NTNU – Trondheim
Norwegian University of
Science and Technology

Design of Nacelle and Rotor Hub for NOWITECH 10MW Reference Turbine

Sandeep Singh Klair

Mechanical Engineering
Submission date: June 2013
Supervisor: Bjørn Haugen, IPM

Norwegian University of Science and Technology
Department of Engineering Design and Materials

NORGES TEKNISK-
NATURVITENSKAPELIGE UNIVERSITET
INSTITUTT FOR PRODUKTUTVIKLING
OG MATERIALER

MASTEROPPGAVE VÅR 2013
FOR
STUD.TECHN. SANDEEP SINGH KLAIR

DESIGN AV NACELLE OG ROTOR HUB FOR NOWITECH 10 MW REFERANSE TURBIN

Design of Nacelle and Rotor Hub for Nowitech 10 MW Reference Turbine.

Utviklingen av vindturbiner har vært formidabel i de siste år. Det finnes kommersielt tilgjengelig 5 MW turbiner med diameter over 120 meter. Disse er også installert offshore på bunnfaste installasjoner. Morgendagens turbiner er muligens enda større og kan være flytende. Dette er et av temaene for forskningssentret NOWITECH. Det jobbes med å utvikle en referanseturbin på 10 MW som skal benyttes i forskningssammenheng på offshore flytende turbiner.

Oppgaven innebærer mekanisk design av en nacelle og rotor hub for en 10 MW offshore vindturbin .

Oppgaven bearbeides ut fra følgende punkter:

1) Litteratursøk
   a) Gjøre seg kjent med "state of the art" av vind turbiner
   b) Gjøre seg kjent med design av vindturbiner og hvordan nacellen er dimensjonert
   c) Gjøre seg kjent med standarden IEC 61400

2) Software kjennskap
   a) Gjøre seg kjent med DAK-programmet NX
   b) Gjøre kjent med styrkeberegning vha Nastran
   c) Gjøre seg kjent med dynamisk analyse vha FAST og Fedem

3) Gjennomføre mekanisk design og dimensjonering av følgende komponenter:
   a) Bunnplate
   b) Hovedaksling
   c) Hub

4) Finne løsning for hovedlager og yaw-lager

Besvarelsen skal ha med signert oppgavetekst, og redigeres mest mulig som en forskningsrapport med et sammendrag på norsk og engelsk, konklusjon, litteraturliste, innholdsfortegnelse, etc. Ved utarbeidelse av teksten skal kandidaten legge vekt på å gjøre teksten oversiktlig og velskrevet. Med henblikk på lesning av besvarelsen er det viktig at de nødvendige henvisninger for korresponderende steder i tekst, tabeller og figurer anføres på begge steder. Ved bedømmelse legges det stor vekt på at resultater er grundig bearbeidet, at de oppstilles tabellarisk og/eller grafisk på en oversiktlig måte og diskuteres utførlig.

Senest 3 uker etter oppgavestart skal et A3 ark som illustrerer arbeidet leveres inn. En mal for dette arket finnes på instituttets hjemmeside under menyen masteroppgave (http://www.ntnu.no/ipm/masteroppgave). Arket skal også oppdateres en uke før innlevering av masteroppgaven.

Besvarelsen skal leveres i elektronisk format via DAIM, NTNUs system for Digital arkivering og innlevering av masteroppgaver.

Kontaktperson: Karl Merz, Nowitech

Torgeir Welo
Instituttleder

NTNU
Norges teknisk-naturvitenskapelige universitet
Institutt for produktutvikling og materialer

Bjørn Haugen
Faglærer

Abstract

Wind turbine development has been formidable in the last years. One of NOWITECH's goals are to do research on large offshore wind turbines. A bed plate and hub for the NOWITECH 10MW reference turbine has been analyzed, according to IEC 61400. The work is based on Ebbe Smith's thesis about bed plate design, and Mohammad Akram Khan's thesis about rotor shaft and rotor hub design.

A completely new bed plate has been designed, and analyzed with more correct boundary conditions including a yaw bearing with contact surfaces and bolt connections between flanges. The hub received minor modifications, and is analyzed with a bolt connection and pitch bearings.

The mass of the new bed plate is 99.6 tons, and a has peak stress of 217.5MPa which is justified in discussion. The peak stress in the hub is 126MPa.

Sammendrag

Utviklingen av vindturbiner har vært formidabel i de siste år. Et av NOWITECHs mål er å gjøre forskning på større vindturbiner som skal plasseres offshore. Det har blitt utført analyse og design av bunnplate og hub, for NOWITECHs 10MW referanse turbin. Dette ifølge IEC 61400 standarden. Arbeidet er basert på Ebbe Smith's oppgave om bunnplate design, og Mohammad Akram Khan's oppgave om hub og aksel design.

En helt ny bunnplate har blitt designet og anlysert med mer korrekte grensebetingelser som inkluderer yaw lager med kontaktoverflater samt boltede flensforbindelser. Huben har fått mindre modifikasjoner og har også blitt anaysert med en boltet forbindelse og pitchlager.

Massen på den nye bunnplaten er 99.6 tonn og har en maksimal spenning på 217.5MPa, som er forsvart i diskusjonskapittelet. Huben har en maksimal spenning på 126MPa.

Contents

Acknowledgment

This thesis is the final work needed in order to obtain a masters degree in Product Development and Materials Engineering from Norwegian University of Science and Technology (NTNU).

Previous work done on the bed plate and the hub lacked correct boundary conditions, which meant that their results were not accurate. This lead to occurrence of this masters thesis. It contains more accurate simulations with better boundary conditions which is clearly illustrated with figures.

When I was looking a for theme for my master, I wanted to write and do something that had a complex analysis part. At the beginning of my work I thought that I had chosen something that wasn't so complex and challenging, but later the modeling and analysis got quite challenging which I really enjoyed. I've learned quite a lot about contact analysis, modeling, and mesh techniques which is really going to help me with my future analyzes.

I would like to thank my office colleagues Simen Riser, Steinar Rudi and Bernardo Figueiredo for keeping a good atmosphere and helping me with various types of issues regarding my work. I would also like to thank my fellow student Magnus Skogsfjord for helping me with modeling issues and taught me some modeling techniques.

The last thanks goes to my Supervisor, Professor Bjørn Haugen. I thank him for being patient and for his supervision. He was available all the time to provide help.

Trondheim,June 10, 2013

Sandeep S. Klair

Sandeep Singh Klair

1 Introduction

1.1 Background

This thesis is about designing rotor hub and bed plate for the Norwegian Research Center for Offshore Wind Technology (NOWITECH) turbine with respect to structural stress. Two theses have already been written on this subject, but with different focuses. The first thesis focused on design of the nacelle bed plate and main bearing selection for the rotor shaft [5], and the second thesis on the hub and rotor shaft design [6]. The main scope in this thesis is to combine the previous work into one single assembly and see how they work together to create realistic stress states. Previous simulations were done with artificially stiff boundary conditions which gave uncertain component stresses. With this in mind, a part of the simulation will go to create better component boundary conditions which will give better stress results. More specifically, bearing areas and bolt connections will also be modeled and simulated using contact analysis.

Wind turbines are machines that convert kinetic energy in a wind stream into mechanical rotational energy[12]. Their energy has been used for a over a millennium to power various mechanisms. In their early years they powered water pumps and grain mills. As more efficient machines were invented, they took over the work, and windmills werent used so much. Nowadays, most of the world is relying on non-renewable fossil fuel which is known to be very polluting, and in that aspect, scientist has been focusing on environmental friendly and renewable energy resources like wind energy in order to reduce global warming, and search for new ways to be more energy efficient.

Figure 1: Offshore Windmills [12]

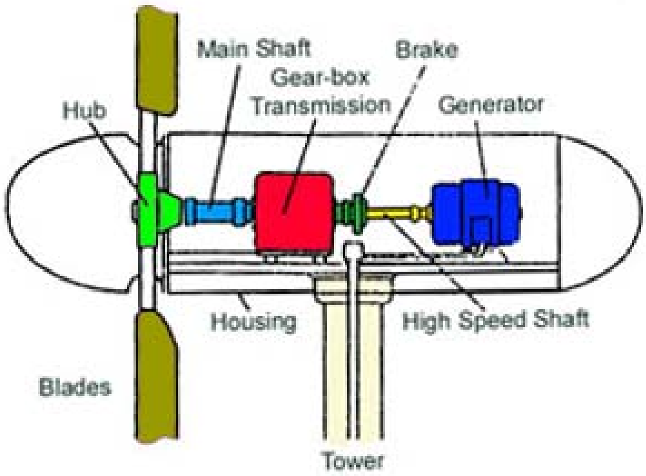

Figure 2: Mechanical principle of a wind turbine [11]

1.2 Goal

The goal of this thesis is to dimension the nacelle/bed plate which transfers loads to the tower, and to dimension the hub which transfers loads to the bed plate. The focus is to create a simulation that contains realistic boundary conditions so more accurate stresses are unveiled. Also, the mass should be kept as low as possible.

1.3 Scope of Work

First, a general introduction will be given before all the theory applied in the thesis will be presented. Then the method of how the bed plate and hub was designed and simulated is explained thoroughly before results and the discussion around it is presented.

2 NOWITECH turbine

The NOWITECH turbine project is a joint project in cooperation with NTNU. The aim of the project is to make a complete model so other firms can compare their results with the projects, in order to make it easier for them to make their own turbines. One purpose behind this would be to promote wind turbine technology and take a step closer to pollution free renewable energy resources [2].

The NOWITECH turbine is a virtual model of a three bladed, horizontally shafted, upwind wind turbine rated at 10MW power. It has been chosen to use a direct drive layout for this turbine (fig. 4). In a direct drive system, the rotor hub is connected directly to the generator through the rotor shaft. In more traditional layouts, the rotor hub must first transfer power to a gearbox through the rotor shaft, and from the gearbox to the generator via the output shaft. With the gearbox removed, the reliability and efficiency is increased. A more complete comparison is shown in figure 3 and more detailed criterias are shown in figure 5.

Advantages of DD	Disadvantages of DD
• No gearbox and related wear and tear on mechanical components	• Larger diameter of generator/nacelle complicates transportation and Installation
• Simpler turbine with fewer parts	• Higher top-mass weight *)
• Higher electrical (PM) and overall drive train efficiency, producing higher energy	• Expensive PM material- potentially uncertain security of
• Lower maintenace and greater reliability with less downtime	• More complex assembly of generator using PMs
• Improved thermal charateristics due to absence of field losses	• Demagnetisation of PM at high temperature
• Full power conversion imporves the turbine's grid compatibility	• More advanced cooling system required
	• Full power rather than partial power conversion makes turbine expensive
Comments: *This comparison is based on a traditional drive train comprising a doubly feed induction generator and 3-4 stage gearbox* **) The general trend towards higher top-mass weight for direct drive turbines seems to have been broken by the new DD turbine SWT 3.0 from Siemens Wind Power* *When comparing the PMG solution to the Enercon DD design, the latter is heavier and the absence of a permanent magnet creates excitation loss when magnetising the coils, but the concept has an impressive track record from more than 20GW of capacity in operation*	

Source: BTM Consult - A Part of Navigant Consulting - March 2011

Figure 3: Advantages and disadvantages of a directdrive turbine [5]

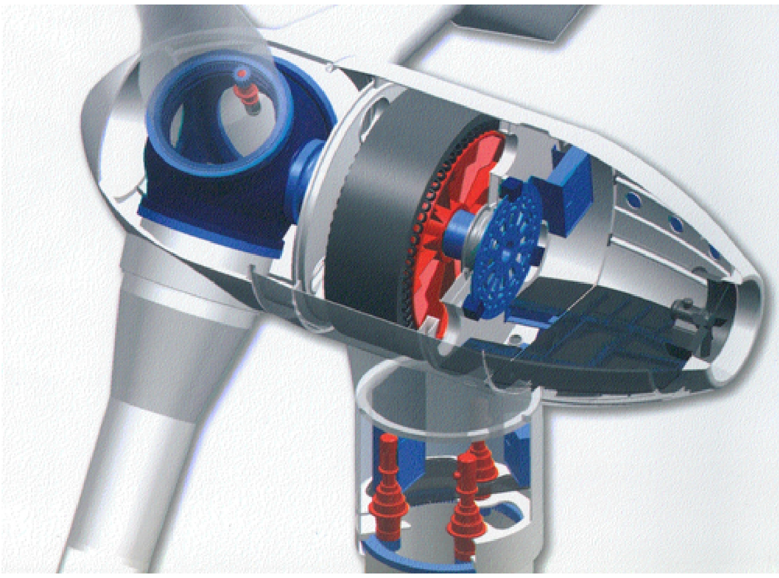

Figure 4: A directdrive turbine with a cut to show the mechanical principle [1]

	Symbol		Unit
Extreme wave height		30	m
Maximum sea current velocity		1.2	m/s
Water depth		60	m
Rated power output	P	10	MW
Electrical Frequency	f_n	50	Hz
Weibull parameter	A	11.75	-
Weibull parameter	k	2.04	-
Density of air	ρ_{air}	1.225	kg/m^3
Density of seawater	ρ_{sea}	1025	kg/m^3
Water salinity		3.5	%
Water temperature (min/max)		0 / 22	°C
IEC turbulence parameter	I_{ref}	0.12	-
IEC reference wind speed	U_{ref}	50	m/s
Average wind speed at hub height	U_{ave}	10.4	m/s
IEC wind shear exponent	α	0.14	-
Rotor diameter	D	141	m
Number of blades		3	-
Hub diameter	D_{Hub}	4.94	m
Length between blade tip and the tower	L	13	m
Maximum rotor speed	n	13.54	rpm
Maximum allowed tip speed		100	m/s
Extreme wind speed, 50 years	U_{e50}	70	m/s
Extreme wind speed, 1 year	U_{e1}	56	m/s
Design wind speed	U_{Design}	13	m/s
Rated wind speed	U_{Rated}	~ 15	m/s
Cut-in wind speed	$U_{Cut\text{-}in}$	4	m/s
Cut-out wind speed	$U_{Cut\text{-}out}$	30	m/s
Optimum tip speed ratio	TSR_{Opt}	7.8	-
Blade pre-curvature		3.06	m
Turbine blade coning angle		2	degrees
Main shaft tilt angle		5	degrees

Figure 5: Criterias of the NOWITECH turbine [5]

2.1 Requirements

The requirements for the bed plate structure are to keep it within the given geometrical constraints, and dimension it according to IEC 61400 standard with a maximum stress of 200MPa. Secondary requirement is to keep the structure mass as low as possible. The maximum mass wanted per part is 50 tonnes or less. Geometrical constrains are shown in the figure 6.

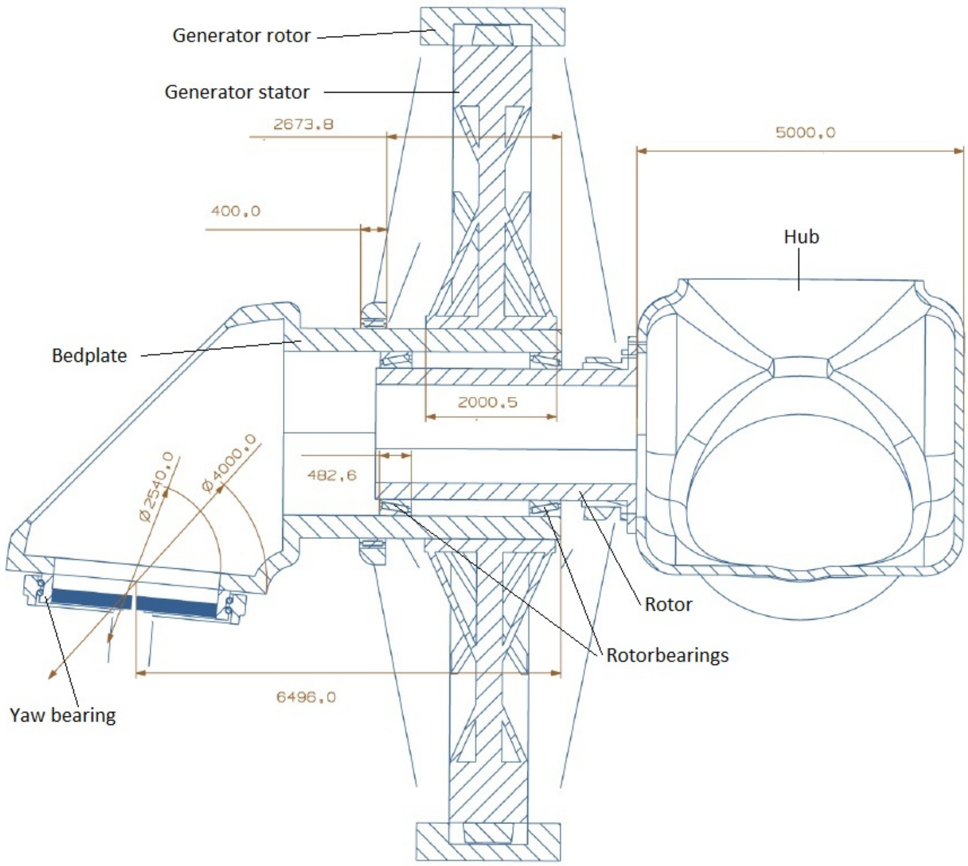

Figure 6: The geometrical constraints for the bed plate, rotor and rotor hub [5]

Further, the bed plate must be strong enough to fully withstand the forces from the generator, rotor with hub and axle, and the blades. The bed plate must then transfer all of the loads to the tower through a yaw bearing.
The requirements for the hub is to keep the maximum stresses below 166MPa, transfer loads to the bedplate via the rotorshaft, and at the same time keep the mass as low as possible.

3 Theory

In this section theory applied to the thesis will be described.

3.1 Wind energy convertors

In general, the only purpose of wind turbines are to convert kinetic wind energy into mechanical energy. And from there we can decide where to put the energy. This implies that the primary part of a wind turbine is the rotor disc and its properties. How a rotor disc extracts energy from a wind stream was first described by Albert Betz [3]. He first describes the power extracted from a wind stream through energy conservation, and secondly power extracted through momentum conservation.

Through energy conservation he describes the power (P) extracted from a airstream as,

$$P = \frac{1}{2}\dot{m}(v_1^2 - v_2^2) \tag{1}$$

where $\dot{m}$ is the mass flow and v_1 and v_2 are velocity in and out of a rotor disc, respectively.

And through momentum conservation,

$$P = \dot{m}(v_1 - v_2)\acute{v} \tag{2}$$

where $\acute{v}$ is the velocity *through* the rotor disc.
Rearranging equation (1) and (2) with respect to $\acute{v}$ gives,

$$\acute{v} = \frac{1}{2}(v_1 + v_2) \tag{3}$$

Combining equation (3) with the mass flow gives,

$$P = \frac{1}{4}\rho A(v_1^2 - v_2^2)(v_1 + v_2) \tag{4}$$

By making a ratio of this and a free reference airstream, it gives us the ideal power coefficient. The maximum value of this is $cp = \frac{16}{27} = 0.593$ which is called the Betz factor [3]. This is the highest efficiency a wind turbine can have. In other words, only about 60 percent of the energy in a wind stream can be extracted.

From this we get the velocity relation,

$$v_2 = \frac{1}{3}v_1 \tag{5}$$

Which means that in order to have maximum energy extraction, the exit velocity should be one third of the incoming velocity. [3]

3.2 Loads

At first glance loads appearing on a wind turbine are quite simple and obvious. An airstream pushing on a rotor disc, which in turn generates torque transferred to a generator. A closer look reveals more loads that first guessed. Along with wind loads, the complete structure is also subjected to inertial, gravitational and turbulent wind/gust loads. One of the most common simplification done is to think that the wind is a constant airstream, all over the rotor disc. The wind is in fact highly non-uniform and its direction will change over time. This creates additional forces and moments around the rotor hub which puts the hub and bed plate into complex load cases. These load cases creates stresses in different areas, and with them, the highest stress needs to be identified. The blades will in fact experience forces in every direction and moments around every axis. Some of these loads are steady, some are cyclic and some are completely non-cyclic. This makes a wind turbine a highly dynamic machine and subjected to fatigue.

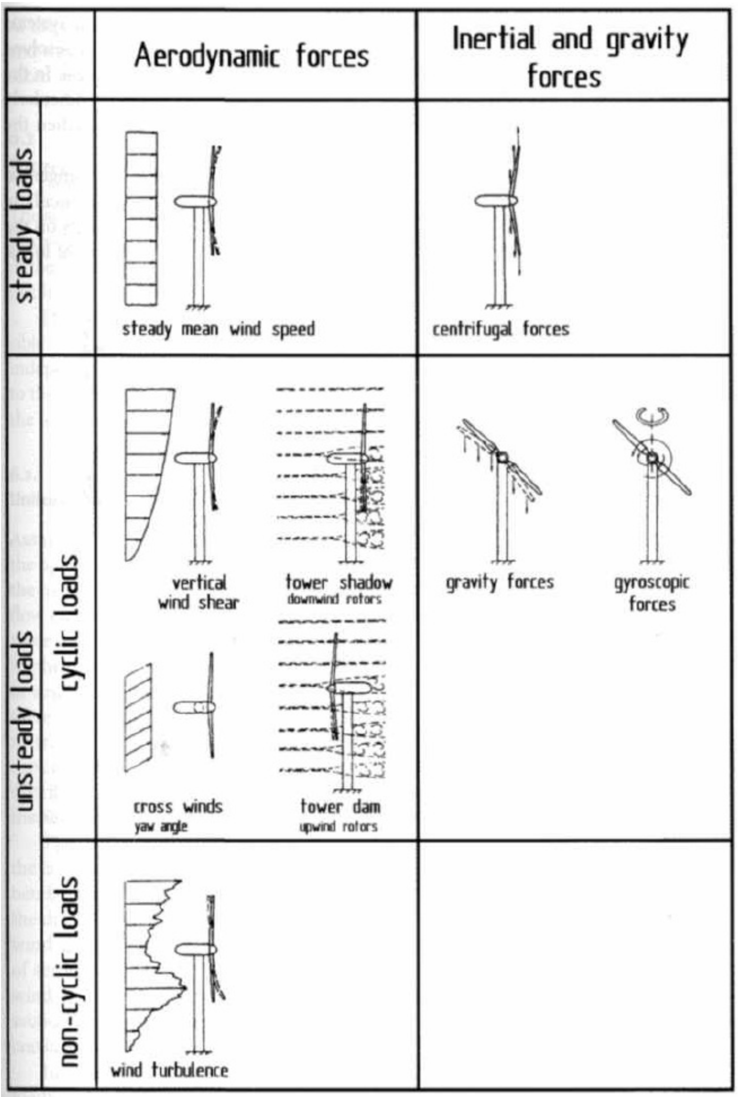

Figure 7: Loads acting on a wind turbine [3]

The Wind turbine has to be designed according to the IEC 61400 standard. The standard specifies several load cases that a wind turbine can experience. A load case contains different load conditions like wind speed, wind model, and different operational status of the turbine like idle, standby or power production. In the early stages of development its difficult to determine whether its fatigue or plastic collapse which gives the dimensioning load. Therefore, a number of different load cases are made to make it possible to identify which condition that gives the highest stress. Load case used in this thesis is DLC (Design Load Case) 1.3ETM (Extreme Turbulence Model) which previously gave the highest peak stress [5] In order to understand how the loads of DLC 1.3 ETM works, a reference coordinate system is presented by figure 8.

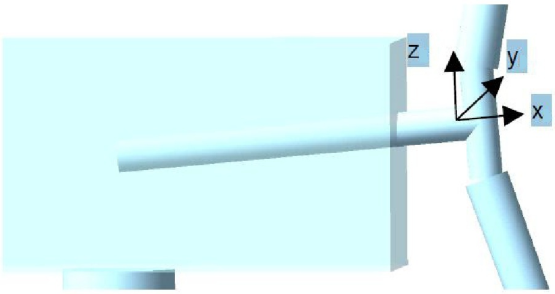

Figure 8: The reference coordinate system MAK

A simple list of what the specific DLC 1.3 is about, is presented by figure 9.

1.3- Extreme Turbulence Model	Wind speed = 13m/s	Fx [kN]	1746	
		Fy [kN]	132	
		Fz [kN]	-1410	-1806
		Mx [kNm]	9129	
		My [kNm]	14550	-10440
		Mz [kNm]	15330	
	Ultimate loads	Fx [kN]	2176	
		Fy [kN]	340	
		Fz [kN]	-1542	-1919
		Mx [kNm]	11690	
		My [kNm]	28660	-16050
		Mz [kNm]	22480	

Figure 9: Design Load Case 1.3 Extreme Turbulence Model [5]

Forces and moments listed in figure 9 are calculated with a software called FAST [9], which is a free aero elastic software which simulates different operating conditions for wind turbines. It is important to understand that the calculated results are working from one single point. More specifically in the coordinate system's origin.

3.3 Dimensioning

To ensure that a part will not fail it must contain enough material to cope with the loads applied. But at the same time it must be as material efficient as possible. This is done through dimensioning where stresses are calculated and evaluated. The dimensioning requirement is expressed [5] :

$$\sigma_D \leq \sigma_R \tag{6}$$

Where σ_D is the design stress and σ_R is the maximum resistance stress.
Because the material's strength is based on statistics, the resistance stress must be reduced by a safety factor. A safety factor also applies to design stress, but instead used increase its numerical value. This gives us,

$$\sigma_D \gamma_s \leq \sigma_R \frac{1}{\gamma_m} \tag{7}$$

Where γ_s is the stress safety factor and γ_m is the material safety factor.
IEC 61400-3 specifies a load safety factor of 1.35 [4]
When a component changes cross section e.g. changing diameter of a rod along its length it will experience multiaxial stress regardless of its loading type. The Von Mises Yield criterion is a method for adding these stresses. The criterion adds up all the stresses and turns them into a single, one dimensional stress. The Von Mises stress is then compared to the maximum allowable stress (resistance stress modified with safety factor).

$$\sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\tau_{xy}^2} \leq \sigma_{Rmod} \tag{8}$$

3.3.1 Stress concentration

A very important thing to consider is stress concentration. A stress concentration is defined as a stress that is higher than the average stress in a component. When there is a change in geometry a stress concentration will occur. A good example is a hole in a plate (which is in fact a change in geometry) where the stress concentration is 3. If we look at the stress as lines in the plate, its easy to understand that the concentration of stress lines will be higher at the edges of the hole. This is easily illustrated by figure 10.

The area in, and around a stress concentration, will contain a multiaxial stress state. Because of this, the exact stress state in these areas is sometimes not easy to tell. This is even more important in fatigue analyzes where it can lead to direct failure of the structure without a warning. In there cases different approaches than Von Mises must be used. However, in simpler fatigue cases this is not a problem.

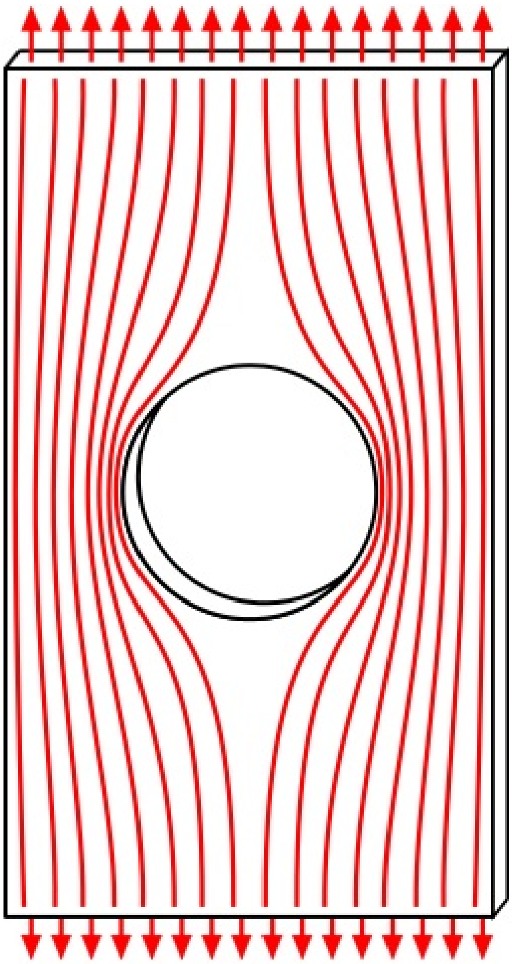

Figure 10: Stresslines in a plate with a hole [8]

4 Method

4.1 The Model

4.1.1 Introduction

The objective of this thesis is to combine Ebbe Smiths (ES) [5] bed plate model with Muhammad Akram Khan's (MAK) [6] rotor model. By the authors personal point of view, the bed plate model was not satisfactory. The main reasons behind this was complex structure surfaces and too flexible transfer points for the yaw bearing. Because of this, the author decided to make a complete new bed plate model with a better transfer area and simpler geometries/surfaces. This has naturally lead to a lot of time going to a part which was not directly the scope of this thesis. The model from MAK also had a shortcoming. The length of the shaft didn't match ES's bearing distance. But with ES bearing distance, the hub did not match. So the solution was to increase the length MAK's shaft to 4250mm between the bearings. With a modeling/simulation task like this, its obvious that its going to take a lot of iterations before one is satisfied with the result. It would been outside the scope of this thesis to describe all the iteration steps done. So instead of detailed steps, a small summary is presented.

4.1.2 Previous Models

The first primitive model was created with two tubes with slightly different diameters. A large flange was then created in the connection points of the tubes. The purpose of the flange was first of all to add material where the expected stresses were to be highest. The flange was also used as a connection point for the eccentricity between the tubes, which created an additional bending stress in the structure. The flange was also placed in such a position that it tangent the rear rotor bearing support. The rear rotor bearing support had a cross section of a I-beam around the rotational axis. The Front rotor bearing support was just the same as ES's. The first models did not contain an inspection hole, but was later added when the design was satisfactory. This hole was placed on the top of the structure at first. It was later decided to move it to the rearmost part. It was also decided not to add a yaw bearing in the first models to keep the simulation time down, so the design iteration could be more productive. Models are shown in figure 11 and 12.

After a couple of analyzes it was realized that the applied loads were without a safety factor. This in turn meant that the structure wasn't stiff enough between the flange and the rear bearing support. With the safety factor applied the stresses were now over 260MPa. In order to cope with these stresses, straight and rectangular bar stiffeners were used. It was decided to use 4 of them to stiffen the transition part where the front tube meets the flange. These four stiffeners work together to make the whole bed plate more stiff in bending. The two other stiffeners where used to reduce bending in the flange, as the flange tended to bend because of different tube diameters.

After a design review, it was decided to redesign the bed plate so the tubes connected concentrically instead of eccentrically. This was first of all chosen to see if it was pos-

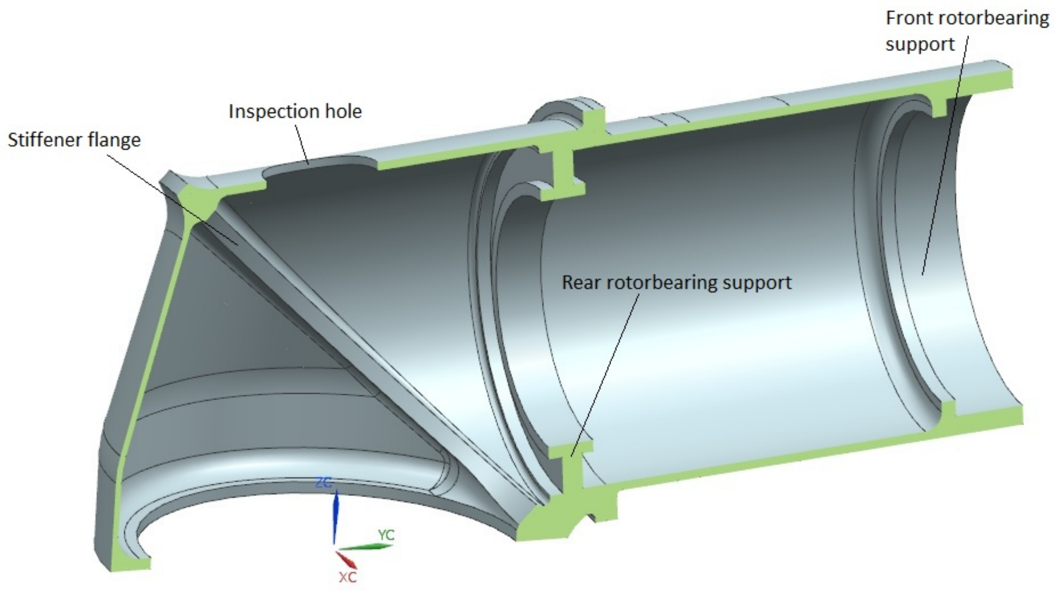

Figure 11: One of earlier models

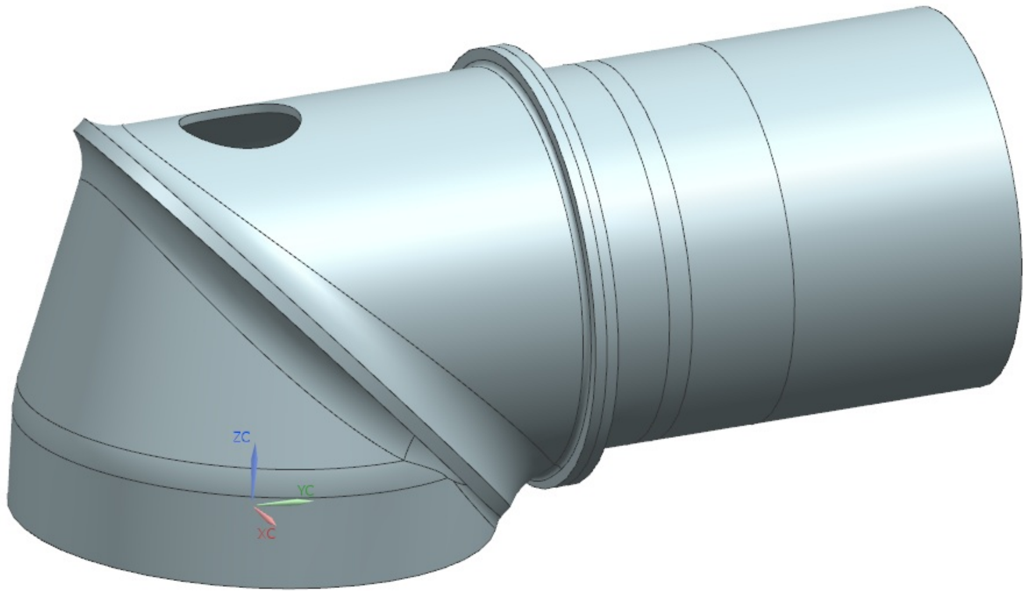

Figure 12: One of the earlier models, closed

sible to remove the internal stiffeners. Still, a centric connection between the tubes weren't enough. This lead to adding stiffeners anyway.
The shaft has just the same geometry as originally by MAK, but with a longer total length. MAK's design didn't match ES's bearing spacing, so it had to be lengthened a bit. But still a little shorter that ES bearing distance, in order to make the hub fit. To simulate bearings, two large cylinders are used. This is done in order to use contact analysis so the total analysis will be more accurate.

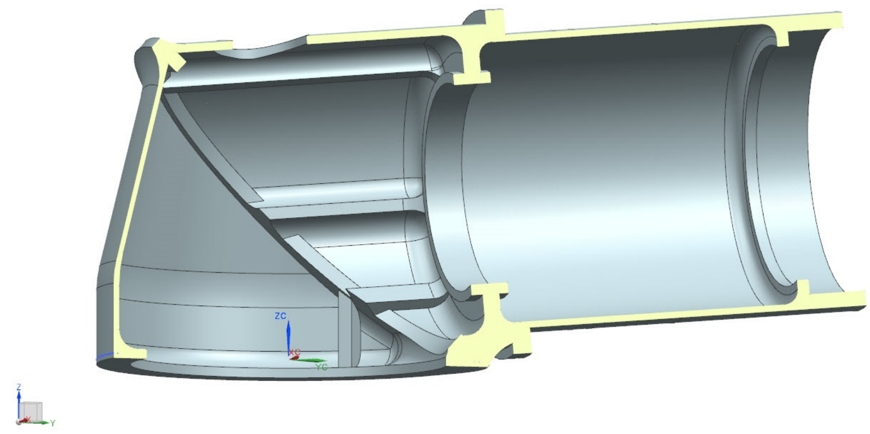

Figure 13: Structure with stiffeners

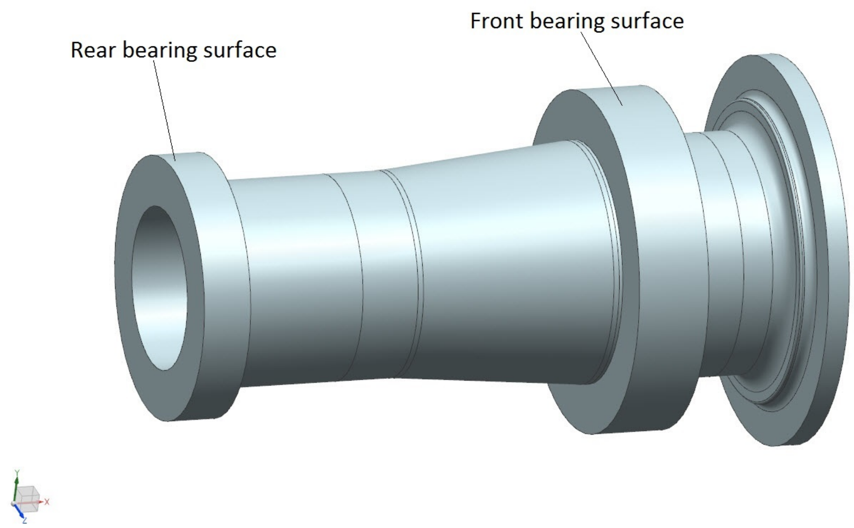

Figure 14: The modified shaft

4.1.3 Final Model

The final bed plate model consist of two parts. The first part to be described is the tower part of the bed plate, then comes the nose part.

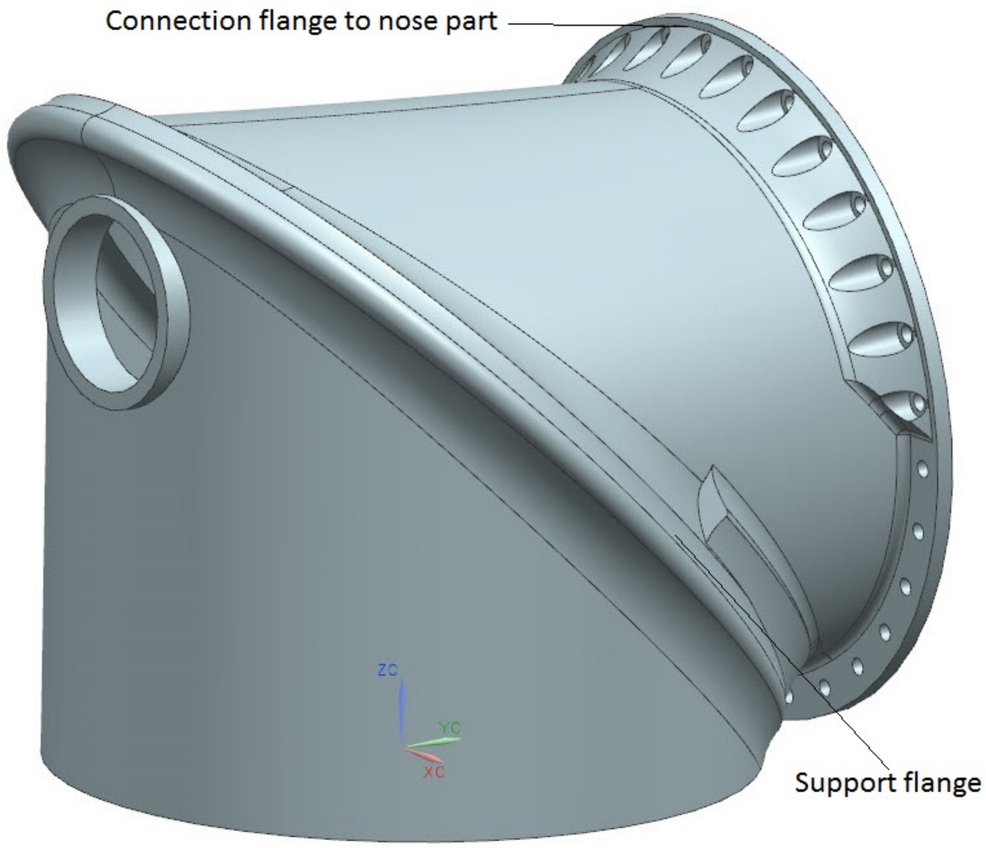

Figure 15: The tower part of the bed plate

The tower part (figure 15 and 16) is the structure that has the highest loads, and have the most complex shape. This translates into high stresses, and its naturally the part where the stresses are highest. It consists mainly of two pipelike structures. The part connected to the yaw bearing area is a simple pipe with a outer diameter of four meters, and has a cut that is approx. 45 degrees to the X-Y plane. From this cut, a pipe shaped structure is swept into the nose connection flange, with a diameter of 3130mm. In other words, it goes from a 45 degree, four meter pipe, into a straight 3130mm pipe. This eliminates the eccentricities that were a problem previously, because the pipes now have the same diameter at their meeting point.

A large support flange is placed between the pipe connection points. The purpose of this flange is to stiffen the bed plate in the pipe meeting area and absorb stresses. The flange has a complex shape in order to make the geometric transitions as smooth as possible. The basic cross section is rectangular with a height of 200mm and a width of

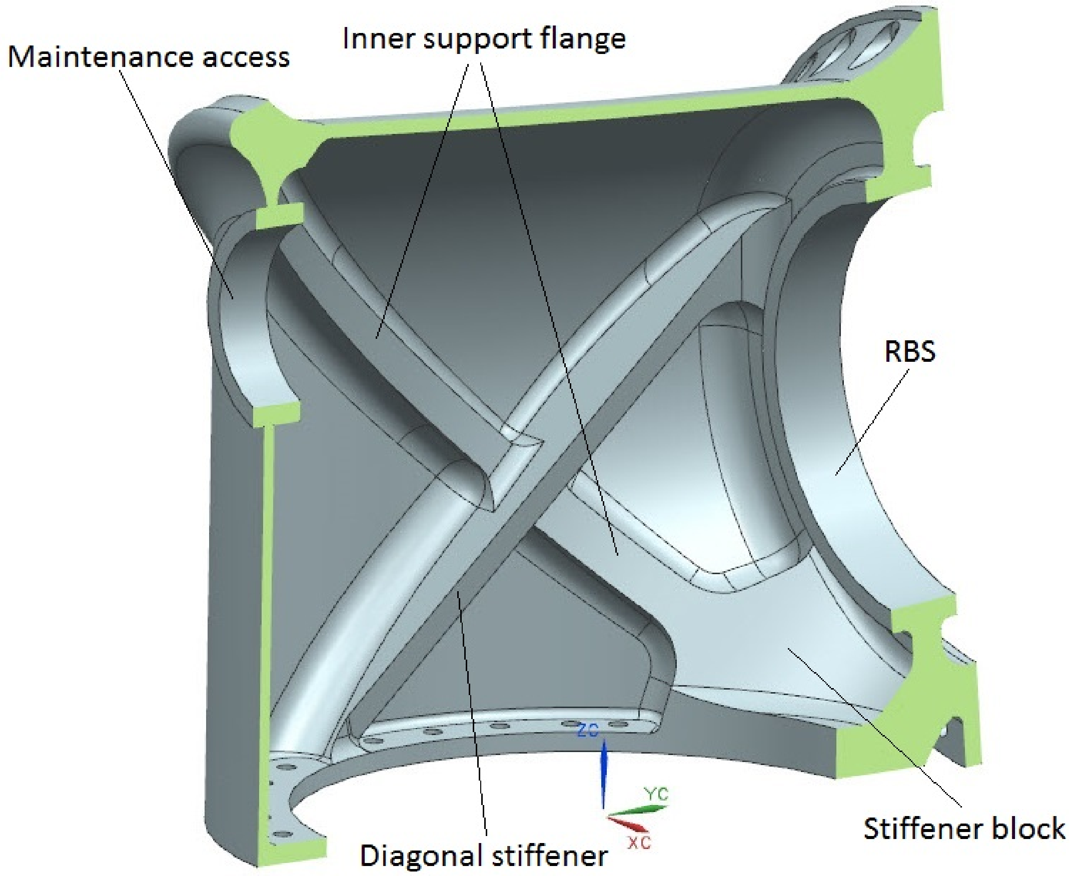

Figure 16: The tower part of the bed plate, with a through cut

400mm. On this rectangle, one corner is expanded into a 75 percent circle with diameter of 200mm. The circle expansion is done to make surface transition smoother.

The front end contains the rear bearing support (RBS) and a another flange to mount it to the nose section. The RBS still has a cross section of an I-beam like the previous models. Right after the RBS, a large stiffener "block" is added to stiffen up this area. Further, this then connects to the inside of the stiffening flange. These integral stiffeners are both thicker and wider than the original flange in order to further stiffen the structure. On the sides of the inside, large diagonal stiffeners are placed to stiffen the structure in both torsional load situations and in bending the structure up or down. These were a key part in increasing the whole stiffness of the tower part.

In the rearmost end of the structure a maintenance access is added. As mentioned in the theory part, the stresses near a hole can be around 3 times the average stress. Because of this, relatively large flanges are placed around the hole. The flange connecting the tower part to the nose part is reinforced on the upper side (fig 18). This flange was originally 100mm in thickness and had a 50mm radius at its connection point to the tower part. In this configuration stresses at the radius reached 300MPa. After the reinforcement, stresses went down to somewhere between 100-110MPa. The bottom obviously contains the flange connection to the yaw bearing. Even with stiffeners placed in right places, the stress magnitude requires increased wall thickness in certain areas (fig 17) near the RBS.

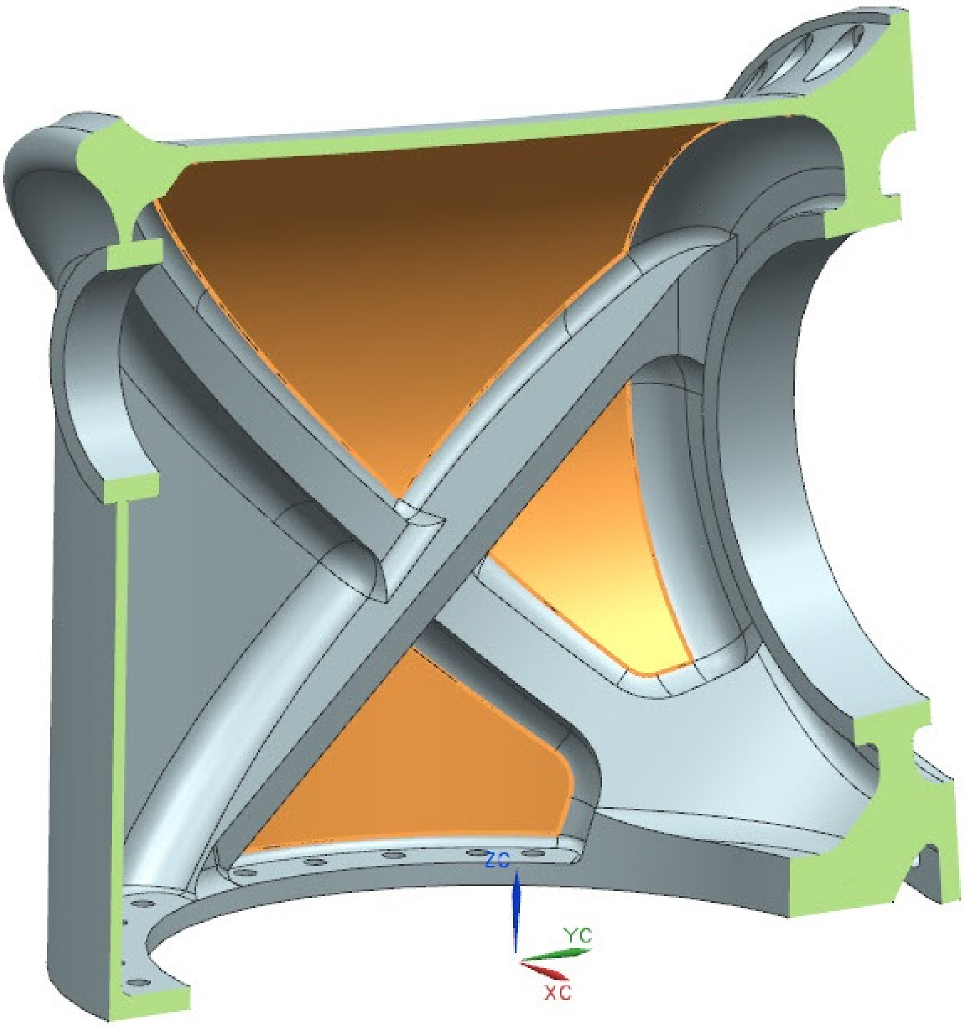

Figure 17: Highlighted areas have a thickness increase of 20mm

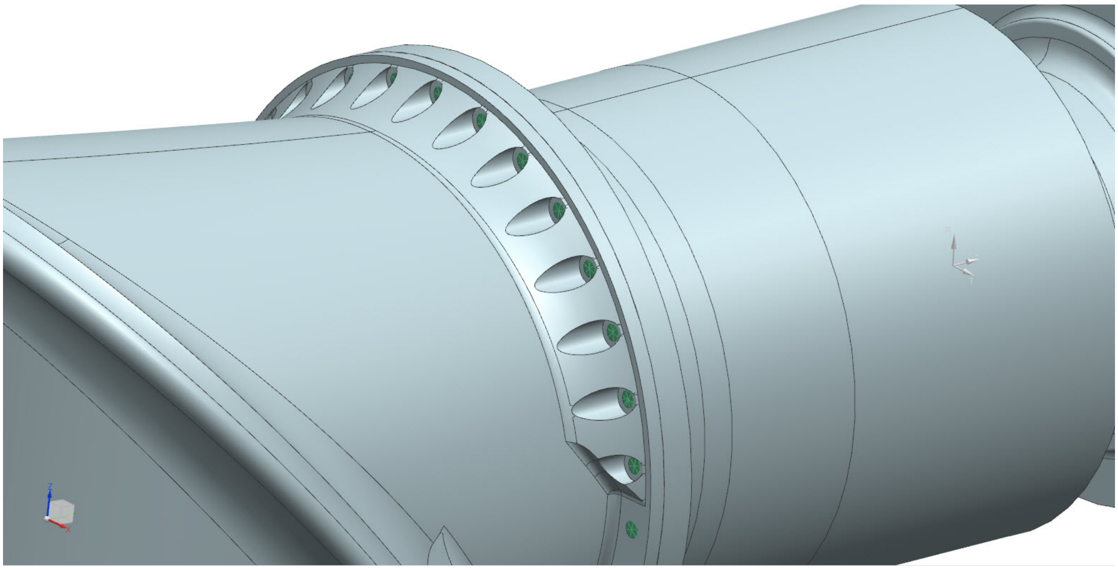

Figure 18: The reinforced flange

Estimated mass of the tower part is 59 tonnes, which is 9 tonnes over the maximum target mass. This part can further be mass optimized in the RBS, the stiffener block, support flange and diagonal stiffeners using geometry optimization. How big the mass advantage can be has not been estimated. The total mass is now 99.6 tons, which is 17.2 tons higher than ES's design. This is mainly due to added stiffness required when the whole structure is placed on a bearing.

The nose part is much simpler in construction. It consists of a simple pipe with a outer diameter of 3130mm (fig. 19). The foremost end contains the Front rotor bearing support (FBS). The dimensions used here are the same as ES used in his analysis. The rearmost end contains a thick mounting flange which mounts against the tower part. This flange is modeled to be 200mm thick in order to avoid bending stresses in the transition zone between the flange and the pure pipe section. Another critical part is the radius that is blending the flange to the pipe. This radius is 190mm. Right under this radius on the inside, a stiffener ring is added to reduce stresses in the radius (fig. 20).

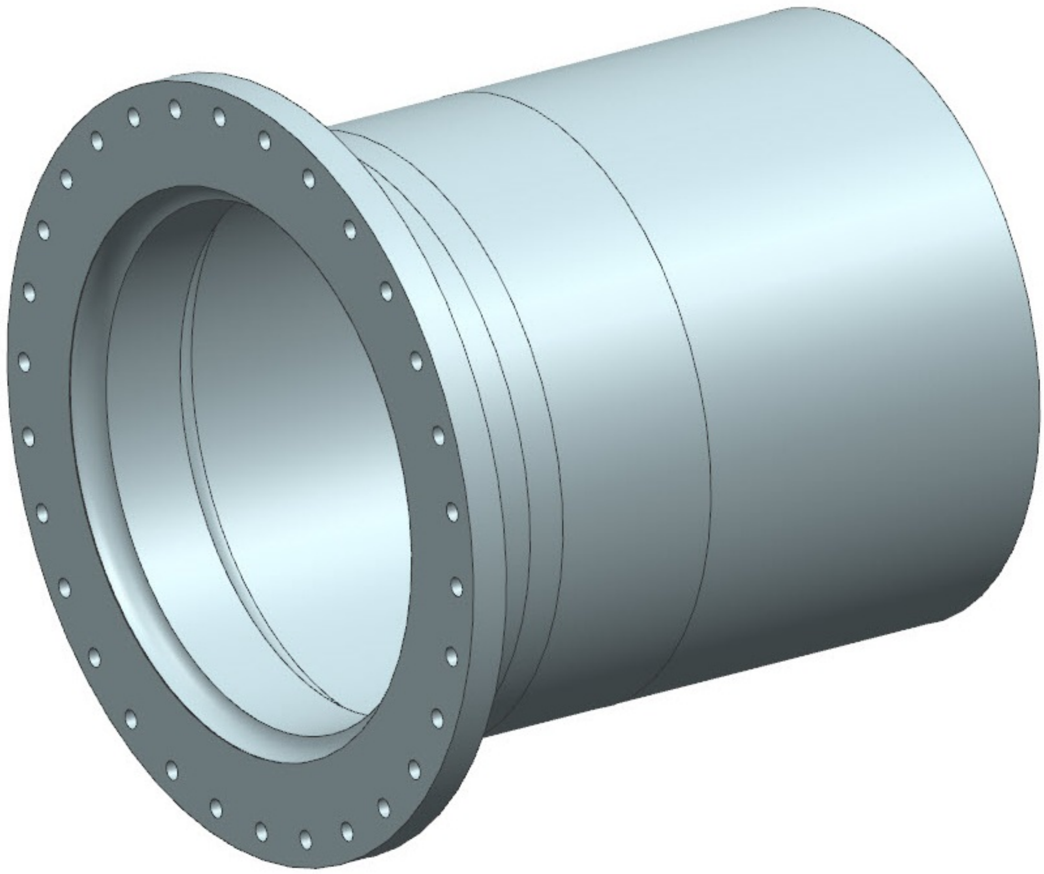

Figure 19: The nose part of the bed plate

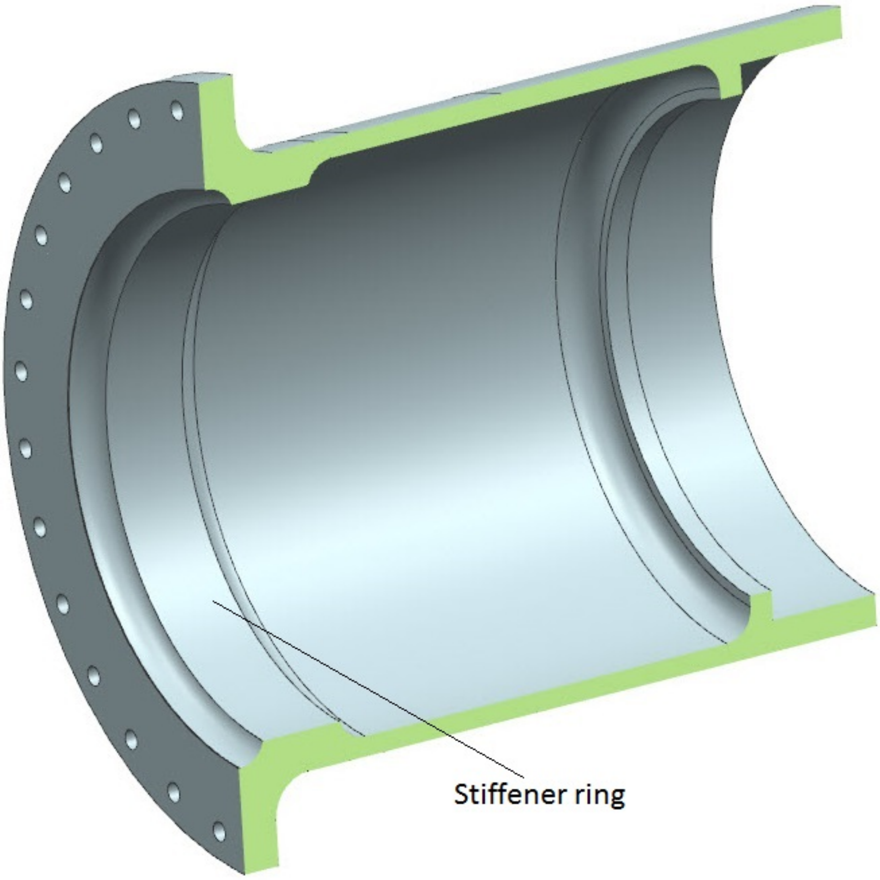

Figure 20: The nose part of the bed plate cut

4.1.4 Yaw Bearing

In order to have realistic stress states, the whole bed plate must be connected to a bearing in order to eliminate the unrealistic infinite stiffness given by a fixed constraint.
The yaw bearing (fig. 21) has been selected to be a double row angular contact ball bearing [5]. The load diameter of this bearing is 4 meters which means that the loads from the bed plate is transferred directly to the rolling elements in the bearing, without an eccentricity. The bearing design is based on the yaw bearing from SKF used in ES's model. The rolling elements has been simplified from spheres to rings in order to simplify simulation. The diameter of the spheres/rings were calculated simply by linear scaling:

$$bearing\ ratio = \frac{ball\ diameter\ of\ the\ bearing}{mean\ diameter} = br = \frac{bd}{D} \tag{9}$$

Which gives,

$$br = \frac{100}{2900} = 0.034 \tag{10}$$

Where Br is a ratio between sphere diameter and mean diameter.
By using these values,

$$0.034 = \frac{sphere_1}{4000} \tag{11}$$

The calculation gives us,

$$sphere_1 = 137.93 \tag{12}$$

Which is rounded up to 140mm.
Other dimensions was further scaled from this value.

The inner bearing ring also contains internal stiffeners in order to make it stiffer in the radial directions (fig. 22). This stiffening action contributes to lower stresses in the bedplate. In order to constrain the whole structure to freely rotate around the global Z-axis, four simple grooves were made and spread around the inside of the inner ring. The outer bearing ring is connected to a simple, but massive support cylinder which simulates the tower. This cylinder also contains four "constraining keys" which fits inside the grooves in the inner ring. These two together makes a lock in yaw direction.

4.1.5 Pitch Bearings

The pitch bearings are identical to the yaw bearing in construction, but naturally have different dimensions to match the hub. It has a mean diameter of 3670mm and its sphere/ring size is 130mm. Their rotational constraint is modeled with a hole in the inner bearing ring and a simple rod. The rod goes through this hole and is fixed in the hub's pitch motor mounts, and prevents the bearing from rotating.

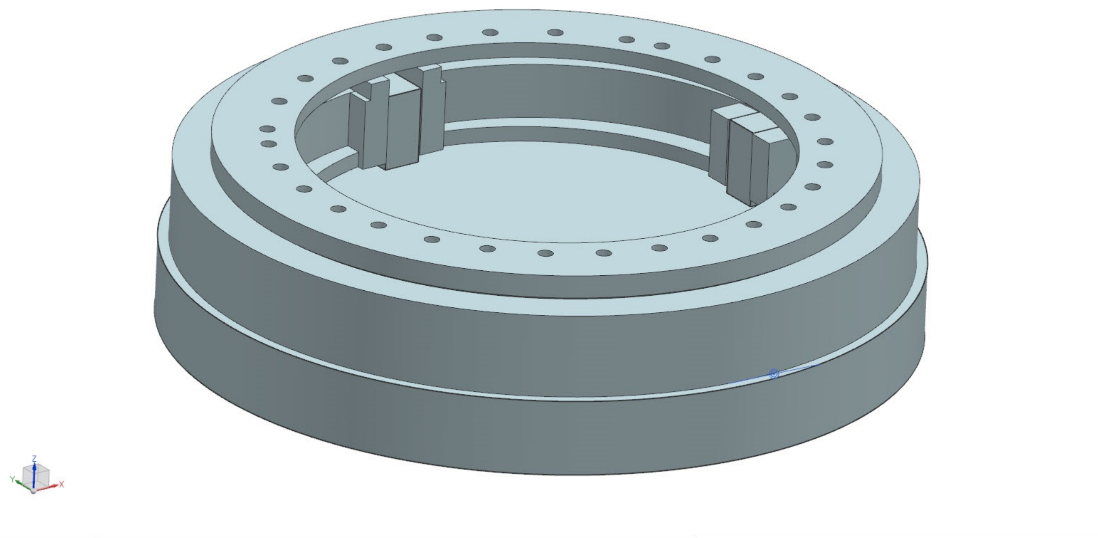

Figure 21: The complete yaw bearing

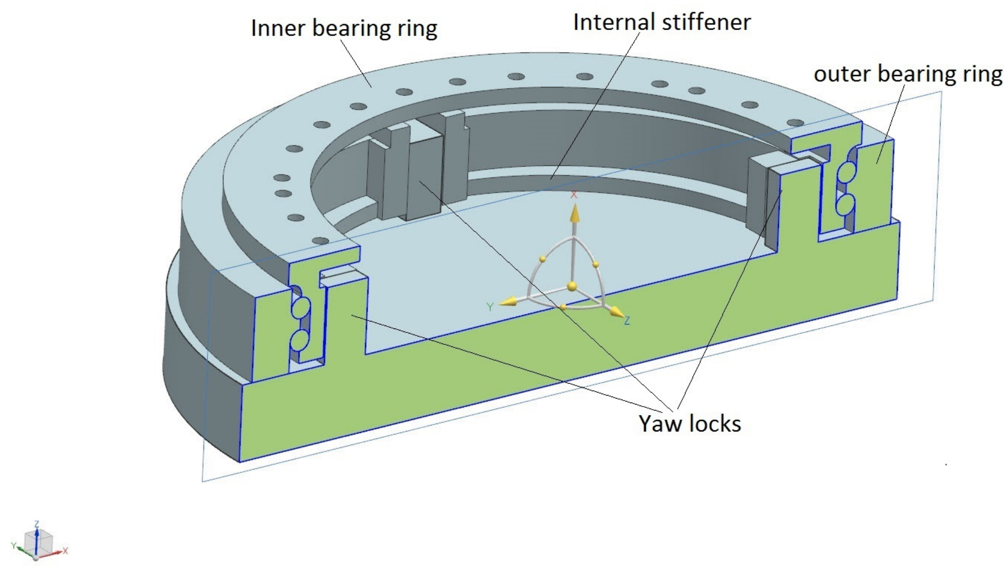

Figure 22: The complete yaw bearing, cut

4.1.6 Bolts

The model was even in late stages modeled with a glued mesh between the tower part and nose part, and between the tower part and inner bearing ring. Just after the model was complete, it was chosen to make a bolt connection instead of mesh mating in mentioned areas. The reason behind this were unrealistic stresses in terms of incorrect force transfer between such components. The bolt diameter chosen for this is 100mm in diameter, and 30 bolts between each connection. The bolt pretension was calculated simply by using a empirical relation:

$$Fpretens = a\sigma_y e \tag{13}$$

Where a is the bolt area, σ_y is the yield stress and e is a tension factor from experience. Normally, a bolt is pretensioned to around 0.7 to 0.8 times its yield strength.
By using $e = 0.75$ and bolt quality of 8.8,
The formula gives,

$$Fpretens = \frac{100^2\pi 640 \cdot 0.75}{4} = 3769911N \tag{14}$$

Which is rounded up to $Pretension = 3770kN$ and used on every bolt connection in the simulation. While the bolt connection between the tower and the nose part is made by bolts with nuts (fig. 23), the situation is slightly different in the connection between the tower part and inner bearing ring. Here, a combination of bolts with nuts, and studs with nuts are used because of the tower part's complex structure (fig. 24).

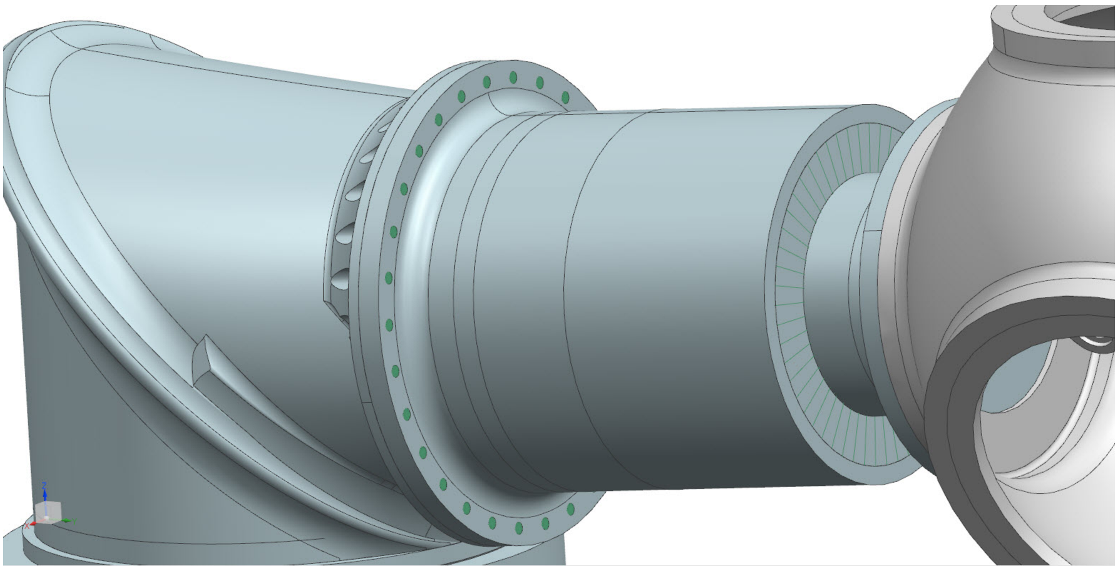

Figure 23: Bolted connection between bed plate parts

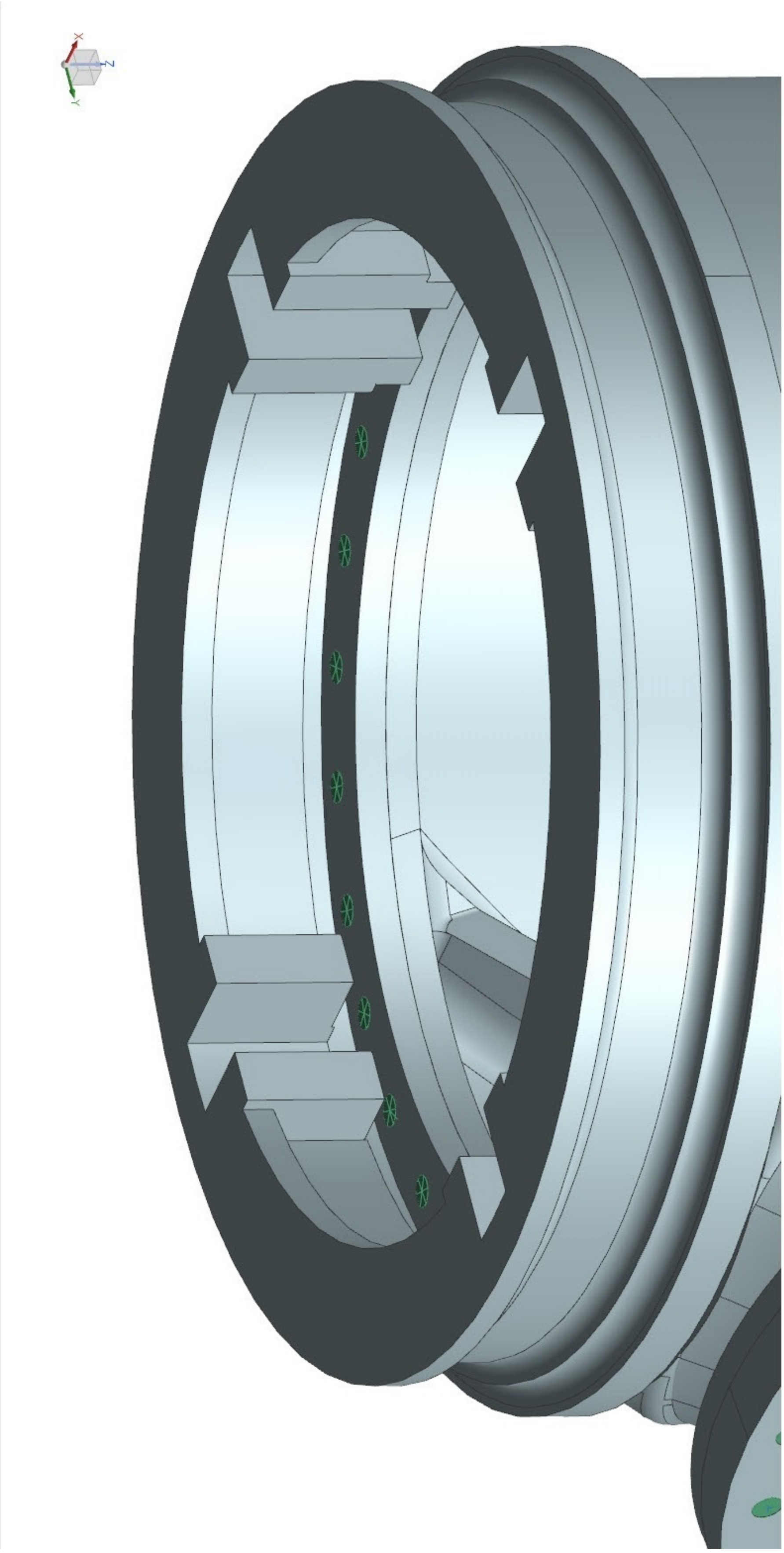

Figure 24: Bolted connection between bearing and bed plate. A stud is used where the diagnoal stiffener meets the flange.

4.1.7 Hub

The hub base is the same as MAK's model. That means few geometry changes is done to it. The pitch motor mounts have an increased thickness of 40mm and their radius towards the edge is increased from 100mm to 250mm. A smaller change is added bolt holes (fig. 25).

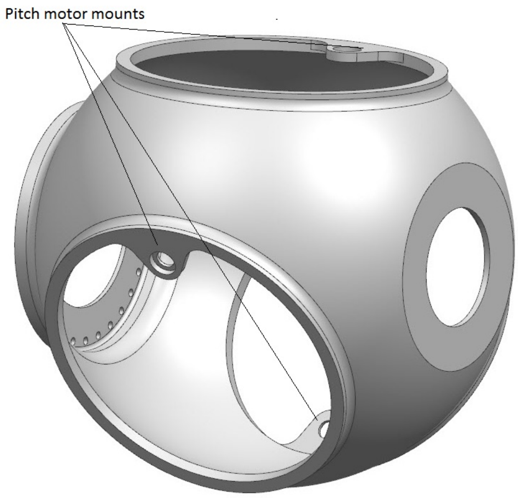

Figure 25: Hub with bolt holes

4.2 Meshing

The mesh in the model contains various types of elements. The bodies are meshed with 3D CTETRA(10) elements in general(fig. 26 and fig. 27), and supplemented with 2D CQUAD8 elements in critical radiuses and bearing surfaces. The average mesh size for the CTETRA10 elements are naturally different from body to body in order to make the analysis as time efficient as possible. The nose part also has a mesh control applied to it. This is done to reduce the number of elements where the stresses are not concentrated. The element size along this control line is specified to be 200mm. A complete 3D element size overview i presented in the tables 1 and 2.

Table 1: 3D Mesh size for bed plate

Where	Size(mm)
Hub	500
Shaft	500
Nose part	325
Tower part	192
inner bearing ring	191
bearing rings	286
outer bearing ring	423
tower connection	428

Table 2: 3D Mesh size for hub

Where	Size(mm)
Hub	271
Shaft	500
inner bearing ring	222
bearing rings	265
outer bearing ring	370

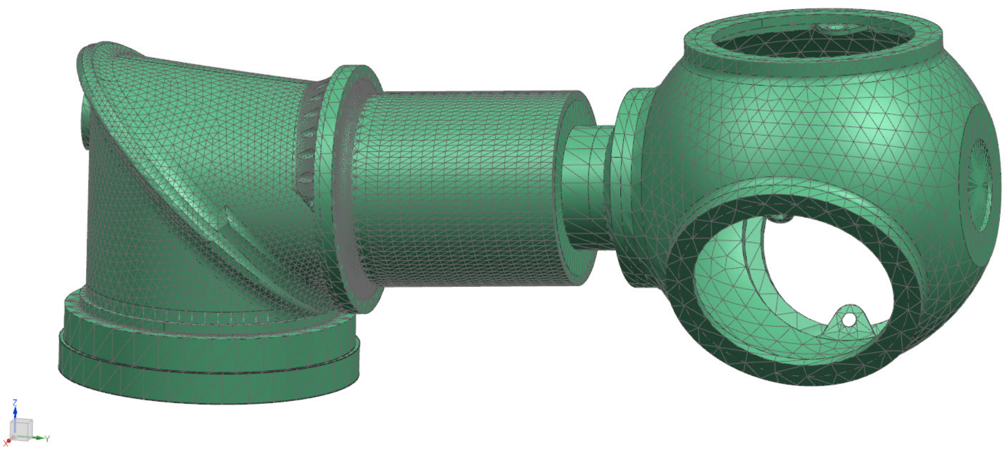

Figure 26: The whole structure meshed

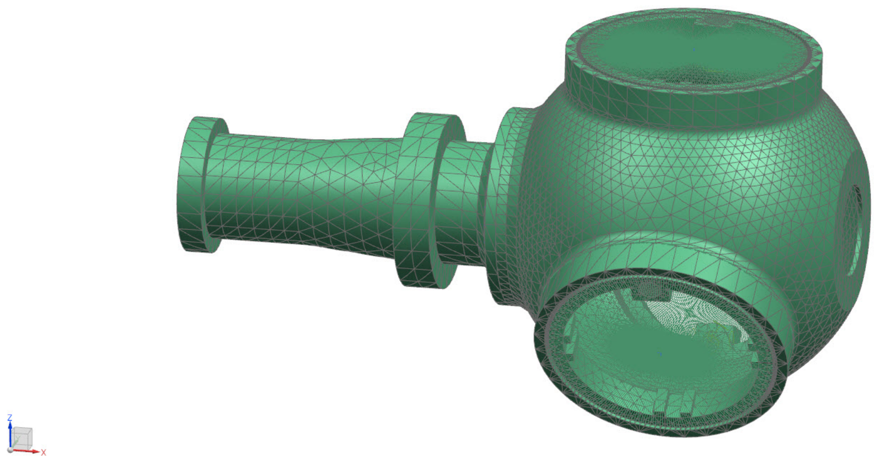

Figure 27: Mesh for hub simulation

In the bearing and flange areas (fig. 28 and 29), 50mm CQUAD8 elements are used. The reason for using this mesh in the bearing areas is to keep the original shape of the area in order to make the load transfer correct. The flanges have this mesh in order to catch the correct stress.

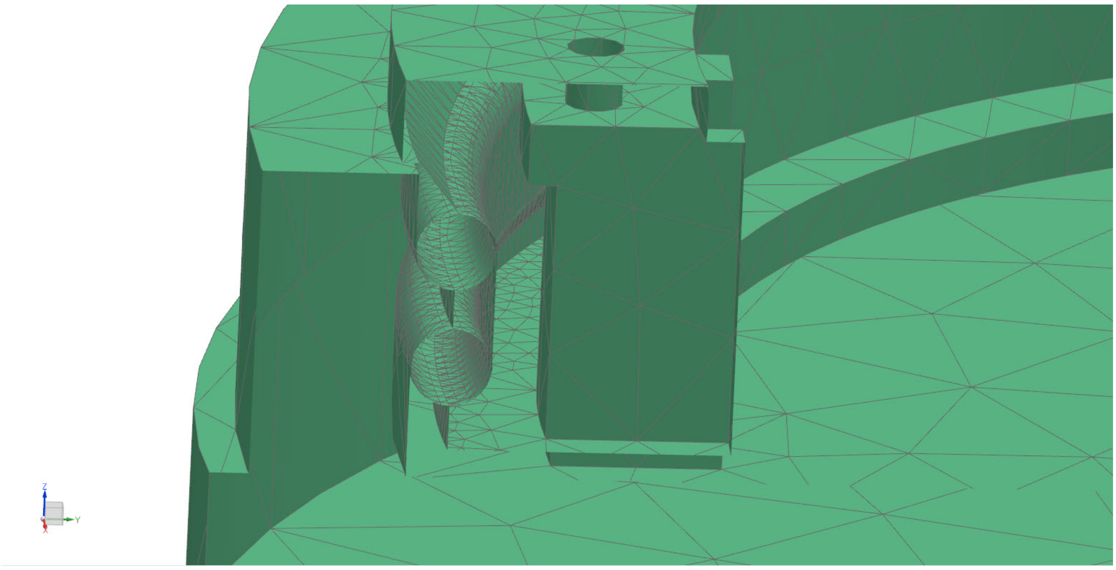

Figure 28: How the Bearing mesh look like

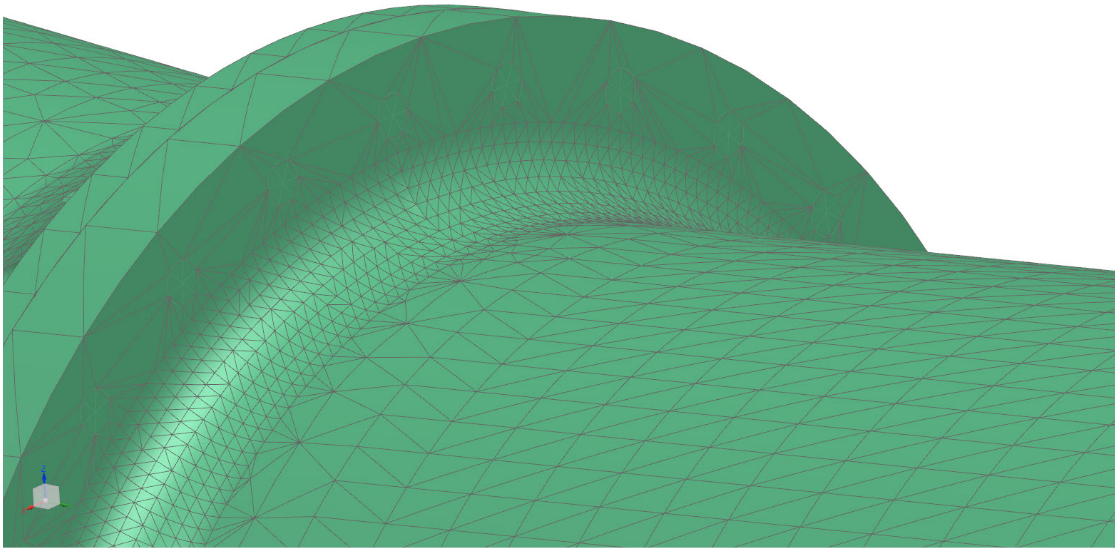

Figure 29: The flange mesh

One RBE3 1D element (fig. 30) on each part are used to connect the rotor shaft to the nose part (fig. 31). The reason why RBE3 are used instead of RBE2 is because of the relative node stiffness with a RBE2 element is higher. In other words, the "‘node legs"’ cannot deform independently which makes them more suitable in pure load transfer situations. In this simulation it is assumed that the structure is flexible enough to justify movement of "‘node legs"’, which means RBE3 elements. One RBE3 element is also used in the front of the hub for transferring the moments applied here.

The bolt connection in the simulation uses one CBEAM element for the bolt shank, and two RBE2 elements as head and nut (fig. 32). Both the head and nut diameter is specified to be 1.5 times the diameter of the bolt shank.

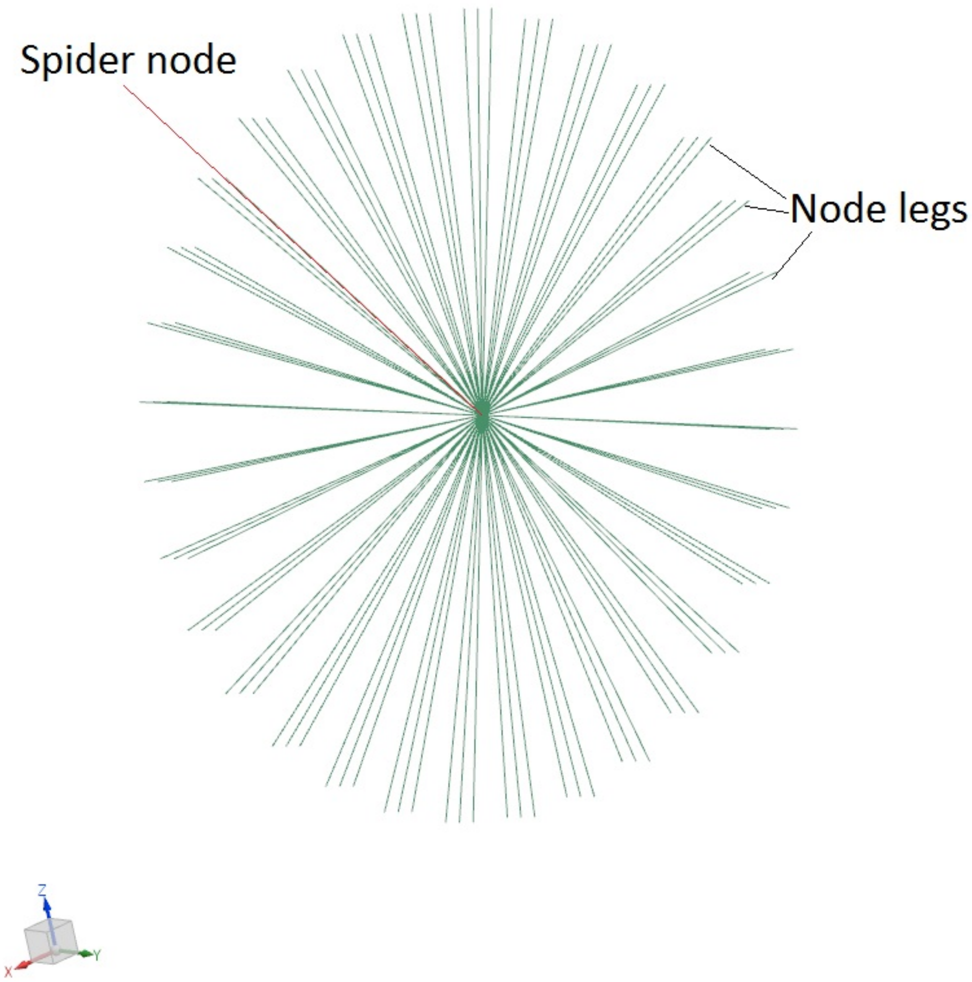

Figure 30: Showing spider node and its legs

Mesh mating is used between the rotor and the hub (valid only for the bed plate simulation), and the yaw bearing and tower platform. In reality they would have been bolted, but the exact stresses in these areas in this simulation is not important. Mesh mating means that the simulation thinks the mesh mated parts are welded or casted in one piece.

For the hub simulation the situation is a bit different. Mesh mating is used between the pitch bearings and the hub. The pitch bearings are in reality bolted to the hub, but because of the small contact area and nearly zero offset bolts, it is chosen to simulate it with mesh mating which should give approximate the same results. Between the hub and the shaft, a bolted connection instead of mesh mating is used in order to "‘catch"’ stresses induced by force eccentricities.

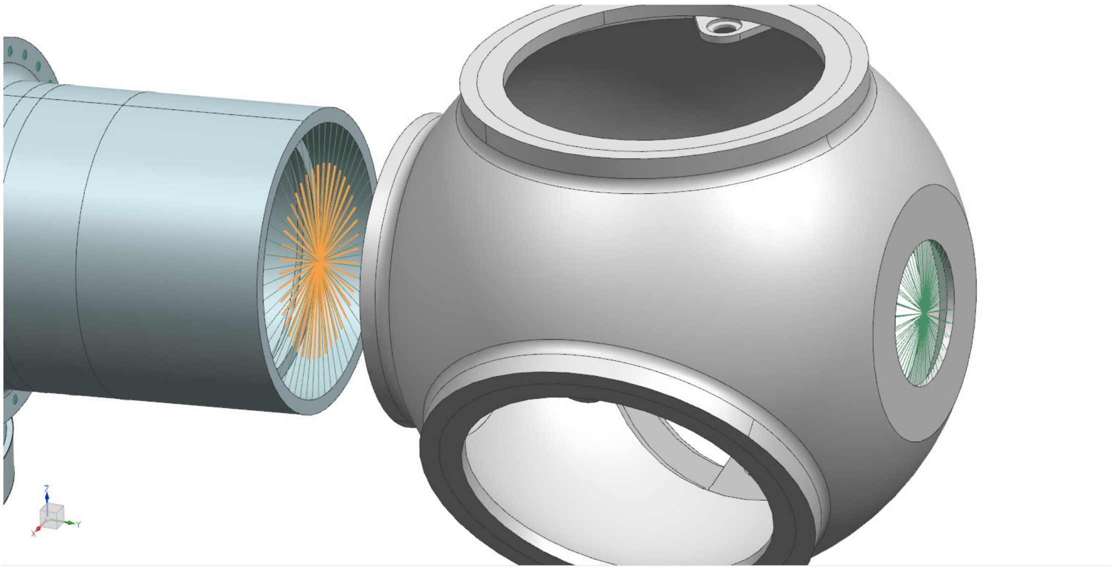

Figure 31: The RBE3s. The highlighted one is RBE3 connected to the shaft (which is hidden)

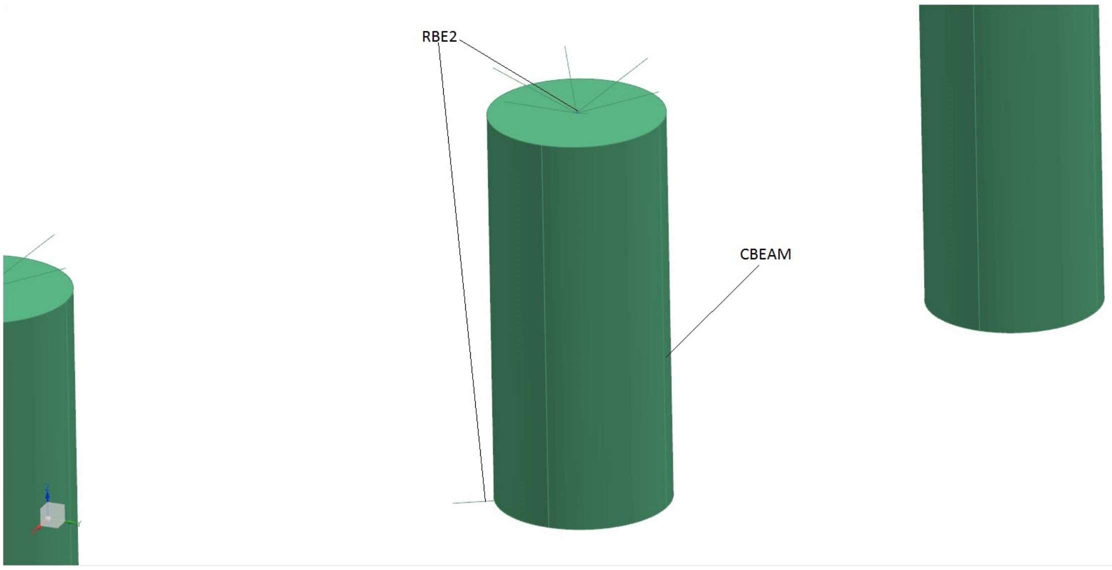

Figure 32: The bolt mesh with RBE2 spider and CBEAM element

4.3 Simulation

This chapter explains how the simulation is put together.

4.3.1 Assembly of The Bed Plate

The assembly consists of the tower part, nose part, rotor shaft, rotor hub and the assembled yaw bearing (fig. 33 and 34).
An important thing to mention is the modification done to the rotor shaft. Two massive cylinders added to the shaft to simulate bearings in order to have a realistic load transfer (14).

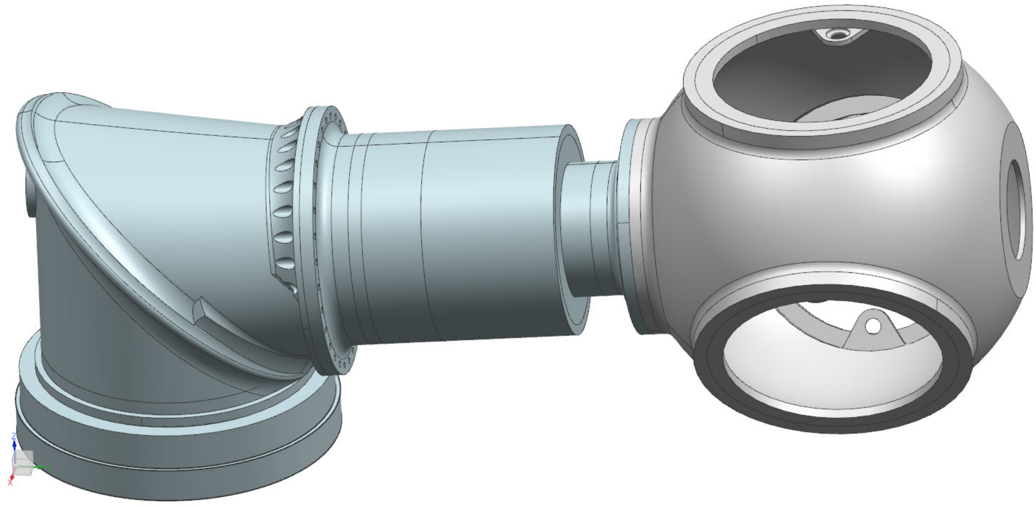

Figure 33: The whole assembly of the structure

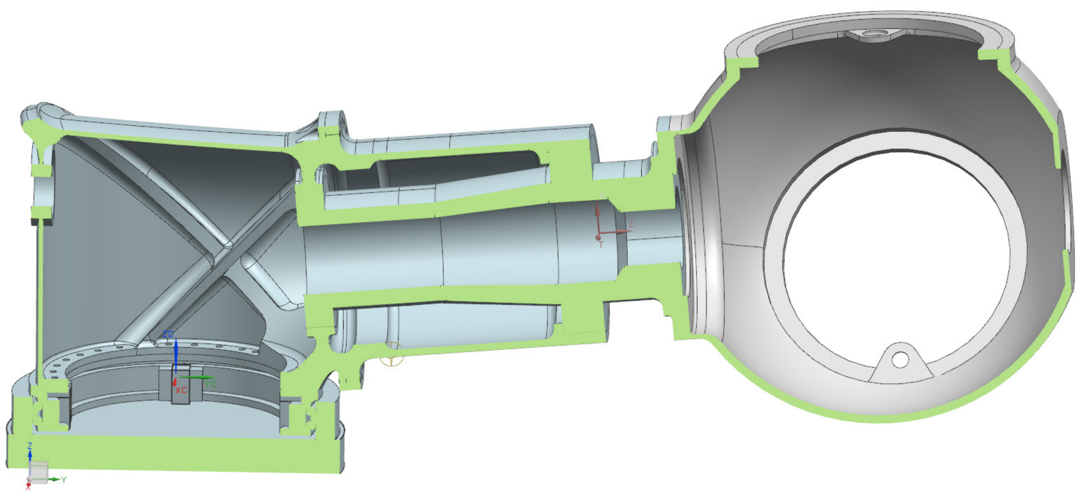

Figure 34: The whole assembly of the structure, cut

4.3.2 Assembly of The Hub

The hub assembly consists of the hub, the rotor shaft, and three pitch bearings (fig. 35).

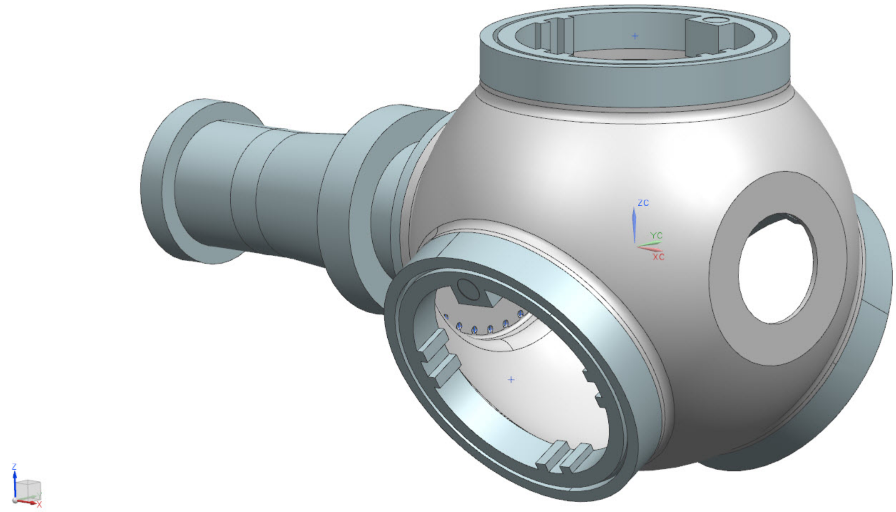

Figure 35: The complete hub

4.3.3 Loads on Bed Plate

The loads are taken from ES thesis appendix A [5] which are results from FAST. These loads are then further scaled up with a safety factor of 1.35, like the IEC 61400 specifies. In order to have realistic stresses, the loads are expanded to be placed right on the hub instead of just being concentrated in one point. Loads are presented in table 3 and shown in fig. 36.

Table 3: Applied loads [5]

Force	FAST value	SF	Applied value	Unit	Applied where
F_x	726	1.35	980.1	kN	Each blade connection
F_y	114	1.35	153.9	kN	Each blade connection
F_z	640	1.35	864	kN	Each blade connection
M_x	4550	1.35	6142.5	kN	Each blade connection, like a force
M_y	28660	1.35	38691	kNm	Concentrated on the hub
M_z	22480	1.35	30348	kNm	Concentrated on the hub
Gen. rotor	295	1.35	398.25	kN	Nose part and Shaft
Gen. stator	1970	1.35	2659.5	KN	Nose part
G	9.81	1.35	13.25	m/s^2	Whole structure

Because the wind direction changes over time, load directions will also change. This means the whole structure must be checked with different load combinations in order to see if the stresses are within the limit. This is basically done by changing the directions of M_y and M_z to make all four possible load combinations. Table 4 below explains how M_y and M_z are applied in the different load directions.

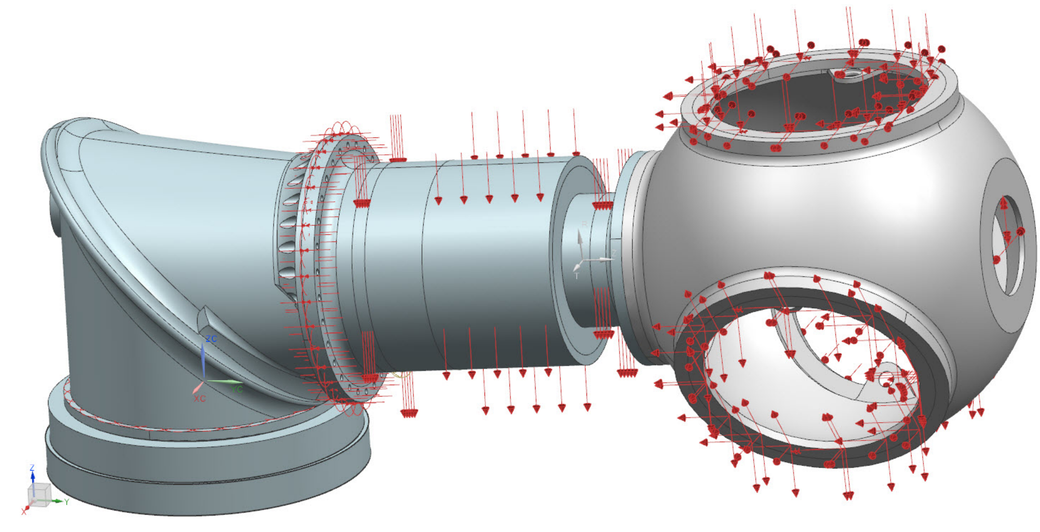

Figure 36: Loads on bed plate assembly

Table 4: Load combinations

Load combination	M_y	M_z
1	+	+
2	-	+
3	-	-
4	+	-

4.3.4 Loads on Hub

The loads acting on the hub (fig. 37) is taken from MAK's appendix 2 [6] DLC 1.2 10 m/s with a safety factor of 1.35 (table 5). According to MAK, DLC 6.1 gave the highest stresses. But after investigating the results for DLC 6.1 in MAK's appendix 2, several questions about the validity of results needs to be answered before it can be used. One major concern is that DLC 6.1 is called "‘parked state"’, but the forces in Z-direction for each blade is the same. That means that the turbine is actually rotating. MAK mentioned early in his project that DLC 6.1 is either totally parked or idling (spinning with blades pitched in feathered position), but what the actual situation is like is not confirmed. Therefore, DLC 1.2 10m/s is used instead of DLC 6.1 which gave MAK the second highest stress.

Table 5: Applied loads on hub [6]

Force	FAST value	SF	Applied value	Unit	Applied where
F_x1	508.6	1.35	686.6	kN	blade 1
F_y1	-347.5	1.35	-469.1	kN	blade 1
F_z1	1580	1.35	2133.3	kN	blade 1
M_x1	-14220	1.35	-19197	kNm	blade 1
M_y1	20450	1.35	27607.5	kNm	blade 1
M_z1	150.5	1.35	213.3	kNm	blade 1
F_x2	478.8	1.35	646.3	kN	blade 2
F_y2	-350.8	1.35	-473.6	kN	blade 2
F_z2	1569	1.35	2118.2	kN	blade 2
M_x2	9198	1.35	12417.3	kNm	blade 2
M_y2	19710	1.35	26608	kNm	blade 2
M_z2	143	1.35	193	kNm	blade 2
F_x3	510.6	1.35	689.3	kN	blade 3
F_y3	-344.9	1.35	-465.6	kN	blade 3
F_z3	1582	1.35	2135.7	kN	blade 3
M_x3	9078	1.35	12255.3	kNm	blade 3
M_y3	21020	1.35	28377	kNm	blade 3
M_z3	144.3	1.35	194.8	kNm	blade 3
G	9.81	1.35	13.25	m/s^2	Whole structure

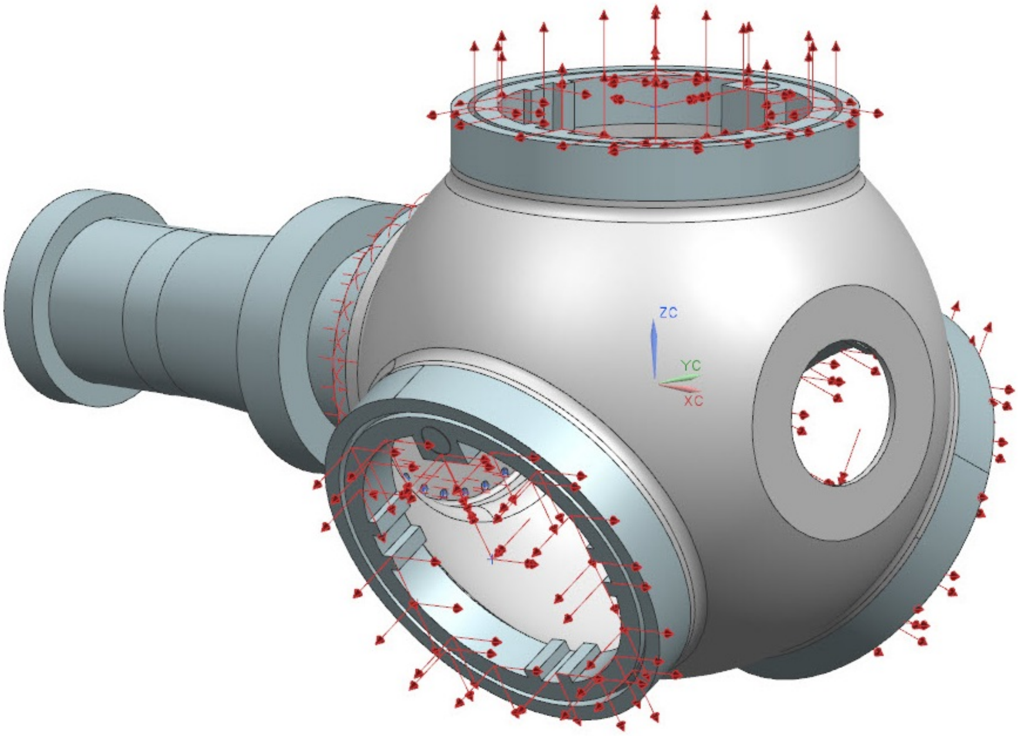

Figure 37: The hub with its loads

4.3.5 Constraints on The Bed Plate

There are two constraints in this model. The first one is applied to the tower structure. All of the loads applied finally end here. The second one is between both RBE3 spiders to connect the shaft to the bed plate. In order to make this connection work, a modeling object called AUTOMPC must be enabled in the solution parameters. This modeling object enables NX to connect two duplicate nodes and share their DOF's. Constraints illustrated in fig. 38.

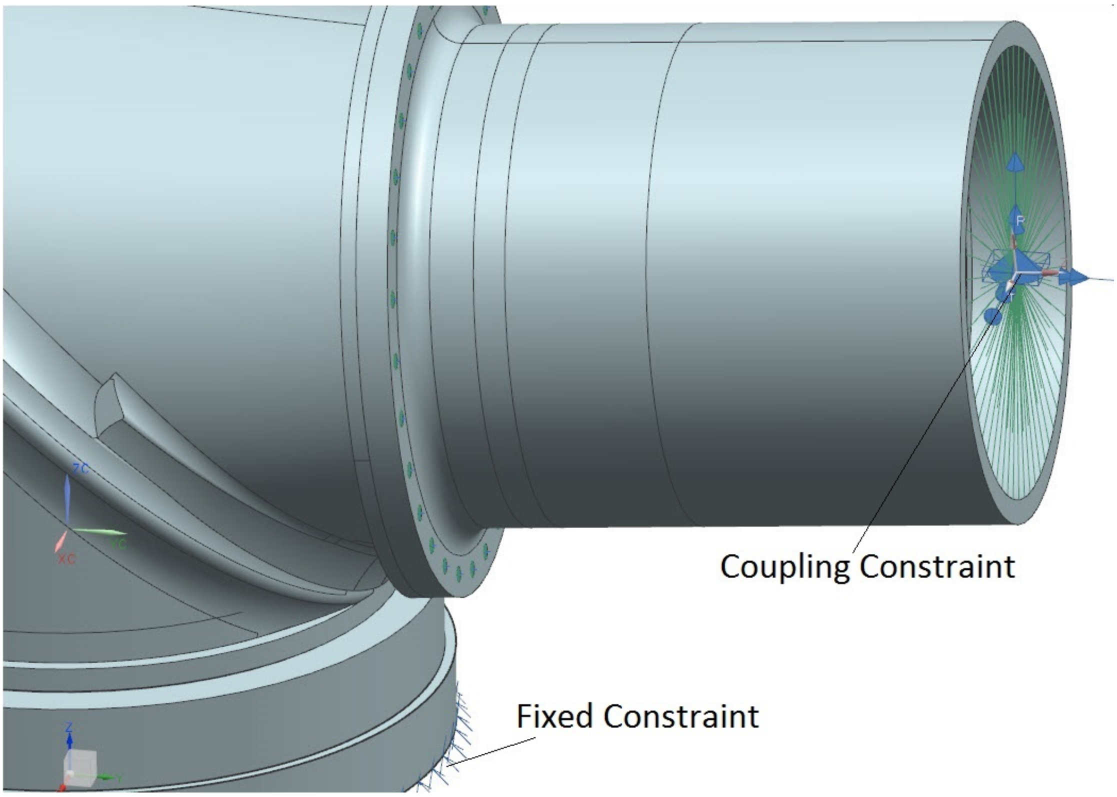

Figure 38: Constraints on bed plate

4.3.6 Constraints on The Hub

The hub simulation is constrained simply by fixed constraints on each rotor bearing surface. Constraints are applied here in order to remove artificial stiffness in the hub. Exactly the same principle is applied on the bed plate. Constraints illustrated in fig. 39.

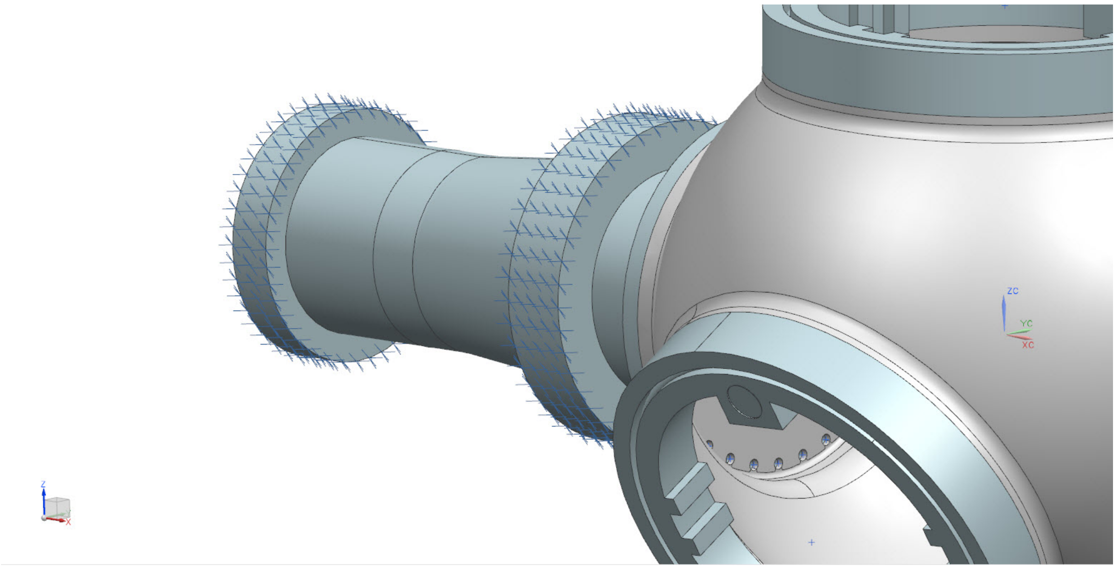

Figure 39: The hub with constraints

4.3.7 Contact Areas on Bed Plate

Contact areas are established so relative motion between certain components can occur. Areas where the possibility of relative motion is wanted are: Between connection flanges (fig. 42), between inner and outer bearing ring and rolling element in the yaw bearing (fig. 41), and between rotor bearings and the bed plate (fig. 40). Yaw locks also contains contact areas in order to constrain the bed plate from yaw motion. The only areas where friction is applied is between the connection flanges. Friction is applied here because its a big part of how a flange connection works in reality. The friction coefficient used here is 0.35, which is a standard friction coefficient between two steel bodies.

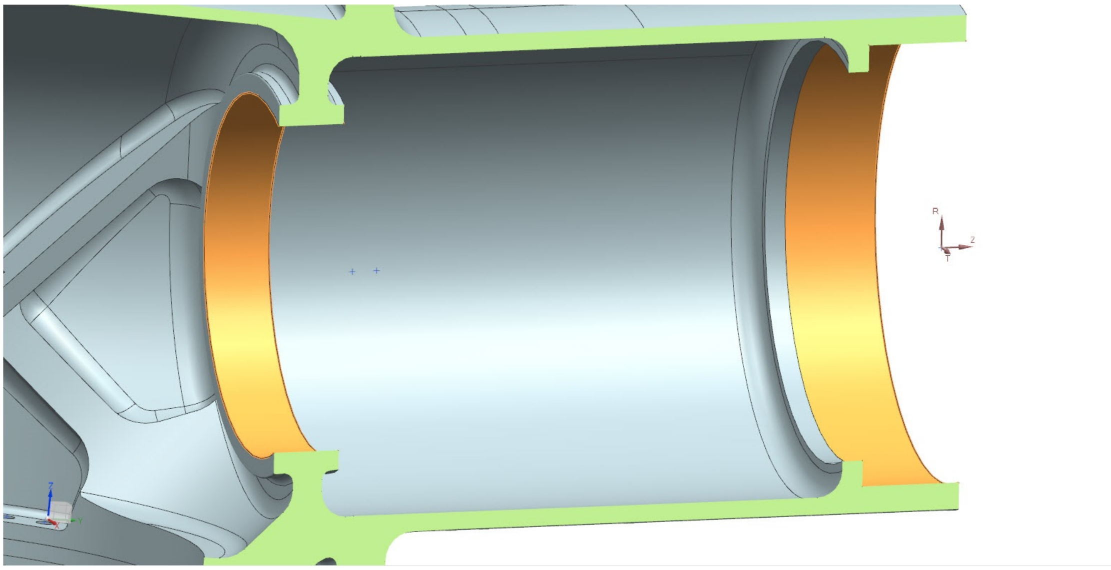

Figure 40: Highlighted areas are rotor bearing contact faces

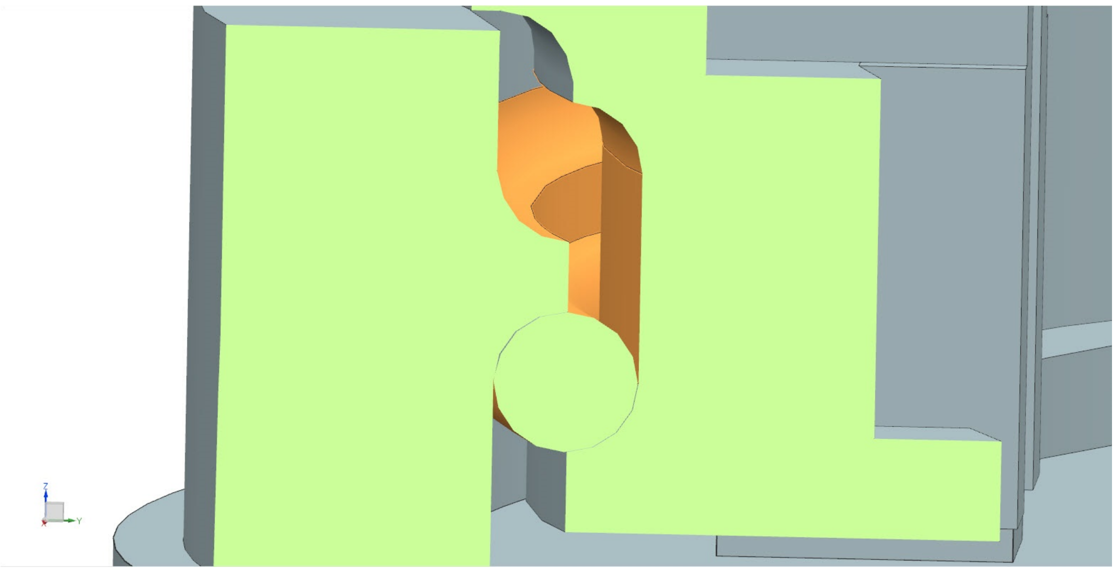

Figure 41: Highlighted areas are Yaw bearing contact faces. Notice that one roller ring is removed

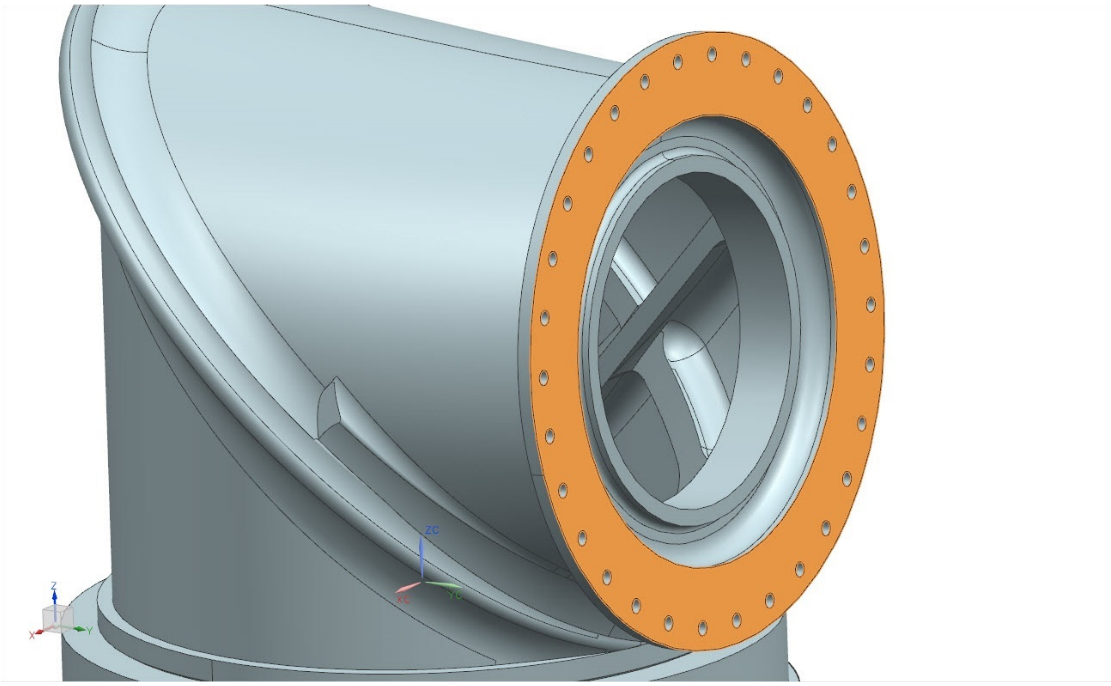

Figure 42: Highlighted area is a flange contact area. The nose part is not shown here.

4.3.8 Contact Areas on Hub

The hub has several contact areas. Most of them are located in the pitch bearings, except contact between the hub and the shaft. Only the contact face on the shaft (fig. 43) and on the yaw lock (fig. 44) are illustrated.

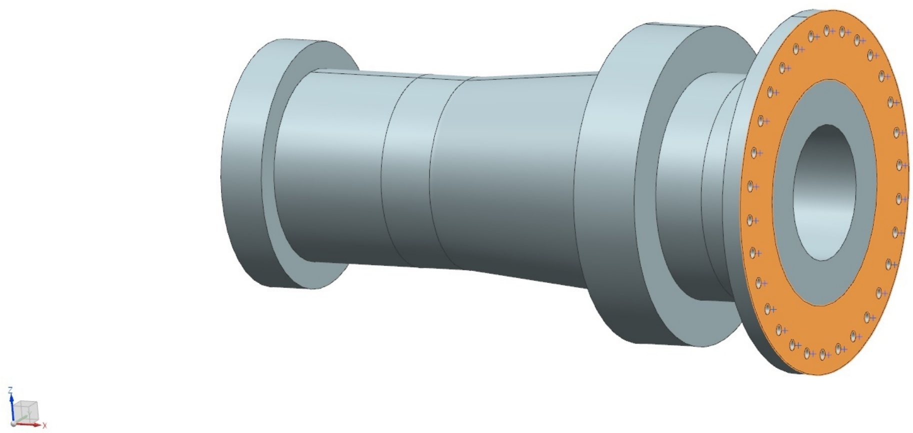

Figure 43: Shaft's contact surface

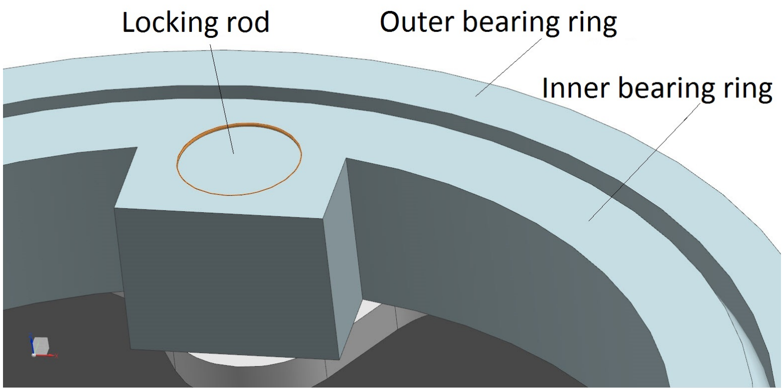

Figure 44: Yaw lock contact area for the pitch bearing is highlighted

4.3.9 Connections in The Bed Plate

The bed plate parts are connected with a bolt connection (fig. 45) of bolts and nuts, instead of mesh mating. Mesh mating gives artificially good load transfer and can in worst case eliminate some load effects. Nearly the same joining technique is used for the connection between the bed plate and the yaw bearing. Here, instead of just bolts, studs are also used. According to Maskindeler [7], the threaded part of the stud should be somewhere between 0.8 or 1.5 times the diameter of the bolt. In this simulation a thread length of 150mm is used.

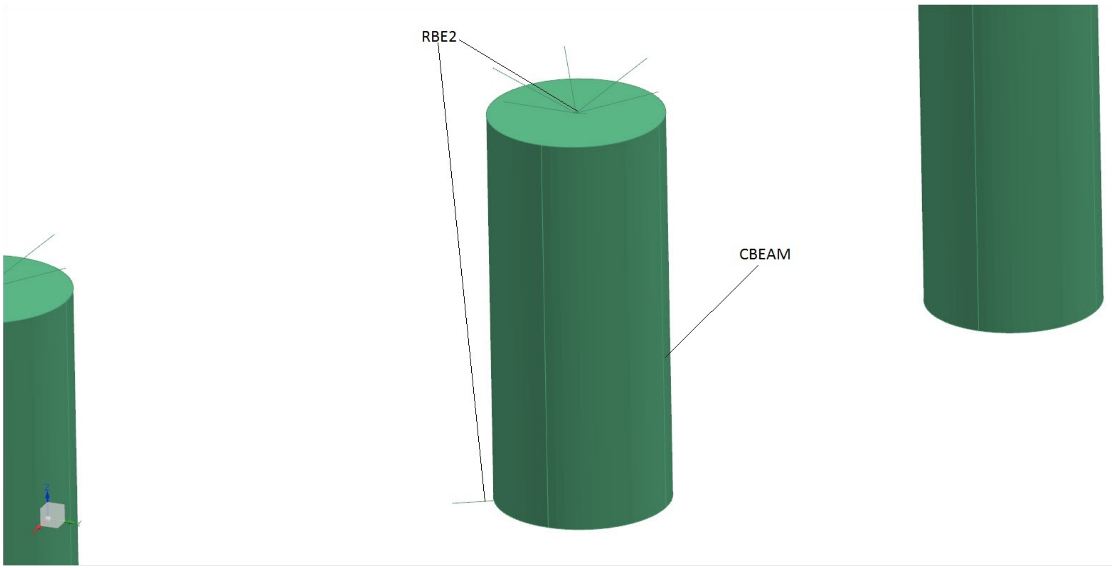

Figure 45: A bolt connection

The rotor hub is connected to the rotor shaft through mesh mating in this simulation (bed plate), in order to make it less complex because these areas are not investigated here. The yaw bearing is connected to the tower structure via mesh mating. Again, this is not an area to be investigated.

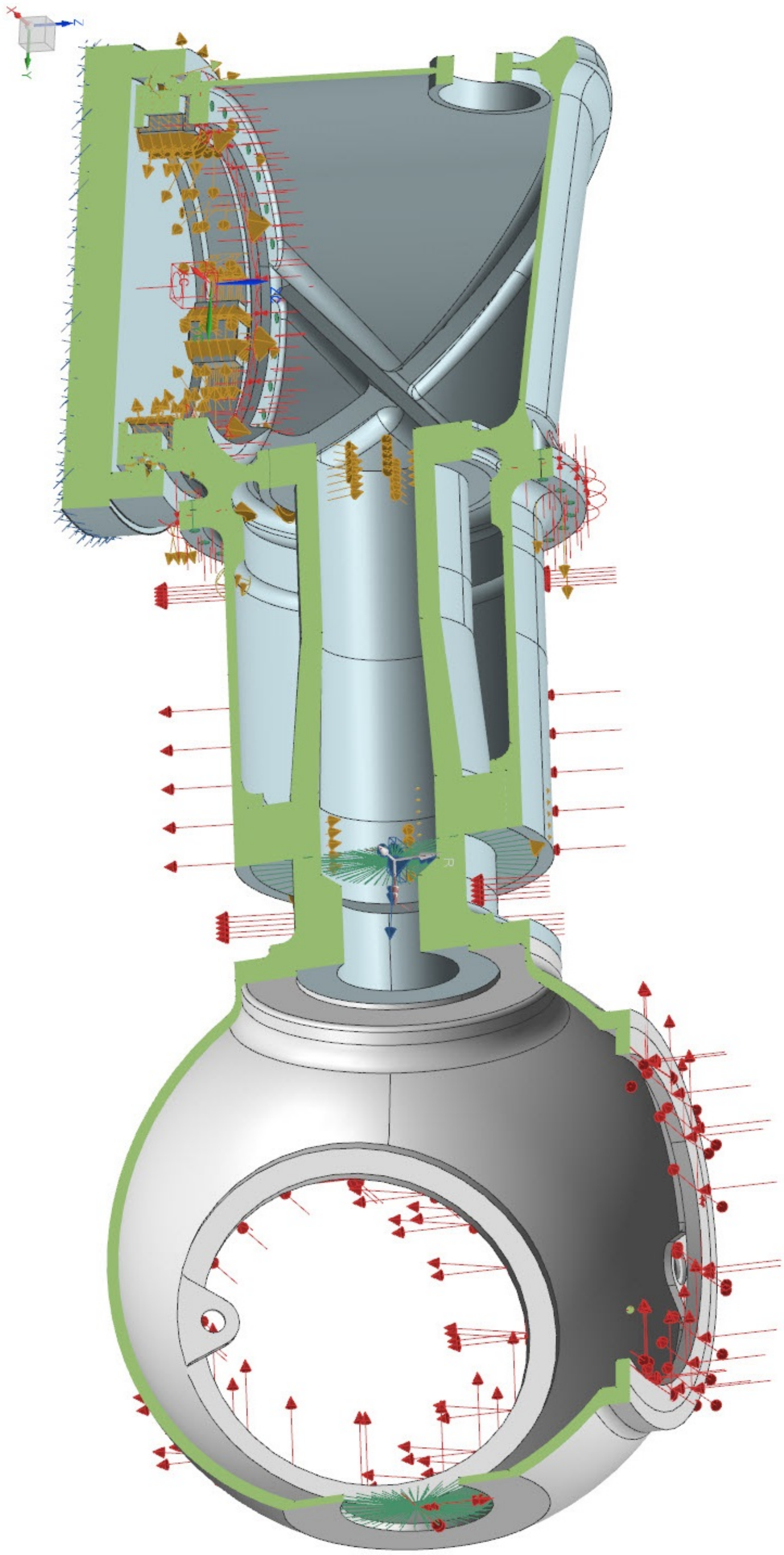

Figure 46: The complete bed plate simulation model with geometries, loads, contacts and constraints

4.3.10 Connections in The Hub

The hub is connected to the shaft through a flange connection with 30 100mm bolts, just like the other flange connections (fig. 47). The pitch bearings are connected to the hub via mesh mating.

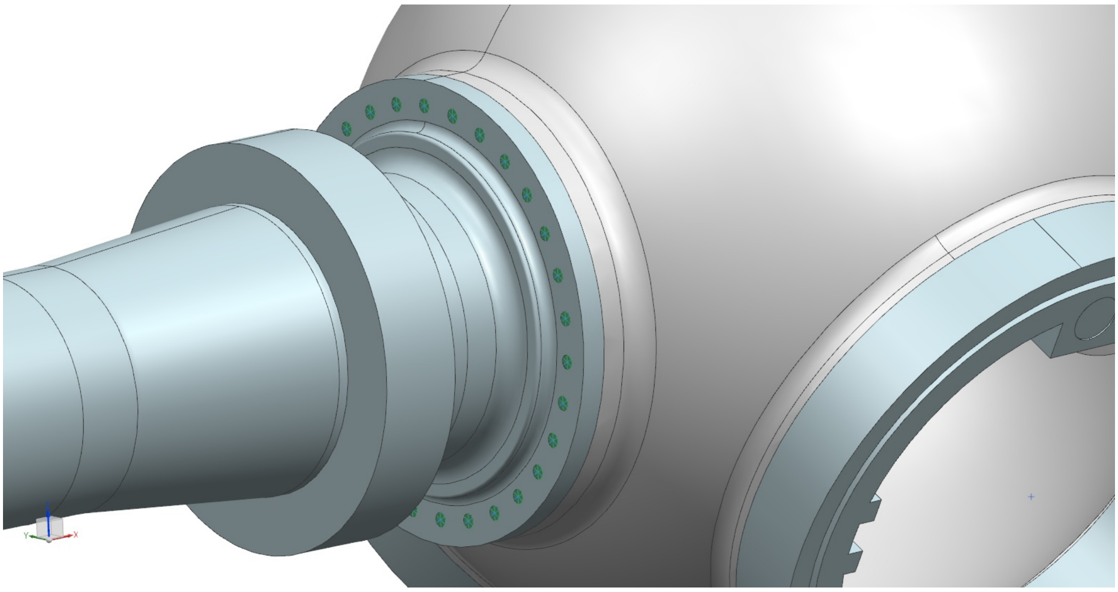

Figure 47: The hub's bolt connection

5 Results Bed plate

5.1 Load combination 1

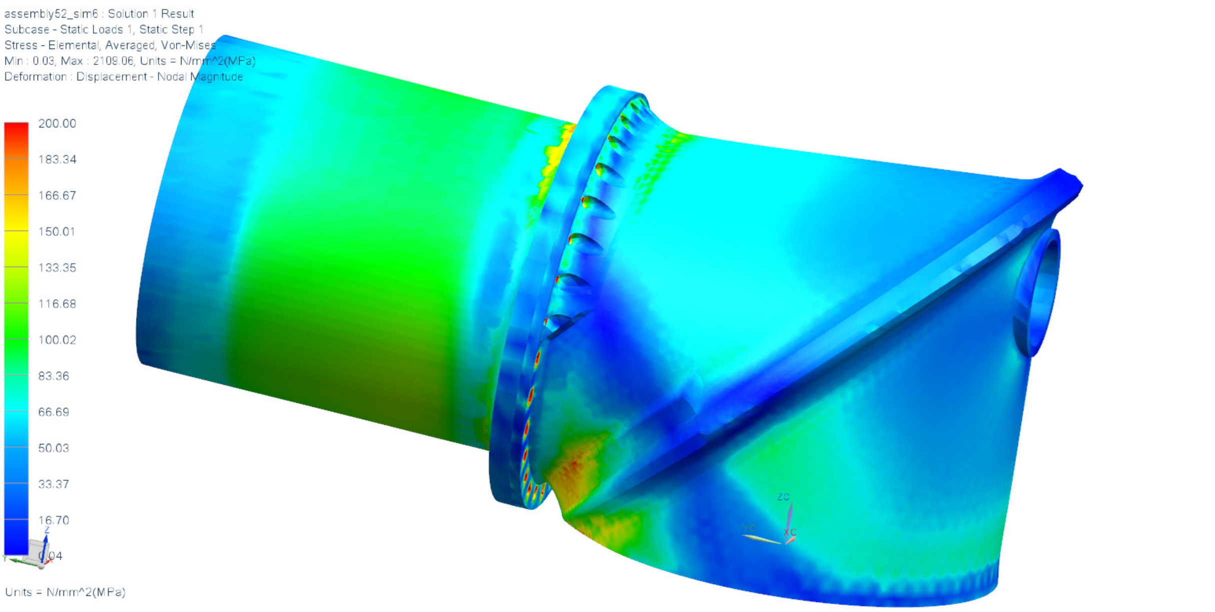

Figure 48: Load combination 1 from one side

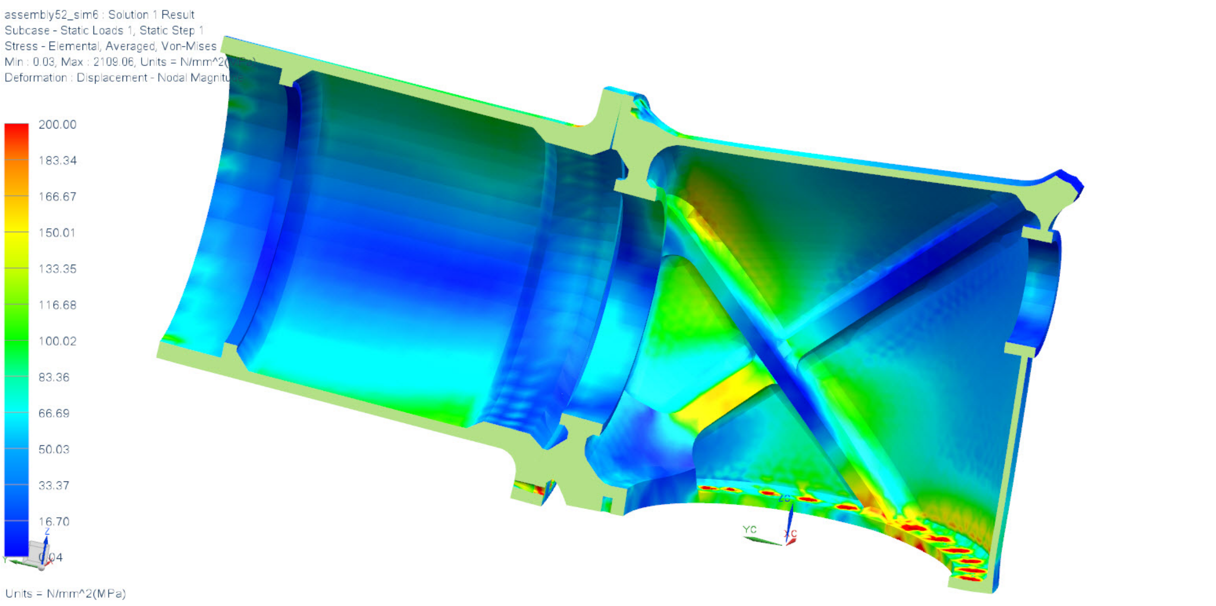

Figure 49: Load combination 1 from one side, cut

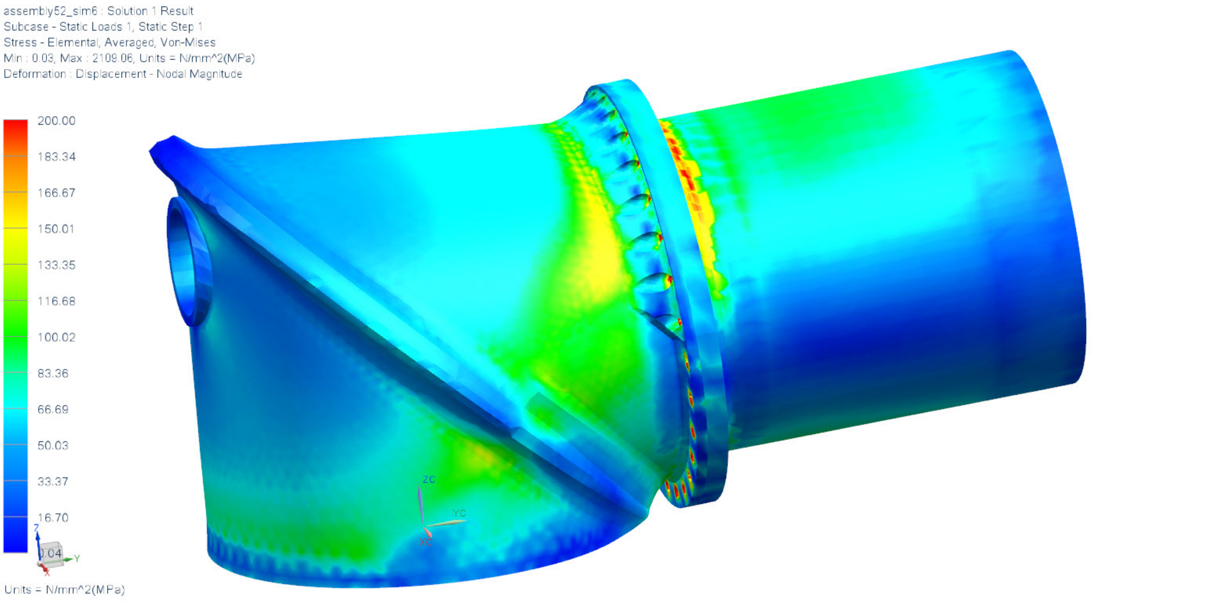

Figure 50: Load combination 1 from the other side. Peak stress area in red

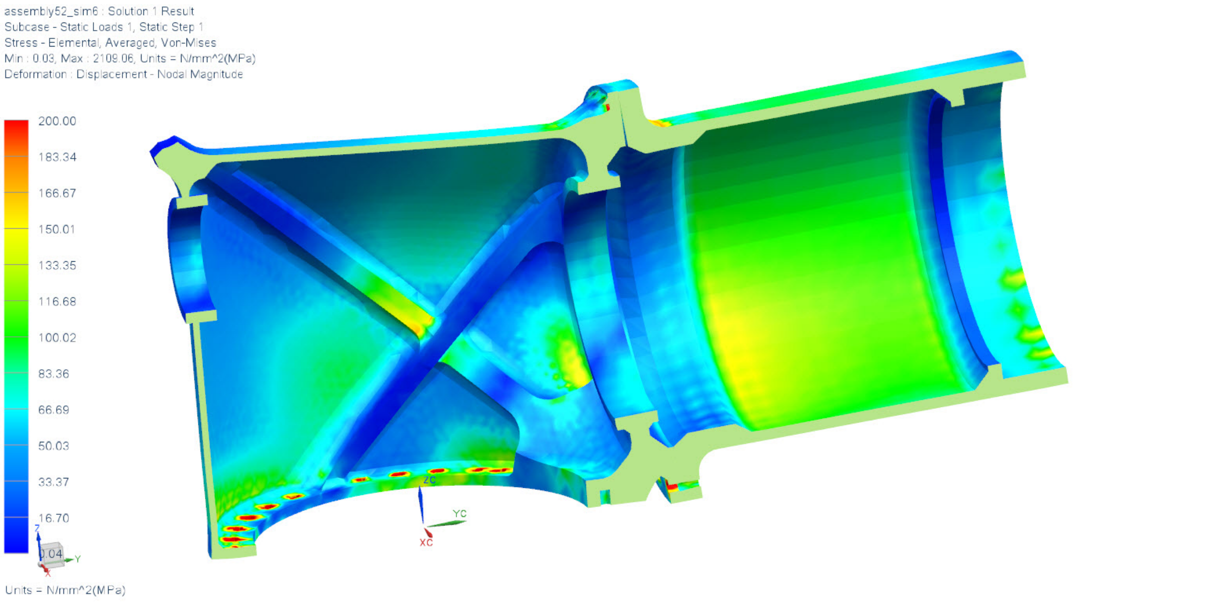

Figure 51: Load combination 1 from the other side, cut

5.2 Load combination 2

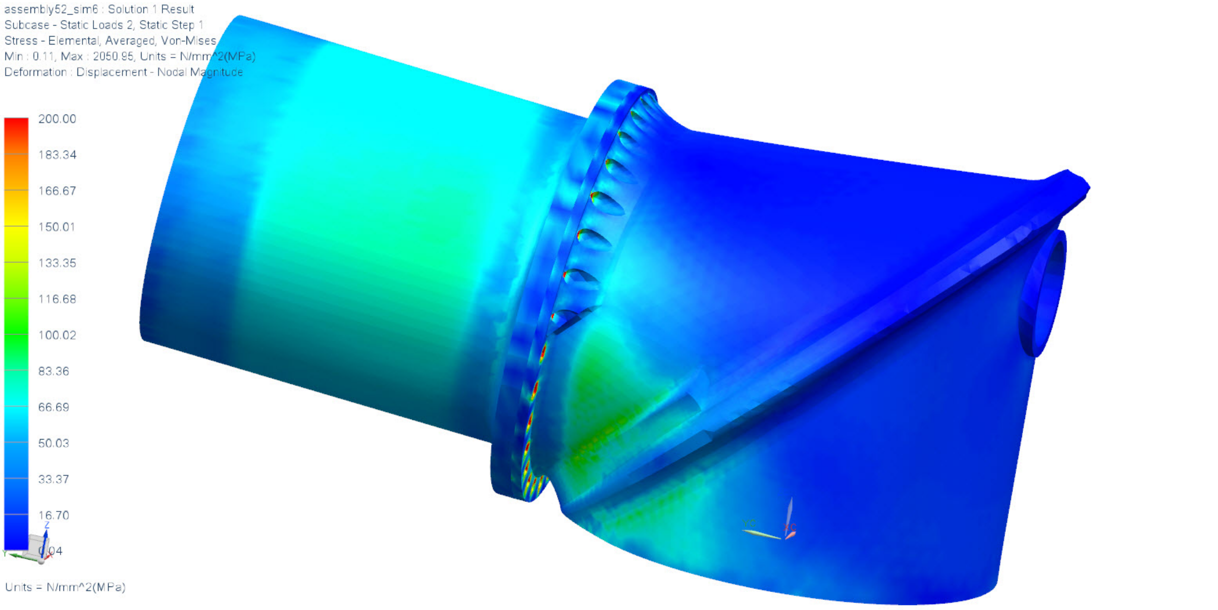

Figure 52: Load combination 2 from one side

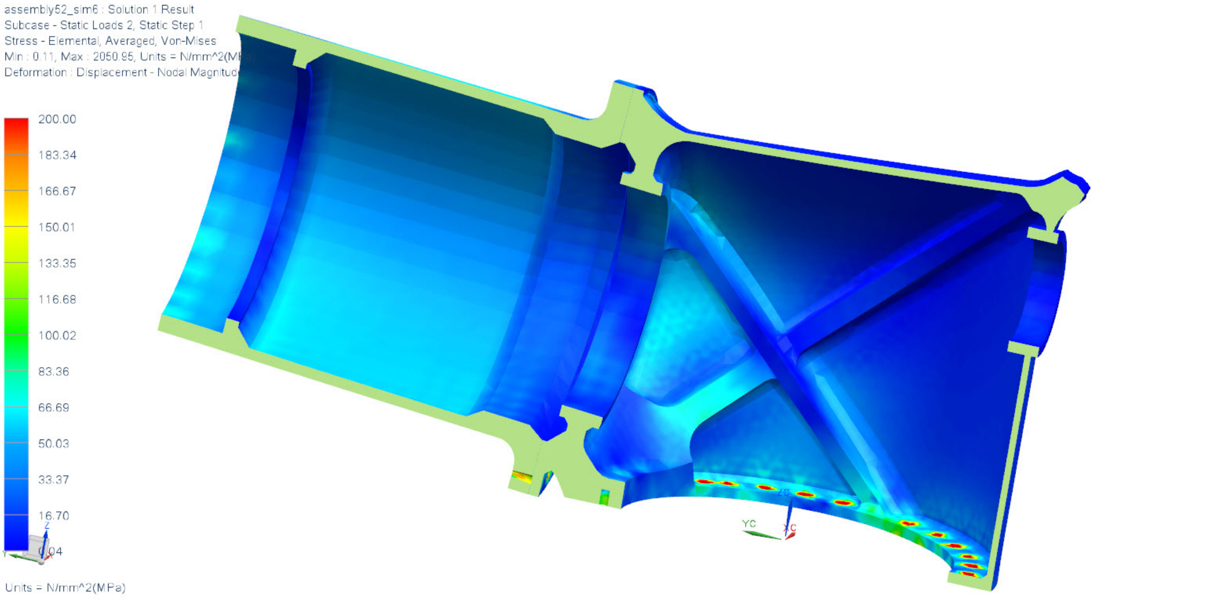

Figure 53: Load combination 2 from one side, cut

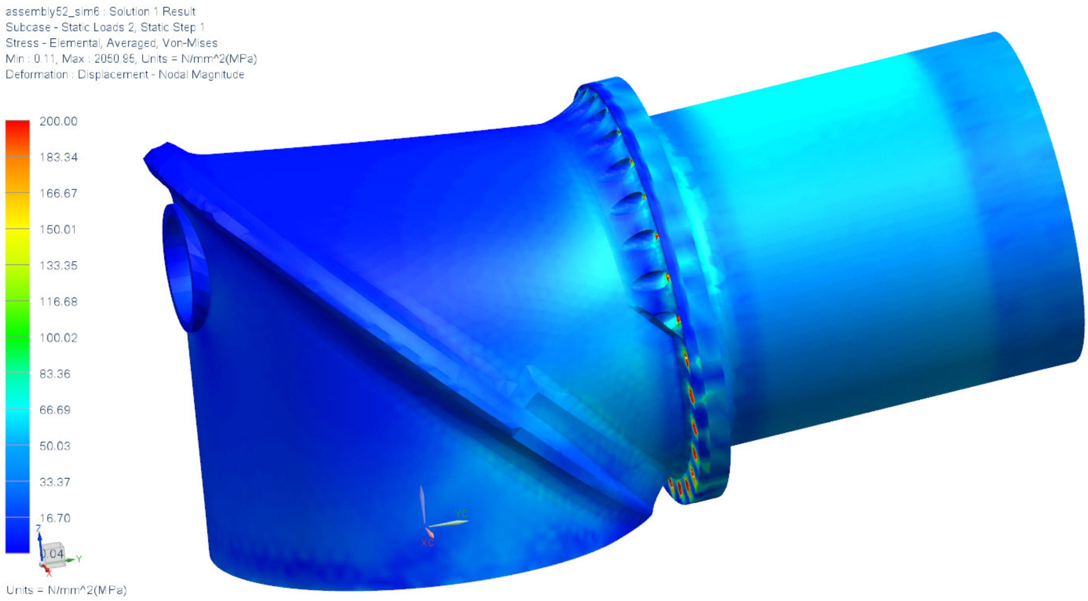

Figure 54: Load combination 2 from the other side

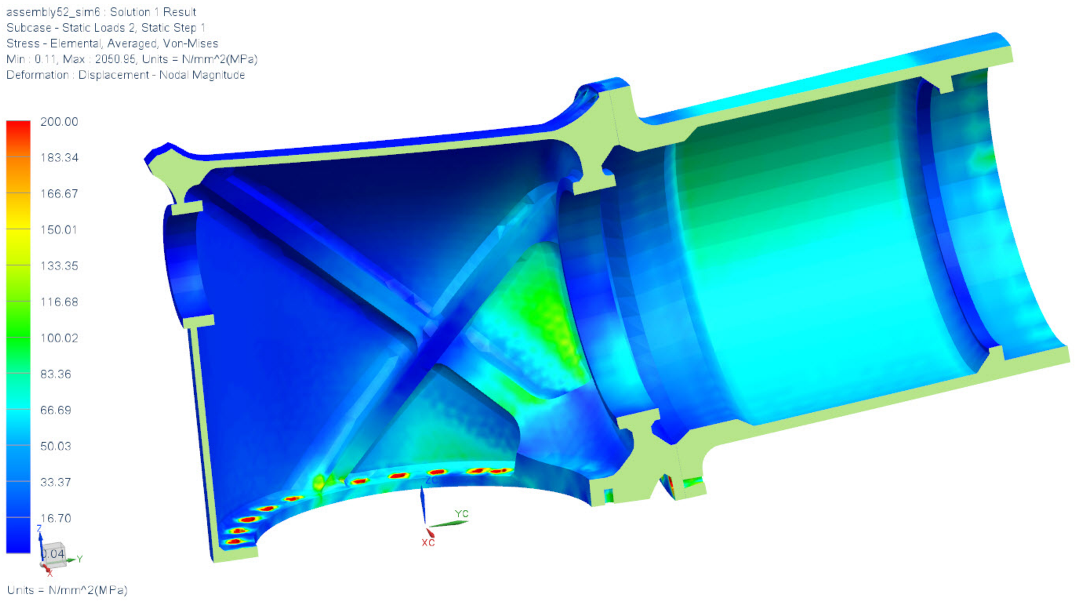

Figure 55: Load combination 2 from the other side, cut

5.3 Load combination 3

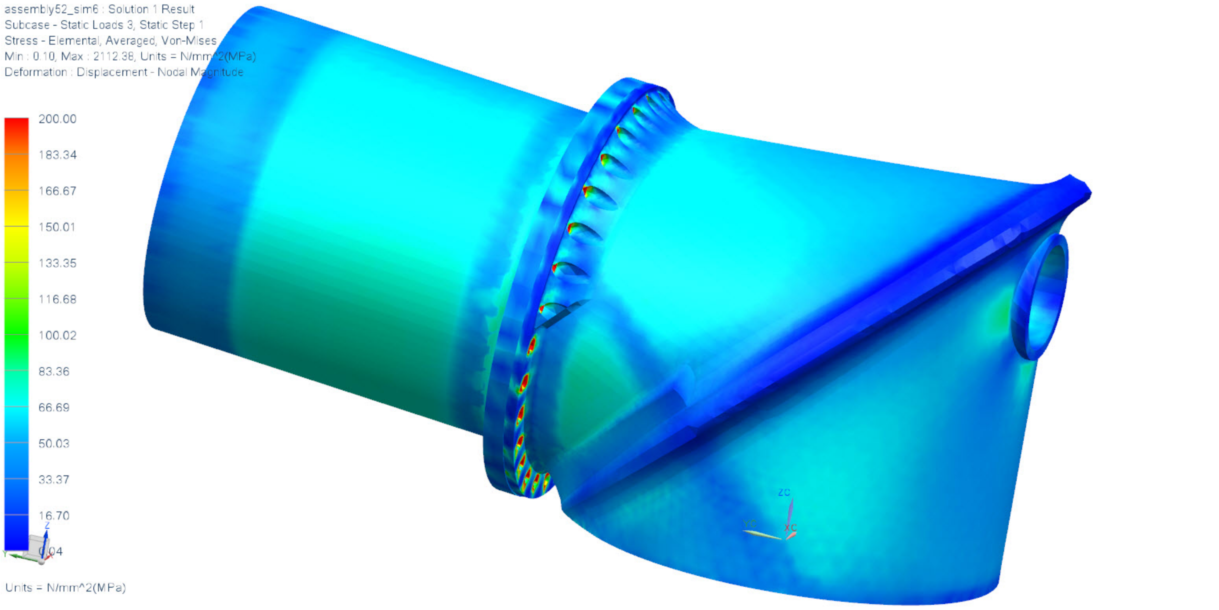

Figure 56: Load combination 3 from one side

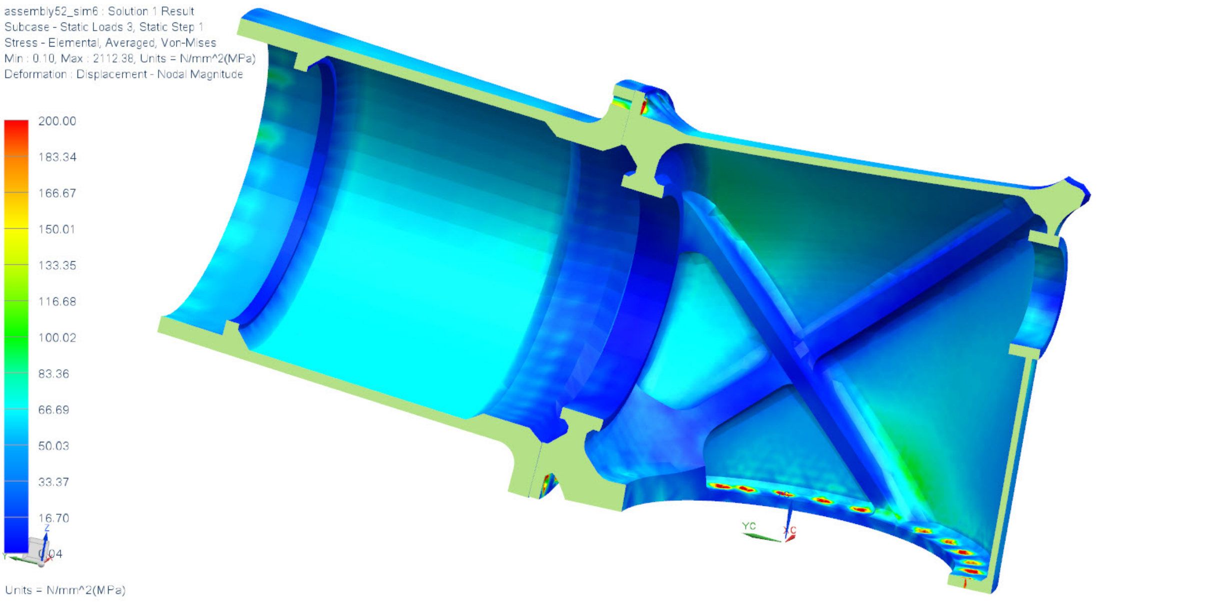

Figure 57: Load combination 3 from one side, cut

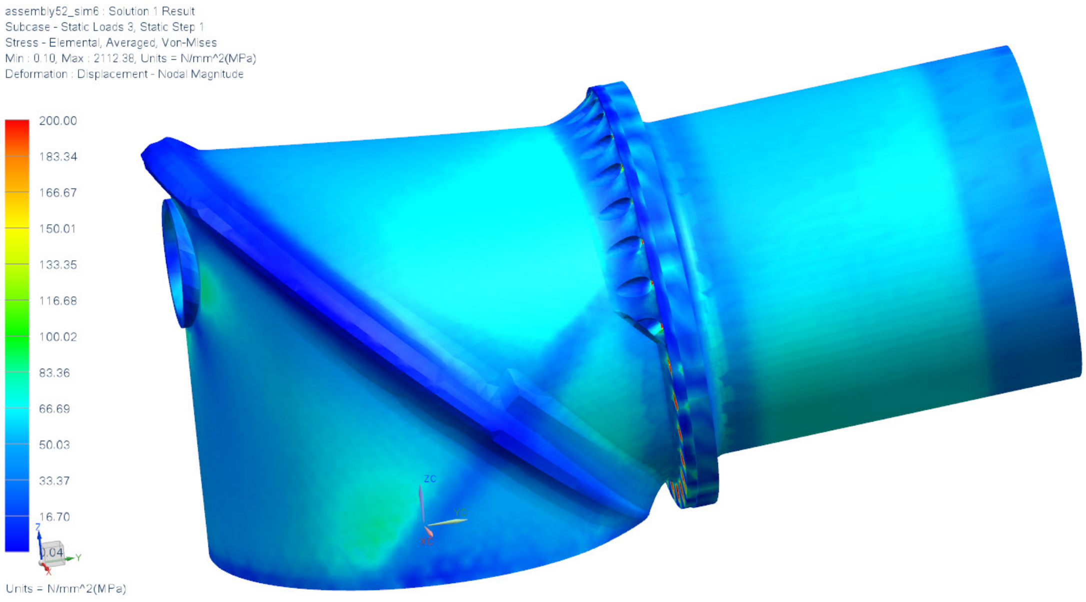

Figure 58: Load combination 3 from the other side

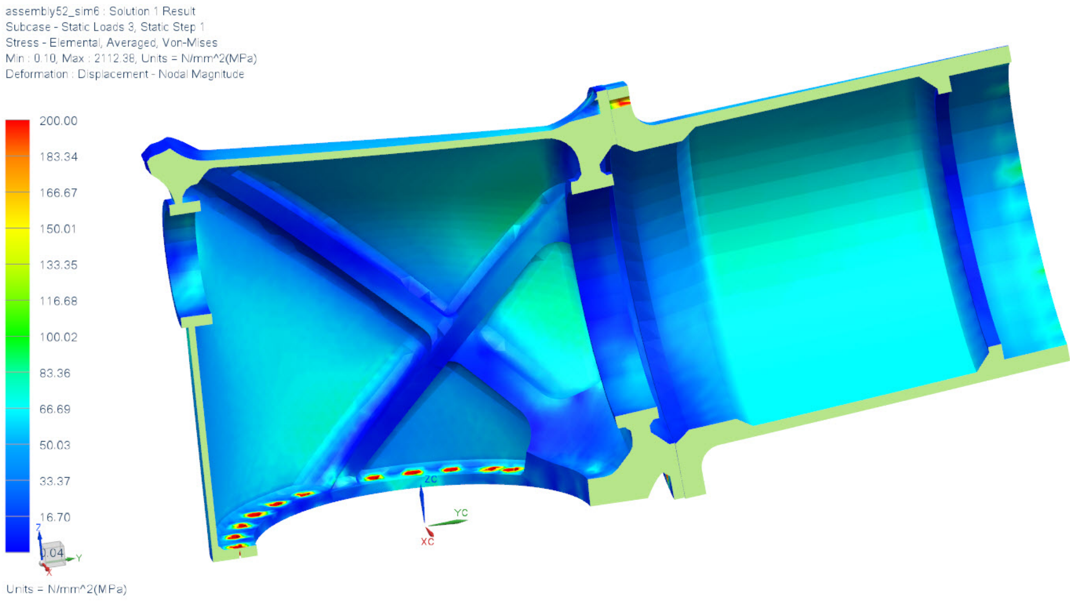

Figure 59: Load combination 3 from the other side, cut

5.4 Load combination 4

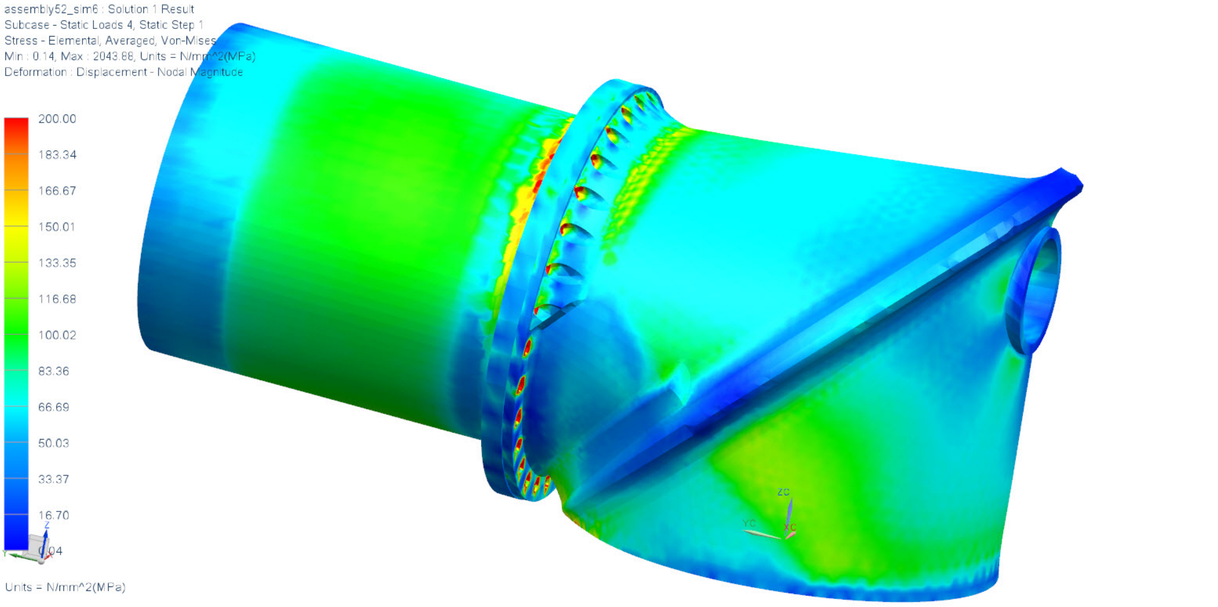

Figure 60: Load combination 4 from one side. Peak stress seen in red

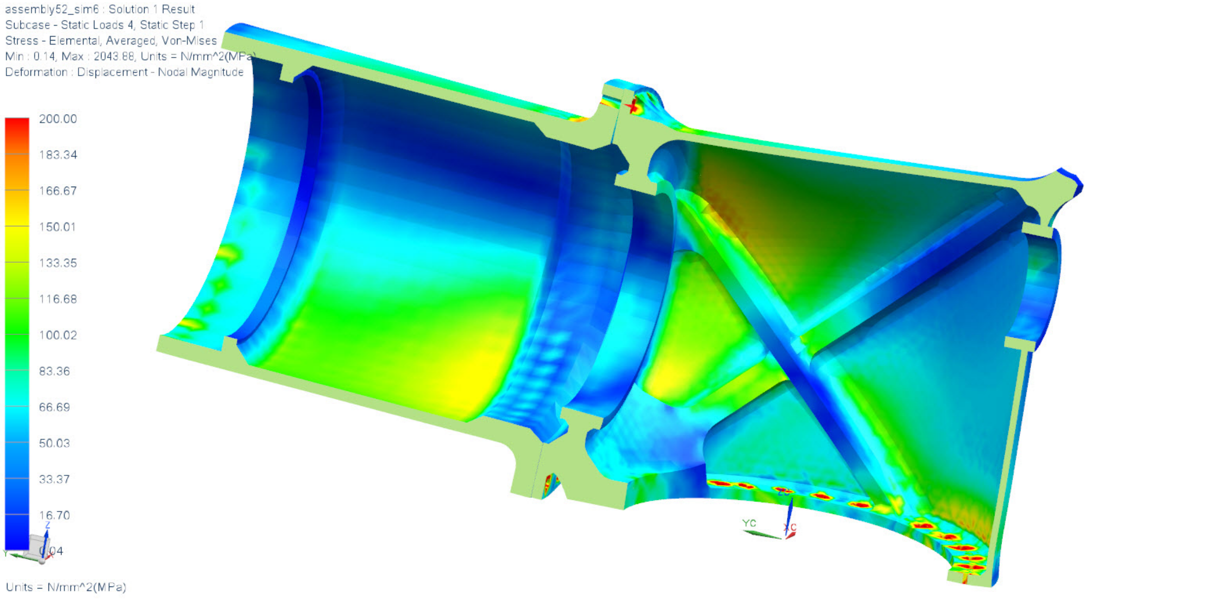

Figure 61: Load combination 4 from one side, cut

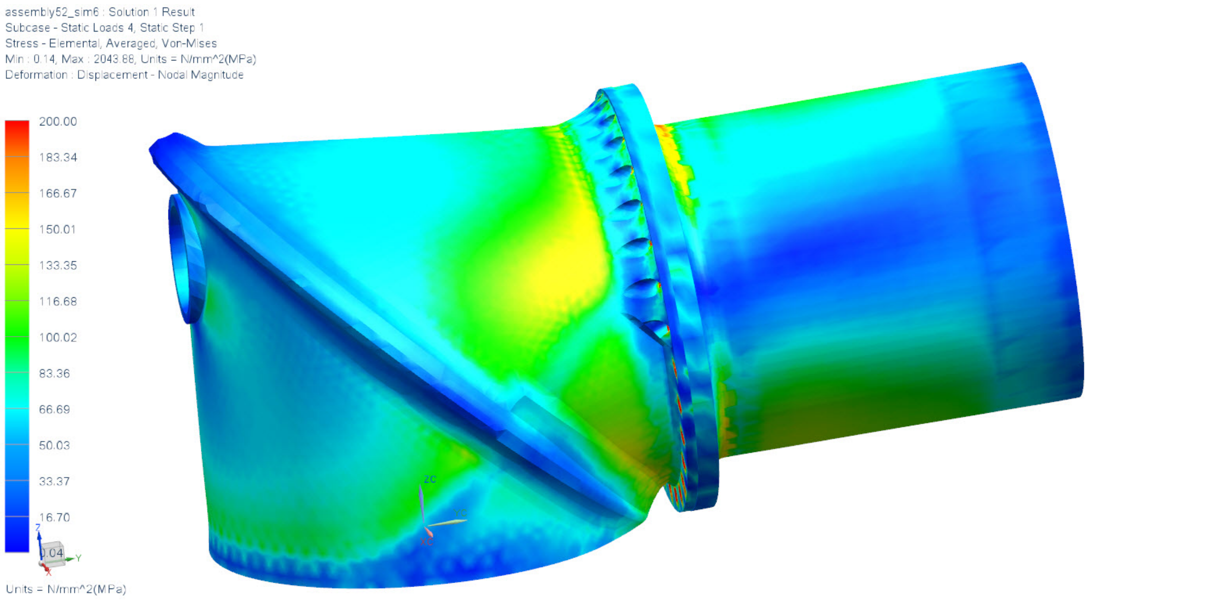

Figure 62: Load combination 4 from the other side

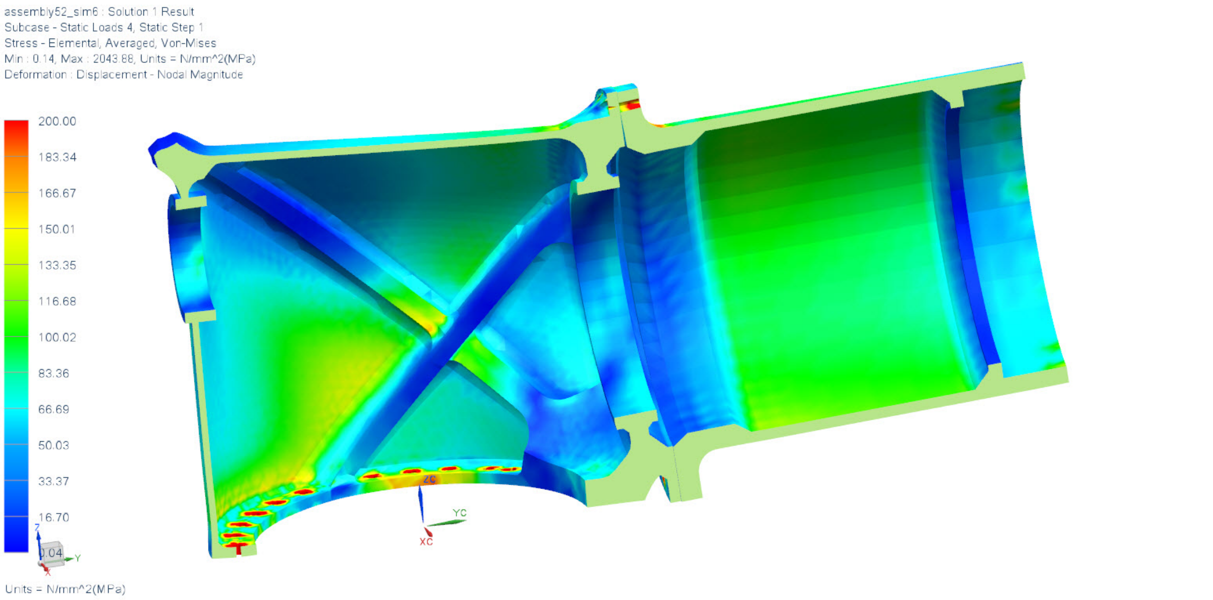

Figure 63: Load combination 4 from the other side, cut

6 Results Hub

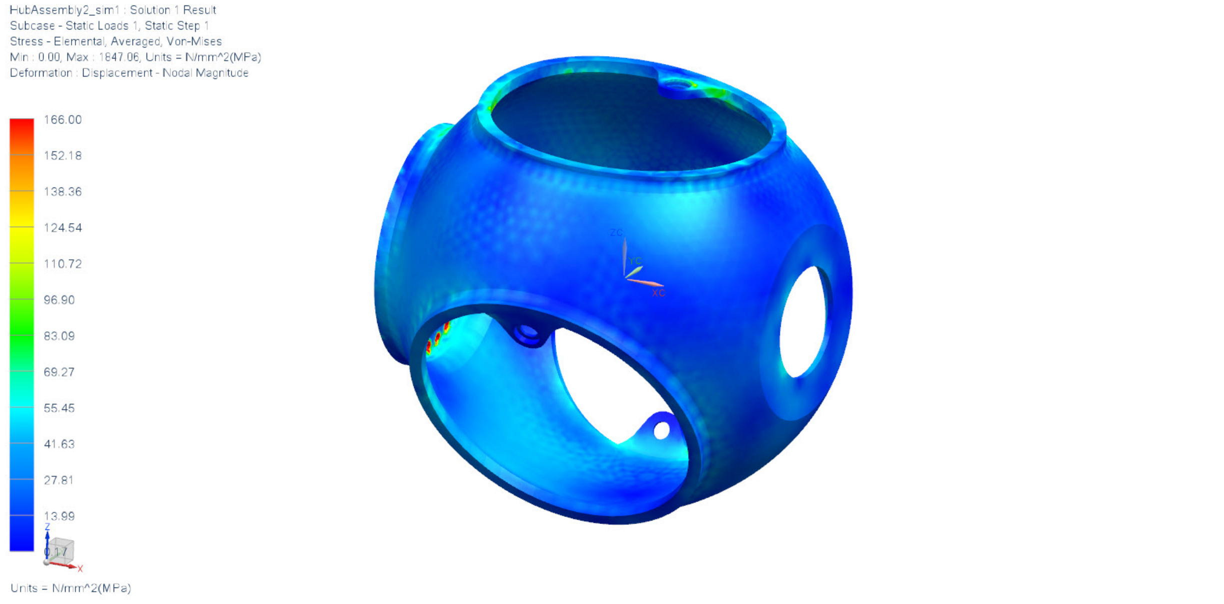

Figure 64: Stress distribution in the hub

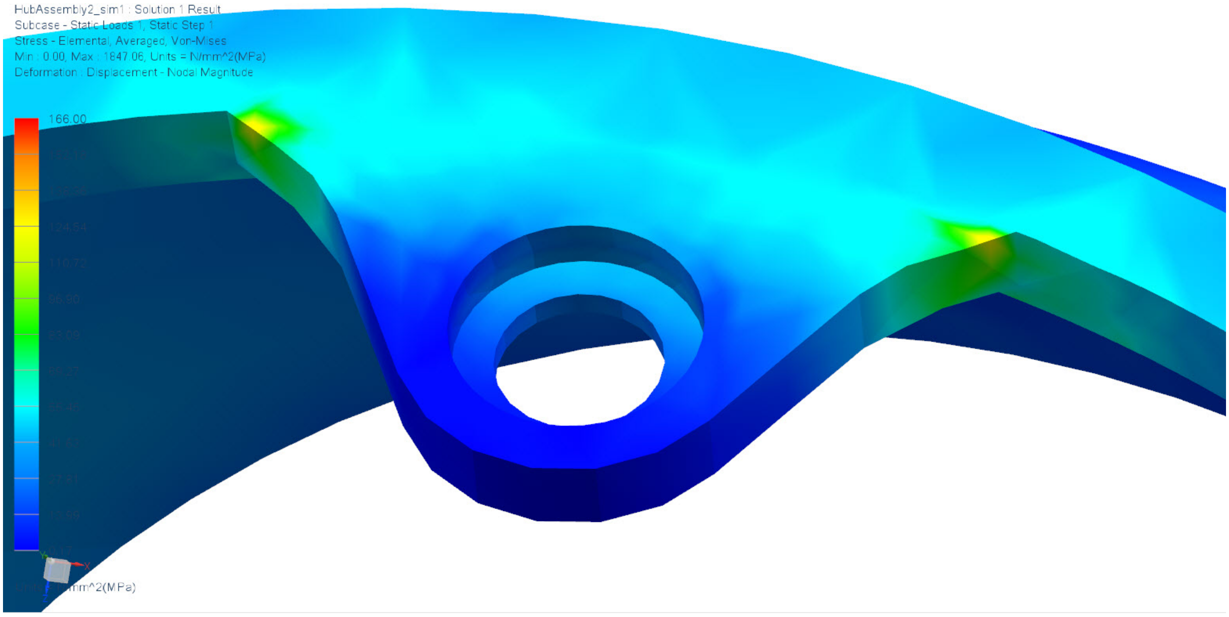

Figure 65: Stress at pitch motor mount, peak stress is 126MPa

7 Discussion

The most critical load state is load combination 1. This load combination gives three weak spots which needs to be considered. The first and worst weak spot is located where the right diagonal stiffener meets the bearing flange (fig. 66). More specifically, it's located on the outer edge of the transition. The stress here is calculated to be as high as 217.5MPa. There are several reasons for questioning its validity. First of all, this is only in one single node. The other nodes have values that are under 200MPa. The second thing is rough mesh quality in order to have an time efficient simulation. In fact, that transition consists of only one CTETRA10 Element. It would be reasonable to say that the peak stress here will be reduced if the mesh size is reduced.

The second weak spot is located at the transition radius between the nose part and its connecting flange (fig. 67). The peak stress here is calculated to be 203MPa which also appears in one single node. The stresses around shows right under 200MPa. If we take the average of the stresses in that area, the average would be under 200MPa. So this should in fact be considered safe.

The third (and most difficult to work with) is located on the transition radius on the stiffener flange, on the outside of the structure (fig. 68). This region shows a peak stress of 195MP, which is under the maximum limit. The reason for discussing this region is the difficulties it has given under the whole process. It has been that area that nearly always has given the highest stresses, because it's double curved. This further gives it higher stresses because of stress concentration, and because the difficulty associated to modeling such surfaces.

The second most critical load state is load combination 4. Only one area needs to be considered here. This region is also mentioned in the previous paragraph, and is located at the transition radius between the nose part and its connection flange (fig. 69). In this area, several nodes show values between 200-203MPa with 206MPa being the peak value in one node. So this is an area that should be slightly reinforced with the same method used on tower part's connecting flange. Other load combinations are not discussed in detail because their peak stresses slightly exceeds 100MPa only.

The reason behind load combination 1 being most critical is naturally because the combination of applied loads sums up favorably. But it is also important to remember that load combination 4 gives nearly the same stresses. In comparison, load combination 2 also gives nearly the same results as load combination 3. The difference between load combination 1 and 4, and 2 and 3, is how M_y is applied (table 4). M_y gives tension on the upper side of the structure in load combination 1 and 4, and the opposite in load combination 2 and 3. This means that the bed plate is much better to handle loads that gives compression on the upper side. This is in fact favorable, because it's more logical that the wind speed would be higher at the top of the rotor disk than at the bottom of it (which gives compression loads on top).
The most efficient part (in terms of stress unloading) of the bed plate structure, must be the diagonal stiffeners which really contributes to the total stiffness. They are so

efficient, that the back part of the structure should be considered to be profiled like them (explained better in "further work"). The final thing to mention is the average stress in the whole bed plate. With all load combinations considered, it's somewhere between 70MPa and 100MPa. From this, it is simple to understand that stress concentrations dominates dimensioning.
Considering the total mass, it is increased from ES's design with 17.2 tons. This is mainly due to the stiffness requirement when the structure is placed on the yaw bearing, but also because of bolt connections. Artificial stiffness is removed (mesh mating, and fixed constraint directly on the bearing flange), and higher stresses are shown. Most of the mass is added (in order to reduce stress) on the inside between RBS and bearing flange, and on the internal flange stiffener. Naturally, some mass has also gone into wall stiffening. The mass can be reduced if the rearmost part of the tower part is redesigned in such a way that it follows the lines of the diagonal stiffeners.
The last thing to be mentioned is the bearing flange stiffness. At the highest load (load combination 1), its outer edge deflects 3mm from the inner bearing ring. In theory, the flange shouldn't move at all. But what the actual allowable limit is, is not known. At least, the bearing doesn't lift directly under the bolt head which is the most important part.

The hub part doesn't show any major weaknesses and is in fact good from the stress point of view. The only part that has been modified is the pitch motor mounts. Before the modification, the stress here peaked at 190MPa, which is a bit over the limit. After the modification the stress went down to 126MPa, which is very good. Mass can actually be removed from the hub in order to make it more weight efficient. This can been seen by its average stress which is around 30MPa. A rough guess is that its mass can be lowered by at least one third of its current weight, as long as one takes care of the pitch motor mounting brackets. But before anything can be fully concluded, the loads from DLC 6.1 must be rechecked and verified.

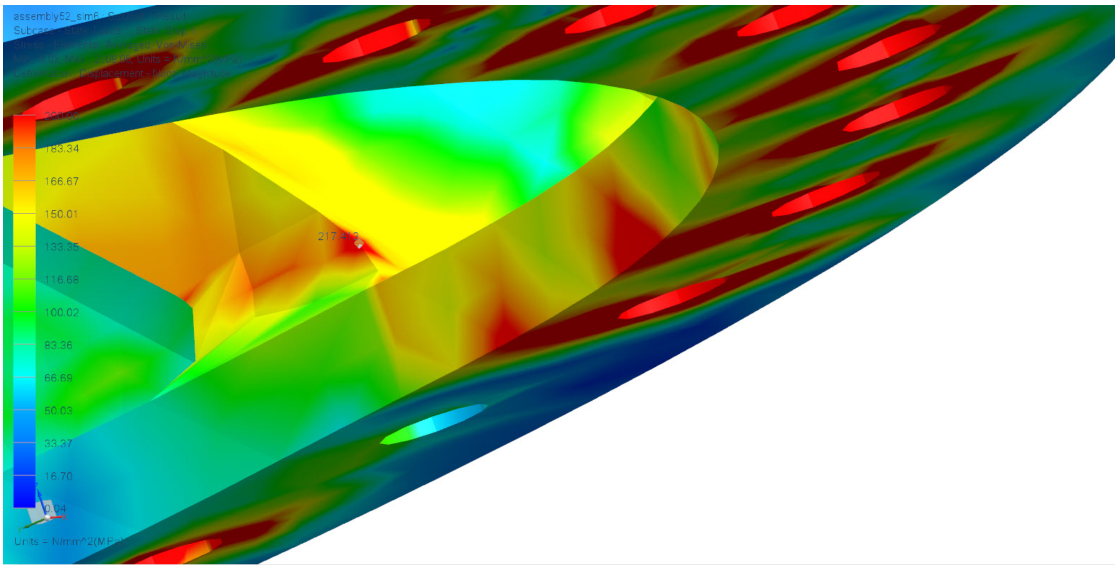

Figure 66: Load combination1, stiffener transition with 217.5MPa

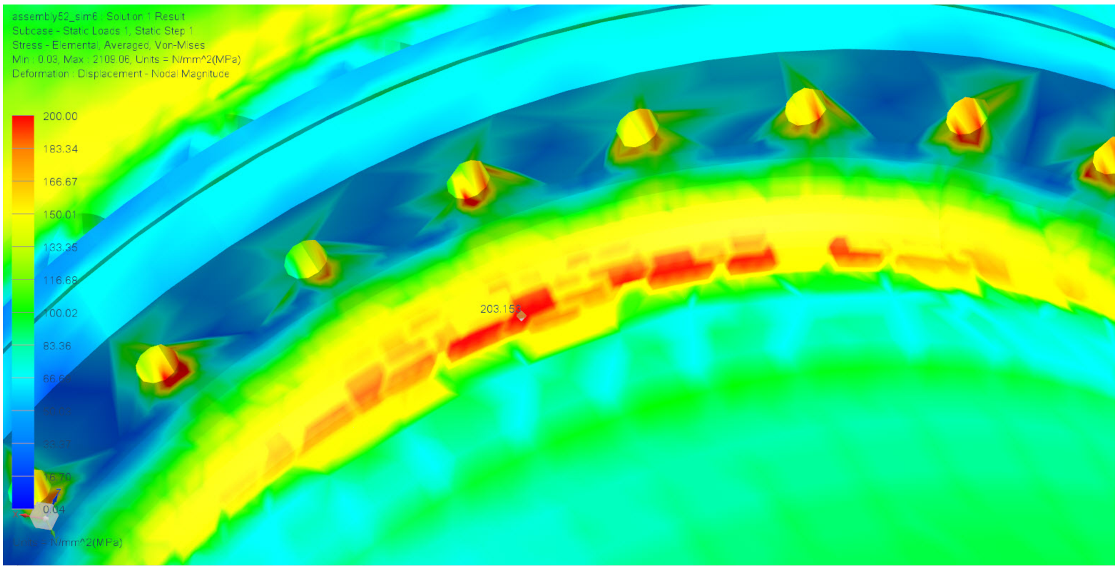

Figure 67: Load combination 1, 203MPa stress in flange radius

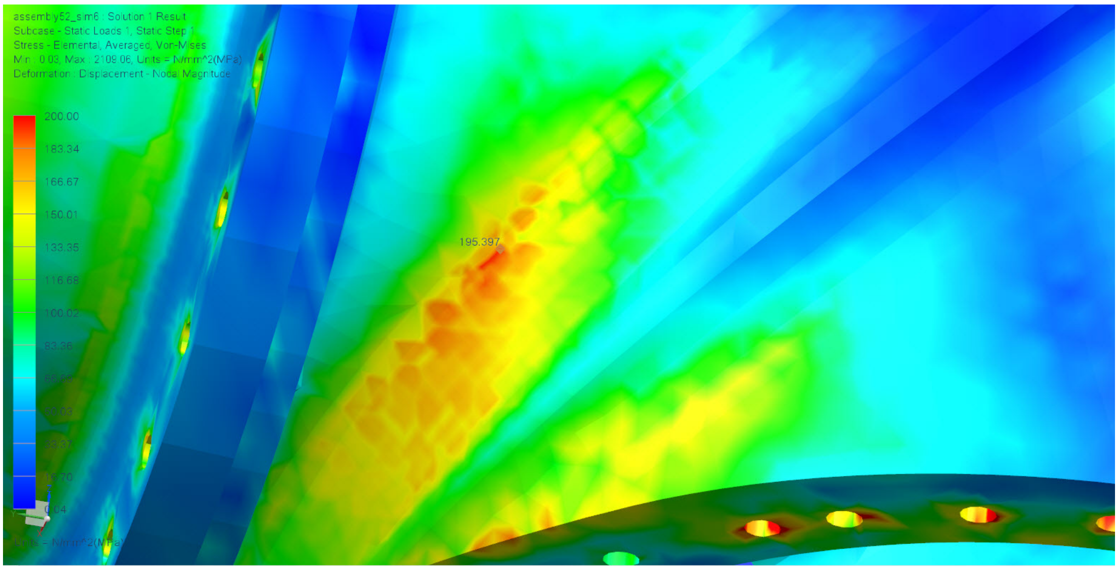

Figure 68: Load combination 1, 195MPa outer transition radius

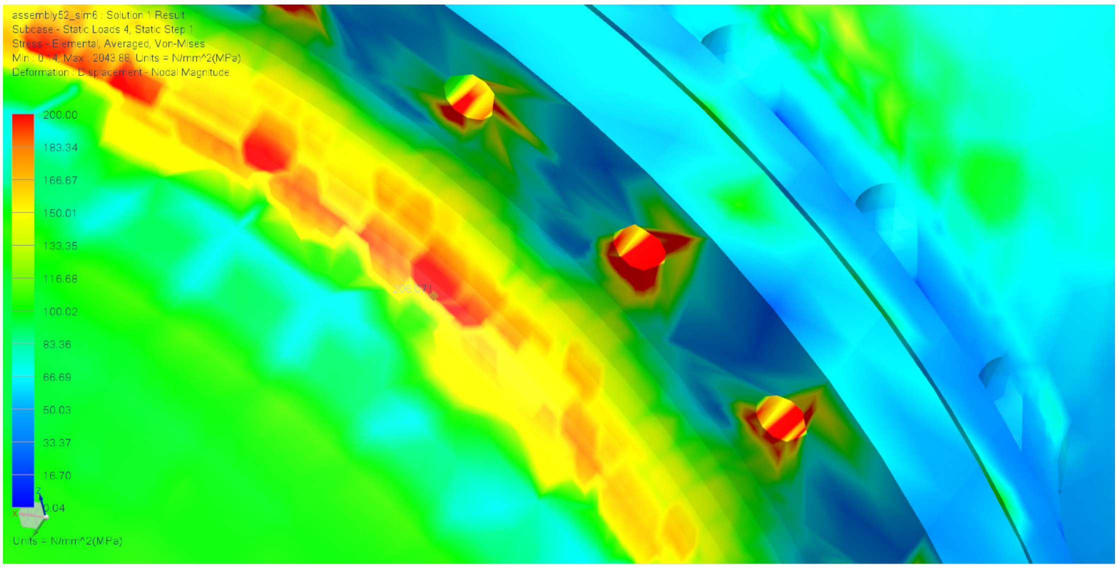

Figure 69: Load combination 4, 206MPa stress in flange radius

8 Conclusion

A bed plate and hub for a 10MW reference turbine with given geometrical constraints has been designed according to IEC 61400 for NOWITECH.
In general, the new bed plate has a good design in terms of load transfer. This is because the bed plate has contact all around the yaw bearing, which means it transfers load through the bearings complete circumference. This in fact means that the bearing is used more efficiently. It also has relatively simple geometry so it should be easier to manufacture than ES's design. The stresses represented in the model are also much more accurate compared to ES because the use of a bearing and bolt connections. The requirements on the other hand is a bit exceeded. The maximum stress is 217.5MPa, which is 17.5MPa above allowed stress. However, this is in an area that is subjected to some uncertainties in terms of element size. In terms of percentage, the mass was exceeded more. The mass of the tower part is exceeded with 17.2 tons. This was in a way expected because reduced stiffness in the boundary conditions means that the structure it self needs to be stiffer. The hub is dimensioned conservatively, but satisfies all requirements.

Personally, the author is satisfied and reached his main goal. And that was in the first place to understand and handle contact/bearing simulations and handle complex analyzes, but he also learned how to simulate bolted connections and got design experience as a bonus.

9 Further work

Further work that can be done is:

Redesign the rear part of the tower part as a 90 degree pipe bend instead of a straight pipe.

Do a geometry optimization on the tower part.

Do a new analysis on the hub with DLC6.1 with present boundary conditions and reduce its mass.

Do a fatigue analysis on both assemblies

References

[1] *http://www.evwind.es/2013/02/21/are-direct-drive-wind-turbines-the-future-of-wind-power/29555*

[2] *http://www.sintef.no/Projectweb/Nowitech/Organization/prosjekt*
http://www.sintef.no/Projectweb/Nowitech/Research-Activities/

[3] Erich Hau, *Wind Turbines*. Springer, 2nd Edition, Pages 81-85 2005.

[4] Erich Hau, *Wind Turbines*. Springer, 2nd Edition, Page 135 2005.

[5] Ebbe Berge Smith, *Design av Nacelle for en 10MW vindturbin*. NTNU, 2012.

[6] Mohammad Akram Khan, *Design of Rotor Hub for NOWITECH 10MW Reference Wind Turbine*. NTNU, 2012.

[7] Gunnar Härkegård, *Dimensjonering av maskindeler*. Tapir akademisk forlag, Chapter 9, 2004.

[8] *http://en.wikipedia.org/wiki/File:HoleForceLines.svg*

[9] *http://wind.nrel.gov/designcodes/simulators/fast/*

[10] *http://www.windfarmbop.com/tag/autocad/*

[11] *http://www.windpowersavings.com/how-do-wind-turbines-work/14/*

[12] *http://en.wikipedia.org/wiki/Wind_turbine*