COMPUTATIONAL ANALYSIS OF I.T. RACK UNDER VIBRATION LOAD

by

FRAMROZ M. BHARUCHA

Presented to the Faculty of the Graduate School of

The University of Texas at Arlington in Partial Fulfillment

of the Requirements

for the Degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON

December 2015

Copyright © by FRAMROZ M. BHARUCHA 2015

All Rights Reserved

Acknowledgements

I would like to take this opportunity to thank my supervising professor Dr. Dereje Agonafer for his constant encouragement, support and guidance during the course of my research and studies at this University. The invaluable advice and support provided by him was the major driving force, which enabled me to complete my thesis.

I take this opportunity to show my gratitude to Dr. Kent Lawrence and Dr. Ratan Kumar for being there as my committee members and also thank them for their valuable comments on my thesis and the work I put in.

I want to sincerely thank CommScope Inc. for giving me the opportunity to work on their project and develop that into a thesis. I am also thankful to industry mentors Mr. Tom Craft and Mr. Al Skrepcinski for their overall help to finish my thesis.

I like to thank all EMNSPC lab members and mechanical department who supported me throughout my graduate studies and helped me get through it all. Last but not the least, my parents, family and friends without whose support none of this would have been possible.

November 12, 2015

Abstract

COMPUTATIONAL ANALYSIS OF I.T. RACK

UNDER VIBRATION LOAD

FRAMROZ M. BHARUCHA, MS

The University of Texas at Arlington, 2015

Supervising Professor: Dereje Agonafer

Structural failure of a rack due to vibration could result in injury to people, damage to IT equipment, or interruption of services that depend on proper functioning of the IT and networking equipment in the rack. In this topic, a computational study of an IT rack and a group of racks placed adjacent to each other under three vibration load scenarios: transportation, office and earthquake vibration is presented.

Each rack can weigh as much as 1600 kg (3500 lb.) when fully populated with IT equipment. Two standards commonly used for testing racks with synthetic seismic loads are the GR-63-CORE Network Equipment Building Systems (NEBSTM) and the International Building Code (IBC). In this paper, the earthquake, office and transportation vibration loads as given in GR-63-CORE are applied on a computer-aided design (CAD) model of the rack mentioned above. The material used is ASTM A36 / A572 series steel. The IT equipment was made in CATIA V5 / Solidworks and then imported into ANSYS Workbench where it was meshed. After meshing, a pre stressed Modal Analysis was run to find the natural frequencies of the body. From the output result of Modal analysis, all modes of vibration were included as input for Response Spectrum analysis (RS-analysis),

as those modes will be the dominant modes for vibration. Harmonic Response is used to analyze office vibrations, as the swept sine wave (Harmonic Response) resembles office environment vibrations. Random vibration analysis is used for transportation vibration. All the acceleration curves and standard for testing are in correlation with GR-63-CORE NEBSTM standard. Other boundary conditions included is that the M8 screw used to bolt the bottom of the IT equipment.

Table of Contents

Acknowledgements iii

Abstract iv

List of Illustrations viii

List of Tables xi

Chapter 1 INTRODUCTION 1

1.1 Earthquake Environment 2

1.2 Transportation Environment 6

1.3 Office Environment 7

Chapter 2 FINITE ELEMENT METHOD 8

2.1 Introduction to Finite Element Method 8

2.2 FEA Problem Solving Steps 9

2.3 Geometry and Material Definition 10

2.4 Meshing Model 11

Chapter 3 FINITE ELEMENT ANALYSIS 13

3.1. Importing CAD model 13

3.2. Checking the model 13

3.3. Contact analysis 14

3.4 Meshing the Model 15

3.5. Material properties 16

3.6 Rack Loading 18

3.7 Finite Element Analysis Results 18

3.7.1. Stress criterion 19

3.7.2 Deformation criterion 20
Chapter 4 CASE STUDY 21
4.1 Case study one 21
4.2 Case study two 31
4.3 Case study three 37
4.4 Case study four 48
Chapter 5 SUMMARY OF RESULTS 53
Chapter 6 CONCLUSION AND FUTURE WORK 54
References 55
Biographical Information 57

List of Illustrations

Figure 1-1 Map of earthquake risk zones 3
Figure 1-2 Earthquake synthesized waveform VERTEQII – Zone 4 4
Figure 1-3 Required Response Spectrum 5
Figure 1-4 Transport Vibration Environment 6
Figure 3-1 Schematic diagram of complete analysis 13
Figure 3-2 Geometry 14
Figure 3-3 Initial contact information 15
Figure 3-4 Tets, bricks, prism and pyramid mesh elements 16
Figure 3-5 Material library in ANSYS workbench 16.1 17
Figure 3-6 Von-Mises and maximum shear stress criterion 19
Figure 4-1 Geometry 21
Figure 4-2 Ansys project tree layout 22
Figure 4-3 Static structural boundary conditions 23
Figure 4-4 Total deformation for static structural 24
Figure 4-5 Equivalent stress for static structural 24
Figure 4-6 Magnified view equivalent stresses for static structural 25
Figure 4-7 Factor of safety 25
Figure 4-8 Total deformation for RS-analysis 26
Figure 4-9 Equivalent stress for RS-analysis 27
Figure 4-10 Magnified view for equivalent stresses RA-analysis 27
Figure 4-11 Deformation for harmonic response 28
Figure 4-12 Equivalent stresses harmonic response 28
Figure 4-13 Total deformation random vibration 29

Figure 4-14 Equivalent stresses random vibration 30
Figure 4-15 Magnified view of equivalent stresses random vibration 30
Figure 4-16 Geometry 31
Figure 4-17 Grouped rack boundary conditions 32
Figure 4-18 Grouped rack deformation 32
Figure 4-19 Grouped rack equivalent stress for static loading 33
Figure 4-20 Grouped rack equivalent stress for static loading 33
Figure 4-21 Grouped old rack deformation (RS analysis) 34
Figure 4-22 Grouped old rack stress (RS analysis) 35
Figure 4-23 Grouped old rack stress (RS analysis) magnified 35
Figure 4-24 Grouped old rack deformation for harmonic response 36
Figure 4-25 Grouped old rack stress for harmonic response 36
Figure 4-26 Geometry 37
Figure 4-27 Boundary conditions 38
Figure 4-28 Front view of boundary conditions 39
Figure 4-29 Total deformation for static loading 40
Figure 4-30 Equivalent stress for static loading 40
Figure 4-31 Magnified view for equivalent stresses for static loading 41
Figure 4-32 Factor of safety for static loading 41
Figure 4-33 Magnified F.O.S 42
Figure 4-34 Total deformation RS-analysis 43
Figure 4-35 Equivalent stress for RS-analysis 44
Figure 4-36 Magnified view equivalent stress for RS-analysis 44
Figure 4-37 Total deformation for harmonic response 45
Figure 4-38 Equivalent stress for harmonic response 45

Figure 4-39 Deformation for random vibration 46
Figure 4-40 Equivalent stress for random vibration 47
Figure 4-41 Magnified view equivalent stress for random vibration 47
Figure 4-42 Boundary conditions 48
Figure 4-43 Deformation for static structural 49
Figure 4-44 Equivalent stresses for static structural 49
Figure 4-45 Magnified view of equivalent stresses for static structural 50
Figure 4-46 Deformation for RS-analysis 50
Figure 4-47 Equivalent stresses for RS-analysis 51
Figure 4-48 Magnified view of equivalent stresses for static structural 51
Figure 4-49 Deformation for harmonic response 52
Figure 4-50 Equivalent stresses for harmonic response 52

List of Tables

Table 1-1 Earthquakes in the United States 3

Table 1-2 Response Spectrum 5

Table 3-1 Material properties 17

Table 5-1 Result table 53

Chapter 1

INTRODUCTION

Data centers are physical infrastructures used for storage of computer systems, servers, telecommunication systems and other components used for the company's information technology needs. For making sure the proper functioning of all the equipment's in the data center, there is proper environmental control is required. Data centers often require backup power supply systems and effective cooling systems such as air conditioning, economizer cooling, since all these IT equipment stored in the data center produces large amount of heat within the data centers. As the whole data center structure is expensive which should be maintained and protected from all those factors that may cause its malfunctioning.

Designing a data center is one of the major tasks for which many aspects are required to be considered. One of the major aspects should be considered are the environmental effects acting externally on data center structures. Earthquakes, wind effects, rain effects, flooding are few of those environmental impacts. Earthquakes have the greatest impact on data center structures.

From a company point of view it is very important for them that this data is available to use at any given point of time. Any down time would mean a huge loss to the company. Thus testing the structural integrity of a component before field implementation is very important.

It is common practice to conform the functionality and integrity of structures experiencing strong ground motion before their field installation. Hence, earthquake, office and transport vibrations analysis becomes necessary.

The purpose of this project is to see the structural integrity of the rack while looking at the benefits of grouping versus single frame response for the available racks. As well as, to check the maximum allowable loading per rack.

The purpose as stated above is to check the rack component placed in the data centers for failures due to vibrations. Earthquake is not the only form of vibration the racks may be affected by. The rack when transported for installation to a new facility will undergo a series of random vibrations which may cause it to generate high stresses and can cause failure. Similarly while operating in a facility the acoustic vibration caused in an office environment can cause failures if the natural frequency of structure is close to the load applied to it.

The different vibrational loads that the I.T. rack is subjected to are as follows;

- Earthquake vibrations (Response-Spectrum analysis)
- Office vibrations (Harmonic Response analysis)
- Transport vibration (Random vibration analysis)

1.1 Earthquake Environment

During an earthquake, telecommunications equipment is subjected to motions that can over-stress equipment framework, circuit boards, and connectors. The amount of motion and resulting stress depends on the structural characteristics of the building and framework in which the equipment is contained, and the severity of the earthquake. Figure 2-2 shows the map of earthquake risk zones. Zone 4 corresponds to the highest risk areas, Zone 3 the next highest, and so on. Geographic areas designated as Zone 0 present no substantial earthquake risk. Equipment to be used in earthquake risk Zones 1 through 4 shall be tested to determine the equipment's ability to withstand earthquakes. No earthquake requirements are provided for Zone 0.

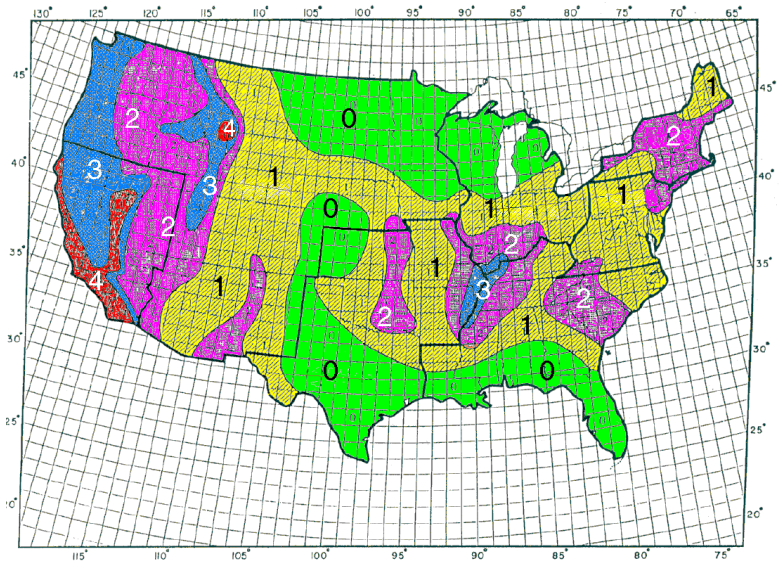


Figure 1-1 Map of earthquake risk zones

A list of largest earthquakes in terms of magnitude in the past 50 years that have occurred in the United States of America are listed below.

Table 1-1 Earthquakes in the United States

Date	State(s)	Magnitude	Further information
27-Mar-64	Alaska	9.2	1964 Alaska earthquake and tsunami
4-Feb-65	Alaska	8.7	1965 Rat Islands earthquake and tsunami
9-Mar-57	Alaska	8.6	1957 Andreanof Islands earthquake and tsunami
9-Jul-58	Alaska	8.3	1958 Lituya Bay earthquakes and megatsunami
18-Apr-06	California	7.9	1906 San Francisco earthquake
3-Nov-02	Alaska	7.9	2002 Denali earthquake
23-Jun-14	Alaska	7.9	2014 Aleutian Islands earthquake
5-Jan-13	Alaska	7.5	2013 Craig earthquake
17-Aug-59	Montana, Wyoming	7.3 – 7.5	1959 Hebgen Lake earthquake

Table 1-1 continued

28-Oct-83	Idaho	7.3	1983 Borah Peak earthquake
28-Jun-92	California	7.3	1992 Landers earthquake

As discussed earlier there are two different ways to measure the response of a body due to earthquake vibrations; Transient and Response-Spectrum analysis.

Transient analyses are more costly in terms of solution time. Also, it is necessary to artificially create the time-acceleration data in such a way that these data are compatible with the smoothed response spectrum. This will give out results on the basis of dynamic equations of equilibrium and hence both compressive (negative) and tensile (positive) stresses will be generated for the whole period of the earthquake.

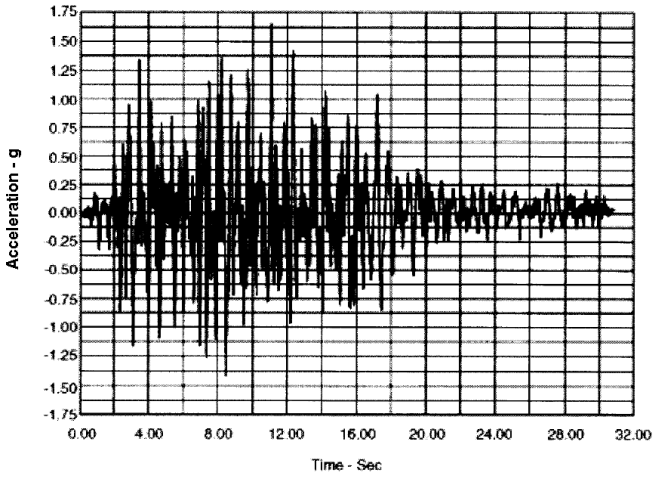


Figure 1-2 Earthquake synthesized waveform VERTEQII – Zone 4

The Response- Spectrum analysis uses the modal result obtained as an input for calculation of earthquake response. To include the response specter data containing the relation between structural acceleration and the structural frequencies we insert the tool

“RS-Acceleration” and include earthquake data as a table (Frequencies [Hz]- Acceleration [g])

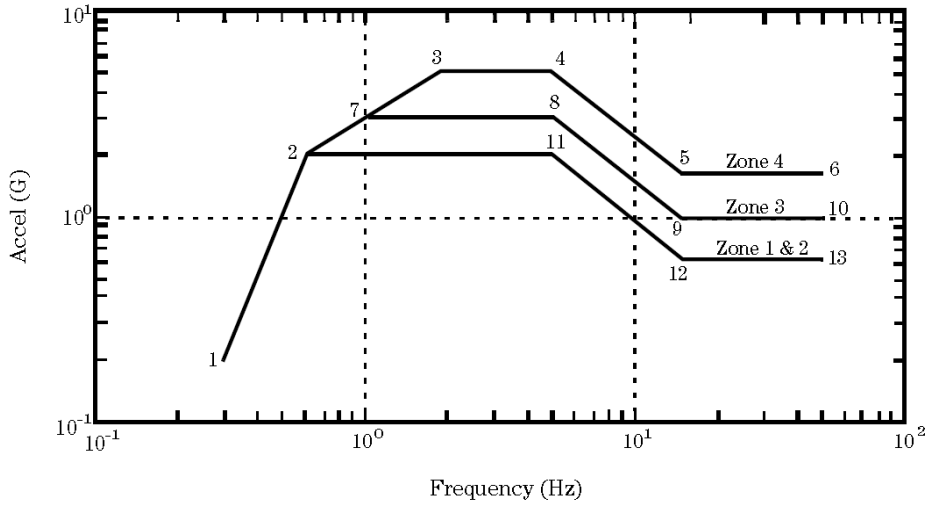


Figure 1-3 Required Response Spectrum

Table 1-2 Response Spectrum

Coordinate point	Frequency (Hz)	Value for Upper Floor Acceleration (g)	Coordinate point	Frequency (Hz)	Value for Upper Floor Acceleration (g)
Zone 1 and 2			Zone 4		
1	0.3	0.2	1	0.3	0.2
2	0.6	2.0	2	0.6	2.0
11	5.0	2.0	3	2.0	5.0
12	15.0	0.6	4	5.0	5.0
13	50.0	0.6	5	15.0	1.6
Zone 3			6	50.0	1.6
1	0.3	0.2			
2	0.6	2.0			
7	1.0	3.0			
8	5.0	3.0			
9	15.0	1.0			
10	50	1.0			

1.2 Transportation Environment

One of the main goals or uses of random vibration testing in industry is to bring a DUT to failure. For example, a company might desire to find out how a particular product may fail because of various environmental vibrations it may encounter. The company will simulate these vibrations on a shaker and operate their product under those conditions. Testing the product to failure will teach the company many important things about its product's weaknesses and ways to improve it. Random testing is the key testing method for this kind of application.

Equipment will generally experience maximum vibration in the non-operating, packaged condition, during commercial transportation. The transit environment is complex. There are low-level vibrations of randomly distributed frequencies reaching 1 to 500 Hz with occasional transient peaks.

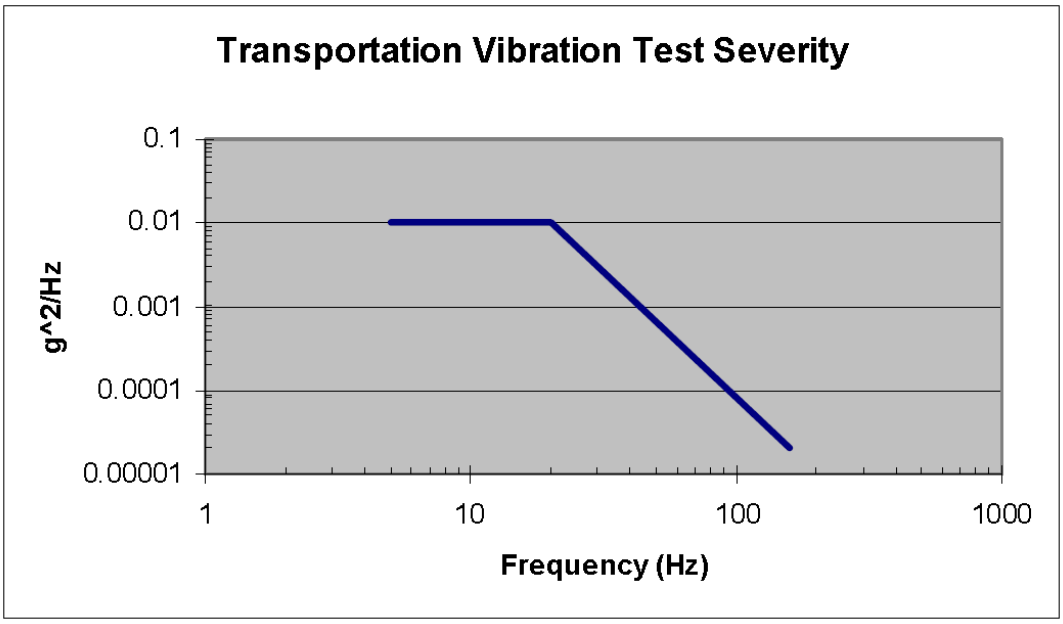


Figure 1-4 Transport Vibration Environment

During transportation the frame is assembled and bolted to the base. It is transported empty as to reduce the stresses generated in body. While transporting the frame, tie straps are also used to hold the frame in place. This practice of transportation allows us to scale down the results obtained by simulation by to up to 10 times.

1.3 Office Environment

The office vibration test is often performed in conjunction with the synthesized waveform test, since the test configuration requirements are the same. In the test, the effects of office vibration are simulated by a swept sine survey.

A sweep-sine acceleration of 0.1g is applied for a range of frequencies (5 Hz to 100 Hz) which represents a working condition in an office environment.

Chapter 2

FINITE ELEMENT METHOD

2.1 Introduction to Finite Element Method

The Finite Element Method is a computational technique used to obtain approximate solution to boundary value problem in engineering. FEM is virtually used in almost every industry that can be imagined. Common application of FEA applications are mentioned here.

- Aerospace/Mechanical/Civil/Automobile Engineering
- Structural Analysis (Static/Dynamic/Linear/Non-Linear)
- Thermal/Fluid Flow
- Nuclear Engineering
- Electromagnetic
- Biomechanics
- Biomedical Engineering
- Hydraulics
- Smart Structures

"The Finite Element Method is one of the most powerful numerical techniques ever devised for solving differential equations of initial and boundary value problems in geometrically complicated regions". Sometimes it is hard to find analytical solution of important problems as they come with complicated geometry, loading condition, and material properties. So FEA is the computational technique which helps in reaching the satisfactory results with all the complex conditions that can't be solved through analytical procedure. There are wide range of sophisticated commercial code available which helps in reaching the approximately close solution in 1D, 2D and 3D. In this FEA method, the

whole continuum is divided into a finite numbers of small elements of geometrically simple shape. These elements are made up of numbers of nodes. Displacement of these nodes is unknown and to find it, polynomial interpolation function is used. External force is replaced by an equivalent system of forces applied at each node. By assembling the mentioned governing equation, results for the entire structure can be obtained.

$$\{F\} = [K]\{u\}$$

Where, {F} = Nodal load/force vector

[K] = Global stiffness matrix

{u} = Nodal displacement

Structure's stiffness (K) depends on its geometry and material properties. Load (F) value has to be provided by user. The only unknown is displacement (u). The way in general FEA works is, it creates the number of small elements with each containing few nodes. There are equations known as Shape function in software, which tells software how to vary displacement (u) across the element and average value of displacement is determined at nodes. Those stress and/or displacement values are accessible at nodes which explains that finer the mesh elements, more accurate the nodal values would be. So there are certain steps that we need to follow during the modeling and simulation in any commercial code to reach approximately true solution, which would be explained hereby. In this study commercially available FEA tool, ANSYS Workbench v15.0 has been leveraged.

2.2 FEA Problem Solving Steps

These five steps need to be carefully followed to reach satisfactory solution to FEA problem:

1) Geometry and Material definition

2) Defining Connection between bodies
3) Meshing the model
4) Defining load and boundary condition
5) Understanding and verifying the results

ANSYS is a general purpose FEA tool which is commercially available and can be used for wide range of engineering application. Before we start using ANSYS for FEA modeling and simulation, there are certain set of questions which need to be answered based on observation and engineering judgment. Questions may be like what is the objective of analysis? How to model entire physical system? How much details should be incorporated in system? How refine mesh should be in entire system or part of the whole system? To answer such questions computational expense must be compared to the level of accuracy of the results that needed. After that ANSYS can be employed to work in an efficient way after considering the following:

- Type of problem
- Time dependence
- Nonlinearity
- Modeling simplification

From observation and engineering judgment, analysis type has to be decided. In this study the analysis type is structural; to be specific out of different other structural problem focus in this study is on Static and Dynamic analysis. Non-linear material and geometrical properties such as plasticity, contact, and tensile strength are available.

2.3 Geometry and Material Definition

Geometrical nonlinearity needs to be considered before analysis. This nonlinearity is mainly of two kinds.

1) Large deflection and rotation: If total deformation of the structure is large compared to the smallest dimension of structure or rotate to such an extent that dimensions, position, loading direction, change significantly, then large deflection and rotation analysis becomes necessary. Fishing rod explains the large deflection and rotation.
2) Stress Stiffening: Stress stiffening occurs when stress in one direction affects the stiffness in other direction. Cables, membranes and other spinning structures exhibit stress stiffening.

Material nonlinearity is also the critical factor of FE analysis, which reflects in the accuracy of the solution. If material exhibit linear stress-strain curve up to proportional limit or loading in a manner is such that it doesn't create stress higher than yield values anywhere in body then linear material is a good approximation. If the material deformation is not within the loading condition range is not linear or it is time/temperature dependent then nonlinear properties need to be assigned to particular parts in system. In that case plasticity, creep, viscoelasticity need to be considered. Apart from that, if a structure exhibit symmetry in geometry, then it needs to be considered when creating model of physical structure which is advantageous in saving the computational time and expense. Once the geometry and material properties are taken care of contacts between different bodies needs to be considered such as rigid, friction, bonded etc.

2.4 Meshing Model

As discussed in section 4.1, large number of mesh counts (elements) provides better approximation of solution. There are chances in some case that excessive number of elements increases the round off error. It is important that mesh is fine or coarse in appropriate region and answer to that question is completely dependent on the physical

system being considered. In some cases mesh sensitivity analysis is also considered to balance computational time with accuracy in solution. Analysis is first performed with certain number of elements and then with twice the elements. Then both the solutions are compared, if solutions are close enough then initial mesh configuration is considered to be adequate. If solutions are different than each other than more mesh refinement and subsequent comparison is done until the convergence is achieved. There are different types of mesh elements for 2D and 3D analysis in ANSYS which can be used based on application

Chapter 3

FINITE ELEMENT ANALYSIS

3.1. Importing CAD model

Finite element method is used to analyze the rack structure modelled in either Solidworks or CATIA V5 by using ANSYS 16.1 simulation software. The simulation process is carried out in ANSYS workbench which acts as a graphical unit interface. For this model, we carry out static structural, modal, response-spectrum, random vibration and harmonic response analysis and the work flow of the analysis used is shown in the picture below.

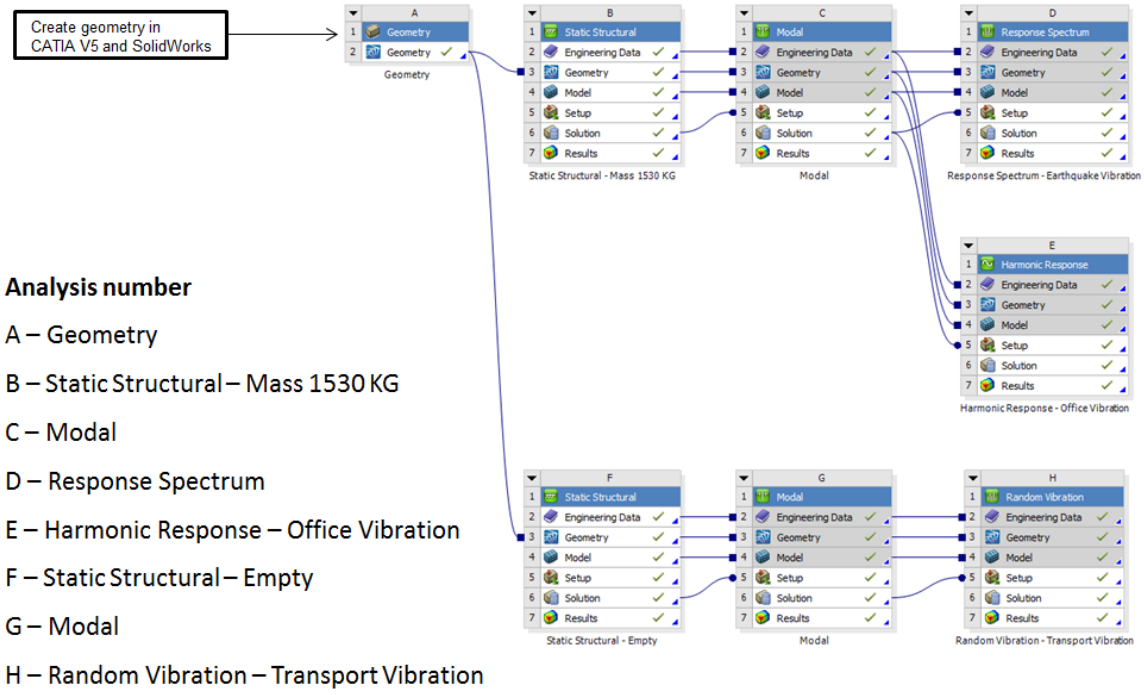


Figure 3-1 Schematic diagram of complete analysis

3.2. Checking the model

The CAD model with assembly parts designed is converted and saved as a STEP file. The STEP file is then imported into ANSYS design modeler to start structural analysis.

It is always recommended to turn on the surface bodies, line bodies option in design modeler to import all parts of the assemble model without losing anything. Then generate the model and the whole assembly is created within design modeler. For this assembly we have 12 parts. If necessary the different parts can be color coded differently to differentiate them and also naming them is an option that can be helpful while analyzing in the mechanical solver. The imported model in ANSYS 16.1 can be seen in the below image.

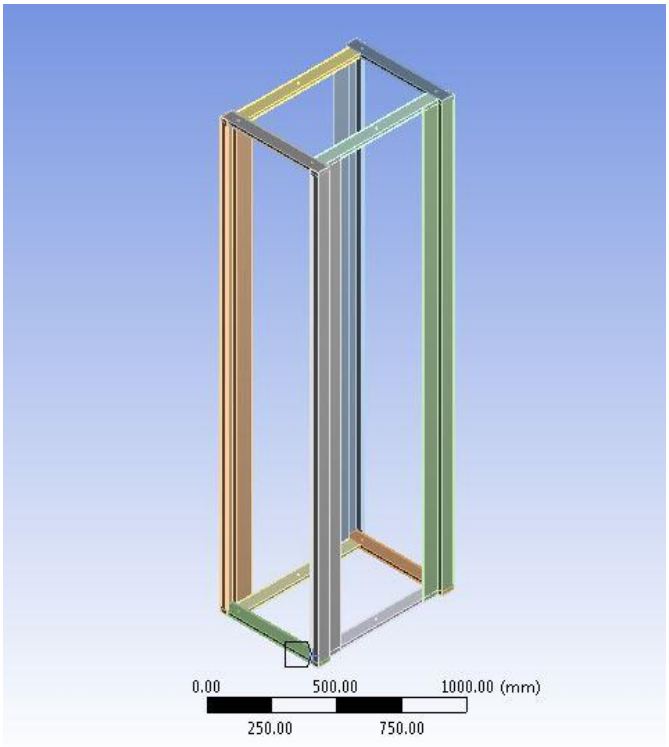
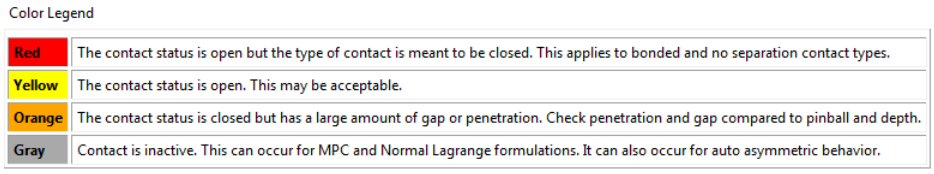


Figure 3-2 Geometry

3.3. Contact analysis

After importing and generating the CAD model in ANSYS, it is moved to the mechanical solver where the model is pre-processed to get structural results. When the model is sent to the solver all the contacts between parts are all generated automatically

based on assembling of parts in CREO parametric. Before going onto analyze it is always good to check the contacts if there are any unnecessary contacts acting between different parts. For this manual check, ANSYS provides us an option named contact information where the program will run throughout all the contacts and produces a data sheet that has the penetration, gap, type of contact, status and pin-ball radius for each contact. Based on that it is easy to pre-check all the contacts and delete the unwanted contacts. Color codes represented to inform the user that the contacts could be far open or too close or proper. In our model all contact types are bonded and hence a linear type problem. The initial contact information generated helped to delete excessive contacts created and also helped increase pin ball radius for contacts those were open but had to be bonded. By increasing pin ball radius based on the gap information it is easy to make the bonded contact more active than being far open. Finally, checking all the contacts is absolutely necessary so that the model would not have more gaps between parts which will create peak stresses and mislead the analysis.


Color Legend

Red	The contact status is open but the type of contact is meant to be closed. This applies to bonded and no separation contact types.
Yellow	The contact status is open. This may be acceptable.
Orange	The contact status is closed but has a large amount of gap or penetration. Check penetration and gap compared to pinball and depth.
Gray	Contact is inactive. This can occur for MPC and Normal Lagrange formulations. It can also occur for auto asymmetric behavior.

Figure 3-3 Initial contact information

3.4 Meshing the Model

Once the model is checked for contacts the next important step to carry out is to mesh the model using different options. There are plenty of meshing methods available in ANSYS 16.1 Each model has to be meshed based on its geometry and structure. There are different types of elements while generating mesh for a body. Tetrahedral, hexahedral (brick) and prism elements are the different types of mesh elements. Brick elements are

the best element to be used since it produces an even mesh over a body part and also reduced the count of elements produced. Some of the element pictures are shown below.

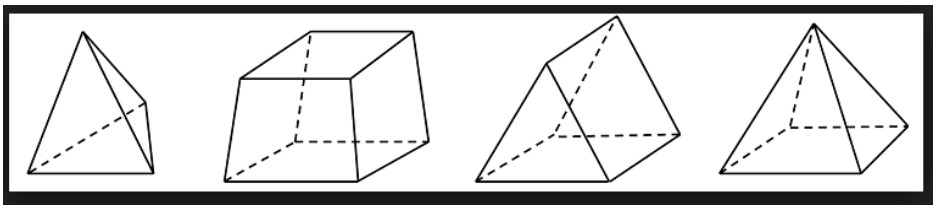


Figure 3-4 Tets, bricks, prism and pyramid mesh elements

While generating mesh for model, there are two strategies to be followed. Using global meshing methods at first and then based on the model requirements local meshing methods can be used.

The geometry being tested has the scope of using mid surfaces planes. This procedure allows us to use a quad mesh for the entire structure. Sizing of element size can be specified for having finer mesh. Locations of connections can be given edge sizing to have mode divisions in particular areas as required. Thus following all the above stated meshing techniques, we finally generated the model with half a million element and nodes count approximately. Once meshing is done, all the contact information should be regenerated because contacts depend on nodes too.

3.5. Material properties

All the required materials are included in the ANSYS workbench library through engineering data. All the required properties like densities, isotropic coefficients, yield strength and ultimate strength can be plugged in from the property chart shown on the left side of the figure below. These materials can then be called into the solver while analyzing easily.

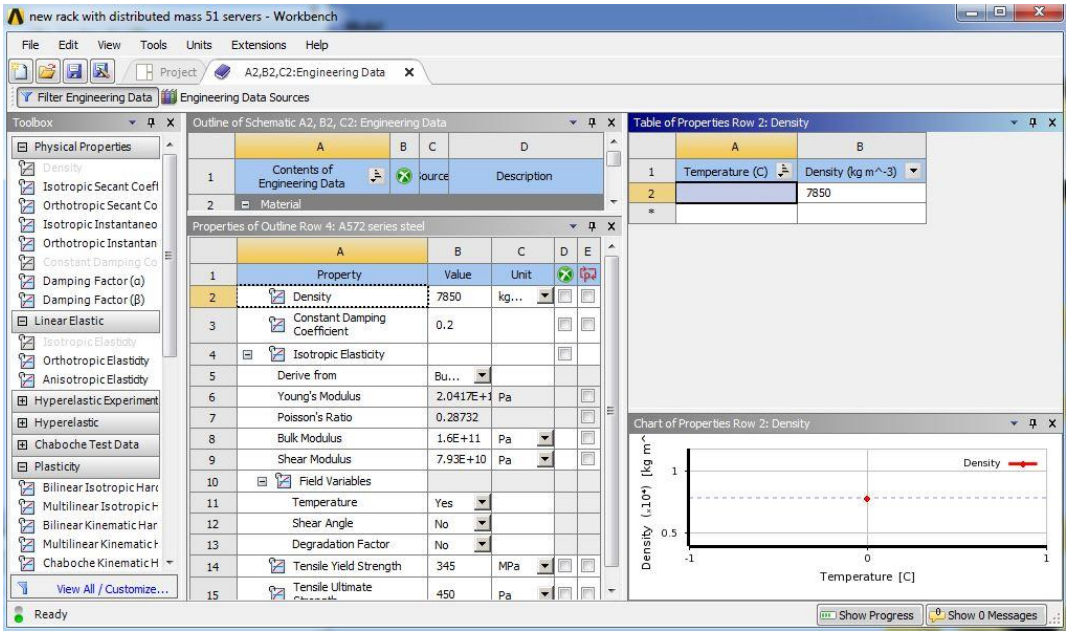


Figure 3-5 Material library in ANSYS workbench 16.1

Material selected should always satisfy ASME standards. The material properties of all the material used and assumed are provided in the tabular data below.

Table 3-1 Material properties

	ASTM-A36 steel	ASTM-A572 series steel					
Grade		**42**	**45**	**50**	**55**	**60**	**65**
Density	7850 kg/m3	7850 kg/m3	7850 kg/m3	7850 kg/m3	7850 kg/m3	7850 kg/m3	7850 kg/m3
Young's modulus of elasticity	200 Gpa	205 Gpa	205 Gpa	205 Gpa	205 Gpa	205 Gpa	205 Gpa

Table 3-1 continued

Poisons Ratio	0.26	0.285	0.285	0.285	0.285	0.285	0.285
Ultimate Tensile Strength	400-550 Mpa	415 Mpa	415 Mpa	450 Mpa	485 Mpa	520 Mpa	550 Mpa
Yield Tensile Strength	250 Mpa	290 Mpa	310 Mpa	345 Mpa	380 Mpa	415 Mpa	450 Mpa

3.6 Rack Loading

The geometry being tested has a fixed base where it is attached to the ground. There are 3 different tests which will be performed on the rack and have been discussed in chapter 2. The loading criteria for all tests are described below

- For RS-analysis the rack is fixed at the bottom and is completely loaded (1530 KG mass). Standard earth gravity acts on the geometry.
- For harmonic response analysis the rack is fixed at the bottom and is completely loaded (1530 KG mass). Standard earth gravity acts on the geometry.
- For random vibration analysis the rack is fixed at the bottom and is completely empty. Standard earth gravity acts on the geometry.

3.7 Finite Element Analysis Results

Finite element analysis is carried out for the mount model in ANSYS workbench 16.1 with all the boundary conditions we got from the Telcordia standards. The stress, deformation and strain results are analyzed and elaborately discussed in this chapter.

3.7.1. Stress criterion

After defining the material properties, contacts between parts and meshing the geometry, we define the boundary conditions before starting to run the simulation. For this model stress results were analyzed and since the materials used are all steel structures which are ductile and also the structure having multiaxial loadings, it is safe to analyze the equivalent von-mises stress instead of normal and shear stresses individually. Von-mises stress is nothing but a logical way to sum of all the directional stresses. Von-mises stress results can be further compared to the yield strength to verify whether the entire model satisfies von-mises stress criterion.

$$\sigma v = \frac{[(\sigma 1 - \sigma 2)\text{^}2 + (\sigma 2 - \sigma 3)\text{^}2 + (\sigma 3 - \sigma 1)\text{^}2]}{2}\text{^}1/2 \geq \sigma y$$

Where, σ1, σ2 and σ3 are principle stresses of the model on all three directions, σv represents the von-mises stress and σy represents the yield strength. Von-mises criterion can be stated as the model will fail when the von-mises stress exceeds the yield strength.

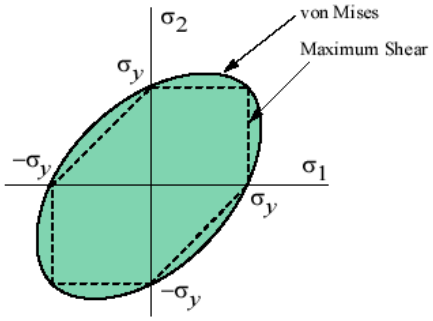


Figure 3-6 Von-Mises and maximum shear stress criterion

3.7.2 Deformation criterion

After defining the problem and computing the results we have two primary areas of interest. Stress as we discussed is one of the main criterions and the second is deformation. In our solution the criteria is well defined in the Telcordia guide. The total deformation of the body is not to exceed a maximum of 3 inches (75 mm) deflection, as it has a possibility to topple over.

Chapter 4

CASE STUDY

4.1 Case study one

Here in case one we see the single frame as shown in the figure below. This is one of the types of frame that we will be testing. The geometry is a simple structure with four base structures that will be bolted to the floor. Four vertical members are used where the servers will be attached and four top supporting members to complete the geometry at the top.

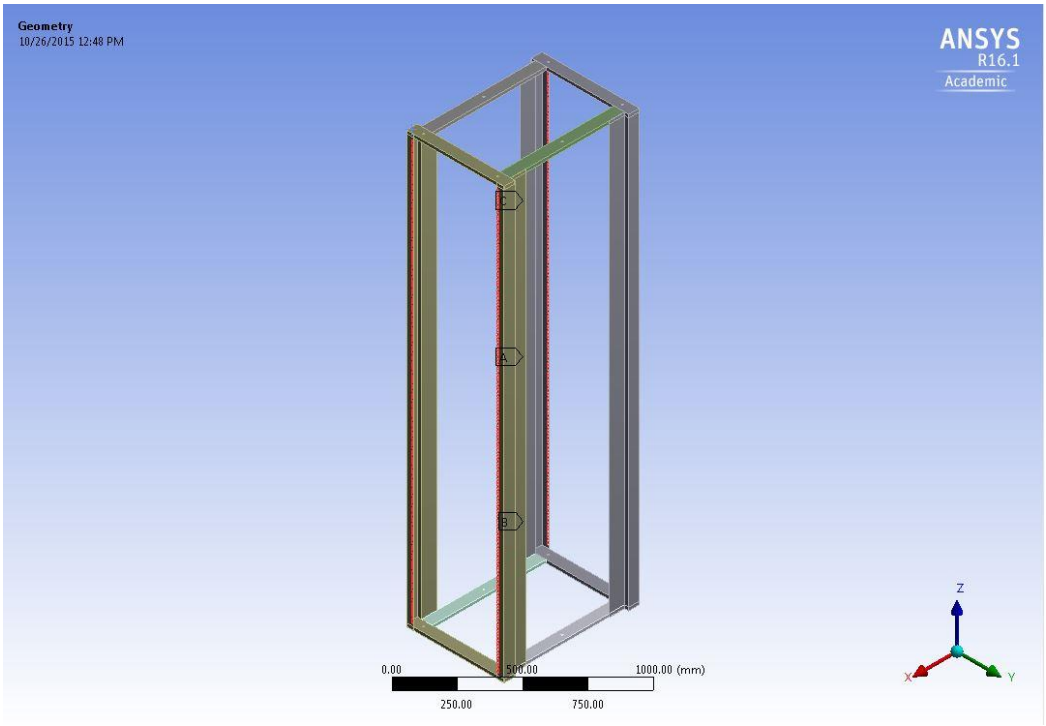


Figure 4-1 Geometry

The best practices to perform analysis suggest that we import the geometry as surfaces instead of solid bodies and define a thickness to the material after it has been

imported for analysis. This procedure allows us to use shell mesh instead of solid mesh which in turn helps us reduce number of nodes in a structure and reduces computational time.

In place of actual servers we will use point mass load of equivalent weight in efforts to make this into a compact model. As each server weighs approximately 30 KG (66 Lbs.) and one rack can hold up to 51 such servers, we apply a total point mass of 1530 KG (3373 Lbs.). This point mass is applied to the exact same location as each server would rest on the vertical member. The figure below shows us how the surface bodies and point masses represented in the body.

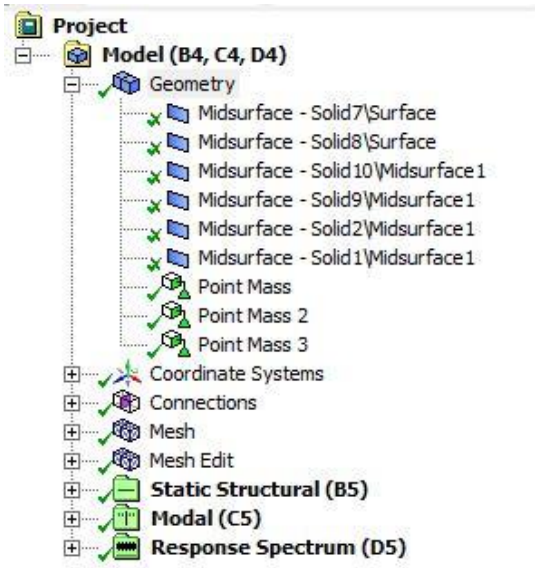


Figure 4-2 Ansys project tree layout

After the geometry is imported and thickness is defined we proceed to forming connections in parts where Ansys has failed to generate the contacts. Once all connections are defined correctly we proceed to meshing. Ansys auto generated mesh for this surface type of modeling is quad mesh. We can define the mesh size for a good quality mesh. After meshing is complete we have to check the model for connections that have not meshed

correctly. We use the edit mesh tool to connect mesh of different parts and connect nodes at for intersecting bodies.

After setting up the mesh we set up for a static loading test. Here we apply standard earth gravity and fixed supports to the geometry. Thus subjecting it to the working environment. The figure below shows geometry subjected to all loads for static structural test.

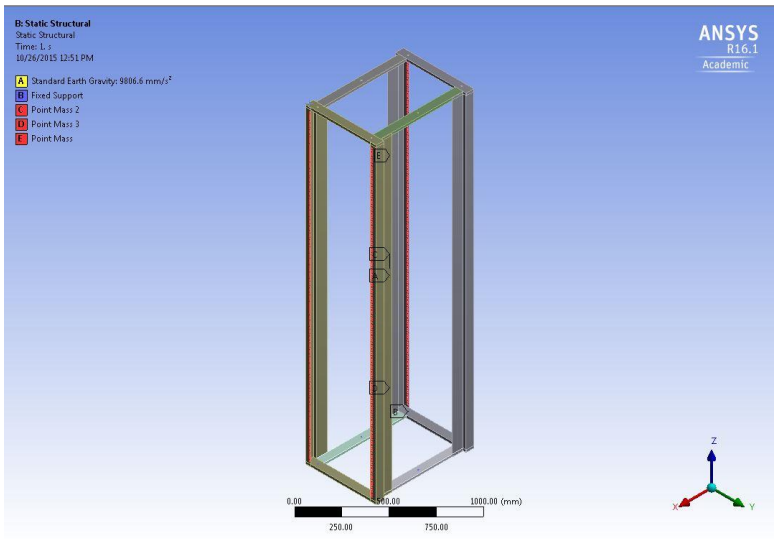


Figure 4-3 Static structural boundary conditions

After applying all the loads we look for deformation and equivalent stress in the body as they are the limiting factors for our design. The figure below shows us maximum deformation and maximum equivalent stress during static loading for this structure. The maximum Deformation occurs near the bottom of the rack and has a maximum value of 0.196 mm. Also maximum equivalent stress occurs at the point of the fixed support and is 372 Mpa. The edge on which this stress is generating is fixed in all degree of freedom and hence shows higher stress than what is occurring. It is noticeable that the stress in the body will not exceed 150 Mpa.

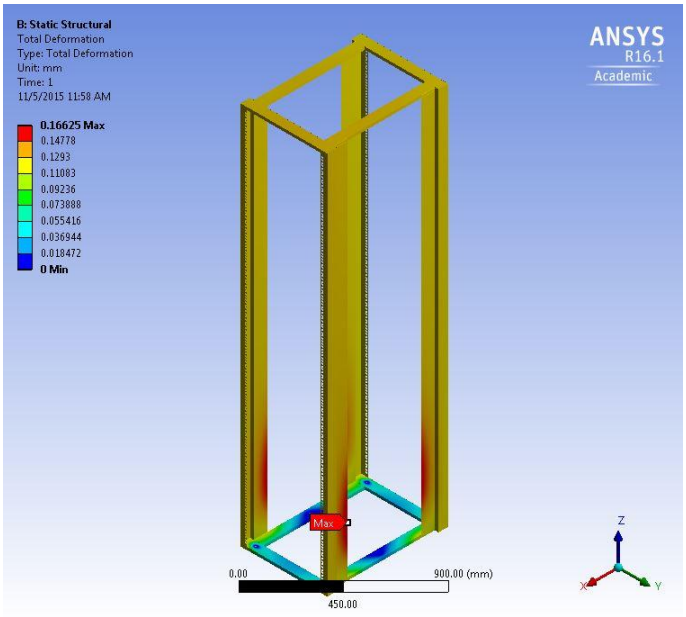


Figure 4-4 Total deformation for static structural

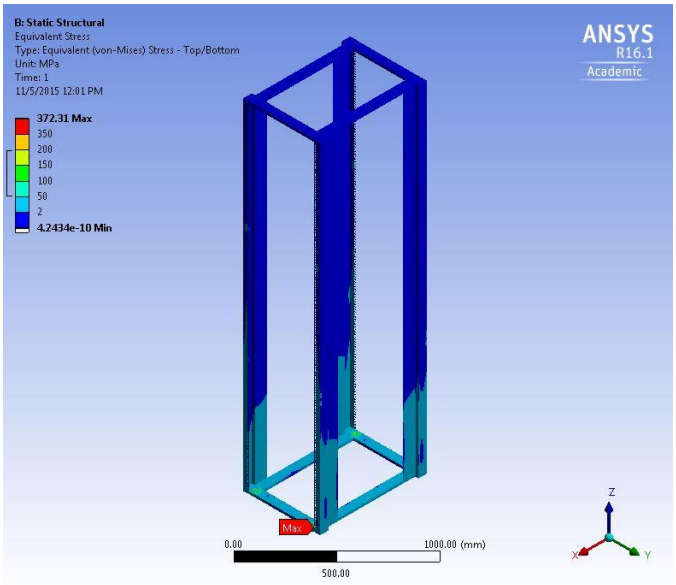


Figure 4-5 Equivalent stress for static structural

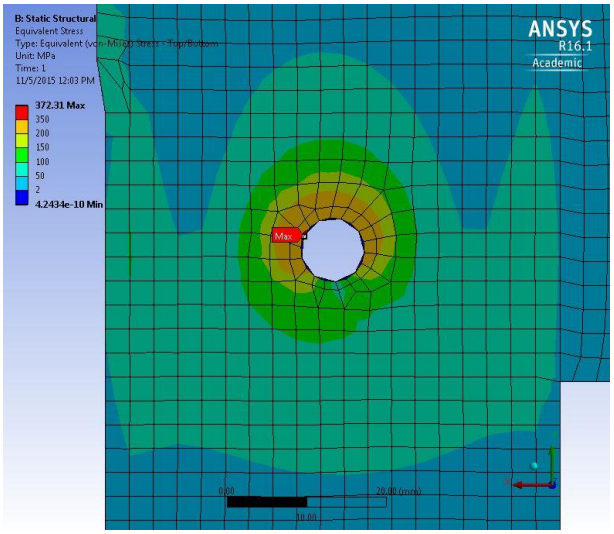


Figure 4-6 Magnified view equivalent stresses for static structural

The factor of safety for the structure is shown in the figure below and it is seen that the design is robust for such kinds of working loads.

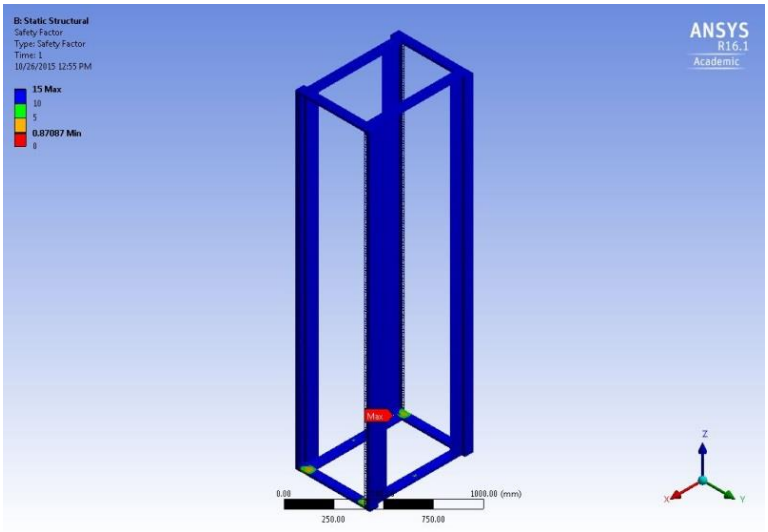


Figure 4-7 Factor of safety

Once the static structural test is completed we couple the results to a modal analysis. The modal analysis gives us the natural frequencies of the body i.e. how the body

would respond when subjected to certain types of frequencies. The reason to couple the static loading test and modal analysis together is so that we have a pre-stressed body while testing.

Modal analysis is a pre-requisite for performing response spectrum analysis. The response spectrum analysis (RS-analysis) is used to subject the body to an earthquake type of acceleration load test. During a RS-analysis the body is subjected to ground motion which is similar to an earthquake.

The figures below show us the Maximum deformation and maximum equivalent stress generated in the body while running this test. The results indicate that maximum deformation is approximately 21.44 mm.

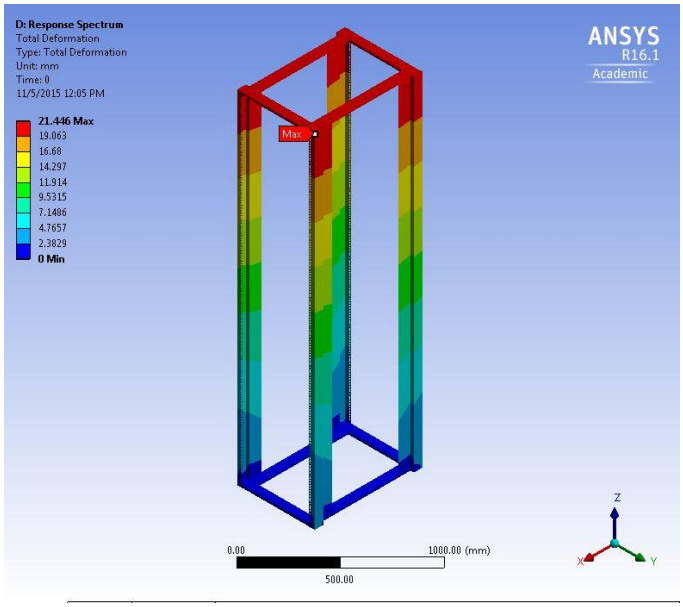


Figure 4-8 Total deformation for RS-analysis

The maximum equivalent stress is 6525 Mpa. The magnified image of the location where these stresses are being generated shows that the stress is caused due to a fully constrained edge. As the edge is not allowed to deform (elastic or plastic deformation) the

stress keeps on rising. By not considering this region it can be seen that stresses in body do not exceed 350 Mpa.

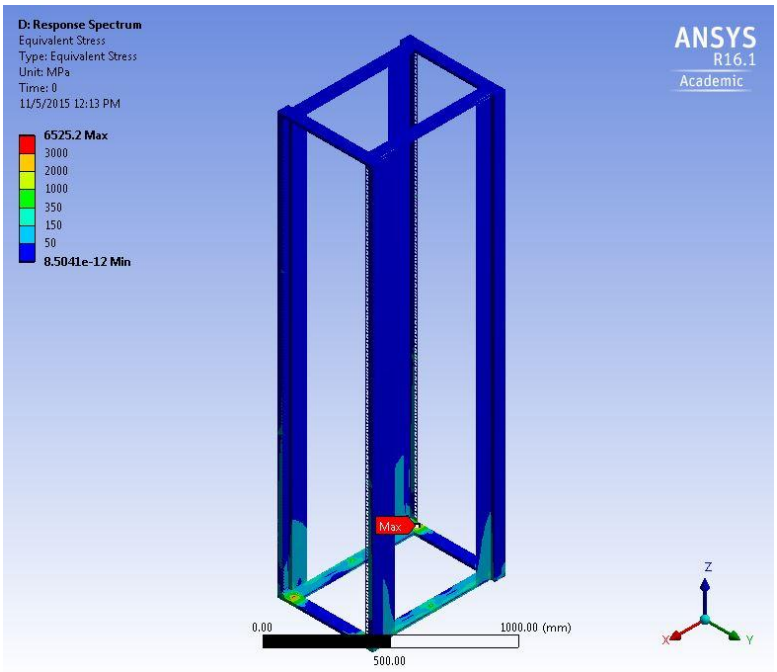


Figure 4-9 Equivalent stress for RS-analysis

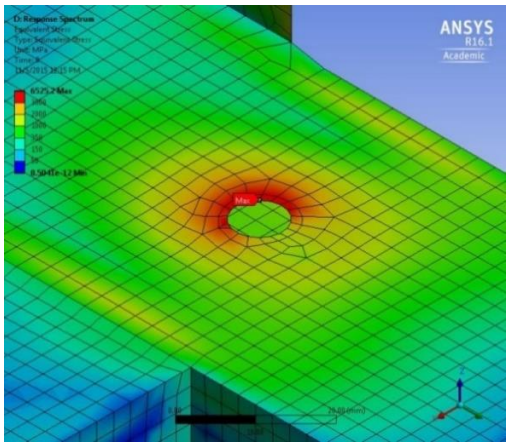


Figure 4-10 Magnified view for equivalent stresses RA-analysis

The next result is of the office vibration (Harmonic response). It is visible that there is very limited amount of deformation and stress being induced in the body. The maximum deformation is 0.0011 mm whereas maximum equivalent stress is 0.35 Mpa.

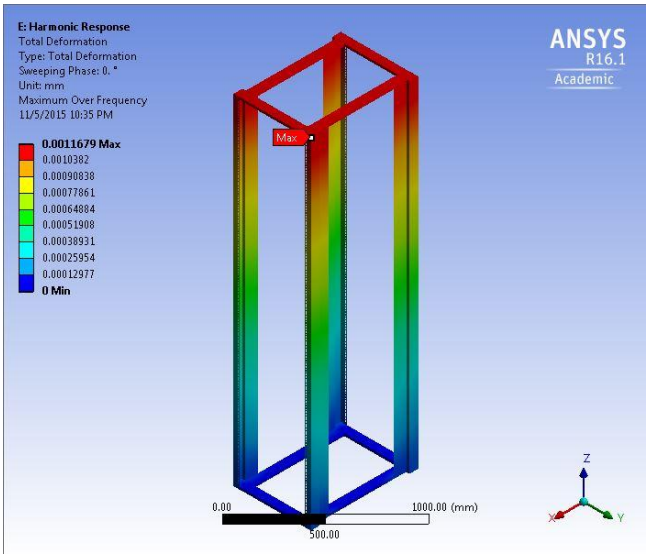


Figure 4-11 Deformation for harmonic response

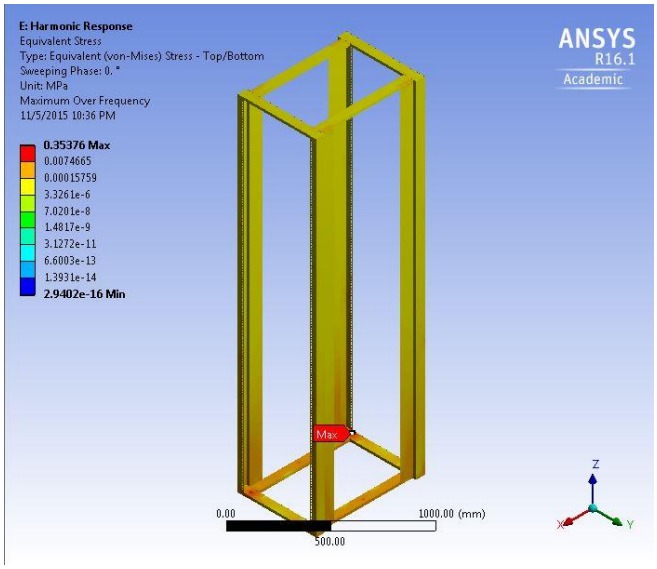


Figure 4-12 Equivalent stresses harmonic response

The results below are for transport vibration. Here the rack is empty and bolted at the base with standard earth gravity acting on it. This is the method in which the rack is transported from one location to another. The results indicate that the maximum deformation is 22.11 mm whereas maximum equivalent stress is 126.99 Mpa

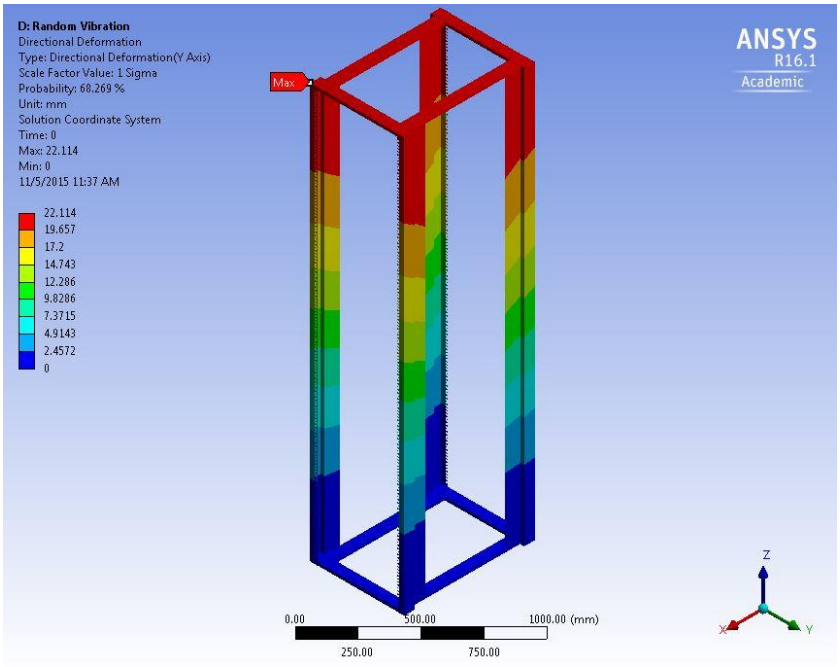


Figure 4-13 Total deformation random vibration

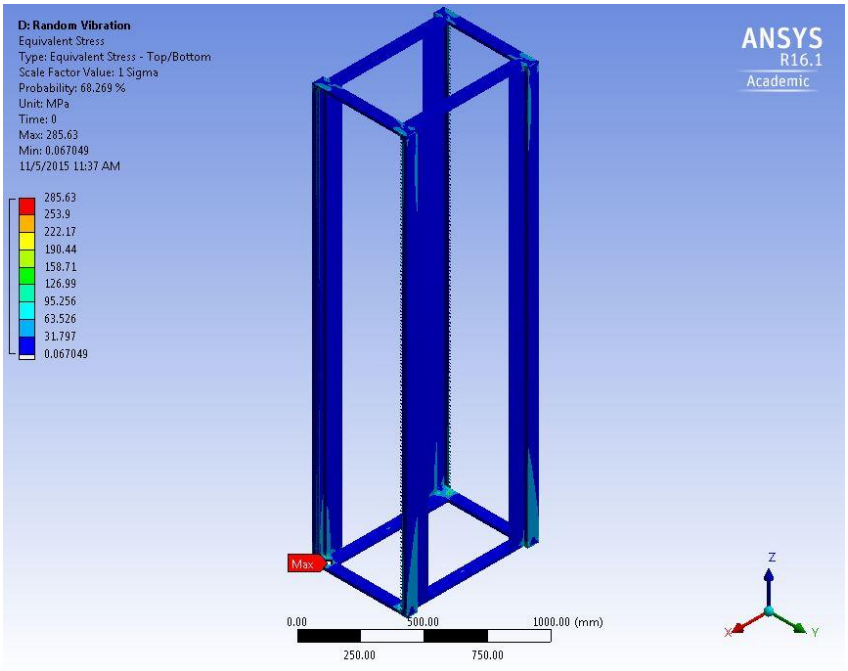


Figure 4-14 Equivalent stresses random vibration

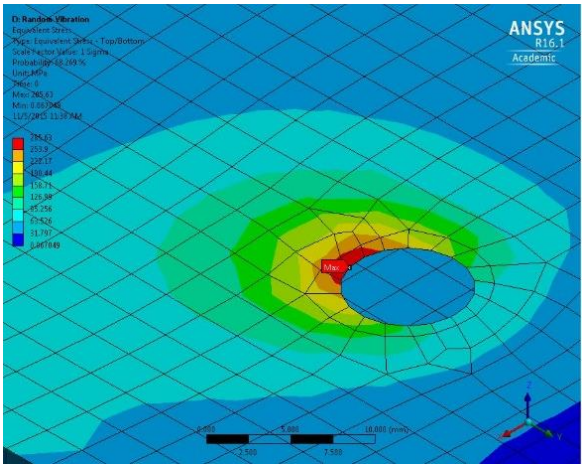


Figure 4-15 Magnified view of equivalent stresses random vibration

4.2 Case study two

Here a group of the old racks are placed side by side with point mass load. In this scenario the racks are bolted together to act as one unit to improve structural stability. This setup is done to check if grouping the racks is a better option as compared to keeping them individually stacked.

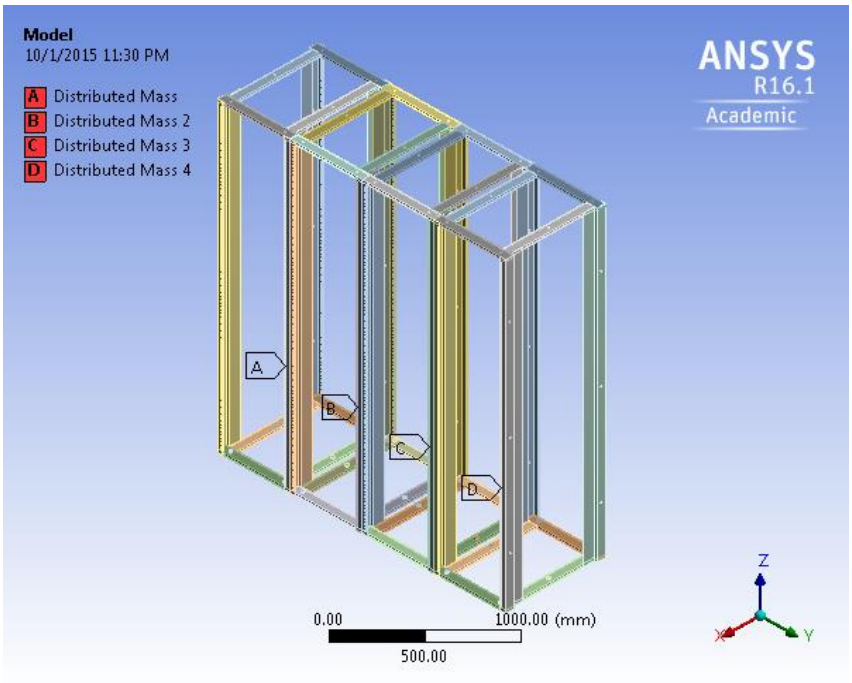


Figure 4-16 Geometry

The body is constrained at the bottom with bolting effect (bonded contact support). A mass load of 1530 Kg is applied to each rack to subject it to maximum loading before testing.

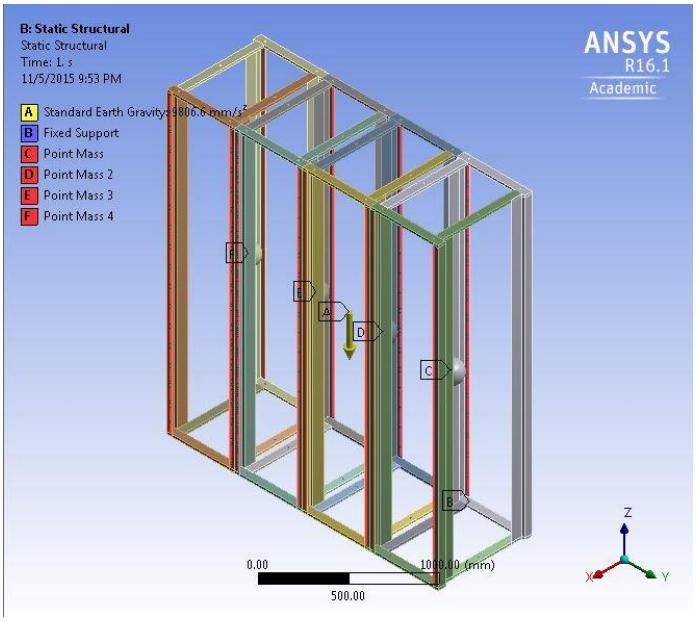


Figure 4-17 Grouped rack boundary conditions

The figure below shows the maximum deformation of 0.15 mm caused by the static structural load of 1530 kg per frame while they are placed side by side.

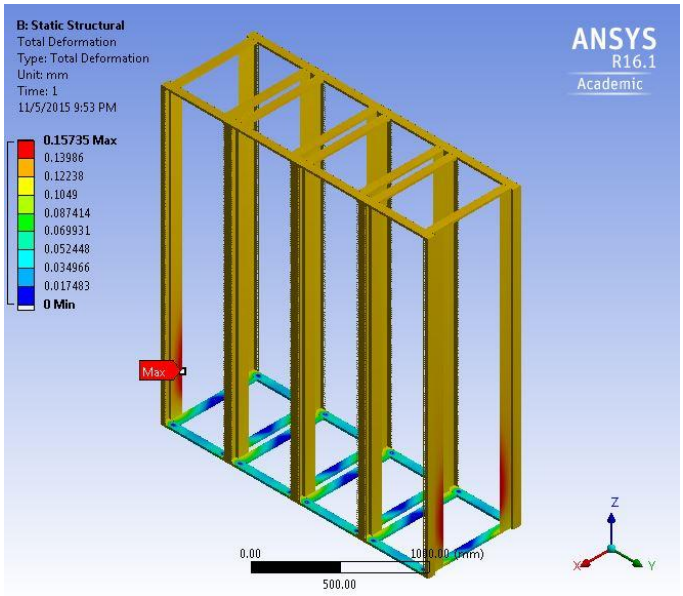


Figure 4-18 Grouped rack deformation

The figure below shows the maximum equivalent stresses of 100 Mpa caused by the static loading of 1530 kg per frame while they are placed side by side.

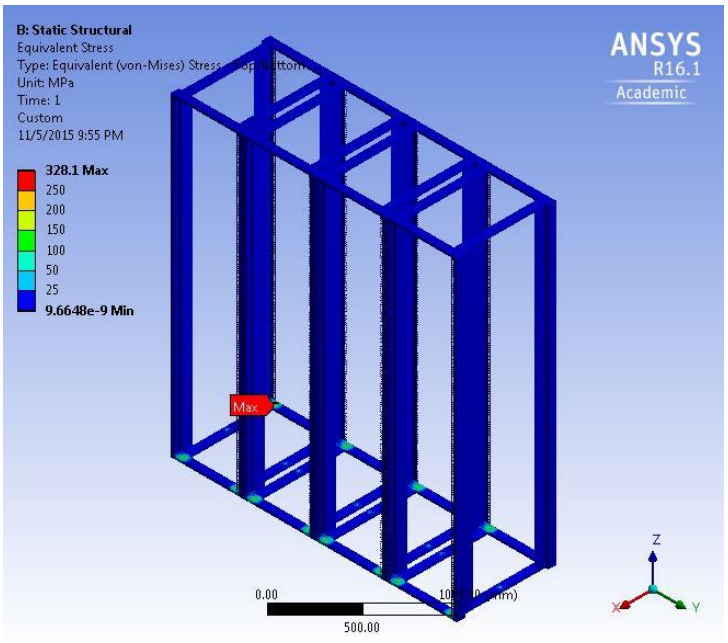


Figure 4-19 Grouped rack equivalent stress for static loading

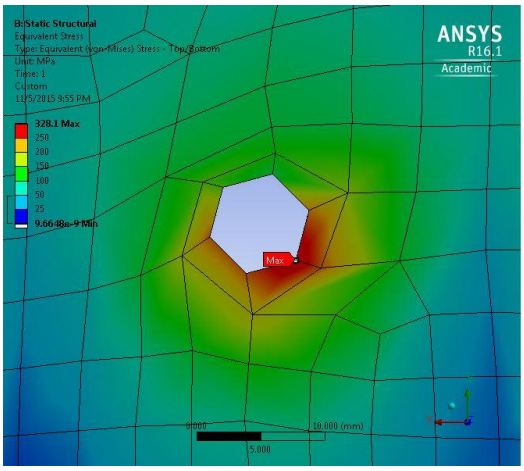


Figure 4-20 Grouped rack equivalent stress for static loading

The figure below shows the maximum deformation of 6.61 mm caused by the response spectrum curve. Here the direction of the accelerated load is in the X direction.

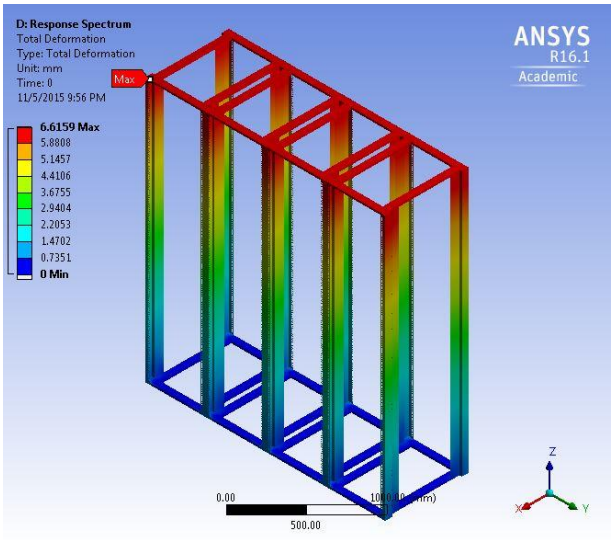


Figure 4-21 Grouped old rack deformation (RS analysis)

The figure below shows the maximum equivalent stress of approximately 350 Mpa caused by the response spectrum curve. Here the direction of the accelerated load is in the X direction.

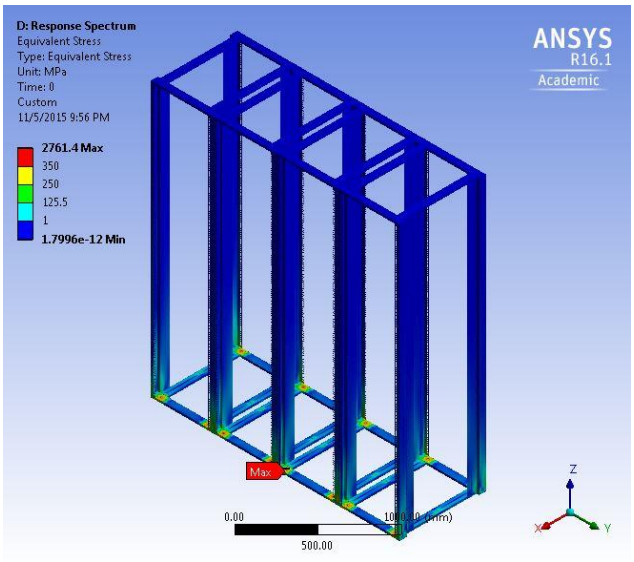


Figure 4-22 Grouped old rack stress (RS analysis)

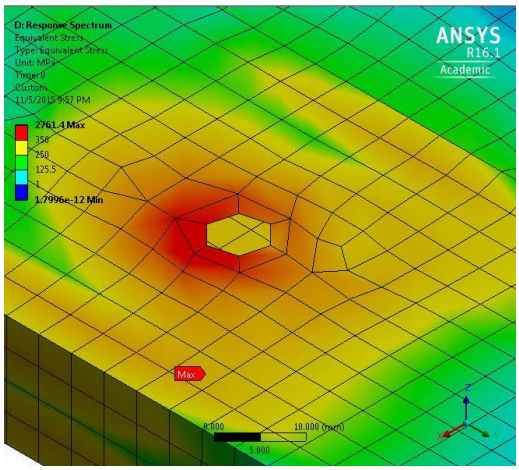


Figure 4-23 Grouped old rack stress (RS analysis) magnified

The next result is of the office vibration (Harmonic response). It is visible that there is very limited amount of deformation and stress being induced in the body. The maximum deformation is 0.0010 mm whereas maximum equivalent stress is 0.43 Mpa.

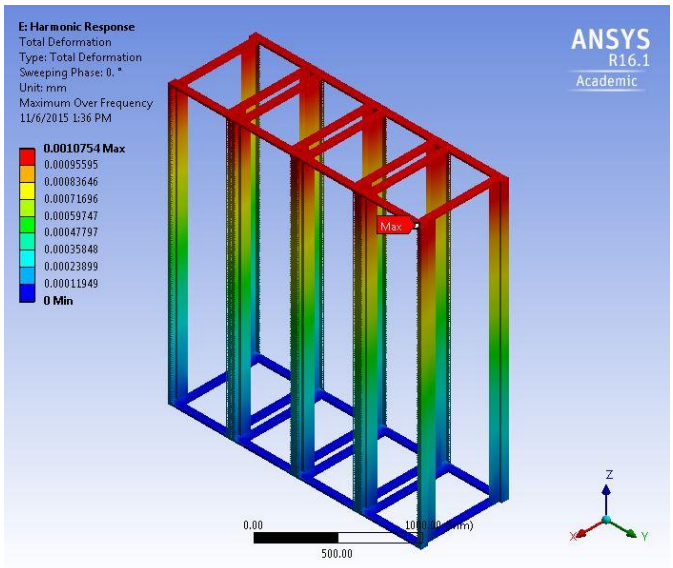


Figure 4-24 Grouped old rack deformation for harmonic response

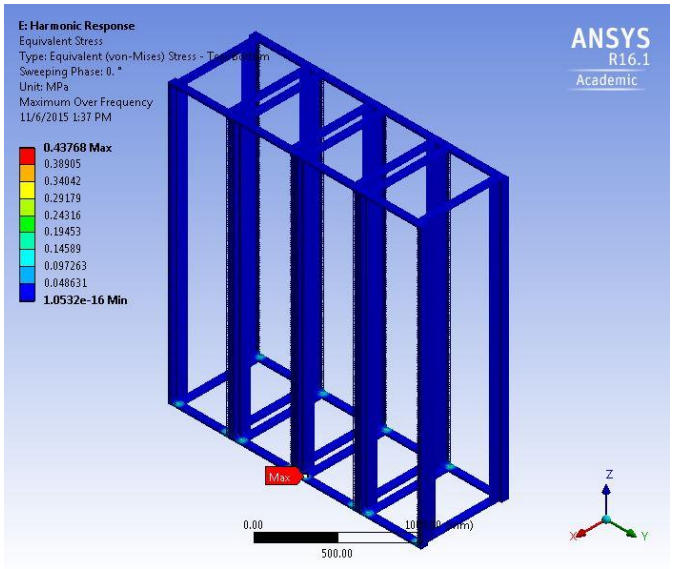


Figure 4-25 Grouped old rack stress for harmonic response

4.3 Case study three

Here in case three we see the single frame as shown in the figure below. This is the second type of the frame that we will be testing. The geometry is a simple structure with four base structures that will be bolted to the floor. Four vertical members where the servers will be attached and four top supporting members. Additional side members are attached to the supply excess support and increase structural stability. The material thickness for the sheet metal has been increased to 3 mm for the base structure to avoid failure due to shearing of the base.

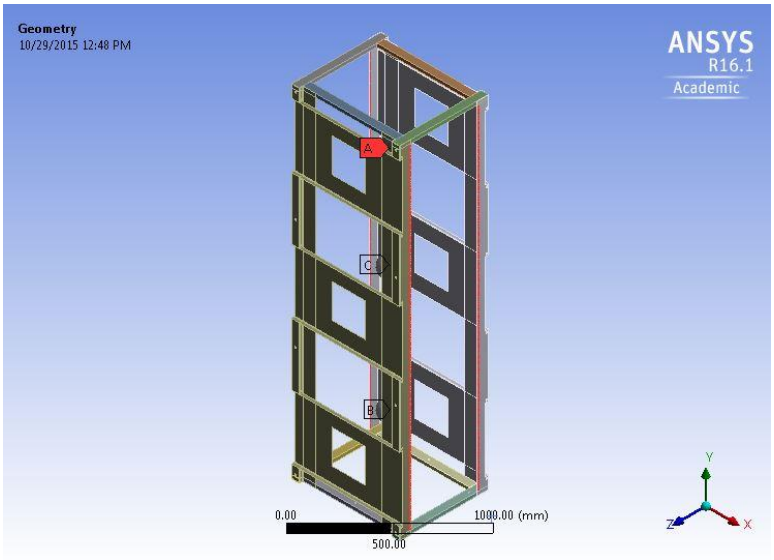


Figure 4-26 Geometry

As discussed earlier in case one, the modelled geometry is of surface kind. Actual servers have been replace by point masses remotely attached to the main body. The total weight being added is 1530 KG (3373 Lbs.) in the form of point loads. The base is fixed at

the bottom as a bonded contact. In actual 6 M8 bolts can be used to obtain more accurate results.

The figure below demonstrated the loading and fixed support acting on the geometry. Point masses 1, 2 and 3 are each 510 KG which add up to the total weight of 1530 KG which is the required maximum mass.

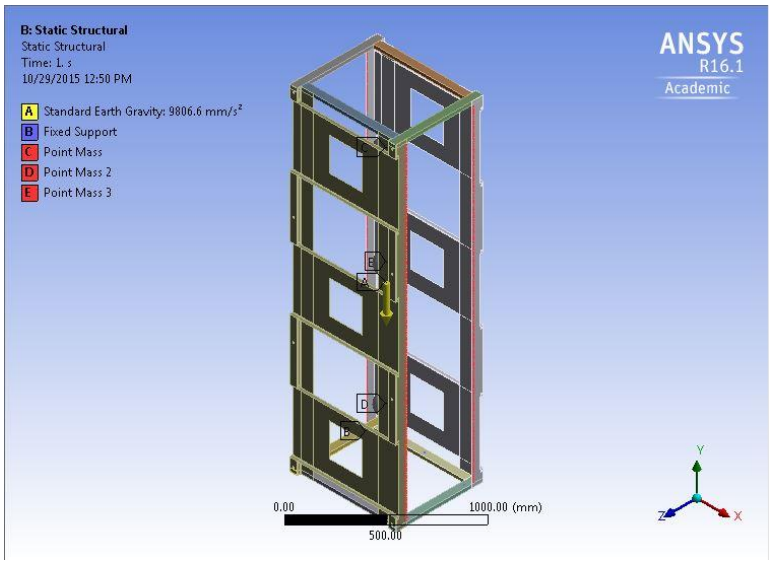


Figure 4-27 Boundary conditions

A front view of the same structure above is shown in the next figure giving us a clearer view of the load setup.

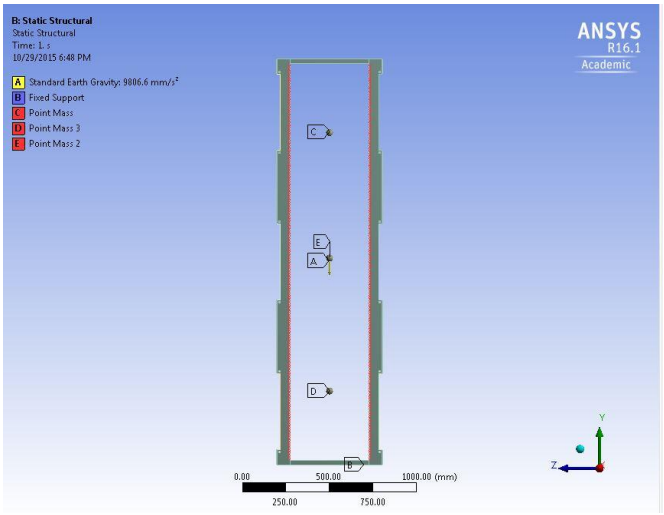


Figure 4-28 Front view of boundary conditions

Once the connections, mesh and static loads have been applied we can start the compilation. We look for deformation and equivalent stress in the body as they are the limiting factors for our design. The figures below shows us maximum deformation and maximum equivalent stress during static loading for this structure. The maximum Deformation occurs in the upper half of the rack and has a maximum value of 0.119 mm. Also maximum equivalent stress occurs at the point of the fixed support and is 429 Mpa. The edge on which this stress is generating is fixed in all degree of freedom and hence shows higher stress than what is occurring. It is noticeable that the stress in the body will not exceed 250 Mpa.

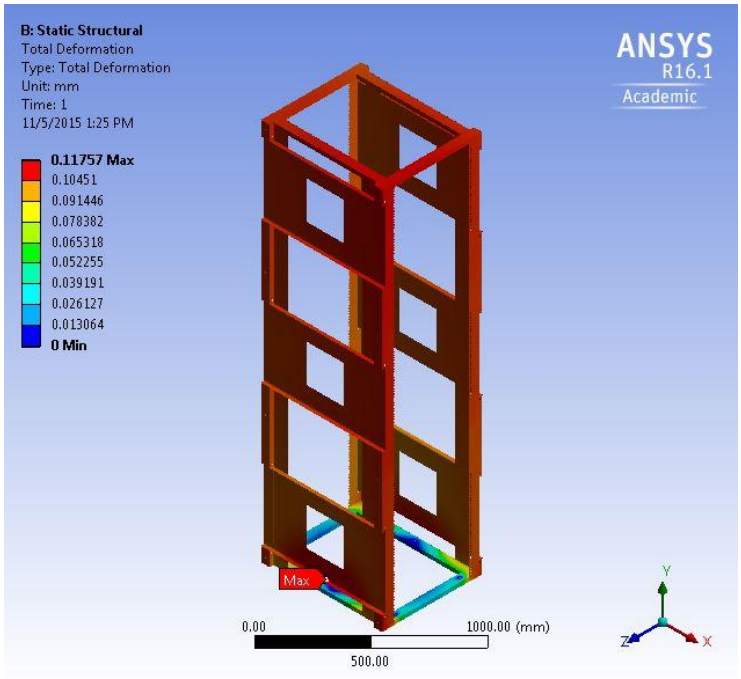


Figure 4-29 Total deformation for static loading

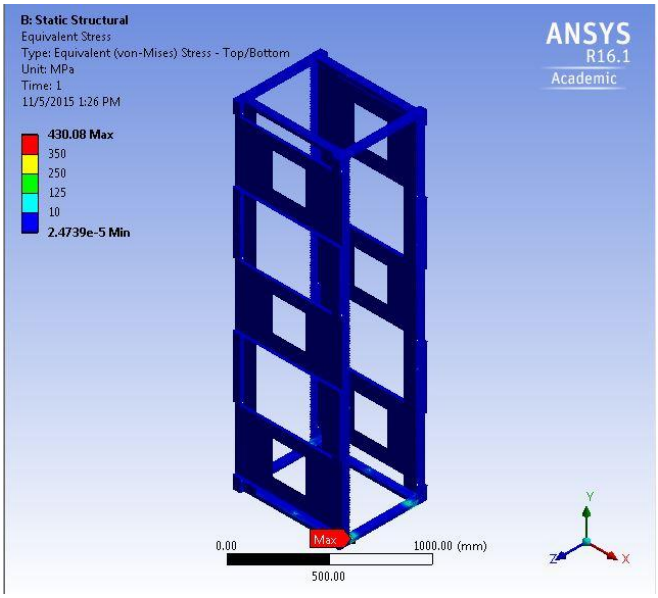


Figure 4-30 Equivalent stress for static loading

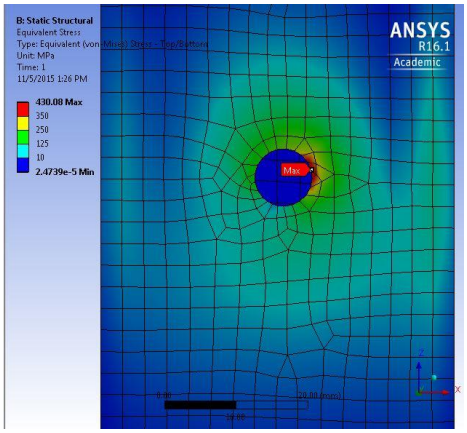


Figure 4-31 Magnified view for equivalent stresses for static loading

It is visible from the figure above that there is stress concentration at a bonded contact. This stress is generated due to the boundary conditions as the edge is fixed and has no allowable elastic or plastic deformation. The factor of safety for the structure is shown in the figure below and it is seen that the design is robust for such kinds of working loads.

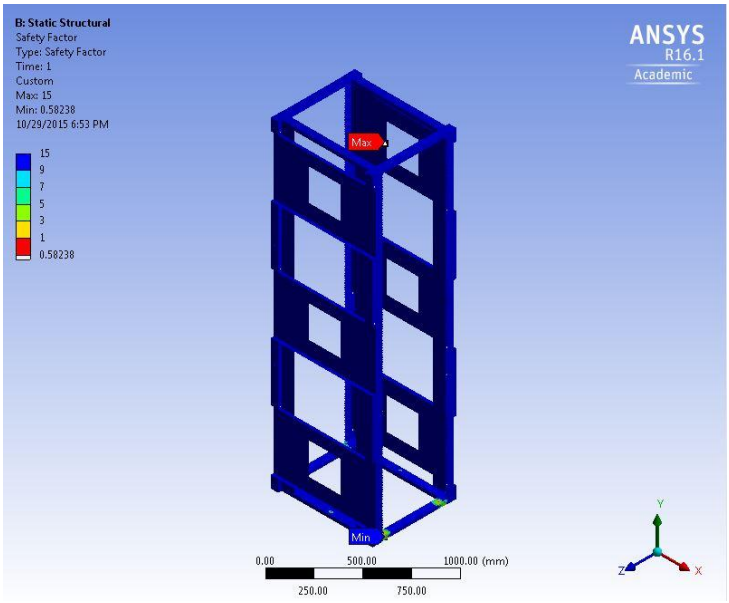


Figure 4-32 Factor of safety for static loading

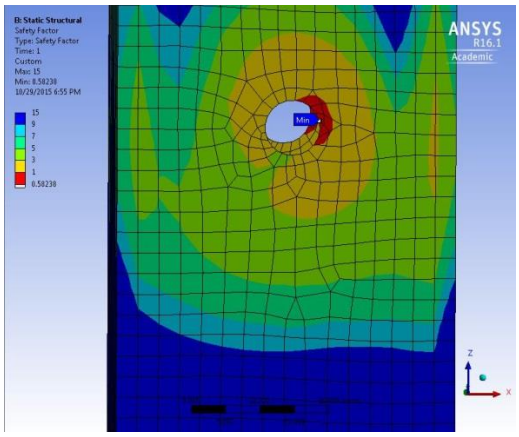


Figure 4-33 Magnified F.O.S

Once the static loading test is completed we can move on to the modal analysis. The solver provides us with the natural modes of vibration and we can couple those results with the RS-analysis tool in workbench to calculate the response of the body due to earthquake vibration.

The figures below show us the Maximum deformation and maximum equivalent stress generated in the body while running this test. The results indicate that maximum deformation is approximately 13 mm.

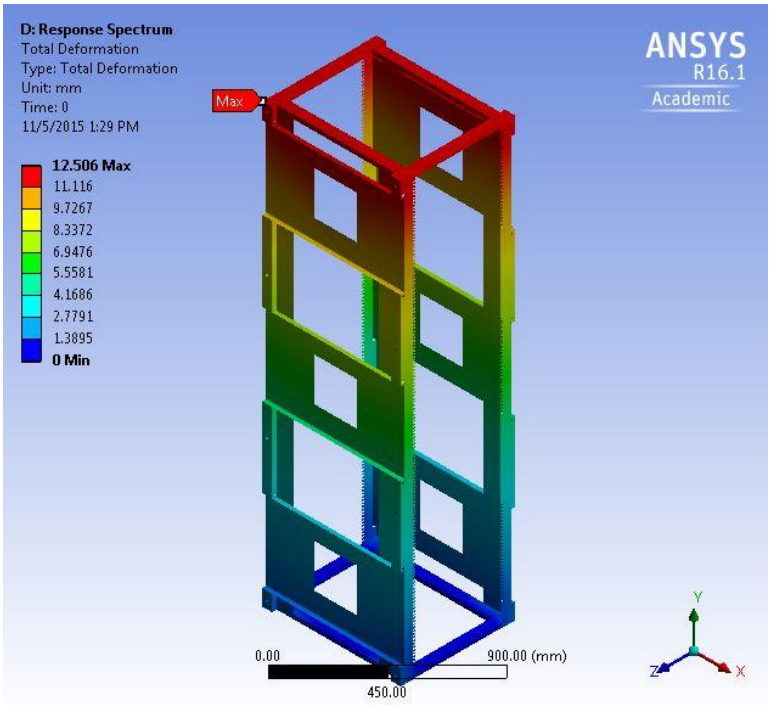


Figure 4-34 Total deformation RS-analysis

The maximum equivalent stress is 4679 Mpa. The magnified image of the location where these stresses are being generated shows that the stress is caused due to a fully constrained edge. As the edge is not allowed to deform (elastic or plastic deformation) the stress keeps on rising. By not considering this region it can be seen that stresses in body do not exceed 350 Mpa.

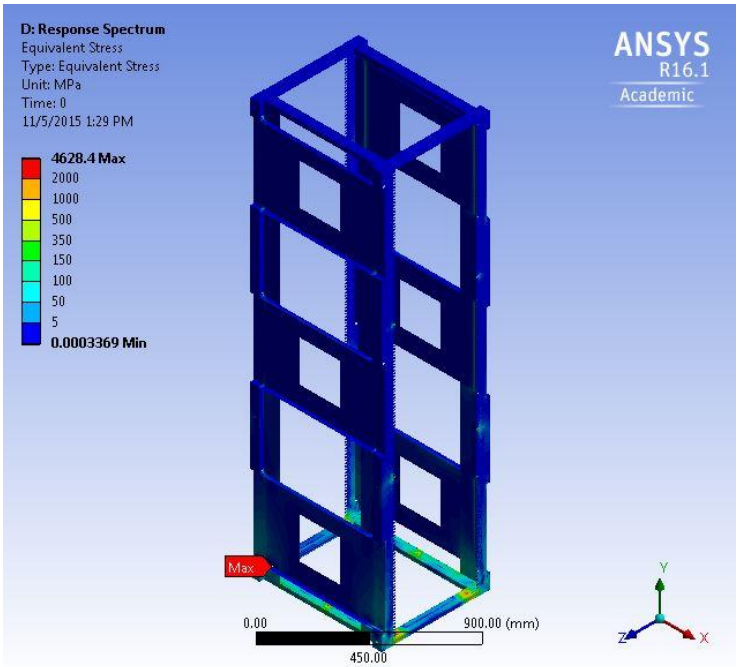


Figure 4-35 Equivalent stress for RS-analysis

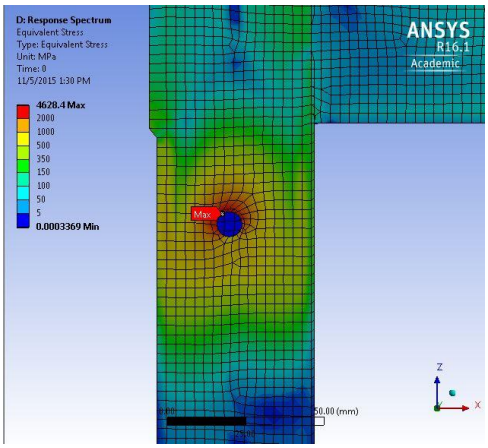


Figure 4-36 Magnified view equivalent stress for RS-analysis

The next result is of the office vibration (Harmonic response). It is visible that there is very limited amount of deformation and stress being induced in the body. The maximum deformation is 0.0023 mm whereas maximum equivalent stress is 0.89 Mpa.

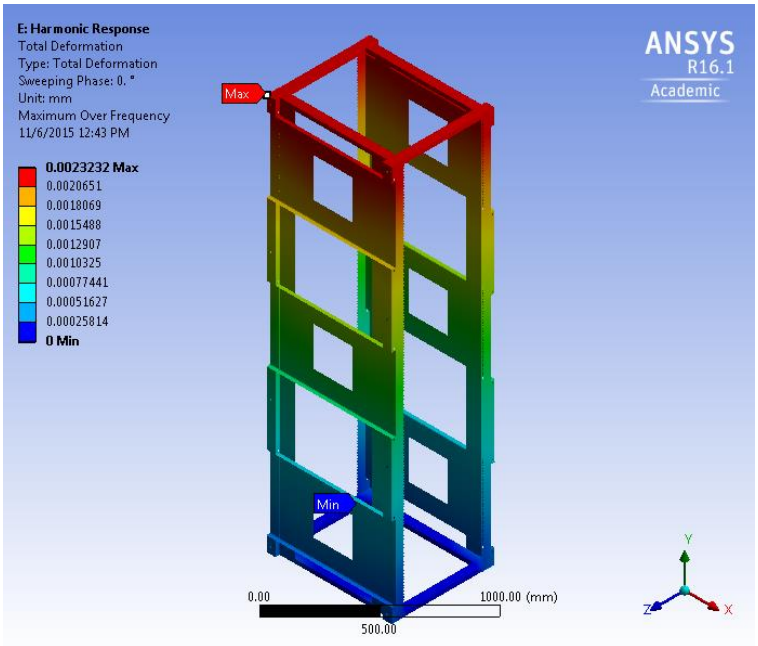


Figure 4-37 Total deformation for harmonic response

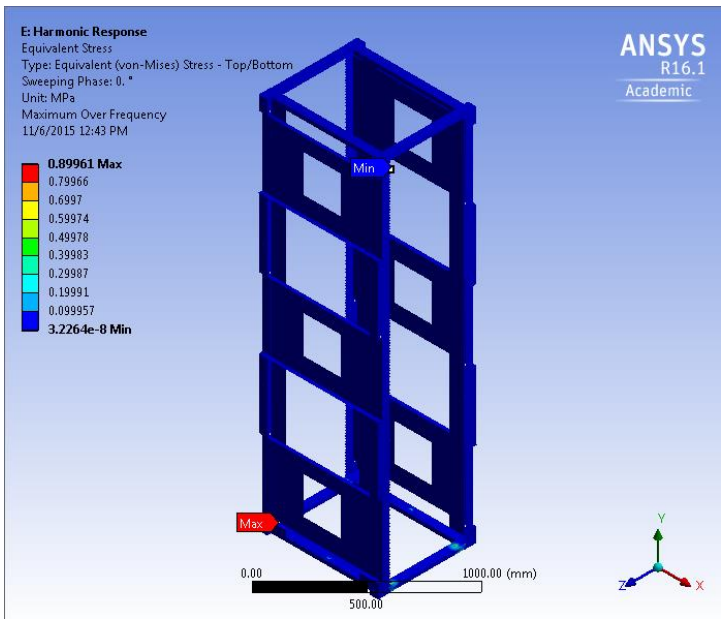


Figure 4-38 Equivalent stress for harmonic response

The results below are for transport vibration. Here the rack is empty and bolted at the base with standard earth gravity acting on it. This is the method in which the rack is transported from one location to another. The results indicate that the maximum deformation is 15.52 mm whereas maximum equivalent stress is 165 Mpa

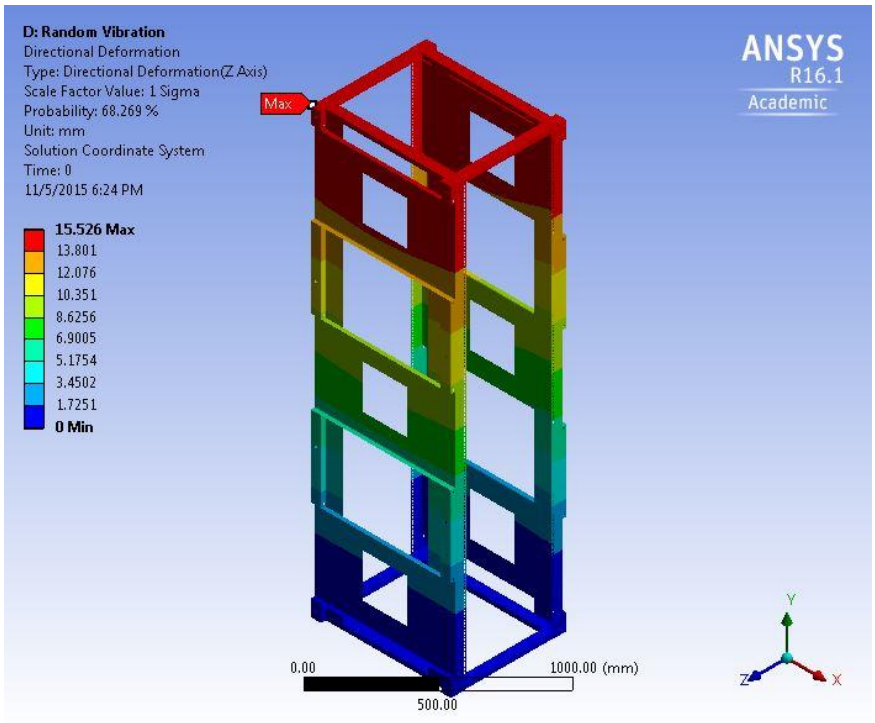


Figure 4-39 Deformation for random vibration

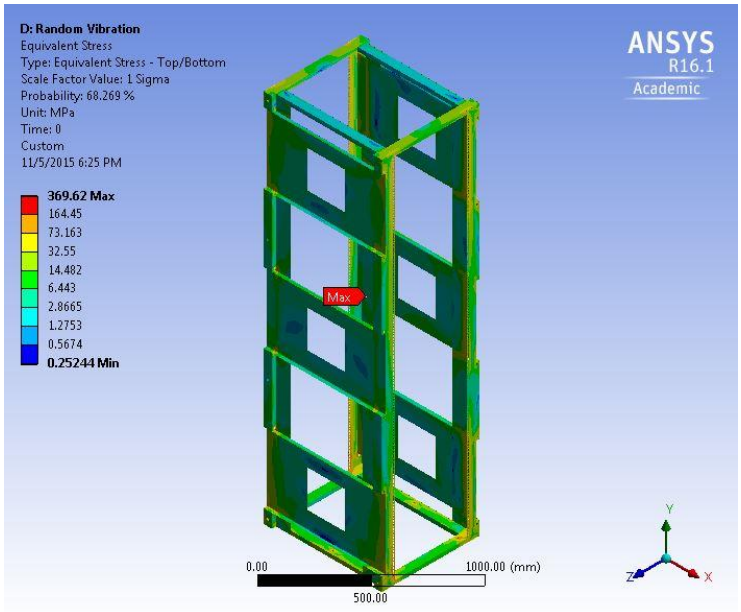


Figure 4-40 Equivalent stress for random vibration

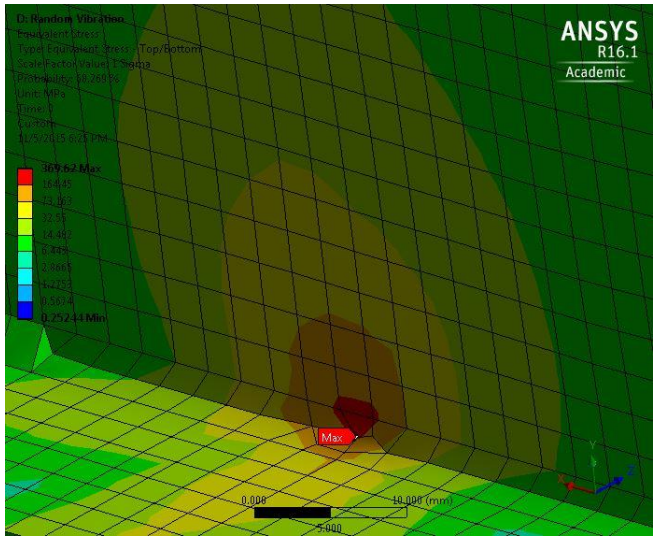


Figure 4-41 Magnified view equivalent stress for random vibration

4.4 Case study four

Here a group of the new racks are placed side by side with point mass load. In this scenario the racks are bolted together to act as one unit to improve structural stability. This setup is done to check if grouping the racks is a better option as compared to keeping them individually stacked.

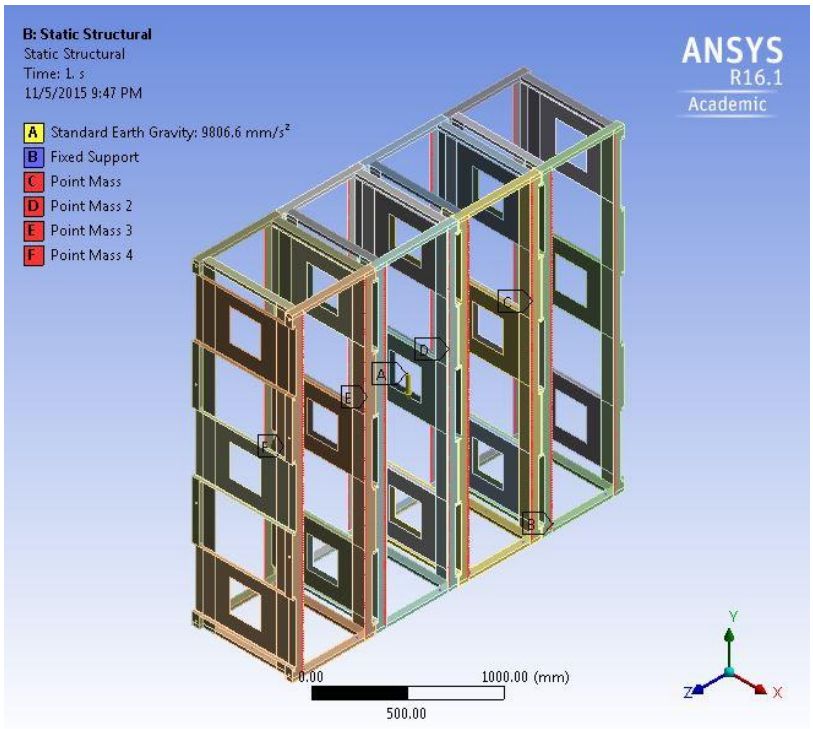


Figure 4-42 Boundary conditions

The body is constrained at the bottom with bolting effect (bonded contact support). A mass load of 1530 Kg is applied to each rack to subject it to maximum loading before testing.

The figure below shows the maximum deformation of 0.11 mm caused by the static structural load of 1530 kg per frame while they are placed side by side.

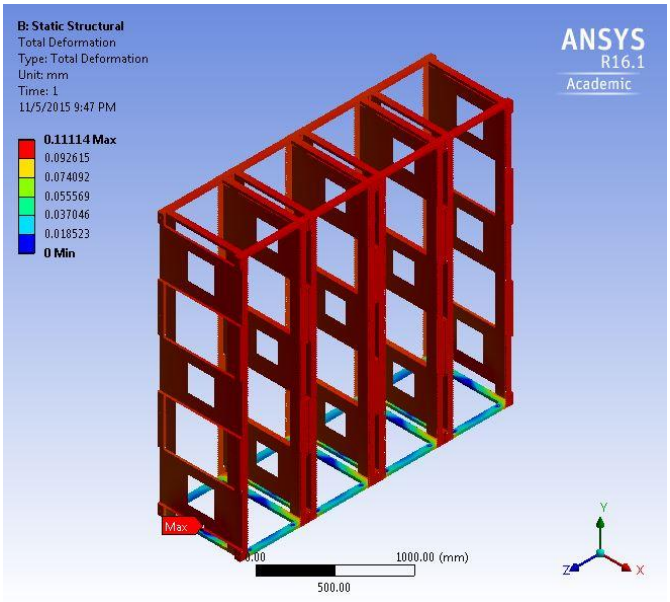


Figure 4-43 Deformation for static structural

The figure below shows the maximum equivalent stresses of 80 Mpa caused by the static loading of 1530 kg per frame while they are placed side by side.

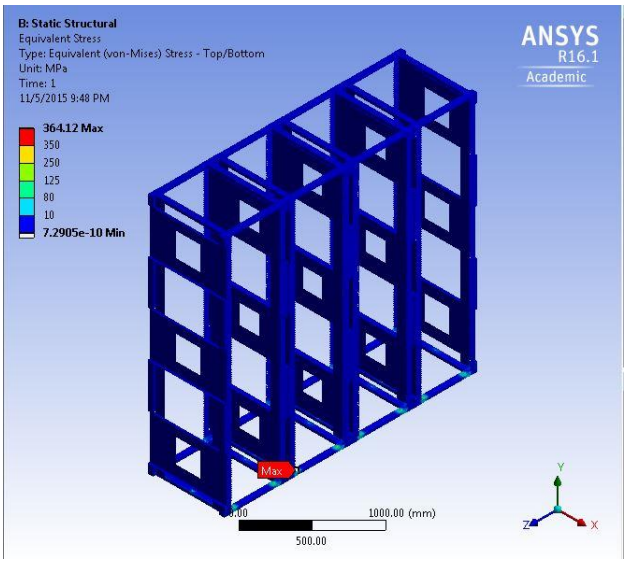


Figure 4-44 Equivalent stresses for static structural

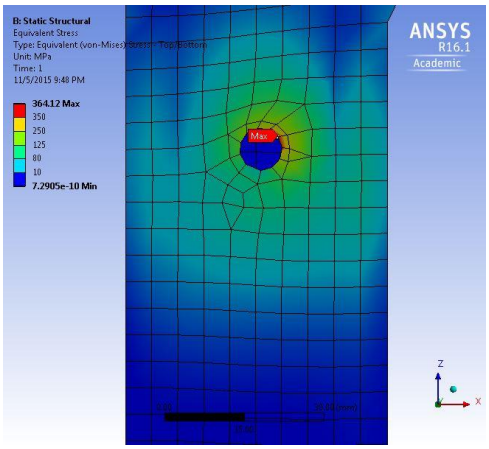


Figure 4-45 Magnified view of equivalent stresses for static structural

The figure below shows the maximum deformation of 4.73 mm caused by the response spectrum curve. Here the direction of the accelerated load is in the X direction.

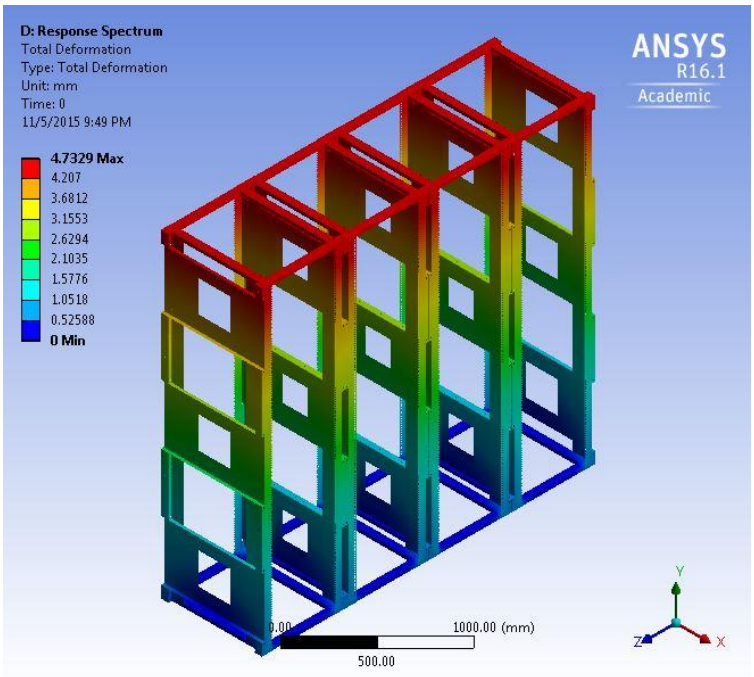


Figure 4-46 Deformation for RS-analysis

The figure below shows the maximum equivalent stress of about 350 Mpa caused by the response spectrum curve. Here the direction of the accelerated load is in the X direction.

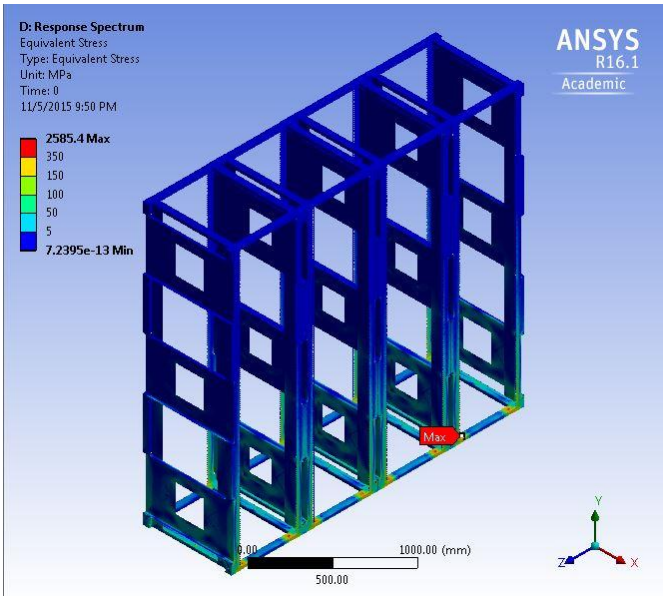


Figure 4-47 Equivalent stresses for RS-analysis

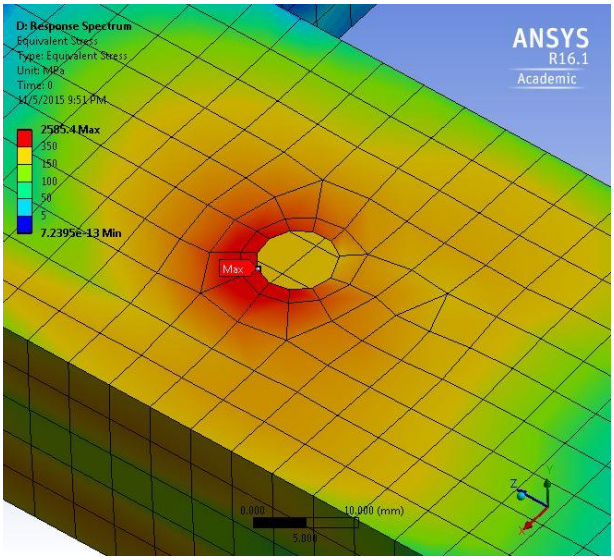


Figure 4-48 Magnified view of equivalent stresses for static structural

The next result is of the office vibration (Harmonic response). It is visible that there is very limited amount of deformation and stress being induced in the body. The maximum deformation is 0.0009 mm whereas maximum equivalent stress is 0.50 Mpa.

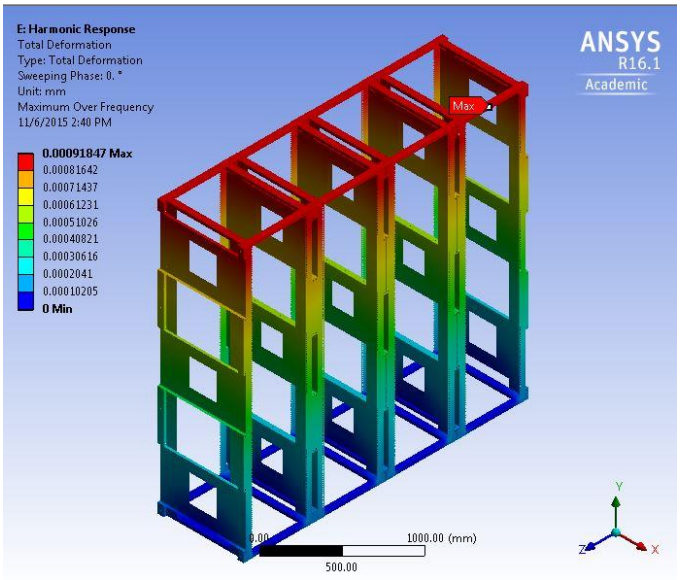


Figure 4-49 Deformation for harmonic response

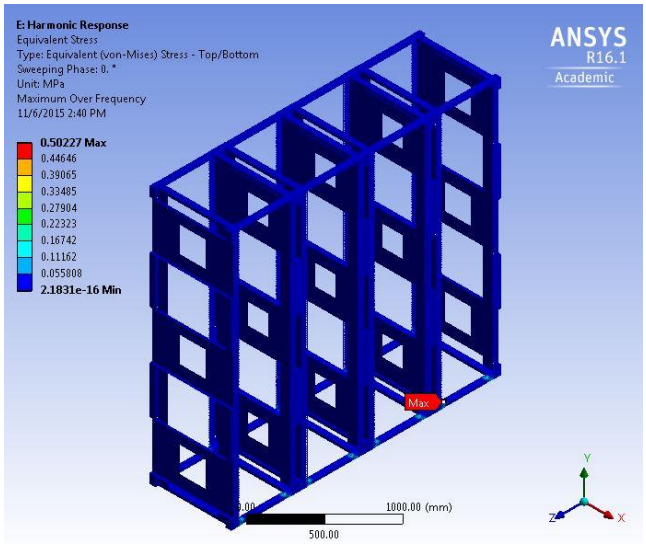


Figure 4-50 Equivalent stresses for harmonic response

Chapter 5

SUMMARY OF RESULTS

The results appear quite acceptable given the nature of the limiting factors. These factors included the meshing and corresponding resource limitations when applied to such large and complex assemblies. For complex structures and models, improving the correlation remains challenging. Following is the result table for the detailed analysis.

Table 5-1 Result table

	Mass 1530 kg added	Case 1	Case 2	Case 3	Case 4	Max Allowable
Deformation (Max)						
Static Structural	Yes	0.16 mm	0.15 mm	0.11 mm	0.11 mm	-
RS-Analysis	Yes	21.44 mm	6.61 mm	12.50 mm	4.73 mm	75 mm
Harmonic Response	Yes	0.0011 mm	0.0010 mm	0.0023 mm	0.0009 mm	-
Random Vibration	No	22.11 mm	N/A	15.52 mm	N/A	-
Equivalent Stress (Max)						
Static Structural	Yes	150 MPa	100 MPa	125 MPa	80 MPa	YS
RS-Analysis	Yes	~ 350 MPa	~ 250 - 350 MPa	~ 350 MPa	~ 150 - 350 MPa	YS
Harmonic Response	Yes	0.35 MPa	0.43 MPa	0.89 MPa	0.5 MPa	YS
Random Vibration	No	126.98 MPa	N/A	164.45 MPa	N/A	YS

Note:
Mesh sensitivity analysis has been done for case 1 and 3
YS = Tensile Yield Strength of Material

Chapter 6

CONCLUSION AND FUTURE WORK

The following conclusion can be made from the analysis:

- It is observed in the summary of result that both the racks are well suited for all types of vibrational loads.
- The maximum stress being generated in the rack does not exceed 350 MPa under any loading.
- For the materials being tested it is safe to conclude that material A572 series steel grade 55 should be used to ensure safety as it's tensile yield strength is 380 MPa.
- Also the maximum deflection is kept much under the limit which is 3 inches (75 mm)
- The results also indicate that the racks grouped side by side improves the structural stability.

As a part of future work,

- The investigation should focus on incorporating bolted connection and improved component stiffness.
- A detailed model can be used in place of a compact model.
- Transient analysis could be conducted to achieve most accurate and precise results.

References

1. Telcordia technologies generic requirements gr-63-core issue 3, 2006. NEBSTM Requirements: Physical Protection.

2. M. Waqas Anjum, Basit Khalid and Abdul Rehman. 2012. Technical Journal, University of Engineering and Technology Taxila, Vibration analysis issue, 2012. Seismic Analysis of Electronic Cabinet using ANSYS.
http://web.uettaxila.edu.pk/techJournal/2012/Seismic%20Analysis%20of%20Electronic%20Cabinet%20using%20A NSYS.pdf

3. ANSYS Tutorial: Earthquake analyses in workbench
http://www.edr.no/blogg/ansys_bloggen/ansys_tutorial_earthquake_analyses_in_workbench

4. Properties and Selection: Irons, Steels, and High Performance Alloys.2005.ASM Handbook Volume 1.Pg 384-385.
http://usuarios.fceia.unr.edu.ar/~adruker/Asm%20Metals%20Handbook%20Volume%201%20-%20Properties%20And%20Selection%20Irons,%20Steels,%20And%20High-Performance%20Alloys.pdf

5. Budy Notohardjono, Richard Ecker, Jiantao Zheng, John Torok. 2012. Structures Congress 2012 © ASCE 2012. SEISMIC TESTING AND ANALYSIS OF MAIN FRAME COMPUTER STRUCTURE.
http://ascelibrary.org/doi/pdf/10.1061/9780784412367.125

6. B. D. Notohardjono J. S. Corbin S. J. Mazzuca S. C. McIntosh H. Welz. 2001. IBM J. RES. & DEV. VOL. 45 NO. 6 NOVEMBER 2001. Modular server frame with robust earthquake retention
http://citeseerx.ist.psu.edu/viewdoc/download?doi=10.1.1.88.6368&rep=rep1&type=pdf

7. Richard Mitchell, Joe Luxmoore, Mark Leddin . Ansys presentation. March 2 2012. Mesh Connections Meshing Best practice.
http://www.ansys.com/staticassets/ANSYS%20UK/staticassets/Release%2014.0%20Downbloads/Mesh%20connections%20and%20best%20practice_v2%20(1).pdf

8. MatWeb. (2015). Online material information. Retrieved from http://www.matweb.com/

9. Dynamic analysis using response spectrum seismic loading. From “Static and Dynamic of Structures” by Ed Wils1on. Modified July 13 2014

10. John Van Baren, Vibration Research Corporation, Jenison, Michigan . What is Random Vibration Testing?

11. United states Geological Survey http://www.usgs.gov/

12. Shock-and-vibration-analysis-using-ansys-mechanical
http://www.ozeninc.com/wp-content/uploads/2015/05/shock-and-vibration-analysis-usingansys-mechanical.pdf

Biographical Information

Framroz Bharucha is a Mechanical engineer graduate student at University of Texas at Arlington who completed his studies in fall, 2015. He had a great opportunity to work for a company projects with CommScope Incorporation and his area of interest has always been design engineering and the thesis work speaks about the interest. Framroz graduated from University of Pune, India as an undergraduate in mechanical engineering in 2013 during which did an internship for a year with Bharat Forge India Ltd., a forging company which mainly deals with manufacturing components for automobiles. Upon graduation, Framroz would love to work for automotive industry as a design engineer and gain, improve knowledge with industrial experiences.