University of Stavanger

Faculty of Science and Technology

MASTER'S THESIS

Study program/ Specialization: **Constructions and materials** **Specialization: Offshore constructions**	Spring semester, 2014 Open
Writer: **Morten Ljosland Løland**	……………………………………… (Writer's signature)

Faculty supervisor:
Sudath C Siriwardane
External supervisor(s):
Dag Holen, Cameron Sense
Frode Stakkeland, Cameron Sense

Thesis title:

Optimization of wire sheave

Credits (ECTS):
30

Key words: **Optimization** **Wire sheave** **ANSYS** **Autodesk Inventor** **Stress** **Buckling** **Fatigue**	Pages: **104** + enclosure: **141** Stavanger 04.06/14 Date/year

PREFACE

This thesis marks the end of my two-year master's program in construction and material with specialization in offshore constructions at the University of Stavanger.

The thesis is to look at the possibility of an optimized 72" (1828,8 mm) wire sheave, and is written in collaboration with Cameron Sense in Kristiansand.

I would like to thank Cameron Sense for sharing information, and having offices and software at my disposal. I also want to thank my instructor at UiS, Sudath C. Siriwardane for his support on this work, and Dag Holen and Frode Stakkeland at Cameron Sense for all guidance, help and discussions during my work.

Kristiansand, 21. Mai 2014

Morten Ljosland Løland

SUMMARY

The objective of this thesis is to optimize a 72” (1828,8 mm) wire sheave with respect to weight and rotational inertia. The sheaves are used as pulleys on derrick structures. Seven different design concepts were designed and analyzed. Table 1 shows the different concepts with their capacities.

Table 1: Results

Concept	Stress capacity	Buckling capacity	Fatigue capacity	Weight [kg]	Moment of inertia [kg*mm²]
1. Double web	52 %	35 %	102 %	647	$2{,}46*10^8$
2. Straight web	72 %	63 %	70 %	622	$2{,}38*10^8$
3. Straight web w/holes	82 %	66 %	67 %	531	$2{,}02*10^8$
4. Double web w/holes	44 %	26 %	84 %	620	$2{,}32*10^8$
5. Thin web w/ stiffeners	70 %	73 %	72 %	531	$1{,}97*10^8$
6. Thin web w/ stiffeners and holes	96 %	71 %	72 %	516	$1{,}92*10^8$
7. Web with decreasing thickness	82 %	84 %	73 %	520	$1{,}90*10^8$

The design concepts may be described as follows:

1. The first design concept, double web, was given by Cameron Sense as an educational purpose to learn the different programs being used. The result of the analysis shows good capacity against side loads and buckling, but the weight is relatively high.

2. The next concept, a straight web, had to be thick enough to withstand the stress at the inner edge. Therefore the weight was not reduced much.

3. In the third concept holes were made in the straight web. Since the stresses increase towards the center of the sheave, the holes had to be put as close as possible to the outer edge of the sheave. The weight was reduced by 90kg when adding eight holes around the sheave.

4. The fourth concept consisted of the double web sheave from Cameron, but with holes in it. Since this concept consisted of two thin webs, cylinders (sleeves) had to be placed in the holes to avoid local buckling. The weight was not reduced much due

to the weight of the cylinders that were added.

5. The next two concepts were a thin web with six stiffeners around the sheave and one with holes between the stiffeners. These concepts had low weight, but the results are questionable. The side load was applied right above one of the stiffener. When the sheave rotates, this load will act on different places, also in between the stiffeners. This will give bending stresses at the web as well. Therefore more investigations will be required to conclude these concepts.

6. The last concept was having a web with a decreasing thickness. With a weight of 520,1kg this was, beside from the concept with thin web and stiffeners and holes, the sheave the lowest mass.

The sheave with decreasing thickness of the web had low weight and good utilization on all the disciplines that were checked. The fatigue capacity was pretty high on all the concepts when checking with the D-curve from DNV. The spectrum supplied by Cameron Sense is intended to drill 100 deep wells over a period of 20 year, which results in 200 million fatigue cycles. When the traveling block is moving up, from the work that it has done, the sheave is not fully loaded. This results in a reduction of fatigue cycles. The fatigue cycles are reduced to 100 million cycles. This gives a higher allowable stress for fatigue. Making design concept 2,3,5,6 and 7 casted will result in a component with no welds. This gives a C-curve and a reduction factor for the stress range. With the new stress range, and the new S-N curve, the fatigue capacities on most of the sheaves are considered acceptable.

Optimizing sheaves and other equipment is essential, since the world is facing difficulty with cost of new constructions. The light sheave that Cameron Sense is using weighs 646,5 kg, but most of the sheaves being used weighs about 800 kg. Using the sheave with a decreasing thickness on the web, the weight is reduced with 126,4 kg from the light sheave Cameron uses, and approximately 280 kg from the regular sheaves being used. In a 14 parts system this will reduce the total weight of sheaves by 3,92 tons and 4,48 tons in a 16 parts system.

ABBREVIATIONS AND SYMBOLS

$DNV = Det\ Norske\ Veritas$

$API = American\ Petroleum\ Institute$

$FE = Finite\ element$

$parts\ system = many\ sheaves\ in\ order\ to\ haul\ more\ load\ with\ the\ same\ input\ force$

$F = wire\ load$

$F_h = side\ load\ from\ wire$

$D = diameter\ of\ the\ sheave$

$f_y = yield\ stress$

$SF_D = safety\ factor$

$M_{Ed} = moment$

$M_{Rd} = moment\ capacity$

$H = moment\ arm$

$P = pressure\ on\ the\ groove$

$F_a = allowed\ stress$

$\mathrm{k} = equivalent\ stress\ rangefactor$

$\Delta\sigma_{cap} = fatigue\ capacity$

$\Delta\sigma = stress\ range\ for\ fatigue\ from\ ANSYS$

$N = total\ number\ of\ fatigue\ cycles$

$F_E = Euler\ load$

$b = hot\ spot\ stress$

$\Delta\sigma_N = Normal\ stress$

$\Delta\sigma_b = bending\ stress$

$\alpha = buckling\ load\ multiplier$

$F_y = Reaction\ force\ vector\ in\ y - direction$

$F_z = Reaction\ force\ vector\ in\ z - direction$

$F_x = Reaction\ force\ vector\ in\ x - direction$

$F = total\ reaction\ force\ vector$

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

1.0 INTRODUCTION

Inside a traveling block there are many sheaves, additional to the sheaves on top of the derrick. The more sheaves we have, the better mechanical advantages we have. An ideal block with a moving block supported by *n* rope sections has the mechanical advantage:

$$\frac{F_B}{F_A} = n$$

Where F_A is the hauling force, or input load, and F_B is the load. [1]
An actual sheave have force loss due to friction and moment of inertia. Especially in floaters, when the sheaves have to compensate for waves, the moment of inertia will cause the sheaves to lose some of the mechanical advantage.
The weights of these sheaves are large, and they are located on top of the derrick up to 125m above sea level. The additional forces on the extra weight from these sheaves gives larger members on the derrick to transfer the forces down through the derrick. Today it is very important to optimize structures as the world is facing difficulty with cost of new constructions. By making the sheaves and other equipment lighter, the derrick can also be optimized.

1.1 BACKGROUND

Cameron is a leading provider of flow equipment products, systems and services to worldwide oil, gas and process industries. [2]
In Kristiansand they have close up to 600 employees and is one of the contributors in the Norwegian Offshore & Drilling Engineering (NODE) cluster that together makes southern Norway a market leader with a market share of more than 90% of all drilling technology globally. [3]
Cameron Sense uses many large wire sheaves in their major drilling packages.
In the 1000 tons system, 72” (1828,8 mm) sheaves are typically used. Eight sheaves in the traveling block, and seven plus four sheaves on top of the derrick.

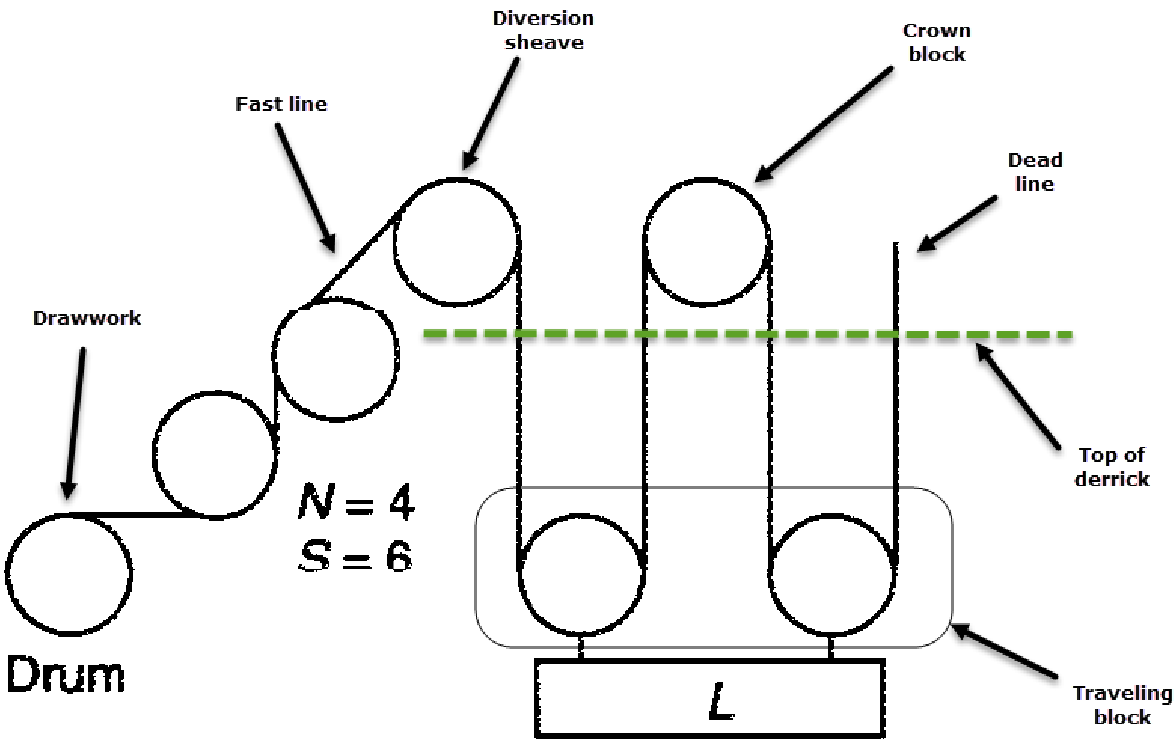


Figure 1: Sheaves on derrick [4]

Figure 1 above shows a line string up and the placement of the sheaves. The traveling block is moving up and down, while the other sheaves are fixed. The figure also shows how the drill line moves to get the mechanical advantage.

These sheaves are customized for a 2” wire (50,8 mm) drill line, which have a maximum load on wire on a 16 parts system of approximately:

$$\frac{1.10*1000*10^3 kg}{16\ parts} \approx 69\ tons.$$

Each sheave weighs about 800kg. Since the speeds of the fastest sheaves are very large, the rotational inertia of the sheave is also large and unfortunate. Because of the high rotational inertia, it takes more torque to increase or stop the rotation.
Cameron usually buy complete certified wire sheaves, including design approval. According to Cameron’s review of this design, the theoretical design life of these sheaves may be limited and shorter than the full design life for the drilling package.
When a new sheave concept is evaluated, it is very important to account all relevant aspects in the design. The current design is a simple welded sheave type with double web. This

double web makes the rim stable sideways and the sheave can take large side loads.

The weld between the web and the hub must be made one-sided, with limited control of the weld root. This area has relatively high stresses, and this type of weld may potentially have a relatively short fatigue life. Calculations show that the sheaves must rotate more than 200 million times over a 20 year period, but the history also show that the main problem with the current type of sheave design has been wear of the wire groove and not fatigue cracks in the welds. In a few cases the double web sheave has been split in two due to excessive wear of the groove. The double web design gives a relatively heavy sheave as each of the two webs must be thick enough to avoid buckling due to compression. And as the stress level increase towards the hub, the thickness of each web must allow a sufficiently long design life of the sheave.

1.2 OBJECTIVE

The objective of this thesis is to look at the possibility of an optimized sheave design concept with particular emphasis on reducing weight and rotational inertia. The sheave must be able to withstand 200 million fatigue cycles over a period of 20 years which is considered a desirable fatigue life for a wire sheave. The design should consist of:

- Stress analyses
- Evaluation of buckling strength
- Evaluation of the design life, fatigue.

In the evaluation of the different concepts, the stress level can be calculated with hand calculations and with simple FE models. The stress level must be compared with the allowable stress defined in API8C.

Buckling should be checked with easy, but accurate, buckling calculations. (ANSYS is a suitable tool for this)

Fatigue life should be checked by using DNV RP C203 in combination with the design load spectrum given for this sheave. This design load spectrum will be supplied by Cameron based on their observations and assumptions.

From observations made in such a sheave design, wear and tear and sudden collapse due to fatigue cracks or sheave buckling is the area of concern. Excessive yield and plastic deformation have not been observed. The design must be based on the requirements given in DNV and API, as shown below:

Design Codes:

- DNV-OS-E101 Drilling Plant
- DNV Standard for cert. 2-22
- DNV RP-C203 Fatigue
- API 8A Drilling and Production Hoisting equipment
- API 8C Drilling and Production Hoisting Equipment (PSL 1 and PSL 2) (ANSI-API 8C-ISO 13535)

References:

- API Spec. 9A Wire Rope
- API RP 9B Application Care, and use of Wire Rope for Oil Field Service

2.0 THEORY AND METHODS

In the following chapter different theories being used are explained for dimensioning and designing the optimized sheave. Hand calculations are shown in section 5.0

2.1 FINITE ELEMENT METHOD

Finite element method assumes that a structure is built up by small elements held together by nodes. The nodes will cause displacements in different directions when the structure is subjected to forces, also known as degrees of freedom. The displacement on the nodes determines the stress and strain in each element.
The equation is expressed in the form of:

$$\mathbb{k}^e * \mathbb{d} = \mathbb{f}$$

Where $\mathbb{k}^e$ is the elements stiffness matrix, $\mathbb{d}$ is the nodal displacement vector of the element and $\mathbb{f}$ is the nodal displacement vector of the elements load vector. These vectors define all the displacements and corresponding forces in the elements node.
All the element stiffness matrixes are put together to a system stiffness matrix. Here the sum of the stiffness to all nodes is obtained. When all elements are put together to a system, the system will get stiffness in the nodes which are the sum of all the nodes stiffness in each element:

$$\mathbb{K} = \sum \mathbb{k}^e$$

And the constitutional relation system matrix is expressed in the form:

$$\mathbb{K} * \mathbb{D} = \mathbb{F}$$

Where $\mathbb{K}$ is the system stiffness matrix, $\mathbb{D}$ is the system displacements vector and $\mathbb{F}$ is the system load vector. [5]

2.2 DESIGNING FOR YIELDING

Yield strength is defined as the stress at which a material begins to deform plastically. Prior to the yield strength the material will deform elastically. When a material is deformed plastically it is permanent and non-reversible, but during the elastic deformation the material will go back to its original position. [6]

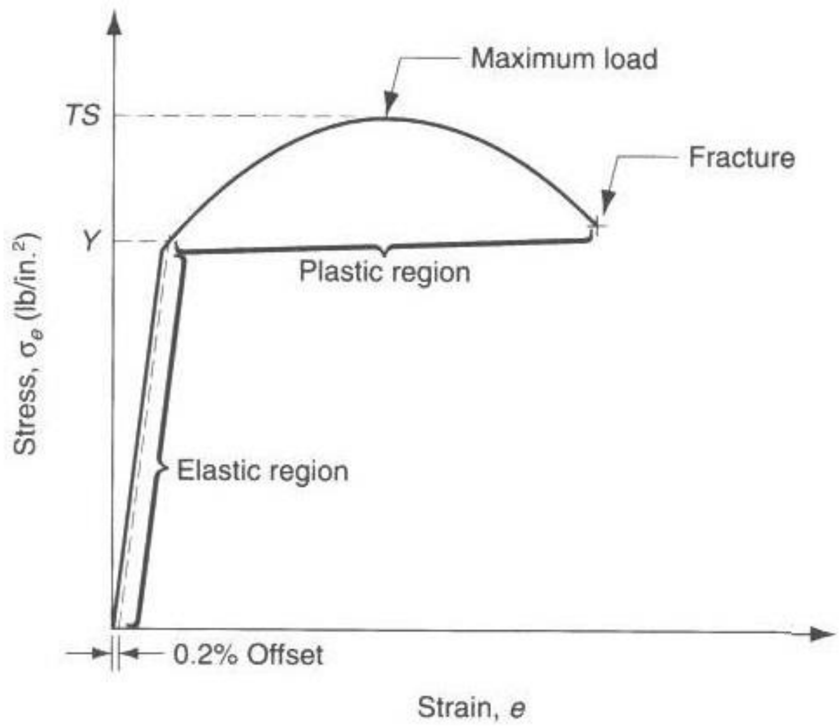


Figure 2: Stress strain behavior of ductile material/steel [7]

Figure 2 shows a typical yield curve from elastic deformation up to fracture.
von Mises yield criterion suggests that yielding of material starts when its von Mises stress reaches a critical value known as the yield strength, σ_y. The criterion is expressed as the following;

$$J_2 = k^2$$

Where k is the yield stress in pure shear, and J_2 is the second deviatoric stress invariant. In the case of pure shear stress, $\sigma_{12} = \sigma_{21} \neq 0$, while all other $\sigma_{ij} = 0$, the von Mises criterion then becomes;

$$\sigma_{12} = k = \frac{\sigma_y}{\sqrt{3}}$$

Where σ_y is the yield strength of the material. The von Mises stress is then set equal to the yield strength and combined with the equations above;

$$\sigma_v = \sigma_y = \sqrt{3J_2} \rightarrow \sigma_v^2 = 3k^2$$

Substituting J_2 with the Cauchy stress tensor components gives:

$$J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

Insert this in the von Mises yield criterion equation and the following equation for von Mises equivalent stress is obtained:

$$\sigma_v = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

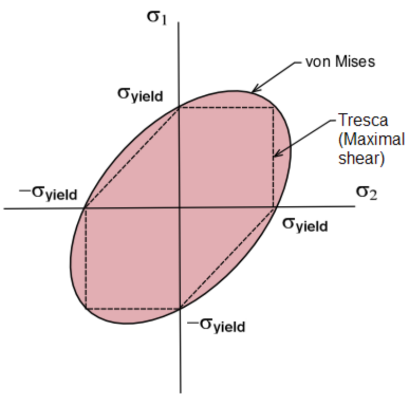


Figure 3: Intersection of the von Mises yield criterion [8]

The σ_v is the equivalent stress equation which is used to predict yielding of materials under multiaxial loading conditions using results from simple uniaxial tensile tests. [9]

2.3 DESIGNING FOR BUCKLING

Buckling is an instability leading to collapse or partially collapse. It is characterized by a sudden failure of a member subjected to high compressive stresses, where the actual compressive stress at the point of failure is less than what the material is capable of withstanding. As a mathematical analysis of buckling it is often used an axial load eccentricity that introduces a secondary bending moment, which is not a part of the primary applied forces on a member. When the load is increased it will ultimately become large enough to cause the member to become unstable and buckle. [10]

In 1757 Leonhard Euler derived a formula for maximal axial load a column can carry without buckling called the Euler load. The load is defined as the value of the axial load which makes that the straight, elastic column is located in an equilibrium position.
The derivation of this formula is based on equilibrium of the column in deflected state and requires the following, [11]:

- The column is straight and without imperfections
- The material is elastic and follows Hook's law
- Load attacks in the columns gravitational axis
- The column displacements are small

Since the column displacements are small, the slope $(w,_x)^2 \ll 1$. From this the curvature can be approximated to:

$$\frac{1}{R} = \frac{w,_{xx}}{(1+w,_x^2)^{\frac{3}{2}}} \approx w,_{xx}$$

This linearization of the curvature expression is the reason that the following theory is described as a linearized buckling analysis.

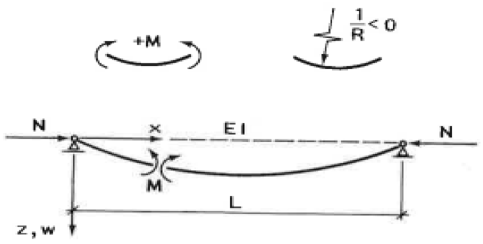


Figure 4: Simply supported column [11]Error! Reference source not found.

Figure 4 shows a simple supported column with directions for moment and curvature. In the deflected state the moment in a section x is given by:

$$M = N * w$$

With the relationship between moment and curvature, and with the directions of positive signs, the following relationship is made:

$$M = -\frac{EI}{R} \approx -EI * w,_{xx}$$

By combining the last two equations, the expression for the linearized differential equation becomes:

$$w,_{xx} + \frac{N}{EI} w = 0$$

Or:

$$w,_{xx} + k^2 w = 0$$

Where:

$$k^2 = \frac{N}{EI}$$

This equation is a homogenous, linear differential equation and has the solution:

$$w = c_1 \sin(kx) + c_2 \cos(kx)$$

The coefficients c_1 and c_2 must be decided by the boundary conditions at the edges of the column.

$$w = 0 \text{ at } x = 0, L$$

With these boundary conditions the solution to the differential equation becomes:

$$\begin{bmatrix} 0 & 1 \\ sin(kL) & \cos(kL) \end{bmatrix} * \begin{bmatrix} c_1 \\ c_2 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

This matrix only has a non-trivial solution when the determinant is zero. This again implies that

$$\sin(kL) = 0$$

This is only possible when:

$$kL = n\pi$$

$kL = n\pi$ represents the eigenvalue for the stability problem. The column will have an infinite amount of values of the axial force in equilibrium, the lowest of these is defined as the Euler load:

$$N_E = \frac{\pi^2 * EI}{L^2}$$

When the column is not simply supported we need to account for the forces at the edge, when we are calculating moment in a section:

$$M = N * w + M_A + V * x = -EI * w_{,xx}$$

This gives the inhomogeneous differential equation

$$w_{,xx} + k^2 w = -\frac{1}{EI} * (M_A + V * x)$$

The homogenous solution is still like the last differential equation, but the particular solution part is:

$$w_p = -\frac{1}{N} * (M_A + V * x)$$

The full equation gives the following:

$$w = c_1 \sin(kx) + c_2 \cos(\mathrm{kx}) - \frac{1}{\mathrm{N}} * (M_A + V * x)$$

The coefficients c_1 and c_2 must be decided by the boundary conditions for displacement, w, and slope, $w_{,x}$, as well as the equilibrium of the column in deformed condition. This will give the relationship between M_A and V.

The side load on the sheave from the wire will create a moment and an addition deflection. This can be considered as an imperfection on the sheave.

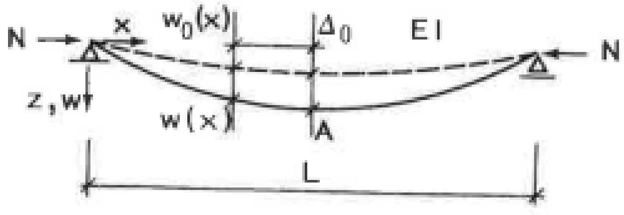


Figure 5: Column with initial deflection [11]

Figure 5 shows a column with an initial deflection or an imperfection in it, $w_0(x)$. By having this it gets an additional deflection, $w(x)$ after the axial load N is applied. The outer moment in a given section is:

$$M = N(w_o + w)$$

and the inner moment is proportional with the curvature from the additional deflection;

$$M = -EI * w_{,xx}$$

By combining these two equations and using $k^2 = \frac{N}{EI}$ the beams differential equation becomes;

$$\frac{d^2w}{dx^2} + k^2 w = -k^2 w_0$$

The homogenous solution for this equation is the same as the first differential equation above, but the particular solution is dependent on the pre deflections variation along the beam. Here it is assumed that the beam has the pre deflection:

$$w_o = \Delta_0 * \sin\frac{\pi}{L} * x$$

Because of the boundary conditions the additional deflection is assumed to be:

$$w = A * \sin\frac{\pi}{L} * x$$

Inserting these two equations into the differential equation and solved for the undetermined coefficient A, we get:

$$w = \Delta_0 * \frac{\frac{N}{N_E}}{1-\frac{N}{N_E}} * \sin\frac{\pi}{L} * x$$

The total displacement then becomes:

$$w_t = w_0 + w = \Delta_0 \left(1 + \frac{\frac{N}{N_E}}{1-\frac{N}{N_E}}\right) * \sin\frac{\pi}{L} * x = \Delta_0 \frac{1}{1-\frac{N}{N_E}} * \sin\frac{\pi}{L} * x$$

We see that the axial force N, have the effect that the pre deflection Δ_0 in enhanced with a factor f, which is:

$$f = \frac{1}{1-\frac{N}{N_E}}$$

Similarly the moment is:

$$M = -EI * w_{,xx} = EI \left(\frac{\pi}{L}\right)^2 \Delta_0 \frac{\frac{N}{N_E}}{1-\frac{N}{N_E}} * \sin\frac{\pi}{L} * x = N * \Delta_0 \frac{1}{1-\frac{N}{N_E}} * \sin\frac{\pi}{L} * x$$

The moment also gets enhanced with the factor f.

To check for secondary degree of buckling the effect from axial load and the effect from the moment has to be less than 1, the following equation has to be fulfilled:

$$\frac{N}{N_E} + \frac{M}{M_{Rd} * \left(1-\frac{N}{N_E}\right)} < 1$$

2.4 DESIGNING FOR MOMENT OF INERTIA

Moment of inertia is a measure of an objects resistance to changes in rotation direction. The formula for moment of inertia, I, can be defined by looking at a simple pendulum.

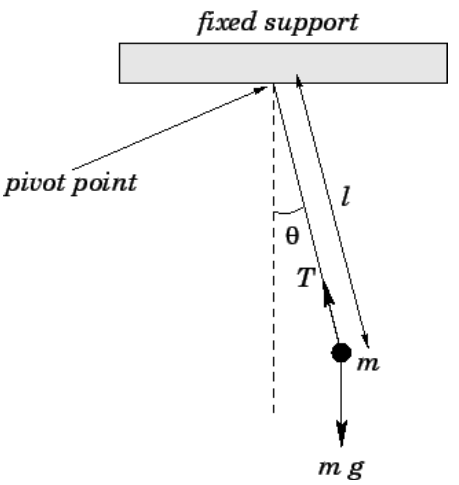


Figure 6: Schematic representation of pendulum [12]

It is a well-known fact that the pendulum is a point mass suspended with a string so that movement is constrained to a circle around a point.

Gravitational force on the mass generates torque on the pendulum around the axis perpendicular to the plane of the pendulum movement;

$$\tau = r \times F$$

Where F is the tangential component of the net forces on the mass, and r is the distance from the axis to the edge of the mass.

Associated with the torque there is an angular acceleration, α, of the string and mass around the axis. Since the mass is constrained to a circular movement, the tangential acceleration of the mass is $a = \alpha \times r$, and $F = ma$, so the equation for torque becomes;

$$\tau = r \times F = r \times (m\alpha \times r) = (mr^2)\alpha = I\alpha e$$

Where e is a unit vector perpendicular to the plane of the pendulum and I is the moment of inertia. $I = mr^2$ also appears in the angular momentum of a simple pendulum, which is calculated from the velocity $v = \omega \times r$ of the pendulum mass around its point. Here ω is the angular velocity of the mass around its point.

The angular momentum is given by;

$$L = r \times (mv) = (mr^2)\omega = I\omega e \qquad \text{(Similar calculation like above)}$$

Similarly the kinetic energy of the pendulum mass is also defined by the pendulum velocity around its point

$$E_k = \frac{1}{2}mv^2 = \frac{1}{2}(mr^2)\omega^2 = \frac{1}{2}I\omega^2$$

This shows that $I = mr^2$ is how the mass combines with the shape of a body to define rotational inertia.

Moment of inertia for an arbitrary shaped body is the sum of mr^2 for all the elements of mass in the body, expressed like

$$\int r^2\, dm$$

Where dm is mass of an infinitesimally small part of the body. [13]

2.5PROGRAMS

To be able to conduct this objective, different set of programs had to be used. Below the different programs are described.

2.5.1 ANSYS

ANSYS is convenient engineering simulation software for structural analysis.
Structures can be modeled in ANSYS or you can import geometry modeled in Inventor.
After the selected loads and boundary conditions are applied, ANSYS uses finite element method to calculate stresses and deformation in the structure.

2.5.2 INVENTOR

The optimized sheave design will be modelled in Autodesk Inventor.
Autodesk Inventor is 3D mechanical design software used to create 3D models used for visualization and simulation of products. After a product is designed in inventor it can be imported to other programs for stress calculations.

3.0 CRITERIA

The sheaves need to have a diameter of 72” (1828,8 mm) and be strong enough for the 1000t systems. The sheave groove should be customized for a 2” (50,8 mm) wire.
In addition to withstand the load from wire, the sheave also needs to withstand fatigue. The sheave must be able to rotate 200 million times with the design load spectrum, over a period of 20 years. The weld between the web and the hub is the area of concern.
The sheave must be designed according to API and checked with respect to API and DNV standards.

3.1 AMERICAN PETROLEUM INSTITUTE, API

The American Petroleum Institute is the largest U.S trade association for the oil and natural gas industry. Its main functions regarding the industry include:

- Advocacy and negotiation with governmental, legal and regulatory agencies
- Research into economic, toxicological and environmental effects
- Establishment and certification of industry standards
- Education outreach

Each year API distributes more than 200,000 copies of its publications. The publications are according to API itself. They are designed to help users improve the efficiency and cost-effectiveness of their operations, while they comply with legislative and regulatory requirements, safety and protect the environment. [14]

3.2 DET NORSKE VERITAS, DNV

Det Norske Veritas is an independent foundation working to safeguard life, property and the environment. It is the leading technical advisory company. They deliver classification, verification, risk management and technical advisory. [15]

4.0 LOADS

Table 1 in API Spec 8C shows the design safety factor for the sheave.

Table 1 — Design safety factor

Load rating R kN (ton)	**Design safety factor** SF_D
1 334 kN (150 short tons) and less	3,00
1 334 kN (150 short tons) to 4 448 kN (500 short tons) inclusive	3,00 – [0,75 × (R – 1 334)/3 114] [a] (3,00 – [0,75 × (R – 150)/350])[b]
Over 4 448 kN (500 short tons)	2,25
[a] In this formula, the value of R shall be in kilonewtons. [b] In this formula, the value of R shall be in short tons.	

Figure 7: Table 1 from API Spec 8C [16]

For a 1000 tons system a design safety factor of $SF_D = 2{,}25$ is needed.

According to API Spec 8C 4.3.5 the maximum allowable stress limit is then becomes:

$$Allowable\ stress_{max} = \frac{TS_{min}}{SF_D} = \frac{355\ Mpa}{2{,}25} = 158\ Mpa = F_a$$

Where TS_{min} is the tensile strength. All of the sheave concepts considered in this thesis must have a stress lower than $F_a = 158$ MPa.

4.1 LOAD ASSUMPTIONS

The sheaves are customized for a 2" wire (50,8 mm), drill line, which have a max load on wire approximately:

$$\frac{1{,}10*1000tons}{16\ parts} \approx 69\ tons \text{ for a 16 parts system.}$$

There is also a 14 parts system which gives a force:

$$F = \frac{1.10*1000*10^3kg}{14\ parts} \approx 78{,}5\ tons.$$

In this thesis a force of $F = 90\ tons$ will be used to account for other systems and uncertainties.

The load is applied where the wire contacts the bottom of the groove and is normal to the face in each point. API 8C Figure 6 shows the contact area of the wire, this corresponds to an angle of 150°.

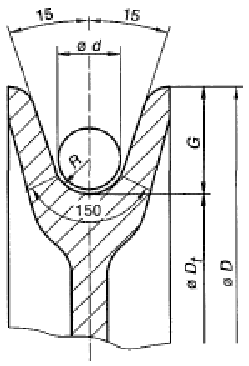


Figure 8: Sheave groove [16]

Pressure at the sheave groove is:

$$P = \frac{2F}{B*D}.$$

Where $2F$ the total line pull on the sheave, D is the outer diameter of the sheave and B is the contact surface in the groove. The variable needed to get the pressure is B.

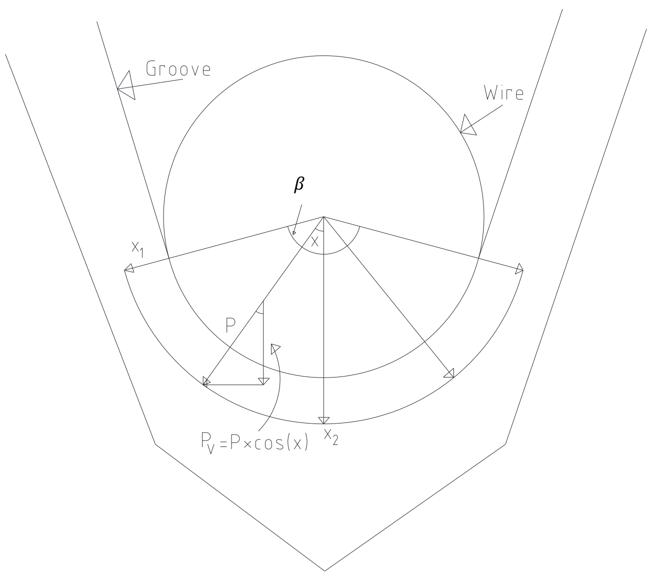


Figure 9: Wire contact surface

Figure 9 shows the wire with 150° contact surface, β. The pressure from the wire is distributed over the 150° contact surface. To get the exact pressure from the wire on the groove, a formula for the pressure has to be derived. From Figure 9 it is seen that the vertical component of the pressure, P, is:

$$P_v = P * \cos(x)$$

The distributed load on the sheave per unit length, q is:

$$q = \frac{2F}{D}$$

The sum of all the P_v components gives the pressure to be applied in the groove:

$$\int_{x_1}^{x_2} P_v * r_{groove} * d\theta$$

By taking the sum of the vertical components for half of the groove, this equation must be equal to half of the distributed load q. Combining the two equations above and using half the distributed load q gives the expression:

$$\frac{q}{2} = \int_{x_1}^{x_2} P_v * r_{groove} * d\theta \rightarrow \frac{F}{D} = P * r_{groove} * \int_{x_1}^{x_2} \cos(x) * d\theta$$

$$\frac{F}{D} = P * r_{groove} * [\sin(x)]_{x_1}^{x_2}$$

Half the pressure of the 150° contact surface goes from 0° to 75°, so the equation becomes:

$$\frac{F}{D} = P * r_{groove} * (\sin(75°) - \sin(0°))$$

From this the pressure is expressed in the form of:

$$P = \frac{F}{D * r_{groove} * \sin(\frac{\beta}{2})}$$

Where D is the diameter of the sheave, r_{groove} is radius in the groove, F is the force from wire and β is the contact surface angle from wire.

By inserting the known values the pressure applied on the groove is;

$$P = \frac{90000kg * 9{,}81\frac{m}{s^2}}{1684mm * 27mm * \sin\left(\frac{150°}{2}\right)} = 20{,}1\ MPa$$

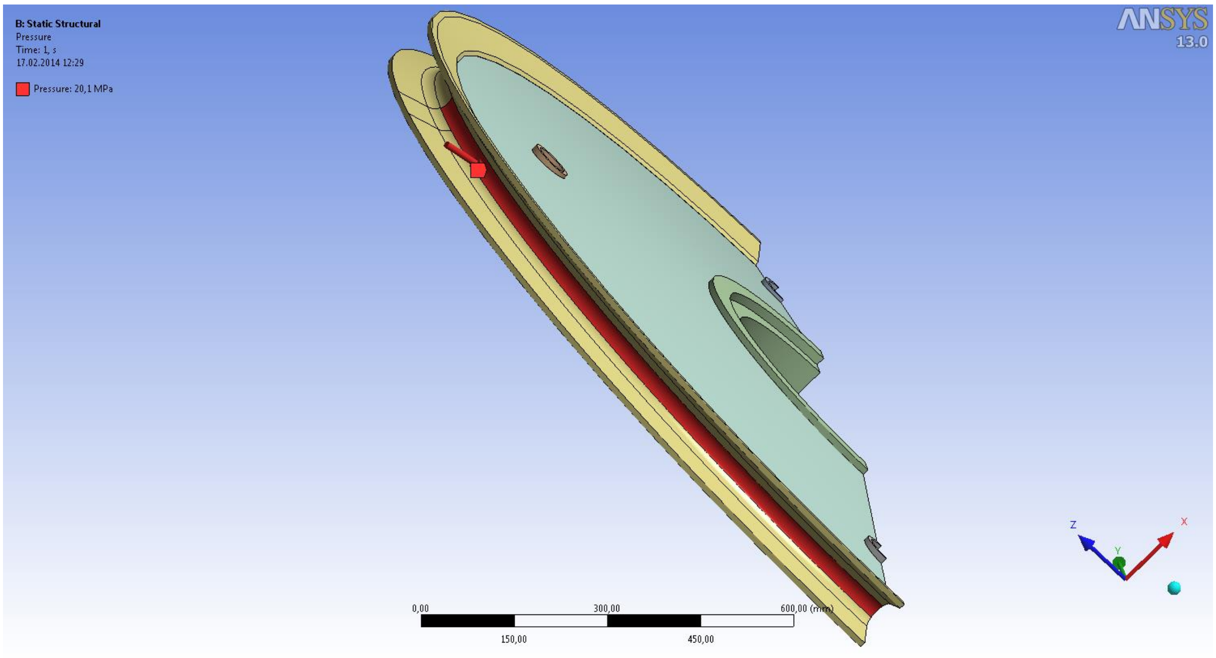


Figure 10: Pressure load from wire to contact face

Figure 10 shows the applied pressure in ANSYS.

An imprint of $r_{groove} * \sin\left(\frac{150°}{2}\right) = 27 * \sin(75) = 26{,}1\ mm$ wide is constructed on the groove to get the right pressure distribution.

4.2 SIDE LOAD

In addition to the pressure normal to the groove there is also a side load from the wire with a max inclination of 2°.

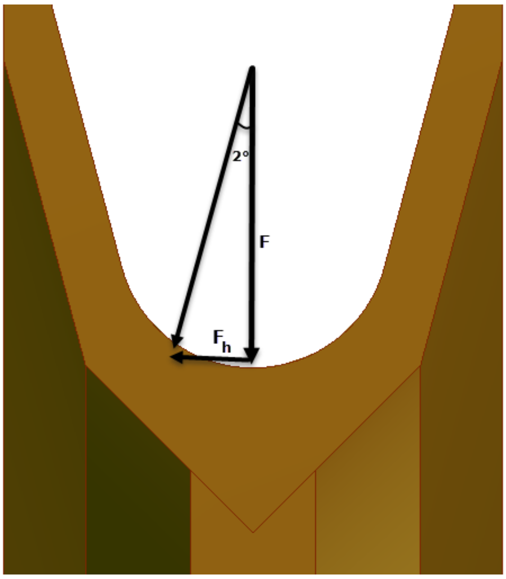


Figure 11: Side load in groove with 2° inclination

The inclination can vary from 0°-2°. The wire can have 1° inclination on each of the sides of the sheave, but having 2° inclination on one side is the worst case scenario.

2° inclination on one side gives a side load of:

$$F_h = F * tan(2°) = 882{,}9\ kN * tan(2°) = 30{,}83\ kN$$

The side load is applied where the wire leaves the sheave, which is the first 6,5° from the top of the sheave. 6,5° corresponds to a distance of;

$$L = 2\pi * r * \left(\frac{\theta}{360°}\right) = 2 * \pi * \frac{1684}{2} * \left(\frac{6{,}5°}{360°}\right) \approx 100\ mm.$$

The side load is therefore applied on a line that is 100 mm long. The pressure applied is:

$$p_{line} = \frac{F_h}{L} = \frac{30{,}83kN}{100\ mm} = \frac{308{,}3}{2}\frac{N}{mm} = 154{,}15\frac{N}{mm}$$

The load is divided by two since only half of a sheave is modeled.

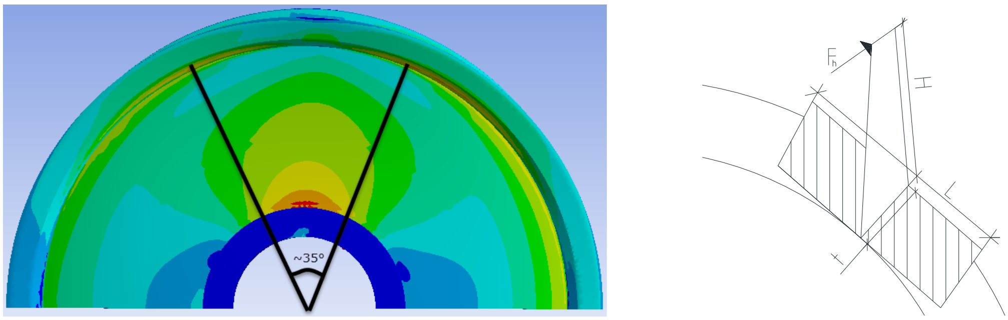


Figure 12: Bending stress

After running an analysis of a sheave with only the horizontal force acting on the sheave, bending stresses will act all over the plate. In this example the bending stress concentration is located in an angle of $\approx 35°$ as shown on the picture on the left in Figure 12. The picture on the right hand side shows an approximation of the cross section of the web. The shaded area represents the approximated area for which the concentrated bending stresses are located. The groove and the support are holding the web/plate, and when the horizontal side load then is applied the bending stress will be spread out over the entire plate. The bending stresses will act in different directions, so calculating them by hand is very comprehensive. As illustrated in Figure 12 the bending stress is distributed over the entire web.

To determine the moment capacity needed regarding second order effect of buckling, tests of different side load value in ANSYS will be carried out. These values and the corresponding stress, when only side load is applied, will be inserted in spreadsheet. The values are linearly extrapolated to make a graph. From this graph the moment capacity for each sheave can be found.

5.0 CALCULATIONS

This chapter shows the different hand calculations used in this thesis.

5.1 AXIAL LOADING

First the thickness of the web has to be established. Figure 13 shows the normal pressure and the diameter of the sheave.

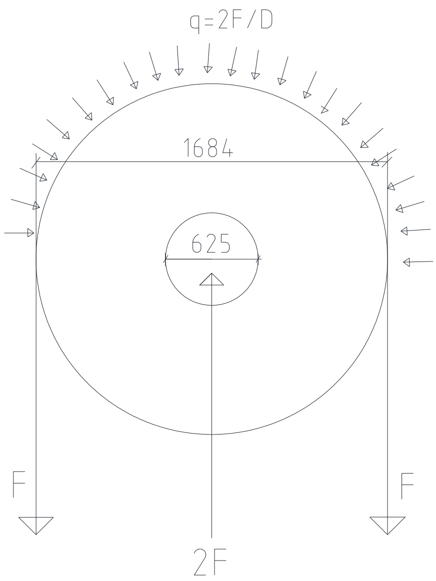


Figure 13: Simple load sketch

As described in the objective description, a 16 parts system is used. There is also a 14 parts system which gives a force; $F = \frac{1.10*1000*10^3 kg}{14\ parts} \approx 78{,}5\ tons$.

In this thesis a force, $F = 90\ tons$, will be used to account for other systems and uncertainties. The pressure at any given point of the sheave is:

$$\sigma = \frac{2F}{D*t}$$

Where D is the diameter and t is thickness of the web. The biggest impact on weight is the thickness of the web. Therefore the thickness is set as the variable.

$$t = \frac{2F}{\sigma*D}$$

With the safety factor $SF_D = 2,25$ as specified in API spec 8C the maximum stress becomes:

$$\sigma = \frac{f_y}{\gamma_m} = \frac{355\ MPa}{2,25} \approx 158\ MPa$$

Inserted in the thickness equation above, this gives the approximately thickness needed at the given diameter of the sheave;

$$t \geq \frac{2*90000\ kg*9,81\frac{m}{s^2}}{158\ MPa*D}$$

At the outer edge of the sheave the thickness has to be a minimum of;

$$t \geq \frac{90000\ kg * 9,81\frac{m}{s^2}}{158\ MPa * 1684mm} = 3,32mm$$

Thickness of the web at the inner edge of the sheave has to be:

$$t \geq \frac{90000\ kg * 9,81\frac{m}{s^2}}{158\ MPa * 600mm} = 9,3mm$$

From this it is seen that the stress increases towards the center of the sheave. The stress distribution on the sheave is shown below in Figure 14, where point 1 shows the stress distribution.

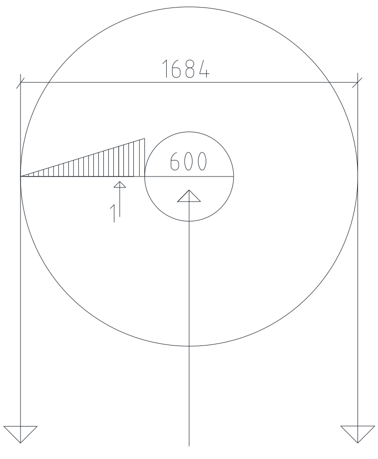


Figure 14: Stress distribution

5.2 SIDE LOADING

The side load also has an effect on the thickness of the plate. The side load creates bending stresses at the inner edge of the sheave.

It is this load that gives the effect of second order buckling. Figure 15 below shows an example of a graph obtained when different moments are applied on the sheave.

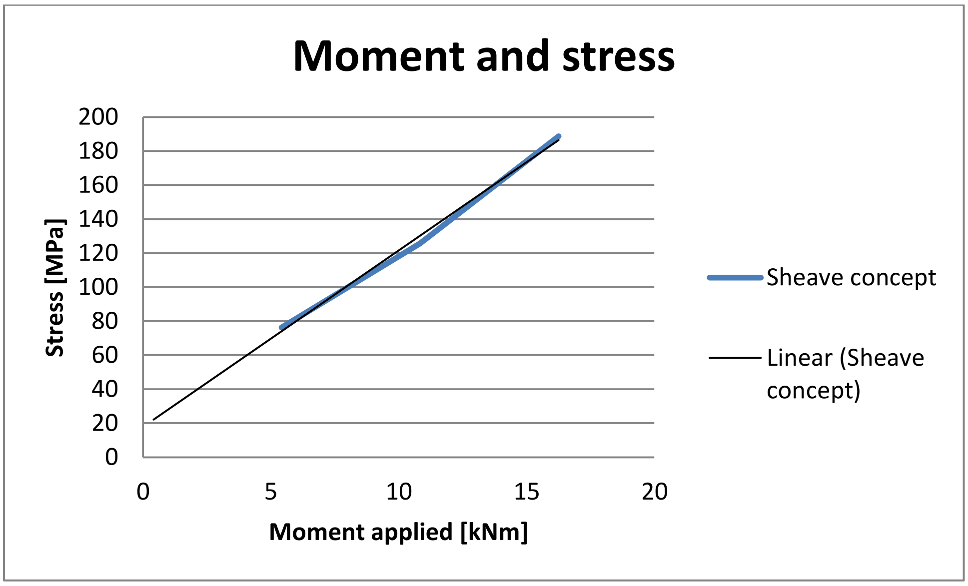


Figure 15: Graph to determine moment capacity

The blue line in the graph illustrates the measured values when different moments are applied on the sheave in ANSYS. The thin black line is the extrapolated value.

From this the moment capacity can be found. At yield stress, when the stress is 158 MPa, the moment is 13,52 kNm. This will then be the moment capacity, M_{Rd}, to be used to check for second order effects of buckling for this concept. The side load gives a maximum moment on sheave of:

$$M_{Ed} = F_h * H = 30831{,}5\ N * 542\ mm = \frac{16{,}7\ kNm}{2} = 8{,}35\ kNm$$

Where H is the distance from where the side load is applied to the inner edge of the web. The moment is divided by two since only half a sheave is modeled. The graph for moment capacity on each design concept can be found in appendix B.

5.3 REACTION FORCES

The reaction forces given by ANSYS have to be checked against the forces that were applied on the model. The loads applied in ANSYS are a pressure of 20,1 MPa normal to the surface in the groove on half the model, and a line pressure of 154,15 $\frac{N}{mm}$ on one of the edges.

To find the reaction forces, the pressure has to be decomposed to z and y components and projected to the surface shown in Figure 16. The thickness is 26,1mm as shown before, and the projected line is:

$$L = \sqrt{r^2 + r^2} = \sqrt{842^2 + 842^2} = \sqrt{2} * 842mm = 1190{,}8\ mm$$

The total reaction force that needs to be checked against ANSYS is then:

$$F = P * A = 20{,}1\ MPa * (1190{,}8 * 26{,}1)mm^2 = 624{,}7\ kN$$

The total force will act with an angle of 45° from the y-axis. The decomposed forces F_y and F_z then becomes:

$$F_y = F_z = F * \cos(45°) = 624{,}7\ kN * \cos(45°) = 441{,}7\ kN$$

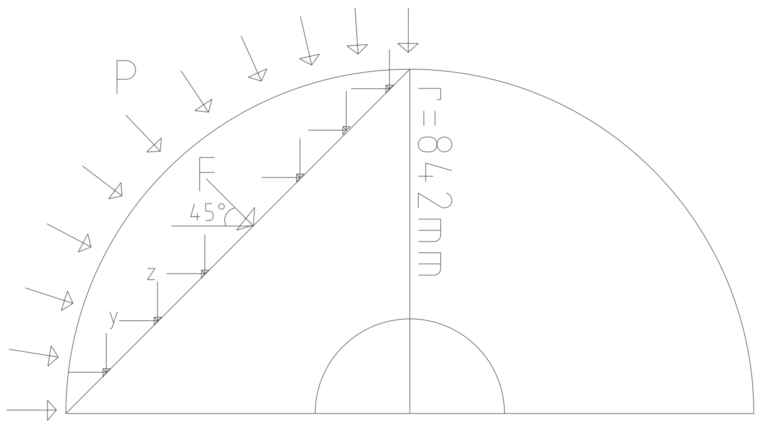


Figure 16: Reaction forces calculation

In x-direction the line pressure needs a reaction force of:

$$F_x = 154{,}15 \frac{N}{mm} * 100mm = 15{,}4\ kN$$

To be certain that the numerical aspect of the calculations is correct, these values need to be checked towards the reaction forces from ANSYS.

5.4 FATIGUE

Fatigue assessment of the sheaves should be done according to DNV-RP-C203.

DNV has made a guidance on how to compute the hot spot stresses with potential to fatigue cracking from the weld toe, using finite element method.

The element mesh size at the hot spot regions has to be from t x t up to 2t x 2t. A larger element mesh size can give non-conservative results.

The standard states that the stress components should be evaluated along the paths illustrated in the figures below:

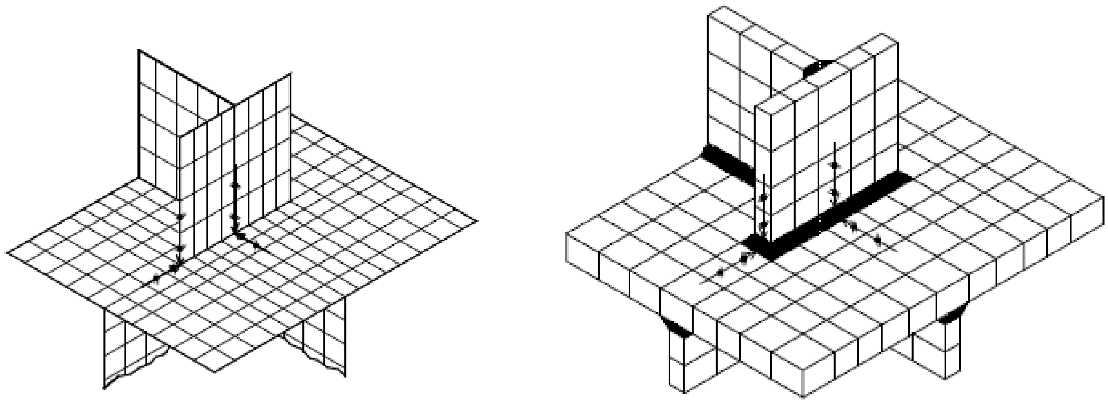


Figure 17: Stress extrapolation in a 3-D FE model to the weld toe [17]

The recommended stress evaluation points are located at a distance of 0,5t and 1,5t away from the hot spot, where t is the plate thickness at the weld toe. These points are also denoted as the stress read out points for the analysis.

Maximum stress in the sheaves are located at the surface. According to DNV the surface stress may be evaluated at the corresponding mid-side points. Thus the stresses at mid side node along line A-B may be used direclty as stress read out points 0,5t and 1,5t.

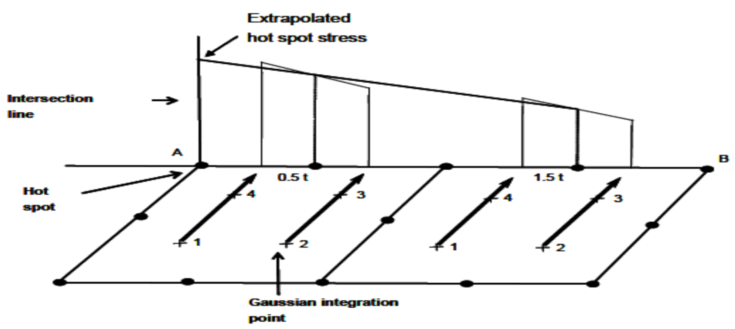


Figure 18: Example of derivation of hot spot stress [17]

After running the analyzis in ANSYS the stress at 0,5t and 1,5t away from the hot spot region, from maximum- and minimum princiapl stresses, are extrapolated to the hots pot. This gives the stress range for fatigue analysis. It is recommended to link the hot spot stress to the D-curve [17]. This stress range is then multiplied with the equivalent stress range given from the fatigue load spectrum, supplied by Cameron Sense in appendix C. The equivalent stress range is derived as follow: [18]

$$D = \sum_{i=1}^{k} \frac{n_i}{N_i} \leq \eta \qquad \text{(Miner's rule)}$$

Where D is the accumulated fatigue damage, n_i is the number of stress cycles in stress block i, N_i is the number of cycles to failure at constrant stress range and η is the usage factor. From equation 2.4.1 in DNV-RP-C203, the expression for the S-N curve gives:

$$\text{Log}(N_i) = \log(\bar{a}) - m * \log(\Delta\sigma_i) \rightarrow$$

$$\text{Log}(N_i) = \log \bar{a}(\Delta\sigma_i)^{-m} \rightarrow$$

$$\text{Log}(N_i) = \log\left(\frac{\bar{a}}{\Delta\sigma_i^m}\right) \rightarrow$$

$$N_i = \frac{\bar{a}}{\Delta\sigma_i^m}$$

Inserting this equation into the accumulated fatigue damge equation gives:

$$D = \frac{1}{\bar{a}} \sum_{i=1}^{k} n_i (\Delta\sigma_i)^m \leq \eta$$

Instead of using different stress at different fatigue cycle, equivalent stress is use, the formula then becomes:

$$D = \frac{1}{\bar{a}} \sum_{i=1}^{k} n_i \left(\Delta\sigma_{eq}\right)^m \leq \eta$$

To get the same accumulated damage, D, $\Delta\sigma_{eq}$ is selected as:

$$\frac{D}{D_{eq}} = \frac{\sum_{i=1}^{k} n_i (\Delta\sigma_i)^m}{\left(\sum_{i=1}^{k} n_i\right) * \left(\Delta\sigma_{eq}\right)^m} = 1$$

$$\Delta\sigma_{eq} = \left(\frac{\sum_{i=1}^{k} n_i * \Delta\sigma_i^m}{\sum_{i=1}^{k} n_i}\right)^{\frac{1}{m}}$$

The equivalent stress range factor can be defined as:

$$k = \frac{\Delta\sigma_{eq}}{\Delta\sigma_{max}}$$

The equivalent stress range for the different sheave concepts is then:

$$\Delta\sigma * \left(\frac{\Delta\sigma_{eq}}{\Delta\sigma_{max}}\right) \leq \Delta\sigma_{cap}$$

All of the concepts are checked for the utilization equation expressed like:

$$\frac{\Delta\sigma_{cap}}{k} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1$$

Where $\Delta\sigma$ is the stress range found in ANSYS. From appendix C the equivalent stress range factor is:

$$\mathrm{k} = 0{,}38$$

The sheave needs to be able to have 200 million cycles over a periode of 20 years. This gives a capacity of:

$$\Delta\sigma_{cap} = 10^{\left(\frac{\log(15{,}606) - \log(2*10^8)}{5}\right)} = 28{,}9\ MPa$$

The values from $\log(\bar{a})$ and m is chosen from table 2-1 in DNV-RP-C203. The fatigue capacity divided on the equivalent stress range factor is then:

$$\frac{\Delta\sigma_{cap}}{k} = \frac{28{,}9}{0{,}38} = 76\ MPa$$

In the double web concepts the weld is on the backside of the webs. Choosing the D-curve in these concepts is a little optimistic, since there can be imperfections in the weld that is not controlled, a lower curve might be selected. If the welds in the design concepts with the weld on the outside of the web are machine flushed, to remove defects and weld overfills, a higher curve might be selected. The straight web design concepts can be casted (2,3,5,6,7), and since the sheave has only compression stress, the stress range from ANSYS can be multiplied with a reduction factor f_m. The factor is expressed in equation below where σ_t is maximum tension stress and σ_c is the maximum compression stress [17]:

$$f_m = \frac{\sigma_t + 0{,}6\sigma_c}{\sigma_t + \sigma_c}$$

6.0 DIFFERENT DESIGN CONCEPTS

In this chapter different sheaves are designed and analized. The first concept is the sheave currently being used by Cameron Sense. This sheave was used to learn the different programs being used. Drawings of this sheave were given by Cameron Sense.

Modelling of the sheaves was done in Inventor, and then exported to ANSYS for analysis. The loads applied are the same in every concept, FE mesh and support is also the same as the first concept. The FE mesh is checked with different values of mesh to ensure credible results. Cross sectional view of every sheave with dimensions can be found in appendix A.

6.1 DOUBLE WEB

The model of the sheave is made with Autodesk Inventor design tool. The model is sliced in half in symmetry plane XY. The sheave is sliced in half since it is symmetric and the calculation time in ANSYS is reduced.
After designing the sheave it is exported to ANSYS workbench for analysis. Bolt holes on the support, for roller bearings cover, are suppressed in Inventor and not included in the analysis.

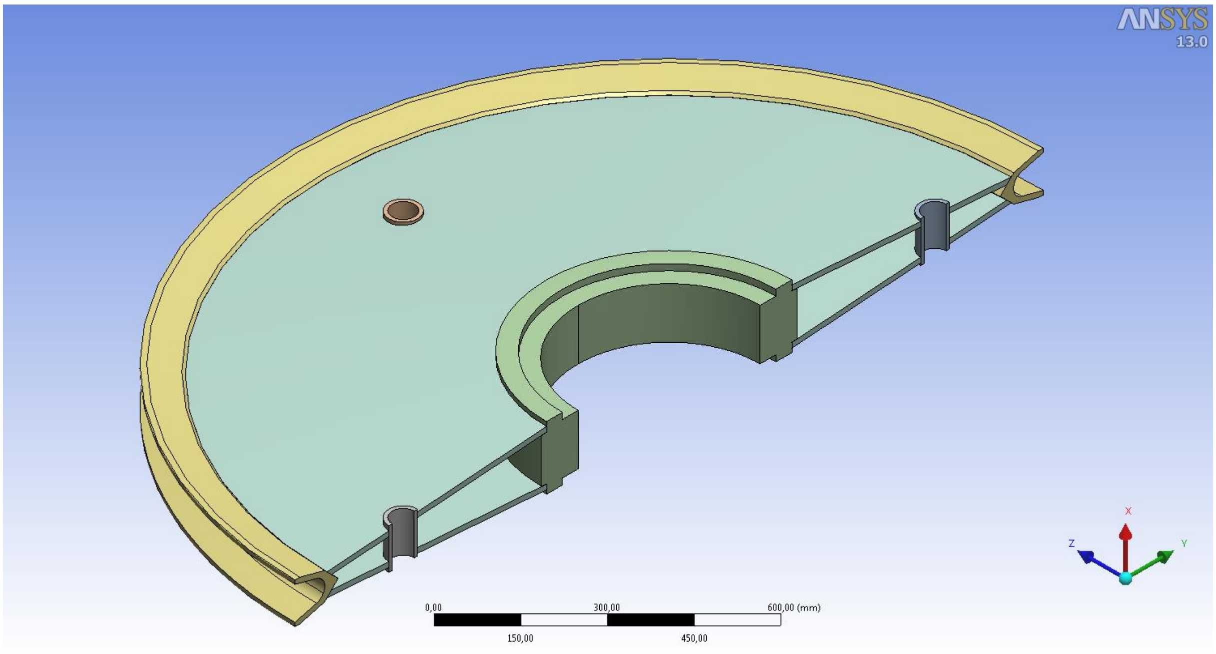


Figure 19: Analysis model of the sheave (ANSYS)

The current sheave design consists of a support, two plates, a groove and four small cylinders. 3D-model of the sheave in ANSYS is shown in Figure 19 above.

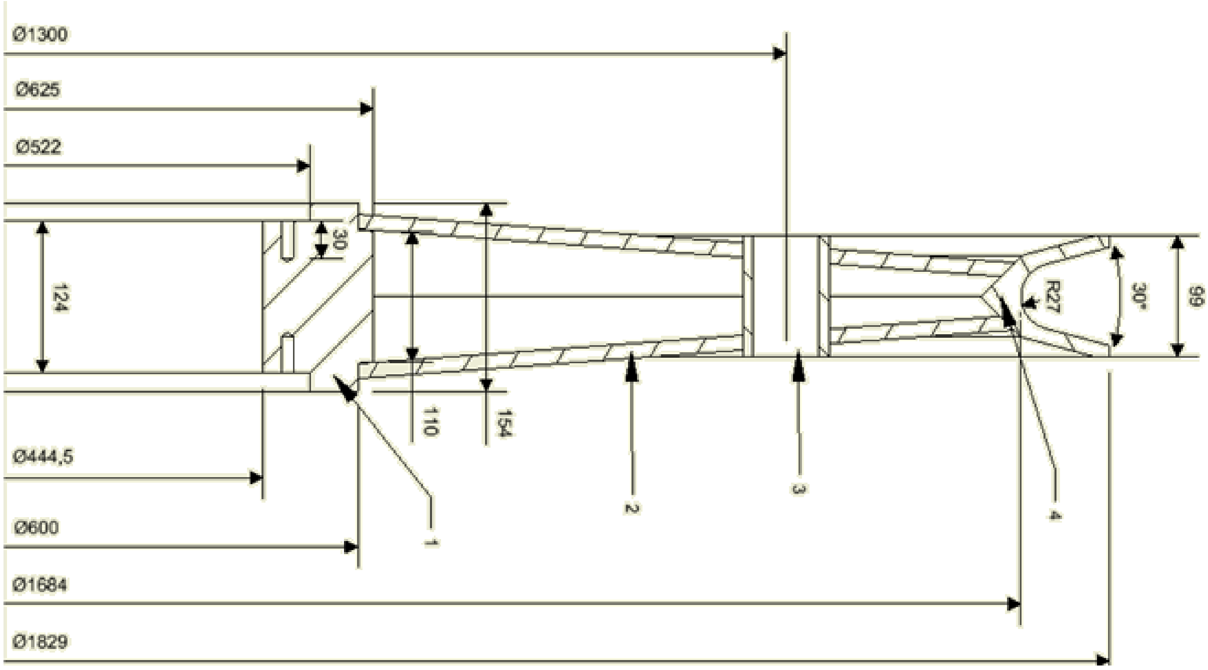


Figure 20: Cross section of the sheave and dimensions

Figure 20 shows cross section of the light sheave. In Figure 20, point 1 is the sheaves support, 2 are the webs, 3 is a hole supported by a cylinder and 4 is the sheave groove. The holes at point 3 are to be able to lock the sheave in place when stationary. There are a total of four of these holes.

According to DNV Standard for cert. 2-22 B600 the sheave diameter for steel wire ropes shall have at least a diameter that corresponds to the ratio $\frac{D_p}{d} = 18$, where D_p is the pitch diameter of the sheave and d is the wire rope diameter. In our case the ratio is $\frac{72}{2} = 36$, so this is considered ok. It is the webs that it will be dimensioned with respect to. It is these that will have to distribute the loads and have the largest effect on the sheaves mass.

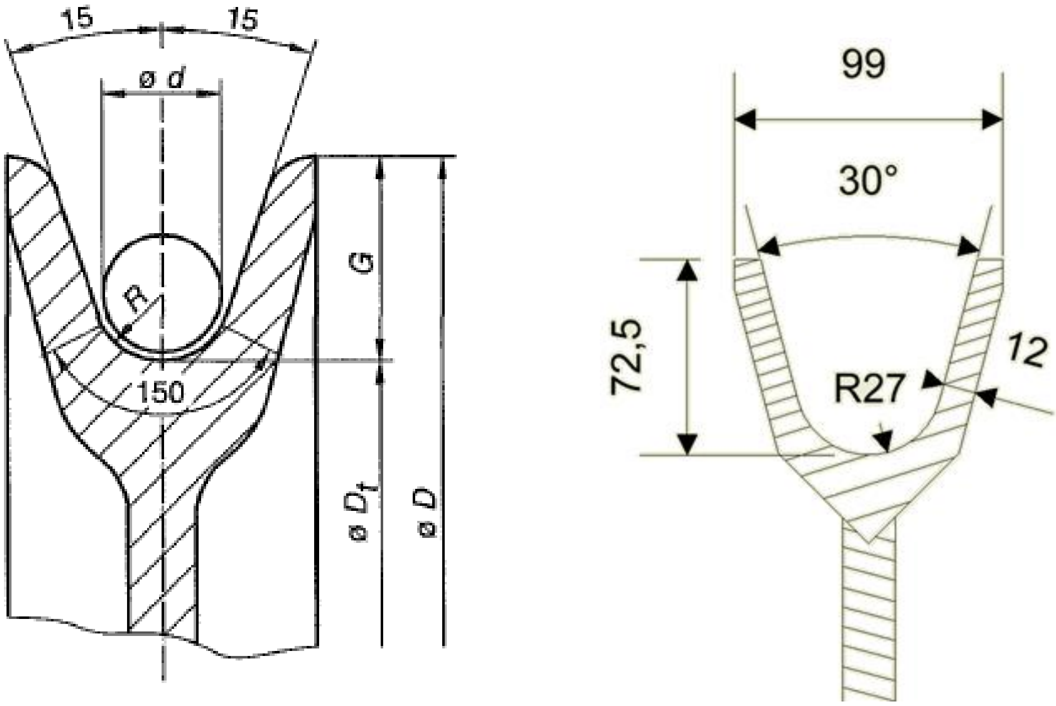


Figure 21: sheave groove API [16]

Figure 21 shows the sheave groove according to API spec 8C. The angle inside of the groove is set to be 30°, the total groove depth, G, shall be a minimum of $1{,}33 * \mathrm{d_{wire}}$ and maximum $1{,}75 * \mathrm{d_{wire}}$. The wire is 50,8 mm which gives a groove depth between:

$$\mathrm{G} = \begin{cases} 67{,}564\ \mathrm{mm} \\ 88{,}9\ \mathrm{mm} \end{cases}$$

The radius on the bottom of the sheave should be between $R_{min} = R_{wire} * 1{,}06$ and $R_{max} = R_{wire} * 1{,}10$

In this thesis the same groove being used for the current state of art sheave will be used on all the different concepts. Cross sectional view with dimensions is shown in right hand side figure above. The groove is 99mm wide at it widest, sides are 72,5mm high, the angle in the groove is 30° and the radius inside the groove is 27mm.

6.1.1 FINITE ELEMENT ANALYSIS OF DOUBLE WEB SHEAVE

After modelling the sheave in Autodesk Inventor it is exported to ANSYS for analysis. Imprints on the sheave are made in inventor to apply the right forces and boundary conditions.

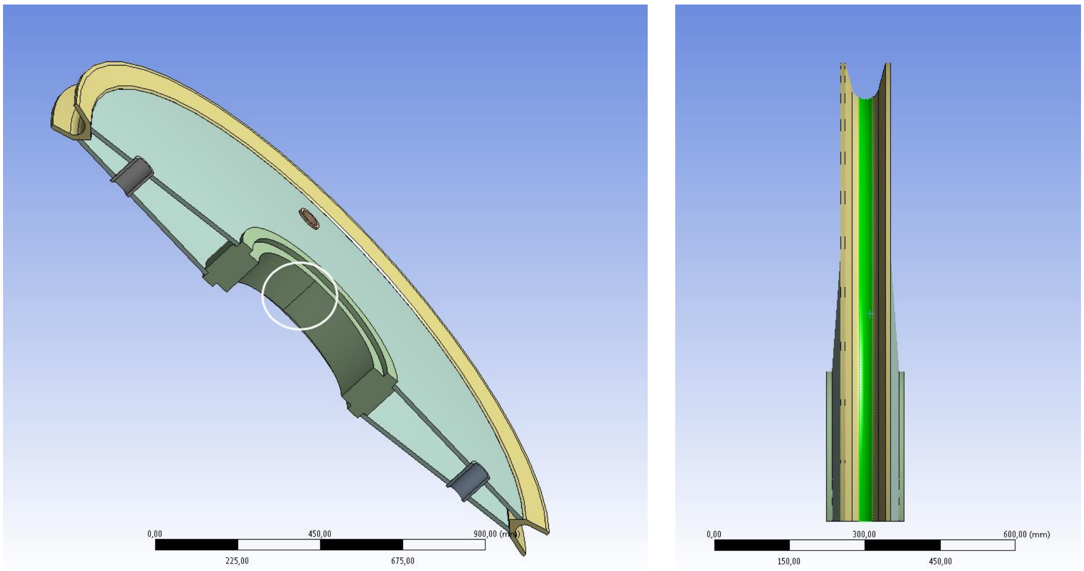


Figure 22: Imprint on support and groove

Figure 22 shows an imprint on the support and groove. The support imprint is at the middle of the half cylindrical surface. The green surface on the right hand side picture is the imprint for the pressure from the wire. On the support body, a cylindrical support is added on one of the faces as shown in Figure 23. The cylindrical support is free in tangential direction to allow rotation. This support is infinitive stiff. Modelling a shaft to be used as a support is more accurate, but a cylindrical support is a very efficient approximation.

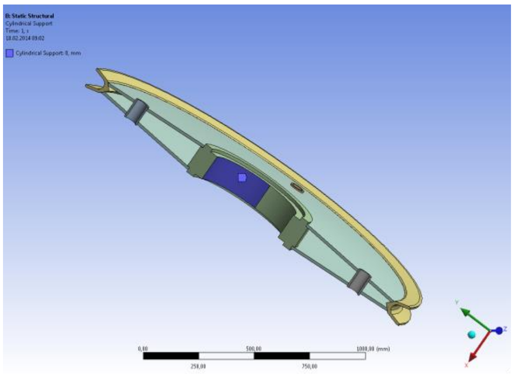


Figure 23: Cylindrical support

Since only half the size was modelled, boundary condition to simulate a symmetry region on the sheave has to be applied.

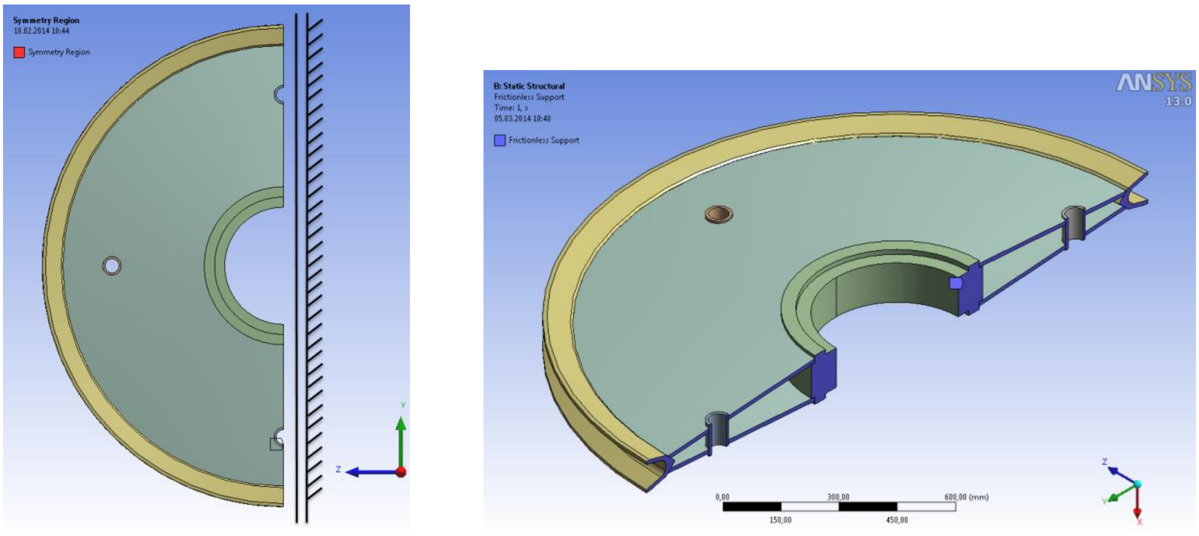


Figure 24: Symmetry region of the model

By adding frictionless supports on the surfaces, shown on the right hand side in Figure 24, the sheave is able to rotate around z-axis. The forces on the sheave are then distributed throughout the entire sheave instead of getting large reaction forces at the section split. The displacements at the frictionless support are zero so the reaction forces at this point become correct.

In Figure 25, pressure on the groove from wire is shown. The pressure is 20,1 MPa as calculated in section 5. Additionally line pressure of 154,15 $\frac{N}{mm}$ from the wire is applied on the line in the groove as shown in Figure 26.

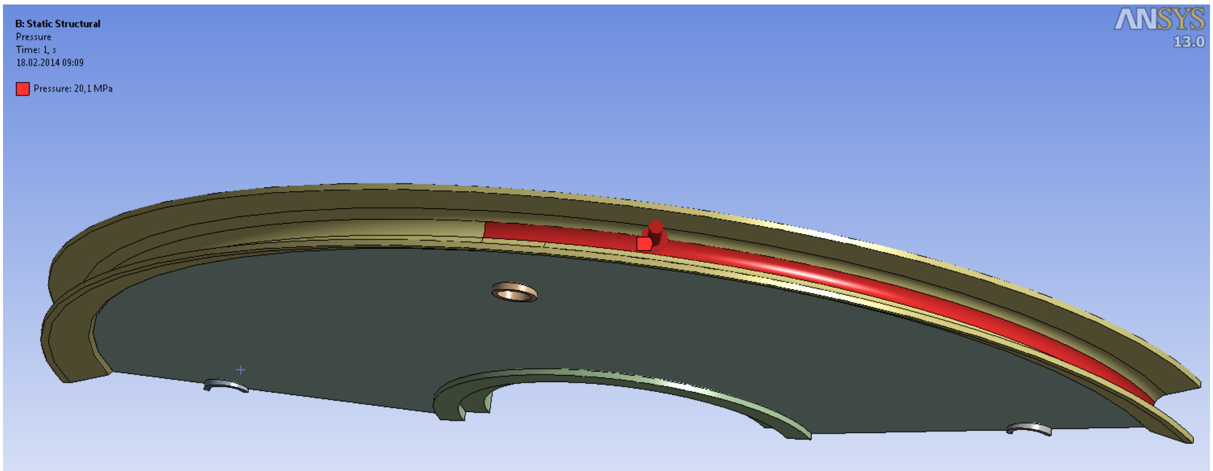


Figure 25: Pressure on groove

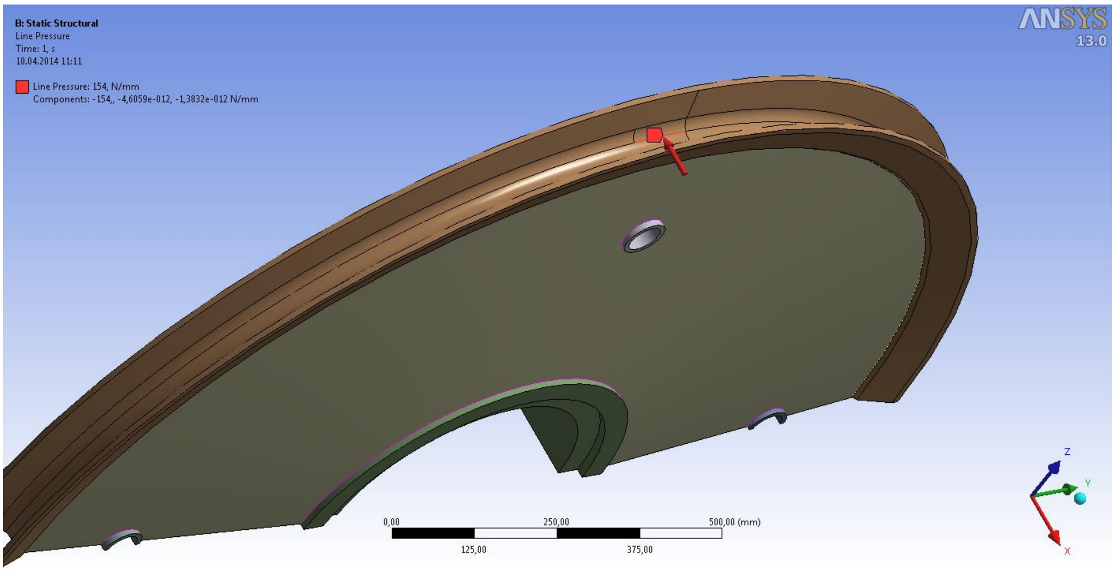


Figure 26: Side pressure from wire

6.1.2 FE MESH

Figure 27 illustrates FE mesh of the model. The FE mesh divides the model into smaller elements to find an accurate solution of the stresses in the sheave. The model must have a minimum of two elements in the web thickness to make sure that it is not to stiff. This model has an element size of maximum 10 mm, which gives a total number of 336635 nodes and 77649 elements within the model.

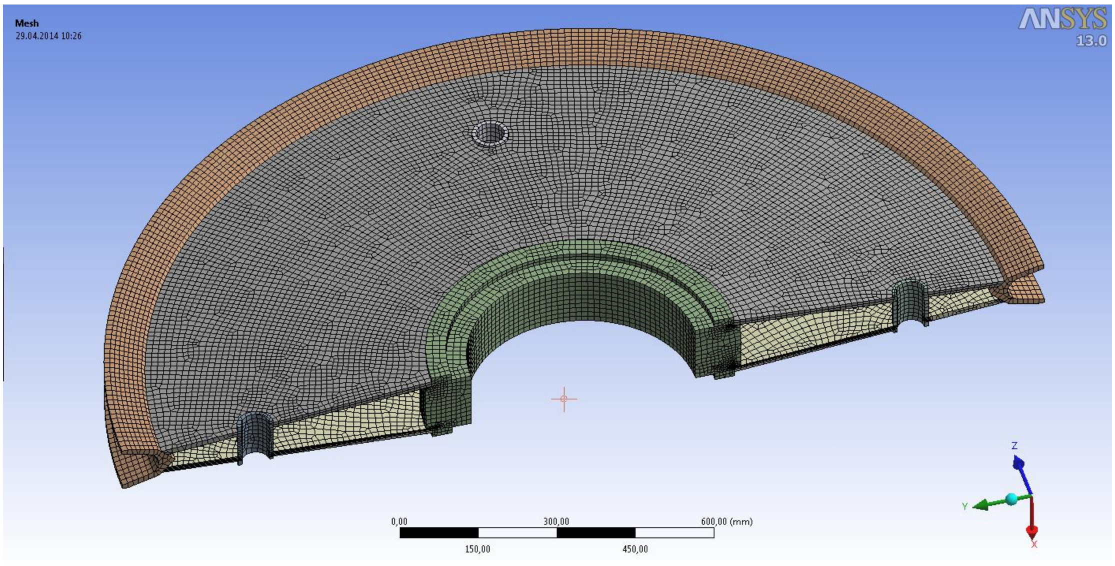


Figure 27: FE mesh of sheave

6.1.3 EQUIVALENT STRESS PLOT

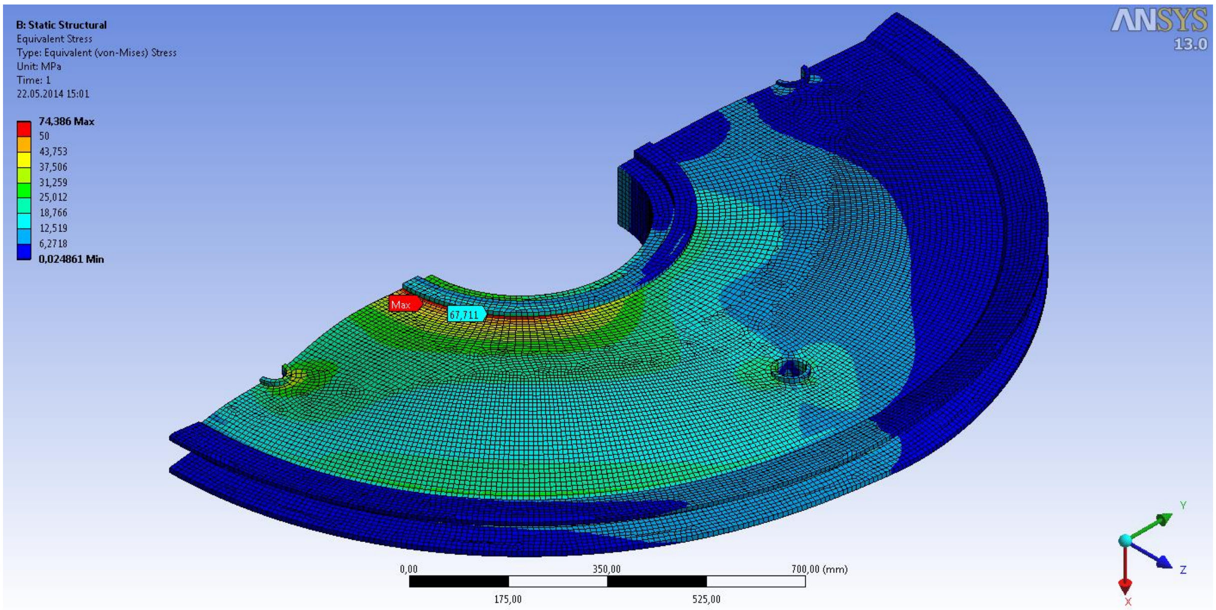


Figure 28: Normal stress plot

Figure 28 illustrates the equivalent stress plot when only the axial load is applied on the sheave.

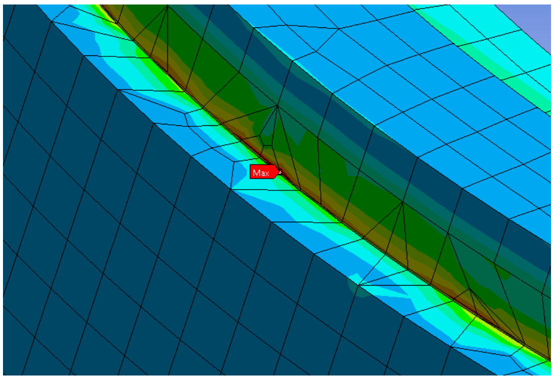


Figure 29: Maximum average equivalent stress

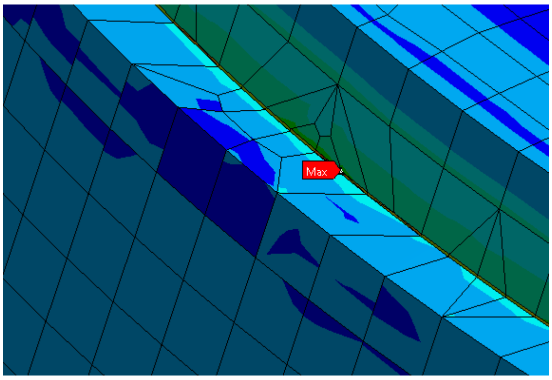


Figure 30: Maximum non-average equivalent stress

Figure 29 shows a close up on where the maximum stress occurs. As seen from the figure, the maximum stress occurs at the support. The left hand side picture is the average equivalent stress, and the right hand side figure is the non-average equivalent stress. It is this stress concentration that gives the maximum equivalent stress at this point. The stress of concern is at the end of one of the web, with a value of 67,7 MPa.

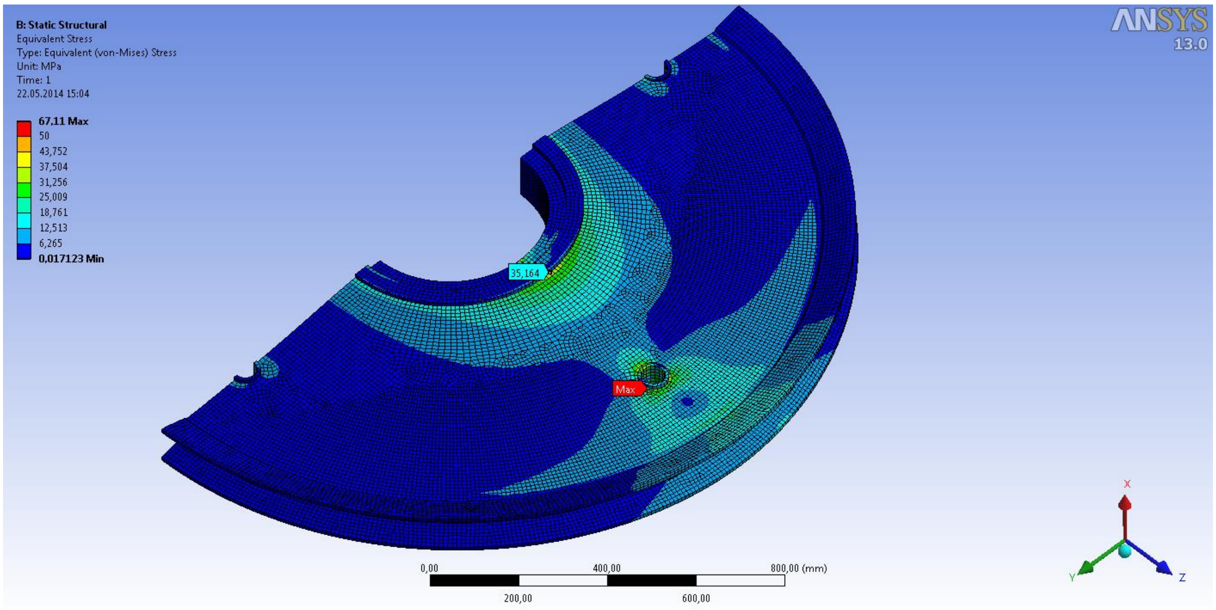


Figure 31: Bending stress plot

Figure 31 shows the stress plot when only side load is applied. Maximum stress occurs at one of the cylinders in the sheave with a value of 67,1 MPa. The stress at the inner edge of the web is 35,16 MPa.

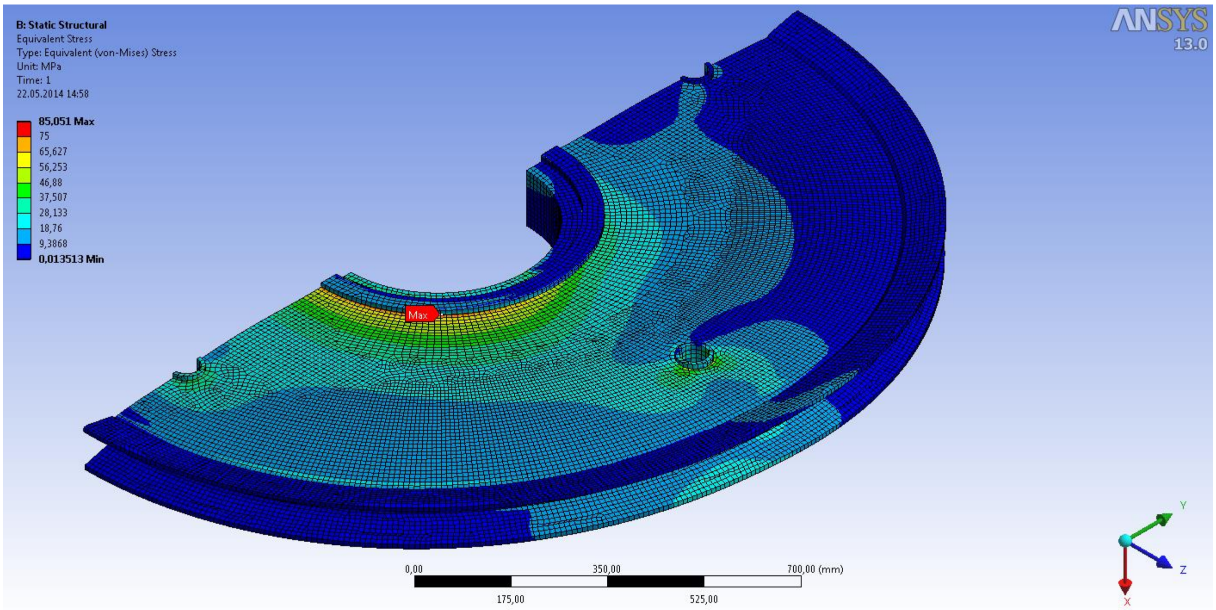


Figure 32: Equivalent stress plot

Figure 32 illustrates the equivalent stress plot with both loads applied. The maximum stress is at the end of the web with a value of 85,05 MPa.

6.1.4 REACTION FORCES

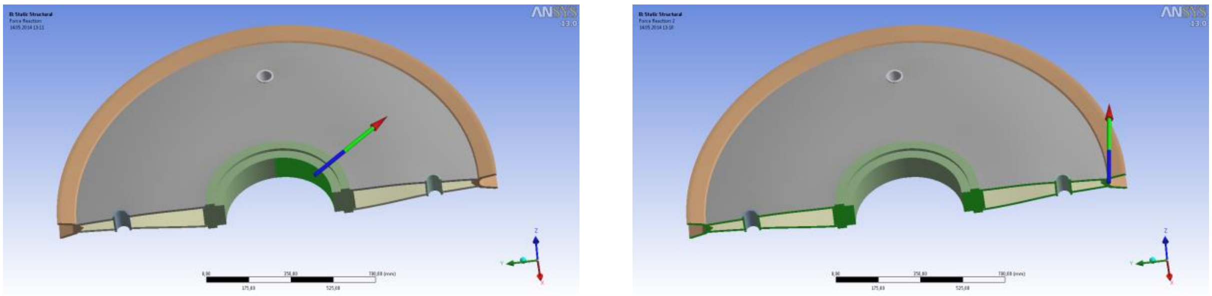


Figure 33: Reaction forces

Reaction forces for the cylindrical support on the left hand side and the frictionless support on the right hand side are shown in Figure 33. The results are presented in Table 2 below.

Table 2: Reaction forces at the cylindrical support

Support	Direction		
	X [kN]	Y [kN]	Z [kN]
Cylindrical	15,4	-442,3	387,8
Frictionless	0	-3,8	51,8
Total	15,4	-446,1	439,6

From Table 2 it is seen that the values corresponds well in x-direction and that the frictionless support takes some of the force in z-direction. The total force vector from ANSYS in y- and z-direction is:

$$F = \sqrt{F_y^2 + F_z^2} = \sqrt{(-446{,}1)^2 + 439{,}6^2} = 626{,}3\ kN$$

Which is approximetly the same as the total force vector calculated in section 5.

So the numerical aspect of the ANSYS calculation is considered ok.

6.1.5 BUCKLING

The model was also analyzed for buckling. ANSYS uses linear buckling calculation to check the stability of the structure according to linear (Euler) theory.

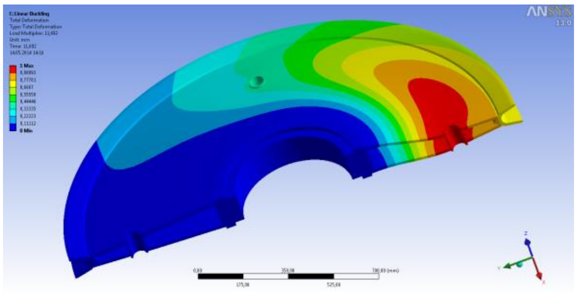


Figure 34: Buckling mode 1

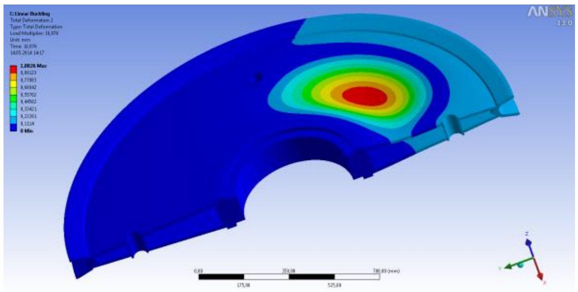


Figure 35: Buckling mode 2

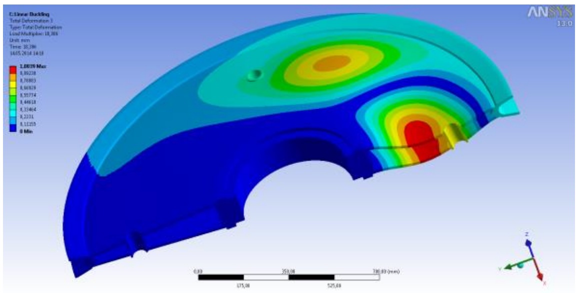


Figure 36: Buckling mode 3

Figures above illustrate the different buckling modes from ANSYS. Three modes of buckling were calculated and gave the results presented in Table 3 below

Table 3: Buckling modes and load multiplier

Buckling mode	Load multiplier
1	11,7
2	16,07
3	18,8

The load multiplier from ANSYS is a factor the applied load on the sheave have to be multiplied with to get the Euler load, $F * \alpha = F_E = \frac{\pi^2 * EI}{L^2}$. Where α is the load multiplier. As shown in Table 3 the sheave has good capacity for nonlinear buckling. Checking for second order effect of buckling, the following equation have to be fulfilled:

$$\frac{F}{F_E} + \frac{M}{M,Rd * \left(1 - \frac{F}{F_E}\right)} \leq 1$$

From the graph in appendix B the moment capacity for this sheave concept is:

$$M_{Rd} = 34{,}63\ kNm$$

All the values needed to check for second order buckling are known, equation above gives:

$$\frac{F}{F_E} + \frac{M}{M{,}Rd*\left(1-\frac{F}{F_E}\right)} = \frac{F}{F*\alpha} + \frac{8{,}35*10^6}{34{,}63*10^6*\left(1-\frac{F}{F*\alpha}\right)} = \frac{1}{11{,}7} + 0{,}24 * \frac{1}{\left(1-\frac{1}{11{,}7}\right)} = 0{,}35 \leq 1 \rightarrow ok!$$

This sheave is considered acceptable regarding second order effect of buckling. Checking for the other buckling modes in not necessary since these values are higher than the first mode, and a higher buckling load multiplier gives a lower value.

6.1.6 FATIGUE

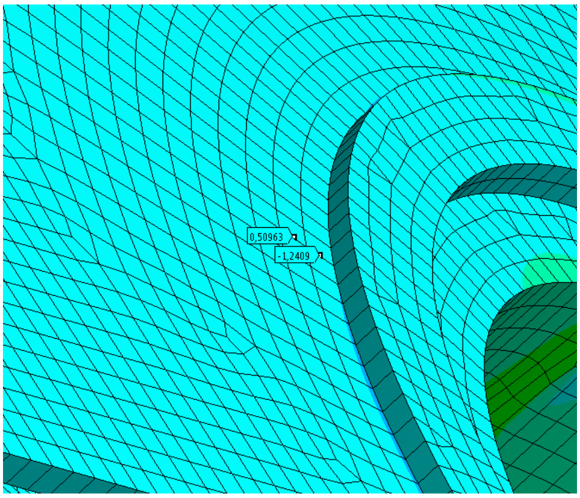


Figure 37: Maximum principal hotspot stress plot

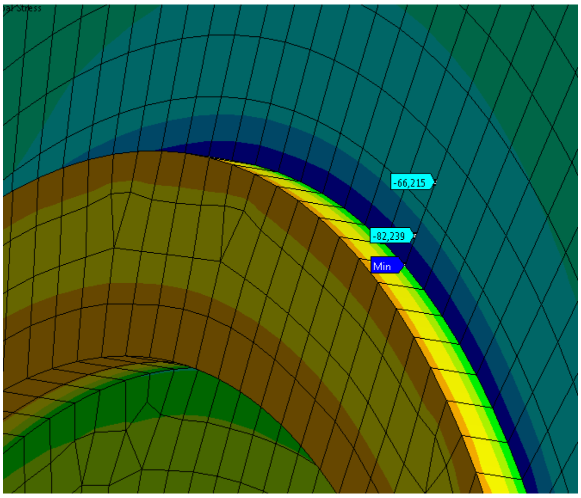


Figure 38: Minimum principal hotspot stress plot

Figures above show the maximum and minimum principal stress plot 0,5t and 1,5t away from the hot spot region. Maximum principal stress occurs around the cylinder inside the plate with a value of 71,75 MPa. The minimum principal stress at this point is very low. Since the minimum principal stress is so low, it will give the highest stress range at the inner edge of the web, even though the maximum hot spot stress is very low.

Maximum hot spot from web to support is then:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -1{,}24\ MPa - \frac{(-1{,}24 - 0{,}5)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -2{,}11\ MPa$$

Minimum hot spot stress from web to support is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -82{,}24\ MPa - \frac{(-82{,}24 - (-66{,}2))MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -90{,}26\ MPa$$

The stress range is then:

$$\Delta\sigma = \sigma_{max} - \sigma_{min} = -2{,}11 - (-90{,}26) = 88{,}15\ MPa$$

The equation below is used to check whether or not the concept is acceptable regarding fatigue:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{\mathrm{k}}\right)} \leq 1 \rightarrow \frac{88{,}15}{\left(\frac{28{,}9}{0{,}38}\right)} = 1{,}16 < 1 \rightarrow not\ ok!$$

With the maximum stress range in this concept the fatigue utilization is 116 % when using the D-curve and 200 million cycles. The spectrum in appendix C has considered the sheaves to have full loading when it goes up and down. On the way up, after the work cycle is done, the sheave is not fully loaded, therefore the spectrum can be halved to 100 million fatigue cycles. This gives a maximum stress from S-N curve of:

$$\Delta\sigma_{max} = 10^{\frac{(log\bar{a} - logN)}{m}} = 10^{\left(\frac{15{,}606 - \log(1*10^8)}{5}\right)} = 33{,}2\ MPa$$

With the new maximum stress the equation to be checked for fatigue becomes:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{\mathrm{k}}\right)} \leq 1 \rightarrow \frac{88{,}15}{\left(\frac{33{,}2}{0{,}38}\right)} = 1{,}02 < 1 \rightarrow not\ ok!$$

The utilization is still not acceptable, but it is very close. As stated before, using the D-curve might not be applicable. The weld can be classified as category F, [17]. This gives a utilization of 129 % shown in appendix C.

6.2 STRAIGHT WEB

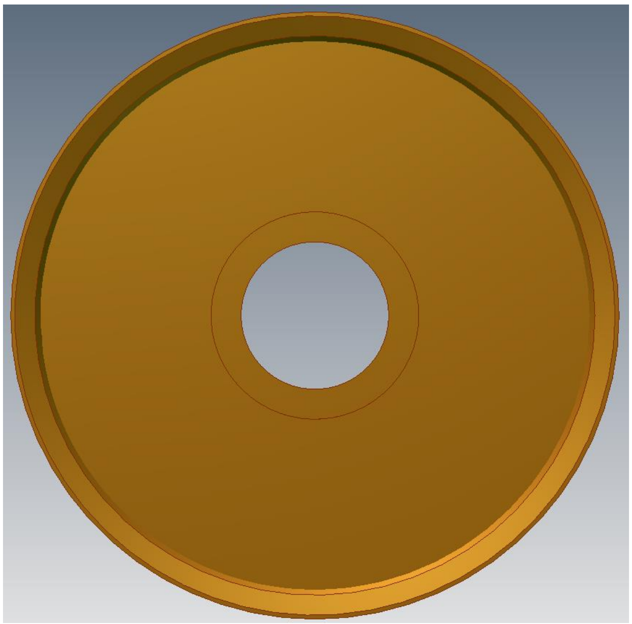
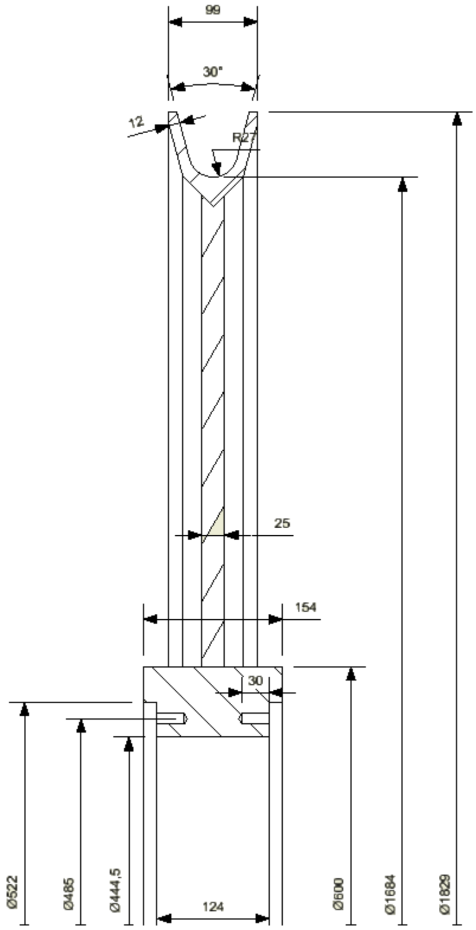


Figure 39: Straight web



Cross section view

This concept consists of a simple straight web, groove and support as shown on the right hand side in Figure 39. First the minimum thickness of the sheave has to be determined, by using the equation derived in section 5:

$$t > \frac{\frac{F}{D}}{\sigma_Y * SF} = \frac{\frac{882900}{600}}{355 * 2,25} = 9,3\ mm$$

Minimum thickness at the inner edge, where the highest stress occurs, is 9,3 mm. This is when only normal stress is applied.

The bending stresses will also contribute to the thickness of the web. After running analysis of the sheave with both load conditions in ANSYS, with different thickness, the thickness is chosen to 25 mm. It is chosen to 25 mm to not fully utilize the sheave in case of unexpected extra loads. The notches on both sides at the support from the current state of art sheave are removed since these are not needed for this concept. The groove of the sheave is the same as the current. Figure 39 shows the concept with dimensions. From the simple hand calculations the stress at the inner edge of the sheave with the thickness chosen should be force divided on area:

$$\sigma = \frac{2*90T*9{,}81\frac{m}{s^2}}{25mm*600mm} = \frac{117{,}72MPa}{2} = 58{,}86\ MPa$$

The normal stress plot from ANSYS is illustrated in Figure 40 below.

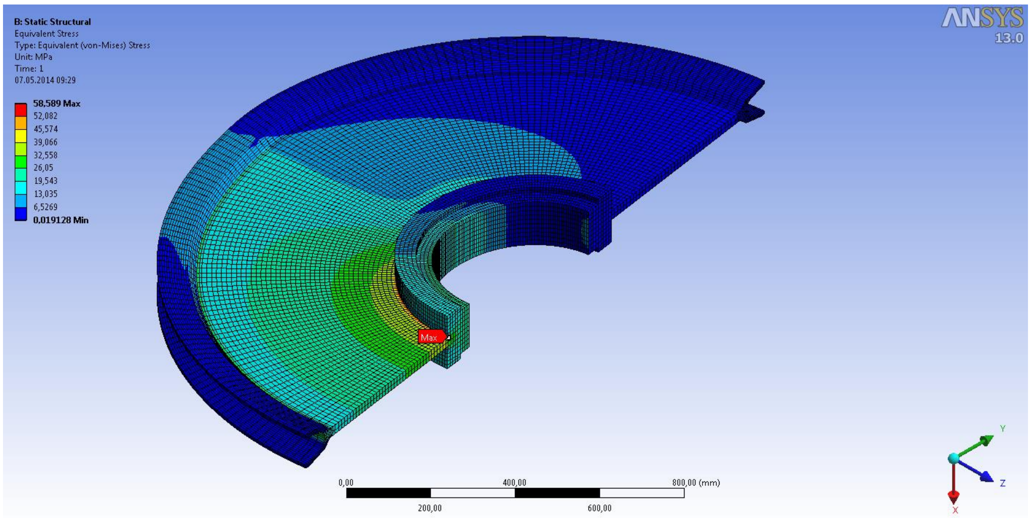


Figure 40: Stress plot for straight web sheave

As seen in Figure 40 the stress at the inner edge of the plate is 58,3 Mpa, which is almost the same as the result from hand calculations. In addition it is seen from the stress plot that a lot of area on the sheave has very small stresses.

The side load will give high stresses at the inner edge of the sheave. After running the analysis in ANSYS it is shown that the concentrated bending stresses are located at the inner edge of the sheave. Figure 41 below shows the bending stress plot from ANSYS, with a maximum bending stress of 83,48 MPa.

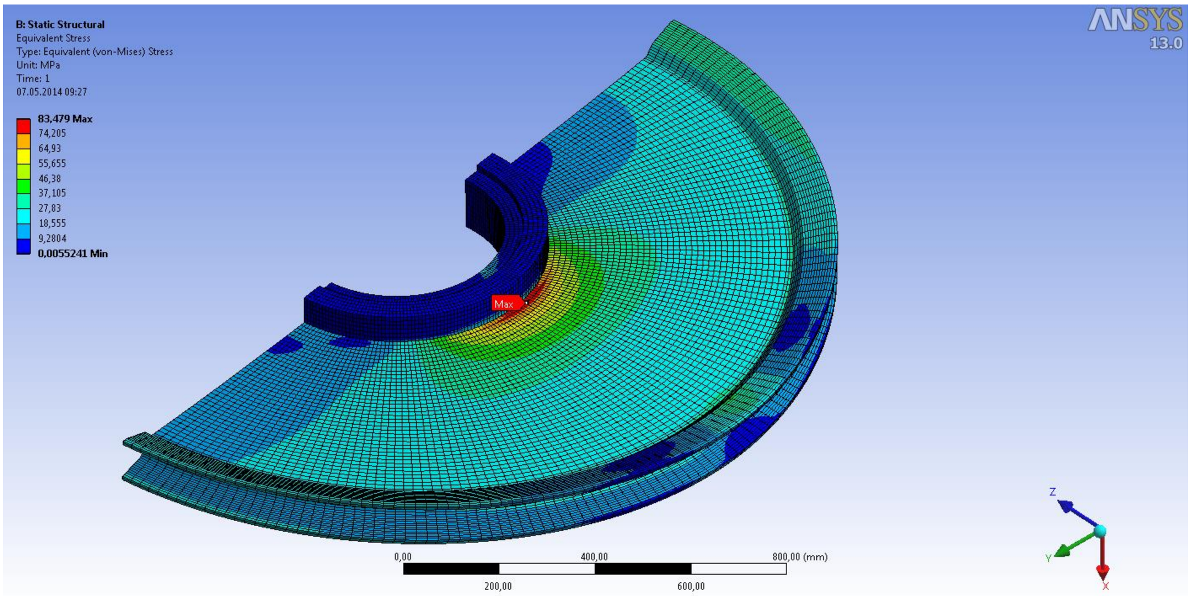


Figure 41: Bending stress plot for straight web sheave

To find the combined equivalent stress, ANSYS uses von Mises stress theory by finding the principal stress vectors $\sigma_1, \sigma_2 and\ \sigma_3$. ANSYS then uses the von Mises equation to find the equivalent stress. The result from ANSYS shows a maximum equivlaent stress of 113 MPa. Adding the maximum normal stress with the maximum bending stress will not give accurate results. This is because the principal stresses, used to get the combined equivalent stress, do not all have the their maximum values in the point where the maximum equivalent stress appears. The combined equivalent stress plot from ANSYS is shown in Figure 42 below.

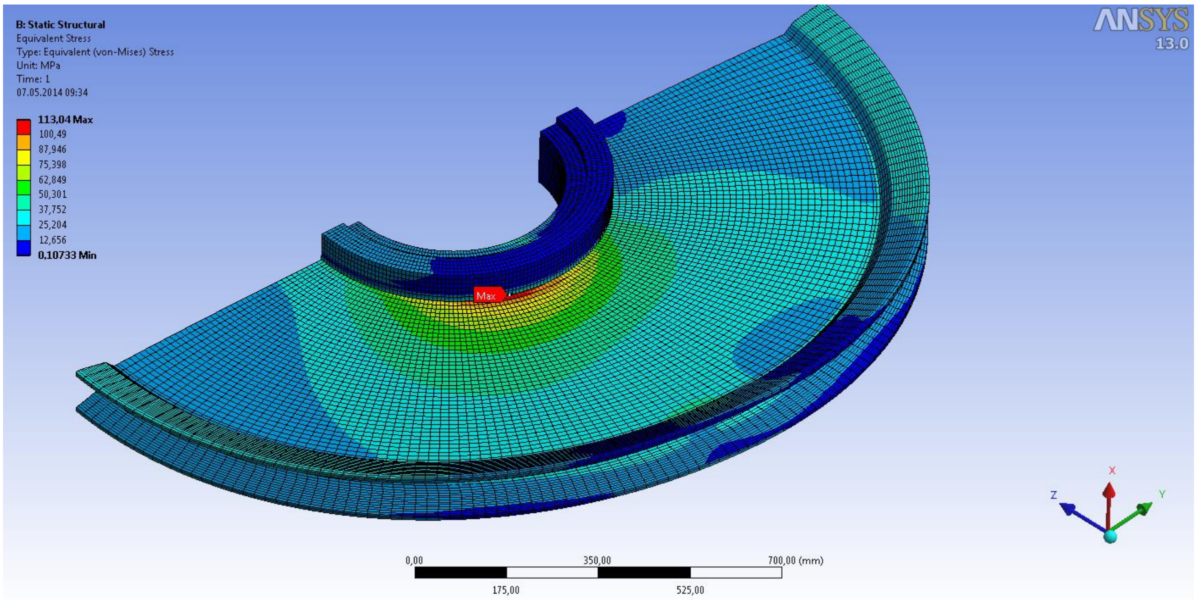


Figure 42: Combined stress plot for straight web sheave

6.2.1 BUCKLING

In ANSYS linear buckling is calculated. Three buckling modes are calculated, from which the first mode is the worst.

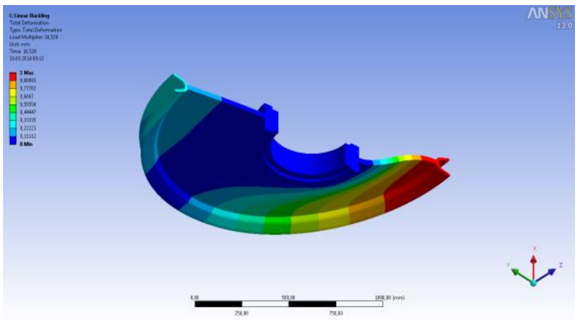


Figure 43: Buckling mode 1

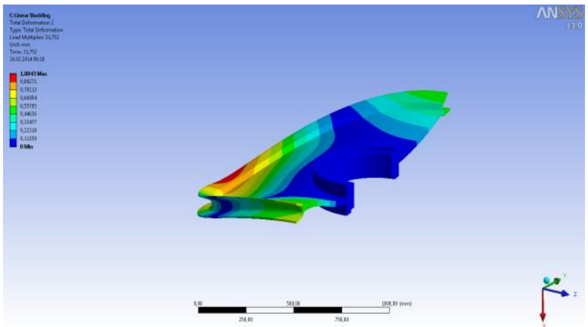


Figure 44: Buckling mode 2

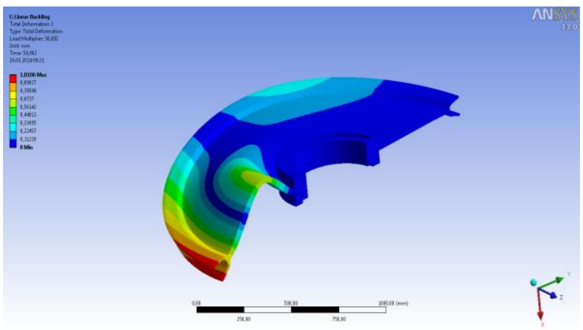


Figure 45: Buckling mode 3

The figures above show the three bukling modes for this concept. The Euler load multiplier is presented in Table 4 below.

Table 4: Buckling modes for straight web sheave

Buckling mode	Load multiplier
1	16,4
2	32,04
3	50,65

Second effects of buckling also has to be checked. As shown in Table 4, the sheave has good capacity for nonlinear buckling. Checking for second order effect of buckling the following equation have to be fulfilled:

$$\frac{F}{F_E} + \frac{M}{M,Rd*\left(1-\frac{F}{F_E}\right)} \leq 1$$

From graph in appendix B the moment capacity at this point is:

$$M_{Rd} = 15{,}57\ kNm$$

All the values needed to check for second order buckling are known, equation above gives:

$$\frac{F}{F_E} + \frac{M}{M,Rd*\left(1-\frac{F}{F_E}\right)} = \frac{F}{F*\alpha} + \frac{8{,}35*10^6}{15{,}57*10^6*\left(1-\frac{F}{F*\alpha}\right)} = \frac{1}{16{,}4} + 0{,}54 * \frac{1}{\left(1-\frac{1}{16{,}4}\right)} = 0{,}63 \leq 1 \rightarrow ok!$$

This concept is considered acceptable regarding second order effect of buckling.

6.2.2 REACTION FORCES

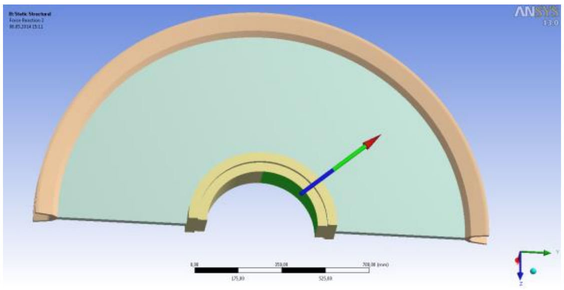


Figure 46: Total cylindrical support reaction force

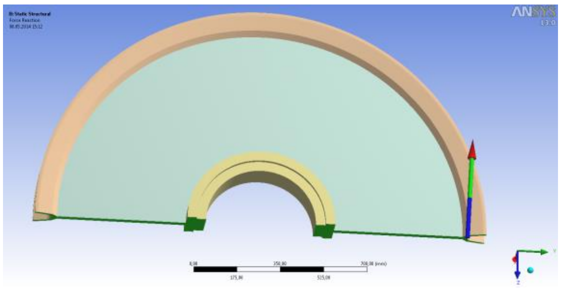


Figure 47: Total frictionless support reaction force

Figures above illustrates the total reaction force from the different support in the model. The left hand side picture shows the cylindrical supports reaction force and the frictionless support reaction force is shown on the right.

The value of the components in different directions are shown in Table 5.

Table 5: Reaction forces from ANSYS for double web

Support	Direction		
	X [kN]	Y [kN]	Z [kN]
Cylindrical	15,45	442,3	-385,4
Frictionless	-0,04	2,02	-54,16
Total	15,41	444,3	-439,54

From Table 5 it is seen that the values corresponds well in x-direction. The total force vector from ANSYS in y- and z-direction is:

$$F = \sqrt{F_y^2 + F_z^2} = \sqrt{441{,}3^2 + (-439{,}54)^2} = 624{,}9\ kN$$

Which is the same as the total force vector calculated above.

6.2.3 FATIGUE

As stated in DNV the stress must be extrapolated from 0,5t and 1,5t to the hot spot region.

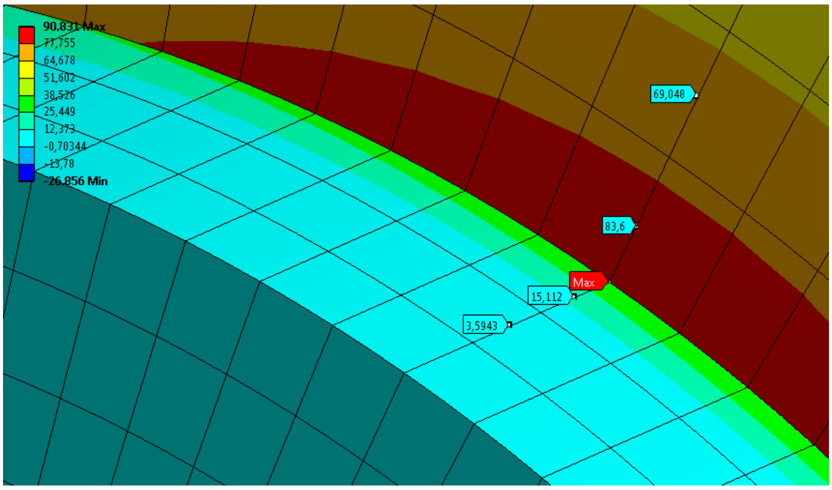


Figure 48: Maximum principal stress plot at 0,5t and 1,5t for straight web sheave

Figure 48 shows the maximum principal stress values at point 0,5t and 1,5t for the straight web concept with mesh size of 25mm (thickness of the web). The formula for the linear curve extrapolation to the hot spot is:

$$y = ax + b$$

Where a is the slope and b is the intersection at y-axis. The slope can be found by the following formula:

$$a = \frac{y_1 - y_2}{x_1 - x_2}$$

Where y_1 and y_2 is the stresses and x_1 and x_2 is the location. Inserted into the equation for linear curve the equation for the hot spot stress becomes:

$$y = \frac{y_1 - y_2}{x_1 - x_2} * x + b \rightarrow b = y - \frac{y_1 - y_2}{x_1 - x_2} * x$$

Where the curve intersects y-axis is where the hot spot stress is located, so b needs to be determined. Solving the formula for b gives the maximum hot spot stress at this point:

$$b = y - \frac{y_1 - y_2}{x_1 - x_2} * x = 83{,}6\ MPa - \frac{(83{,}6 - 69{,}1)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = 90{,}4\ MPa$$

The hot spot stress from maximum principal stress from the web to the support is 90,4 MPa. The hot spot stress from support to web is:

$$b = y - \frac{y_1 - y_2}{x_1 - x_2} * x = 15{,}1\ MPa - \frac{(15{,}1-3{,}6)MPa}{0{,}5t-1{,}5t} * 0{,}5t = 20{,}85\ MPa$$

To get the stress range, also the minimum principal stress has to be determined.

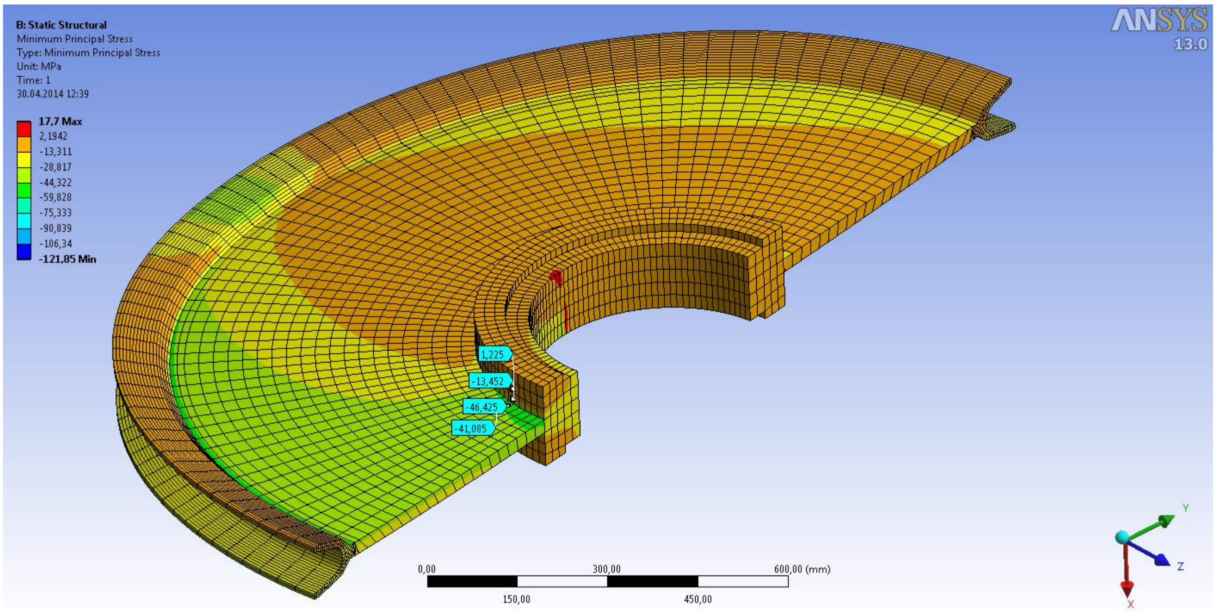


Figure 49: Minimum principal stress plot at 0,5t and 1,5t for straight web sheave

As Figure 49 illustrates, the minimum stress is lower towards the symmetry region of the sheave. Since the sheave is rotating back and forth, this will be the position for the minimum principal stress. It is important to be careful when using values close to boundary conditions since there is a support at the end which is infinitive stiff. The stresses around the edge can be higher than what they actually are. The stresses chosen in Figure 49 are almost the peak stress and they are some distance away from the support.

The same formula as above is used to calculate the hot spot stress in the joint:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = -46{,}4\ MPa - \frac{(-46{,}4-(-41{,}1))MPa}{-0{,}5t-1{,}5t} * 0{,}5t = -49{,}1\ MPa$$

Hot spot stress from minimum principal stress from web to support is -49,1 MPa. The stresses from support to web is:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = -13{,}5\ MPa - \frac{(-13{,}5-1{,}22)MPa}{-0{,}5t-1{,}5t} = -20{,}9\ MPa$$

Now both σ_{max} and σ_{min} are determined, so the stress range to be used for fatigue calculations can be found. The stress range for the path from web to support is:

$$\Delta\sigma = \sigma_{max} - \sigma_{min} = \big(90{,}4 - (-49{,}1)\big)MPa = 139{,}5\ MPa$$

From support to web:

$$\Delta\sigma = \sigma_{max} - \sigma_{min} = \big(20{,}85 - (-20{,}9)\big)MPa = 41{,}75\ MPa$$

This are the values received from a mesh size of 25mm (thickness of web). Having only one element in the web thickness can make the sheave act stiffer than it really is. Even though DNV suggest using a txt mesh, tests with smaller mesh size is carried out to see if the values vary.

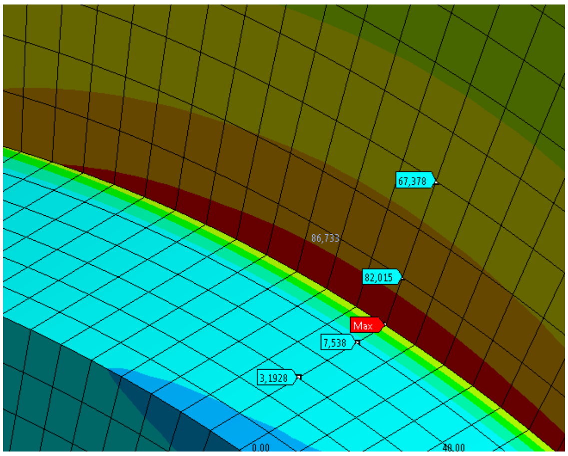


Figure 50: Maximum principal stress plot

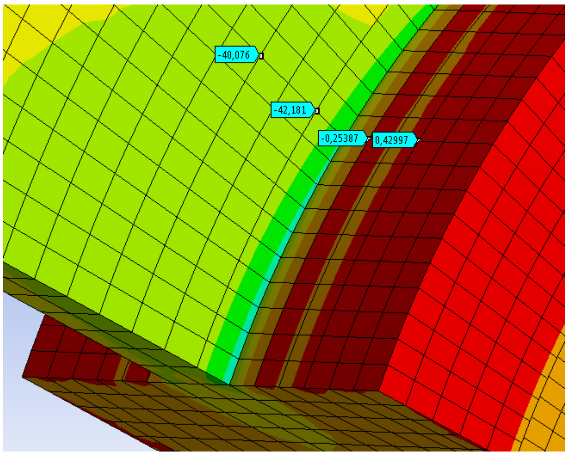


Figure 51: Minimum principal stress plot

The figures above shows the principal stress plot with 0,5tx0,5t element size. The values shown in the figures are representing 0,5t and 1,5t position from the txt mesh. Extrapolation of the maximum principal stresses on path from web to support gives:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = 82{,}01\ MPa - \frac{(82{,}01 - 67{,}4)MPa}{-1t - 3t} * 1t = 89{,}31\ MPa$$

From support to web:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = 7{,}54\ MPa - \frac{(7{,}54 - 3{,}2)MPa}{-1t - 3t} * 1t = 9{,}71\ MPa$$

Extrapolation of the minimum principal stresses gives:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = -42{,}2\ MPa - \frac{(-42{,}2 - (-40{,}1))MPa}{-1t - 3t} * 1t = -43{,}25\ MPa$$

And from on the path from support to web:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = -0{,}25\ MPa - \frac{(-0{,}25 - 0{,}43)MPa}{-1t - 3t} * 1t = -0{,}6\ MPa$$

With these values, the stress range from web to support is:

$$\Delta\sigma = \sigma_{max} - \sigma_{min} = \big(89{,}31 - (-43{,}25)\big)MPa = 132{,}6\ MPa$$

And from support to web:

$$\Delta\sigma = \sigma_{max} - \sigma_{min} = \big(9{,}71 - (-0{,}6)\big)MPa = 10{,}31\ MPa$$

As seen from the different mesh size, there is not much varying between the different stress ranges. Next a test with mesh size of 0,25tx0,25t is carried out to see if the stresses vary more with an even denser mesh.

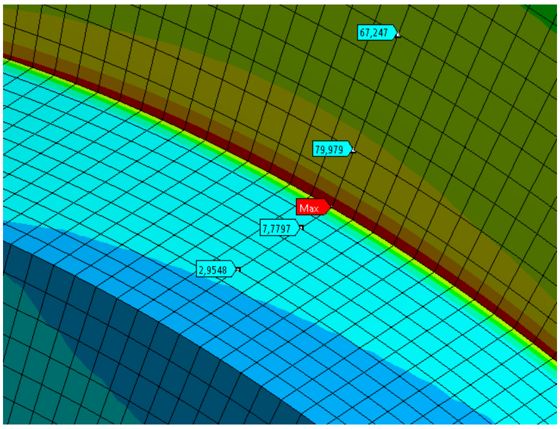


Figure 52: Maximum principal stress plot

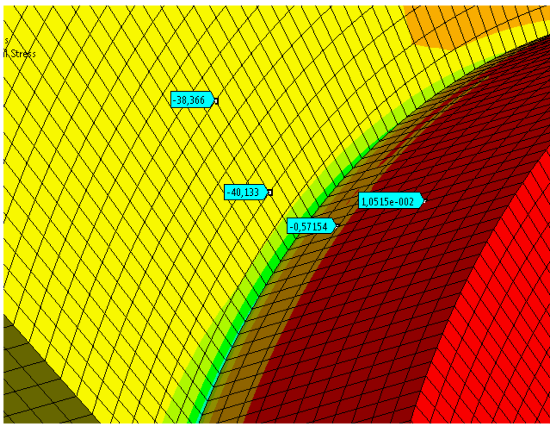


Figure 53: Minimum principal stress plot

Figures shows the principal stresses with an element size of 0,25tx0,25t.

Extrapolation of the maximum principal stress on path from web to support gives a hot spot stress of:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = 80\ MPa - \frac{(80 - 67{,}25)MPa}{-2t - 6t} * 2t = 86{,}4\ MPa$$

From support to web:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = 7{,}78\ MPa - \frac{(7{,}78 - 3)MPa}{-2t - 6t} * 2t = 10{,}2\ MPa$$

Extrapolation of the minimum principal stresses from web to support gives:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = -40{,}13\ MPa - \frac{(-40{,}13 - (-38{,}4))MPa}{-2t - 6t} * 2t = -41\ MPa$$

And from support to web:

$$b = y + \frac{y_1 - y_2}{x_1 - x_2} * x = -0{,}6\ MPa - \frac{(-0{,}6 - (-0{,}01))MPa}{-2t - 6t} * 2t = -0{,}9\ MPa$$

The stress range from web to support is then:

$$\Delta\sigma = \sigma_{max} - \sigma_{min} = \big(86{,}4 - (-41)\big)MPa = 127{,}4\ MPa$$

And from support to web we get:

$$\Delta\sigma = \sigma_{max} - \sigma_{min} = \big(10{,}2 - (-0{,}9)\big)MPa = 11{,}1\ MPa$$

Table 6 shows the stress ranges with the different mesh sizes. The ranges in Y-direction (from web to support) are nearly constant and the range in X-direction (support to web) has a relatively large increase with a mesh size of txt. Choosing a smaller mesh size than txt as DNV states will not affect the results much, so for the next concepts a mesh size of txt will be used. The largest of the X and Y stress range is chosen as the stress range, so the stress range to be used for fatigue is 139,5 MPa.

Table 6: Summary of stress range with different mesh size

Mesh	**Stress range [MPa]**	
Direction	X	Y
t x t	41,75	139,5
0,5t x 0,5t	10,31	132,6
0,25t x 0,25t	11,1	127,4

The equation below is used to check whether or not the concept is acceptable regarding fatigue:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{139{,}5}{\left(\frac{28{,}9}{0{,}38}\right)} = 1{,}83 < 1 \rightarrow \mathit{not\ ok}!$$

With the maximum stress range in this concept the fatigue utilization is 183 % when using a D-curve with 200 million cycles. As stated in the previous chapter the traveling block is not fully loaded on the way up, therefore the fatigue cycles is reduced to 100 million cycles. This concept can be casted. A casted sheave has no welds, therefore the stress range can be reduced by a factor of:

$$f_m = \frac{\sigma_t + 0{,}6\sigma_c}{\sigma_t + \sigma_c}$$

Where σ_t is maximum tension stress and σ_c is maximum compression stress. When the sheave is loaded the wire will only give compression stress to the sheave, the tension stress is zero. The reduction factor is then:

$$f_m = \frac{0 + 0{,}6\sigma_c}{0 + \sigma_c} = \frac{0{,}6}{1} = 0{,}6$$

The reduction factor is multiplied with the stress range [17]. From appendix in DNV-RP-C203 table A.5 the weld can be categorized as C.

With this category and reduction factor the equation below to check whether or not the concept is acceptable regarding fatigue becomes:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq f_m * \Delta\sigma \rightarrow \frac{\mathrm{f_m} * \Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{139{,}5 * 0{,}6}{\left(\frac{46}{0{,}38}\right)} = 0{,}70 < 1 \rightarrow \ ok!$$

With the reduction factor as above and using category C, the fatigue utilization of this concept 70 %.

6.3 STRAIGHT WEB WITH HOLES

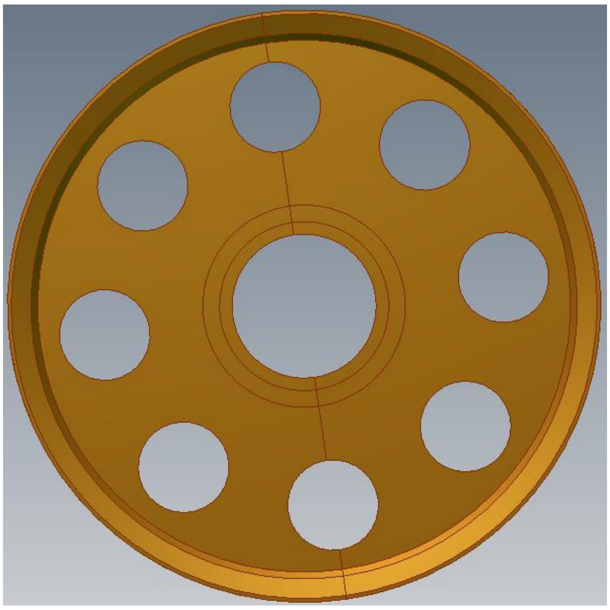


Figure 54: Straight with holes

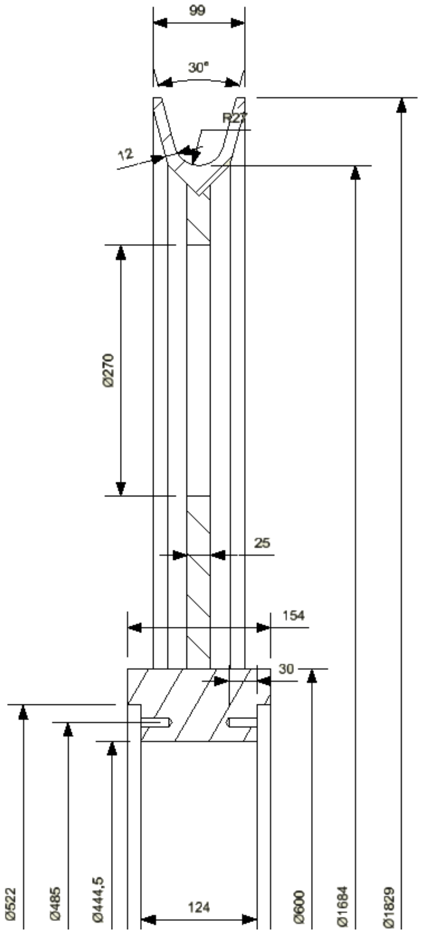


Figure 55: Cross section view

This concept is the same as the previous, but with holes to reduce the weight of the sheave. Inserted in the sheave are eight holes with a diameter of 270mm. This gives a removed area on the sheave of:

$$A_{removed} = \frac{\pi * D^2}{4} = \frac{\pi * 270^2}{4} = 57255{,}53\ mm^2$$

The thickness of the plate is 25mm and with the steel density of 7850 $\frac{kg}{m^3}$, the removed mass on sheave is:

$$m_{removed} = A_{removed} * t * \rho * n = 57255{,}53\ mm^2 * 25\ mm * 7850\,\frac{kg}{m^3} * 8 = 89{,}9\ kg$$

From stress plot of the previous concept it is seen that the holes must be close to the outer edge of the sheave, since that is where the lowest stresses are. Normal stress plot from ANSYS is shown below.

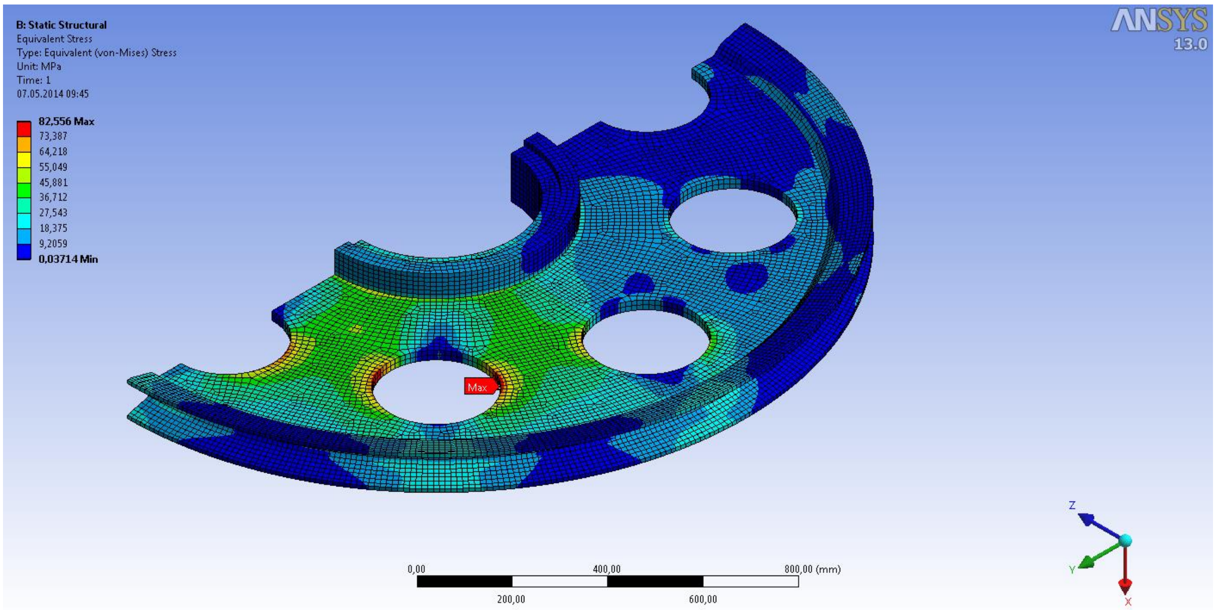


Figure 56: Normal stress plot for straight web with holes sheave

From Figure 56 the maximum normal stress occurs around the holes that were made. The stresses have a maximum value of 82,55 MPa. From the stress plot it is seen that a lot more area on the sheave is utilized and the utilization of the sheave are better.
The maximum stress occurs at the end of the holes. This is where the stress concentration is highest, which can be shown by the formula:

$$\sigma_{N,max} = \sigma_N \left(1 + 2 * \frac{a}{b}\right) \qquad [19]$$

Where a is the hole diameter and b is the width. At the edge around the hole$a = b$, the expression is then:

$$\sigma_{N,max} = \sigma_N * 3$$

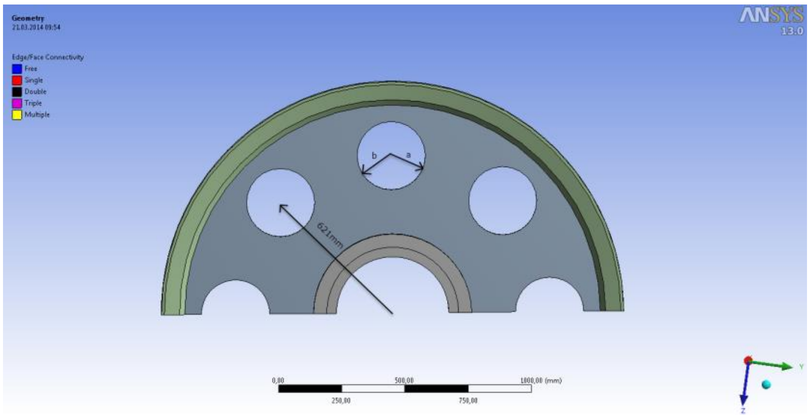


Figure 57: Measures for stress around holes

σ_N, at the point where the holes are, is then:

$$\sigma_N = \frac{F}{A} = \frac{90000kg*9{,}81\frac{m}{s^2}}{2*621mm*25mm} = 28{,}43\ MPa$$

$\sigma_{N,max}$ is then:

$$\sigma_{N,max} = 3 * \sigma_N = 3 * 28{,}43\ MPa = 85{,}3\ MPa$$

This corresponds well with the stress plot from ANSYS.

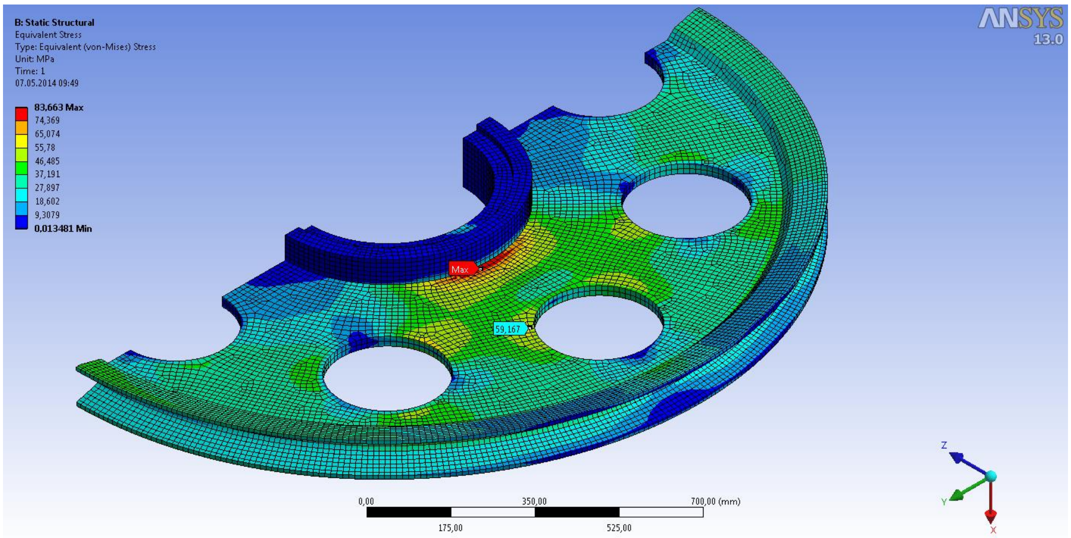


Figure 58: Bending stress plot for straight web with holes sheave

Figure 58 shows the bending stress with only the horizontal force acting on the sheave.

The bending stress has its maximum at the hub of the sheave, with a value of 83,66 MPa.

The bending stress around the holes are also shown in the figure. This value is approximetly 60 MPa.

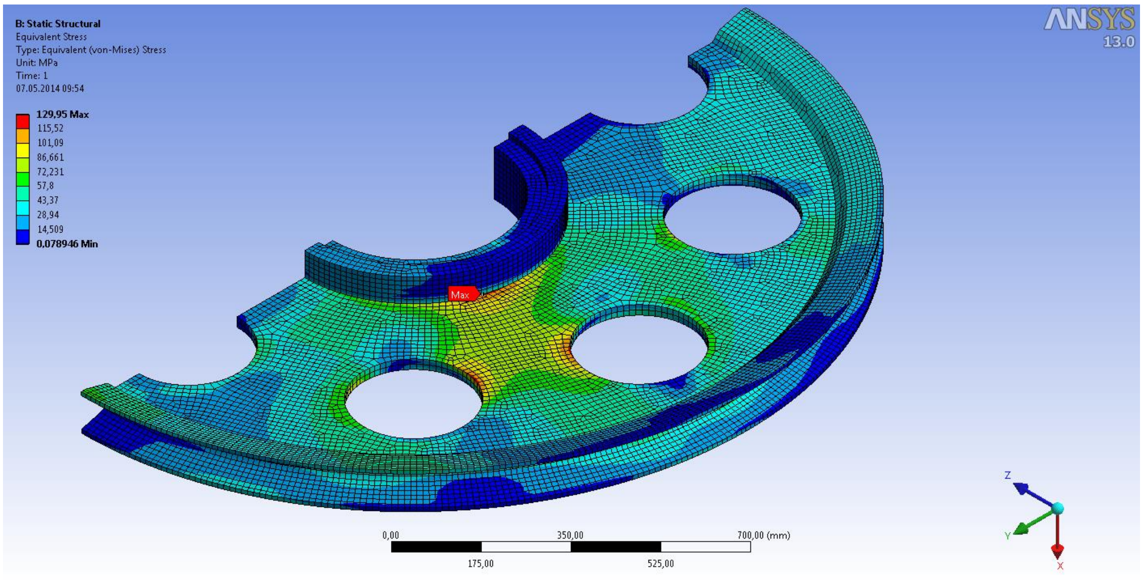


Figure 59: Combined equivalent stress plot for concept 2

Figure 59 illustrates the equivalent von Mises stress plot of this concept. It has the maximum stress at the inner edge of the sheave like the previous concept.

The maximum value is 130 MPa. The stress plot shows that the entire sheave is much more utilized.

6.3.1 BUCKLING

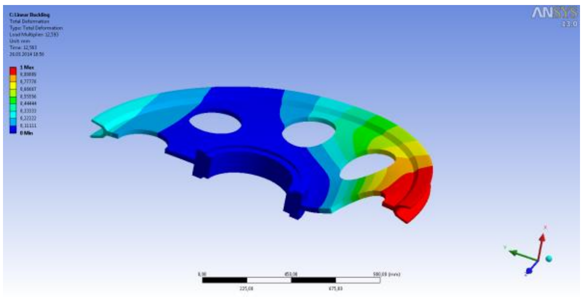


Figure 60: Buckling mode 1

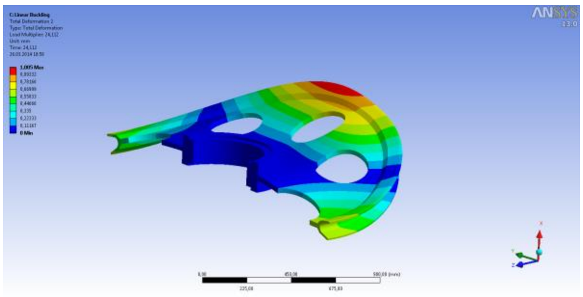


Figure 61: Buckling mode 2

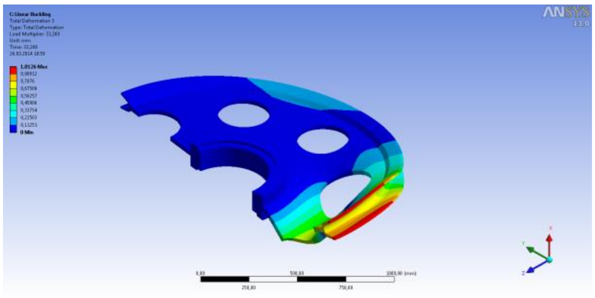


Figure 62: Buckling mode 3

The three buckling modes for concept 2 is shown in the figures above, the load multiplier to the respective mode is shown in Table 7 below;

Table 7: Buckling modes for straight web with holes sheave

Buckling mode	Load multiplier
1	11,9
2	23,9
3	31,3

From the graph in appendix the moment capacity for this sheave concepts is:

$$M_{Rd} = 15{,}8\ kNm$$

The load multiplier α from ANSYS is shown in Table 7. As seen from this table the sheave has good capacity for nonlinear buckling, as the previous concept. Checking for second order effect of buckling the following equation has to be fulfilled;

$$\frac{F}{F_E} + \frac{M}{M_{Rd} * \left(1 - \frac{F}{F_E}\right)} \leq 1$$

All the values needed to check for second order buckling are known, the equation becomes:

$$\frac{F}{F_E} + \frac{M}{M{,}Rd * \left(1 - \frac{F}{F_E}\right)} = \frac{F}{F * \alpha} + \frac{8{,}35 * 10^6}{15{,}8 * 10^6 * \left(1 - \frac{F}{F * \alpha}\right)} = \frac{1}{11{,}9} + 0{,}53 * \frac{1}{\left(1 - \frac{1}{11{,}9}\right)} = 0{,}66 \leq 1 \rightarrow ok!$$

This concept is considered acceptable regarding second order effect of buckling.

6.3.2 REACTION FORCES

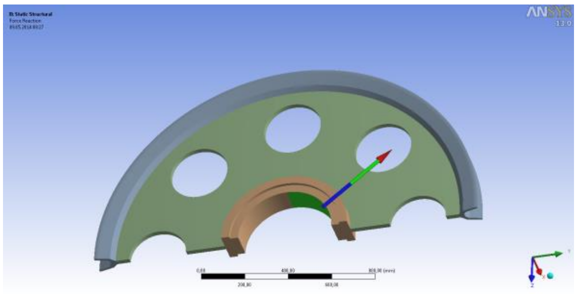


Figure 63: Total reaction force on cylindrical support

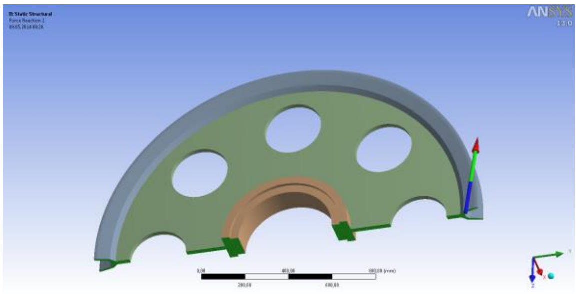


Figure 64: Total reaction force on frictionless support

Table 8: Reaction forces for straight web with holes sheave

Support	Direction		
	X [kN]	Y [kN]	Z [kN]
Cylindrical	15,45	442,3	-395,2
Frictionless	-0,1	4,35	-43,43
Total	15,35	446,65	-438,63

These values are approximetly the same as the values in the previous concept, so the numerical aspect of the calculations from ANSYS are consdierd to be correct.

6.3.3 FATIGUE

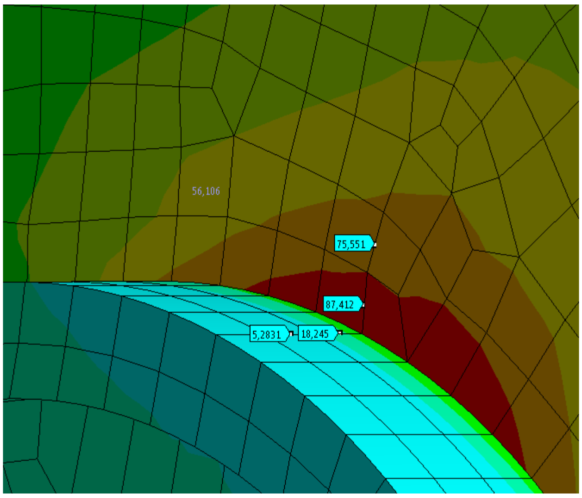


Figure 65: Maximum principal hotspot stress plot

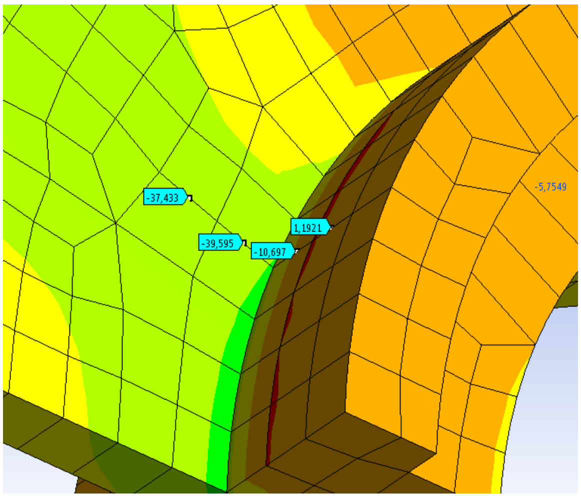


Figure 66: Minimum principal hotspot stress plot

Figures above show the maximum and minimum principal stress plot 0,5t and 1,5t away from the hot spot region. The maximum hot spot stress from web to support is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 87{,}4\ MPa - \frac{(87{,}4 - 75{,}5)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = 93{,}35\ MPa$$

From support to web:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 18{,}24\ MPa - \frac{(18{,}24 - 5{,}3)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = 24{,}7\ MPa$$

Minimum hot spot stress from web to support is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -39{,}6\ MPa - \frac{(-39{,}6 - (-37{,}4))MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -40{,}7\ MPa$$

And from support to web, the minimum hot spot stress is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -10{,}7\ MPa - \frac{(-10{,}7 - 1{,}2)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -16{,}65\ MPa$$

Stress ranges are shown in Table 9 below

Table 9: Stress range for straight web with holes sheave

Direction	$\Delta\sigma$ [MPa]
Web to support	134,04
Support to web	41,4

The equation below is used to check whether or not the concept is acceptable regarding fatigue:

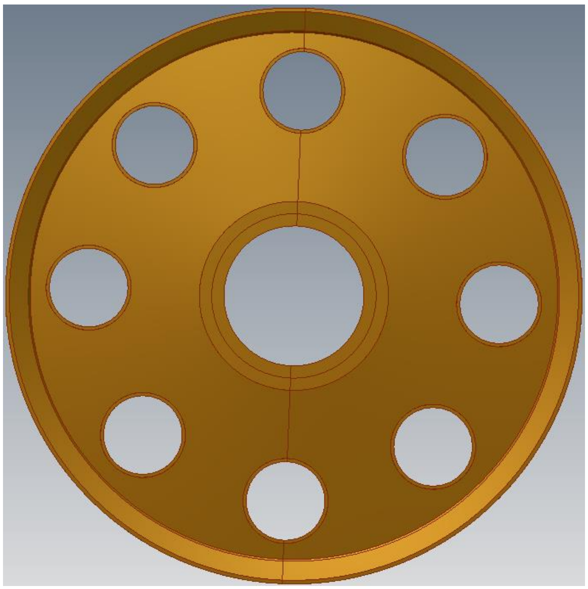
$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{134{,}04}{\left(\frac{28{,}9}{0{,}38}\right)} = 1{,}76 < 1 \rightarrow not\ ok!$$

With the maximum stress range in this concept the fatigue utilization is 176 %. This design concept can also be casted, no welds. With the same conditions as previous concept, 6.2.3, the equation below to check whether or not the concept is acceptable regarding fatigue becomes:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq f_m * \Delta\sigma \rightarrow \frac{\mathrm{f_m} * \Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{134 * 0{,}6}{\left(\frac{46}{0{,}38}\right)} = 0{,}67 < 1 \rightarrow\ ok!$$

With the reduction factor and using category C, the fatigue utilization of this concept 67 %.

6.4 DOUBLE WEB WITH HOLES



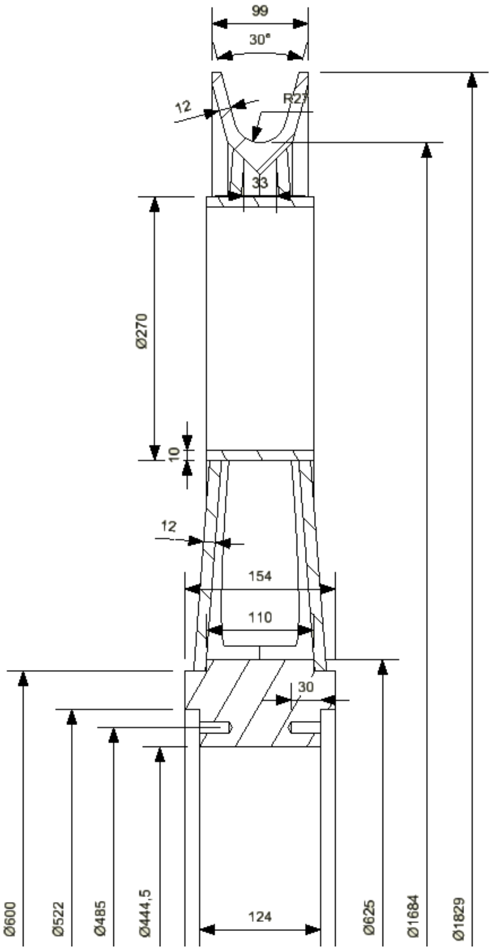


Figure 67: Double slanted plate with holes

Figure 68: Cross section view

The double webbed sheave that is currently being used is shown above. A version of this sheave is the next concept, double webbed sheave with holes in it.

Adding holes to the sheave will reduce the weight. The holes added to the sheave needs to be supported. The cylinders that are inserted in the holes also have a mass, so the holes have to be large enough to remove more mass than what is add with the cylinders.
In this design, 8 holes with a diameter of 270mm are added. The outer edge of the holes is a distance of 50mm from the end of the web. With a steel density $\rho = 7850 \frac{kg}{m^3}$ this gives a removed mass on each web of:

$$m_{removed} = \rho * \frac{d^2 * \pi}{4} * t * n = 7850\frac{kg}{m^3} * \frac{270^2 mm^2 * \pi}{4} * 12mm * 8\ holes = 43{,}15\ kg$$

Where ρ is the steel density, d is the diameter of the hole, t is the thickness of the web and n is number of holes in the web. The concept consists of two plates, so the total mass removed with the holes dimension chosen is:

$$m_{removed,tot} = 43{,}15\ kg * 2 = 86{,}3\ kg$$

The cylinders supporting the holes are 270mm in outer diameter, the wall thickness is 10mm and they are 110mm high. This gives the total added mass of:

$$m_{added} = \frac{\pi(d_o^2 - d_i^2)}{4} * h * \rho * n = \frac{\pi(270^2 - 250^2)mm^2}{4} * 120mm * 7850\frac{kg}{m^3} * 8 = 61{,}5\ kg$$

So in total the removed mass is approximately 25 kg.

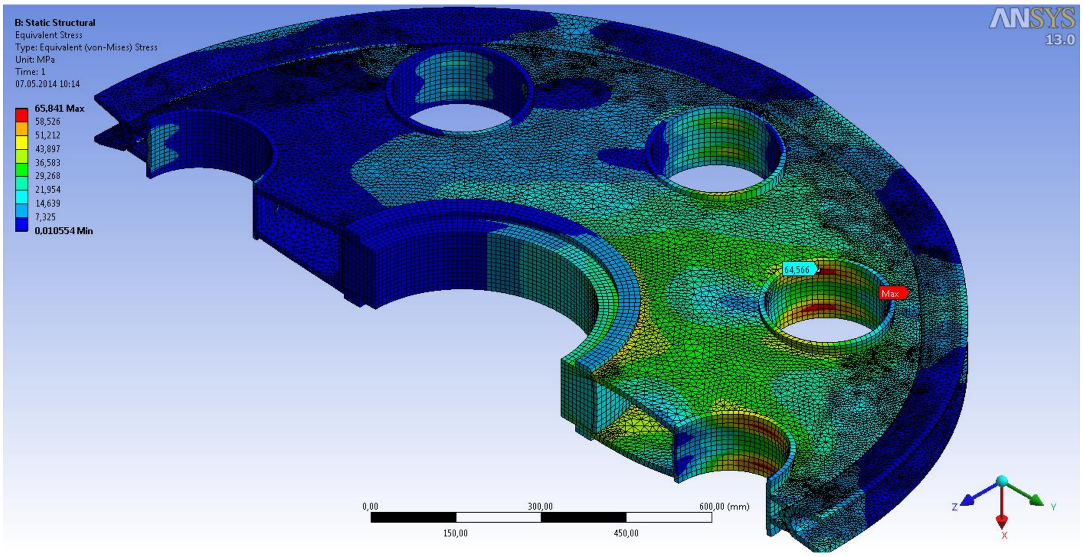


Figure 69: Normal stress plot for double web with holes sheave

Figure 69 shows the normal stress plot from ANSYS. From stress analysis it is seen that the maximum stress is located at the top of one of the web. This is a stress concentration in three elements. The stress of interest is around the holes in the sheave. The stress here has a maximum value of 64,56 MPa. From hand calculations the normal stress at this point is:

$$\sigma_N = \frac{F}{A} = \frac{90000kg * 9{,}81\frac{m}{s^2}}{652mm * 2 * 12mm * 2} = 28{,}2\ MPa$$

The normal stress is 28,2 MPa in the layer where maximum stress occurs. The maximum stress around the holes is then:

$$\sigma_{N,max} = \sigma_N * 3 = 3 * 28,2\ MPa = 84,6\ MPa$$

The calculated stress is slightly higher than the result from ANSYS. This is because the cylinders supporting the holes are not included in the hand calculation.

The bending stress results from ANSYS are shown in Figure 70. Cylinders in between the webs, and the slanted webs, will stiffen the plate and make it very suitable for bending loads. The bending stresses in this concept are low compared to other concepts. Maximum bending stress occurs at the support where the cylindrical support is located. The stress at the end of the web is

30,5 MPa.

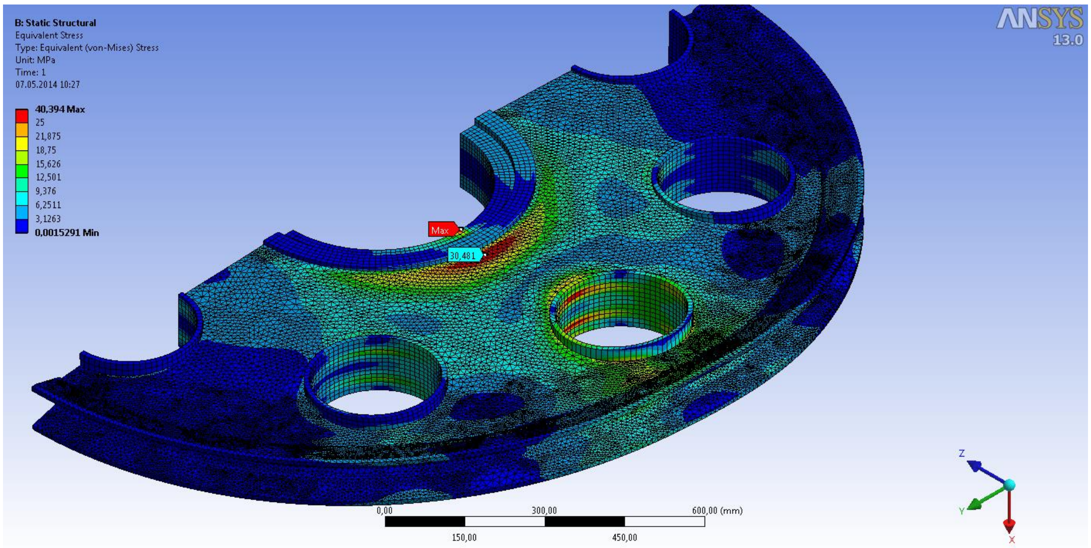


Figure 70: Bending stress plot for double web with holes sheave

The combined equivalent stress plot from ANSYS is illustrated in Figure 71 below

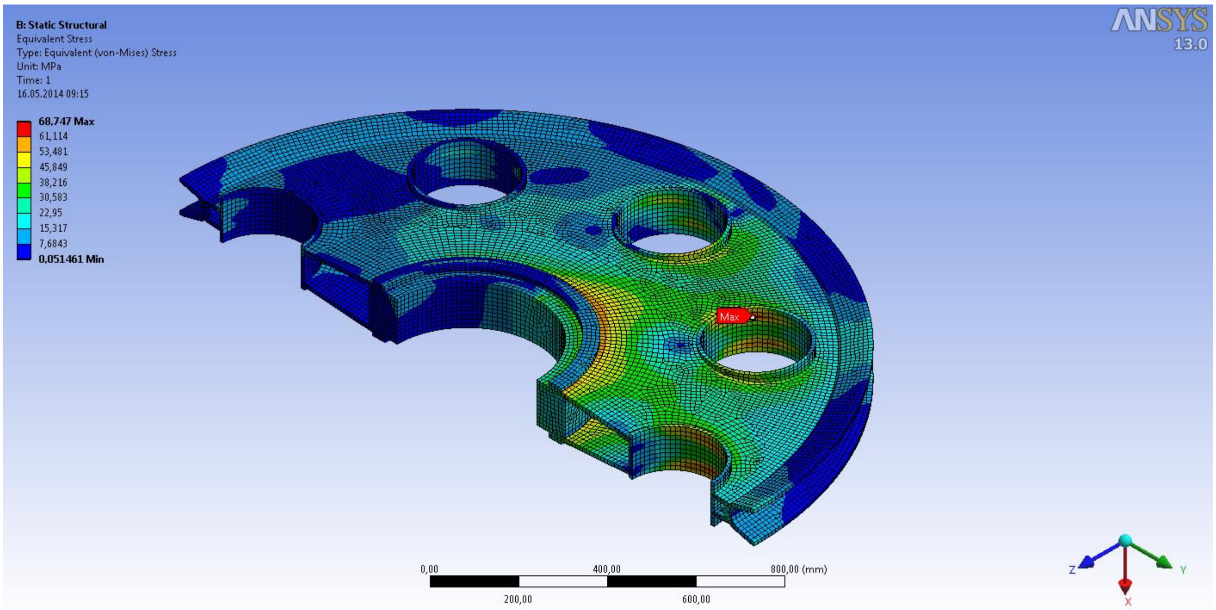


Figure 71: Stress plot for double web with holes sheave

As shown in figure above the combined equivalent stress is highest on the cylinders around the holes, with a maximum value of 68,75 MPa. This sheave has the lowest equivalent stress of all the concepts tested. But the weight is relativly high compared to the other concepts.

6.4.1 BUCKLING

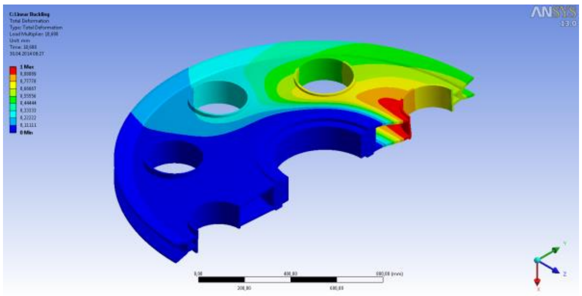


Figure 72: Buckling mode 1

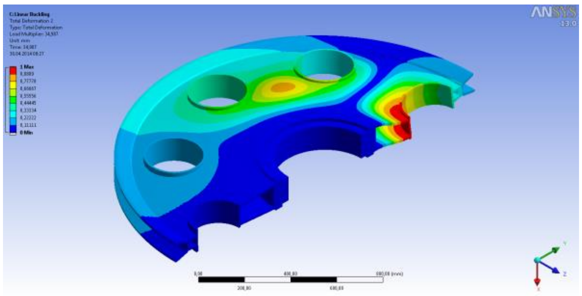


Figure 73: Buckling mode 2

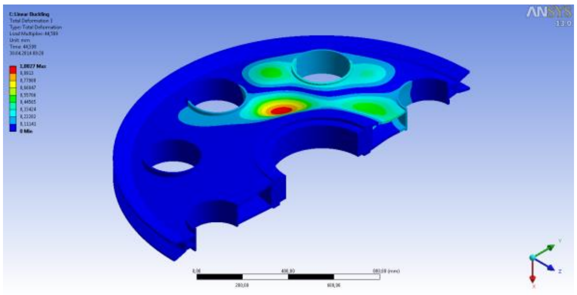


Figure 74: Buckling mode 3

Figures above show the three buckling modes for this concept. The load multiplier for the respective buckling mode is shown in Table 10

Table 10: Buckling modes for double web with holes sheave

Buckling mode	Load multiplier
1	18,7
2	35
3	44,6

From graph in appendix the moment capacity at this point is:

$$M_{Rd} = 42{,}5\ kNm$$

All the values needed to check for second order buckling are known:

$$\frac{F}{F_E} + \frac{M}{M_{Rd} * \left(1 - \frac{F}{F_E}\right)} = \frac{F}{\alpha * F_E} + \frac{8{,}35}{42{,}5 * \left(1 - \frac{F}{\alpha * F}\right)} = \frac{1}{18{,}7} + 0{,}19 * \frac{1}{\left(1 - \frac{1}{18{,}7}\right)} = 0{,}26 < 1 \rightarrow ok!$$

This concept is considered acceptable regarding second order effect of buckling.

6.4.2 REACTION FORCES

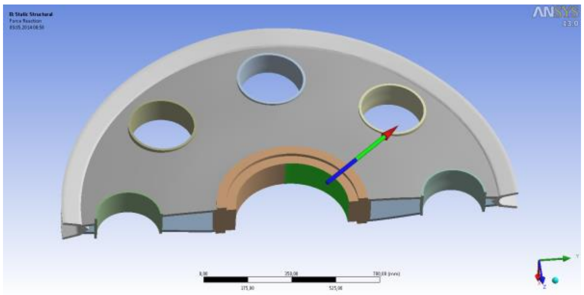


Figure 75: Total cylindrical reaction force

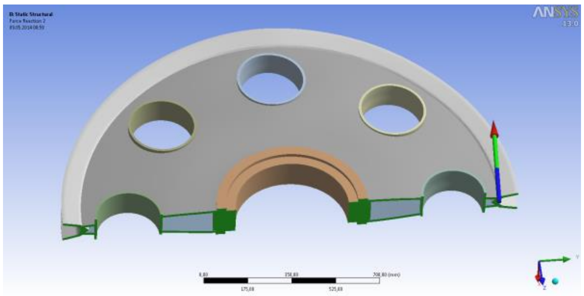


Figure 76: Total frictionless reaction force

Figures above shows the total cylindrical support reaction force on the left hand side, and the frictionless support reaction force on the right hand side.

Table 11: Reaction forces for double web with holes sheave

Support	Direction		
	X [kN]	Y [kN]	Z [kN]
Cylindrical	15,45	442,3	-391,7
Frictionless	-0,1	3,08	-48
Total	15,35	445,38	-439,7

These values are approximetly the same as the values in the previous concept, so the numerical aspect of the calculations from ANSYS are consdierd to be correct.

6.4.3 FATIGUE

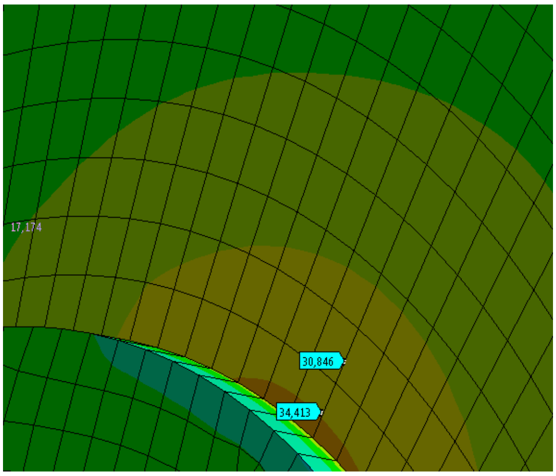


Figure 77: Maximum principal hotspot stress plot

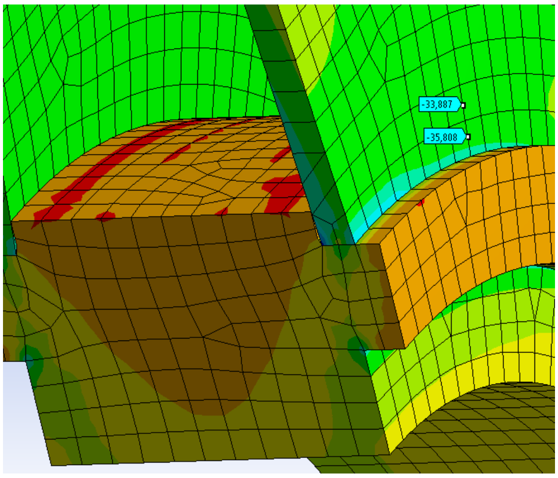


Figure 78: Minimum principal hotspot stress plot

Figures above show the maximum and minimum principal stress plot 0,5t and 1,5t away from the hot spot region. The maximum hot spot stress from web to support is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 34{,}4\ MPa - \frac{(34{,}4 - 30{,}85)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = 36{,}2\ MPa$$

Minimum hot spot stress from web to support is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -35{,}8\ MPa - \frac{(-35{,}8 - (-33{,}9))MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -36{,}8\ MPa$$

Stress range is shown in Table 12 below

Table 12: Stress range for double web with holes sheave

Direction	$\Delta\sigma$ [MPa]
Web to support	73

The equation below is used to check whether or not the concept is acceptable regarding fatigue:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{73}{\left(\frac{28{,}9}{0{,}38}\right)} = 0{,}96 < 1 \rightarrow\ ok!$$

With the maximum stress range in this concept the fatigue utilization is 96 %.

As stated in section 6.1.6, the weld category could be defined as F. This gives a utilization of 107 %.

6.5 THIN WEB WITH STIFFENERS

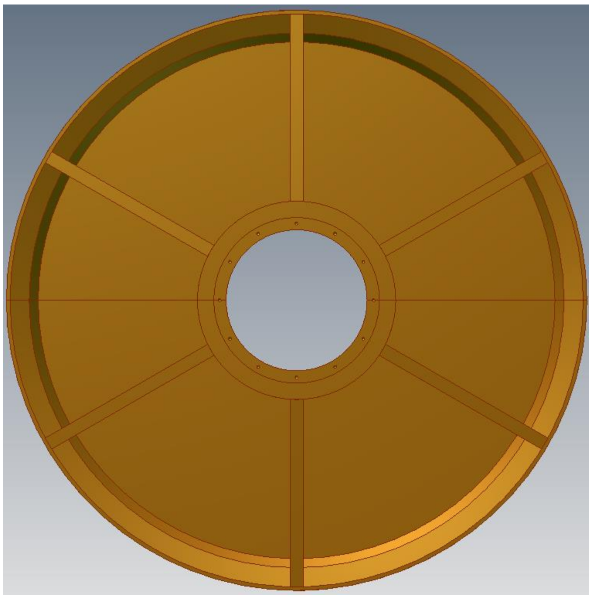


Figure 79: Sheave with thin plate and stiffeners

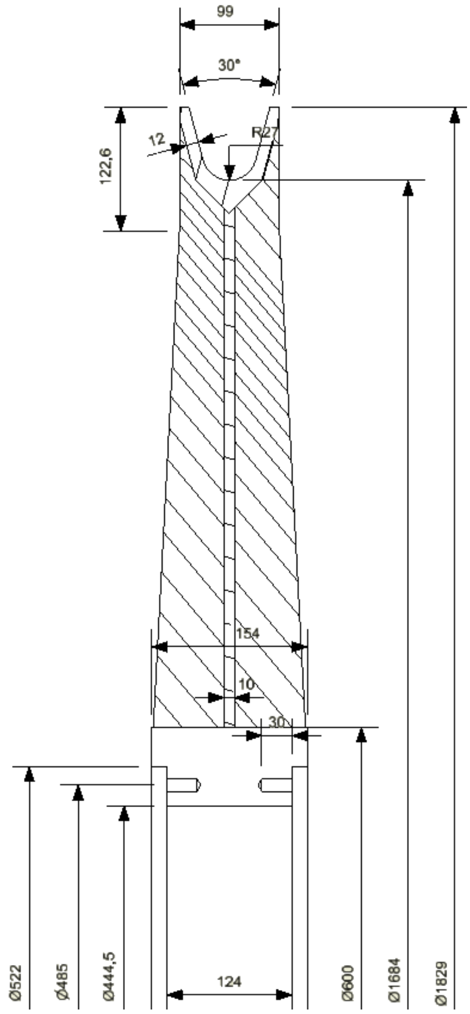


Figure 80: Cross sectional view

A different concept of sheave design is to have a thin web supported by stiffeners around the sheave as shown in Figure 79. The web is 10mm thick and the stiffeners go from outer edge of support to the top of the groove as shown in Figure 79.

In this concept there are six stiffeners supporting a thin web. By selecting a thin web, the mass of the sheave is greatly reduced. Normal stress plot from ANSYS is illustrated in Figure 81 below. ANSYS shows a maximum normal stress of 77 MPa. This is located where the stiffener meets the groove. This is a stress concentration as shown further down in this section. Hand calculations give a maximum normal stress at the inner edge of:

$$\sigma_N = \frac{F}{A} = \frac{90000kg*9{,}81\frac{m}{s^2}}{625mm*10mm} = \frac{141{,}27MPa}{2} = 70{,}63\ MPa$$

From Figure 81 it is shown that the stresses at the end of the web is 69,7 MPa. This corresponds well with the hand calculated stresses.

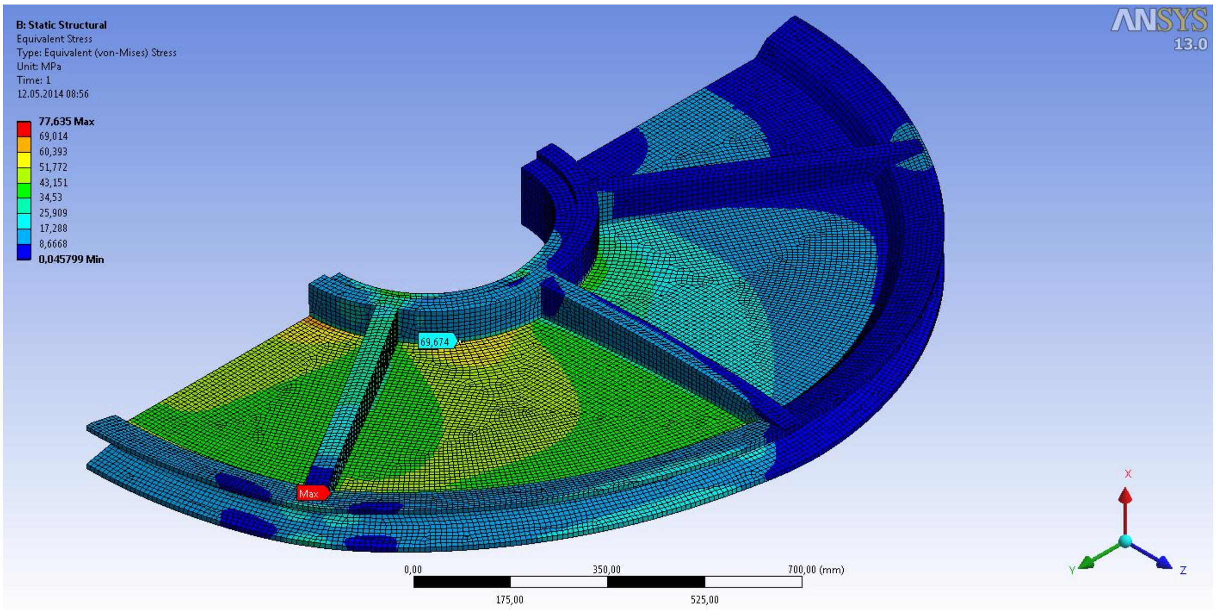


Figure 81: Normal stress plot for thin web with stiffeners sheave

In the bending stress plot, shown in Figure 82, it is shown that the stiffener takes nearly all the bending stresses. From ANSYS the maximum bending stress at this point is 87,8 MPa.

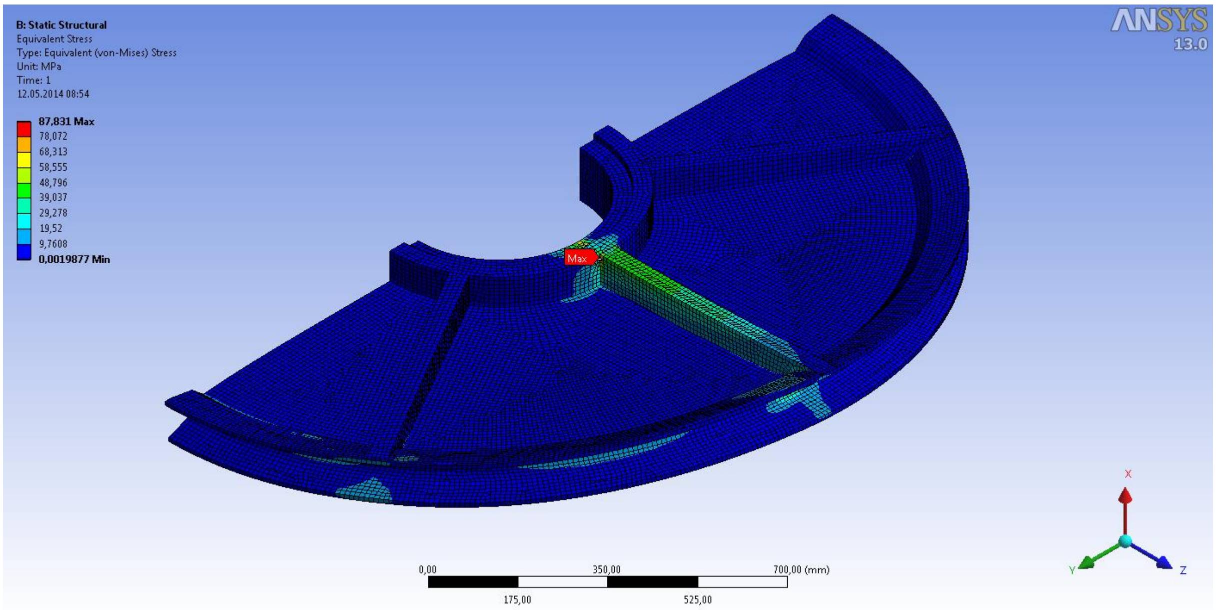


Figure 82: Bending stress plot for thin web with stiffeners sheave

Figure 83 shows the combined equivalent stress plot from ANSYS. The maximum stress of interest occurs at the same place as the highest bending stresses were, with a value of 111,2 MPa.

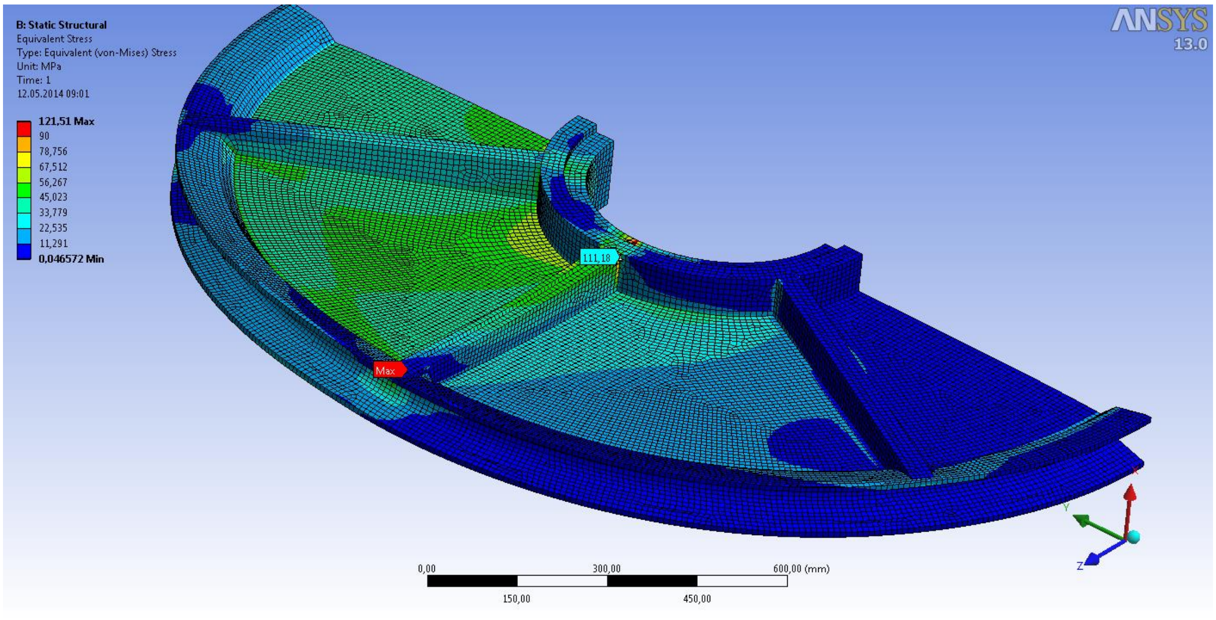


Figure 83: Combined equivalent stress plot for thin web with stiffeners sheave

Maximum stresses in some of the pictures above occur at the top of the middle stiffener. Figure 84 shows the average- and non-average maximum stress at this point. The right hand side picture shows the non-average stress plot. As shown in this picture, the maximum non-average stress at this point is 336 MPa. Averaging this stress to the other elements on the groove gives high stresses at this point. The other elements have normal stress values around this stress concentration. Therefore the stresses on the elements in figures below are not considered as the maximum stress value.

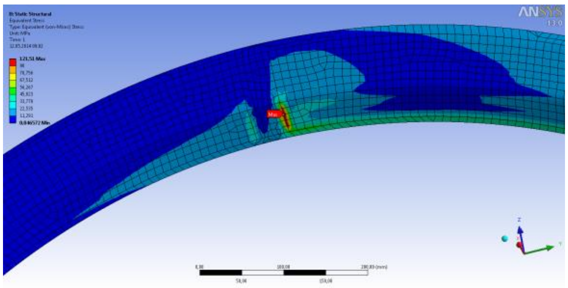


Figure 84: Average maximum stress

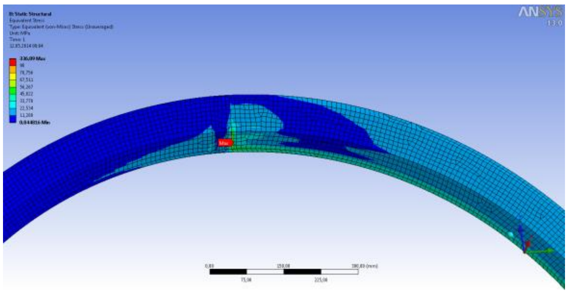


Figure 85: non-average maximum stress

6.5.1 BUCKLING

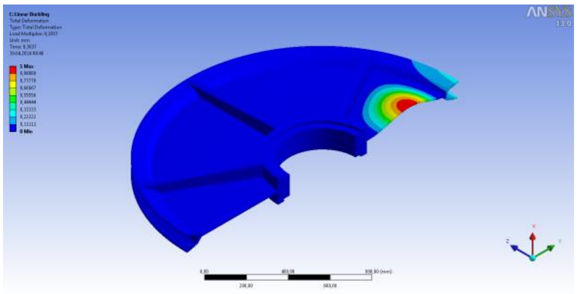


Figure 86: Buckling mode 1

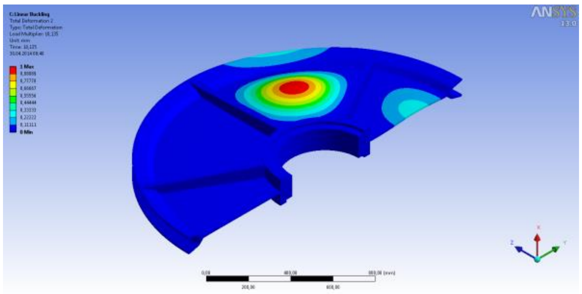


Figure 87: Buckling mode 2

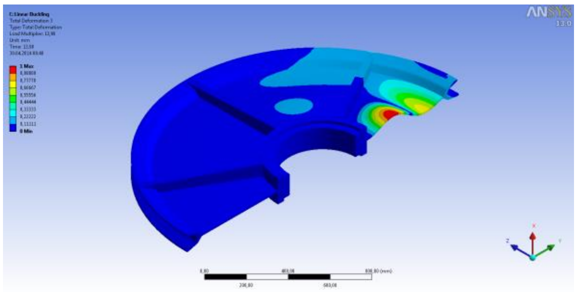


Figure 88: Buckling mode 3

The three buckling modes are shown in figures above and their load multiplier is presented in Table 13 below:

Table 13: Buckling modes for thin web with stiffeners sheave

Buckling mode	Load multiplier
1	9,3
2	10,2
3	14

From graph in appendix the moment capacity at this point is:

$$M_{Rd} = 15,02$$

Checking for second order effect of buckling:

$$\frac{F}{F_E} + \frac{M}{M_{Rd} * \left(1 - \frac{F}{F_E}\right)} = \frac{F}{F * \alpha} + \frac{8,35}{15,02 * \left(1 - \frac{F}{F * \alpha}\right)} = \frac{1}{9,3} + 0,56 * \frac{1}{\left(1 - \frac{1}{9,3}\right)} = 0,73 \leq 1 \rightarrow ok!$$

This concept is considered acceptable regarding second order effect of buckling.

6.5.2 REACTION FORCES

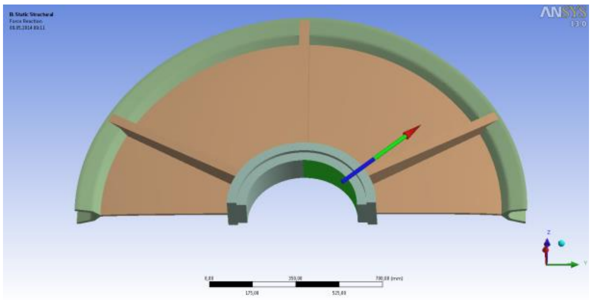


Figure 89: Total reaction force on cylindrical support

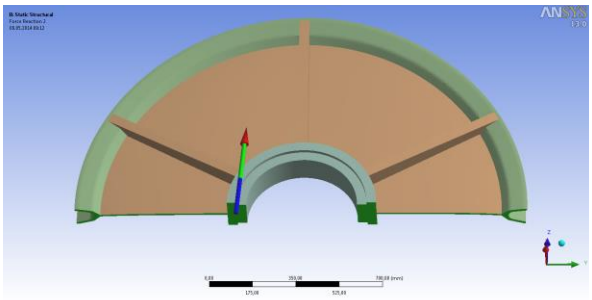


Figure 90: Total reaction force on frictionless support

Figures above shows the total cylindrical support reaction force on the left hand side, and the frictionless support reaction force on the right hand side for concept 4.

Table 14: Reaction forces for thin web with stiffeners sheave

Support	Direction		
	X [kN]	Y [kN]	Z [kN]
Cylindrical	-15,45	442,3	412,5
Frictionless	0	2,08	26,5
Total	15,45	444,38	439

These values are approximetly the same as the values in the previous concept, so the numerical aspect of the calculations from ANSYS are consdierd to be correct.

6.5.3 FATIGUE

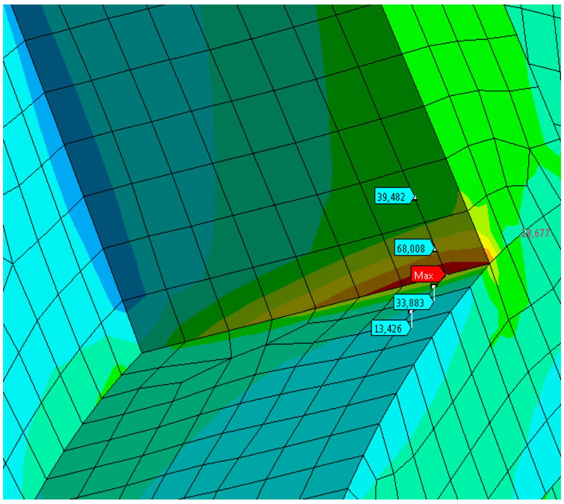


Figure 91: Minimum principal stress

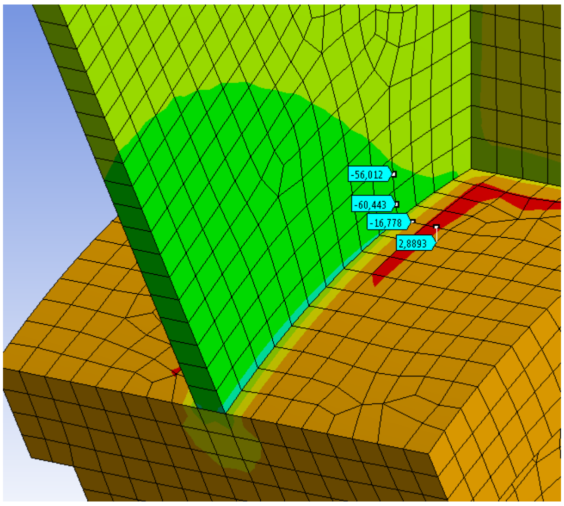


Figure 92: Maximum principal stress

Figures above show the maximum and minimum principal stress plot 0,5t and 1,5t away from the hot spot region. The maximum hot spot stress from stiffener to support is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 68\ MPa - \frac{(68-39,5)MPa}{0,5t-1,5t} * 0,5t = 82,25\ MPa$$

From support to stiffener:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 33,8\ MPa - \frac{(33,8-13,4)MPa}{0,5t-1,5t} * 0,5t = 44\ MPa$$

Minimum hot spot stress from web to support:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -60,4\ MPa - \frac{(-60,4-(-56,01))MPa}{0,5t-1,5t} * 0,5t = -62,6\ MPa$$

And from support to web, the minimum hot spot stress is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -16,8\ MPa - \frac{(-16,8-2,9)MPa}{0,5t-1,5t} * 0,5t = -26,65\ MPa$$

Stress range are shown in Table 15 below:

Table 15: Stress range for thin web with stiffeners sheave

Direction	$\Delta\sigma$ [MPa]
Web/stiffener to support	144,9
Support to web/stiffener	70,64

The equation below is used to check whether or not the concept is acceptable regarding fatigue:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{144{,}9}{\left(\frac{28{,}9}{0{,}38}\right)} = 1{,}9 < 1 \rightarrow not\ ok!$$

With the maximum stress range in this concept the fatigue utilization is 190 %. This design concept can be casted as the straight web design concepts. By using the same conditions as straight web concept, 6.2.3, the equation below to check whether or not the concept is acceptable regarding fatigue becomes:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq f_m * \Delta\sigma \rightarrow \frac{\mathrm{f_m} * \Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{145 * 0{,}6}{\left(\frac{46}{0{,}38}\right)} = 0{,}72 < 1 \rightarrow\ ok!$$

With the reduction factor and using category C, the fatigue utilization of this concept 72 %.

6.6 THIN WEB WITH STIFFENERS AND HOLES

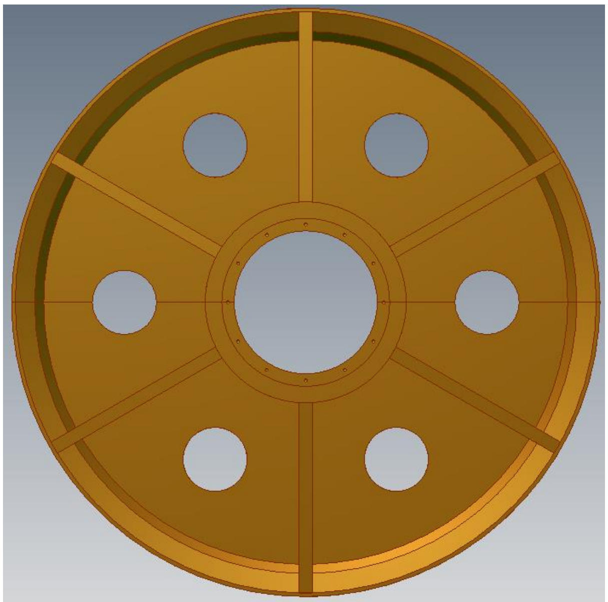


Figure 93: With stiffeners and holes

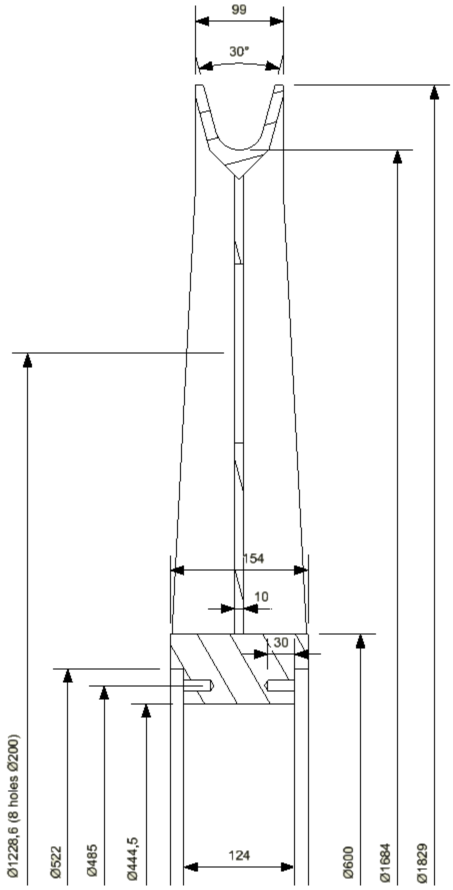


Figure 94: Cross section view

This is the same concept as previous, but with holes in between the stiffeners. Again the holes are added to reduce the mass of the sheave and make it more utilized.

As shown from the stress plot of previous concept in Figure 81, the thin web has a decent utilization on its entire length. Adding holes to the web has to be done with caution to not make the sheave incapable to the load conditions.

A total of six holes between each stiffene,r with a diameter of 200mm were added. They are located 100mm from the outer edge of the web. Adding six holes in the 10mm plate will remove a mass of:

$$m_{removed} = \rho * A * t * n = 7850\,\frac{kg}{m^3} * \frac{\pi * 200^2}{4} mm^2 * 10mm * 6 = 14{,}8\;kg$$

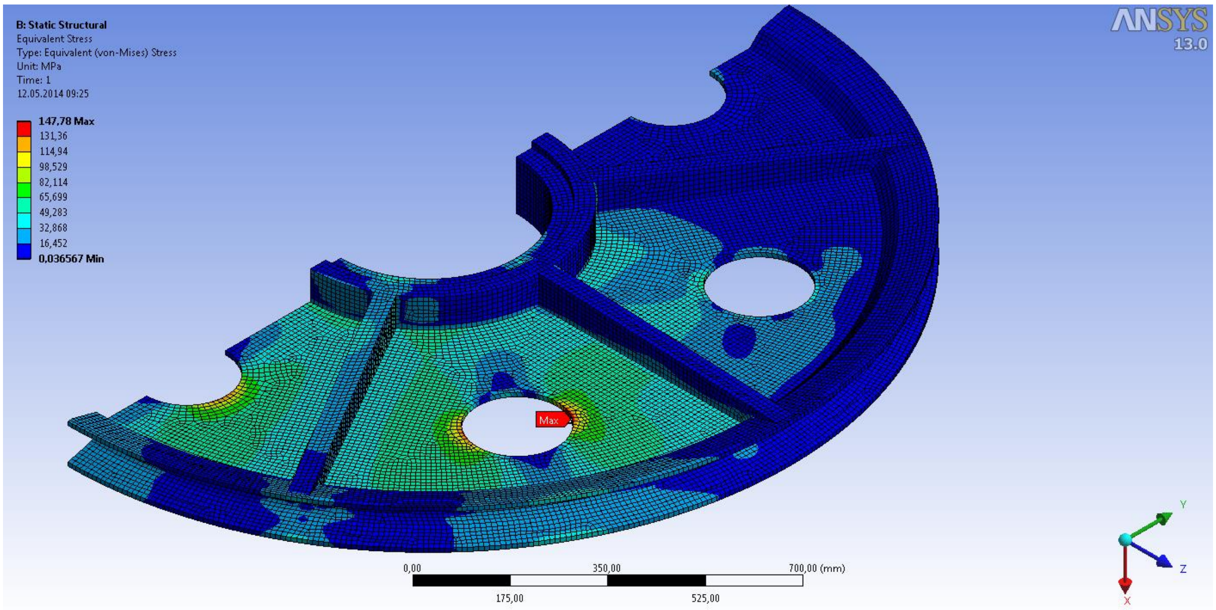


Figure 95: Normal stress plot for thin web with stiffeners and holes sheave

Figure 95 illustrates the normal stress plot of this concept. Maximum stress occurs around the holes with a value of 147,8 MPa.

Maximum normal stress around the holes from hand calculations gives:

$$\sigma_{N,max} = \sigma\left(1 + 2 * \frac{a}{b}\right) = \sigma * 3 = \frac{90000kg*9{,}81\frac{m}{s^2}}{2*614mm*10mm} * 3 = 215{,}7\ MPa$$

This is much higher than the results from ANSYS. In hand calculations, the stiffeners are not taking into account. The stiffeners will take some of the normal stresses and the stresses around the holes will therefore go down.

ANSYS bending stress plot is shown in Figure 96 with a maximum value of 85,5 MPa. This is at the same place as previous concept, at the end of the stiffener.

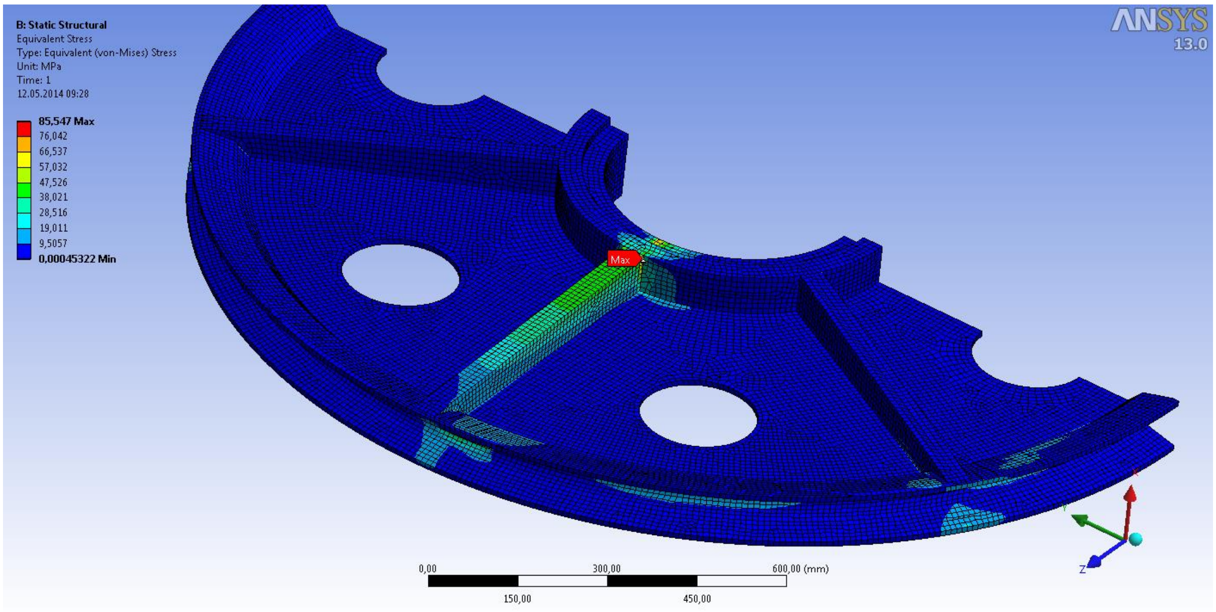


Figure 96: Bending stress plot for thin web with stiffeners and holes sheave

Combined stress plot from ANSYS of this concept is shown in Figure 97 below. Maximum stress is at the same place as maximum normal stress, with a value of 151 MPa. The holes nearly don’t get any bending stress since the stiffeners are taking up the entire side load.

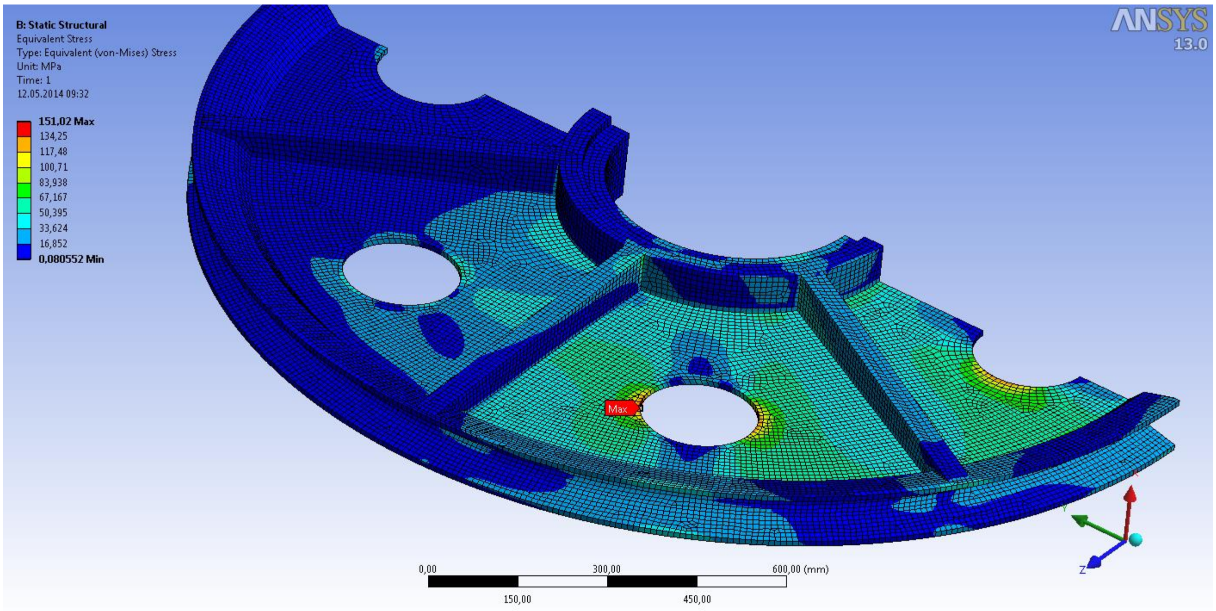


Figure 97: Stress plot for thin web with stiffeners and holes sheave

6.6.1 BUCKLING

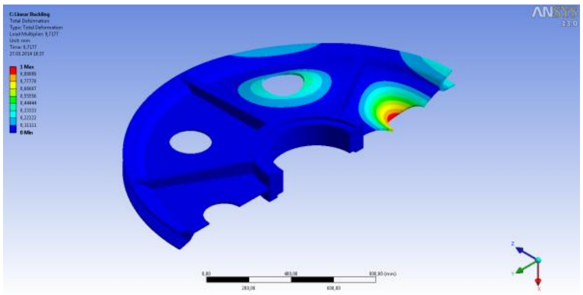


Figure 98: Buckling mode 1

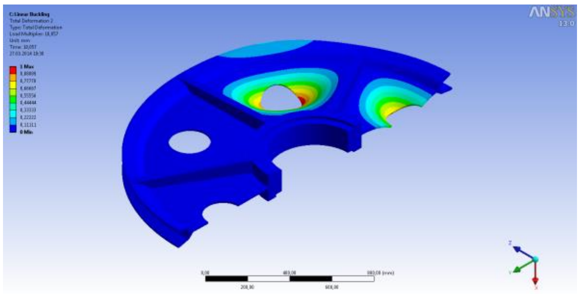


Figure 99: Buckling mode 2

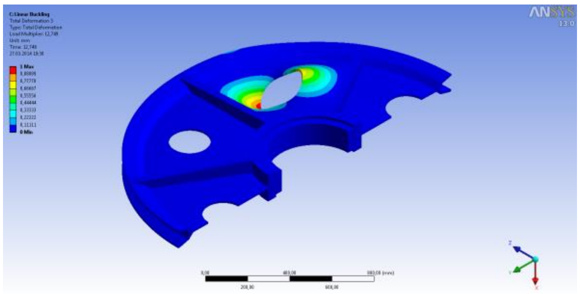


Figure 100: Buckling mode 3

Above the three buckling modes from ANSYS are shown. The load multiplier to each mode is presented in Table 16 below.

Table 16: Buckling modes for thin web with stiffeners and holes sheave

Buckling mode	Load multiplier
1	9,72
2	10,06
3	12,75

From graph in appendix the moment capacity at this point is:

$$M_{Rd} = 15{,}4\ kNm$$

Checking for second order effect of buckling;

$$\frac{F}{F_E} + \frac{M}{M_{Rd} * \left(1 - \frac{F}{F_E}\right)} = \frac{F}{F * \alpha} + \frac{8{,}35}{15{,}4 * \left(1 - \frac{F}{F * \alpha}\right)} = \frac{1}{9{,}72} + 0{,}54 * \frac{1}{\left(1 - \frac{1}{9{,}72}\right)} = 0{,}71 \leq 1 \rightarrow ok!$$

This concept is considered acceptable regarding second order effect of buckling.

6.6.2 REACTION FORCE

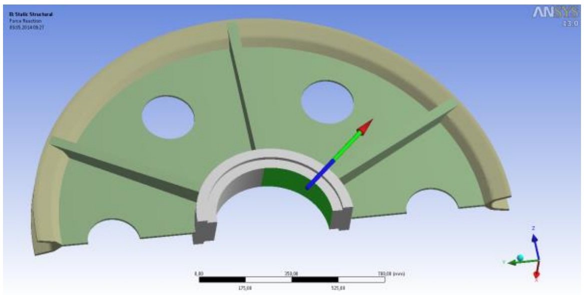


Figure 101: Total reaction force on cylindrical support

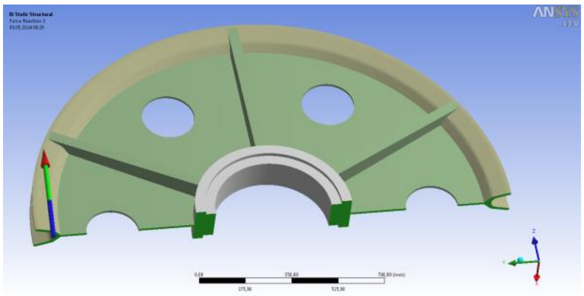


Figure 102: Total reaction force on frictionless support

Figures above shows the total cylindrical support reaction force on the left hand side, and the frictionless support reaction force on the right hand side for concept 5.

Table 17: Reaction forces for thin web with stiffeners and holes sheave

Support	Direction		
	X [kN]	Y [kN]	Z [kN]
Cylindrical	15,45	-442,3	416,7
Frictionless	0	-1,9	22,2
Total	15,45	-444,2	438,9

These values are approximetly the same as the values in the previous concept, so the numerical aspect of the calculations from ANSYS are consdierd to be correct.

6.6.3 FATIGUE

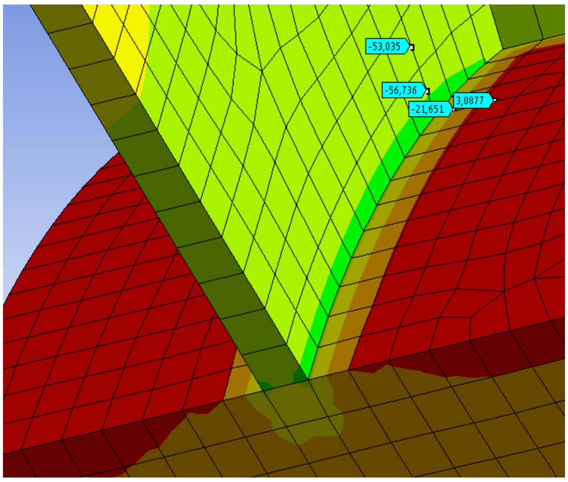


Figure 103: Minimum principal stress

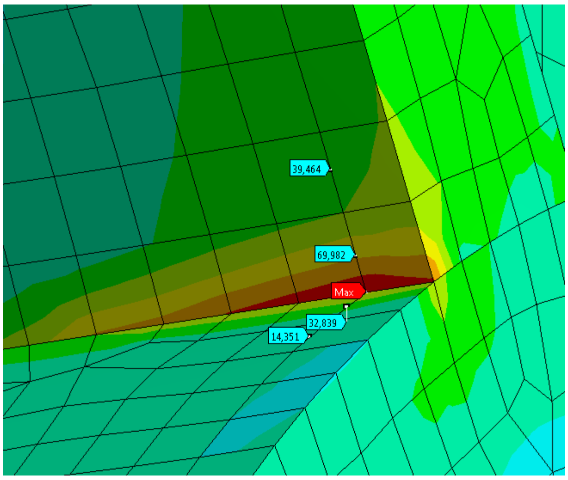


Figure 104: Maximum principal stress

Figures above show the maximum and minimum principal stress plot 0,5t and 1,5t away from the hot spot region. The maximum hot spot stress from stiffener to support is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 69{,}9\ MPa - \frac{(69{,}9 - 39{,}5)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = 85{,}1\ MPa$$

From support to stiffener:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 32{,}8\ MPa - \frac{(32{,}8 - 14{,}35)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = 42{,}03\ MPa$$

Minimum hot spot stress from web to support:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -56{,}74\ MPa - \frac{(-56{,}74 - (-53{,}03))MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -58{,}6\ MPa$$

And from support to web, the minimum hot spot stress is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -21{,}65\ MPa - \frac{(-21{,}65 - 3{,}08)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -34{,}01\ MPa$$

Stress range isshown in Table 18 below:

Table 18: Stress range for thin web with stiffeners and holes sheave

Direction	$\Delta\sigma$ [MPa]
Web/stiffener to support	143,7
Support to web/stiffener	76,04

The equation below is used to check whether or not the concept is acceptable regarding fatigue:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{143{,}7}{\left(\frac{28{,}9}{0{,}38}\right)} = 1{,}89 < 1 \rightarrow not\ ok!$$

With the maximum stress range in this concept the fatigue utilization is 189 %.

By using the same conditions as straight web concept, 6.2.3, the equation below to check whether or not the concept is acceptable regarding fatigue becomes:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq f_m * \Delta\sigma \rightarrow \frac{\mathrm{f_m} * \Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{144 * 0{,}6}{\left(\frac{46}{0{,}38}\right)} = 0{,}72 < 1 \rightarrow\ ok!$$

With the reduction factor and using category C, the fatigue utilization of this concept 72 %.

6.7.0 WEB WITH DECREASING THICKNESS

By evaluating the different concptes above and from the calculations made, it is shown that the stresses are low at the outer edge of the sheave and increases toawards the sheave hub. Beacause of this stress distribution, a concecpt where the thickness of the web increases towards center is designed. Having a sheave with decreasing thickness gives a lower mass and thereby smaller moment of inertia.

As shown in the simple calculations above the sheave have to be atleast 9,3 mm at the inner edge of the sheave. At the outer edge of the sheave, the thickness had to be larger than 4 mm by using the same eqatuion when only the normal stress is acting on the sheave.

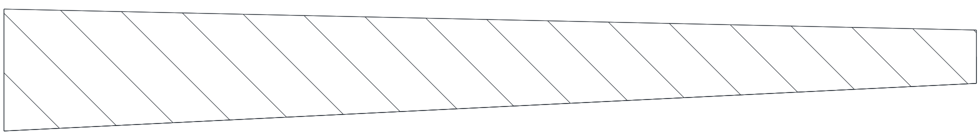


Figure 105: Web with decreasing thickness

Figure 105 above shows the web of the sheave. It has a larger thickness at the left hand side and decreases towards the edge.

From analyzes of the previous concepts it is shown that the side load in the groove gives large bending stresses towards the center of the sheave. And from concept 2, a straight web, with a web thickness of 25mm the sheave get a pretty high equivalent stress at the end of the web. With these observations in mind a web with the values of 26mm at the inner edge of and 12mm at the outer edge is designed.

By changing the cross section to a sheave like this, instead of having a rectangular cross section with the maximum thickness needed across its entire length, a lot of mass is removed. A rectangular cross section weighs approximately:

$$\rho * A = 7850 \frac{kg}{m^3} * 26mm * 500mm \approx 102 \frac{kg}{m}$$

while a cross section like the figure above have a mass of:

$$7850 \frac{kg}{m^3} * (12mm * 500mm + \frac{2}{2}(7mm * 500mm) \approx 74{,}5 \frac{kg}{m}$$

So the webs cross section is approximately 25 $\frac{kg}{m}$ lighter with a decreasing web thickness comapred to a constant web thickness.

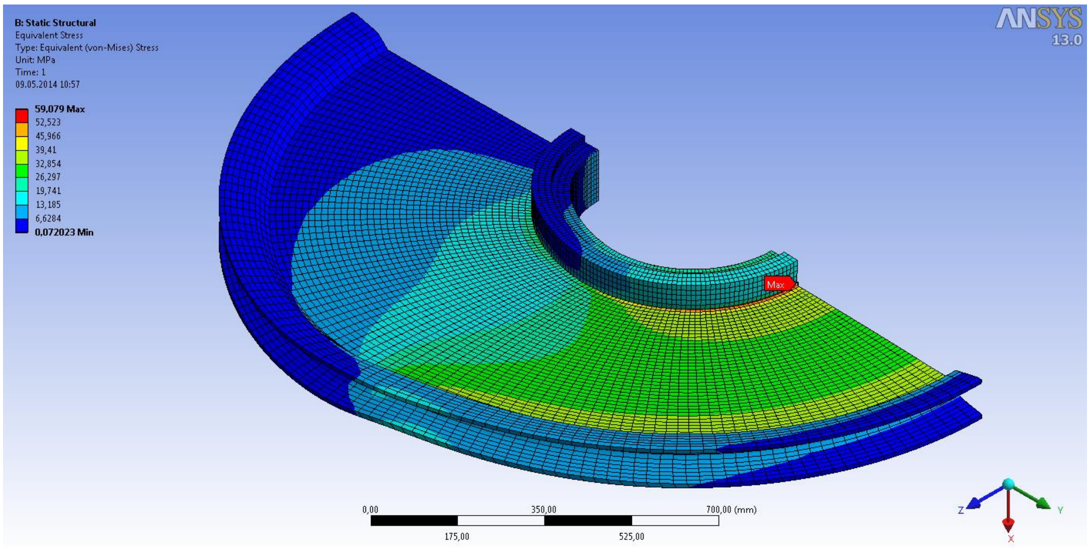


Figure 106: Normal stress plot for decreasing thickness of web sheave

Figure 106 shows the normal stress plot from ANSYS when only axial force is applied. From hand calculations the stress at the inner edge is:

$$\sigma_N = \frac{2*90000kg*9{,}81\frac{m}{s^2}}{600mm*26mm*2} = 56{,}6\ MPa.$$

This corresponds well with the results from ANSYS that have a maximum value of 59,08 MPa.

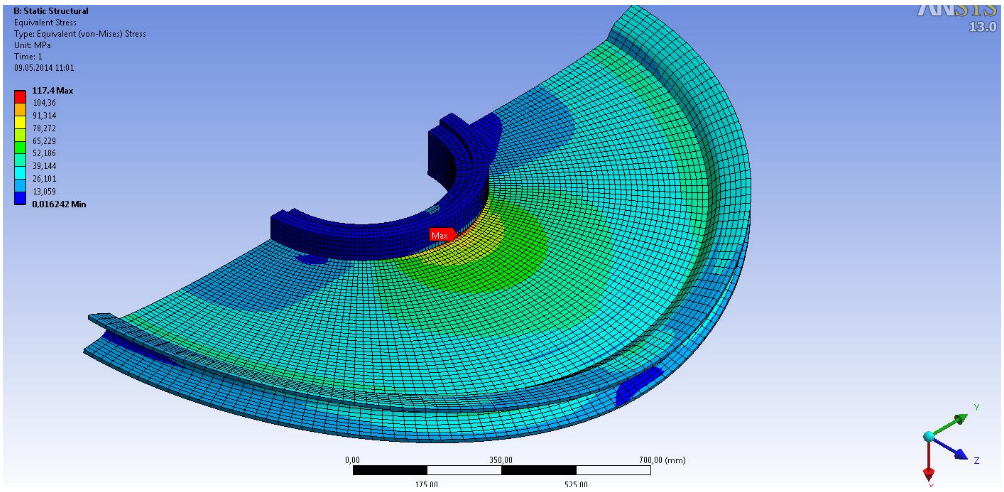


Figure 107: Bending stress plot for decreasing thickness of web sheave

Figure 107 shows the equivalent bending stress when only the line pressure is applied. The bending stresses has its maximum at the inner edge of the web with a value of 117,4 MPa.

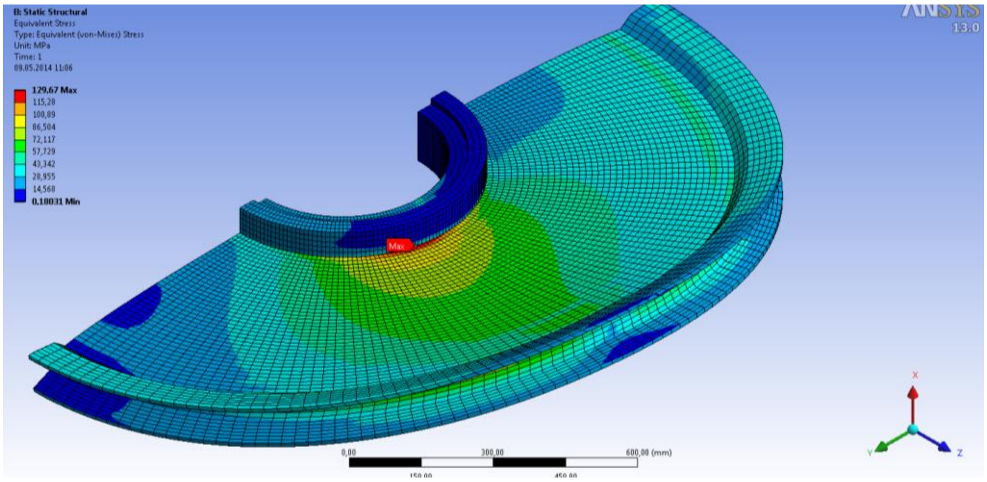


Figure 108: Combined equivalent stress plot for decreasing thickness of web sheave

The equivalent stress plot for both forces acting on the web with decreasing thickness is shown in Figure 108. The maximum stress occurs at the intersection between the web and the support. The maximum value is 129,7 MPa.

As the equivalent stress plot from ANSYS illustrates, the sheave have good utilization over the entire length of the sheave.

6.7.1 BUCKLING

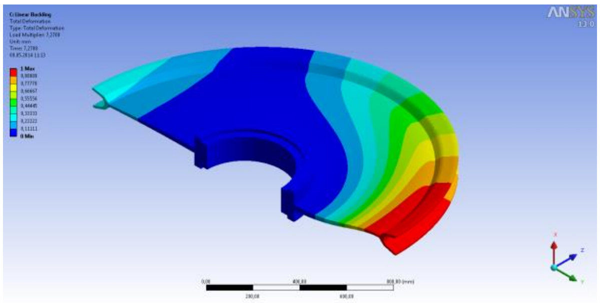


Figure 109: Buckling mode 1

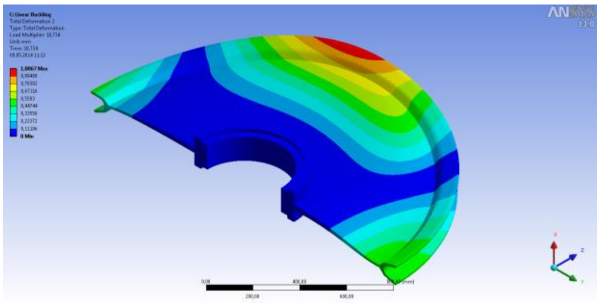


Figure 110: Buckling mode 2

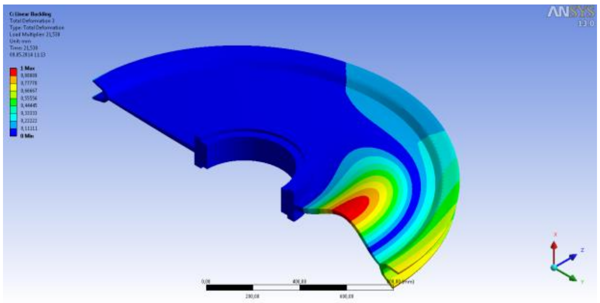


Figure 111: Buckling mode 3

The three buckling modes for this concept is shown in the figures above, the load multiplier to the respective mode is shown in Table 19 below;

Table 19: Buckling modes for decreasing thickness of web sheave

Buckling mode	Load multiplier
1	7,3
2	18,7
3	21,5

From graph in appendix the moment capacity at this point is:

$$M_{Rd} = 13{,}5\ kNm$$

Checking for second order effect of buckling:

$$\frac{F}{F_E} + \frac{M}{M{,}Rd*\left(1-\frac{F}{F_E}\right)} = \frac{F}{F*\alpha} + \frac{8{,}35*10^6}{13{,}5*10^6*\left(1-\frac{F}{F*\alpha}\right)} = \frac{1}{7{,}64} + 0{,}62*\frac{1}{\left(1-\frac{1}{7{,}64}\right)} = 0{,}84 \leq 1 \rightarrow ok!$$

This concept is considered acceptable regarding second order effect of buckling.

6.7.2 REACTION FORCE

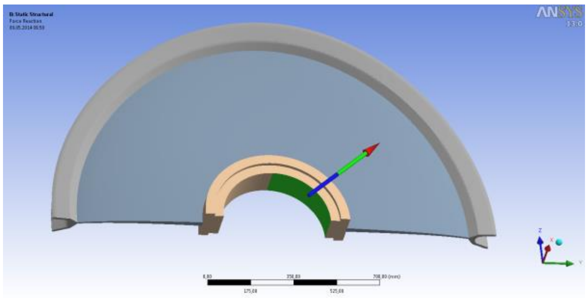


Figure 112: Total reaction force on cylindrical support

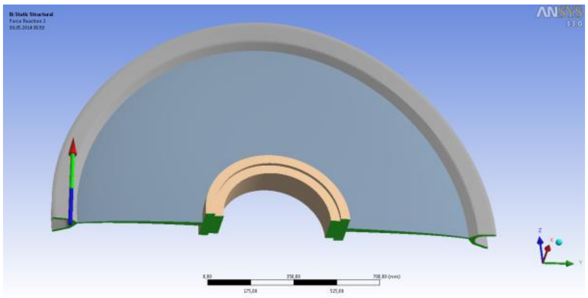


Figure 113: Total reaction force on frictionless support

Figures above shows the total cylindrical support reaction force on the left hand side, and the frictionless support reaction force on the right hand side for the concept with decreasing thickness of web.

Table 20: Reaction forces for decreasing thickness of web sheave

Support	Direction		
	X [kN]	Y [kN]	Z [kN]
Cylindrical	-14,1	449,2	377,5
Frictionless	0,17	6,62	48
Total	-14	455,82	425,5

These values are approximetly the same as the values in the previous concept, so the numerical aspect of the calculations from ANSYS are consdierd to be correct.

6.7.3 FATIGUE

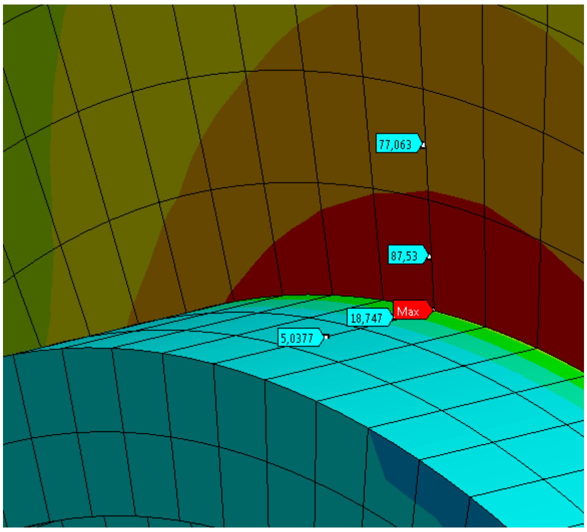


Figure 114: Maximum principal stress

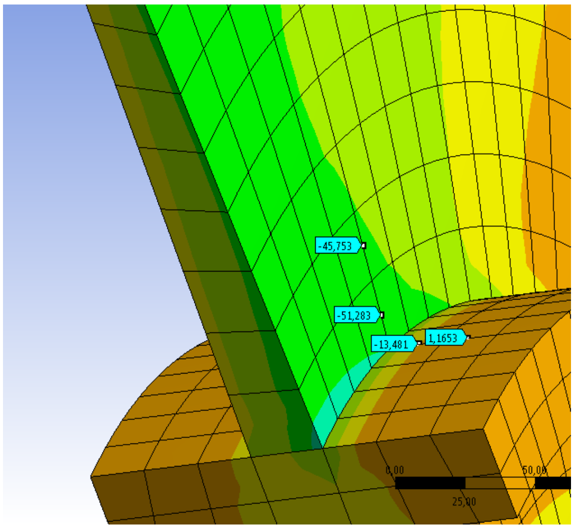


Figure 115: Minimum principal stress

Figures above show the maximum and minimum principal stress plot 0,5t and 1,5t away from the hot spot region. The maximum hot spot stress from web to support becomes:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 87{,}53\ MPa - \frac{(87{,}53 - 77{,}06)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = 92{,}77\ MPa$$

From support to web:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = 18{,}75\ MPa - \frac{(18{,}75 - 5{,}04)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = 25{,}61\ MPa$$

Minimum hot spot stress from web to support is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -51{,}3\ MPa - \frac{(-51{,}3 - (-45{,}75))MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -54{,}1\ MPa$$

And from support to web, the minimum hot spot stress is:

$$b = y_1 - \frac{y_1 - y_2}{x_1 - x_2} * x_1 = -13{,}5\ MPa - \frac{(-13{,}5 - 1{,}2)MPa}{0{,}5t - 1{,}5t} * 0{,}5t = -20{,}85\ MPa$$

Stress ranges are shown in Table 21 below

Table 21: Stress range for decreasing thickness of web sheave

Direction	$\Delta\sigma$ [MPa]
Web to support	146,9
Support to web	46,46

The equation below is used to check whether or not the concept is acceptable regarding fatigue:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq \Delta\sigma \rightarrow \frac{\Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{146{,}9}{\left(\frac{28{,}9}{0{,}38}\right)} = 1{,}97 < 1 \rightarrow not\ ok!$$

With the maximum stress range in this concept the fatigue utilization is 197 %. By using the same conditions as previous concept, with a reduction factor of 0,6, the equation below to check whether or not the concept is acceptable regarding fatigue becomes:

$$\frac{\Delta\sigma_{cap}}{\mathrm{k}} \geq f_m * \Delta\sigma \rightarrow \frac{\mathrm{f_m} * \Delta\sigma}{\left(\frac{\Delta\sigma_{cap}}{k}\right)} \leq 1 \rightarrow \frac{147 * 0{,}6}{\left(\frac{46}{0{,}38}\right)} = 0{,}73 < 1 \rightarrow\ ok!$$

With the reduction factor and using category C, the fatigue utilization of this concept 73 %.

7.0 CONCLUSION AND DICUSSION

From the different concepts created it is seen that the sheaves with slanted double webs have the lowest equivalent stress. This is because of the slanted webs, they will have good capacity of side loads and bending stress. These design concepts have a large weight compared to the other concepts because each web has to be thick enough to withstand buckling.

The straight web concept is also relatively heavy and the outer edge of the plate gets a low utilization. Since the web has a constant thickness, determined from the highest stress at the inner edge of web, the outer edge of web will get a low utilization.
By adding holes on the straight web concept the utilization of the sheave becomes better, but the holes create large axial stresses around the holes and the buckling capacity goes down. The holes are added as close as possible to the outer edge since the stresses have their lowest value here.

The concept with a thin web and six stiffeners placed around the sheave has a small weight. As seen from the stress plot for this concept, it is the stiffeners that get the highest stresses. The side load is placed right above on of the stiffeners, and the bending stress on the thin web is almost zero. As the sheave rotates, the side load will act on all places on the sheave, and give relatively large bending stresses between the stiffeners.
Adding holes in this concept brings the weight down approximately 15 kg, but this gives large axial stresses around the holes, and the bending stresses will also act around the holes when the sheave is rotating. Because of this, these concepts are not very good.

The last design concept was having a web with decreasingly thickness toward the outer edge of the sheave. By decreasing the web thickness towards the inner edge, the stress utilization becomes good on the entire sheave and the weight goes down a lot. Beside from the sheave with thin web with stiffeners and holes, this is the lightest sheave of the different concepts, with a weight of approximately 520 kg.

In this thesis seven different sheave concepts were analyzed. All concepts are checked for stress, buckling and fatigue. The objective of the thesis was to optimize the sheave by bringing the weight down.

Table 22: Summary of comparison of each sheave design concept

Design concept	Stress capacity	Buckling capacity	Fatigue capacity	Weight [kg]	Moment of inertia [kg*mm²]
1. Double web	52 %	35 %	102 %	647	$2{,}46*10^8$
2. Straight web	72 %	63 %	70 %	622	$2{,}38*10^8$
3. Straight web w/holes	82 %	66 %	67 %	531	$2{,}02*10^8$
4. Double web w/holes	44 %	26 %	84 %	620	$2{,}32*10^8$
5. Thin web w/ stiffeners	70 %	73 %	72 %	531	$1{,}97*10^8$
6. Thin web w/ stiffeners and holes	96 %	71 %	72 %	516	$1{,}92*10^8$
7. Web with decreasing thickness	82 %	84 %	73 %	520	$1{,}90*10^8$

Table 22 shows the different concepts with their capacities on stress, buckling and fatigue. The current sheave that Cameron Sense uses, double web, has very good capacity in each category, but the weight is high compared to the others. The moment of inertia is also quite large. As seen from the Table 22 the concept with a thin web and stiffeners and holes have the lowest weight. But the stress utilization is very high, and as discussed above the side load will act in different positions as the sheave rotates.

Therefore the concept with a decreasing thickness on the web is the best concept. This has a good utilization in all the design criteria and the weight is low. It also has the lowest moment of inertia of all the sheaves.

With the load spectrum supplied by Cameron the sheaves had a very high utilization regarding fatigue. With this load spectrum the sheaves have to withstand drilling 100 deep wells over a period of 20 year. The spectrum is also considering that the sheave is fully loaded when it goes up and down. On the way up the sheave is not fully loaded, therefore the spectrum can be halved to 100 million fatigue cycles. This gives an increased allowable stress.

The straight web sheaves and the thin web sheaves can be casted (concept 2,3,5,6 and 7). Casting the design concepts will result in a sheave without any welds. Therefore the sheave can be categorized as class C from DNV-RP-C203. As the sheaves is in compression loading only the reduction factor of 0,6 is also used. Using the C-curve and the reduction factor when fatigue checking the single web design concepts with respect to fatigue results in an acceptable result.

Calculating with the side load acting is very conservative. The sheaves should in principal not have any side load acting on it. But failure of a sheave, because of side load, is very dangerous. Therefore some side load is applied in the design concept calculations.

Optimizing sheaves and other equipment is essential, since the world is facing difficulty with cost of new constructions. The light sheave that Cameron Sense is using weighs 647 kg, but most of the sheaves being used weigh about 800 kg. Using the sheave with a decreasing thickness on the web we have removed 126,4 kg from the lightest sheave, and approximately 280 kg from the regular sheaves being used. In a 14 parts system this will reduce the total weight of sheaves by 3,92 tons and 4,48 tons in a 16 parts system.

Recommendation for further work is to do more investigations on the design concepts with stiffeners. Calculations with the line pressure on different locations should be carried out. Tests by heat treating welds to get rid of residual stresses can be carried out to see if the reduction factor can be used on welded sheave concepts as well.

The sheave should also be compared with respect to cost.

8.0 REFERENCES

[1]. Wikipedia (read 20.02.14). Mechanical advantage. Available http://en.wikipedia.org/wiki/Block_and_tackle

[2]. Cameron (read 20.05.2014). Available: http://www.c-a-m.com/forms/AboutUs.aspx

[3]. Cameron (read 20.05.2014). Available: http://www.cameroncareers.no/

[4]. API RP 09B Wire Rope Specification 1999

[5]. Robert D. Cook, David S. Malkus, Michael E. Plesha, Robert J. Witt, Concepts and applications of finite element analysis. Madison: University of Wisconsin, 4th edition.

[6]. Wikipedia (read 03.03.2014). Yield (engineering). Available: http://en.wikipedia.org/wiki/Yield_(engineering)

[7]. Pictutre: Stress/strain curve (Read 28.05.14). Available: http://goo.gl/wqHqPp

[8]. Picture: Intersection of the von Mises yield criterion (Read 28.05.14). Available: http://goo.gl/SVZtCf

[9]. Wikipedia (read 10.03.2014). von Mises yield criterion. Available: http://en.wikipedia.org/wiki/Von_Mises_yield_criterion

[10]. Wikipedia (read 20.03.2014). Buckling. Available: http://en.wikipedia.org/wiki/Buckling

[11]. Per Kr. Larsen, Dimensjonering av stålkonstruksjoner. Trondheim: Tapir Akademisk Forlag. 4th edition, 2004.

[12]. Picture: Simple pendulum (Read 28.05.14). Available: http://goo.gl/FzuKsN

[13]. Wikipedia (read 24.03.2014). Moment of inertia. Available: http://en.wikipedia.org/wiki/Moment_of_inertia

[14]. Wikipedia (read 07.02.2014). American Petroleum Institute. Available: http://en.wikipedia.org/wiki/American_Petroleum_Institute

[15]. Wikipedia (read 07.02.2014). Det Norske Veritas. Available: http://en.wikipedia.org/wiki/Det_Norske_Veritas

[16]. API Spec 08C Drilling Hoisting Equipment 2003

[17]. Det Norske Veritas. DNV-RP-C203, april 2010.

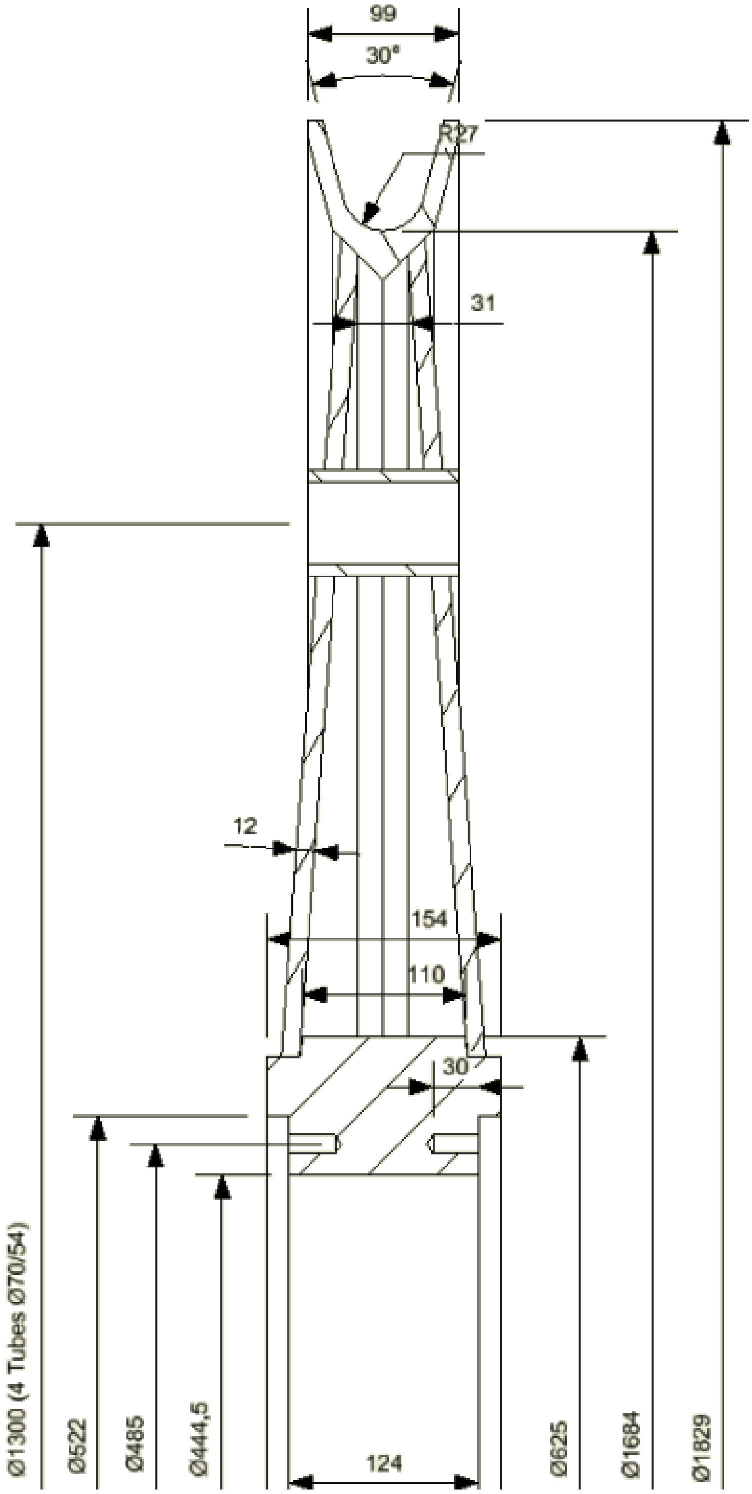
[18]. Norsk standard. NS-3472, Prosjektering av stålkonstruksjoner – Beregning og dimensjonering. 2. opplag, april 1985

[19]. Wikiversity (read 30.05.14). Stress concentration factor around holes. Available: http://en.wikiversity.org/wiki/Introduction_to_Elasticity/Plate_with_hole_in_tension

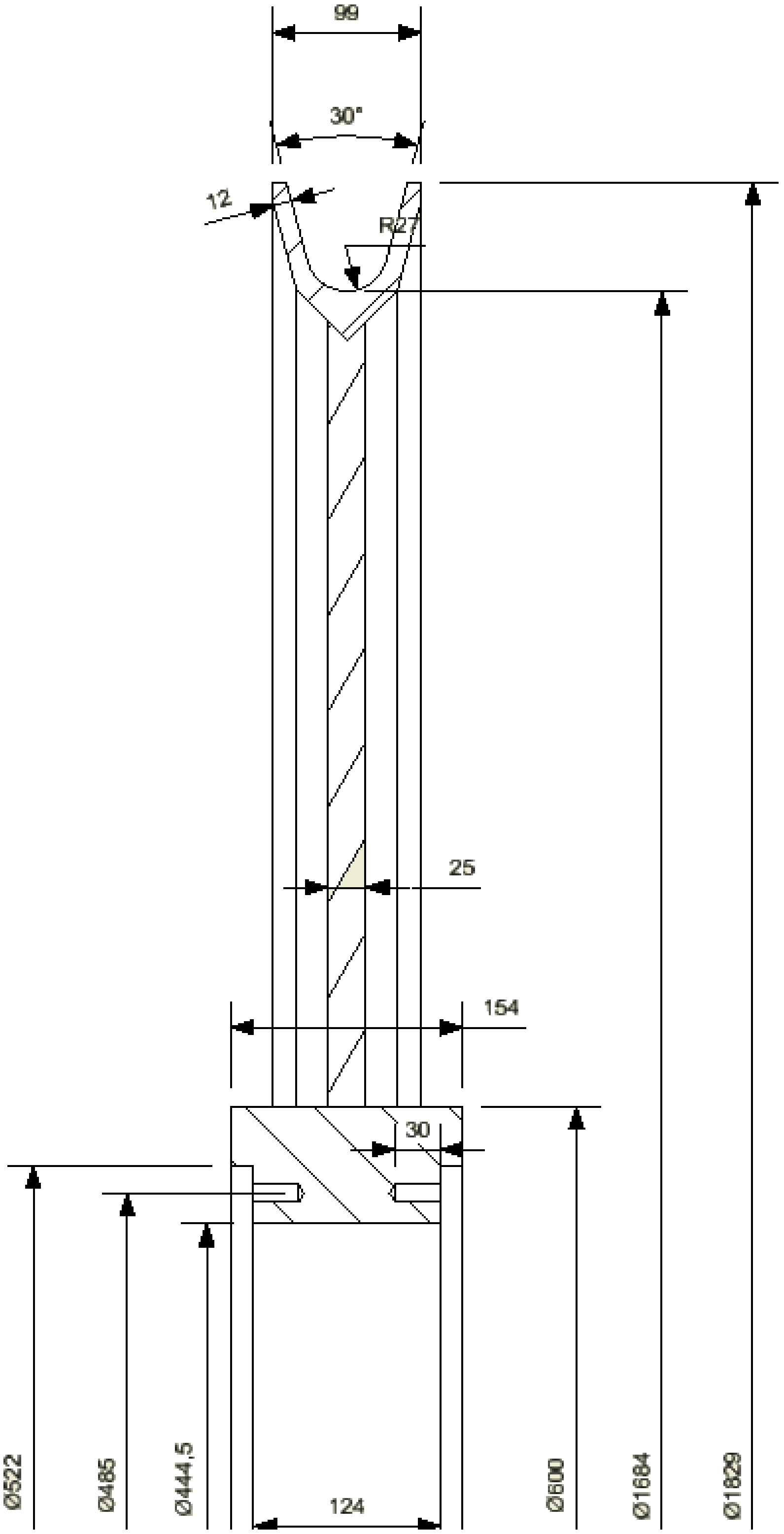
APPENDIX

APPENDIX A: FIGURES

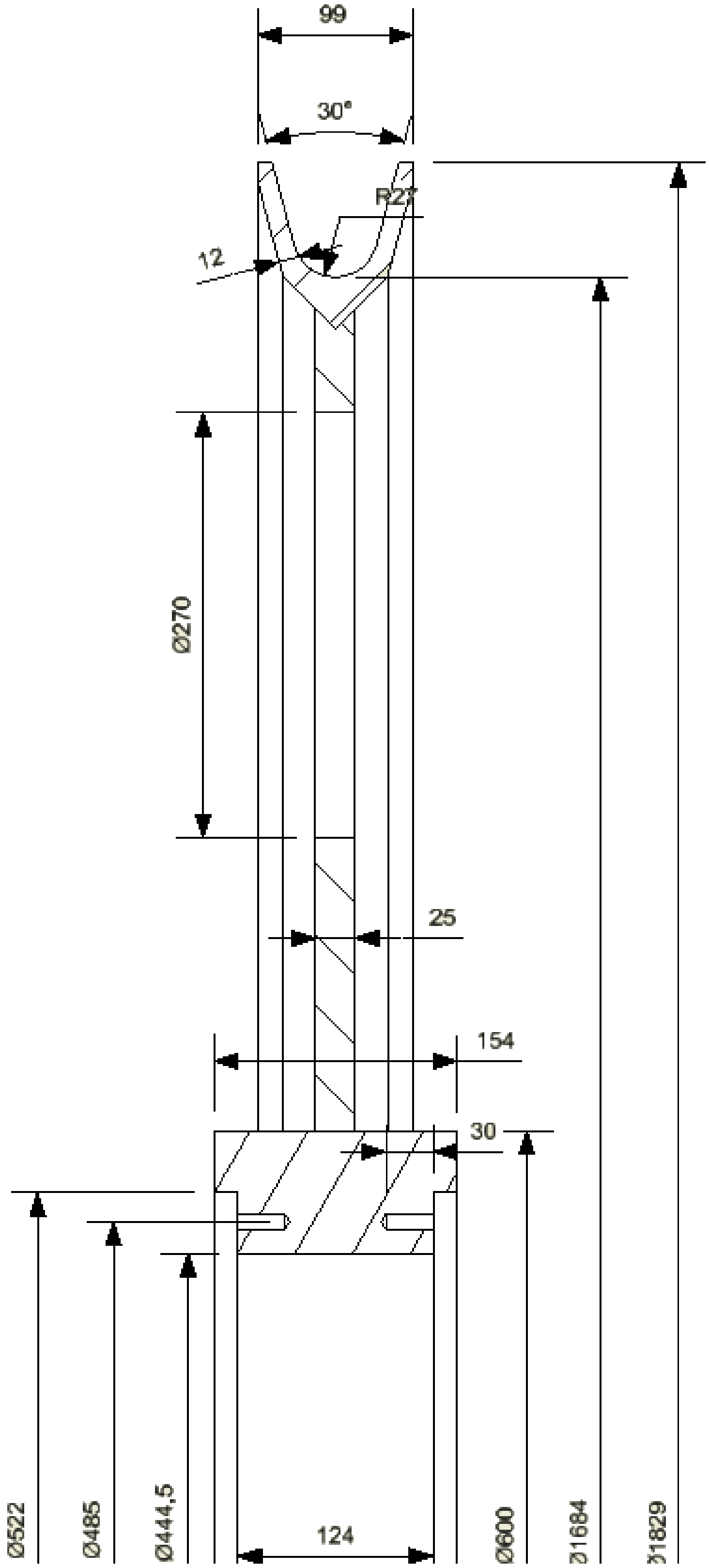
A.1 DOUBLE WEB



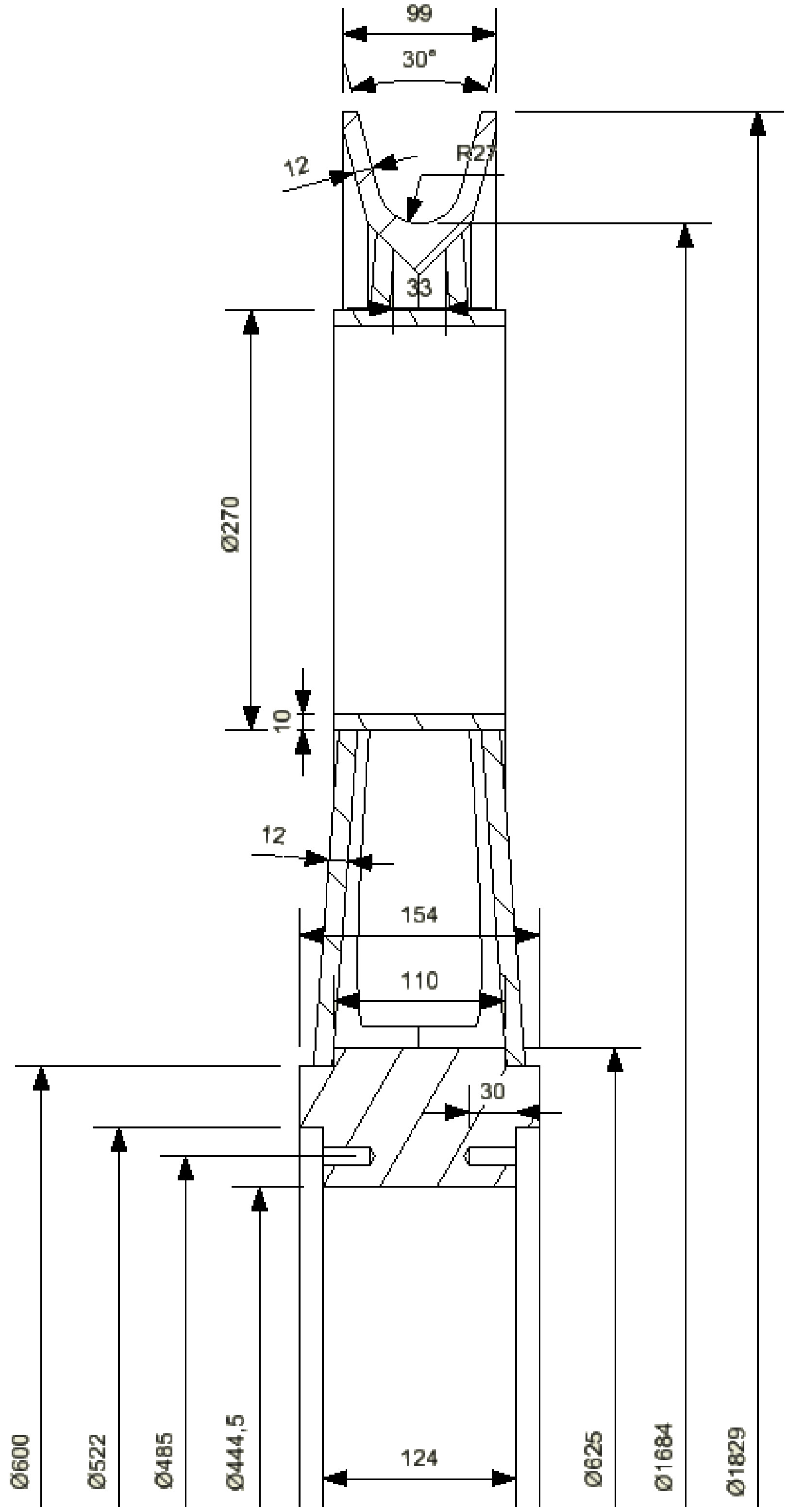
A.2 STRAIGHT WEB



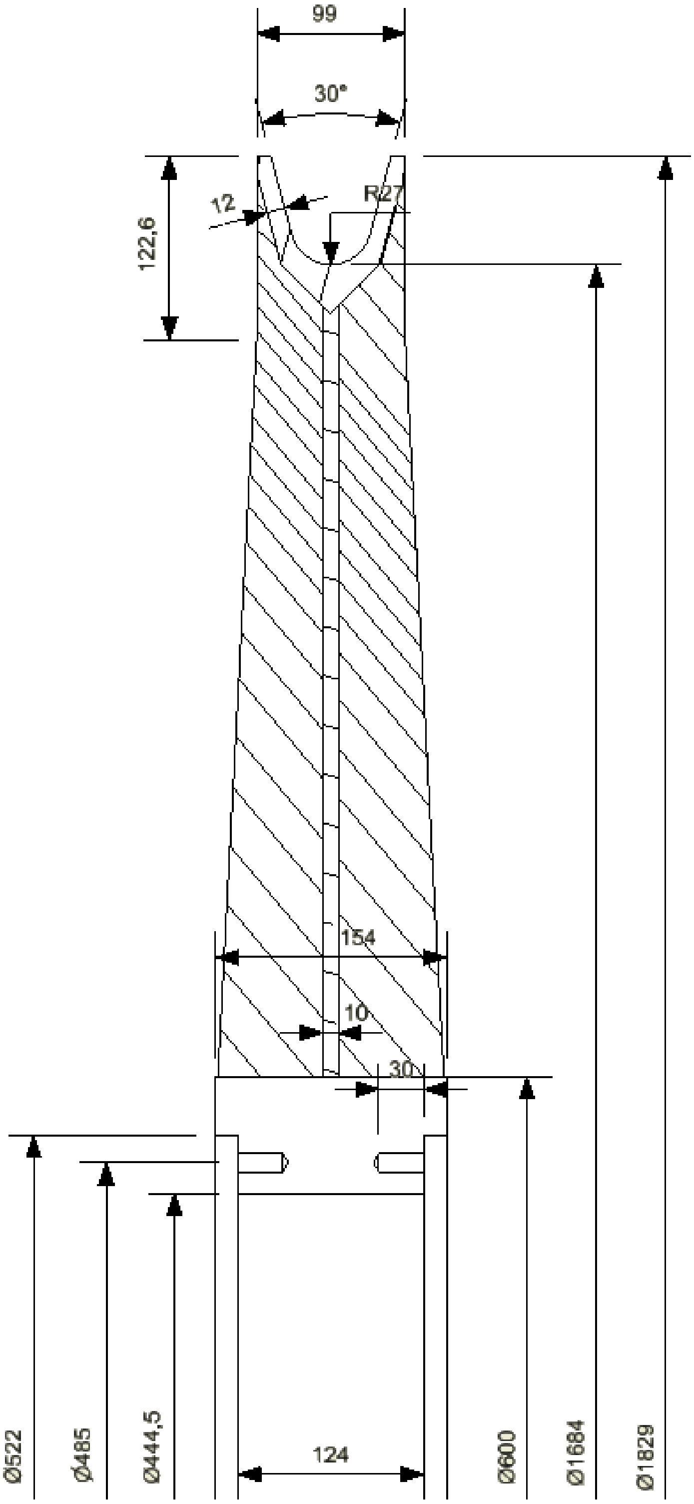
A.3 STRAIGHT WEB WITH HOLES



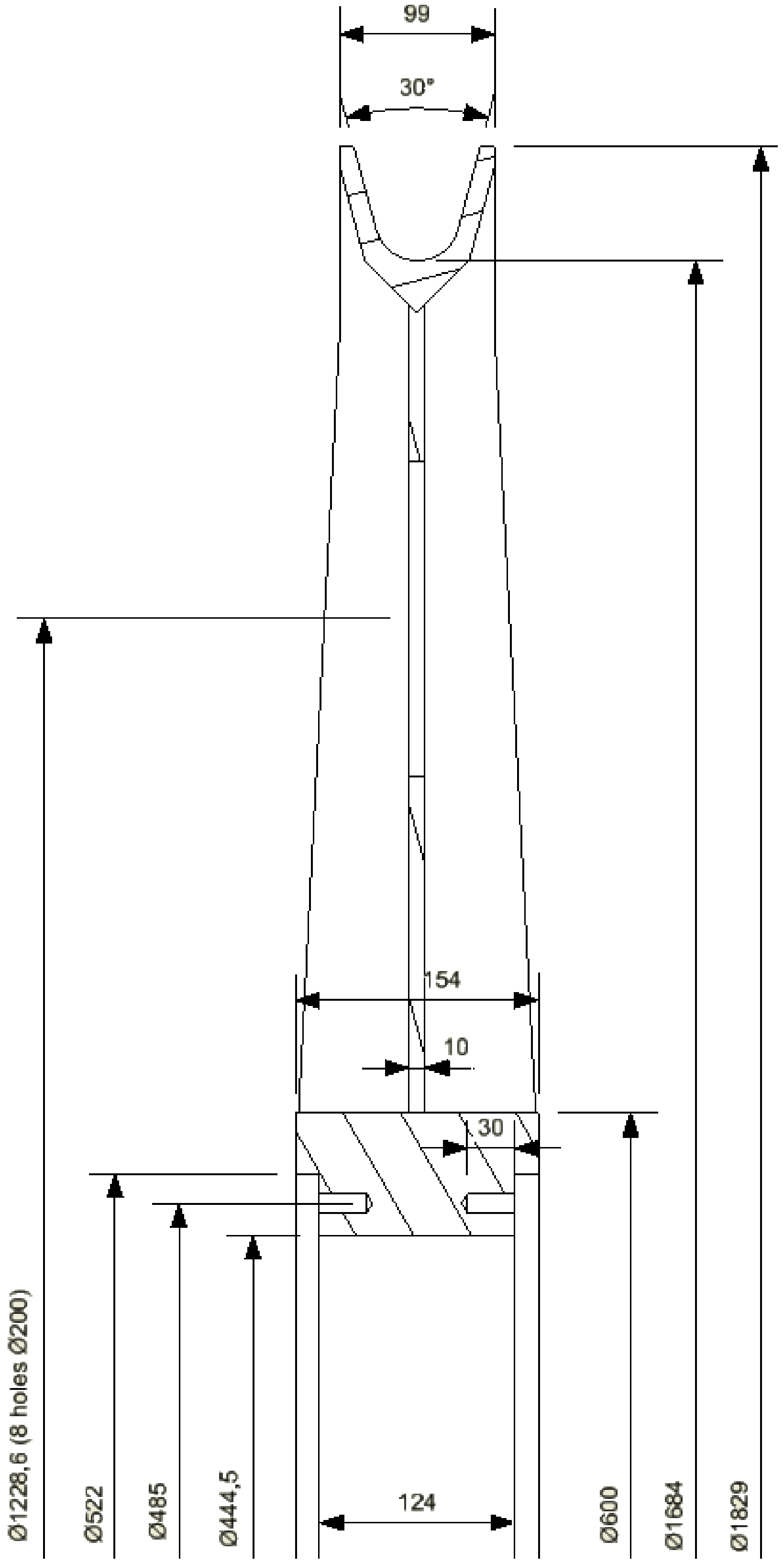
A.4 DOUBLE WEB WITH HOLES



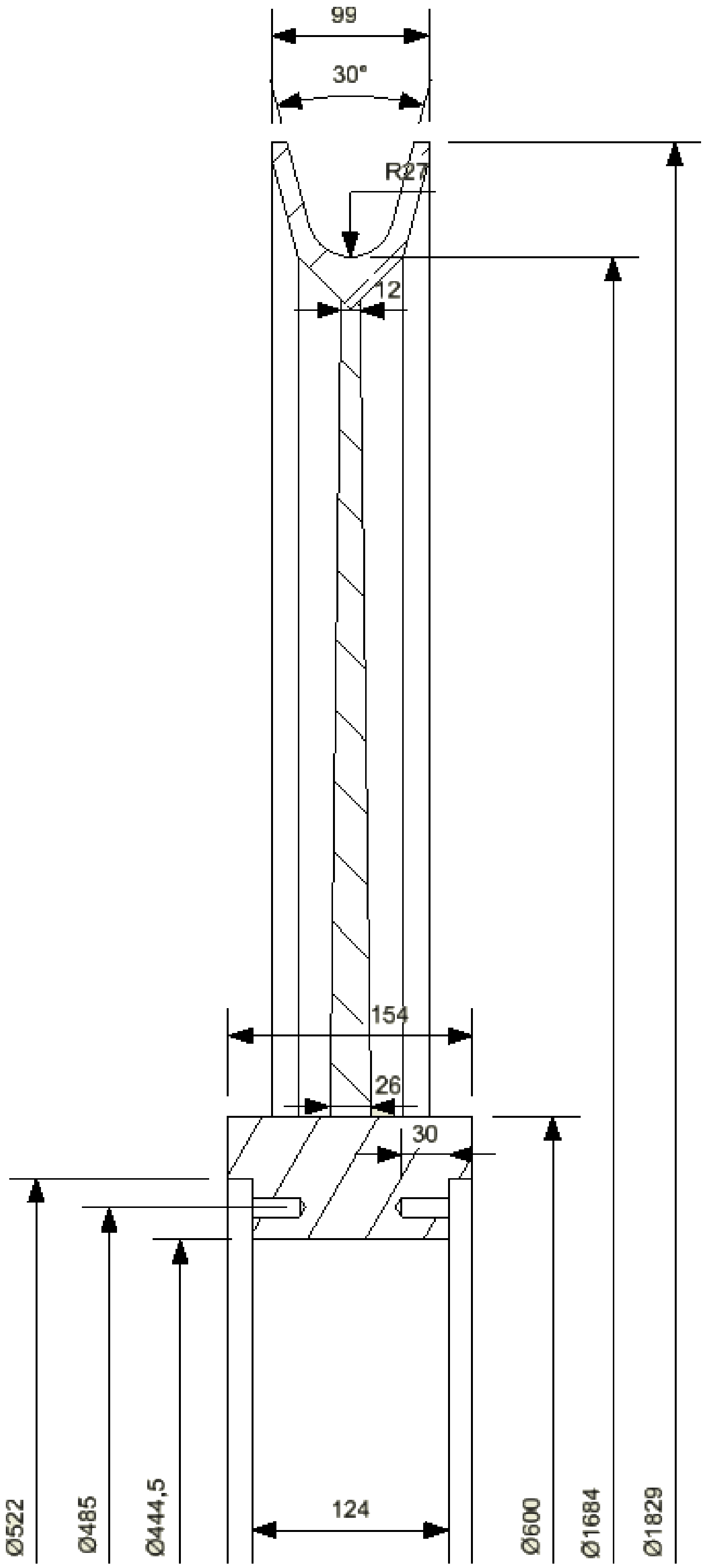
A.5 THIN WEB WITH STIFFENERS



A.6 THIN WEB WITH STIFFENERS AND HOLES

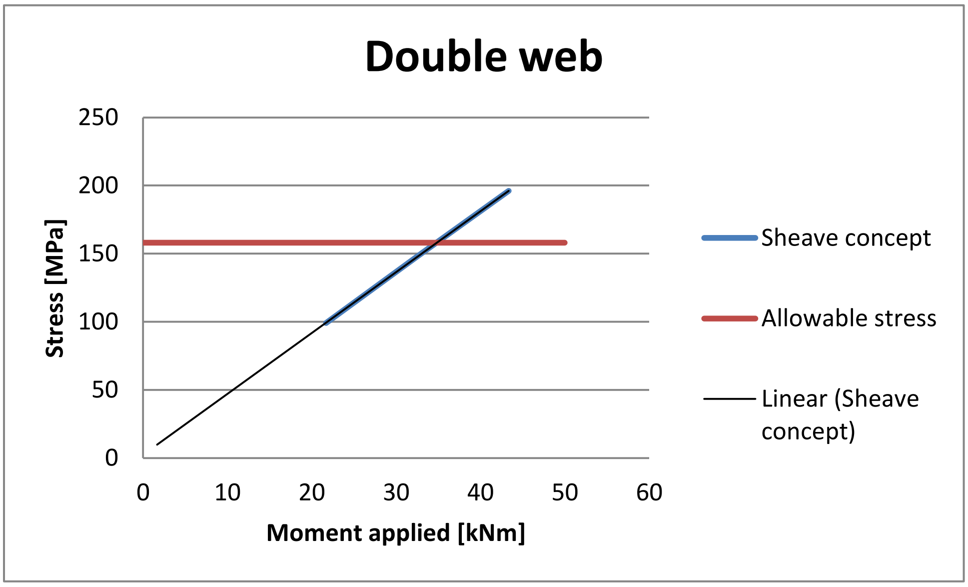


A.7 CHOSEN CONCEPT



APPENDIX B: MOMENT CAPACITY

B.1 DOUBLE WEB



Allowable stress is:

$$F_a = \frac{F_y}{SF_D} = \frac{355}{2,25} = 158\ MPa$$

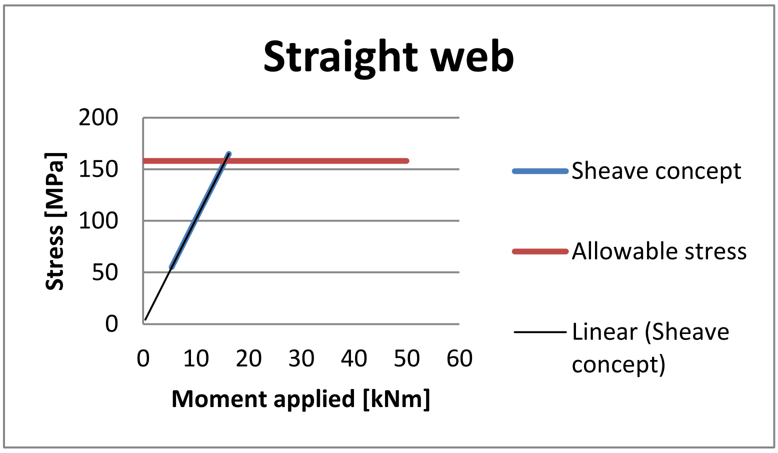
The linear extrapolated line is expressed in the form of:

$$y = 4,5x + 2,16$$

Intersection point with the yield stress is when $y = 158\ MPa$, this gives a moment capacity:

$$y = 4,5x + 2,16 \rightarrow x = \frac{y - 2,16}{4,5} = \frac{158 - 2,16}{4,5} = 34,63 = M_{Rd}$$

B.2 STRAIGHT WEB



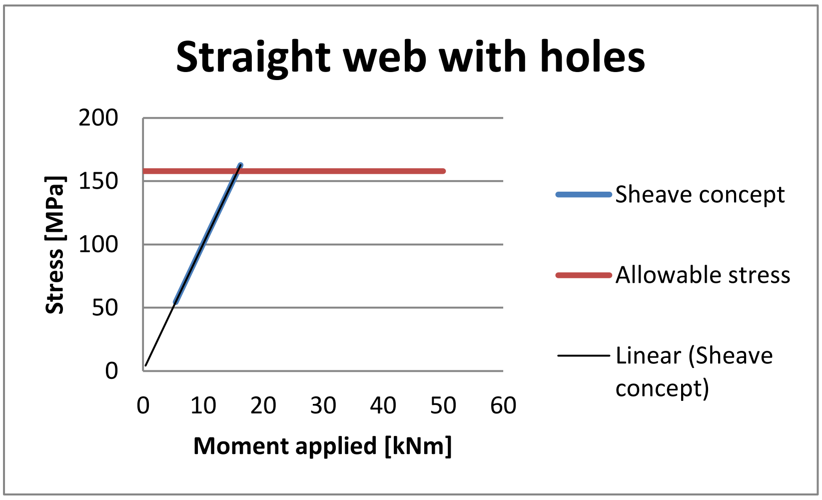
The linear extrapolated line is expressed in the form of:

$$y = 10{,}15x - 0{,}02$$

Intersection point with the yield stress is when $y = 158\ MPa$, this gives a moment capacity:

$$y = 10{,}15x - 0{,}02 \rightarrow x = \frac{y+0{,}02}{10{,}15} = \frac{158+0{,}02}{10{,}15} = 15{,}56 = M_{Rd}$$

B.3 STRAIGHT WEB WITH HOLES



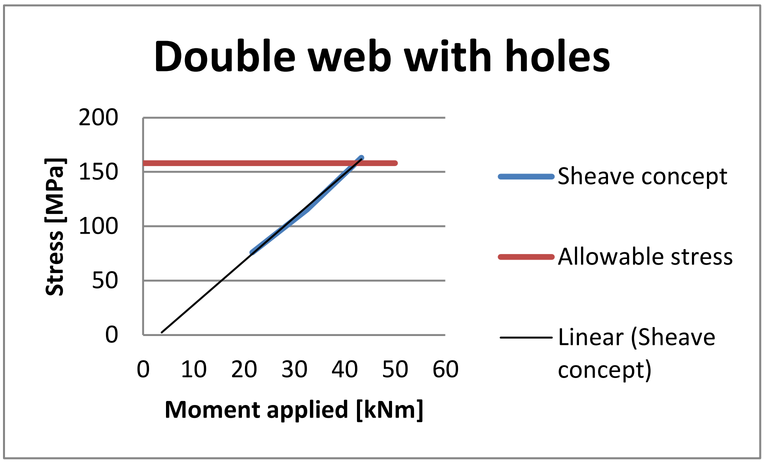
The linear extrapolated line is expressed in the form of:

$$y = 10{,}01x + 0{,}05$$

Intersection point with the yield stress is when $y = 158\ MPa$, this gives a moment capacity:

$$y = 10{,}01x + 0{,}05 \rightarrow x = \frac{y-0{,}05}{10{,}01} = \frac{158-0{,}05}{10{,}01} = 15{,}8 = M_{Rd}$$

B.4 DOUBLE WEB WITH HOLES



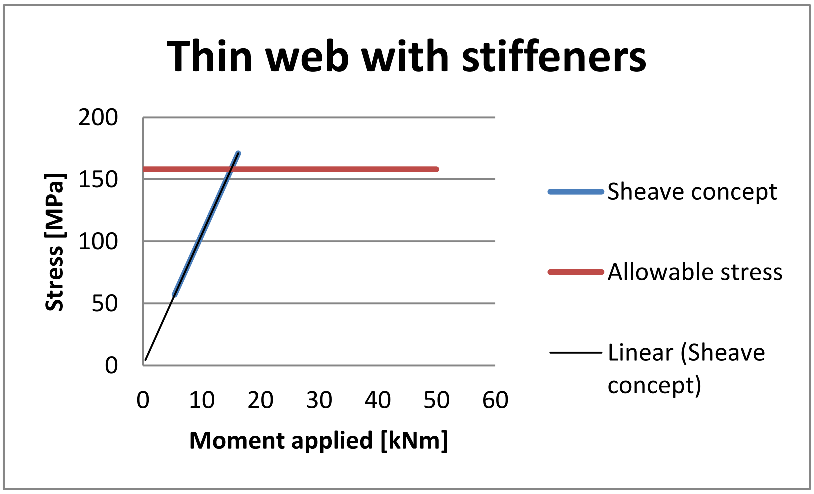
The linear extrapolated line is expressed in the form of:

$$y = 4{,}013x - 12{,}43$$

Intersection point with the yield stress is when $y = 158\ MPa$, this gives a moment capacity:

$$y = 4{,}013x - 12{,}43 \rightarrow x = \frac{y+12{,}43}{4{,}013} = \frac{158+12{,}43}{4{,}013} = 42{,}5 = M_{Rd}$$

B.5 THIN WEB WITH STIFFENERS



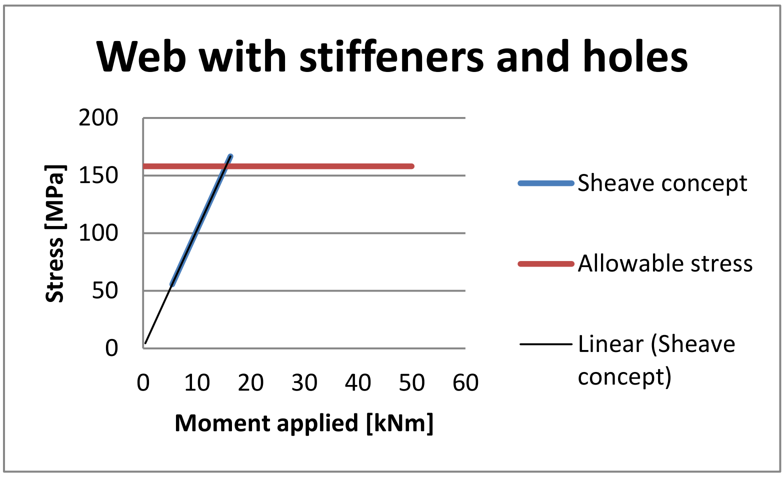
The linear extrapolated line is expressed in the form of:

$$y = 10{,}516x$$

Intersection point with the yield stress is when $y = 158\ MPa$, this gives a moment capacity:

$$y = 10{,}516x \rightarrow x = \frac{y}{10{,}516} = \frac{158}{10{,}516} = 15{,}02 = M_{Rd}$$

B.6 STRAIGHT WEB WITH STIFFENERS AND HOLES



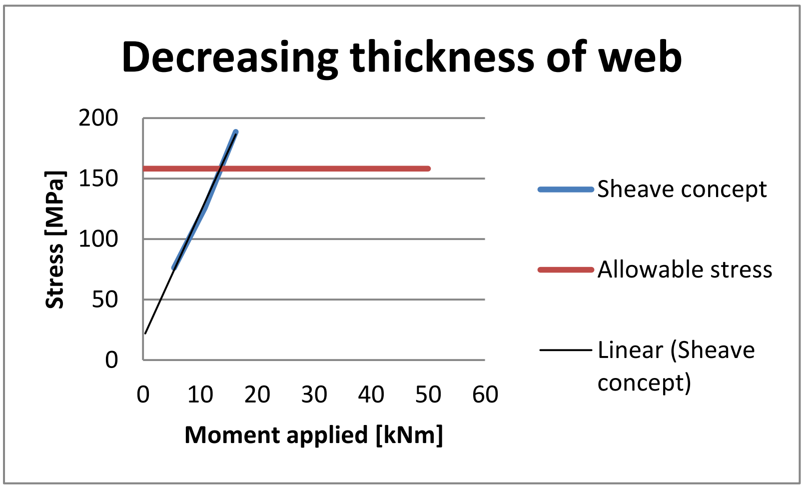
The linear extrapolated line is expressed in the form of:

$$y = 10{,}24x$$

Intersection point with the yield stress is when $y = 158\ MPa$, this gives a moment capacity:

$$y = 10{,}24x \rightarrow x = \frac{y}{10{,}24} = \frac{158}{10{,}24} = 15{,}43 = M_{Rd}$$

B.7 DECREASING THICKNESS ON WEB



The linear extrapolated line is expressed in the form of:

$$y = 10{,}37x + 17{,}7$$

Intersection point with the yield stress is when $y = 158\ MPa$, this gives a moment capacity:

$$y = 10{,}37x \rightarrow x = \frac{y-17{,}7}{10{,}37} = \frac{158-17{,}7}{10{,}37} = 13{,}52 = M_{Rd}$$

APPENDIX C: FATIGUE LOAD SPECTRUM

Below it is shown the drilling load spectrum from Cameron Sense for fatigue calculations. $\Delta\sigma$ is the stress design from ANSYS and η is the utilization.

The spectrum represents a typical deep well for a 1000 shT hoist system.

C.1 SPECTRUM, 1000SHT HOOK LOAD

Total number of wells:		100						
Total number of cycles		1,20E+06						
σ from other loads		0	N/mm^2					
σ from weight of heaviest lift		310	N/mm^2					
Σσ max		310	N/mm^2					
Wohler slope, c:		5,00						
			Each well	All wells		Sec 2.1.4.3		
Block	**Description**	**kN**	**ni**	**ni**	**Σσi**	**(Σσi / Σσ Max)^c*ni/n**	**ni/N**	**kN/Max**
1	Tripping	700	2160	2,16E+05	24	0,0000	17,98 %	0,08
2	Tripping	1600	2880	2,88E+05	56	0,0000	23,98 %	0,18
3	Tripping	2250	2880	2,88E+05	78	0,0002	23,98 %	0,25
4	Tripping	2900	1920	1,92E+05	101	0,0006	15,98 %	0,33
5	Tripping	3800	1680	1,68E+05	132	0,0020	13,99 %	0,43
6	Casing	4500	120	1,20E+04	157	0,0003	1,00 %	0,51
7	BOP	5400	240	2,40E+04	188	0,0016	2,00 %	0,61
8	BOP / Misc.	6750	120	1,20E+04	235	0,0025	1,00 %	0,76
9	Misc.	8900	12	1,20E+03	310	0,0010	0,10 %	1,00
							100,00 %	
		Max		N		Ksp		
		8900,0	12012	1,20E+06		0,0083		
						Equiv. Load	**Max*Equiv. Load**	
						0,38	3417	

Figure above shows the load spectrum supplied from Cameron Sense. This spectrum represents a typical deep well for a 1000 sHT hoist system. This spectrum has a total fatigue cycles of 1.2 million. In the hoist system, the sheaves rotates X-times per hoist system cycle. The sheaves should have a drilling load spectrum corresponding to 200 million fatigue cycles. Multiplying each block with $\frac{200mill}{1{,}2mill} = 166{,}67$ will give a total fatigue cycles of 200 million.

The drilling load spectrum for the sheaves, multiplied with 166,67 is shown in figure below.

Total number of wells:		100						
Total number of cycles		2,00E+08						
σ from other loads		0	N/mm^2					
σ from weight of heaviest lift		310	N/mm^2					
Σσ max		310	N/mm^2					
Wohler slope, c:		5,00	Booklet 9					
			Each well	All wells		Sec 2.1.4.3		
Block	**Description**	**kN**	**ni**	**ni**	**Σσi**	**(Σσi / Σσ Max)^c*ni/n**	**ni/N**	**kN/Max**
1	Tripping	700	360000	3,60E+07	24	0,0000	17,98 %	0,08
2	Tripping	1600	480000	4,80E+07	56	0,0000	23,98 %	0,18
3	Tripping	2250	480000	4,80E+07	78	0,0002	23,98 %	0,25
4	Tripping	2900	320000	3,20E+07	101	0,0006	15,98 %	0,33
5	Tripping	3800	280000	2,80E+07	132	0,0020	13,99 %	0,43
6	Casing	4500	20000	2,00E+06	157	0,0003	1,00 %	0,51
7	BOP	5400	40000	4,00E+06	188	0,0016	2,00 %	0,61
8	BOP / Misc.	6750	20000	2,00E+06	235	0,0025	1,00 %	0,76
9	Misc.	8900	2000	2,00E+05	310	0,0010	0,10 %	1,00
							100,00 %	
		Max		N		Ksp		
		8900,0	2002000	2,00E+08		0,0083		
						Equiv. Load	Capacity	
						0,38	28,91	

With this spectrum, and using the D-curve, the sheave concepts stress ranges are checked versus the fatigue capacity, from DNV-RP-C203, divided on the equivalent load range. The results are shown in figure below:

Concept	Decription	Δσ	Cap/equiv.load	η
1	Double web	88	86	102 %
2	Straight web	140	86	162 %
3	Straight web w/holes	134	86	155 %
4	Double web w/holes	73	86	84 %
5	Thin web w/ stiffeners	145	86	168 %
6	Thin web w/ stiffeners and holes	144	86	167 %
7	Web w/ decreasing thickness	147	86	170 %

As seen the utilization is high. Since the sheaves are not loaded on the way up the fatigue cycles is halved to 100 million cycles. The spectrum with 100 million cycles and its maximum allowed stress is shown in figure below:

Total number of wells:		**100**						
Total number of cycles		**1,00E+08**						
σ from other loads		0	N/mm^2					
σ from weight of heaviest lift		310	N/mm^2					
Σσ max		310	N/mm^2					
Wohler slope, c:		5,00						
			Each well	All wells				
Block	**Description**	**kN**	**ni**	**ni**	**Σσi**	**(Σσi / Σσ Max)^c*ni/n**	**ni/N**	**kN/Max**
1	Tripping	700	180000	1,80E+07	24	0,0000	17,98 %	0,08
2	Tripping	1600	240000	2,40E+07	56	0,0000	23,98 %	0,18
3	Tripping	2250	240000	2,40E+07	78	0,0002	23,98 %	0,25
4	Tripping	2900	160000	1,60E+07	101	0,0006	15,98 %	0,33
5	Tripping	3800	140000	1,40E+07	132	0,0020	13,99 %	0,43
6	Casing	4500	10000	1,00E+06	157	0,0003	1,00 %	0,51
7	BOP	5400	20000	2,00E+06	188	0,0016	2,00 %	0,61
8	BOP / Misc.	6750	10000	1,00E+06	235	0,0025	1,00 %	0,76
9	Misc.	8900	1000	1,00E+05	310	0,0010	0,10 %	1,00
							100,00 %	
		Max		N		Ksp		
		8900,0	1001000	1,00E+08		0,0083		
						Equiv. Load	Allowable stress	
						0,38	33	

The stress range from ANSYS multiplied with the reduction factor in the straight web design concpepts are shown in figure below:

	Reduction factor fm=0,8		
Concept	**Decription**	**Δσ**	**Δσ*fm**
1	Double web	88	
2	Straight web	140	84
3	Straight web w/holes	134	80
4	Double web w/holes	73	
5	Thin web w/ stiffeners	145	87
6	Thin web w/ stiffeners and holes	144	86
7	Web w/ decreasing thickness	147	88

The design concepts with respect on different S-N curves and its utilization is shown in excel sheet below where $\sigma_{allow} = \frac{\sigma_{max}}{k}$:

				Concept utilization						
Curve	log a	σ max	σ allow	1	2	3	4	5	6	7
C	16,32	46	120		93 %	89 %		97 %	96 %	98 %
C1	16,081	41	108		130 %	125 %		135 %	134 %	137 %
C2	15,835	37	96							
D	15,606	33	86	102 %	162 %	155 %	84 %	168 %	167 %	170 %
E	15,35	30	77							
F	15,091	26	68	129 %			107 %			
F1	14,832	23	61	145 %			121 %			
F3	14,576	21	54	164 %			136 %			
G	14,33	18	48							
W1	14,101	17	43							
W2	13,845	15	38							
W3	13,617	13	35							

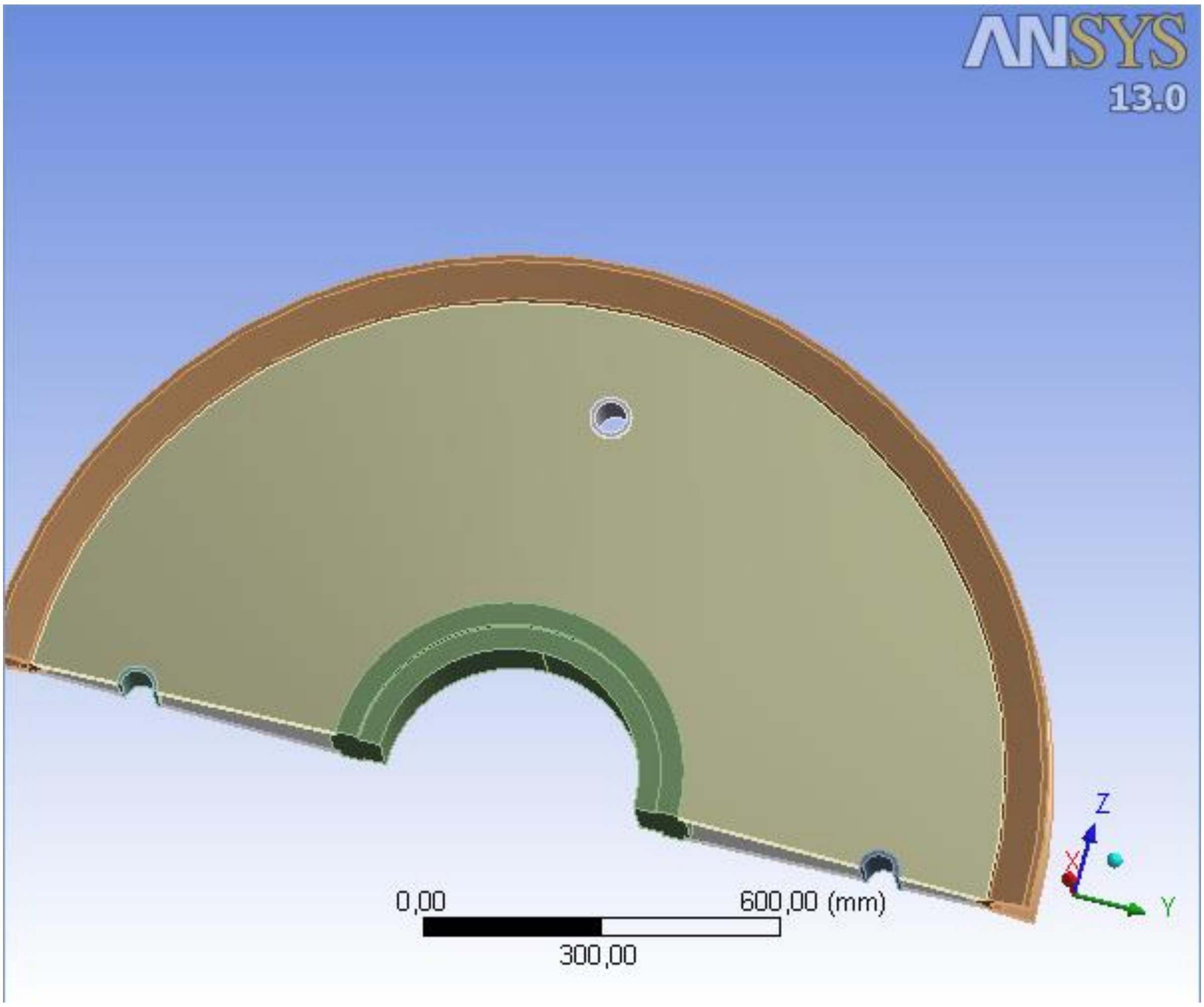
APPENDIX D: ANSYS REPORT

Below the report from ANSYS for all concepts are shown.

D.1 DOUBLE WEB

Project

First Saved	Tuesday, March 11, 2014
Last Saved	Wednesday, May 14, 2014
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (B4, C4)

Geometry

TABLE 2
Model (B4, C4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Users\166864\Documents\masteroppgave\ansys\doubleweb_files\dp0\Geom\DM\Geom.agdb
Type	Inventor
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	154, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	4,1176e+007 mm³
Mass	323,23 kg
Scale Factor Value	1,
Statistics	
Bodies	7
Active Bodies	7
Nodes	237257
Elements	57436
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\166864\AppData\Local\Temp
Analysis Type	3-D

Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3

Model (B4, C4) > Geometry > Body Groups

Object Name	*plate180:1 Bodies*
State	Meshed
Graphics Properties	
Visible	Yes
Definition	
Suppressed	No
Assignment	Structural Steel
Coordinate System	Default Coordinate System
Bounding Box	
Length X	1540, mm
Length Y	18290 mm
Length Z	9145, mm
Properties	
Volume	4,1176e+010 mm³
Mass	323,23 kg
Centroid X	13193 mm
Centroid Y	-29859 mm
Centroid Z	25728 mm
Moment of Inertia Ip1	8,0056e+006 kg·mm²
Moment of Inertia Ip2	1,8984e+006 kg·mm²
Moment of Inertia Ip3	6,2017e+006 kg·mm²
Statistics	
Nodes	237257
Elements	57436
Mesh Metric	None

TABLE 4
Model (B4, C4) > Geometry > plate180:1 Bodies > Parts

Object Name	*pipe:1*	*plate180:1*	*plate180:1*	*pipe180:2*	*support 180:1*
State	Meshed				
Graphics Properties					
Visible	Yes				
Glow	0				
Shininess	1				
Transparency	1				
Specularity	1				
Definition					

Suppressed	No				
ID (Beta)	157	159	161	163	165
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	Structural Steel				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	99, mm	50,728 mm		99, mm	154, mm
Length Y	70, mm	1673, mm		70, mm	625, mm
Length Z	70, mm	836,5 mm		35, mm	312,5 mm
Properties					
Volume	1,5426e+005 mm^3	1,1264e+007 mm^3		77134 mm^3	1,0235e+007 mm^3
Mass	1,2109 kg	88,421 kg		0,60551 kg	80,345 kg
Centroid X	1319,2 mm	1357,7 mm	1280,8 mm	1319,2 mm	1319,7 mm
Centroid Y	-2985,9 mm			-2335,9 mm	-2985,9 mm
Centroid Z	2858,6 mm	2595, mm		2228,4 mm	2380,3 mm
Moment of Inertia Ip1	1172,8 $kg \cdot mm^2$	2,1151e+007 $kg \cdot mm^2$		383,11 $kg \cdot mm^2$	3,5096e+006 $kg \cdot mm^2$
Moment of Inertia Ip2	1572,7 $kg \cdot mm^2$	3,9983e+006 $kg \cdot mm^2$	3,9988e+006 $kg \cdot mm^2$	573,59 $kg \cdot mm^2$	7,04e+005 $kg \cdot mm^2$
Moment of Inertia Ip3	1572,2 $kg \cdot mm^2$	1,7171e+007 $kg \cdot mm^2$	1,7172e+007 $kg \cdot mm^2$	811,32 $kg \cdot mm^2$	3,0622e+006 $kg \cdot mm^2$
Statistics					
Nodes	1233	74406	62655	907	44439
Elements	208	17342	12776	217	12336
Mesh Metric	None				

TABLE 5
Model (B4, C4) > Geometry > plate180:1 Bodies > Parts

Object Name	*groove180:1*	*pipe180:1*
State	Meshed	
Graphics Properties		
Visible	Yes	
Glow	0	
Shininess	1	
Transparency	1	
Specularity	1	
Definition		

Suppressed	No	
ID (Beta)	167	169
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		
Assignment	Structural Steel	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	99, mm	
Length Y	1829, mm	70, mm
Length Z	914,5 mm	35, mm
Properties		
Volume	8,1048e+006 mm³	77133 mm³
Mass	63,622 kg	0,6055 kg
Centroid X	1319,2 mm	
Centroid Y	-2985,9 mm	-3635,9 mm
Centroid Z	2755,4 mm	2228,4 mm
Moment of Inertia Ip1	2,8346e+007 kg·mm²	248,42 kg·mm²
Moment of Inertia Ip2	4,5906e+006 kg·mm²	570,74 kg·mm²
Moment of Inertia Ip3	2,3871e+007 kg·mm²	707,83 kg·mm²
Statistics		
Nodes	55464	860
Elements	14375	182
Mesh Metric	None	

Coordinate Systems

TABLE 6
Model (B4, C4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 7
Model (B4, C4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 8
Model (B4, C4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	5,1267 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 9
Model (B4, C4) > Connections > Contacts > Contact Regions

Object Name	*Contact Region 2*	*Contact Region 4*
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Contact	2 Faces	
Target	2 Faces	
Contact Bodies	plate180:1	
Target Bodies	groove180:1	
Definition		
Type	Bonded	
Scope Mode	Automatic	
Behavior	Symmetric	

Trim Contact (Beta)	Program Controlled
Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

Mesh

TABLE 10
Model (B4, C4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	12,0 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	0,295210 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default

Statistics	
Nodes	237257
Elements	57436
Mesh Metric	None

TABLE 11
Model (B4, C4) > Mesh > Mesh Controls

Object Name	*Hex Dominant Method*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Geometry	7 Bodies
Definition	
Suppressed	No
Method	Hex Dominant
Element Midside Nodes	Use Global Setting
Free Face Mesh Type	Quad/Tri
Control Messages	Yes, Click To Display...

Static Structural (B5)

TABLE 12
Model (B4, C4) > Analysis

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 13
Model (B4, C4) > Static Structural (B5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Restart Analysis	
Restart Type	Program Controlled
Status	Done
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled

Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	Yes
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\doubleweb_files\dp0\SYS\MECH\
Future Analysis	Prestressed analysis
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 14
Model (B4, C4) > Static Structural (B5) > Loads

Object Name	*Cylindrical Support*	*Frictionless Support*	*Line Pressure*	*Pressure*
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face	16 Faces	1 Edge	1 Face
Definition				
ID (Beta)	65	67	139	185
Type	Cylindrical Support	Frictionless Support	Line Pressure	Pressure
Radial	Fixed			
Axial	Fixed			
Tangential	Free			
Suppressed	No			
Define By			Vector	Normal To
Magnitude			154, N/mm (ramped)	20,1 MPa (ramped)
Direction			Defined	

FIGURE 1
Model (B4, C4) > Static Structural (B5) > Line Pressure

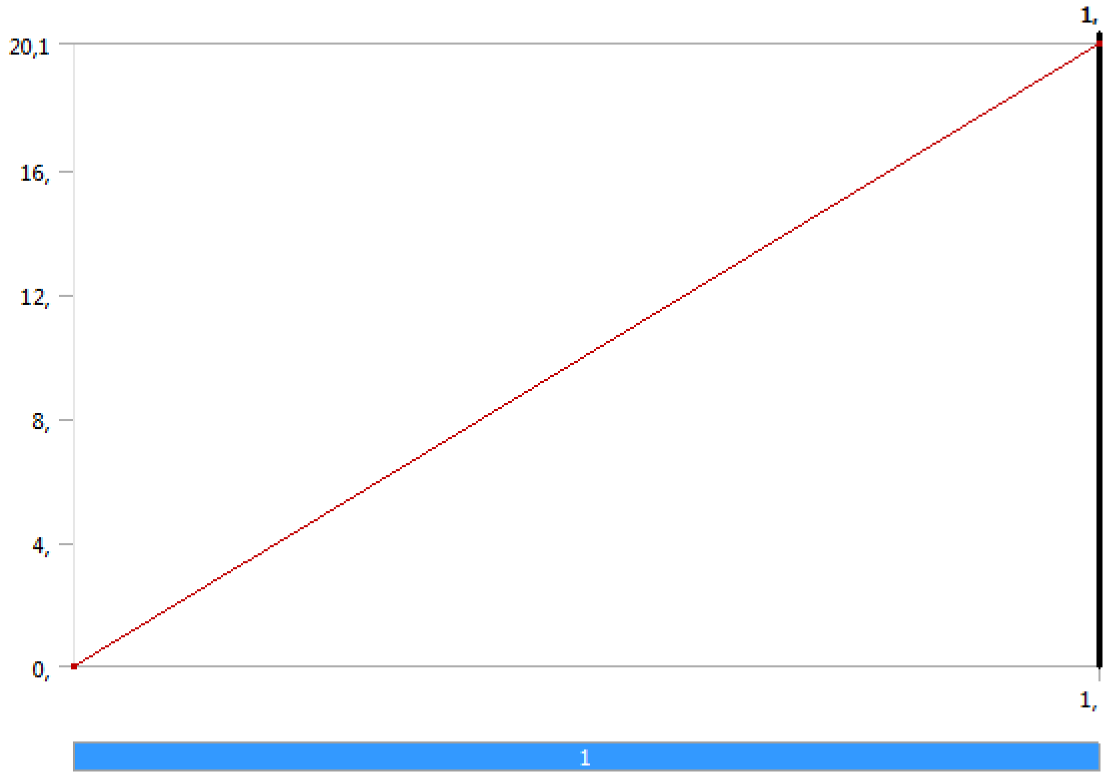
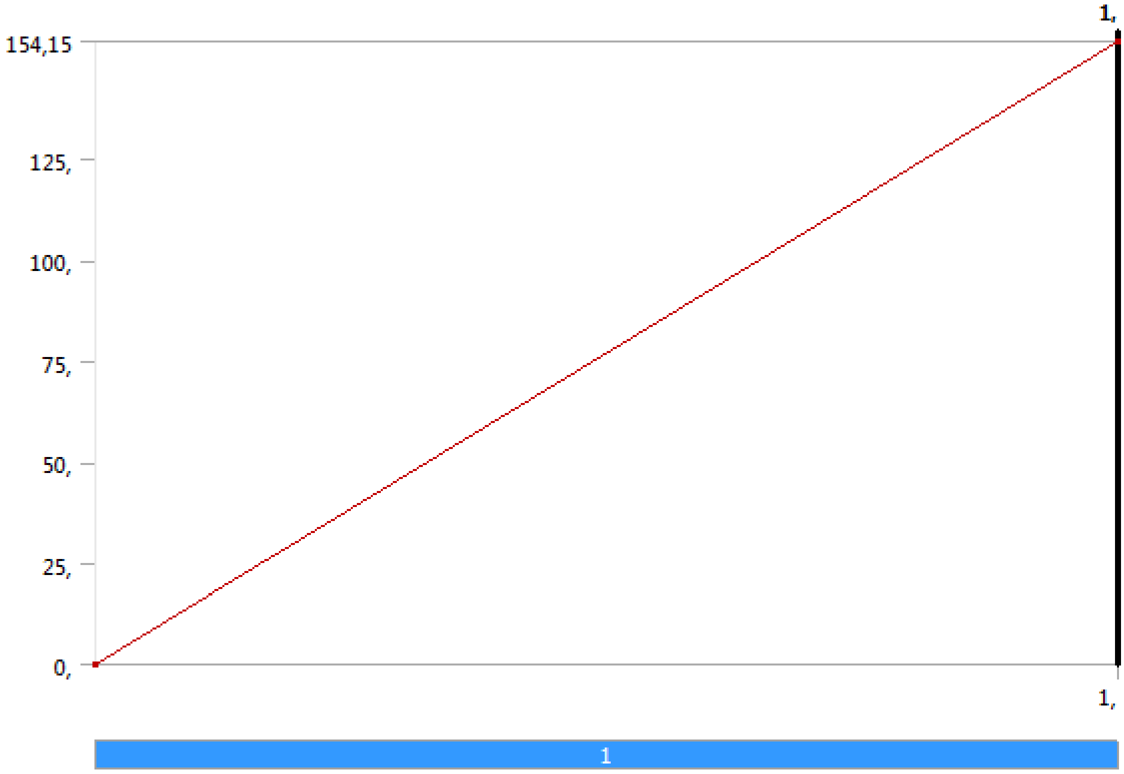


FIGURE 2
Model (B4, C4) > Static Structural (B5) > Pressure



Solution (B6)

TABLE 15
Model (B4, C4) > Static Structural (B5) > Solution

Object Name	*Solution (B6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 16
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 17
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Results

Object Name	*Equivalent Stress*	*Maximum Principal Stress*	*Minimum Principal Stress*	*Maximum Principal Stress 2*
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	All Bodies			
Definition				
Type	Equivalent (von-Mises) Stress	Maximum Principal Stress	Minimum Principal Stress	Maximum Principal Stress
By	Time			
Display Time	Last			
Calculate Time History	Yes			
Identifier				
Integration Point Results				
Display Option	Averaged			Unaveraged
Results				
Minimum	2,2477e-002 MPa	-23,877 MPa	-101,84 MPa	-28,439 MPa
Maximum	85,578 MPa	71,738 MPa	23,245 MPa	102,08 MPa
Minimum Occurs On	groove180:1		plate180:1	groove180:1
Maximum Occurs On	support 180:1	plate180:1		support 180:1
Information				
Time	1, s			
Load Step	1			
Substep	1			
Iteration Number	1			

TABLE 18
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Probes

Object Name	*Force Reaction*	*Force Reaction 2*
State	Solved	
Definition		
Type	Force Reaction	
Location Method	Boundary Condition	
Boundary Condition	Cylindrical Support	Frictionless Support
Orientation	Global Coordinate System	
Options		
Result Selection	All	
Display Time	End Time	
Results		
X Axis	15430 N	-9,9333 N
Y Axis	-4,4229e+005 N	-3860,6 N

Z Axis	3,878e+005 N	51810 N
Total	5,8843e+005 N	51954 N
Maximum Value Over Time		
X Axis	15430 N	-9,9333 N
Y Axis	-4,4229e+005 N	-3860,6 N
Z Axis	3,878e+005 N	51810 N
Total	5,8843e+005 N	51954 N
Minimum Value Over Time		
X Axis	15430 N	-9,9333 N
Y Axis	-4,4229e+005 N	-3860,6 N
Z Axis	3,878e+005 N	51810 N
Total	5,8843e+005 N	51954 N
Information		
Time	1, s	
Load Step	1	
Substep	1	
Iteration Number	1	

Linear Buckling (C5)

TABLE 19
Model (B4, C4) > Analysis

Object Name	*Linear Buckling (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Linear Buckling
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

TABLE 20
Model (B4, C4) > Linear Buckling (C5) > Initial Condition

Object Name	*Pre-Stress (Static Structural)*
State	Fully Defined
Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time
Contact Status	Use True Status

TABLE 21
Model (B4, C4) > Linear Buckling (C5) > Analysis Settings

Object Name	*Analysis Settings*

State	Fully Defined
Options	
Max Modes to Find	3,
Output Controls	
Calculate Stress	No
Calculate Strain	No
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\doubleweb_files\dp0\SYS-1\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	nmm

Solution (C6)

TABLE 22
Model (B4, C4) > Linear Buckling (C5) > Solution

Object Name	*Solution (C6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

FIGURE 3
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

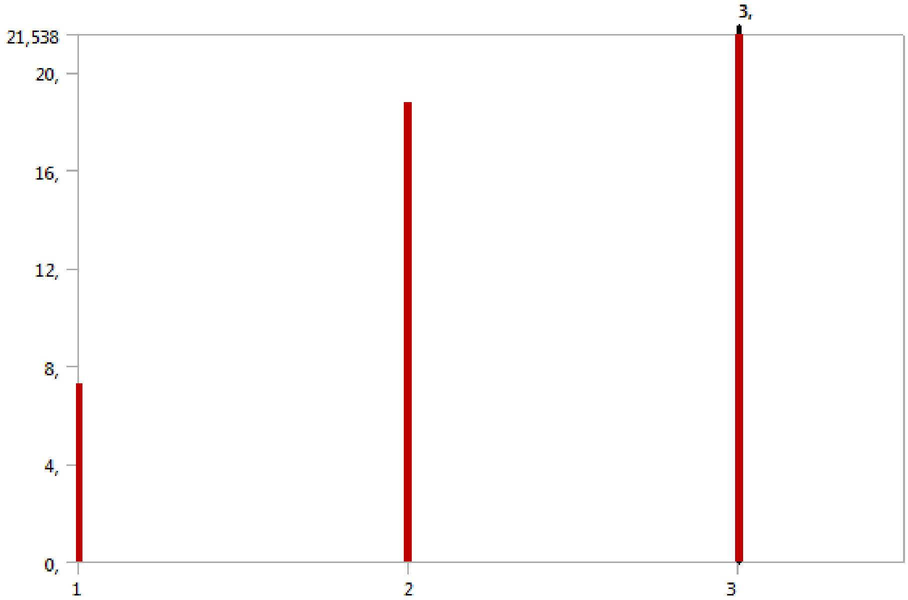


TABLE 23
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

Mode	Load Multiplier
1,	11,692
2,	16,076
3,	18,306

TABLE 24
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 25
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Results

Object Name	*Total Deformation*	*Total Deformation 2*	*Total Deformation 3*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation		
Mode	1,	2,	3,
Identifier			

Results			
Load Multiplier	11,692	16,076	18,306
Minimum	0, mm		
Maximum	1, mm	1,0026 mm	1,0039 mm
Minimum Occurs On	support 180:1		
Maximum Occurs On	pipe180:1	plate180:1	

TABLE 26
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation

Mode	Load Multiplier
1,	11,692
2,	16,076
3,	18,306

TABLE 27
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 2

Mode	Load Multiplier
1,	11,692
2,	16,076
3,	18,306

TABLE 28
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 3

Mode	Load Multiplier
1,	11,692
2,	16,076
3,	18,306

Material Data

Structural Steel

TABLE 29
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 30
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 31
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa

250

TABLE 32
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
250

TABLE 33
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
460

TABLE 34
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 35
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 36
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 37
Structural Steel > Isotropic Elasticity

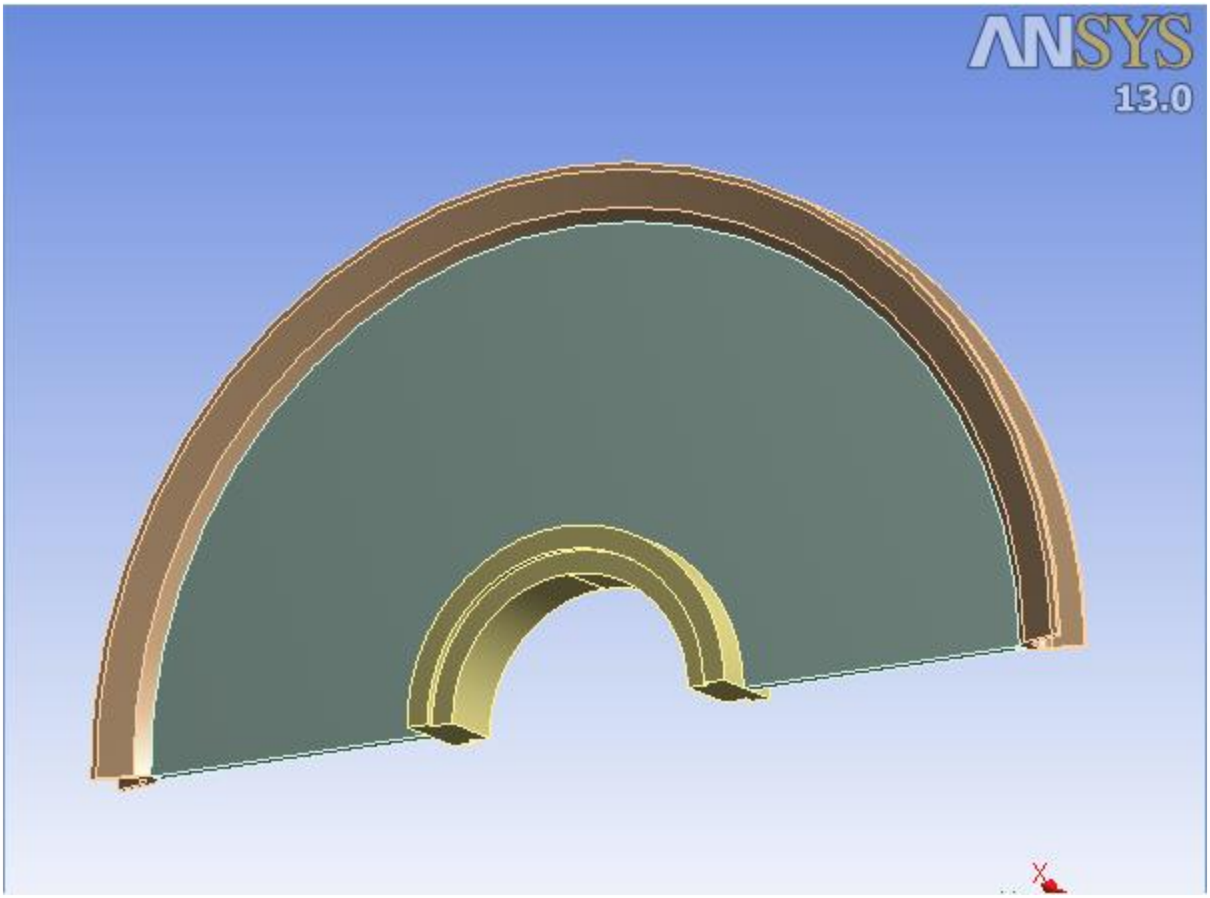
Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

TABLE 38
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

Project

First Saved	Thursday, February 27, 2014
Last Saved	Tuesday, May 13, 2014
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (B4, C4)

Geometry

TABLE 2
Model (B4, C4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Users\166864\Documents\masteroppgave\ansys\180rettplate_files\dp0\Geom\DM\Geom.agdb
Type	Inventor
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	200, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	3,9628e+007 mm³
Mass	311,08 kg
Scale Factor Value	1,
Statistics	
Bodies	4
Active Bodies	3
Nodes	157399
Elements	29988
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\166864\AppData\Local\Temp
Analysis Type	3-D

Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (B4, C4) > Geometry > Body Groups

Object Name	*Part*
State	Meshed
Graphics Properties	
Visible	Yes
Definition	
Suppressed	No
Assignment	Structural Steel
Coordinate System	Default Coordinate System
Bounding Box	
Length X	1540, mm
Length Y	18290 mm
Length Z	9145, mm
Properties	
Volume	3,9628e+010 mm³
Mass	311,08 kg
Centroid X	9719,3 mm
Centroid Y	-19196 mm
Centroid Z	-3662,1 mm
Moment of Inertia Ip1	7,7168e+006 kg·mm²
Moment of Inertia Ip2	1,7822e+006 kg·mm²
Moment of Inertia Ip3	5,9716e+006 kg·mm²
Statistics	
Nodes	157399
Elements	29988
Mesh Metric	None

TABLE 4
Model (B4, C4) > Geometry > Part > Parts

Object Name	*groove180:1*	*plate600:1*	*support600:1*
State	Meshed		
Graphics Properties			
Visible	Yes		
Glow	0		
Shininess	1		
Transparency	1		
Specularity	1		
Definition			
Suppressed	No		
ID (Beta)	689	691	693
Stiffness Behavior	Flexible		
Coordinate System	Default Coordinate System		
Reference Temperature	By Environment		

Material			
Assignment	Structural Steel		
Nonlinear Effects	Yes		
Thermal Strain Effects	Yes		
Bounding Box			
Length X	99, mm	25, mm	154, mm
Length Y	1829, mm	1643,5 mm	600, mm
Length Z	914,5 mm	821,76 mm	300, mm
Properties			
Volume	8,1048e+006 mm³	2,2583e+007 mm³	8,94e+006 mm³
Mass	63,622 kg	177,28 kg	70,179 kg
Centroid X	971,89 mm	971,93 mm	
Centroid Y	-1919,5 mm	-1919,6 mm	-1919,7 mm
Centroid Z	-546,7 mm	-379,76 mm	-168,36 mm
Moment of Inertia Ip1	2,8323e+007 kg·mm²	4,1056e+007 kg·mm²	2,9369e+006 kg·mm²
Moment of Inertia Ip2	4,5869e+006 kg·mm²	7,7857e+006 kg·mm²	5,9746e+005 kg·mm²
Moment of Inertia Ip3	2,3852e+007 kg·mm²	3,3289e+007 kg·mm²	2,5757e+006 kg·mm²
Statistics			
Nodes	42325	61335	57285
Elements	7056	10836	12096
Mesh Metric	None		

TABLE 5
Model (B4, C4) > Geometry > Parts

Object Name	*HUB for support:1*
State	Suppressed
Graphics Properties	
Visible	No
Definition	
Suppressed	Yes
ID (Beta)	696
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	200, mm
Length Y	444,25 mm
Length Z	222,12 mm
Properties	
Volume	1,55e+007 mm³
Mass	121,68 kg
Centroid X	971,93 mm
Centroid Y	-1919,6 mm

Centroid Z	-94,058 mm
Moment of Inertia Ip1	1,903e+006 kg·mm²
Moment of Inertia Ip2	8,1944e+005 kg·mm²
Moment of Inertia Ip3	1,891e+006 kg·mm²
Statistics	
Nodes	0
Elements	0
Mesh Metric	None

Coordinate Systems

TABLE 6
Model (B4, C4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 7
Model (B4, C4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 8
Model (B4, C4) > Connections > Contacts

Object Name	*Contacts*
State	Suppressed
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies

Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	5,1366 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 9
Model (B4, C4) > Connections > Contacts > Contact Regions

Object Name	*Frictional - support600:1 To HUB for support:1*
State	Suppressed
Scope	
Scoping Method	Geometry Selection
Contact	2 Faces
Target	No Selection
Contact Bodies	support600:1
Target Bodies	HUB for support:1
Definition	
Type	Frictional
Friction Coefficient	0,2
Scope Mode	Automatic
Behavior	Symmetric
Trim Contact (Beta)	Program Controlled
Suppressed	No
Advanced	
Formulation	Pure Penalty
Interface Treatment	Adjust to Touch
Normal Stiffness	Program Controlled
Update Stiffness	Each Iteration
Pinball Region	Program Controlled
Time Step Controls	None

Mesh

TABLE 10
Model (B4, C4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off

Relevance Center	Coarse
Element Size	12,50 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	9,35590 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Kept
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	157399
Elements	29988
Mesh Metric	None

TABLE 11
Model (B4, C4) > Mesh > Mesh Controls

Object Name	*Body Sizing*	*Face Sizing*	*Body Sizing 2*
State	Fully Defined		Suppressed
Scope			
Scoping Method	Geometry Selection		
Geometry	1 Body	5 Faces	No Selection
Definition			
Suppressed	No		
Type	Element Size		
Element Size	6, mm	12,5 mm	25, mm
Behavior	Soft		
Active			No, Suppressed Geometry

Static Structural (B5)

TABLE 12
Model (B4, C4) > Analysis

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 13
Model (B4, C4) > Static Structural (B5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	Yes
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points

Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180rettplate_files\dp0\SYS\MECH\
Future Analysis	Prestressed analysis
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 14
Model (B4, C4) > Static Structural (B5) > Loads

Object Name	*Pressure*	*Frictionless Support*	*Line Pressure*	*Cylindrical Support*
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face	6 Faces	1 Edge	1 Face
Definition				
ID (Beta)	45	110	402	756
Type	Pressure	Frictionless Support	Line Pressure	Cylindrical Support
Define By	Normal To		Vector	
Magnitude	20,1 MPa (ramped)		154,15 N/mm (ramped)	
Suppressed	No			
Direction			Defined	
Radial				Fixed
Axial				Fixed
Tangential				Free

FIGURE 1

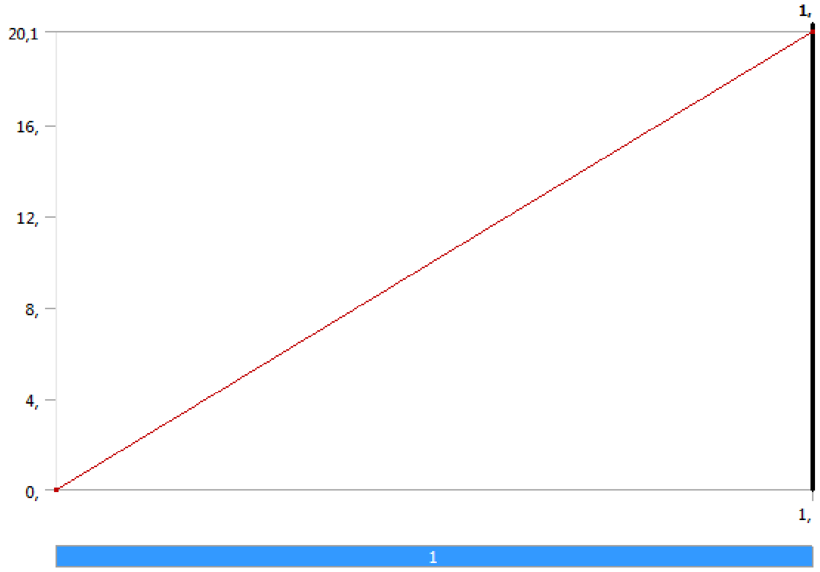
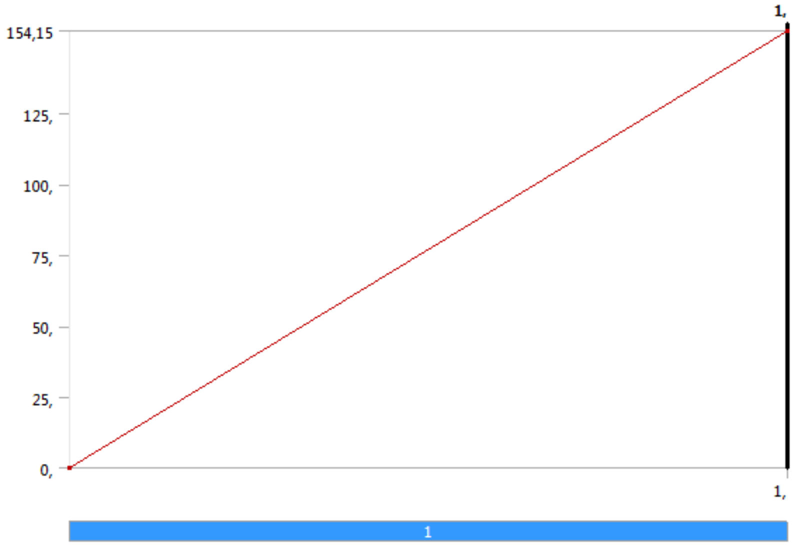


FIGURE 2
Model (B4, C4) > Static Structural (B5) > Line Pressure



Solution (B6)

TABLE 15
Model (B4, C4) > Static Structural (B5) > Solution

Object Name	*Solution (B6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 16
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 17
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Results

Object Name	*Equivalent Stress*	*Total Deformation*	*Maximum Principal Stress*	*Maximum Principal Stress 2*	*Minimum Principal Stress*
State	Solved				
Scope					
Scoping Method	Geometry Selection				
Geometry	All Bodies				
Definition					
Type	Equivalent (von-Mises) Stress	Total Deformation	Maximum Principal Stress		Minimum Principal Stress
By	Time				
Display Time	Last				
Calculate Time History	Yes				
Identifier					
Integration Point Results					
Display Option	Averaged		Unaveraged	Averaged	
Results					
Minimum	7,6937e-002 MPa	0, mm	-28,088 MPa	-28,087 MPa	-152,51 MPa
Maximum	116,53 MPa	3,1191 mm	103,73 MPa		18,71 MPa
Minimum Occurs On	support600:1		plate600:1		
Maximum Occurs On	plate600:1	groove180:1	plate600:1		
Information					
Time	1, s				
Load Step	1				
Substep	1				
Iteration Number	1				

TABLE 18
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Stress Safety Tools

Object Name	*Stress Tool*
State	Solved
Definition	
Theory	Max Equivalent Stress
Stress Limit Type	Tensile Yield Per Material

TABLE 19
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Stress Tool > Results

Object Name	*Safety Factor*	*Stress Ratio*
State	Solved	
Scope		
Scoping Method	Geometry Selection	
Geometry	All Bodies	
Definition		
Type	Safety Factor	Stress Ratio
By	Time	
Display Time	Last	
Calculate Time History	Yes	
Identifier		
Integration Point Results		
Display Option	Averaged	
Results		
Minimum	2,5316	2,608e-004
Minimum Occurs On	plate600:1	support600:1
Maximum		0,39501
Maximum Occurs On		plate600:1
Information		
Time	1, s	
Load Step	1	
Substep	1	
Iteration Number	1	

TABLE 20
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Probes

Object Name	*Force Reaction 2*	*Force Reaction*
State	Solved	
Definition		
Type	Force Reaction	
Location Method	Boundary Condition	
Boundary Condition	Cylindrical Support	Frictionless Support
Orientation	Global Coordinate System	
Options		
Result Selection	All	
Display Time	End Time	
Results		
X Axis	15451 N	-42,909 N

Y Axis	4,4229e+005 N	2023,7 N
Z Axis	-3,854e+005 N	-54160 N
Total	5,8685e+005 N	54197 N
Maximum Value Over Time		
X Axis	15451 N	-42,909 N
Y Axis	4,4229e+005 N	2023,7 N
Z Axis	-3,854e+005 N	-54160 N
Total	5,8685e+005 N	54197 N
Minimum Value Over Time		
X Axis	15451 N	-42,909 N
Y Axis	4,4229e+005 N	2023,7 N
Z Axis	-3,854e+005 N	-54160 N
Total	5,8685e+005 N	54197 N
Information		
Time	1, s	
Load Step	1	
Substep	1	
Iteration Number	1	

Linear Buckling (C5)

TABLE 21
Model (B4, C4) > Analysis

Object Name	*Linear Buckling (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Linear Buckling
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

TABLE 22
Model (B4, C4) > Linear Buckling (C5) > Initial Condition

Object Name	*Pre-Stress (Static Structural)*
State	Fully Defined
Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time
Contact Status	Use True Status

TABLE 23
Model (B4, C4) > Linear Buckling (C5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Options	
Max Modes to Find	3,
Output Controls	
Calculate Stress	No
Calculate Strain	No
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180rettplate_files\dp0\SYS-1\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	nmm

Solution (C6)

TABLE 24
Model (B4, C4) > Linear Buckling (C5) > Solution

Object Name	*Solution (C6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

FIGURE 3
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

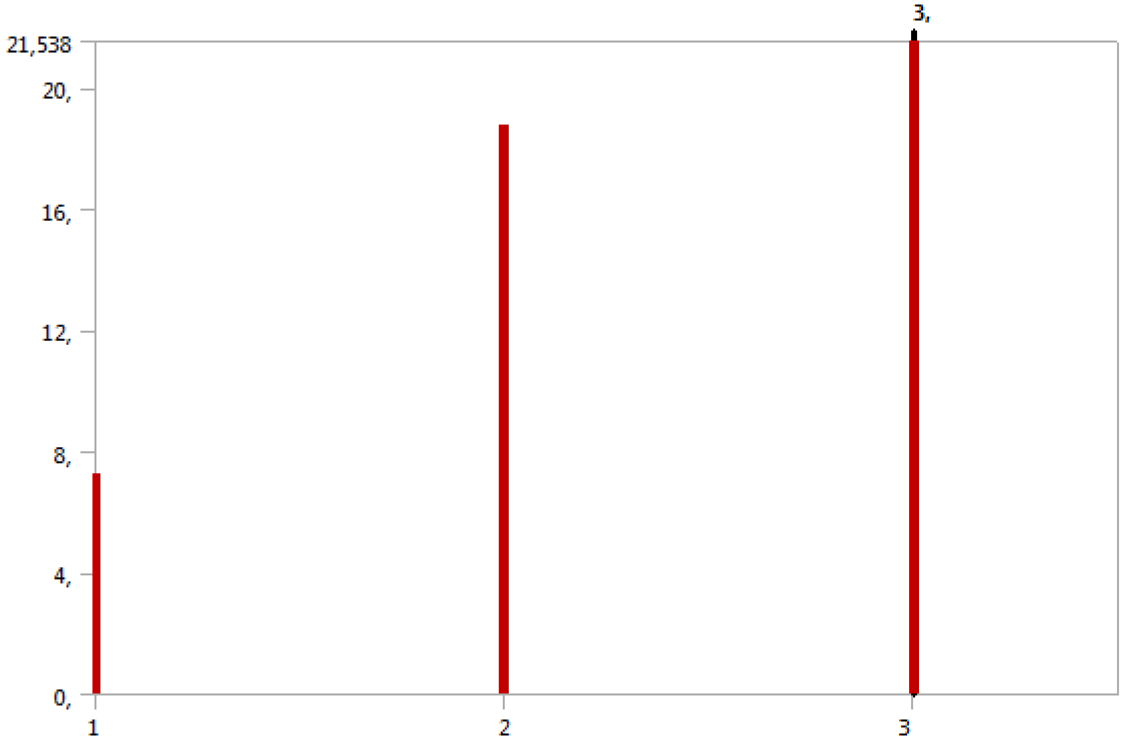


TABLE 25
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

Mode	Load Multiplier
1,	15,594
2,	31,38
3,	47,954

TABLE 26

Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 27
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Results

Object Name	*Total Deformation*	*Total Deformation 2*	*Total Deformation 3*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation		
Mode	1,	2,	3,
Identifier			
Results			
Load Multiplier	15,594	31,38	47,954
Minimum	0, mm		
Maximum	1,0001 mm	1,0041 mm	1,0105 mm
Minimum Occurs On	support600:1		
Maximum Occurs On	groove180:1		

TABLE 28
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation

Mode	Load Multiplier
1,	15,594
2,	31,38
3,	47,954

TABLE 29
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 2

Mode	Load Multiplier
1,	15,594
2,	31,38
3,	47,954

TABLE 30
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 3

Mode	Load Multiplier
1,	15,594
2,	31,38
3,	47,954

Material Data

Structural Steel

TABLE 31
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 32
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 33
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa
295

TABLE 34
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
295

TABLE 35
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
470

TABLE 36
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 37
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 38
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 39
Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

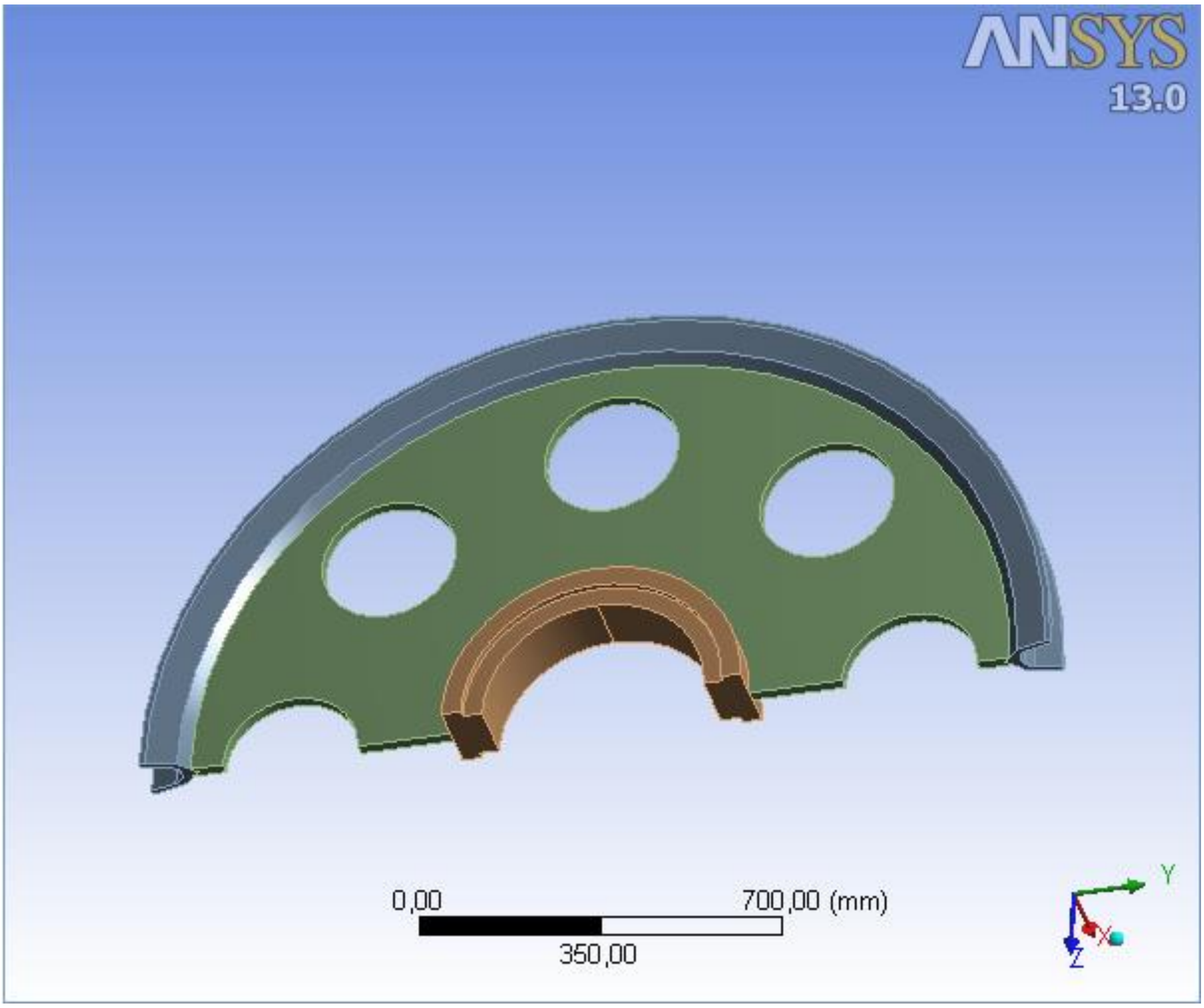
TABLE 40
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

D.3 STRAIGHT WEB WITH HOLES

Project

First Saved	Thursday, February 27, 2014
Last Saved	Tuesday, May 13, 2014
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (B4, C4)

Geometry

TABLE 2
Model (B4, C4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Users\166864\Documents\masteroppgave\ansys\180RettPlateMedHull_files\dp0\Geom\DM\Geom.agdb
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	154, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	3,3902e+007 mm³
Mass	266,13 kg
Scale Factor Value	1,
Statistics	
Bodies	3
Active Bodies	3
Nodes	137336
Elements	30252
Mesh Metric	None
Preferences	
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\166864\AppData\Local\Temp
Analysis Type	3-D
Enclosure and Symmetry Processing	Yes

TABLE 3

Model (B4, C4) > Geometry > Body Groups

Object Name	*Part*
State	Meshed
Graphics Properties	
Visible	Yes
Definition	
Suppressed	No
Assignment	Structural Steel
Coordinate System	Default Coordinate System
Bounding Box	
Length X	154, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	3,3902e+007 mm³
Mass	266,13 kg
Centroid X	894,83 mm
Centroid Y	-1035,5 mm
Centroid Z	-362,36 mm
Moment of Inertia Ip1	6,6259e+007 kg·mm²
Moment of Inertia Ip2	1,5737e+007 kg·mm²
Moment of Inertia Ip3	5,0888e+007 kg·mm²
Statistics	
Nodes	137336
Elements	30252
Mesh Metric	None

TABLE 4
Model (B4, C4) > Geometry > Part > Parts

Object Name	*groove180:1*	*plate600:1*	*support600:1*
State	Meshed		
Graphics Properties			
Visible	Yes		
Glow	0		
Shininess	1		
Transparency	1		
Specularity	1		
Definition			
Suppressed	No		
ID (Beta)	297	299	301
Stiffness Behavior	Flexible		
Coordinate System	Default Coordinate System		
Reference Temperature	By Environment		

Material			
Assignment	Structural Steel		
Nonlinear Effects	Yes		
Thermal Strain Effects	Yes		
Bounding Box			
Length X	99, mm	25, mm	154, mm
Length Y	1829, mm	1643,5 mm	600, mm
Length Z	914,5 mm	821,76 mm	300, mm
Properties			
Volume	8,1048e+006 mm³	1,6858e+007 mm³	8,94e+006 mm³
Mass	63,622 kg	132,33 kg	70,179 kg
Centroid X	894,8 mm	894,84 mm	
Centroid Y	-1035,4 mm	-1035,5 mm	-1035,6 mm
Centroid Z	-546,7 mm	-376,61 mm	-168,37 mm
Moment of Inertia Ip1	2,8323e+007 kg·mm²	3,0169e+007 kg·mm²	2,9369e+006 kg·mm²
Moment of Inertia Ip2	4,5869e+006 kg·mm²	5,7226e+006 kg·mm²	5,9746e+005 kg·mm²
Moment of Inertia Ip3	2,3852e+007 kg·mm²	2,446e+007 kg·mm²	2,5757e+006 kg·mm²
Statistics			
Nodes	44001	67077	29297
Elements	9038	14056	7158
Mesh Metric	None		

Coordinate Systems

TABLE 5
Model (B4, C4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 6
Model (B4, C4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 7
Model (B4, C4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	5,1267 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

Mesh

TABLE 8
Model (B4, C4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	12,50 mm
Initial Size Seed	Active Assembly
Smoothing	Medium

Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	9,35590 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	137336
Elements	30252
Mesh Metric	None

TABLE 9
Model (B4, C4) > Mesh > Mesh Controls

Object Name	*MultiZone*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Geometry	3 Bodies
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Free Mesh Type	Hexa Dominant
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source	Program Controlled
Advanced	
Mesh Based Defeaturing	Off
Minimum Edge Length	9,3559 mm

Write ICEM CFD Files	No

Static Structural (B5)

TABLE 10
Model (B4, C4) > Analysis

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 11
Model (B4, C4) > Static Structural (B5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Restart Analysis	
Restart Type	Program Controlled
Status	Done
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	Yes
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes

Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180RettPlateMedHull_files\dp0\SYS\MECH\
Future Analysis	Prestressed analysis
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 12
Model (B4, C4) > Static Structural (B5) > Loads

Object Name	*Pressure*	*Cylindrical Support*	*Frictionless Support*	*Line Pressure*
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face		8 Faces	1 Edge
Definition				
ID (Beta)	38	88	90	295
Type	Pressure	Cylindrical Support	Frictionless Support	Line Pressure
Define By	Normal To			Vector
Magnitude	20,1 MPa (ramped)			154,15 N/mm (ramped)
Suppressed	No			
Radial		Fixed		
Axial		Fixed		
Tangential		Free		
Direction				Defined

FIGURE 1
Model (B4, C4) > Static Structural (B5) > Pressure

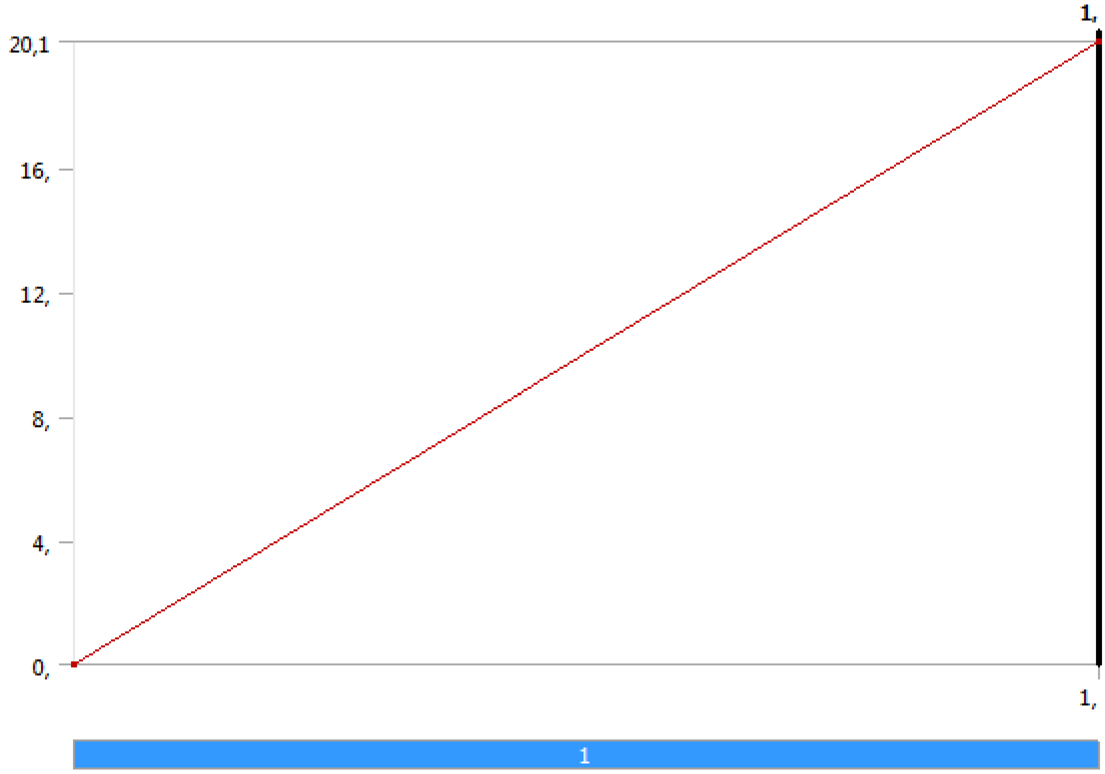
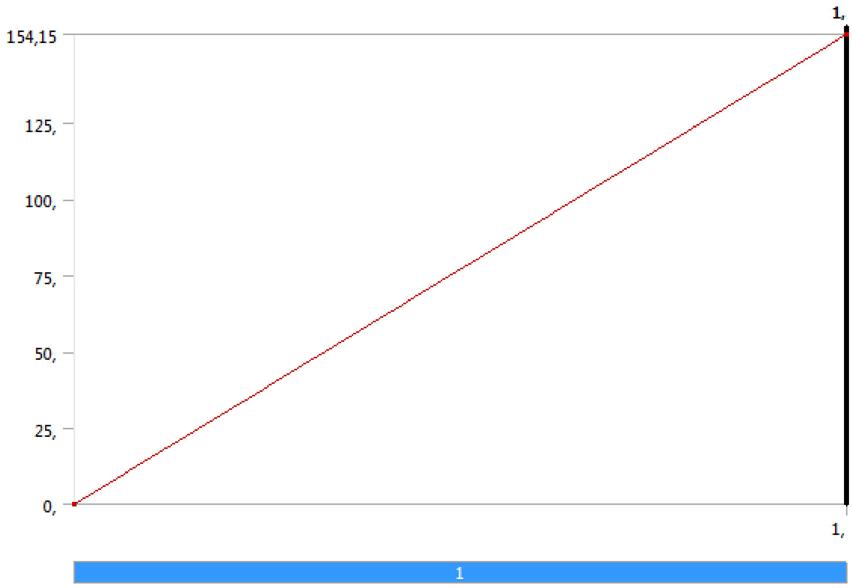


FIGURE 2
Model (B4, C4) > Static Structural (B5) > Line Pressure



Solution (B6)

TABLE 13
Model (B4, C4) > Static Structural (B5) > Solution

Object Name	*Solution (B6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 14
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 15
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Results

Object Name	*Equivalent Stress*	*Maximum Principal Stress*	*Minimum Principal Stress*	*Maximum Principal Stress 2*
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	All Bodies			
Definition				
Type	Equivalent (von-Mises) Stress	Maximum Principal Stress	Minimum Principal Stress	Maximum Principal Stress
By	Time			
Display Time	Last			
Calculate Time History	Yes			
Identifier				
Integration Point Results				
Display Option	Averaged			Unaveraged
Results				
Minimum	7,8946e-002 MPa	-34,318 MPa	-171,2 MPa	-34,844 MPa
Maximum	129,95 MPa	112,64 MPa	23,666 MPa	148,59 MPa
Minimum Occurs On	support600:1	plate600:1		
Maximum Occurs On	plate600:1			

Information	
Time	1, s
Load Step	1
Substep	1
Iteration Number	1

TABLE 16
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Stress Safety Tools

Object Name	*Stress Tool*
State	Solved
Definition	
Theory	Max Equivalent Stress
Stress Limit Type	Tensile Yield Per Material

TABLE 17
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Stress Tool > Results

Object Name	*Safety Factor*
State	Solved
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Safety Factor
By	Time
Display Time	Last
Calculate Time History	Yes
Identifier	
Integration Point Results	
Display Option	Averaged
Results	
Minimum	1,9238
Minimum Occurs On	plate600:1
Information	
Time	1, s
Load Step	1
Substep	1
Iteration Number	1

TABLE 18
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Probes

Object Name	*Force Reaction*	*Force Reaction 2*
State	Solved	
Definition		
Type	Force Reaction	
Location Method	Boundary Condition	
Boundary Condition	Cylindrical Support	Frictionless Support
Orientation	Global Coordinate System	
Options		

Result Selection	All	
Display Time	End Time	
Results		
X Axis	15451 N	-89,43 N
Y Axis	4,4229e+005 N	4354,9 N
Z Axis	-3,9519e+005 N	-43432 N
Total	5,9333e+005 N	43650 N
Maximum Value Over Time		
X Axis	15451 N	-89,43 N
Y Axis	4,4229e+005 N	4354,9 N
Z Axis	-3,9519e+005 N	-43432 N
Total	5,9333e+005 N	43650 N
Minimum Value Over Time		
X Axis	15451 N	-89,43 N
Y Axis	4,4229e+005 N	4354,9 N
Z Axis	-3,9519e+005 N	-43432 N
Total	5,9333e+005 N	43650 N
Information		
Time	1, s	
Load Step	1	
Substep	1	
Iteration Number	1	

Linear Buckling (C5)

TABLE 19
Model (B4, C4) > Analysis

Object Name	*Linear Buckling (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Linear Buckling
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

TABLE 20
Model (B4, C4) > Linear Buckling (C5) > Initial Condition

Object Name	*Pre-Stress (Static Structural)*
State	Fully Defined
Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time

Contact Status	Use True Status

TABLE 21
Model (B4, C4) > Linear Buckling (C5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Options	
Max Modes to Find	3,
Output Controls	
Calculate Stress	No
Calculate Strain	No
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180RettPlateMedHull_files\dp0\SYS-1\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	nmm

Solution (C6)

TABLE 22
Model (B4, C4) > Linear Buckling (C5) > Solution

Object Name	*Solution (C6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

FIGURE 3
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

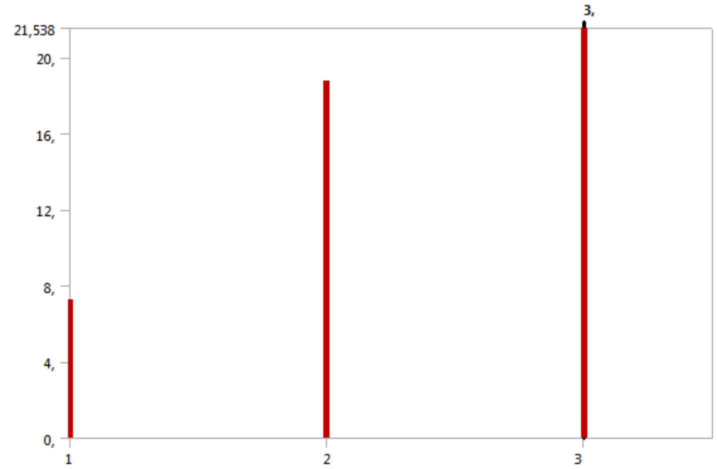


TABLE 23
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

Mode	Load Multiplier
1,	11,919
2,	23,884
3,	31,31

TABLE 24
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 25
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Results

Object Name	*Total Deformation*	*Total Deformation 2*	*Total Deformation 3*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation		
Mode	1,	2,	3,
Identifier			
Results			
Load Multiplier	11,919	23,884	31,31
Minimum	0, mm		
Maximum	1, mm	1,005 mm	1,0128 mm

Minimum Occurs On	support600:1	
Maximum Occurs On	plate600:1	groove180:1

TABLE 26
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation

Mode	Load Multiplier
1,	11,919
2,	23,884
3,	31,31

TABLE 27
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 2

Mode	Load Multiplier
1,	11,919
2,	23,884
3,	31,31

TABLE 28
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 3

Mode	Load Multiplier
1,	11,919
2,	23,884
3,	31,31

Material Data

Structural Steel

TABLE 29
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 30

Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 31
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa

250

TABLE 32
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
250

TABLE 33
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
460

TABLE 34
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 35
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 36
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 37
Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

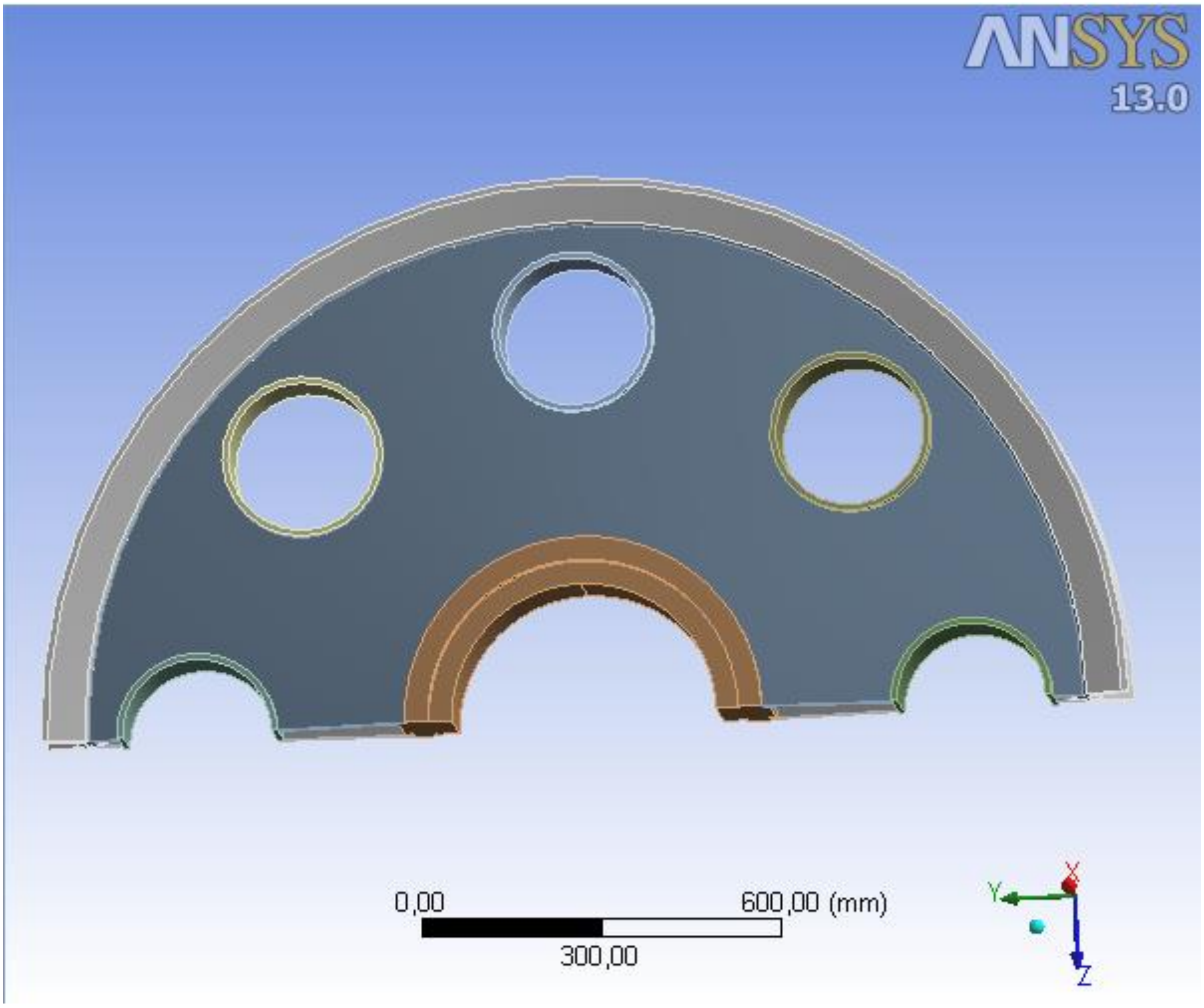
TABLE 38
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

D.4 DOUBLE WEB WITH HOLES

Project

First Saved	Friday, February 28, 2014
Last Saved	Tuesday, May 13, 2014
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (B4, C4)

Geometry

TABLE 2
Model (B4, C4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Users\166864\Documents\masteroppgave\ansys\180dobbelweb8hull_files\dp0\Geom\DM\Geom.agdb
Type	Inventor
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	154, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	3,9493e+007 mm³
Mass	310,02 kg
Scale Factor Value	1,
Statistics	
Bodies	11
Active Bodies	9
Nodes	245492
Elements	67636
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes

Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\166864\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (B4, C4) > Geometry > Body Groups

Object Name	*plate180med5hull:1*
State	Meshed
Graphics Properties	
Visible	Yes
Definition	
Suppressed	No
Assignment	Structural Steel
Coordinate System	Default Coordinate System
Bounding Box	
Length X	1540, mm
Length Y	18290 mm
Length Z	9145, mm
Properties	
Volume	3,9493e+010 mm³
Mass	310,02 kg
Centroid X	-4027,4 mm
Centroid Y	-19987 mm
Centroid Z	-3630,2 mm
Moment of Inertia Ip1	7,6512e+006 kg·mm²
Moment of Inertia Ip2	1,8134e+006 kg·mm²
Moment of Inertia Ip3	5,9154e+006 kg·mm²
Statistics	
Nodes	245492
Elements	67636
Mesh Metric	None

TABLE 4
Model (B4, C4) > Geometry > plate180med5hull:1 > Parts

Object Name	*pipefor5hull:3*	*plate180med5hull:1*	*plate180med5hull:1*	*pipefor5hull180:1*	*support180:1*
State	Meshed				
Graphics Properties					
Visible	Yes				
Glow	0				
Shininess	1				
Transparen	1				

cy					
Specularity	1				
Definition					
Suppressed	No				
ID (Beta)	637	639	641	643	645
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	Structural Steel				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	120, mm	51,71 mm		120, mm	154, mm
Length Y	270, mm	1673, mm		270, mm	625, mm
Length Z	270, mm	836,5 mm		135, mm	312,5 mm
Properties					
Volume	9,8002e+005 mm³	8,6025e+006 mm³	8,6024e+006 mm³	4,9008e+005 mm³	1,0263e+007 mm³
Mass	7,6931 kg	67,53 kg	67,529 kg	3,8472 kg	80,564 kg
Centroid X	-402,73 mm	-442,8 mm	-362,67 mm	-402,74 mm	
Centroid Y	-1538, mm	-1998,7 mm	-1998,8 mm	-2650,2 mm	-1998,7 mm
Centroid Z	-460,56 mm	-379,05 mm	-379,04 mm	-82,602 mm	-171,75 mm
Moment of Inertia Ip1	1,3014e+005 kg·mm²	1,5586e+007 kg·mm²	1,5585e+007 kg·mm²	38706 kg·mm²	3,5216e+006 kg·mm²
Moment of Inertia Ip2	74305 kg·mm²	2,9649e+006 kg·mm²	2,9647e+006 kg·mm²	10814 kg·mm²	7,0665e+005 kg·mm²
Moment of Inertia Ip3	74346 kg·mm²	1,2638e+007 kg·mm²	1,2637e+007 kg·mm²	37153 kg·mm²	3,0729e+006 kg·mm²
Statistics					
Nodes	6380	68471	58633	3479	42386
Elements	1372	19990	16926	801	10578
Mesh Metric	None				

TABLE 5
Model (B4, C4) > Geometry > plate180med5hull:1 > Parts

Object Name	*pipefor5hull180:2*	*groove180:1*	*groove180:1*	*pipefor5hull:2*	*groove180:1*
State	Meshed	Suppressed		Meshed	
Graphics Properties					
Visible	Yes	No		Yes	

Glow	0				
Shininess	1				
Transparency	1				
Specularity	1				
Definition					
Suppressed	No	Yes		No	
ID (Beta)	647	649	651	653	655
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	Structural Steel				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	120, mm	10,913 mm		120, mm	99, mm
Length Y	270, mm	1673, mm		270, mm	1829, mm
Length Z	135, mm	836,5 mm		270, mm	914,5 mm
Properties					
Volume	4,9014e+005 mm³	8,1048 mm³	8,1044 mm³	9,8002e+005 mm³	8,1047e+006 mm³
Mass	3,8476 kg	6,3623e-005 kg	6,3619e-005 kg	7,6932 kg	63,622 kg
Centroid X	-402,73 mm	-424,41 mm	-381,18 mm	-402,74 mm	-402,73 mm
Centroid Y	-1347,3 mm	-1994,3 mm	-1981,8 mm	-2459,4 mm	-1998,8 mm
Centroid Z	-82,591 mm	-521,39 mm	-517,45 mm	-460,71 mm	-546,62 mm
Moment of Inertia Ip1	38661 kg·mm²	30,398 kg·mm²	27,873 kg·mm²	1,3019e+005 kg·mm²	2,8354e+007 kg·mm²
Moment of Inertia Ip2	10799 kg·mm²	4,9134 kg·mm²	4,4389 kg·mm²	74316 kg·mm²	4,5916e+006 kg·mm²
Moment of Inertia Ip3	37115 kg·mm²	25,486 kg·mm²	23,436 kg·mm²	74390 kg·mm²	2,3878e+007 kg·mm²
Statistics					
Nodes	3320	0		6240	55913
Elements	682	0		1287	14654
Mesh Metric	None				

TABLE 6
Model (B4, C4) > Geometry > plate180med5hull:1 > Parts

Object Name	*pipefor5hull:1*
State	Meshed
Graphics Properties	
Visible	Yes

Glow	0
Shininess	1
Transparency	1
Specularity	1
Definition	
Suppressed	No
ID (Beta)	657
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	120, mm
Length Y	270, mm
Length Z	270, mm
Properties	
Volume	9,8003e+005 mm³
Mass	7,6932 kg
Centroid X	-402,75 mm
Centroid Y	-1998,7 mm
Centroid Z	-651,55 mm
Moment of Inertia Ip1	1,3015e+005 kg·mm²
Moment of Inertia Ip2	74298 kg·mm²
Moment of Inertia Ip3	74361 kg·mm²
Statistics	
Nodes	6298
Elements	1346
Mesh Metric	None

Coordinate Systems

TABLE 7
Model (B4, C4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	

X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 8
Model (B4, C4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 9
Model (B4, C4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	5,1267 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 10
Model (B4, C4) > Connections > Contacts > Contact Regions

Object Name	*Contact Region*	*Contact Region 3*	*Bonded - Multiple To Multiple*	*Bonded - Multiple To Multiple*
State	Suppressed		Fully Defined	
Scope				
Scoping Method	Geometry Selection			
Contact	1 Face			
Target	1 Face			
Contact Bodies	plate180med5hull:1		Multiple	
Target Bodies	support 180:1		Multiple	
Definition				

Type	Bonded	
Scope Mode	Automatic	Manual
Behavior	Symmetric	
Trim Contact (Beta)	Program Controlled	
Suppressed	Yes	No
Advanced		
Formulation	Pure Penalty	
Normal Stiffness	Program Controlled	
Update Stiffness	Never	
Pinball Region	Program Controlled	

Mesh

TABLE 11
Model (B4, C4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	12,0 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	2,7196e-004 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	0
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	

Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	245492
Elements	67636
Mesh Metric	None

TABLE 12
Model (B4, C4) > Mesh > Mesh Controls

Object Name	*Hex Dominant Method*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Geometry	9 Bodies
Definition	
Suppressed	No
Method	Hex Dominant
Element Midside Nodes	Use Global Setting
Free Face Mesh Type	Quad/Tri
Control Messages	Yes, Click To Display...

Static Structural (B5)

TABLE 13
Model (B4, C4) > Analysis

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 14
Model (B4, C4) > Static Structural (B5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s

Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	Yes
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180dobbelweb8hull_files\dp0\SYS\MECH\
Future Analysis	Prestressed analysis
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 15
Model (B4, C4) > Static Structural (B5) > Loads

Object Name	*Pressure*	*Cylindrical Support*	*Frictionless Support*	*Line Pressure*
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face		16 Faces	1 Edge
Definition				
ID (Beta)	40	218	220	741

Type	Pressure	Cylindrical Support	Frictionless Support	Line Pressure
Define By	Normal To			Vector
Magnitude	20,1 MPa (ramped)			154,15 N/mm (ramped)
Suppressed	No			
Radial		Fixed		
Axial		Fixed		
Tangential		Free		
Direction				Defined

FIGURE 1
Model (B4, C4) > Static Structural (B5) > Pressure

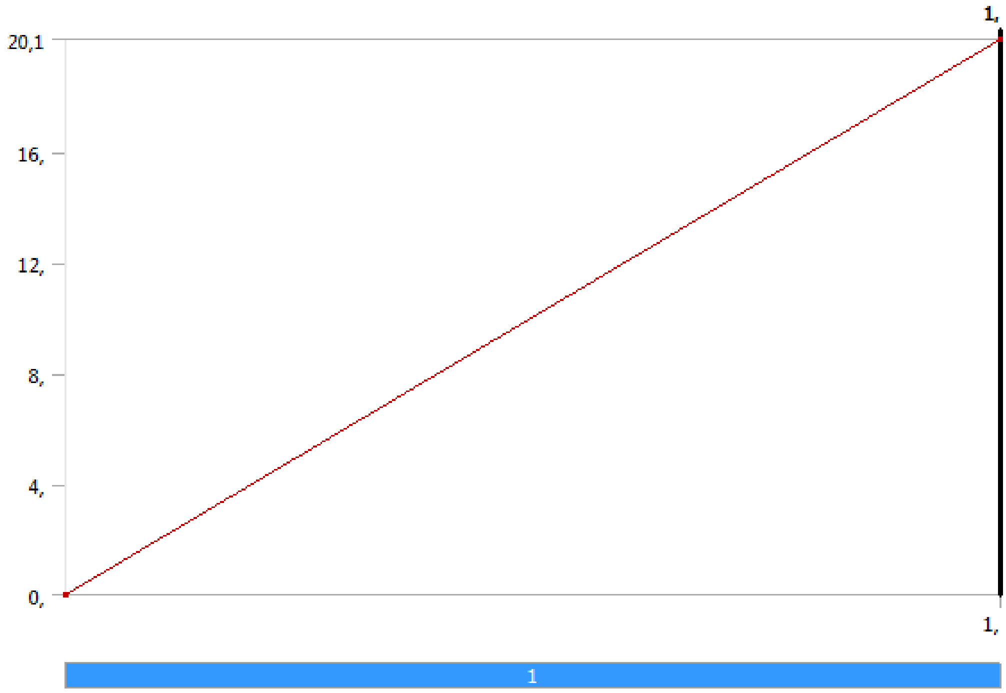
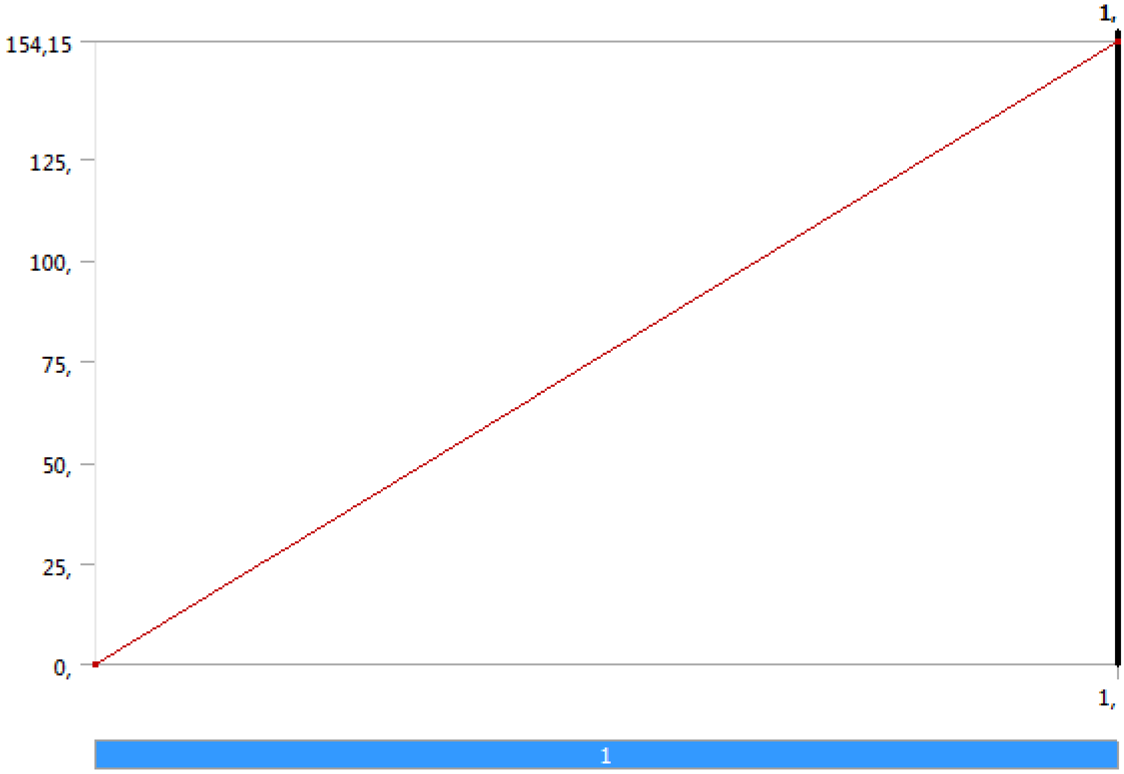


FIGURE 2
Model (B4, C4) > Static Structural (B5) > Line Pressure



Solution (B6)

TABLE 16
Model (B4, C4) > Static Structural (B5) > Solution

Object Name	*Solution (B6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 17
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 18
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Results

Object Name	*Equivalent Stress*	*Total Deformation*	*Maximum Principal Stress*	*Minimum Principal Stress*	*Maximum Principal Stress 2*
State	Solved				
Scope					
Scoping Method	Geometry Selection				
Geometry	All Bodies				
Definition					
Type	Equivalent (von-Mises) Stress	Total Deformation	Maximum Principal Stress	Minimum Principal Stress	Maximum Principal Stress
By	Time				
Display Time	Last				
Calculate Time History	Yes				
Identifier					
Integration Point Results					
Display Option	Averaged		Averaged		Unaveraged
Results					
Minimum	5,1461e-002 MPa	0, mm	-24,683 MPa	-79,369 MPa	-32,436 MPa
Maximum	68,747 MPa	0,41621 mm	51,663 MPa	10,135 MPa	73,923 MPa
Minimum Occurs On	groove180:1	support 180:1	groove180:1	plate180med5hull:1	groove180:1
Maximum Occurs On	pipefor5hull:3	groove180:1	support 180:1		
Information					
Time	1, s				
Load Step	1				
Substep	1				
Iteration Number	1				

TABLE 19
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Results

Object Name	*Equivalent Stress 2*
State	Solved
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Equivalent (von-Mises) Stress
By	Time

Display Time	Last
Calculate Time History	Yes
Identifier	
Integration Point Results	
Display Option	Unaveraged
Results	
Minimum	5,7899e-002 MPa
Maximum	101,31 MPa
Minimum Occurs On	groove180:1
Maximum Occurs On	pipefor5hull:3
Information	
Time	1, s
Load Step	1
Substep	1
Iteration Number	1

Linear Buckling (C5)

TABLE 20
Model (B4, C4) > Analysis

Object Name	*Linear Buckling (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Linear Buckling
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

TABLE 21
Model (B4, C4) > Linear Buckling (C5) > Initial Condition

Object Name	*Pre-Stress (Static Structural)*
State	Fully Defined
Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time
Contact Status	Use True Status

TABLE 22
Model (B4, C4) > Linear Buckling (C5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Options	
Max Modes to Find	3,

Output Controls	
Calculate Stress	No
Calculate Strain	No
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180dobbelweb8hull_files\dp0\SYS-1\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	nmm

Solution (C6)

TABLE 23
Model (B4, C4) > Linear Buckling (C5) > Solution

Object Name	*Solution (C6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

FIGURE 3
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

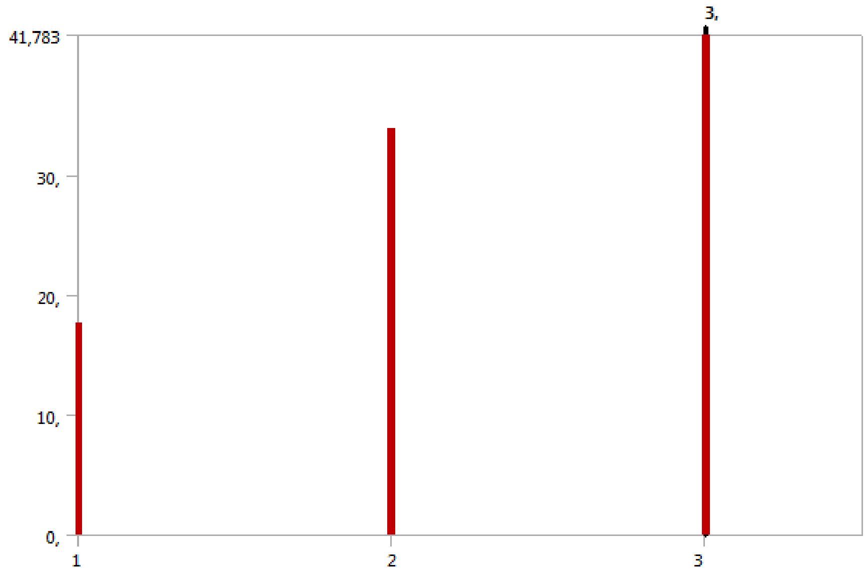


TABLE 24
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

Mode	Load Multiplier
1,	17,675
2,	33,929
3,	41,783

TABLE 25
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 26
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Results

Object Name	*Total Deformation*	*Total Deformation 2*	*Total Deformation 3*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation		
Mode	1,	2,	3,
Identifier			
Results			

Load Multiplier	17,675	33,929	41,783
Minimum	0, mm		
Maximum	1, mm		1,0026 mm
Minimum Occurs On	support 180:1		
Maximum Occurs On	pipefor5hull180:2		plate180med5hull:1

TABLE 27
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation

Mode	Load Multiplier
1,	17,675
2,	33,929
3,	41,783

TABLE 28
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 2

Mode	Load Multiplier
1,	17,675
2,	33,929
3,	41,783

TABLE 29
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 3

Mode	Load Multiplier
1,	17,675
2,	33,929
3,	41,783

Material Data

Structural Steel

TABLE 30
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 31
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 32
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa
250

TABLE 33
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
250

TABLE 34
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
460

TABLE 35
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 36
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 37
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 38
Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

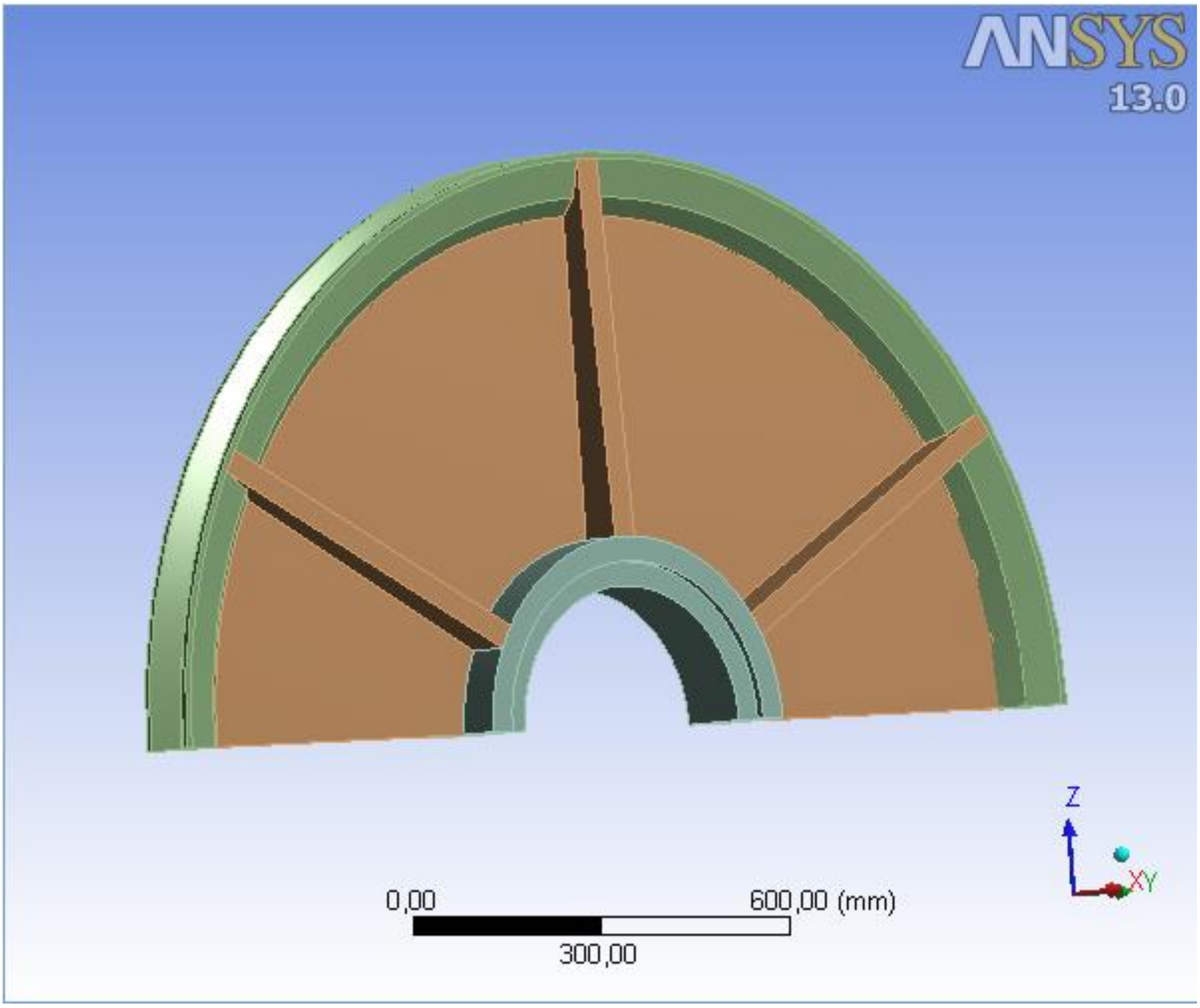
TABLE 39
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

D.5 THIN WEB WITH STIFFENERS

Project

First Saved	Thursday, February 20, 2014
Last Saved	Tuesday, May 13, 2014
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (B4, C4)

Geometry

TABLE 2
Model (B4, C4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Users\166864\Documents\masteroppgave\ansys\18rettplatemedspiler_files\dp0\Geom\DM\Geom.agdb
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	154, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	3,3788e+007 mm³
Mass	265,24 kg
Scale Factor Value	1,
Statistics	
Bodies	9
Active Bodies	3
Nodes	252936
Elements	59855
Mesh Metric	None
Preferences	
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\166864\AppData\Local\Temp
Analysis Type	3-D
Enclosure and Symmetry Processing	Yes

TABLE 3

Model (B4, C4) > Geometry > Body Groups

Object Name	*Part*
State	Meshed
Graphics Properties	
Visible	Yes
Definition	
Suppressed	No
Assignment	Structural Steel
Coordinate System	Default Coordinate System
Bounding Box	
Length X	154, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	3,3789e+007 mm³
Mass	265,24 kg
Centroid X	1746,9 mm
Centroid Y	-479,28 mm
Centroid Z	966,2 mm
Moment of Inertia Ip1	6,4117e+007 kg·mm²
Moment of Inertia Ip2	1,4953e+007 kg·mm²
Moment of Inertia Ip3	4,9712e+007 kg·mm²
Statistics	
Nodes	252936
Elements	59855
Mesh Metric	None

TABLE 4
Model (B4, C4) > Geometry > Part > Parts

Object Name	*groove180:1*	*groove180:1*	*groove180:1*	*groove180:1*	*groove180:1*
State	Suppressed				
Graphics Properties					
Visible	No				
Glow	0				
Shininess	1				
Transparency	1				
Specularity	1				
Definition					
Suppressed	Yes				
ID (Beta)	513	515	517	519	521
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				

Reference Temperature	By Environment				
Material					
Assignment	Structural Steel				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	44,5 mm				
Length Y	96,915 mm	40, mm	96,914 mm	96,915 mm	96,914 mm
Length Z	79,048 mm	89,035 mm	79,048 mm		
Properties					
Volume	147,62 mm³				
Mass	1,1588e-003 kg				
Centroid X	1717,9 mm			1776, mm	
Centroid Y	255,76 mm	-479,28 mm	-1214,3 mm	256,06 mm	-1214,6 mm
Centroid Z	1029,4 mm	1453,8 mm	1029,4 mm		
Moment of Inertia Ip1	2,1921 kg·mm²	2,0167 kg·mm²	2,1922 kg·mm²	2,1735 kg·mm²	2,1739 kg·mm²
Moment of Inertia Ip2	0,48765 kg·mm²	1,703 kg·mm²	0,48788 kg·mm²	0,48437 kg·mm²	0,48441 kg·mm²
Moment of Inertia Ip3	1,7453 kg·mm²	0,36159 kg·mm²	1,745 kg·mm²	1,7285 kg·mm²	1,7291 kg·mm²
Statistics					
Nodes	0				
Elements	0				
Mesh Metric	None				

TABLE 5
Model (B4, C4) > Geometry > Part > Parts

Object Name	*groove180:1*	*groove180:1*	*platefortyntsteg:1*	*support600:1*
State	Suppressed	Meshed		
Graphics Properties				
Visible	No	Yes		
Glow	0			
Shininess	1			
Transparency	1			
Specularity	1			
Definition				
Suppressed	Yes	No		
ID (Beta)	523	525	527	529
Stiffness Behavior	Flexible			
Coordinate System	Default Coordinate System			
Reference Temperature	By Environment			
Material				
Assignment	Structural Steel			
Nonlinear Effects	Yes			

Thermal Strain Effects	Yes			
Bounding Box				
Length X	44,5 mm	99, mm	154, mm	
Length Y	40, mm	1829, mm	1628,5 mm	600, mm
Length Z	89,035 mm	914,5 mm	903,05 mm	300, mm
Properties				
Volume	147,62 mm³	8,1039e+006 mm³	1,6744e+007 mm³	8,94e+006 mm³
Mass	1,1588e-003 kg	63,615 kg	131,44 kg	70,179 kg
Centroid X	1775,8 mm	1746,9 mm		
Centroid Y	-479,28 mm	-479,21 mm	-479,28 mm	-479,35 mm
Centroid Z	1453,8 mm	1151,7 mm	979,27 mm	773,53 mm
Moment of Inertia Ip1	2,0177 kg·mm²	2,8246e+007 kg·mm²	2,8103e+007 kg·mm²	2,951e+006 kg·mm²
Moment of Inertia Ip2	1,7042 kg·mm²	4,5773e+006 kg·mm²	4,9586e+006 kg·mm²	5,9992e+005 kg·mm²
Moment of Inertia Ip3	0,36175 kg·mm²	2,3784e+007 kg·mm²	2,334e+007 kg·mm²	2,5884e+006 kg·mm²
Statistics				
Nodes	0	78108	124562	54163
Elements	0	19401	26381	14073
Mesh Metric	None			

Coordinate Systems

TABLE 6
Model (B4, C4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 7
Model (B4, C4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 8
Model (B4, C4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	5,1267 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 9
Model (B4, C4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Multiple To Multiple*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Contact	18 Faces
Target	18 Faces
Contact Bodies	Multiple
Target Bodies	Multiple
Definition	
Type	Bonded
Scope Mode	Manual
Behavior	Symmetric

Trim Contact (Beta)	Program Controlled
Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

Mesh

TABLE 10
Model (B4, C4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	10,0 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	1,6926e-004 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default

Statistics	
Nodes	252936
Elements	59855
Mesh Metric	None

TABLE 11
Model (B4, C4) > Mesh > Mesh Controls

Object Name	*Hex Dominant Method*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Geometry	3 Bodies
Definition	
Suppressed	No
Method	Hex Dominant
Element Midside Nodes	Use Global Setting
Free Face Mesh Type	Quad/Tri
Control Messages	Yes, Click To Display...

Static Structural (B5)

TABLE 12
Model (B4, C4) > Analysis

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 13
Model (B4, C4) > Static Structural (B5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Restart Analysis	
Restart Type	Program Controlled
Status	Done
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled

Weak Springs	Off
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	Yes
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\18rettplatemedspiler_files\dp0\SYS\MECH\
Future Analysis	Prestressed analysis
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 14
Model (B4, C4) > Static Structural (B5) > Loads

Object Name	*Cylindrical Support*	*Pressure*	*Frictionless Support*	*Line Pressure*
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face		6 Faces	1 Edge
Definition				
ID (Beta)	43	45	73	534
Type	Cylindrical	Pressure	Frictionless	Line Pressure

	Support		Support	
Radial	Fixed			
Axial	Fixed			
Tangential	Free			
Suppressed		No		
Define By		Normal To		Vector
Magnitude		20,1 MPa (ramped)		154,15 N/mm (ramped)
Direction				Defined

FIGURE 1
Model (B4, C4) > Static Structural (B5) > Pressure

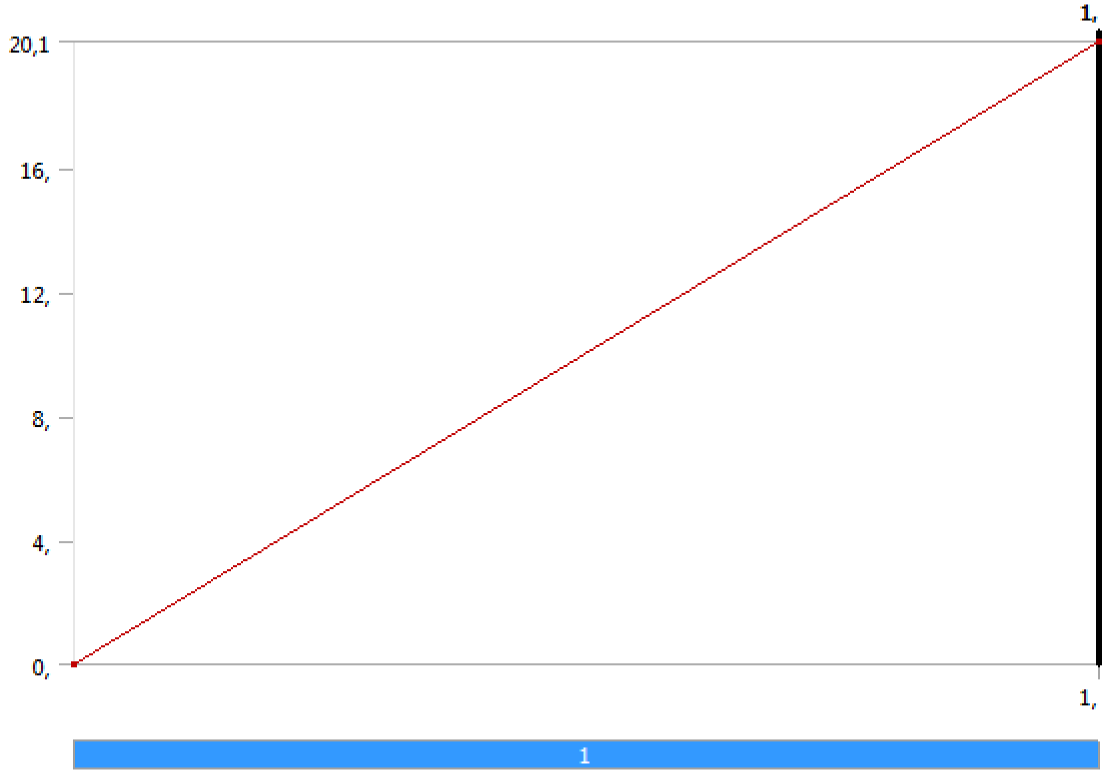
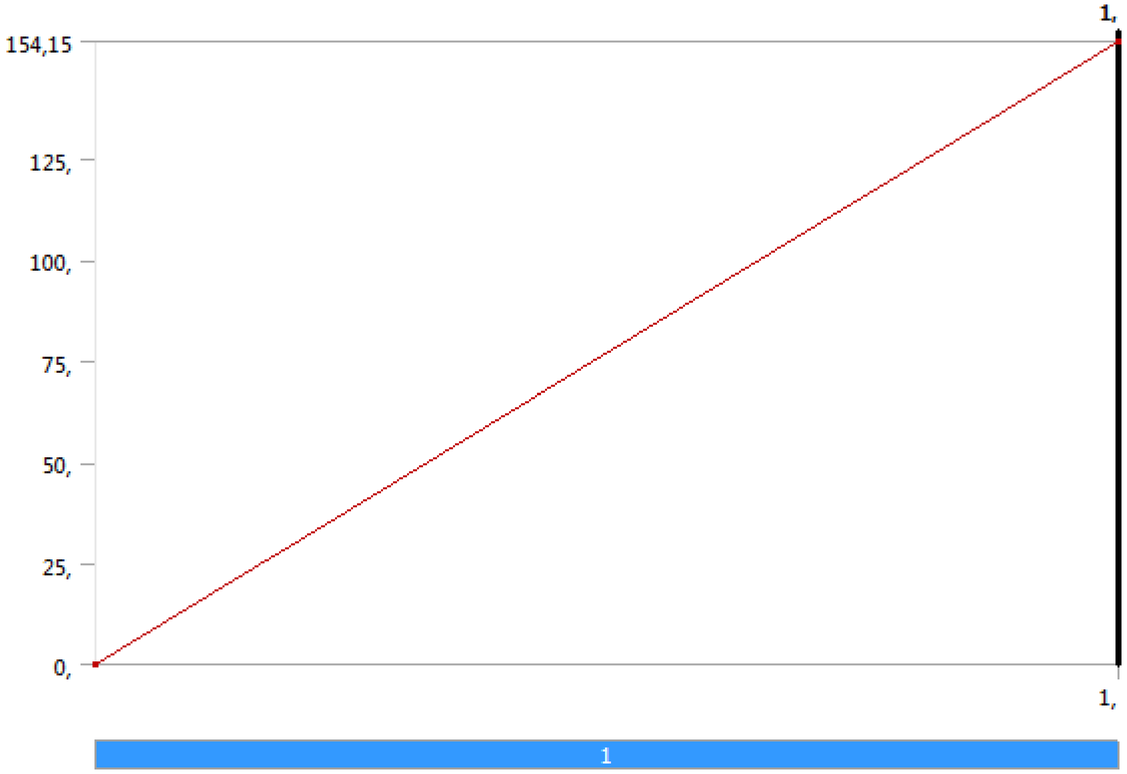


FIGURE 2
Model (B4, C4) > Static Structural (B5) > Line Pressure



Solution (B6)

TABLE 15
Model (B4, C4) > Static Structural (B5) > Solution

Object Name	*Solution (B6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 16
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 17
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Results

Object Name	*Equivalent Stress*	*Total Deformation*	*Maximum Principal Stress*	*Minimum Principal Stress*	*Maximum Principal Stress 2*
State	Solved				
Scope					
Scoping Method	Geometry Selection				
Geometry	All Bodies				
Definition					
Type	Equivalent (von-Mises) Stress	Total Deformation	Maximum Principal Stress	Minimum Principal Stress	Maximum Principal Stress
By	Time				
Display Time	Last				
Calculate Time History	Yes				
Identifier					
Integration Point Results					
Display Option	Averaged		Averaged		Unaveraged
Results					
Minimum	4,6572e-002 MPa	0, mm	-30,652 MPa	-149,26 MPa	-33,55 MPa
Maximum	121,51 MPa	0,79267 mm	103,54 MPa	20,587 MPa	226,2 MPa
Minimum Occurs On	support600:1				groove180:1
Maximum Occurs On	groove180:1		platefortyntsteg:1	support600:1	groove180:1
Information					
Time	1, s				
Load Step	1				
Substep	1				
Iteration Number	1				

TABLE 18
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Probes

Object Name	*Force Reaction*	*Force Reaction 2*
State	Solved	
Definition		
Type	Force Reaction	
Location Method	Boundary Condition	
Boundary Condition	Cylindrical Support	Frictionless Support
Orientation	Global Coordinate System	
Options		

Result Selection	All	
Display Time	End Time	
Results		
X Axis	-15451 N	47,774 N
Y Axis	4,4229e+005 N	1983,7 N
Z Axis	4,1323e+005 N	26654 N
Total	6,0549e+005 N	26728 N
Maximum Value Over Time		
X Axis	-15451 N	47,774 N
Y Axis	4,4229e+005 N	1983,7 N
Z Axis	4,1323e+005 N	26654 N
Total	6,0549e+005 N	26728 N
Minimum Value Over Time		
X Axis	-15451 N	47,774 N
Y Axis	4,4229e+005 N	1983,7 N
Z Axis	4,1323e+005 N	26654 N
Total	6,0549e+005 N	26728 N
Information		
Time	1, s	
Load Step	1	
Substep	1	
Iteration Number	1	

Linear Buckling (C5)

TABLE 19
Model (B4, C4) > Analysis

Object Name	*Linear Buckling (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Linear Buckling
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

TABLE 20
Model (B4, C4) > Linear Buckling (C5) > Initial Condition

Object Name	*Pre-Stress (Static Structural)*
State	Fully Defined
Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time

Contact Status	Use True Status

TABLE 21
Model (B4, C4) > Linear Buckling (C5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Options	
Max Modes to Find	3,
Output Controls	
Calculate Stress	No
Calculate Strain	No
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\18rettplate medspiler_files\dp0\SYS-1\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	nmm

Solution (C6)

TABLE 22
Model (B4, C4) > Linear Buckling (C5) > Solution

Object Name	*Solution (C6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

FIGURE 3
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

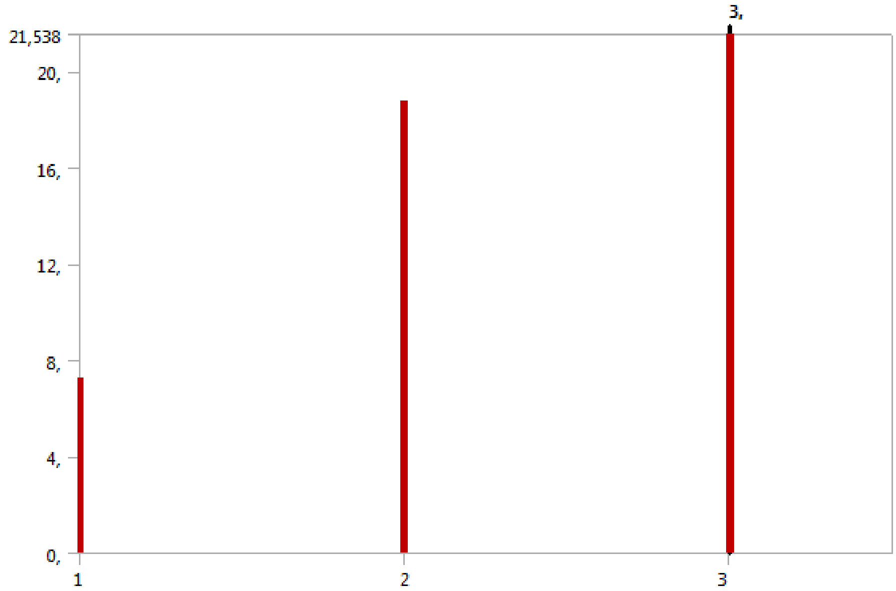


TABLE 23
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

Mode	Load Multiplier
1,	9,2286
2,	10,036
3,	13,858

TABLE 24
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 25
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Results

Object Name	*Total Deformation*	*Total Deformation 2*	*Total Deformation 3*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation		
Mode	1,	2,	3,
Identifier			

Results			
Load Multiplier	9,2286	10,036	13,858
Minimum	0, mm		
Maximum	1, mm		
Minimum Occurs On	support600:1		
Maximum Occurs On	platefortyntsteg:1		

TABLE 26
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation

Mode	Load Multiplier
1,	9,2286
2,	10,036
3,	13,858

TABLE 27
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 2

Mode	Load Multiplier
1,	9,2286
2,	10,036
3,	13,858

TABLE 28
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 3

Mode	Load Multiplier
1,	9,2286
2,	10,036
3,	13,858

Material Data

Structural Steel

TABLE 29
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 30
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 31
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa

250

TABLE 32
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
250

TABLE 33
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
460

TABLE 34
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 35
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 36
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 37
Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

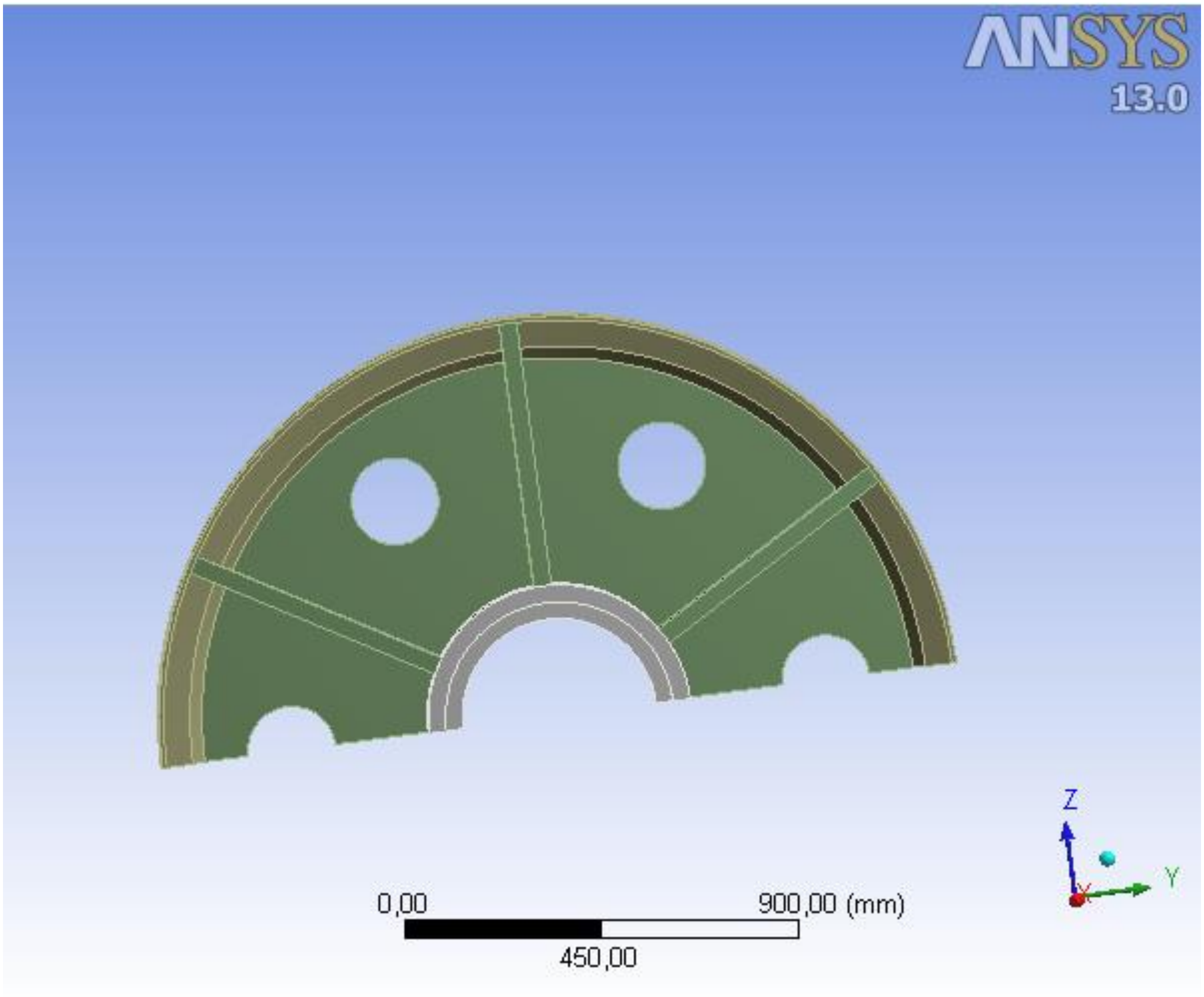
TABLE 38
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

D.6 THIN WEB WITH STIFFENERS AND HOLES

Project

First Saved	Thursday, February 20, 2014
Last Saved	Tuesday, May 13, 2014
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (B4, C4)

Geometry

TABLE 2
Model (B4, C4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Users\166864\Documents\masteroppgave\ansys\180rettplatemedspileroghull_files\dp0\Geom\DM\Geom.agdb
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	154, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	3,2846e+007 mm³
Mass	257,84 kg
Scale Factor Value	1,
Statistics	
Bodies	9
Active Bodies	3
Nodes	246967
Elements	59400
Mesh Metric	None
Preferences	
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes

Temporary Directory	C:\Users\166864\AppData\Local\Temp
Analysis Type	3-D
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (B4, C4) > Geometry > Body Groups

Object Name	*Part*
State	Meshed
Graphics Properties	
Visible	Yes
Definition	
Suppressed	No
Assignment	Structural Steel
Coordinate System	Default Coordinate System
Bounding Box	
Length X	154, mm
Length Y	1829, mm
Length Z	914,5 mm
Properties	
Volume	3,2847e+007 mm³
Mass	257,85 kg
Centroid X	-775,78 mm
Centroid Y	-932,6 mm
Centroid Z	1450,9 mm
Moment of Inertia Ip1	9,3131e+007 kg·mm²
Moment of Inertia Ip2	6,3458e+007 kg·mm²
Moment of Inertia Ip3	5,5186e+007 kg·mm²
Statistics	
Nodes	246967
Elements	59400
Mesh Metric	None

TABLE 4
Model (B4, C4) > Geometry > Part > Parts

Object Name	*platefortyntsteg:1*	*groove180:1*	*groove180:1*	*groove180:1*	*groove180:1*
State	Meshed	Suppressed			
Graphics Properties					
Visible	Yes	No			
Glow	0				
Shininess	1				
Transparency	1				
Specularity	1				
Definition					
Suppressed	No	Yes			
ID (Beta)	206	254	256	258	260
Stiffness Behavior	Flexible				

Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	Structural Steel				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	154, mm	44,5 mm			
Length Y	1628,5 mm	96,915 mm	40, mm	96,914 mm	96,915 mm
Length Z	903,05 mm	79,048 mm	89,035 mm	79,048 mm	
Properties					
Volume	1,5802e+007 mm^3	147,62 mm^3			
Mass	124,04 kg	1,1588e-003 kg			
Centroid X	-1029,9 mm	-1001, mm		-1000,9 mm	-1059, mm
Centroid Y	-1238,1 mm	-1973,1 mm	-1238,1 mm	-503,07 mm	-1973,4 mm
Centroid Z	2000,6 mm	2050,4 mm	2474,8 mm	2050,4 mm	
Moment of Inertia Ip1	2,6289e+007 kg·mm^2	2,1907 kg·mm^2	2,0166 kg·mm^2	2,1908 kg·mm^2	2,1718 kg·mm^2
Moment of Inertia Ip2	4,5524e+006 kg·mm^2	0,48773 kg·mm^2	1,7035 kg·mm^2	0,48733 kg·mm^2	0,48442 kg·mm^2
Moment of Inertia Ip3	2,1933e+007 kg·mm^2	1,7439 kg·mm^2	0,36188 kg·mm^2	1,7443 kg·mm^2	1,7269 kg·mm^2
Statistics					
Nodes	118038	0			
Elements	26078	0			
Mesh Metric	None				

TABLE 5
Model (B4, C4) > Geometry > Part > Parts

Object Name	*groove180:1*	*groove180:1*	*groove180:1*	*support600:1*
State	Suppressed		Meshed	
Graphics Properties				
Visible	No		Yes	
Glow	0			
Shininess	1			
Transparency	1			
Specularity	1			
Definition				
Suppressed	Yes		No	
ID (Beta)	262	264	266	268
Stiffness Behavior	Flexible			
Coordinate System	Default Coordinate System			
Reference	By Environment			

Temperature				
Material				
Assignment	Structural Steel			
Nonlinear Effects	Yes			
Thermal Strain Effects	Yes			
Bounding Box				
Length X	44,5 mm		99, mm	154, mm
Length Y	96,914 mm	40, mm	1829, mm	600, mm
Length Z	79,048 mm	89,035 mm	914,5 mm	300, mm
Properties				
Volume	147,62 mm³		8,1039e+006 mm³	8,94e+006 mm³
Mass	1,1588e-003 kg		63,615 kg	70,179 kg
Centroid X	-1059, mm	-1058,8 mm	-1029,9 mm	
Centroid Y	-502,76 mm	-1238,1 mm	-1238,2 mm	-1238, mm
Centroid Z	2050,4 mm	2474,8 mm	2172,7 mm	1794,5 mm
Moment of Inertia Ip1	2,172 kg·mm²	2,0143 kg·mm²	2,8246e+007 kg·mm²	2,9534e+006 kg·mm²
Moment of Inertia Ip2	0,48419 kg·mm²	1,7005 kg·mm²	4,5774e+006 kg·mm²	6,0038e+005 kg·mm²
Moment of Inertia Ip3	1,7272 kg·mm²	0,36124 kg·mm²	2,3784e+007 kg·mm²	2,5906e+006 kg·mm²
Statistics				
Nodes	0		77730	55089
Elements	0		19187	14135
Mesh Metric	None			

Coordinate Systems

TABLE 6
Model (B4, C4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 7
Model (B4, C4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 8
Model (B4, C4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	5,1267 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 9
Model (B4, C4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Multiple To Multiple*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Contact	18 Faces
Target	18 Faces
Contact Bodies	Multiple
Target Bodies	Multiple
Definition	
Type	Bonded
Scope Mode	Manual
Behavior	Symmetric
Trim Contact (Beta)	Program Controlled
Suppressed	No

Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

Mesh

TABLE 10
Model (B4, C4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	10,0 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	6,6718e-005 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	246967

Elements	59400
Mesh Metric	None

TABLE 11
Model (B4, C4) > Mesh > Mesh Controls

Object Name	*Hex Dominant Method*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Geometry	3 Bodies
Definition	
Suppressed	No
Method	Hex Dominant
Element Midside Nodes	Use Global Setting
Free Face Mesh Type	Quad/Tri
Control Messages	Yes, Click To Display...

Static Structural (B5)

TABLE 12
Model (B4, C4) > Analysis

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 13
Model (B4, C4) > Static Structural (B5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Restart Analysis	
Restart Type	Program Controlled
Status	Done
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled

Weak Springs	Off
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	Yes
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180rettplate medspileroghull_files\dp0\SYS\MECH\
Future Analysis	Prestressed analysis
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 14
Model (B4, C4) > Static Structural (B5) > Loads

Object Name	*Cylindrical Support*	*Pressure*	*Frictionless Support*	*Line Pressure*
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face		8 Faces	1 Edge
Definition				
ID (Beta)	43	45	63	343
Type	Cylindrical Support	Pressure	Frictionless Support	Line Pressure
Radial	Fixed			
Axial	Fixed			
Tangential	Free			
Suppressed	No			
Define By		Normal To		Vector

Magnitude		20,1 MPa (ramped)		154,15 N/mm (ramped)
Direction				Defined

FIGURE 1
Model (B4, C4) > Static Structural (B5) > Pressure

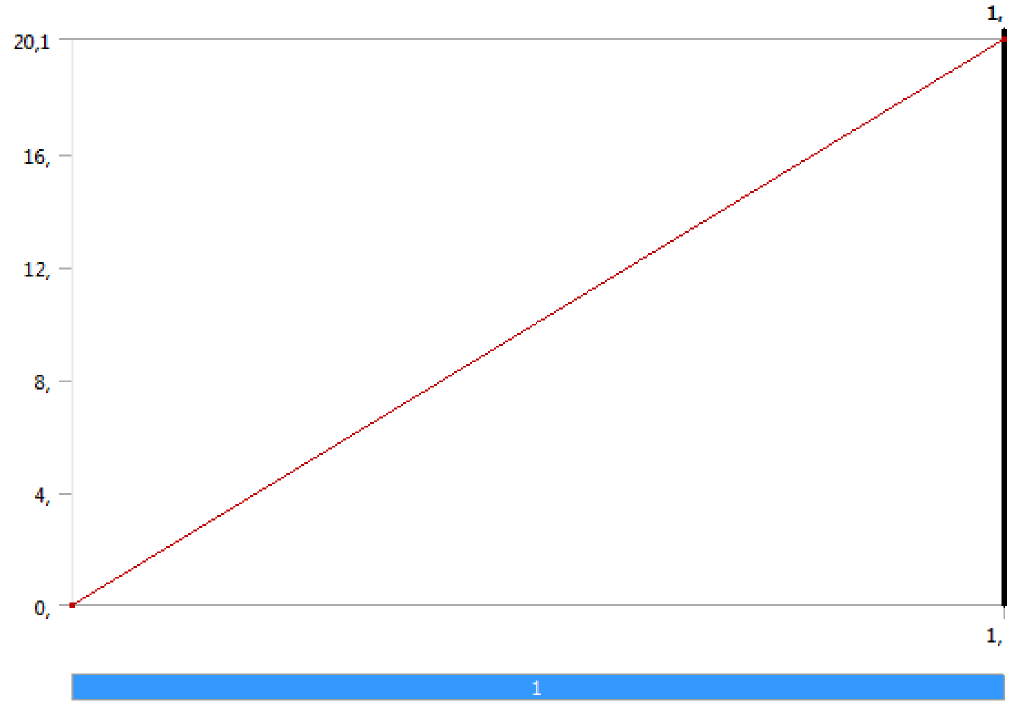
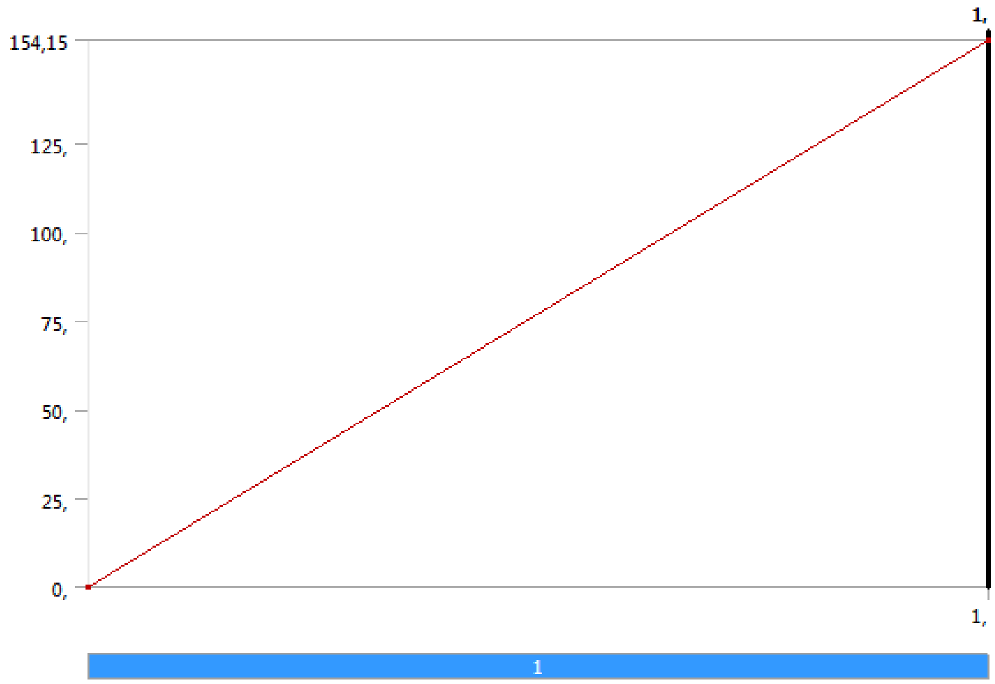


FIGURE 2
Model (B4, C4) > Static Structural (B5) > Line Pressure



Solution (B6)

TABLE 15
Model (B4, C4) > Static Structural (B5) > Solution

Object Name	*Solution (B6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 16
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 17
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Results

Object Name	*Equivalent Stress*	*Maximum Principal Stress*	*Minimum Principal Stress*	*Maximum Principal Stress 2*
State	Solved			
Scope				

Scoping Method	Geometry Selection			
Geometry	All Bodies			
Definition				
Type	Equivalent (von-Mises) Stress	Maximum Principal Stress	Minimum Principal Stress	Maximum Principal Stress
By	Time			
Display Time	Last			
Calculate Time History	Yes			
Identifier				
Integration Point Results				
Display Option	Averaged			Unaveraged
Results				
Minimum	8,0552e-002 MPa	-35,112 MPa	-155,2 MPa	-44,642 MPa
Maximum	151,02 MPa	103,86 MPa	18,463 MPa	263,73 MPa
Minimum Occurs On	support600:1			groove180:1
Maximum Occurs On	platefortyntsteg:1		support600:1	groove180:1
Information				
Time	1, s			
Load Step	1			
Substep	1			
Iteration Number	1			

TABLE 18
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Probes

Object Name	*Force Reaction*	*Force Reaction 2*
State	Solved	
Definition		
Type	Force Reaction	
Location Method	Boundary Condition	
Boundary Condition	Cylindrical Support	Frictionless Support
Orientation	Global Coordinate System	
Options		
Result Selection	All	
Display Time	End Time	
Results		
X Axis	15451 N	-43,4 N
Y Axis	-4,4229e+005 N	-1997,3 N
Z Axis	4,1745e+005 N	22315 N
Total	6,0838e+005 N	22404 N
Maximum Value Over Time		
X Axis	15451 N	-43,4 N
Y Axis	-4,4229e+005 N	-1997,3 N
Z Axis	4,1745e+005 N	22315 N

Total	6,0838e+005 N	22404 N
Minimum Value Over Time		
X Axis	15451 N	-43,4 N
Y Axis	-4,4229e+005 N	-1997,3 N
Z Axis	4,1745e+005 N	22315 N
Total	6,0838e+005 N	22404 N
Information		
Time	1, s	
Load Step	1	
Substep	1	
Iteration Number	1	

Linear Buckling (C5)

TABLE 19
Model (B4, C4) > Analysis

Object Name	*Linear Buckling (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Linear Buckling
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

TABLE 20
Model (B4, C4) > Linear Buckling (C5) > Initial Condition

Object Name	*Pre-Stress (Static Structural)*
State	Fully Defined
Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time
Contact Status	Use True Status

TABLE 21
Model (B4, C4) > Linear Buckling (C5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Options	
Max Modes to Find	3,
Output Controls	
Calculate Stress	No
Calculate Strain	No
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180rettplatemedspileroghull_files\dp0\SYS-1\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	nmm

Solution (C6)

TABLE 22
Model (B4, C4) > Linear Buckling (C5) > Solution

Object Name	*Solution (C6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

FIGURE 3
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

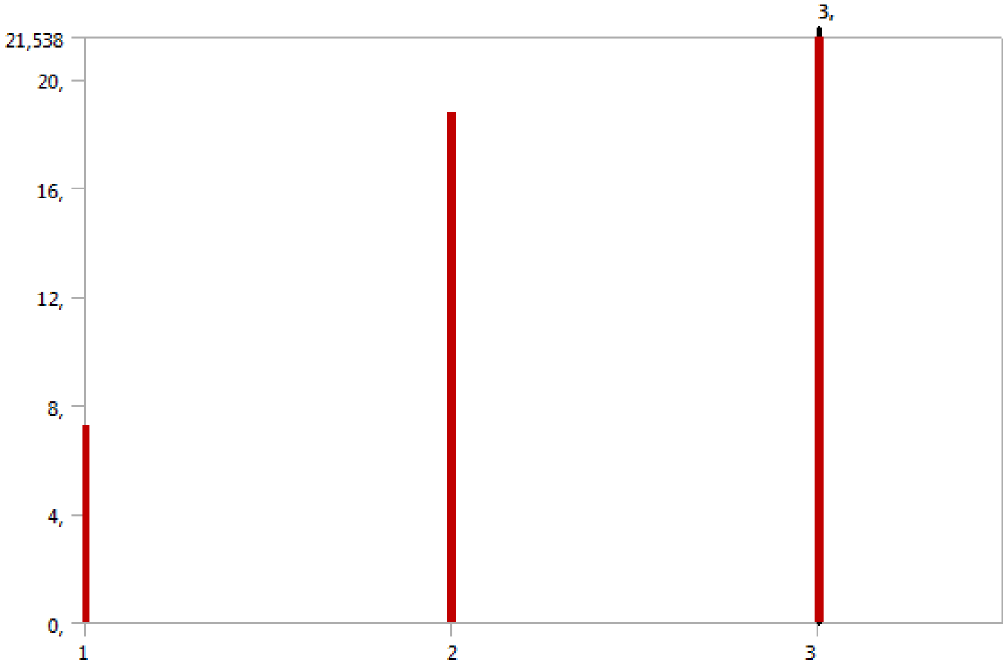


TABLE 23
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

Mode	Load Multiplier
1,	9,4142
2,	9,7825
3,	12,279

TABLE 24
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output

Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 25
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Results

Object Name	*Total Deformation*	*Total Deformation 2*	*Total Deformation 3*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation		
Mode	1,	2,	3,
Identifier			
Results			
Load Multiplier	9,4142	9,7825	12,279
Minimum	0, mm		
Maximum	1,0005 mm	1,0002 mm	1,0004 mm
Minimum Occurs On	support600:1		
Maximum Occurs On	platefortyntsteg:1		

TABLE 26
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation

Mode	Load Multiplier
1,	9,4142
2,	9,7825
3,	12,279

TABLE 27
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 2

Mode	Load Multiplier
1,	9,4142
2,	9,7825
3,	12,279

TABLE 28
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 3

Mode	Load Multiplier
1,	9,4142
2,	9,7825
3,	12,279

Material Data

Structural Steel

TABLE 29
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 30
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 31
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa
250

TABLE 32
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
250

TABLE 33
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
460

TABLE 34
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 35
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0

214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 36
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 37
Structural Steel > Isotropic Elasticity

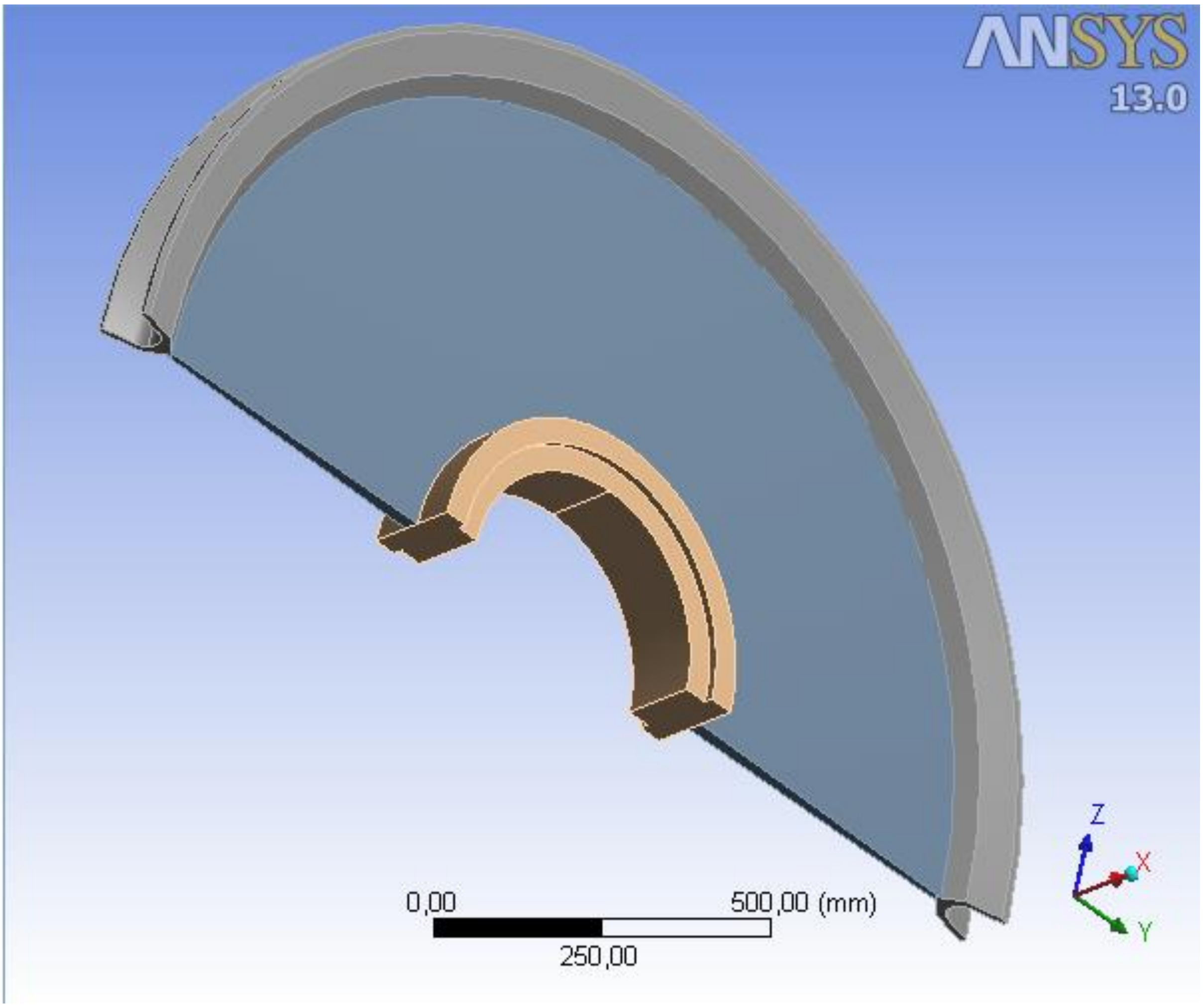
Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

TABLE 38
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

Project

First Saved	Wednesday, March 05, 2014
Last Saved	Friday, May 09, 2014
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (B4, C4)

Geometry

TABLE 2
Model (B4, C4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Users\166864\Documents\masteroppgave\ansys\180valgtplatekonsept_files\dp0\Geom\DM\Geom.agdb
Type	Inventor
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	155,8 mm
Length Y	1829,3 mm
Length Z	914,5 mm
Properties	
Volume	3,313e+007 mm³
Mass	260,07 kg
Scale Factor Value	1,
Statistics	
Bodies	3
Active Bodies	3
Nodes	27664
Elements	19776
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes

Temporary Directory	C:\Users\166864\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (B4, C4) > Geometry > Body Groups

Object Name	*Part*
State	Meshed
Graphics Properties	
Visible	Yes
Definition	
Suppressed	No
Assignment	Structural Steel
Coordinate System	Default Coordinate System
Bounding Box	
Length X	155,8 mm
Length Y	1829,3 mm
Length Z	914,5 mm
Properties	
Volume	3,313e+007 mm³
Mass	260,07 kg
Centroid X	908,64 mm
Centroid Y	-1717,4 mm
Centroid Z	353,47 mm
Moment of Inertia Ip1	6,2452e+007 kg·mm²
Moment of Inertia Ip2	1,5066e+007 kg·mm²
Moment of Inertia Ip3	4,7747e+007 kg·mm²
Statistics	
Nodes	27664
Elements	19776
Mesh Metric	None

TABLE 4
Model (B4, C4) > Geometry > Part > Parts

Object Name	*groove180:1*	*plateforvalgtkonsept:1*	*support600:1*
State	Meshed		
Graphics Properties			
Visible	Yes		
Glow	0		
Shininess	1		
Transparency	1		
Specularity	1		
Definition			
Suppressed	No		
ID (Beta)	370	372	449
Stiffness Behavior	Flexible		

Coordinate System	Default Coordinate System		
Reference Temperature	By Environment		
Material			
Assignment	Structural Steel		
Nonlinear Effects	Yes		
Thermal Strain Effects	Yes		
Bounding Box			
Length X	104,49 mm	27,8 mm	155,8 mm
Length Y	1829,3 mm	1630,6 mm	600,46 mm
Length Z	914,5 mm	815,26 mm	300, mm
Properties			
Volume	8,1048e+006 mm³	1,6086e+007 mm³	8,94e+006 mm³
Mass	63,622 kg	126,27 kg	70,179 kg
Centroid X	908,65 mm	908,64 mm	
Centroid Y	-1717,4 mm		
Centroid Z	546,66 mm	358,96 mm	168,46 mm
Moment of Inertia Ip1	2,831e+007 kg·mm²	2,641e+007 kg·mm²	2,9526e+006 kg·mm²
Moment of Inertia Ip2	4,5867e+006 kg·mm²	5,0987e+006 kg·mm²	6,0048e+005 kg·mm²
Moment of Inertia Ip3	2,3839e+007 kg·mm²	2,1319e+007 kg·mm²	2,5892e+006 kg·mm²
Statistics			
Nodes	5096	13000	10816
Elements	2987	8446	8343
Mesh Metric	None		

Coordinate Systems

TABLE 5
Model (B4, C4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 6
Model (B4, C4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 7
Model (B4, C4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	5,1277 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

Mesh

TABLE 8
Model (B4, C4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	13,0 mm
Initial Size Seed	Active Assembly

Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	8,48530 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Dropped
Straight Sided Elements	
Number of Retries	0
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	27664
Elements	19776
Mesh Metric	None

Static Structural (B5)

TABLE 9
Model (B4, C4) > Analysis

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 10
Model (B4, C4) > Static Structural (B5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Restart Analysis	
Restart Type	Program Controlled
Status	Done
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	Yes
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180valgtplatekonsept_files\dp0\SYS\MECH\
Future Analysis	Prestressed analysis
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 11
Model (B4, C4) > Static Structural (B5) > Loads

Object Name	*Cylindrical Support*	*Pressure*	*Frictionless Support*	*Line Pressure*
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face		6 Faces	1 Edge
Definition				
ID (Beta)	42	119	156	592
Type	Cylindrical Support	Pressure	Frictionless Support	Line Pressure
Radial	Fixed			
Axial	Fixed			
Tangential	Free			
Suppressed	No			
Define By		Normal To		Vector
Magnitude		20,1 MPa (ramped)		154,15 N/mm (ramped)
Direction				Defined

FIGURE 1
Model (B4, C4) > Static Structural (B5) > Pressure

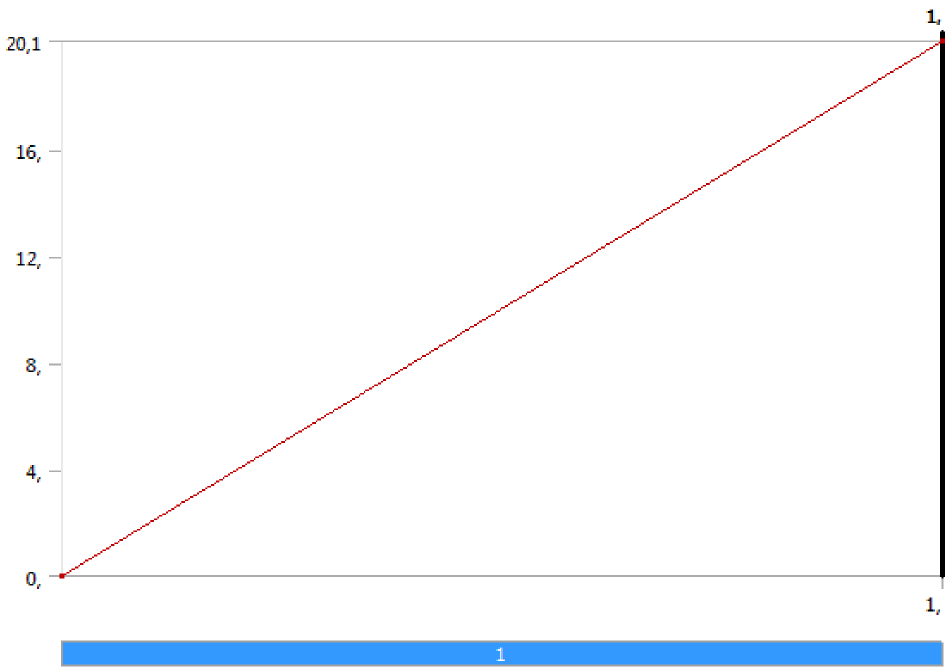
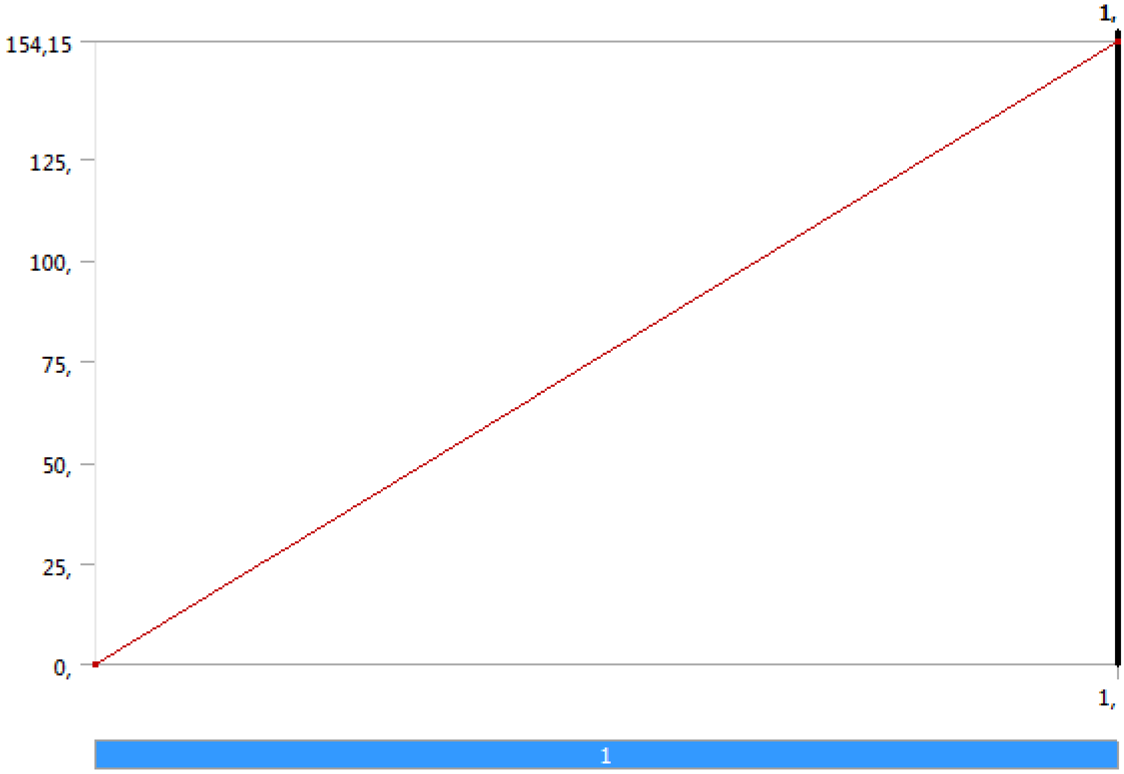


FIGURE 2
Model (B4, C4) > Static Structural (B5) > Line Pressure



Solution (B6)

TABLE 12
Model (B4, C4) > Static Structural (B5) > Solution

Object Name	*Solution (B6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 13
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 14
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Results

Object Name	*Equivalent Stress*	*Maximum Principal Stress*	*Minimum Principal Stress*	*Maximum Principal Stress 2*
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	All Bodies			
Definition				
Type	Equivalent (von-Mises) Stress	Maximum Principal Stress	Minimum Principal Stress	Maximum Principal Stress
By	Time			
Display Time	Last			
Calculate Time History	Yes			
Identifier				
Integration Point Results				
Display Option	Averaged			Unaveraged
Results				
Minimum	0,18031 MPa	-19,218 MPa	-159,98 MPa	-19,449 MPa
Maximum	129,67 MPa	121,74 MPa	14,4 MPa	122,25 MPa
Minimum Occurs On	support600:1	plateforvalgtkonsept:1		
Maximum Occurs On	plateforvalgtkonsept:1		support600:1	plateforvalgtkonsept:1
Information				
Time	1, s			
Load Step	1			
Substep	1			
Iteration Number	1			

TABLE 15
Model (B4, C4) > Static Structural (B5) > Solution (B6) > Probes

Object Name	*Force Reaction*	*Force Reaction 2*
State	Solved	
Definition		
Type	Force Reaction	
Location Method	Boundary Condition	
Boundary Condition	Cylindrical Support	Frictionless Support
Orientation	Global Coordinate System	
Options		
Result Selection	All	
Display Time	End Time	
Results		

X Axis	-14123 N	220,1 N
Y Axis	4,4248e+005 N	7365,2 N
Z Axis	3,8034e+005 N	50865 N
Total	5,8365e+005 N	51396 N
Maximum Value Over Time		
X Axis	-14123 N	220,1 N
Y Axis	4,4248e+005 N	7365,2 N
Z Axis	3,8034e+005 N	50865 N
Total	5,8365e+005 N	51396 N
Minimum Value Over Time		
X Axis	-14123 N	220,1 N
Y Axis	4,4248e+005 N	7365,2 N
Z Axis	3,8034e+005 N	50865 N
Total	5,8365e+005 N	51396 N
Information		
Time	1, s	
Load Step	1	
Substep	1	
Iteration Number	1	

Linear Buckling (C5)

TABLE 16
Model (B4, C4) > Analysis

Object Name	*Linear Buckling (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Linear Buckling
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

TABLE 17
Model (B4, C4) > Linear Buckling (C5) > Initial Condition

Object Name	*Pre-Stress (Static Structural)*
State	Fully Defined
Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time
Contact Status	Use True Status

TABLE 18
Model (B4, C4) > Linear Buckling (C5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Options	
Max Modes to Find	3,
Output Controls	
Calculate Stress	No
Calculate Strain	No
Cache Results in Memory (Beta)	Never
Analysis Data Management	
Solver Files Directory	C:\Users\166864\Documents\masteroppgave\ansys\180valgtplatekonsept_files\dp0\SYS-1\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	nmm

Solution (C6)

TABLE 19
Model (B4, C4) > Linear Buckling (C5) > Solution

Object Name	*Solution (C6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

FIGURE 3
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

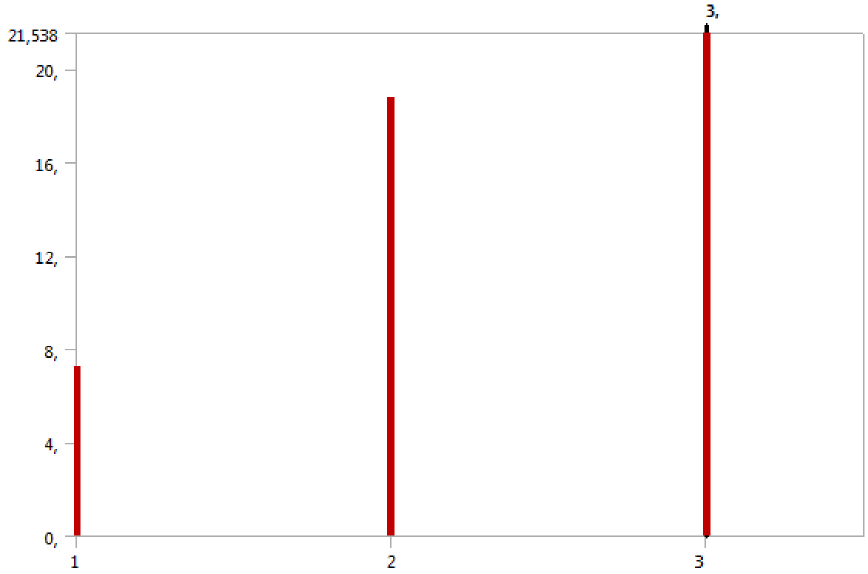


TABLE 20
Model (B4, C4) > Linear Buckling (C5) > Solution (C6)

Mode	Load Multiplier
1,	7,2708
2,	18,734
3,	21,538

TABLE 21
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 22
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Results

Object Name	*Total Deformation*	*Total Deformation 2*	*Total Deformation 3*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation		
Mode	1,	2,	3,
Identifier			
Results			

Load Multiplier	7,2708	18,734	21,538
Minimum	0, mm		
Maximum	1, mm	1,0067 mm	1, mm
Minimum Occurs On	support600:1		
Maximum Occurs On	groove180:1		plateforvalgtkonsept:1

TABLE 23
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation

Mode	Load Multiplier
1,	7,2708
2,	18,734
3,	21,538

TABLE 24
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 2

Mode	Load Multiplier
1,	7,2708
2,	18,734
3,	21,538

TABLE 25
Model (B4, C4) > Linear Buckling (C5) > Solution (C6) > Total Deformation 3

Mode	Load Multiplier
1,	7,2708
2,	18,734
3,	21,538

Material Data

Structural Steel

TABLE 26
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 27
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 28
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa
250

TABLE 29
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
250

TABLE 30
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
460

TABLE 31
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 32
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 33
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 34
Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

TABLE 35
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000