# VERIFICATION AND VALIDATION OF A COMMERCIAL SOFTWARE PRODUCT THAT USES A NEW TYPE OF FINITE ELEMENT METHOD IN THE DESIGN CYCLE OF A WECDIS CONSOLE PROTOTYPE

by
Özgür Erkan İncirci

Submitted to Graduate School of Natural and Applied Sciences
in Partial Fulfillment of the Requirements
for the Degree of Master of Science in
Mechanical Engineering

Yeditepe University
2018

VERIFICATION AND VALIDATION OF A COMMERCIAL SOFTWARE PRODUCT THAT USES A NEW TYPE OF FINITE ELEMENT METHOD IN THE DESIGN CYCLE OF A WECDIS CONSOLE PROTOTYPE

APPROVED BY:

Assist. Prof. Dr. Ali Fethi Okyar .....................................
(Thesis Supervisor)

Assoc. Prof. Dr. Ali Gökşenli .....................................

Assoc. Prof. Dr. Nezih Topaloğlu .....................................

DATE OF APPROVAL: ..../..../2018

# ACKNOWLEDGEMENTS

This study was carried out with the support of TÜBİTAK Project No. 3140763 and also under approval of MILSYS Defence Technologies LLC.

(Bu çalışma 3140763 nolu TÜBİTAK Projesi Desteği ve MILSYS Savunma Teknolojileri Ltd. Şti. onayı ile yapılmıştır)

# ABSTRACT


## VERIFICATION AND VALIDATION OF A COMMERCIAL SOFTWARE PRODUCT THAT USES A NEW TYPE OF FINITE ELEMENT METHOD IN THE DESIGN CYCLE OF A WECDIS CONSOLE PROTOTYPE

WECDIS is an electronic chart display system for warships that is being used commonly around the world navies. Since the introduction of GENESIS and MILGEM Projects in mid-2000's one of the top priorities in the Turkish Navy's agenda has been the development of national technologies for new vessels as well as modernizing the old ones. In order to fulfil one of these necessities, a TUBİTAK R&D Project was initiated with the aim of building a custom WECDIS Console as its end product. The current study focuses on the design process cycle of this product by utilizing a new FEM approach called the external finite element approximations to meet highly demanding MIL-STD standards that are necessary because of the harsh conditions of naval environment. Research and analysis were performed to validate and verify the new approach and compare it with the traditional FEM-based tools to clarify and evaluate the feasibility and convenience of the former. Anticipated result of the new approach offers a cut down CPU usage by 95 per cent and decrease the RAM usage by 88 per cent. The drawback in accuracy is only expected to be around 10 per cent.

# ÖZET


## YENİ BİR TİP SONLU ELEMANLAR METODU KULLANNAN TİCARİ YAZILIM ÜRÜNÜNÜN WECDIS KONSOL PROTOTİP TASARIM AŞAMASINDA TAHKİK VE TASDİK EDİLMESİ

WECDIS dünya çapındaki donanmaların savaş gemilerinde kullanılan, elektronik harita gösterim sistemidir. 2000'li yılların ortalarında GENESIS ve MILGEM Projeleriyle birlikte TC Deniz Kuvvetlerinin birinci önceliği inşaa edilecek yeni gemiler ya da modernize edilecek eski gemiler için yerli teknoloji geliştirmek olmuştur. Bu ihtiyaçlardan birisi olan WECDIS Konsolu üretimini gerçekleştirmek için TÜBİTAK destekli bir AR-GE Projesi başlatılmıştır. Bu çalışma gemi ortamının sert koşulları nedeniyle gerekli olan MIL-STD Standartlarını karşılamak için Harici Sonlu Elemnalar adı verilen yeni bir sonlu elemanlar metodu yaklaşımı kullanarak bu ürünün tasarım sürecine odaklanmaktadır. Yeni yaklaşımın geçerliliğini tahkik ve tasdik etmek ve geleneksel sonlu elemanlar metodu tabanlı araçlar ile fizibilite ve kullanım kolaylığını karşılaştırmak için araştırma ve analiz yapıldı. Yeni yaklaşımın CPU kullanımını yüzde 95, RAM kullanımını da yüzde 88 azaltacağı öngörülmektedir. Sonuçların doğruluğu açısından sadece yüzde 10'luk bir hata payı öngörülmektedir.

# TABLE OF CONTENTS

# LIST OF FIGURES

# LIST OF TABLES

# LIST OF SYMBOLS/ABBREVIATIONS

| | |
|---|---|
| A | Differential Operator |
| $a_i$ | Unknown Factors |
| B | Load Vector |
| $K$ | Stiffness Matrix |
| L | Differential Operator |
| $p_i$ | Basis Approximation Functions |
| V | Subspace of Sololev Space |
| | |
| Ω | Domain |
| δ | Operator |
| γ | Operator |
| | |
| ASME | American Society of Mechanical Engineers |
| BVP | Boundary Value Problem |
| CAD | Computer Aided Design |
| CFD | Computational Fluid Dynamics |
| DOF | Degrees of Freedom |
| EMI | Electro Magnetic Interference |
| FEM,FEA | Finite Element Method, Finite Element Analysis |
| NAFEMS | National Agency for Finite Element Methods |
| PSM | Precise Solids Method |
| WECDIS | Electronic Chart Display System for Warships |

# 1. INTRODUCTION

The design process of military grade naval equipment requires knowhow from different fields of engineering for optimization. The device's working environment is what makes it distinct from products we use in our daily lives. In order to classify a product as mil-grade or marine type there are certain tests and certifications that are not required for industrial counterparts.

The subject of this study is a WECDIS Console, an electronic chart display system designed to be used in warships. Therefore there are requirements for both mil-grade and marine type. Some of the requirements are;

- The console needs to withstand extreme temperatures.
- The console needs to be rigid enough to function in naval environment.
- The console should to be protected against dust and water.
- The console should be protected against electromagnetic interference (EMI).

For temperature constraints a detailed CFD analysis is required in order to decide the method of cooling and also the tools to monitor and heat exchange of the system. However this study is focused on the mechanical design aspect which, resides on area of solid mechanics. The protection against dust, water and EMI are in the area of solid mechanics but they mostly depend upon material selection and product selection.

This study is focuses on the rigidity of the console. In order to evaluate this feature it needs to withstand the shock and vibration test shown in the Table 1.1.

The mechanical design process comes down to two objectives. Design for manufacturing a system assembled to fulfil its functionality and also analysing its structural health to make sure it has the necessary rigidity for testing, The tool that being used for the second objective is the finite element analysis.

Finite Element Analysis is widely used in product development projects. Idea is to betterment of a product in certain ways that already manufactured and sometimes being actively used. In that regard these projects do not require that much of a man hour since the testing procedures and previous analysis input and outputs are already known. Only slight variations are made in the known procedure.

Table 1.1. Examples of environmental tests and their documentation

| **Test** | **Standard** |
|---|---|
| High Temperature Test | MIL-STD-810G [1] |
| Low Temperature Test | MIL-STD-810G |
| Dip proof Test | MIL-STD-108E [2] |
| Vibration Test | MIL-STD-167-1 [3] |
| Shock Test | MIL-STD-810E |
| Acoustic Noise Test | ISO 3744 [4] |
| Magnetic Field Emission Test | MIL-STD-461F [5] |
| Electrical Field Emission Test | MIL-STD-461F |
| Conducted Emission Test | MIL-STD-461F |

Research and development projects or completely new design project as the study follows on the other hand use finite element analysis throughout all project timeline. From material selection to manufacturing methods, part geometry and fastener types, lots of critical design decisions require this analysis.

For the large assemblies the amount of computing power is also much higher. Even this requirement is fulfilled, the necessary man hour for the analysis generally higher than the actual design.

After the design process is finished, prototype production and testing are the projects most challenging and cost effective periods. For a product similar to the one shown in this research, estimated cost and time for testing is € 40000 and 6 months. Because of that reason, lesser error occurs in finite element analysis lesser additional resource will be spent on testing.

With all that in mind an altenative appoach to traditional finite element method is investigated. The reseach was based on different meshing technices and meshless methods as well.

The new approach needs to have these features;

- Requires less computing power.
- Easy to transition between design to analysis since it needs to be used frequently throught the design process.
- Reqires less time to perform the analysis.
- Interpret result which are compatilbe with the MIL-STD test results.
- Needs to give accurate results to reduce testing and prototype production cost.

The next chapter consists various of alternative finite element approaches that are investigated for this study.

## 2. LITERATURE SURVEY

History of finite element method goes back 1940’s although it did not share a common name or set of principles at that time. What made them similar is that one trait which is mesh discretization of a continuous domain into discrete sub domains. These subdomains are generally called elements.

The work of W.Ritz [6] in 1909 and Galerkin [7] in 1915 which is approximate solution of boundary value problems is considered the foundations for finite element method. However some alternative finite element approaches has surfaced around those decades.

In 1926 Trefftz Method [8] was introduced as distinctly using fully discontinuous functions and boundary value problems with prescribed jumps. Later in 1977 Jirousek generalized Trefftz Method and implicated into finite element method. In Jirousek Method [9] the elements that are used are known as ‘’Trefftz Type’’ or ‘’T-element’’.

In 90’s several new approaches has surfaced as lots of them are based on Trefftz Type elements generally considered as meshless finite element method; Element Free Galerkin (EFG) [10] in 1994, Cloud-Based hp-Finite Element Method [11] in 1996, Partition of Unity [12] in 1997.

This study focuses on Method of External Finite Element Approximations introduced in 1991 by Apanovitch [13]. This approach inspired to have the necessary traits needed for analysing the design and the commercial software product was used to evaluate the feasibility and convenience.

# 3. THE METHOD OF EXTERNAL FINITE ELEMENT APPROXIMATIONS

There are two commercial software products that is based on The Method of External Finite Element Approximations and both of them are property of Apanovitch who is also the founder of the theory behind it.

The first software is named Procision and it is introduced by Vicror Apanovitch and Paul Kurowski in year 2000. The new approach is introduced as precise solids method, based on The Method of External Finite Element Approximations. However for unknown reasons this software is discontinued and the term ‘’ precise solids method’’ is cannot be found anywhere in the literature.

The second software is called Simsolid and introduced by Victor Apanovitch and Ken Welch. Exact date when the software is introduced is not displayed by the owners of the company but it is approximately around 2015. This estimation is based on release notes and the publication of the software. This software again based on The Method of External Finite Element Approximations and offers meshless finite element analysis.

The exact connection of these two software products is unknown. However since they both have the same theoretical background and the author of the theory is co-founder of the both software products, it is assumed that Simsolid is the successor of Procision or at least more advanced version of this its predecessor.

Because of that reason two approaches were included in this research in order to have better understanding of the theory behind it. Of course for the variation and the verification of the method only second software was used since first one was discontinued as mentioned before.

## 3.1. PRECISE SOLIDS METHOD (PSM)

The Precise Solids Method is described here from various paper Analysis Tools for Design Engineers [14] as opposing the h and p elements used in finite element method.

However by describing the new approach it is not suggested to use it to replace the old ones. Rather pros and cons are evaluated and suggested which approach is better suited in that particular condition.

For the finite element method, meshing is required in order to perform analysis. In order to give the CAD geometry mesh ability it needs to be discretized into either two geometries.

The h-method uses first or second order polynomials. However this limits the shapes of the elements and this causes discretization of the domain into large number of small elements. The element shapes are shown in Figure 3.1.

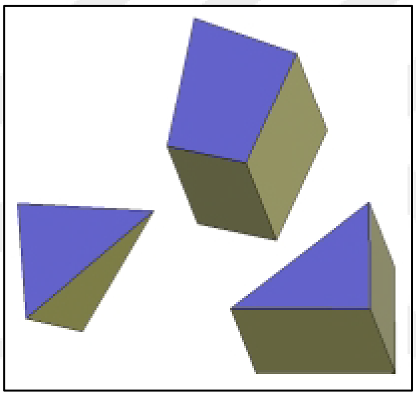

Figure 3.1. Elements of h-method [14]

The h-method is being used nearly 50 years. Another biggest downside is meshing the thin features is this method.

The other method that being used is p-method which ha higher order polynomials up to $9^{th}$ order. This gives the advantage of larger and less number of elements. Also geometry is likely to represent the real geometry better as it has more complex element shapes shown in Figure 3.2.

The p-method can cover linear analysis and also has limited non-linear capabilities. The advantage of complex shapes gives the advantage of not simplifying the geometry as much as the h-method. That means less time consumed to turn geometry into mesh with more precision.

The third method, which is the new method based on the External Finite Element Approximation Method does not need analysis specific geometry. It is meshless but of course there is still discretization. Discretized parts are not called mesh but they are subparts of the geometry that consists of 12$^{th}$ order polynomials and non-algebraic stress concentration functions. Examples of subparts of PSM are shown in Figure 3.3.

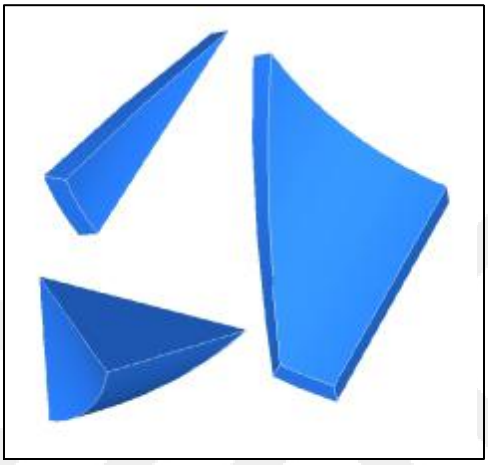

Figure 3.2. Elements of p-method [14]

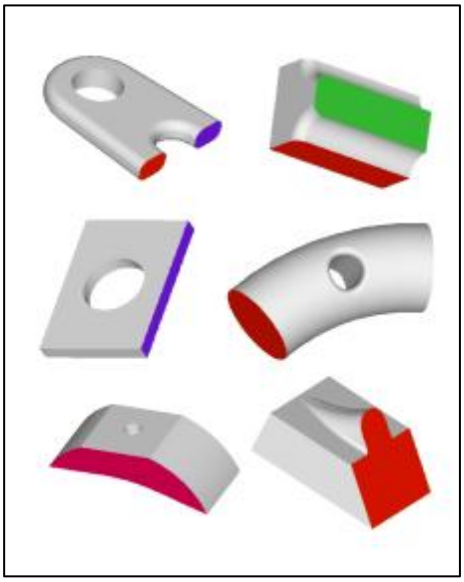

Figure 3.3. Elements of PSM [14]

The disadvantage of this method is, it has to work with solid CAD geometry and it limits the choice of modelling techniques. But of course has the advantage of not converting CAD data into analysis specific shapes.

The paper details which methods suit for the given problem;

- The h-method is suitable for analysists, not for design engineers. It is too demanding analysis tool for concurrent design analysis.
- The p-method is suitable for thin structures with shell or beam elements. Example of shell element is shown in Figure 3.4.
- PSM method is suitable for solid geometries with number of elements.  Otherwise it would take major amount of time to make the geometry meshable. But in PSM there is no need for a simplifying geometry because the assembly can be used as is. Example of PSM discretization is shown in Figure 3.5.

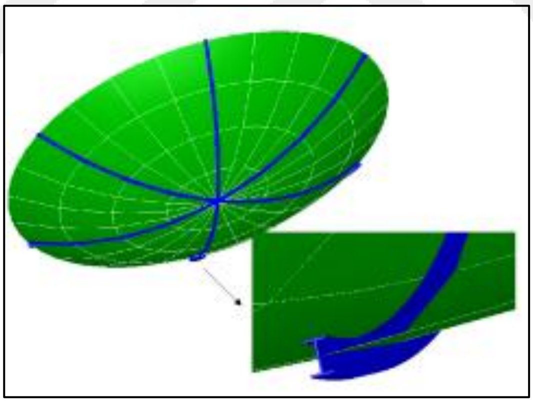

Figure 3.4. Shell element in p-method [14]

The applicability of p-method and PSM is also detailed in a graph shown in Figure 3.6 as complexity of geometry vs level of geometry idealization. If the geometry consists of solids both approaches are suitable. If solid geometry is complex then PSM is the better choice. If the assembly consists of shells of beams then p-method is the better choice.

The paper or other documents found in the literature does not give specific mathematical background for PSM other than being based on the method of external approximations.

And since the software product Procision is discontinued, its performance and feasibility is not discussed here.

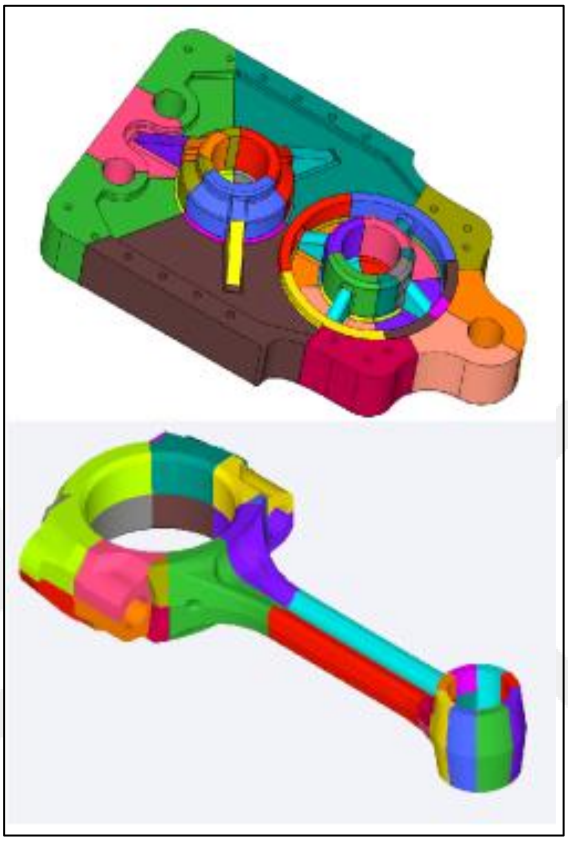

Figure 3.5. Discretized PSM elements in large assemblies [14]

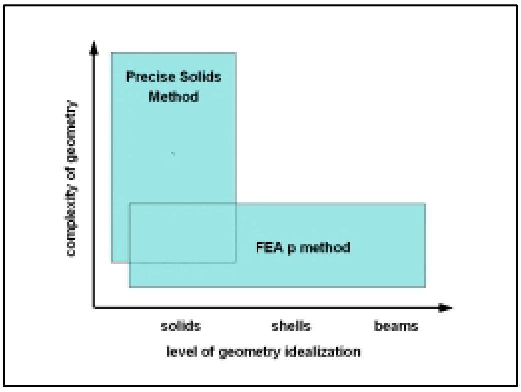


Figure 3.6. The applicability of PSM [14]

The only aspect of this approach shows that external approximations can be used in large assemblies with number of parts and claims to provide necessary convenience for concurrent design.

Also the term meshless finite element approach observed as discretizing the model not the known geometries but to simple parts. It obviously gives a huge advantage because discretization will take less time with less effort.

The results of PSM made the study continue to examine External Finite Element Method with its software product Simsolid, to decide whether to be used in console analysis.

## 3.2. EXTERNAL APPROXIMATIONS OF FINITE ELEMENTS METHOD

The following chapter is taken from the paper [15] that explains the theory behind Simsolid software. First the difference of interal and external approximations are defined. Then, traditional finite element method is briefly explained in order to clarify the difference of external approximations by finite elements.

### 3.2.1. Differences of External and the Internal Approximations

Ritz and Galerkin method assumes functions that approximate the solution are analytical functions defined on the domain. In simple cases these functions have infinite derivatives as they are trigonometric functions or polynomials. However in more complex cases like in real life problems, these functions cannot create a system with required accuracy.

In later years the Finite Element Method improved Ritz-Galerkin method by using functions with local support called finite elements. This improvement solved the problem of numerical instability. Just like before local basis functions of finite elements have infinitively differentiable polynomials but global basis functions did not have it. Their first derivatives are discontinuous.

Finite element method proved that continuity requirement of approximation functions is only necessary to create finite energy which translated to energy functional of the boundary value problem. In 1930’s the space which these functions inhabit introduced by Sobolev.

If the approximation function is in the Sobolev Space than it has finite energy and the approximation is the internal approximation. In that case the approximation is refined state. Solution is converging to exact solution. Approximations are in Sobolev Space.

However in external approximations, the approximations are not in Sobolev Space. Since they have infinite energy it does not converge to exact solution. But if the limit where the number of degrees of freedom tends to infinity, the limit function must belong to the corresponding Sobolev space.

### 3.2.2. Theory of Finite Element Method

Fomulation of abstract boundary value problem is to find a function U which fulfils equations; (A and L are differential operators)

$$\mathrm{AU} = \mathrm{f} \text{ inside a domain } \Omega \tag{3.1}$$

$$\mathrm{LU} = \mathrm{g} \text{ at the domain boundary} \tag{3.2}$$

Ritz method proposed finding approximate solution of boundary value problem by doing approximation with linear combination of basis functions.

$$U_h = \sum a_i\, p_i, \qquad i = 1,2, \ldots n \tag{3.3}$$

$a_i$ are unknown factors and $p_i$ are basis approximation functions.

Finding the mimimum value of energy functional gives the value of $a_i$

$$F\left(\sum a_i\, p_i\right) = min \tag{3.4}$$

The situation where the bonundary value problem is a linear one, minimizing the equation (3.4) gives a linear algebraic equation with respect to factors $a_i$

$$Ka = B \tag{3.5}$$

In (3.5) K is a symmetric matrix, a is vector of unknown factors and B is the right hand side.

In finite element method these symbols have physical meanings. K is the stiffness matrix, B is the load vector and $a_i$ are degrees of freedom.

Later Galerkin expanded the solution to the boundary value problem of (3.1) and (3.2). His solution the U is approximated by

$$U_n = U_0 + \sum a_i p_i, \qquad i = 1,2, \dots n \tag{3.6}$$

$U_o$ is a function that fulfills nonhomogenous the condition (3.2), $p_i$ are approximation functions that fullfils the homogenous conditions.

If (3.6) is substitued intro (3.1) result in residual

$$R = AU_0 + \sum a_i Ap_i - f \tag{3.7}$$

$a_i$ are found from equation system

$$\int_\Omega Rp_i d\Omega = 0\,, \qquad i = 1,2, \dots n \tag{3.8}$$

Similar to Ritz if boundary value problem is linear then (3.8) is a linear algebraic equations.

These methods are accepted as effective solutions to various engineering problems. However mathematical justification has become a setback and only solved after functional analysis is introduced as a dicipline.

Ritz-Galerkin method later modernized and it is based on weak solution of boundary value problem which is finding a function u ϵ V from the Sobolev Space that fulfils an abstract variational equation

$$a(u,v) = f(v)\ for\ any\ function\ v\ \in V \tag{3.9}$$

V is the subspace of Sololev Space, a(u,v) is an unsymmetrical bilinear that is continuous on the space prodcut VxV and f(v) is linear form of V.

For exapmle in structural anaylsis Sobolev Space is a space of functions which has finite strain energy. In Ritz-Galerkin method V space is approximated with $X_h$ the dimensional space and approximation solution is found with form of (3.3) where functions $p_i$ belong to space $X_h$. In conclusion the discretized formulation of BVP is find the function $U_h \epsilon X_h$ which fullfils the equation

$$a(U_h, V_h) = f(V_h) \; for \; any \; function \; V_h \; \in \; X_h \tag{3.10}$$

The substution of (3.3) into (3.10) results in linear algebraic equation and from which the unknown factors $a_i$ are found.

In the classic representation of Ritz-Galerkin $X_h$ is the space of analytical functions that defined on the domain Ω. The factors $a_i$ has no physical meaning.

In traditional finite element method $X_h$ is the space of piecewise polynomials. The factors $a_i$ are values of function $U_h$ in the nodes of finite elements. In case of structural analysis they are the displacement of the nodes.

### 3.2.3. External Approximations by Finite Elements

The distinctive feature of internal approximation is that it is built on functions that belong to Sobolev Space. That creates the necessity of continuity conditions on inter element boundaries. For example in theory of elasticity problems the function need to be continuous between finite elements.

This situation creates the restriction of using only very simple shapes as finite elements. For this reason incompatible finite elements were introduced. In order to use those elements some other polynomials are used with the interpolation basis functions of element with standard shape. The additional functions are added to create a discontinuity around inter element boundaries. The setback of this method is difficulty of mathematical proof and no consistency in results.

The external approximation has a different approach to finite element method in general. In theory finite element is only described as a sub domain inside a domain. No actual shape is detailed. If we consider the assembly as the domain, part of the assembly can be a sub domain.

Another issue of the terminology is that, approximation functions inside finite element does not have to be a polynomial. It only has to be arbitrary. The necessary condition is that it needs to belong to corresponded Sobolev Space. Only in that case it can be sufficiently smooth inside element.

The term external approximations suggests converging the exact solution of boundary value problem outside of Sobolev Space. In order to consider approach external this condition need to be satisfied as;

$$< \delta, \gamma U > = 0 \tag{3.11}$$

(3.11) shows the duality pairing of functional spaces that defined on inter element boundaries.

δ and γ are opeators and U are approximation functions which are inside element. This condition constraints limit approximation function to belong to the corresponding Sobolev Space. This will bring the necessary smoothness properties.

If the condition (3.11) implemented into inner product in other space of functions;

$$(g, \gamma U) = 0 \tag{3.12}$$

In (3.12) g is the function defined on inter element boundaries, they are the boundary functions. They are the functions of surface parameteres. They generate boundary degrees of freedom, which are the integrals of prodcuts of boundary functions onto finite element basis functions over finite element boundary;

$$\int_{\gamma} g_k \gamma U d\gamma, \qquad k = 1,2, \dots N \tag{3.13}$$

ɣ is the boundary of finite element, $g_k$ are the functions defined on the boundary of the finite element. U is the function to be approximated on the element. For expample in structural analysis they are the displacements. In traditional FEM value of degree of freedom is the function U in the node i;

$$U(x_i, y_i, z_i) \tag{3.14}$$

Functionals shown in (3.13) are the boundary degrees of freedom and they do not have a physical meaning. Their purpose is to limit the approximation functions when the degrees of freedom tend to infinity.

It is boundary degrees of freedom's responsibility to meet inter-element continuity conditions.

(3.13) is the external DOF (degrees of freedom). The internal DOF is also defined in external approximations. They are linked to finite element volume and they are defined automatically when solution of approximation is being built.

The final approximation function U is;

$$U_h = \sum a_i\,(U) p_i + \sum (\int_{\gamma} g_k \gamma U d\gamma,) p_k \qquad (3.15)$$

The elements are;

- $a_i$ is internal DOF of element
- $p_i$ is basis functions of internal DOF
- $\int_{\gamma} g_k \gamma U d\gamma,$ are boundary DOF
- $p_k$ is the basis functions of boundary DOF

The basis functions of $p_i$ and $p_k$ are not pre defined since the element is in arbitary shape. They are built during the solution.

The paper list the general algorithm for building basis functions of an element;

- A number of boundary functions $g_k$ is defined
- Space P is defined as approximation functions as set of generic basic functiıns. For example for second degree polynomials for 3D problems space of space of polynomails are;

  $$\{1, x, y, z, x^2, xy, y^2, xz, z^2, yz\}$$
- Generic basis functions are built automatically during the solution. This process happens when stiffness matrix of the sub-domain is evaluated.
- $p_i$ and $p_k$, which are the are basis functions are found by solving system of linear algebric equation.

When basis functions are found the stiffness matrix and load vector are just like in traditional FEM by integrating energy over element volume and loads over the element boundary.

Unlike PSM, this method does not give any specification of discretization of the domain in paper shown above or any other publication. The finite element can be part of an assembly or the domain itself as long as certain conditions are met. Even though an algorithm shows the reasoning behind the theorem, some operators are not defined at all. One of the reason is that some functions are defined once the 3D CAD model is inserted and built on the fly.

However the method and the software product is inspired to have the features wanted for analysis of the console. Since the mathematical solution is in the closed system certain examination was needed to move forward with the design process.

# 4. NATURAL FREQUENCIES OF A RECTANGULAR THIN PLATE

The design study focuses on analysing a complex structure that cannot be calculated analytically in preparation of a real life test. However as explained in the later chapters the real life test was discontinued for this study and new finite element approach was compared to the traditional one.

In order to have some sort of validation for both results, a single part was chosen to conduct a plate bending analysis for natural frequencies in later chapters.

This chapter focuses on formulation of plate bending problem and its finite element representation.

## 4.1. ANALYTICAL SOLUTION OF RECTANGULAR PLATE VIBRATIONS

Finding the natural frequency is explained [16] by using the balance of energy as strain and kinetic. Strain energy is derived by coupling bending and torsional moments over plate area

$$U = \frac{D}{2}\int_{-b}^{b}\int_{-a}^{a}\begin{bmatrix}\left(\frac{\partial^2 W}{\partial x^2}\right)^2 + \left(\frac{\partial^2 W}{\partial y^2}\right)^2 + 2v\left(\frac{\partial^2 W}{\partial y^2}\right)\left(\frac{\partial^2 W}{\partial y^2}\right) + \\ 2(1-v)\left(\frac{\partial^2 W}{\partial x \partial y}\right)^2\end{bmatrix} dxdy \qquad (4.1)$$

W=W(x,y) is the natural mode, D=Eh$^3$ / [12/(1-v$^2$)] is the flexural rigidiry.

E and v are Young’s Modulus and Poisson’s Ratio.

The kinetic energy is defined as

$$K = \frac{\omega^2}{2}\int_{-b}^{b}\int_{-a}^{a} mW^2\, dxdy \qquad (4.2)$$

Where m is the mass per unit area ω is the natural frequency.

Vibration mode of a rectangular plate is shown in figure below;

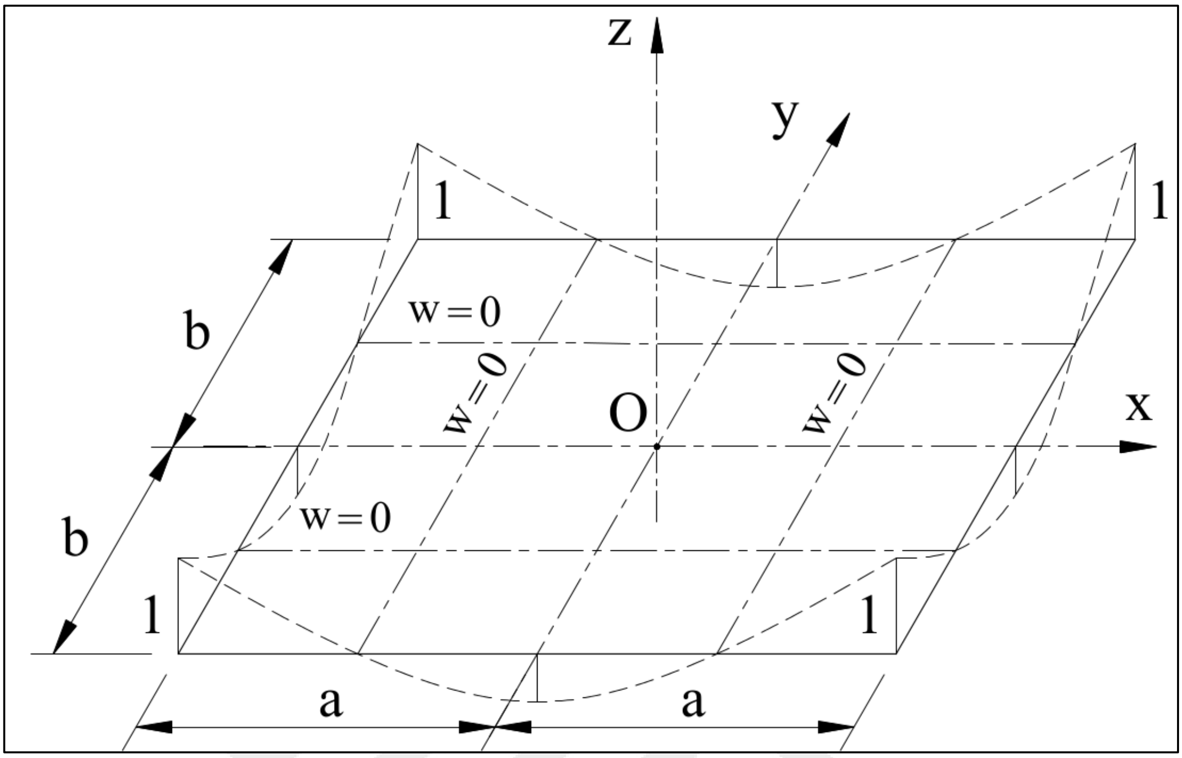


Figure 4.1. Vibration mode of a rectangular plate [16]

The seperation of variables is used for equation (4.1) as $W_{mn}(x,y) = X_m(x)Y_n(y)$. The strain energy equations becomes

$$U_{mm} = \frac{D}{2}\int_{-b}^{b}\int_{-a}^{a}\Big[(X''_m)^2{Y_n}^2 + {X_m}^2(Y''_n)^2 + 2vX''_mX_mY''_nY_n + 2(1 - v)(X'_mY'_n)^2\Big]\,dxdy \tag{4.3}$$

The integral solution is shown in [17] and it gives the simplified strain energy as

$$U_{mn} = \frac{D}{8ab}u_{mn} \tag{4.4}$$

where

$$u_{mn} = (\alpha_m^4 + \beta_n^4)a^2b^2 + 2v\alpha_m a\beta_n btha\alpha_m ath\beta_n b(1 - a_m atha_m a)(1 - \beta_n bth\beta_n b) + 2(1 - v)\alpha_m a\beta_n btha_m ath\beta_b(3 + \alpha_m atha_m a)(3 + \beta_n bth\beta_n b) \tag{4.5}$$

For uniform mass distribution, the kinetic energy can be simplified as

$$K = \frac{\omega^2 m}{2} \int_{-b}^{b} \int_{-a}^{a} X_m{}^2 Y_n{}^2 \, dxdy = \frac{1}{8} \omega^2 m \, a \, b \tag{4.6}$$

With balance of energy (U=K) natural frequency is

$$\omega^2{}_{mn} = \frac{D}{m} \frac{u_{mn}}{a^2 b^2} \tag{4.7}$$

## 4.2. FINITE ELEMENT REPRESANTATION OF PLATE BENDING FREQUENCIES

The finite element formulation of rectangular plate bending is explained in [17] and the plate geometry is shown as in the figure below;

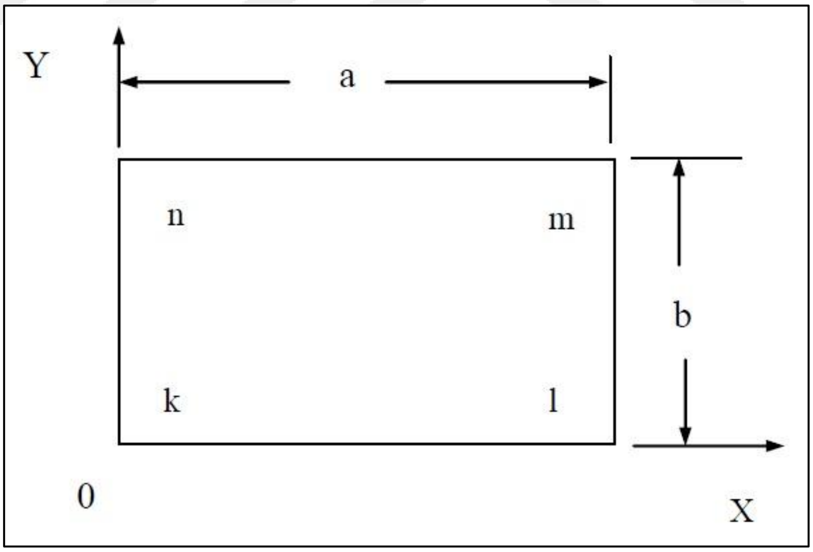


Figure 4.2. Rectangular plate [17]

u is defined as out of plane displacement.

Strain energy U is

$$U = \frac{D}{2}\int_0^b\int_0^a\left[\left(\frac{\partial^2 u}{\partial x^2}\right)^2 + \left(\frac{\partial^2 u}{\partial y^2}\right)^2 + 2\mu\left(\frac{\partial^2 u}{\partial y^2}\right)\left(\frac{\partial^2 u}{\partial y^2}\right) + 2(1 - \mu)\left(\frac{\partial^2 u}{\partial x \partial y}\right)^2\right]dxdy \quad (4.8)$$

D is defined as stiffness factor with E as elastic modulus, h as plate thickness and μ as Poisson's ratio.

$$D = \frac{Eh^3}{12(1-\mu^2)} \quad (4.9)$$

Each corner has one translational and two rotational DOF. That means there are 12 DOF in total. Rotational components are about x and y axes. The corners are named as k,l,m and n as shwon in Figure 4.2.

For example the rotational component θ at corner l are

$$\theta_{xl} = \frac{\partial u_l}{\partial x} \quad (4.10)$$

$$\theta_{yl} = \frac{\partial u_l}{\partial y} \quad (4.11)$$

The diplacement equation has 12 contraints and 12 continuity conditions

$$u(x,y,t) = a_1 + a_2x + a_3y + a_4x^2 + a_5xy + a_6y^2 + a_7x^3 + a_8x^2y + a_9xy^2 + a_{10}y^3 + a_{11}x^3y + a_{12}xy^3 \quad (4.12)$$

The displacement equation is written as

$$u(x,y,t) = \{A\}^T\{Z\} \quad (4.13)$$

Where

$$\{A\}^T = [a_1\ a_2\ a_3\ a_4\ a_5\ a_6\ a_7\ a_8\ a_9\ a_{10}\ a_{11}\ a_{12}] \tag{4.14}$$

$$\{Z\}^T = [1\ x\ y\ x^2\ xy\ y^2\ x^3\ x^2y\ xy^2\ y^3\ xy^3\ xy^3] \tag{4.15}$$

The strain energy formulation from (4.8) than becomes

$$U = \frac{D}{2}\{A\}^T\left(\int_0^b\int_0^a [S(x,y)]\,dxdy\right)\{A\} \tag{4.16}$$

Where

$$S(x,y) = \left(\frac{\partial^2 u}{\partial x^2}\right)^2 + \left(\frac{\partial^2 u}{\partial y^2}\right)^2 + 2\mu\left(\frac{\partial^2 u}{\partial y^2}\right)\left(\frac{\partial^2 u}{\partial y^2}\right) + 2(1-\mu)\left(\frac{\partial^2 u}{\partial x\partial y}\right)^2 \tag{4.17}$$

Again

$$u(x,y,t) = a_1 + a_2x + a_3y + a_4x^2 + a_5xy + a_6y^2 + a_7x^3 + a_8x^2y + a_9xy^2 + a_{10}y^3 + a_{11}x^3y + a_{12}xy^3 \tag{4.18}$$

$$\frac{\partial}{\partial x}u(x,y,t) = a_2 + 2a_4x + a_5y + 3a_7x^2 + 2a_8xy + a_9y^2 + 3a_{11}x^2y + a_{12}y^3 \tag{4.19}$$

$$\frac{\partial}{\partial y}u(x,y,t) = a_3 + a_5x + 2a_6y + a_8x^2 + 2a_9xy + 3a_{10}y^2y + a_{11}x^3 + 3a_{12}xy^2 \tag{4.20}$$

The displacements and derivatives of each corner are

$$u|_{\{0,0,t\}} = a_1 \tag{4.21}$$

$$\left.\frac{\partial u}{\partial x}\right|_{\{0,0,t\}} = a_2 \tag{4.22}$$

$$\left.\frac{\partial u}{\partial y}\right|_{\{0,0,t\}} = a_3 \tag{4.23}$$

$$u|_{\{a,0,t\}} = (a_1) + a(a_2) + a^2(a_4) + a^3(a_7) \tag{4.24}$$

$$\left.\frac{\partial u}{\partial x}\right|_{\{a,0,t\}} = (a_2) + 2a(a_4) + 3a^2(a_7) \tag{4.25}$$

$$\left.\frac{\partial u}{\partial y}\right|_{\{a,0,t\}} = (a_3) + a(a_5) + a^2(a_8) + a^3(a_{11}) \tag{4.26}$$

$$\begin{aligned} u|_{\{a,b,t\}} = {} & a_1 + a_2a + a_3b + a_4a^2 + a_5ab + a_6b^2 + a_7a^3 + a_8a^2b \\ & + a_9ab^2 + a_{10}b^3 + a_{11}a^3b + a_{12}ab^3 \end{aligned} \tag{4.27}$$

$$\begin{aligned} \left.\frac{\partial u}{\partial x}\right|_{\{a,b,t\}} = {} & (a_2) + 2a(a_4) + b(a_5) + 3a^2(a^7) + 2ab(a_8) + b^2(a_9) \\ & + 3a^2b(a_{11}) + b^3(a_{12}) \end{aligned} \tag{4.28}$$

$$\begin{aligned} \left.\frac{\partial u}{\partial y}\right|_{\{a,b,t\}} = {} & (a_3) + a(a_5) + 2b(a_6) + a^2(a^8) + 2ab(a_9) + 3b^2(a_{10}) \\ & + a^3(a_{11}) + 3ab^2(a_{12}) \end{aligned} \tag{4.29}$$

$$u|_{\{0,b,t\}} = (a_1) + (a_3)b + (a_6)b^2 + (a_{10})b^3 \tag{4.30}$$

$$\left.\frac{\partial u}{\partial x}\right|_{\{0,b,t\}} = (a_2) + (a_5)b + (a_9)b^2 + (a_{12})b^3 \tag{4.31}$$

$$\left.\frac{\partial u}{\partial y}\right|_{\{0,b,t\}} = (a_3) + 2b(a_6) + 3b^2(a_{10}) \tag{4.32}$$

The displacement vector for element i

$$\{u\}_i{}^T = \left[u_k\ \theta_{xk}\ \theta_{yk}\ u_l\ \theta_{xl}\ \theta_{yl}\ u_m\ \theta_{xm}\ \theta_{ym}\ u_n\ \theta_{xn}\ \theta_{yn}\right] \tag{4.33}$$

The displacement vector is written as

$$\{u\}_i = [B]\{A\}_i \tag{4.34}$$

where

$$[B] = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & a & 0 & a^2 & 0 & 0 & a^3 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 2a & 0 & 0 & 3a^3 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & a & 0 & 0 & a^2 & 0 & 0 & a^3 & 0 \\ 1 & a & b & a^2 & ab & b^2 & a^3 & a^2b & ab^2 & b^3 & a^3b & ab^3 \\ 0 & 1 & 0 & 2a & b & 0 & 3a^2 & 2ab & b^2 & 0 & 3a^2b & b^3 \\ 0 & 0 & 1 & 0 & a & 2b & 0 & a^2 & 2ab & 3b^2 & a^3 & 3ab^2 \\ 1 & 0 & b & 0 & 0 & b^2 & 0 & 0 & 0 & b^3 & 0 & 0 \\ 0 & 1 & 0 & 0 & b & 0 & 0 & 0 & b^2 & 0 & 0 & b^3 \\ 0 & 0 & 1 & 0 & 0 & 2b & 0 & 0 & 0 & 3b^2 & 0 & 0 \end{bmatrix} \tag{4.35}$$

Solving for A

$$\{A\}_i = [B]^{-1}\{u\}_i \tag{4.36}$$

Let

$$[c] = [B]^{-1} \tag{4.37}$$

$$\{A\}_i = [c]\,\{u\}_i \tag{4.38}$$

The strain energy formulation of (4.16) than becomes

$$U = \frac{D}{2}\{u\}_i{}^T[c]^T\left(\int_0^b\int_0^a [S(x,y)]\,dxdy\right)[c]\{u\}_i \tag{4.39}$$

The kinetic energy becomes

$$K = \frac{\rho h}{2}\int_0^b\int_0^a \dot{u}^2\,dxdy \tag{4.40}$$

Recall the displacement

$$u(x,y,t) = \{A\}^T\{Z\} \tag{4.41}$$

Then velocity is

$$\dot{u} = \{\dot{A}\}^T\{Z\} \tag{4.42}$$

$$[\dot{u}]^2 = \{\dot{A}\}^T\{Z\}\{Z\}^T\{\dot{A}\} \tag{4.43}$$

Recall

$$\{A\} = [c]\,\{u\}_i \tag{4.44}$$

$$[\dot{u}]^2 = \{\dot{u}\}_i{}^T[c]^T\{Z\}\{Z\}^T[c]\{\dot{u}\}_i \tag{4.45}$$

Let

$$P(x,y) = \{Z\}\{Z\}^T \tag{4.46}$$

$$[\dot{u}]^2 = \{\dot{u}\}_i{}^T[c]^TP(x,y)[c]\{\dot{u}\}_i \tag{4.47}$$

The kinetic energy becomes

$$K = \frac{\rho h}{2}\{\dot{u}\}_i{}^T[c]^T\int_0^b\int_0^a P(x,y)\,dxdy[c]\{\dot{u}\}_i \tag{4.48}$$

Work due to the nodal forces and moment

$$W = \{F\}_i{}^T\{u\}_i = \{u\}_i{}^T\{F\}_i \tag{4.49}$$

where

$$\{F\}_i{}^T = \left[F_k\ M_{xk}\ M_{yk}\ F_l\ M_{xl}\ M_{yl}\ F_m\ M_{xm}\ M_{ym}\ F_n\ M_{xn}\ M_{yn}\right] \tag{4.50}$$

Hamilton’s principle yields to the equation of motion

$$[m]\{\ddot{u}\}_i + [k]\{u\}_i = \{F\}_i \tag{4.51}$$

For undamped free vibration u can be written in harmonic solution form with Φ as eigenvector (mode shape) and ω as natural frequency

$$\{u\} = \{\Phi\}\sin\omega t \tag{4.52}$$

For homogenous equation 4.51 than becomes

$$-\omega^2[m]\{\Phi\}\sin\omega t + [k]\{\Phi\}\sin\omega t = 0 \tag{4.53}$$

Local mass matrix m and stiffness matrix k are given as

$$[m]_i = \rho h[c]^T\int_0^b\int_0^a[P(x,y)]dxdy[c] \tag{4.54}$$

$$[k]_i = D[c]^T\int_0^b\int_0^a[S(x,y)]dxdy[c] \tag{4.55}$$

Assembled stiffness and mass matricies are found as

$$[M] = \sum[m]_i \tag{4.56}$$

$$[K] = \sum [k]_i \tag{4.57}$$

The generalized eigenvalue problem becomes

$$\{[K] - \omega^{2}[M]\}\{\Phi\} = 0 \tag{4.58}$$

- K is the global stifness matrix
- M is the global mass matrix
- ω is the natural frequency
- $\Phi$ is the corresponding mode shape (eigenvector)

# 5. NUMERICAL RESULTS AND FEM VALIDATION

The previous chapter shows background information about the alternative FEM approach. However the commercial software of this finite element method SIMSOLID, is not an open source system. That's why in order to use it for console's MIL-STD test analysis verification for its accuracy was tested.

The reason this part of study was initiated is to test the new approach's performance and accuracy with experiment data. The guideline followed was Guide for Verification and Validation in Computational Solid Mechanics from ASME PTC 60 / V&V 10 shown below;

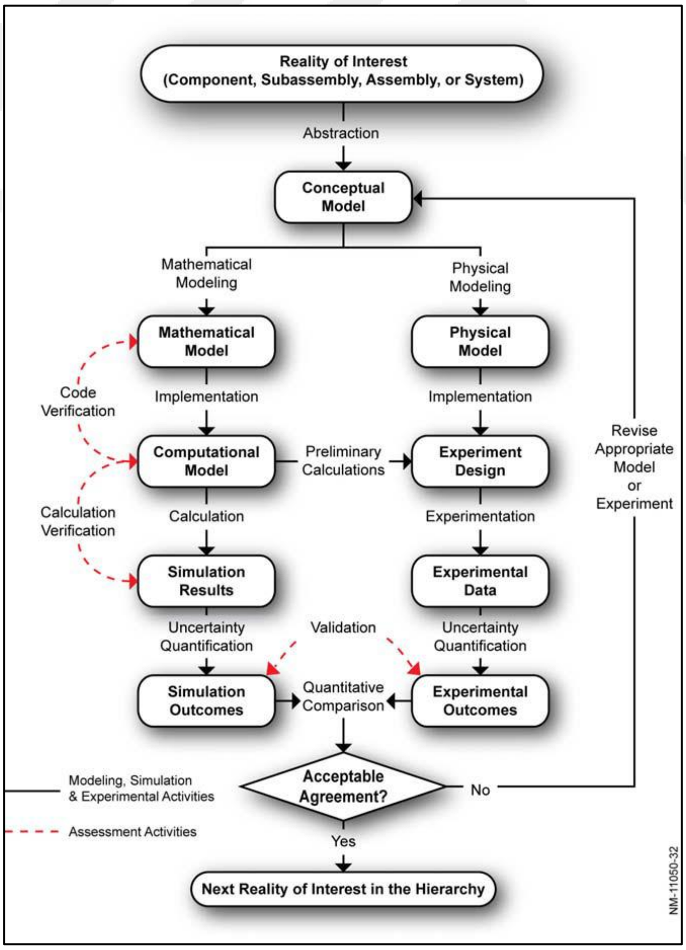


Figure 5.1. Verification & Validation activities and outcomes [18]

As we can see there are two objectives with three steps shown in red in the Figure 5.1

- Verification
    - Code Verification; mathematical model and solution algorithms are accurate
    - Calculation Verification; discrete solution and mathematical model is accurate
- Validation; model is accurate representation of the real world counterpart.

As explained in the beginning of the chapter, code verification cannot be implemented. That’s why two experiments were planned;

- An experiment where the ‘’simulation results’’ can be calculated mathematically for calculation verification.
- A more complex experiment where the ‘’simulation outcomes’’ cannot be calculated mathematically but can be compared with experimental outcomes for the validation.

The final decision for the experiments are;

- NAFEMS Benchmark Test where the experiment outcomes are known and used for various finite element software to test validity.
- WECDIS Console Prototype Test for Shock and Vibration featured in MIL-STD-810G and MIL-STD-167-1.

However for certain reasons the real life test for the console is discontinued and computational analysis was decided instead. Because of that reason the validation test translated to traditional FEM vs external FEM from experiment outcome vs external FEM. An example of real life test is shown in Figure 5.2. and an example of the vibration test of MIL-STD-167-1 is shown in Appendix A since this test couldn’t be a part of this study.


Figure 5.2. Example of ship vibration test [19]

## 5.1. NAFEMS BENCMARK TEST

NAFEMS (National Agency for Finite Element Methods) was found in 1983 in order to validate analysis softwares and grading them in the process. The agency demonstrates real life experiments and make calculations on different fields of engineering backgrounds.

The tests and results are shared online to use in various purposes. A NAFEMS benchmark test was choosen instead of real life experiment. The test model and its dimensions are shown in Figure 5.3.

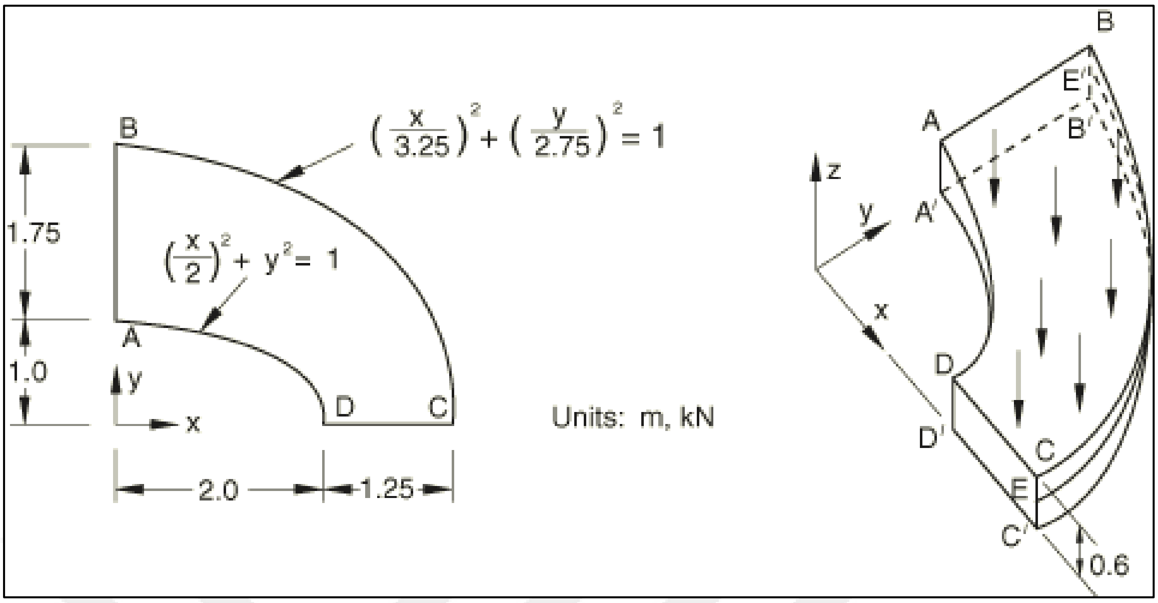


Figure 5.3. NAFEMS LE10 test model dimensions [20]

The test details are;

- Material properties; Young's modulus = 210 Gpa, Poisson's ratio = 0.3, density = 7800kg/$m^3$, linear elastic.
- Boundary conditions; $u_y$=0 on face DCD'C', $u_x$=0 on face ABA'B', $u_x$=$u_y$=0 on face BCB'C', $u_z$=0 on line EE'.
- Loading; Uniform normal pressure of 1 MPa on the upper surface of the plate.

In order to understand the results better, a three way comparison was initiated;

- The reference solution
- Traditional FEM with Solidworks Simulation, the software (Solidworks CAD) which 3D model is also designed.
- External FEM with Simsolid.

Also the traditional FEM was calculated 3 times as coarse, medium and high mesh to compare with the meshless external FEM.

The mesh details are compared in Table 5.1 and meshed models are shown in Figures 5.4-5.6.

Table 5.1. Mesh density chart

| | **Coarse** | **Medium** | **High** |
|---|---|---|---|
| Element Size | 0.296825 m | 0.148413 m | 0.0742063 m |
| Total Node | 1766 | 10876 | 75547 |
| Total Element | 998 | 6951 | 52014 |

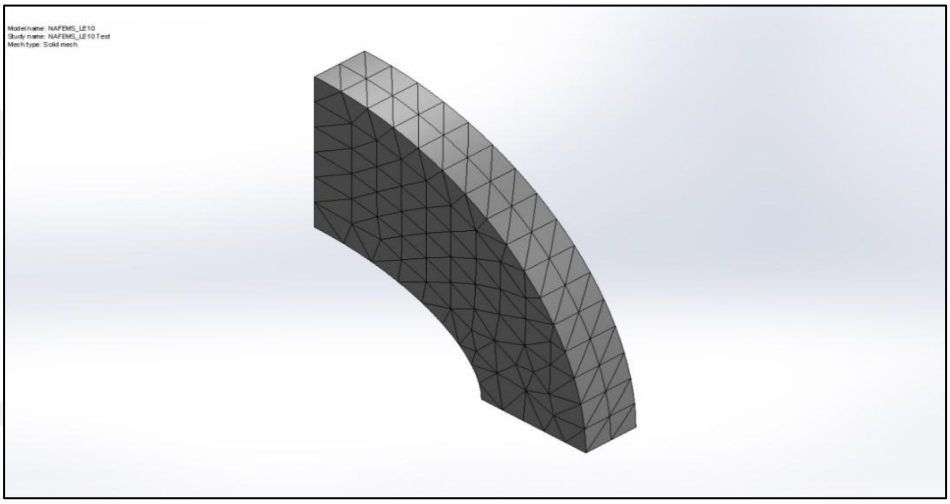


Figure 5.4. Coarse mesh density

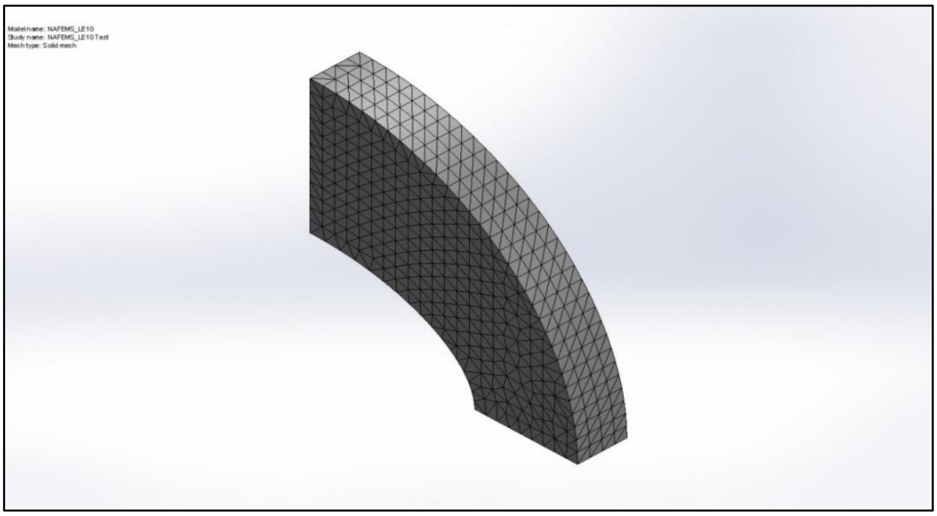


Figure 5.5. Medium mesh density

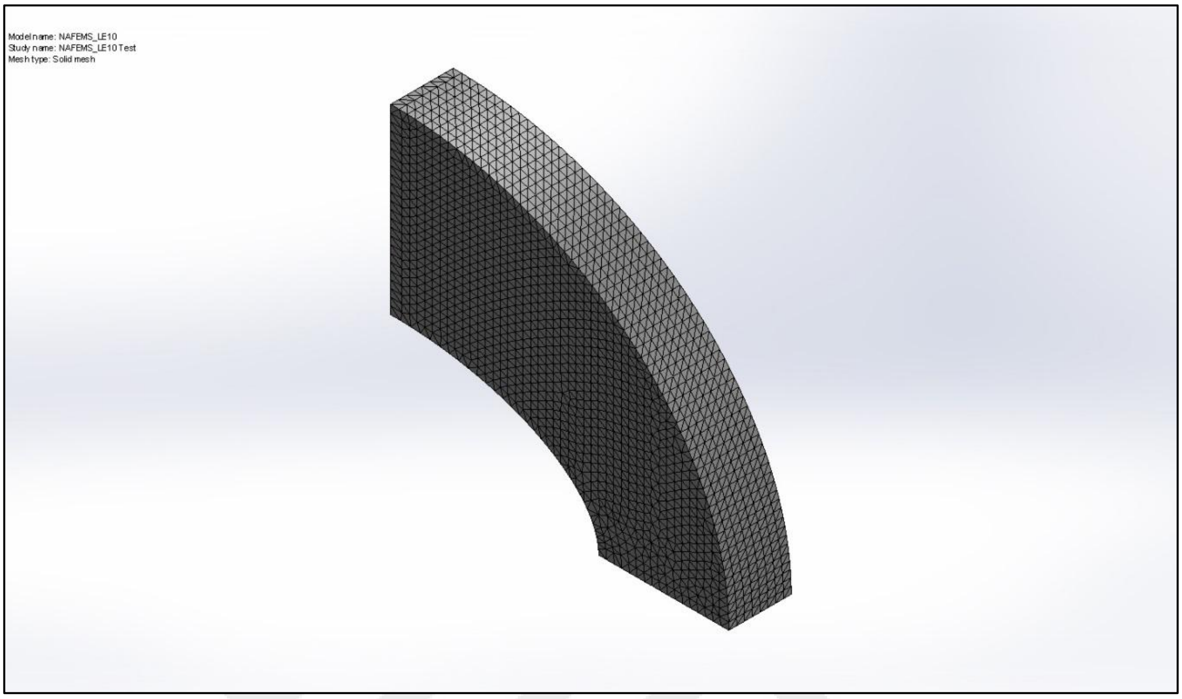


Figure 5.6. High mesh density

The results are calculated for four different scenarios. The goal as the test defined is to find the stress at point D in y direction under given boundary conditions with uniform pressure on the given axis. Compared results are shown in Table 5.2, stress distribution for each analysis are also shown in Figures 5.7-5.10.

Table 5.2. NAFEMS Benchmark test results

| | **Syy in point D (MPA)** | **Difference %** |
|---|---|---|
| Reference Solution | -5,380 | - |
| SIMSOLID | -5,341 | 0,72 |
| Solidworks-Coarse Mesh Density | -5,677 | -5,52 |
| Solidworks-Medium Mesh Density | -5,470 | -1,67 |
| Solidworks-High Mesh Density | -5,379 | 0,02 |

From the result of the analysis two arguments can be made;

- When compared to the reference results, external FEM has 0.72 per cent error which can be viewed as a success on majority of cases.
- When compared to the traditional FEM, only high density mesh could outperformed the external FEM. In other words, external FEM does not give most accurate results only if the traditional method maximize the mesh density.

Performance wise there is not much to compare since the model is only one part and has basic geometry.

For this study in particular, ASME guideline’s calculation verification is a success.

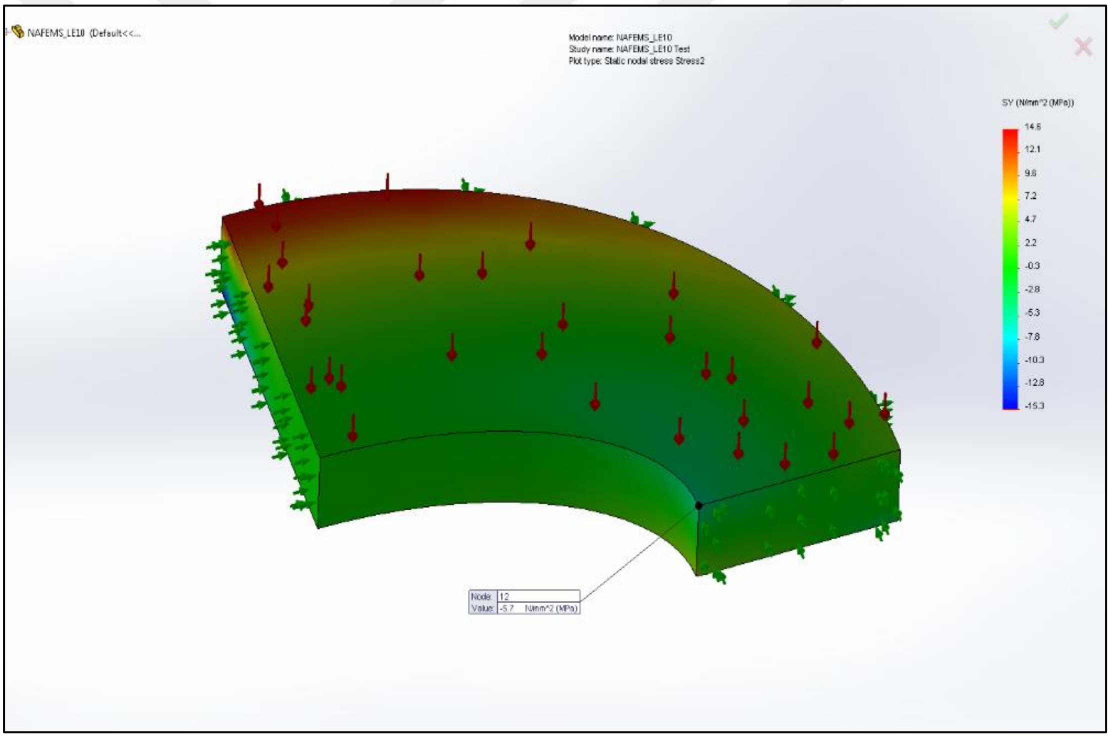


Figure 5.7. Coarse mesh density analysis result

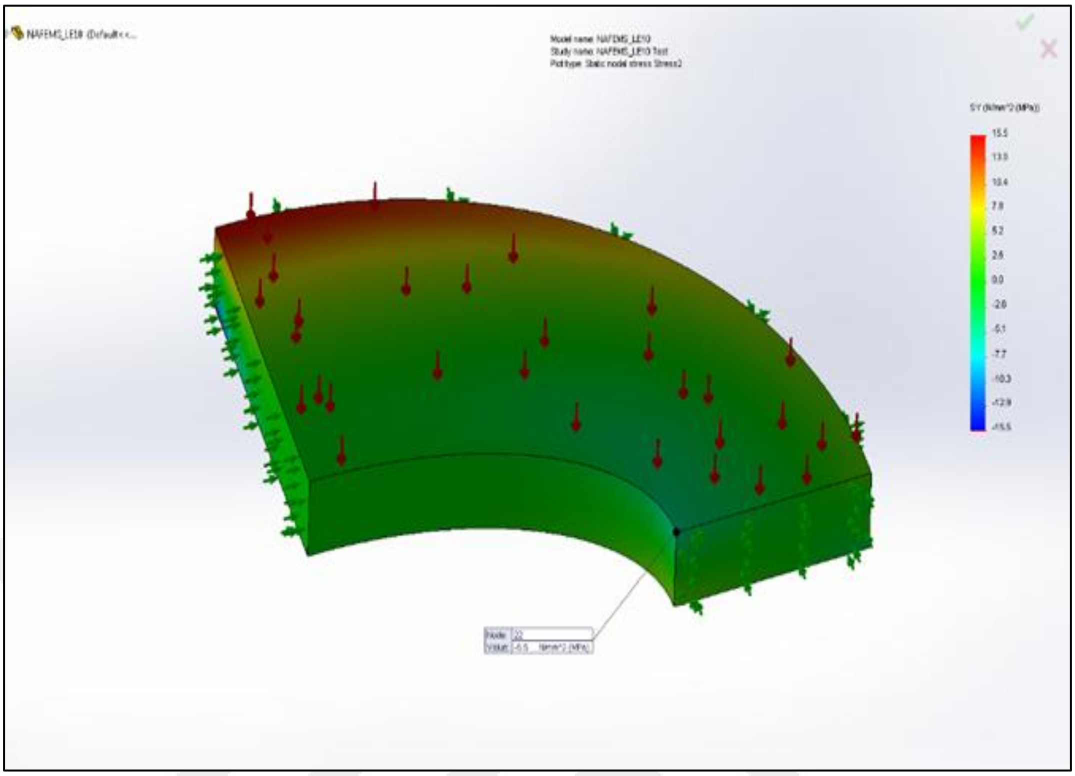


Figure 5.8. Medium mesh density analysis result

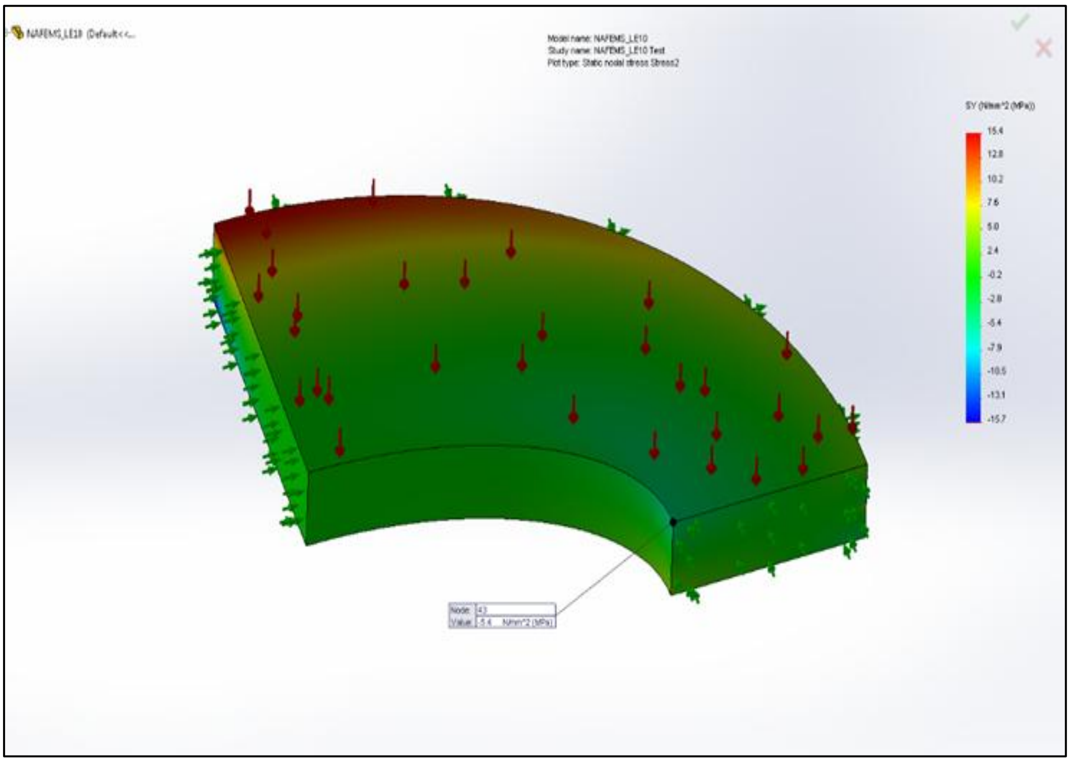


Figure 5.9. High mesh density analysis result

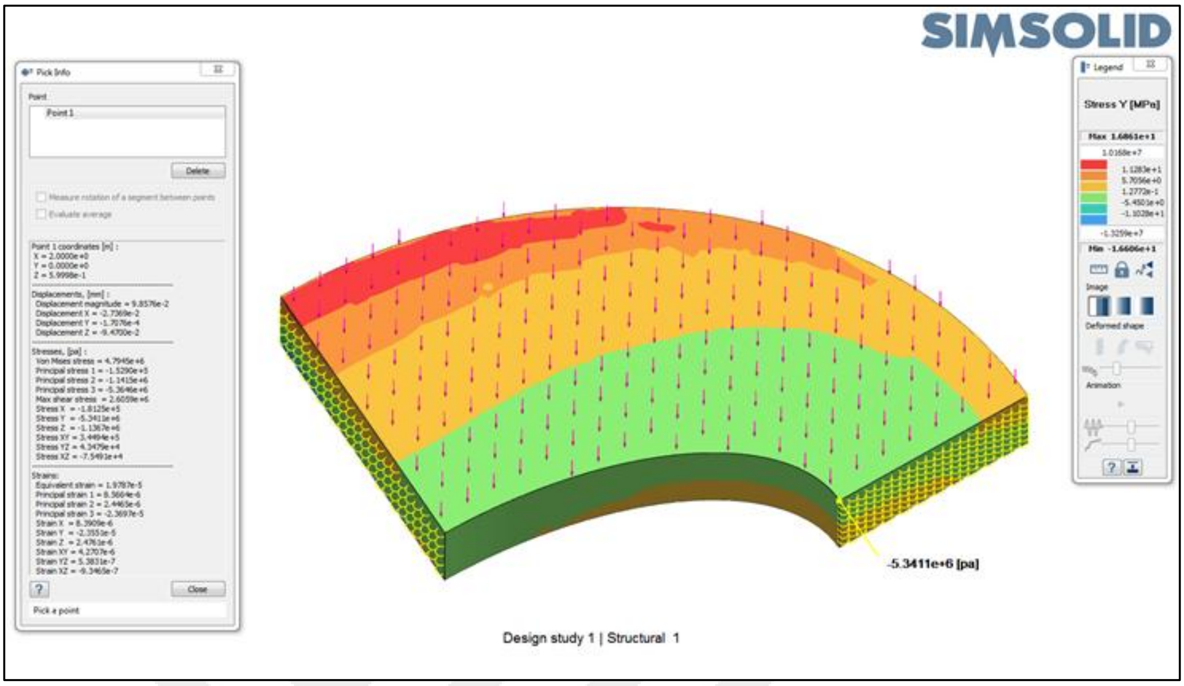


Figure 5.10. Meshless analysis result

## 5.2. WECDIS CONSOLE PROTOTYPE TEST

The process of vibration test begins with finding the natural frequencies of the device as explained in the document MIL-STD-167-1. The device will be exposed to vibration between 4-50 Hz on the naval environment, if the device has natural frequencies between those numbers it would be exposed to endurance test for two hours. If not the vibration test would be performed on the most critical natural frequency.

The necessary optimization is;

- The device’s first natural frequency should be above 50 Hz.
- Naval environment requires the device as light as possible.

That makes the conclusion of first natural frequency should be above 50 Hz but not an exaggeration value since that makes strength of the device is unnecessarily high and probably heavier than required weight.

Simsolid software has three modes; static, modal and thermal analysis. There is no dynamic analysis mode and this issue is discussed in later chapters. However nodal analysis mode can calculate natural frequencies which is the first part of the vibration test procedure.

In conclusion vibration and shock test procedures cannot be simulated on this software. But in order to test external finite element method with multipart complex assembly, the validation procedure of ASME guideline was tested on the nodal analysis of the console prototype.

3D model of the device was designed on Solidworks CAD shown in Figure 5.11.

Figure 5.11. Console CAD model render

Traditional FEM analysis was performed on ANSYS. The pre analysis preparations took nearly 9 hours and solving took extra 2 hours with distributed solver with 3 computers. Meshed model, natural frequency values and its graphics are shown in Figures 5.12-5.13 and Table 5.3.

Another side of the analysis was performed on SIMSOLID without mesh. A single computer with average specs was used and it took 10 minutes for making preparations and solving the problem. Advantages of this analysis are;

Figure 5.12. Console meshed model

Table 5.3. ANSYS Analysis natural frequencies

| **Mode** | **Frequency [Hz]** |
|---|---|
| 1 | 55.734 |
| 2 | 72.443 |
| 3 | 90.622 |
| 4 | 92.991 |
| 5 | 97.472 |
| 6 | 104.43 |

- Material properties and connections are already given since compatible CAD program was used.

- Global contact was chosen because the assembly was already ‘’mated’’.
- Only additional step was defining the rigid parts with their weight since they are COTS (commercially off the shelf) devices and their internal deformation was not the concern, their effect on the system was.

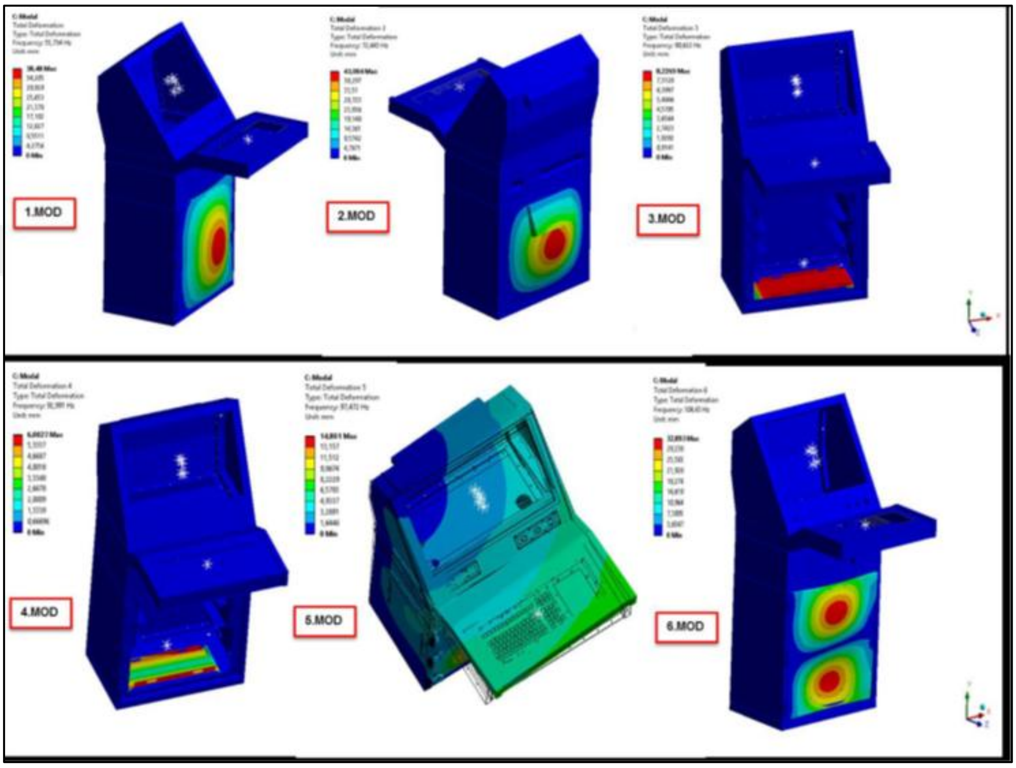


Figure 5.13. ANSYS natural frequencies graphics

Other than those inputs, required number of nodes was entered and constraints were defined as shown in Figure 5.14. Natural frequencies and its graphics are also shown in Figure 5.15 and Table 5.4.

The graphics outputs were also given to compare the modes with each other in the form of their participation for each mode in each axis. These data were categorized as;

- Modal participation factor; the scalar value that measures the interaction between modes and the directional excitation in the given reference frame. Larger the values stronger the contribution.

- Effective mass; the amount of system mass participating in that mode with given direction. They were given as percent of the total system mass. In case of systems general response, mode with the larger effective mass is more significant.

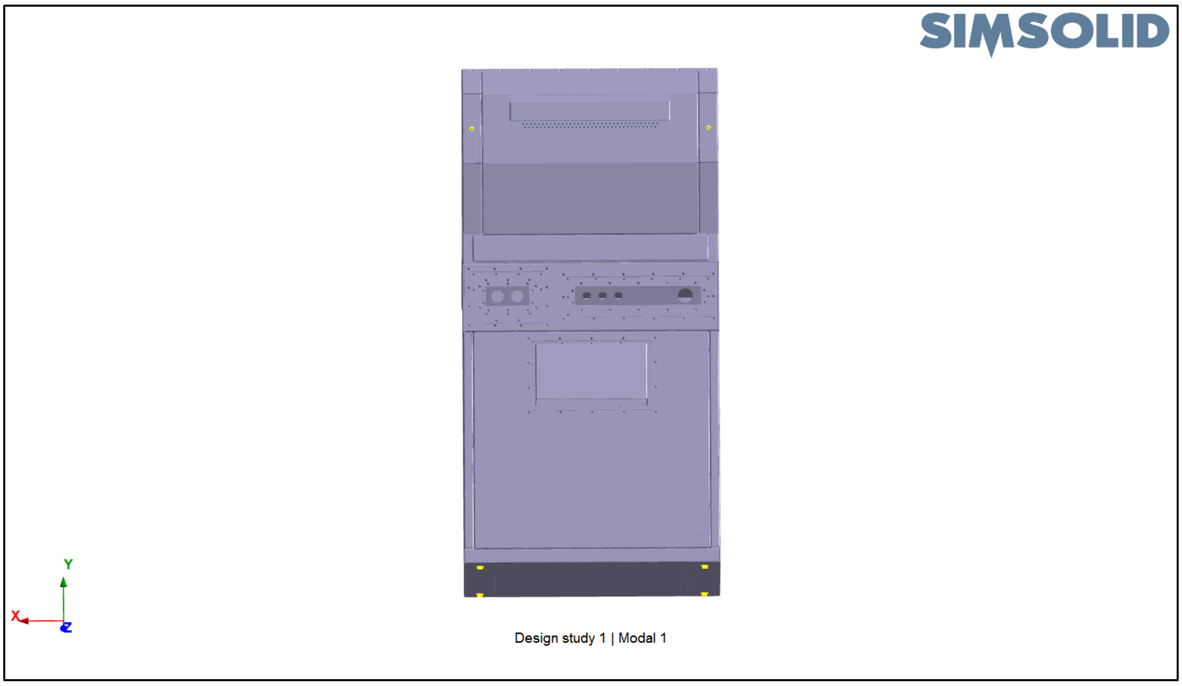


Figure 5.14. SIMSOLID model constraints

Table 5.4. SIMSOLID Analysis natural frequencies

| Mode | Frequency [Hz] |
|---|---|
| 1 | 53.74 |
| 2 | 74.20 |
| 3 | 83.08 |
| 4 | 92.14 |
| 5 | 102.96 |
| 6 | 118.74 |

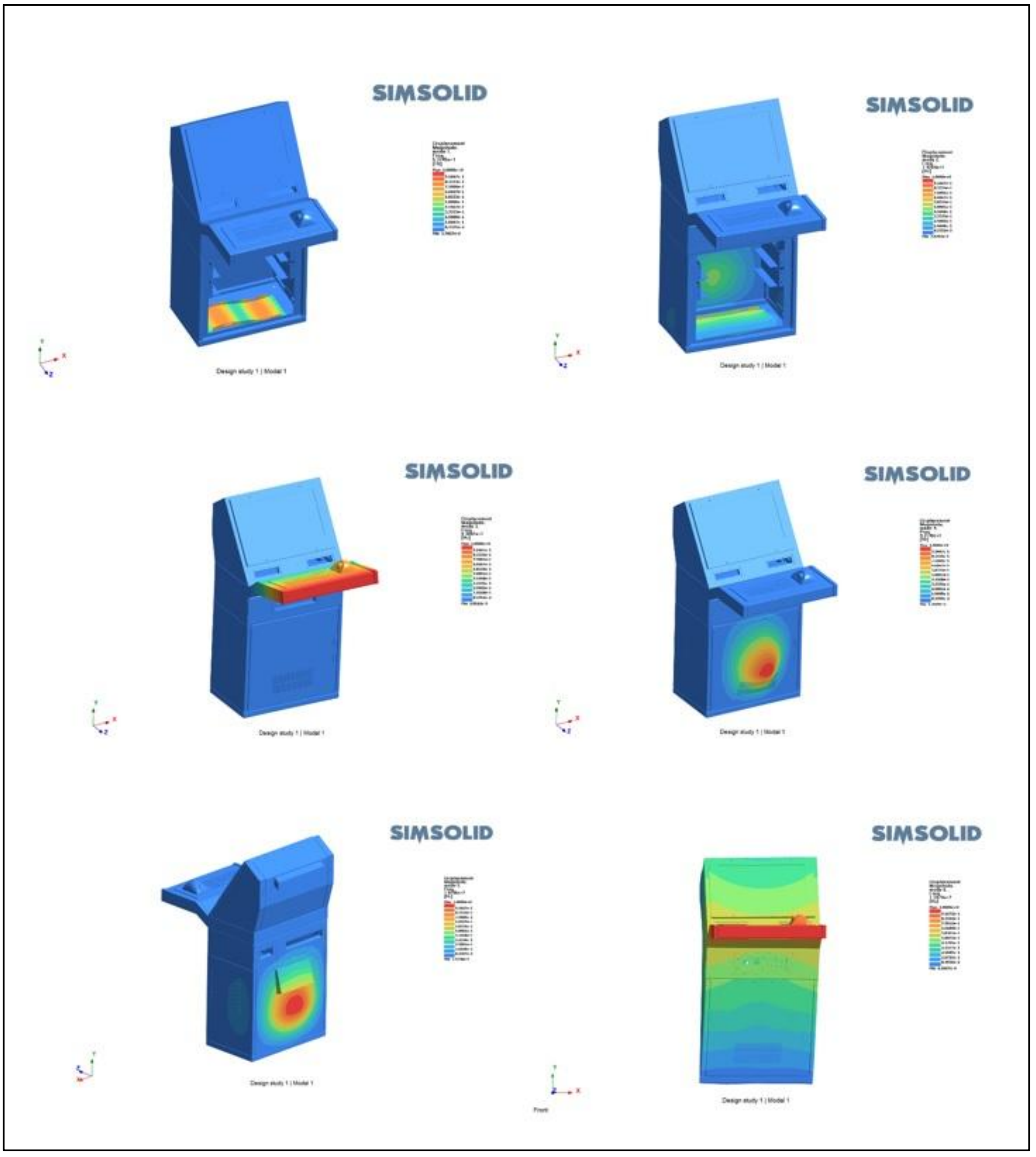

Figure 5.15. SIMSOLID natural frequencies graphics for first 6 modes from left to right

- Cumulative mass; for mode n is the sum of all effective mass from mode 1 to mode n. This feature is generally used to decide how many modes needs to be calculated in order to analyse necessary percentage of the system. It is generally 80 per cent of the cumulative mass is sufficient to be accepted as whole system vibration study.

The system's X, Y and Z directions are given in Figure 5.16.

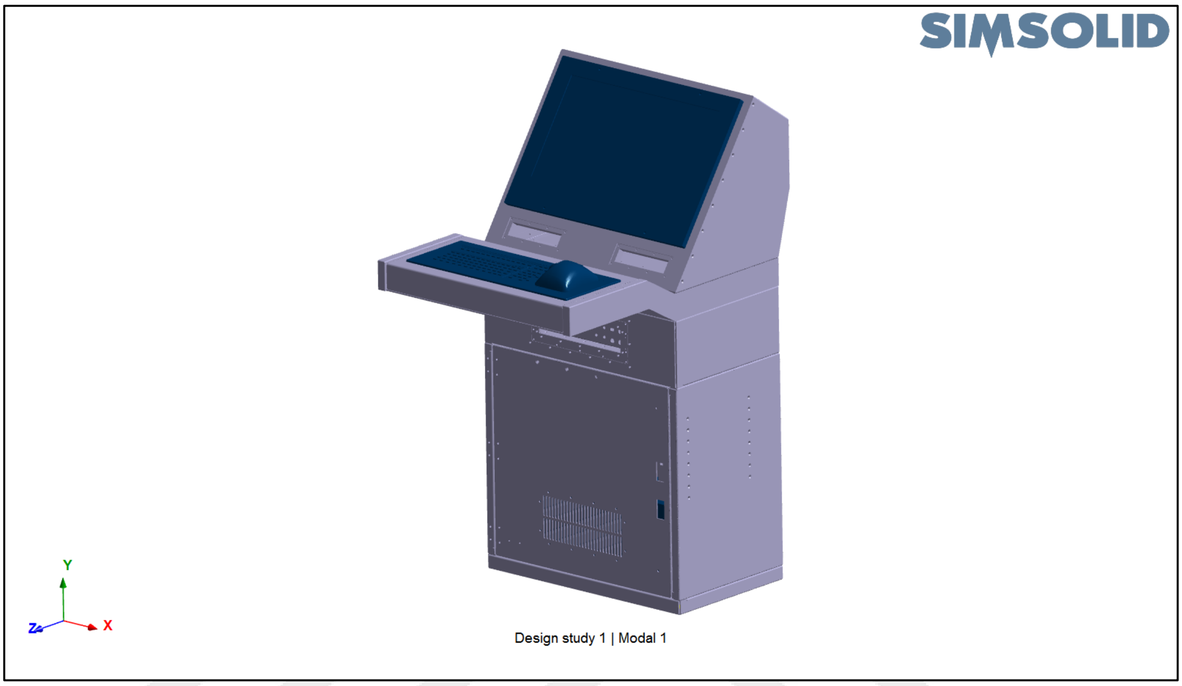


Figure 5.16. Three axis of the system

The modal participation factors, effective mass and cumulative mass tables for the three axis of the system are shown in the Figures 5.17-5.19

Critical areas of the system were determined as;

- Mode 6 for X direction. Whole system but especially the desktop is affected.
- Mode 3 for Y direction, only the desktop is affected.
- Mode 2 and 5 for Z direction, only the back panel is affected.

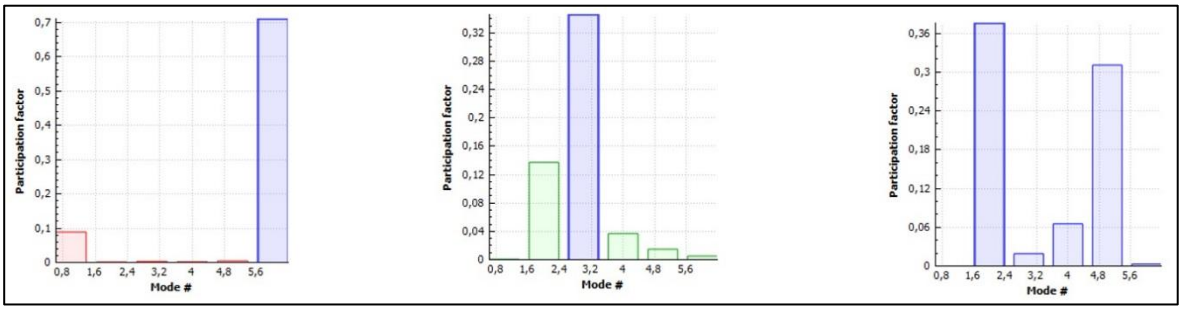


Figure 5.17. Modal participation factor for six modes in X, Y and Z axis from left to right

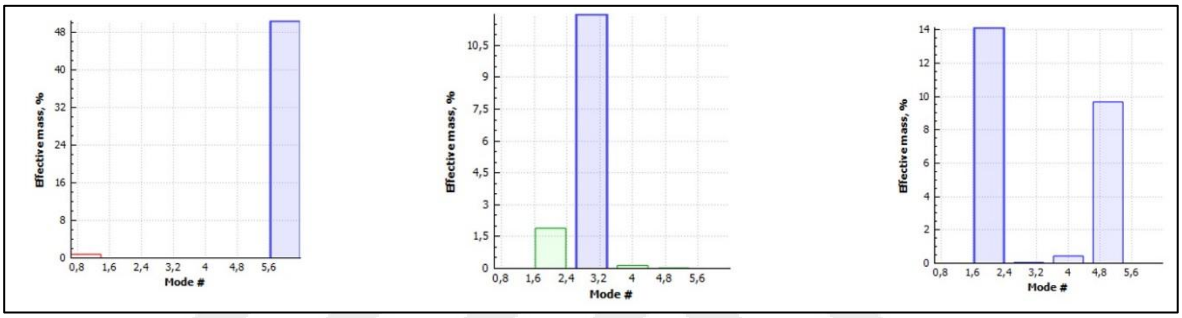


Figure 5.18. Effective mass for six modes in X, Y and Z axis from left to right

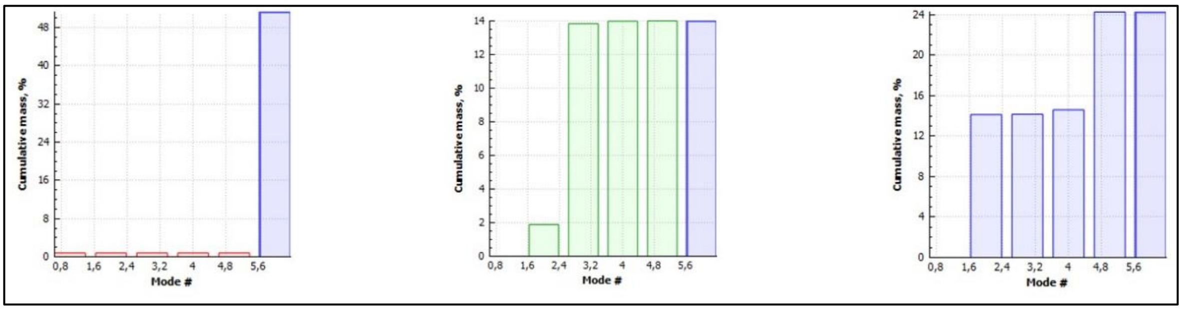


Figure 5.19. Cumulative mass factor for six modes in X, Y and Z from axis from left to right

## 5.3. REFERENCE SOLUTION FOR CONSOLE TEST

There are no real life experiment results to evaluate the console test outcomes as mentioned previously. Unlike NAFEMS test there are no benchmark results as well, since it is an original design project.

It is also impossible analytically formulate a 3D CAD assembly. However a single part can be analysed as known geometry in FEM terminology. A single part of the assembly was chosen for this study. The part can be seen in Figure 5.11 as the cabinet door beneath the desktop. The special conditions of this part are;

- It is the only part in the assembly that does not connect the other ones directly.
- It holds its position only with the defined boundary conditions.
- Rest of the assembly can be viewed as single piece beam and this part is a single plate.

There are also some expectancy for this particular comparison of two analogies;

- Both single plate and 3D part would have same length, width and thickness and also same material properties. However certain cut-outs on the 3D model makes the part more complex and due to that difference it is expected to have slightly different results.
- Mode shapes are expected to have a similarity. For plate bending problem only first and second mode need to have this trait since the part only had 2 modes in traditional FEM and 1 in external FEM. However since which mode shape appears in which mode is unclear first 6 modes were calculated.

The plate has 50 cm witdth and depth and has 3 mm thickness. It was discretized for 100 nodes and 81 elements as shown in Figure 5.20.

The FEM formulation in the previous chapter was used to evaluate first 6 modes of plate bending freqeuncies with material properties 70.3 GPa Young's Modulus, 2.67 g/cm$^3$ mass density and 0.33 Poisson's Ratio.

The boundary conditions are fixed as was in the FEM software solution. Results are shown in Figure 5.21 and Table 5.5.

Remarks for the analysis results are;

- The mode shapes are compatible with software solutions.
- First mode shape of the plate bending appears in mode 1 of ANSYS result and second mode shape appears in mode 6 of ANSYS result. Comparison is shown in Figure 5.22.

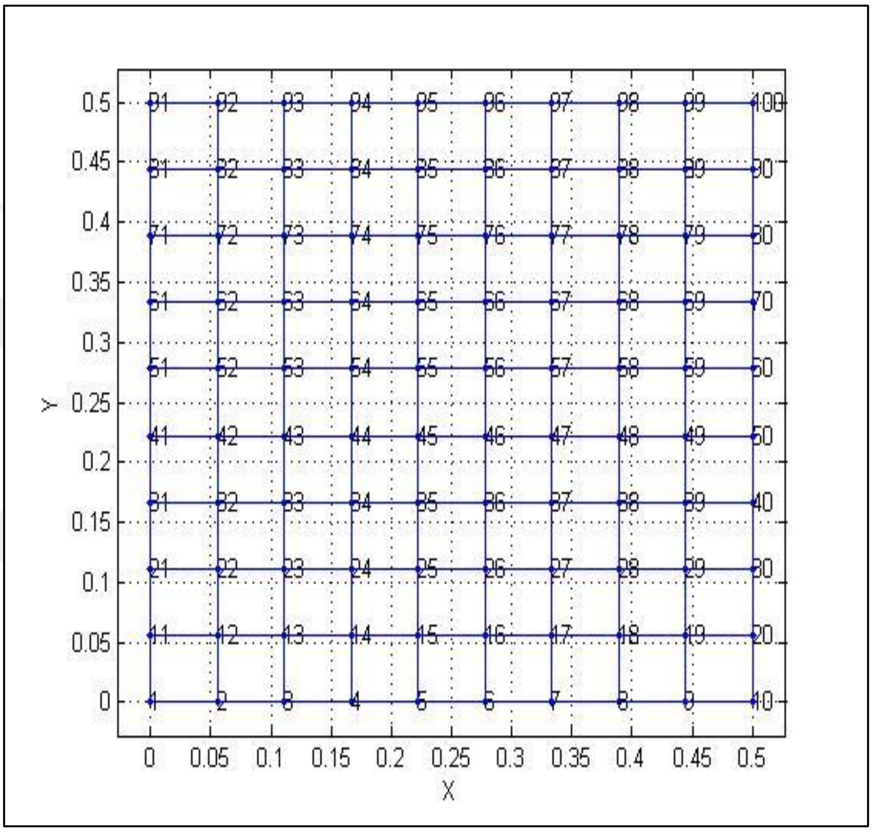


Figure 5.20. 100 Nodes and 81 elements discretized for 0.5 x 0.5 meter plate

- The frequency values are different but is it expected to be in that condition. First mode has 5 per cent difference and second mode has up to 40 per cent difference between each other. It was expected due to complexity of the 3D model.
- SIMSOLID solution has only one mode shape compatible with the plate bending problem. Mode 4 of this analysis has the same mode shape as the two other solutions but its frequency is twice the size of them.

The results can be interpreted as valid in case of mode shapes and their order as well. Frequency values are not compatible with each other for the nature of their interpretation but even then there is a certain pattern for their compatibility to prove its correctness.

Table 5.5. Plate bending analysis natural frequencies

| **Mode** | **Frequency [Hz]** |
|---|---|
| 1 | 58.77 |
| 2 | 146.40 |
| 3 | 146.40 |
| 4 | 230.80 |
| 5 | 292.7 |
| 6 | 292.7 |

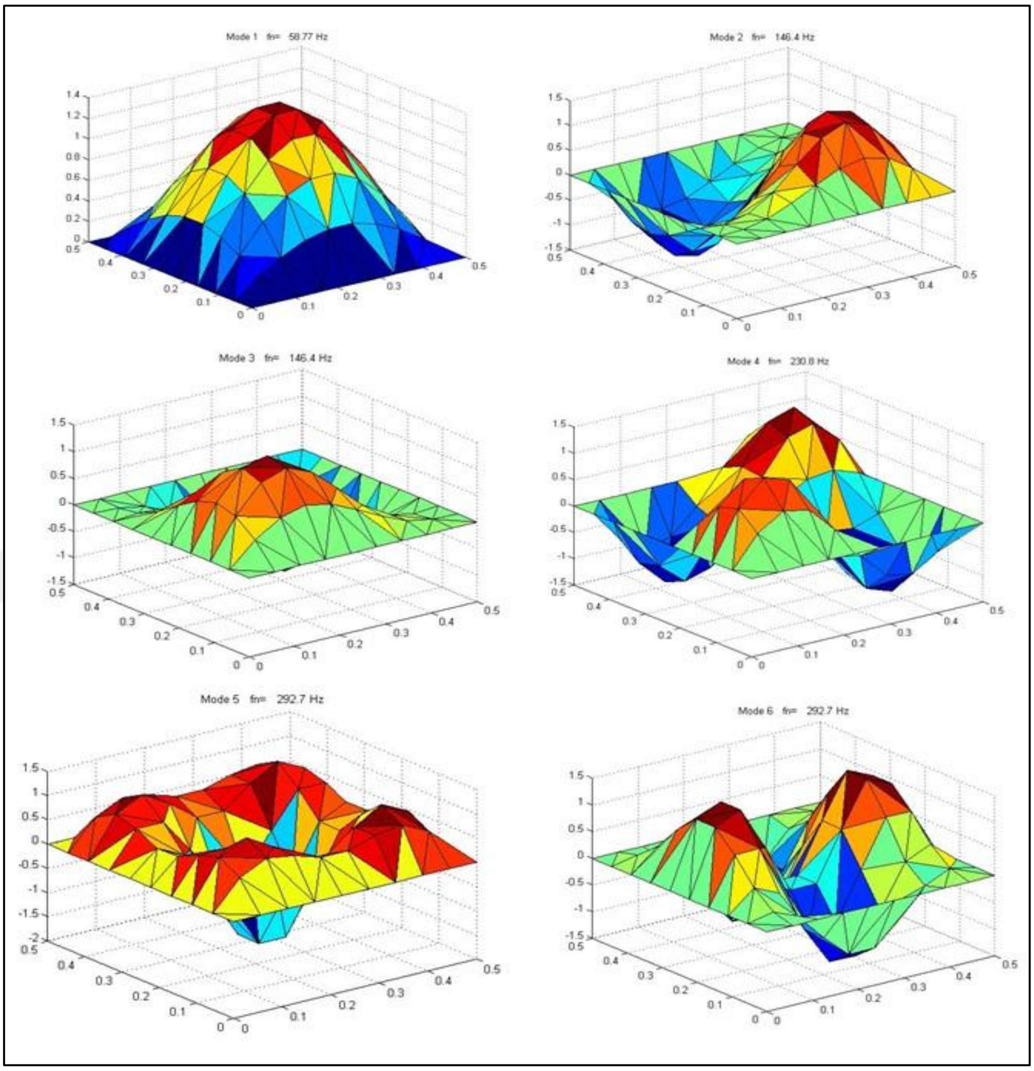


Figure 5.21. First six modes for 0.5 x 0.5 meter plate from left to right

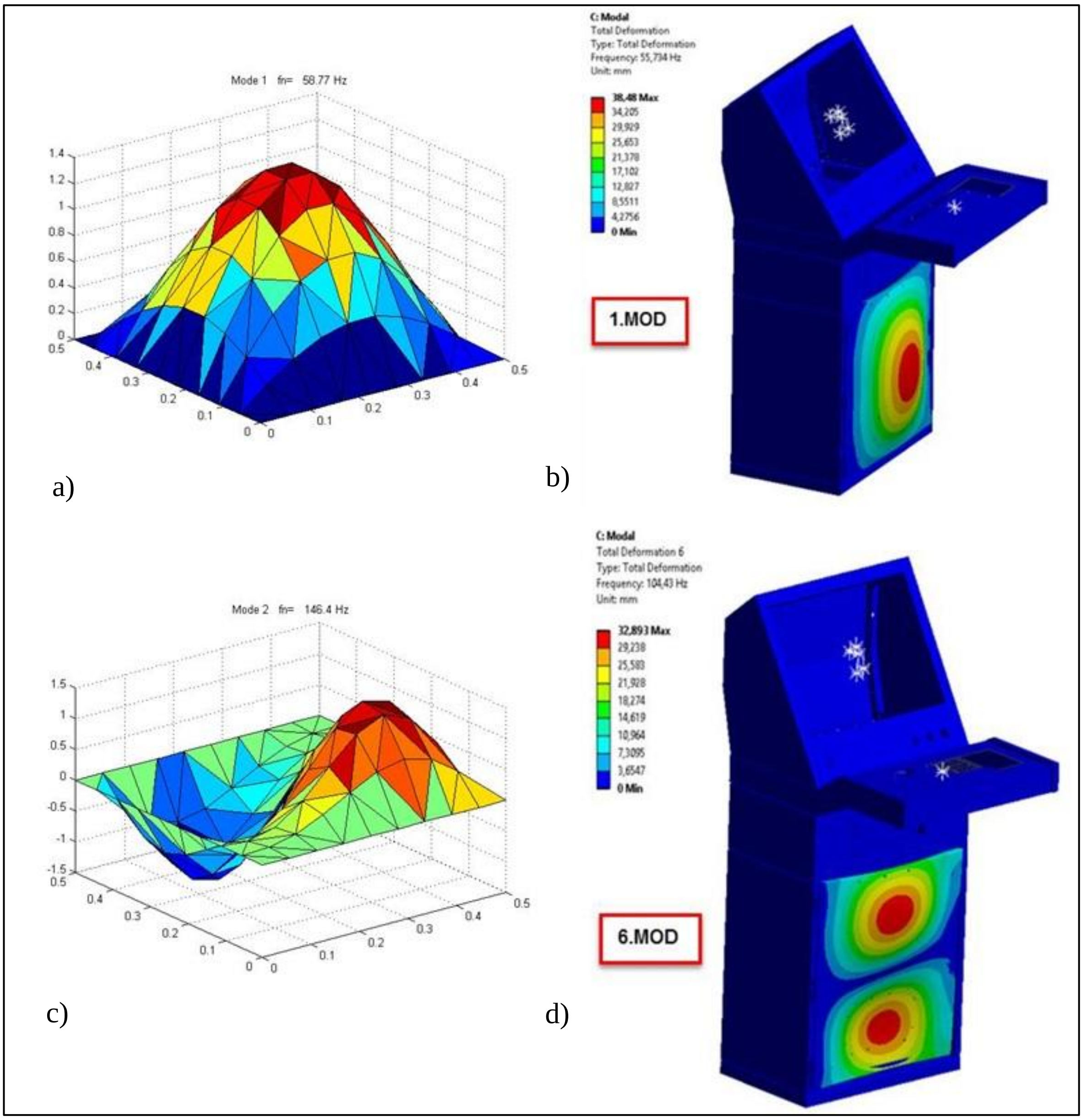


Figure 5.22. Mode shape comparison of plate bending problem and 3D FEM results. a) mode 1 of plate bending b) mode 1 of ANSYS c) mode 2 of plate bending d) mode 6 of ANSYS

# 6. DISCUSSION

In order to compare different outputs from different analysis, only comparable ones are chosen. These are modal participation factors, cumulative mass and effective mass. Modal participation factor represents biggest participation mode as 1 and rate the other ones according to that. Outputs of the two different analyses for three axis are shown in Table 6.1.

Table 6.1. Participation factor for all directions comparison chart

| | ANSYS | | | | | SIMSOLID | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| X DIRECTION | MODE | FREQUENCY | RATIO | EFFECTIVE MASS % | CUMULATIVE MASS % | MODE | FREQUENCY | RATIO | EFFECTIVE MASS % | CUMULATIVE MASS % |
| | 1 | 55,7342 | 0,01 | 0,00274 | 0,0027 | 1 | 53,74 | 0,12 | 0,79 | 0,79 |
| | 2 | 72,4427 | 0,02 | 0,01670 | 0,019 | 2 | 74,2 | 0,0019 | 0 | 0,79 |
| | 3 | 90,6216 | 0,90 | 22,09 | 22,11 | 3 | 83,08 | 0,0047 | 0 | 0,79 |
| | 4 | 92,9909 | 0,24 | 1,52 | 23,63 | 4 | 92,14 | 0,0026 | 0 | 0,79 |
| | 5 | 97,4716 | 1 | 27,31 | 50,94 | 5 | 102,96 | 0,0062 | 0 | 0,79 |
| | 6 | 104,431 | 0,50 | 6,72 | 57,66 | 6 | 118,74 | 1 | 50,48 | 51,27 |
| Y DIRECTION | MODE | FREQUENCY | RATIO | EFFECTIVE MASS % | CUMULATIVE MASS % | MODE | FREQUENCY | RATIO | EFFECTIVE MASS % | CUMULATIVE MASS % |
| | 1 | 55,7342 | 0,01 | 0,000017 | 0,000017 | 1 | 53,74 | 0,002 | 0 | 0 |
| | 2 | 72,4427 | 0,10 | 0,0032 | 0,0032 | 2 | 74,2 | 0,40 | 1,89 | 1,89 |
| | 3 | 90,6216 | 0,15 | 0,0068 | 0,010 | 3 | 83,08 | 1 | 11,96 | 13,85 |
| | 4 | 92,9909 | 0,15 | 0,0066 | 0,017 | 4 | 92,14 | 0,11 | 0,14 | 13,99 |
| | 5 | 97,4716 | 1 | 0,28 | 0,29 | 5 | 102,96 | 0,04 | 0,02 | 14,01 |
| | 6 | 104,431 | 0,03 | 0,00025 | 0,29 | 6 | 118,74 | 0,01 | 0 | 14,01 |
| Z DIRECTION | MODE | FREQUENCY | RATIO | EFFECTIVE MASS % | CUMULATIVE MASS % | MODE | FREQUENCY | RATIO | EFFECTIVE MASS % | CUMULATIVE MASS % |
| | 1 | 55,7342 | 0,46 | 2,99 | 2,99 | 1 | 53,74 | 0,0002 | 0 | 0 |
| | 2 | 72,4427 | 0,47 | 3,09 | 6,08 | 2 | 74,2 | 1 | 14,12 | 14,12 |
| | 3 | 90,6216 | 0,28 | 1,08 | 7,16 | 3 | 83,08 | 0,05 | 0,04 | 14,16 |
| | 4 | 92,9909 | 1 | 14,07 | 21,23 | 4 | 92,14 | 0,17 | 0,43 | 14,59 |
| | 5 | 97,4716 | 0,77 | 8,24 | 29,47 | 5 | 102,96 | 0,83 | 9,67 | 24,26 |
| | 6 | 104,431 | 0,11 | 0,18 | 29,65 | 6 | 118,74 | 0,0081 | 0 | 24,26 |

From the graphics of the two analyses and also from Table 6.1, a number of inferences are made;

- From the MIL-STD-167-1 point of view, the test is either pass or fail. If there are no natural frequencies below 50 Hz, the device does not need to enter into the endurance test. Both analyses give it a pass on that regard and the device continues into vibration test at critical natural frequencies.
- The ASME guideline's validation examination was observed on this test. As mentioned before accepting traditional FEM as a reference point, the device acts a similar pattern has ability to withstand below 50 Hz frequency. On that regard validation is a success.
- Deformed shapes that are parts of the system are very similar in all modes.
- For X direction mode 3 and 5 of ANSYS calculation has nearly same effective mass of SIMSOLID calculation's mode 6. Also mode shape of ANSYS mode 5 and SIMSOLID mode 6 have much similarities with 20 per cent frequency value difference shown in Figure 6.1.

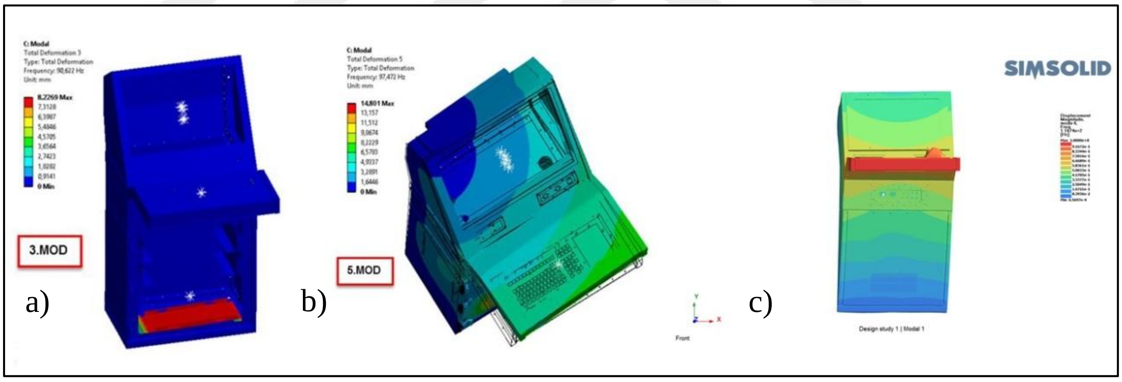


Figure 6.1. Critical modes for X direction for both analyses. a) mode 3 of ANSYS b) mode 5 of ANSYS c) mode 6 of SIMSOLID

- For Y direction mode 5 of ANSYS solution and mode 3 of SIMSOLID solution has similar mode shapes with 16 per cent differences of frequency values. But there is a huge difference of effective mass for both analysis shown in Figure 6.2.
- For Z direction mode 4 of ANSYS solution and mode 2 of SIMSOLID solution has similar mode shapes with 25 per cent difference of frequency value. However effective mass of both analyses are nearly identical as shown in Figure 6.3

- Parts where maximum deformation occurs are similar but happens on different modes.

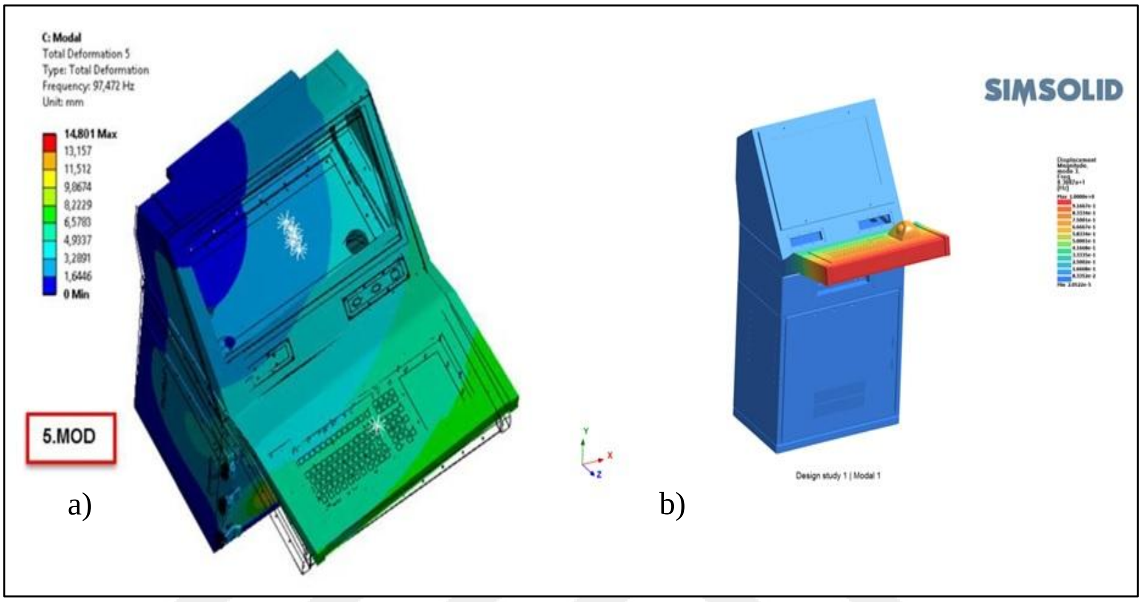


Figure 6.2. Critical modes for Y direction for both analyses. a) mode 5 of ANSYS b) mode 3 of SIMSOLID

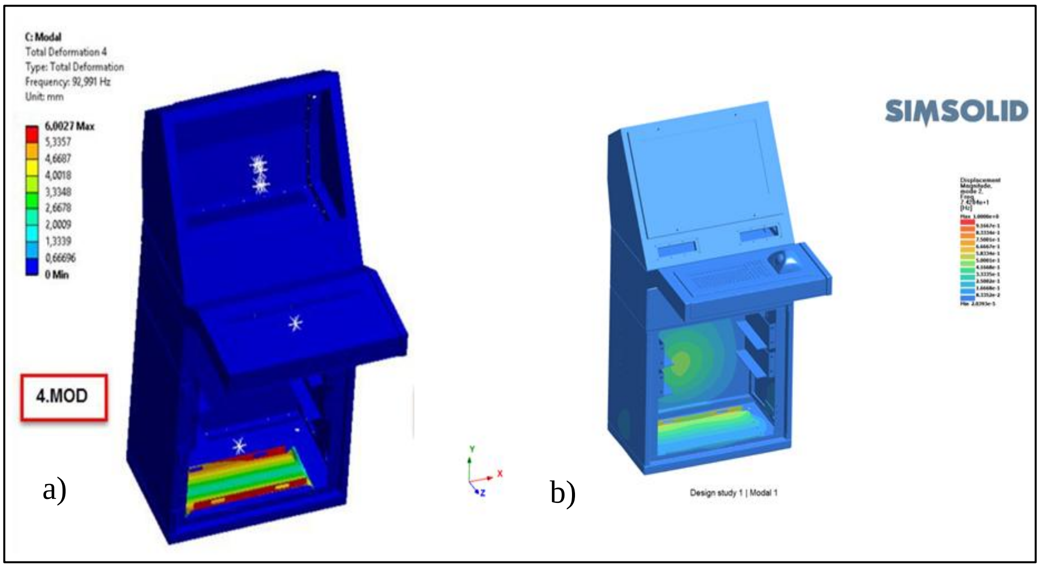


Figure 6.3. Critical modes for Z direction for both analyses. a) mode 4 for ANSYS b) mode 2 for SIMSOLID

On performance wise there is a considerable difference shown in Tables 6.2-6.3

Table 6.2. PC specs

| PC Specs | |
|---|---|
| **Ansys** | **Simsolid** |
| 128GB DDR3 RAM | 16GB DDR3 RAM |
| Intel Xeon 3.70 Ghz 6 Cores 12 threads | Intel Core i7-3770 3.40 Ghz 4 cores 8 threads |
| SSD Disk | HDD Disk |
| Distributed Solver | N/A |

Table 6.3. Analysis performance comparison

| | **Analysis Span (minutes)** | | **Comments** |
|---|---|---|---|
| | **Ansys** | **Simsolid** | |
| Simplify Geometry | 360 | 0 | Compatible CAD software only |
| Create Mesh | 150 | 0 | N/A |
| Setup Analysis | 15 | 10 | Node number, constraint, rigid parts entered Material, connections, contacts already given |
| Solve | 120 | 0,33 | N/A |
| Interpret Results | 20 | 20 | N/A |
| Sum= | 665 | 30,33 | App. 22 times less |

- PC specs show that the amount of RAM and processing power the traditional FEM requires. It usually requires number of workstations working together as distributed solver. However external FEM can operate on a single common PC.
- Simplifying geometry is required to create mesh easier thus reducing meshing time. If compatible software is used as CAD software this process disappears as well as time requires for meshing.
- Since geometry does not require simplification, analysis can be used thought the design project effortlessly. Unlike traditional method shown in Figure 6.4.

The introduction chapter mentions the budget and man-hour required for testing and analysing on these types of projects. Also it is very important not to forget the analysis is constant throughout the design process. If proven successful external finite element method can have a great impact on reducing prototype construction and project time spans.

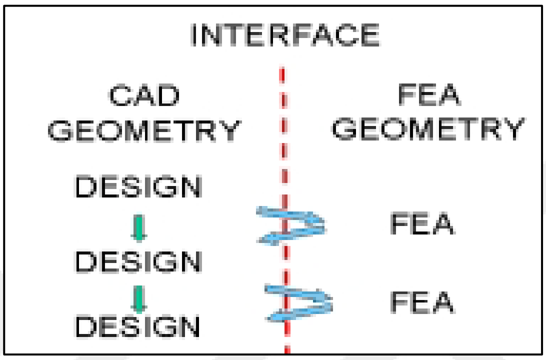


Figure 6.4. CAD and FEA geometry transition in traditional FEA [14]

# 7. CONCLUSION

This study was initiated to decide whether a different approach of FEM which is claimed to be more practical, feasible in a design project like mil-grade naval console design. In that regard the answer can be positive if certain conditions are met.

In the time of this study there is no dynamic study available other than nodal analysis. So the shock and vibration simulations of MIL-STD test were not performed with external approximations of FEM. However the nodal analysis which results in finding the natural frequencies are successfully found with accuracy as well. If the design problem requires in what value the natural frequencies occur, than this method is recommended. The console prototype’s manufacture was completed when it passed the shock and vibration tests and on that regard external approximations of FEM was validated with precondition of vibration test.

Figure 7.1. WECDIS console prototype

In case of NAFEMS test the method is shown to be effective in simple and single part analysis. The accuracy is well received as well with only 0.72 per cent error and supressing the medium and coarse mesh of the traditional FEM approach.

The performance of the commercial software is proven to be most effective result in this study. For concurrent design or optimization projects it can create a huge advantage over cost and man hour for the project.

The disadvantage of external FEM approach is that its results can be unreliable at certain times. As explained before using functions outside of Sobolev Space can create incompatible finite elements and it is difficult to mathematically prove its accuracy. External method claims to have some sort of correction measure for this problem but because of lack of information this study cannot deny or confirm those claims.

Since the formulation is created on the fly with CAD geometry it is advised to test the software with additional geometries and assemblies. Because only reference is the CAD geometry itself and experiment results can be cross referenced to decide to use this method with planned projects.

In conclusion this method can be useful for analysists who work on simple geometries and design engineers who work on similar projects and confirmed accuracy of this method before with additional software or experimental results. For design engineers who work on different assemblies with shapes and sizes and also analysing other than static, nodal and thermal this method can be seen as ineffective.

# REFERENCES


1. MIL-STD-810G, Department of defense test method for environmental engineering considerations and laboratory tests. US department of defense; 31 Oct 2008.

2. MIL-STD-108E, Military standard definitions of and basic requierements for enclosures for electric and electronic equipment. US department of defense; 4 August 1966.

3. MIL-STD-167-1(SHIPS), Mechanical vibrations of shipboard equipment. Naval ship systems command department of the navy; 1 May 1974.

4. ISO 3744, Acoustics determination of sound power levels of noise sources using sound pressure-Engineering method in an essentially free field over a reflecting plane. Türk standartları enstitüsü; 07 April 1977.

5. MIL-STD-461F, Requirements for the control of electromagnetic interference characteristics of subsystems and equipment. US department of defense; 10 December 2007.

6. Ritz W. Über eine neue methode zur lösung gewisser variationsprobleme der mathematischen physik. *Journal für die reine und angewandte mathematik*. 1909;135:1-61.

7. Galerkin B.G. Series solution of some problems in elastic equilibrium of rods and plates. *Vestnik Inzh*. 1915;19:897-908.

8. Trefftz E. Ein gegenst¨uck zum ritzschen verfahren. *Proc 2nd int cong appl mech*. 1926;24:131-137.

9. Jirousek J. Basis for development of large finite elements locally satisfying all field equations. *Computer methods in appllied mechanics and engineering*. 1978;14:65-92.

10. Belytschko T, Organ D, Krongauz Y. A coupled finite elemnt-element free Galerkin Method. *Computational mechanics*. 1995;17:186-195.

11. Oden J, Duarte C, Zienkiewicz O. A New cloud-based hp-finite element method. 1996.

12. Babuska I, Melenk JM. The partition of unity method. *International Journal for Numerical Methods in Engineering*. 1997;40:727-758.

13. Apanovitch V. *The method of external finite element approximations* Minsk: Вышэйшая школа; 1991.

14. Kurowski P. Analysis Tools for Design Engineers. *Society of Automotive Engineers Inc.* 2001;235:1-6.

15. Simsolid Technology Overview. SIMSOLID Corporation. 2015.

16. Senjanović I, Tomić M, Vladimir N, Hadžić N. An Analytical Solution to Free Rectangular Plate. *Transaciton of Famena*. 2016;40:1-18.

17. Irvine T. Plate bending frequencies via the finite element method with rectengular element Revision A. 2011.

18. The American Society of Mechanical Engineers. Guide for verification and validation in computational solid mechanics. 2006.

19. MIL-STD-167 Type I, Vibration test report for 901D of the argon ship exploration equipment increment E (SSEE-E) system. Rustburg: Dynamic Testing • A division of DTI Holdings, LLC. 2003.

20. Abaqus 6.14 Online Documentation. [Online].; 2014 [cited 2018 February 13]. Available from: http://abaqus.software.polimi.it/v6.14/books/bmk/default.htm?startat=ch04s02anf10.html .

## APPENDIX A:  A SHIP VIBRATION TEST STUDY

Following work belongs to Dynamic Testing (A Division of DTI Holdings), LLC and taken from the report MIL-STD-167, TYPE I VIBRATION TEST REPORT FOR 901D OF THE ARGON SHIP EXPLORATION EQUIPMENT INCREMENT E (SSEE-E) SYSTEM.

Only a part of the test is included as additional information.

- Test Procedure; Vibration Three orthogonal axes from 4 to 33 hertz.
- Instrumentation; 4 accelometer located as seen in Figure A.1.
- Results of the Vertical Frequency Test is shown in Table A.1.

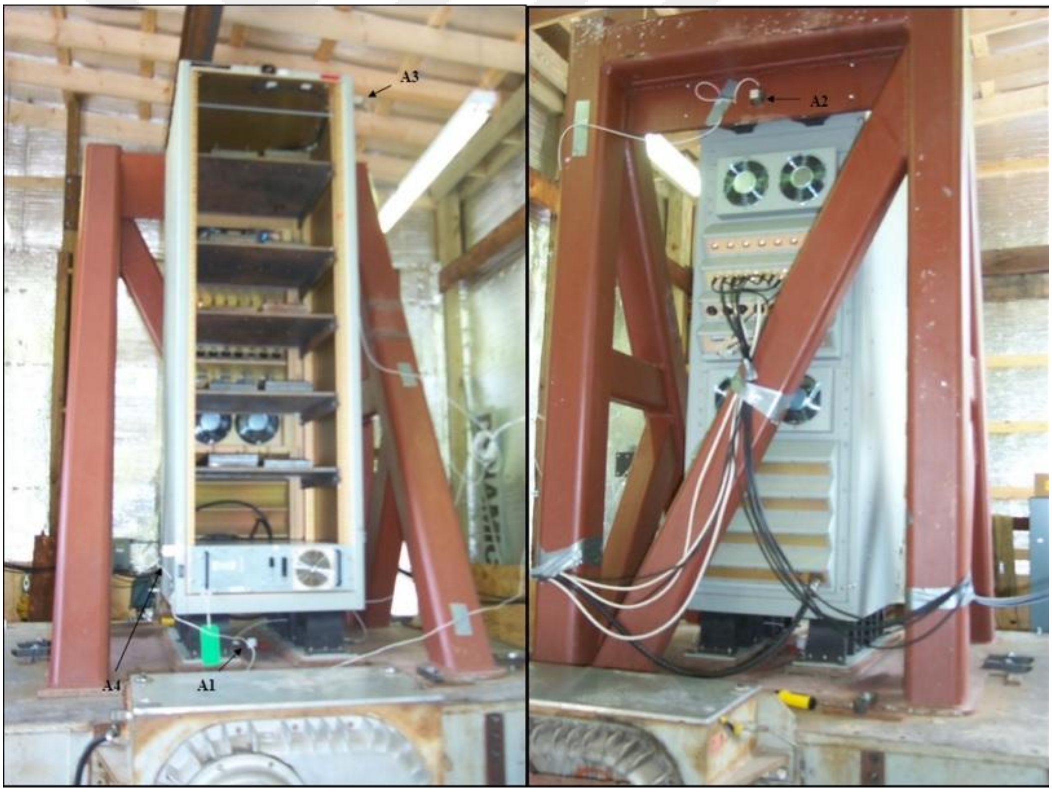


Figure A.1. Accelometers on testing device [19]

- The endurance test should be performed at either resonant frequency between 4-33 Hz or at 33 Hz if no resonant frequency occurs.

- The endurance test for this device was conducted for 8.0 and 9.0 Hertz for one hour each.

Table A.1. Vertical drequency test results

| **Table Input Frequency** | **Table Input Amplitude** | **Back Brace of Fixture** | **Top Right Side of Cabinet** | **Bottom Left Side of Cabinet** | **Q of A3 (Volume accelaration)** |
|---|---|---|---|---|---|
| (Hertz) | A1 (Gs) | A2 (Gs) | A3 (Gs) | A4 (Gs) | $m^3/s^2$ |
| 4,0000 | 0,4800 | 0,0460 | 0,0570 | 0,0580 | 1,1875 |
| 5,0000 | 0,0600 | 0,0580 | 0,0820 | 0,8200 | 1,36667 |
| 6,0000 | 0,0900 | 0,0870 | 0,1480 | 0,1530 | 1,644444 |
| 7,0000 | 0,1243 | 0,1203 | 0,3340 | 0,3234 | 2,687047 |
| 8,0000 | 0,1650 | 0,1570 | 0,6920 | 0,6710 | 4,193939 |
| 9,0000 | 0,0293 | 0,2770 | 1,0700 | 1,2430 | 3,651877 |
| 10,0000 | 0,0360 | 0,3620 | 0,7950 | 0,7990 | 2,208333 |
| 11,0000 | 0,4050 | 0,4100 | 0,6280 | 0,6160 | 1,550617 |
| 12,0000 | 0,4390 | 0,4390 | 0,4850 | 0,4840 | 1,104784 |
| 13,0000 | 0,5460 | 0,5390 | 0,4640 | 0,4640 | 0,849817 |
| 14,0000 | 0,5870 | 0,6070 | 0,4110 | 0,4080 | 0,70017 |
| 15,0000 | 0,6230 | 0,6730 | 0,3690 | 0,3610 | 0,592295 |
| 16,0000 | 0,5950 | 0,7460 | 0,3410 | 0,3410 | 0,573109 |
| 17,0000 | 0,6880 | 0,9290 | 0,3670 | 0,3610 | 0,53343 |
| 18,0000 | 0,8060 | 0,8100 | 0,2760 | 0,2740 | 0,342432 |
| 19,0000 | 0,7940 | 0,8190 | 0,2290 | 0,2350 | 0,288413 |
| 20,0000 | 0,9130 | 0,9240 | 0,2270 | 0,2340 | 0,248631 |
| 21,0000 | 0,8680 | 0,9027 | 0,1887 | 0,1925 | 0,217396 |
| 22,0000 | 1,1000 | 0,9630 | 0,1830 | 0,1790 | 0,166364 |
| 23,0000 | 1,1800 | 1,0600 | 0,1750 | 0,1630 | 0,148305 |
| 24,0000 | 1,1500 | 1,1700 | 0,1820 | 0,1460 | 0,158261 |
| 25,0000 | 1,2600 | 1,3200 | 0,2580 | 0,1550 | 0,204762 |
| 26,0000 | 0,7680 | 0,8400 | 0,1710 | 0,1050 | 0,222656 |
| 27,0000 | 0,7690 | 0,8180 | 0,1120 | 0,1370 | 0,145644 |
| 28,0000 | 0,7740 | 0,8760 | 0,0370 | 0,1480 | 0,047804 |
| 29,0000 | 0,9100 | 0,9060 | 0,0240 | 0,1390 | 0,026374 |
| 30,0000 | 0,8840 | 1,0360 | 0,0070 | 0,1287 | 0,007919 |
| 31,0000 | 0,9438 | 1,1500 | 0,0040 | 0,1170 | 0,04238 |
| 32,0000 | 0,9940 | 1,2300 | 0,0030 | 0,1070 | 0,003018 |
| 33,0000 | 1,2600 | 1,2500 | 0,0100 | 0,0970 | 0,007937 |