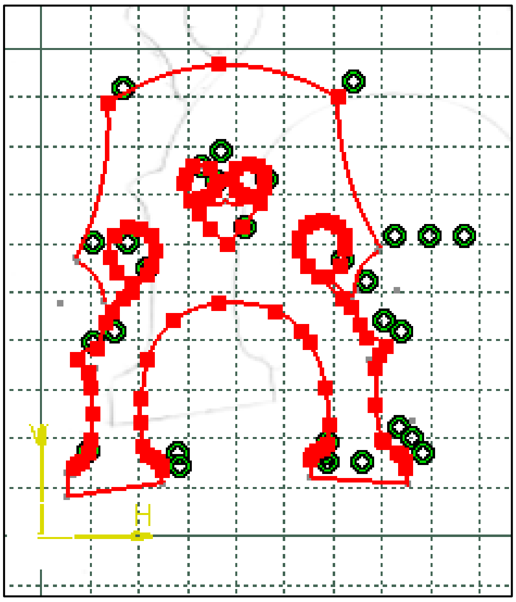


Figure 40: 2D Drawing of the Bridge

10. The sketch was extruded using  icon. Next, lines were drawn on the side view of the bridge. The top width will be 1.00 cm while the bottom is 2.40 cm in width. The shape was extruded with thickness similar to the width of the bridge. Boolean operation was applied.

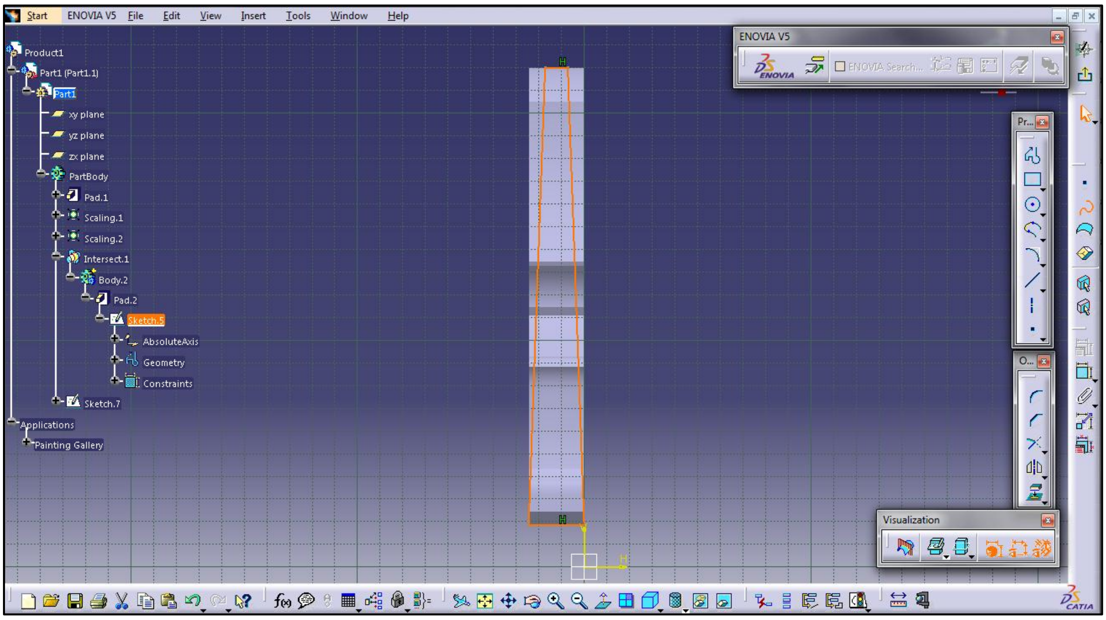


Figure 41: ANSYS Interface

11. 3D modelling of the bridge will be completed once the shape was extruded.

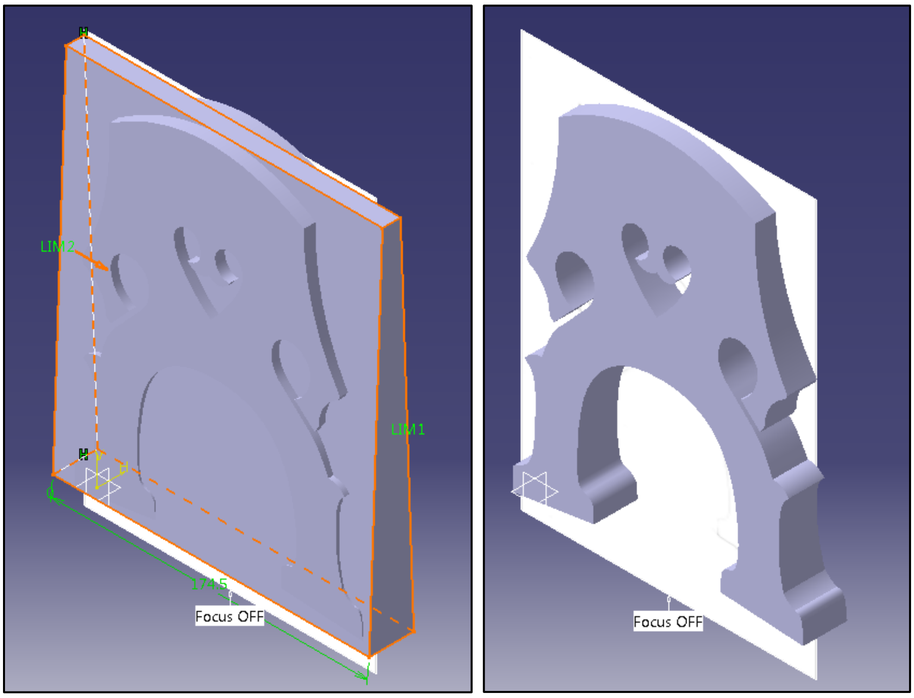


Figure 42: 3D Solid Modeling of the Bridge

APPENDIX D

Procedures followed throughout the analysis in ANSYS:

1. 3D solid model of the bridge in CATIA V5 was imported into ANSYS.

File > Import > CATIA V5

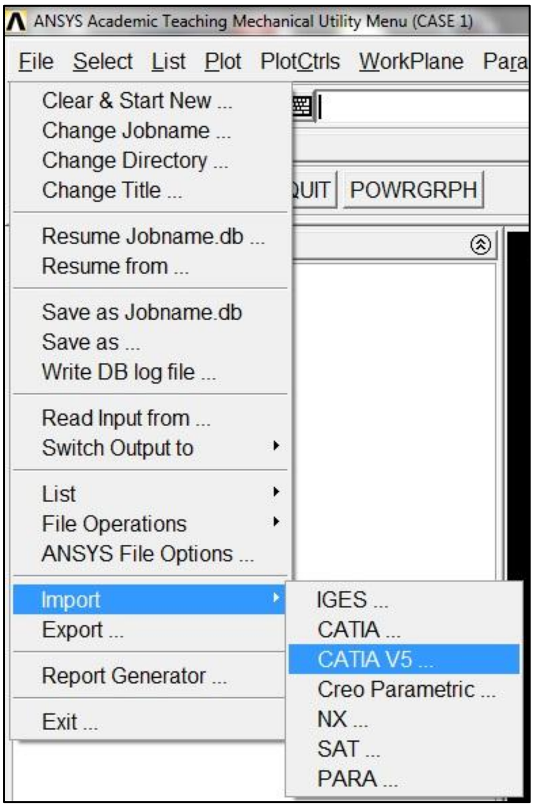


Figure 43: Start Menu Bar

Browse > Open "file name" > Allow model defeaturing > Geometry Type: All

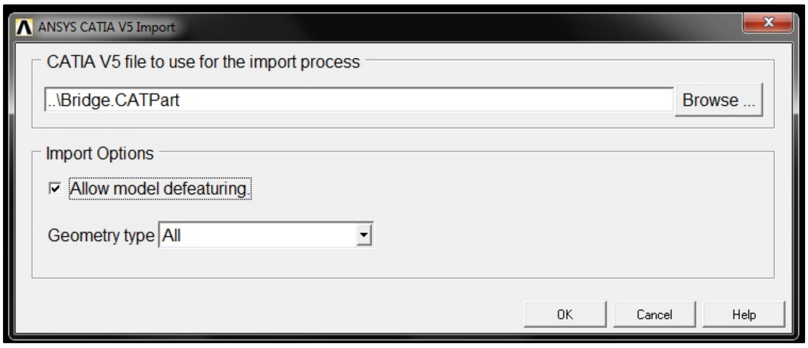


Figure 44: ANSYS CATIA V5 Import

2. Element type and material properties of the model were defined.

Pre-processor > Element Type > Add / Edit / Delete > NONE DEFINED > SOLID285 > OK

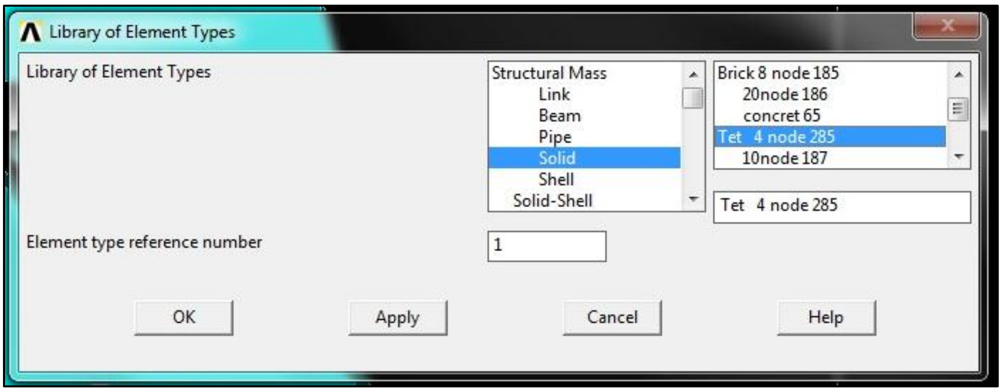


Figure 45: Library of Element Types

Pre-processor > Material Props > Material Models > Material Model Number 1 > Structural > Linear > Elastic > Isotropic > EX = 1.26E10 > OK

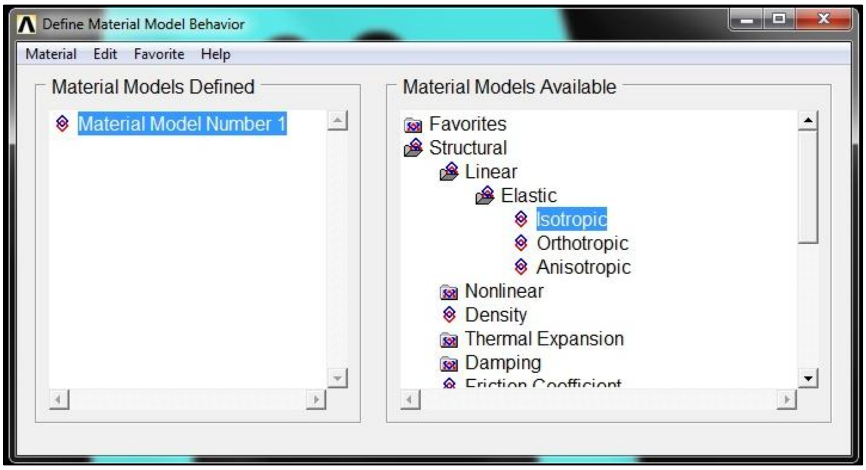


Figure 46: Define Material Model Behavior

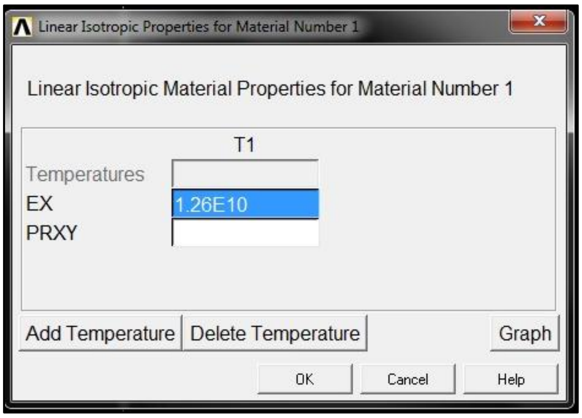


Figure 47: Linear Isotropic Properties for Material Number 1

3. Next, the model was meshed using the mesh tool.

Pre-processor > Meshing > Mesh Tool > Mesh > Select Object > OK

Figure 48: Mesh Tool

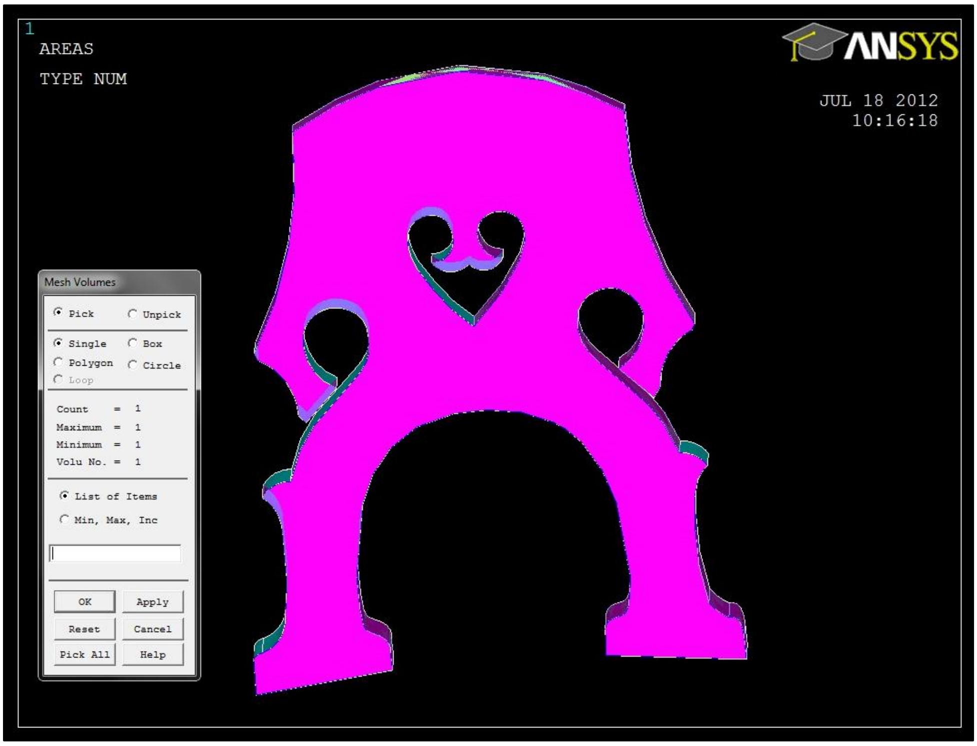


Figure 49: Select Object to be Meshed

4. Analysis type was set to static analysis.

Solution > Analysis Type > New Analysis > Static > OK

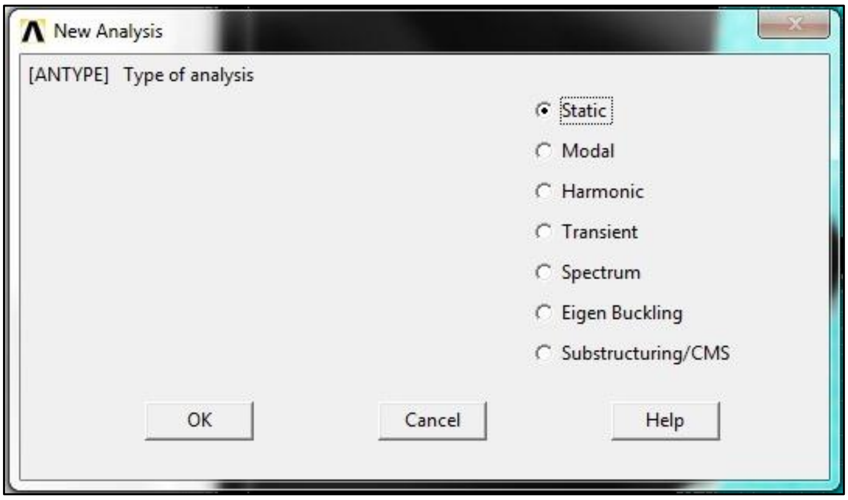


Figure 50: New Analysis

5. Fixed support at the bottom of the bridge was defined.

Solution > Define Loads > Apply > Structural > Displacements > On Areas > Select Areas > OK

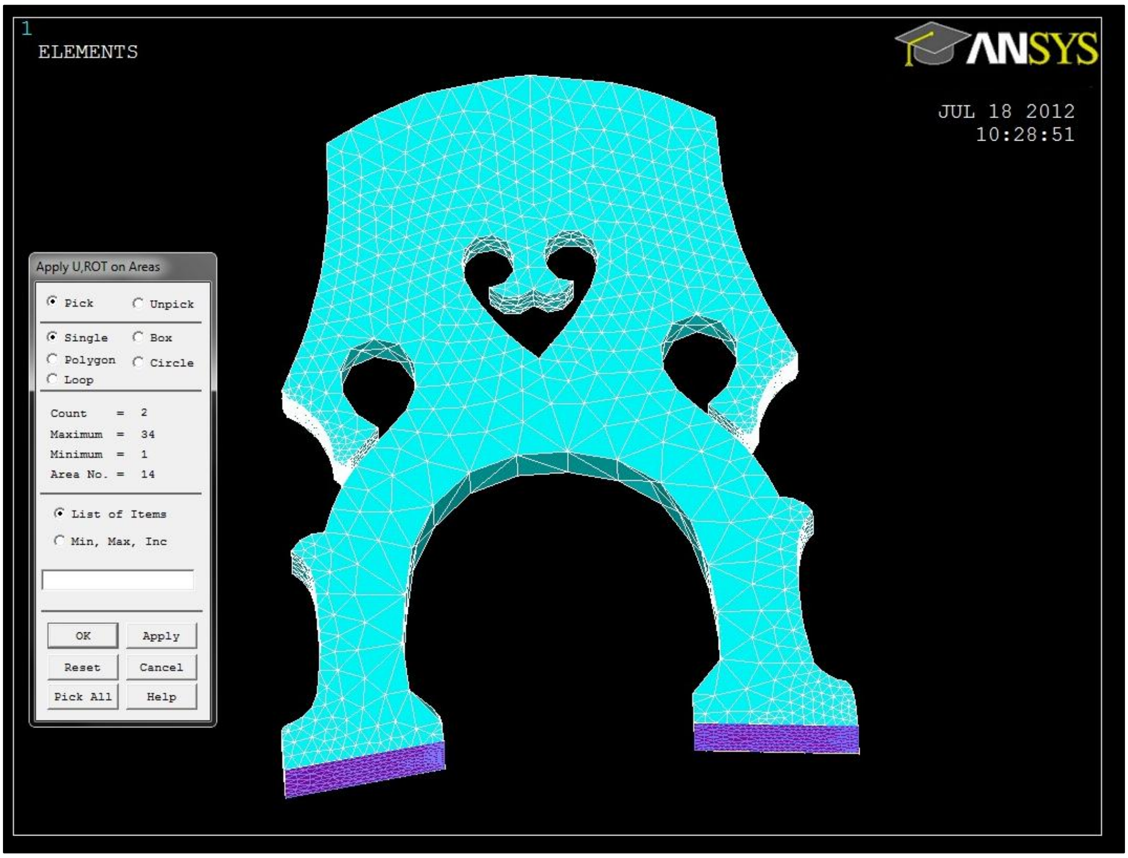


Figure 51: Fixed Support on the Bridge

6. Forces acting at the top of the bridge were defined.

Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes > Select Nodes > OK

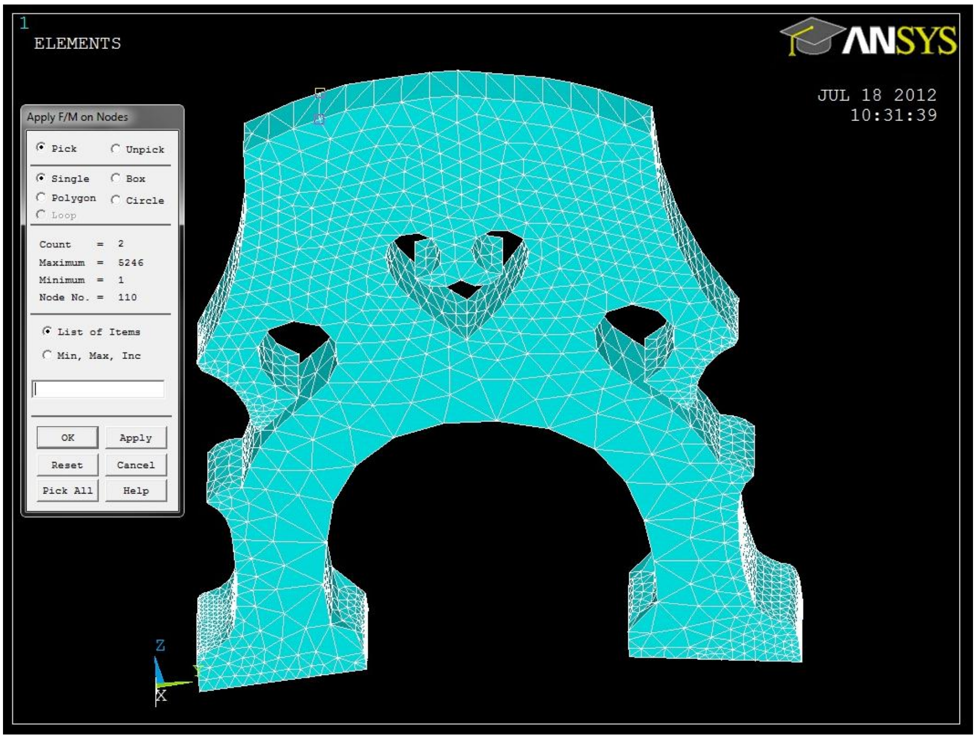


Figure 52: String Tension acting on Top of the Bridge

Direction of Force / Moment > FZ > Fill up Force / Moment value (refer to Table for the force value) > OK

Table 6: String Tension

STRING	G	D	A	E
Force (N)	291.98	293.37	301.99	299.26

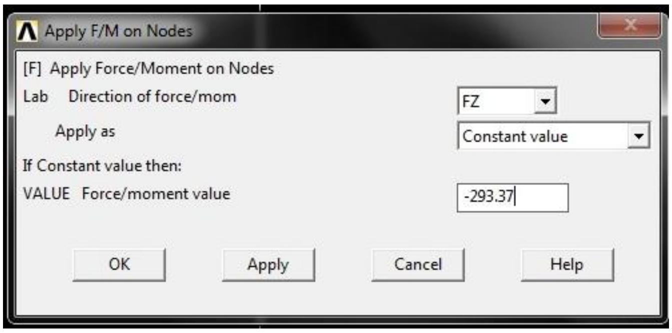


Figure 53: Apply Force/Moment on Nodes

7. Proceed to solve the analysis.

Solution > Solve > Current LS > OK

Figure 54: Solve Current Load Step

General Postproc > Read Results > Last Set

General Postproc > Plot Results > Nodal Solution > von Mises stress > OK

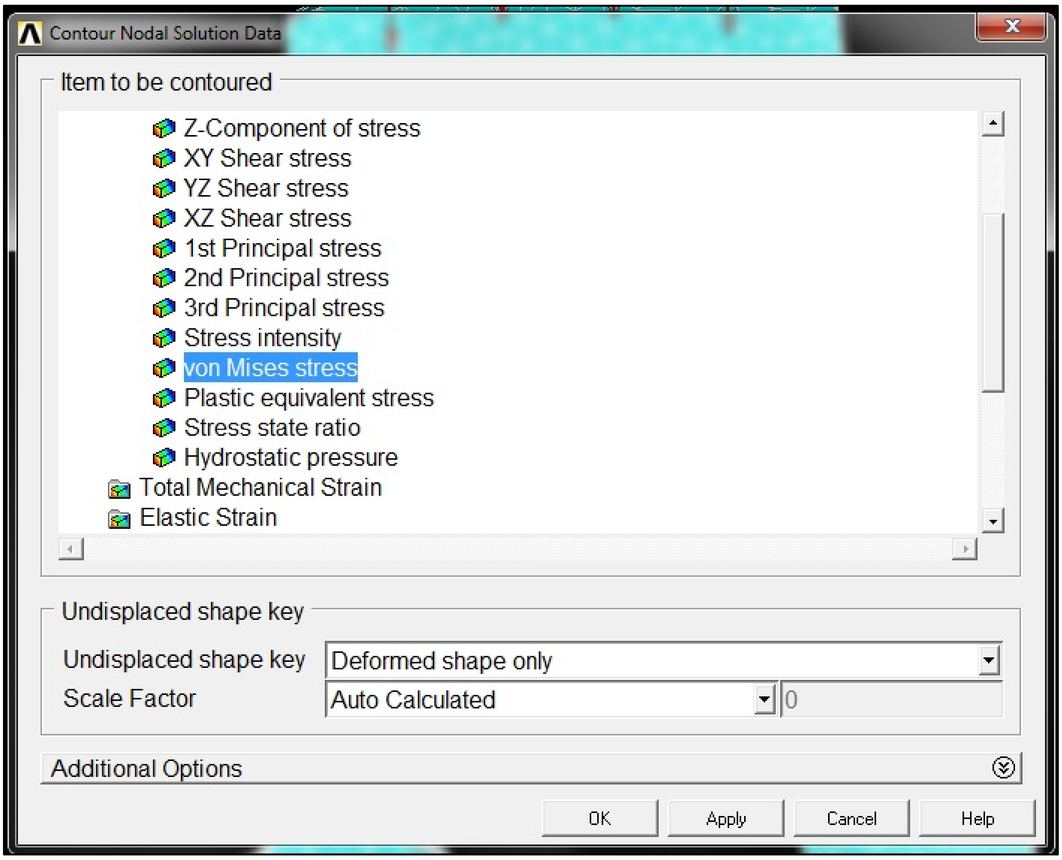


Figure 55: Contour Nodal Solution Data

Once OK button was clicked, the stress contour diagram will be displayed.