Structural Analysis and Design of a Warehouse Building

HAMK
UNIVERSITY OF APPLIED SCIENCES

Bachelor's thesis

Degree Programme in Mechanical Engineering and Production Technology

Name of unit spring/autumn year

BEMPNU10A6/Autumn 2014

Harun Mugo Thande

Clarification of signature

HAMK
UNIVERSITY OF APPLIED SCIENCES

ABSTRACT

BEMPNU10
Mechanical Engineering and Production Technology
Design of Mechanics

Author Harun Mugo Thande **Year** 2014

Subject of Bachelor's thesis Structural Analysis and Design of a Warehouse building.

ABSTRACT

The thesis was about structural analysis of identified parts of a warehouse building. The parts analysed included: roof truss, columns and the joints of the structure. The parts of the building chosen were considered to be the most crucial especially given the loading conditions. The three major loads acting on the structure included: snow load, wind load and the structure´s own weight. The main purpose of the analysis was to identify parts of the building which experienced high stresses.

Until recently, analysis of complex structures proved to be difficult and consumed too much time. Credit to the highly powerful computers and research work, efficient methods of structural analysis have been developed.

Finite element method was the analysis method chosen for this thesis. The method investigated the most vulnerable parts due to high stresses. This type of analysis initially simplified the problem in order to see the bigger picture. Details of the structure were then singled out and analysed. For example the truss and column were first analysed then the individual members of the truss were independently investigated.

Parts of the structure that experienced high stresses were redesigned in order to reduce the stress levels. For example for the truss members, cross section properties were changed which increased the second moment of area. The type of material used on the beams was changed to increase the stiffness. In a nut shell the analysis showed the most vulnerable parts due to high stresses. The next step was to isolate the members and manipulate the design so as to cater for the high stresses.

The analysis covers majority of the thesis and is the most important part. There were many dependent steps involved in the analysis. Ninety percent of the structure did not experience high stresses. Redesigning was crucial to support the analysis.

Keywords Finite element method. Stress analysis. Modelling.

Pages 73 p. + appendices 13 p.

CONTENTS

1 INTRODUCTION

The objective of the thesis is to improve the design of a warehouse building. Structural analysis is important as the reliability of the structure is investigated. Can the building withstand the loading conditions? That is the question asked during most of the analysis. The structural analysis is essential since it identifies the critical parts that need special attention. Furthermore, the analysis helps to understand the design of the structure in more detail. Every part of the structure has a purpose and this should be identified before any adjustments are made. Figures one and two below show the physical real building including the interior parts that will be analysed.

Figure 1 Outside view of the building

Figure 2 Interior truss members

The structure to be analysed is a warehouse building used to store farming equipment and products. The building experiences a lot of stresses in different parts due to various loading conditions. It is not practical to analyse the building as a whole. For more detailed information, the structure is broken down to different smaller parts for easier examination. Also, different parts of the building serve more important roles than others. In this thesis the roof truss, column support and joints are assumed to be the most crucial parts. One of the technical drawings of the building is shown in figure 3 below. This technical drawing is modelled in CAD software and analysis then takes place. This is demonstrated later.

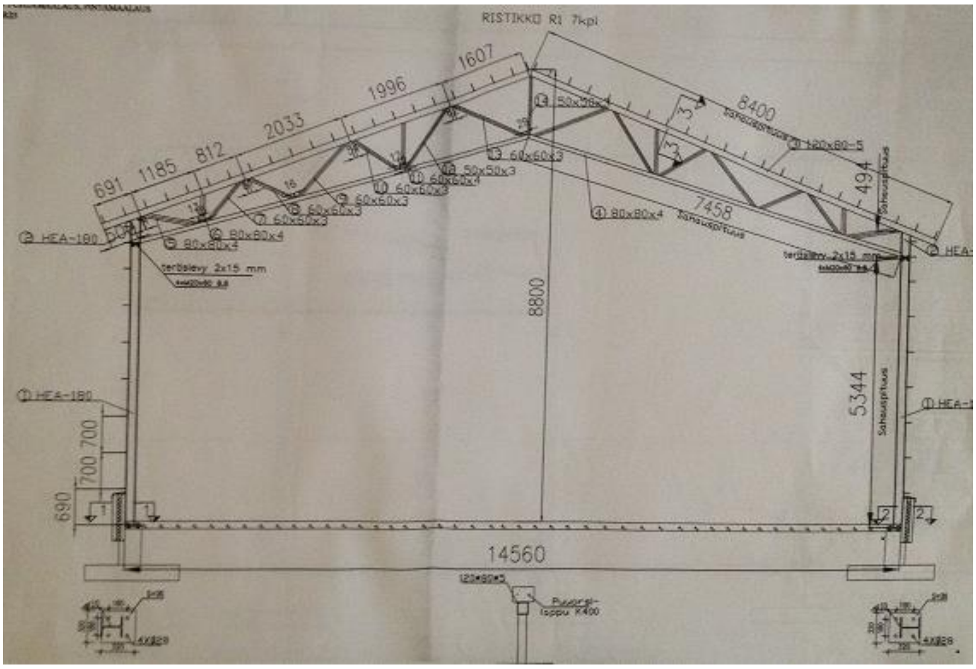


Figure 3 Technical drawing of truss and column assembly

Loads act on a structure in many directions. Sometimes the load acts alone and sometimes simultaneously. Worst case loading conditions which result into the highest stresses are used to analyse the building. Parts of the structure that experience high Von-Mises stresses or high buckling are isolated for further analysis.

Engineering principles mainly used in this analysis derives knowledge from the following topics:

- Statics
- Strength of materials
- Machine elements
- Material selection
- Finite element method for mechanical engineers

The new designs are later introduced to the analysis to check if the high stresses are reduced. Redesigning the members is an important part of the analysis.

In addition to the redesign, a new office structure is designed from a concept idea to a real structure. The new structure is an office for the warehouse manager.

1.1 Brief Introduction of The Analysis

The method of analysis used is highly crucial since the results almost entirely depend on the procedure used to examine the structure. It could be a daunting and error prone task if this examination is done manually.

It is for that reason that finite element method (FEM) is used for this entire analysis. This method is a convenient and faster way to carry out the structural analysis.

Individual drawings of the structures to be analysed are modelled in the software to replicate the real structure. The forces and constraints are also modelled in the software. Figure four below shows the difference between a real structure and modelled structure.

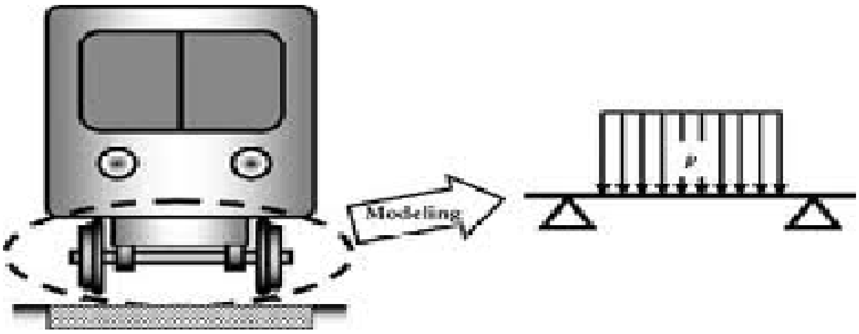


Figure 4 Model of a train´s axle shaft (Tadeusz Storlaski, 2006)

Modelling involves replicating the drawing in the computer software. The most important information in the models include: beam cross section properties, forces and constraints (boundary conditions) of the structure. However, caution is exercised during this process since inaccurate modelling of the structures in the software can lead to accumulated errors.

The next step is to simulate the model to find out the results. Results show the Von-Mises stresses, buckling and displacements of the model. Figure five below shows an example of a simulated model. The results are displayed graphically. The regions with red indicate high stresses or displacements while the blue ones show low stresses or displacements.

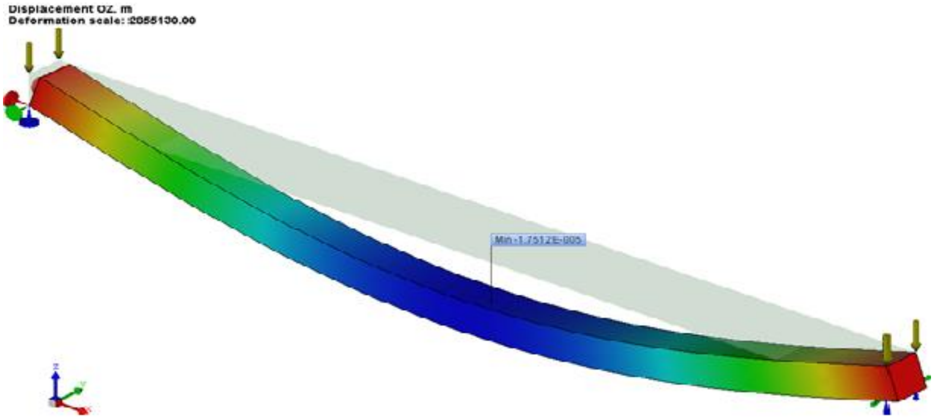


Figure 5 Simulated model (Auto fem, n.d)

This is how part of the analysis process is done but more detailed information is explained in the next chapter.

1.2 Expected Challenges

The process of analysis is not straight forward even with the help of highly advanced computer software. The two main challenges anticipated are maintaining high accuracy and consistency in the procedures. All the results should make sense and are expected to be realistic. If not careful, very bizarre results and conclusions are obtained. A familiar saying used when using computer software “garbage in garbage out” should be always remembered. That is why it is good to know the challenges beforehand so that they can be tackled when necessary.

It is highly important that the level of accuracy remains at a high level throughout the thesis. As mentioned before, accumulated errors lead to inaccurate results and wrong designs. There are many situations that can lead to wrong data being used for the analysis. For example during modelling some details might be omitted from the drawings which affect the final results.

All the steps are double checked just to make sure everything is correct. Simple strength of materials calculations help to improve the accuracy levels. The results should be reasonable to increase the confidence levels.

The method of solving problems should be the same throughout the analysis because it is then easier to compare results and designs. For example the units used in the beginning of the analysis should be used during the whole thesis work. It could be catastrophic if English units were used in one calculation and all the others used metric units.

Avoiding errors and having a good consistency is important during this thesis work.

2 THE FINITE ELEMENT METHOD

2.1 Introduction

Physical problems exist in different categories of engineering for example; solid and fluid mechanics, electronics, dynamics and thermodynamics. Numerical analysis is a technique used by engineers to solve differential equations which best describe the physical models. Finite element method (FEM) is an example of a numerical technique that is used to solve the physical problems.

FEM analysis can be used in many fields. Some of the fields include:

- Structural analysis (stress, strain, buckling and modal)
- Temperature analysis
- Magnetic and electrical analysis
- Crush simulations
- Connected problems (wind load on a building causing deflections)

In structural analysis, FEM is used to investigate how the applied forces will affect the product design. Complex structures are analysed better with the help of FEM method because hand calculations are not able find the solution. Highly powerful computers are essential since FEM method involves solving numerous simultaneous equations.

The residential building is a complex structure. Calculations that yield direct solutions during analysis are not possible due to the complex residential building which has complex loading conditions and geometries. This is why numerical methods such as FEM are used to find the solutions.

Numerical method procedures involve providing a sequence of approximations by repeating the procedure again and again. (Numerical Methods, n.d) In short, direct basic mathematics such as calculus cannot solve the problems. Numerical methods which find the approximations of the solutions are used. This is only the basic information concerning the mathematical theory behind finite element theory. The focus is on how the software is used but not the theoretical background of FEM.

There is an increasing trend in simulation of designs and FEM is a major reason for this capability. The simulations are important especially when experimenting with new designs. They eliminate the cost of testing since the actual design (prototype) is not produced and tested in reality. So not only does the FEM method aid in the analysis, it is also capable of testing the new designs in simulations.

FEM involves breaking down the problem into small elements and finding individual solutions. The elements are connected by nodal points and boundary conditions are explicitly defined in the beginning. The nodes can translate, rotate or remain fixed and this is clearly defined when modelling the structure. If the nodes experience displacement the elements also shift position. The finite elements are again joined together to find the final approximate solution for the whole structure.

Figure six shows the real physical object and a discretized model divided into finite elements with boundary conditions. The results from FEM software are well presented and easy to interpret. However, the results should be double checked.

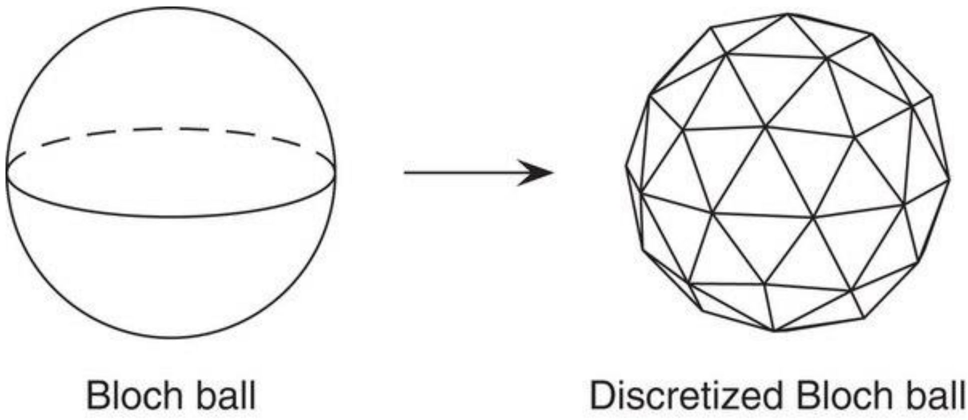


Figure 6 Physical model and discretized model (Stephanie Wehner, 2013)

2.1.1 FEM Computer Software

There are many computer programs that perform structural analysis. The programs are divided into two: small specific programs and heavy duty commercial programs.

The specific programs are designed to solve singled out problems and cannot handle different types of problems. These programmes are cheaper and widely available. On the other hand the commercial programs can solve a wide variety of problems and perform design simulations.

The analysis in this thesis uses both types but mostly the commercial program because of the complex nature of the analysis. The programs used are: Math-cad based program and Creo-simulate. The smaller program used is a Mathcad based developed by Esa Murtola who is a HAMK lecturer.

Several other commercial programmes are available and can be used to perform the same analysis. They include:

- ANSYS
- Abaqus
- ANSA
- ALGOR

The list is very long and the choice of program that is used for this thesis is explained in detail later in the next sub-chapter.

2.1.2 FEM Analysis Procedure

The following steps are followed when using the FEM software Creo simulate. (Toogood, 2012, 2-5)

- Create the geometry with Creo parametric (CAD software)
- Transfer the model to Creo simulate
- Add the simulation parameters (material properties, model constraints and loads)
- Run the model (model is discretized to form finite element mesh)
- Display desired results

The steps are also illustrated in the figure 7 below.

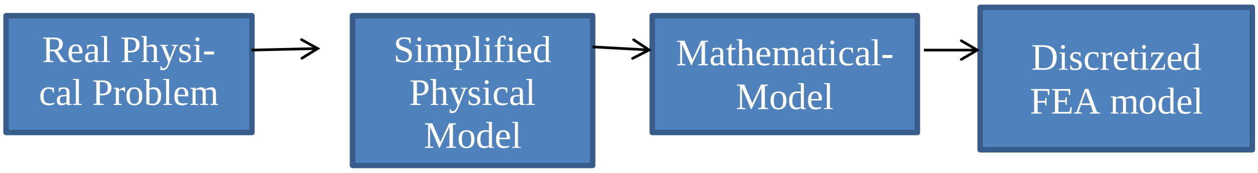


Figure 7 Steps to obtaining results in FEM analysis (Toogood, 2012, 2-5)

When the final results are calculated, it is essential to verify if the results are accurate. This verification is done by a procedure known as convergence analysis. Convergence uses the same problem to display different results so as to determine which results are accurate and should be accepted.

The method used for convergence analysis splits FEM programs into two groups. One group uses h- elements while the other uses p-elements for convergence. Creo-Simulate use p-elements while other programs such as ANSYS use h-elements for convergence analysis.

<u>H- Elements</u>

Mesh refinement is the convergence method to achieve better results in these FEM programmes. Mesh refinement involves making the elements smaller in order to obtain more accurate results. Specific sections of the models are highlighted and the mesh refinement takes place to get more reliable results.

The nodes connect the small elements together and therefore obtaining the solution starts by calculating the displacement of the nodes. Mathematically, the displacement of the node in its simplest form is a linear function. The derivative of the displacement obtains the strain and by using the modulus of elasticity the stress is obtained. This is demonstrated by the following two equations.

$$\varepsilon_x = \frac{d}{dx} u \qquad (1)$$

$$\sigma_X = E \cdot \varepsilon_X \tag{2}$$

With large elements the stress distribution is the same in the structure which is not true in reality. (Toogood, 2012, 2-8)

This is represented in figure 8 below where the nodes are at a wider distance to each other before the mesh refinement.

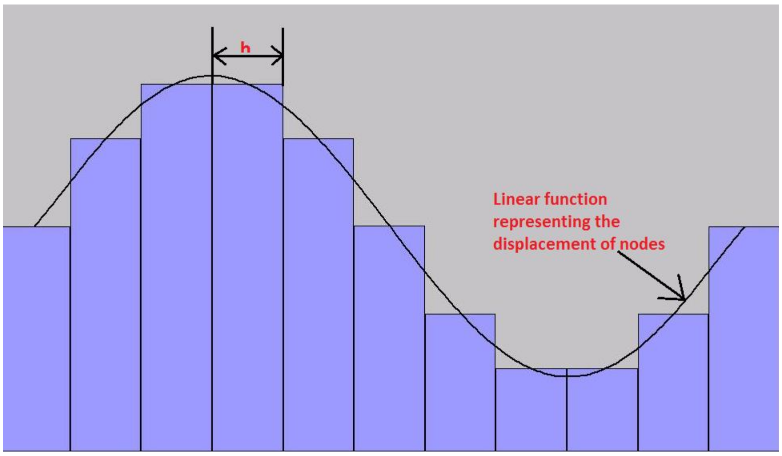


Figure 8 Big elements represented by the wide bars (H method, n.d)

The mesh refinement involves reducing the size of the elements which consequently reduces the length of the nodes shown in figure 8 as h. This then makes the distribution of stress in the structure to be more accurate and unevenly distributed.

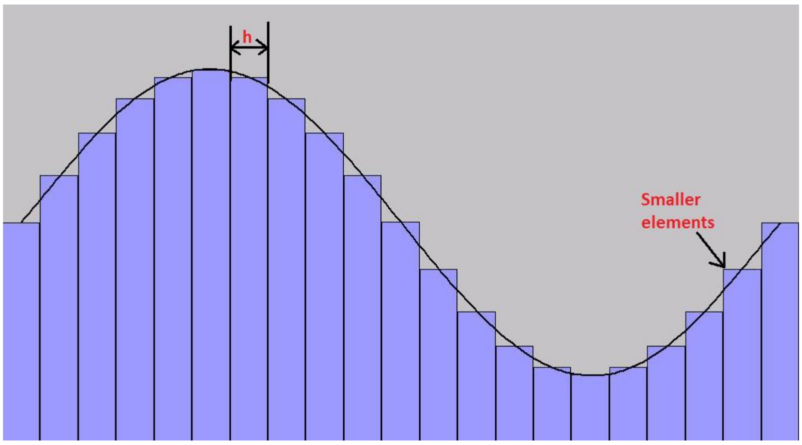


Figure 9 Smaller h length of nodes (H method, n.d)

The mesh refinement is represented in figure 9 and this leads to more accurate results. The only downside to this mesh refinement is that the smaller elements increase the computational time. A compromise has to be made between time and accuracy. (Toogood, 2012, 2-10)

P- Elements

In p elements, the size of the elements remain the same. The order of interpolating polynomials in the elements affects the accuracy of the results. Lower order polynomials result to inaccurate results and higher order polynomials have better results. Convergence occurs by changing the order of the polynomials. The size of the element remains the same but the complexity of the element changes.

2.2 Typical sources of errors in Finite Element Method

There are many steps involved in the process of analysis. Every detail or assumption must not lead to false results. Loss of information as one proceeds from one step to another also causes errors in analysis. During this thesis, every step is reviewed before proceeding to the next one just as a precaution. Figure 10 below shows origins of typical error sources.

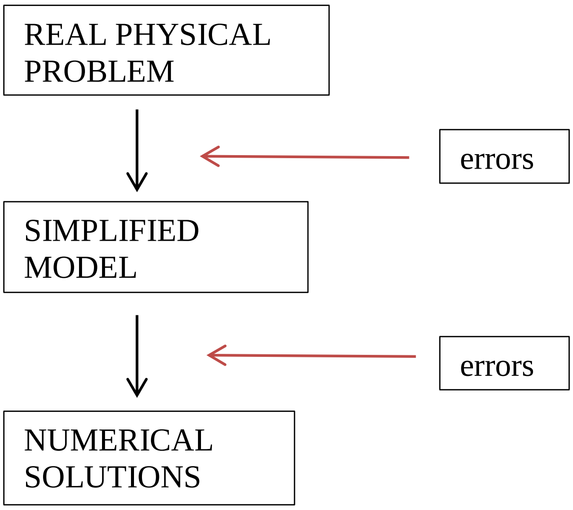


Figure 10 Error sources

Physical Modelling

Sketching of the physical model into CAD software is difficult especially for complicated structures. Information might be lost when making the 3-D model. For instance the dimension can be different from the real model. This can lead to minor errors in the solution.

Simulation Parameters

This is the main source of errors. Setting up the parameters determines the type of results received. For instance big structures have a lot of beams and every beam must be assigned at the correct position. During the modelling the magnitude of the loads and direction should depict the real situation. Defining the constraint is a critical step and should also be well defined to avoid errors.

Rounded off values

As stated earlier on, numerical methods are used to compute the problems in the calculations. These problems are very complex and involve huge numbers which are rounded off. These accumulated errors from rounding off can lead to erroneous results.

Convergence analysis is done at the end of the simulation to reduce these errors. However, during this thesis, simple calculations based from strength of materials are used to back up some of the results.

2.3 Difference between CAD and FEM models

In modelling, knowing the difference between a CAD model and a FEM model increases the accuracy of the analysis. A CAD model entails all the details of a physical model including the chamfers and the rounding. A CAD model is mainly used for the purpose of manufacturing. Simple CAD models which do not have too much detail might also be used simultaneously in FEM and CAD programmes.

A FEM model on the other hand is used for structural analysis. Details such as surface finish might not be required. In some cases, symmetry plays a big part in analysis. Symmetry simplifies the problem and reduces the amount of elements and equations to be calculated leading to more accurate results.

The following figures 11 and 12 help explain in more detail the difference between a CAD and FEM model.

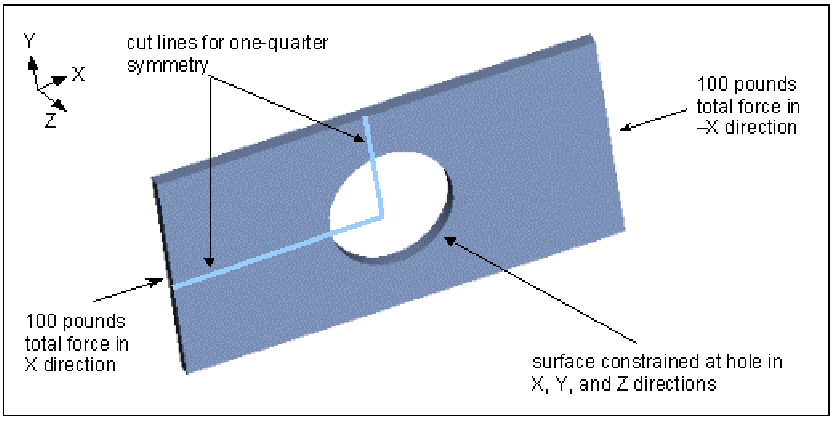


Figure 11 CAD model (Using mirror symmetry, n.d)

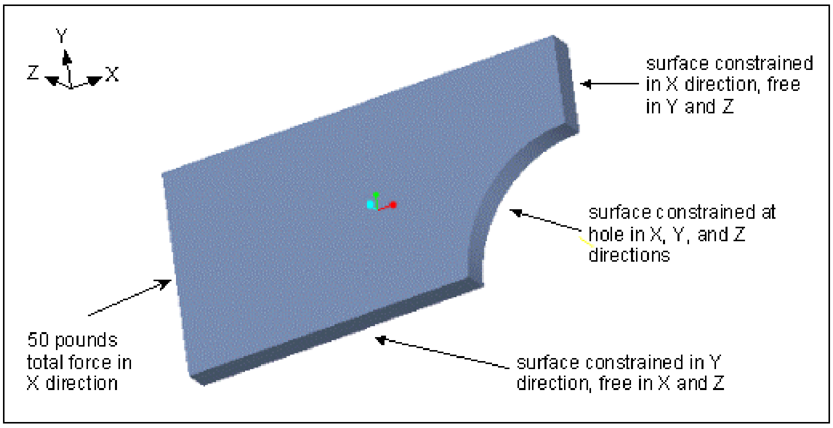


Figure 12 FEM model (Using mirror symmetry, n.d)

3 DEFINATION OF LOADS AND STANDARDS

3.1 Introduction

It is common knowledge that buildings should be designed to resist forces that might cause damage. Structures should be strong and stiff to withstand the stresses caused by the loads. It is therefore very important to know the anticipated loading conditions. Calculating the loads acting on a structure determines the allowable stress values for design. These values determine the design of the joints, columns and beams used in constructing the building.

Buildings are designed for a specific purpose. For instance a disco and a residential building are designed differently because they have different loading conditions. A disco has many people and equipment so it is designed to experience higher loads. On the other hand a residential building has less furniture and people compared to the disco so it is designed to withstand less loading.

Euro-code standard states that there are different quantities of loads for different types of buildings. “Areas in residential, social, commercial and administrative buildings shall be divided into categories according to their specific uses.” (EN 1991-1-1:2002:20-21).

Loads on buildings are classified into two major categories. Gravity loads and lateral loads. Gravity loads pull vertically downwards due to gravity while lateral loads act in the horizontal direction. They in turn have sub categories as shown in figure 13.

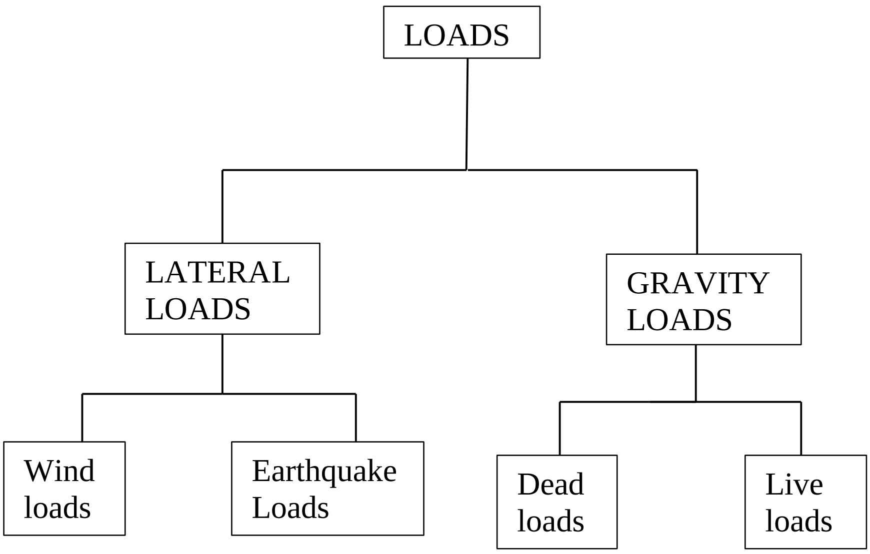


Figure 13 Classification of loads

These are the major loading types which act on a building. They can act alone or like in many cases occur together. Tracing the loads from one part of the building is very important since not all structure elements experience direct forces. Loads on a surface area are expressed in Newton´s per square meter (N/m^2) while those on linear elements such as a beam are in Newton´s per meter (N/m).

3.2 Dead Load

As the name suggests this type of load does not change over time and acts permanently on the building. In the euro code standards, dead loads are referred to as permanent actions. The definition states that “the self-weight of construction works should be classified as a permanent fixed action.” (EN 1991-1-1:2002, 12) Permanently fixed structure such as finishing that remains fixed is also classified as dead load.

The total weight of a structure might not be directly available in most cases. Also, redesigning the structure leads to change in total weight. Material properties such as density and volume of the individual members of the structure are used to calculate the weight. The following formulas 3 and 4 are used to calculate the weight of structures.

$$\text{Mass} = \text{Density } (\text{Kg/mm}^3) * \text{Volume } (\text{mm}^3) \quad (3)$$

$$\text{Weight} = \text{Mass } (\text{Kg}) * \text{Gravity } (\text{m/s}^2) \quad (4)$$

3.3 Live Load

These loads change over time and are temporarily attached to a building. They result from using and occupying the building. Environmental or human interactions are examples that cause live loads.

3.3.1 Snow Loads

They are a sub category of gravity loads and hence acts vertically on the roof. Snow load varies and changes with the location of a building. Therefore, different designs due to the snow loads are required. Unaffected snow measured from the ground is a good estimate of how much snow is on the roof. The figure below shows an example of accumulated snow load on a building.

Figure 14 Snow load accumulated on a building (North roof load zone, n.d.)

(EN 1991-1-3: 2002, 17) Accumulation of snow on the roof is influenced by the following factors.

– the shape of the roof
– Heat generated below the roof
– Distance of close by buildings
– Surface roughness of the roof
– The surrounding terrain.

Standards are used to calculate the snow load due to the many factors listed above. The snow load on roofs is determined by formula 5.

$$S = \mu_i * C_e * C_t * S_k \qquad (5)$$

Where μ_i is snow load shape co-efficient, C_e is the exposure co efficient, C_t is the thermal coefficient and S_k is value of snow load on the ground depending on the geographical position. (EN 1991-1-3: 2002, 18).

3.3.2 Wind Load

Wind acts horizontally on a structure and changes in magnitude and direction with time. Wind pressure might lead to dynamic responses from the building. Hence in some cases it might lead to fatigue stresses especially on the foundation. Wind load effects on a structure are affected by the following factors:

– The height above the ground; obstacles on the ground level reduce wind speed.
– Exposure of the building to its surroundings; trees and other tall buildings block the wind speed.

The wind load is mainly resisted by proper anchoring of the foundation and adding stiffening elements. Lateral forces tend to force structures to move horizontally and this makes the foundation to experience high stresses. Stiffening elements such as braces help to maintain columns into their original position. The figures 15, 16 and 17 below demonstrate the effects of wind pressure on a structure.

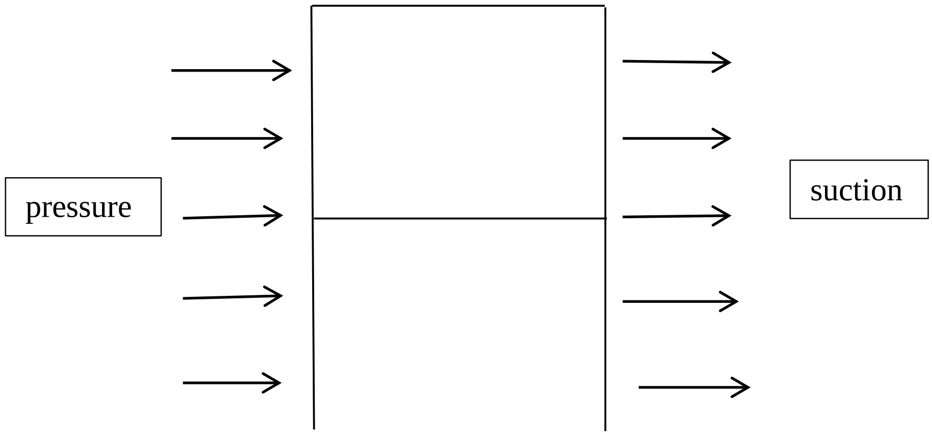


Figure 15 Wind pressure and suction on a building (The effects of imperfections, n.d)

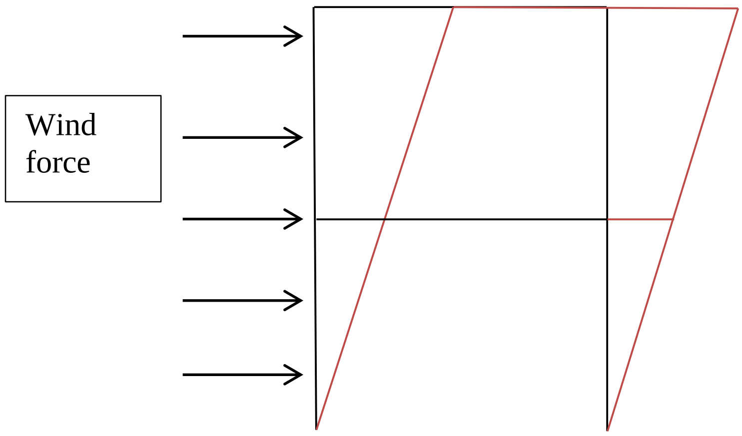


Figure 16 Deformed building due to wind force (The effects of imperfections, n.d)

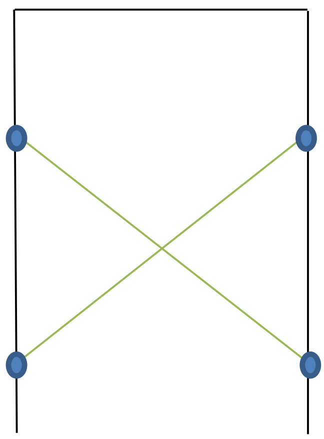


Figure 17 Bracing elements between columns (The effects of imperfections, n.d)

3.4 Earthquake Loads

An earthquake is a vibration that travels on the ground. Several modes of vibration are expected to occur depending on the height of the building. Earthquakes vary in magnitude depending on the geographical location of a building. The earthquake load induces dynamic loading on the foundation of a building leading to shear and fatigue stresses and also causes deformation of a structure. Design of the building requires that the structure can withstand some levels of displacement at the base (Murty, n.d, 1-5)

The inertia force experienced leads to the damaging of the structure. It happens so that the base of the building moves while the upper part moves in the opposite direction leading to inertia force on the roof. This causes buckling on the columns of the building. That is the basic way how the damage occurs due to the earthquake. It is important that the columns are designed to withstand high buckling forces.

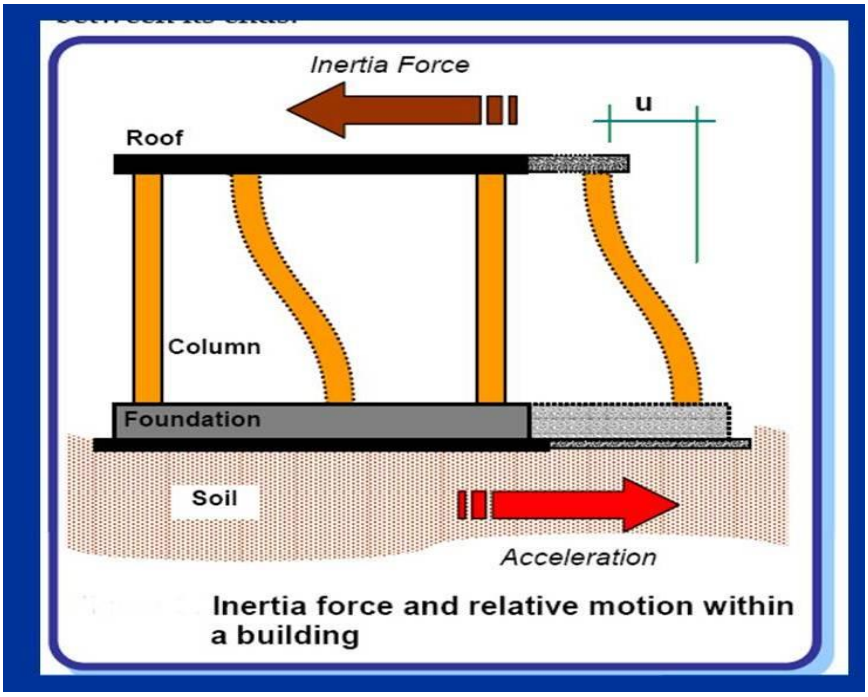


Figure 18 Effects of earthquake on a building (Flowing events, nd)

3.4.1 Designing to prevent earthquake

(Murty, n.d, 2) Earthquake resistance design is based on the following principles.

- The building can experience minor shaking
- Moderate shaking with minor movements of the building
- Extreme shaking but without total collapse of the building to protect human life and property

Damping the vibration is the basic principle of designing against earthquake forces (damping absorbs vibration). The base of the building should have damping structures installed. Efficient design can also be achieved by installing damping devices on the bracing of the building.

Choice of construction material is important to reduce the effects of the earthquake that occurs after vibration. Ductile materials deform over a longer period than brittle materials. Materials such as structural steel are highly recommended. Materials with high stiffness are essential especially for the columns since they prevent the buckling effect.

Bracings in between columns help to restrict lateral movements of the columns. Installing damping devices on the bracing of the building makes the resistance even higher.

3.5 Tracking and defining the loads

It is very important that the loads are accurately defined in the beginning. This is usually a major source of errors for the analysis. Basic hand calculations are used to define the forces caused by the loads. Standards are also used to further determine the exact loads.

3.5.1 Basic calculations for determining the loads

In the standard manuals, most of the loads are given in pressure units for example $(kN/m\text{^}2)$. In some cases the pressure load needs to be converted to a uniformly distributed line load (kN/m). The line load as well is in some cases need to be in form of a point load (N). It is important that the techniques of converting the loads are well defined.

Pressure load to line load

To find a uniformly distributed load on a pressure surface, equation 6 is used. Choosing the length depends on the axis that you wish the line load to be. If line load one is desired, the length perpendicular to the line load for example b is used and vice versa as it is shown in figure 19.

$$\text{Pressure load }\left(\frac{kN}{m^2}\right) * \text{ length(m)} = \text{Line load}\left(\frac{\text{kN}}{\text{m}}\right) \qquad (6)$$

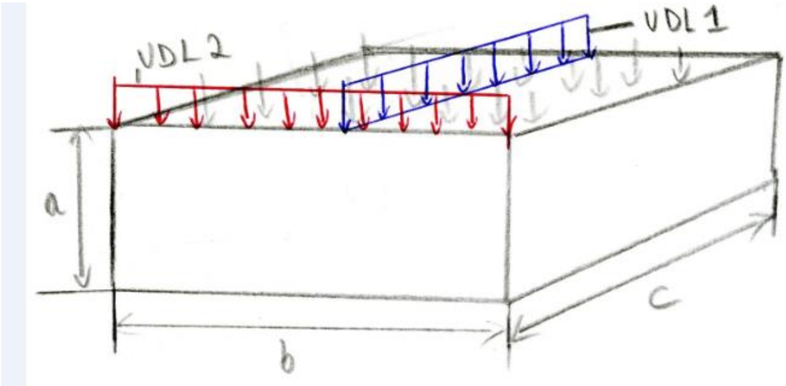


Figure 19 Pressure load to uniformly distributed load

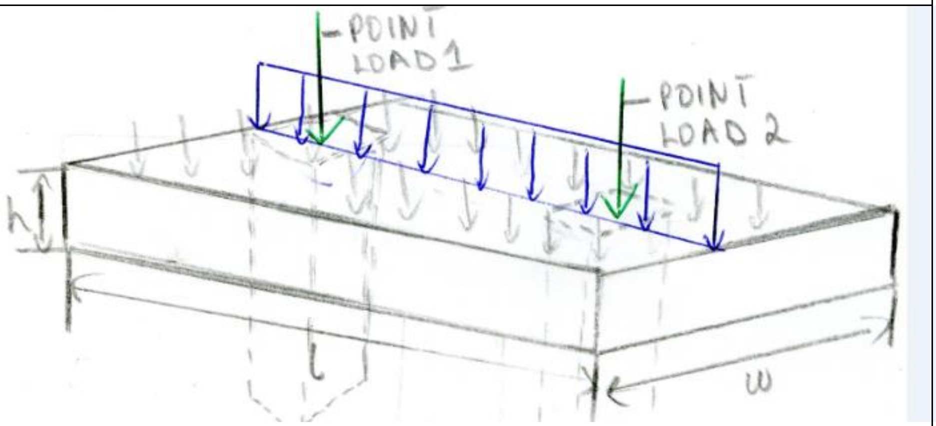
Line load to point load

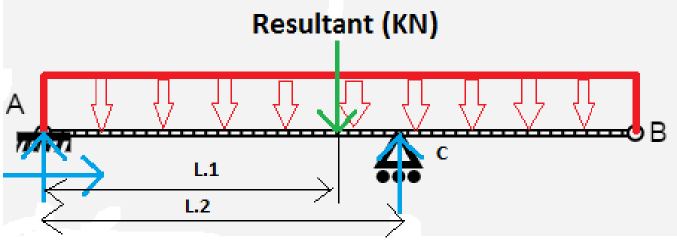
Since the structure is static, equations of equilibrium are used to determine the point loads. Equation 7 is used to determine one of the forces involved in the statics calculation.

$$\text{Line load } \left(\frac{kN}{m}\right) * length(m) = Resultant\ load(kN) \qquad (7)$$

Table 1 below demonstrates the calculation involved.

Table 1 Line load to point load steps

Stages	Schematics and free body diagrams
Stage 1:Physical problem	

<table>
<tr><td>Stage 2: Free body diagram</td><td>
</td></tr>
<tr><td>Stage 3: Equations of equilibrium</td><td>$\Sigma\, f(x) = 0 \quad \Sigma\, f(y) = 0 \quad \Sigma\, M = 0$

$\Sigma f(y) := A_y + C_y - \text{Resultant}$

$M_A := C_y \cdot l_2 - \text{Resultant} \cdot l_1$

$C_y = \frac{\text{Resultant} \cdot l_1}{l_2}$

$A_y = \text{Resultant} - C_y$</td></tr>
</table>

The force calculated in equation 7 is used to find the resultant forces at A and B which represents the point loads of the two columns.

3.6 Conclusion

It is important to identify all the loads that act on a building for the purpose of stress calculations. Loads can be combined to find the stress if necessary. Most common loads that are combined are for example the snow load and wind load. Earthquake loads do not occur frequently in Finland and therefore design against this type of loads is not highly emphasized in the analysis.

4 SIMPLE DEMONSTRATIONS OF FEM ANALYSIS PROCEDURE

4.1 Introduction

So far, most of the information about FEM analysis has been purely theoretical. It is important to demonstrate how the analysis works using simple physical problems. These demonstrations also serve the purpose of increasing the confidence level for the use of FEM software. These problems can be solved using statics and strength of materials hand calculations and that is why they are good for comparison with Creo-Simulate.

First, the physical problems are calculated based on statics principles followed by simple strength of materials calculations. The results are later compared with the FEM model results. Convergence analysis is also demonstrated. The same techniques of analysis are used in the structural analysis of the residential building.

The two physical problems include:

– Cantilever beam
– Simple truss

The cantilever beam is important as it demonstrate how solid modelling techniques are used. The simple truss demonstrates the beam and frames style of modelling and also introduces the small FEM software discussed in chapter 2.

4.2 Cantilever beam

4.2.1 Simple hand calculations

The physical problem is a cantilever beam structural steel (S355) with a point force at one end of the beam. The main aim is to find the maximum stresses and deflection of the beam. Figure 20 shows the physical problem and cross section.

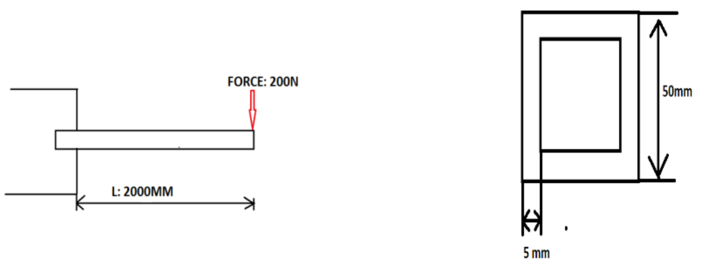


Figure 20 Cantilever beam details

Table 2 Properties of the beam and the forces applied

PROPERTIES	VALUE (UNITS)
Young´s Modulus (E)	210,000MPa
Beam length (L)	2000mm
Centroid of beam (y)	25mm
Second moment of area (I)	307500mm^4
Force (N)	200N
Area (mm^2)	900mm^2
Maximum bending moment (M max)	400KN.mm

The following formulas are used to calculate the maximum stress, deflection distance, allowable deflection and allowed stress values respectively.

$$\sigma_{max} = \frac{M_{max} \cdot y}{I} \qquad (8)$$

$$\delta_{max} = \frac{F \cdot L^3}{3 \cdot E \cdot I} \qquad (9)$$

$$D_{allowable} := \frac{L}{360} \qquad (10)$$

$$Sf := \frac{\delta_y}{2} \qquad (11)$$

The results are shown in table 3 below. The detailed calculations are found in appendix 1.

Table 3 Results of the simple cantilever analysis

PROPERTIES	VALUE (UNITS)
Maximum bending stress (σ max)	32.52MPa
Maximum deflection (mm)	8.259mm
Allowable deflection (mm)	5.556mm
Allowed Stress (MPa)	177.5MPa

4.2.2 FEM analysis results

The cantilever is first modelled in Creo-parametric as a 3-Dimensional model as shown in figure 21.

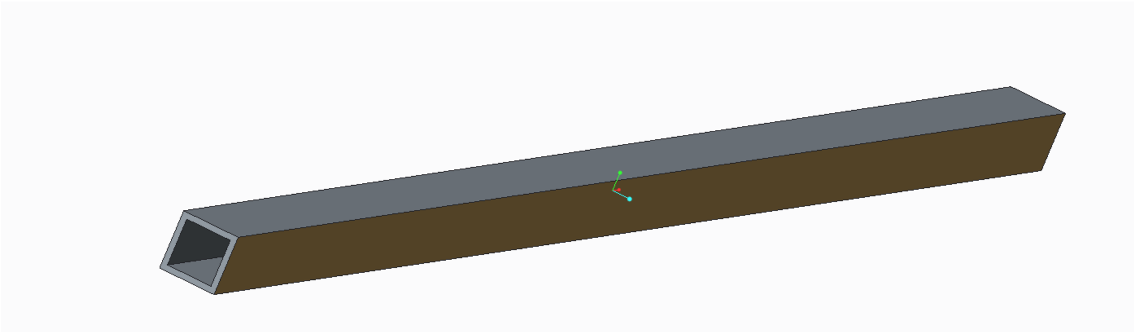


Figure 21 3-D model of cantilever beam

The beam is transferred to Creo-simulate which is the FEM part of the Creo software. As demonstrated, one of the advantages of using Creo is, it is easy to transfer the CAD model to the FEM model. The next step is setting up the parameters. The constraints, boundary conditions (material and displacements) and forces are incorporated to the problem. This part is very important and is the main source of errors for most problems. The result is as shown in the figure 22 below.

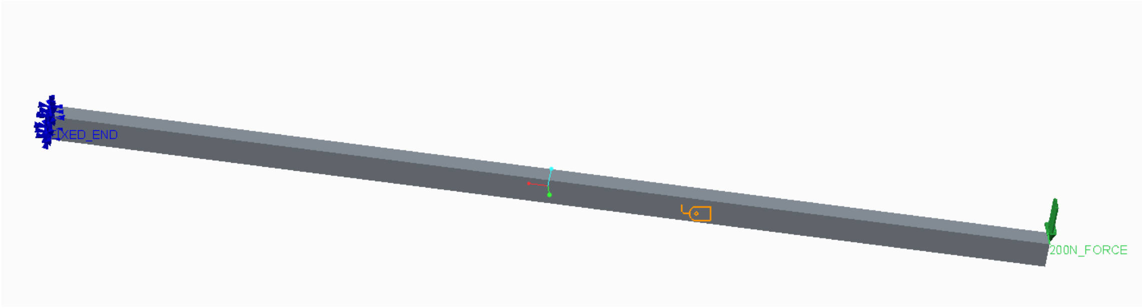


Figure 22 Model in FEM before analysis

The simulation process starts soon after the parameters are double checked. Running the problem means the software is actually solving the numerous differential equations and the solution to these equations is displayed graphically as shown below in figures 23 and 24.

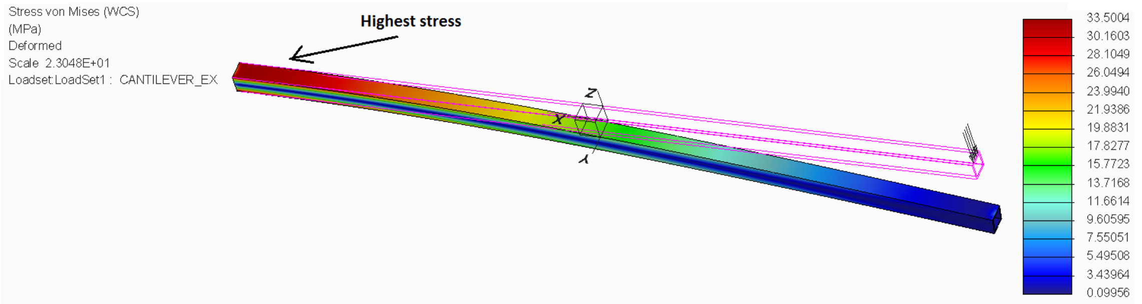


Figure 23 Stress results

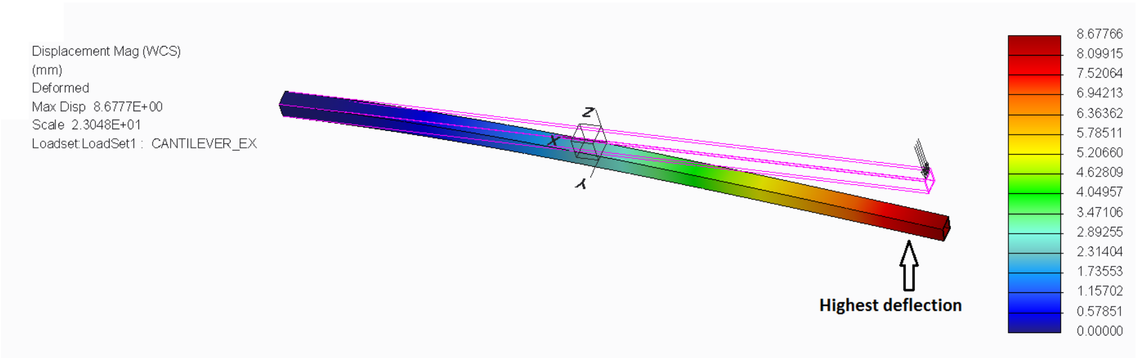


Figure 24 Displacement solution

The hand calculations results and FEM results are compared in the table 4 below.

Table 4 FEM and hand calculated results comparison

PROPERTIES	CALCULATED RESULTS	FEM RESULTS
Maximum bending stress (δ max)	32.52MPa	33.5004MPa
Maximum deflection (mm)	8.259mm	8.678mm
Allowable deflection (mm)	5.556mm	5.556mm
Allowed Stress (MPa)	177.5MPa	177.5MPa

The results from the analysis show the highest stresses will be at a value of 33.5004MPa. The result from FEM is very close to the theoretical value. The difference does not have any effect on the allowable stress so the results are acceptable.

Same case applies to the deflection values. They have very small differences. The conclusion however is the same. The deflection has exceeded the allowed deflection.

This demonstration shows how reliable is the FEM analysis if the sources of errors are minimized. Moreover, the FEM software has some added advantages over the hand calculations. One, it displays the stress and displacement results for all the sections of the cantilever. Also if the dimensions of the cross section need to be changed, then it is much easier and faster to do this in the FEM software.

4.3 Simple Truss Analysis

Truss strength analysis is a very crucial part of the analysis of the warehouse building. It is therefore necessary to demonstrate the truss analysis using the beam idealization method. The following demonstration helps to understand the basic concepts of truss analysis.

4.3.1 Results from a specific FEM program

The smaller FEM program is dedicated to solve 2-dimensional truss problems. The nodal point, cross section properties and length of the elements are typed in the software and the results are displayed. The details are as shown in appendix 2.

Figure 25 below shows a simple truss structure with five elements and four nodes.

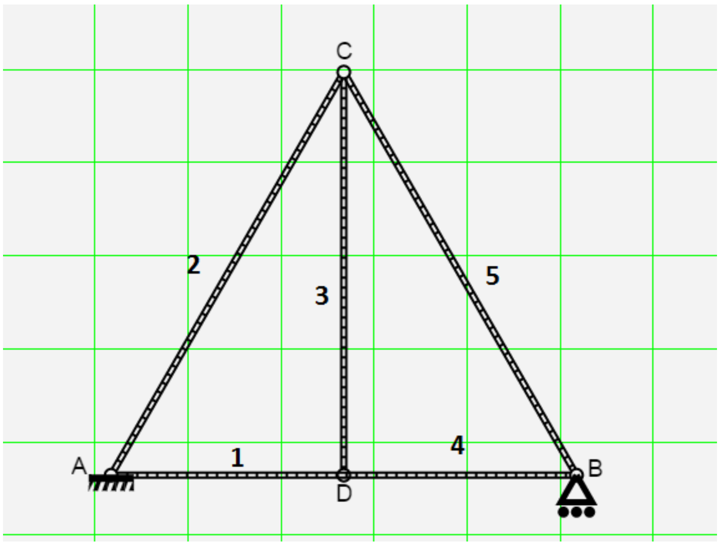


Figure 25 Simple truss

Figure 26 below shows the reaction forces when the load is applied at node C.

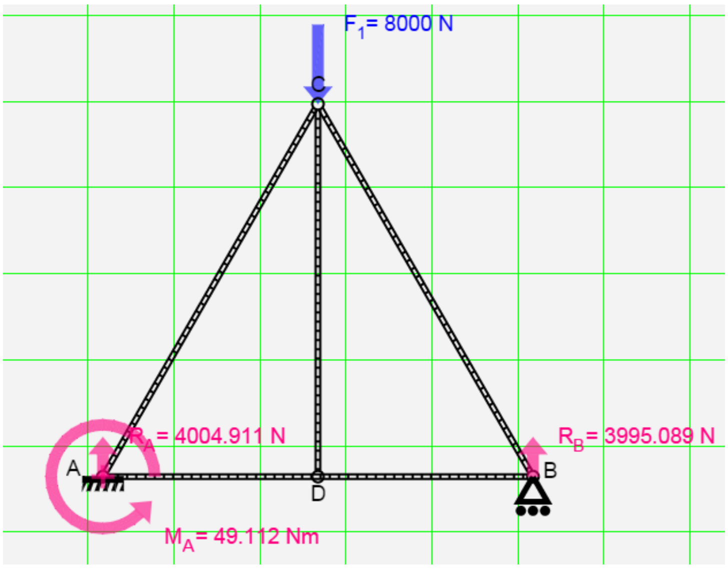


Figure 26 Results after force is applied

In the small FEM analysis software the stresses for each element are calculated and are shown in the table 5. The material used for the truss members is structural steel with yield strength of 355Mpa. The theoretical safety factor for this example is 300MPa which is just below the yield strength.

Table 5 Stress values for the truss elements

ELEMENTS NUMERS	VALUE (UNITS)
Element 1	4.619MPa
Element 2	-9.238MPa
Element 3	0
Element 4	4.619MPa
Element 5	-9.238MPa

4.3.2 Creo- Simulate Analysis

The truss is idealized as beam members and the 3-dimensional members are not modelled in the beginning. Beams represent the 3-dimensional members with similar cross section properties, material and length. Each beam is treated as an element as shown in the figure 27 below.

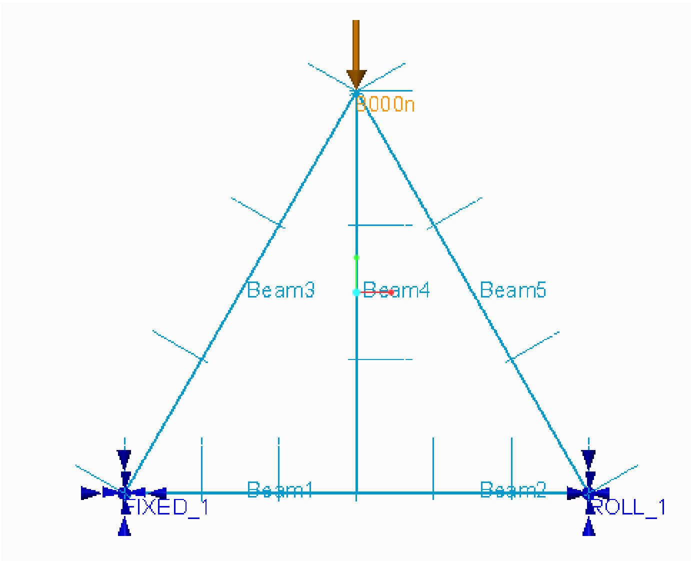


Figure 27 Idealized truss beams

After running the problem, the results are displayed as shown below in figure 28. The high stresses are on beam 3 and beam 5. This is similar to the results from the smaller FEM program.

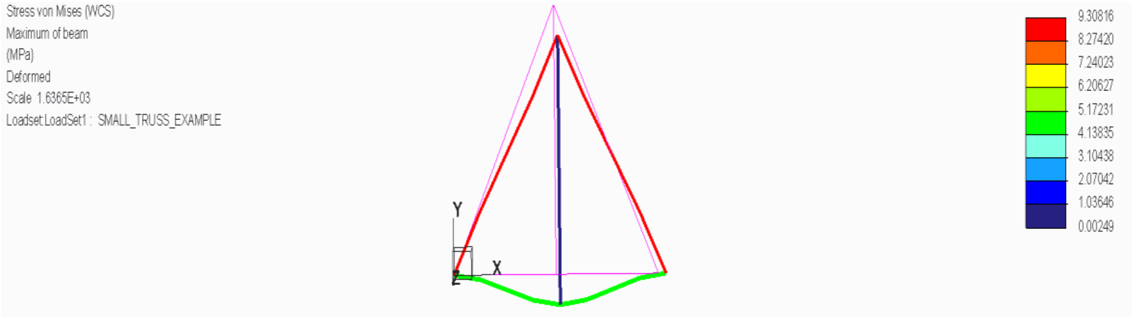


Figure 28 Simulated model

Table 6 Comparison of the values

ELEMENTS NUMERS	VALUE (MPa)	VALUE (MPa)
Element 1	4.619	4.655
Element 2	-9.238	-9.30816
Element 3	0	0
Element 4	4.619	4.65533
Element 5	-9.238	-9.30816

The values are almost identical which illustrates the authenticity of the FEM programs.

4.4 Conclusion

FEM software like Creo-simulate is a very useful tool in structural analysis. This occurs if the errors are avoided or kept at a minimum. The minor differences in the results are sometimes acceptable especially when the allowable stresses are avoided. The demonstrations done above are a huge confidence booster for the main analysis of the residential building.

5 ANALYSIS OF WAREHOUSE BUILDING

5.1 Introduction

The analysis is focused on the “skeleton” part of the warehouse building. This includes the truss column assembly, beams connecting truss, bracings and joints. The technical drawings of the buildings are used to model the “skeleton” in CAD software as shown in figure 29 below.

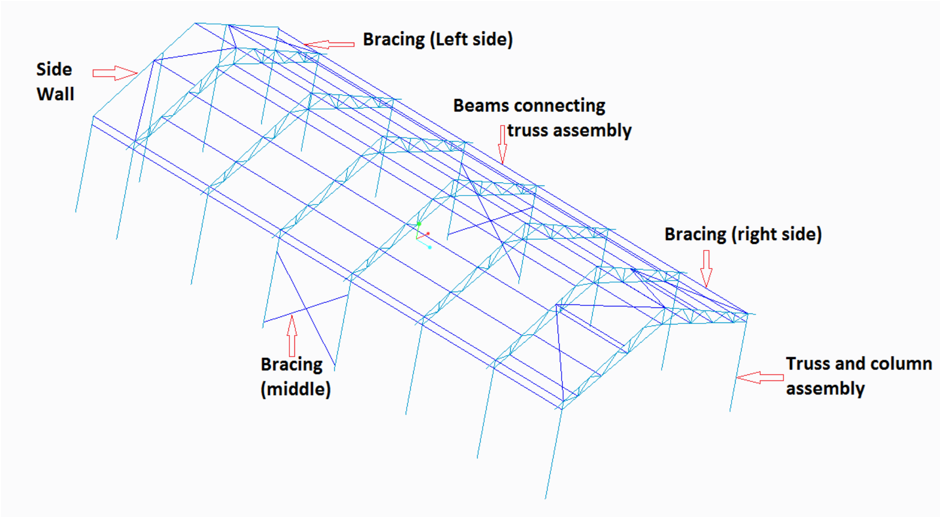


Figure 29 “Skeleton” model of the warehouse building

The analysis process starts by modelling the structure in a Computer Assisted Design (CAD) software. This marks the start of the modelling process. The analysis starts by simplifying the structure and assuming the residential building is in a two dimensional view. Three dimensional analysis of the structure then follows.

The elements are joined by nodes and together all the elements appear like the real drawing. Other parts to be analysed like the column and joints are modelled together and when required separated. Each element has clearly defined geometric properties such as the length and cross section properties. Each line elements will only have two nodes. The interconnected elements then represent the structure.

After sketching of the structure, constraints on the nodes are defined. The node can be free in all degrees or can be totally fixed. Some nodes are fixed in one axis while free in other axis. This depends on how the real structure is set up and the expected results. The loading is then defined on the model. The magnitude and direction of the loading should be clearly stated.

After all the simulation parameters are set up, the model is run. A convergence analysis estimates the errors in the simulation. In Creo - parametric, three options provide different convergence methods. They include: quick check, single pass adaptive and multi-check adaptive. More accurate results are achieved by using multi-check adaptive convergence method.

If there are any errors after simulation, it is possible to go back to the simulation parameters and edit the information. The results can be displayed in many forms if necessary. They can be viewed graphically, deformed view or animated. The results must be critically reviewed and are not blindly accepted. The shape of the deflected model and the animation help to check if the correct parameters were used.

5.2 Column Analysis

The column (HEA-180) is very important as it links the roof truss and the foundation as shown in figure 29. It mainly has two sources of stress, wind load and snow load. The snow load results to buckling of the column and the wind load causes the column to act like a cantilever beam with a distributed load.

European standard EN 1991-1-3 is used to find the snow load. The snow load that is distributed on the roof is determined using equation 5 and is calculated as shown below.

$$S := \mu_i \cdot C_e \cdot C_t \cdot S_k = 1.92 \times 10^3 \, Pa \qquad (12)$$

The uniform distributed load on the roof is calculated using equation 6 and is presented below.

$$UDL := L \cdot P_{force} = 9.21 \times 10^3 \, N/m \qquad (13)$$

Where L is the average length (4.797m) of the building in meters and P (force) is the pressure force from equation 12.

5.2.1 Buckling analysis using theory

The first step is to identify the point load at the column using the steps shown in table 1. The figure below illustrates the free body diagram of the top frame that is directly acted upon by the UDL. The main aim is to identify the reaction at (A) where the column is pinned.

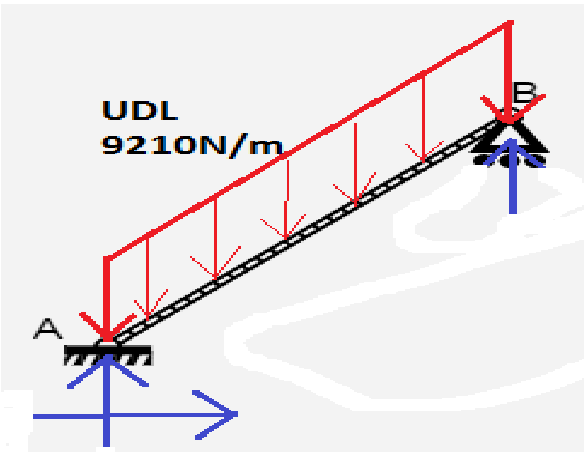
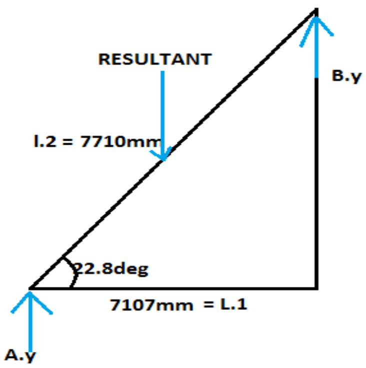


Figure 30 Free body diagram of top beam

The problem is static and the equations of equilibrium are once again utilised. Using the equations of equilibrium the point force at A which will act as the axial force on the column is calculated. The table 7 below shows how the force is evaluated.

Table 7 Force at top column

Description of procedure	Diagrams and calculations
Schematic presentation (FBD)	
Resultant force	$UDL := 9.21\times 10^{3}\frac{N}{m}$ $Resultant := L_2 \cdot UDL = 7.101\times 10^{4} N$
Equilibrium formulae and statics calculations	$\Sigma f(x) = 0 \quad \Sigma f(y) = 0 \quad \Sigma M = 0$ $B_y \cdot L_1 - Resultant \frac{L_1}{2} = 0 \text{ solve}, B_y \rightarrow 35504.55N$ $B_y := 35504.55 \cdot N$ $A_y := Resultant - B_y = 3.55\times 10^{4} N$

The applied axial force (P applied) on the column is 35500N acting axially on the column. The next step is to perform a buckling analysis on the column to find out if the column is stiff enough to avoid high buckling levels.

Euler´s buckling formula is used to find the theoretical buckling load.

$$P_{cr} := \frac{\pi^2 \cdot E \cdot I}{(2 \cdot L)^2} \qquad (14)$$

$E := 200000MPa$

$$I := 9250000\,mm^4$$

$$L := 5838\,mm$$

$$P_{cr} := \frac{\pi^2 \cdot E \cdot I}{(2 \cdot L)^2} = 1.339 \times 10^5\,N$$

According to the theory above, the column will buckle if the load exceeds 133900N.

5.2.2 Buckling analysis using FEM

The FEM analysis software is used to analyse the column to find out if the same critical buckling load will occur and to visualize the simulation. The analysis is modelled in Creo- Simulate as shown in figure 31 and 32 below.

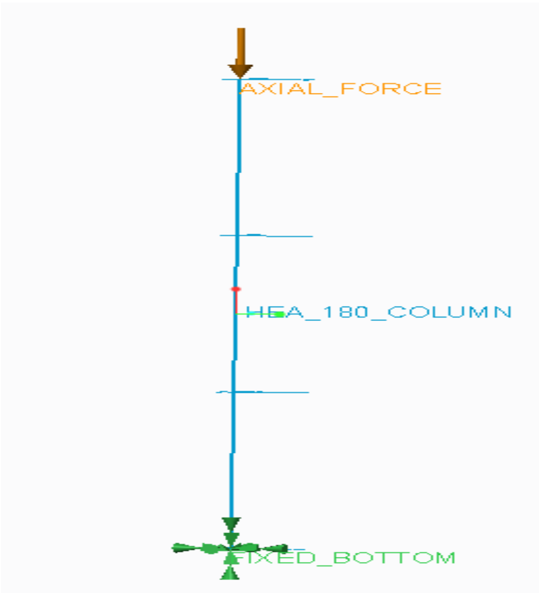


Figure 31 Idealized model showing axial force

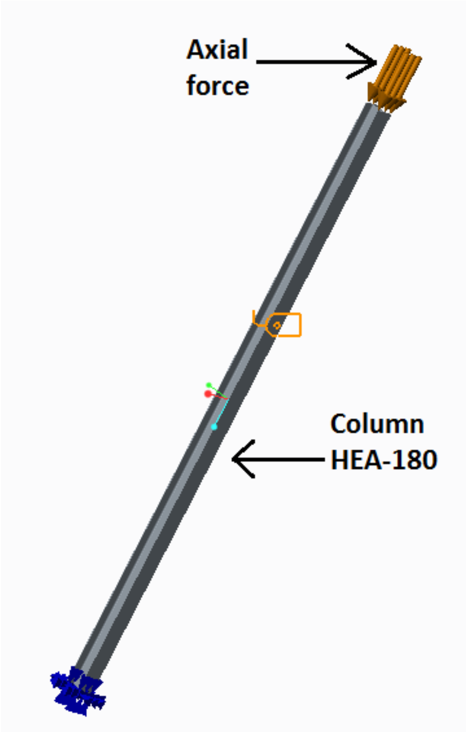


Figure 32 3 dimensional model

When inserting the parameters, a force of 1N is used as the axial force so as to anticipate a buckling load factor (B.L.F) of 133,392. The following figure shows the results after simulation.

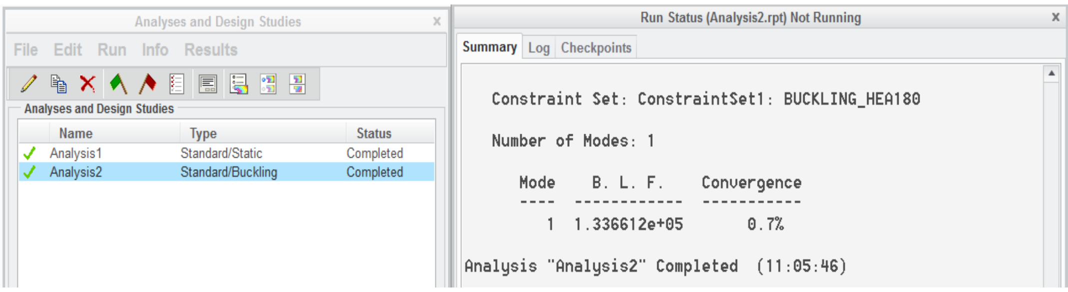


Figure 33 Analysis results

The B.L.F is 133,661 which are almost similar to the theoretical results. Therefore, the critical load (P.cr) that will cause the column to buckle is 1N*B.L.F.

From the FEM analysis and theoretical calculations, the buckling load is estimated to be 133,900N. A safety value is applied to the critical load and the allowed axial load is as shown below.

$n = \text{Safetyfactor}$

$$P_{allowed} := \frac{P_{cr}}{2} = 6.697 \times 10^{4} \, N$$

$P_{allowed} < P_{cr}$

The applied load at the top of the column is 35,500N which is less than the allowed load of 66,970N. Therefore, it is concluded that the load on the column caused by the snow load will not cause the column to buckle.

5.2.3 Wind Load on Column

First, the wind load has to be defined according to the standards. The wind pressure on an external surface is calculated using the following formula. (EN – 1991 – 1 -4, 43)

$$W_e = q_p(Z_e) \cdot C_{pe} \qquad (15)$$

Where $q_p(z_e)$ is the peak velocity pressure and C_{pe} is the pressure co-efficient for external surface.

Peak velocity pressure = 600 N/m^2

Pressure co-effecient = 1.4

We = 840 N/m^2

Finally, the UDL (N/m) on the column is solved using equation 6 as follows.

Pressure load = 840 N/m^2

Length = 4.797m

$$UDL := pressureload \cdot Length = 4.029 \times 10^3$$

The problem is modelled in Creo – Simulate as shown in figure 34 below.

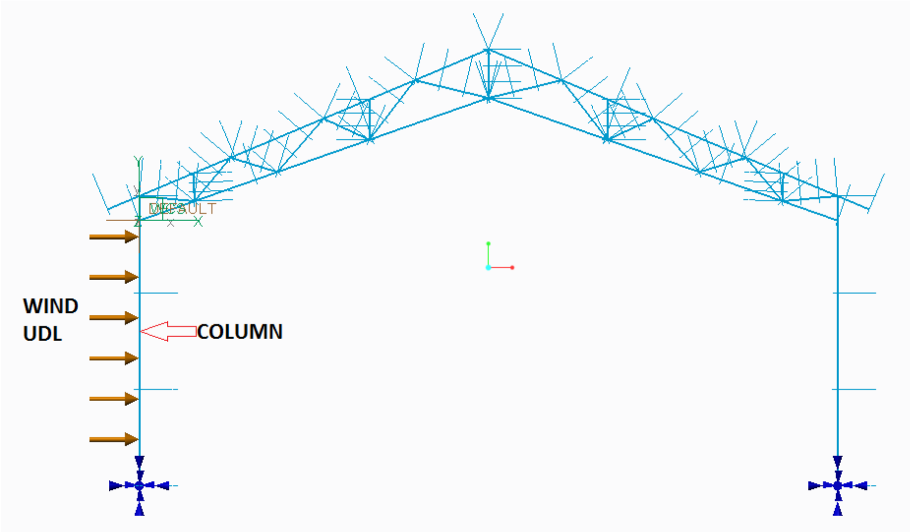


Figure 34 Wind UDL on the building

Solution to the problem is simulated and the maximum Von-Mises stress occurs at the base of the column at a value of 18MPa. This is a considerably low stress given that it is nowhere near the yield stress.

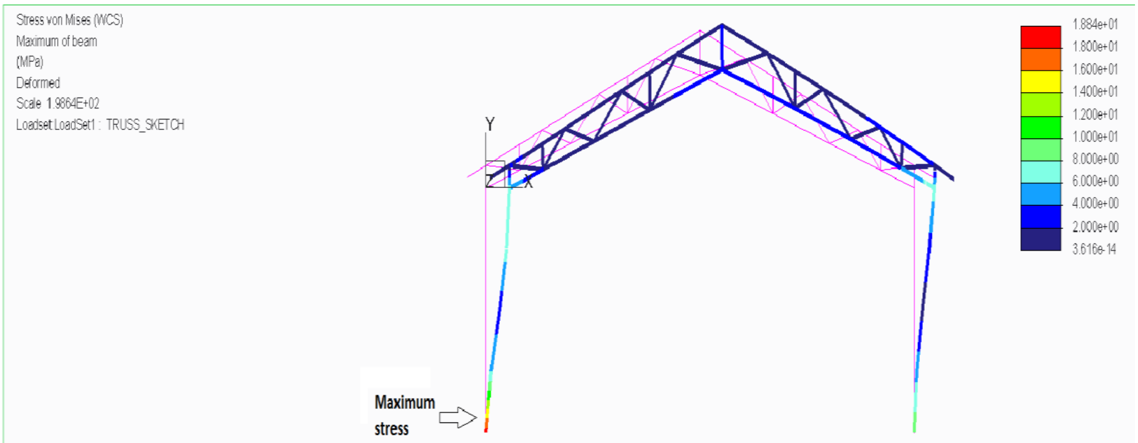


Figure 35 Simulated model

The next step is to isolate the column in order to perform even a more detailed analysis. 3 dimensional model of the column part is simulated and the simulated parameters (load, fixed positions) are as shown in the figure 36 below.

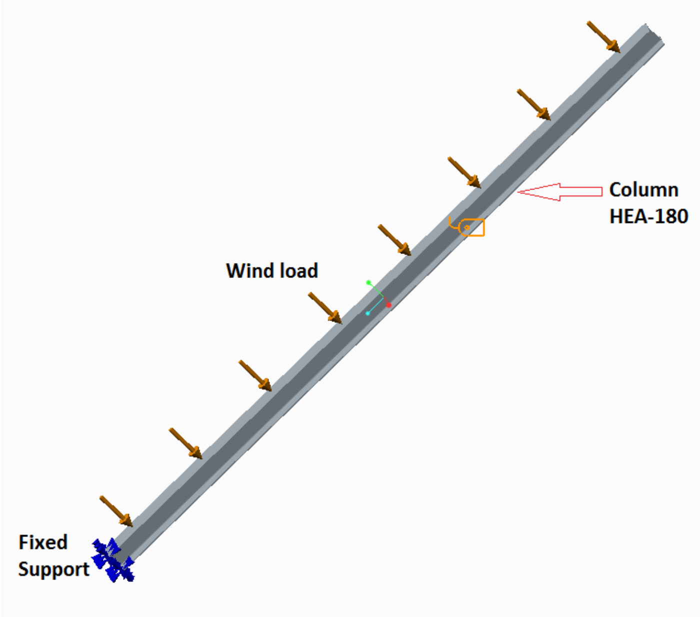


Figure 36 Column showing simulation parameters

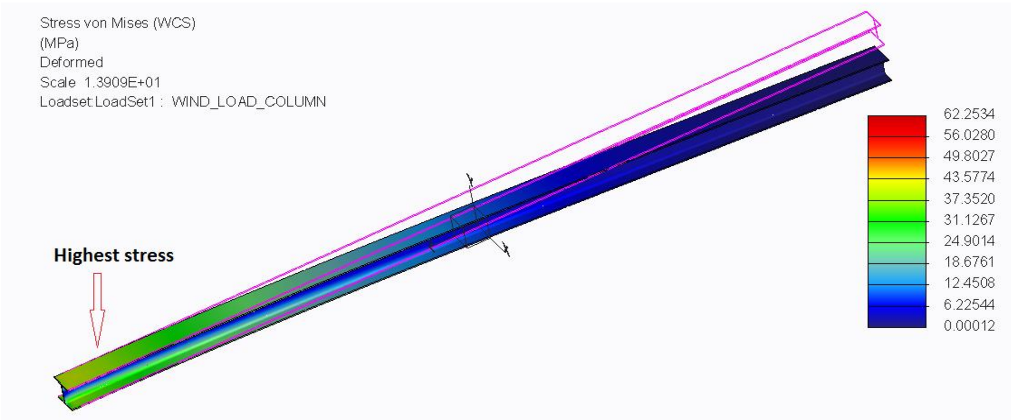


Figure 37 Simulated model

The high stresses occur at the base of the column similar to the earlier model shown in figure 35. However, there is a huge difference in the values of the maximum stress. The new value of the stress is 37.52MPa compared to the earlier 18MPa. The new value is considerably low and is nowhere near to the critical level. This huge difference is due to the different technique used in the modeling. In the first case the roof truss is still connected to the column. The load of the truss and snow reduces the bending of the column when the wind load is acting on the column. This is why the model in figure 37 experiences high stress compared to the one in figure 35. The conclusion is the same since both stresses are way below the critical stresses set according to the safety factor and yield stress.

5.2.4 Column base Joint

The base of the column is supposed to be very stable to provide the rigidity required during minor vibrations. The base supports the column which in turn supports the roof truss and walls of the building. This tells that the base strength integrity should be very high.

The base joint includes anchor bolts, a plate and the concrete base support. The anchor bolts are inside the concrete and are held in position by the plate. The base is in the technical drawing is as shown in figure 38 below.

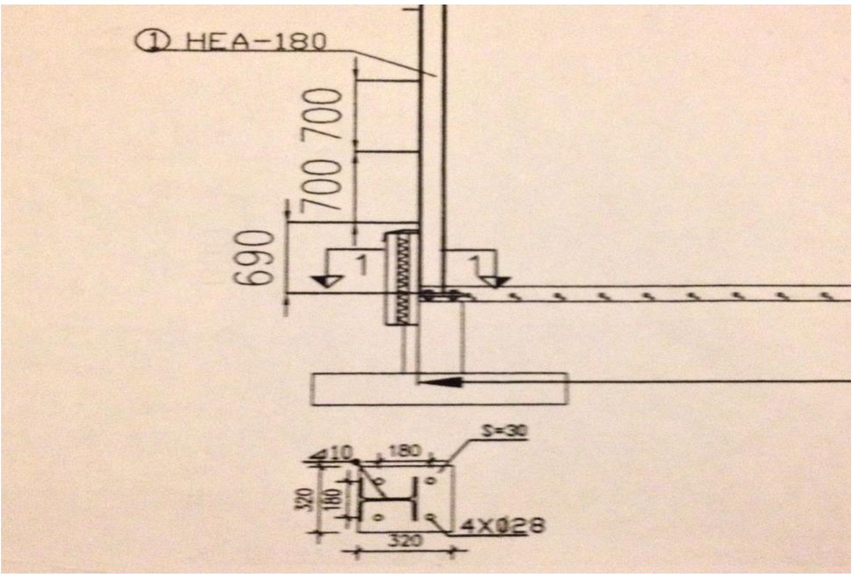


Figure 38 Column base

The bolts used for the joint are unique since their main function is to act as anchors to the column. Figure 39 below shows an example anchor bolt in use connected with concrete.

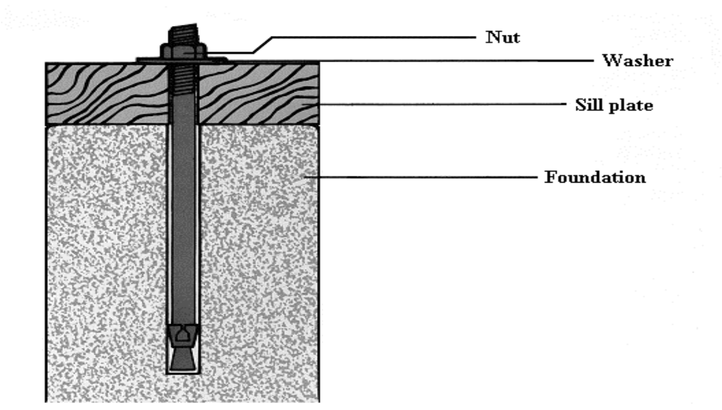


Figure 39 Anchor bolt in concrete (A word about anchor bolts, 1992)

The loads that occur on the bolt depend on the loads that are on the column. These loads include; roof snow load, wind load and column weight. These loads exert tension, compression and shear forces on the bolt. Sometimes the forces might act together for example tension and compression or the forces can act alone. Bearing stress on the bolt due to the plate is also experienced on the bolt.

The wind load on the side of the column produces a bending moment at the base. The bending stress of the bolt results to one half of the bolt experiencing tension and the other half compression. The point load at the top of the column due to the snow load and load of the column, results to compression at the base hence helping in the anchoring. This in turn reduces the bending moment of the bolts caused by the wind. This leads to the conclusion that the wind load is most critical load acting on the bolts.

Analysis of the bolts starts with the simplified model shown below. It should be noted that only the critical load which is the wind is used to analyze the bolts.

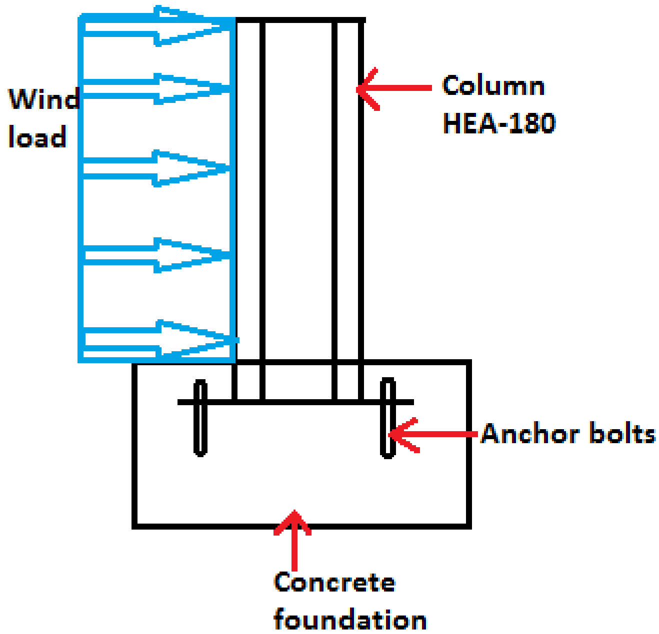


Figure 40 Simplified model of column and the base

Appendix three show the detailed calculation to find the stresses of the bolt. The axial stress of one anchor bolt is approximately 309MPa. This is a high value considering the bolt material has a yield strength of 355MPa. However, this value is significantly reduced by the column weight and point load force at the top of the column.

5.3 Roof truss analysis

As shown in figure 29, the roof truss and column assembly forms most of the “skeleton” of the structure. Different loading conditions that replicate real weather conditions are applied. The main aim is to find regions of high stresses in the truss. The model in figure 41 below is the first loading type to be modelled and simulated.

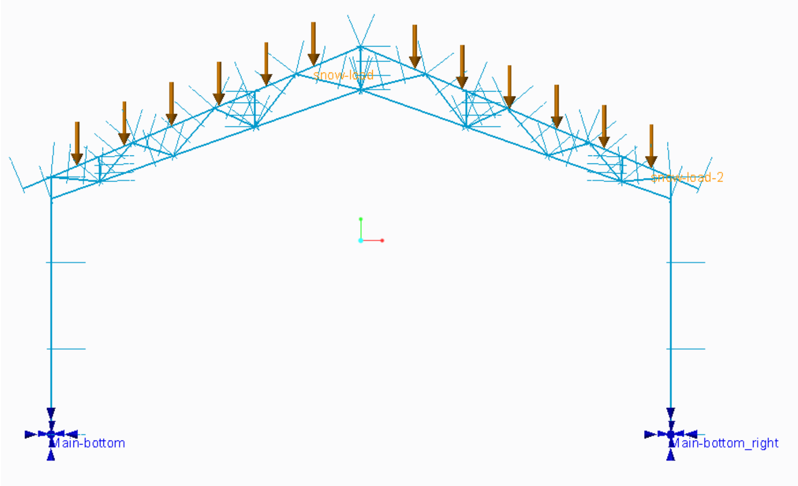


Figure 41 Snow load on the roof

The snow load on both sides the roof exerts a lot of stress on the truss. The truss members experience different stress levels depending on their locations and properties. The simulated model in figure 42 shows the truss members with the highest stress levels.

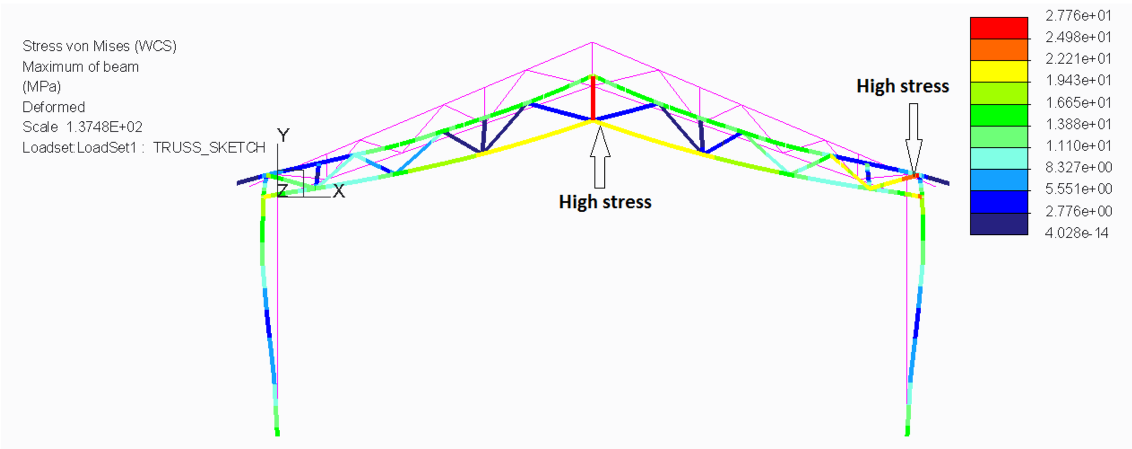
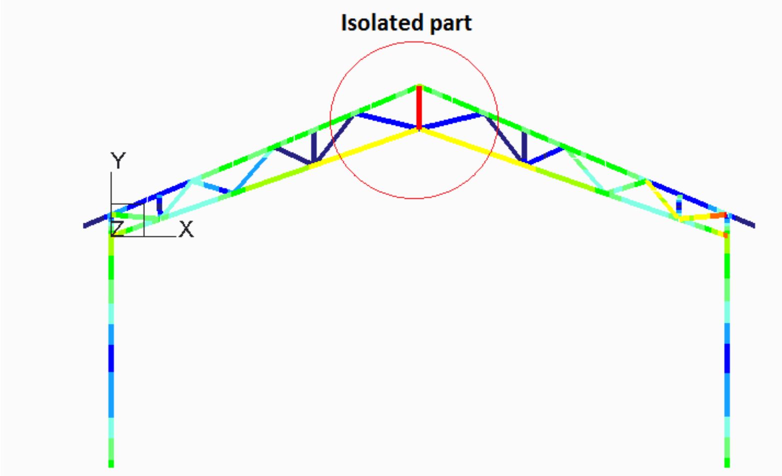
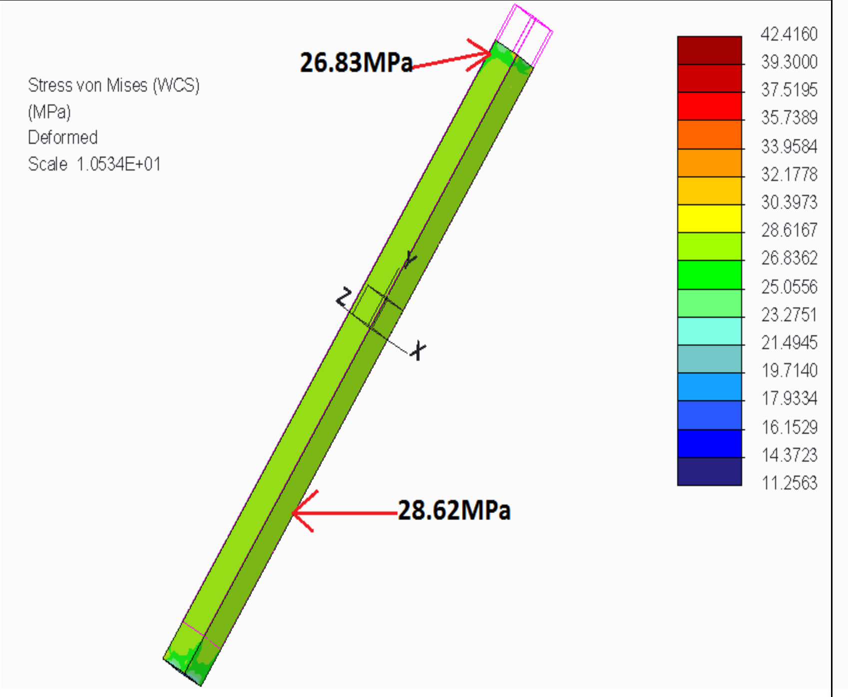


Figure 42 Results of simulated model

The results of the simulation reveal that the truss is stiff and strong enough to hold the snow load. Only one member experiences high stresses at a value of 27.76MPa. The material used for the truss members is structural steel with yield strength of 355MPa. The resulting stress is well below the allowed stress value of 177.5MPa.

Even more detailed modelling of the truss is necessary to make sure that the most accurate results are calculated. The table below shows the isolation process of regions of high stress.

Table 8 Detailed modelling process

Part description	Model
High stress region is first identified	
High stress member is isolated and analysis is done again	

The second simulation results of the model show even more specific results. For example the higher stresses are revealed to be in the middle section of the member although the average of the two stresses results to 27.73MPa. The stress levels are good enough but the deflection of the truss still needs to be investigated. Figure 43 below shows the deflected truss member.

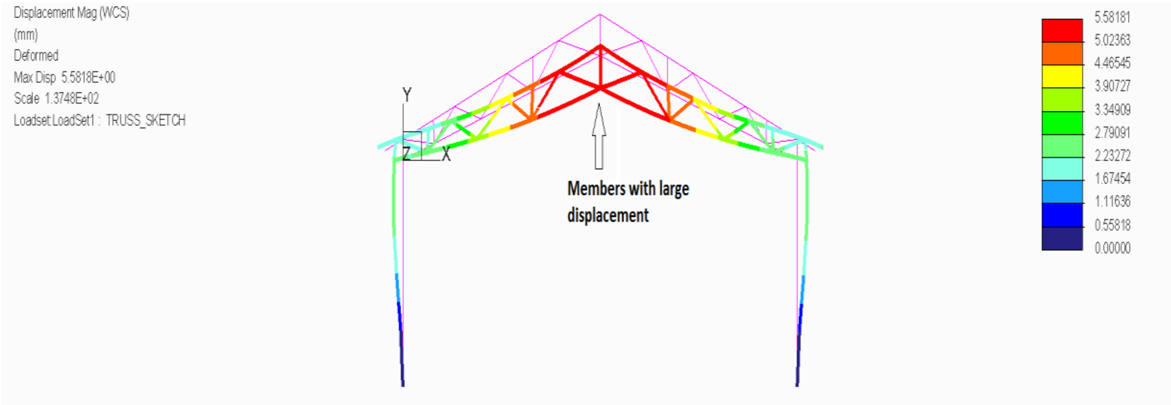


Figure 43 Truss deflection

The main parts of the truss that deflected are in the middle. The highest deflection value is 5.58181mm. Using the top member of the truss as a reference length, the allowed deflection is determined. The calculations are as below.

L = length of the top column (120*80*5)

$L := 16.800m$

$$\frac{L}{360} = 0.047m$$

The allowable deflection is nowhere close to the applied deflection of the building. Therefore, the truss is stiff enough for the loading condition above.

A different loading condition that has snow on one side of the roof is then modelled. This can happen in situations where there is a structure obstructing the snow fall on one side of the building. Figure 44 below illustrates the physical problem.

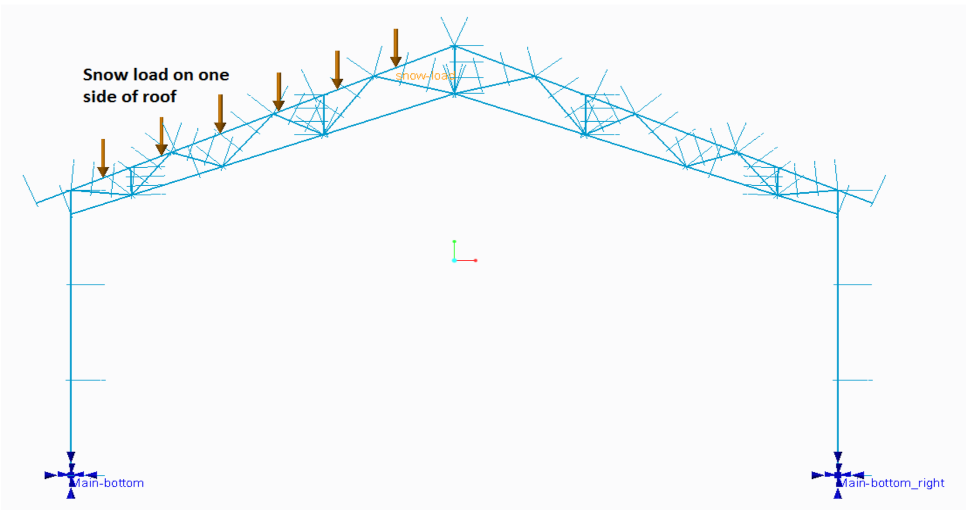


Figure 44 Model of snow load on one side of the roof

The aim of this kind of model is to find out if the truss stresses will change due to the different loading case. The simulated models showing the deflection and stress are shown below in figure 45 and 46.

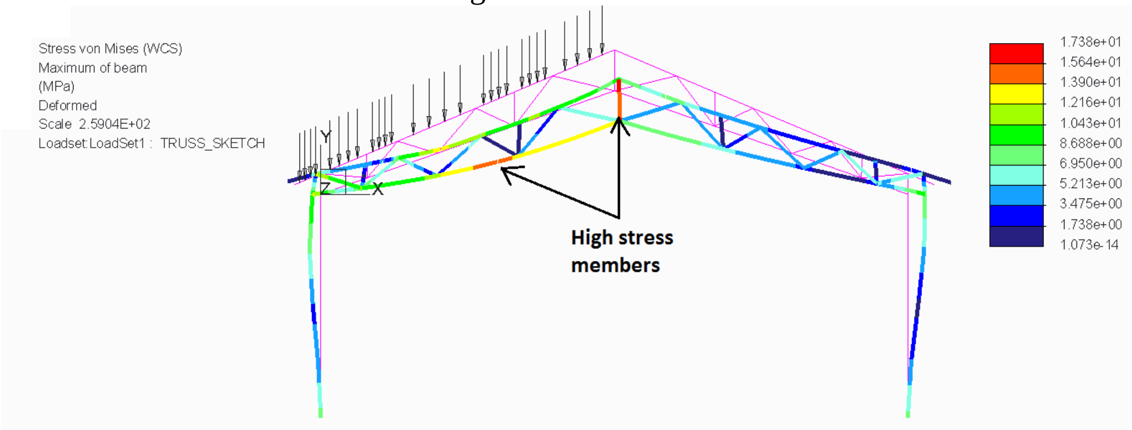


Figure 45 Stress results

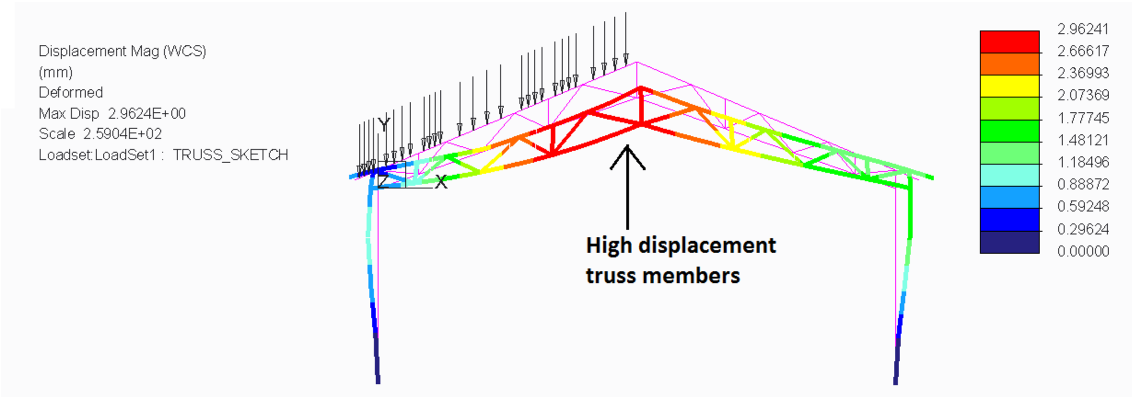


Figure 46 Total displacement results

The stress and displacements are less than those of the initial load conditions. The conclusion therefore is that the second loading condition has insignificant effects on the truss structure.

5.4 Side wall analysis

5.4.1 Snow load

The side-wall assembly as shown in figure 29 is on the left side of the building. It is the first structure to get in contact with the wind and hence is connected with the bracing to the right of the structure. Wind load is very crucial for this part as well as the snow load. The side-wall is modelled for the snow load analysis as shown below in figure 47.

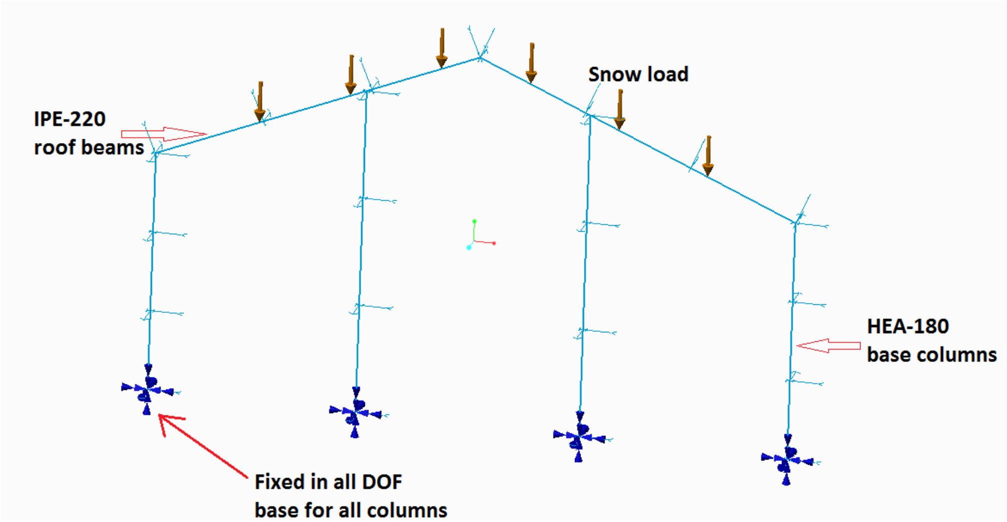


Figure 47 Snow load model

The results that are expected for this simulation are that the deflection is small enough, the stresses are low and the columns will not buckle under the snow load. The results of the simulation for all three parameters are shown and explained in the following figures 48, 49 and 50.

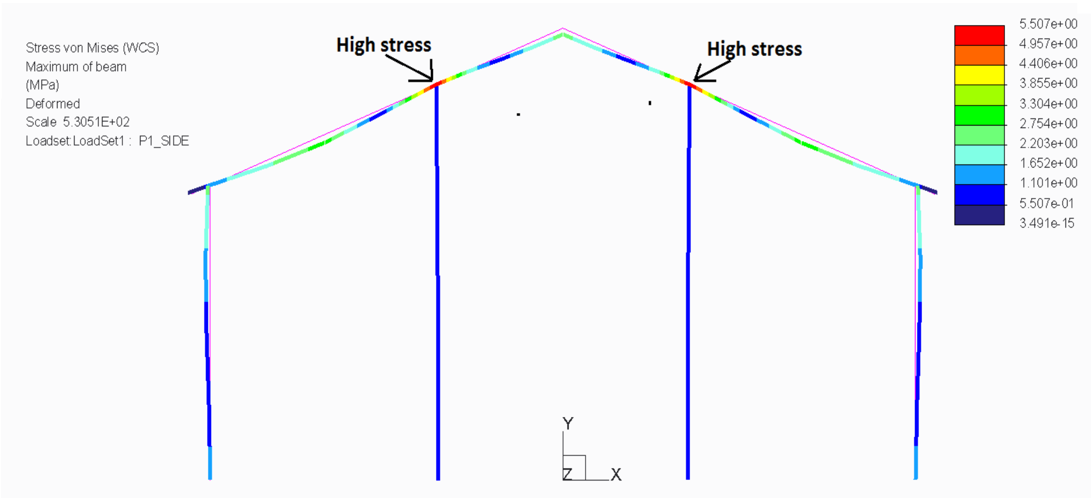


Figure 48 Stress results

The highest stress value is very low at a value of 5.507MPa. This occurs at the point where the column and roof beam join. The value is way below the unacceptable stress value therefore it is not a major concern.

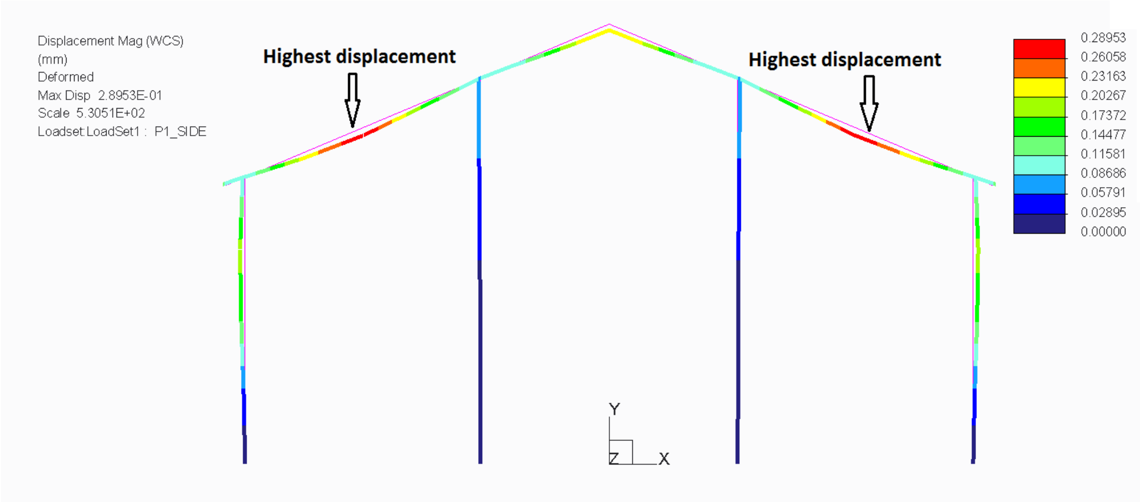


Figure 49 Simulated deflection results

The deflection is very low at a value of 0.28953mm. This is safe enough for the design since it is way below the allowable deflection value.

The next step is to find the buckling load factor. Recall that the critical load is the product of the buckling load factor and the applied load.

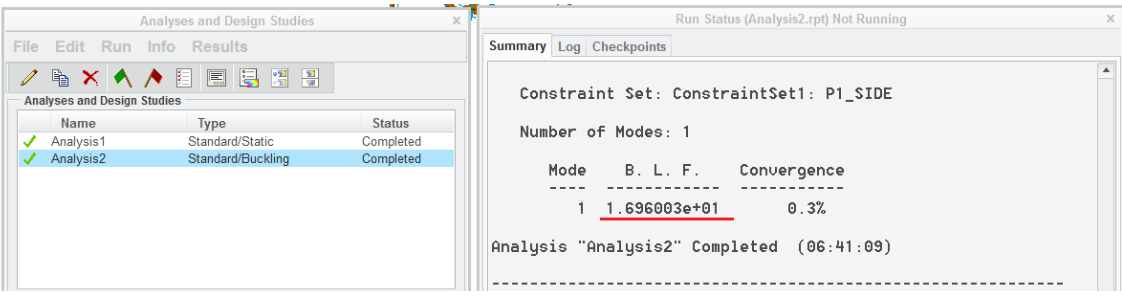


Figure 50 Buckling load factor after simulation

The buckling load factor is 16.96 which is a very high value. This value multiplied by the applied snow load results to 156.201KN which will be the critical buckling load for the structure above. The applied load is way less than the allowed load. This shows that the structure is very stiff and strong to handle the snow load.

5.5 Wind load

The model for the wind load analysis is as shown in figure 51 below.

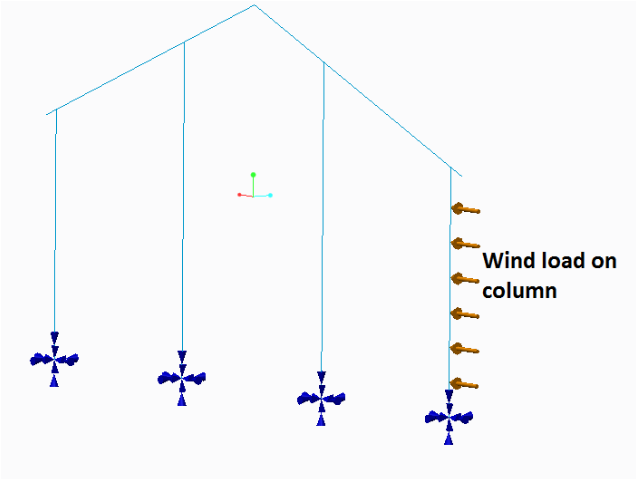


Figure 51 Model of wind load

The base of the columns is fixed in all degrees of freedom and the wind load is applied against the structure. The stress and displacement results are shown in figure 52 below.

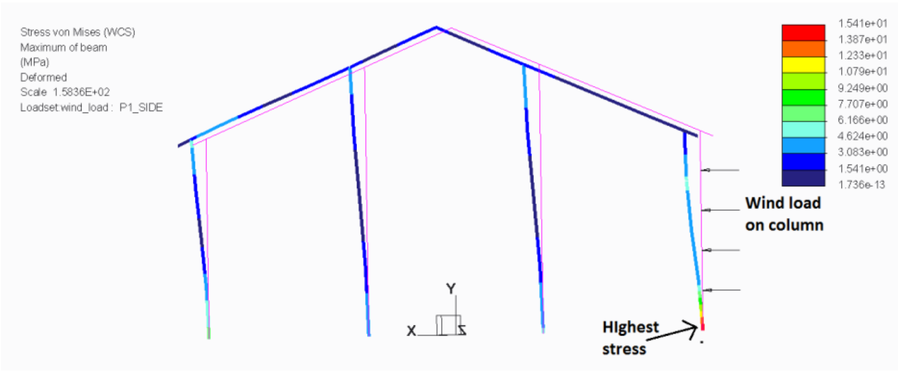


Figure 52 Stress results

The highest stress values are found to be at the base of the columns due to the effects of the wind load. The values are very low at a value of 15.41MPa.

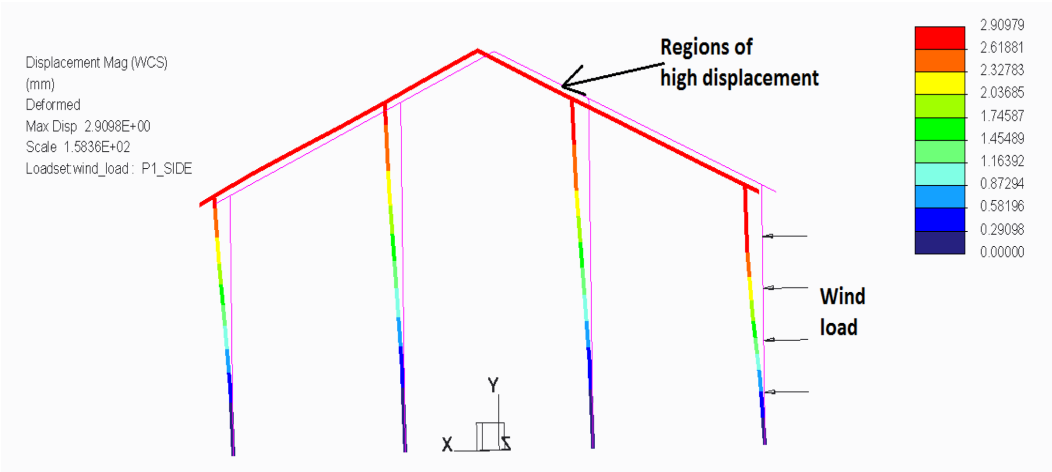


Figure 53 Displacement simulation results

The total displacement is very low at a value of 2.91mm. This is not a value to be concerned about and the structure is considered to be stiff and safe enough.

5.6 Bracing analysis

5.6.1 Wind load

The bracings are mainly designed to protect the columns and truss assemblies from the wind pressure. They are found on the sides of the building where the wind first comes into contact. The model to this problem is as shown in figure 54 below.

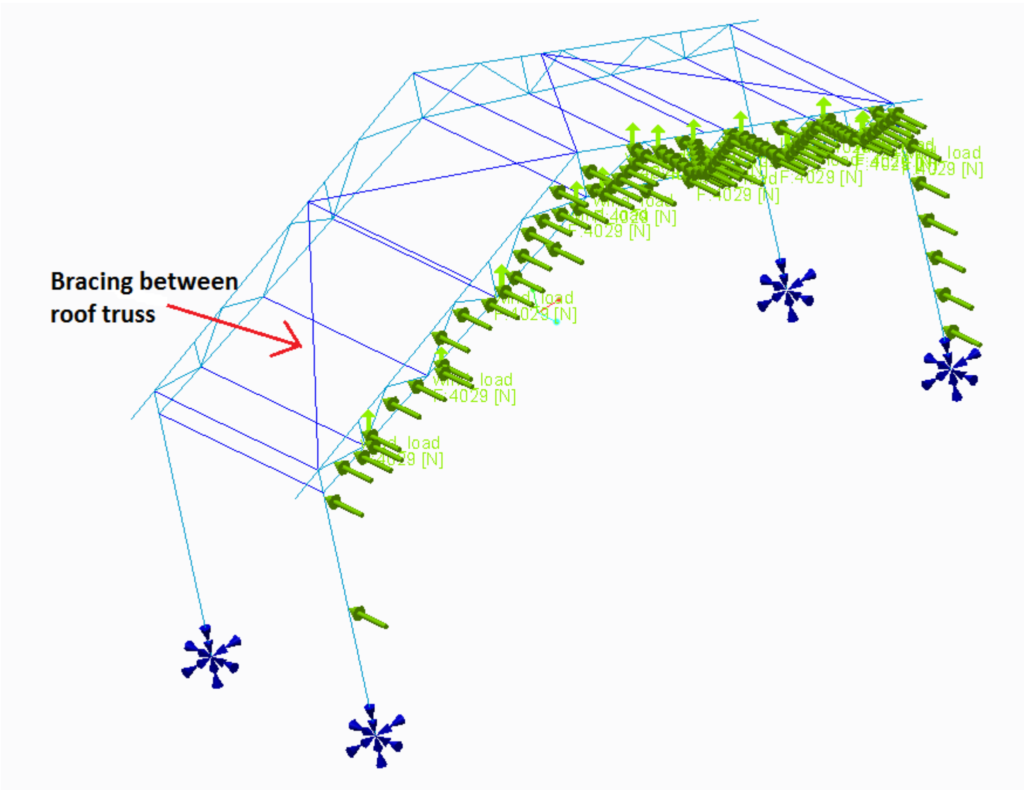


Figure 54 Model of wind load on bracing

The wind load is acting on the first truss and column assembly on the side of the building. The member in the bracing absorbs some of the force caused by the wind load. This prevents the force being transferred to other truss and column assemblies. The results of the simulation are displayed in figure 55 below.

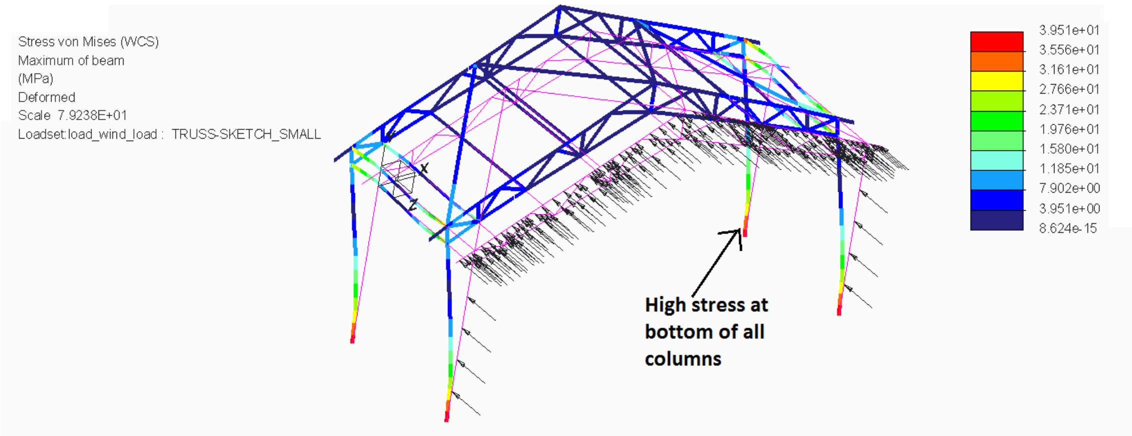


Figure 55 Simulated results

The results reveal that the base of the column has the highest Von-Mises stress values of 39.51MPa. This is an allowable load since it is less than the yield strength. The truss has very small stresses due to the bracing. This is a good example of the effect that the bracing has on strengthening the truss.

5.6.2 Snow load

The following simulation tests if the bracing is strong enough for the snow load. The members of the bracing should be stiff enough to resist the snow load.

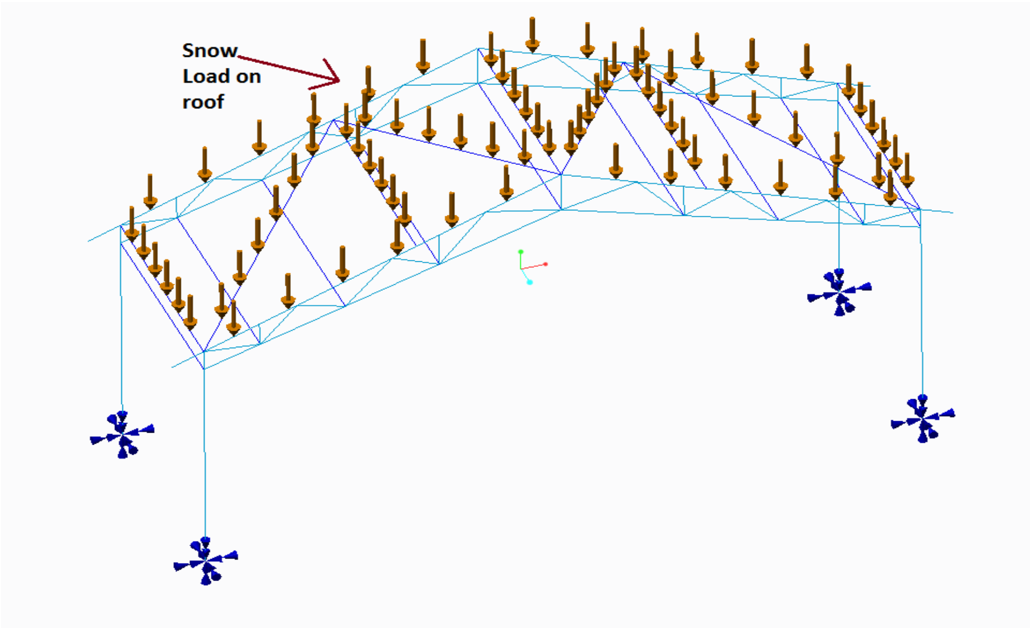


Figure 56 Model before simulation

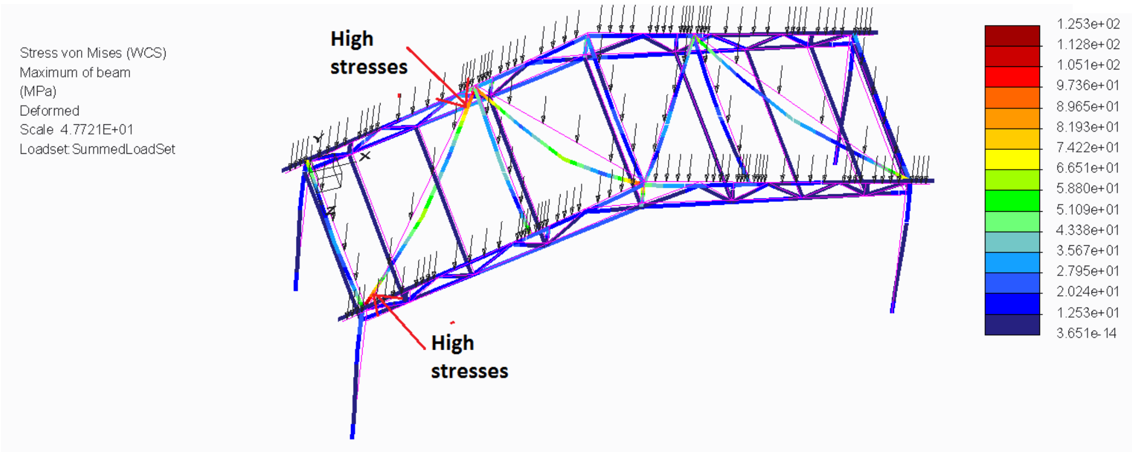


Figure 57 Model after simulation showing stress values

The highest stress occurs at the joints of the bracings as shown. The value is 125.3MPa which is less than the safety stress value. Therefore, some of the members might require modification if higher snow loading conditions are anticipated.

Snow loads might lead to deflection of the bracings. Therefore, the simulation of the displacements is necessary.

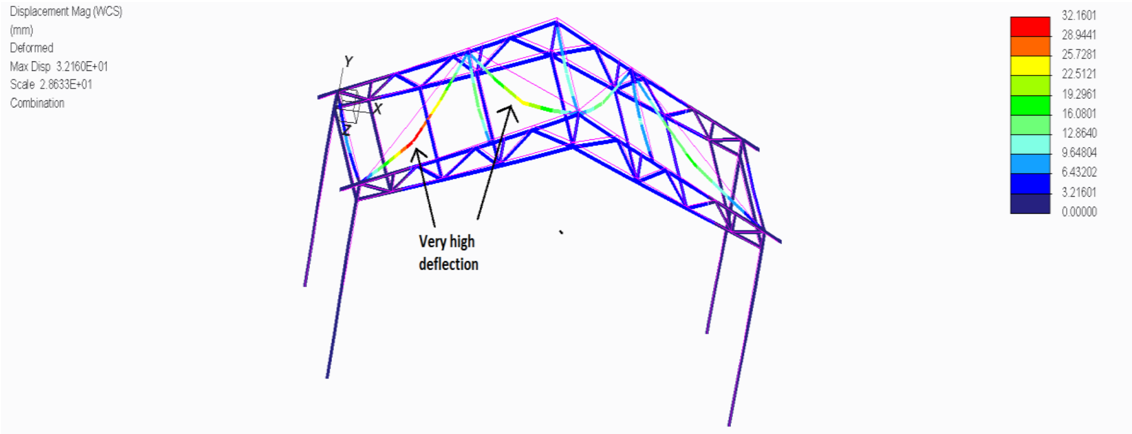


Figure 58 Simulated model showing deflection values

The same members that are under high stress in the figure 57 are the same members with high deflection values. The highest deflection value is 32.1601mm which is less than the safety value but still very close. Clearly, the parts of the bracing need to be redesigned to increase the reliability of the structure.

5.6.3 Combined snow load and wind load

In some extreme weather conditions, both the snow load and the wind load occur at the same time. This may lead to more stresses and deflections so that is why it is important to imitate the conditions. The model of the two combined loads is shown below.

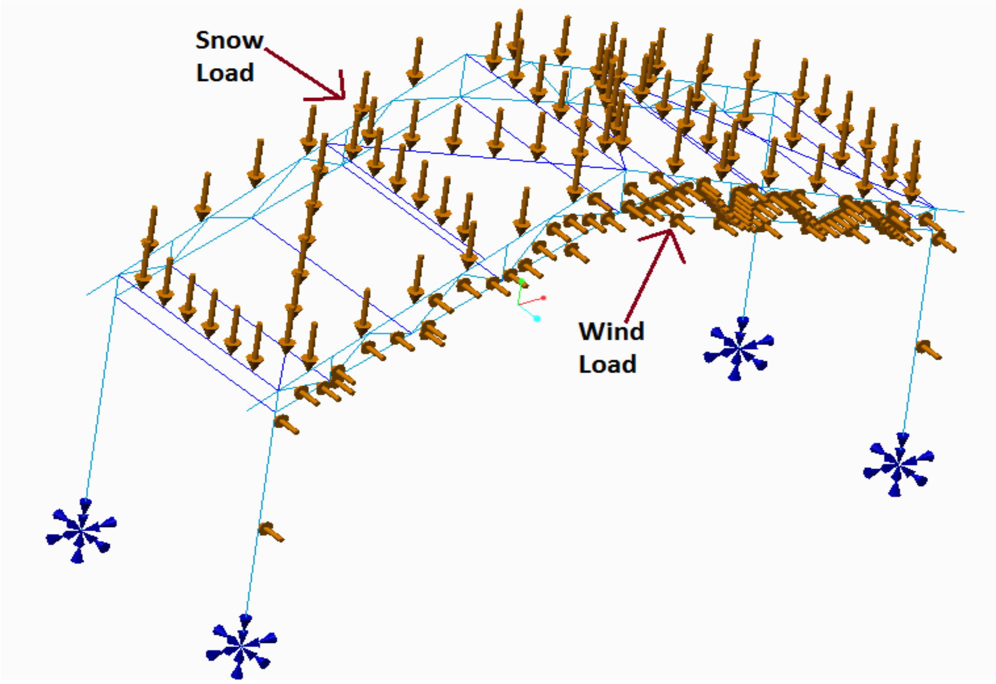


Figure 59 Model of both loads before simulation

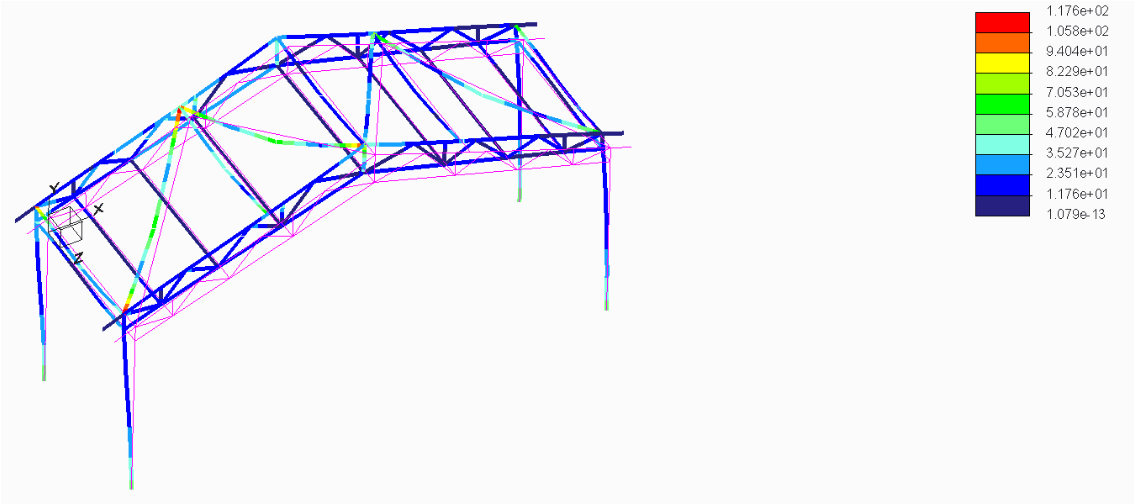


Figure 60 Model after simulation

There are no major changes to the stress levels of the model and the result values are similar to the snow load results on the bracing. The values for the displacement are shown below.

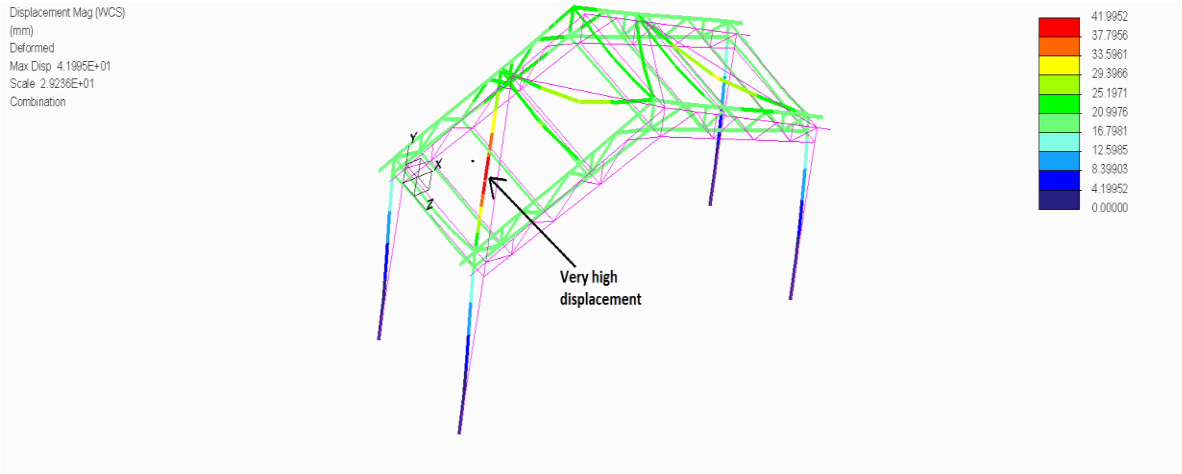


Figure 61 Model showing displacement

As shown above the model for the displacement is very different from the one with the snow load or wind load. However, the same bracing member that was under high deflection is still the same. This time around the value of the displacement is 41.99mm. This is a very high value and is not acceptable. The main aim of the bracing is to hold firm against wind forces. Since the bracing is deflecting at high values, this causes the truss members to deflect at higher values as well. The value of the truss deflection is 25mm which is different from the previous models.

The members that have high stresses and deflections are isolated and analysed to investigate even more about the stress distribution. The figure below shows the simulation results of the middle member which experienced the very high stresses.

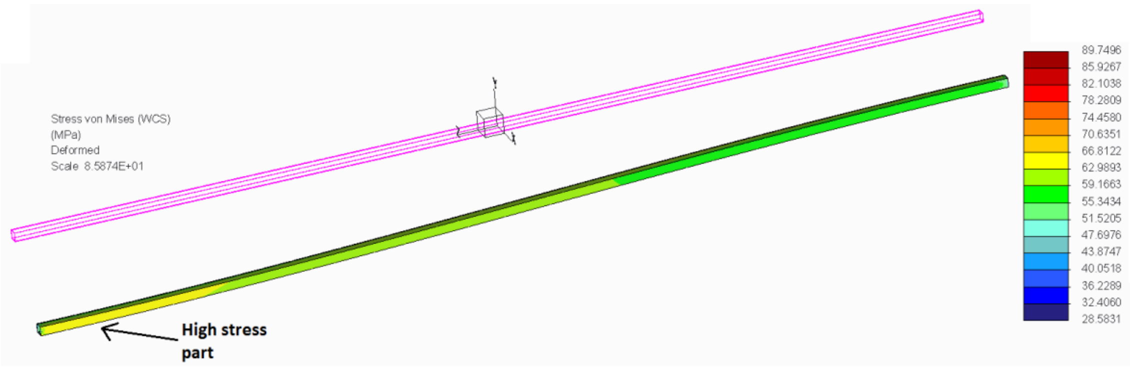


Figure 62 Isolated results of the member

The stress varies from one side of the member to another side. By using figure above, the highest stresses are 70.46MPa which is a lower value compared to the values obtained before. Smaller elements in the member show higher stresses. The left side end is zoomed in and the stress is as shown below. The highest stress is 89.75MPa on the zoomed part which shows the stress variety in the member.

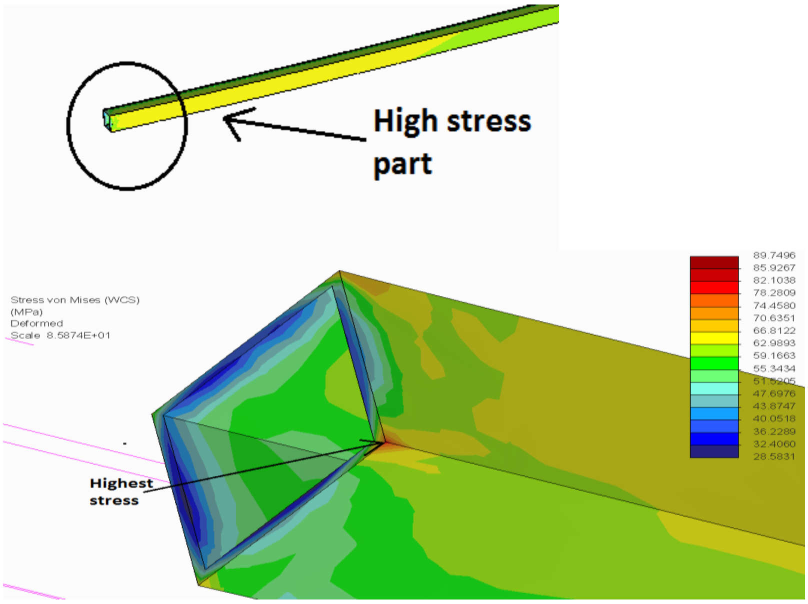


Figure 63 Detailed model

The bracing needs to be redesigned so that it is more stiff and stronger to resist any kind of combined loads.

5.7 Truss connection analysis

Part of the model shown in the bracing analysis is located on the side ends of the building. In between the building, there exists other type of connection. The beams connecting the assembly will be analysed next.

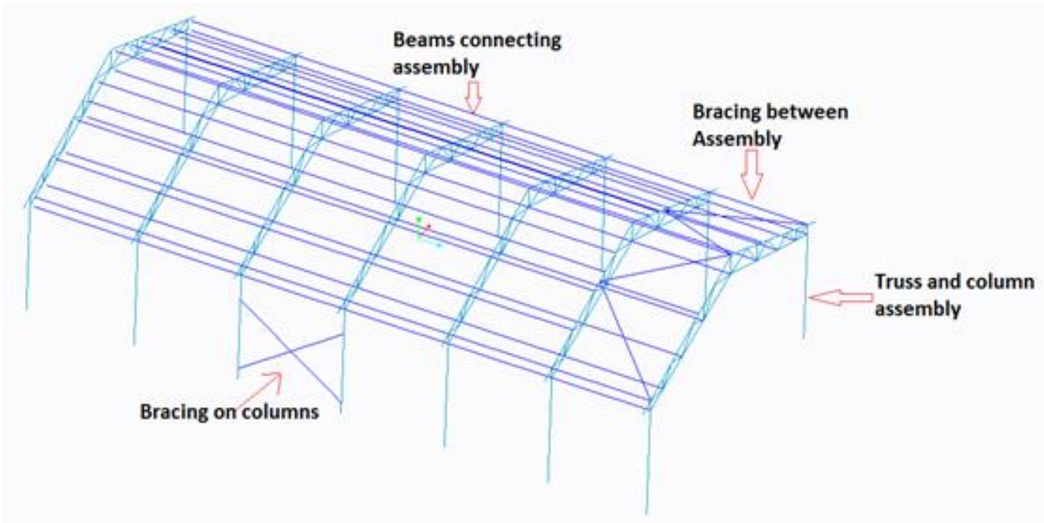


Figure 64 ”Skeleton” model

The beams connecting the assembly should be stiff and strong enough to resist the load caused by the snow load. The beams are also used to join the roof material (sheet metal or brick roof) together with the truss. The beams are expected not to deflect or have high stresses due to their important use.

5.7.1 Snow load analysis

The simulated figure is shown in figure 65 below.

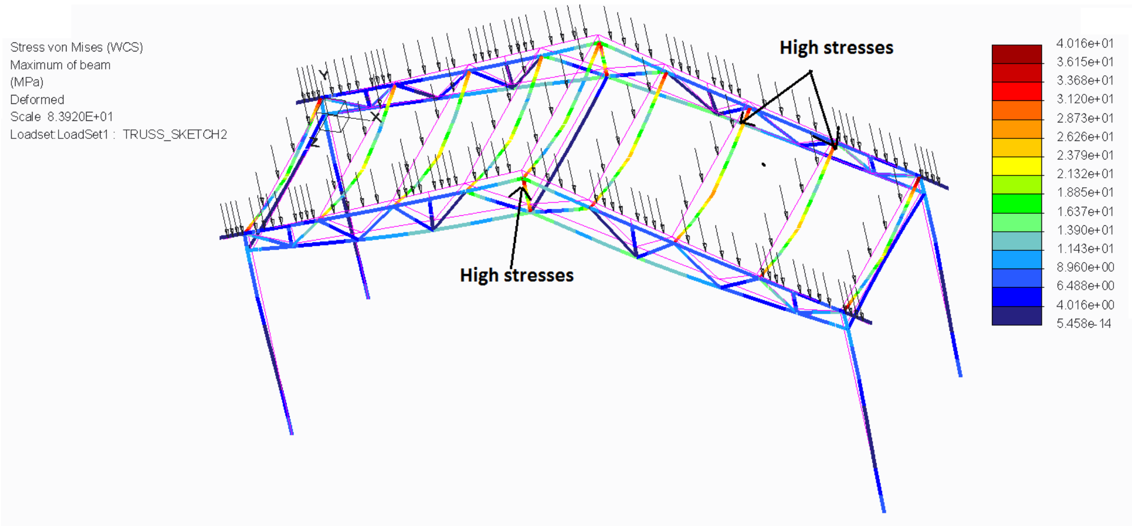


Figure 65 Model showing stress levels

High stress occurs at the end joints of the beam as shown above. The stress value is 40.16MPa which is below the allowable stress value. The results of deflection of the beams are then simulated.

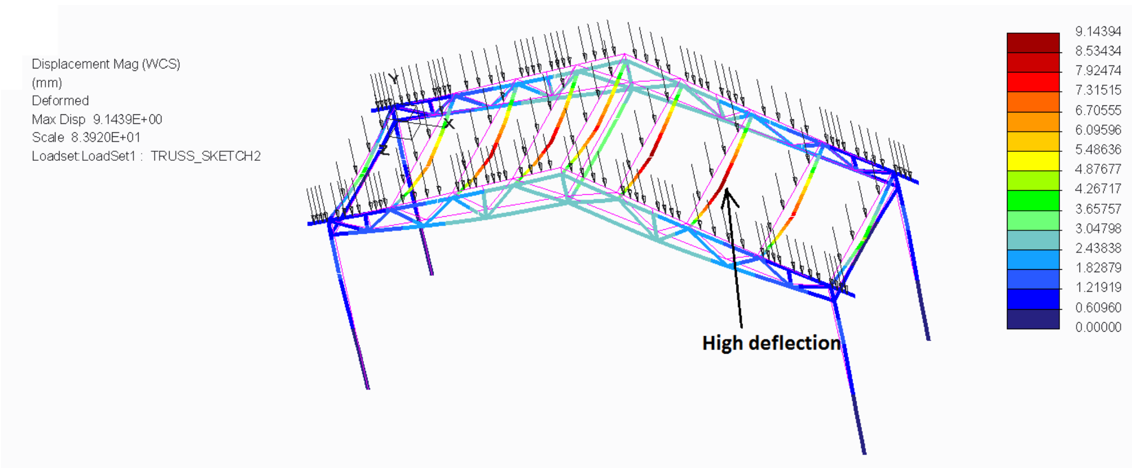


Figure 66 Model showing displacement

The connecting beams have the highest total displacement of 9.14mm as shown above. The high values of stress and deflection occur on the same beams. The connecting beam is isolated and more analysis is done to find out more about the stresses and deflection on this critical member.

This can cause damages especially if there is an unexpected load value. The deflection is not acceptable and this calls for a redesign of the connecting beams.

5.7.2 Combined loads analysis

The wind loads and snow loads are then combined and simulated to investigate if the structure will hold the loads. The problem of the model is as shown in figure 67 below.

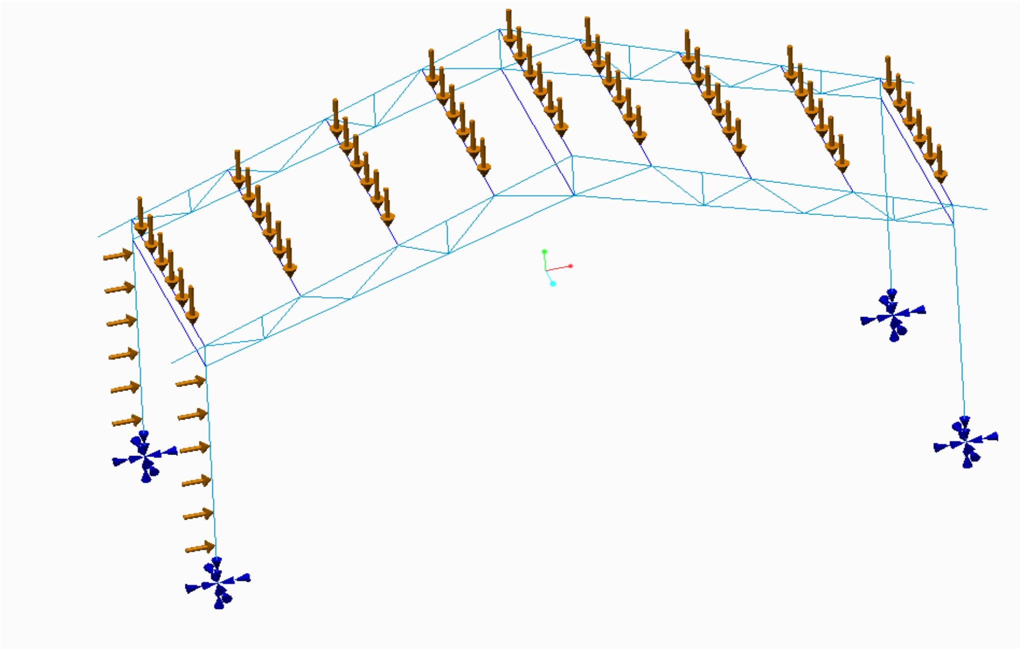


Figure 67 Model set up

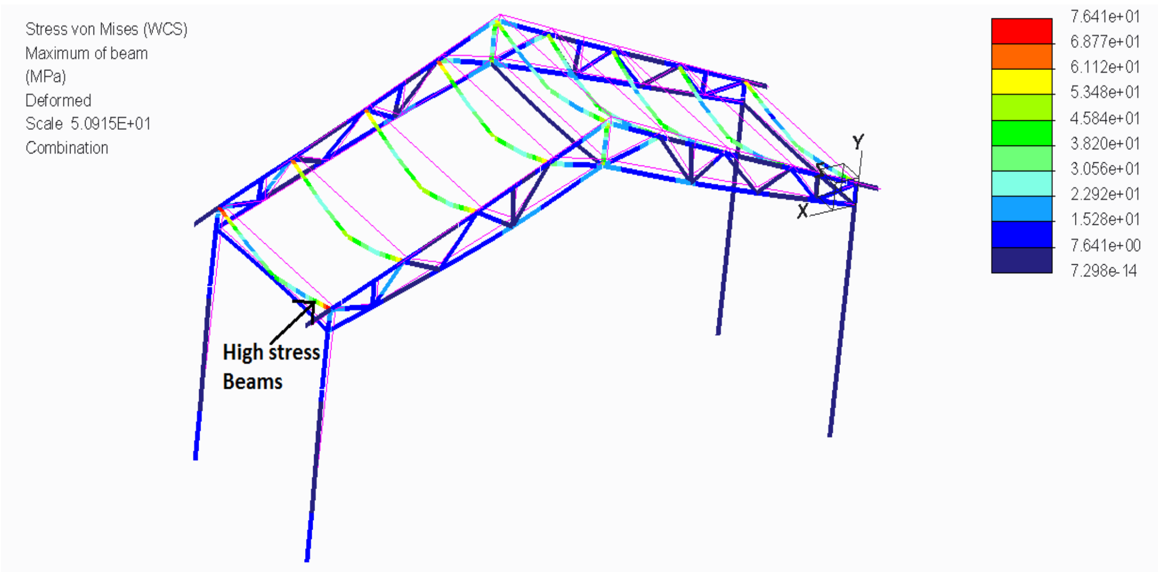


Figure 68 Simulated model (stress)

There is no much difference in the stress and displacement values for the combined loads. However the conclusion is still the same that the beams have to be redesigned.

5.8 Bracing and beams connected trusses

The next step in the analysis is to model and simulate a larger part of the building to check for possible faults. The model comprises of the bracing and connected beams as shown in figure 69 below.

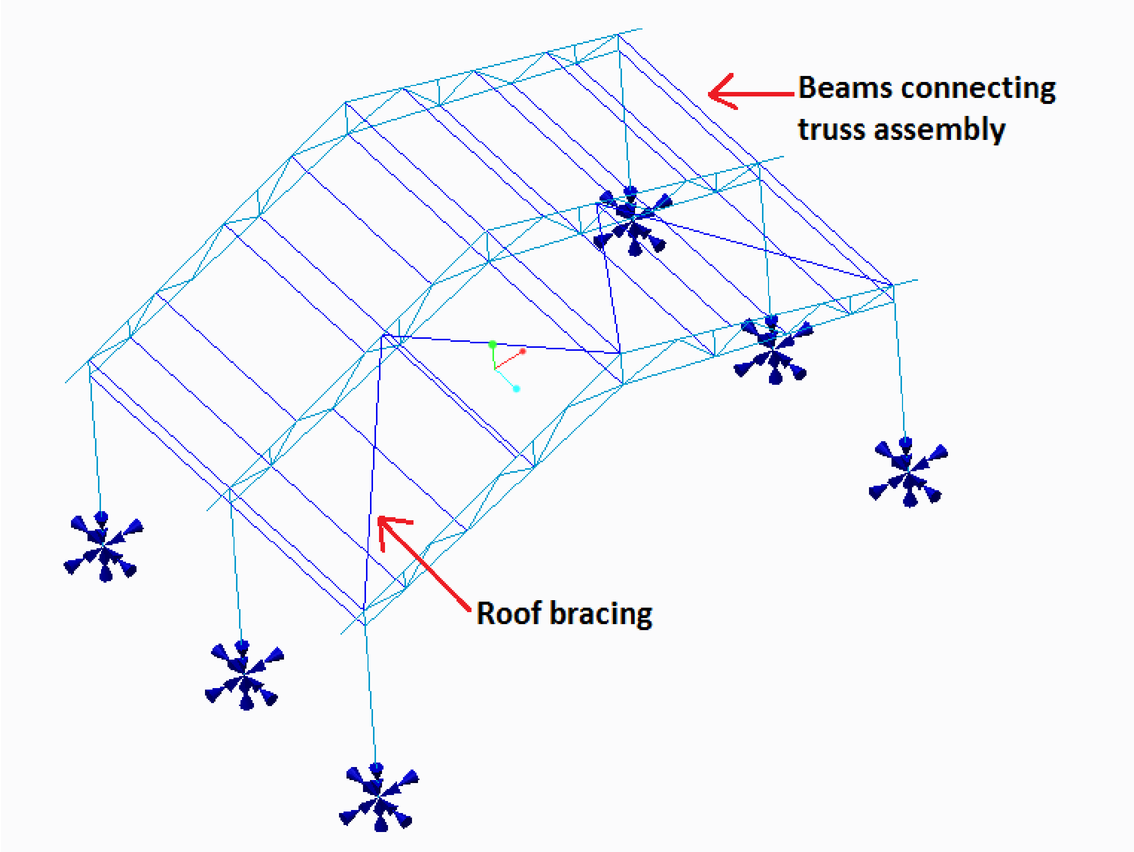


Figure 69 Larger assembly model

The same procedure of analysis follows where first the snow load and wind load are analyzed separately and then both are combined.

<u>Wind Load</u>

First the parameters for the wind load model are set up and then the simulation follows. The model set up and simulation result is as shown below.

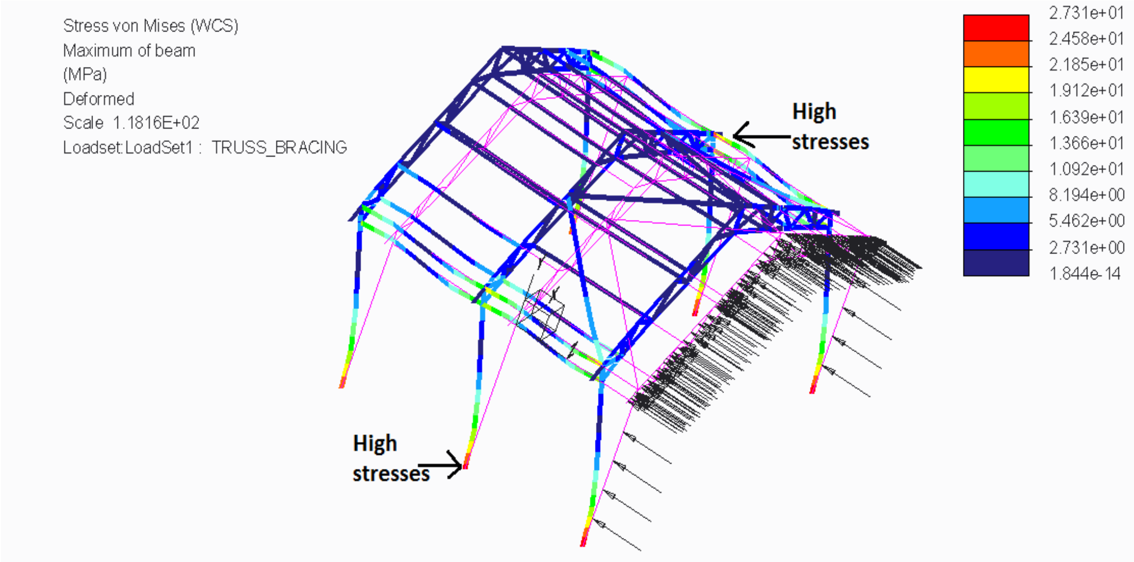


Figure 70 Stress results for wind load

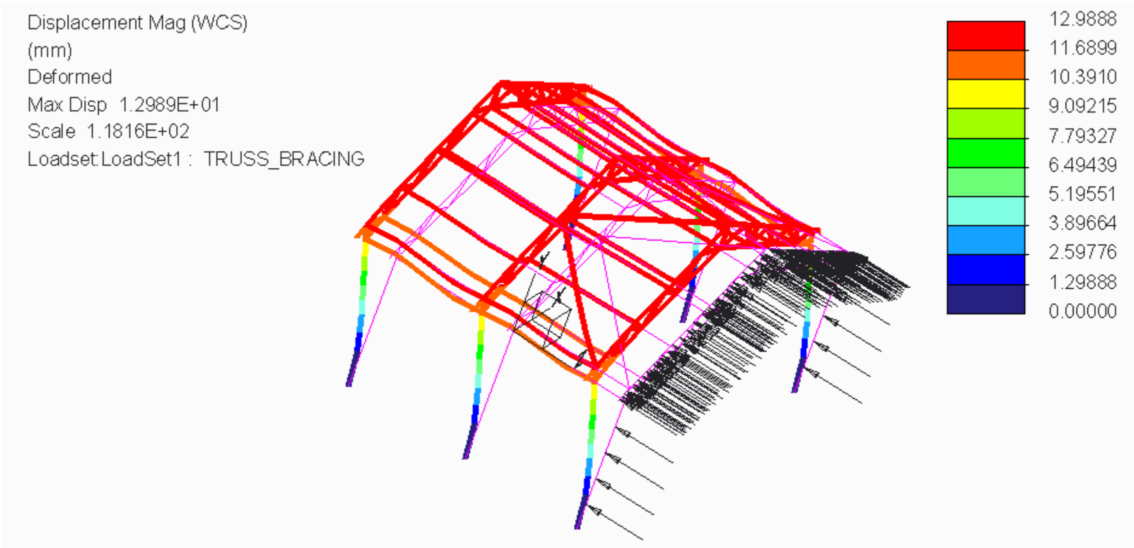


Figure 71 Deflection results for wind load

The total deflection of 12.88mm is quite high and the main reason for this is because of the bracing members. They should be stiffer to reduce the deflection and hence the redesign should be done for the members.

Snow Load

Snow load analysis for the model is set up as below.

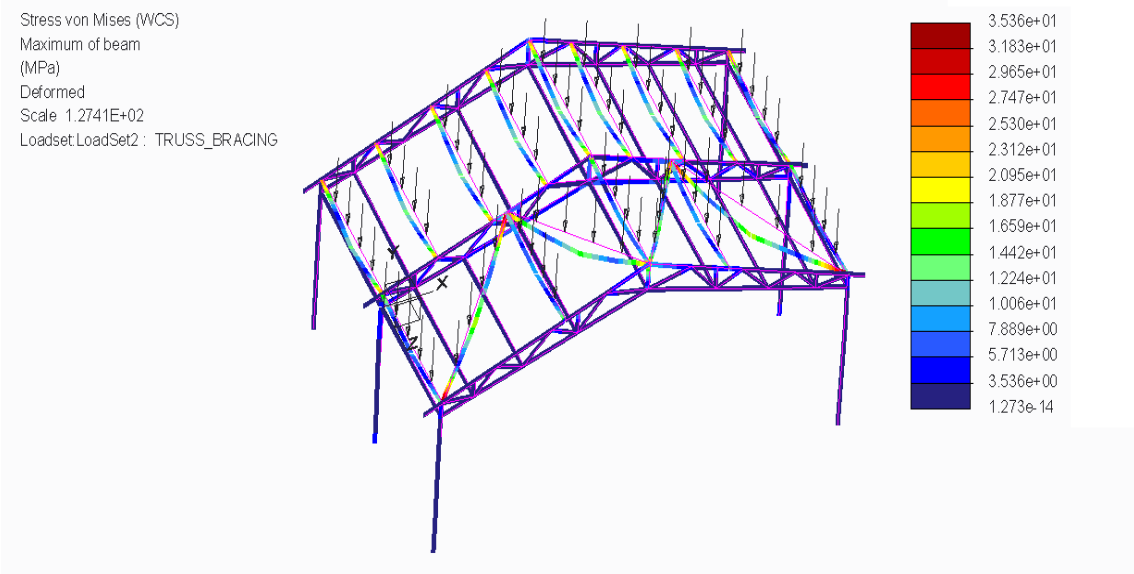


Figure 72 Stress results for snow load

High stress occurs at the bracing whose value is low at a value of 35.36MPa which is below the yield point. The deflection result is as below.

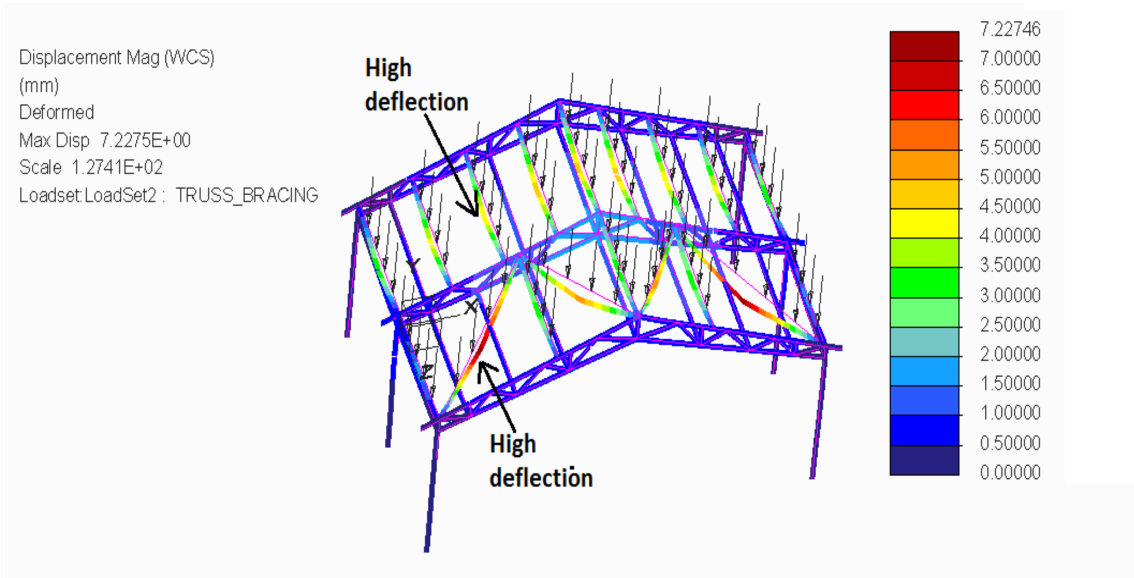


Figure 73 Deflection results

The deflection is low compared to the allowable deflection and it is almost similar to the bracing analysis done earlier.

Combined Loads Analysis

As done before, the loads are combined to imitate some severe weather condition that might occur. The model set up and results are shown below.

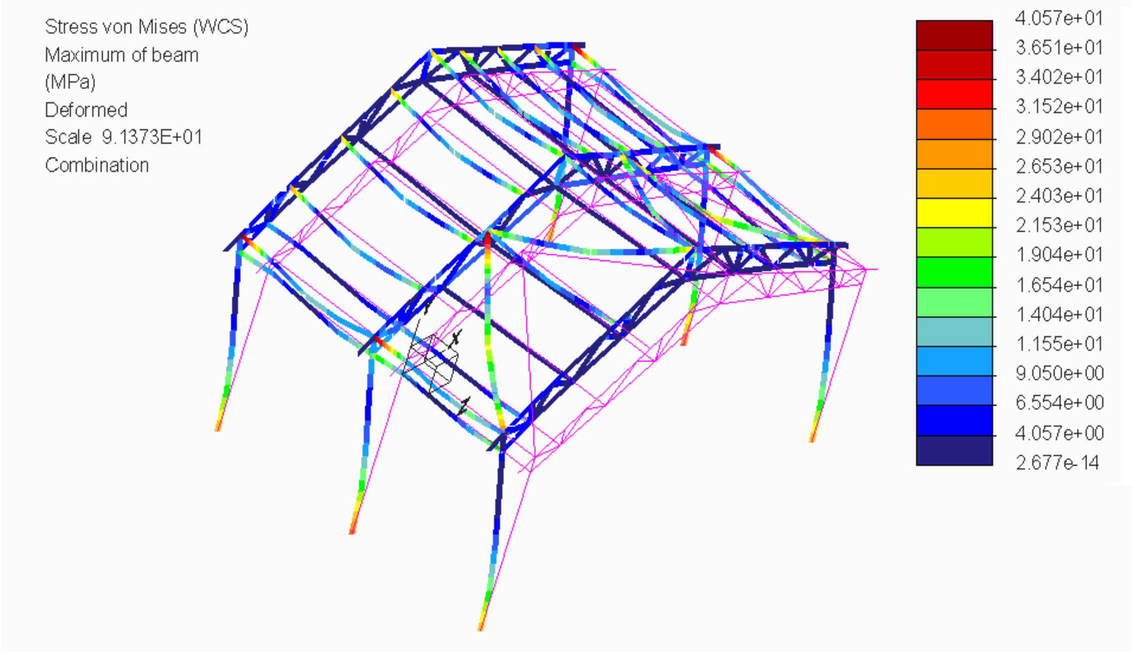


Figure 74 Results of simulation

5.9 Summary

The building´s design for strength before analysis is quite impressive because most of the beams can resist the forces caused by the given loads. Different loading conditions are applied on the structure to find out if the structure will still be stable. However, some beams under certain loading conditions tend to very have high stresses and deflection.

Safety values for stress and deflection are calculated in the beginning of the analysis. If the structure experiences values of stress and deflection that are very close to the safety values, then this calls for redesign of the members of the structure.

6 REDESIGN OF THE WAREHOUSE BUILDING

6.1 Introduction

Some members of the building are under undesirable stresses and deflection. Even though these values do not lead to failure, there is need for redesign to raise the reliability levels in terms of strength. In order to improve the strength, two properties of the structure members have to be changed. One is the material property of the beam member (E) and the other is the cross section properties of the beam (I). These make up the flexural stiffness of the beam. Increasing the flexural stiffness reduces the stress in the beams.

The material used for the beams in the building is structural steel (S355). This material is considerably strong and stiff having a high level of young´s modulus. It is also cheap and highly available which makes it economically viable to be used in this warehouse building. There exist other types of materials that have better strength and stiffness values than the structural steel. Even more materials have better strength to weight ratios than the material used in this building. However, these materials are much more expensive than structural steel and do not make economic sense to use them in the structure. The table below illustrates the difference in prices of high strength metals.

Table 9 Metal price comparisons (Roy Beardmore, 2010)

Material	Cost/tonne	Relative Cost (weight)	Relative Cost (volume)
	�/tonne		
Steel (Billet) LME-Nov-2010	321	1	1
Steel (Hot Rolled Plate)-MEPS-July-2010	505	1,6	1,6
304 Steel (Hot Rolled Plate)-MEPS-July-2010	2 536	7,9	7,9
316 Steel (Hot Rolled Plate)-MEPS-July-2010	3 535	11	11
Tin- LME-Nov-2010	15 458	48	45
Aluminium Alloy - LME-Nov-2010	1 407	4,.4	1,5
Aluminium - LME-Nov-2010	1 425	4,4	1,5
Copper - LME -Nov-2010	5 279	16,4	18,7
Zinc - LME -Nov-2010	1 412	4,4	4,0
Nickel - LME -Nov-2010	14 398	44	51
Lead - LME-Nov-2010	1 414	4,4	6,4
Titanium (ingot 6AL-4V)	15 700	49	28

Clearly, steel is much more realistic to use than other materials which have a higher young´s modulus than other materials such as titanium. The table below shows the different modulus of elasticity for selected materials.

Table 10 Young´s modulus comparison (Roy Beardmore, 2010)

Material	Young´s modulus (GPa)
Aluminium	69,5
Steel	210
Titanium	110,3

It is therefore concluded that changing the material for the beam members is not an option due to the price of the alternative materials.

Changing the cross section properties is the next viable option to improve the strength of the beam members. It is common knowledge for engineers to understand that the higher the second moment of area, the higher the strength of the beam member. Increasing the second moment of area reduces the bending stresses. The second moment of area is changed by altering the dimensions of cross section or choosing a different shape of cross section.

6.2 Truss member redesign

In the roof truss analysis the middle section member has the highest stress resulting from the snow load. If dramatic snow fall occurs, then the beam would be very close to failure. This raises the need for redesign.

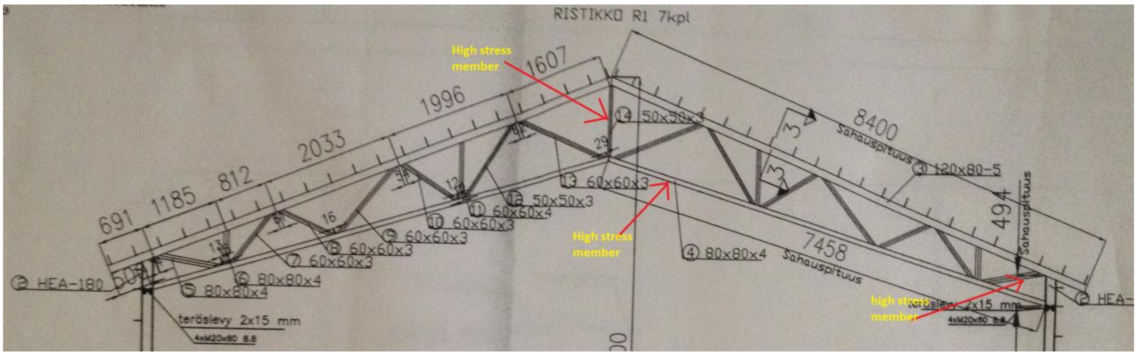


Figure 75 High stress members to be redesigned

The cross-section of the highlighted members is changed to decrease the stress values. This is illustrated in the figures below.

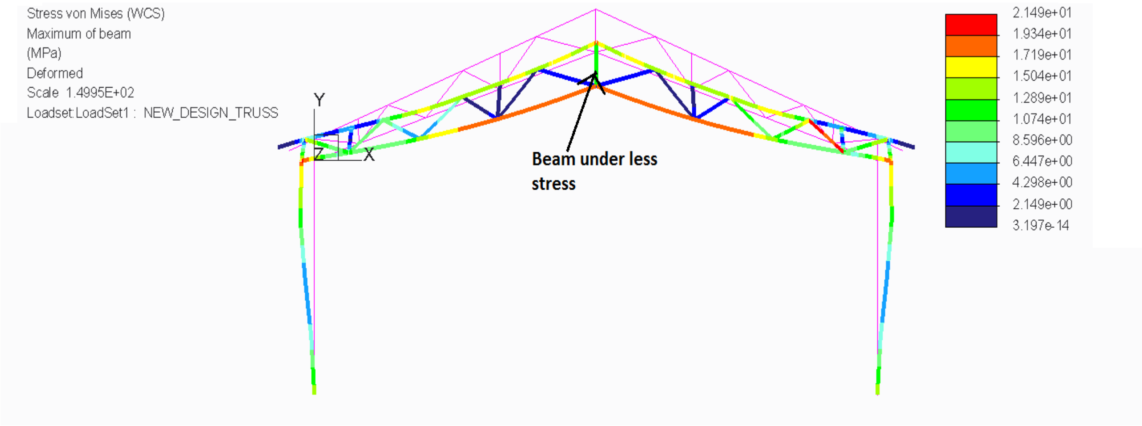


Figure 76 Simulated model after redesign

The cross section of the beam shown above was 50mm by 50mm by 3mm with second moment of area of 19.47 mm^4. The new design has a cross section of 70mm by 70mm by 5mm with second moment of inertia of 84.63mm^4. As shown in the figure 76, the Von Mises stress is now 12.89MPa compared to the 27.76MPa that was calculated in figure 42. This illustrates that the truss is stronger and stiffer than before due to the changes made of the cross section.

6.3 Bracing beam members

Some of the bracing beam members experienced high stresses and deflection almost too close to the safety values. This calls for a mandatory change to the design of the bracings. The obvious change for this problem is to increase the flexural stiffness of the beam members. The second option is to increase the number of beam members to increase the overall strength. This latter option has a negative effect on the overall weight of the structure. The higher the number of members, the more the weight increases. The best option is to increase the flexural stiffness by increasing the second moment of area.

As shown in figure 61, the two beam members of the bracing experience very high stresses. The beam members have a cross section of 60mm by 60mm by 3mm. The high stresses of up to 125MPa are too dangerous. The beam cross section is changed to 80mm by 80mm by 5mm and the simulated results for the snow load are shown below. The stress has decreased by more than 100MPa to 47.28MPa by changing the cross section properties of the beams. The new result is acceptable and safer.

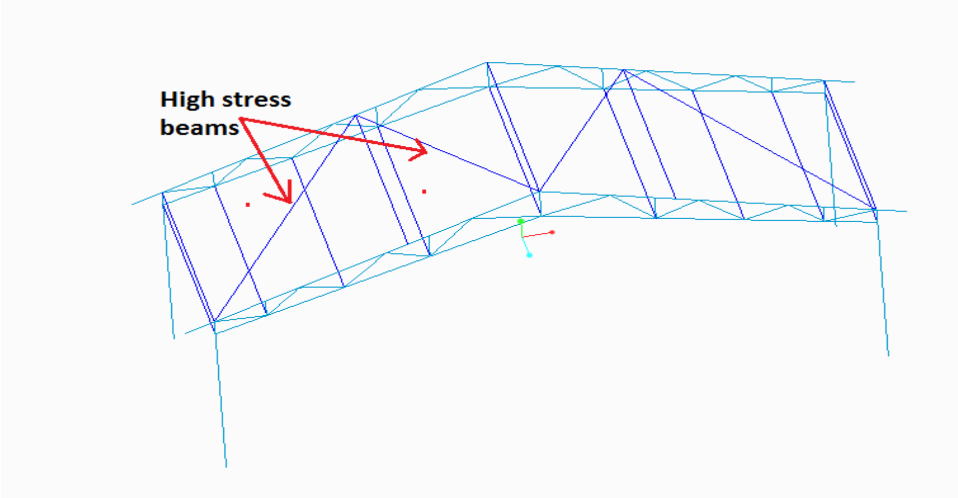


Figure 77 High stress beams.

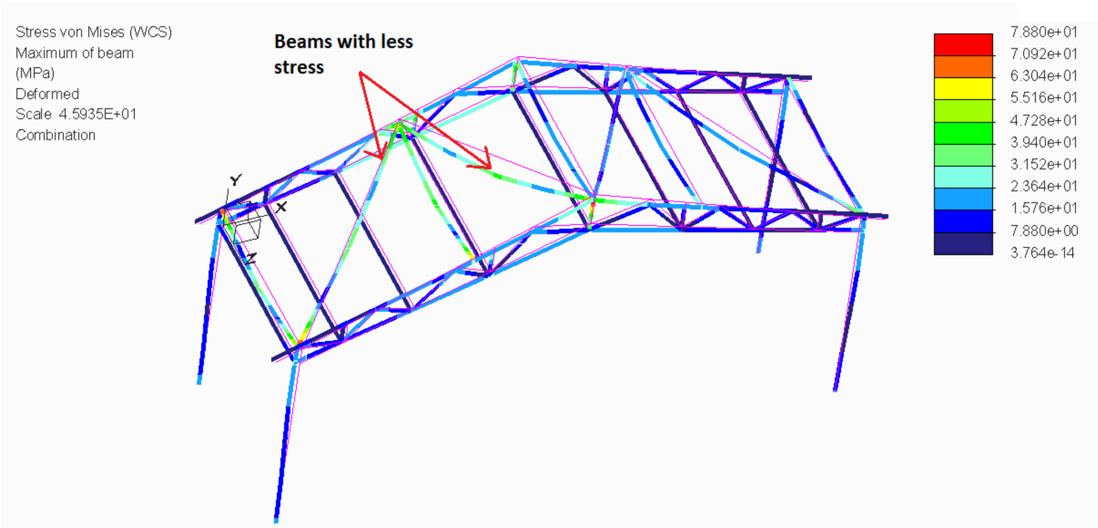


Figure 78 Simulated results

6.4 Beams connecting the trusses

The beams connecting the trusses in the middle of the structure are next redesigned. The main issue with these beams is the deflection and overall stiffness. The same procedure is used for making the new design. The cross sections of the beams before are 60mm by 60mm by 3mm. The simulated results of the new design are shown in the figures below.

The cross section value of the new design is 80mm by 80mm by 3mm. The new cross section results to a higher flexural stiffness causing reduced stresses and deflections. The new high stress and deflection values are 36.3MPa and 9.69mm respectively. These values are less than the values displayed in figure 79 and 80.

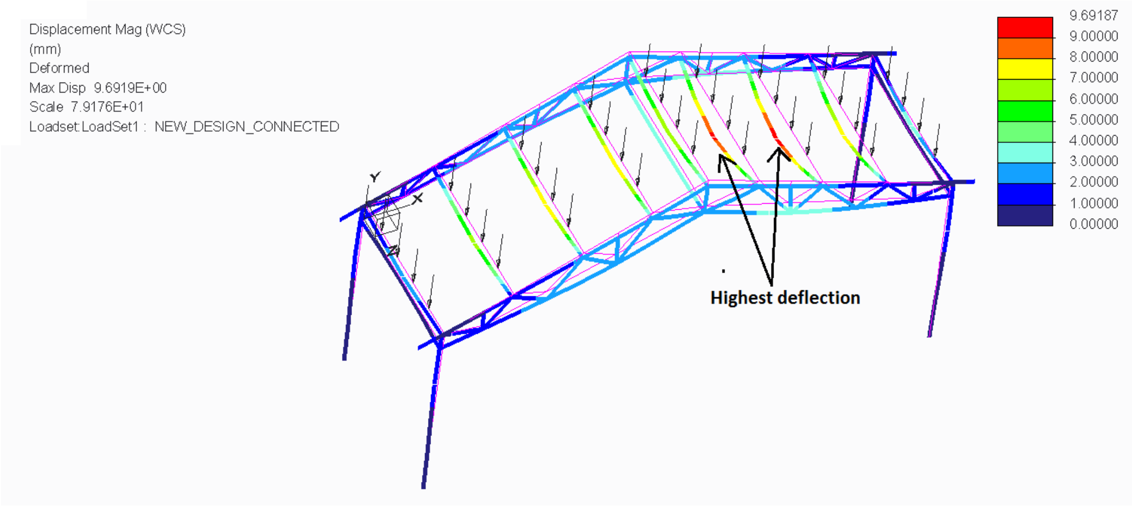


Figure 79 Deflection results

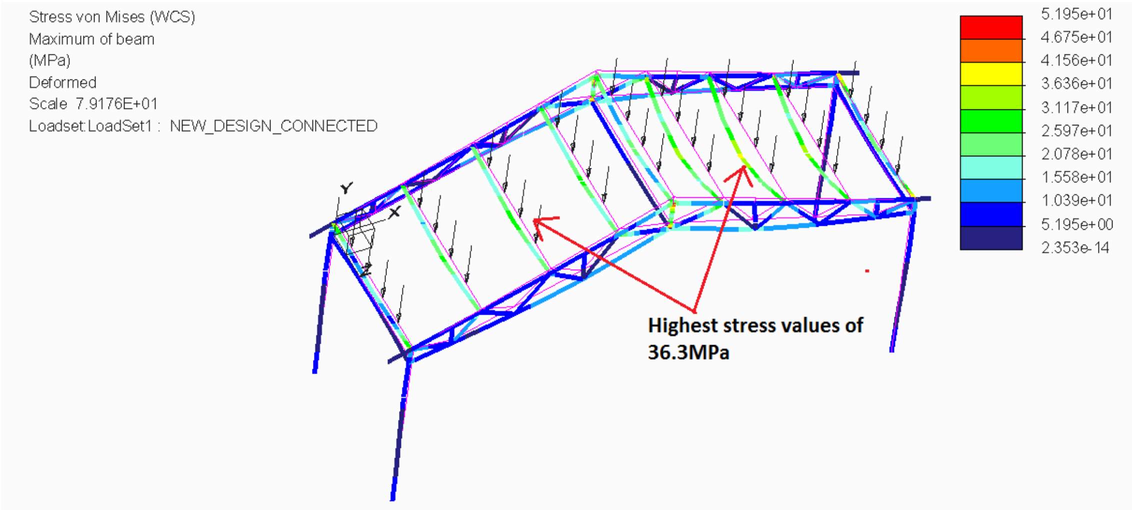


Figure 80 Stress results

6.5 Conclusion

There are several ways of performing the redesign of the members. The most economical and safe method is always preferred. By increasing the flexural stiffness (E*I) of the beams in the structure, high deflection and stress levels are greatly reduced. The new beam designs are incorporated in the structures and a new analysis is done. The difference is very clear in the results and the new design is accepted with a high level of confidence.

7 NEW STRUCTURE (OFFICE) IN THE BUILDING

7.1 Introduction

After analysis and redesign of the building, it is concluded that the structure is strong and stiff enough to carry the given loads. The next step is to push the limits of the strength of the building by adding a new structure. The structure to be designed is an office for the warehouse manager. The "skeleton" part of the office is designed and analysed.

The office is designed so that it is attached to the side wall of the warehouse as shown in figure 81 and 82.

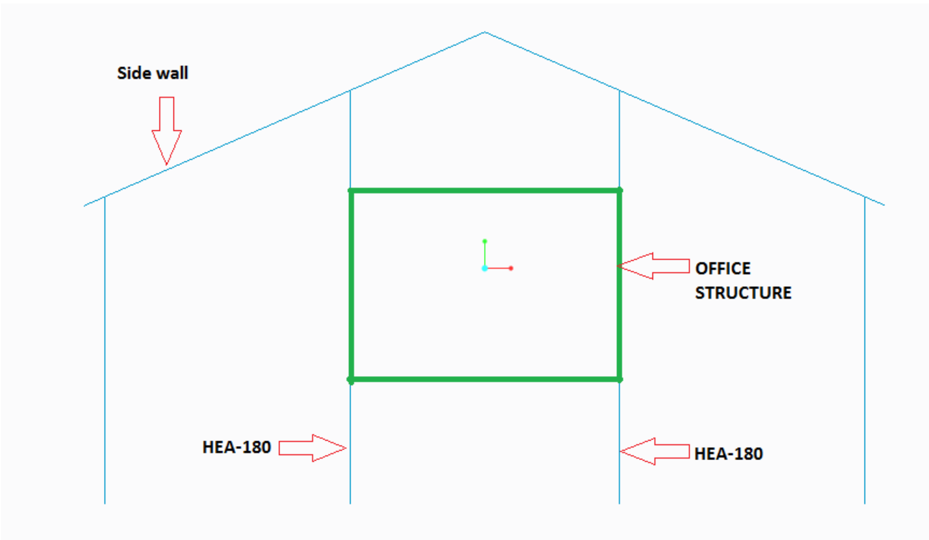


Figure 81 Office attached to side wall (front view)

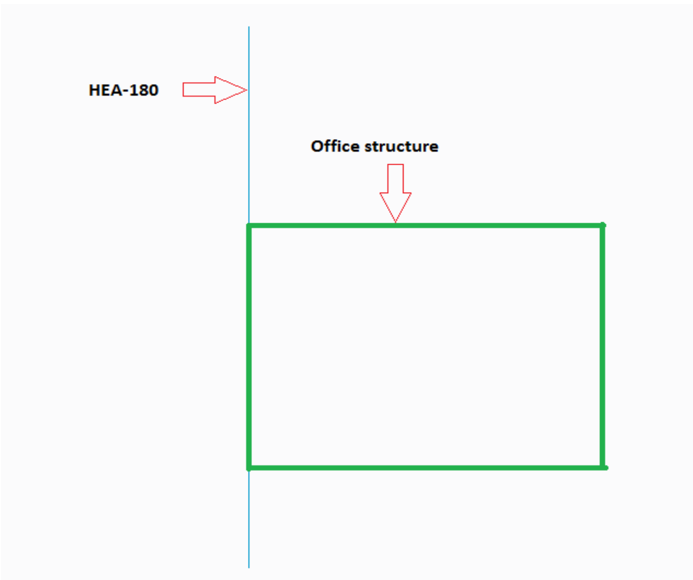


Figure 82 Office structure side view

7.2 Design of the office structure

7.2.1 Main beam design

The first step in the design is to calculate the loads that will act on the structure. According to the euro code standard EN 1991-1-2:2002, this structure falls into category B which is office areas. The pressure load for this category is 2 to 3 KN/m^2. The highest pressure load which is 3 KN/m^2 is chosen as the design load.

The length of the beams supporting the floor of the office from the column HEA 180, are designed to be 3m and the width is the distance between them which is 4.6m.

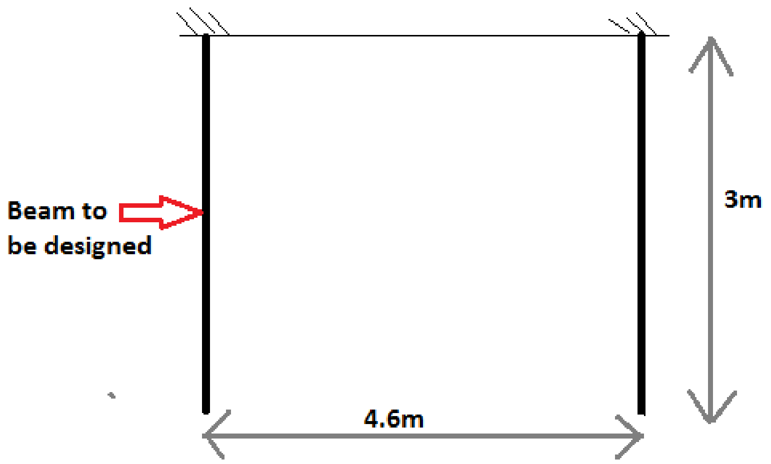


Figure 83 Supporting beam concept design

The next step is to convert the pressure load to a line load on the beam. The line load equals the pressure load multiplied by the perpendicular distance (4.6m) which results to 13.8 KN/m. Since there are two beams, the line load is to divide by two. The calculations for the beam selection are shown in appendix 4 based on the simplified model below.

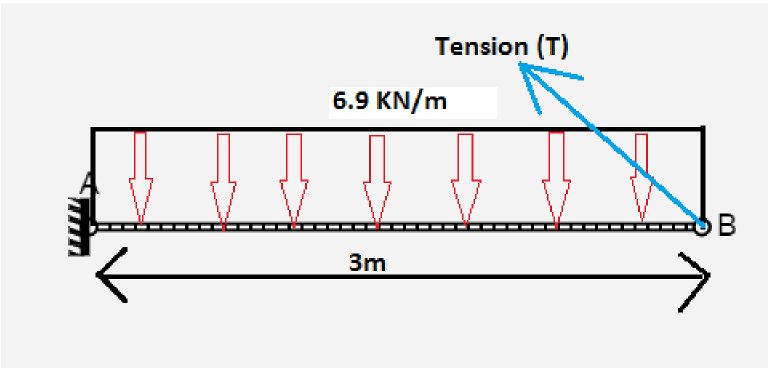


Figure 84 Simplified model of the beam

Based on the calculations in appendix 4, the needed section modulus that will be used to choose the beam is 87.46cm^3. A beam that has a value higher than section modulus calculated is chosen using the chart in appendix 4. I beam section is preferred because of its good geometrical stiffness properties. Section IPE 160 is chosen to act as the beam at the base.

7.2.2 Tension rod design

The beam which is connected to the column on the side wall requires an extra support to support the weight. The tension rod therefore is designed to reduce the stress on the beam. The tension rod is connected to the side wall column as shown in figure 85 below.

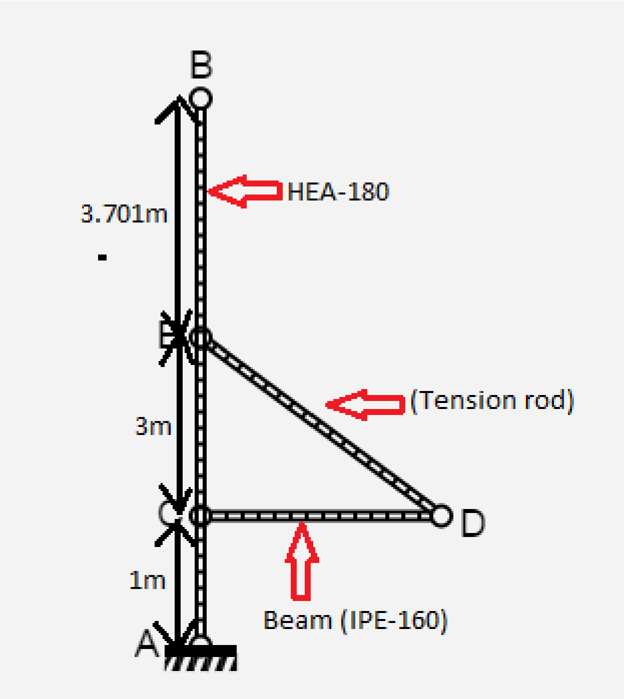


Figure 85 Tension rod attached to column

Calculations to find the exact dimensions required for the rod are in appendix 5. It is concluded the rod should be made of steel and have a diameter of approximately 10.00 mm.

7.2.3 Connecting beams design

The design proceeds to design middle beams which connect the two IPE 160 beams. These beams will act as the main office floor support and are shown in figure 86 below. Detailed calculations and selection chart are in appendix 5.

The needed section modulus for the beam is 30.197cm^3. A rectangular beam of 100 by 60 by 6 is chosen according to the chart in appendix 6.

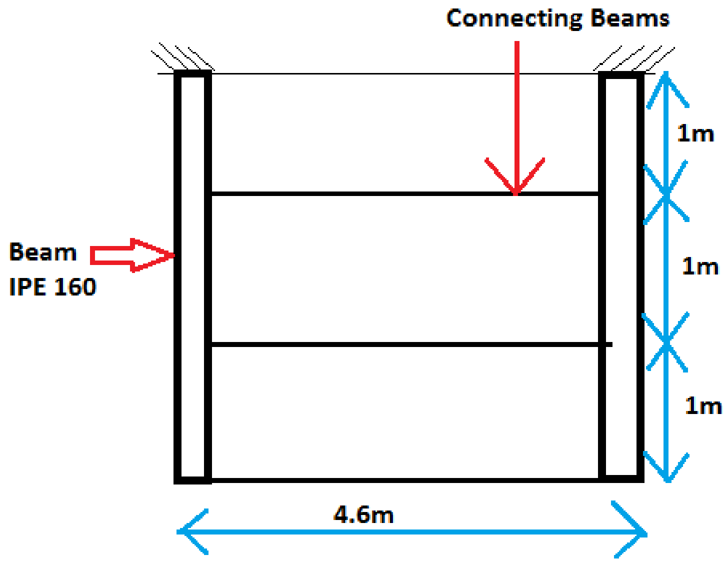


Figure 86 Sketch showing connecting beams

7.3 Analysis of new design

To verify if the new design is strong enough to handle the loads, the new structure has to be analyzed. The analysis also shows the effect of the new design to the already existing structures.

The tension rod and main supporting beam are first analyzed to find out the effects they have on the column. The model in creo-simulate program is as shown in figure 87.

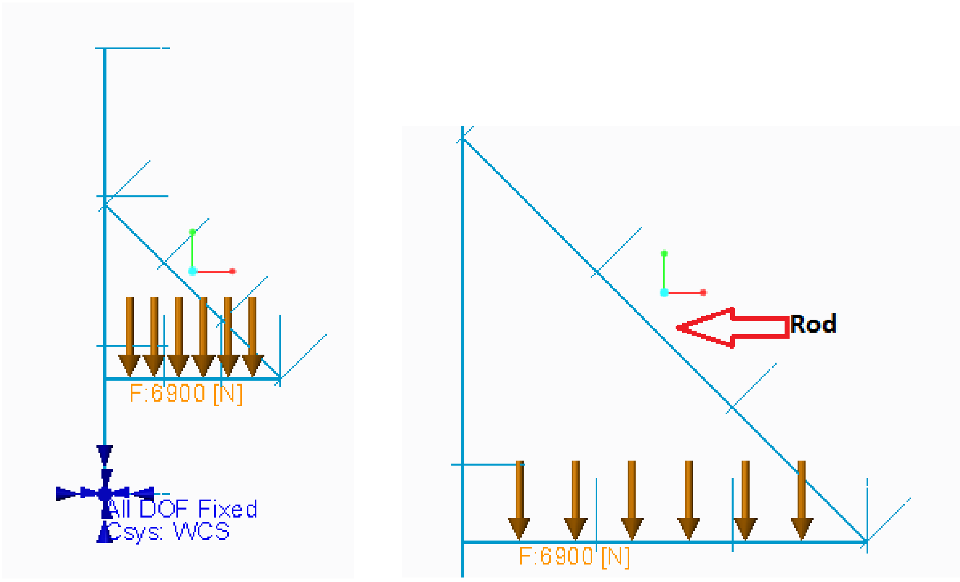


Figure 87 Side view model of new design

A distributed load is applied on the beam and the effects are as shown in figure 88 below.

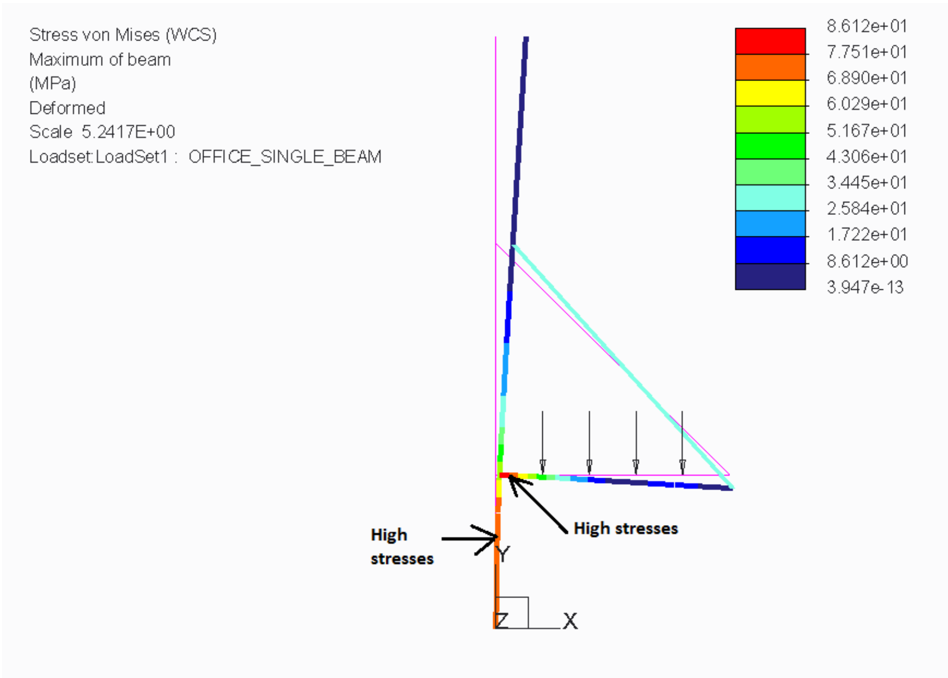


Figure 88 Simulated model showing the stress results

The highest stress are at the bottom of column HEA 180 with a value of 86.12MPa. This value is not very high and therefore is accepted since the strength has not reached a value higher than the yield strength.

The connecting middle beams as shown in figure 86 are then analyzed to check if they can support the loads. Figures 89 and 90 below show the model in creo-simulate.

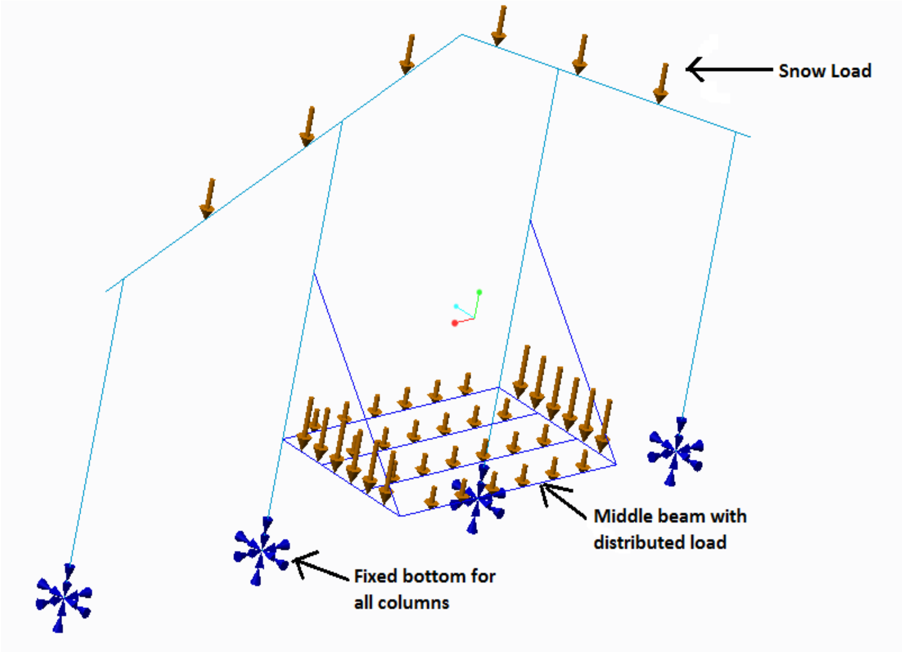


Figure 89 Model set up

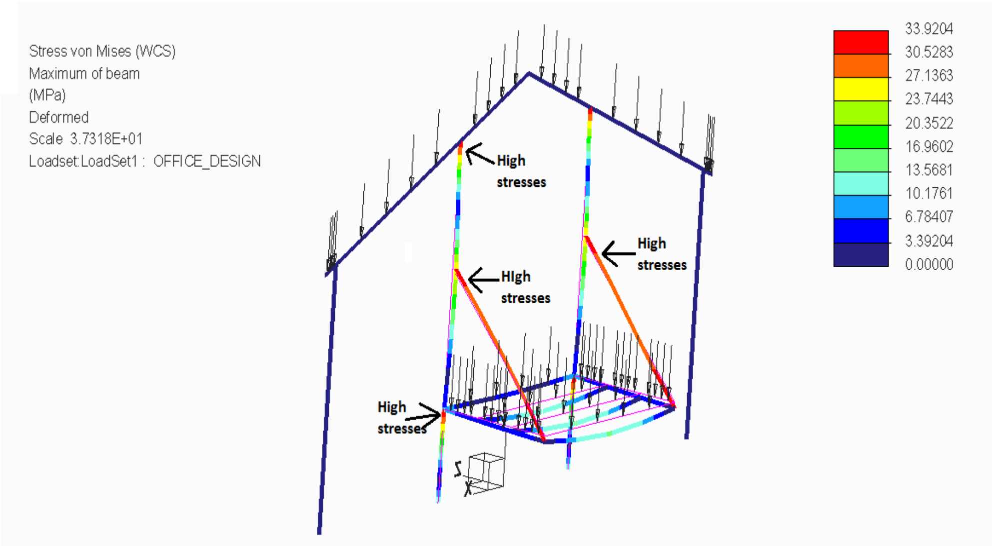


Figure 90 Simulated model showing stress

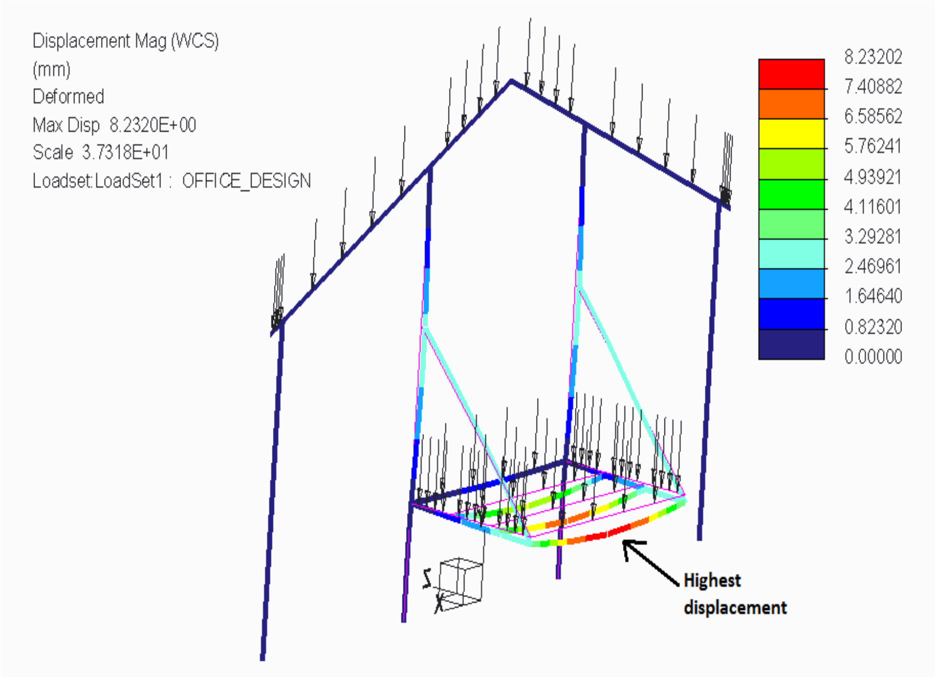


Figure 91 Simulated model showing deflection of middle beams

The highest stress value is 33.92MPa which is below the yield strength of the structural steel used in the building. The values are not close to the yield strength and are therefore acceptable. The highest displacement value in the middle beam is 8mm approximately. The displacement and stress due to the applied forces on the beams are very low. Conclusion to this is that the middle beams are well designed to support the floor of the office. The drawings to the I-beams and middle beams are found in appendix.

The beams which have the highest stresses and displacements are analyzed in more detail so that the level of confidence can be increased. The middle beam is modelled in 3-dimension and then analyzed. In a nut shell, idealized beams are converted to 3 dimensional models to achieve more accurate results.

The 3 dimensional model uses the results from figure 90 to set up the constraints. The left and right end surfaces are displaced with values obtained earlier in the figure 90. The middle beam is then simulated to find out the specific stresses and deflection and how they are distributed in the beam. Figure 92 below show the model and simulated results.

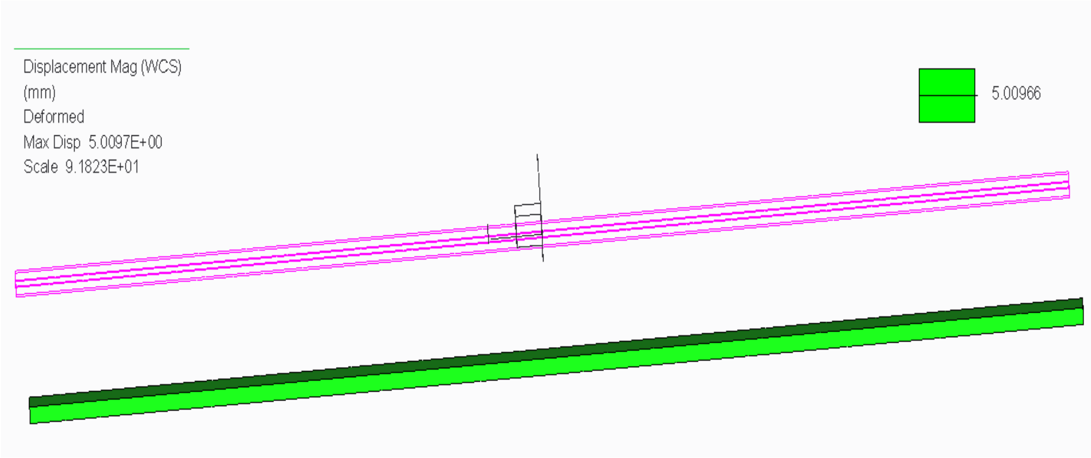


Figure 92 Detailed model

The middle part of the beam has the highest deflection as it was shown in the idealized beams. However, the value is more precise with a value of 5.009mm.

The final 3 dimensional model is as shown in figure 93 below.

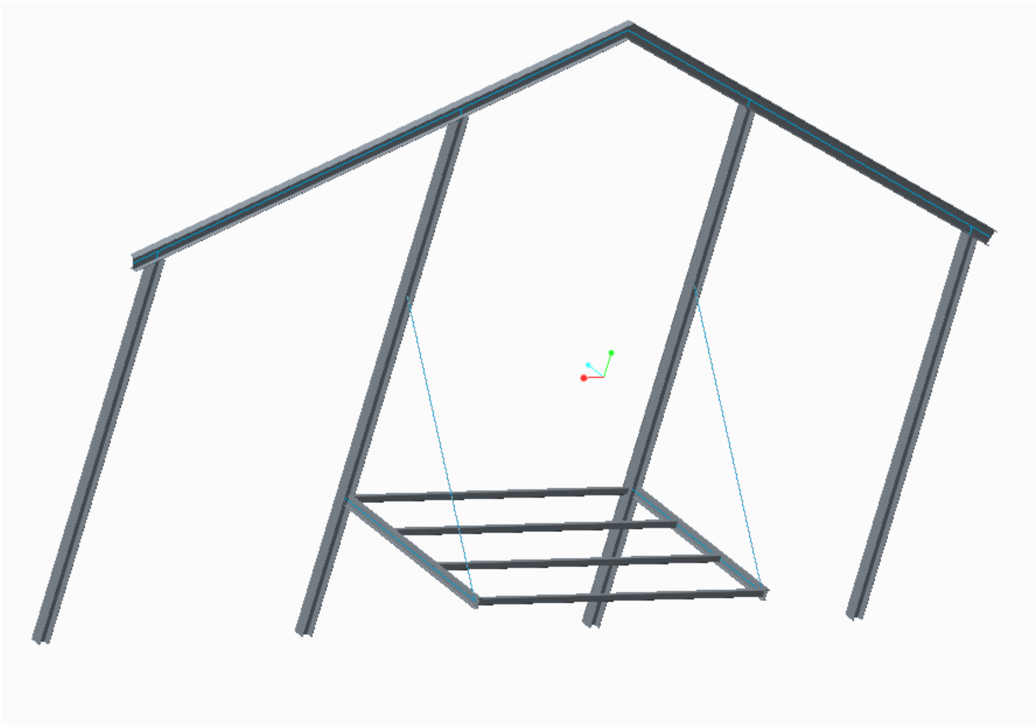


Figure 93 Assembly of new office design and wall

7.4 Conclusion

The new design does not compromise the strength of the already existing side wall structure. The office design as well is strong enough to hold the calculated loads. The side walls, floor, roof and interior strictures will also not destabilize the office structure.

8 CONCLUSION

The main aim of this thesis was to analyse a residential building using the finite element method. The method of analysis was chosen carefully since there are many numerical techniques of doing the analysis. The residential building was critically investigated and the necessary changes to the design were made. The main question that was asked throughout was, could the building sustain the forces generated by the loads? Changes were made using strength of materials and other design of mechanics knowledge. The analysis improved the integrity of the building and after the new design the residential building is regarded as more reliable in terms of strength.

This method of structural analysis has been thoroughly criticized at times due to the analyst´s mistakes. A lot of caution was taken at the start of the analysis so that no errors accumulate. The example problems that were analysed before the main analysis were very important. This was because they show in a nut shell, how the whole process is followed. Another example was how the loads flow from one part of the building to another. Identifying the correct loads is essential as this acts as the start of analysis and if the loads are incorrect then wrong designs are made.

Most parts of the building analysed, were found to be very strong and safe. There were only a few beams that raised high concerns about their reliability. However, when different loading conditions are applied, more beams experience higher stresses. The solution is to design the vulnerable beams to higher safety values than before. The new changes improved the strength of the residential building and hence it was more reliable. The new models with new designs further support the design.

The finite element analysis as a method of structural analysis is the most important tool in the thesis work. As long as the errors are kept to minimum, the method can be used extensively to investigate structures. This method made possible what could have been impossible if hand calculations were made. For example, the stress is well presented graphically throughout the beams and columns leading to very comprehensive conclusions of the analysis. Structural analysis of the residential building using FEM in this thesis is a good demonstration of how this method of analysis is effective.

However, there is more room for improvement for the structural analysis using FEM. For instance the thermal analysis for the structure could be carried out to find if the temperature changes affect the strength of the building. Also, modal analysis that investigates the vibrations especially due to earthquake forces could be done. This greatly depends on the location of the building.
The analysis done in this thesis work was successful since the goals set in the beginning were achieved. Many have warned about the use of FEM software as a tool of analysis due to the deceiving graphics and presentations. But by taking precautions and confirming the results using hand calculations, it has been proven that this method is reliable if the analyst is knows

the topic well. Therefore, the FEM analysis method could be used for structural analysis for complicated structures as it has been proved in chapter five.

The new design and analysis can be presented to interested parties with a high level of confidence.

SOURCES

Tadeusz Storlaski, 2006. Global specs electronics. Application of Ansys to stress analysis. Accessed 16th September 2014.
http://www.globalspec.com/reference/42954/203279/html-head-chapter-3-application-of-ansys-to-stress-analysis

Auto fem, n.d. Deflection of a beam under a umiformly sitributed load. Accessed 16th September 2014.
http://www.autofemsoft.com/examples/deflection_of_a_beam_under_a_u.html

Numerical methods n,d. What is a Numerical Method. Accessed 20th May 2014.
http://www3.ul.ie/~mlc/support/CompMaths2/files/WhatNumericalMathod.pdf

Stephanie Wehner, 2013. Macmillan publishers. An information-theoretic principle implies that any discrete physical theory is classical. Accessed 16th September 2014.
http://www.nature.com/ncomms/journal/v4/n5/full/ncomms2821.html#/author-information

Toogood, 2012, 2-5. Creo Simulate Tutorials Releases 1.0 and 2.0. Schroff Development Corporation.

Toogood, 2012, 2-8. Creo Simulate Tutorials Releases 1.0 and 2.0. Schroff Development Corporation.

H method, nd. Dr Eric klang. Accessed 30th October 2010.
http://www.mae.ncsu.edu/klang/courses/mae533/Reference/Methods.htm

Using mirror symmetry, n.d. Simulation Pro Engineer mechanica. Accessed 16th September 2014.
http://www3.eng.cam.ac.uk/DesignOffice/cad/proe-wild3/usascii/proe/promec/getstart/getstart/reference/ex_std_sym.htm

EN 1991-1-1:2002, 20-21. 6.3 Characteristic values of imposed loads. 6.3.1.1 Categories. Accessed 22.5.2014.
file:///C:/Users/MUGO/Downloads/load%20on%20buildings%20(1).pdf
EN 1991-1-1:2002, 12 Section 2: Classification of actions. 2.1: Self weight. Accessed 22.5.2014.
file:///C:/Users/MUGO/Downloads/load%20on%20buildings%20(1).pdf

EN 1991-1-3: 2002, 17-18. Section 5 Snow load on roofs. 5-1 Nature of the load. Accessed 22.5.2014.
file:///C:/Users/MUGO/Downloads/load%20on%20buildings%20(1).pdf

EN 1991-1-3: 2002, 18. Section 5 Snow load on roofs. 5-1 Nature of the load. 3P. Accessed 22.5.2014.
file:///C:/Users/MUGO/Downloads/load%20on%20buildings%20(1).pdf

North roof load zone, n.d. How we build. Accessed 22.5.2014. http://www.fairmonthomes.com/snow.php

The effects of imperfections, n.d. Braced frames. Accessed 30th October 2014. http://www.steelconstruction.info/Braced_frames

Flowing events, n d. Accessed 29.5.2014. http://www.flowingevents.com/2012/01/some-must-read-rule-to-save-build-ing.html

Murty, n.d, 1-5. C.V.R Murty, Rupen Goswami, A.R Vijayanayarayan, Vipul V. Mehta. Earthquake behaviour of buildings. Accessed 30th October 2014. http://www.iitk.ac.in/nicee/IITK-GSDMA/EBB_001_30May2013.pdf

Murty, n.d, 2. C.V.R Murty, Rupen Goswami, A.R Vijayanayarayan, Vipul V. Mehta. Earthquake behaviour of buildings. Accessed 30th October 2014. http://www.iitk.ac.in/nicee/IITK-GSDMA/EBB_001_30May2013.pdf

EN – 1991 – 1 -4, 43. Section 5 Wind actions. 5.2 Wind pressure on surfaces. Accessed 20.9.2014. file:///C:/Users/MUGO/Downloads/wind%20load%20 (3).pdf

Roy Beardmore, 2010. Metal costs. Last accessed 9th October 2014. http://www.roymech.co.uk/Useful_Tables/Matter/Costs.html

Roy Beardmore, 2010. Properties of materials. Last accessed 9th October 2014. http://www.roymech.co.uk/Useful_Tables/Matter/prop_metals.htm.

A word about anchor bolts, 1992. Foundation and foundation to wall connections. Last accessed 23rd October 2014. http://www.mcvicker.com/twd/apa/eqguide/eqguid08.htm

Appendix 1

Simple cantilever calculation

$E := 210000 \frac{N}{mm^2}$ $I := 307500 \cdot mm^4$ $L := 2000 \cdot mm$ $F := 200 \cdot N$

$A := 900 \cdot mm^2$ $M_{max} := 400000 \cdot N \cdot mm$ $y := 25 \cdot mm$ $\delta_y := 355 \frac{N}{mm^2}$

$Sf := \frac{\delta_y}{2}$ $D_{allowable} := \frac{L}{360}$

$$\sigma_{max} := \frac{M_{max} \cdot y}{I} = 32.52E+006 Pa$$

$$\theta := \frac{F \cdot L^2}{2 \cdot E \cdot I} = 6.194E-003$$

$$\delta_{max} := \frac{F \cdot L^3}{3 \cdot E \cdot I} = 8.259E-003 m$$ OR 8.259mm

$Sf = 177.5E+006 Pa$ $D_{allowable} = 5.556 \times 10^{-3} m$ OR 5.556mm

Appendix 2

Small FEM programme evaluation procedure

SMALL TRUSS THESIS EXAMPLE

INPUT (BLUE BOXES)

Nodal coordinates (x- coord , y-coord)

$$Solmut := \begin{pmatrix} 0 & 5 & 10 & 5 \\ 0 & 0 & 0 & 8.66 \end{pmatrix}^T$$

x-coord

y-coord

Topologimatrix for elements from node Sn to node En

$$Elementit := \begin{pmatrix} 1 & 1 & 2 & 2 & 3 \\ 2 & 4 & 4 & 3 & 4 \end{pmatrix}^T$$

start

end

Properties of elements (Young modulus E , cross section area A and second moment of area I)

$E := 210 \cdot 10^9$ $\quad A_2 := 5 \cdot 10^{-4}$ $\quad I_2 := 5.4167 \cdot 10^{-8}$

cross section is 30*30*5 hollow tube

Material is S355 structural steel

$A_1 := 5 \cdot 10^{-4}$ $\quad I_1 := 5.4167 \cdot 10^{-8}$

$$A := \begin{pmatrix} A_1 & A_2 & A_2 & A_2 & A_2 \end{pmatrix}^T$$

$$I := \begin{pmatrix} I_1 & I_2 & I_2 & I_2 & I_2 \end{pmatrix}^T$$

Nodal loads (X and Y directions)

$$Voimat := \begin{pmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -8 \end{pmatrix}^T \cdot 10^3$$

Allowable stress

$\sigma_{sall} := 300 \cdot 10^6$

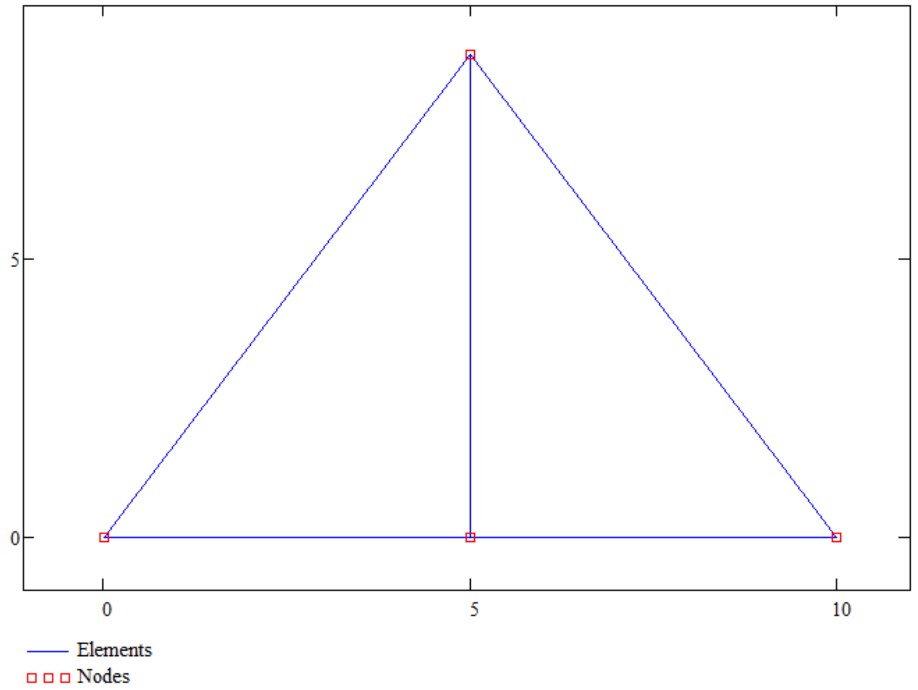
Safety for buckling

(only for graphs)

$nnurj := 2$

The Structure

Kuvan piirto



Calculation

Jaykkyysmatriisin muodostaminen

$$K = \begin{pmatrix} 4.2\times10^{7} & 0 & -2.1\times10^{7} & 0 & 0 \\ 0 & 1.212\times10^{7} & 0 & 0 & -1.212\times10^{7} \\ -2.1\times10^{7} & 0 & 2.363\times10^{7} & -2.625\times10^{6} & 4.547\times10^{6} \\ 0 & 0 & -2.625\times10^{6} & 5.25\times10^{6} & 0 \\ 0 & -1.212\times10^{7} & 4.547\times10^{6} & 0 & 2.787\times10^{7} \end{pmatrix}$$

Kuormitusvektorin muodostaminen

$$R^{T} = \begin{pmatrix} 0 & 0 & 0 & 0 & -8\times10^{3} \end{pmatrix}$$

Nodal displacements {U} from stiffness equation

$U := K^{-1} \cdot R$

$U^T = \begin{pmatrix} 1.1 \times 10^{-4} & -5.714 \times 10^{-4} & 2.199 \times 10^{-4} & 1.1 \times 10^{-4} & -5.714 \times 10^{-4} \end{pmatrix}$

solmusiirtymat elementeittain

$$u^T = \begin{pmatrix} 0 & 0 & 1.1 \times 10^{-4} & 1.1 \times 10^{-4} & 2.199 \times 10^{-4} \\ 0 & 0 & -5.714 \times 10^{-4} & -5.714 \times 10^{-4} & 0 \\ 1.1 \times 10^{-4} & 1.1 \times 10^{-4} & 1.1 \times 10^{-4} & 2.199 \times 10^{-4} & 1.1 \times 10^{-4} \\ -5.714 \times 10^{-4} & -5.714 \times 10^{-4} & -5.714 \times 10^{-4} & 0 & -5.714 \times 10^{-4} \end{pmatrix}$$

1 row normal forces, 2 row buckling forces

$$N^T = \begin{pmatrix} 2.309 \times 10^{3} & -4.619 \times 10^{3} & 0 & 2.309 \times 10^{3} & -4.619 \times 10^{3} \\ 4.491 \times 10^{3} & 1.123 \times 10^{3} & 1.497 \times 10^{3} & 4.491 \times 10^{3} & 1.123 \times 10^{3} \end{pmatrix}$$

normal stresses

$$\sigma := \overrightarrow{\frac{N^{\langle 1 \rangle}}{A}}$$

$\sigma^T = \begin{pmatrix} 4.619 \times 10^{6} & -9.238 \times 10^{6} & 0 \times 10^{0} & 4.619 \times 10^{6} & -9.238 \times 10^{6} \end{pmatrix}$

buckling stresses

$$\sigma_{nurj} := \overrightarrow{\frac{-N^{\langle 2 \rangle}}{A}} \qquad n_{nurj} := \overrightarrow{\frac{\sigma_{nurj}}{\sigma - 0.1 \cdot 10^{-9}}}$$

lisäys nollasauvoja varten jakajassa

$\sigma_{nurj}^T = \begin{pmatrix} -8.981 \times 10^{6} & -2.245 \times 10^{6} & -2.994 \times 10^{6} & -8.981 \times 10^{6} & -2.245 \times 10^{6} \end{pmatrix}$

safety for buckling

(negative values for tension)

$n_{nurj}^T = \begin{pmatrix} -1.944 & 0.243 & 2.994 \times 10^{16} & -1.944 & 0.243 \end{pmatrix}$

Appendix 4/2

$$F = \begin{pmatrix} -2.309\times10^{3} & 2.309\times10^{3} & 0 & -2.309\times10^{3} & -2.309\times10^{3} \\ 0 & 4\times10^{3} & 0 & 0 & 4\times10^{3} \\ 2.309\times10^{3} & -2.309\times10^{3} & 0 & 2.309\times10^{3} & 2.309\times10^{3} \\ 0 & -4\times10^{3} & 0 & 0 & -4\times10^{3} \end{pmatrix}$$

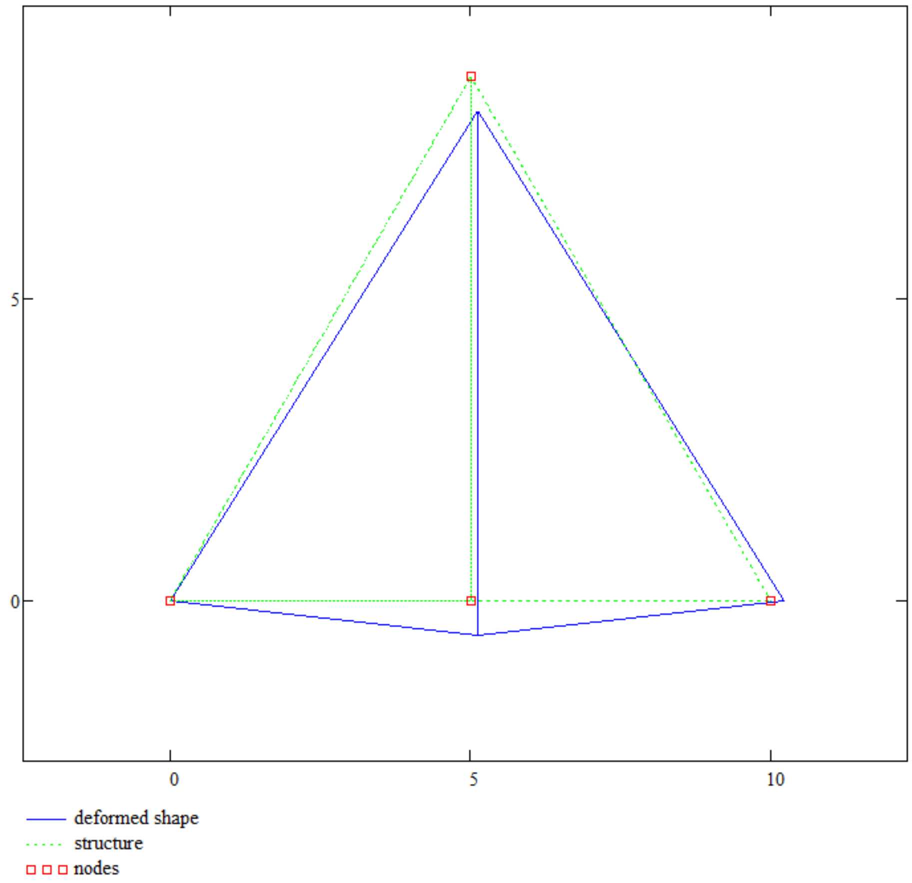
solmuissa vaikuttavat voimat

$$T^{T} = \begin{pmatrix} -4.547\times10^{-13} & 0 & 4.547\times10^{-13} & 0 \\ 4\times10^{3} & 0 & 4\times10^{3} & -8\times10^{3} \end{pmatrix}$$

solmu 1 solmu 2 jne .

Displacements scale := 100 **choose the scale factor**

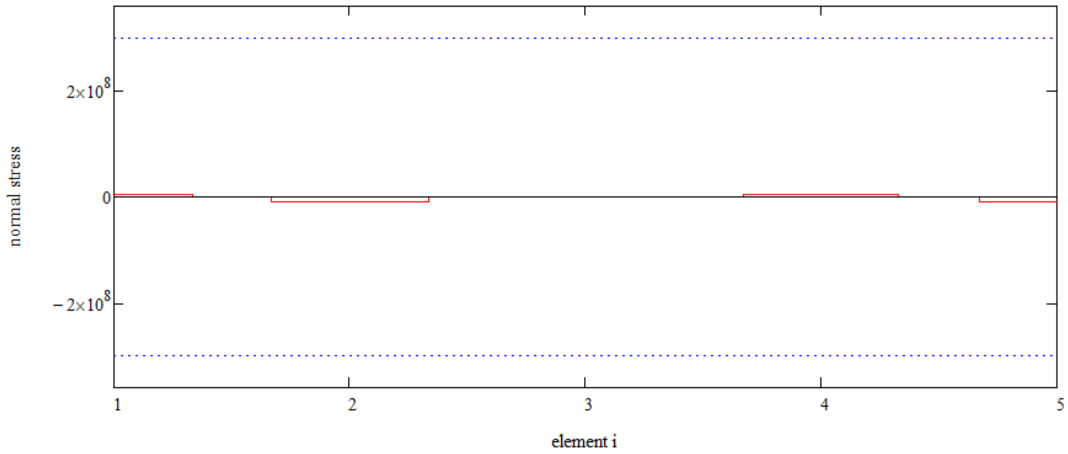
displacements



Results

stresses

$\sigma_{sall} = 300 \times 10^{6}$

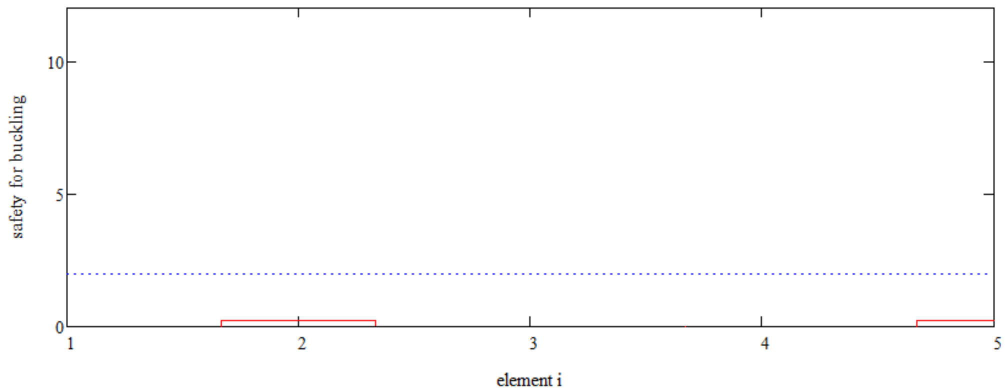


stresses

$$\sigma^{T} = \begin{pmatrix} 4.619 \times 10^{6} & -9.238 \times 10^{6} & 0 \times 10^{0} & 4.619 \times 10^{6} & -9.238 \times 10^{6} \end{pmatrix}$$

safety for buckling

scale for y - coord $yr := 12$



safety for buckling (Euler II) (negative on tension)

$$n_{nurj}{}^{T} = \begin{pmatrix} -1.944 & 0.243 & 2.994 \times 10^{16} & -1.944 & 0.243 \end{pmatrix}$$

displacements

$$U^{T} = \begin{pmatrix} 109.975 \times 10^{-6} & -571.429 \times 10^{-6} & 219.949 \times 10^{-6} & 109.975 \times 10^{-6} & -571.429 \times 10^{-6} \end{pmatrix}$$

Appendix 6/2

$$u^T = \begin{pmatrix} 0 & 0 & 1.1\times10^{-4} & 1.1\times10^{-4} & 2.199\times10^{-4} \\ 0 & 0 & -5.714\times10^{-4} & -5.714\times10^{-4} & 0 \\ 1.1\times10^{-4} & 1.1\times10^{-4} & 1.1\times10^{-4} & 2.199\times10^{-4} & 1.1\times10^{-4} \\ -5.714\times10^{-4} & -5.714\times10^{-4} & -5.714\times10^{-4} & 0 & -5.714\times10^{-4} \end{pmatrix}$$

x direc / y direc — start node; x dir / y dir — end node

normal forces

$$N^{\langle 1\rangle T} = \begin{pmatrix} 2.309\times10^{3} & -4.619\times10^{3} & 0\times10^{0} & 2.309\times10^{3} & -4.619\times10^{3} \end{pmatrix}$$

element forces

$$F = \begin{pmatrix} -2.309\times10^{3} & 2.309\times10^{3} & 0 & -2.309\times10^{3} & -2.309\times10^{3} \\ 0 & 4\times10^{3} & 0 & 0 & 4\times10^{3} \\ 2.309\times10^{3} & -2.309\times10^{3} & 0 & 2.309\times10^{3} & 2.309\times10^{3} \\ 0 & -4\times10^{3} & 0 & 0 & -4\times10^{3} \end{pmatrix}$$

x direc / y direc — start node; x dir / y dir — end node

Loading for nodes

$$T^T = \begin{pmatrix} -454.747\times10^{-15} & 0\times10^{0} & 454.747\times10^{-15} & 0\times10^{0} \\ 4\times10^{3} & 0\times10^{0} & 4\times10^{3} & -8\times10^{3} \end{pmatrix}$$

compare

support

$$Tuenta^T = \begin{pmatrix} 1 & 0 & 0 & 0 \\ 1 & 0 & 1 & 0 \end{pmatrix}$$

nodal forces

$$Voimat^T = \begin{pmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -8\times10^{3} \end{pmatrix}$$

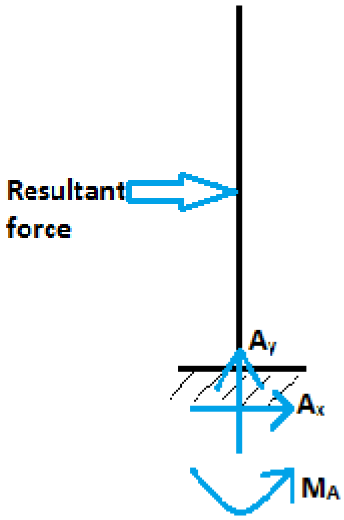
Appendix 3

Column base calculations

Equations of equilibrium

$$\Sigma f(x) = 0 \qquad \Sigma f(y) = 0 \qquad \Sigma M = 0$$

Free body diagram

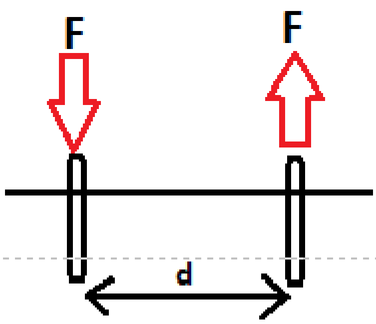


$UDL = 4029.48 \frac{N}{m}$ $\qquad l := 5.838m$

$$R := 4029.48 \frac{N}{m} \cdot 5.838m = 23.524 \times 10^3 N$$

$\Sigma M_A = -\left(R \cdot \frac{l}{2}\right) + M_A$ $\qquad M_A := R \cdot \frac{l}{2} = 68.667 \times 10^3 J$

Details at the base to find bolt forces



$d := 180mm$

$F \cdot d = M_A$

$$F := \frac{M_A}{d} = 381.483 \times 10^3 N$$

Two bolts on each side

$$\frac{F}{2} = 190.741 \times 10^3 N$$

Appendix 2/3

Stress calculation

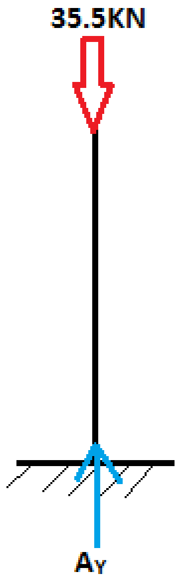
Axial stress = Force/Area

$$\text{Area} := \frac{\pi\cdot(28\text{mm})^2}{4} = 6.158\times10^{-4}\,\text{m}^2 \qquad A := 6.158\times10^{-4}\,\text{m}^2$$

$$A_{stress} := \frac{\frac{F}{2}}{A} = 3.097\times10^{8}\,\text{Pa}$$

Axial stress on each bolt will experience 309MPa

Effect of point load on bolts



$$\Sigma F_y = 0$$

$$A_y := 35500\text{N}$$

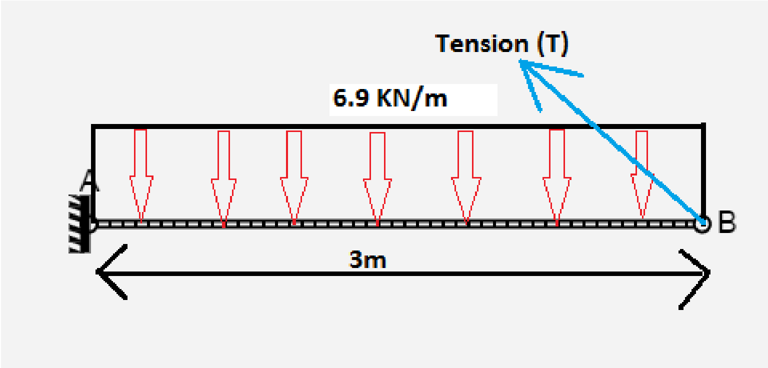
$$A_{stress2} := \frac{A_y}{A} = 5.765\times10^{7}\,\text{Pa}$$

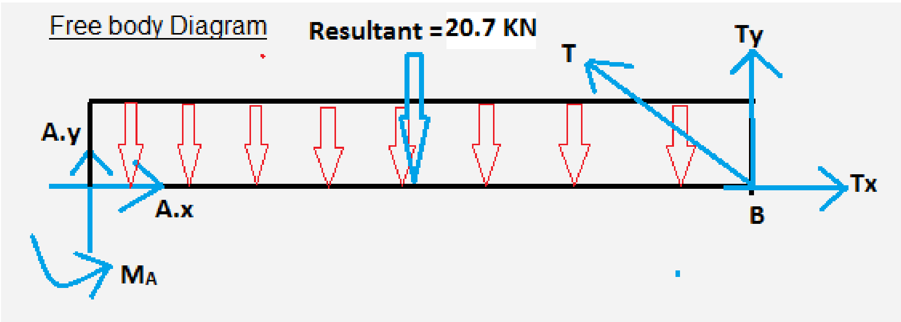
Axial stress of the bolts is reduced by approximately 57.65MPa

Appendix 4

Office beam design calculations

Simplified model





$\Sigma f(x) = 0 \qquad \Sigma f(y) = 0 \qquad \Sigma M = 0$

Angle between T = θ

$\theta := 45$

Equations of equilibrium

$\Sigma F_x = 0 \qquad \Sigma F_y = 0 \qquad \Sigma M = 0$

Statics calculation

$$\Sigma M_A = 0$$

$$T_y \cdot 3m - 20.7kN \cdot 1.5m = 0 \text{ solve}, T_y \rightarrow 10.35 \cdot kN$$

$$T_y := 10.35 \cdot kN$$

Equations of equilibrium

$\Sigma F_x = 0 \qquad \Sigma F_y = 0 \qquad \Sigma M = 0$

Statics calculation

$$\Sigma M_A = 0$$

$$T_y \cdot 3m - 20.7kN \cdot 1.5m = 0 \text{ solve}, T_y \rightarrow 10.35 \cdot kN$$

$$T_y := 10.35 \cdot kN$$

$$\Sigma F_y = 0$$

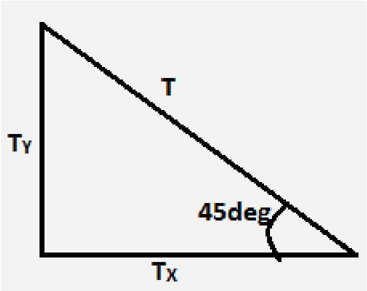
$$T_y + A_y - 20.7kN = 0 \text{ solve}, A_y \rightarrow 10.35 \cdot kN$$

$$A_y := 10.35 \cdot kN$$

$$\Sigma F_x = 0$$

$$T_x + A_x = 0$$

Appendix 2/4



$\sin(45deg) = 0.707 \quad \cos(45.deg) = 0.707$

$\sin(45deg) = \frac{T_y}{T} \quad 0.707 = \frac{T_y}{T} \text{ solve}, T \rightarrow 14.64 \cdot kN$

$T := 14.64kN$

$\cos(45) = \frac{T_x}{T} \quad 0.707 = \frac{T_x}{T} \text{ solve}, T_x \rightarrow 10.35 \cdot kN$

$T_x := 10.35kN$

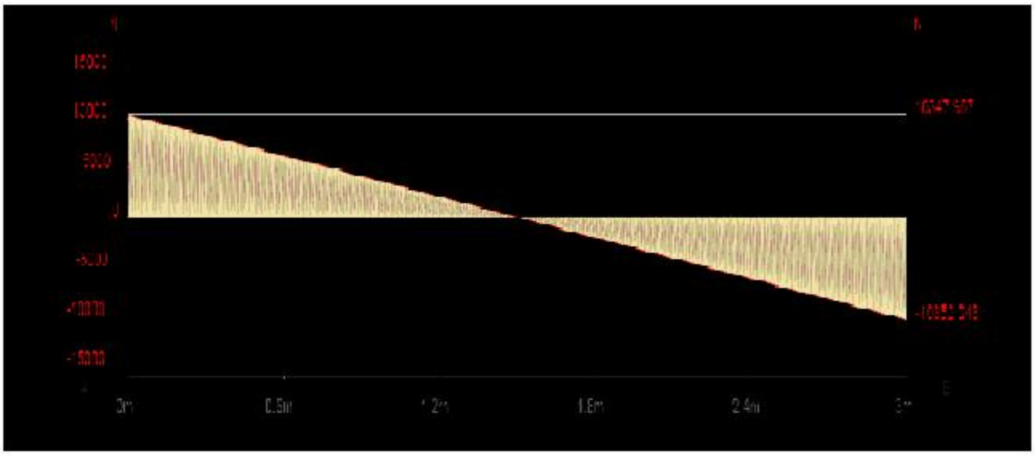
$\Sigma F_x = 0$

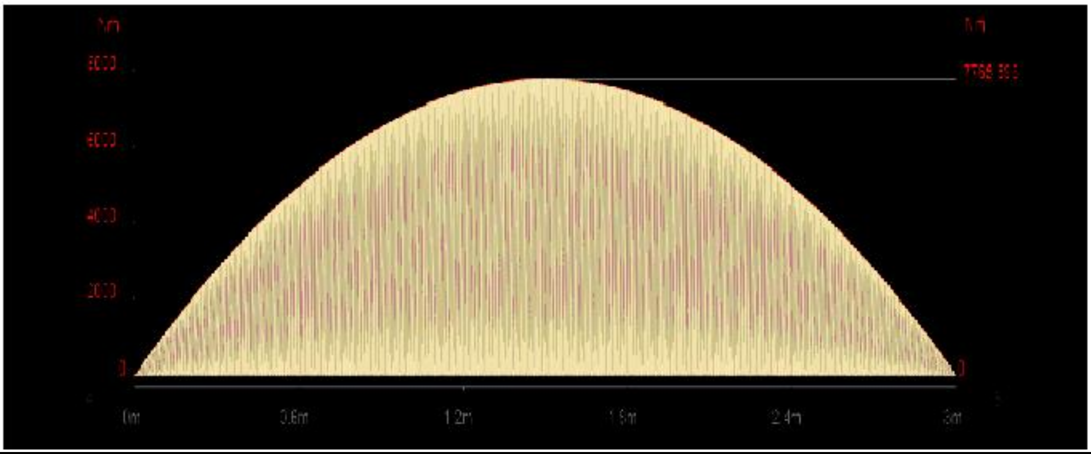
$T_x + A_x = 0 \text{ solve}, A_x \rightarrow -10.35 \cdot kN$

$A_x := -10.35kN$

Appendix 3/4

Shear force and moment diagram





Beam selection

$$\sigma_{max} = \frac{M_{max} \cdot y}{I}$$

Z = Section modulus Z = I/y

$$\sigma_{max} = \frac{M_{max}}{Z}$$

$$\sigma_{max} \leq \sigma_{all} = \frac{355}{2} MPa$$

$M_{max} := 7.765 kN \cdot m$ $\sigma_{max} := 177.5 MPa$

$$Z := \frac{M_{max}}{\sigma_{max}} = 4.375 \times 10^{-5} m^3$$

Needed section modulus = $8.746 \times 10^{-5} m^3$

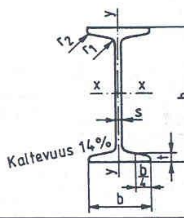
Appendix 5

Beam selection table

KUUMAVALSSATUT KAPEAT I-TANGOT

SFS 2028 ja DIN 1025 mukaan

I = neliömomentti
W = taivutusvastus
i = hitaussäde = $\sqrt{I/A}$
A = poikkipinnan ala
m_l = pituusmassa; ϱ = 7850 kg/m³
U = profiilin koko vaippapinta
S_x = poikkipinnan puolikkaan staattinen momentti
$s_x = I_x/S_x$ = veto- ja puristusjännityksen resultanttien etäisyys



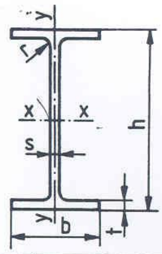
264

Tunnus I	h mm	b mm	s mm	t mm	r_1 mm	r_2 mm	A 10^3mm^2	m_l kg/m	U m^2/m	x-x I_x 10^6mm^4	x-x W_x 10^3mm^3	x-x i_x mm	y-y I_y 10^6mm^4	y-y W_y 10^3mm^3	y-y i_y mm	S_x 10^3mm^3	s_x mm
80	80	42	3,9	5,9	3,9	2,3	0,757	5,94	0,304	0,778	19,5	32,0	0,0629	3,00	9,1	11,4	68,4
100	100	50	4,5	6,8	4,5	2,7	1,06	8,34	0,370	1,71	34,2	40,1	0,122	4,88	10,7	19,9	85,7
120	120	58	5,1	7,7	5,1	3,1	1,42	11,1	0,439	3,28	54,7	48,1	0,215	7,41	12,3	31,8	103
140	140	66	5,7	8.6	5,7	3,4	1,82	14,3	0,502	5,73	81,9	56,1	0,352	10,7	14,0	47,7	120
160	160	74	6,3	9,5	6,3	3,8	2,28	17,9	0,575	9,35	117	64,0	0,547	14,8	15,5	68,0	137
180	180	82	6,9	10,4	6,9	4,1	2,79	21,9	0,640	14,5	161	72,0	0,813	19,8	17,1	93,4	155
200	200	90	7,5	11,3	7,5	4,5	3,34	26,2	0,709	21,4	214	80,0	1,17	26,0	18,7	125	172
220	220	98	8,1	12,2	8,1	4,9	3,95	31,1	0,775	30,6	278	88,0	1,62	33,1	20,2	162	189
240	240	106	8,7	13,1	8,7	5,2	4,61	36,2	0,844	42,5	354	95,9	2,21	41,7	22,0	206	206
260	260	113	9,4	14,1	9,4	5,6	5,33	41,9	0,906	57,4	442	104	2,88	51,0	23,2	257	223
280	280	119	10,1	15,2	10,1	6,1	6,10	47,9	0,966	75,9	542	111	3,64	61,2	24,5	316	240
300	300	125	10,8	16,2	10,8	6,5	6,90	54,2	1,03	98,0	653	119	4,51	72,2	25,6	381	257
320	320	131	11,5	17,3	11,5	6,9	7,77	61,0	1,09	125,1	782	127	5,55	84,7	26,7	457	274
340	340	137	12,2	18,3	12,2	7,3	8,67	68,0	1,15	157,0	923	135	6,74	98,4	28,0	540	291
360	360	143	13,0	19,5	13,0	7,8	9,70	76,1	1,21	196,1	1090	142	8,18	114	29,0	638	307
380	380	149	13,7	20,5	13,7	8,2	10,7	84,0	1,27	240,1	1260	150	9,75	131	30,2	741	324
400	400	155	14,4	21,6	14,4	8,6	11,8	92,4	1,33	292,1	1460	157	11,60	149	31,3	857	341
425	425	163	15,3	23,0	15,3	9,2	13,2	104	1,41	369,7	1740	167	14,40	176	33,0	1020	362
450	450	170	16,2	24,3	16,2	9,7	14,7	115	1,48	458,5	2040	177	17,30	203	34,3	1200	383
475	475	178	17,1	25,6	17,1	10,3	16,3	128	1,55	564,8	2380	186	20,90	235	36,0	1400	404
500	500	185	18,0	27,0	18,0	10,8	17,9	141	1,63	687,4	2750	196	24,80	268	37,2	1620	424
550	550	200	19,0	30,0	19,0	11,9	21,2	166	1,80	991,8	3610	216	34,90	349	40,2	2120	468
600	600	215	21,6	32,4	21,6	13,0	25,4	199	1,92	1390,0	4630	234	46,70	434	43,0	2730	509

KUUMAVALSSATUT PUOLILEVEÄT I-TANGOT IPE

SFS 2029 JA DIN 1025 MUKAAN

I = neliömomentti
W = taivutusvastus
i = hitaussäde = $\sqrt{I/A}$
A = poikkipinnan ala
m_l = pituusmassa; ϱ = 7850 kg/m³
U = profiilin koko vaippapinta
S_x = poikkipinnan puolikkaan staattinen momentti
$s_x = I_x/S_x$ = veto- ja puristusjännityksen resultanttien etäisyys



265

Tunnus	h mm	b mm	s mm	t mm	r mm	A $10^3 mm^2$	m_l kg/m	U m^2/m	x-x I_x $10^6 mm^4$	x-x W_x $10^3 mm^3$	x-x i_x mm	y-y I_y $10^6 mm^4$	y-y W_y $10^3 mm^3$	y-y i_y mm	S_x $10^3 mm^3$	s_x mm
IPE 80	80	46	3,8	5,2	5	0,764	6,00	0,328	0,801	20,0	32,4	0,0849	3,69	10,5	11,6	69,0
IPE 100	100	55	4,1	5,7	7	1,03	8,10	0,400	1,71	34,2	40,7	0,159	5,79	12,4	19,7	86,8
IPE 120	120	64	4,4	6,3	7	1,32	10,40	0,475	3,18	53,0	49,0	0,277	8,65	14,5	30,4	105
IPE 140	140	73	4,7	6,9	7	1,64	12,90	0,551	5,41	77,3	57,4	0,449	12,3	16,5	44,2	123
IPE 160	160	82	5,0	7,4	9	2,01	15,80	0,623	8,69	109	65,8	0,683	16,7	18,4	61,9	140
IPE 180	180	91	5,3	8,0	9	2,39	18,80	0,698	13,2	146	74,2	1,01	22,2	20,5	83,2	158
IPE 200	200	100	5,6	8,5	12	2,85	22,40	0,768	19,4	194	82,6	1,42	28,5	22,4	110	176
IPE 220	220	110	5,9	9,2	12	3,34	26,20	0,848	27,7	252	91,1	2,05	37,3	24,8	143	194
IPE 240	240	120	6,2	9,8	15	3,91	30,70	0,922	38,9	324	99,7	2,84	47,3	26,9	183	212
IPE 270	270	135	6,6	10,2	15	4,59	36,10	1,04	57,9	429	112	4,20	62,2	30,2	242	239
IPE 300	300	150	7,1	10,7	15	5,38	42,20	1,16	83,6	557	125	6,04	80,5	33,5	314	266
IPE 330	330	160	7,5	11,5	18	6,26	49,10	1,25	117,7	713	137	7,88	98,5	35,5	402	293
IPE 360	360	170	8,0	12,7	18	7,27	57,10	1,35	162,7	904	150	10,4	123	37,9	510	319
IPE 400	400	180	8,6	13,5	21	8,45	66,30	1,47	231,3	1160	165	13,2	146	39,5	654	354
IPE 450	450	190	9,4	14,6	21	9,88	77,60	1,61	337,4	1500	185	16,8	176	41,2	851	397
IPE 500	500	200	10,2	16,0	21	11,6	90,70	1,74	482,0	1930	204	21,4	214	43,1	1100	439
IPE 550	550	210	11,1	17,2	24	13,4	106,00	1,88	671,2	2440	223	26,7	254	44,5	1390	482
IPE 600	600	220	12,0	19,0	24	15,6	122,00	2,01	920,8	3070	243	33,9	308	46,6	1760	524

Appendix 6

Tension rod calculation and selection

Force = Tension in rod

$$\sigma_{axial} \leq \sigma_{all} = \frac{355}{2} MPa$$

$$\sigma_{axial} = \frac{Force}{Area} \qquad T := 14640N$$

$$\sigma_{axial} := 177.5 \frac{N}{mm^2}$$

$$\sigma_{axial} = \frac{T}{Area} \text{ solve}, Area \rightarrow 82.479E+000 \cdot mm^{2E+000}$$

$$A := 82.47mm^2$$

$$A = \frac{\pi \cdot d^2}{4} \text{ solve}, d \rightarrow \begin{pmatrix} \frac{\sqrt{105004065254308866727} \cdot mm}{1000000000} \\ -\frac{\sqrt{105004065254308866727} \cdot mm}{1000000000} \end{pmatrix} = \begin{pmatrix} 0.01 \\ -0.01 \end{pmatrix} m$$

required diameter = 10mm