

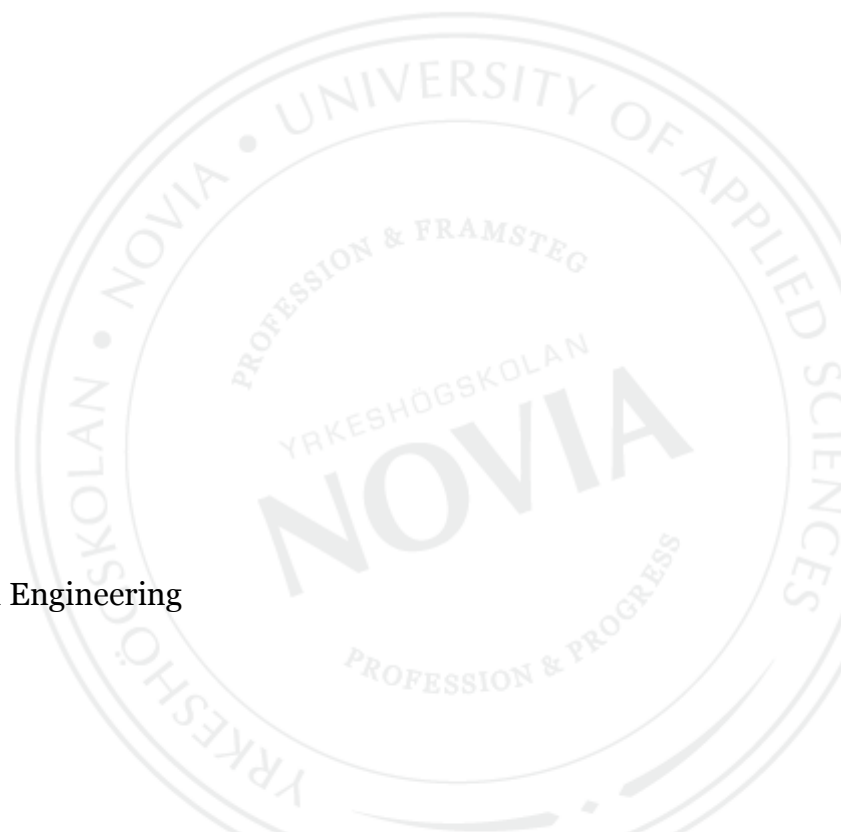
# **Mechanical design of an advanced self-aligning mounting system**

Sakarias Widner

Bachelor's Thesis

Mechanical and Production Engineering

Vaasa 2013



# BACHELOR'S THESIS

Author: Sakarias Widner  
Degree programme: Mechanical and Production Engineering, Vaasa  
Specialization: Mechanical Construction Systems  
Supervisors: Kaj Rintanen & Tomas Södö

Title: *Mechanical design of an advanced self-aligning mounting system*

---

Date: 15 April 2013

Number of pages: 32

Appendices: 3

---

## Abstract

This Bachelor's thesis work has been conducted in cooperation with Wärtsilä Ship Power. The main focus of this study has been placed on designing the engine mounting brackets, which are included in a flexible mounting system for a Wärtsilä 12V46F engine. Air springs are used for the engine mounting in order to improve the degree of insulation and to eliminate the frequent engine re-alignment that is needed when soft rubber mounts are used.

The study is based on two former theses. The main focus of the initial study was placed on the forces that the air springs are exposed to. The result was a force calculation tool in Excel, which calculates the reaction force acting on the air springs. The second study was focused on the mechanical construction of the mounting system. The result was a complete 3D assembly of the mounting system.

This thesis includes a theoretical part where classification rules regarding the design of ship engines are presented. The Finite Element Method is explained whereafter a structural analysis is carried out in order to calculate the stress and the displacement in the structure caused by engine torque and inclinations due to sea movements.

An appropriate design of the engine mounting brackets is developed through this study. A cost calculation, which compares the manufacturing cost of the obtained design with a previous construction, is also carried out.

---

Language: English

Key words: FEA, FEM, mounting system

---

Filed at Theseus.fi.

# EXAMENSARBETE

Författare: Sakarias Widner  
Utbildningsprogram och ort: Maskin- och produktionsteknik, Vasa  
Inriktningsalternativ: Maskinkonstruktion  
Handledare: Kaj Rintanen & Tomas Södö

Titel: *Konstruktion av en avancerad självlinjerande uppställning*

---

Datum: 15.4.2013

Sidantal: 32

Bilagor: 3

---

## Abstrakt

Detta examensarbete har utförts i samarbete med Wärtsilä Ship Power. Tyngdpunkten i arbetet ligger på konstrueringen av infästningarna mellan motordynorna och motorblocket, som ingår i en flexibel uppställning av en Wärtsilä 12V46F motor. I uppställningen används luftbälgar som flexibla element, för att förbättra isoleringen av vibrationer och för att eliminera den återkommande linjeringen av motorn som krävs när mjuka gummielement används.

Arbetet baserar sig på två tidigare examensarbeten. Tyngdpunkten i den första studien låg på de krafter som luftbälgarna utsätts för. Resultatet var ett Excel-verktyg som beräknar reaktionskrafterna som verkar på luftbälgarna. Tyngdpunkten i den andra studien låg på konstrueringen av motoruppställningen. Resultatet var en komplett 3D-sammanställning av motoruppställningen.

Detta arbete innehåller en teoribeskrivning där klasskrav för design av fartygsmotorer presenteras. Finita elementmetoden presenteras varefter en strukturell analys utförs för att beräkna spänningarna och förskjutningen som uppstår i strukturen, till följd av motorns belastning samt lutningar orsakade av sjögång.

Genom arbetet erhålls en ändamålsenlig design av motorinfästningarna. En kostnads kalkyl, som jämför tillverkningskostnader för den erhållna konstruktionen med en tidigare konstruktion, presenteras i slutet.

---

Språk: engelska

Nyckelord: FEA, FEM, motoruppställning

---

Examensarbetet förvaras i webbliblioteket Theseus.fi.

# OPINNÄYTETYÖ

Tekijä: Sakarias Widner  
Koulutusohjelma ja paikkakunta: Kone- ja tuotantotekniikka, Vaasa  
Suuntautumisvaihtoehto: Koneensuunnittelu  
Ohjaajat: Kaj Rintanen & Tomas Södö

Nimike: *Kehittynyt koneen linjaus ja kiinnitysjärjestelmän suunnittelu*

---

Päivämäärä: 15.4.2013

Sivumäärä: 32

Liitteet: 3

---

## Tiivistelmä

Tämä opinnäytetyö on tehty yhteistyössä Wärtsilä Ship Powerin kanssa. Tämän työn tarkoitus oli suunnitella moottorityynyjen ja moottorilohkon välissä sijaitsevat kiinnikkeet, joita käytetään Wärtsilän 12V46F-moottorin joustavassa kiinnitysjärjestelmässä. Kiinnitysjärjestelmässä käytetään ilmatyynyjä joustavina kiinnityselementteinä parantamaan moottorin eristystä ja välttämään moottorin toistuvia uudelleenlinjauksia, jota pehmeät elementit vaativat.

Opinnäytetyö perustuu kahteen aiempaan tutkimukseen. Ensimmäisen tutkimuksen tarkoitus oli tutkia voimia, joille ilmatyynyt ovat alttiina. Tuloksena oli Excel-työkalu, joka laskee ilmatyynyihin vaikuttavat reaktiovoimat. Toinen tutkimus keskittyi kiinnitysjärjestelmän suunnitteluun. Tuloksena oli täydellinen 3D-kokoonpano kiinnitysjärjestelmästä.

Tämä opinnäytetyö sisältää teoriaosion, jossa selitetään viranomaisten vaatimuksia laivamoottoreiden suunnittelua varten. Elementtimenetelmä selitetään ja sitä käytetään rakenteellisessa analyysissä laskemaan jännitteet ja siirtymä, jotka moottorin kuorma ja aallokoiden kaltevuudet aiheuttavat.

Tämän tutkimuksen lopputulos on suunnittelu moottorikiinnikkeistä. Lopussa esitetään kustannusarvio joka vertailee tuotantokustannuksia tuloksena olevasta rakenteesta aiemmin tehtyyn rakenteeseen.

---

Kieli: englanti

Avainsanat: FEA, FEM, moottorin kiinnitys

---

Opinnäytetyö arkistoidaan verkkokirjastossa Theseus.fi.

# Table of contents

1	Introduction .....	1
1.1	Background .....	1
1.2	Problem formulation .....	1
1.3	Goal.....	2
1.4	Limitations .....	2
1.5	The Wärtsilä 46F engine .....	3
1.6	Outline .....	4
2	Earlier research .....	5
2.1	Results of earlier research .....	6
3	Theoretical background .....	9
3.1	Rules made by classification societies .....	9
3.1.1	Det Norske Veritas AS .....	9
3.1.2	Summary of the classification rules.....	10
3.2	The Finite Element Method .....	11
3.2.1	Finite Element Analysis considerations .....	11
3.2.2	Types of Finite Elements .....	12
3.2.3	Finite Element Analysis procedure .....	13
3.2.4	Conclusion .....	14
4	Method.....	15
4.1	Modelling.....	15
4.2	Engine mounting bracket .....	15
4.3	Force calculation.....	17
4.4	Finite Element Analysis.....	19
4.4.1	Idealizing the geometry .....	19
4.4.2	Defining a material and creating the mesh .....	21
4.4.3	Applying loads and constraints.....	22
4.4.4	Solving the model and viewing the analysis results .....	23
5	Results .....	24
5.1	Initial results .....	24
5.2	Geometry improvement .....	25
5.3	Final results.....	26
5.4	The final design .....	28
5.5	Cost calculation.....	29

6	Conclusion.....	30
6.1	Reaching the goal.....	30
6.2	Use of FEA .....	31
6.3	Utility.....	31
6.4	Future research.....	31
7	List of sources.....	32

## **Appendices**

Appendix 1 Force calculation tool.

Appendix 2 Stress and displacement as a result of the initial FEA.

Appendix 3 Stress and displacement as a result of the final FEA.

## **Basic terminology**

Constraint	A fixation of the model which emulates how it would be held in real life.
FEA	Finite Element Analysis.
FEM	Finite Element Method.
Mesh	A network of elements and nodes making up the FEM model.
Node	The vertices that make up an element.



## Symbols, quantities and units

Symbol	Quantity	Unit
F	Force	N
M	Moment	Nm
n	Rotational speed	1/min
P*	Power	kW
r	Radius	m
Sf	Safety factor	
v	Velocity	m/s
$\sigma_v$	Von Mises stress	N/mm <sup>2</sup>
$\sigma_y$	Yield strength	N/mm <sup>2</sup>
$\omega$	Angular velocity	rad·s <sup>-1</sup>

\* kW for equation (1) only, otherwise the unit should be W.

## **Preface**

Conducting this thesis work has been challenging but truly interesting. It is a great experience that completes my studies in mechanical engineering at Novia University of Applied Sciences. I want to thank Tomas Södö, my supervisor at Wärtsilä Ship Power, for giving me the opportunity to do this thesis work. I also want to thank Kaj Rintanen, my supervisor at Novia University of Applied Sciences, for giving me clear guidelines and feedback throughout the study.

Vaasa, 15 April 2013

*Sakarias Widner*

# **1 Introduction**

This Bachelor's thesis will deal with the mechanical design of a steel structure in an engine mounting system. The design will be explained and carried out with regards to previous constructions, specific demands and numeric methods.

The construction has to fulfil the specific demands of the usage area. Rules provided by classification societies will therefore be considered in order to ensure that the construction is appropriate from a safety point of view. Analytical calculations based on the classification rules are performed in order to calculate the forces acting on the structure. The theoretical part will deal with a method used to calculate stress and displacement and to ensure that failure will not occur under the effect of applied loading.

## **1.1 Background**

This thesis work has been conducted in cooperation with Wärtsilä Ship Power, which is a Finnish corporation that provides customers with marine products, integrated solutions and service. The main products are medium-speed four-stroke engines and low-speed two-stroke engines.

This thesis is a part of a project regarding a flexible mounting system for ship engines. The project was started in order to make a new concept for resilient mounting. Resilient mounting is often used in marine main-engine installations with strict vibration and noise requirements, such as passenger ships and research vessels. Resilient mounting is used in such applications in order to reduce vibrations and noise transfer to the ship foundation.

## **1.2 Problem formulation**

Today, Wärtsilä uses different types of soft rubber elements in their flexible mounting systems. The problem with soft rubber elements is the high creep rate which causes the engine to move from its initial position after some time. This will require a frequent engine re-alignment using shims to get the correct alignment, which can be costly and time consuming for the ship owner.

To get round the frequent engine re-alignment that the conventional system requires, air springs were selected as engine mounts in the new concept. Air springs can provide the highest degree of insulation compared to other types of vibration insulators and their natural frequencies can be as low as 1 Hz. The new system will be self-aligning in order to keep the correct alignment between the engine and the generator. This is achieved by regulating the pressure in the air springs. Sensors will detect the height of the air spring and keep it in the right position at all times during engine operation.

An appropriate type of air springs has been selected and a preliminary model of the system has already been designed, but strength calculations have only been carried out on a few parts. Therefore, further design requires strength calculations in order to ensure that an appropriate design is obtained.

### **1.3 Goal**

The main objective of this study is to design major steel parts for an engine mounting system using air springs for the Wärtsilä 12V46F engine. Main dimensions and data of the engine are presented in chapter 1.5. The result will be detailed drawings of the major steel parts and an assembly drawing of the mounting system. A cost calculation will eventually be made in order to investigate the manufacturing cost of the major steel parts.

### **1.4 Limitations**

This thesis has been limited to one engine type, i.e. the W12V46F. A V-configuration has been selected for the air springs due to advantages compared to a vertical configuration. This will be explained thoroughly in chapter 2.1. Clearance to other parts has to be considered since the engine and essential parts of the system have been designed earlier.

One of the major steel parts in the mounting system is the engine mounting bracket, which connects the air spring to the engine block. A 12V cylinder configuration would require ten engine mounting brackets with the same design. The main focus has therefore been placed on designing this part. The brackets in the corners of the engine will also be attached to air springs which will constrain longitudinal movements. However, these will not be included in this study since the air springs that they will be attached to have not been selected yet.

Structure, material and dimensions will be determined according to a previous design of a flexible mounting system for a similar engine. An existing 3D model of the mounting system will be used and further developed.

The forces acting on the bracket will be calculated according to classification rules, which give system requirements and prescribe minimum requirements for materials and design.

## 1.5 The Wärtsilä 46F engine

The Wärtsilä 46F is a medium-speed, 4-stroke, non-reversible, turbocharged and intercooled diesel engine with direct fuel injection.

Main data and output:

- Cylinder bore 460 mm
- Stroke 580 mm
- Speed 600 rpm
- Maximum continuous output 14400 kW (IMO Tier 2)

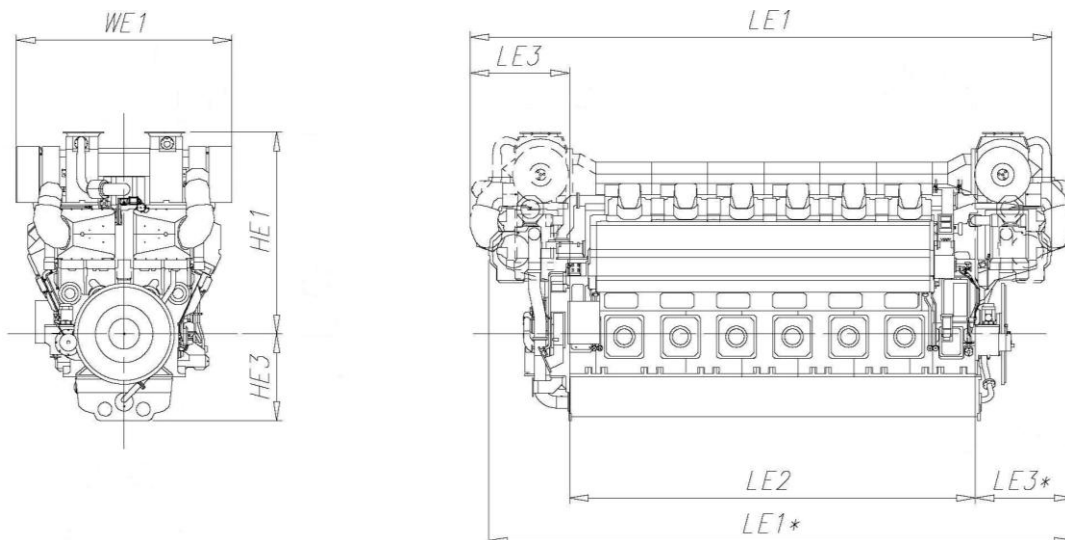


Figure 1. V-engine. (Wärtsilä 2011, p. 4).

Table 1. Dimensions and weight.

Engine	LE1*	LE1	LE2	LE3*	LE3	HE1	HE3	WE1	WE3	Weight [ton]
12V46F	10945	10284	7600	1830	1952	3765*/ 3770	1620	4040*/ 4026	1820	177

\* Turbocharger in flywheel end. All dimensions are in mm. The weight is dry weight of a rigidly mounted engine without flywheel. (Wärtsilä 2011, p. 4).

## 1.6 Outline

The contents of the chapters are described as follows.

- Chapter 1 begins with a short introduction to the subject, followed by the background and the problem formulation. A goal is set and the limitations are prescribed. The chapter ends with a short introduction to the engine.
- Chapter 2 presents earlier research. The results of two previously conducted theses are presented.
- Chapter 3 introduces the theoretical background. Essential rules of four different classification societies are presented and one society is studied thoroughly. A conclusion of the classification rules is thereafter presented. The chapter ends with an explanation of the Finite Element Method.
- Chapter 4 describes how the modelling has been carried out whereafter the obtained design is explained. The use of the force calculation and the FEA procedure is explained at the end of the chapter.
- Chapter 5 presents the results from the initial FEA. A second FEA is carried out and presented after interpreting the initial results.
- Chapter 6 summarizes the way of reaching the goal whereafter the results are discussed. The utility of the study is mentioned and possible future research is finally suggested.

## **2 Earlier research**

Two studies of the same subject have previously been carried out. The first one was done in 2011 by Anders Wasberg. His study was a Bachelor's thesis done at Novia University of Applied Sciences in Vaasa. It was a pre-study for a mounting system using air springs in a V-configuration. The main focus of the study was the forces that the air springs are exposed to. The major result of his study was a force calculation tool made in Excel. The force calculation tool will be used in this study, and it will be explained thoroughly in chapter 2.1.

The second study was done in 2012 by Filip Långbacka. His study was also a Bachelor's thesis done at Novia University of Applied Sciences in Vaasa. The purpose of his study was to design a self-aligning mounting system for the Wärtsilä 12V46F engine. The main focus was placed on the mechanical design of the mounting system. The result was a preliminary 3D model of the mounting system made in NX Unigraphics. Studying the classification rules made by DNV and using the force calculation tool enabled him to calculate the forces acting on the air springs. Strength calculations were also carried out on some parts of the mounting system. An air spring supplier was selected and an appropriate model was finally chosen.

## 2.1 Results of earlier research

A V-configuration of the mounting system was selected in the first study due to great advantages with respect to the roll and trim that occur in ships due to sea movements. The characteristics of a V-configuration will prevent the engine from rolling. Besides this, it will also obstruct the force that is produced due to engine torque. Another benefit is that if the stiffness centre is near the centre of the flexible coupling, rather small movements will occur at the flexible coupling. (Wasberg 2011, p. 7 & 15).

The force calculation provides the forces that the air springs are exposed to. The result is achieved by adding the following required parameters:

- Transverse inclination (roll) and longitudinal inclination (trim)
- Engine weight
- Torque caused by the engine
- All distances on the engine and mounting system
- Number of air springs
- Element angle

The result is the forces acting on the engine mounts in a worst case scenario. The worst case scenario is considered when the engine is exposed to both roll and trim at the same time plus maximum torque. The picture below shows the radial force ( $F_{1x}$  and  $F_{2x}$ ) and the axial force ( $F_{1y}$  and  $F_{2y}$ ) which were used in order to select an appropriate air spring. In this thesis, the reaction forces ( $F_1$  and  $F_2$ ) will be used to calculate the stress and the displacement of the structure.

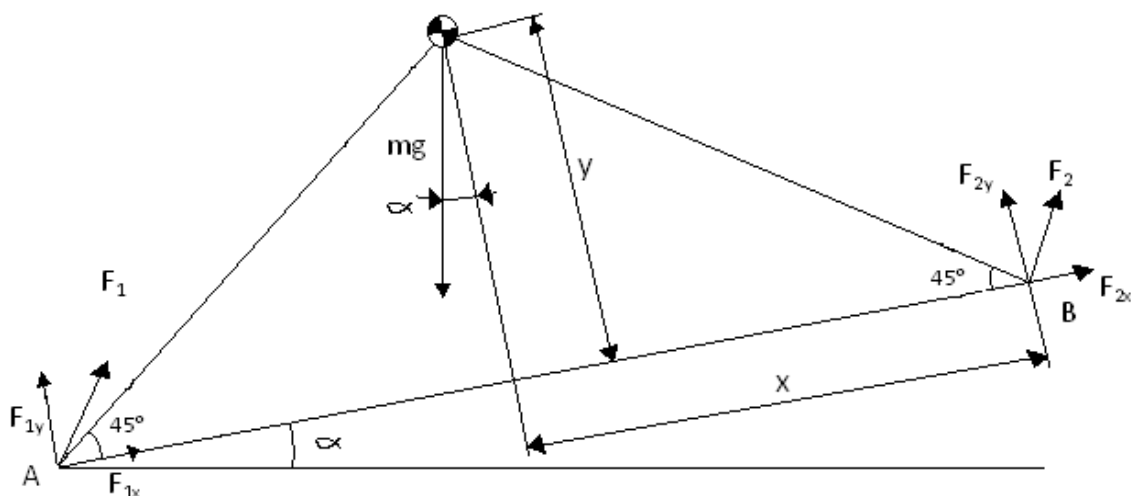
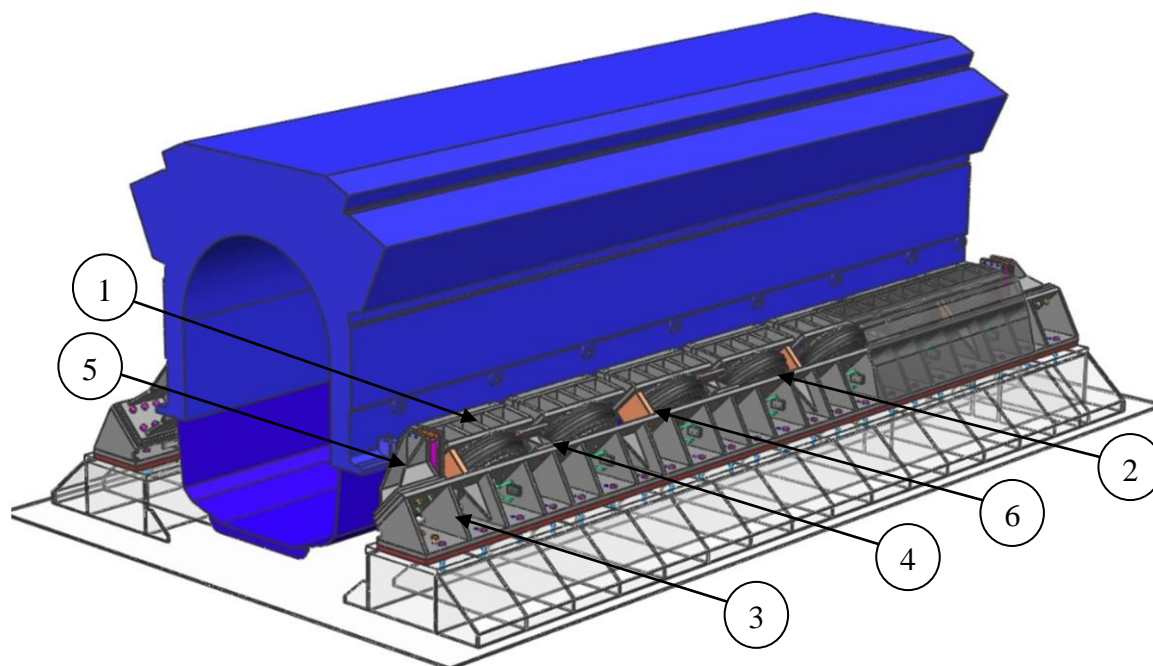


Figure 2. An illustration used in the force calculation. (Wasberg 2011, appendix 1).



The figure below shows the 3D model of the mounting system as a result of the second study. 3D models of the different parts were made in NX Unigraphics, except for the engine block and standard parts such as bolts, nuts and washers. These had been modelled earlier in I-deas. The study began by simplifying the block of a W12V46F engine in I-deas after which it was exported to NX. A top-down modelling method was used, which means that you start with the part closest to the engine block and work yourself down to the tank top of the ship. (Långbacka 2012, p. 10).



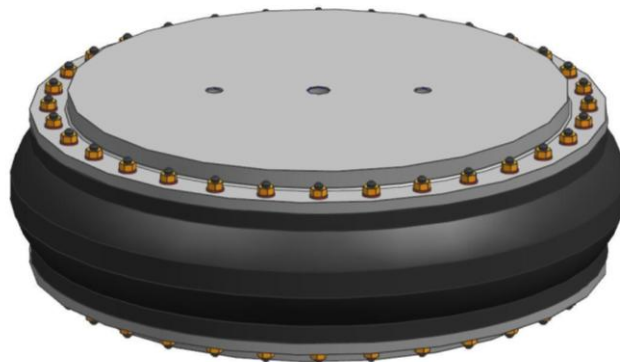
*Figure 3. Complete assembly of the mounting system with air springs. (Långbacka 2012, p. 11).*

#### Location of different parts

1. Engine mounting bracket
2. Air spring
3. Fixing rail
4. Height/lateral buffer
5. Longitudinal buffer
6. Transport bracket

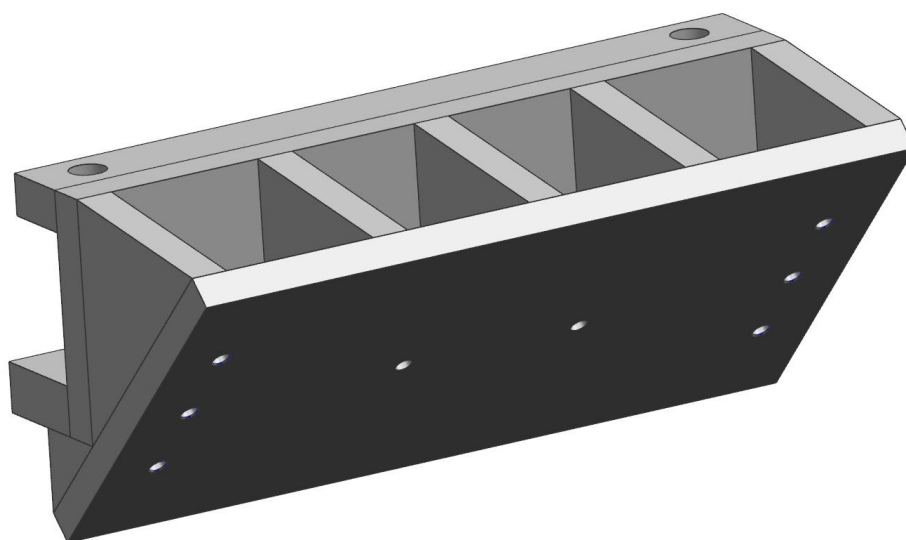
When the concept design of the mounting system was complete, strength calculations were performed for the height/lateral buffers and the transport brackets. Strength calculations were also carried out for the bolts connecting the engine mounting bracket and the fixing rail to the transport brackets. The parts that were over- or under-dimensioned were eventually modified. (Långbacka 2012, p. 10).

An appropriate air spring could be selected by using the force calculation tool. The selected air spring is manufactured by ContiTech and the model is FS 2870-16 RS. It is a single convolution type with removable bead rings. A picture of the air spring is shown below. (Långbacka 2012, p. 12).



*Figure 4. Air spring with bead rings.*

A preliminary model of the engine mounting bracket was designed in the second study. The design was based on a similar part which was designed by Wärtsilä earlier for a 12V46F using flexible mounts in a vertical configuration. The new bracket is, contrary to the part developed earlier by Wärtsilä, designed as a separate bracket for a V-configuration.



*Figure 5. Engine mounting bracket.*

### 3 Theoretical background

This chapter presents the demands made by classification societies. A method used to calculate stress and displacement is presented at the end of the chapter.

#### 3.1 Rules made by classification societies

Rules by a few different classification societies have been issued and one of them has been studied more thoroughly. The essential rules in this case are the required design angles of the inclination for machinery systems. The inclination is caused by sea movements and it is important, because it determines the size of the reaction force. The requirements regarding inclination angles are stated in the table below.

*Table 2. Required inclination angles for main and auxiliary machinery.*

Classification society	Angle of inclination [°] <sup>1</sup>			
	Athwartships		Fore and aft	
	Static	Dynamic	Static	Dynamic
American Bureau of Shipping	15	22,5	5	7,5
Det Norske Veritas	15	22,5	5	7,5
Germanischer Lloyd	15	22,5	5	7,5
Lloyds Register	15	22,5	5	7,5

<sup>1</sup> Athwartships and fore and aft inclinations may occur simultaneously.

(ABS 2006, part 4, chap. 1, p. 1).

(DNV 2012b, part 4, chap. 1, p. 31).

(GL 2012, part 1, chap. 2, pp. 1-2).

(LR 2011, part 5, chap. 1, s. 3).

##### 3.1.1 Det Norske Veritas AS

The DNV demands for the design of engine components have been studied thoroughly. The most essential rules to be applied in this case are presented below.

The rules give system requirements and prescribe minimum requirements for materials and design. The design should take into account the materials used in the construction, the purpose for which the construction is intended and the work which it will be subjected to. (DNV 2012b, part 4, chap. 1, pp. 5-6).

Welded structural components shall be designed with care in order to avoid fatigue cracking due to the stresses in operation. Full penetration welds are to be applied in the most fatigue exposed areas. (DNV 2012a, part 4, chap. 3, p. 14).

Bolts and nuts exposed to vibrations and dynamic forces are to be properly secured (DNV 2012b, part 4, chap. 1, p. 11).

Bolt connections shall be designed to prevent fatigue. Particular attention must be given to possible fretting of joining surfaces. (DNV 2012a, part 4, chap. 3, p. 16).

Components are to be designed according to the loads and ambient conditions which are expected to occur. Generally recognized standards and engineering principles may be applied regarding the choice of material. Generally accepted safety margins are to be used. (DNV 2012b, part 4, chap. 1, p. 13).

According to the demands of resilient mounting, the elastic mounts shall be able to support the reaction force caused by the mass of the engine, the maximum engine torque and the worst environmental conditions. The worst environmental conditions are the maximum inclination caused by sea movements. (DNV 2012a, part 4, chap. 3, p. 38).

All machinery, components and systems are to be designed to operate under the following environmental conditions:

- Ambient air temperature in the machinery space between 0 °C and 55 °C.
- Relative humidity of air in the machinery space up to 96 %.
- List, rolling, trim and pitch according to table 2.

(DNV 2012b, part 4, chap. 1, p. 31).

### **3.1.2 Summary of the classification rules**

All classification societies state similar demands regarding required design angles of inclination for engine systems. The product guide for the Wärtsilä 46F engine states quite the same inclination angles at which the engine will operate satisfactorily. The only difference is that the maximum permanent angle for trim is 10°. The worst case scenario will therefore be a combination of inclinations stated by classification societies and by Wärtsilä's product guide.

## **3.2 The Finite Element Method**

Design is a process where products are created and modified, and it includes all activities from the original concept to the finished product. With the advancements in computer technology, it is nowadays possible to analyze three-dimensional designs without building physical models. The creation of three-dimensional models on computers offers a wide range of benefits. Computer models are easier to interpret and can easily be altered. Simulations of real-life loads can be applied and the results can be graphically displayed. (Shih 2004, chap. 1, p. 2).

The Finite Element Method (FEM) enables calculations of stresses, displacements, natural frequencies and dynamic responses of structures. The name comes from the basic idea of dividing a continuous structure into a finite number of sub-regions. FEM was first applied on problems of structural mechanics but moreover it is a method to solve partial differential equation systems approximately. (Heinze & Schmidt 2004, p. 3).

### **3.2.1 Finite Element Analysis considerations**

Finite Element Analysis (FEA) of an engineering problem requires the idealization of the problem into a mathematic model. Since the idealization will affect the results, it is crucial to select an appropriate mathematical model that will most closely represent the actual situation. It is also important to realize that the response cannot be predicted exactly, because it is impossible to create a mathematical model that will represent all the information contained in the actual system. (Shih 2004, chap. 1, p. 3).

A general rule is to begin the modelling with a simplified model. Once a mathematical model has been solved accurately and the results have been interpreted, it is possible to create a more refined model. This is done in order to increase the accuracy of the results. In a structural analysis, the application of the actual loads and constraints into the appropriate model can drastically change the results of the analysis. The results from the simplified model, combined with an understanding of the behaviour of the system, will assist in determining if the model needs further refinements. (Shih 2004, chap. 1, p. 3).

### 3.2.2 Types of Finite Elements

The finite element method is a numerical solution technique that finds an approximate solution by dividing a continuous structure into a finite number of sub-regions. The division of the structure enables the governing equations of each sub-region to be reached much more simply compared to the equation needed for the entire region. The sub-regions are called elements and they are connected through interconnecting points, called nodes. The elements make up the element mesh, see the figure below. (Shih 2004, chap. 1, p. 4).

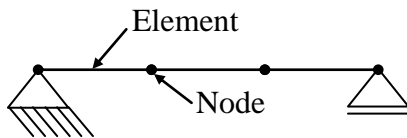


Figure 6. Mesh consisting of three elements and four nodes.

There are numerous types of finite elements and they can be divided into three categories depending on the geometry:

1. One-dimensional line elements: Truss, beam, and boundary elements.



Figure 7. One-dimensional element.

2. Two-dimensional plane elements: Plane stress, plane strain, axisymmetric, membrane, plate and shell elements.



Figure 8. Two-dimensional element.

3. Three-dimensional volume elements: tetrahedral, hexahedral, and brick elements.

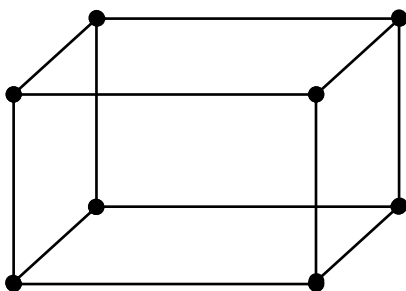


Figure 9. Three-dimensional element.

Finite element solutions using one-dimensional line elements are usually as accurate as solutions obtained using conventional truss and beam theories. Generally, it is easier to get results using FEA compared to manual calculations using conventional theories. Regarding two-dimensional elements, very few closed-form solutions exist and almost none exist for three-dimensional solid elements. (Shih 2004, chap. 1, p. 5).

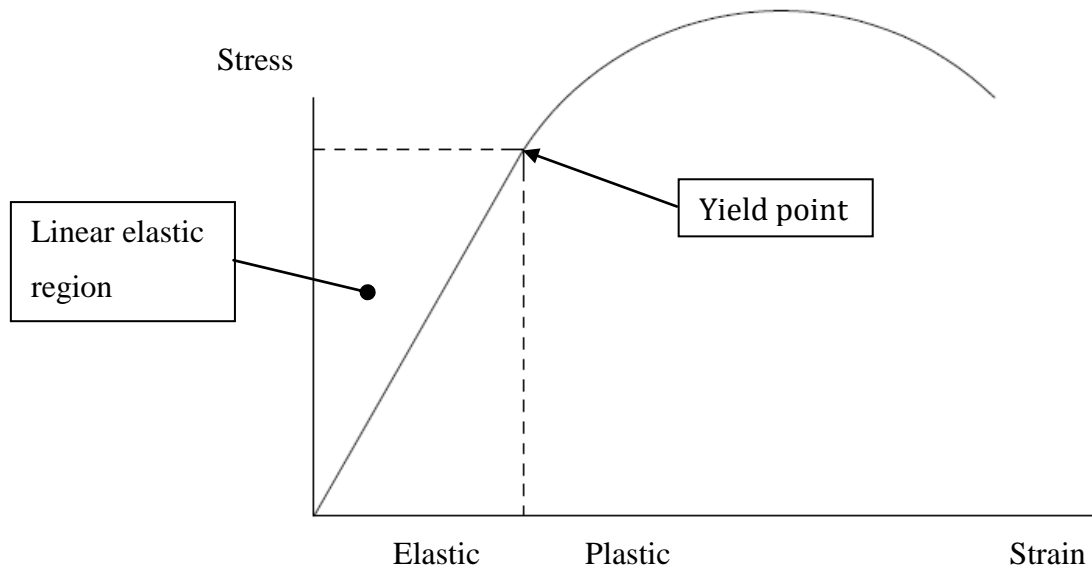
In theory, all structures can be modelled with three-dimensional volume elements. However, this is not practical since many designs can be simplified with reasonable assumptions in order to achieve accurate FEA results. An advantage with using simplified models is less required time and effort to reach FEA solutions. (Shih 2004, chap. 1, p. 5).

A finite element model using three-dimensional solid elements may look the most realistic when compared to the other types of elements. However, this type of analysis requires more elements, which requires more mathematical equations, effort and time. (Shih 2004, chap. 10, p. 2).

### **3.2.3 Finite Element Analysis procedure**

The main goal of a finite element analysis is to calculate the stresses and displacements under specified loading conditions. Another important issue is to determine if failure will occur under the effect of the applied loading. In the elastic region of the material, the deformation of the system is recoverable. If the system is stressed beyond the elastic limit, even in a small region of the system, the deformation is no longer recoverable. However, this does not necessarily mean that the system cannot carry any further load. (Shih 2004, chap. 10, pp. 4-5).

There are several theories which define when failure occurs. However, all these theories provide quite similar results. The most commonly used failure criterion is the Von Mises yield criterion. Von Mises is a scalar quantity, and when compared with the yield stress of the material, it indicates if the system has exceeded the elastic state. The diagram on the next page shows a stress - strain curve for a typical ductile material. (Shih 2004, chap. 10, p. 2).



*Figure 10. Stress - strain diagram of a typical ductile material.*

### 3.2.4 Conclusion

When examining the FEA results, one should first examine the deformed shape to check for proper placement of boundary conditions and to determine if the deformation of the model is reasonable. The mesh can be refined in high-stress areas in order to obtain more accurate results. Besides confirming that the system remains in the elastic regions for the applied loading, other considerations are also important such as large displacements. (Shih 2004, chap. 10, p. 31).

The Finite element analysis has rapidly become a vital tool for design engineers. It should be emphasized that the use of this tool does not guarantee correct results. The design engineer is still responsible for making approximate calculations, using good design practice, and applying good engineering judgement to the problem. It is hoped that FEA will supplement these skills to ensure that an appropriate design is obtained. (Shih 2004, chap. 10, p. 31).



## **4 Method**

The modelling and the design of the engine mounting bracket will be explained in this chapter. The use of the force calculation tool to calculate the reaction force is also described. A FEA is explained and carried out at the end of the chapter.

### **4.1 Modelling**

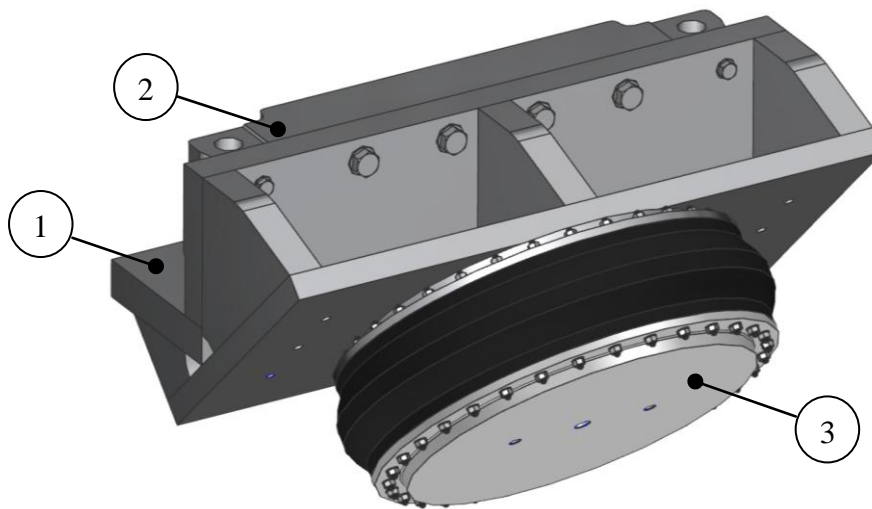
The modelling has been performed in NX Unigraphics 7.5. The preliminary 3D assembly was created in order to give an overall picture of the flexible mounting system using air springs. The next step would be to bring the design down to a detailed level. The previous 3D model was based on a simplified engine block. Since there is a lot of piping attached to the engine block, this has to be simulated before conducting any further design on a detail level. Therefore, the modelling began by exporting a W12V46F engine with all parts included from I-DEAS. Having a complete engine as a 3D model ensures that further design will be made with enough clearance in respect to other parts and that there is enough space for service of the engine.

### **4.2 Engine mounting bracket**

Most attention has been paid to the brackets connecting the air springs to the engine block. The brackets have, in contradiction to the lower fixing rail, been designed as separate parts. Since the length of the engine differs according to cylinder configuration, the amount of required brackets varies. For example a 12V46F engine would need a total amount of ten similar brackets plus one bracket in each corner. The advantage with separate brackets is that they can be mounted on all W46F engines independently of the configuration. Using similar brackets for different engine configurations is also a benefit from an economical point of view.

The material and all dimensions have been selected on the basis of a previous fixing rail for a similar engine. The bracket consists of a welded steel structure, and the selected material is S275. Fatigue cracking might occur in welded structural components due to the stresses in operation. Full penetration welds are therefore to be applied.

The new design can be seen in the picture below. The amount of steel plates has been decreased compared with the preliminary model. It will be confirmed in chapter 5 if this modification affects the stiffness and the strength. To give an idea of the size, it could be mentioned that the weight is about 500 kg and the length is about 1 m. The thickness of the steel plates varies from 40 to 60 mm.



*Figure 11. The new design of the engine mounting bracket.*

Location of different parts:

1. Bracket
2. Flat bar
3. Air spring with bead rings

The flat bar has been designed earlier by Wärtsilä. The flat bar is bolted to the bracket in order to eliminate additional tensions that would occur if the parts were welded together. This will also facilitate the mounting of the bracket to the engine block. Particular attention must be given to possible fretting of joining surfaces. Therefore, the holes for the bolts are made with a clearance, so that the machined surfaces of the flat bar and the bracket will be properly aligned. The bolts will be exposed to vibrations and must therefore be properly secured. This is achieved by counter-boring the holes in the bracket with a smooth surface finish. Conical washers, which provide an axial force when deformed, will be applied as locking devices.

All machinery components must be designed to operate in a relative humidity of air of up to 96 %. The bracket will therefore be painted with an epoxy paint to provide corrosion protection.

### 4.3 Force calculation

The next step will be to calculate the reaction force acting on the bracket. Since the reaction force varies according to the inclination caused by sea movements, the worst theoretical case must be considered. The size of the reaction force is achieved by inserting a few required parameters into the force calculation tool in Excel.

The first required parameter is the inclination of the engine, which is a combination of both roll and trim. The inclination in a worst case scenario will be 22.5° roll and 10° trim, according to previously stated demands.

The second parameter is the element angle, which is 45°.

The third parameter is the engine weight. The engine alone weighs 177 tons, but adding the weight of the brackets will result in an estimated total weight of 185 tons.

The fourth parameter is the torque caused by the engine. Based on the formula below, the calculated torque will be 229200 Nm.

$$(1) M = 9550 \cdot \frac{P^*}{n} \quad * \text{ [kW]}$$

(Valtanen 2010, p. 981).

The factor 9550 is derived from the following equations:

$$(2) P = F \cdot v$$

(Valtanen 2010, p. 202).

$$(3) v = r \cdot \omega$$

(Valtanen 2010, p. 199).

$$(4) \omega = \frac{2 \cdot \pi \cdot n}{60}$$

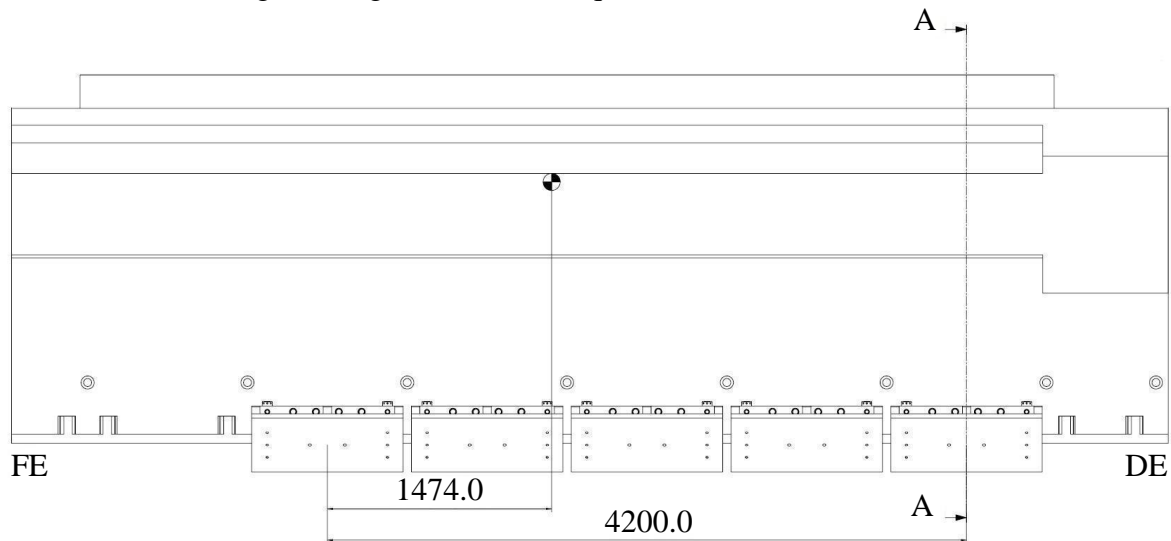
(Valtanen 2010, p. 200).

$$v = \frac{r \cdot 2 \cdot \pi \cdot n}{60}$$

$$P = \frac{F \cdot r \cdot 2 \cdot \pi \cdot n}{60} \rightarrow P = \frac{M \cdot 2 \cdot \pi \cdot n}{60} \rightarrow M = \frac{60 \cdot P}{2 \cdot \pi \cdot n} = 9.549 \cdot \frac{P}{n}$$

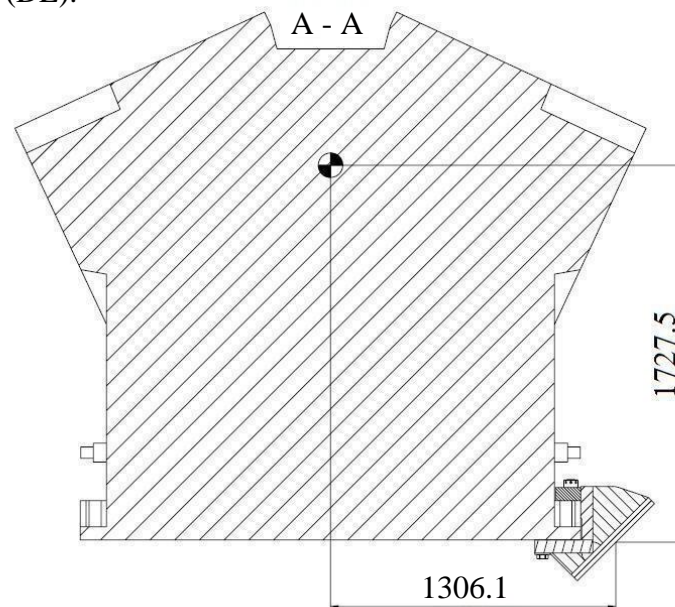
The fifth parameter, which is the number of elements per engine side, will be 5 according to the number of brackets with the same design.

The final parameter is all the distances in the engine and mounting arrangement. The required distances are measured by creating a 3D assembly of the brackets and the engine block. The following drawings indicate the required distances.



*Figure 12. Side view of the mounting arrangement.*

The distances above are measured from the centre of the brackets. The upper dimension (1474.0) is the distance from the bracket in the free end (FE) to the centre of gravity. The lower dimension (4200.0) is the distance between the brackets located in the free end and in the driving end (DE).



*Figure 13. Section view of the mounting arrangement.*

The dimensions above are the horizontal and the vertical distances between the centre of the brackets and the centre of gravity.

The final reaction force is calculated to be 718915 N after the required distances have been entered. A copy of the force calculation tool can be found in appendix 1.

## 4.4 Finite Element Analysis

Having a model of the structure and knowing the size of the reaction force allow you to perform an FEA. The result of the analysis will say how the structure, with regard to both stress and displacement, can be improved.

A structural finite element analysis has been performed in NX UGS 6 with the Advanced Simulation package. The structural analysis can generally be divided into four different steps:

1. Idealizing the geometry.
2. Defining a material and creating the mesh.
3. Applying loads and constraints.
4. Solving the model and viewing the analysis results.

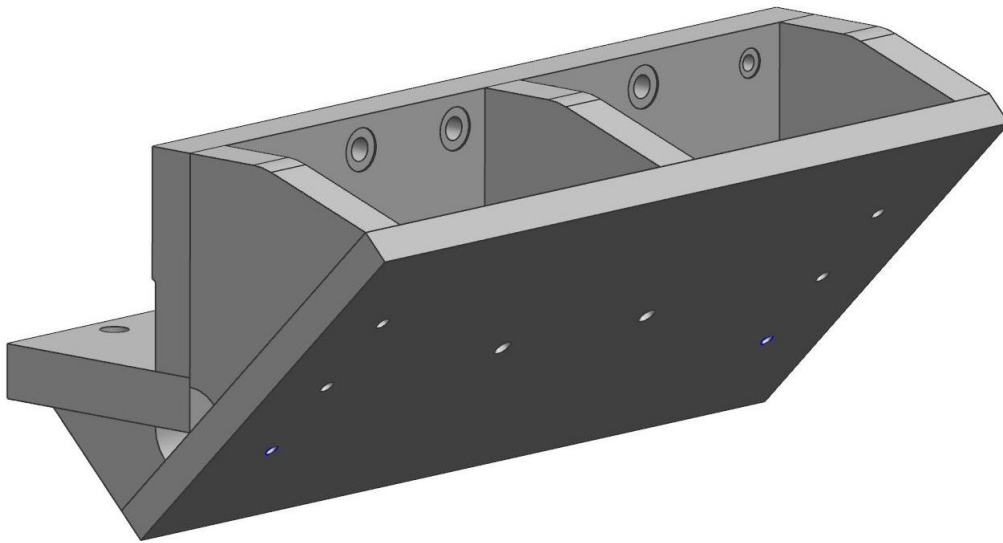
Once the model of the bracket has been created, the Advanced Simulation package is started and a “New FEM and Simulation” is created. The solver is NX Nastran, the analysis type is structural and SESTATIC 101 is the solution type. SESTATIC 101 is used in order to find stress and displacement for static situations. As explained earlier, the reaction force is derived from inclinations caused by dynamic sea movements. However, these sea movements will be very slow (less than 2 Hz) and a static solution type can therefore be selected. The selected reference temperature is 55 °C since machinery components are to be designed for a temperature between 0 °C and 55 °C. The highest temperature is selected since the yield strength of the material will decrease with rising temperature.

### 4.4.1 Idealizing the geometry

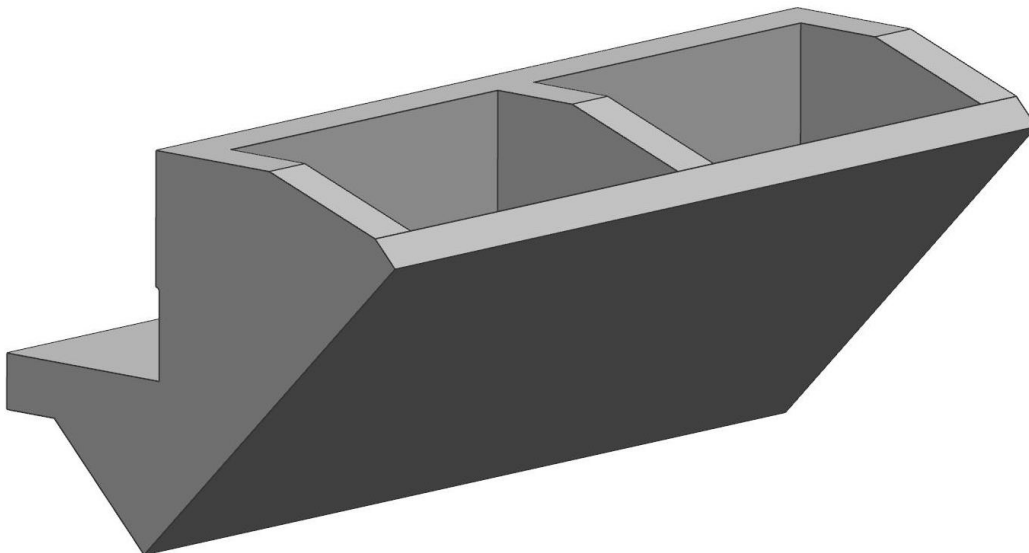
The first step is to idealize in order to get a simplified 3D model. As explained earlier, this is done in order to increase the accuracy of the results. Since the bracket consists of thick steel plates, two-dimensional shell elements are not an option when creating the finite element mesh. Three-dimensional volume elements with a tetrahedral shape would be most appropriate in this case.

A body without radiuses or holes can be considered a simplified geometry. Geometry that is not significant in the analysis is simplified or removed in order to facilitate the creation of the elements. In this case, every radius has been replaced with a sharp edge and all holes have been removed completely.

The pictures below show the transformation from the actual model to the idealized model.



*Figure 14. The actual model.*



*Figure 15. The idealized model.*

A few more modifications are required before creating the mesh. The surfaces that will be bolted to the engine block and to the flat bar will be fixed in the simulation model. Two separate contours will therefore be sketched on the bracket in order to mark out the surfaces. A contour of the bead ring will also be sketched on the surface of the bracket, because the reaction force will be applied to this surface.

The bracket consists of steel plates connected by full penetration welds. Therefore, the welding seams have been created by uniting all steel plates so that they are modelled as one solid piece. However, the picture on the next page shows two seams which have to be modelled in order to get a realistic structure.

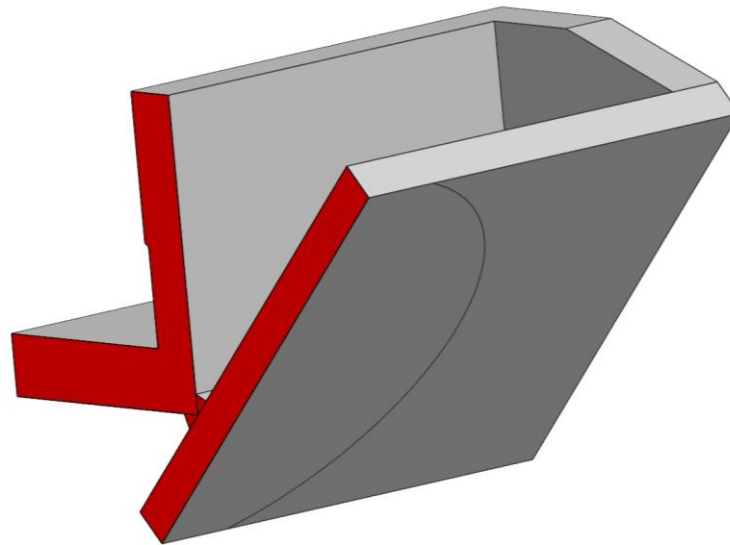


Figure 16. A section view of the bracket with the applied seams in the middle. The semicircle is the contour of the bead ring.

#### 4.4.2 Defining a material and creating the mesh

Rolled steel was selected as material since the structure consists of steel plates. The material properties have to be modified in order to match the properties for the selected material, S275. As explained earlier, the yield strength of the material will decrease with rising temperature. Based on both material thickness and a reference temperature of 55 °C, the selected yield strength is 213 N/mm<sup>2</sup> according to the diagram below.

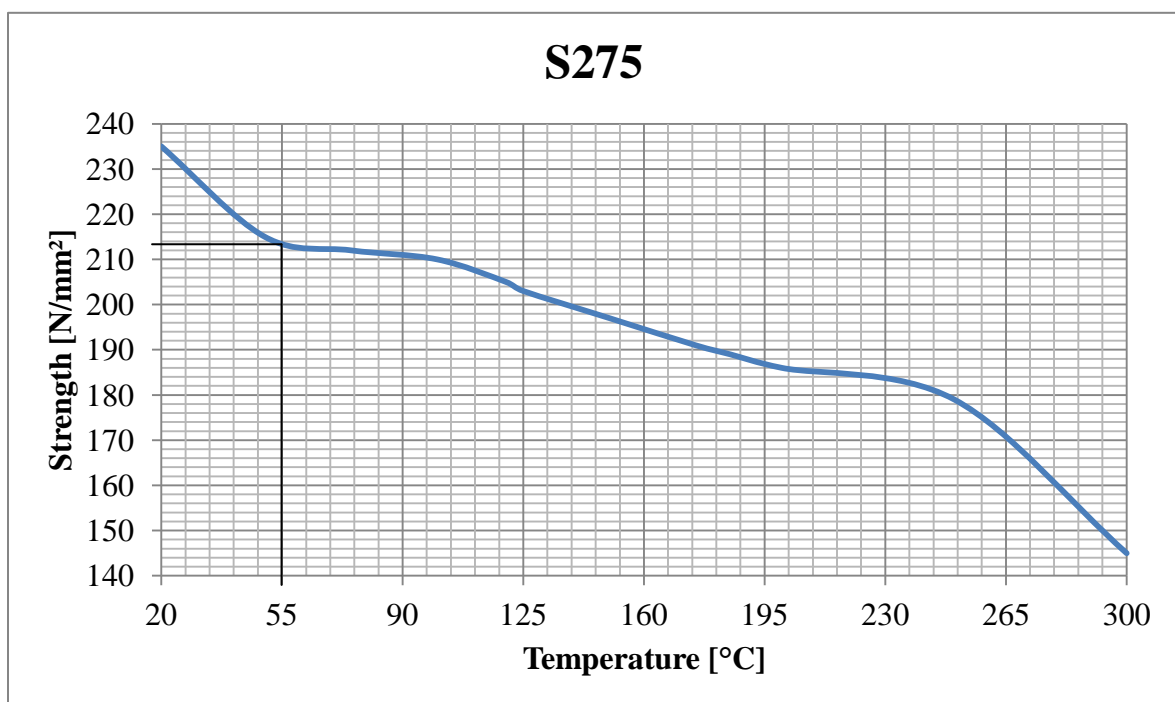
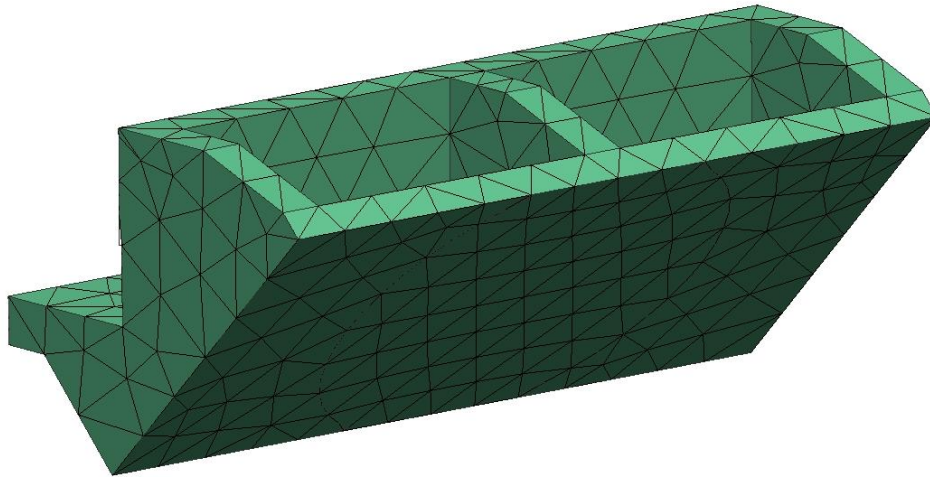


Figure 17. Material strength versus reference temperature. (Valtanen 2010, p. 1114).

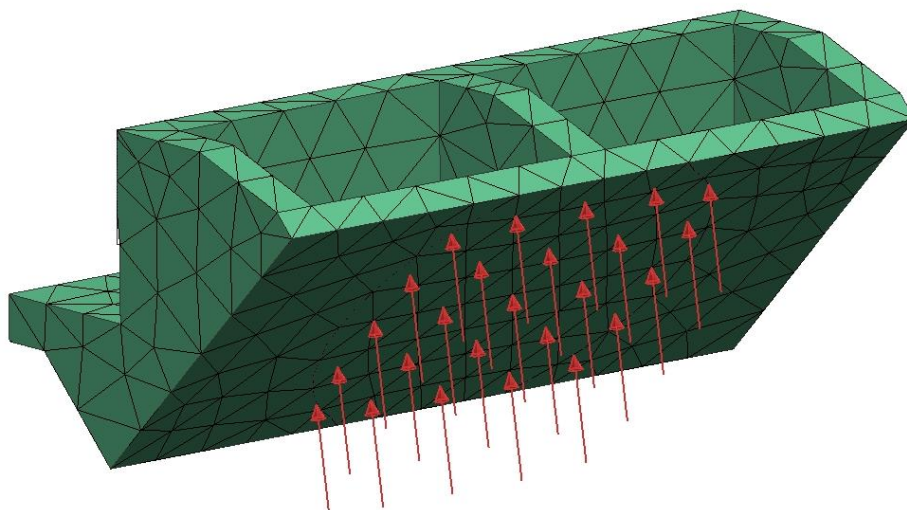
Furthermore, a few parameters are required to create the mesh. Citera 10 is specified as the tetrahedral element type. The number 10 comes from the amount of nodes in each tetrahedral element. Citera 4 would be another option, but this type is too stiff for a structural analysis due to a small amount of nodes. The size of each element is defined as 78.7 mm by the software. A mesh consisting of about 1500 elements is finally created. A finite element check ensures that all elements have been successfully created. The picture below shows the created mesh.



*Figure 18. A finite element mesh consisting of tetrahedral elements.*

#### **4.4.3 Applying loads and constraints**

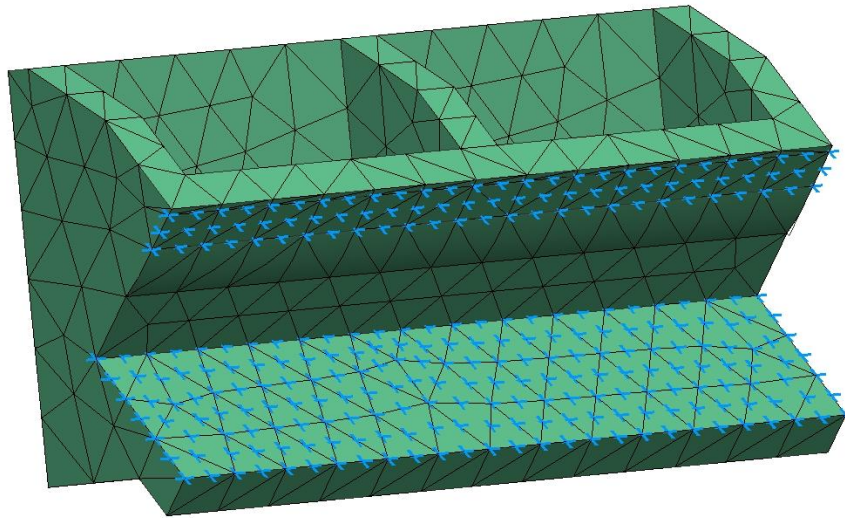
The next step is to apply loads and constraints. The reaction force is applied to the surface of the bracket with the same inclination angles as in the worst case scenario, i.e.  $22.5^\circ$  roll and  $10^\circ$  trim. The force is spread over a surface equalling that of the bead ring.



*Figure 19. The reaction force applied to the surface of the bracket.*



Applying constraints means that you fix nodes in the mesh in one or several degrees of freedom. In this case, the surfaces touching the engine block and the flat bar are to be considered fixed constraints. This means that the surfaces will not be able to move in any direction during the simulation. The picture below shows the surfaces fixed by constraints.



*Figure 20. Lines marked with blue are fixed constraints.*

#### **4.4.4 Solving the model and viewing the analysis results**

Since the mesh consists of 3D elements, an iterative solver is recommended. The total amount of equations required to solve the model grows rapidly when 3D elements are used. An iterative solver provides an indirect solution of the model and a maximum number of iterations must therefore be specified. The maximum number of iterations is set to 999 in this case. The model can finally be solved and the results are presented in the next chapter.

## 5 Results

In the following chapter the results of the FEA will be presented in two sections: initial and final results. The final design of the bracket and an illustration of the mounting system are presented in figures 26 and 27. A cost calculation will be carried out at the end of the chapter.

### 5.1 Initial results

The result of the FEA is presented in Appendix 2. The overall model confidence level is 84 %. The calculated Von Mises stress is  $79.4 \text{ N/mm}^2$ . Comparing the stress with the yield strength of the material ( $213 \text{ N/mm}^2$ ) indicates that the system has not exceeded the elastic state. This confirms that the deformation of the system is recoverable. As stated before, one important thing when examining the results is to determine if the deformation of the model is reasonable. The deformation might seem quite huge in this case, because it is automatically scaled 10:1 in order to clarify the displacement. One should observe that the maximum stress is concentrated on a very small area. This is confirmed by comparing the colours with the colour legend in Appendix 2. The location of the maximum stress can be seen in the picture below.

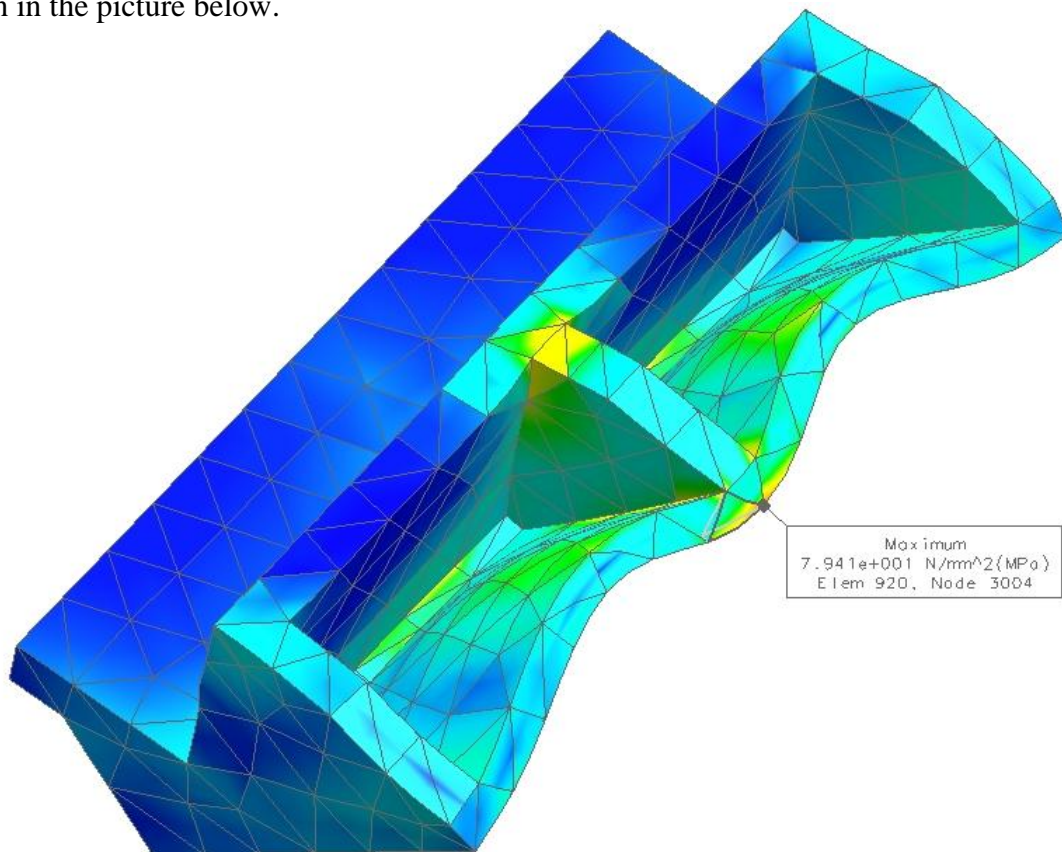
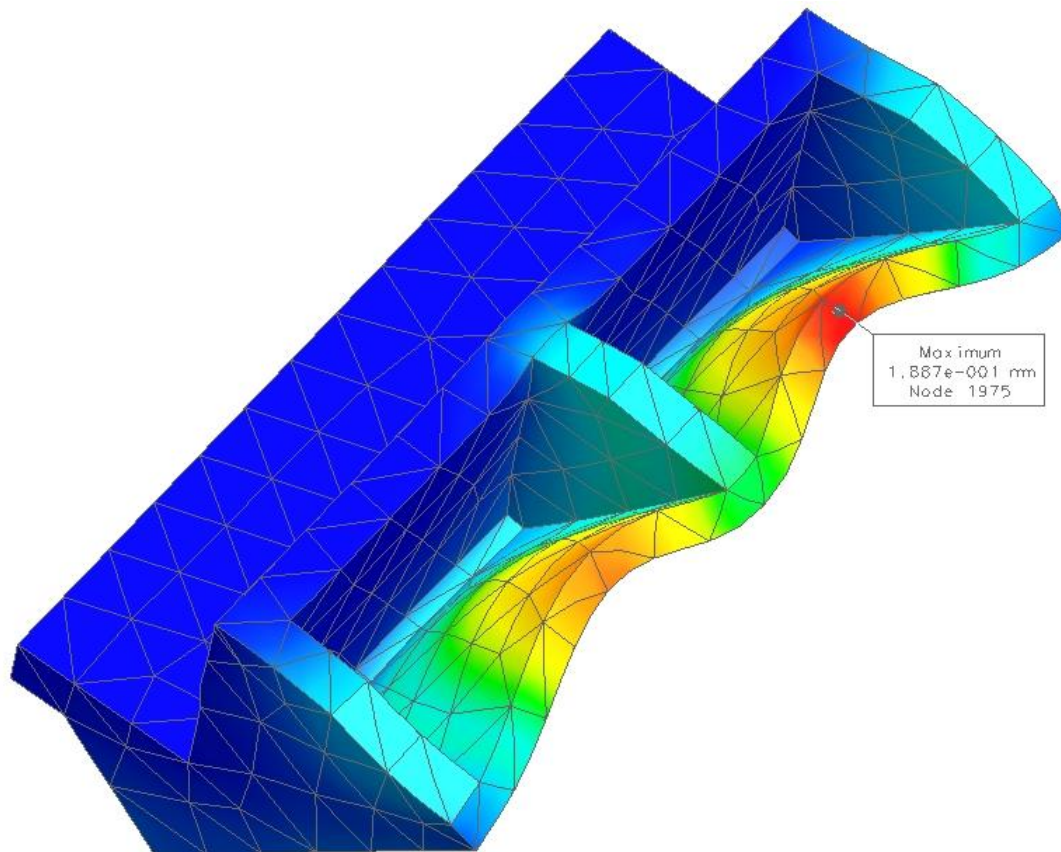


Figure 21. Location of the maximum stress.

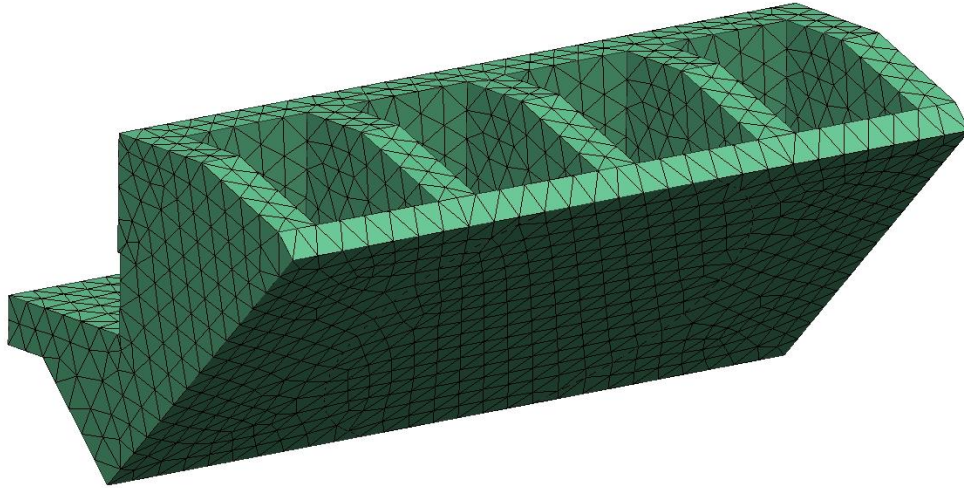
Besides confirming that the system remains in the elastic regions for the applied loading, the displacement must also be considered. The calculated displacement is 0.19 mm. This is not a very big displacement, but it indicates that the bracket is quite flexible. Flexibility is a desired property for an engine mount, but not for an engine bracket. Therefore, the bracket geometry will be improved in order to make it stiffer. The picture below shows the location of the maximum displacement.



*Figure 22. Location of the maximum displacement.*

## **5.2 Geometry improvement**

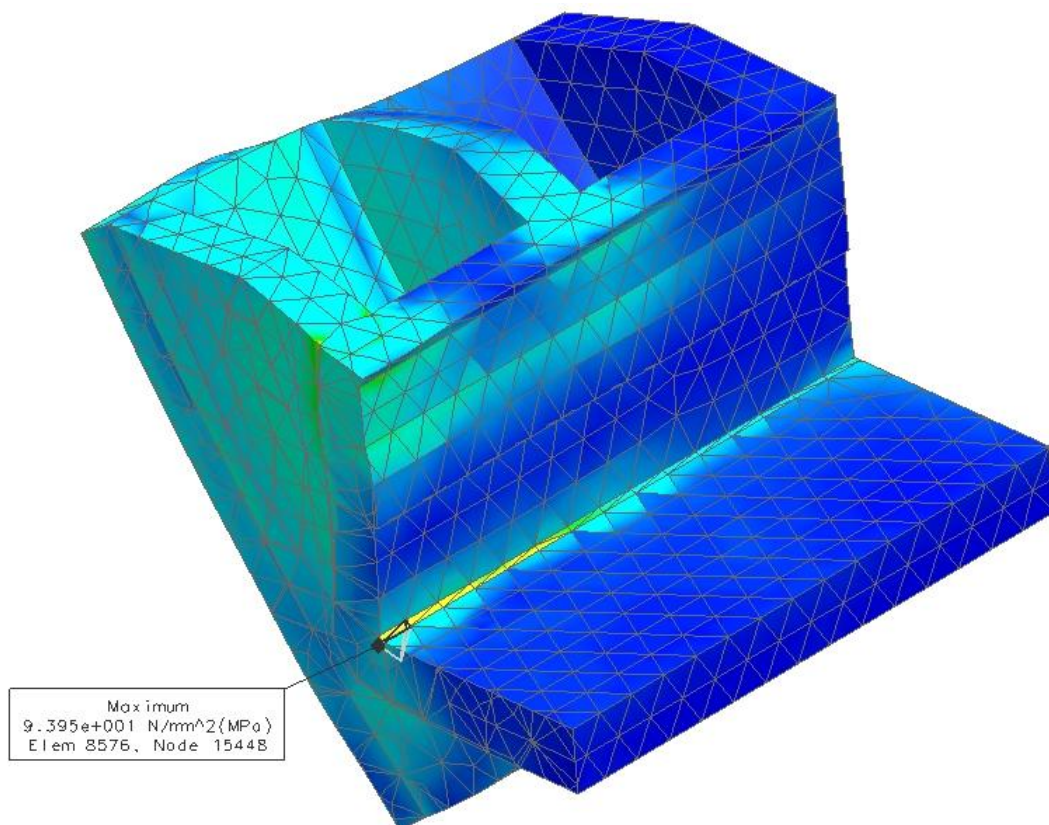
The geometry is improved by adding perpendicular steel sheets in the area where the maximum displacement is located. The constraints and the applied force remain as in the previous simulation. Since further mesh refinement was recommended, the automatic element size will now be halved. The picture on the next page shows the new refined mesh consisting of 8700 tetrahedral elements.



*Figure 23. A refined mesh with an increased number of steel sheets.*

### **5.3 Final results**

The result of the second FEA is presented in Appendix 3. The overall model confidence level has now increased to 88 %. The calculated Von Mises stress is  $94.0 \text{ N/mm}^2$ , and the location has moved compared to the previous simulation model. The picture below shows the location of the maximum stress.



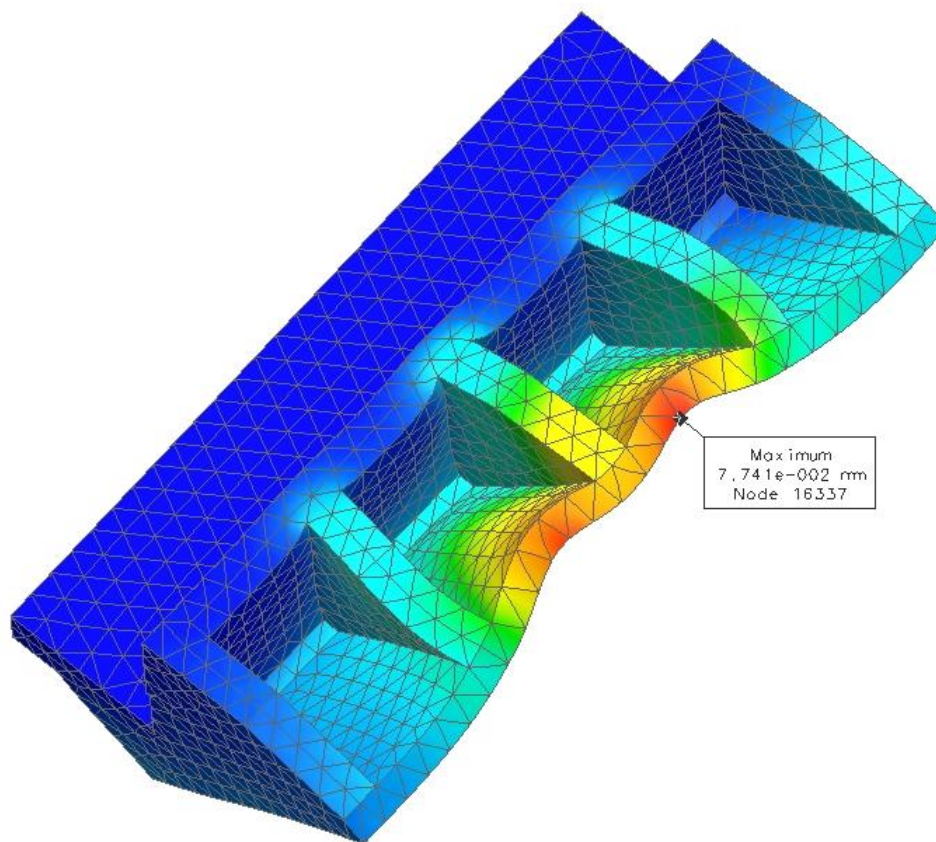
*Figure 24. A section view showing the location of the maximum stress.*

It might seem strange how the stress increases when the structure is made stiffer. The reason is that the stress is spread more efficiently due to a stiffer structure. This causes a concentrated stress in the perpendicular angle between the steel plates. Comparing the colours with the legend in Appendix 3 shows that the stress is only concentrated on the surface and spread efficiently inside the structure.

This deformation of the system will be recoverable since the maximum stress does not override the yield strength of the material. A safety factor can be calculated when the stress and the yield strength are known. Using a minimum safety factor of 1.5 is a general rule of thumb when designing steel structures. Based on the formula below, the calculated safety factor is 2.3.

$$S_f = \frac{\sigma_y}{\sigma_v}$$

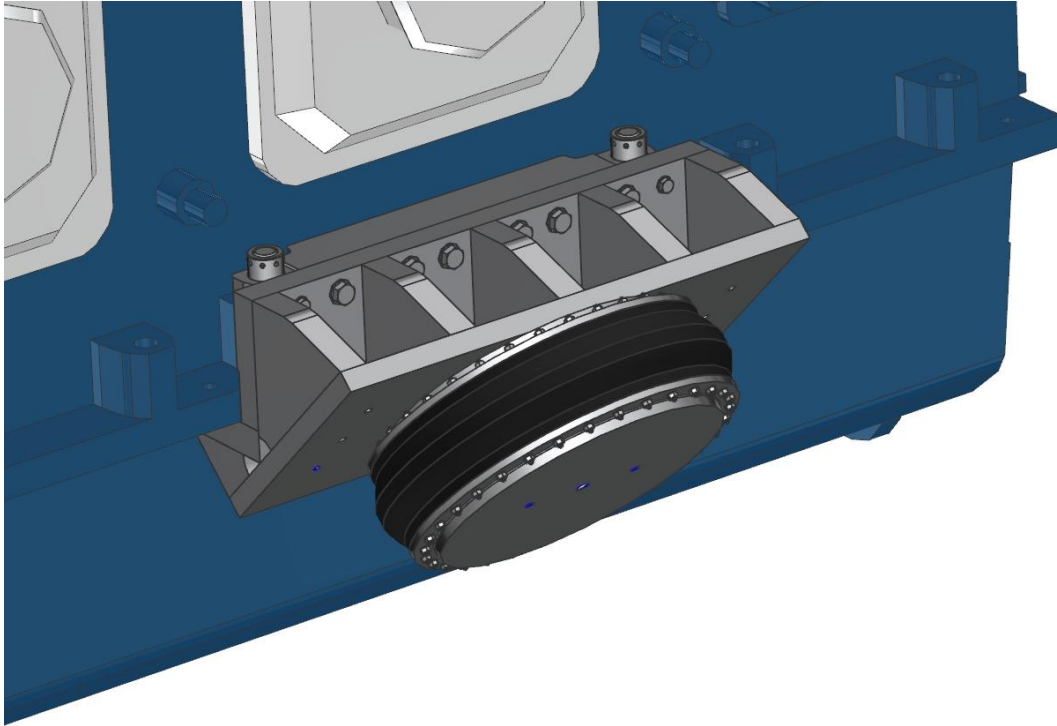
An increased stiffness of the structure can be confirmed by the displacement, which has decreased to 0.08 mm. The picture below shows the location of the maximum displacement.



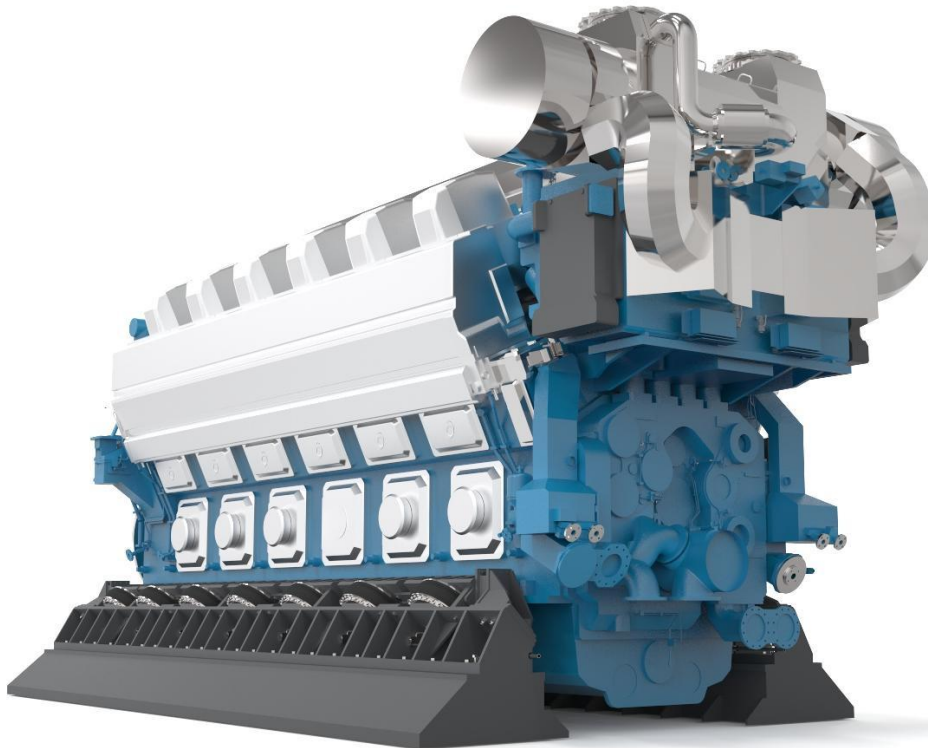
*Figure 25. Location of the maximum displacement.*

## 5.4 The final design

An appropriate design is finally obtained. The picture below shows the bracket connecting the air spring to the engine block. The second picture is an illustration of a resiliently mounted W12V46F engine with air springs as engine mounts.



*Figure 26. The bracket connected to the engine block.*



*Figure 27. An illustration of a resiliently mounted W12V46F engine. (Wärtsilä R&D).*

## 5.5 Cost calculation

A production drawing of the bracket was made after the final design was obtained. A local subcontractor was contacted in order to get a cost estimate with regard to the following manufacturing costs:

- Raw material
- Welding
- Machining

The cost estimate was made according to the batch size required for a W12V46F engine configuration.

It was earlier stated that an advantage with separate brackets was that it would be a benefit from an economical perspective. The following cost calculation will prove the truthfulness of that statement.

The cost calculation is a comparison between two different options for a resiliently mounted W12V46F engine:

- a) The brackets designed as separate parts for a flexible mounting system using air springs. This is the design developed through this thesis. A 12V46F engine would require 14 separate brackets.
- b) The brackets designed as two long fixing rails for a flexible mounting system using conventional soft rubber mounts. This design is a prototype which has been designed and manufactured earlier by Wärtsilä. The fixing rails (one per engine side) consist of welded steel plates and will therefore be an appropriate structure to compare with.

The result of the cost calculation indicates that separate brackets would be 36 % cheaper to manufacture compared to long fixing rails. One reason is that the long fixing rails require more machining due to a large amount of rubber mounts. Another reason is that surface treatment was not considered in the cost estimate for the newly designed bracket. Adding the cost of surface treatment will probably make the newly designed brackets approximately 20 % cheaper to manufacture.

## **6 Conclusion**

This chapter describes how the goal was reached. The method and the results will be discussed from a slightly critical point of view. The utility of this study for the company is also presented. A suggestion for future research will be made at the end of the chapter.

### **6.1 Reaching the goal**

The initial phase was a thorough study of the demands made by different classification societies. This was time consuming but of great importance, since it ensures that the final design will be made with safety in mind.

The earlier performed studies have been two essential building blocks in this thesis. The calculation of the reaction force could easily be performed thanks to the force calculation tool. Furthermore, having a preliminary model of the structure was useful when the detailed design was carried out.

FEA was used to calculate the stress, the displacement and to ensure that failure will not occur under the effect of applied loading. After receiving the initial results, which proved that the bracket was too flexible, the model could easily be modified and solved again. The use of this tool was an effective way to obtain an appropriate design.

A production drawing was made and sent to a local manufacturer in order to get a cost estimate. The cost estimate was used to carry out a cost calculation, which compared the newly designed bracket with a similar part designed as a long fixing rail. The result indicated that designing the brackets as separate parts made the manufacturing cheaper in this case.



## **6.2 Use of FEA**

The FEA was carried out as a static load case since the force acting on the structure derived from slow but still semi-dynamic sea movements. The results proved that failure will not occur under the effect of applied loading. Besides this, a dynamic load case could have been carried out in order to simulate how the dynamic forces derived from the engine are absorbed by the structure.

One must be aware that performing an FEA does not guarantee correct results. Therefore, approximate calculations should be performed to validate the results. In this case, performing analytical calculations to calculate stress and displacement would have been very time consuming and challenging. Another way to validate the results would be to measure stress levels by means of “Strain Gauge” measurements. That would be a reliable but expensive way of validating the results.

## **6.3 Utility**

The engine mounting bracket developed through this thesis is one of the major steel parts in the engine mounting system. Obtaining a design that is appropriate from an economical, safe and durable point of view will hopefully be useful to Wärtsilä when it comes to future testing and development of the mounting system.

## **6.4 Future research**

A good continuation regarding future research would be to calculate the natural frequency of the bracket. Vibrations play a critical role in engine components, especially in the engine mounting brackets. Vibrations in this area may lead to structural failure if the vibrations are severe and excessive. This can be avoided by ensuring that the natural frequency of the bracket is well above the frequency band in which excitation exhibits most of the vibratory energy.

## 7 List of sources

American Bureau of Shipping (2006). *Rules for Building and Classing Steel Vessels*.

Det Norske Veritas AS (2012a). *Rules for classification of High Speed, Light Craft and Naval Surface Craft*.

Det Norske Veritas AS (2012b). *Rules for classification of Ships*.

Germanischer Lloyd (2012). *Rules for Classification and Construction - Ship Technology*.

Heinze P. & Schmidt I. (eds.) (2012). *Finite Element Method an Introduction*. Hochschule Wismar.

Lloyd's Register (2011). *Rulefinder Version 9.15 – Rules and Regulations for the Classification of Ships*.

Långbacka, F. (2012). *Mechanical design of advanced self-aligning mounting system*. Bachelor's degree in Mechanical Construction Systems. Novia University of Applied Sciences, Mechanical and Production Engineering, Vaasa.

Shih, R. (2004). *Introduction to Finite Element Analysis using I-DEAS 11*. Oregon Institute of Technology.

Valtanen, E. (2010). *Tekniikan taulukkokirja*. (18. Edition) Mikkeli: Genesis-Kirjat.

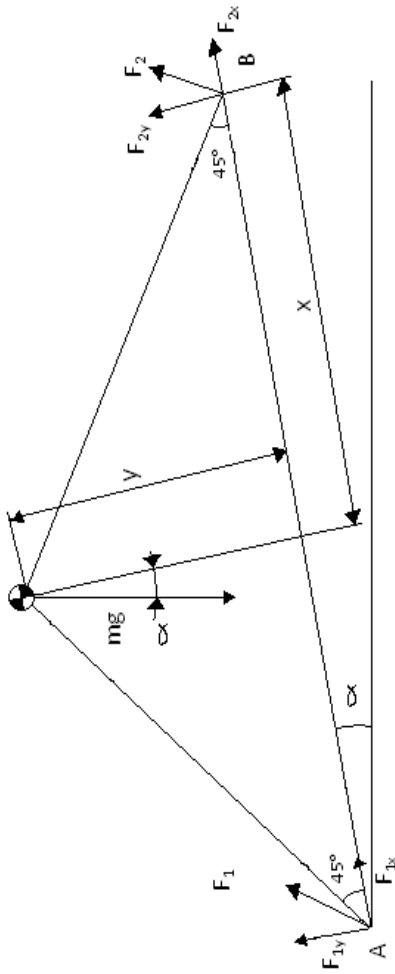
Wasberg, A. (2011). *Advanced self-aligning mounting system*. Bachelor's degree in Vehicle Technology. Novia University of Applied Sciences, Mechanical and Production Engineering, Vaasa.

Wärtsilä (2011). *Wärtsilä 46F - Product Guide*.

Roll	22,5 [°]
Trim	10 [°]
Element angle ( $\alpha$ )	45 [°]
Engine weight	185000 [kg]
Torque	229200 [Nm]
Distance c.o.g to element center (horizontal)	1,3061 [m]
Distance c.o.g to element center (vertical)	1,7275 [m]
Element per engine side	5
Distance c.o.g to F element	1,474 [m]
Distance F to D element	4,2 [m]

Roll + Torque	
mg	362,97 [kN]
F <sub>m</sub> (to element)	17,55 [kN]
F <sub>1y</sub>	294,63 [kN]
F <sub>1x</sub>	69,45 [kN]
$\beta_1$	76,74 [°]
F <sub>2y</sub>	58,26 [kN]
F <sub>2x</sub>	69,45 [kN]
$\beta_2$	39,99 [°]

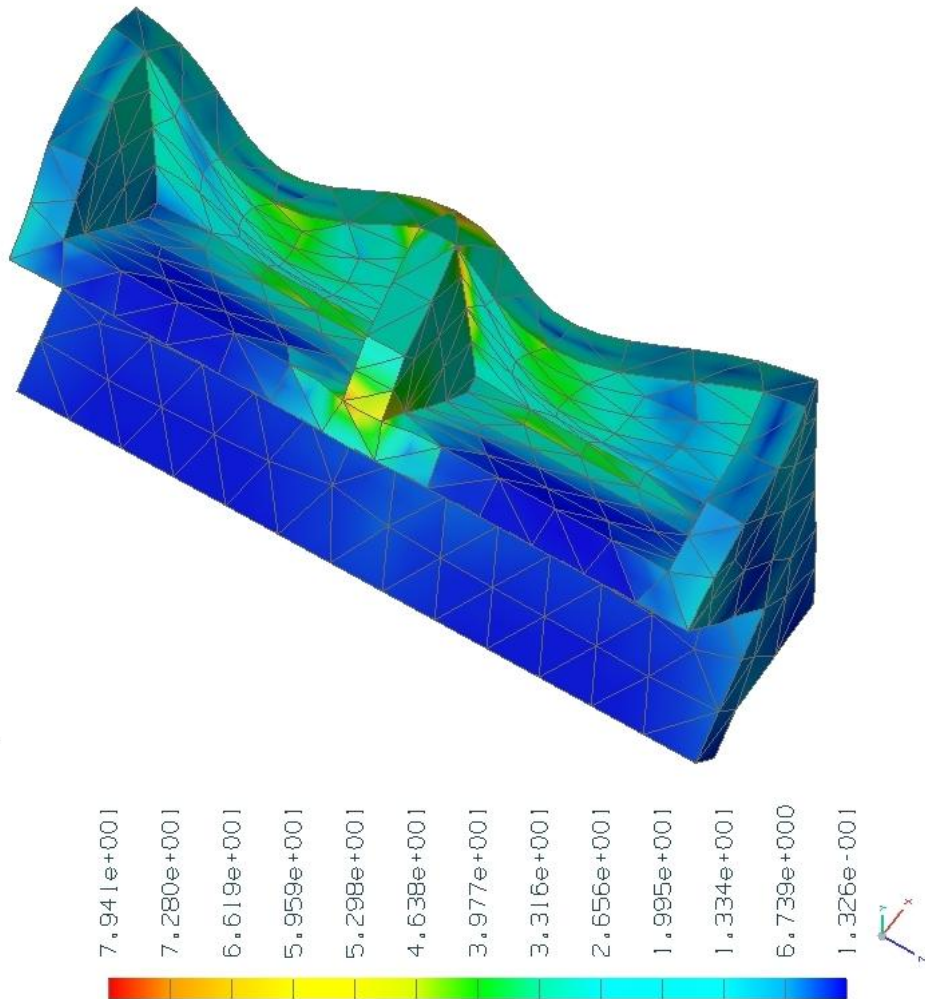
Roll and Trim + Torque	
x	0,30 [m]
l <sub>1</sub>	1,15 [m]
l <sub>2</sub>	3,05 [m]
m'g	661,88 [kN]
F <sub>1y</sub>	508,35 [kN]
F <sub>1x</sub>	126,64 [kN]
$\beta_1$	76,01 [°]
F <sub>2y</sub>	120,69 [kN]
F <sub>2x</sub>	126,64 [kN]
$\beta_2$	43,62 [°]



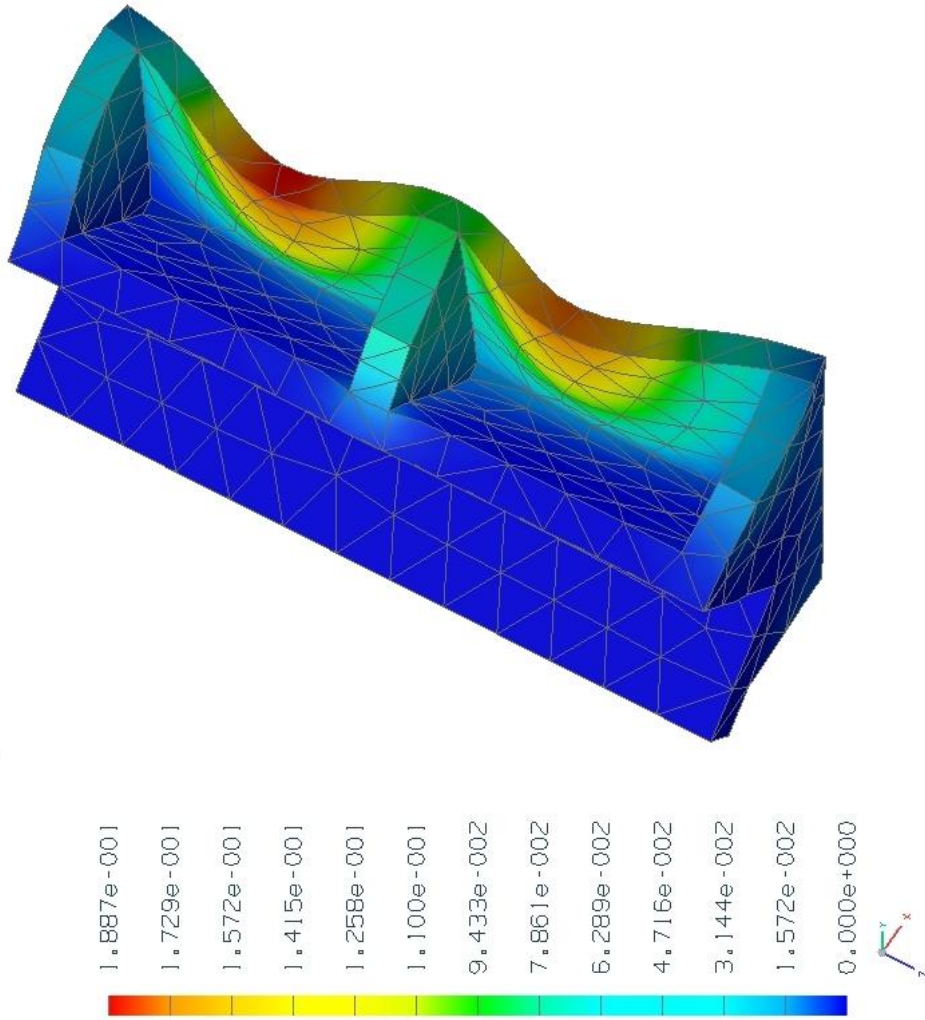
Case Roll		Bellow	
F1	416,66 [kN]	F1 axial	354,37 [kN]
F2	82,40 [kN]	F1 radial	219,17 [kN]
		F2 axial	82,08 [kN]
		F2 radial	-7,19 [kN]

Case Roll and Trim		Bellow	
F1	718,91 [kN]	F <sub>A</sub> axial	616,16 [kN]
F2	170,69 [kN]	F <sub>A</sub> radial	370,38 [kN]
		F <sub>B</sub> axial	170,64 [kN]
		F <sub>B</sub> radial	-4,11 [kN]

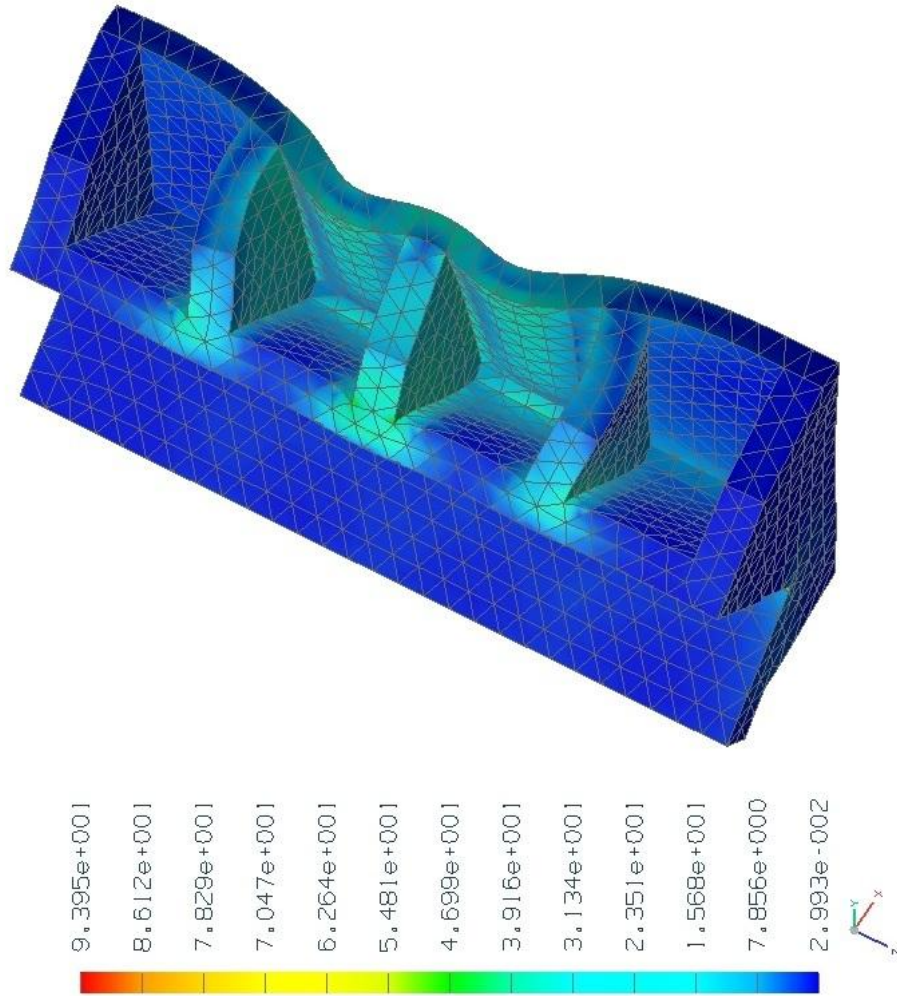
Middle bracket FEM\_sim1 ; Solution 1 Result  
 Load Case 1, Static Step 1  
 Stress - Element-Nodal, Unaveraged, Von-Mises  
 Min : 1.326e-001, Max : 7.941e+001, N/mm<sup>2</sup>(MPa)  
 Deformation : Displacement - Nodal



Middle bracket FEM\_sim1 ; Solution 1 Result  
 Load Case 1, Static Step 1  
 Displacement - Nodal, Magnitude  
 Min : 0.000e+000, Max : 1.887e-001, mm  
 Deformation : Displacement - Nodal



Middle bracket FEM 2\_sim] ; Solution 1 Result  
 Load Case 1, Static Step 1  
 Stress - Element-Nodal, Unaveraged, Von-Mises  
 Min : 2.993e-002, Max : 9.395e+001, N/mm<sup>2</sup>(MPa)  
 Deformation : Displacement - Nodal



Middle bracket FEM 2\_sim] ; Solution 1 Result  
 Load Case 1, Static Step 1  
 Displacement - Nodal, Magnitude  
 Min : 0.000e+000, Max : 7.741e-002, mm  
 Deformation : Displacement - Nodal

