CHAPTER 1
INTRODUCTION

Chapter 1 introduces and briefly explains the Final Year Project of "Stress Analysis on Four Battered-legs Offshore Jacket Structures". This chapter comprises of the Background of the project, Problem Statement, the Significance of conducting this project, followed by the Objectives and Scope of Work of this Final Year Project and last but not least, the Feasibility of this project.

1.1 Background

Jacket is a structure that supports and positions the upper part of an oil platform at a predetermined height above sea level, and at the exact location above an oil reservoir. An arrangement of tubular steel columns make-up most of the jacket structure, with the principle vertical or sometimes slanting (*batter)* columns that constitute the foundation of the jacket normally referred to as the *jacket legs*. In between these jacket legs, arrays of smaller steel grill, normally arranged in a criss-cross manner, make-up the entire jacket structure. Jackets are connected to topsides by means of welding and by utilizing the combination of parts such as *transition pieces, leg-mating units (LMU)* and *sandscan/receptor.* The strength of the jacket legs are very important to withstand any forces or stresses that act to the jacket legs thus to avoid the oil platform to collapse.

1.2 Problem Statement

Jackets are one of the most important and main structures for offshore oil platform. Jacket is the structure that holds the topside of the oil platform in place and above sea level. The welded steel members are supporting beams which consist of compression and tension members that are welded together to the main steel structure.

Jacket structures are also submerged underwater and very vulnerable to corrosion, environmental loads (static and dynamic) and marine growth, which the structure must be able to withstand if they are to support the topside without experiencing failures and collapses. In this project, study will cover the static analysis of deck loadings that is acting on the jacket structures only. These structures will be analyzed based on the stresses generated on respective nodes and deformation. Study will also cover the relevancy of types of suitable structure to withstand respective amount of deck loading.

1.3 Significance of the Project

Due to inherent random nature of various input parameters affecting the response of these structures, reliability analysis assumes greater importance in the design of offshore structures. The results from this project would provide further understanding regarding offshore jacket durability to withstand extreme deck loads and to overcome, analyse, compare and choose the best between two different types of structures. This will eliminate any possible of accidents or mishaps on offshore oil platform. Study will be commenced on the relationship between stress sustained by the structures and load.

The purpose of this project is to propose a jacket structure with same or potentially higher durability, ability to withstand extreme loading, lighter by weight, using lesser materials to be built and cost-effective . In this study, the analysis of jacket structures with different measurements would enable potential benefits and improvements of jacket structures in the future. This project will use two (2) different measurements of jacket structure to produce two (2) jacket structures with different cross-sectional areas.

1.4 Objectives and Scope of Study

The objective of this project is to apply the knowledge of stress analysis (particularly static analysis) to propose a better jacket structure to be able to withstand extreme deck loads and to compare between two (2) different dimensions of jacket structures and evaluate the results obtain.

The objectives of this project are the following:

1. To draw the two jacket structures using CATIA V5 R18.
2. To apply two deck loads acting on the two (2) jacket structures.
3. To run static analysis on the two jacket structure drawings.
4. To simulate the response of the structure using ANSYS 12.0.
5. To evaluate maximum stresses and Von Mises stresses of the structure.

Final step will be determining the best jacket structure to withstand those two (2) deck loadings.

1.5 Feasibility of the Project

The time frame given for this project is one year. This project is divided into two parts: First semester will be Final Year Project (FYP) 1, second semester will be Final Year Project 2 (refer to Chapter 3, section 3).

CHAPTER 2
LITERATURE REVIEW

2.1 Topside and Jacket

Offshore platform consists of 2 main structures: (i) Topside and, (ii) Jacket.

Topside is the main deck that is installed on top of the jacket structure (refer to Figure 2.1). It consists of main deck, helideck, galley, living quarters/accommodation, control centre, support equipments, hosting the separators that separate the gas and oil from unwanted substances. The steel jacket type platform on a pile foundation is by far the most common kind of offshore structure and exists worldwide.

Jackets, known as “substructure”, are fabricated from steel welded pipes and are pinned to the sea floor with steel piles, which are driven through piles guides on the outer members of the jacket (refer to Figure 2.2). The piles are thick steel pipes of 1 to 2 meters diameter and can penetrate as much as 100 m into the sea bed. The jacket can weigh up to 20,000 tonnes and able to support topside that weights up to 21,000 tonnes.

To ensure that the installation will last for the required service life, maintenance must be carried out including the cathodic protection.

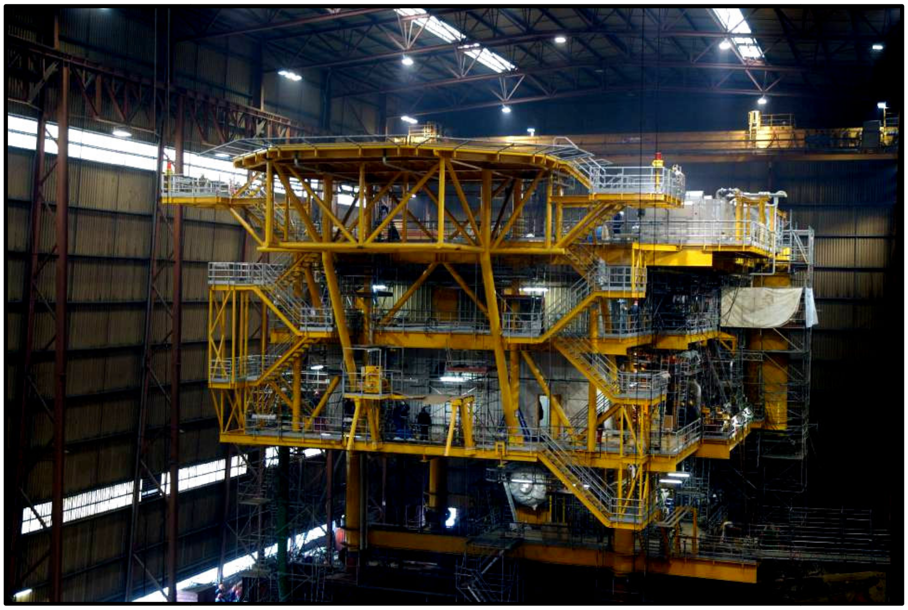

Figure 2.1: Topside Hasdrubal ‘A’ by Heerema Fabrication Group (HFG). Adapted from Rigzone.com

Figure 2.2: Jacket structure

2.2 Stress Analysis

Stress analysis is an engineering discipline that determines the stress in materials and structures subjected to static or dynamic forces or loads.

The aim of the analysis is usually to determine whether the element or collection of elements, usually referred to as a structure, can safely withstand the specified forces. This is achieved when the determined stress from the applied force is less than the ultimate tensile strength, ultimate compressive strength or fatigue strength the material is known to be able to withstand, though ordinarily a factor of safety is applied in design.

The factor of safety is a design requirement for the structure based on the uncertainty in loads, material strength, and consequences of failure. Often a separate factor of safety is applied to the yield strength and to the ultimate strength. The factor of safety on yield strength is to prevent detrimental deformations and the factor of safety on ultimate strength is to prevent collapse. The factor of safety is used to calculate a maximum allowable stress.

When performing stress analysis, a factor of safety is calculated to compare with the required factor of safety. The factor of safety is a design requirement given to the stress analyst. The Analyst calculates the design factor. Margin of safety is another way to express the design factor. A key part of analysis involves determining the type of loads acting on a structure, including tension, compression, shear, torsion, bending, or combinations of such loads.

Sometimes the term stress analysis is applied to mathematical or computational methods applied to structures that do not yet exist, such as a proposed aerodynamic structure, or to large structures such as a building, a machine, a reactor vessel, a piping system or a large beams structure.

A stress analysis can also be made by actually applying the force to an existing element or structure and then determining the resulting stress using sensors, but in this case the process would more properly be known as testing. In this case special equipment, such as a wind tunnel, or various hydraulic mechanisms, or simply weights is used to apply the static or dynamic loading.

When forces are applied, or expected to be applied, repeatedly, nearly all materials will rupture or fail at a lower stress than they would otherwise. The analysis to determine stresses under these cyclic loading conditions is termed fatigue analysis and is most often applied to aerodynamic structural systems.

The evaluation of loads and stresses within structures is directed to finding the load transfer path. Loads will be transferred by physical contact between the various component parts and within structures. The load transfer may be identified visually, or by simple logic for simple structures. For more complex structures, more complex methods such as theoretical solid mechanics or by numerical methods may be required. Numerical methods include direct stiffness method which is also referred to as the finite element method.

The object is to determine the critical stresses in each part, and compare them to the strength of the material. For parts that have broken in service, a forensic engineering or failure analysis is performed to identify weakness, where broken parts are analyzed for the cause or causes of failure. The method seeks to identify the weakest component in the load path. If this is the part which actually failed, then it may corroborate independent evidence of the failure. If not, then another explanation has to be sought, such as a defective part with a lower tensile strength than it should for example.

2.3 Static Analysis

Statics analysis is the analysis of statics mechanism that acted on rigid bodies, deformable bodies and fluid mechanics. Statics deals with the equilibrium of bodies, either at rest or move with constant velocity. Statics involves the Newton's Three (3) Laws of Motion. They apply to the motion of a particle as measured from a non accelerating reference frame. (Hibbler, R.C, 2004)

Statics is used in the analysis of structures. Strength of materials is a related field of mechanics that relies heavily on the application of static equilibrium. A key concept is the center of gravity of a body at rest which represents an imaginary point at which all the mass of a body resides. The position of the point relative to the foundations on which a body lies determines its stability towards small movements.

If the center of gravity exists outside the foundations, then the body is unstable because there is torque acting: any small disturbance will cause the body to fall or topple. If the center of gravity exists within the foundations, the body is stable since no net torque acts on the body. If the center of gravity coincides with the foundations, then the body is said to be metastable.

2.4 CATIA V5 R18

CATIA V5 R18 is an engineering virtual product design that is applied to wide varieties to industries, from aerospace, automotive and industrial machinery to electronics, shipbuilding, plant design and consumer goods, even clothing. CATIA facilitates reuse of product design knowledge and shortens development cycles, helping enterprises accelerate their response to market needs.

CATIA V5 R18 extends the end-to-end composites process from design to simulation to manufacturing for automotive and aerospace industries. CATIA V5 R18 also enables an effective Mechanical CAD with its new improved technology and collaboration. Its structural functional design has improved and changed multiply. The drafting solution for classification is vastly upgraded and its performance has been upgraded by 50% on average.

CATIA V5 R18 has enhanced Tooling capabilities and efficiency with dedicated functionality. It provides productivity gains for toolmakers, as it streamlines the Design procedures of complex Mold Tooling and gives more freedom and automation tools. It also enhances design to manufacturing performance with the introduction of a new assembly feature that generates a single "machined" part from an assembly of parts without modifying the reference part. It also has a strengthened 3D master product representation and maximizes users' productivity. The 3D master process enables users to precisely produce the same product that is conceptualized at the earlier stages with a unique associative approach.

CATIA V5 R18 allows users to easily perform 2D and 3D conceptual designs in a complex environment, including relational design with existing components. The software continues dramatically improve design productivity with revolutionary capability in the market. It delivers a new, unique breakthrough, Auto-draft, which complements the unmatched Auto-fillet capability introduced in the previous versions. It brings stunning productivity to automate powertrain and chassis designers to foundry tooling design and optimizes the resulting casting parts. The drafting activity is shortened considerably as all drafts are computed in a single operation.

2.5 ANSYS 12.0

ANSYS, Inc. is an engineering simulation software provider founded by software engineer John Swanson. It develops general-purpose finite element analysis and computational fluid dynamics software. While ANSYS has developed a range of computer-aided engineering (CAE) products, it is perhaps best known for its ANSYS Mechanical and ANSYS Multiphysics products.

ANSYS Mechanical and ANSYS Multiphysics software are non exportable analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a graphical user interface.

These are general-purpose finite element modeling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

ANSYS Mechanical technology incorporates both structural and material non-linearities. ANSYS Multiphysics software includes solvers for thermal, structural, CFD, electromagnetics, and acoustics and can sometimes couple these separate physics together in order to address multidisciplinary applications. ANSYS software can also be used in civil engineering, electrical engineering, physics and chemistry.

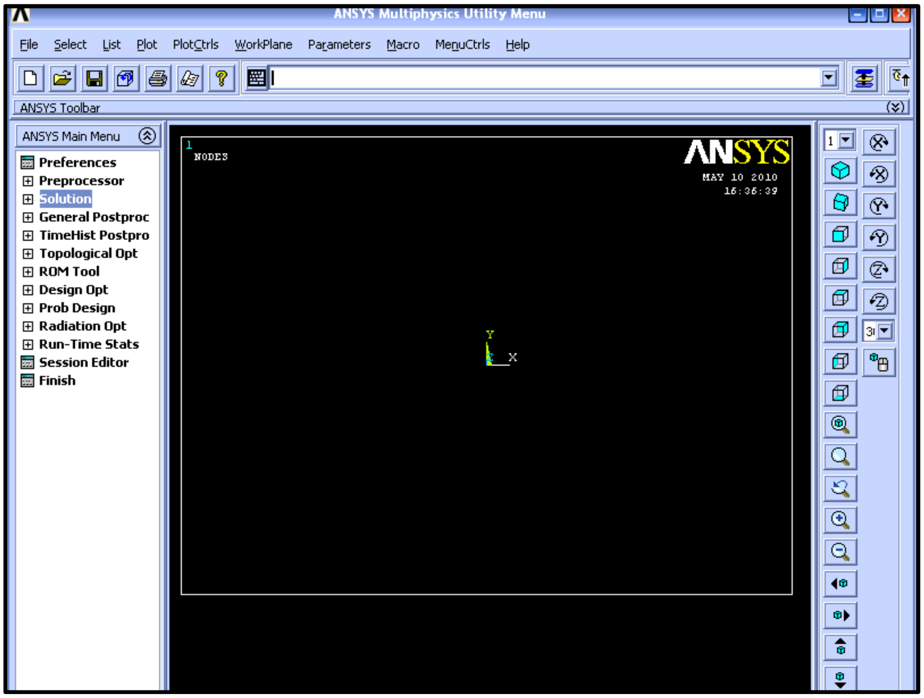

Figure 2.3: Picture of ANSYS 12.0 workbench

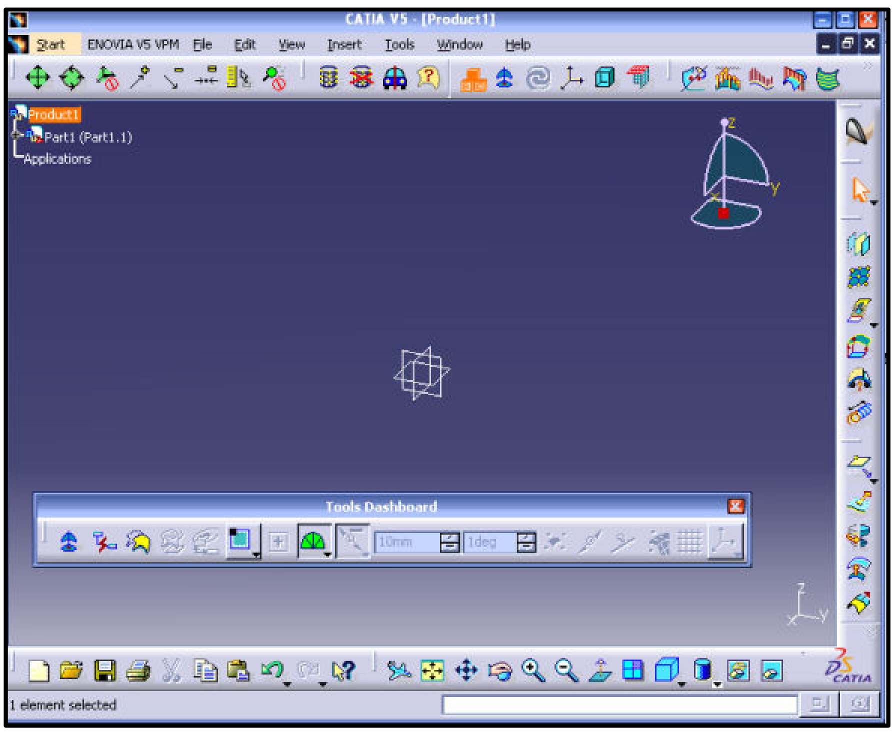

Figure 2.4: Picture of CATIA V5 R18 workbench.

2.6 High Strength Alloy Steel A514

Steel A514 is a particular type of high strength steel, which is quenched and tempered alloy steel, with basic strength of 100,000 psi (100 ksi, where 1 ksi = 1,000 psi) and its maximum yield strength is up to 700 MPa. A514 is primarily used as a structural steel for building construction.

The tensile yield strength of A514 alloys is specified as at least 100 ksi (690 MPa) for thicknesses up to 2.5 inch (64 mm) thick plate, and at least 110 ksi (760 MPa) ultimate tensile strength, with a specified ultimate range of 110–130 ksi (760–900 MPa). Plates from 2.5 to 6.0 inches (64 to 152 mm) thick have specified strength of 90 ksi (620 MPa) (yield) and 110–130 ksi (760–900 MPa) ultimate tensile strength.

A514 steels are used where a weld able, be able to machined, very high strength steel is required to save weight or meet ultimate strength requirements. It is normally used as a structural steel in building construction, jackets, cranes, or other large machines supporting high loads. (Manual of Steel Construction, 8th Edition, 2nd revised printing, American Institute of Steel Construction (1987), chapter 1 page 1-5). The table properties of Steel A514 are stated below:

Table 2.1: Mechanical Properties of Steel A514

Properties	**Values**
Density (x 1000 kg/m³)	8.03
Poisson's Ratio	0.30
Young's Modulus (GPa)	205
Yield Strength (MPa)	690
Ultimate Yield Strength (MPa)	760

CHAPTER 3
METHODOLOGY

This project was divided into two sections of the time course given for completion of the project. The first section is Final Year Project I and the second section is Final Year Project II.

3.1 Final Year Project I

Before proceeding with the experiment itself, a project plan was drawn out to clear out on the flow of the project (refer Figure 3.1). A rough design of a prototype of the jacket structure has been made by using CATIA V5 R18. During this time period, information on the Jacket leg beams and the knowledge pertaining to this project was searched and obtained. Sources of knowledge include past journals, research and previous studies performed by engineering researchers, internet, and books from the libraries. The collections of technical details regarding stress analysis, and the data of maximum stresses that act on Jacket legs was acquired. Besides data pooling, computerized analyzer such as CATIA and ANSYS softwares were familiarized for application in the second section of the Final Year Project II.

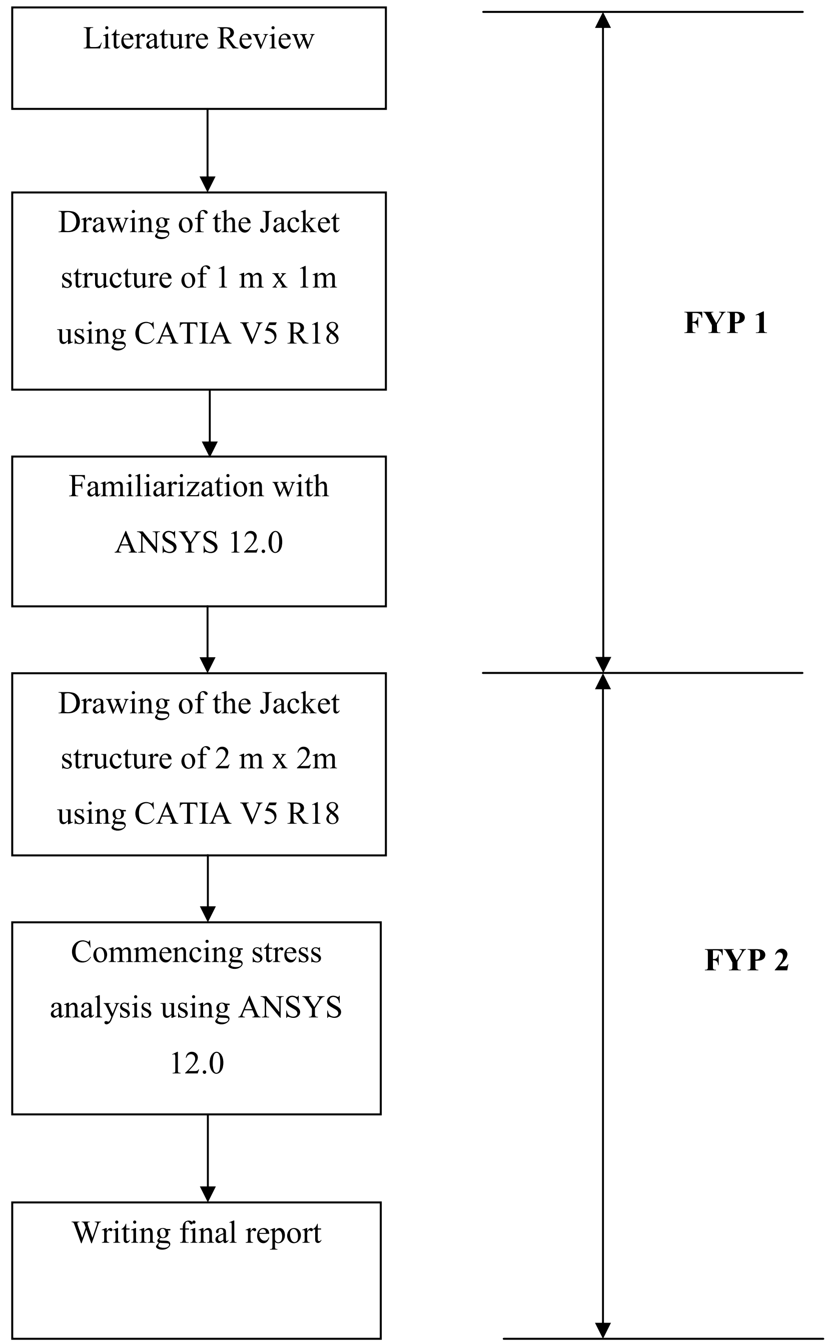

Figure 3.1: Flowchart of Project Plan.

3.2 Final Year Project II

FYP II initially involves the application of CATIA V5 R18 to model jacket structure drawings of two jacket structures related for this project, which are:

(i) Jacket Structure with 1 m x 1 m cross-sectional area steel members,
(ii) Jacket Structure with 2 m x 2 m cross-sectional area steel members.

The drawings were then imported from CATIA to ANSYS to be meshed in order to begin simulation of the jacket structures. In the simulation, variable deck loads of 20 MN and 200 MN were applied onto each jacket structure. Subsequently, static analysis was conducted by using the ANSYS software. Through the application of ANSYS, the stress endured by the jacket structure, the beams of jacket legs and the response from both the structure and the jacket beams are better illustrated and explained. Finally, results from the simulations were obtained and the comparisons of the two jacket structures were made.

3.2.1 Manual Calculations

a) Deck loading calculations

Using Newton's 2nd Law, F = mass x acceleration:

i) Deck load of 2000 tonnes:

F1 = mass x gravity
= (2000x 1000kg) x g
= 2000000kg x 9.81 m/s²
= 19620000 kg.m/s² = 20 MN

ii) Deck load of 20000 tonnes:

F2 = mass x gravity
= (20000 x 1000kg) x g
= 20000000 kg x 9.81 m/s²
= 196200000 kg.m/s² = 200 MN

b) Deck loading on each nodes calculations

These nodes are the nodes located on the top surface of the head jacket and each node was experiencing these amounts of forces stated below:

i) Jacket Structure 1 m x 1 m (123 nodes)

20 MN → = 20 MN / 123

= 162601.63 N / node

200 MN → = 200 MN / 123

= 1626016.26 N / node

ii) Jacket Structure 2 m x 2 m (132 nodes)

20 MN → = 2 MN / 132

= 15151.52 N / node

200 MN → = 200 MN / 132

= 1515151.5 N / node

3.2.2 Jacket Structures

Since this project focuses on the comparison of (a) Jacket Structure with 1m x 1m cross-section steel members and (b) Jacket Structure with 2m x 2m cross-section steel members, application of CATIA V5 R18 was used for two jacket structures. In CATIA V5 R18 utilization, typical four battered-leg jacket structure installed in medium water depth which is 37.45 m in average was used in this study. It measures 21 m x 21 m at the base and 12 m x 12 m at the top of the head of the jacket. The height of the jacket structure is 47.08 m. Jacket legs are battered to provide larger base for the structure at the mud line and therefore assist in resisting higher environmentally induced overturning moments. The structure is supported by a single pile at each leg. The deck will be fixed on top of these steel members of 12m x 12m. In this project we have two (2) drawings of jacket structures:

(a) Jacket structure of cross-sectional area of 1 m x 1 m steel members

The area of the top head jacket, A1:

$A1 = (12\ m \times 12\ m) - (10\ m \times 10\ m)$
$A1 = 144\ m^2 - 100\ m^2$
$A1 = \underline{44\ m^2}$

(b) Jacket structure of cross-sectional area of 2 m x 2 m steel members

The area of the top of the jacket, A2:

$A2 = (12\ m \times 12\ m) - (8\ m \times 8\ m)$
$A2 = 144\ m^2 - 64\ m^2$
$A2 = \underline{80\ m^2}$

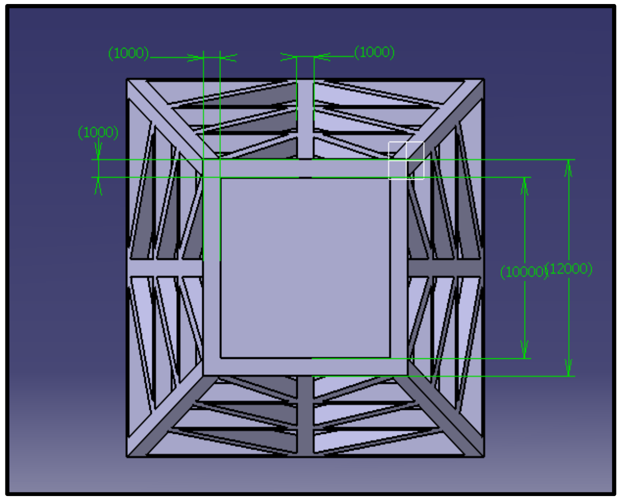

Figure 3.2: Top view of jacket structure of 1 m x 1 m steel members with dimensions.

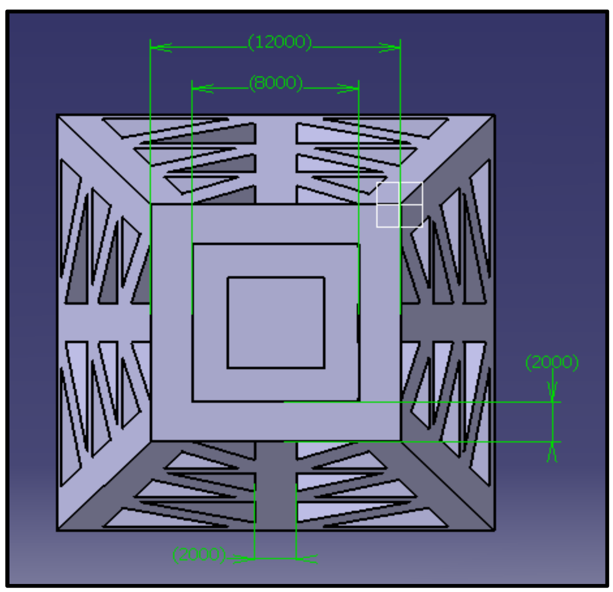

Figure 3.3 Top view of jacket structure of 2 m x 2 m steel members with dimensions.

3.2.3 Meshing

After the jacket drawings are completed, both of the drawings were imported from CATIA to ANSYS. In ANSYS, these drawings have to be meshed to obtain the nodes and before forces will be applied on the jacket structures. Before meshing these drawings, at ANSYS Main Menu, the Material Model number 1 of structural, linear, elastic and isotropic are being selected. The main material of this jacket structure is High Strength Alloy Steel ASTM A514, with the Young's Modulus of 205 GPa and Poisson ratio of 0.3 (Since the dimension of CATIA drawings are in mm, the Young's Modulus will be executed inside the 'Properties of Material' in Figure 3.4 in MPa instead of Pa). After done setting these two (2) parameters, the element type Tetrahedral 10 nodes 187 of Structural Mass Solid were chosen. The next step is to mesh the drawings. To mesh, Global element attributes was chosen because of the complex drawing and followed by meshing the structures by volume. After executing the mesh, Jacket with 1 m x 1 m cross-sectional area of steel members has the total of 37655 nodes. Jacket with 2 m x 2 m cross-sectional area steel members has the total of 146132 nodes.

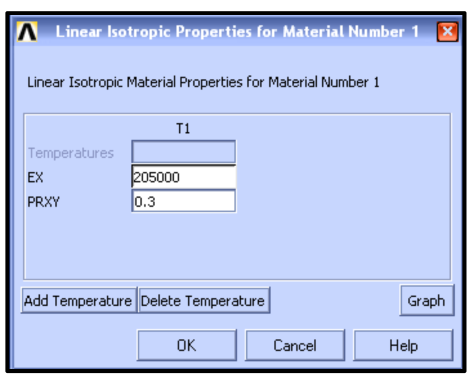

Figure 3.4: Properties of Material.

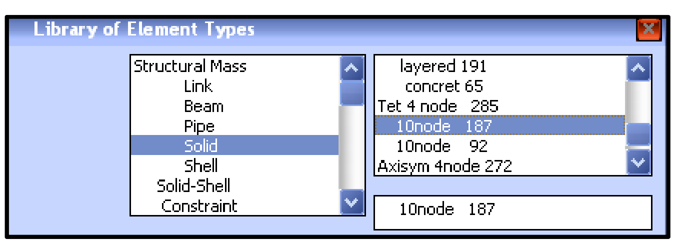

Figure 3.5: Element type selection.

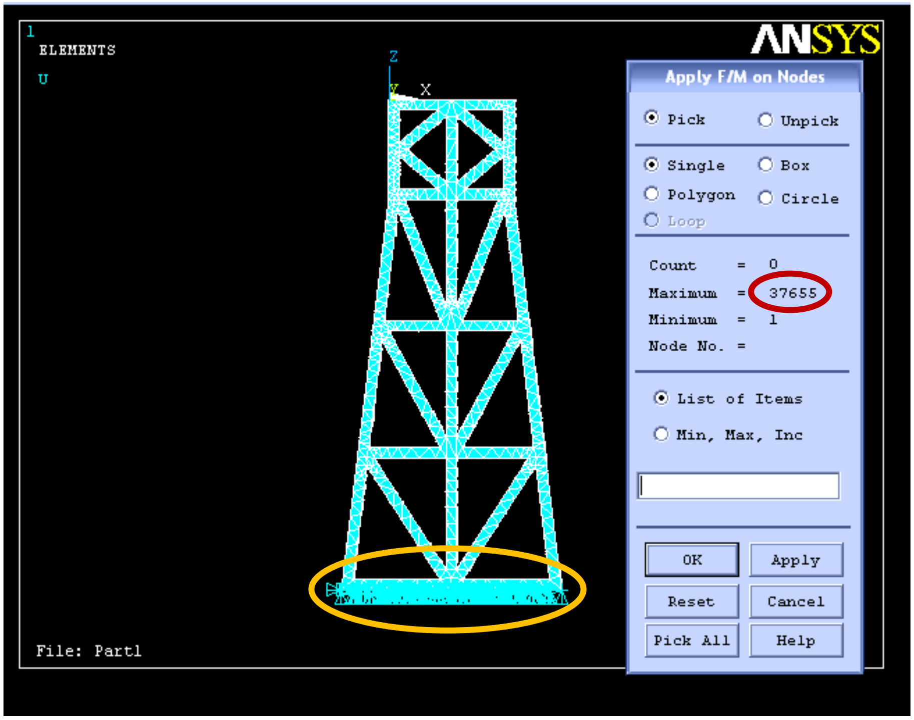

Figure 3.6: Meshed drawing of Jacket structure of 1 m x 1 m cross-sectional area steel members with number total of nodes (marked in red circle).

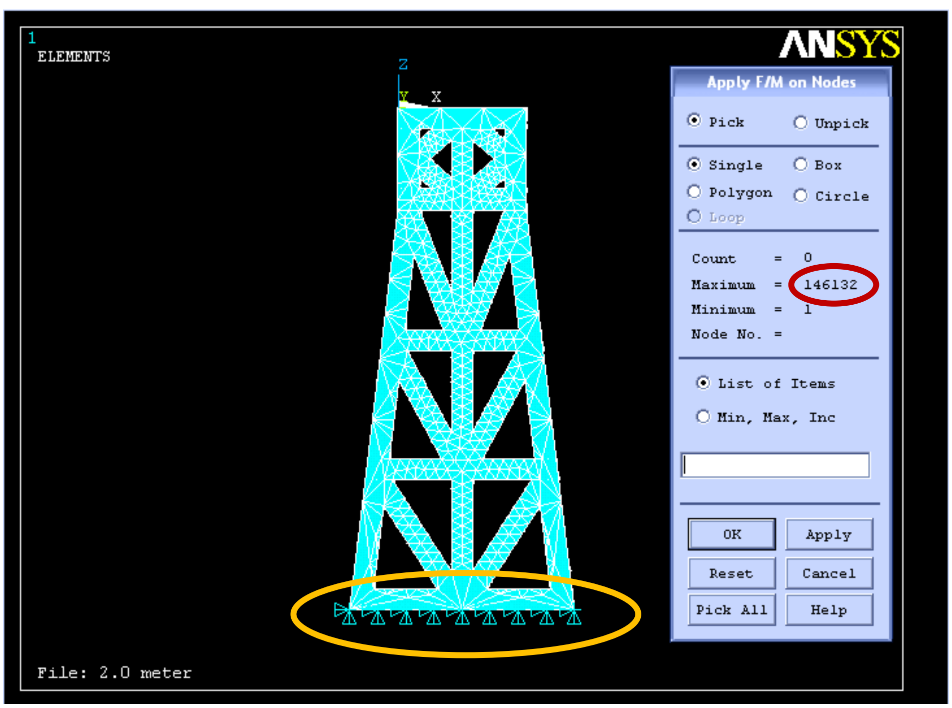

Figure 3.7: Meshed drawing of Jacket structure of 2 m x 2 m cross-sectional area steel members with number total of nodes (marked in red circle).

3.2.4 Stress Analysis (Static)

After meshing, the next step is to select the intended fixed nodes to apply the Degrees of Freedom (DOF). After executing, two (2) blue perpendicular triangles emerged at the nodes selected (refer to Figure 3.4 and 3.5, marked by yellow oval). The type of analysis for both analyses is static analysis. Forces of 2 MN and 20 MN were applied to both meshed drawings (refer to Chapter 3, section 3.2.1.a). These forces were applied on every each top nodes (refer to Chapter 3, section 3.2.1.b). All forces that were acting on the negative z-direction to indicate the forces acting alongside gravity pull.

After these forces were applied and the analyses were solved, the result can be seen by selecting the nodal solution Contour plot at the main menu "General Postproc". The results are displayed in Von Mises Criterion Contour Plot and the parameters are SMX, SMN, DMX, SINT and SEQV. SEQV is the Stress Equivalent, which is the average reading of the stresses that is acting on the structure and it is taken as the final reading to determine the Von Mises criterion.

3.2.5 Equipment Tools

i) CATIA V5 R18
ii) ANSYS 12.0

3.2.6 Comparison of Results

Results from both jacket structures were compared and evaluated. The parameters considered in comparisons were:

(i) Cross-sectional area of the steel members between two drawings
(ii) Deformed shape between two simulated drawings
(iii) Von Mises Contour Plot between two simulated drawings
(iv) Stress Intensity (SINT) and Stress Equivalent (SEQV)

CHAPTER 4
RESULTS AND DISCUSSIONS

4.1 Drawings using CATIA V5 R18

Jacket structure drawings obtained through application of CATIA V5 R18. The drawings consist of the rectangle head structure which is connected with four battered legs and completes with its bottom. For Figure 4.1 and 4.2, all of the steel members are sized of 1 m x 1 m. As in for Figure 4.3 and 4.4, the steel members are sized of 2 m x 2 m.

These drawings were drawn in a simplified manner. It is necessary because of the used of meshing and also because of the lack of accuracy geometric information for some components. Common components of a jacket structures such as buoyancy tanks, leg pile slots, mudmat, spider-rail and ballast tanks at the legs all were neglected while commencing the drawings. Since the needed results from the analysis was the amount of stress acting on the structures, small lacking in accuracy of the drawings were deemed inconsequential.

“Part design” under “Mechanical design” was chosen to draw these structures. Most applications such as blocks, constraint, mirror, translation extrude and lines were used.

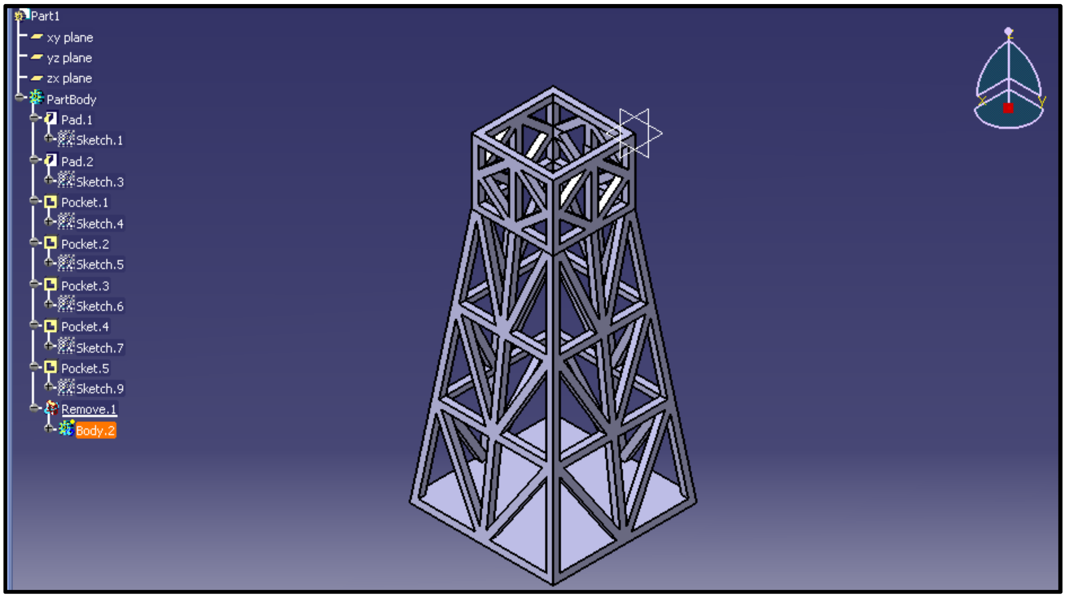

Figure 4.1: Jacket Structure drawing (isometric view) with 1 m x 1 m cross-sectional area of steel members.

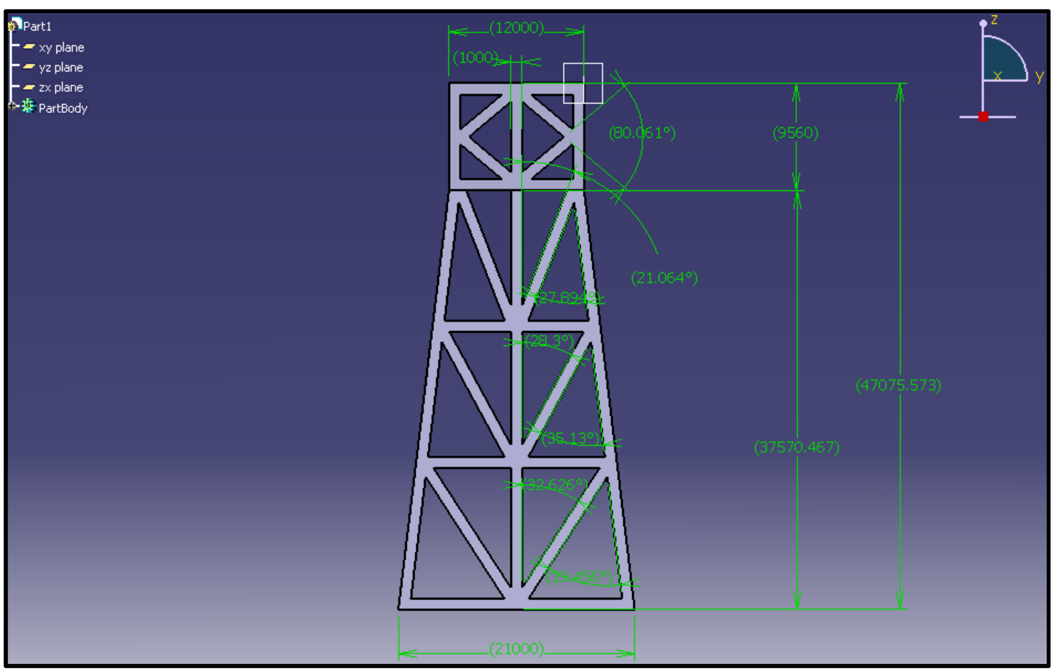

Figure 4.2: Jacket Structure drawing with 1 m x 1 m cross-sectional area of steel members with dimensions.

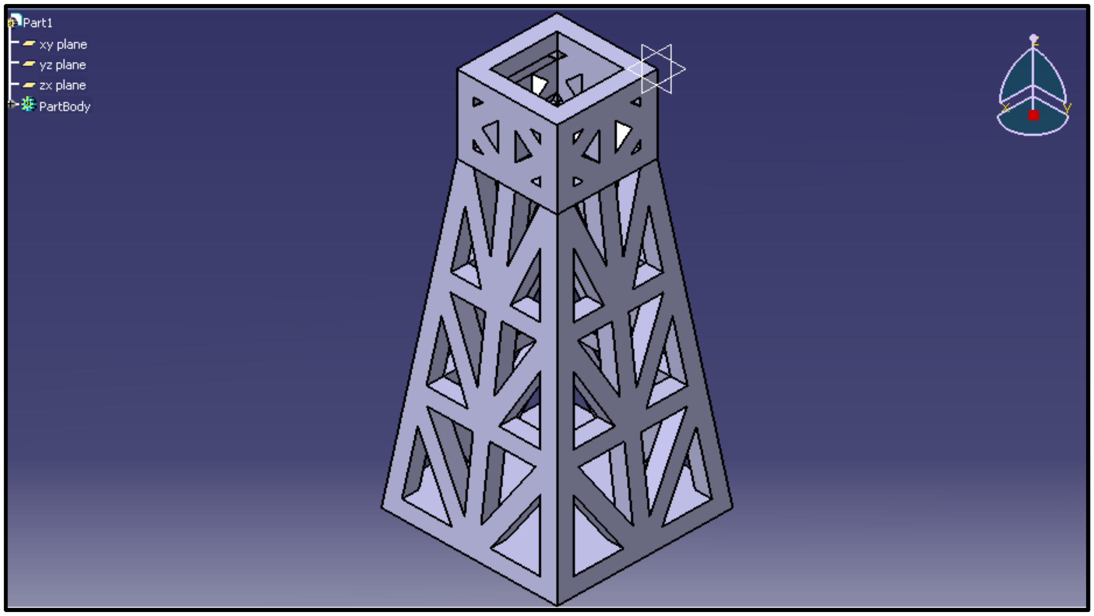

Figure 4.3: Jacket Structure drawing (isometric view) with 2 m x 2 m cross-sectional area of steel members.

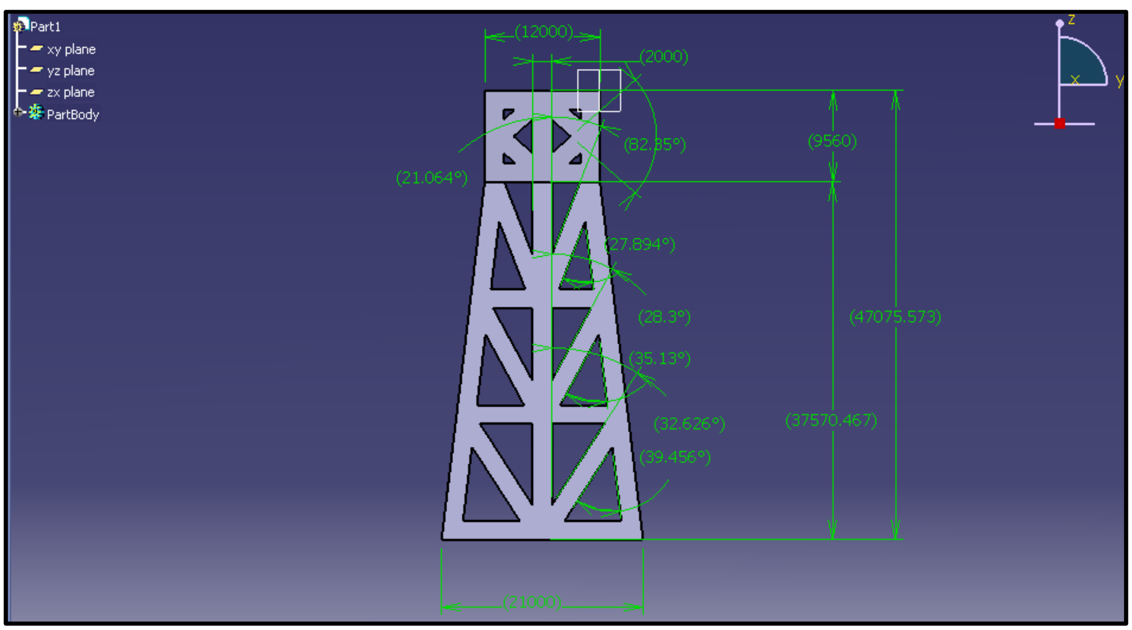

Figure 4.4: Jacket Structure drawing with 2 m x 2 m cross-sectional area of steel members with dimensions.

4.2 Jacket Structure Analysis Using ANSYS 12.0

4.2.1 Jacket Structure with 1m x 1m cross-sectional area Steel Members

In this study, the jacket structure was exerted with deck loads of 20 MN and 200 MN. The following graphics below are the product of ANSYS 12.0. For jacket structure with cross sectional area of 1 m x1 m, the amount of nodes on the area of the top of the jacket structure after being meshed are 123 nodes. (refer to Chapter 3 above)

a) Application of Deck Load of 20 MN on the nodes at the top of the jacket structure

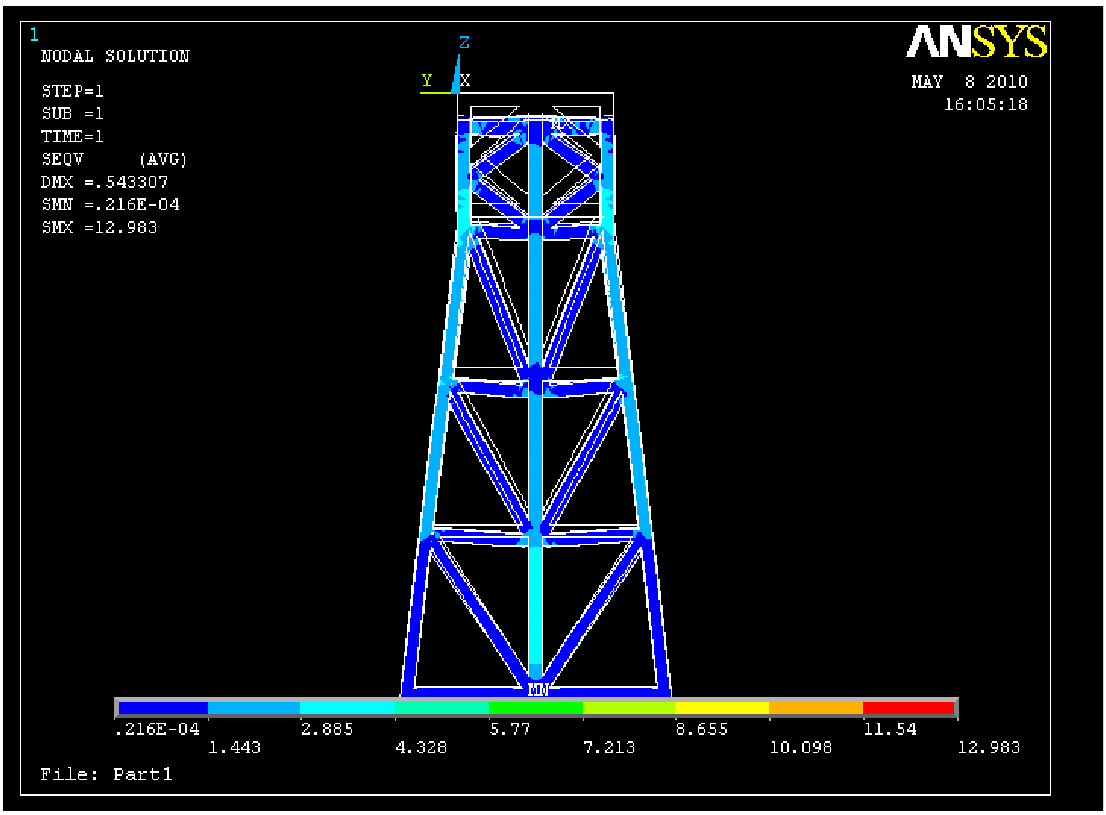

Figure 4.5: Von Mises nodal solution (Contour plot normal view) for Jacket Structure with 1 m x 1 m cross-sectional area steel members at deck load of 20 MN.

In Figure 4.5, it can be observed that the structure is deformed from its initial shape (white line). The Stress Maximum (SMX) obtained is 12.983 MPa. It is relatively too low for this huge structure to be evaluated. Therefore, the ideal weight of the deck load is switched to 200 MN.

b) Application of Deck Load of 20 MN on the nodes at the top of the jacket structure

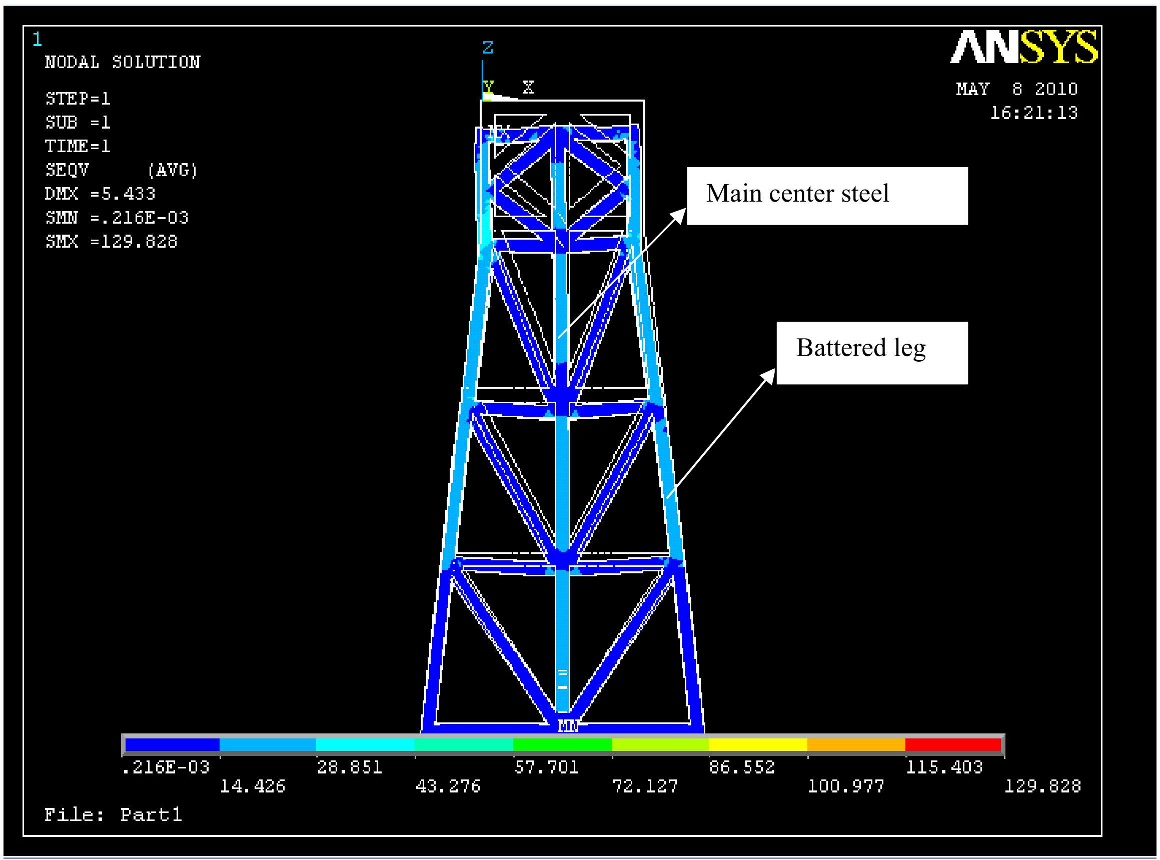

Figure 4.6: Von Mises nodal solution (Contour plot normal view) for Jacket Structure with 1 m x 1 m cross-sectional area steel members at deck load of 200 MN.

Based on Figure 4.6, the structure is deformed from its initial shape (white line) after being applied the deck load of 200 MN (1626016.26 N on every each node in the negative z-direction) on top of the jacket (refer to Chapter 3). Stress Maximum (SMX) obtained is 129.828 MPa. It is observed that stresses intensity are focused on the battered leg of the jacket and also at four main center steel members of the jacket structure. This can be determined by looking at the Figure 4.7, different contour coloration in which from dark blue coloration that represents 0.216E-03 MPa to light blue/cyan coloration that represents ranges between 28.851 MPa to 43.276 MPa. This is more logical and acceptable compared to results from Figure 4.5.

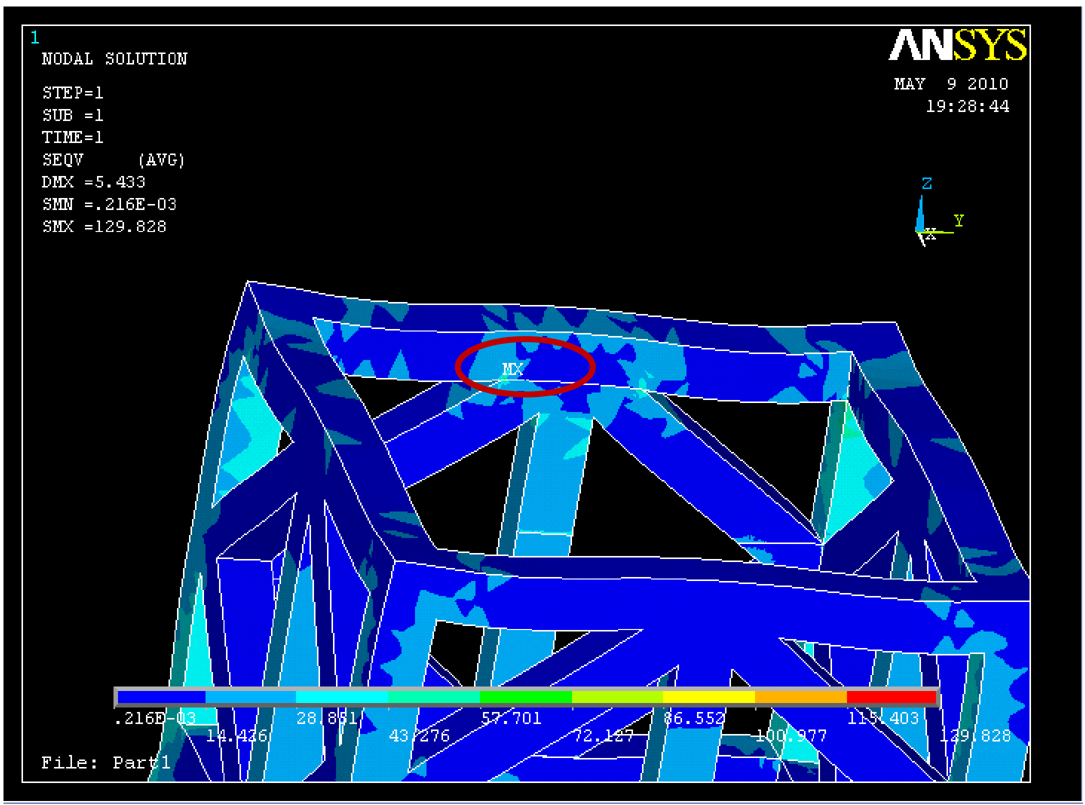

Figure 4.7: Von Mises nodal solution Contour plot (location of SMX) for Jacket Structure with 1 m x 1 m cross-sectional area steel members at deck load of 200 MN.

From Figure 4.7, SMX is located in between joint of steel members at the head area of the jacket.

MINIMUM VALUES					
NODE	6099	6099	6098	19719	19719
VALUE	-21.967	-37.650	-119.97	0.69750E-03	0.60415E-03
MAXIMUM VALUES					
NODE	10175	10175	10137	6319	6319
VALUE	52.006	11.899	2.3768	111.49	107.23

Figure 4.8: List results of nodal solution (minimum and maximum values) for jacket structure with 1 m x 1 m cross-sectional area steel members at deck load of 200 MN.

Figure 4.8 illustrates the maximum value of Stress Equivalent (SEQV) is averaged at node 6319 with the value of 107.23 MPa. The maximum value of Stress Intensity (SINT) at node 6319 is 111.49 MPa.

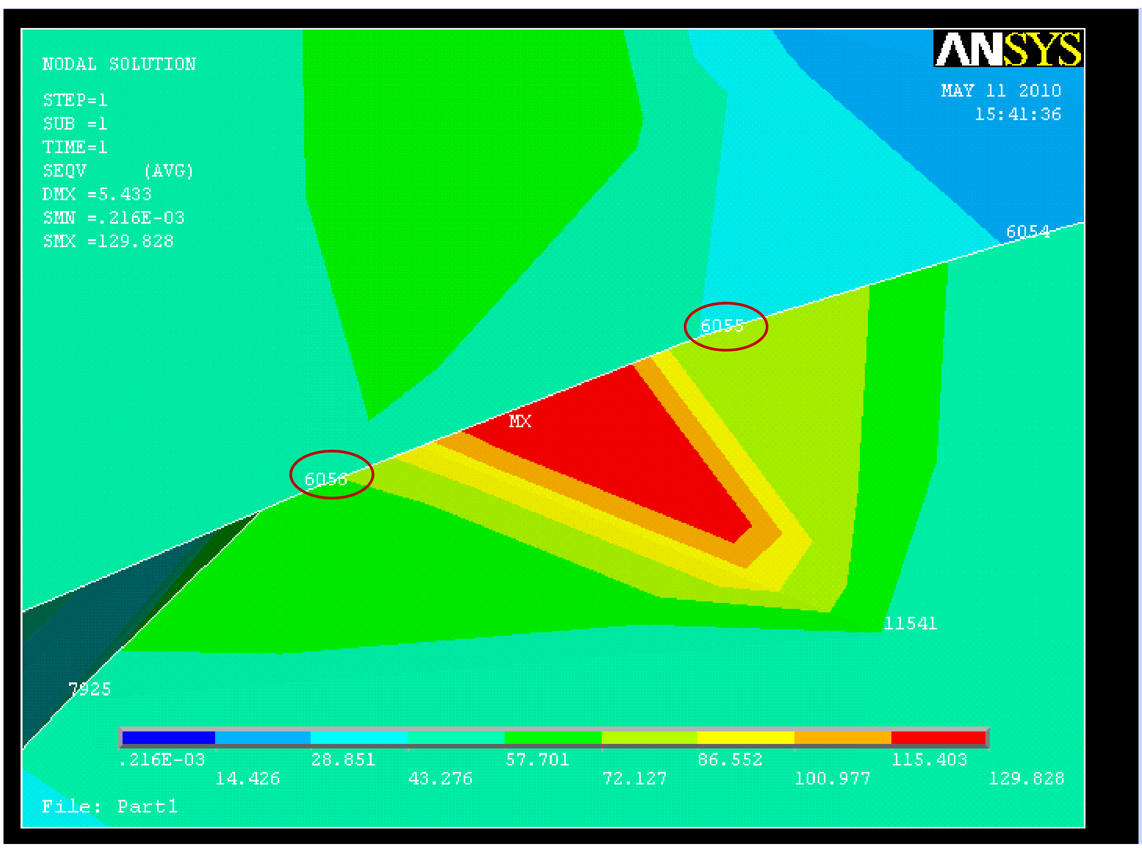

Figure 4.9: Contour plot of nodal solution (node 6055 and 6056) for jacket structure with 1 m x 1 m cross-sectional area steel members at deck load of 200 MN.

6051	-0.33157	-18.712	-22.287	21.956	20.410
6052	-0.10000	-20.548	-24.493	24.393	22.688
6053	0.15977	-18.178	-20.405	20.565	19.547
6054	2.8401	-17.103	-26.516	29.357	26.074
6055	4.9541	-19.535	-43.565	48.519	42.048
6056	6.4755	-6.5098	-54.683	61.158	55.890
6057	10.282	2.3169	-35.940	46.222	42.801
6058	6.4314	0.23938	-21.743	28.175	25.646
6059	7.0866	-0.49789	-11.621	18.707	16.297

Figure 4.10: List results of nodal solution (node 6055 and 6056) for jacket structure with 1 m x 1 m cross-sectional area steel members at deck load of 200 MN.

There are no specific nodes which lies on the exact contour SMX (red coloration), therefore nearest nodes will be considered. In Figure 4.9 shows that node 6056 is chosen because it is one of the nearest nodes located to the SMX Contour Plot (red coloration) and its value of stress lies in between light green coloration (57.701 MPa) and yellow coloration (86.552 MPa). To be more specific, the readings from the list result nodal solution of node 6056 are SINT = 61.158 MPa and SEQV = 55.890 MPa. Both readings are proven to be in between the range of 57.701 MPa to 86.552 MPa.

4.2.2 Jacket Structure with 2 m x 2 m cross-sectional area Steel Members

In this study, the jacket structure was exerted with deck loads of 20 MN and 200 MN. The following graphics below are the product of ANSYS 12.0. For jacket structure of 2 m x 2 m cross-sectional area, the amount of nodes on the area of the top of the jacket structure obtained after being meshed are 132 nodes (refer Chapter 3).

a) Application of Deck Load of 20 MN on the nodes at the top of the jacket structure.

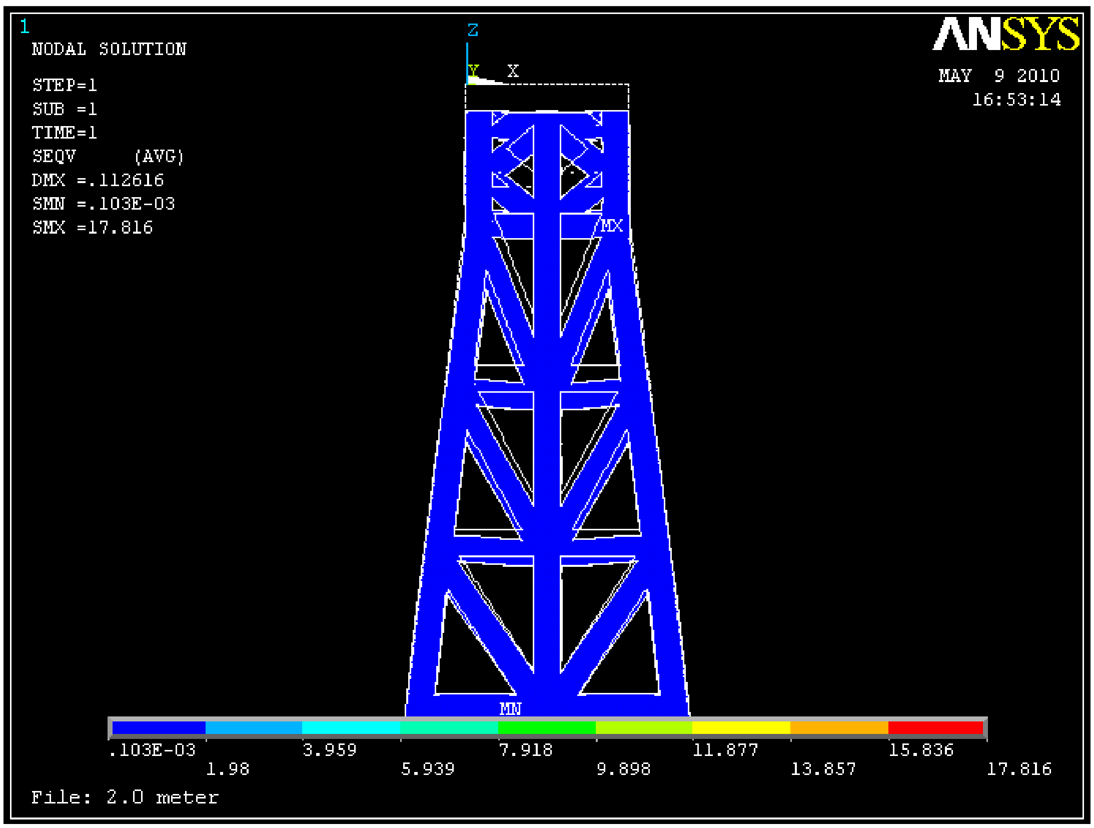

Figure 4.11: Von Mises nodal solution (Contour plot normal view) for Jacket Structure with 2 m x 2 m cross-sectional area steel members at deck load of 20 MN.

Based on Figure 4.11, it can be observed that the structure is deformed from its initial shape before load is applied (white line). The stress maximum obtained is 17.816 MPa. It is relatively too low for this huge structure to be evaluated. Therefore, the ideal weight of the deck load is switched to 200 MN.

b) Application of Deck Load of 200 MN on the nodes at the top of the jacket structure.

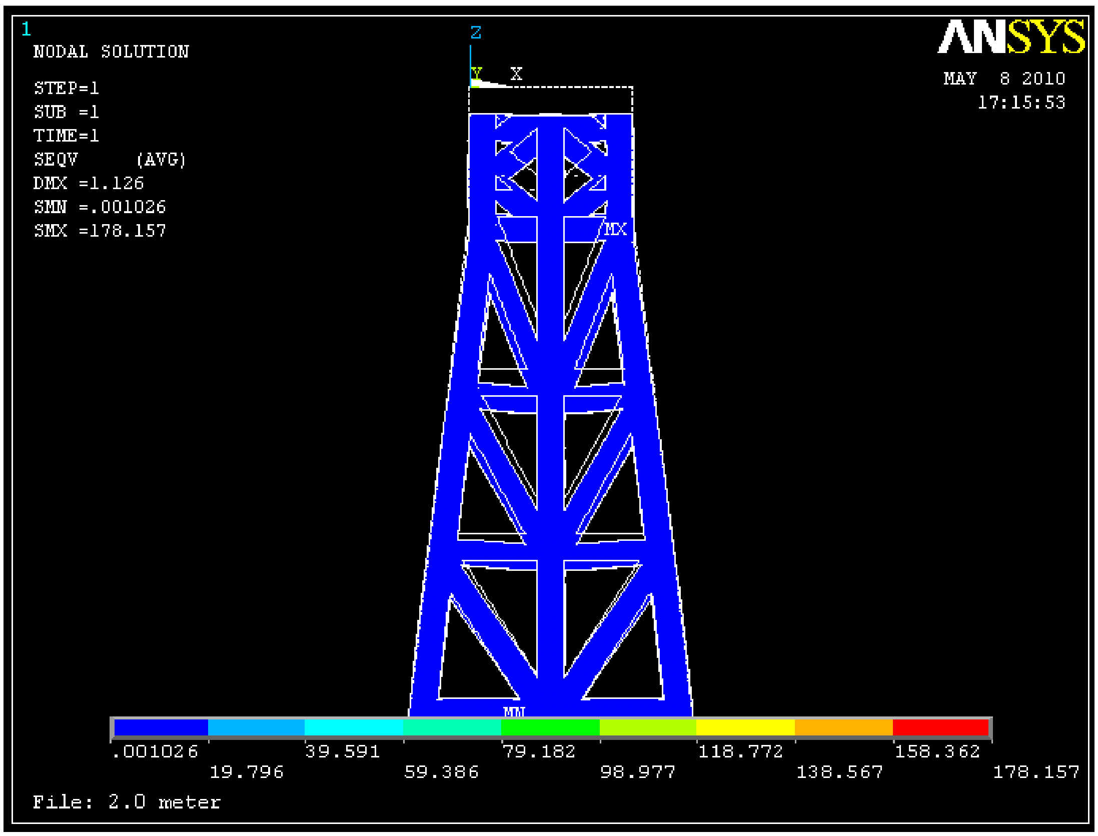

Figure 4.12: Von Mises nodal solution (Contour plot normal view) for Jacket Structure with 2 m x 2 m cross-sectional area steel members at deck load 200 MN.

Statement from 4.2.1 (a) is being adapted , based on Figure 4.12, the structure is deformed from its initial shape (white line) after being applied the deck load of 200 MN (1515151.5 N on every each node in the negative z-direction) on top of the jacket. Stress Maximum (SMX) obtained is 178.157 MPa. This is more logical and acceptable compared to results from Figure 4.10. The contour coloration at figure above is dark blue which representing 0.001026 MPa and does not showing the difference in color.

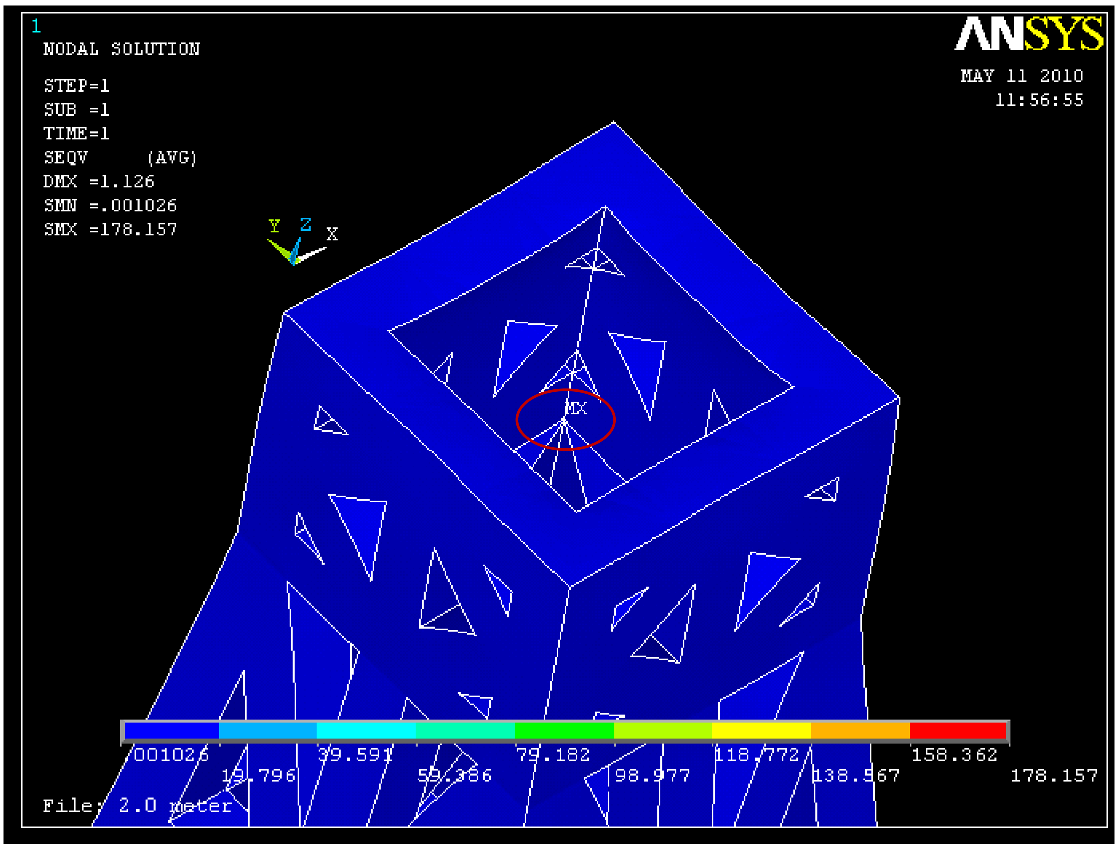

Figure 4.13: Von Mises nodal solution Contour plot (location of SMX) for Jacket Structure with 2 m x 2 m cross-sectional area steel members at deck load of 200 MN.

Figure 4.13 shows the SMX concentrates on the corner and the contour is gradually increasing when approaching the corner. So the range of the stresses is in between 0.001026 MPa to 178.157 MPa.

MINIMUM VALUES					
NODE	27666	5175	5175	34040	33889
VALUE	-30.987	-42.478	-132.46	0.99920E-02	0.92056E-02
MAXIMUM VALUES					
NODE	548	885	885	5175	27666
VALUE	46.013	12.755	8.8195	102.28	97.221

Figure 4.14: List results of nodal solution (minimum and maximum values) for jacket structure with 2 m x 2 m cross-sectional area steel members at deck load of 200 MN. Software used was ANSYS under PRINSOL Command.

From figure above, the maximum value of Stress Equivalent (SEQV) is averaged at node 27666 with the value of 97.221 MPa. The maximum value of Stress Intensity (SINT) at node 5175 is 102.28 MPa.

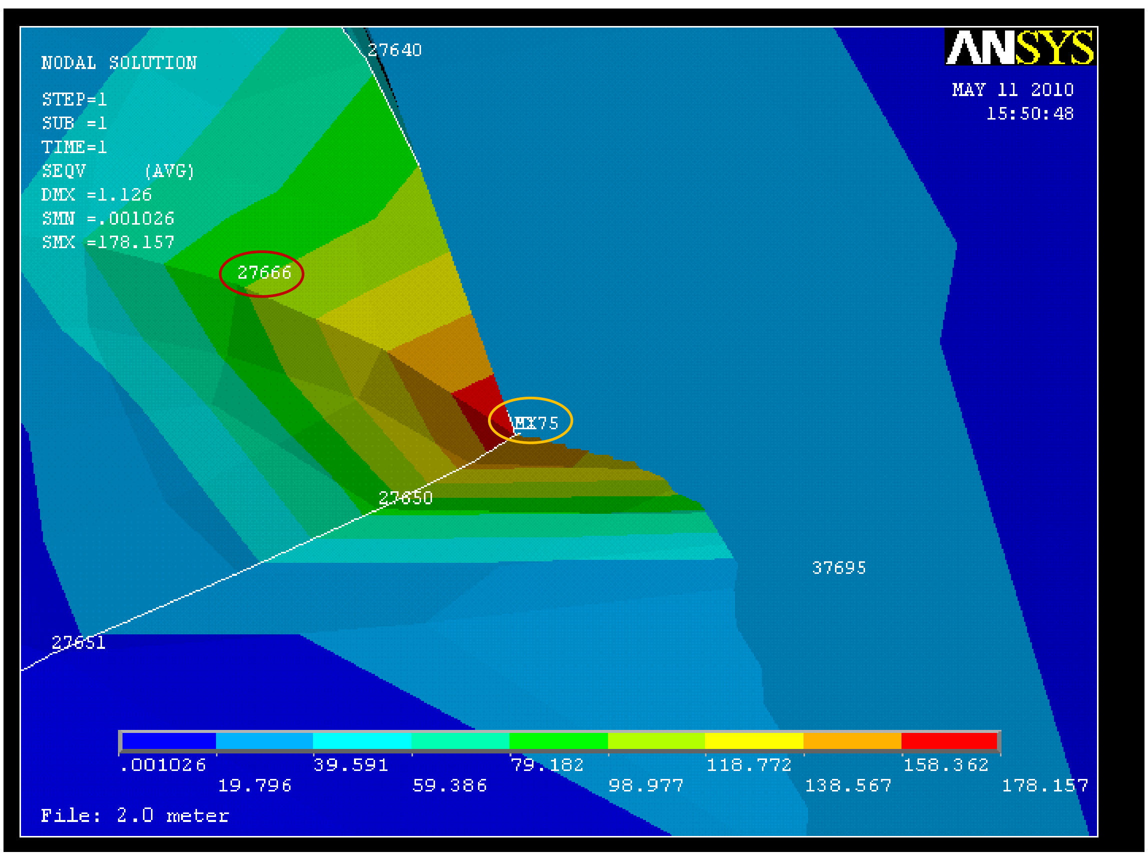

Figure 4.15: Contour plot of nodal solution (node 5175 and 27666) for jacket structure with 2 m x 2 m cross-sectional area steel members at deck load of 200 MN.

5174	1.6547	0.15774	-1.6207	3.2754	2.8561
5175	-30.177	-42.478	-132.46	102.28	96.935
5176	-1.4183	-2.4567	-13.817	12.399	11.916
5177	-0.93356	-3.3622	-13.979	13.045	12.016
5178	-1.3445	-3.8385	-14.598	13.253	12.201
5179	-1.2030	-4.5137	-15.777	14.574	13.248
27665	19.179	12.413	3.8731	15.306	13.285
27666	-30.987	-38.960	-131.95	100.96	97.221
27667	4.3010	1.1802	-5.1672	9.4683	8.3570
27668	13.153	5.6860	0.47387	12.679	11.038
27669	2.0036	-5.2787	-25.557	27.560	24.736
27670	-9.5690	-15.741	-64.879	55.310	52.497

Figure 4.16: List results of nodal solution (node 5175 and 27666) for jacket structure with 2 m x 2 m cross-sectional area steel members at deck load of 200 MN.

There were no specific nodes which lie on the exact contour SMX (red coloration); therefore nearest nodes will be considered. Based in Figure 4.15, node 5175 and 27666 are one of the few nodes that are nearest to the SMX contour. The value of SEQV of node 27666 is averaged at 97.221MPa which lies in between green coloration of 79.182 MPa and green yellow coloration of 98.977 MPa. The SINT is recorded at 102.28 MPa.

4.3 Comparison and Discussion

Based on Figure 4.6, the battered leg of the jacket and also at four main center steel members of the jacket structure coloration indicates that those parts experiencing the more stress intensity overall. This occurs because the battered leg of the jacket and also at four main centers steel members of the jacket structure are the main parts of the structure that holds the weight of the topside.

From the results obtained, the jacket structure of 2 m x 2 m possesses a slightly lower Stress Equivalent (SEQV) and Stress Intensity (SINT) than the jacket structure of 1 m x 1 m steel members (refer Table 4.1). However, the jacket structure of 2 m x 2 m experiences more Stress Maximum (SMX) and Stress Minimum (SMN) than jacket structure of 1 m x 1m.

Based on Figure 4.6 and 4.11, the structure in Figure 4.11 exhibits a constant coloration of dark blue compared to Figure 4.6 despite both structures showing deformation. This is because of the structure in Figure 4.11 has a more rigid body. Its cross-sectional steel members are also bigger than the structure in Figure 4.6. Hence, more materials are used to build the structure, which contributes to increment of the weight and cost of the structure.

Despite showing some deformations, both of the structures are still strong enough to withstand the deck loadings acting on them because fixed platform can sustain deck loading up to 21000 tonnes (refer Chapter 2, section 2.1) and the Von Mises criterions does not exceeds the yield strength of the Steel A514 of 690 MPa (refer Chapter 2, section 2.6). Therefore, these structures are not experiencing any failures or damages.

Table 4.1: Results of application of deck loading of 200 MN.

Jacket Structure cross-sectional steel members	**Stress Minimum**	**Stress Maximum**	**Stress Intensity**	**Stress Equivalent**
1 m x 1 m	0.216E-03 MPa	129.828 MPa	111.49 MPa	107.23 MPa
2 m x 2 m	0.001026 MPa	178.157 MPa	102.28 MPa	97.221 MPa

CHAPTER 5
CONCLUSION & RECOMMENDATIONS

5.1 Conclusion

Basically this project is all about the static analysis on the jacket structures, to study more on stresses that are acted on the structures. From the results obtained, few recommendations can be proposed for possible solutions.

Conclusively, the goals and objectives of this study have been clearly defined at the beginning of the project and the achievements of the goals and objectives were well monitored. The first objective would be analyzing the static analysis that runs on the two (2) jacket structure with different cross-sectional areas of steel members by means of drawing using CATIA V5 R18 and simulating using ANSYS 12.0. This objective has been successfully achieved by means of manual calculations in order to obtain useful parameters for executing the ANSYS 12.0. The results show that the smaller structure sustaining slightly more deformation than the bigger structure which can be determined by looking at contour plots coloration differences. Since the deck load of 200 MN is chosen for comparison, the SEQV and SINT values between two structures are at the difference of approximately10MPa, which is relatively small and does not make any difference for a huge structure like a jacket. Both structures are not experiencing any failures or collapse because the Von Mises stresses obtained are very relatively lower than the yield stress of Steel A514.

The second objective would be to use the results of the study to make suggestions regarding choosing a potentially better jacket structure to withstand a deck load of 200 MN in a 37.5 m depth of seawater and also lighter by weight, using lesser materials to be built and cost-effective.

5.2 Limitations

The limitation factors that may have affected the experimental results could have been the cause of the big deviation of values and large standard deviation values. First cause of such variation of values could be the lack of operational skills and knowledge in using analytical softwares. There are very few lecturers, lab technicians and post-graduate students know how to operate the ANSYS 12.0. Even their knowledge and skills in operating ANSYS 12.0 are very limited. The author had to study without any complete assistance on how to operate the ANSYS 12.0 besides by means of guidelines, tutorials and journals. The second cause would be the limited resources. The author had difficulties in finding similar journals that can be related to this project. Journals from the library and internet are not helping enough. However, journals from library are easy to gain access to compare to the internet because of online purchase prior to obtain the journals. The third cause would be the time constraint to complete this project. The total time interval of completing this project is given for approximately 8 months (2 semesters). With the lack of sources, skills, knowledge and references, it is almost impossible to successfully complete the project to perfection in this given time.

5.3 Recommendations

From the results, discussion, conclusion and limitations, there are two recommendations from the writer to help improving the course of completing any projects in the near future.

5.3.1 Recommendations on improving the accessibility of informational resources, references and published journals.

Due to insufficient informational sources, references and published journals regarding the scope of this study, the writer was obligated to be dependent on the Internet for references and information resources.

This is considerably inaccurate as uncertified information from the internet is an unreliable source. Hence, this causes the acquisition of uncertified information to be very tedious and time-consuming. For future student researchers, it is suggested that the university to prepare and supply students with accessibility to online collections of published scientific and other informational websites. This could improve the quality and increase productivity of student's research project papers.

5.3.2 Recommendations on ANSYS software implementation for further studies.

Lacking in skills and proficiency in using ANSYS 12.0 has caused some assumptions to be made to a few values and applications required in defining parameters for ANSYS calculations and therefore the results obtained might not be accurate. If this project was to be further studied, the operational and application skills for ANSYS are essential. Simplifications done in the project may be the very main cause of inaccuracy in the result. Therefore, it is suggested that the university to encourage students to learn and familiarize ANSYS, such as implementation on the usage of ANSYS into their course syllabus.

REFERENCES

ANSYS, Inc (2010). ANSYS, Engineering Simulation for the 21st Century. Adopted from the Official Website of ANSYS Inc., www.ansys.com.

ANSYS, University of Alberta. (Basic tutorials of using ANSYS) Adopted from http://www.mece.ualberta.ca/tutorials/ansys/BT/BT.html

Arizona Board of Regents, 2010. Conversion of wind speed, University of Arizona. Adopted from http://www.lpl.arizona.edu.

Avallone, Eugene A.; Baumeister, Theodore; Sadegh, Ali; Marks, Lionel Simeon (2006), *Mark's Standard Handbook for Mechanical Engineers* (11th, Illustrated ed.), McGraw-Hill Professional.

Benson, Tom (2008). Shape Effects of Drag, National Aeronautics and Space Administration (NASA). Adopted from NASA Official Website, www.grc.nasa.gov.

Bungale S. Taranath,. Wind and Earthquake Resistance Buildings, Structural Analysis and Design. CRC Press (Nov 2004) page 5-6.

Darling, David (2010). Wind Power Density, from Encyclopedia of Astrobiology, Astronomy, and Spaceflight) and the Encyclopedia of Alternative Energy and Sustainable Living, American Wind Energy Association. from. www.daviddarling.info/encyclopedia.

CATIA V5R18 Fact Sheet. Dassault Systemes.

eFunda, Inc (2010). Alloy Steel ASTM A514 Type A. Adapted from Engineering Fundamentals, the Online Destination for Engineering Communities, www.efunda.com. Copyright 2010.

Engineer's Edge (2000) Factor of Safety Review. Adapted from http://www.engineersedge.com. Copyright 2000 - 2010, by Engineers Edge, LLC All rights reserved.

International Steel Group, Inc. (2010). Tension Yield Strength and Ultimate Tensile Strength of Steel A514. Adapted from the Official site for International Steel Group www.arcelormittal.com

Jusoh, Iberahim (1997). Stress Utilization of a Jacket Structure. P.Eng MIEM, MSUT, MSSSS, Department of Applied Mechanics, Faculty of Mechanical ENgineering, Universiti Teknologi Malaysia.

Manual of Steel Construction, 8th Edition, 2nd revised printing, American Institute of Steel Construction, 1987, Chapter 1 page 1-5

Richie, Douglas G. (1977). Experimental Stress Analysis in Massive Offshore Structures. Experimental Mechanics, Volume 17, Number 9 / September, 1977.

Steel Jacket Structure, Engineering, University of Starthclyde, United Kingdom. Adapted from http://www.strath.ac.uk/esru

Stress Analysis. Adopted from Wikipedia Encyclopedia at http://en.wikipedia.org

Van Raaij *et al.* (2004). Simplified dynamic analysis relevant for jackets exposed to wave-in deck loading. Faculty of Science and Technology, Stavenger University College, Stavenger, Norway.

APPENDICES

Gantt Chart

Year 2009	Semester July 2009					
Project Activities	Jul	Aug	Sept	Oct	Nov	Dec
Literature Review	+	+	+	+	+	
Preliminary Report write-up and submission		+				
Seminar 1 Presentation			+			
Progress Report write-up and submission			+			
Methodology • Identify Methods • Drawing 1st Structure	+			+		
Researches and journals collection	+	+	+	+	+	
Interim Report write-up and submission				+	+	
FYP 1 Final Presentation					+	

Year 2010	Semester January 2010					
Project Activities	Jan	Feb	Mar	Apr	May	Jun
Progress Report 1 write-up and submission		+				
Progress Report 2 write-up and submission			+			
Seminar 2 Presentation			+			
Methodology • Drawing 2nd Structure • Meshing. • Simulation. • Calculation	+	+	+	+	+	
Results & Data collecting		+	+	+	+	
Dissertation Report write-up and submission					+	+
FYP 2 Final Presentation						+