Personal Project Portfolio

Neeraj Kumar Jha

FEA Lead Engineer

July 21, 2024

Contents

List of Figures

1 Introduction

In this portfolio I am showcasing few projects that I have worked on, with general understanding of physics and is intended to elaborate my skills and domains of expertise in structural analysis and product development of structural components.

2 Project 1: Fatigue hotspot mitigation

2.1 Problem definition

This problem concerns with fatigue mitigation for the component named Bottom cross (BC) in Rear Frame (RF), the Global model (GM) with position of BC and geometry of BC is shown in Figure 1 and Figure 2.

For fatigue evaluation I use cyclic loading in terms of linear accelerations, rotational velocities and rotational accelerations in X,Y,Z and apply these loads to the RF for evaluating the fatigue hotspots, the magnitude of these loads is in unit quantity and I call them Unit load case (ULC).

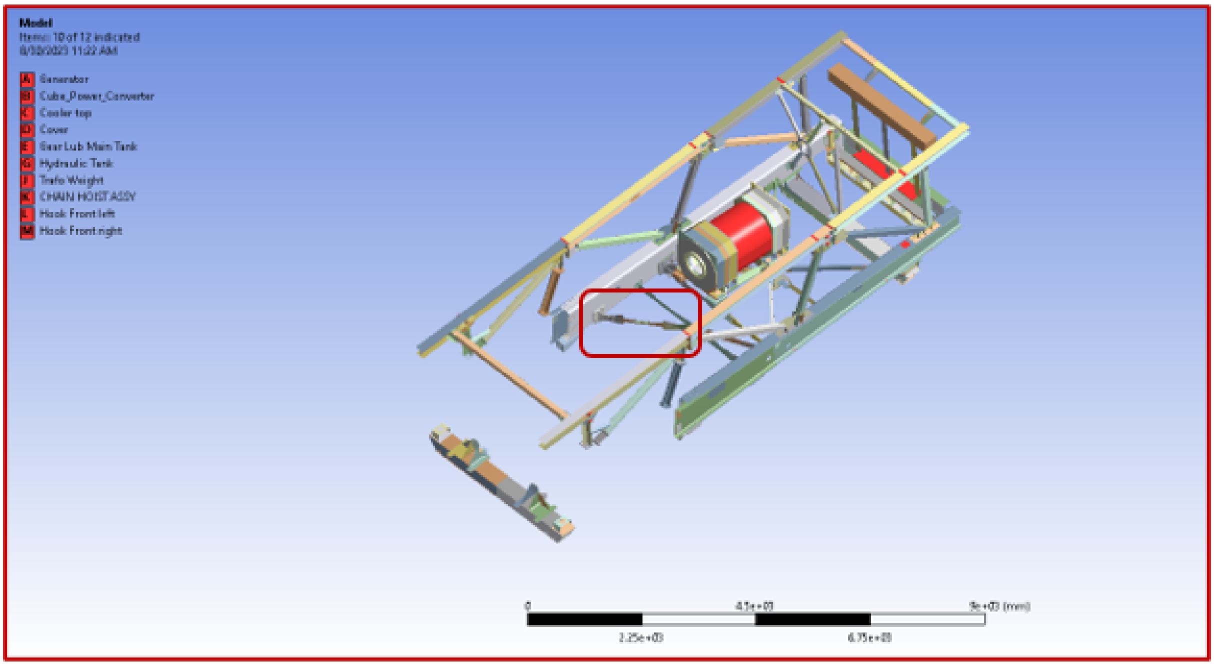


Figure 1: Position of bottom cross GM

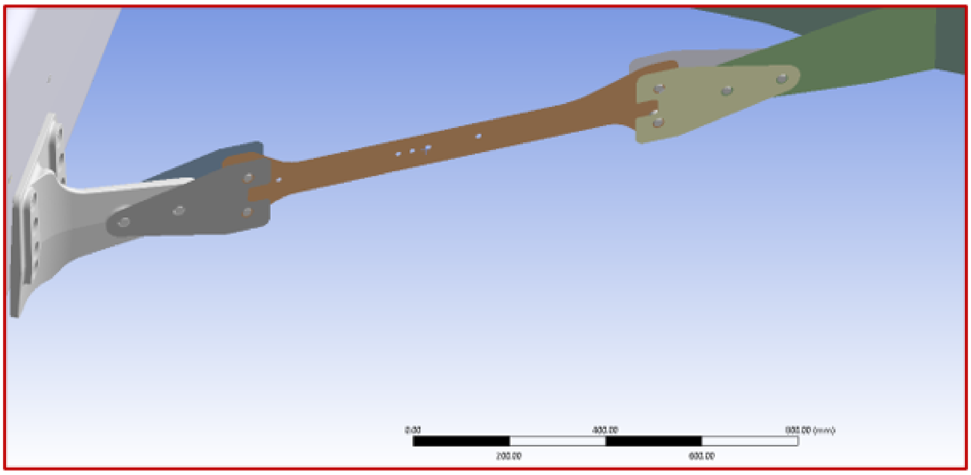


Figure 2: Geometry of Bottom cross

As the component goes into tension-compresssion cycles, there are chances that components might fail much before yield, hence it's very important to study the components in RF from fatigue perspective. In my personal experience, I can expain fatigue to anyone who has done cycling that they must have felt this tension-compression load in their legs and if they do it for a longer duration(cycles) then their legs feel that fatigue.

Coming back to the component of my interest BC, where I experienced hotspots in the ULC in GM. The SN curve data which has fatigue strengths on Y axis and Number of cycles on X, is shown below in Figure 3 and is taken from Euro code 3.

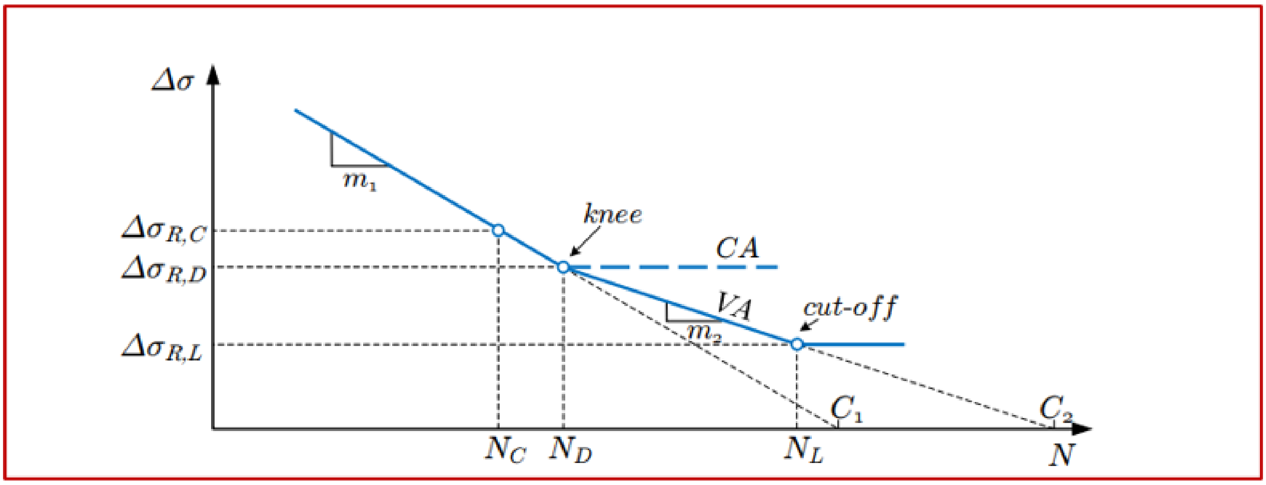


Figure 3: SN curve details

The stresses in terms of 6 tensors (3 Normal and 3 shear components) at nodes from the BC were extracted in FEA and for the maximum stress I calculated the fatigue life from the SN curve and a comparison with the number of load cycles, the data of which comes from sensors, gives me the damage value from Palmgren-Miner rule.

2.2 Flowchart of the process

The process of mitigating the fatigue hotspots starts with extraction of 6 tensors of stresses through a script, after running the ULC in FEA. The SN curve data is taken from Eurocode 3, from the maximum stresses on all the nodes extracted, I calculate the fatigue life in terms of number of cycles. From sensor data I get the load time series, then

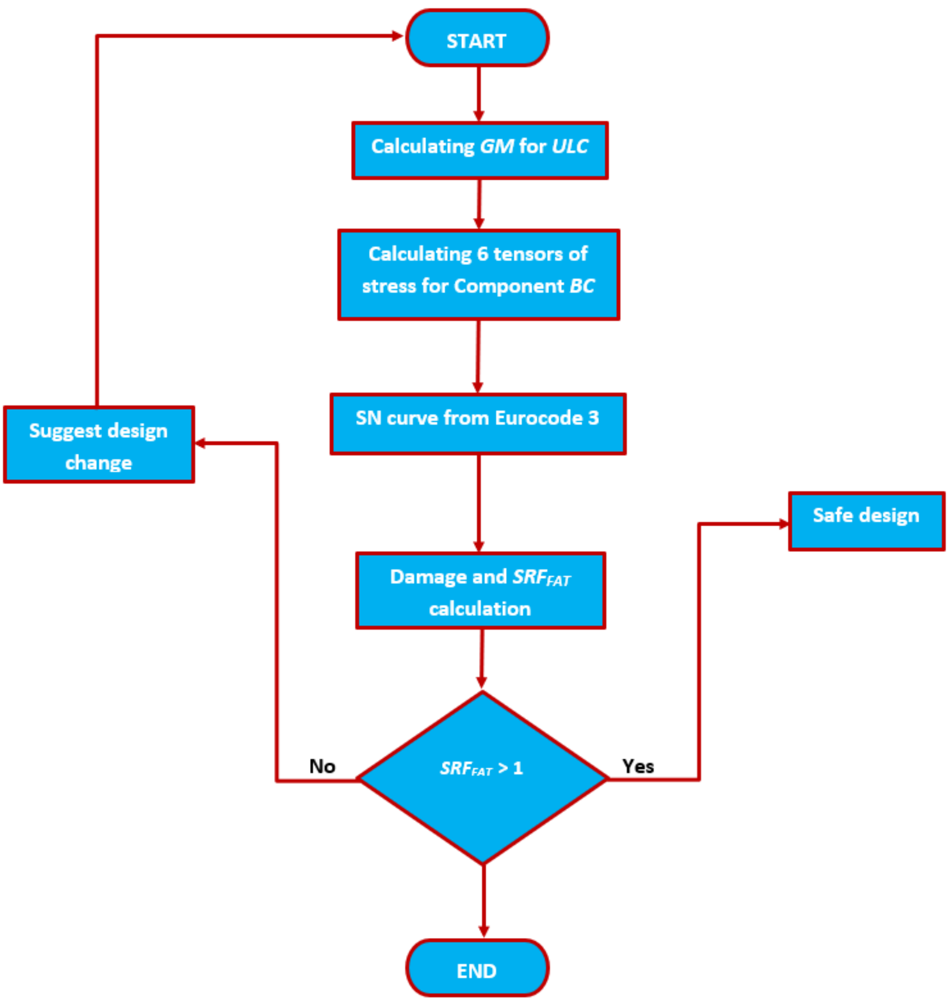


Figure 4: Fatigue mitigation process Flowchart

I take reference from Palmgren-Miner rule and compare the two to calculate fraction of life consumed by a stress cycle, this is also called as damage. Once I calculate the damage, I take inverse of it for all nodes to calculate for SRF_{FAT} (Stress reserve factor fatigue), the success criteria for the SRF_{FAT} is they must be all greater than 1, in case the SRF_{FAT} are below 1, I look for redesign and follow the same steps again, the process flowchart is shown in Figure 4 above.

2.3 Stress tensor extraction and SN curve

First of all, nodes were selected and given a named selection in FEA, for all these nodes, 6 stress tensors were extracted, 3 Normal and 3 shear stresses, the APDL script that I formulated. The SN curve data which gives the behavior of structural steel under a cyclic loading, in terms of number of cycles and fatigue strength, I take Detail Category (DC) curve from Euro-code 3.

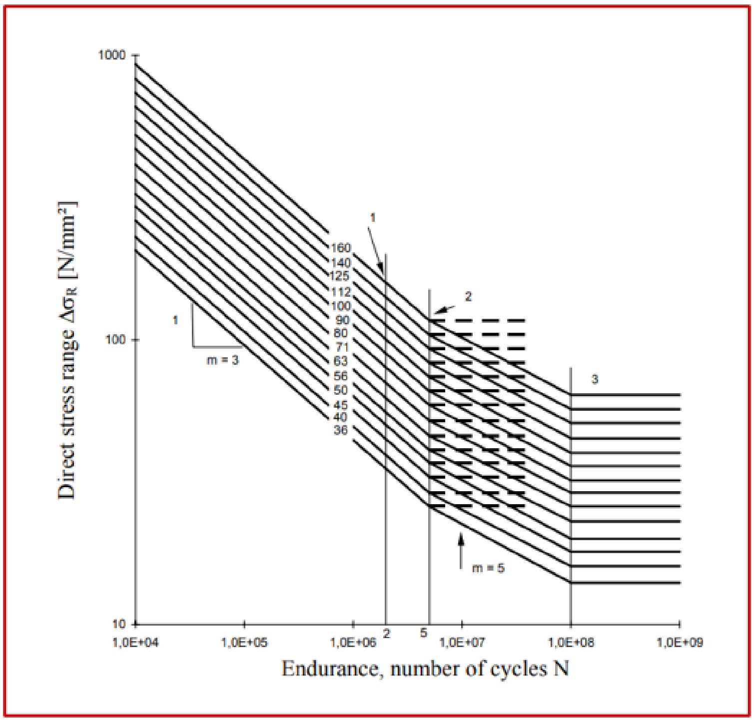


Figure 5: SN curve for different DC category

Since in the global model I saw that the hotspots were related to edges in BC, hence I took the SN curve for DC 125, this can also be seen with the SRF_{FAT} results shown below for BC in Figure 6, this image clearly shows that hotspots originating from slot edges.

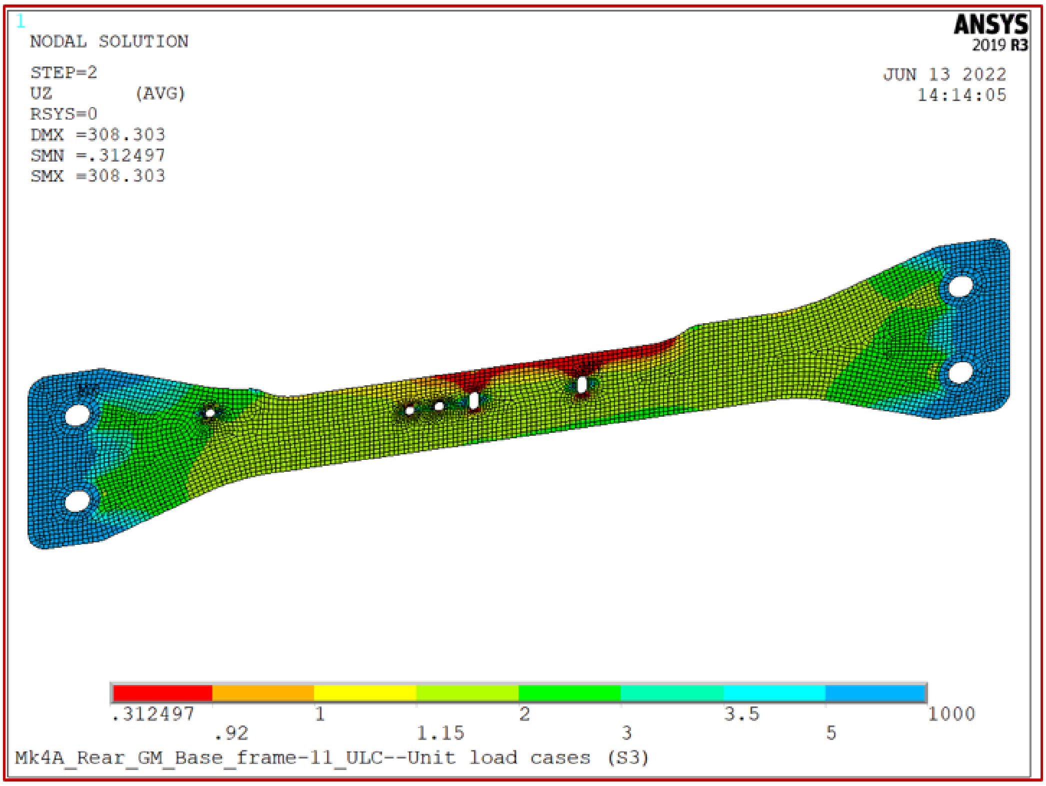


Figure 6: First iteration for Bottom cross

2.4 Relation between SN curve and Load-time series

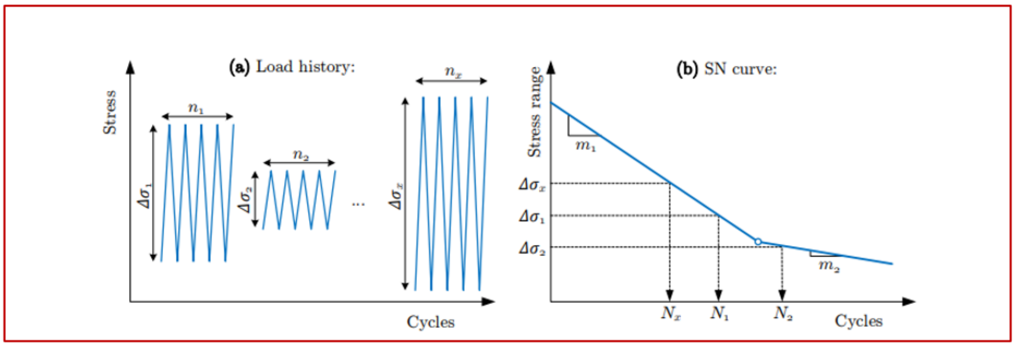


Figure 7: Load-time series and SN curve

After finalizing which SN curve data that I need to use, I needed to import the time series data from the sensor load files that I get from loads department, which they collect from sensors installed in Wind turbine. In Figure 6, the loads operating for a certain number of cycles is shown in left side, and for each of these stresses ($\Delta\sigma_x$) fatigue life is calculated in terms of number of cycles (N_x) from SN curve, shown in right side of Figure 6.

The cumulative damage (D) is calculated through Eq.1 shown below.

$$D = \frac{n_1}{N_1} + \frac{n_2}{N_2} + \frac{n_3}{N_3} + - - - \frac{n_x}{N_x} \tag{1}$$

Failure is assumed when the damage sum crosses 1, and I follwed this as success criteria. In vestas, I needed to inverse this damage and found out SRF_{FAT} and it has to be above 1 for all nodes. In further sections, I will put forth the iterations performed and results for fatigue mitigation in terms of SRF_{FAT}.

2.5 Design Iterations for Bottom cross (BC)

In this section I will explain the various design iterations and how this fatigue hotspot was mitigated, from Figure 5, it can be seen that these hotspots are related to edges of the slots, so the obvious choice for me was to reduce the slot sizes in vertical direction and use oversized holes instead. First what I tried was provide some extra material towards

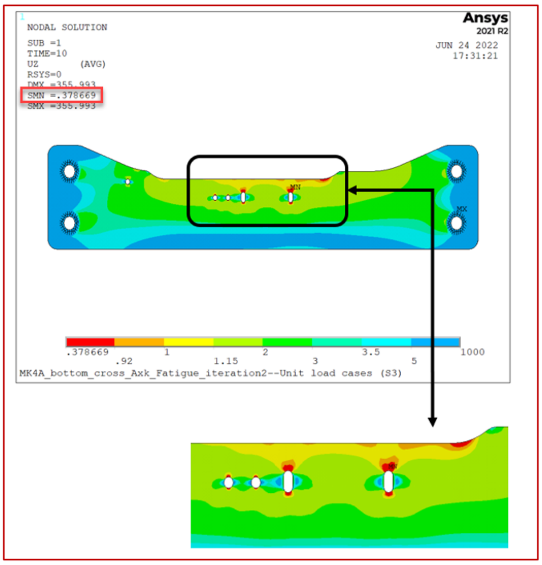


Figure 8: Second iteration for Bottom cross

the edge, and evaluate if there was any improvement, without changing the slots, this is shown in Figure 7 below. Even in this case the SRF_{FAT} was below 1 and hence not safe.

In third iteration, I removed the slots and used oversized holes and calculated again to see if the hotspots reduce, this is shown below in Figure 9.

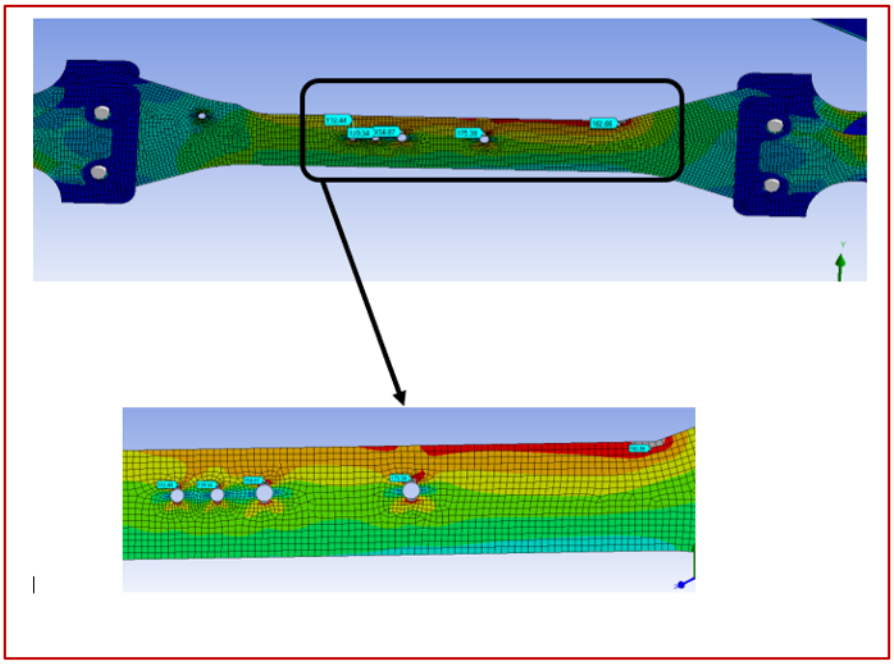


Figure 9: Third iteration for Bottom cross

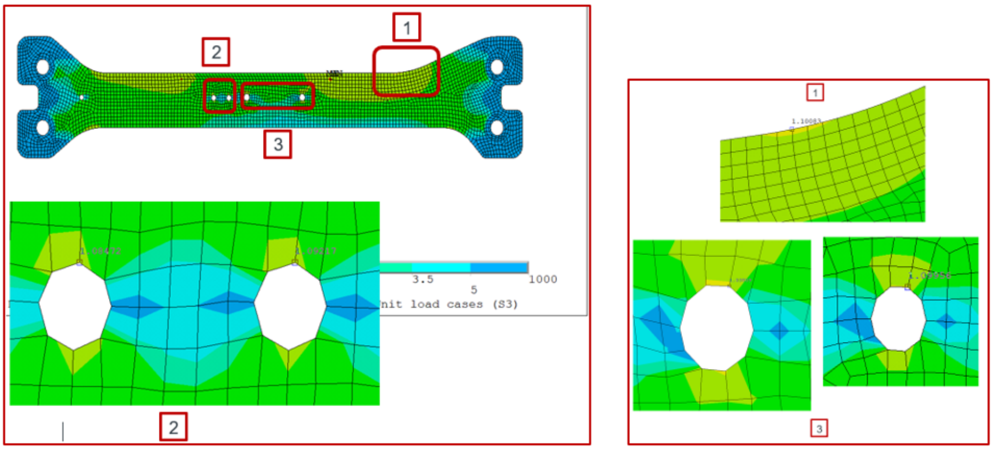


Figure 10: Fourth iteration Bottom cross

Even after 3rd iteration the fatigues hotspots were not mitigated, so in 4th iteration, I increased the plate thickness from 30mm to 40mm and re-calculated under ULC, the result for this is shown below in Figure 9, where the SRF_{FAT} are shown through probes and all SRF_{FAT} were found to be above 1.
So all the hotspots were mitigated for the Bottom cross (BC) in fatigue by redesigning the component and FEA simulation.

2.6 Summary

The fatigue mitigation project demanded skills and knowledge in the field of fatigue, in fatigue components can fail much below yield limit, and it's very difficult to detect with any pointers before the fracture happens, hence it's pertinent to have the components calculated in FEA before making the prototype.
The fatigue mitigation task involved changing the design of the BC, so as to improve the hotspots, this project was interesting in the sense that I got a chance to develop the component from scratch and do design as well as FEA.
The final results of the fatigue mitigation for BC were part of Design verification report (DVRE), I also regularly discussed with my team lead on the FEA, calculation results and DVRE, this report was used for getting certification from the DNV for the new turbine. My manager gave me this work, as I had earlier design and development experience and along-with my FEA knowledge I was able to mitigate the fatigue hotspots.

3 Project 2: Weld extrapolation for toe failure

3.1 Problem definition

This problem concerns with weld extrapolation for welded components which showed hotspots near the toe or the root of the weld in the Rear Frame (RF) of Nacelle in a Wind Turbine Generator (WTG). The Global model for the RF is shown in Figure 1 below. As can be seen the RF consists of many welded components which needs validation through FEA under various LCs.

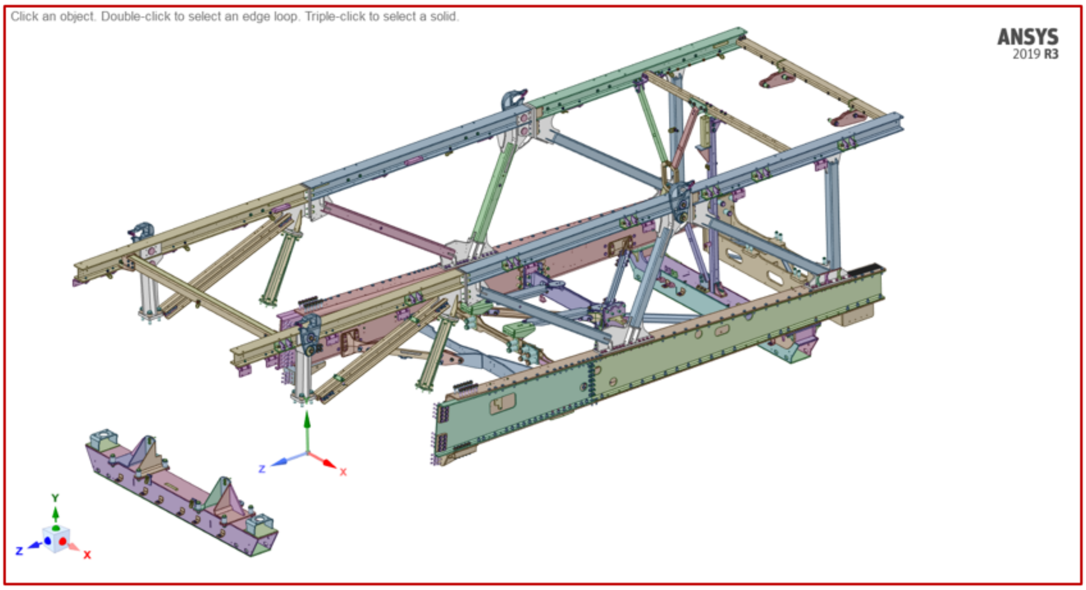


Figure 11: Global model GM

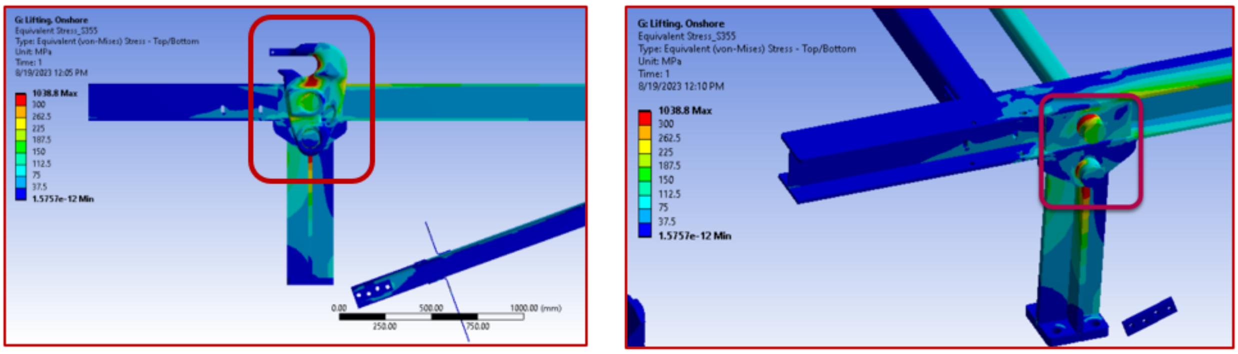


Figure 12: Detail of the hotspot around lifting pins

I work in the Finite Element Analysis (FEA) of the RF components as Lead Engineer, where I have to analyse the GM for various LCs, the LCs are mainly decided on what load would be coming to these components during operation, handling and during the service lifetime of the WTG.

Lifting scenarios (onshore and offshore), transport loads while handling, wind loads during operation, fatigue (tension-compression cycle) and other extreme LCs are simulated in FEA, these loads are applied in terms of accelerations to the RF components.

Von-Mises stress plot for the Lifting onshore LC near front hook is shown in Figure 12, details of the hotspot is also shown in Figure 12, I needed to perform this first-hand screening for all LCs, for the welds in the GM, and if I observe any hotspots then I go into more detailed study like weld extrapolation for root or toe failure check for each of these hotspots at submodel level.

I extrapolate the stresses as per Hot-spot stress method (HSS), given in IIW for toe failure check or J.D.Sørensen method for root failure check.

3.2 Flowchart of the process

The weld extrapolation task involves calculation for the GM in all LCs and then evaluating weld related hotspots in terms of stresses, once I see any hotspots in after running the GM, I would prepare a sub-model with solid plates and welds with exact representation.

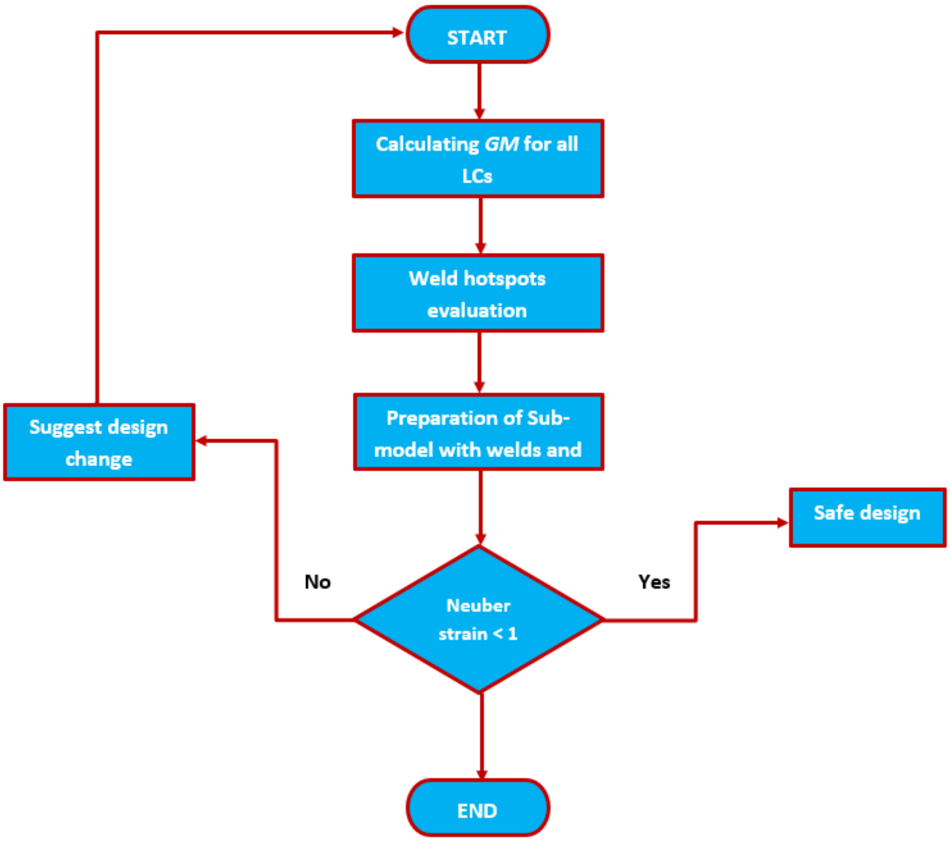


Figure 13: Weld extrapolation Process Flowchart

So, if there is a fillet weld or penetration weld the FEA model contains both, also the GM is a shell model, but the sub-model has solid plates in-order to be more accurate. A flowchart of the process followed by me is given in Figure 13.

3.3 Weld extrapolation for Toe failure

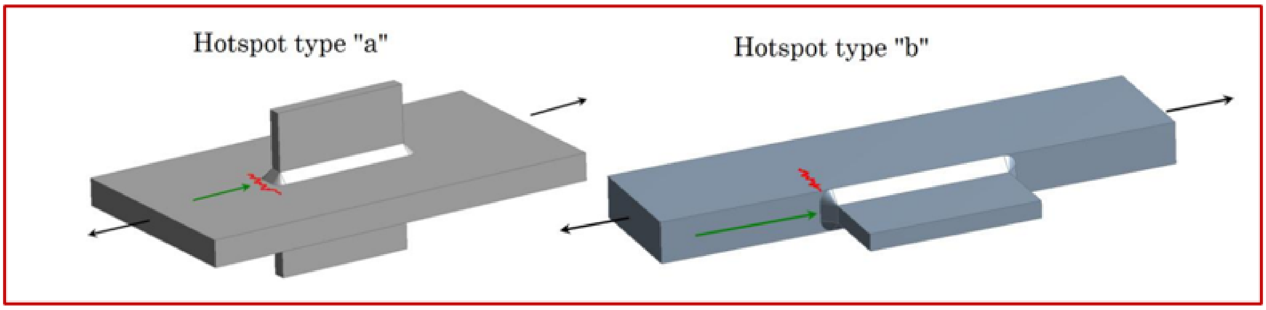


Figure 14: Type "a" and type "b" classification by IIW

Once I simulate for a particular LC, and using proper contraints in the FEA, I observe for weld related hotspots, something which is shown above in Figure 14.

I first evaluated which kind of weld failure the component can go through either toe or root failure.

The toe failure given by HSS method, the type"a" and type"b" hotspot classification comes from where is the hotspot and what kind of crack it can induce, if the hotspot can induce a crack on the weld toe it's type "a" and if the hotspot can induce the crack on the plate then it's type "b", shown in Figure 14.

The stresses calculated at weld toe can increase infinitely as I reduce the mesh size, hence it's always advisable to extrapolate stresses away from the weld. For type a extrapolation thickness of the plate comes into picture and I define two paths one at 0.4·t and another at 1·t distances from the toe of the weld, for type "b" the paths are independent of plate thickness and are defined at 4mm, 8mm and 12mm from the toe of the weld. The two extrapolation techniques are also shown below in Figure 15.

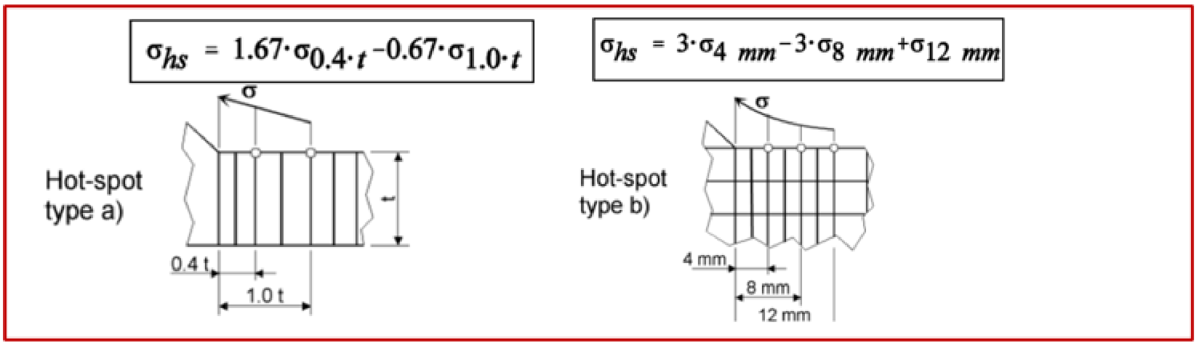


Figure 15: Type "a" and type "b" extrapolation

3.4 Weld extrapolation for root failure

The weld root failure is more common in fillet welds and I usually recommend designers to go for penetration weld where toe failure would be more common, the reason being that toe failure is easily inspectable whereas the root failure is not.

For some places in GM, where I see fillet welds and thickness of plates is not much, there I extrapolate for root failure. The method for root failure extrapolation is given by

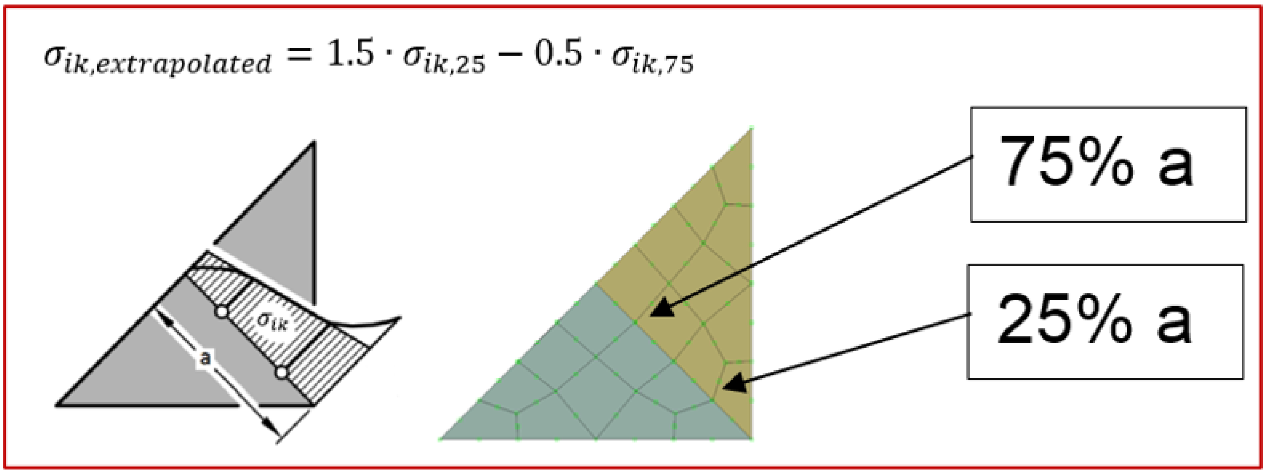


Figure 16: Extrapolation for root failure

Sørensen, J. D. and is also shown in Figure 16.

3.5 Neuber strain

Once I have the maximum principal stresses calculated along the predefined paths, I calculate the Neuber strains for the maximum extrapolated stress. Neuber strain is a linear way to calculate for strains even when components go into plasticity, it's a linear way in which I don't need to introduce material non-linearity in the FEA model, the success criterion in this case is Neuber strains < 1%.

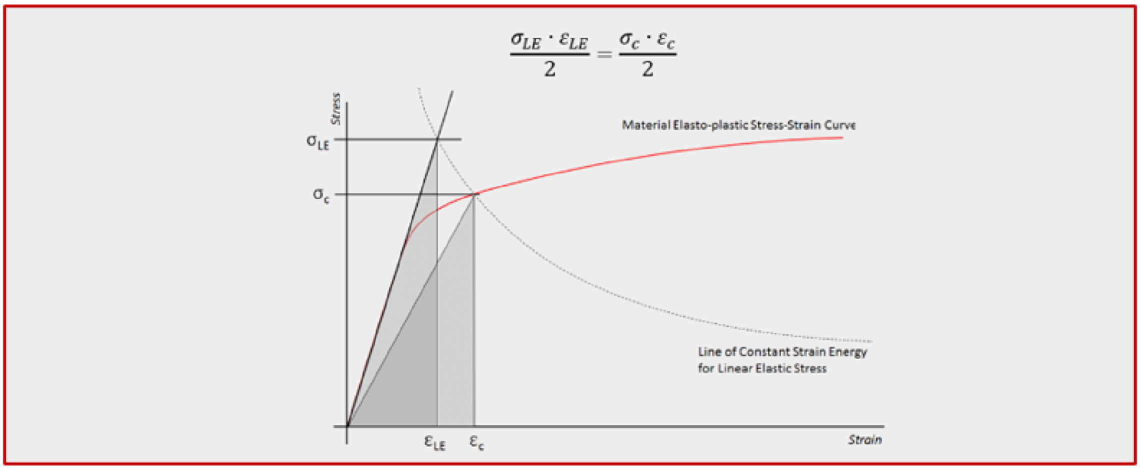


Figure 17: Neuber strain

The idea of Neuber strain is shown below in Figure 17, where strain energies in elasto-plastic and the linear case are equated.

Once I calculate the Neuber strain for maximum extrapolated stress then the strain must be below 1%, then only I consider it safe. If the strains are higher than 1% then I suggest for redesign in terms of weld sizes or changing the plates stiffness.
Below in Figure 18, the hotspot in the GM is shown which was calculated for lifting LC, once I observed the hotspot at the plate joints and pin joints, then I prepared a submodel with solid members and fillet welds, details of the fillet welds were taken from the manufacturing drawing.

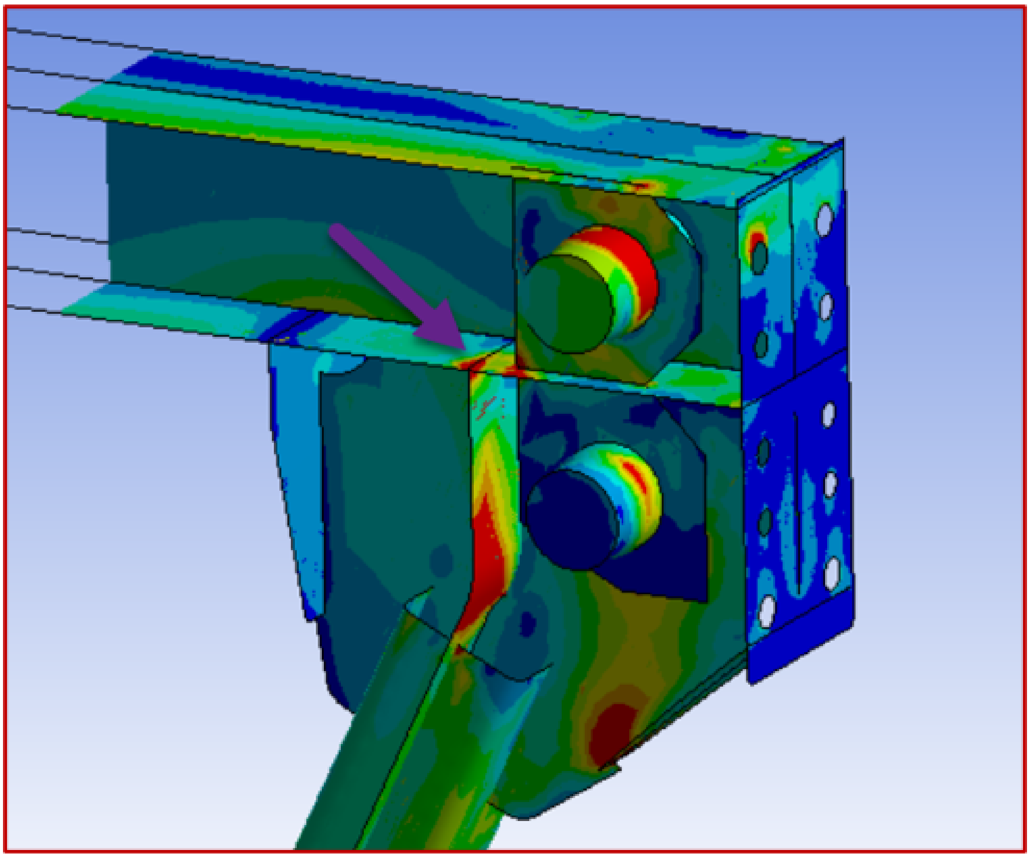


Figure 18: Weld hotspot near plates in GM

The sub-model with the actual fillet welds and solid plates in shown in Figure 19, the paths for weld stress extrapolation is shown in Figure 19 with the help of arrows, as this was type "a", hence I made splits at 0.4·t and 1·t from the toe of the weld and extrapolated for maximum principal stresses along these paths.

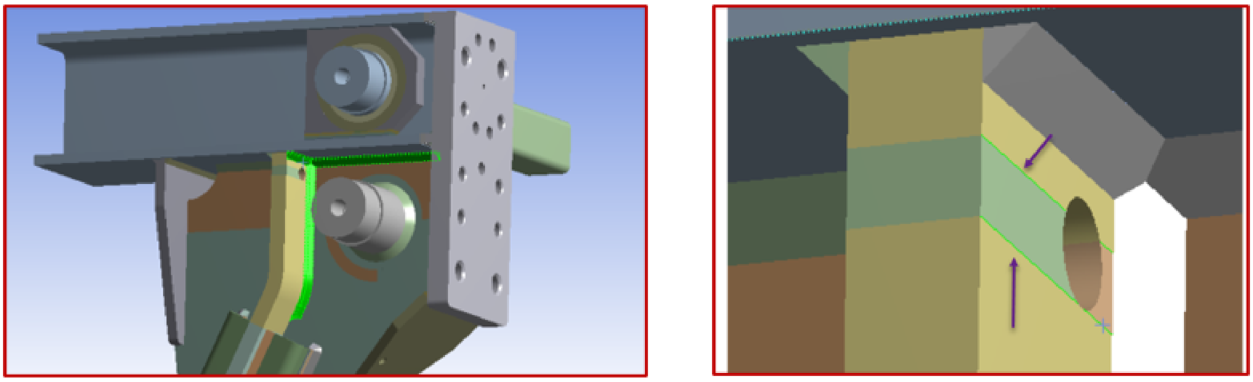
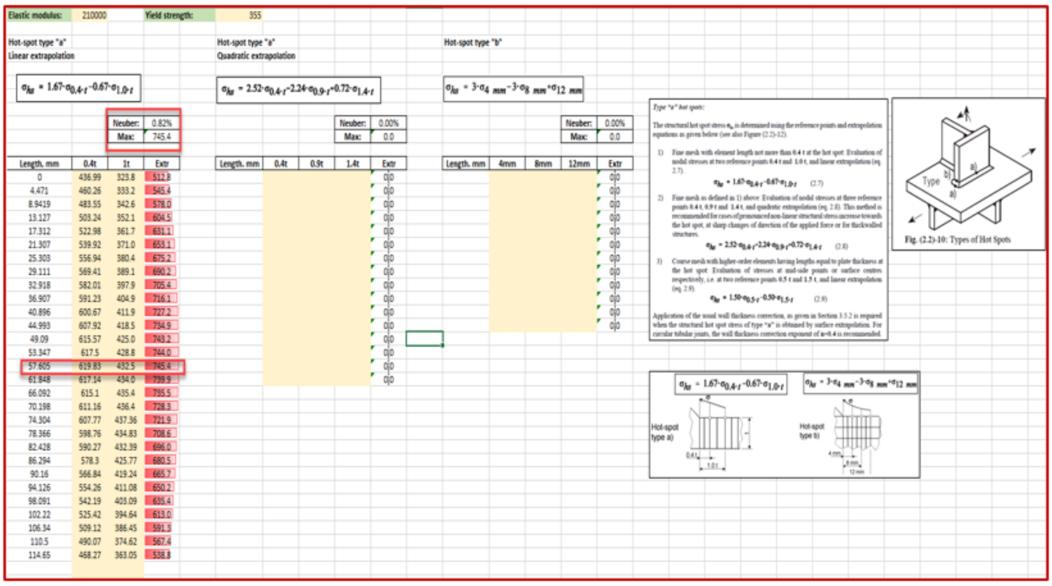


Figure 19: Fillet welds and extrapolation paths

Figure 20, below shows the extrapolation sheet where I put all the extrapolated principal stresses along 0.4·t and 1·t and calculating for Neuber strain with maximum principal stress and checking if it crosses 1% limiting value.



Length, mm	0.4t	1t	Extr
0	436.99	323.8	512.8
4.471	460.26	333.2	545.4
8.9419	483.55	342.6	578.0
13.127	503.24	352.1	604.5
17.312	522.98	361.7	631.1
21.307	539.92	371.0	653.1
25.303	556.94	380.4	675.2
29.111	569.41	389.1	690.2
32.918	582.01	397.9	705.4
36.907	591.23	404.9	716.1
40.896	600.67	411.9	727.2
44.993	607.92	418.5	734.9
49.09	615.57	425.0	743.2
53.347	617.5	428.8	744.0
57.605	619.83	432.5	745.4
61.848	617.14	434.0	739.9
66.092	615.1	435.4	735.5
70.198	611.16	436.4	728.3
74.304	607.77	437.36	721.9
78.366	598.76	434.83	708.6
82.428	590.27	432.39	696.0
86.294	578.3	425.77	680.5
90.16	566.84	419.24	665.7
94.126	554.26	411.08	650.2
98.091	542.19	403.09	635.4
102.22	525.42	394.64	613.0
106.34	509.12	386.45	591.3
110.5	490.07	374.62	567.4
114.65	468.27	363.05	538.8

Figure 20: Calculation spreadsheet

3.6 Summary

This project for weld extrapolation was given to me by my manager, who saw capability in me to solve this. Preparing the analytical spreadsheet, the GM filtering for weld hotspots, the extrapolations in FEA sub-model, was a lengthy and challenging task. To get the model close to accurate I had modelled all things as it was in real.

This work was documented in the form of a Design verification report (DVRE), which goes for certification from bodies like DNV, preparation of this report was also my task, I regularly discussed with my team lead about the FEA results and DVRE. As these tasks where something that I really enjoyed, so most of the challenges I faced were mitigated by looking through more standards.

Whenever I saw some welds were not passing, then I would talk to the designer for a possible change in the type, size of the welds and if there was any possibility for a design change in the plates itself.

4 Bolted joint verification

4.1 Problem definition

This problem concerns with bolted joints calculation for the Rear Frame of Nacelle in a Wind Turbine Generator. Nacelle structure is mainly meant to carry the gearbox, the generator, along-with other electrical components, and to mount these components you need some supporting structure.

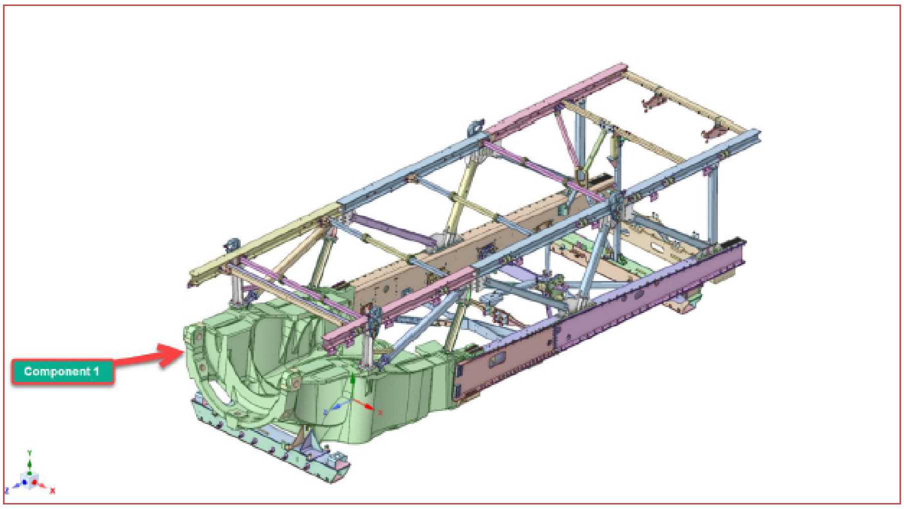


Figure 21: Nacelle RearFrame

We in WTG industry divide these supporting structures into two main categories, structure that supports gearbox is called a Base-Frame (Component 1 in Figure 21), which is mainly a casted component. To support the generator, transformer, control cabinets, towards the rear side of the nacelle, we use a structure that has welded and bolted components, we call this structure as Rear Frame (RF) Figure 22.

I work in the Finite element analysis (FEA) of the RF components as Lead Engineer, one of the first step was to reduce the solid RF model to shell and beams, in-order to reduce the size of the problem. For different Load Cases (LCs), the bolted joints were calculated for slip and tension by me.

The LCs are mainly decided on what load would be coming to these components during operation, handling and during the lifetime of the WTG. Lifting scenarios (onshore and

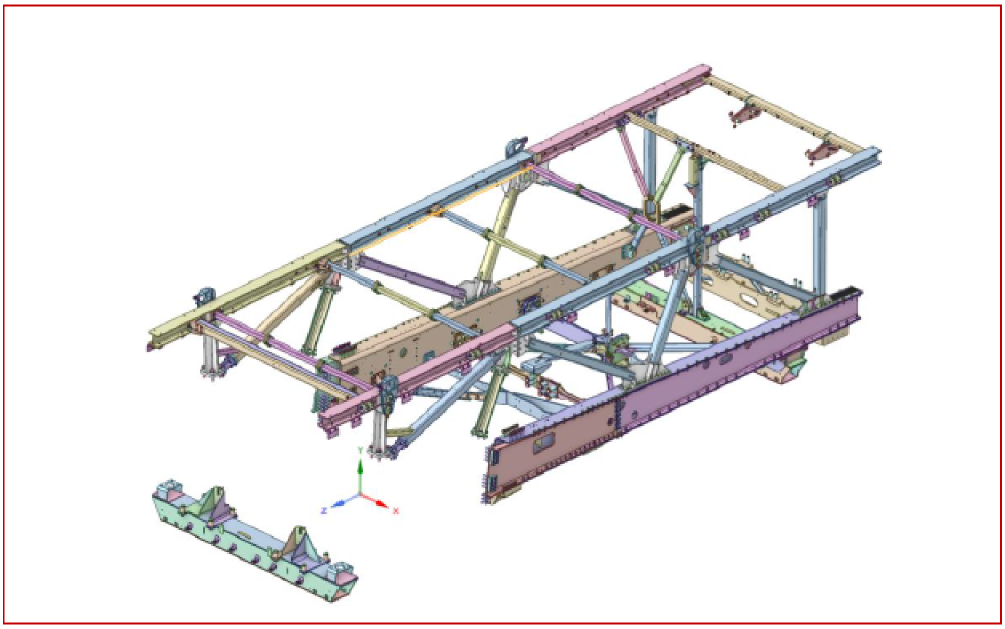


Figure 22: Nacelle RearFrame

offshore), transport loads while handling, wind loads during operation, and other extreme LCs are simulated in FEA, these loads are applied in terms of accelerations to the RF components.

4.2 Flowchart of the process

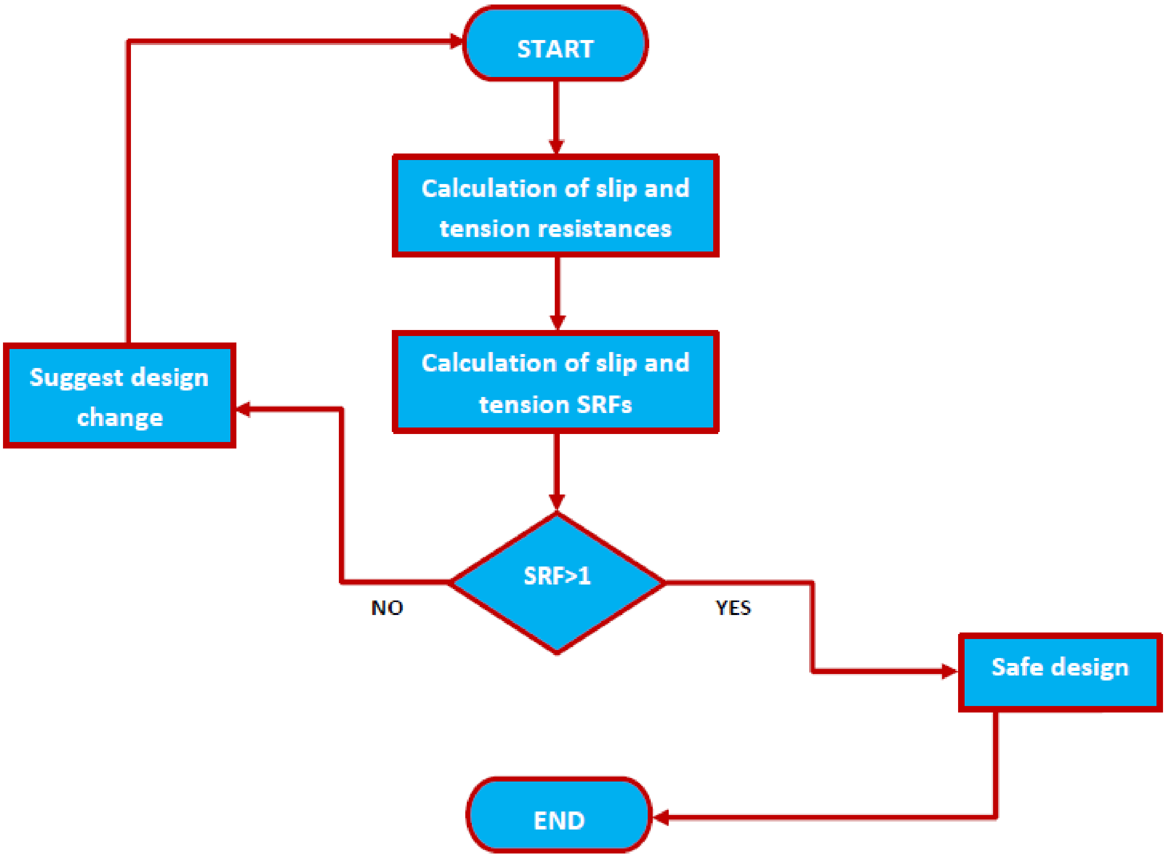


Figure 23: Process Flowchart

The bolt verification task involves calculation for shear and axial forces in the bolts modelled as beams in the FEA, from different LC, then taking these forces and analytically calculating for the slip and tension resistance check for all these bolts in RF.
The resistances were made non-dimensional by comparing them with the tension and shear forces and that parameter we call as Safety reserve factor (SRF). A flowchart of the process followed is given in Figure 23.

4.3 Beam force extraction

Once we simulate for a particular LC, and using proper contraints in the FEA, we can extract the forces through beam probe or scripts. The extraction gives us the shear and axial forces which are starting points for our calculation. Figure 24 shows the position for one such beam force extraction, and Figure 25 shows the axial and shear forces extracted.

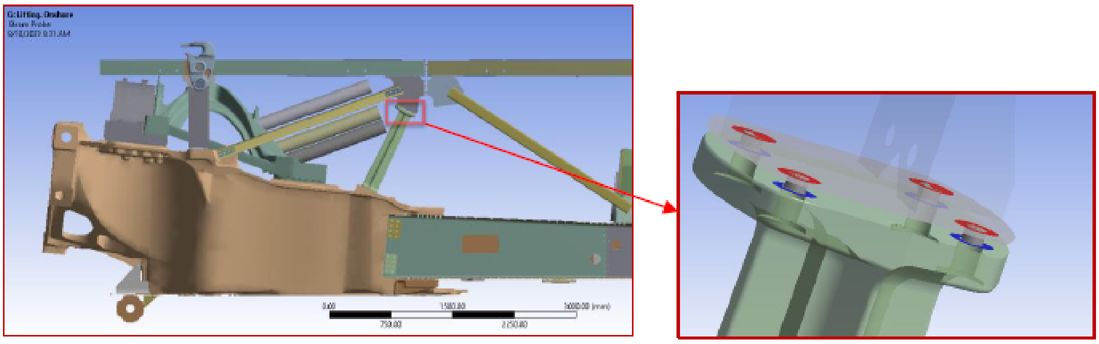


Figure 24: Beam connection shown

Definition	
Type	Beam Probe
Boundary Condition	G1010101. 3|Rear frame)
Suppressed	No
Options	
Result Selection	All
Display Time	End Time
Results	
Maximum Value Over Time	
Axial Force	28836 N
Torque	-28343 N·mm
Shear Force At I	8487.7 N
Shear Force At J	8487.7 N
Moment At I	7.4826e+005 N·mm
Moment At J	7.1255e+005 N·mm

Figure 25: Axial and shear force from Beam

4.4 Calculation of the Joint stiffness factor/Load factor Φ

After I got the forces, the next important parameter was Load factor (Φ), which speaks about the portion of total axial force coming to the bolts during the operation.
So, once I know this share, I could move to the next step of reducing the clamping force by this magnitude of force coming to the plates in order to understand the remaining clamp force, this is also shown below in Eq. 2.

$$\Phi = \frac{F_{SA}}{F_A} \tag{2}$$

Where,

F_{SA} is the force coming to the Bolt

F_A is the axial force on the joint (From beam force extraction)

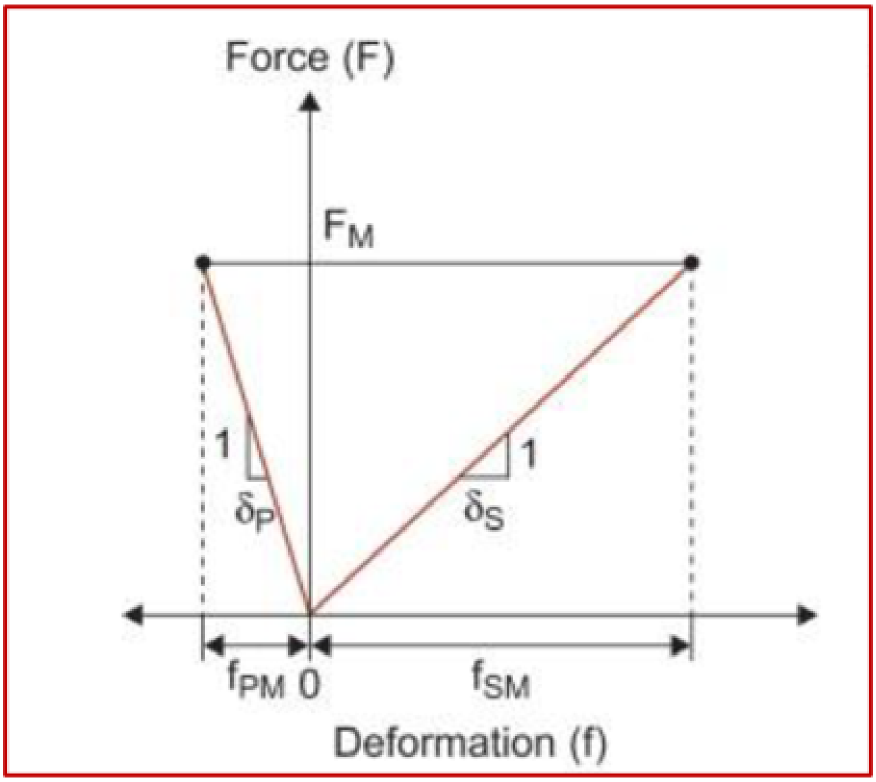


Figure 26: Bolt joint loading diagram under preload

The joint loading diagram (Figure 27) comes from VDI 2230 part 1, it gives the blueprint for the calculation and understanding of how joints behave when in assembly. In this diagram, Force Vs Deformation graph is plotted, FM is the preload on the members and drawn in absence of an axial force.

The left portion of this graph talks about plate which would be under compression, and the right side talks about the bolts under tensile loading. The positive deformation for the bolts (f_{SM}) and negative deformation for the plates (f_{PM}) are represented in this diagram.

Figure 28 shows the joint diagram of members under axial tensile load (F_A) coming to the assembly.

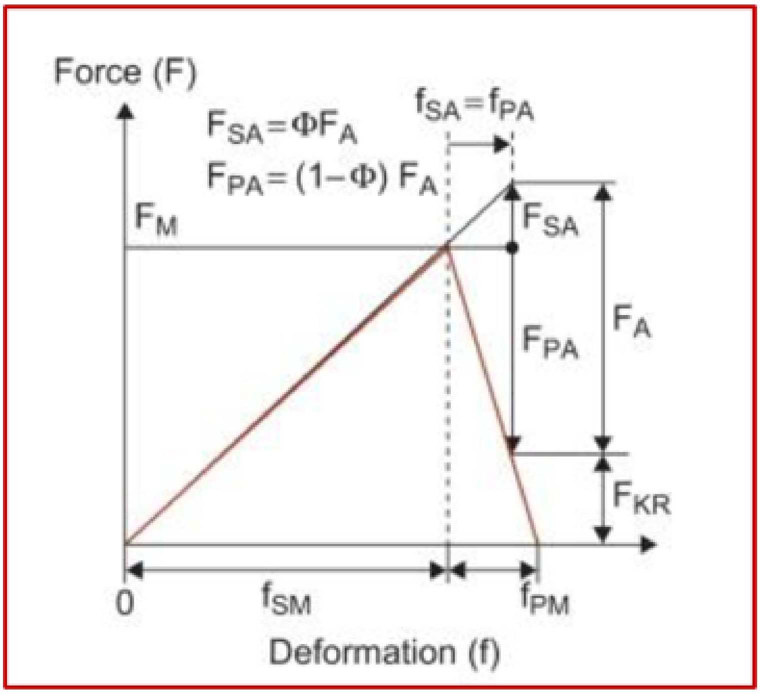


Figure 27: Bolt joint diagram under axial load

This load is shared by the bolts and the plates and mathematically can be written as:

$$F_A = F_{SA} + F_{PA} \tag{3}$$

$$\Phi = \frac{F_{SA}}{F_A} = \frac{K_b}{(K_b + \ K_{m)}} \tag{4}$$

Where,

K_b is the stiffness of the bolts

K_m is the stiffness of the member (plates)

The bolt stiffness would come from it's tensile stress area, it's nominal diameter and it's length, calculation of which is quite similar to that of a spring in tension.
On the other hand, for the members (plates), the stiffness is calculated in an integral sense, as clamp area keep on changing from the bolt head to the nut, this clamping area is in the form of a truncated cone, shown in Figure 29 below.

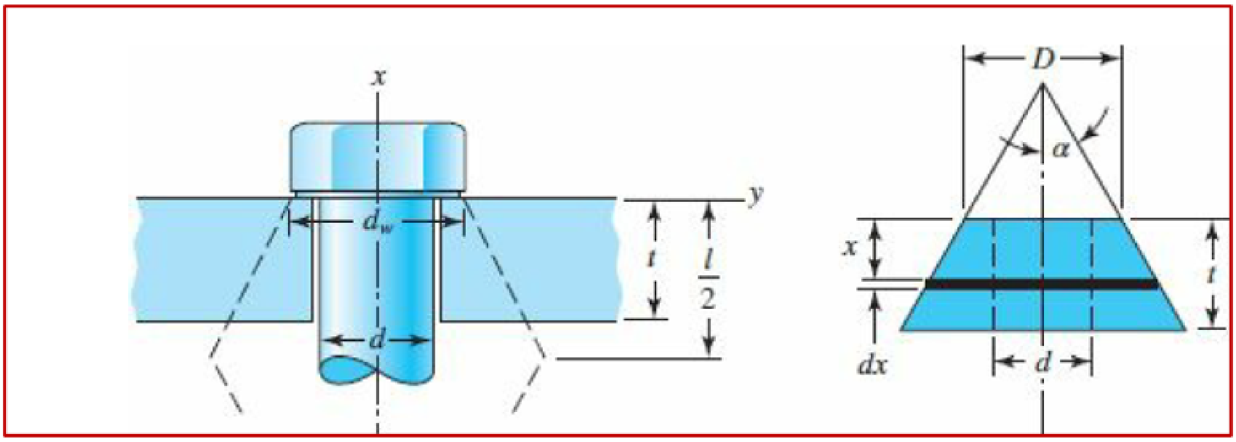


Figure 28: Pressure cone

$$F_{pc} = F_A[F_p - (1 - (\Phi)] \tag{5}$$

Once I got hold of the Load factor Φ, then my focus goes to the load coming to the members (plates). As I am now aware that the Load factor Φ, is the share of axial load coming to bolts, I calculated for the force coming to the members as this tensile force would decompress and, in a sense, relieve the clamping done initially on the members. Below in Eq.6 the residual clamp force in the members is shown.

For any sliding to happen, static friction must be overcome. Figure 29 shows the general physics for two plates with the residual clamp force and static friction force.

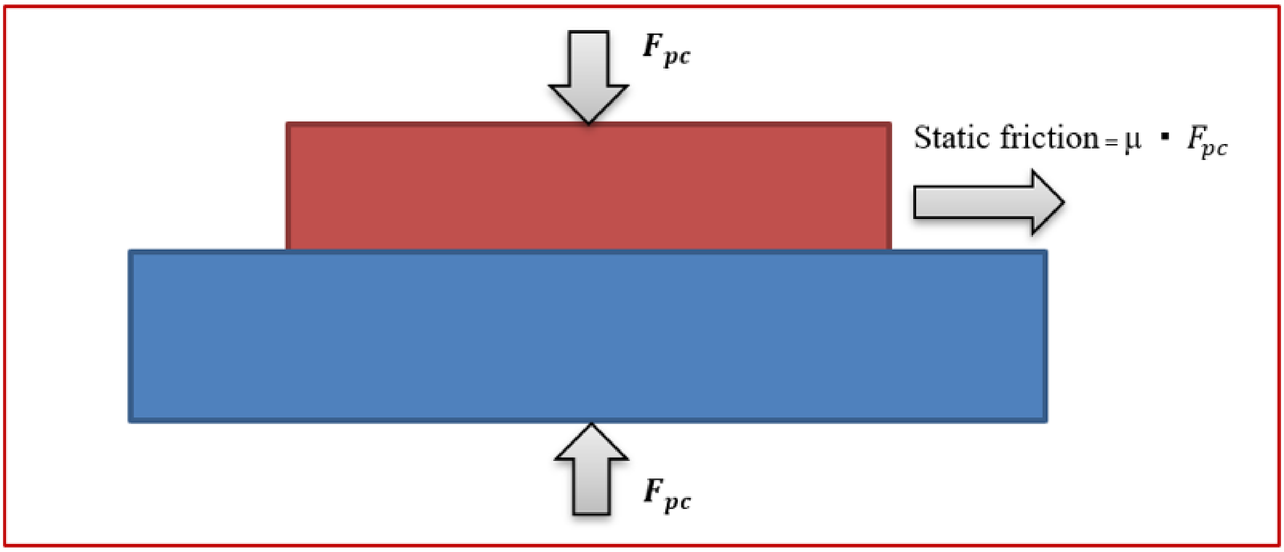


Figure 29: Static sliding between two plates

Once I calculated the limiting slip load, I include few factors such as slip resistance factor (K_s) and factor of safety to arrive at the Design Slip Resistance ($F_{S,Rd}$).

Design slip resistance ($F_{S,Rd}$) is calculated as follows:

$$F_{S,Rd} = K_s n \mu [\frac{F_p - (1 - \Phi) F_A}{\gamma_{M3}}] \tag{6}$$

Where,

K_s is the slip resistance factor, dependent on the hole shape (slotted, normal)

Partial safety factor, slip resistance at ultimate state, $\gamma_{M3} = 1.25$

Comparison is made with the shear force $F_{v,Ed}$ coming to the joints, and in order to have the joints to function properly, the shear force should always be less than the slip resistance.

$Slip_{SRF}$ and success criterion:

$$Slip_{SRF} = \frac{F_{s,Rd}}{F_{v,Ed}} > 1 \tag{7}$$

4.5 Tension resistance calculation

Bolts get stretched under the preload, then if the axial tensile force comes to the bolt then it adds up the stresses in the bolt, in-order that the bolts function safely under this acting load, I needed to first understand what is bolt's tension resistance from it's material properties (ultimate strength, f_{ub}) and cross sectional parameters (tensile stress area, A_s).

The bolt's tension resistance $F_{t,Rd}$ is calculated as:

$$F_{t,Rd} = \frac{k_2.f_{ub}.A_s}{\gamma_{M2}} \tag{8}$$

Where,

A_s is tensile stress area of the Bolt

f_{ub} is the ultimate strength of the bolt, for 10.9 bolt, $f_{ub} = 1000$ MPa

Partial safety factor, resistance of the bolts, $\gamma_{M2} = 1.25$

$K_2 = 0.63$ for countersunk bolts

$K_2 = 0.90$ otherwise

The total axial load coming to the bolt is calculated as

$$F_{t,Ed} = \ F_p + \Phi . F_A \tag{9}$$

Where,

F_p is the preload applied on the bolt

F_A is the axial force coming to the overall assembly

Φ is the load factor

Tension SRF and success criterion:

$$SRF_{Tension} = \ \frac{F_{t,Rd}}{F_{t,Ed}} > 1 \tag{10}$$

Again comparing the tensile force coming to the bolt (Eq.9), with the tension resistance (Eq.8), I calculated for $SRF_{Tension}$ with Eq.10.

4.6 Summary

The project demanded skills and knowledge from both the numerical FEA and analytical domains. As this was an individual project, at first I had to prepare a spreadsheet with all these formulas for tension and slip checks.

The bolt checks helped me in preparing the Design verification reports (DVRE) for the bolts in the WTG assembly, these were used to attain certification from DNV, this played a crucial role in certification of the Nacelle and overall wind turbine.

As I already had good exposure to bolt calculation with the VDI 2230 and Euro-code 3, it was an obvious choice for my manager to give me this responsibility, I also regularly discussed with my team lead on the FEA, calculation results and DVRE. With my work these bolts were evaluated and played a crucial role in the overall wind turbine certification and deployment.

5 Buckling analysis

5.1 Problem definition

As the rear frame is operation under various extreme loads, there are always chances of channels to buckle under compressive and bending loads.

This has to be checked by extracting forces in the beams in normal direction. The exact steps are mentioned in the subsequent subsection.

5.2 Design Buckling resistance, $\alpha_{b,Rd}$

Step 1

- Load multiplier check

The first step is to check the load multiplier from the Eigen-Value Buckling analysis of the assembly.

BS EN 1993-1-1:2005
EN 1993-1-1:2005 (E)

5.2 Global analysis

5.2.1 Effects of deformed geometry of the structure

(1) The internal forces and moments may generally be determined using either:

- first-order analysis, using the initial geometry of the structure or
- second-order analysis, taking into account the influence of the deformation of the structure.

(2) The effects of the deformed geometry (second-order effects) should be considered if they increase the action effects significantly or modify significantly the structural behaviour.

(3) First order analysis may be used for the structure, if the increase of the relevant internal forces or moments or any other change of structural behaviour caused by deformations can be neglected. This condition may be assumed to be fulfilled, if the following criterion is satisfied:

$$\boxed{\begin{aligned} \alpha_{cr} &= \frac{F_{cr}}{F_{Ed}} \geq 10 \quad \text{for elastic analysis} \\ \alpha_{cr} &= \frac{F_{cr}}{F_{Ed}} \geq 15 \quad \text{for plastic analysis} \end{aligned}} \tag{5.1}$$

where α_{cr} is the factor by which the design loading would have to be increased to cause elastic instability in a global mode

F_{Ed} is the design loading on the structure

F_{cr} is the elastic critical buckling load for global instability mode based on initial elastic stiffnesses

NOTE A greater limit for α_{cr} for plastic analysis is given in equation (5.1) because structural behaviour may be significantly influenced by non linear material properties in the ultimate limit state (e.g. where a frame forms plastic hinges with moment redistributions or where significant non linear deformations from semi-rigid joints occur). Where substantiated by more accurate approaches the National Annex may give a lower limit for α_{cr} for certain types of frames.

Figure 30: Load Multiplier from EC-3

This is done in pre-stressed state of any particular extreme load case. As per EN 1993-1-5:2005, the critical load should be over and above 10 times from the applied load, in the elastic analysis, this is also shown in Figure 30.

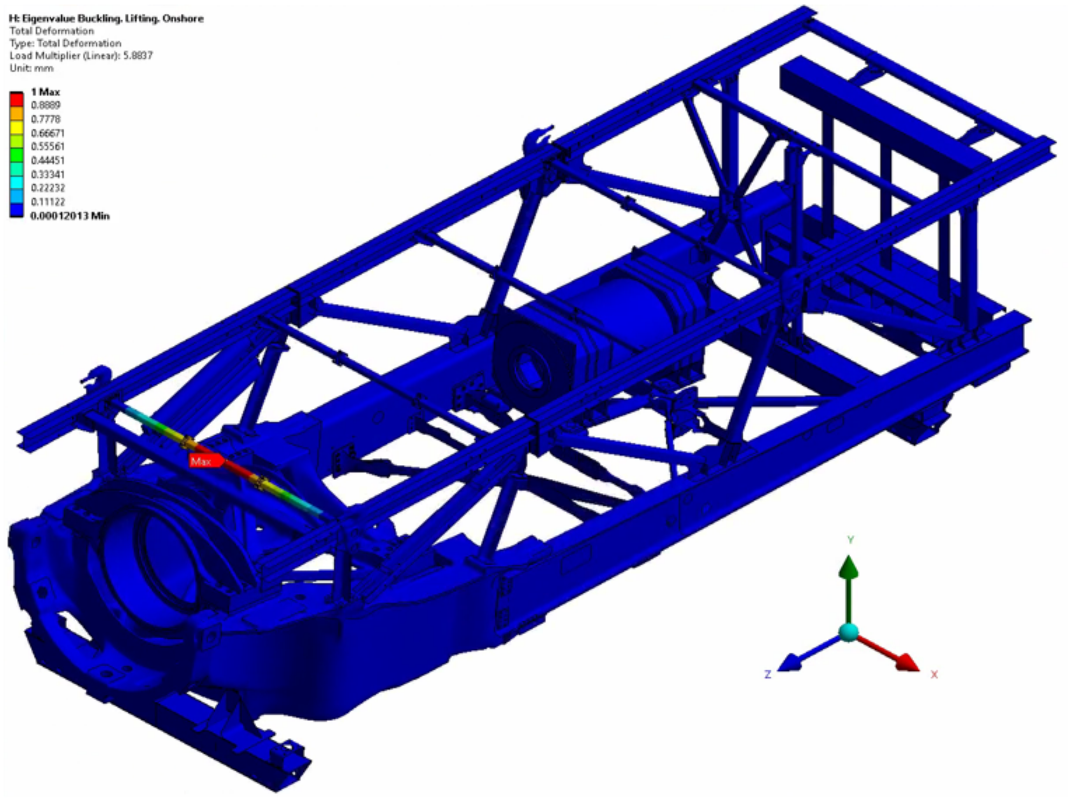


Figure 31: Load Multiplier Eigen-Value Buckling

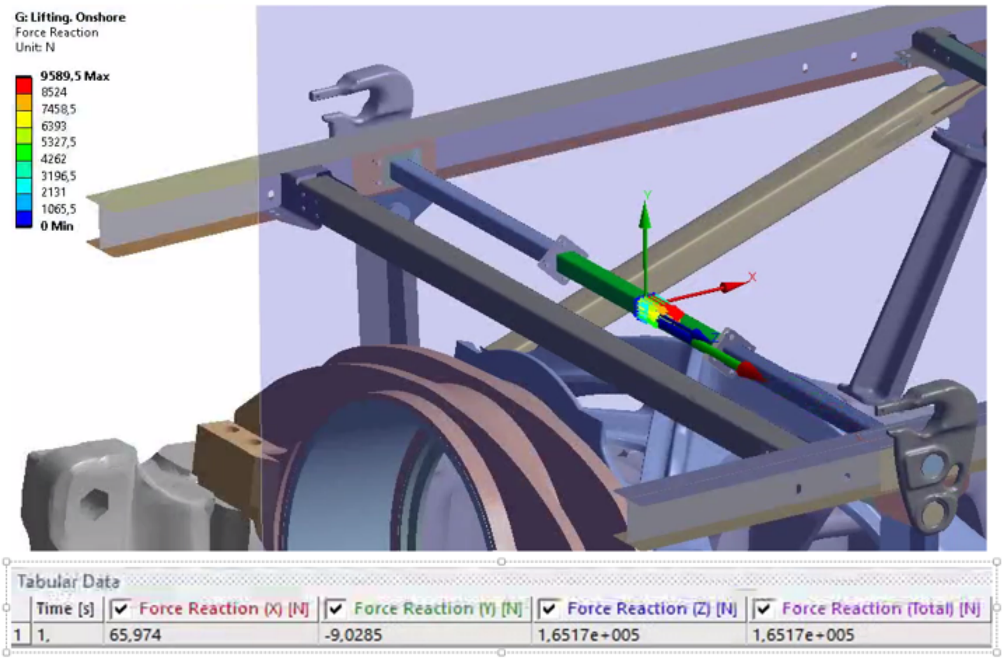
In the Eigen-Value Buckling pre-stressed state if I observe load multiplier less than 10 (Figure 31), then I need to validate this structure either through analytical calculations or through re-design.

Step 2

- Compression load and section properties

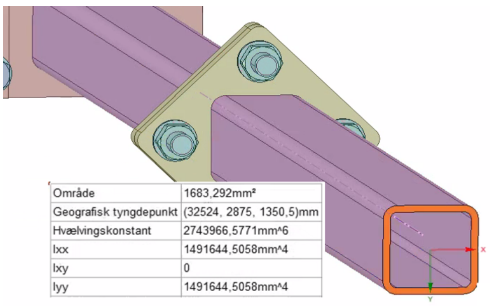
In this part once I observe that load multiplier is less than 10, then I calculate through analytical formulas given in EN 1993-1-5 : 2005 for Design Buckling resistance (α).

In this regard first I extract the compression force coming on the channel in FEA tool, shown in Figure 32.



	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	1,	65,974	-9,0285	1,6517e+005	1,6517e+005

Figure 32: Max compression force on channel/Beam



Område	1683,292mm²
Geografisk tyngdepunkt	(32524, 2875, 1350,5)mm
Hvælvingskonstant	2743966,5771mm^6
Ixx	1491644,5058mm^4
Ixy	0
Iyy	1491644,5058mm^4

Figure 33: Moment of inertia

I also take the moment of inertia for this channel from SpaceClaim (Figure 33)

Step 3

- Slenderness ration(λ)

$$\lambda = \sqrt{\frac{Af_y}{F_{cr}}}$$

Where,

$$f_y = \text{Yield limit}$$

$$F_{cr} = \frac{\pi^2 EI}{{l_e}^2}$$

Step 4

- Calculation of parameter ϕ

$$\phi = 0.5[1 + \alpha(\lambda - 0.2) + \lambda^2]$$

Where,

α = is the imperfection factor and depends on Buckling curves

Buckling curve*	a_0	a	b	c	d
Imperfection factor	0.13	0.21	0.34	0.49	0.76

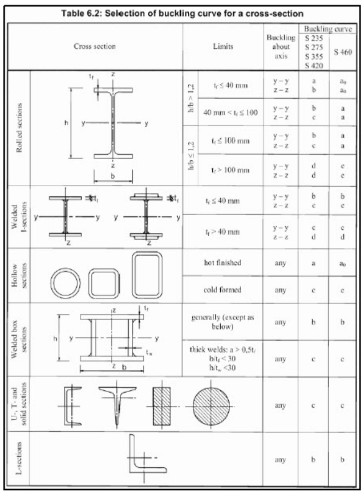


Table 6.2: Selection of buckling curve for a cross-section

Cross section		Limits		Buckling about axis	Buckling curve S 235 S 275 S 355 S 420	Buckling curve S 460
Rolled sections		h/b > 1,2	$t_f \le 40$ mm	y – y z – z	a b	a_0 a_0
		h/b > 1,2	40 mm < $t_f \le 100$	y – y z – z	b c	a a
		h/b ≤ 1,2	$t_f \le 100$ mm	y – y z – z	b c	a a
		h/b ≤ 1,2	$t_f > 100$ mm	y – y z – z	d d	c c
Welded I-sections		$t_f \le 40$ mm		y – y z – z	b c	b c
		$t_f > 40$ mm		y – y z – z	c d	c d
Hollow sections		hot finished		any	a	a_0
		cold formed		any	c	c
Welded box sections		generally (except as below)		any	b	b
		thick welds: $a > 0{,}5t_f$ $b/t_f < 30$ $h/t_w < 30$		any	c	c
U-, T- and solid sections				any	c	c
L-sections				any	b	b

Figure 34: Buckling Curves

Step 5

- Reduction factor (χ)

$$\chi = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}$$

Step 6

- Design Buckling factor, $\alpha_{b,Rd}$

$$\alpha_{b,Rd} = \frac{\chi A f_y}{F_{ed}}$$

Where,

F_{ed} = Max compression force coming from FEA

Step 7

- Design Buckling factor, $\alpha_{b,Rd}$

$$\alpha_{b,Rd} = \frac{\chi A f_y}{F_{ed}}$$

If,

$\alpha_{b,Rd} > 1$, design accepted

$\alpha_{b,Rd} < 1$, design not-accepted. Non-linear analysis needs to be performed

6 Fracture mechanics

6.1 Problem definition

In fracture mechanics I have solved one problem given in a paper ”Predicting extended service Life durability and damage tolerance” by David R. Dearth. The geometry of the problem is shown below in Figure 35.

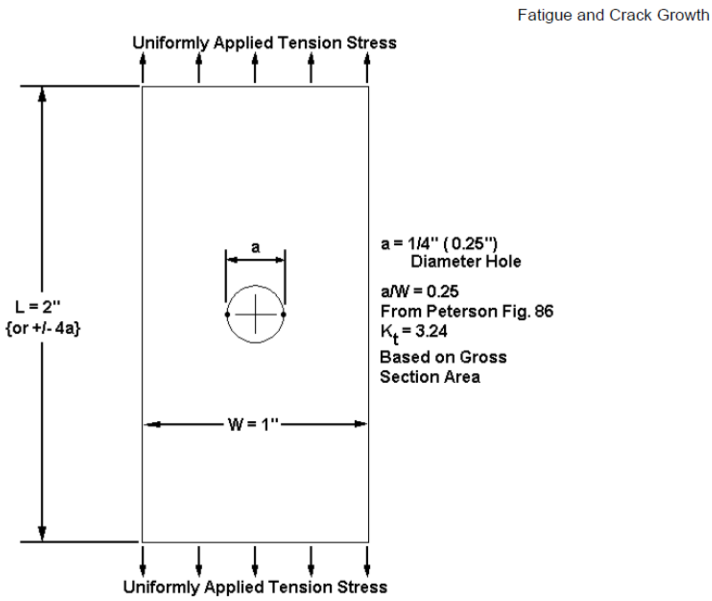


Figure 35: Geometry of plate with hole

In this geometry and loading a stress concentration factor(SCF) K_t effect is created near the edge of the hole. The plate is fabricated from aluminum alloy 2024-T4 extruded plate and is subjected to constant amplitude cyclic loading of 0 psi Minimum, to 12,000 psi Maximum.

6.2 Calculation of SCF K_t

The theorectical value can be seen from Petersons work on stress concentration factor. With the same geometric dimensions and loads I simulated to see the stress concentration factor(K_t) in FEA calculation.

Figure 36: Maximum principal stress

Taking the ratios of the maximum principal stress and the load applied.

$$K_t = \frac{39386}{12000} = 3.28$$

Stress concentration factor Kt as per R.E. Peterson = 3.24

So, the percentage deviation from theory to FEA is

$$\frac{3.28}{3.24} - 1 = 1.2\%$$

6.3 Fracture toughness and fatigue tool

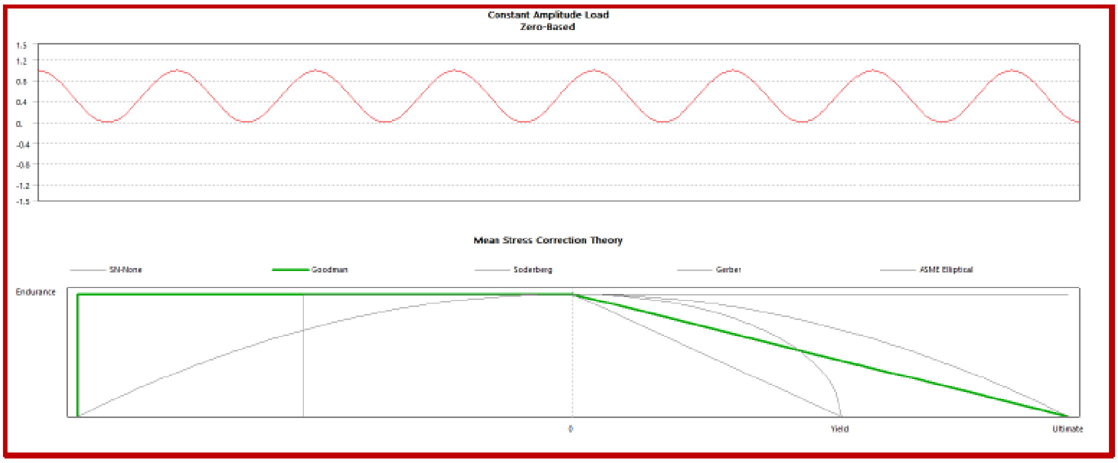


Figure 37: Goodman mean stress correction

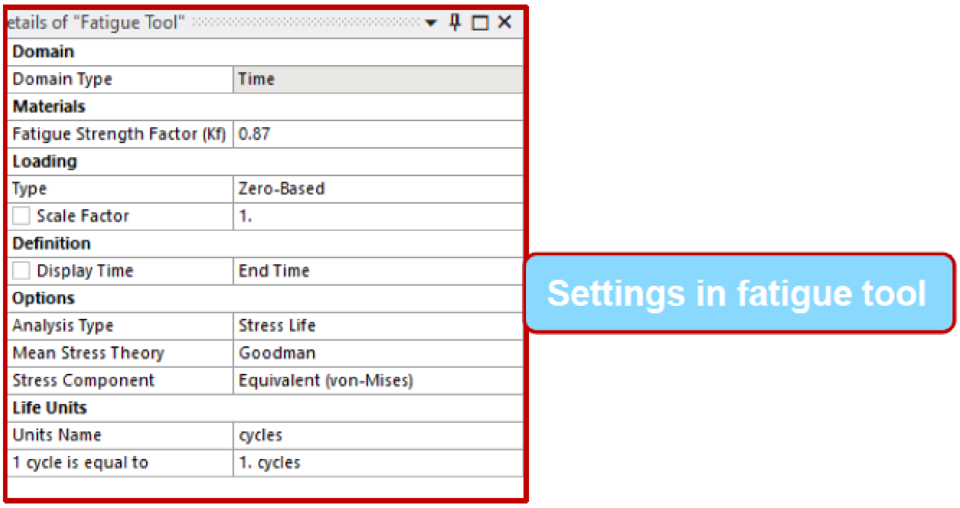


Figure 38: Fatigue tool settings

With the goodman mean stress correctin and the settings for he fatigue tool, where the load is zero-max-zero(zero based) and the analysis type is stress-based , the fatigue strength of 0.87 was taken from the paper by David R. Dearth.

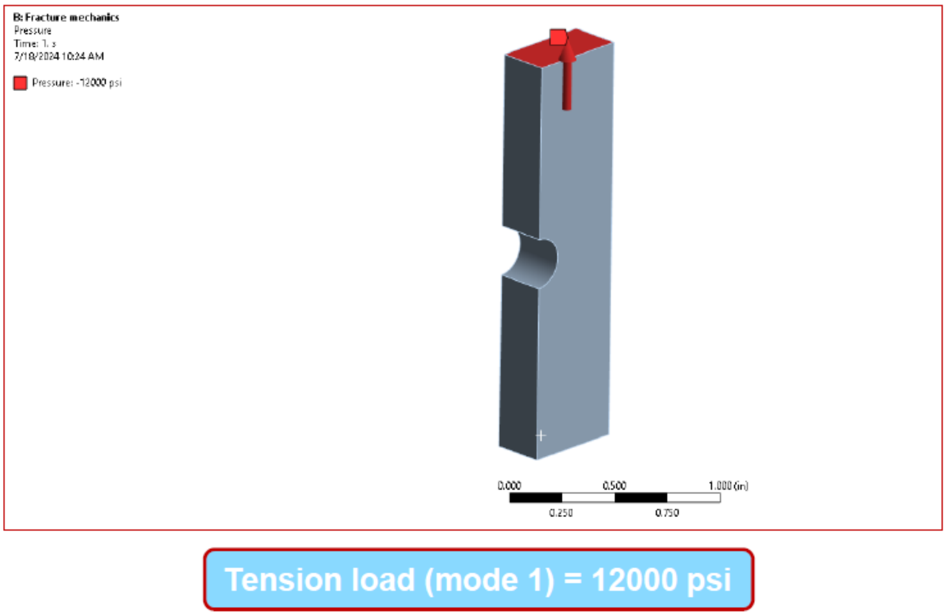


Figure 39: Tensile loading: Mode I

I modeled an elliptical crack through the samart rack tool in ansys and calculated for the stress intensity factor mode 1 as that is the dominant mode, this is also shown in Figure 40 below.

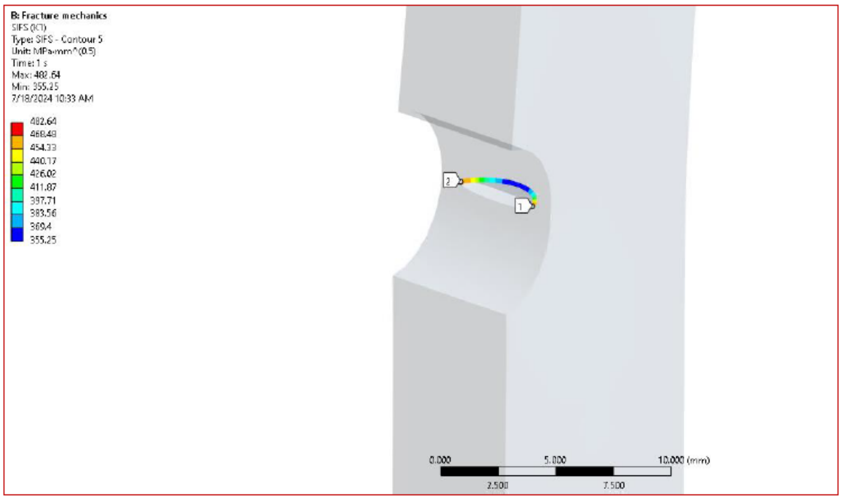


Figure 40: Elliptical crack

Definition	
Type	SIFS
Subtype	K1
Active Contour	5
Location Method	Crack Length Percentage
Crack Length Percentage	50
Suppressed	No
Options	
Display Time	End Time
Spatial Resolution	Use Maximum
Results	
SIFS (K1)	10225 psi·in^(0.5)
Minimum Value Over Time	8491.9 psi·in^(0.5)
Maximum Value Over Time	10225 psi·in^(0.5)

Figure 41: Stress intensity factor K_I

The fracture toughness of the material(K_{IC}) was 30000 psi in$^{0.5}$

The stress intensity factor in dominant mode < K$_{IC}$