Strength assessment of bolted connections

The effect of geometric non linearity and stress stiffening on bolt stresses

Author:
Moniek Strijdveen
s1009311

Supervisor Mecal Wind/Energy:
Ir. Annette op de Weegh

Supervisor University of Twente:
Prof. Dr. Ir. André de Boer

UNIVERSITY
OF TWENTE.

Abstract

The development of wind turbines is a complex task as they should meet several European and Global safety norms regarding the construction, maintenance, performance and grid connection. A certificate is necessary for the manufacturer in order to demonstrate that the turbine satisfies the norms. MECAL Wind Turbine Design (WTD) is an engineering company which has contributed to the development of many different turbine technologies. Part of their work is the strength assessment of bolted connections using the Finite Element Method (FEM).

During this internship the effect of two phenomena: geometric non linearity and stress stiffening on the strength assessment procedure of bolted connection is investigated. Geometric non linearity refers to taking into account large displacements. Stress stiffening refers to an increment in bending stiffness of the structure as a consequence of an axial force. It is mainly visible in structures with a low ratio of bending/axial stiffness. Regarding bolted connections, it is expected that taking into account stress stiffening during bolt assessment will result in higher bending stresses. The axial force which is responsible for the increment in bending stiffness equals the pretension force.

The translation of the bolt geometry to a Finite Element (FE) model is done by MECAL using beam elements. The stresses at the transition of the thread to the shaft and the ends of the bolts are assessed. An alternative option is to model the bolt with solid elements however these elements have some disadvantages regarding the degrees of freedom (DoF) and consequently the calculation time.

Based on the assessed geometries, it is not necessary to account for geometric non linearities. However one should keep in mind that it is possible that geometric non linearities should be taken into account in the near future as a consequence of the increase in the size of the turbines and the reduction of the relative stiffness of components due to mass optimization. For these situations it is recommended to use beam elements for modelling bolts based on the complexity level of obtaining sectional forces and moments compared to solid elements.

In FEM stress stiffening is taken into account by augmenting an additional matrix to the normal stiffness matrix of the structure. In order to identify when stress stiffening should be taken into account a analytical FEM simulation is executed. A simple 2D plane frame element, accompanying stiffness and geometric stiffness matrices are used for this simulation. A analytical expression of the bending stiffness is obtained. Plotting the bending stiffness against the slenderness ratio of the beam resulted in a curve from which could be concluded at which slenderness ratio it is relevant to take stress stiffening into account.

In order to show the effect of stress stiffening in for example a pitch bearing connections, results of the simulations where stress stiffening had/had not been taken into account should be compared. Based on the comparison one can conclude if it is nesecarry to include the effect of stress stiffening in future strength assessments of bolted connections or that the safety factor already covers the increment in bending stresses as a consequence of stress stiffening.

Contents

Chapter 1 Introduction

The development of wind turbines is a complex task as they should meet several European and Global safety norms regarding the construction, maintenance, performance and grid connection. A certificate is necessary for the manufacturer in order to demonstrate that the turbine satisfies the norms [1].

MECAL Wind/Energy is an engineering company, which has contributed to the development of many different wind turbine technologies. For over 20 years they have been developing competences in design & engineering, loads & control and simulation & analysis. Part of their work is the strength assessment of bolted connections using Finite Element Method (FEM). Typical bolted connections present in a turbine (fig.1.1) are located between the blade and the hub (1), hub and main shaft (2) and mainframe and tower top flange (6). In between clamped components (1) and (3) a bearing is present, enabling blade pitching and turbine yawing respectively. By means of the clamping force, introduced through the preloading of the bolts, forces and moments can be transmitted between the clamped components. The bolted connections are considered to be critical and should for this reason be assessed in detail.

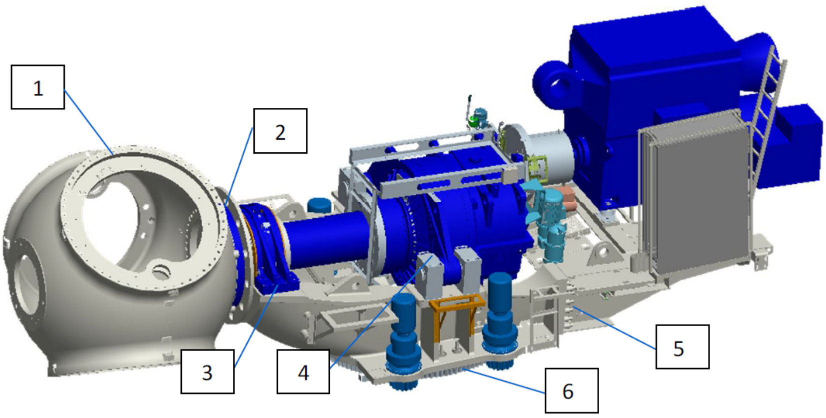


Figure 1.1: Typical bolted connections wind turbine [2]

During this internship the effect of two phenomena: geometric non linearity and stress stiffening, on the strength assessment procedure of bolted connections are investigated. Both phenomena will be extensively explained in chapter 4 and 5, a brief description will be given here. Geometric non linearity refers to taking into account large displacements. In case of small displacements, the strain-displacement relations can be linearised. When large displacements do occur, linearisation is not possible anymore and a quadratic expression is used to describe the strain-displacement relations correctly. Stress stiffening is an increase in the lateral stiffness of a structure due to an axial tensile force. It is mainly visible in thin structures with a low ratio of bending stiffness compared to axial stiffness. A guitar string illustrates the effect of stress stiffening perfectly. Stretching the string first and applying a lateral force sequentially will result in less deflection and a higher tone compared to the case where only a lateral force is applied.

Chapter 2 provides background information about bolted connections in order to support the information in the subsequent chapters. It addresses MECAL's strength assessment procedure and corresponding 3D FEM models of bolted connections. Chapter 3 presents an initial study done by MECAL and the reason why the effect of geometric non-linearity and stress stiffening in the strength assessment procedure should be investigated.

The effect of geometric non-linearity and stress stiffening will be discussed in chapter 4 and 5 respectively. The report finalizes with conclusions and recommendations for the strength assessment procedure, described in chapter 6.

Chapter 2 Background

This chapter will elaborate on MECAL's strength assessment procedure of bolted connections. Topics addressed are the representation of a bolted connection in FEM, what elements are used and which boundary conditions are needed? In addition the assessed load combinations and the behaviour of a bolted connection to these load combinations are discussed.

2.1 Representation of bolted connection in FEM

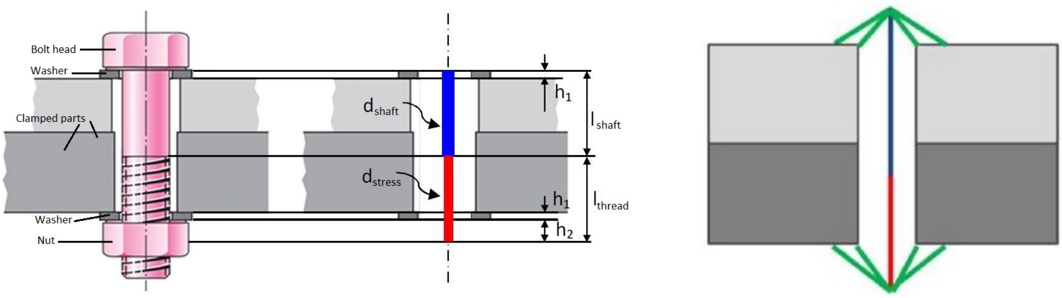
A bolted connections consists of at least three components indicated in figure 2.1a:

- bolt with head and nut
- washers
- clamped parts

MECAL uses the FEM software ANSYS to represent the components present in the wind turbines. For bolted connections a few assumptions and simplifications are made:

- bolt heads and nuts are not represented
- non-loaded parts of the bolt are not modelled
- threads are not modelled, but are represented by the bolt stress diameter (d_{stress})
- the first two threads attached to a nut or threaded hole do not carry any load.

The translation of the bolt geometry to a bolt FEM model is illustrated in figure 2.1a. The bolts is modelled using beam elements. Beam elements (BEAM188) having a solid circular cross section with the shaft d_{shaft} or stress diameter d_{stress} represent the bolt shaft and thread respectively. From figure 2.1a can be seen that the FEM model represents bolt up to the head/washer and nut/washer sections.



(a) Left: Actual bolt geometry, Right:modelled bolt geometry using beam elements

(b) Spoke wheel connection

Figure 2.1: FE model of bolted connection

BEAM188 elements have two nodes with each six degrees of freedom (DoF): Three translations in x, y and z direction and three rotations about these axis. As a consequence of ignoring the bolt head and nut, a connection between the end nodes of the bolt and nodes located at the inner and outer ring of the washers is made. This is illustrated in figure 2.1b. Elements used for this connection are solid circular beam elements (BEAM188) having the nominal diameter [2].

Bolt pretension is applied by means of PRETS179 elements. These elements have a single translational degree of freedom, oriented parallel to the bolt axis. By the definition of a pretension element, a constraint equation between two coincident nodes is defined in the pretension direction.

2.1.1 Contact interfaces

A bolted connection has three contact interfaces:

1. bolt head - washer
2. nut - washer
3. between clamped parts

The behaviour of these interfaces due to working loads is different. Interface 1 and 2 will never open as a consequence of a working load. However this is not the case for interface 3. For this reason the head/nut washer interfaces are modelled as bonded contact, which means that opening and sliding is not possible. The interface between the clamped parts is described with rough contact: a interface which can slide and open. It is stressed that the behaviour of the last type of contact is non-linear. For this reason a non-linear solving procedure is necessary.

The contact interfaces are modelled with elements CONTA174 and TARGET170. These elements have the same DoF as the underlying solid element faces. In most cases these are SOLID186 elements, having 20 nodes with each 3 DoF.

2.2 Load assessment cases

In order to make sure that the bolted connections can resist all conceivable circumstances, the ultimate bolt strength, fatigue bolt strength and reserve on slip is calculated.

In general the bolt strength is assessed at four locations: twice at the diameter transitions (2 and 3) and twice at the interfaces with the components (1 and 4). These locations are shown in figure 2.2. The reason for these assessment points is that these location will give the maximum bolt stress for the corresponding bolt section.

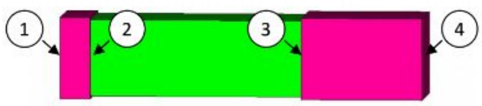


Figure 2.2: Assessment points bolt. Bolt thread and shaft are shown in pink and green respectively

2.2.1 Ultimate assessment

A set of extreme load combinations is determined to which the wind turbine could be subjected, for instance a whirlwind. Finite element calculations are performed in order to determine the bolt stresses occurring as a consequence of these extreme load cases. The yield strength of bolts is documented in ISO 898. Calculated stresses from the ultimate assessment are compared with the values from ISO 898 in order to determine if the structure is able to resist these extreme cases.

2.2.2 Fatigue assessment

The fatigue assessment of a bolted interface in general is done using the bending moment in the dominant load direction(s). Depending on the tightening method of the bolt, there is a scatter of the nominal pretension. Note that the ultimate strength of a bolt is determined using the maximum pretension level. The fatigue strength of the bolt, as well as the resistance against slip is determined using the minimum pretension. By means of FE analysis the non-linear behaviour of the bolted connection is analysed for the complete range of values of the dominant load during the design life of the turbine. These analyses are used to construct the bolt curvefigure 2.3 [3]. Three phases can be identified:

1. Linear part of the curve. The load is carried by both the bolt and the contact interface remains closed.
2. Non linear behaviour, at this point the pretension is exceeded and the interface will start gapping.
3. Linear part of the curve, interface is gapping and bolt carries the external force completely.

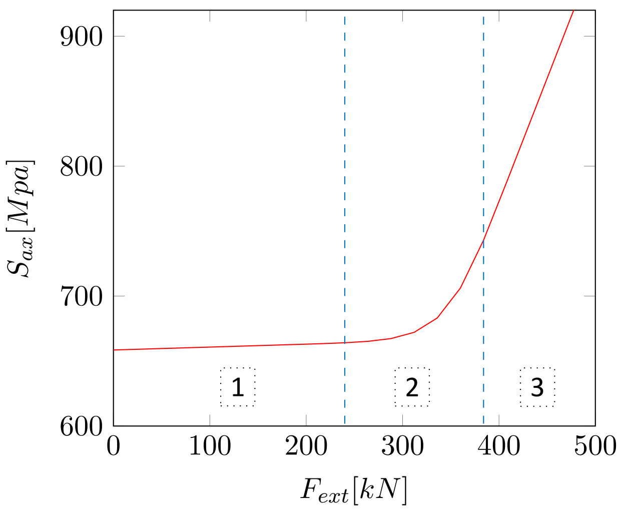


Figure 2.3: Bolt Curve

The bolt curve slopes are combined with the corresponding load from the load time series, in order to obtain a stress time signal. Rainflow counting is performed on this signal. The fatigue strength of the bolts is calculated for the minimum bolt pretension level, by using the appropriate bolt S/N curve, which gives te relation between stress range and number of cycles.

Non linear behaviour of the bolt (phase 2) should be prevented as this lead to unproportional high stress ranges and thus to a low bolt strength. For this reason the importance of an appropriate preload is stressed. In general a high loaded bolt is pre tensioned to a level of 60 to 80% of the yield strength (σ_y). Besides the prevention of decompression, pretension also provides resistance against sliding in the interface. In order to determine whether the resistance is sufficient, the occurring slip force should be compared to the allowable slip force. The allowable slip force depends on the bot pretension and the friction coefficient of the interface[2].

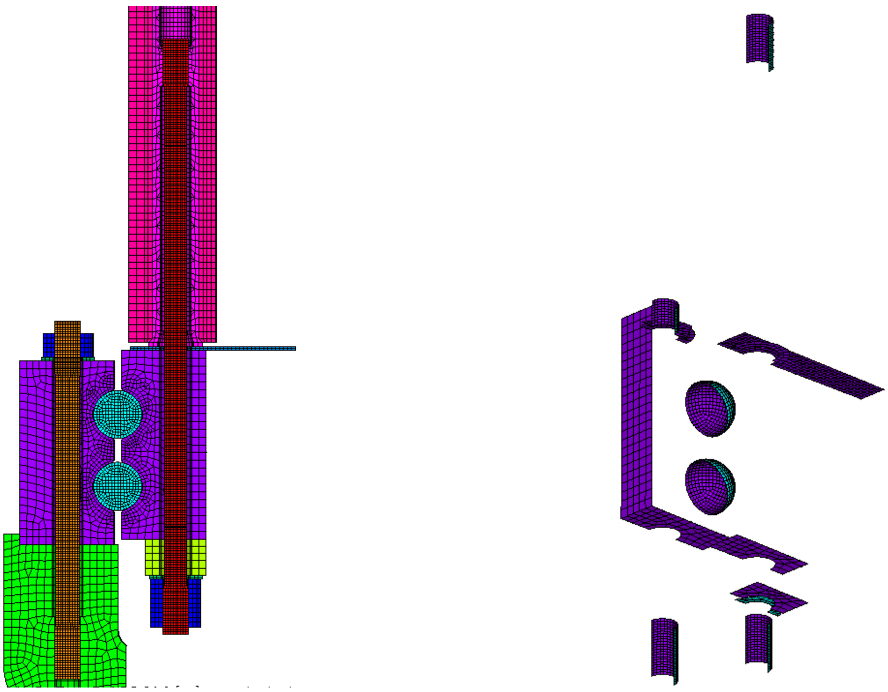
Chapter 3 Assignment

This chapter will illustrate why the effect of geometric non linearity and stress stiffening on the strength assessment procedure should be investigated. The outcome of an initial bolt study executed by MECAL was the reason for prescribed investigation and is presented first.

3.1 Initial bolt study

At the moment it is not necessary to take into account the effect of geometric non-linearities in the assessment of wind turbine components. However, as turbines increase in size and mass optimization reduces the relative stiffness of components, it might be necessary to take into account geometric non-linearities in the near future. In order to study the effect of geometric non-linearity, MECAL already examined a bolted connection in which geometric non-linear behaviour was included. This is obtained in Ansys by switching the command `nlgeom` 'on'.

The structure is depicted in figure 3.1a shows part of the blade (pink), pitch bearing (purple, light blue) and hub (green) of the turbine all modelled with solid elements (SOLID186). As the name implies, this bearing induces a pitch motion of the blade, in other words rotating it in such a way that it catches the wind best as possible. Only a small part of the structure is modelled, this is called a section model. The complete geometry: hub, pitch bearing including bearing balls and one blade can be obtained by mirroring the section model at the symmetry plane and then copying the pie-piece a number of times. 3.1a.



(a) Pitch bearing bolted connection, close up. Frontal view of plane of symmetry

(b) Contact interfaces pitch bearing connection. Isometric view of plane of symmetry.

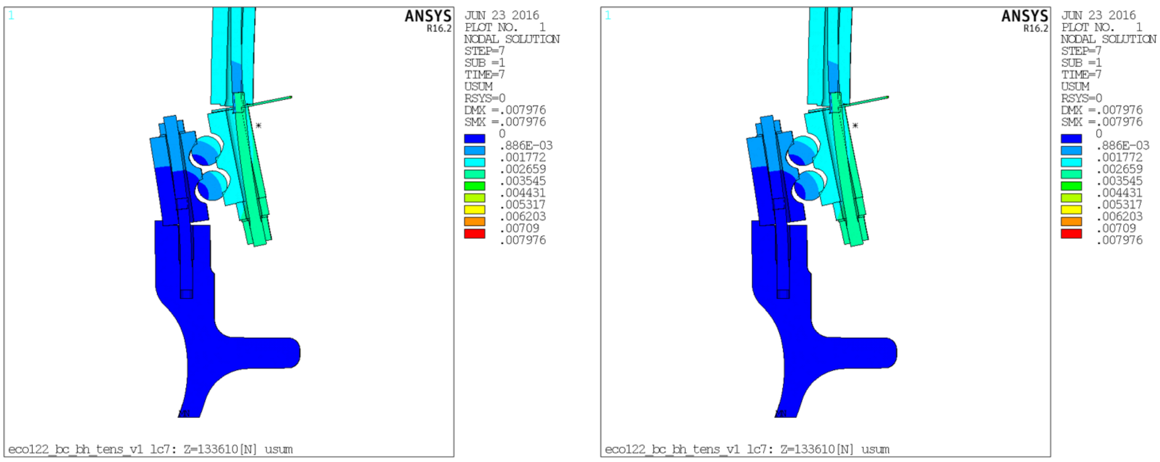
Figure 3.1: Geometry pitch bearing model

Figure 3.1a shows the bolts which clamps the pitch bearing between the blade and hub by means of a pretension force. The hub bolt is depicted in orange and the blade bolt in red. Nuts and washer are shown in blue and green respectively. One should note that these bolts are modelled with solid elements (SOLID186) instead of beam elements. The contact interfaces (CONTA174,TARGE170) present in this structure are depicted in figure 3.1b. All of them are rough contact interfaces.

The structure is loaded on top of the blade with a vertical force. This force induces axial and bending moments in the bolts. The stresses occurring in th blade bolt are discussed in the next section.

3.1.1 Results and Conclusion

Figure 3.2 shows the displacements of the situations where geometric non-linearity had been/ had not been taken into account. From these plots can be concluded that it is not necessary to account for large displacements by switching `nlgeom,on`.



(a) Usum with `NLGEOM OFF`

(b) Usum with `NLGEOM ON`

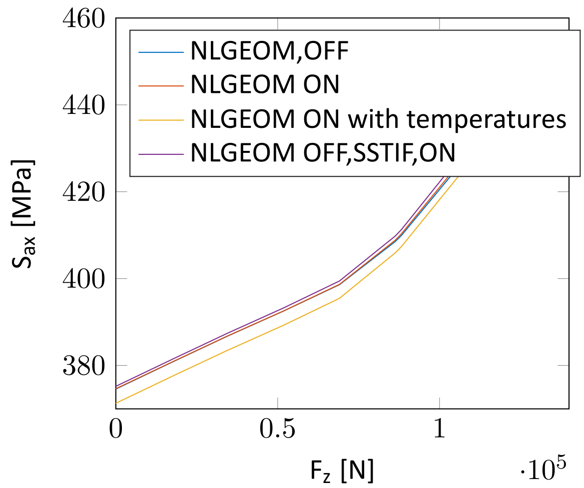
Figure 3.2: Displacements pitch bearing connection

The axial and bending stress occurring in the bolt are depicted in figure 3.3. The stresses are calculated at the four locations described in section 2.2. Unfortunately discrepancies occurred in the bending stresses of the bolt between both solving types, `nlgeom,off` and `nlgeom,on`. Only the results of the location 1 and 4 are shown in figure 3.3 as location 2-3 showed similar discrepancies.

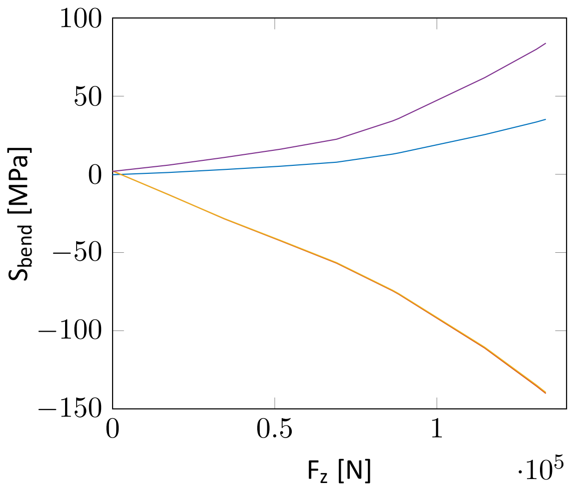
Two additional tests are executed in order to exclude the effect of stress stiffening and the orientation of the pretension elements during the deformation. The latter is done by means of a temperature increment. From 3.3 can be concluded that the discrepancies between `nlgeom,on/off` are not only caused by the effect of stress stiffening `sstif,on` or the way the bolt was pre tensioned `nlgeom,on temperatures`. Apparently something else changes by switching `nlgeom,on`.

3.2 Assignment

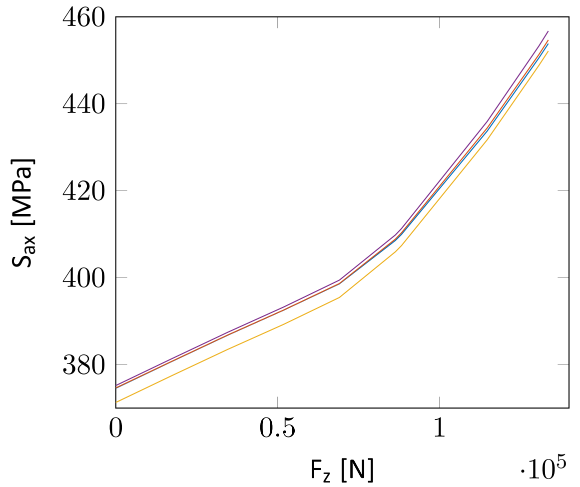
The results from the initial bolt study have led to the formulation of two research topics: Geometric non-linearity and stress stiffening. First will be investigated what exactly happens during the solution



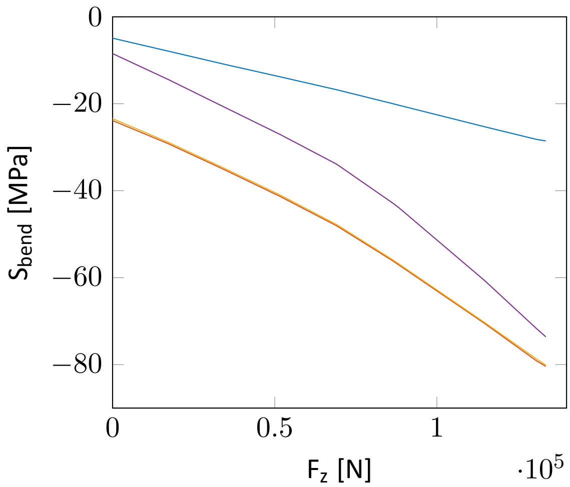
(a) Axial stress location 1



(b) Axial and bending stresses location 1



(c) Axial and bending stresses location 4



(d) Axial and bending stresses location 4

Figure 3.3: Results initial bolt study

procedure by switching `nlgeom,on`. This will be done using a simple geometry where the bolt is subjected to bending. Secondly the effect of stress stiffening will be studied.

From figure 3.3 it is clear that stress stiffening increases the bending stresses. In addition solving a structure with `nlgeom,on` also accounts for stress stiffening, based on this one should expect the same bolt curves for small displacements. The goal of the study is to define at which bolt length stress stiffening should be taken into account and to give recommendation on the (minimum) number of elementsthat should be used to model the bolts.

The outcome of these studies will probably lead to some changes in the way bolted connections are assessed at MECAL.

Chapter 4 Geometric non-linearity

In this chapter the effect of taking into account geometric non linearity in a strength assessment is investigated. As mentioned in the introduction geometric non linearity refers to situations where displacements cannot be approximated with linear terms any more. In addition the equilibrium equations used in FEM should be written with respect to the deformed configuration.

Section 3.1 showed an increment in bending stresses as a results of taking into account geometric non linearity (`nlgeom,on`) The geometry where the problem occurred is quite complex and as a reason of this a simpler model was built to be able to have a thorough understanding of the behaviour of the structure. A list of requirements for the simpler model is formulated, based on the section model shown in figure 3.1a:

- Bolt should be subjected to bending
- Possibility to introduce contact interfaces
- Possibility to change element type of the bolt

This list is based on the train of thought that the type of elements used are responsible for the discrepancy in bending stresses. If this is indeed the reason for the discrepancy between the bending stresses, the problem (sec. 3.1) could be reproduced with the simpler geometry.

4.1 Geometry 1

The geometry depicted in figure 4.1 is used to exclude the possible effect of contact elements (CONTA174, TARGE170) on bending stresses by switching `nlgeom on`. A block is clamped between a bolt of length 0.1m. Only the shaft with a diameter of 0.036m is modelled, so no thread has be taken into account. The bolt, washer and left part of the block (purple) all have a young modulus equal to $E = 2.11e^{11}$ Pa. The part of the block depicted in light blue has a youngs modulus of $E = 1e^9$ Pa. The combination of the different materials present in the block introduce bending stresses in the bolt after applying pretension. The whole geometry is meshed as one part and as a reason of this no contact interfaces are present.

4.1.1 Boundary conditions and loads

Both vertical ends of the geometry are constrained for all DoF. The bolt is pre tensioned with a force of 670 kN, which equals 70% of the yield strength (σ_y) of the bolt: 940MPa.

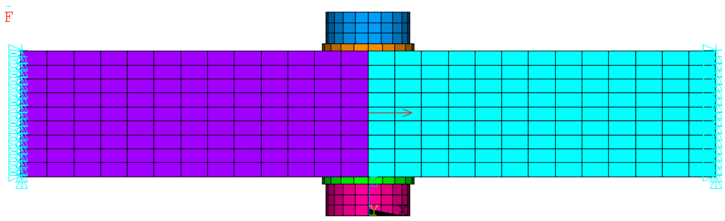


Figure 4.1: Geometry 1

4.1.2 Test cases

To study the effect of different elements, the bolt is modelled with either solids (SOLID186) or beam elements (BEAM188). Depending on the type of elements of the bolt, different methods can be used to extract the bolt stresses. Possible effect of these commands is also investigated. The command `fsum` can be used for both solid and beam elements and will sum the forces and moments at a section around a specified point (`spoint`). `etab` can only be used in case of beam elements and extracts forces and moments from the end nodes of the beam element.

In short, three cases will be tested for both `nlgeom,off` and `nlgeom,on`. These are listed below:

1. Bolt modelled with SOLID186 elements, use `fsum` to extract forces and moments
2. Bolt modelled with BEAM188 elements, use `fsum` to extract forces and moments
3. Bolt modelled with BEAM188 elements, use `etab` to extract forces and moments

4.1.3 Results and Conclusion

The forces, moments and corresponding stresses are obtained at the transition between the bolt shaft and head or nut, location 1 and 2 respectively. Due to symmetry of the structure only the stresses at location 1 are shown in table 4.1. Column 4 and 5 show the bending stresses in y direction for the cases `nlgeom` ON and OFF. Column 6 indicates the % difference between these two cases. From the list of % differences can be concluded that the effect described in section 3.1.1 is not reproduced by this geometry as the % difference is not comparable with the % difference in bending stresses observed in 3.1a. Another geometry is necessary in order to identify the discrepancy between the bending stresses in case of `nlgeom` ON and OFF. This will be explained in section 4.2.

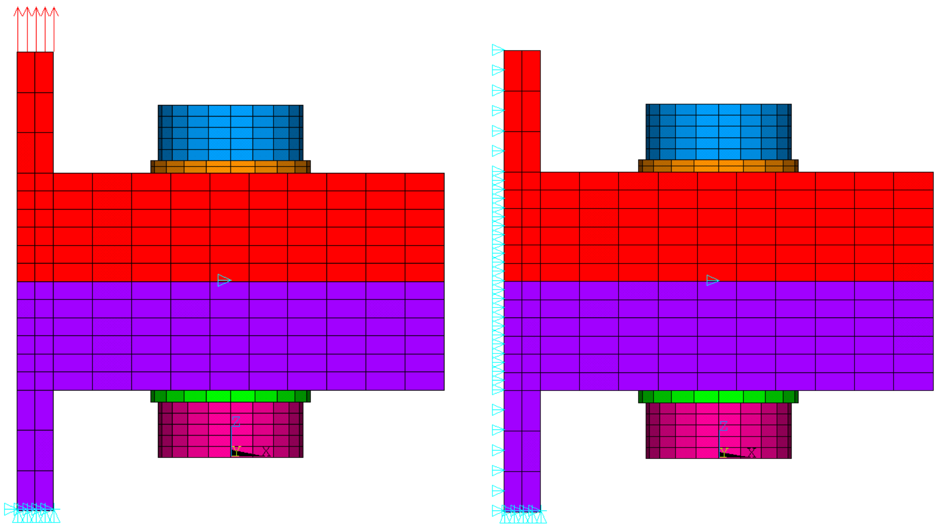
Case	El.type	Method	NLGEOM OFF	NLGEOM ON	% Difference
1.	SOLID186	FSUM	-7.6216E+08	-7.5690E+08	-0.0069
2	BEAM188	FSUM	-8.4511E+08	-8.4808E+08	0.0035
3	BEAM188	ETAB	-8.4686E+08	-8.5515E+08	0.0098

Table 4.1: Bending stresses [Pa] geometry 1, location 1, loadcase: only pretension

4.2 Geometry 2

The structure depicted in figure 4.2 consists of three parts: bolt including washers, top part flange (red) and bottom part flange (purple) all having a youngs modulus of $E = 2.11E^9$ Pa. The bolt has a length of 0.1m and a diameter of 0.036m. Also in this case, the thread is not taken into account.

In contradiction to geometry 1, contact interfaces are introduced by meshing the parts separately. These interfaces are modelled as bonded contact which means that the interface is not able to open or slide, the interface only transmits forces between the three parts. The model resembles a bolted flange connections, e.g. a tower flange. The bending stress is introduced by applying a force on top of the flange. (fig.4.2a).



(a) pretension and F_z (b) pretension and U_x

Figure 4.2: Geometry 2

4.2.1 Boundary conditions and loads

The structure is constrained for all DoF at the bottom of the flange as indicated in figure 4.2. Two load cases are evaluated: force F on top the flange (fig4.2a) and a translation of 0.001 m to the right (fig.4.2b. The bolt is pre tensioned with a force of 670 kN, which equals 70% of σ_y: 940 MPa.

4.2.2 Results and conclusion

The forces, moments and corresponding stresses are obtained at the transition between the bolt shaft and head or nut, location 1 and 2 respectively (tab.4.2). In case 4 and 5 a vertical force has been applied, case 6 and 7 represent a case where the structure has been translated to the right. Column 4 and 5 of table 4.2 show the bending stresses in y direction for the cases `nlgeom` ON and OFF. Column 6 indicates the % difference between these two cases. One should note that in case 6 the effect has been reproduced, a large % difference occurred between the bending stress of cases `nlgeom` ON and OFF. From the results of case 6 and 7 can be concluded that using the `fsum` command causes the increment in bending stress when `nlgeom` is switched ON. This is strange as the bending stresses should not change after applying a horizontal translation.

Case	El.type	Method	NLGEOM OFF	NLGEOM ON	% Difference
4.	SOLID186	FSUM	-1.3368E-01	-7.6108E+00	55.9330
5	BEAM188	FSUM	-7.2154E-03	1.0723E+00	147.6127
6	SOLID186	FSUM	-5.3803E+04	1.4667E+08	2725.0562
7	BEAM188	ETAB	-2.2995E+05	-2.2110E+05	-0.0385
8.	SOLID186	FSUM	-5.3803E+04	-1.9142E+05	2.5578

Table 4.2: Bending stresses geometry 2, location 1, loadcase: pretension and F_z (4-5) or U_x (6-7)

The command `spoint` is always used in combination with `fsum` and it is plausible that the location of the spoint is the reason for the increment in bending stresses. An increment of the bending moment equal to exactly the pretension force ($F_{prets} = 670kN$) times the displacement ($u_x = 0.01m$) could be observed when switching `nlgeom,on`. This increase indicates that the location of the `spoint` is not moving with the geometry. It is defined in the *undeformed* geometry and stays at this location after a translation of 0.01m has been applied, the *deformed* geometry.

Case 8, shown in table 4.2 confirms that the location of the `spoint` is indeed the reason for discrepancy in bending stresses between `nlgeom,on` and `nlgeom,off`. The `spoint` is defined in the deformed configuration and the difference in bending stresses has almost been disappeared. However, it must be noted that the use of beam elements in combination with the command `etab` still results is more accurate calculations of the bending stresses. For this reason it is recommended to use beam elements for modelling the bolts and use command `etab` to extract sectional forces and moments.

Chapter 5 Stress stiffening

In this chapter the phenomenon stress stiffening is further investigated. It refers to the influence of membrane forces on lateral deflection. Compressive membrane forces weaken the structure whereas tensile membrane forces result in stiffening [4].

In order to account for stress stiffening in a geometric linear analysis, meaning small deformations and rotations, a stress stiffening matrix (K_g) should be augmented to the regular stiffness matrix (K). These matrices are derived from strain-displacement matrices. It must be stressed that it is not necessary to run an geometric non linear analysis in order to obtain accurate solutions if rotations and deformations are small. Accounting for stress stiffening is sufficient for these cases.

The (non)-linear strain-displacement matrix is the derivative of a set of interpolation functions. These set of functions describe the curve of the beam in a deformed configuration. A set of cubic shape functions results in quadratic functions in the strain-displacement matrix. As the strains and stresses are also derived from this matrix these are able to vary quadratically between the nodes.

Regarding the equivalence of cables with long slender bolts, it is expected that taking into account stress stiffening during bolt assessment will result in higher bending stresses as a consequence of stress stiffening. This discrepancy between the situation with stress stiffening/no stress stiffening should be less visible in shorter thicker bolts [4]. However up to now it is unclear at which bolt length it is necessary to take into account stress stiffening. More insight can be obtained using numerical simulations however this will take a lot time. Another option is to use the analytical expressions for the stiffness and geometric stiffness matrix for beam elements and solve the equation $F = Ku$ using a symbolic solver. From the obtained solution an expression for the bending stiffness can be derived, which depends on the bolt length, L. Whit this expression one is able to show at which bolt length stress stiffening has dominant effects.

5.1 Analytical model

A combination of a bar and beam element, accounting for axial and bending deformation respectively are combined in a plane frame element (fig.5.1). The resistance to bending as a result of a force is called rotational stiffness (K_{rot}), deflection as a consequence of applying a moment is related to the flexural stiffness (K_{flex}). Analytical expressions of the stiffness and geometric stiffness matrix of this element, k and k_g respectively, are provided in equation 5.2 and 5.5 and correspond to the 2D Timoshenko beam and 1D bar element. The element stiffness matrix is a function of the youngs modulus (E), cross sectional area (A) first moment of area (I), length of the beam element (L), shear modulus (G) and effective shear area (A/k_y), where k_y depends on the geometry of the structure. The geometric stiffness matrix is a function of the length of the beam elements L and the pretension force (P). The use of 3D beam stiffness matrices was also possible however it would make the expression unnecessary complex. The description of displacement field in the v and w direction are the same if a beam with a symmetric cross section along y and z axis is used.

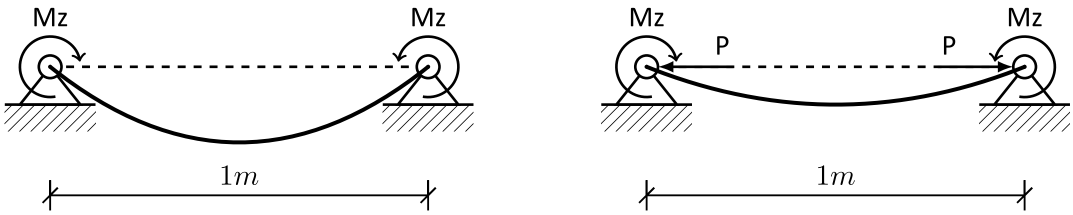
The structures shown in figure 5.2 and 5.3 are used to obtain the prescribed relation between K_{rot} /K_{flex} and L respectively. The corresponding system of equations is given in equation 5.1 with K and K_g the global stiffness and geometric stiffness. These global matrices are sparse matrices and

Figure 5.1: Degrees of Freedom 2D plane frame element

assembled from n element stiffness and geometric stiffness matrices [4]. Note the difference in notation between the global and element matrices.

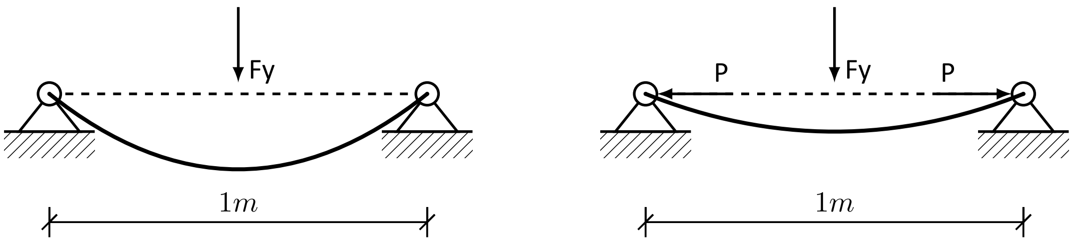
5.1.1 Changing I and E

As can be seen in matrix 5.2 and 5.5, K_{flex} and K_{rot} are functions of E, A, I, L, G, k_y and P. E is assumed to be constant, as bolts are made of steel. However changing I, or actually the shaft diameter d_n will influence the bending stiffness. The more slender the beam, the less bending stiffness is has. Instead of having a critical length at which the difference between K and K_g exceeds 5% it is more valuable to obtain a relation between K_{rot}/K_{bend} and the slender ratio $\frac{L}{A}$.



(a) No pretension (b) No pretension

Figure 5.2: Load case 1 to obtain K_{rot}



(a) No pretension (b) Pretension

Figure 5.3: Load case 2 to obtain K_{flex}

$$\underbrace{\begin{Bmatrix} F_{x1} \\ F_{y1} \\ M_{z1} \\ \vdots \\ F_{xn} \\ F_{yn} \\ M_{zn} \end{Bmatrix}}_{\text{Force vector}} = \underbrace{\begin{bmatrix} K_{1,1} & K_{1,2} & \cdots & K_{1,n} \\ K_{2,1} & K_{2,2} & \cdots & K_{1,n} \\ \vdots & \vdots & \ddots & \vdots \\ K_{n,1} & K_{n,2} & \cdots & K_{n,n} \end{bmatrix}}_{\text{Global stifness matrix}} + \underbrace{\begin{bmatrix} K_{g_{1,1}} & K_{g_{1,2}} & \cdots & K_{g_{1,n}} \\ K_{g_{2,1}} & K_{g_{2,1}} & \cdots & K_{g_{2,n}} \\ \vdots & \vdots & \ddots & \vdots \\ K_{g_{1,n}} & K_{g_{2,n}} & \cdots & K_{g_{n,n}} \end{bmatrix}}_{\text{Global geometric stifness matrix}} \underbrace{\begin{Bmatrix} u_1 \\ v_1 \\ \theta_1 \\ \vdots \\ u_n \\ v_n \\ \theta_n \end{Bmatrix}}_{\text{DoF}} \tag{5.1}$$

$$k = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EIL^3}{(1+\Phi)} & \frac{6EIL^3}{(1+\Phi)}L^3 & 0 & -\frac{12EI_zL^3}{(1+\Phi)} & \frac{6EI_zL^2}{(1+\Phi)} \\ 0 & \frac{6EIL^2}{(1+\Phi)} & \frac{(4+\Phi)EIL}{(1+\Phi)} & 0 & -\frac{6EI}{(1+\Phi)L^2} & \frac{(2-\Phi)EIL}{(1+\Phi)} \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{12EI}{(1+\Phi)L^3} & -\frac{6EIL^2}{(1+\Phi)} & 0 & \frac{12EIL^3}{(1+\Phi)} & -\frac{6EI}{(1+\Phi)}L^2 \\ 0 & -\frac{6EI}{(1+\Phi)L^2} & \frac{(2-\Phi)EIL}{(1+\Phi)} & 0 & -\frac{6EI}{(1+\Phi)L^2} & \frac{(4+\Phi)EIL}{(1+\Phi)} \end{bmatrix} \tag{5.2}$$

$$\text{with } \Phi = \frac{12EIk_y}{AGL^2} \quad ; \quad E = 2.11e^{11}Pa \quad ; \quad A = 0.001 \tag{5.3}$$

$$m^2 \quad ; \quad I = 8.24e^{-8}m^4 \quad G = 79.3e9Pa; \quad k_y = 1.1; \tag{5.4}$$

$$k_g = \frac{P}{L}\begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & \frac{6}{5} & \frac{1}{10}L & 0 & -\frac{6}{5} & \frac{1}{10}L \\ 0 & \frac{1}{10}L & \frac{2}{15}L^2 & 0 & -\frac{1}{10}L & -\frac{1}{30}L^2 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & -\frac{6}{5} & -\frac{1}{10}L & 0 & \frac{6}{5} & -\frac{1}{10}L \\ 0 & \frac{1}{10}L & -\frac{1}{30}L^2 & 0 & -\frac{1}{10}L & -\frac{2}{15}L^2 \end{bmatrix} \tag{5.5}$$

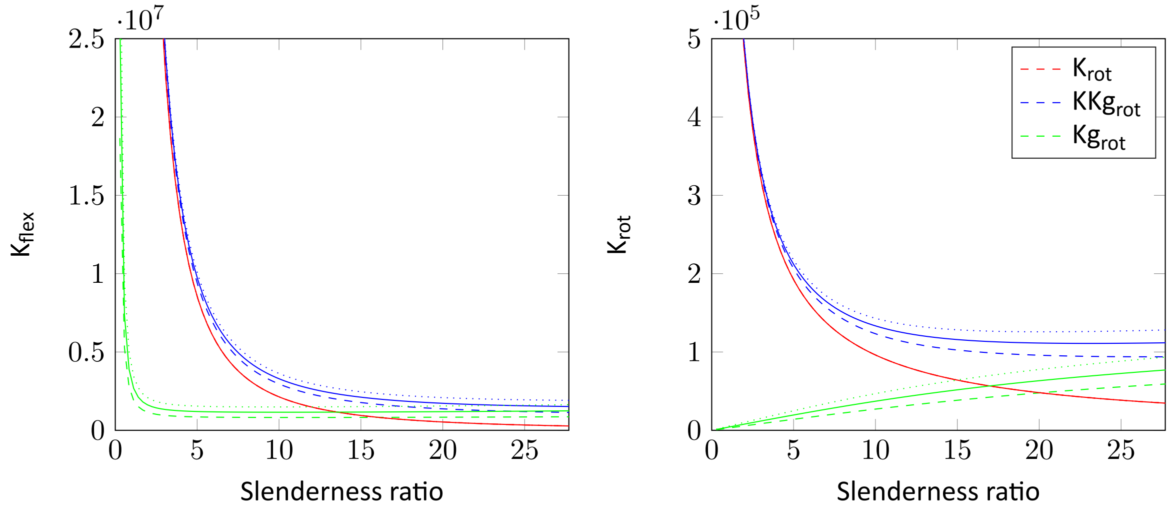
$$\text{with } P = 670kN \tag{5.6}$$

5.1.2 Boundary conditions and loads

The beam is modelled with two elements ($n = 2$ in eq. 5.1) resulting in 3 nodes and a total of 9 DoF. All cases shown in figure 5.2 and 5.3 have 5 DoF: two rotational ones (θ_1, θ_3) at node 1 and 3 and 3 DoF (u_2, v_2, θ_2) at node 2. The force vector is zero except for M_{z_1},M_{z_3} and F_{y_2}. In case of pretension F_{x_2} equals P.

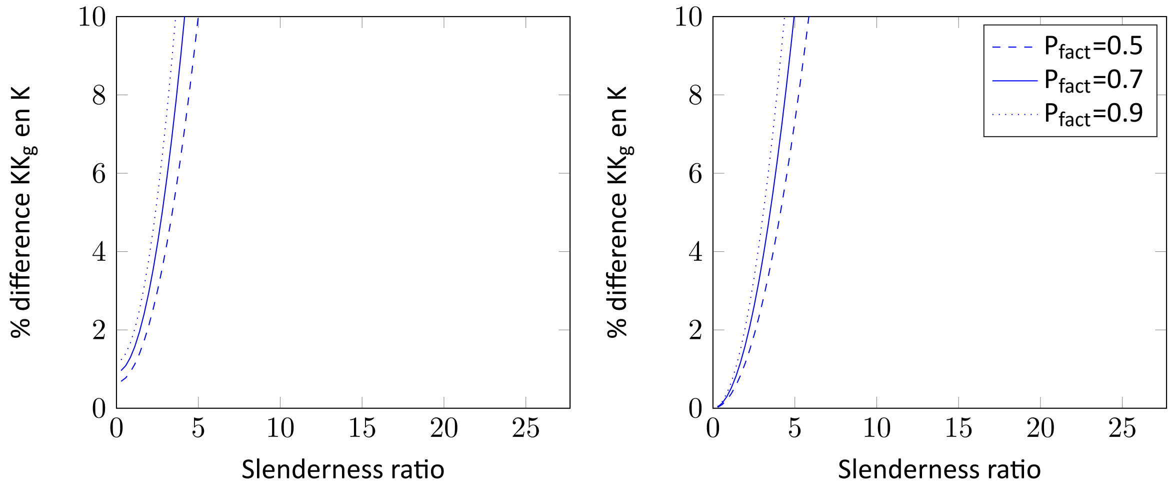
5.2 Results

The values for E, I, A, G, k_y and P are according to equation 5.4 and 5.6. L is varied between 0 and 1 m. Figure 5.4 shows the dependence of K_{flex}/K_{rot} on the slenderness ratio. K refers to the original stiffness, K_g indicates the geometric stiffness matrix and KK_g is the sum of K and K_g. The latter is the matrix used in FEM when stress stiffening has been taken into account. The percentage of K_g with respect to K is calculated and depicted in figure 5.4. For a pretension factor of P=0.7 of σ_y and slenderness ratio of 2.8 and 3.5 this K_g is more than 5 % of K for K_{flex} and K_{rot} respectively. Lower and higher values of P, representing the variation in pretension, result in higher and lower slenderness ratios respectively where stress stiffening should be taken into account.



(a) K_{flex} (b) K_{rot}

Figure 5.4: Bending stiffness dependent on slenderness ratio.
K_g,KK_g,K



(a) Percentage Kg_{flex} of K_{flex} (b) Percentage Kg_{rot} of K_{rot}

Figure 5.5: Percentage of K_g with respect to K

5.3 Validation

The results shown in section 5.2 are validated with Ansys 16.2. The same model, as presented in section 5.1, is built. Instead of two 2D planar frame elements, two 3D planar frame element are used: BEAM188. A BEAM188 element has 6 DoF at each node ($u_x, u_y, u_z, rot_x, rot_y, rot_z$) however it is constrained in such a way that it is similar to a 2D plane frame element described in section 5.1. The nodes at the ends (1 and 3) have only a rotational DoF (rot_z) and the node in the middle (2) has 3 DoF (u_x, u_y, rot_z). The length of the beam is 0.61m and is modelled using 2, 4 and 10 elements. To have a fair comparison with the results obtained from the analytical model, cubic shape functions are used to describe the displacement field (KeyOpt(3)=3).

A comparison of the results between Ansys and the analytical model is shown in table 5.1-5.3. From this table is observed that no difference occurred in the lateral deflection between cases where the bolt is modelled with 2, 4 and 10 beam elements. In addition the analytical and Ansys model gave the same results. From this may be concluded that the derived expression between L/A and K_{rot}/K_{flex} will be valid.

	K	K+Kg	SSTIF OFF	SSTIF ON
Rotz$_{left}$	-0.433	-0.134	-0.434	-0.140
Rotz$_{right}$	0.433	0.134	0.434	0.140
v	-0.108	-0.020	-0.109	-0.020
K$_{bend}$	3.46e4	1.12e5	3.46e4	1.07e5

Table 5.1: Matlab vs. Ansys 2 beams

	K	K+Kg	SSTIF OFF	SSTIF ON
Rotz$_{left}$	-0.433	-0.138	-0.434	-0.139
Rotz$_{right}$	0.433	0.138	0.434	0.139
v	-0.108	-0.020	-0.108	-0.020
K$_{bend}$	3.46e4	1.08e5	3.46e5	1.06e5

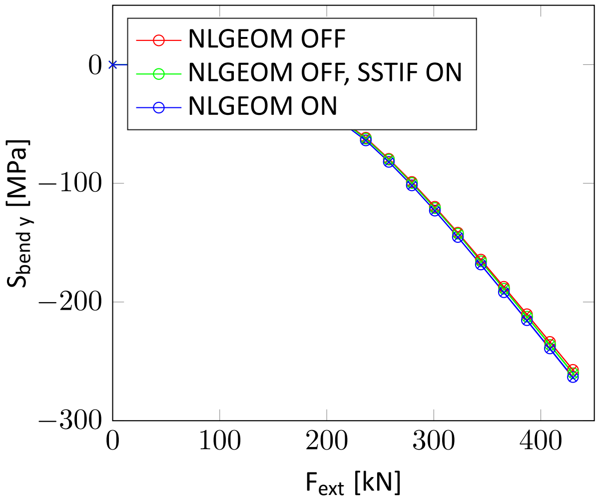
Table 5.2: Matlab vs. Ansys 4 beams

	K	K+Kg	SSTIF OFF	SSTIF ON
Rotz$_{left}$	-0.433	-0.139	-0.434	-0.140
Rotz$_{right}$	0.433	0.139	0.434	0.140
v	-0.108	-0.020	-0.109	-0.020
K$_{bend}$	3.46e4	1.08e5	3.46e4	1.08e5

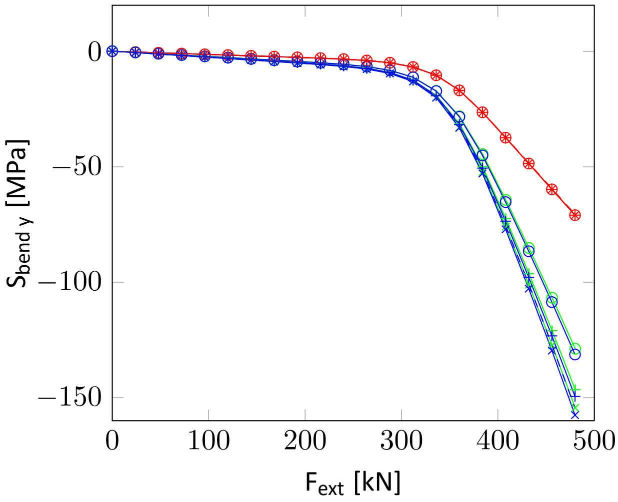
Table 5.3: Matlab vs. Ansys 10 beams

In the previous validation a model is used in which no solid elements and contact interfaces were present. This does not represent a realistic case and for this reason also a more complex model is used in order to identify if the effects of stress stiffening indeed should be taken into account for longer bolts. Geometry 2 (fig. 4.2), presented in chapter 4, is used for this simulation and again a bolt with length 0.1 m and 0.61 m are tested modelled with 2, 4 and 10 beams.

Figure 5.6 and 5.7 show the results of prescribed simulations. In case of quadratic shape functions, no differences occurred for the bolt of length 0.1 m between the situations where stress stiffening has/has not been taken into account. However, from figure 5.6b can be concluded that stress stiffening indeed plays a dominant role. In addition also the number of beams used to model the bolt affected the bending stress curve in the simulations where is accounted for the effect of stress stiffening. The more beams used, the higher the observed bending stresses. In case of cubic shape functions (fig.5.7) these differences are not present as these functions are able to exactly describe the deformation of a beam element [4]. One should note that cubic shape functions (KeyOpt(3)=3) are only used if `etcontrol,off`, otherwise ANSYS changes the element setting automatically to quadratic shape functions.

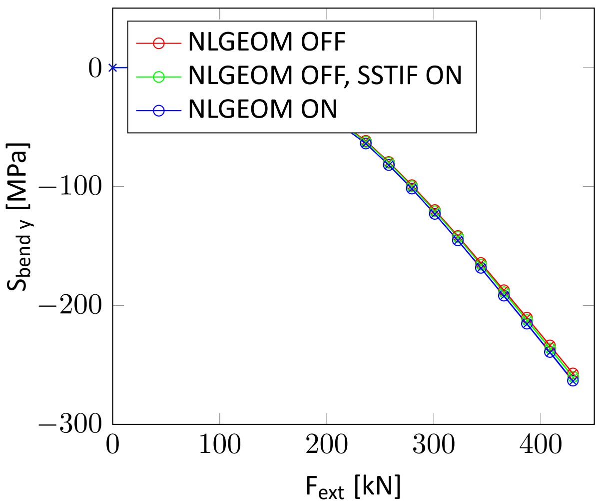


(a) S_{bend} L=0.1m

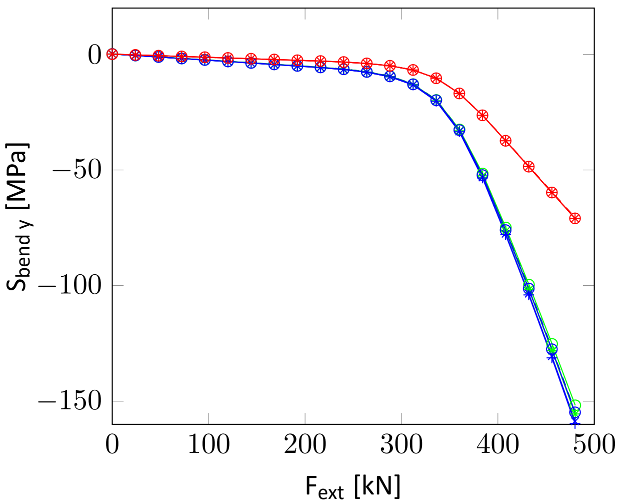


(b) S_{bend} L=0.61m

Figure 5.6: Analysis with quadratic shape functions.
Marker indicates number of beams: o=2 beams, +=4 beams, x=10 beams, –=solid elements



(a) S_{bend} L=0.1m



(b) S_{bend} L=0.61m

Figure 5.7: Analysis with cubic shape functions.
Marker indicates number of beams: o=2 beams, +=4 beams, x=10 beams, –=solid elements

5.4 Conclusion

In chapter 4 it is recommended to asses bolted connections with beam elements and obtain the forces moments occurring at the assessment points (fig.2.2) using the command `etab` in ANSYS. As mentioned in section 2.1 the general number of elements used is 4. In order to have a good approximation of the displacement field and corresponding stresses and strains, the use of cubic shape functions during strength assessment of bolted connections is strongly recommended. Cubic shape functions are defined by the commands `KeyOPt(3)=3` and `etcontrol,off`. An alternative option is to use quadratic shape functions under the condition that at least 4 beam elements represent the bolt. This conclusion is based on the similarity of the curves from figure 5.6 with 4 beam elements and solid elements and axial stresses below σ_y.

Regarding the effect of stress stiffening and the average slenderness ratio of the hub and blade, is recommended to include the effect of this phenomenon in the strength assessment procedure of bolted connections by switching `sstif,on`.

Chapter 6 Conclusion and recommendations

The effects of geometric non linearity and stress stiffening on the strength assessment of bolted connections have been discussed in the previous chapters. Based on this it is recommended to change MECAL's strength assessment procedure. The changes related to geometric non linearity and stress stiffening will be described in section 6.1 and 6.2.

6.1 Geometric non-linearity

It is not necessary to account for geometric non linearities based on the results of the initial bolt study presented in section 3.1. However, as stated in chapter 3, it is possible that geometric non linearities should be taken into account in the near future as a consequence of the increase in the size of the turbines and the reduction of the relative stiffness of components due to mass optimization. For these situations, where `nlgeom` is ON, it is recommended to model the bolts with beam elements and use the command `etab` to obtain the forces and moments occurring in the bolts. An alternative option is to model the bolt with solid elements and extract the sectional forces and moments with the command `fsum`. In the latter case, it is stressed that the summation point for the forces and moments, `spoint`, should be defined in the deformed geometry.

6.2 Stress stiffening

Regarding the effect of stress stiffening and the average slenderness ratio of the hub and blade bolt, is recommended to include the effect of stress stiffening in the strength assessment procedure of bolted connections by switching `sstif,on`. It is advised to model the bolt with 4 beam elements based on cubic or quadratic shape functions.

6.3 Recommendations

In order to show the effect of stress stiffening in for example a pitch bearing bolted connection, results of the simulation where is/is not accounted for the effect of stress stiffening should be compared. Based on this comparison one can conclude if it is necessary to include the effect of stress stiffening in the strength assessment of bolted connections in the future or that the increase of the bending stresses is already covered by the safety factor.

Bibliography

[1] R. voor ondernemend nederland. Normen en certificaten. organization. [Online]. Available: http://www.rvo.nl/onderwerpen/duurzaam-ondernemen/duurzame-energie-opwekken/windenergie-op-land/wetten-en-regels/normen-en-certificaten

[2] E. Leijten, *WTD-WI-025: Bolt connections strength assessment*, MECAL.

[3] *Systematic calculation of high duty bolted joints joints with one cylindrical bolt. Systematische Berechnung hochbeanspruchter Schraubenverbindungen Zylindrische Einschraubenverbindungen*. Berlin: VDI, 2003. [Online]. Available: https://cds.cern.ch/record/1478944

[4] R. D. Cook *et al.*, *Concepts and applications of finite element analysis*. John Wiley & Sons, 2007.

[5] I. ANSYS, *Theory reference for the mechanical APDL and Mechanical Applications*, ANSYS.