CT3000: Bachelor Thesis Report, Izik Shalom (4048180)

Computations of stresses with volume-elements in rectangular and HE sections

Supervisors: dr. ir. P.C.J. Hoogenboom en Ir. R. Abspoel

June 2013

Preface

This Bachelor thesis contains the results of a nine weeks study into the modelling of rectangular and HE sections in Finite Element software with 20-nodes volume-elements. This study is part of obtaining the degree of Bachelor of Science at the faculty of Civil Engineering of Delft University of Technology and was carried out under the supervisors dr. ir. P.C.J. Hoogenboom and Ir. R. Abspoel. I would like to express my gratitude towards my supervisors, in particularly Mr. Hoogenboom , for supplying the facilities during the study, for helping to understand fundamental issues regarding Finite Element Analysis and of course for listening and guiding me through the problems that occurred during the study. I would also like to express my gratitude towards Mr. Abspoel for directing me to the vital issues concerning the HE section.

For questions, feedback or comments don't hesitate to contact me.

Izik Shalom
Izik_shalom@hotmail.com

Delft
June 2013

.

Abstract

The purpose of this study is to calculate the critical stresses in a beam by using the finite element analysis. The study focuses on optimization of the number of 20-nodes volume-elements required to model rectangular beam and HE beam with a high accuracy. The optimal number of elements are directly derived through looking at the nodal forces, which follow from a linear elastic finite element analysis. The finite element method is a mathematical technique that splits up complex problems into greater number of simple problems which can approximate the solution. The study was divided into two parts, the first part concerning rectangular beam subjected to torsion differs considerably from the second part, HE-section subjected to bending moment, shear and torsion. In the first part of the study, a parameters study has been conducted, where approximately 300 models have been generated. This parameters study has been conducted in the search for the influence of every parameter on the accuracy of the solution. Evaluating the assembled results of this part gave the accuracy for every element configurations with respect to the computation time. In the second part, involving HE300A section a different working method has been used: The adaptive mesh refinement. In this method the models didn't have a uniform mesh but standard coarse mesh with additional finer submesh regarding the magnitude of the stresses at that point. The highlights of this study are the output of mesh design tables for each loading case for rectangular and HE sections. Structural engineers may be using those tables to guide them through modelling in ANSYS with certain efficiency and accuracy. Modelling beams with FEM led to the following main conclusions:

- Modelling a HE-beam subjected to shear force or bending moment with a maximal of 1% can be done with relative few elements;
- Stresses caused by torsion moment will need the finest mesh in order to reach an accuracy of less than 10%;
- Therefore, while modelling a beam subjected to shear, bending a torsion load, the mesh that belongs to the torsion moment should be applied;
- Not only the total number of elements is important, but also the ratio between the elements in the height and in the width;
- Transforming the peak shear stresses in the HE-section to the scalar quantity of Von-Mises stresses are a realistic representation of the accuracy of the stated mesh density.

Contents

1. Introduction

This Bachelor thesis explains the method developed by engineers to analyze the stresses in a structure and its principles. It contains background information about the numerical finite element calculation and describes briefly the basics of the structural mechanics theories. The finite element analysis (FEA) is a way of getting a numerical solution to a problem that is usually being solved analytically by using formulas. The FE analysis is cutting the structure into several pieces and calculating for every element the strain, stress and the displacements with a high accuracy. However analyzing a beam or complete structure with FE method requires a significant computing capacity from the computer. In this Bachelor thesis we'll find the optimal number of volume-elements required to analyse a beam with high precision and with effective computer computation.

1.1 Background

The Finite Element method as known today was provided in 1973 with the publication by Strang and Fix. The method has since been generalized for numerical modeling of physical systems like in structural engineering. By the late 1980s the software of FEA was available. In the recent years FEA has become popular, and form now a big industry. Numerical solutions of stress problems can now be obtained by using FEA with a powerful computer simulation. This Bachelor thesis will use the finite element computer-program ANSYS given in *figure 1.1.*

Figure 1-1: ANSYS version 13.0

ANSYS is engineering simulation software with a variety of tools. One of the tools being used in this research is the simulation for structural physics that simulate static problems. The software will solve the structure by generating a mesh that divides the structure into small elements. For every element ANSYS computes the stresses and deflection values with 3D numerical equations. These results can be presented in a list or directly plotted on the structure showing the deflections and the stresses contours.

1.2 Problem formulation

A 3D beam can be modeled by dividing it into volume-elements. The shape, size, amount and configuration of elements have to be chosen carefully such that the original body is simulated as closely as possible without decreasing computation effectiveness and accuracy.

The problem in the current working method within the collaboration of the structural engineer and the architect comes from inefficient working procedure. For example an Architect designs a structure in AutoCAD software. The design is sent forward to the constructor for further static analysis of the structure. The problem occurs because the constructor needs to draw the structure again in static analysis's software such as Matrixframe as line element and afterwards to determine the sections properties of each line element. A straight-forward solution is to solve the structure with the ANSYS program by meshing the existing AutoCAD design. The density of the mesh determines on one hand the accuracy of the solution but on the other hand the calculation time and the memory use required from the computer. In this case it is preferred to use much volume elements with small dimensions instead of a small number of elements with larger dimensions. (Figure 1-2, Figure 1-3)

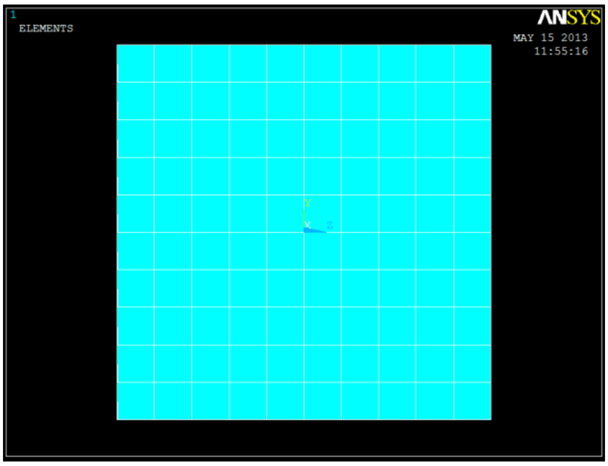


Figure 1-2: 10 by 10 grid

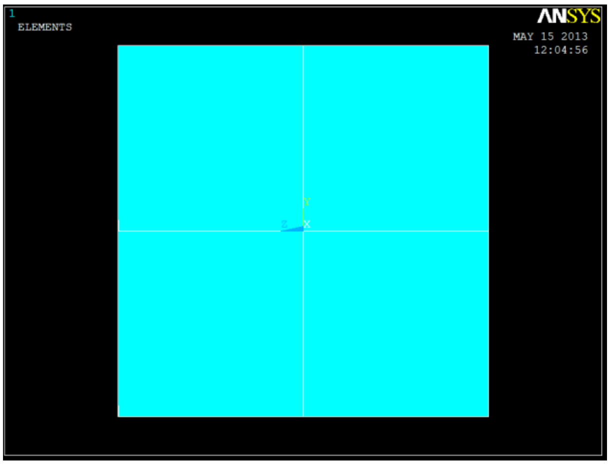


Figure 1-3: 2 by 2 grid

Although the left model is favorable, it is inefficient to use such a dens mesh for every section type or for every loading case. Therefore, for a stated section type an efficient mesh density and size is required. However, we know that on a structure a variety of loads are acting. For every loading case like bending moment or shear load the stresses in the structure behave differently. The ideal mesh will be the governing mesh that satisfies each loading case. For instance a mesh for HE-section subjected to shear and bending moment will require a different mesh than rectangular cross-section subjected to shear, torsion and bending load. The problems can be summarized into four sections:

Problem 1: The current working method between the engineer and the architect is cost and time inefficient.

Problem 2: Meshing an existing design will result in deviations of the values of the stresses.

Problem 3: A fine mesh with small spacing between the nodes is accurate but time inefficient while a course mesh with larger spacing between the nodes is less accurate but time effective. How the ideal mesh can be defined for different loading cases?

Problem 4: For every loading type, for torsion, shear and bending, a different mesh is required. Which mesh is governing?

Now, when the problems are described, the evaluation of the solution is necessary. How do we know if the chosen mesh is safe? Or in other words: What should be the deviation of the numerical solution from the exact solution in order to obtain a safe structure design that also meets the current design standards?

1.3 The Finite Element Analysis

The research problem is divided into two sub-problems:

1.3.1 Rectangular cross-section

To model a *rectangular* beam in ANSYS, it is divided into volume-elements. Use a 20-nodes element like a Quadratic *Hexahedron.* Apply torsion load on the beam, calculate the largest stresses and compare these with the exact stresses values. Change the number of elements in the width and in the height of the rectangular cross-section. Determine the minimal number of volume-elements such that the torsion stresses in the critical cross-section can be calculated with a deviation of 1%-10%. Exhibit the deviation of the stresses in a table as a function of the number of elements in the width and the height. Confirm the current Thumb-rule of 2 elements in the width and 5 elements in the height are correct. (Figure 1-4)

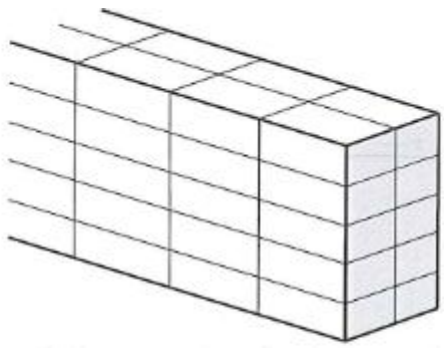


Figure 1-4: Element configuration for an error of 1%

1.3.2 Section HE300A

In the industry a standard HE-section is often being used due to the high strength of the steel and the material efficient design it has in comparison to a massive cross-section. For a HE300A section a FEA model can be also created from a mesh of 20-nodes volume-elements. However, the distribution of these elements in the web and the flange is yet unknown. The purpose is to find the minimal amount of volume-elements in the web and the flanges with a deviation of 1%-10% under a torsion, shear and bending moment load. (Figure 1-5)

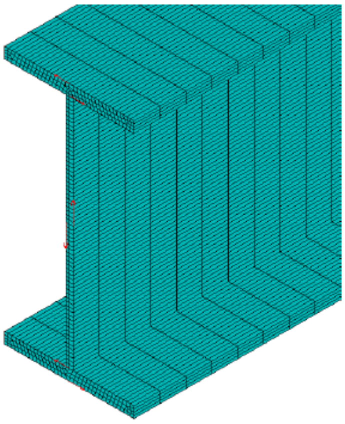


Figure 1-5: FEA model of I-section

1.3 Problem statement

How can a beam, loaded under shear, moment, normal and torsion force, be divided into volume-elements as efficient as possible with sufficient accuracy of the results?

1.4 Aim of the research

With the aid of this last paragraph it is now possible to determine the aim of this research.

Primary objective: Design a set of Thumb-rules for the necessary amount of volume-elements for an accurate calculation of the stresses.

Secondary objective: Finding for the range of the accuracy for each loading case

Tertiary objective: Finding the rate of convergence of the mesh density when its size goes to zero for rectangular cross-section subjected to torsion load and for HE300A section subjected to torsion, shear and bending moment.

1.5 ANSYS software

Modelling the beam with ANSYS software is done by using the Finite Element Analysis. Working with ANSYS software suggests three working methods: The first method is by means of the *graphical user interdace* (GUI). This method is an interactive working mode where the user enters data about the structure in the main menu of ANSYS which contains the primary ANSYS functions. (*Figure 1-6*)

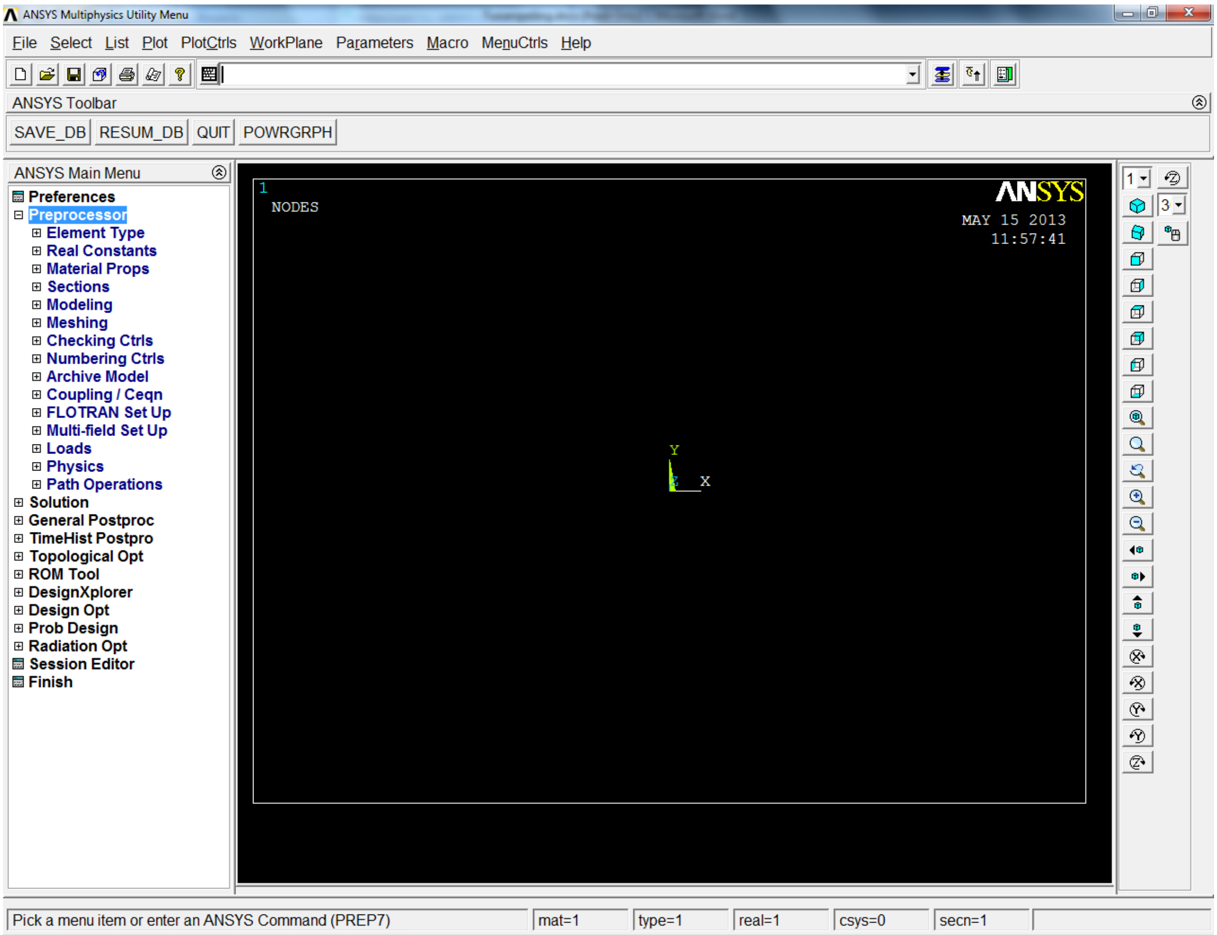


Figure 1-6: ANSYS GUI with the menu on the left side

In this working mode every step of the user represents a program command. From this menu the vast majority of modeling commands are issued. The user can save his work as a txt file containing all of the used commands.

The second working mode is by means of command files. The user writes a text file containing commands at the system level. ANSYS can import the text file and runs it to perform the required analysis. (*Figure 1-7*)

```
ax = 3000                     ! Lenght                      mm
ay = 400                      ! Height                      mm
az = 400                      ! Width                       mm
factor1 = 0.237

E  = 210e3                    ! Young's modulus             N/mm^2
nu = 0.1                      ! Poisson's ratio             -
EZ =  2               ! Number of elements in width
EY = 2                ! Number of elements in height
dEX = 25
M  = 20e6                     ! Force                       Nmm
Iz = (1/12)*az*ay*ay*ay       ! Moment of inertia           mm^4

/PREP7
MPTEMP,,,,,,,,                ! Material: isotropic
MPTEMP,1,0
MPDATA,EX,1,,E
MPDATA,PRXY,1,,nu
ET,1,SOLID186                 ! Element type: 8 node

nz=2*EZ+1                     ! Number of nodes in x, y and z direction
ny=2*EY+1
s=ax
nx=1
*DOWHILE,s
  s=s-2*dEX
  nx=nx+4
*ENDDO
EX=(nx-1)/2

nodenr=1
*DO,i,1,nx                    ! Put nodes
  x=(i-1)*(ax/(nx-1))
  *DO,j,1,ny
    y=(ny-j)*(ay/(ny-1))-ay/2
    *DO,k,1,nz
      z=(nz-k)*(az/(nz-1))-az/2
      N,nodenr,x,y,z,,,
      nodenr=nodenr+1
    *ENDDO
  *ENDDO
*ENDDO
*DO,i,1,nx
  *DO,j,2,ny-1,2
    *DO,k,2,nz-1,2
      NDELE,(i-1)*ny*nz+(j-1)*nz+k
    *ENDDO
  *ENDDO
*ENDDO
*DO,i,2,nx-1,2
  *DO,j,1,ny,2
    *DO,k,2,nz-1,2
      NDELE,(i-1)*ny*nz+(j-1)*nz+k
    *ENDDO
  *ENDDO
*ENDDO
*DO,i,2,nx-1,2
  *DO,j,2,ny-1,2
    *DO,k,1,nz,2
      NDELE,(i-1)*ny*nz+(j-1)*nz+k
    *ENDDO
  *ENDDO
*ENDDO

*DO,i,1,EX                    ! Put elements
  *DO,j,1,EY
    *DO,k,1,EZ
      E,(i-1)*2*ny*nz+j*2*nz+k*2+1,(i-1)*2*ny*nz+j*2*nz+k*2-1,i*2*ny*nz+j*2*nz+k*2-1,i*2*ny*nz+j*2*nz+k*2+1,(i-1)*2*ny*nz+(j-1)*2*nz+k*2+1,(i-1)*2*ny*nz
+(j-1)*2*nz+k*2-1,i*2*ny*nz+(j-1)*2*nz+k*2-1,i*2*ny*nz+(j-1)*2*nz+k*2+1
```

Figure 1-7: An example of ANSYS text script

The advantage of writing a script, the second working mode, is the ability of ANSYS to read the commands and solve the structure without the need of using the software toolbars. It is also the favorable working method for a parameter study/optimization study.

The last working mode and the most user friendly one is by means of the ANASYS Workingbench (*Figure 1-8*). A complete environment for geometry modeling, mesh manipulations and optimization which is integrated with CAD package. This working mode is easy to use due to a project workflow scheme.

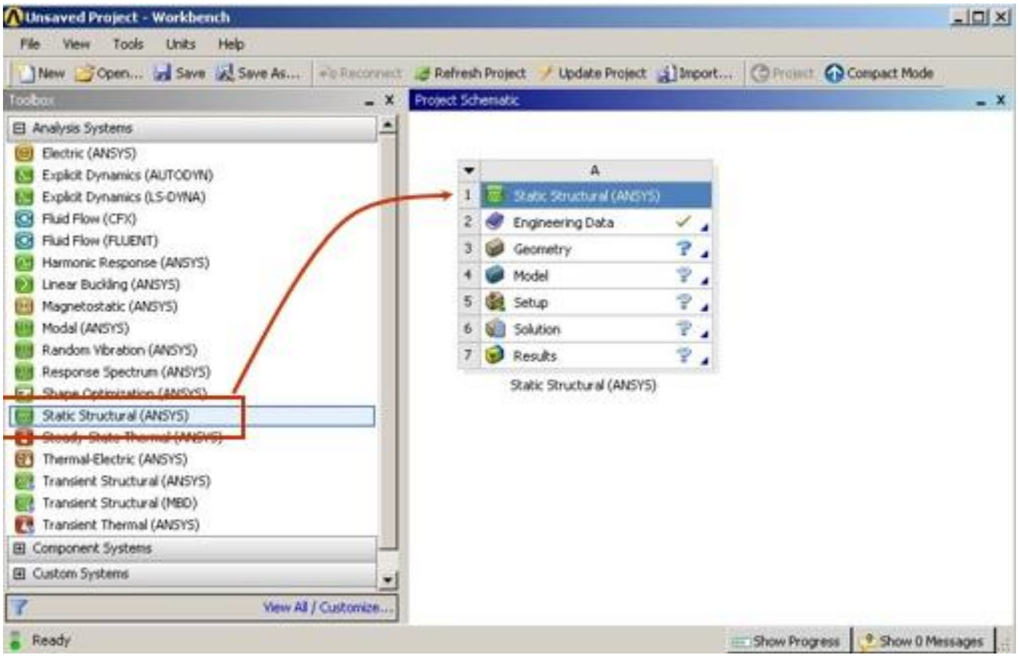


Figure 1-8: The Workingbench

In this bachelor thesis it is preferred to use the second working mode to simulate the rectangular beam and the first working method to simulate the HE-section. In this case the script will automatically sketch and solve the rectangular beam with regard to the mesh density that will be changed in every solving cycle. In such a way an optimum study can be easily conducted. For example, consider a rectangular cross-section of a beam with the following dimensions: L=2000mm h=200mm and b=300mm. The beam is loaded. And we want to know which mesh will simulate the stresses in the beam as accurate as possible. Therefore we'll start with a rough mesh of two by two cross-sectional elements and twenty elements in the length of the beam. ANSYS computes the stresses in every element of the beam. Comparing those stresses with the analytical exact stresses gives the accuracy of the chosen mesh.

1.6 Thesis outline

This thesis begins in $\oint 1$ where detailed introduction of the problem, thesis goelm, working method and fundumental knowledge over the finite element method in ANSYS software. The backround of this method, type of volume elements and the error obtained by it will be discussed in $\oint 2$. $\oint 3$ will subsequently discuss the hand-calculation solutions of considered beams of which the exact stresses values can be founded. In $\oint 4$ the rectangular beam properties will be described. Moreover, the FEA-beam will be considered with the relevant basics principles related to interpretation and understanding the results from ANSYS. Subsequently, the output of the optimization study will be given in $\oint 5$ where the influence of parameters variation will be summarized in meshing design tables. $\oint 6$ continues with an overview of the HE300A section and the working method will be considered during the study of this section. In $\oint 7$ $\oint 8$ $\oint 9$ the HE300A section will be subjected to bending moment, shear force and torsion moment. The results will be evaluated and summarized in meshing design table for each loading case. This thesis finishes in $\oint 10$ with a overview of the conclusions and recommendations.

2. Theory

The Numerical calculation with Finite Element Method is using volume elements to divide the structure into small pieces. A consequence of this approach is that all six possible stresses must be taken into account (three normal and three shear). The displacement field involves three possible components for the x, y and z direction $u\ \ v\ and\ \ w$. Typical 3D elements for solid structures are tetrahedral and hexahedron. Of course the number of elements and their shape vary with the type of the structure and it size. In this research linear analysis behaviour of the structure will be implemented. Initial stress due to temperature changes will be ignored in this study.

2.1 The numerical FEA

The FEA is applicable for the solution of equilibrium (static) problems. The FE method divides the structure into small pieces and for every small piece it computes the displacement and stresses. These elements are considered to be interconnected at specified nodal points. The variation of the displacement field can be approximated by a simple function. The function is used to represent the behaviour of the solution. The accuracy of the solution is obtained by increasing the order of the function polynomial. Therefore a polemical of infinite order corresponds to the exact solution. (Figure 2-1)

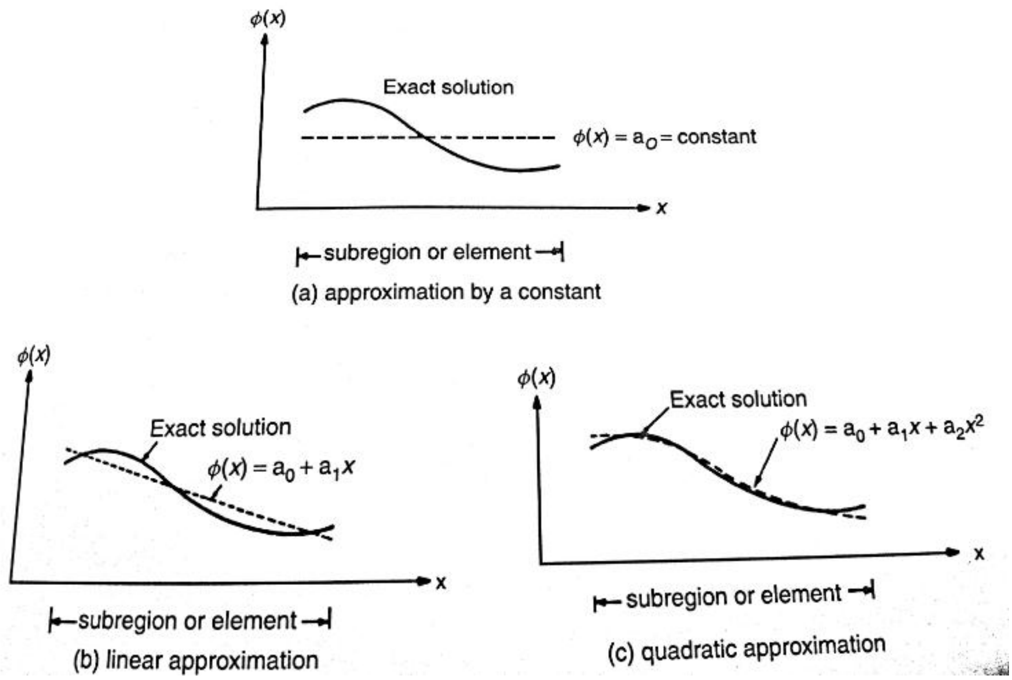


Figure 2-1: Approaching the exact solution [Rao, Singiresu S, The finite element method in the engineering]

We shall consider a cubic hexahedron, an element that is known as a "higher order element" with extra interior nodes besides the primary corner nodes and a rectangular prism with only corner nodes. (*Figure 2-1*)

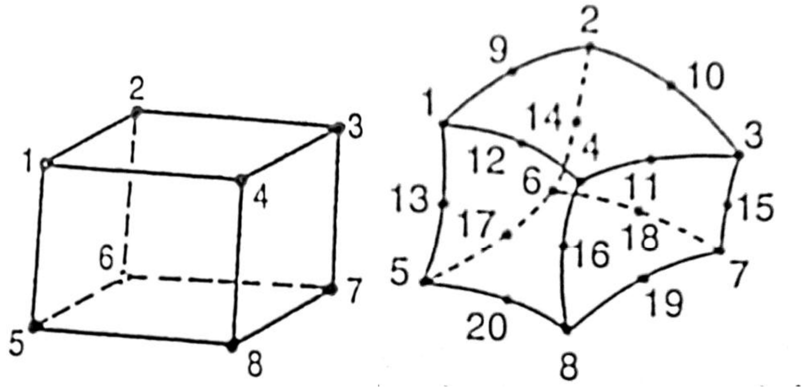


Figure 2-1: Solid185 (left) versus Solid186 (right), [Rao, Singiresu S, The finite element method in the engineering]

The boundary conditions of the displacement field are known as the Kinematical boundary conditions and the forces on the structure are taking into account the dynamical boundary conditions. For the unknown displacement field we assume a polynomial type of variation for the displacement field $N(x, y, z)$ in a three-dimensional linear cubic element, where $\alpha_1 \ldots . \alpha_m$ are the coefficients of the polynomial.

$$N(x, y, z) = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 z + \alpha_5 x^2 + \alpha_6 y^2 + \alpha_7 z^2 + \alpha_8 xy + \alpha_9 yz + \alpha_{10} xz + \alpha_{11} x^3 \ldots . + \alpha_{19} yz^2 + \alpha_{20} xyz$$

With:

N Displacement field in local coordinate system

α_m Coefficients of the polynomial

n The degree of the polynomial (For cubic n=3)

m Number of polynomial coefficient (for *Solid186* 20-nodes cubic is 20)

The number of the polynomial coefficients m should be equal to the number of nodal Degrees Of Freedom α. The nodal DOF is treated as unknown, with this nodal DOF as the displacement of every element that will be determent later. The next step in the solution procedure is to derive the strain-displacements relations with the nodal DOF as parameters. Afterwards we can derive according to the Hook's Law the stress-strain relations with the nodal DOF as parameters. The last phase is to substitute the stresses expressed in the nodal DOF in the equilibrium equation of every element.

The primary aim of any stress analysis is to find the distribution of the displacements and the stress under the stated loading and boundary condition. For a 3D problem the following basic equations need to be considered:

Type of equations	*In 3D problem*
Equilibrium equations	3
Stress-strain equations	6
Strain-displacements relations	6
Total number of equations	15

Table 2-1: Equations

The unknown quantities whose number is equal to the number of equations available, are given below:

Unknowns	*In 3D problems*
Displacements	u, v, w
Stress	$\sigma_{xx}, \sigma_{yy}, \sigma_{zz}$
Strains	$\varepsilon_{xx}, \varepsilon_{yy}, \varepsilon_{zz}, \varepsilon_{xy}, \varepsilon_{yz}, \varepsilon_{zx}$
Total number of unknown	15

Table 2-2: Unknowns

In practice we'll also have to satisfy additional equations such as a compatibility equation and boundary conditions. Those equations are essential in deriving the element solutions. In order to find the displacements of every cubic volume-element under static load we should first use the equilibrium equations.

2.1.1 Equilibrium equation

For a 3D problem we consider the stress acting on the cubic volume-element on the six faces of the cubic as given in *Figure 2-2*:

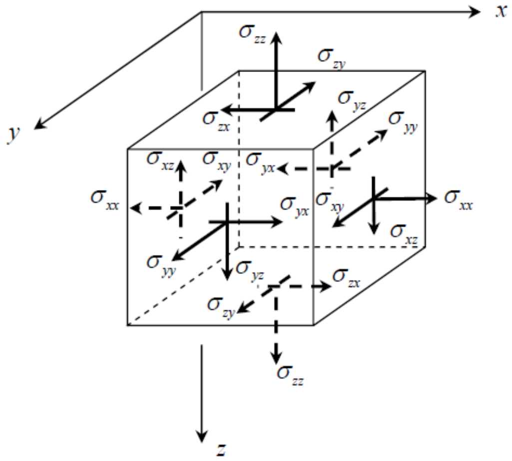


Figure 2-2: Axis definition

The equilibrium equations for this 3D body are:

$$\frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \sigma_{xy}}{\partial y} + \frac{\partial \sigma_{zx}}{\partial z} + \ \phi_x = 0$$

$$\frac{\partial \sigma_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \sigma_{yz}}{\partial z} + \ \phi_y = 0$$

$$\frac{\partial \sigma_{zx}}{\partial x} + \frac{\partial \sigma_{yz}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} + \ \phi_z = 0$$

2.1.2 Stress-strain relations

According to the constitutive linear theory of elasticity the Stress-Strain relations are also known as *Hooke's Law,* where $\bar{\sigma}$ and $\bar{\varepsilon}$ are vectors with 6 components, as appears in figure below:

$$\bar{\sigma} = E\bar{\varepsilon}$$

$$\bar{\sigma} = \begin{matrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{zx} \end{matrix} \qquad \bar{\varepsilon} = \begin{matrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \varepsilon_{xy} \\ \varepsilon_{yz} \\ \varepsilon_{zx} \end{matrix}$$

2.1.3 Stiffness matrix

The stiffness matrix K is derived from combining the governing equations, so the only unknown will be the displacements. This can be done by using Hook's constitutive equations to replace the stresses in the equilibrium equations by the strain, and then using the kinematic equations to replace the strain by the displacements.

The elastic stiffness matrix is expressed in constants such as the young's modulus E and Poisson's ratio v. The Poisson's ratio is the negative ratio of transverse to axial strain.

$$\bar{\sigma} = K\bar{\varepsilon}$$

$$v = -\frac{d\varepsilon_y}{d\varepsilon_x} = -\frac{d\varepsilon_z}{d\varepsilon_x}$$

$$\frac{1}{E}\begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\ 0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix}$$

Figure 2-3: The elastic stiffness matrix $\mathrm{K_e}$ ***[Wikipedia]***

The elastics stiffness matrix K_e in *figure 2-3* is necessary to determine the forces that are set up in the volume-element by the displacements of the nodes. With Gusse interpolation points we can approximate direct the displacement of every point within the element in terms of the nodal displacements. An example in one-dimension is presented in *Figure 2-4* where function $u(x)$ describes the nodal displacement and $N(X)$ describes the interpolation function.

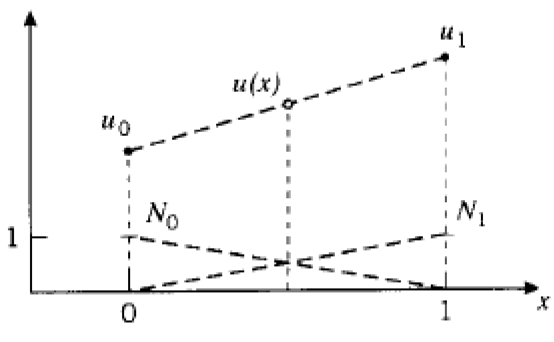


Figure 2-4: Interpolation in one dimension.), [Rao, Singiresu S, The finite element method in the engineering]

For a three dimensional cubic element there are $2 \times 2 \times 2$ integrations points. (*Figure 2-5*)

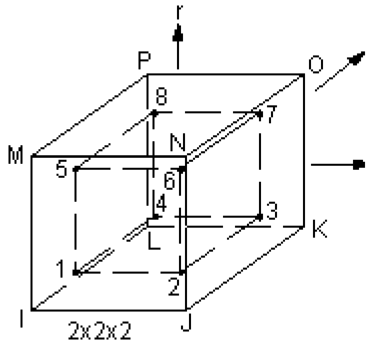


Figure 2-5: [Rao, Singiresu S, The finite element method in the engineering]

2.2 Solution procedure with FEA

Preliminary Decisions: The most common type of analysis in the civil engineering: Static analysis is defined. The type of volume element will be defined. In this rapport we'll work only with 20-nods volume-elements since they've been proved to be more accurate. In ANSYS it called *Solid186.*

Pre-processing: In this stage the model of the structure is entered into ANASYS. The geometry is divided into a number of sub-regions connected with each other at the nodes. The boundary conditions are specified by fixing the displacement of several nodes. In addition external load on the structure is added to the nodes (torsion, shear or bending moment load). Material properties are given to the volume-elements. The last step is to solve the model with the solver.

Post processing: Reviewing the results obtained by ANASYS. That includes viewing cross-sectional results and plotting data.

2.3 20-nods element

The *Solid186* element is an element that exhibits quadratic displacement behaviour. This element is defined by 20 nodes having three degrees of freedom per node: Translation in the x, y, and z, that is 60 DOF's per element. (*Figure 2-6*)

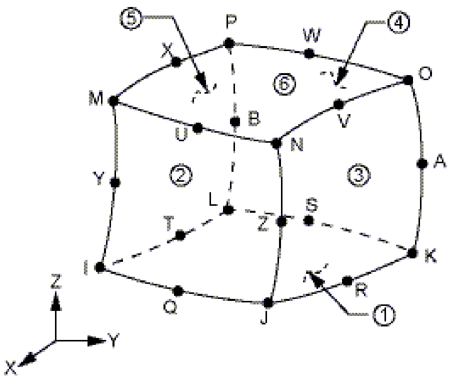


Figure 2-6: Solid186 , [Rao, Singiresu S, The finite element method in the engineering]

Because the *Solid186* has midside nodes, the displacement varies parabolically, rather than linearly along that edge. The 20-nodes element has an interesting trait, which is that the displacement at the midside node will always be greater than at the corner nodes. Therefore it is usually better to pick the critical point at the midside node. The solid186 exhibits quadratic displacement behaviour. For a 3D problem the following polynomial is given:

$$u_x(x,y,z) = D_1 + D_2x + D_3y + D_4z + D_5x^2 + D_6xy + D_7xz + D_8y^2 + D_9z^2 + D_{10}yz$$

Stresses components can be derived:

$$\varepsilon_{xx} = \frac{\partial u_x}{\partial x} \Rrightarrow \sigma_{xx} = E * \frac{\partial u_x}{\partial x}$$

$$\varepsilon_{xy} = \frac{\partial u_x}{\partial y} \Rrightarrow \sigma_{xy} = E * \frac{\partial u_x}{\partial y}$$

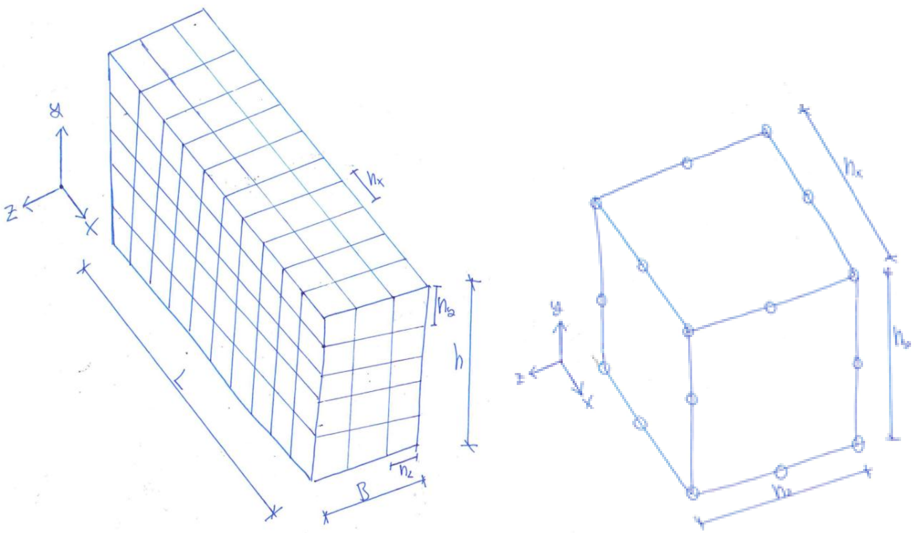
$$\varepsilon_{xz} = \frac{\partial u_x}{\partial z} \Rrightarrow \sigma_{xz} = E * \frac{\partial u_x}{\partial z}$$

2.4 Degrees Of Freedom

The Degrees of Freedom (DOF) of a beam assemble volume-elements indicates the amount of iterations needed by ANSYS to solve the structure. Of course, more DOF means more iteration that leads to longer calculation time. Therefore, the DOS will help us to take the time factor into account. A high DOS leads to unrealistic calculation time that the engineer should avoid. There are two types of DOF: Translational DOF, indicates that forces are transmitted through the nodes and a rotational DOF, that indicates that moments are transmitted through the nodes. For arbitrary cross-section in our beam we define EY, EZ and EX as numbers of elements in Y, Z and X direction in the local coordinates system.

The total DOF's in a beam is determent according to the following formula:

$$(EX+1)*(3*EY*EZ+2EY+2EZ+1)*3+EX(EY*EZ+EY+EZ+1)*3$$



2.5 Idealization study

Figure 2-7: The beam in the local coordinate system

Stress analysis is conducted with ANSYS to find the ideal number of volume elements. The purpose of this study is to find the optimum number of volume-elements. The number of elements with respect to the DOF versus the exact stress solution is plotted in *figure 2-8* to show what this study is exactly willing to achieve concerning the accuracy of the stresses computation in ANSYS.

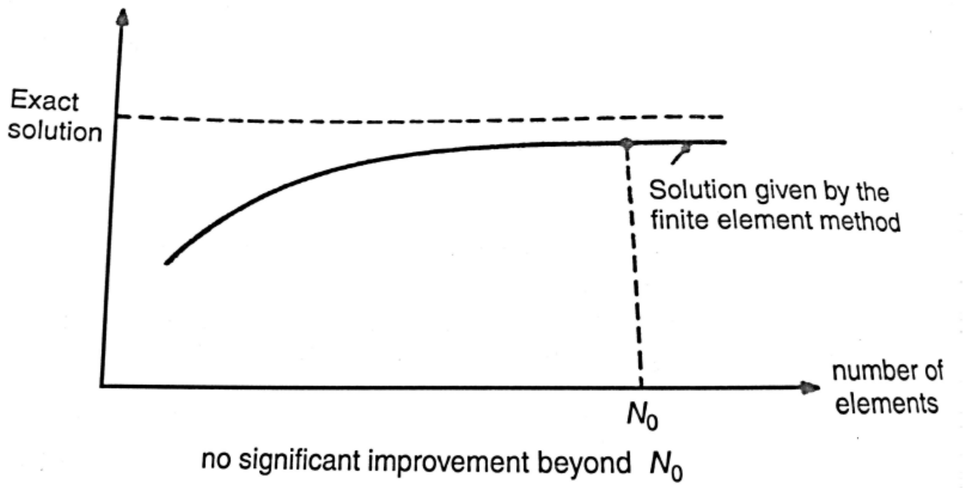


Figure 2-8: Approaching the exact solution with FEA method, [Rao, Singiresu S, The finite element method in the engineering]

N_0 represents the number of cross-sectional volume elements EX, EY and EZ , required in order to approach the exact stresses values (dashed line). Moreover, during the study the trendline given in - *figure 2-8* is expected to appear with as the output: N_0 value for several loading cases. Using knowledge from previous studies indicates that the number of elements in the x-direction EX also influences the accuracy of the solution. Therefore, in order to simplify the study, EX will be high enough in order to avoid deviation as result from too coarse mesh in this direction.

2.6 Error in an FEA Model

2.6.1 Accuracy solutions

In all mathematical processes errors occur. The errors are unavoidable, however it is essential to understand the different types of errors. The numerical error considered in this rapport is the difference between the exact analytical (mathematical) solution and the approximated numerical solution obtained when simplifications in the numerical computation abbreviate the computation time. Because a numerical computation is still an approximation depending on the number of iterations and because we want to restrict the number of iterations while keeping the deviation as small as possible, appropriate approximation methods will be applied. These are as follows:

1. The absolute error: $|numerical\ value - exact\ value|$

2. The relative error: $\frac{|numerical\ value - exact\ value|}{|exact\ value|} * 100$

2.6.2 Mesh discretization error

When calculating the stresses in a beam subjected to a load, the solution from the stress-strain relations are continuous. This means that the stresses in the cross-section are continuous distributed. However, a FEA model is a discrete problem that approximates the continuous problem. The question might be: what is the error due to an inadequate mesh density?

The mesh will be initially coarse with $EY * EZ = 2 * 2$, the number of elements will increase approaching the exact state of stress. The error of every refinement step of the mesh is defined by:

$$Relative\ error = \frac{|Numerical\ referamce\ valeu - numerical\ valeu\ |}{Numerical\ referance\ valeu} * 100$$

3. Analytical solution

The analytical stress solution of the beam under stated load is associated with the exact value of the stresses in the beam. Hand calculations or an accurate numerical solution will form the required reference values during the study. Our beam is a homogenous beam with linear elastic behaviour. In this chapter the theoretical formulas will be applied in the search for the analytical stress solution for each of the loading cases described in $\oint 1$.

3.1 Torsion

The beam is subjected to torsion load. The torsion stiffness GI_t of the cross-section is defined as the torsion moment M_t times the length of the beam divided by the rotation φ.

$$GI_t = \frac{M_t * L}{\varphi}$$

The following common situation is being analysed. Torsion force is applied on a beam due to load from a transverse beam. *(Figure 3-1*)

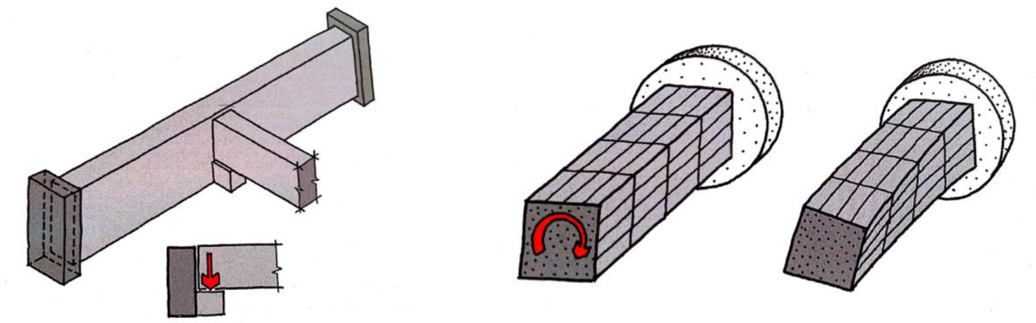


Figure 3-1: The beam under torsion moment. [Kracht en Vorm]

Before sketching the mechanical model, boundary conditions are chosen:

Torsion boundary condition	*Physical meaning*	*Mathematical meaning*
Pinned (Free end warping)	The cross-section can't twist but can warp freely	$\varphi = 0\,, \frac{d^2\varphi}{dx^2} = 0$
Fixed	The cross-section can't twist or wrap	$\varphi = 0\,, \quad \frac{d\varphi}{dx} = 0$
Free	The cross-section can twist and warp freely	$\frac{d^2\varphi}{dx^2} = 0, M_t = 0$

Table 3-1: Boundary conditions

Let us consider that the rectangular bar is subjected to torsion as shown in *figure 3-1*. As mentioned in the table above we can approach the problem with three mechanical models. Let us focus from now on the *both side fixed beam* (*figure 3-2*). In this model, the cross-section at the fixed supporting point can't twist or wrap. In this case there will be a warping resistance at the fixed supporting point. According to the theory of Vlasov, there is a deviation in the torsion values within a distance of five times the beam height from the fixed supporting point, due to mentioned warping resistance. (*Figure 3-2)*

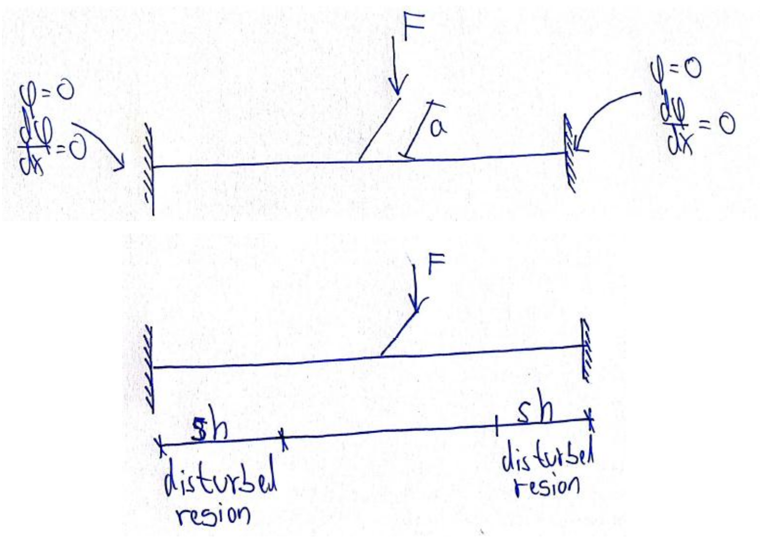


Figure 3-2: mechanical model

Due to symmetry we consider only one side of the beam. For T/2 it holds:

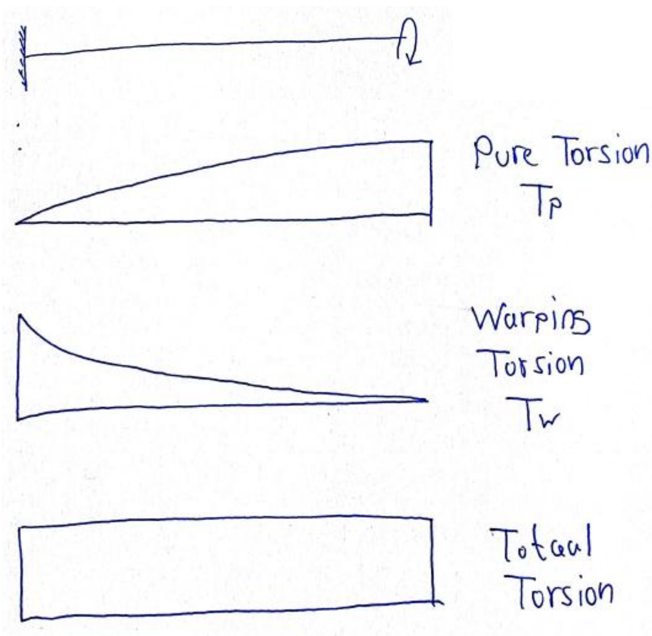


Figure 3-3: The torsion moment distribution

The cross-section in the middle of the beam will be analysed where the warping can be neglected. In this study the rectangular beam and HE300A beam will be considered. Of course, for each cross-section the calculation will be different. As mentioned, the warping effect will not be taken into account. This means that the second part of the formula bellow will has to be zero:

$$T_{(x)} = GI_t \frac{d\phi}{dx} - EI_w \frac{d^3\phi}{dx^3}$$

As results, the deformation effect caused by the secondary torsion moment (Bi-moment) will not be taken into account. This assumption is taken in order to avoid a complex model. This assumption is thus necessary in order to simplify the model and restrict the research objective. A further detailed research on the effect of warping resistance on the accuracy of the numerical computation with FEA will need to be conducted.

3.1.1 Table values rectangular profile

The torsion stresses of rectangular cross-section are derived in *table 3-1*:

$\frac{b}{h}$	$\frac{I_w}{bh^3}$	$\frac{I_p}{bh^3}$	$\frac{M_w}{\tau_{max} bh^2}$	$\frac{M_w}{\tau_2 bh^2}$	$1000\frac{C_w}{b^3h^3}$	$\frac{100\,B}{\sigma_{max} b^2 h^2}$
1,0	0,141	0,167	0,210	0,210	0,134	0,368
1,2	0,166	0,203	0,221	0.237	0,352	0,565
1,4	0,187	0,247	0,230	0.262	0,838	0,987
1,6	0,204	0,297	0,237	0.281	1,418	1,37
1,8	0,218	0,353	0,243	0.299	2,000	1,69
2,0	0,229	0,417	0,249	0.314	2,540	1,94
2,5	0,250	0,604	0,261	0.342	3,640	2,35
3,0	0,264	0,833	0,271	0.362	4,416	2,59
4,0	0,281	1,417	0,288	0.388	5,354	2,82
5,0	0,292	2,167	0,299	0.398	5,865	2,90
10,0	0,314	8,417	0,323	0.400	6,642	2,94
50,0	0,331	208,417	0,329	0.400	6,931	2,82
∞	$\frac{1}{3}$	∞	$\frac{1}{3}$	$\frac{2}{5}$	$\frac{25}{36}$	$\frac{5}{18}$

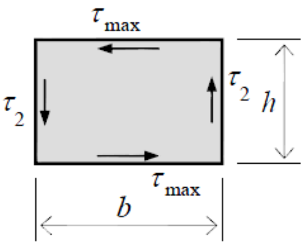


Table 3-2: Table values for rectangular cross-section [Reader Torsion, dr.ir.P.Hoogenboom]

The torsion moment in the cross section can be easily calculated with this table. However, there is one exception: For non-circle cross-section the polar torsion stiffness is unequal to the torsion stiffness. So it holds:

$$GI_p \neq GI_w$$

3.1.2 Torsion HE300A section

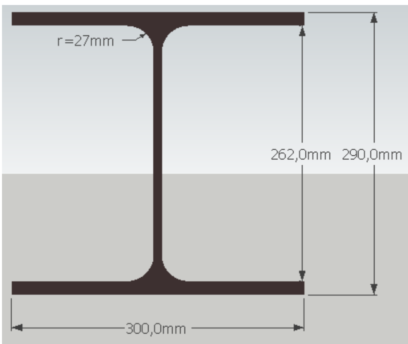


Figure 3-4: HE300A section subjected to torsion moment $\mathbf{M = 10e6\ Nmm}$

The approximation of the maximal torsion stress in a HE-section is calculated for a bar which can warp freely and where only shear stress occurs.

Torsion constant:

$$J = \frac{1}{3}\left(\left(h - t_f\right)t_w^3 + 2bt_f^3\right) = \frac{1}{3}\left((290 - 14)8.5^3 + 2 * 300 * 14^3\right) = 605299.5\ mmn^4$$

Maximal shear stress $\tau_{max} = \frac{M_t * t_f}{J} = \frac{10e6 * 14}{605299.5} = 231.29 \frac{N}{mm^2}$

3.1.3 Ideal beam length

In order to analyse the stresses in the cross-section it is important to ensure that this critical cross-section is located far enough from the disturbed areas. These areas are most likely to be at the support points and at the force loading point. The theory of Valsov defines the resistance length where the shear stresses in the beam deviate due to warping resistance of the fixed supporting point. The distance lc (Valsov region) can be calculated according to the given formula:

$$l_c = \sqrt{\frac{EC_w}{GI_w}} \qquad (3\text{-}4)$$

The characteristic bar length according the formula of Valsov is computed:

Shear modulus: $G = \frac{E}{2(1+v)} = \frac{210000}{2(1+0.1)} = 95454.54 \frac{N}{mm^2}$

Warping constant: $C_w = \frac{factor * b^3 * h^3}{1000} = \frac{1.418 * 200^3 * 320^3}{1000} = 3.71720e11mm^6$

Moment of inertia: $I_w = factor * b * h^3 = 0.204 * 300 * 200^3 = 4.896e8\ mm^4$

Characteristic length: $l_c = \sqrt{\frac{EC_t}{GI_t}} = \sqrt{\frac{210000 * 3.71720e11}{95454.54 * 4.896e8}} = 40.9mm$

Generally, for deviation smaller than 1% Valsov region should be equal to equal to 6*lc. That’s 245.4mm for a bar with h=320mm and b=200mm.

3.2 Bending moment

From the constitutive, equilibrium and kinematic relationship of linear elastically behaviour the bending moment can be described for a cross-section where the x-axis coincides with the member axis and the NC coincides with the YZ origin.

Moment of inertia rectangular cross-section: $I_y = \frac{1}{12} * b * h^3$

Curvature: $k_y = \frac{M_y}{EI_y}$

Moment : $M_y = EI_y k_y$

Stress: $\sigma_y = \frac{M_y Z_{na}}{I_y}$

The cross-section can be considered as thin-walled or thick walled. Thin walled means that the material is concentrated in the centre lines: $t_w \ll h \; and \; t_f \ll b$. This means that the moment of inertia $I_{y\,(cneter)}^{flange}$ will be zero.

Thin walled Moment of inertia $I_y = 2 * I_{y\,(Steiner)}^{flange} + I_{y\,(centre)}^{web}$

Thick walled Moment of inertia $I_y = \left(I_{y\,(centre)}^{flange} + I_{y\,(steiner)}^{flange}\right)2 + I_{y\,(centre)}^{web}$

HE300A	*A [mm2]*	*I_y [mm4]*
Thick walled	10627	1.7196e8
Thin walled	10746	1.7486e8
Table book	11300	1.8260e8

Table 3-2: Area's and moment of inertia

3.3 Shear stress

The cross-sectional shear stresses are stresses that are working on the cross-section plane. This can be determined thanks to the equilibrium relationship on s small rectangular block *(figure 3-5*).

In the cross-sectional plane the shear stresses in two perpendicular planes are equal (*figure 3-6*). For HE-sections subjected to shear force V, the shear stress can be calculated with the following formula:

$$\tau = -\frac{VS_y^a}{b^a I_y}$$

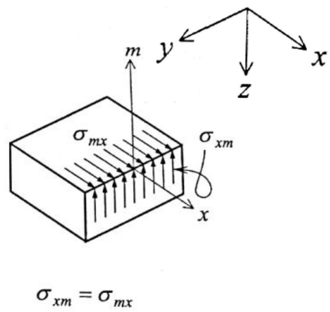


Figure 3-5: [C.Hartsuijker Toegepaste Mechanica, deel 2]

S_y^a represents the statical moment of the sliding element. In this way the shear stresses can be determined in each point of the cross-sectional plane. In a case of a thin-walled HE section the web and the flange will be considered separately. In order to calculate the static moment of the flange, a symmetrical double cut has been introduced. Where m_2 is the location of the cut *(figure 3-7*). This can be processed into formula:

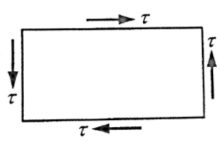


Figure 3-6: [C.Hartsuijker Toegepaste Mechanica, deel 2]

$$S_y^a = -t_f b m_2$$

$$b^a = 2t_f$$

$$\tau = \frac{V \cdot t_f b m_2}{2t_f \cdot I_y}$$

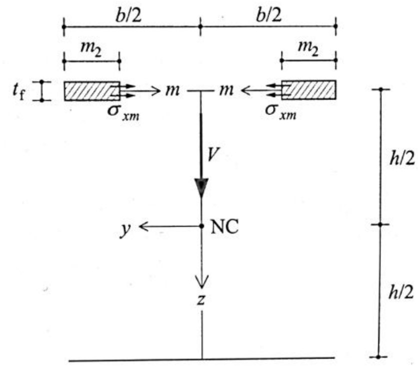


Figure 3-7: C.Hartsuijker Toegepaste Mechanica, deel 2]

As for the web *(figure 3-8)*:

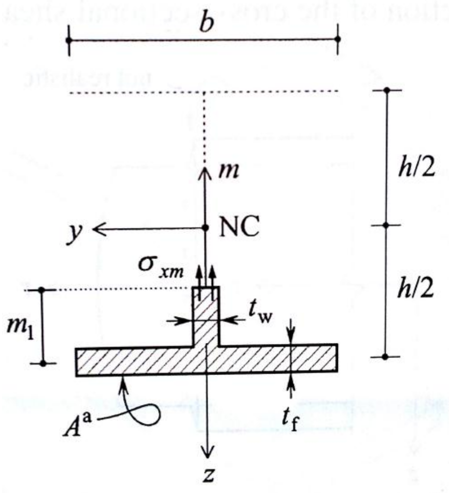


Figure 3-8: :[C.Hartsuijker Toegepaste Mechanica, deel 2]

$$S_y^a = t_f b \cdot \frac{1}{2}h + t_w m_1 \cdot (\frac{1}{2}h - \frac{1}{2}m_1)$$

$$b^a = t_w$$

$$\tau = \frac{V \cdot (t_f b \cdot \frac{1}{2}h + t_w m_1 \cdot \left(\frac{1}{2}h - \frac{1}{2}m_1\right))}{t_w \cdot I_y}$$

The shear stress diagram is presented in *figure 3-8*. The shear stress flow from the edges of the upper flange towards the web, and flow out below to the edges of the lower flange. The shear stress is constant along the wall thickness but increases linear from the edge to the centre of the flange. The shear stress at the web increases parabolic with top value at the level of NC. (*Figure 3-9*)

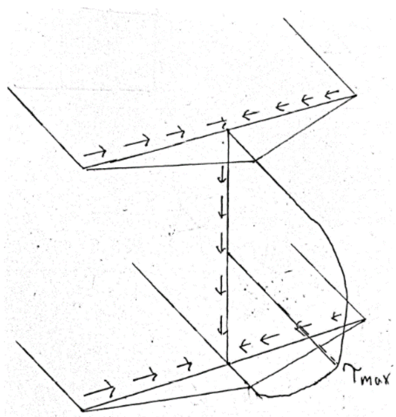


Figure 3-9: Shear stress flow

4. The rectangular cross-section

The beam consists of a steel rectangular cross-section with variable width and height. At the centre of the beam, at L/2, torsion loading is applied. From analytical computation of the applied torsion loading, the resulting stresses in the cross-section can be calculated. The stresses due to the same torsion loading can be calculated with FEM. The precision of the stresses distribution calculated with FEM is strongly depending on the number, size and ratio of volume-elements applied in the cross-section. In order to verify the optimal number of elements in the cross-section, the stress distribution of both Analytical (exact) and Numerical (approximation) methods need to be compared. The deviation from the exact solution needs to be as small as possible in order to meet the norms. This chapter will introduce the FEA model and will explain the method which the data from the model will be analysed. In the following next chapter $\oint 5.$ the database will be assembled and analysed and the recommendation will be given.

4.1 Material properties

The material properties of steel are been used. The material properties consist of the young's modulus E and the Poisson's ratio v. Regarding isotropic, linear elastic material the following properties are obtained:

E $\quad 210{,}000 \ \frac{N}{mm^2}$

v $\quad 0.1$

The Poisson's ratio has a direct influence on the stiffness matrix $\oint 2.3.1$ obtained in FEA. The relation between the Poisson's ratio and the Von-Mises stresses on an arbitrary point of a beam subjected to torsion moment is given in the following table:

v	*0,1*	*0,15*	*0,2*	*0,25*	*0,3*	*0,35*
$Von\ Mises\ \frac{N}{mm^2}$	312,66	307,968	303,397	298,937	294,573	290,28

Table 4-1: Von-Mises stresses for variable Poisson's ratio for an arbiter point of HE300A section

Hereby the table values are plotted to confirm the linear relation between the nodal stresses and the Poisson's ratio.

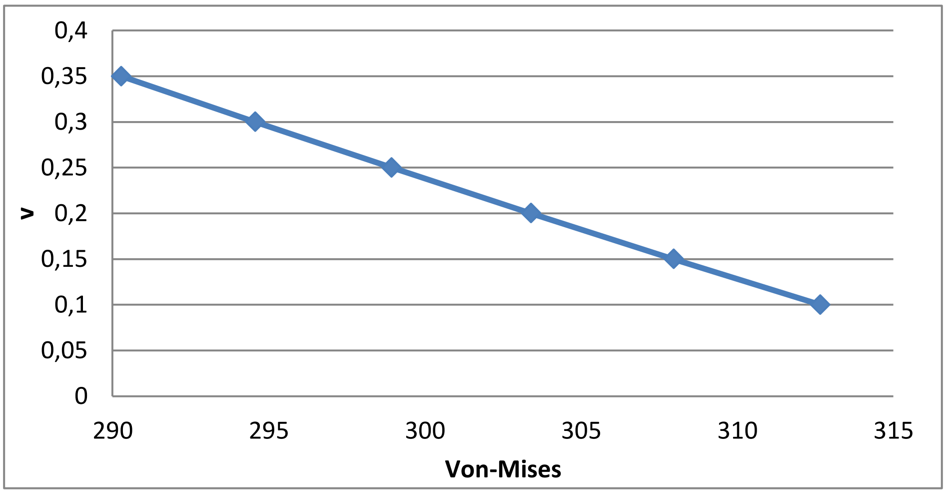


Figure 4-1: Linear relation between Poisson's ratio and the nodal stresses

4.2 Constraint

In order to simplify the problem, the beam will be modelled as a two sides fixed beam. This means that the boundary conditions for this problem are a fixed support applied on both side of the beam. These kinematical constraints will result in a simple and continues displacement distribution that meets the geometrical conditions of the beam. Further information can be found in $\oint 3.1.3$.

4.3 Forces

To simulate torsion moment acting on the 3D beam, a horizontal concentrated force is applied on the upper side and on the bottom side of the beam, in opposite directions. The magnitude of the applied force must not exceed the yield stress $f_y = 235\ \frac{N}{mm^2}$ in order to stay in the linear elastic stress distribution (*Figure 4-2*).

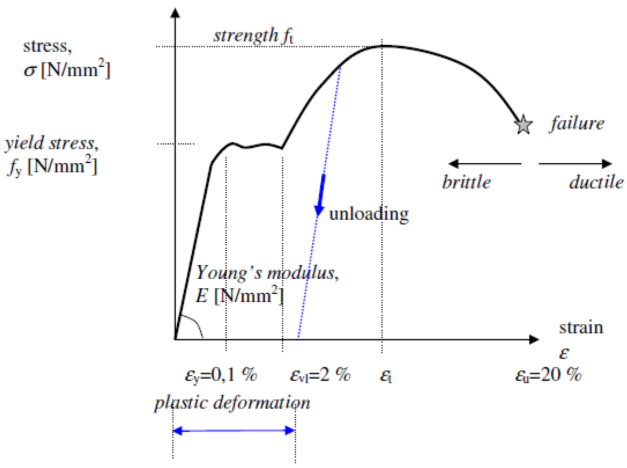


Figure 4-2: Material properties steel

In the example a cross section with $EY = 2$ and $EX = 2$ is drawn, the concentrated force is applied on the nodes of the beam as it appears in *figure 4-3*:

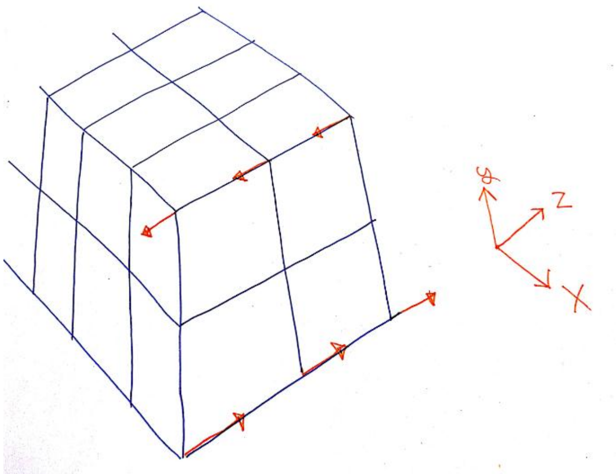


Figure 4-3: Beam with EY=2 and EZ=2 under torsion load

The magnitude of the force F_z is derived from the way which the force applied on the nodes:

$$F_z * n * h = M$$

$$n = EZ + 1$$

$$F_z = \frac{M}{h * (EZ + 1)}$$

The magnitude of M is arbitrary because we're only interested in the deviation of the stresses in our finite element model.

4.4 Modelling of the beam

The 3D beam (*figure 4-4)* is being modelled in ANSYS. Modelling the dimensions of the beam involves several variables such as:

L	Length of the beam
b	Cross-sectional width
h	Cross-sectional high

While modelling the mesh of the beam involves the following factors:

EZ	Number of elements in the width of the beam
EY	Number of elements in the high of the beam
EX	Number of elements in the length of the beam
L_x	Length of volume-element in the x-direction
L_y	Length of volume-element in the y-direction
L_z	Length of volume-element in the z-direction

Where:

$L = L_x \cdot EX$

$b = L_z \cdot EZ$

$h = L_y \cdot EY$

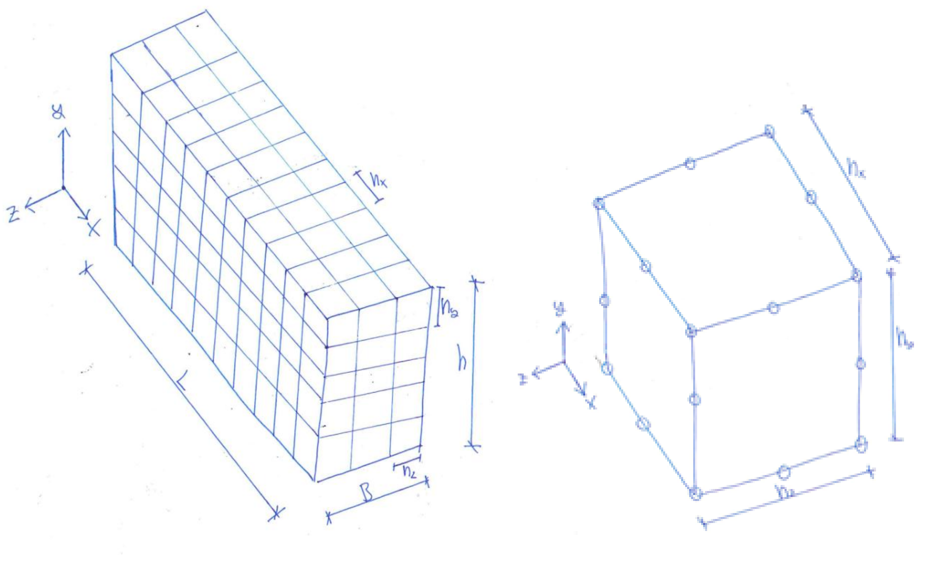


Figure 4-4: FEA model of rectangular beam in the global coordinate system

The length of the beam L is derived directly from the number of volume-elements in the x-direction EX. The applied loading on the beam is restricted to pure torsion. We want to neglect any variation of the numerical stress solution due to the number of elements in the length of the beam. Therefore, the beam will be modelled with length that goes to infinity, or in other words the ratio $\frac{EX}{L}$ will be as small as possible. Furthermore, during the FEA only the critical cross-section will be analysed, this cross section is located far enough from the beam supporting point at $L/_2$ where the warping resistance effect can be neglected.

Modelling the beam in ANSYS is a significant stage in this research, since it is an optimization study for the best mesh density and size, several of mesh configurations will be considered during the study. Also the discretisation of the beam dimensions will be analysed.

4.5 Stress field

The concept introduced in $\oint$ 2.3 about the displacement field polynomial will be drawn-out. The unknown displacement field u^h can be described by using basis shape function and discrete nodal values N_i which represent the amplitude of the shape function. The displacement at any point in the structure is determined in terms of discrete number of values which are stored at the nodes (known as DOF) and basis functions. The shape function for one dimensional (*figure 4-5)* bar is given by:

$$u^h(x) = N_1(x)a_1 + N_2(x)a_2 \qquad \text{(5-1)}$$

$$N_1 = -\frac{x}{L} + 1$$

$$N_2 = \frac{x}{L}$$

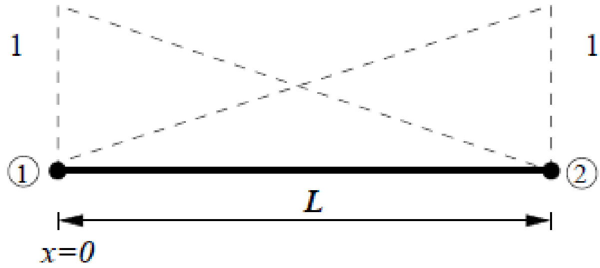


Figure 4-5: Shape function for linear element

For a higher-order quadrilateral element such as 20-nodes elements, the so-called Lagrange element, the displacement will be given by matrix notation. The Function $u^h(x, y, z)$ is the function describing the displacement field for every point of the specified beam:

$$u^h = Na_e$$

With:

$$\varepsilon^h = \frac{\partial u^h}{\partial(x, y, z)}$$

The strain field is given by:

$$\varepsilon^h = Ba_e$$

Where the stress field is given by:

$$\sigma = E * \varepsilon^h$$

With N as a 3×20 matrix (dimensions*Number of nodes) and $B = \frac{\partial N_i}{\partial(x,y,z)} \quad i = 0 \dots 20$

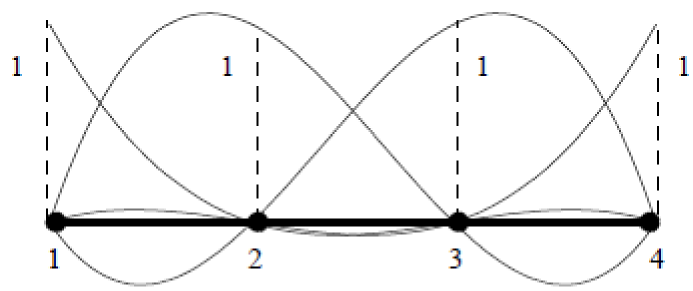
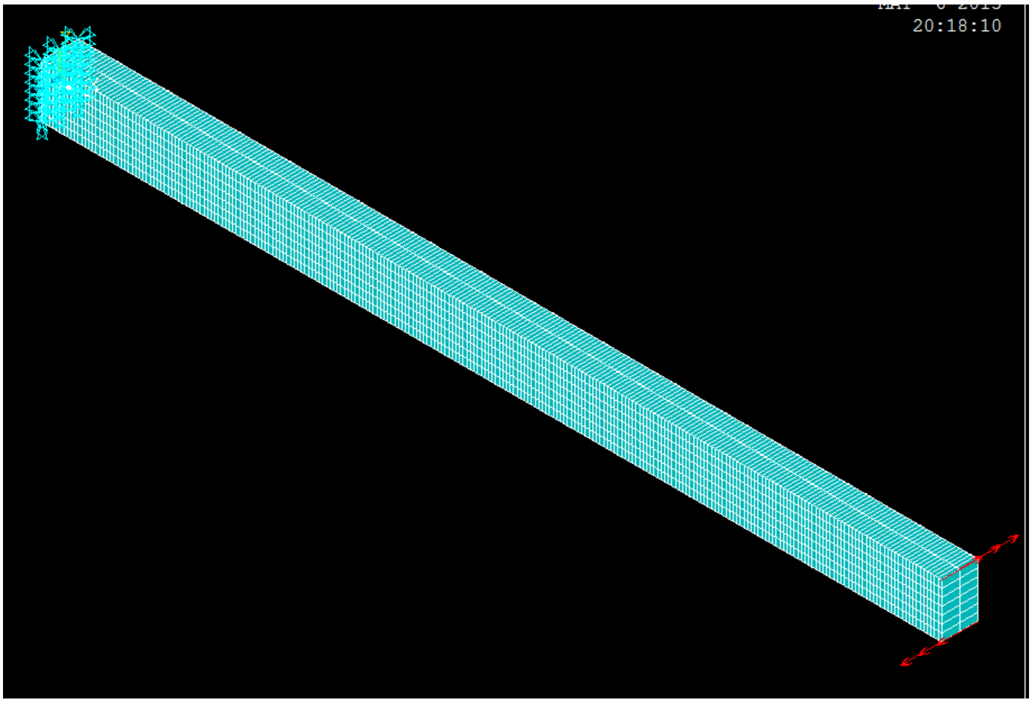


Figure 4-6: Higher order shape function

4.6 Torsion stresses

The aim of the study is to anticipate the stress values in the beam with high accuracy. But how can we find this numerical solution with an as small as possible deviation for a beam under torsional load? To answer this question the stresses distribution in the critical cross-section will be analyzed. At the critical cross-section located at L/2 the stresses values are being measured. The correct location of this reference point is important to avoid any unwanted measurement error (Observational error).



$L_E = 25mm$

$L = 3000mm$

$E = 210e6 \frac{N}{mm^2}$

$v = 0.1$

$M = 20e6\ Nmm$

Figure 4-7: The Beam in ANASYS

The location of the analytical and numerical reference point must be identical. At these points the shear stresses τ_{xy} and τ_{xz} will be the corresponding points P1 and P2. These reference points are given in *figure 4-8*.

$P_1\,(y,z) = \left(0, -{}^{b}\!/_{2}\right)$ and $P_2\,(y,z) = (-{}^{h}\!/_{2}\,, 0)$

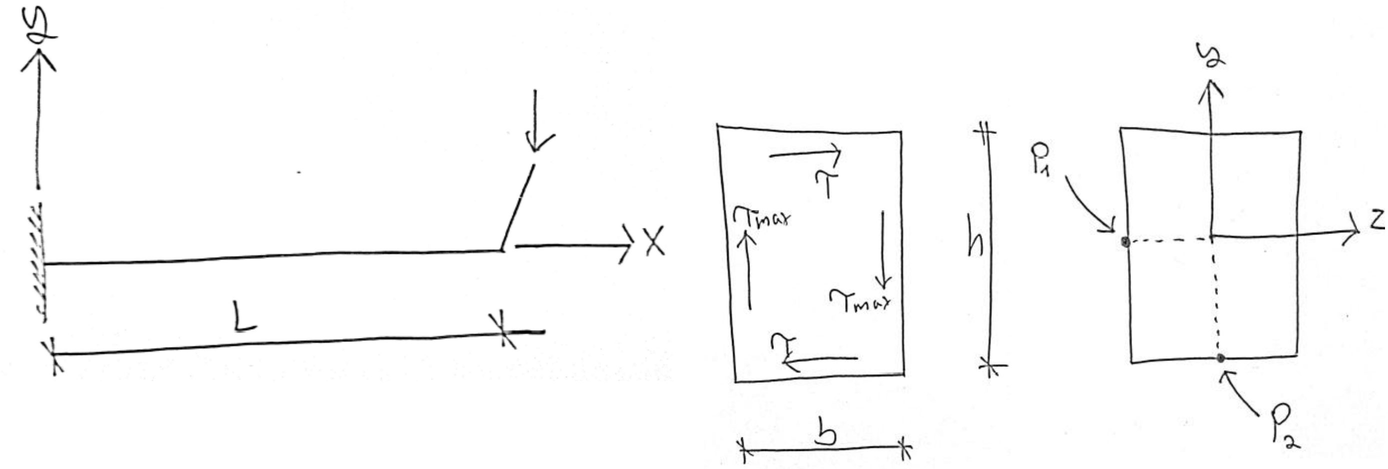


Figure 4-8: The location of the reference points p1 and p2

Points P1 and P2 in *figure 4-8* are located where the torsion stresses are maximal. As showed In *figure 4-9* and *figure 4-10* :

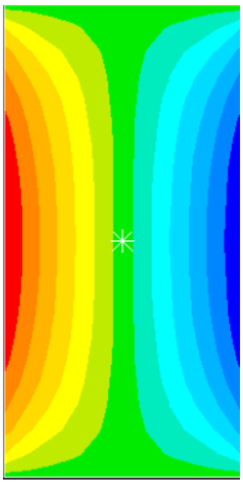


Figure 4 -9: τ_{xy} stress torsional stresses distribution [Reader P.Hoogenboom]

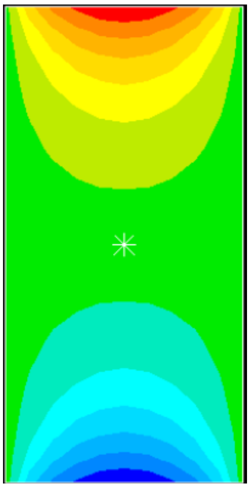


Figure 4-10: τ_{xz} stress torsional stresses distribution [Reader P.Hoogenboom]

4.7 Reference FEA model

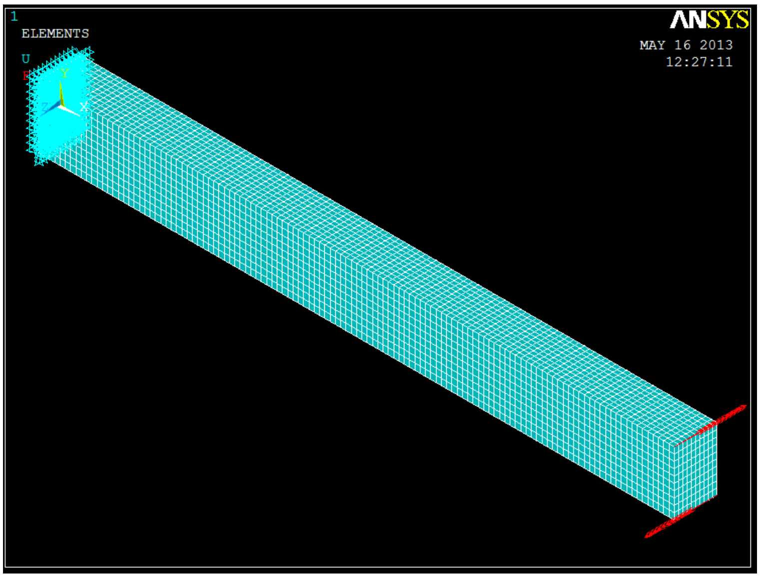


Figure 4-11: The reference model in ANSYS

The reference model is required to reach exact stress values. These values will be from now on the reference values regarding the deviation of every FEA model. The reference FEA model should be as accurate as possible. Therefore a 10 by 10 element has been chosen. In addition, the present of a node in the midside of the cross-section, due to even number of elements in the height and the width will contribute the accuracy of the reference model. The FEA reference model results in:

- Mesh of 10 by 10 elements: $EZ = 10 \quad EY = 10$
- The length of beam $L = 3000mm$
- 120 elements in the x-direction $EX = 120 \;\; L_x = 25mm$
- The $YZ - plane$ has 1023 DOF's which indicate the time it will take for the computer to solve the model. High DOF means longer computation time.
- $v = 0.1$
- $M = 20e6Nmm^2$
- *Solid186* element
- $h \; and \; b$ are variables

4.8 Element stresses distribution

Now, when the cross sectional reference points $P1$ and $P2$ are defined and the reference model is established, the numerical stresses solutions for the model can be calculated. Yet, ANSYS program provides two ways of reading stresses values. The nodal stresses and the elements stresses. These two will be explained in the following paragraph.

Initially, the stresses contour with respect to the elements will be discussed. *Figure 4-12* illustrates the stresses contour at x=0 for a cross-section with $EZ = 10 \; and \; EY = 10$.

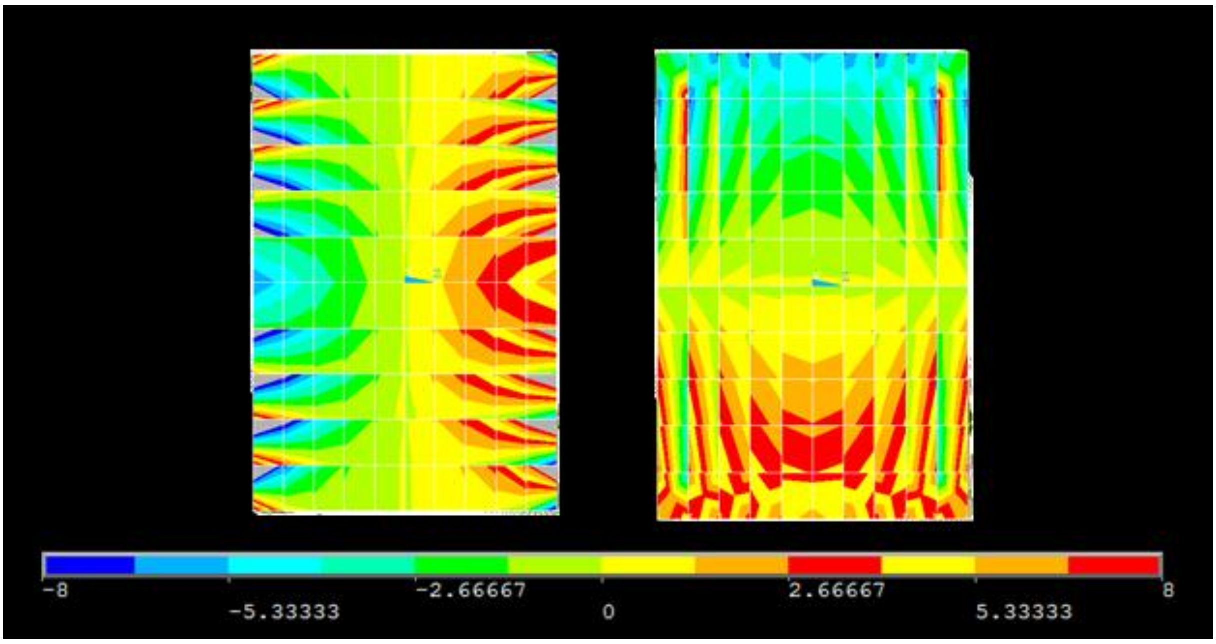


Figure 4-12: Element stresses contour at x=0 for τ_{xz} on the right and τ_{xy} on the left side a fixed support point (x=0) for b=200mm, h=320mm, M=20e06 Nmm2, L=3000mm, E=210e06 N/mm2 and v=0.1

To understand the results above it is significant to comprehend the FEA solving procedure. The FEA software takes the considered beam of h=300mm, b=200mm and solves every region (element) individually with simple linear equations. Because the solutions are not continuous (Figure xx), the software creates linear approximations that strive to map, as closely as possible, the true continuous solution. For a 10 by 10 cross-section with 121 corner nodes and 220 midside nodes ANSYS will calculate the displacements at each node given the loading and constraints on the model. Next, as a secondary operation, ANSYS approximates the stress contour in each element by looking at the relative displacement of the nodes of each of the 100 elements. In this manner a stress contour is determined for each element. In this manner the stress contour will be discontinue from one element to the next. This discontinuity results in the so called discrezation error. The error is reduced when the elements size reduced towards zero. In the search for the minimal number of elements with minimal error, it can be concluded that elements stress contour will therefore not be used. In the next paragraph a more precise FEA solution method will be introduced.

4.9 Nodal stresses solution

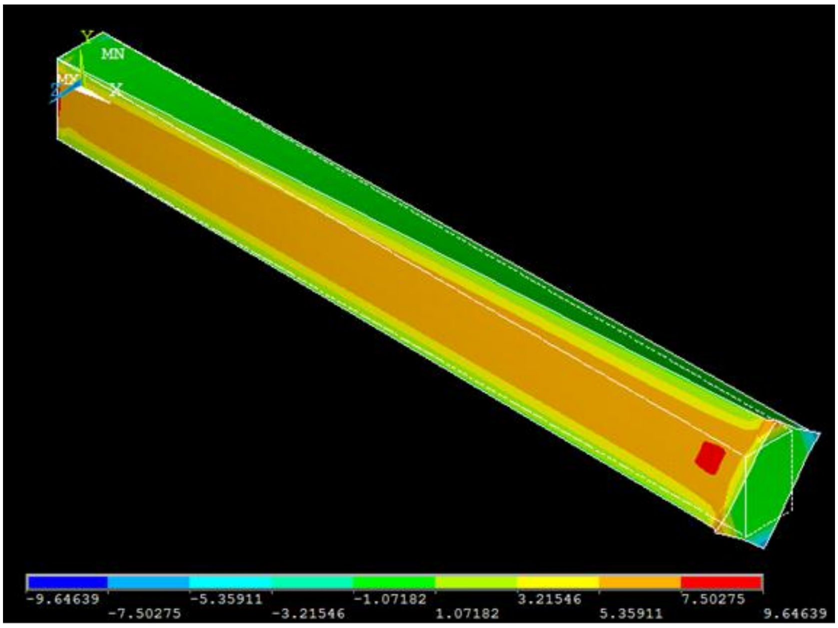


Figure 4-13: Torsion shear stresses

For the FEA with ANSYS software the same beam as mentioned is 4.7.1 will be analysed. The beam consist of 10 by 10 *Solid186* 20-noods element. For each node ANSYS will calculate three displacements U_x U_y U_z and three rotations θ_x θ_y θ_z. The stresses will be calculated for each node for the corresponding displacement value. *(Figure 4-14)*

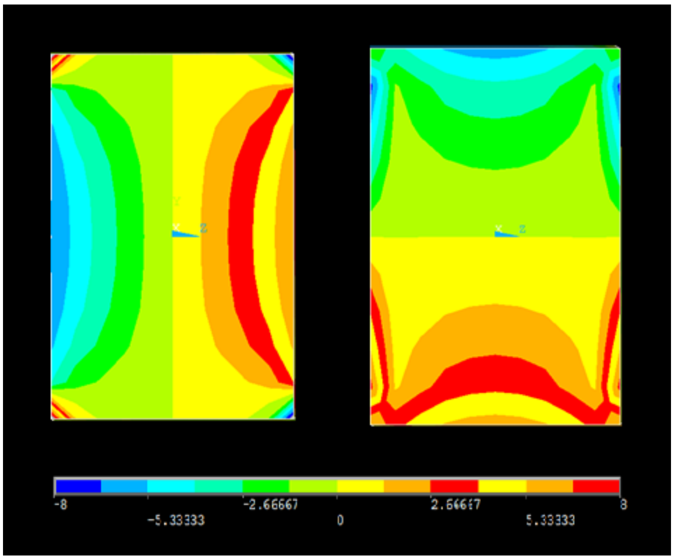


Figure 4-14: Nodal stress contour at x=0 for τ_{xz} on the right and τ_{xy} on the left at fixed support point (x=0) for b=200mm, h=320mm, M=20e06 Nmm2, L=3000mm, E=210e06 N/mm2 and v=0.1

Figure 4-14 shows the nodal stress contour, The total picture is much more transparent than the elements stress-contour given in *figure 4-12*. For this new situation the coordinate system is defined in *figure 4-15*:

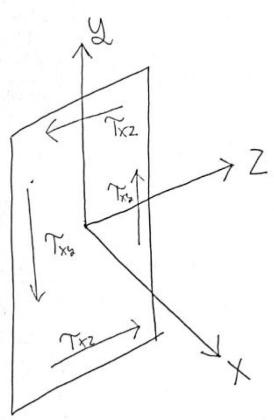


Figure 4-15: the global coordinate system

Splitting the cross-section into elements can be done in two ways, *odd* number of elements or *even* number of elements. It's important to distinguish between those two scenarios because the error for each partitioning is approximated differently. As shown *figure 4-16* there are even numbers of elements at the Y and Z-direction. At the midside the maximal shear stresses can be directly read from the middle node.

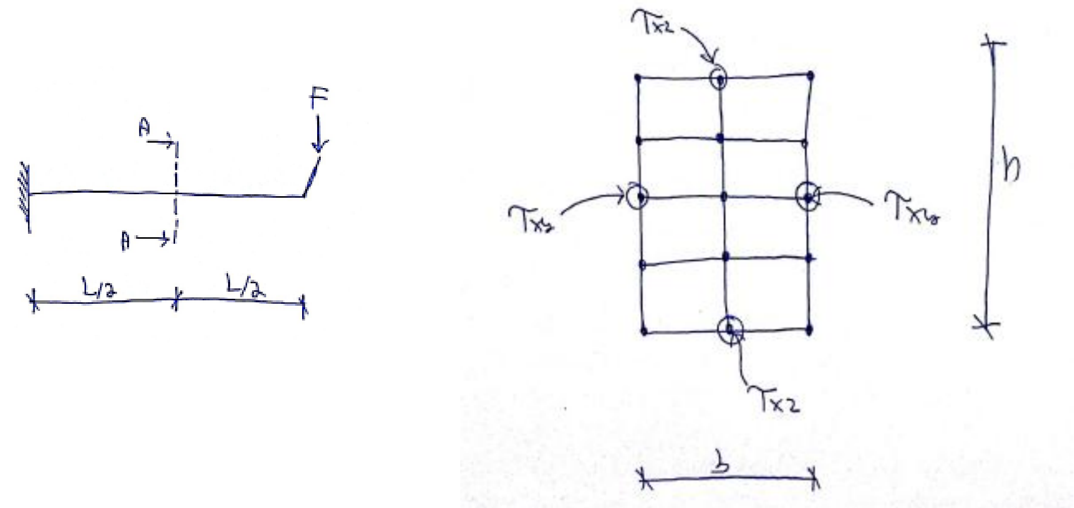


Figure 4-16: Cross-section A-A with even number of elements: $\mathbf{EZ=2}$ ***and*** $\mathbf{EY} = \mathbf{4}$

The reference coordinate where the stresses in the YZ plane can be read is at:

$\tau_{xz}(x, y, z) = (0.5L, \pm 0.5h, 0)$

$\tau_{xy}(x, y, z) = (0.5L, 0, \pm 0.5b)$

In a case of odd number of nodes as shown in *figure 4-17* the unknown numerical stresses values at midside of the outline can't be red. There are no nodes at the midline.

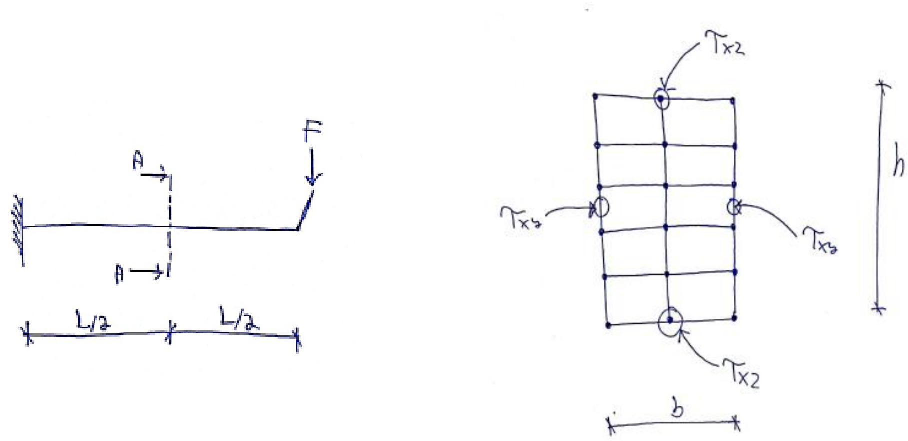


Figure 4-17: Cross-section A-A with odd number of elements in the height of the beam

The cross-section in *figure 4-17* is modelled in ANSYS with the purpose of understanding the influence of odd number of elements on the error. *(Figure 4-18)*

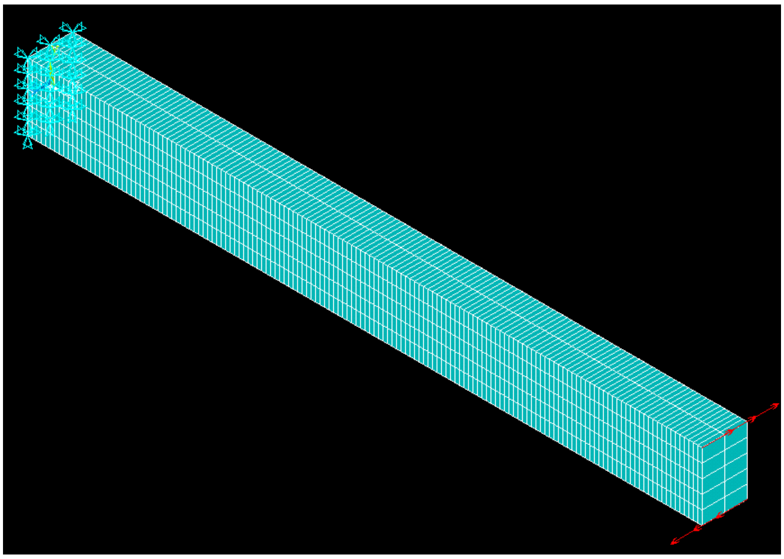


Figure 4-18: The beam consist of $\mathbf{EZ = 2, EY = 5}$, $\mathbf{L = 3000mm}$, $\mathbf{h = 320mm}$ $\mathbf{b = 200}$

We zoom in on the critical cross-section located at $x = L/2$. There, the two shear stress components τ_{xy} and τ_{xz} will be plotted. For five elements in the height there are six discrete points and for two elements in the width there are three discrete points. Notice that only the stresses on the outline of the cross-section are considered.

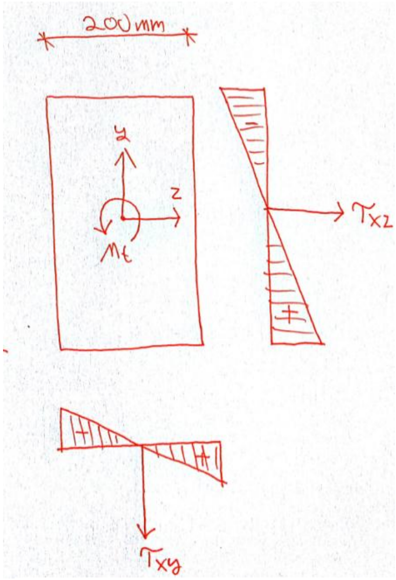


Figure 4-19: τ_{xy} ***and*** τ_{xz} ***stresses***

The numerical analysis of the torsion stresses in the beam provides the nodal stresses in the critical cross-section indicated with a broken red line in *figure 4-20:*

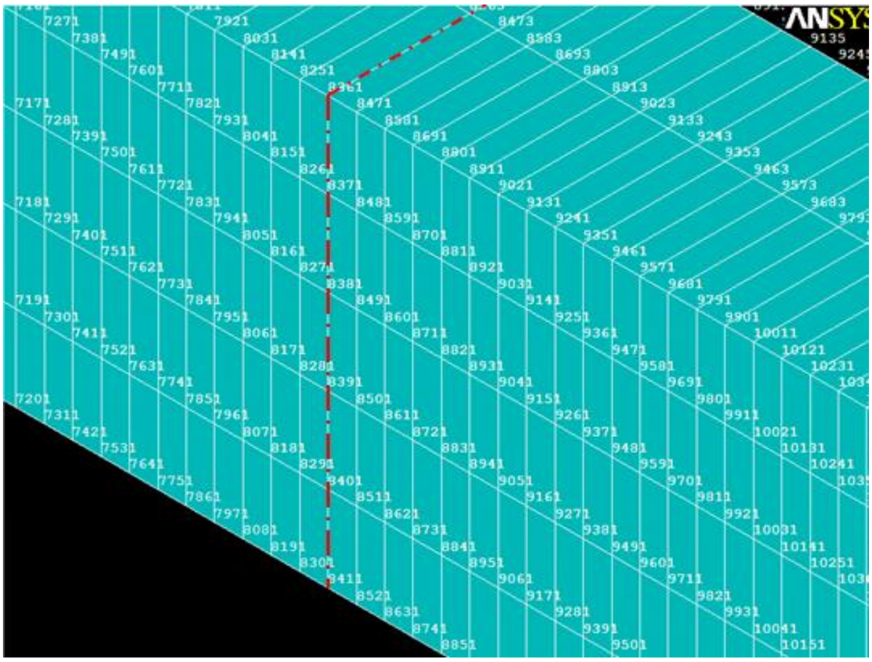


Figure 4-20: 3D ANSYS image of the considered cross-section

The XY and XZ stresses components have the maximal stresses values along the outline of the rectangular. As mentioned before, the stresses in the nodes will be the design stresses. Theses stresses change from one node to the other along the cross-sectional outline. The stresses are presented in the following tables:

Node	*Numerical [N/mm2]*	*Numerical ref [N/mm2]*	*Deviation [-]*	*Table value [N/mm2]*	*Deviation [-]*
1	0.6308	-	-	-	-
2	5.2294	-	-	-	-
3	6.5067	6.6819	2.6%	6.5928	1.3%
4	6.5067	6.6819	2.6%	6.5928	1.3%
5	5.2294	-	-	-	-
6	0.6308	-	-	-	-

Table 4-2: τ_{xy} stress analysis: L=3000mm, h=320mm, b=200mm, E=210e3N/mm2, v=0.1, EZ=2 and EY=5

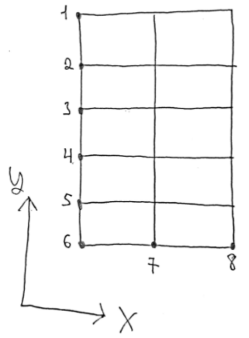


Figure 4-21: the cross-section

Node	*Numerical [N/mm2]*	*Numerical ref [N/mm2]*	*Deviation [-]*	*Table value [N/mm2]*	*Deviation [-]*
6	1.2878	-	-	-	-
7	6.3991	5.5777	12.8%	5.5605	13.1%
8	1.2878	-	-	-	-

Table 4-3: : τ_{xz} stresses analysis : L=3000mm, h=320mm, b=200mm, E=210e3N/mm2, v=0.1, EZ=2 and EY=5

The data in the tables above shows the nodal cross-sectional shear stresses solution due to torsion moment. The data implies that for the current number of elements an accurate approximation of the stress can be achieved depending on which reference value is being chosen. The deviation of the numerical values from the reference values is larger than the deviation from the torsion table values given in $\oint$ 3.1.1. The differences in stresses values with identical beam properties might lead to the wrong conclusions. The variation of the numerical reference model from the *table values* for the stated cross-section is presented in *table 4-4:*

$h/_b$	*1*	*1,6*	*2*	*2,5*	*3*
D_{xy}	0,9%	1,3%	1,5%	1,49%	1,5%
D_{xz}	0,9%	0,3%	0,3%	0,4%	1,1%

Table 4-4: Deviation of the reference value from the table value

This undesirable deviation of our numerical reference values from the *table values* occurs due to the modelling technique being used to derive the table values. The *torsion table values* is assembled from a model in FEA software just like our reference model. However, there are differences between the two models. (Table 4-5)

	Reference model	*Table model*
Equations	3D differential equations	2D differential equations
Length	Finite	Infinite
Boundary conditions	Influence fixing and load at beam end	No influence
Elements	High accuracy (20 nodes hexahedron)	Low accuracy (3 nodes triangle)
Cross-sectional DOF's	Approximately 300	100000

Table 4-5: Comparison Reference model with Table value model

The table values are obtained from a 2D model with significant amount of DOF's , which explains the accurate results obtained from this model. To conclude, the table values are probably more accurate, therefore table values will be used as a reference for the rectangular beam concerning the calculation of the error.

4.10 Error approximation

In $\oint$ 4.9 the deviation of the given rectangular beam has been analysed. Now the magnitude of the deviation (error) will be checked for this given beam. This working method will be applicable for $\oint$ 5.0 where the discretization of the beam in the height and in the width will take place. Below both shear stress components are given in two forms:

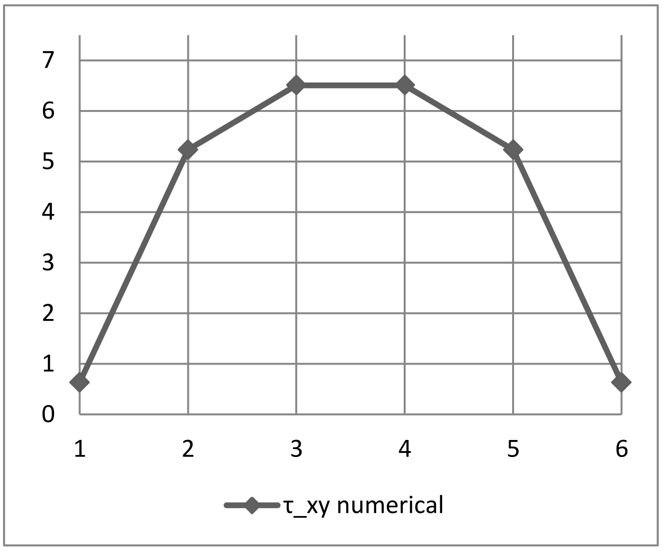


Figure 4-22: Linear stresses distribution between the nodes

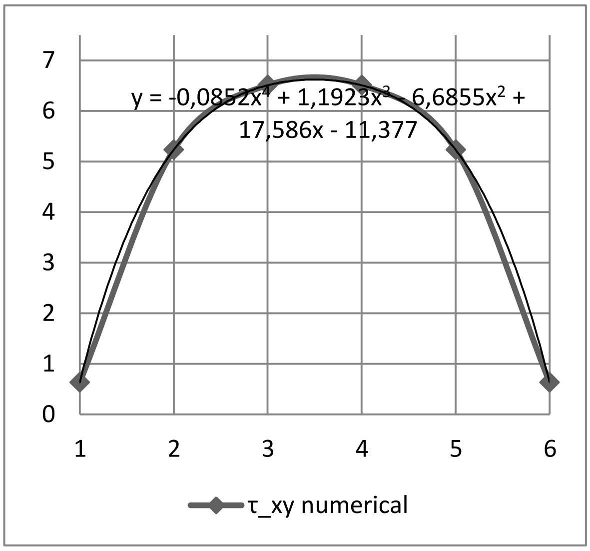


Figure 4-23: Stresses replotted in a 4-oreder polynomial to approach the solution between node 3 and node 4.

The stresses distribution along the long side of the cross-section uses stresses data from six nodes. For 8-nodes elements the stresses would be constant between the nodes. For 20-nodes elements the same stresses values are distributed with a linear change between the nodes (figure 4-22). From the basic structural mechanics we know that torsional stresses distributed parabolically. Thus, we should consider using numerical manipulations to construct a new data point at the midside of the xy-plane within the range of the discrete set of known points, or in other words interpolation. The two possibilities, linear and quartic functions, given in *figure 4-22* and *figure 4-23* will be compared in *table 4-6* and explained in $\oint$ 4.10.1 and $\oint$ 4.10.2.

	Table value	Linear (the mean of node 3 and 4)	4-order polynomial
Stress value [N/mm2]	6.5928	6.6819	6.6111
Error [-]	0%	1.3%	0.28%

Table 4-6: Error for τ_{xy} at point p1(y, z)=(0,b/2)

Analysing the content of *table 4-2* reveal that the stresses by an odd number of elements can be approximated from the nodes lying nearby the midside, in our case it is node 3 and node 4 in *figure 4-21*. However this conclusion will be checked for smaller number of odd element where the error might be large.

4.10.1 Mean value for odd element number

This is the average of the τ_{xy} stresses in two midside nodes. The deviation of the mean value of two correspond nodes holds: $\tau_{xy} = \frac{\tau_{xy;3}+\tau_{xy;4}}{2}$ or in more general expression:

$$\overline{\tau_{xy}} = \frac{1}{2}(\tau\big((n_{y,x=L/2} + 1)/2\big) + \tau\big((n_{y,x=L/2} - 1)/2\big))$$

With:

$n_{y,x=L/2}$ number of element in the y-direction for cross-section x=L/2

$\tau\left(\frac{n_{y,x=L/2}+1}{2}\right)$ stresses at the specified node

Because the formula above is calculating the mean value of two points, it is essential to check the standard deviation of the measurements:

$$\sigma = \sqrt{\left[\tau\left(\frac{n_{y,x=\frac{L}{2}} - 1}{2}\right) - \overline{\tau_{xy}}\right]^2 + \left[\tau\left(\frac{n_{y,x=\frac{L}{2}} - 1}{2}\right) - \overline{\tau_{xy}}\right]^2}$$

$$\tau_{xy} = \overline{\tau_{xy}} \pm \sigma$$

4.10.2 4-order polynomial

Using excel results in *quartic function* results in formula 4.xx where$\tau_{p1} = \tau_{xy}(x = 3.5)$.

$$\tau_{xy} = a_1x^4 + a_2x^3 + a_3x^2 + 4x + a_5 \qquad (4.x)$$

$$\tau_{xy} = -0.0852x^4 + 1.1923x^3 - 6{,}6855x^2 + 17{,}586x - 11.377 \qquad (4.x)$$

Higher-order functions like 5-order or 6-order appear to not have any effect on the interpolation. It is remarkable that this equation looks like the approach of Roark's formula for Stress&Strain that holds:

$$\tau_{xy} = \frac{3T}{8hb^2}[1 + 0.6095\frac{h}{b} + 0.8865\left(\frac{h}{b}\right)^2 - 1.8023\left(\frac{h}{b}\right)^3 + 0.9100\left(\frac{h}{b}\right)^4]$$

5. Optimization study

The length of the rectangular beam in the 3D space influence the accuracy of the obtained numerical stresses values. In $\oint 3.3.1$ the ideal length of the beam considering the Vlasov regions is founded. The warping resistance is therefore neglected by modelling a beam with sufficient length. In this case the length should be larger than 1000mm. Afterwards in $\oint 4.9$, the method of which the stress database should be assembled have been founded. The next step in the FEA is to run ANSYS several of times with respect to the discrete variables $EX, EZ\ , EY\ , b, and\ h$. Thanks to the optimized use of ANSYS, it is possible to conduct this calculations relative fast. The only restriction is the memory capacity of the computer which being reflected in long computation time. It is of course also an option to increase or decrease the Poisson's ratio or the Young's modulus. The correlation of the Poisson's ratio with the stresses values has been presented in $\oint 4.1$.

5.1 Length

Initially, we consider the error due to variable beam length and cross-sectional dimensions. The purpose is to find the numerical solution changes in respect to the ratio of beam length and cross-sectional dimensions (Table 5-1). The error is given by the following formula:

$$D_\tau = \frac{|Numerical\ solution - Referance\ valeu|}{Referance\ valeu} * 100$$

$L/max(h \mid b)$	*1.42*	*2*	*3.33*	*4*	*6.66*	*10*	*20*	*30*
$D_{\tau_{xy}}(\ z = 0.5b\ ;\ y = 0\ )$	9.9	5.2	1.4	1.1	1.0	1.0	1.0	1.0
$D_{\tau_{xz}}(\ z = 0\ ;\ y = 0.5h\ )$	7.0	3.0	0.7	1.0	1.1	1.0	1.0	1.0

Table 5-1: The relative error 'D' for a square cross-section at x=L/2 for: E=210e3N/mm2, v=0.1, L=3000mm, Mt=20e6 Nmm and $\mathbf{L_x = 25mm}$

Table 5-1 indicates that the error decrease as function of the beam ratio. For $L/h < 2$ a relative larger error is obtained. This can also refer to the length of Vlasov region and to the disturbed zone due to the concentrated force and the boundary conditions. Therefore, L/h ratio of 9.375 will be used with $EX = 120$

5.2 YZ-plane dimensions

The dimensions of the cross-section might have influence on the deviation of the solution. The rectangular beam is modelled in ANSYS with variable h/b ratio. The relation between the h/b ratio and the cross-sectional stresses distribution for a constant number of cross sectional elements EY and EZ is illustrated in *table 5-2*:

h=300mm b =... [mm]	*100*	*200*	*250*	*300*	*350*	*400*
$\tau_{xy}\ [\frac{N}{mm^2}]$	25.0	7,0	4,6247	3,4468	2,4743	1,9353
$Ref\ \tau_{xy}\ [\frac{N}{mm^2}]$	25.0	7.2305	4,8786	3,5592	2,7442	2,2045
Error [%]	0,05	3,21	5,49	3,26	10,91	13,91

Table 5-2: **The relative error at x=L/2 for $\mathbf{EZ = 2, EY = 5}$, E=210e06, v=0.1, $\mathbf{L_x = 25mm}$ and Mt=20e06Nmm**

The error for $b > h$ is increasing with respect to b. For the cross-sectional dimensions the following rule is applied: $\frac{h}{b} \geq 1$ for τ_{xy}

5.3 Number of elements

Optimization study of the number of cross-sectional elements was critical to determine the distribution of the error. From this study conclusion can be made over the minimal number of elements for a certain error. The error is computed with as reference value, the torsion table values described in $\oint 3.1.1$. It seems that the error is relative large for cross-section with only two elements in the height. This is valid for all : h/b ratio and for both stress components τ_{xy} and τ_{xz}. According to *figure 5-1* based on *table 5-3* it seems that the error convergence to zero for $EY \approx 5$. It can be concluded that for the highest accuracy: $EY > 5$ Furthermore, it should be noticed that besides the total number of elements also the ratio between the EY and EZ influence the accuracy. For example:

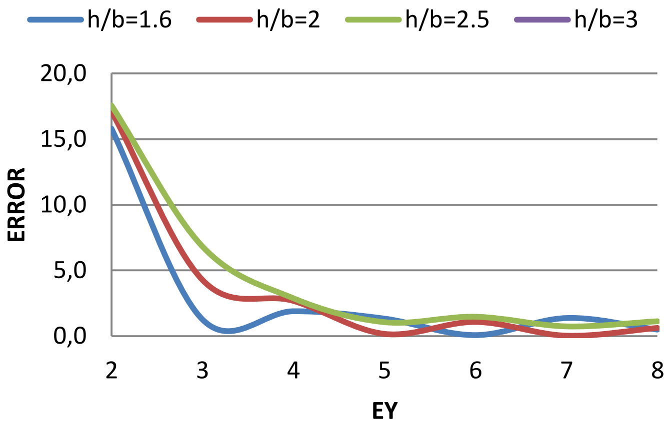


Figure 5-1: $\tau_{\max}$ ***error for variable EY, EZ=2 and EX=160***

- $D_{\tau_{xy}} : h/b = 1.6\,, n_b = 4\; n_h = 5$ is 0,2%
- $D_{\tau_{xy}} :\; h/b = 1.6, n_b = 5\; n_h = 4$ is 3,2%

In addition, as for the maximal stress component τ_{xy}, it seems that the error will increase if the number of elements in the transverse direction will increase. While for τ_{xz} , the error will decrease as the number of elements will increase for $EY > 2$ *and* $EZ > 2$. It can be concluded that for τ_{max} the following design rule valid: $\frac{EY}{EZ} \geq 1.$ The diagram showed in *figure 5-2* presents the change in the error as function of number of elements in the width for constant number of element in the height. The error is large for higher values of EZ . *Table 5-2* and 5-*3* present the data collected during the optimization study of the discrete beam. The data is valid for *Solid186* volume elements for constant beam length of 3000mm, constant element length in the x-direction $L_x = 25mm$ and constant torsion load of $20e6Nmm$.

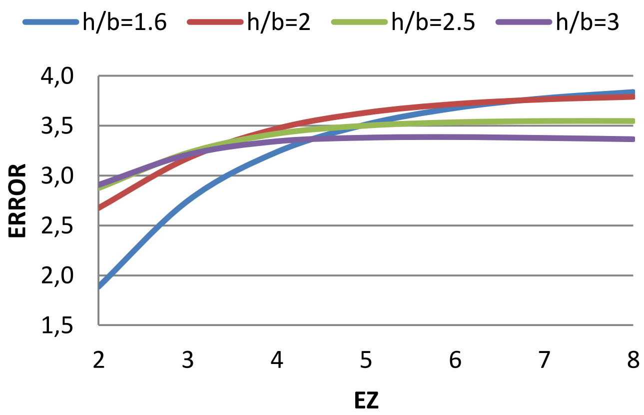


Figure 5-2: $\tau_{\max}$ ***error for variable EZ EY=4 and EX=160***

$h/b = 1$	EZ 2	3	τ_{xy} 4	5	6	7	8
EY 2	10,0	13,4	14,6	15,1	15,3	15,5	15,5
3	7,1	2,3	0,4	0,6	1,2	1,5	1,7
4	2,7	0,3	1,7	2,5	2,9	3,2	3,4
5	7,1	3,6	2,0	1,2	0,7	0,4	0,2
6	4,9	1,6	0,1	0,6	1,0	1,3	1,5
7	6,9	3,4	1,8	1,0	0,6	0,3	0,1
8	5,6	2,2	0,7	0,0	0,5	0,7	0,9

$h/b = 1{,}6$	EZ 2	3	4	5	6	7	8
EY 2	15,8	16,3	16,4	16,5	16,5	16,5	16,5
3	1,3	2,9	3,6	3,9	4,0	4,1	4,2
4	1,9	2,7	3,2	3,5	3,7	3,8	3,8
5	1,3	0,3	0,2	0,5	0,7	0,8	0,9
6	0,1	1,0	1,4	1,7	1,8	1,9	1,9
7	1,4	0,3	0,2	0,4	0,6	0,6	0,7
8	0,5	0,5	1,0	1,2	1,3	1,4	1,5

$h/b = 2$	EZ 2	3	4	5	6	7	8
EY 2	17,6	17,2	17,1	17,0	16,9	16,9	16,9
3	4,3	5,1	5,5	5,6	5,7	5,7	5,7
4	2,7	3,2	3,5	3,6	3,7	3,8	3,8
5	0,2	0,7	1,0	1,2	1,3	1,3	1,4
6	1,1	1,5	1,7	1,8	1,9	2,0	2,0
7	0,0	0,5	0,7	0,9	0,9	1,0	1,0
8	0,6	1,1	1,3	1,4	1,5	1,6	1,6

$h/b = 2.5$	EZ 2	3	4	5	6	7	8
EY 2	17,6	17,2	17,1	17,0	16,9	16,9	16,9
3	6,8	7,2	7,3	7,3	7,3	7,2	7,2
4	2,9	3,2	3,4	3,5	3,5	3,5	3,5
5	1,1	1,4	1,5	1,6	1,7	1,7	1,8
6	1,5	1,7	1,8	1,8	1,9	1,9	1,9
7	0,7	1,0	1,1	1,1	1,2	1,2	1,2
8	1,1	1,4	1,5	1,5	1,5	1,6	1,6

$h/b = 3$	EZ 2	3	4	5	6	7	8
EY 2	17,3	16,9	16,7	16,7	16,6	16,6	16,6
3	8,6	8,6	8,6	8,6	8,5	8,5	8,5
4	2,9	3,2	3,3	3,4	3,4	3,4	3,4
5	1,6	1,8	1,9	2,0	2,0	2,0	2,1
6	1,6	1,7	1,8	1,8	1,8	1,9	1,9
7	1,1	1,3	1,3	1,4	1,4	1,4	1,4
8	1,4	1,5	1,5	1,6	1,6	1,6	1,6

Table 5-3: Relative error for $\mathbf{M = 20e6 \frac{N}{mm^2}\ v = 0.1\ E = 210e6\ N/mm^2}$ ***L=3000mm and EX=120***

$h/_b = 1$	EZ 2	3	τ_{xz} 4	5	6	7	8
EY 2	10,0	7,1	2,7	7,1	4,9	6,9	5,6
3	13,4	2,3	0,3	3,6	1,6	3,4	2,2
4	14,6	0,4	1,7	2,0	0,1	1,8	0,7
5	15,1	0,6	2,5	1,2	0,6	1,0	0,0
6	15,3	1,6	2,9	0,7	1,0	0,6	0,5
7	15,5	1,5	3,2	0,4	1,3	0,3	0,7
8	15,5	1,7	3,4	0,2	1,5	0,1	0,9

h/b=1.6	EZ 2	3	4	5	6	7	8
EY 2	1,5	20,6	13,2	18,5	15,7	18,1	16,5
3	7,9	9,8	5,1	9,7	7,2	9,4	7,9
4	11,4	5,3	1,5	5,8	3,5	5,5	4,2
5	13,1	2,9	0,3	3,8	1,6	3,5	2,3
6	14,0	1,5	1,4	2,6	0,5	5,5	1,2
7	14,5	0,6	2,1	1,9	0,1	1,6	0,5
8	14,8	0,0	2,5	1,4	0,6	1,2	0,0

$h/_b = 2$	EZ 2	3	4	5	6	7	8
EY 2	9,5	29,4	20,4	26,1	22,9	25,5	23,7
3	4,2	14,5	8,8	13,8	11,1	13,4	11,9
4	9,3	8,2	3,6	8,2	5,8	7,9	6,5
5	11,9	4,8	1,0	5,3	3,0	5,0	3,7
6	13,3	2,8	0,6	3,6	1,4	3,3	2,1
7	14,2	1,5	1,6	2,5	0,4	2,2	1,0
8	14,7	0,6	2,2	1,7	0,3	1,5	0,3

$h/_b = 2.5$	EZ 2	3	4	5	6	7	8
EY 2	20,4	41,1	30,1	36,2	32,4	35,2	33,2
3	1,4	21,2	14,5	19,9	17,0	19,4	17,8
4	5,6	12,8	7,3	12,2	9,5	11,8	10,3
5	9,4	8,1	3,5	8,0	5,6	7,7	6,3
6	11,6	5,2	1,2	5,6	3,3	5,3	4,0
7	13,0	3,3	0,2	4,0	1,8	3,7	2,4
8	13,9	2,0	1,2	2,9	0,7	2,6	1,4

$h/_b = 3$	EZ 2	3	4	5	6	7	8
EY 2	31,2	52,4	39,5	45,9	41,5	44,5	42,2
3	7,3	28,2	20,5	26,3	23,1	25,7	23,9
4	1,7	17,6	11,2	16,4	13,6	15,9	14,4
5	6,7	11,5	6,2	11,0	8,4	10,6	9,2
6	9,7	7,7	3,2	7,7	5,3	7,4	6,0
7	11,6	5,3	1,2	5,6	3,3	5,3	4,0
8	12,9	3,6	0,1	4,1	1,9	3,8	2,6

Table 5-4: Relative error for $\mathbf{M = 20e6 \frac{N}{mm^2}\ v = 0.1\ E = 210e6\ N/mm^2}$ ***L=3000mm and EX=160***

Looking at the error tables in the previous page will reveal that the correlation between the number of elements and the error is quite strong. The smallest error is seen at the largest index. But now as an alternative, the degrees of freedom of the cross-section will be plotted. The DOF of the cross-section can be calculated as mentioned in $\oint 2.4$. It holds:

		EY						
		2	3	4	5	6	7	8
EY	2	63	87	111	135	159	183	207
	3	87	120	153	186	219	252	285
	4	111	153	195	237	279	321	363
	5	135	186	237	288	339	390	441
	6	159	219	279	339	399	459	519
	7	183	252	321	390	459	528	597
	8	207	285	363	441	519	597	675

Table 5-5: Cross-sectional DOF's

The error is now plotted with respect to cross-sectional DOF and compared with different h_e/h ratio's. (Figure 5-3)

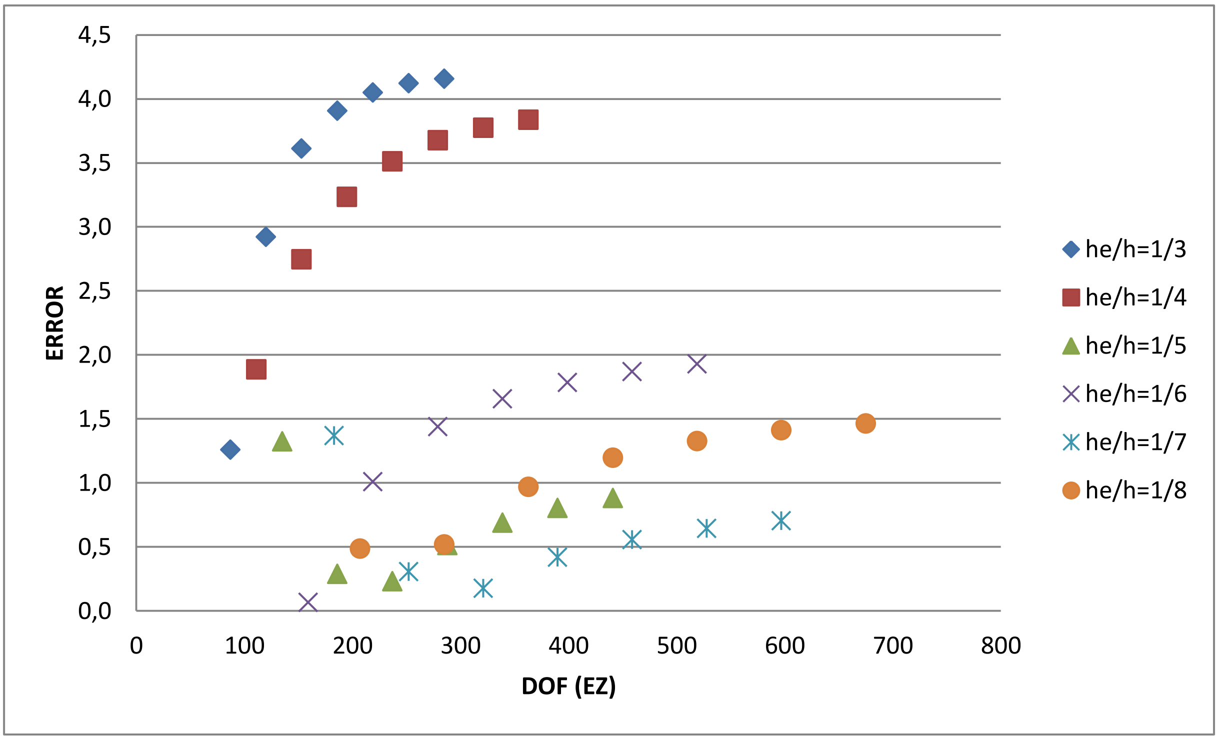


Figure 5-3: Relative error distribution versus cross-sectional DOF for $\mathbf{h/b = 1.6}$**.**

The error decreases for constant h_e/h ratio by increasing the number of elements in the width. Furthermore, the error becomes smaller for smaller values of h_e/h. In the second diagram (Figure 5-4) the b_e/b will be constant while the number of elements in the height will increases. It is remarkable that in this case the error will decrease for higher values of EY

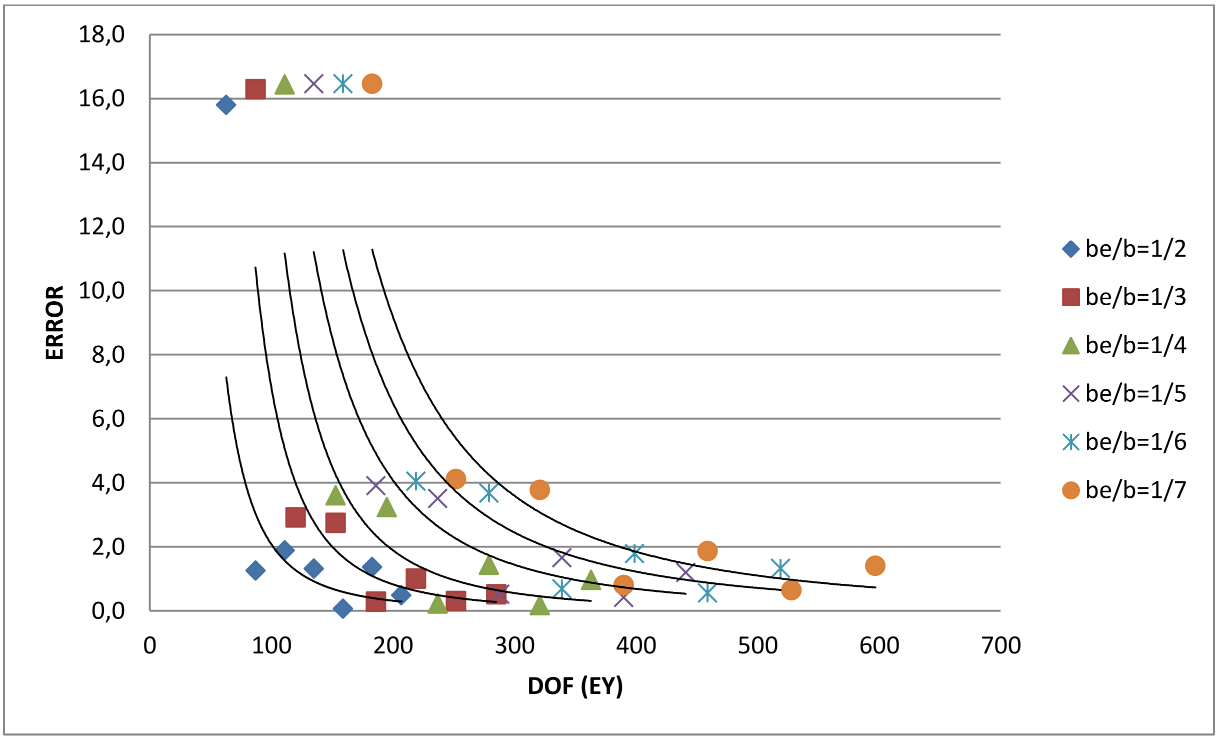


***Figure 5-4: Relative error distribution versus cross-sectional DOF for* $\mathbf{h/b = 1.6}$**

Both of the diagrams proofs what have been mentioned in the beginning of this paragraph. The ratio of the elements is critical for the accuracy and not only the number of elements (DOF's). This conclusion is valid for the maximal shear stress component. Consequently, in order to approach τ_{xy} for $\frac{h}{b} \geq 1$, the number of elements in the height are critical to reduce the error whereas the number of elements in the width will increase the error. As for the secondary shear components it can be concluded that while approaching the τ_{xz} for $\frac{h}{b} \geq 1$,increasing the number of elements in the height or in the width will reduce the error.

5.4 Sorting results

Sorting the results in *table 5-3* and *table 5-4* regarding the smallest error up to the largest error will help us to draw the conclusion related not to the accuracy but also to the computation time. As mentioned before, this computation time is equivalent to the DOF. In this way the minimal number of elements can be determine with taking into account the effectiveness of the computation time. *Table 5-6* and *table 5-7* present the relative error for $h/b = 1.6$. According to *table 5-6* the vertical shear stress component τ_{max} will be approached with accuracy of 99.9% through applying $EY = 6$ and $EZ = 2$ with only 159 DOF's. In $\oint$ 5.5 the meshing design tables for torsion moments will be presented on the basis of this representation.

EY	EZ	ERROR	DOF's	EY	EZ	ERROR	DOF's
6	2	0,1	159	6	5	1,7	339
5	4	0,2	237	6	6	1,8	399
7	4	0,2	321	6	7	1,9	459
5	3	0,3	186	6	8	1,9	519
7	3	0,3	252	4	2	1,9	111
7	5	0,4	390	3	2	2,7	87
8	2	0,5	207	3	3	2,9	120
8	3	0,5	285	4	4	3,2	195
5	5	0,5	288	4	5	3,5	237
7	6	0,6	459	3	4	3,6	153
7	7	0,6	528	4	6	3,7	279
5	6	0,7	339	4	7	3,8	321
7	8	0,7	597	4	8	3,8	363
5	7	0,8	390	3	5	3,9	186
5	8	0,9	441	3	6	4	219
8	4	1	363	3	7	4,1	252
6	3	1	219	3	8	4,2	285
8	5	1,2	441	2	2	15,8	63
8	6	1,3	519	2	3	16,3	87
5	2	1,3	135	2	4	16,4	111
3	2	1,3	87	2	5	16,5	135
6	4	1,4	279	2	6	16,5	159
8	7	1,4	597	2	7	16,5	183
7	2	1,4	183	2	8	16,5	207
8	8	1,5	675				

Table 5-6: relative of $\boldsymbol{\tau_{xy}}$ ***for*** $\mathbf{h/b = 1.6}$

EY	EZ	ERROR	DOF's	EY	EZ	ERROR	DOF's
8	2	0	207	4	6	3,5	279
8	8	0	675	5	5	3,8	288
7	6	0,1	459	4	8	4,2	363
5	4	0,3	237	3	4	5,1	153
6	6	0,5	399	4	3	5,3	153
7	8	0,5	597	6	7	5,5	459
7	3	0,6	252	4	7	5,5	321
8	6	0,6	519	4	5	5,8	237
8	7	1,2	597	3	6	7,2	219
6	8	1,2	519	3	8	7,9	285
7	7	1,2	528	3	2	7,9	87
6	4	1,4	279	3	7	9,4	252
8	5	1,4	441	3	5	9,7	186
6	3	1,5	219	3	3	9,8	120
2	2	1,5	63	4	2	11,4	111
4	4	1,5	195	5	2	13,1	135
5	6	1,6	339	2	4	13,2	111
7	5	1,9	390	6	2	14	159
7	4	2,1	321	7	2	14,5	183
5	8	2,3	441	8	2	14,8	207
8	4	2,5	363	2	6	15,7	159
6	5	2,6	339	2	8	16,5	207
5	3	2,9	186	2	7	18,1	183
5	7	3,5	390				

Table 5-7 : relative error of $\boldsymbol{\tau_{xz}}$ ***for*** $\mathbf{h/b = 1.6}$

5.5 Conclusion

From table 5-3 in ∮ 5.3 , only element configuration correspond with an error of up to 3% are taken into account. Moreover, the errors given in the meshing design table are compared to reject any chance that the obtained error has been founded accidently and valid only for the specific configuration with the specific dimensions. In addition, element configuration with large DOF will be neglected because only effective element configuration is wanted. This result in “FEA Meshing Design table” for rectangular beam subjected to pure torsion moment, without taking into account the warping resistance. The design table should be applied for the Ultimate limit state where the maximal shear force is governing. In our case it is τ_{xy} . In addition, the table is valid for load within the linear elastic stress distribution. The following parameters have been used to compute this design table:

$$\frac{EX}{L} = \frac{1}{120} \qquad E = 210e6\frac{N}{mm^2} \qquad v = 0.1 \qquad b = 200mm$$

τ_{xy}	EY	EZ	Error	DOF's
$h/b = 1$	6	4	0.1	279
	4	3	0.3	153
	6	3	1.6	219
	4	2	2.7	111
$h/b = 1{,}6$	5	3	0.3	186
	6	3	1	219
	5	2	1.3	135
	4	2	1.9	111
	4	3	2.7	153
$h/b = 2$	5	2	0.2	135
	5	3	0.7	186
	4	2	2.7	111
	4	3	3.2	153
$h/b = 2.5$	5	2	1.1	135
	5	3	1.4	186
	4	2	2.9	111
$h/b = 3$	5	2	1.6	135
	6	3	1.7	219
	4	2	2.9	111
	4	3	3.2	153

Table 5-8: FEA Design table for rectangular cross-section subjected to torsion

6. The HE300A section

6.1 Relevant information from previous report

The report of B.M. van de Weerd [December 2007] describes the numerical solution using FEA model for several of I-sections subjected to shear stress. The measurements of the shear stresses in the critical cross-section imply that these stresses will be largest at the mid-wedge as expected. Furthermore, it shows peaks of the stresses at the wedge-flange connection as shown in *figure 6-1*

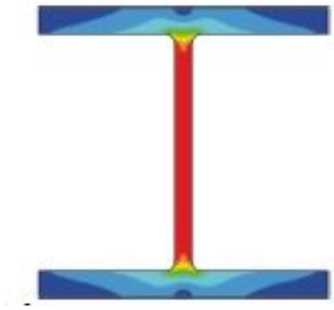


Figure 6-1: Shear stresses contour in ANSYS [B.M van de Weerd 2008]

6.2 Concept HE-beam model

In the previous chapter, the rectangular cross-section subjected to torsion moment has been analyses in the search fot optimal number volume elements. The rectangular cross-section was relative simple to model. Due to the rectangular shape there are few peak stresses concentrations. Those peak stresses are located at the corners of the cross-section and cause an infinite stresses values. The same logic applies for simplified HE-section model with rectangular planes. (Figure 6-2)

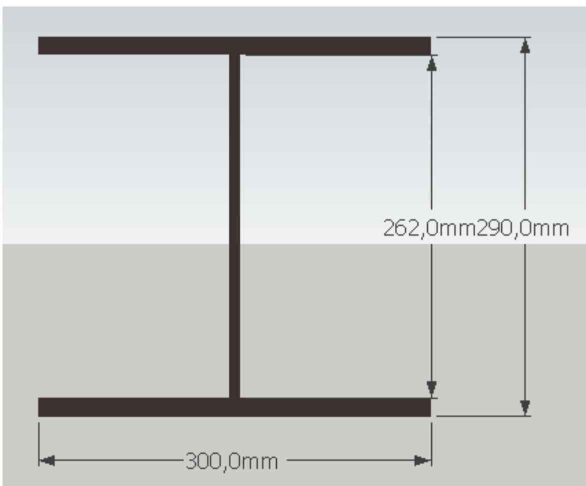


Figure 6-2: Simplified model for r=0 consist of rectangular geometry

The stresses contour of the simple HE-section in *figure 6-3* will contain peak stresses consecration at the web-flange connection and at the edges of the flanges. Letting ANSYS to analyses this model will result in infinite stresses values in those locations. That is the reason why realistic model will be introduced, including the rounded corners with $r = 27mm$. (figure 6-3)

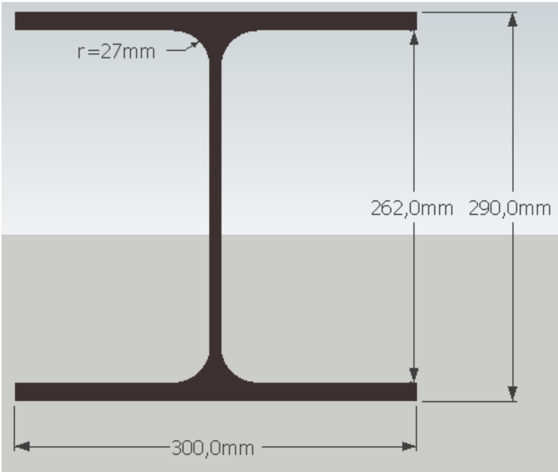


Figure 6-3: Realistically model for r=27mm

In this model the rounded corners reducing the peak stresses concentration. This singularities can be explained according to the elastic membrane analogy [Ludwig Prandtl, 1903] which describes the he stress distribution on a long bar in torsion. The theory uses the term $\phi - slope$ which directly proportional to the stresses. In the corner of the model (figure 6-3) the $\phi - slope$ will be large, rounding these corners will reduce the slope and the stresses peak concentration. In this research only the realistic HE-section with rounded corners will be analyzed. In the future, it is recommended to compare the stresses results of the simple model with the stresses results of the realistic model, in order to find the accuracy of the simple model.

6.3 Elements type

In this phase we'll analyze an FEA model of HE300A section, this common steel section is being used as compression element or for beam when the structure height has to be limited. However, modeling HE-section considering the mesh located at the rounded corners will require the use of volume elements like Prism (*figure 6-4)* due to the parabolic shape.

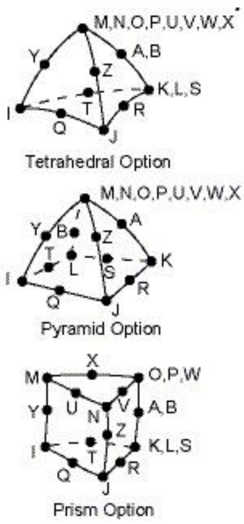


Figure 6-4: Various Solid186 shapes

6.4 The simple FEA model

To remind, the goal of this research is not to make the most accurate mesh, but to make a new accessible working method for the civil engineer. The simple HE-model from *figure 6-2* meet this requirement because it can be meshed with only hexahedron volume elements. In this case the model consist of rectangular sections with uniform mesh. That should be a straightforward mesh configuration with limited number of elements. In this report the reliability of this model will not be checked due to lack of time. However, it been recommended to check the accuracy of this model in future researches. *Figure 6-5* shows the simple FEA model in ANSYS.

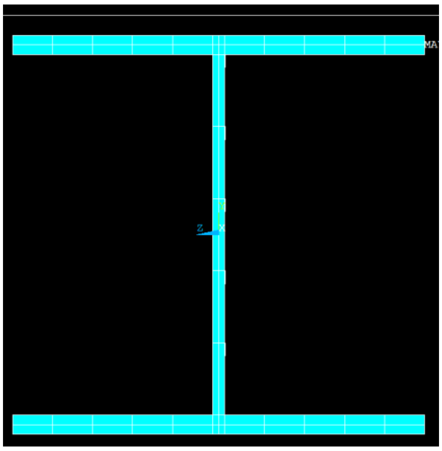


Figure 6-5 The simplified model with r=0

6.5 The realistic model

To meet the shape requirements related to the use of the rounded corners, this model will consist of 20-nodal hexahedron volume-element in the flange and web while the rounded flange-web corners will consist of 20-nodes *Prism* volume-elements. (Figure 6-6)

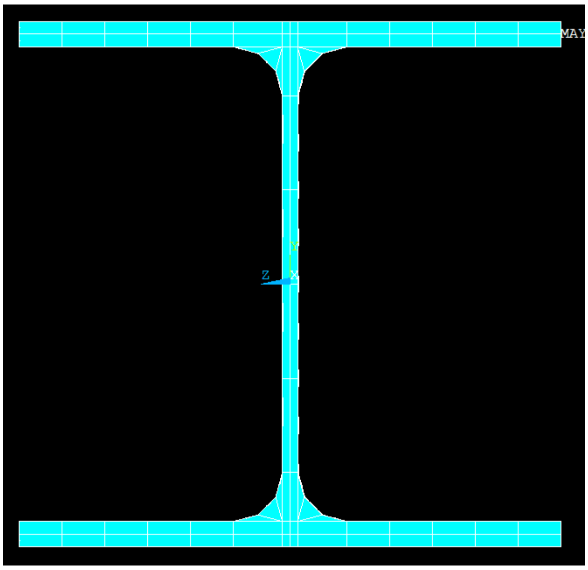


Figure 6-6: The realistic model with RoC=27mm

6.6 The reference models

In the search for the optimal HE300A FEA model the numerical, yet unknown values, will be compared with a reference value. This reference value should supply the exact stress value of which the deviation will be measured. The reference model consist of a fine mesh comparing to the coarse mesh of the investigated model. For each loading case the reference model will be different with respect to the pattern being discovered in the analysis. For example, the reference model of the torsion moment case has been resolved by more than one million differential equation and 9123 cross sectional DOF's. Such a model has been solved within more than one hour on a Pentium 4 computer.(Figure 6-7)

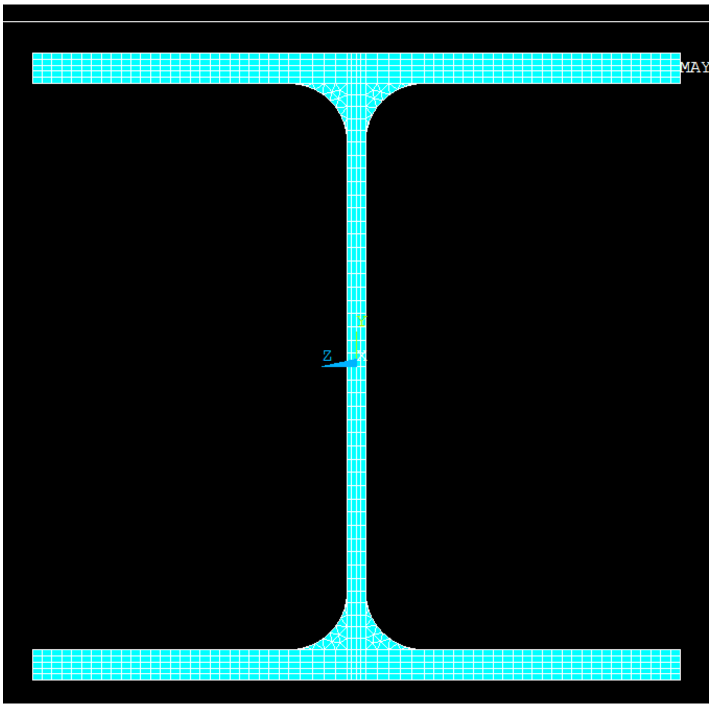


Figure 6-7: The reference model

6.7 Working method

The design of the HE300A beam in ANSYS occurs in a different method than from the previous rectangular beam. In this case, few HE300A FEA-models will be individually computed between themself. The working method suggests beginning with a basic coarse mesh which will be refined in every new model. The results of this working method will be the reducing/increasing the deviation with every new mesh density. This working method will adapt the mesh density until an adequate deviation will be achieved.

6.7.1 Adaptive Mesh Refinement

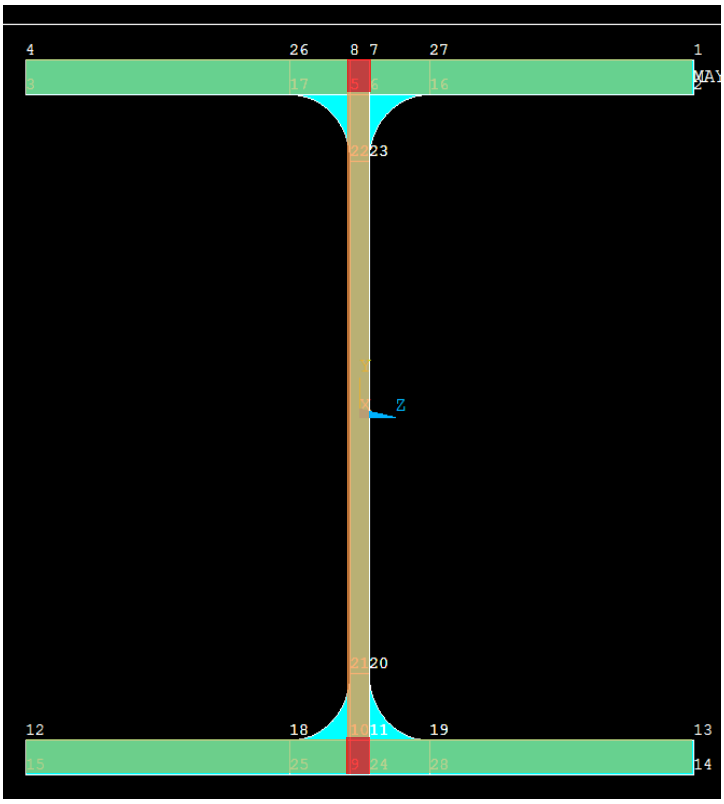


Figure 6-8: Cross-section is divided into subregions

Sub region	*Description*
Green	Flange mesh
Red	Peak stress region
Orange	Web mesh
Blue	Rounded corners (R)

Table 6-1: Legend for figure 6-8

For a proper modeling the mesh will not be uniform however, it will be denser where the stresses are maximal. Alternative approach considers a FEA model with a standard uniform mesh. The unknown nodal stresses values for this grid are estimated by ANSYS. The spacing of the mesh determines the accuracy of these values. Although the mesh of the cross-section is uniform, the stresses contour is not uniform. There are regions in the cross-section where a finer mesh is required to reduce the deviation from the real stresses values. The Adaptive Mesh Refinement is using submesh to fix computations error in difficult regions. The first step is identifying the peak stresses of every load case. Indubitably, every loading case like torsion, bending moment and shear obtain different stresses contour and required a different mesh. The idealized mesh will be then non-uniform mesh. The mesh density will be determined upon the peak stresses. The location of the peaks is well known from the theory and from early researches as mentioned in ∮ 6.1

6.7.2 Meshing variation

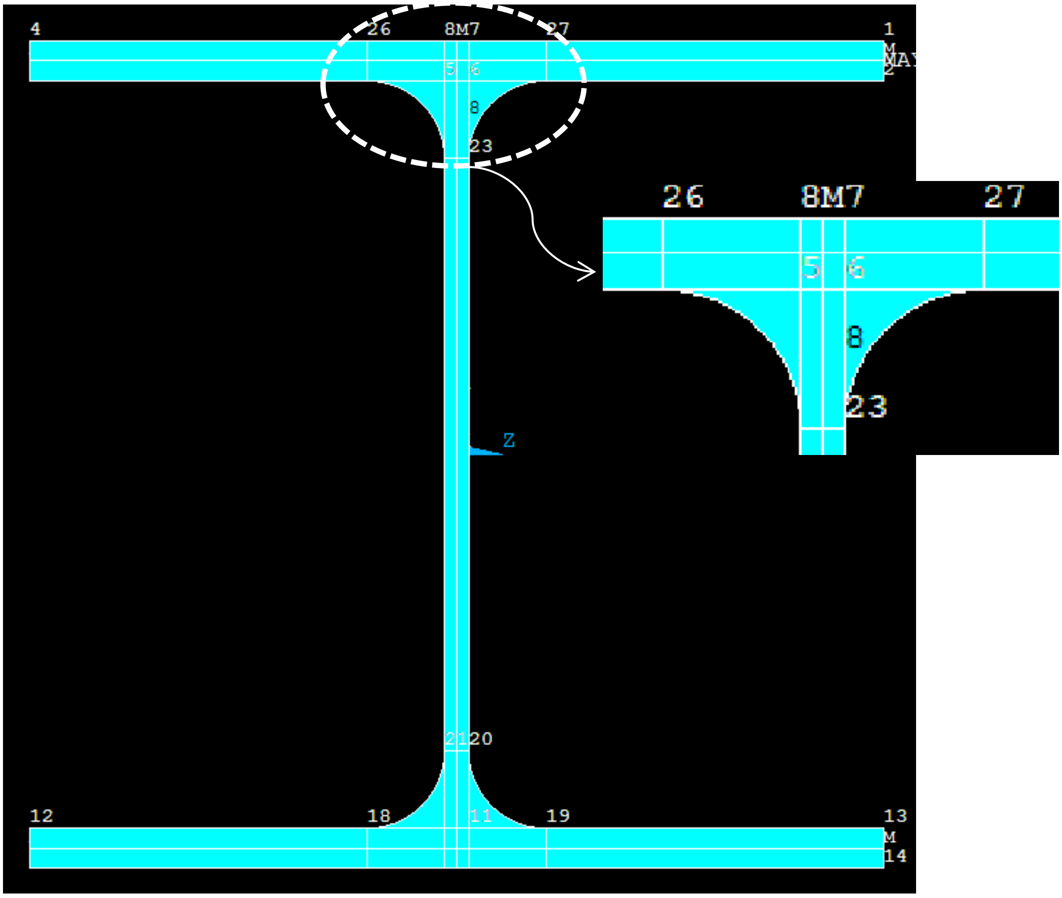


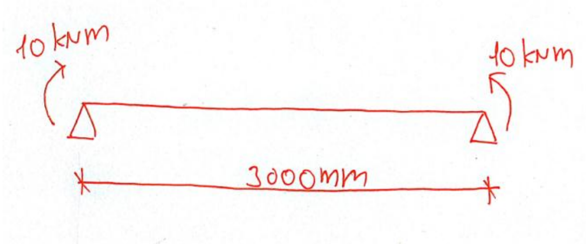
Figure 6-9: Initial mesh with $\mathbf{EY_{fl} = 2\ and\ EZ_w = 2}$

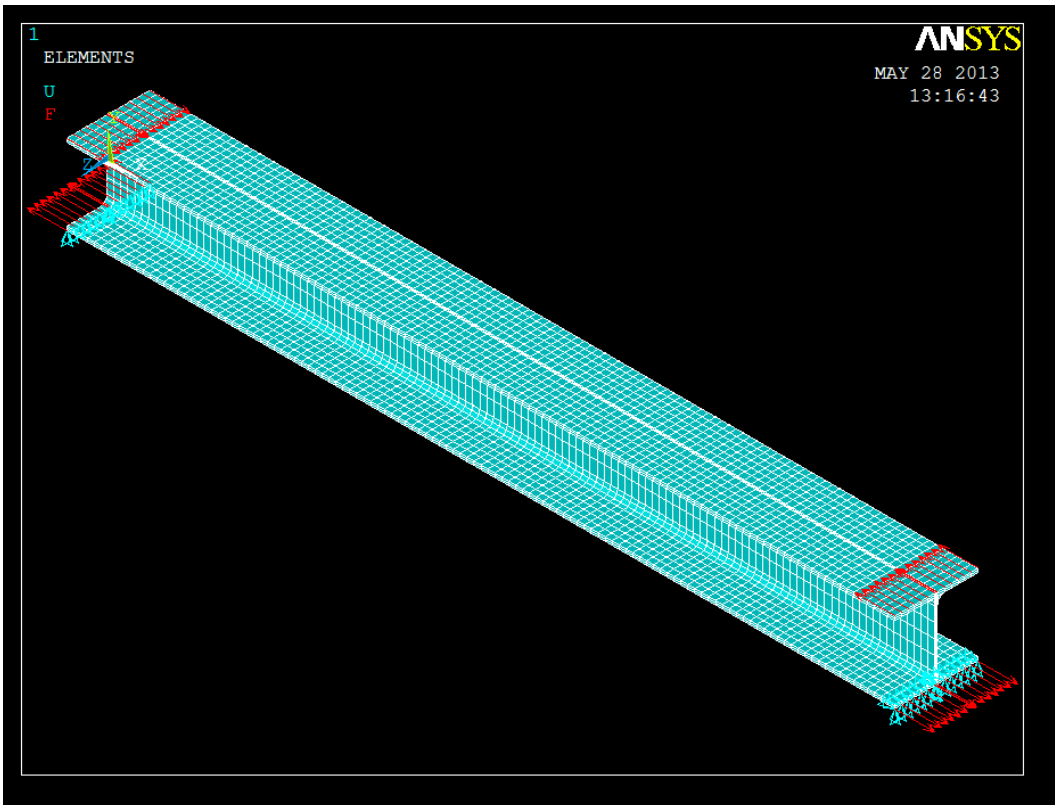
Due to modeling limitation with ANSYS software, the area around the web-flange connection consists of four fixed regions given in the small figure above (table 6-1). Within the boundaries of each region a local mesh refinement will be generated. Meshing the cross-section will be executed in such a way that the EZ of the web coincide with EY of the flange in the red subregion mentioned in $\oint$ 6.7.1. Or in other words, the vertical mesh of the flange coincides with the horizontal mesh of the web in the red subregion will be called from now on *peak-region*. The *peak-region* is confine by the mesh density of the web and the flange. Because the peak stresses are likely to occur in the *peak region*, the meshing components $EZ_{web}\ and\ EY_{flange}$ will have a crucial influence on the deviation of the peak stresses values. Analyzing the stresses results in this report pointed that the minimum mesh for the peak region should be:

$$EY_p = 2\ and\ EZ_p = 2$$

Therefor the initial mesh of every *FEA Model* will be as given in *figure 6-9.*

7. Bending moment





$L_E = 25mm$

$L = 3000mm$

$E = 210e6 \frac{N}{mm^2}$

$v = 0.1$

$M = 10e6\ Nmm$

Figure 7-1: The FEA bending moment model in ANSYS

For bending moment without an additional normal force, the bending stresses will be approached with numerical equation. The normal bending stresses are uniform distributed along the beam length. The characteristic cross-section will be at halfway the beam length in order to neglect the influence of boundary conditions on the stresses. In this cross-section the outer fibers are loaded maximally. The HE-sections have broad flanges to use maximum material in the flanges subjected to bending. Assumption: the magnitude of the applied bending moment will not cause yielding of the steel. In that case plastic deformation will not occur and the stress distribution will stay linear as shown in *figure 7-2*. The applied moment will increase the curvature within the linear elastic stress phase (figure 7-3).

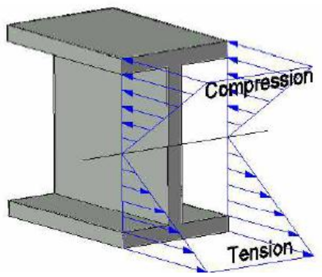


Figure 7-2: Bending stresses [H.Welleman]

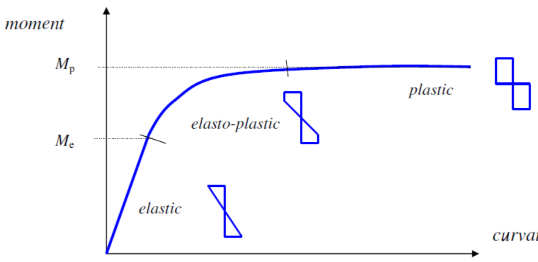


Figure 7-3: Moment (y-axis) versus the curvature (x-axis) for the different stresses distribution [H.Welleman]

When a slender member is subjected to axial force due to flexure the top side and the bottom side will be subjected to axial compression or axial tension force. Because the stated member in *figure 7-1* is not supported in lateral direction, the beam will fail due to lateral buckling of the compression flange. This will occur for a critical value of the flexural load. In addition, if the compression flange buckles laterally, the cross section will also twist in torsion, resulting in a failure mode known as *lateral-torsional buckling.* These two failure mechanism will not be discussed in this research. The governing bending moment load in this case will be small enough preventing the mentioned mechanism to occurring .

7.1 The FEA models

According to the adaptive meshing refinement working method explained in ∯ 6.7, the models will be refined or coarsen according to the obtained results of the first model *"Model 1"*. Analyzing stresses results for the first model emphasis the need for a second model. Therefore three FEA models are introduced from coarse to fine mesh when the third model will be the reference model. (Table 7-1)

FEA bending models	*DOF's flange*	*DOF's web*	*DOF's RoC*	*Total DOF's*
Model 1	351	129	33	963
Model 2	543	225	33	1443
Model 3 (Ref)	1497	507	69	3777

Table 7-1: Bending moment FEA models

7.1.1 Model 1 (963 DOF's)

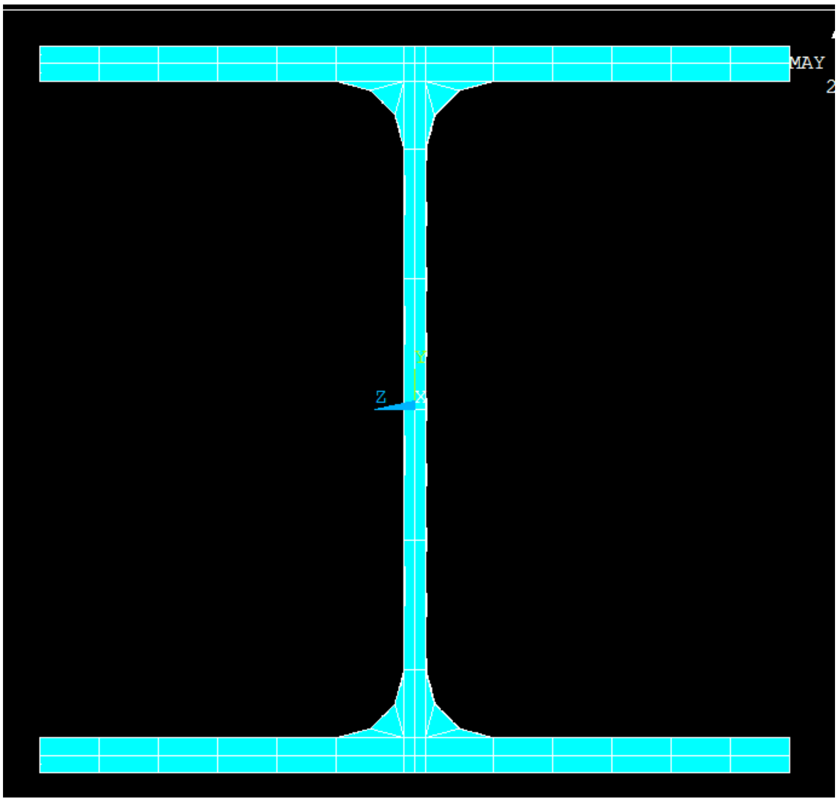


Figure 7-4: The first model

The normal stresses due to pure bending moment are checked for FEA model with the following mesh refinement:

$EY_{web} = 6$ $\quad EZ_{web} = 2$ $\quad b_E^{web} = 4.25mm$ $\quad h_E^{web} = 43.67mm$

$EY_{flange} = 2$ $\quad EZ_{flange} = 12$ $\quad b_E^{fl} = 24.3mm$ $\quad h_E^{fl} = 7mm$

$EY_{peak}^{fl} = 2$ $\quad EZ_{peak}^{fl} = 2$ $\quad b_E^{p} = 4.25mm$ $\quad h_E^{p} = 7mm$ $\quad E_R = 3$

7.1.2 Model 2 (1443 DOF's)

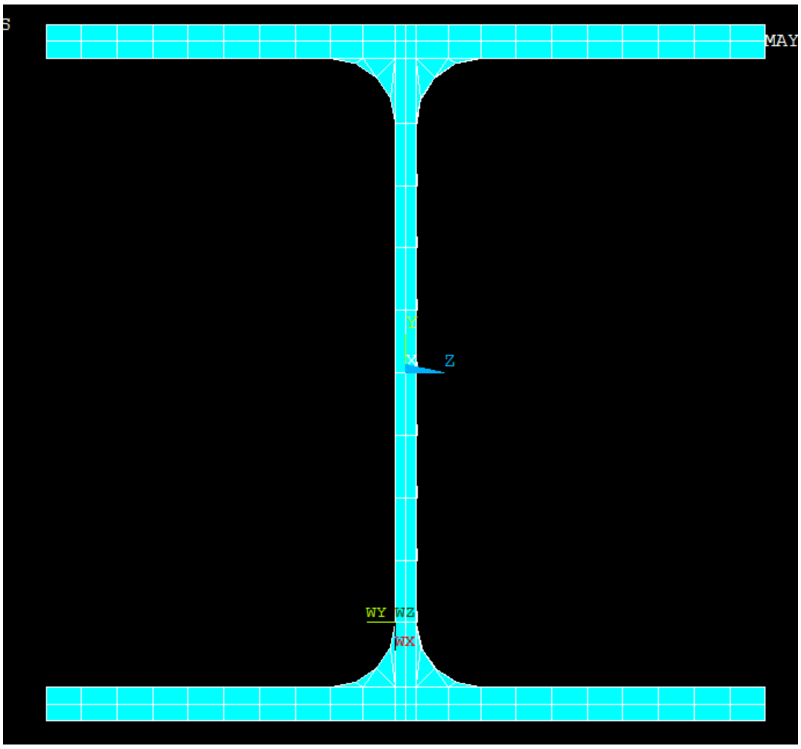


Figure 7-5: The second model

During the FEA of the first model, deviation in the stresses distribution has been found. Yet, this deviation occurs in the web of the cross-section and not in the flange. The deviation can be referred to the element height in the web. As mentioned before, the critical stresses occur in the outer fiber, therefore the normal stresses values in the web are less interesting. This second model has been added to this analysis in order to observe the change in stresses values due to decreasing elements height.

$EY_{web} = 10$ $EZ_{web} = 2$ $b_E^{web} = 4.25mm$ $h_E^{web} = 26.2mm$

$EY_{flange} = 2$ $EZ_{flange} = 20$ $b_E^{fl} = 14.575mm$ $h_E^{fl} = 7mm$

$EY_{peak}^{fl} = 2$ $EZ_{peak}^{fl} = 2$ $b_E^{p} = 4.25mm$ $h_E^{p} = 7mm$

$E_R = 5$

7.1.3 Model 3 (Ref)

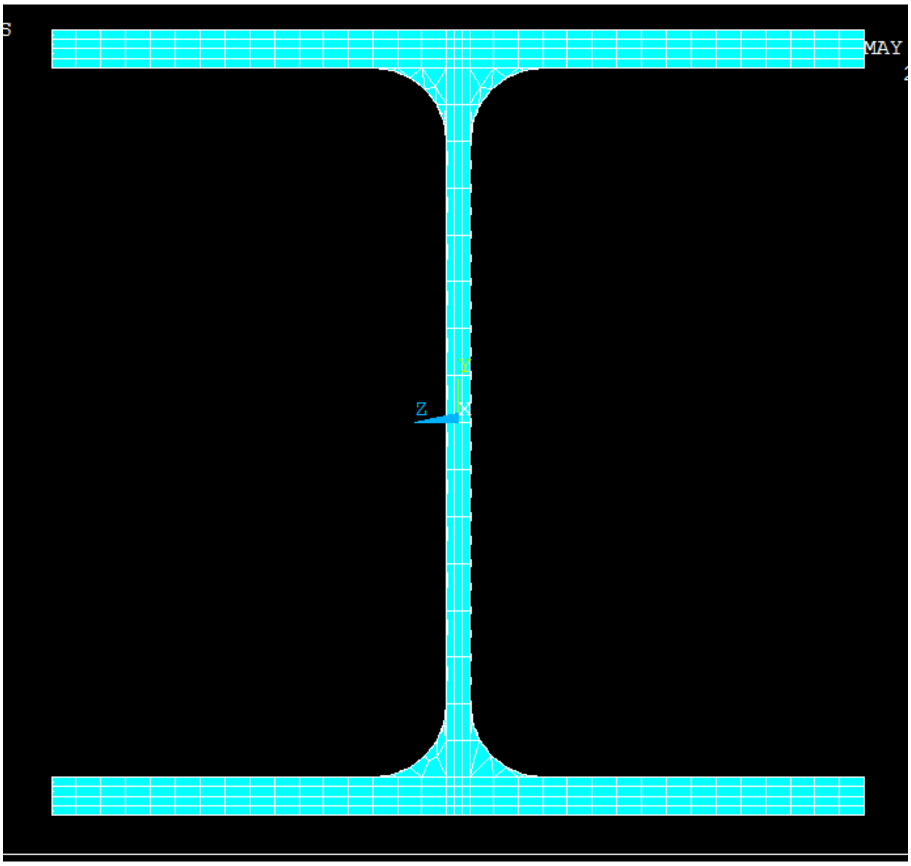


Figure 7-6: The bending reference model

The reference model concerns this loading case involves 3777 cross-sectional DOF. The mesh consists of *Solid186 Hexahedron* and *Prism* elements with the following configurations:

$EY_{web} = 16$ $EZ_{web} = 3$ $b_E^{web} = 2.8mm$ $h_E^{web} = 16.375mm$

$EY_{flange} = 4$ $EZ_{flange} = 32$ $b_E^{fl} = 9.1mm$ $h_E^{fl} = 3.5mm$

$EY_{peak}^{fl} = 4$ $EZ_{peak}^{fl} = 3$ $b_E^{p} = 2.6mm$ $h_E^{p} = 3.5mm$

$E_R{}^1 = 11$

[1] Total number of elements at rounded corners

7.2 Stresses distribution

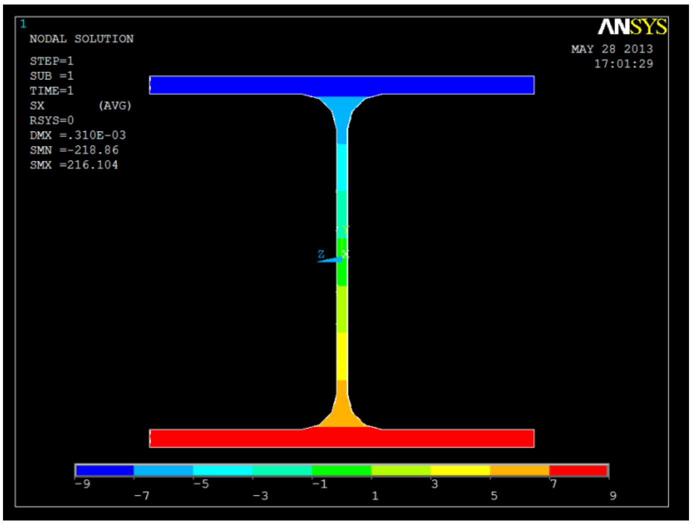


Figure 7-7: Nodal normal stress contour at x=L/2

Only the normal stress components (perpendicular to the cross-sectional plane) are presented in the figure above. This stress contour is a representation of the actual normal bending stresses in the HE-section, in a cross section located half way the beam length. The stress contour shows linear stresses distribution according to the theory. This is conformed in *figure 7-8*. The maximum stresses occur as stated before at the outer fiber, where at the middle of the web the stresses are zero. The nodal normal stress values in *figure 7-8* are plotted for $Z = 0$ and along the y-axis.

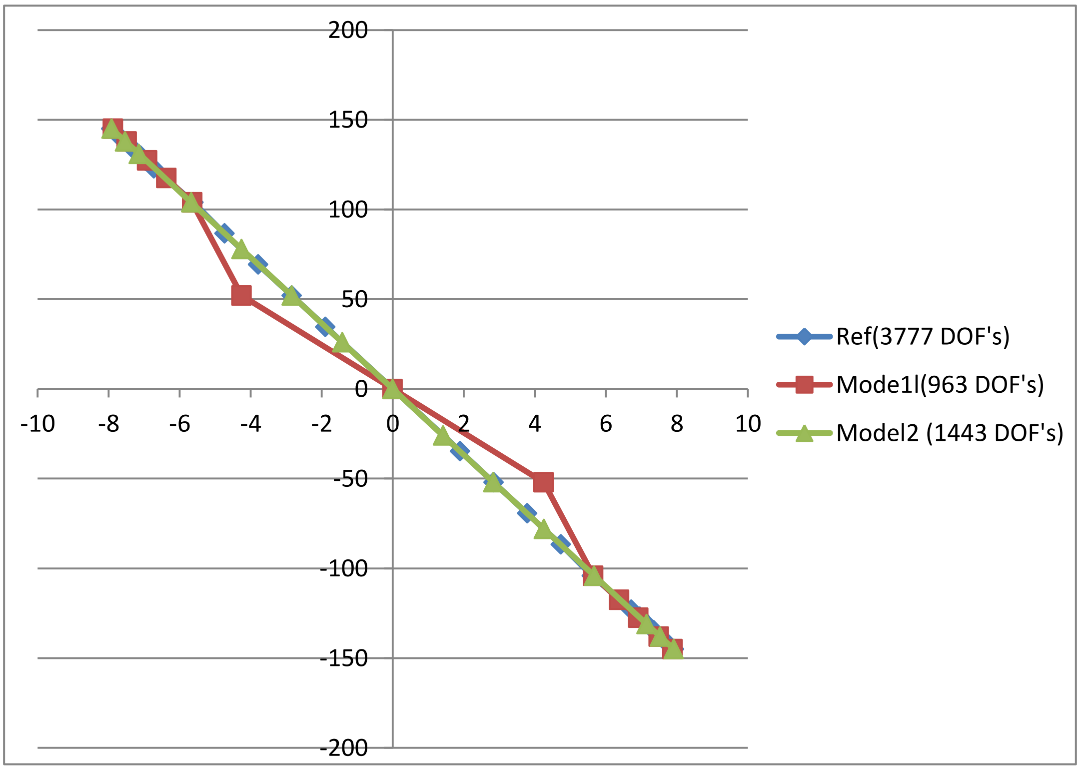


Figure 7-8: Normal stresses for z=0 at x=L/2 for M=10e6 Nmm

Remarkable from the diagram in *figure 7-8* is the accuracy of the critical stresses values at the outer fibre. It seems that *Model 1* and *Model 2* are successfully approaching the actual stresses at the outer fibres. However, as expected the web of model 1 deviate from the actual stresses values due to larger element height. The error for z=0 and y=0.5h is given in the following table:

	Reference	*Model 1*	*Model 2*	*Analytical*
$\sigma_x \quad N/mm^2$	7.9238	7.8766	7.9180	7.94085
Error	-	0.6%	0.07%	0.2%

Table 7-2: Stresses values at (x,y,z)=(0.5L, 0.5h,0) for M=10e6 Nmm E=210e6 N/mm2 v=0.1 and L=3000mm for HE300A section.

The first model succeeds to approach the stress with error smaller than 1%. Adding elements to the web and the flange while unchanging the Red-subregion mesh where the stresses at the outer fibre are measured result in extremely small error 0.07%.

7.3 Stresses distribution in the width

Investigating the stresses distribution along the width of the beam in *model 1* and in *model 2* reveals constant stresses along the flange width.

7.4 Number of elements

7.4.1 Web

As mentioned before, the height of the element has a direct influence on the accuracy of the FEA-model. Since this research is carried out on specific cross section, the cross-sectional dimensions are not discrete. Nevertheless ,it is essential to be able to produce a more global conclusions that can be applied on every HE or I-section. Therefore the ratio $\frac{h_E^{web}}{h_{web}}$ has been plotted with the relative error.

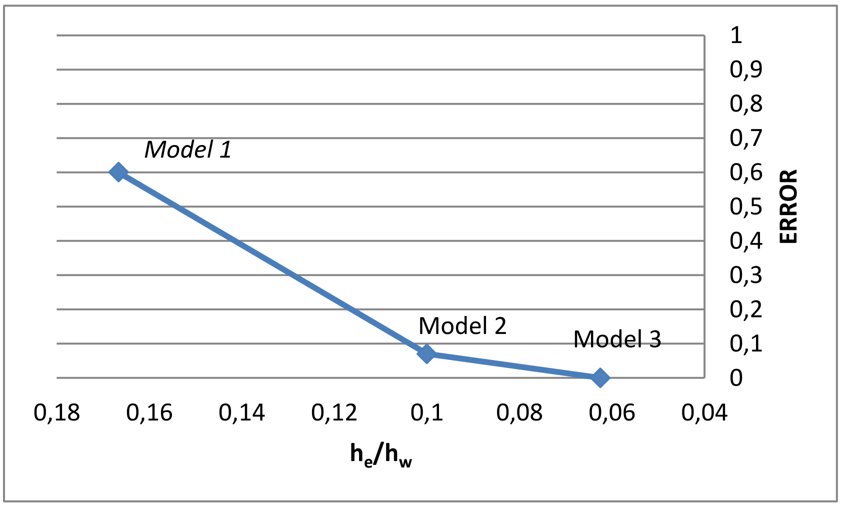


Figure 7-9: Relative error

The error convergences to zero for smaller values of the $\frac{h_E^{web}}{h_{web}}$ ratio with $h_E^{web} \to 0$. This is logical because when the number of elements in the height of the web increases the error will be smaller.

7.5.2 Flange

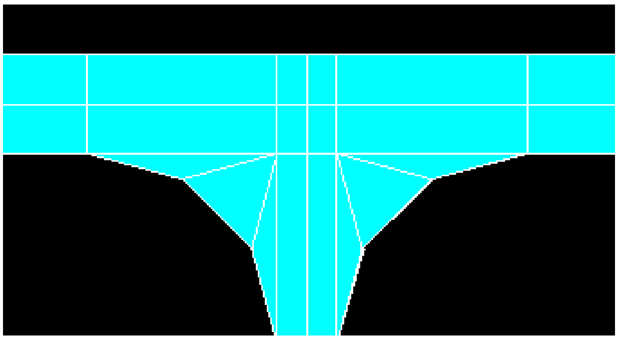


*Figure 7-10: Peak region meshed with 2*2 elements result in up to 1% error*

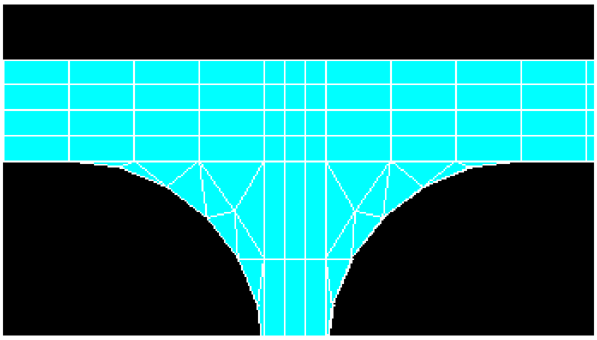


*Table 7-11 Reference mesh with 3*4 elements*

Zooming on the area where the peak stresses are measured showed in *figure 7-10* and comparing this mesh with the reference mesh in *figure 7-11* reveal that $EY * EZ = 2 * 2$ elements with the ratio $\frac{h_E^{fl}}{h_{fl}} = 0.5$ are sufficient in order to achieve error with upper value of 1%. However as mentioned before in ∮ 6.7.2 meshing this *peak region* holds:

$$EZ_{peak}^{fl} = EZ_{web} \; and \; EY_{flange} = EY_{peak}^{fl}$$

As for the number of element in the width of the flange EZ_{fl} . Reducing the EZ_{fl} from 12 to 8 and to 6 result in negligible fluctuation of the peak stresses values. It can be explained due to the constant stresses distribution along the beam width mentioned in ∮ 6.4.

7.6 Conclusions bending moment

During the FEA of the pure bending moment three models are carried out in ANSYS. In the search for the minimal number of elements with minimal deviation from exact stresses values the following Rule of thumb should be used for an HE or I-section subjected to pure bending moment:

- Required mesh concerning $\sigma_{xx,max}$ at the outer fiber for $EZ_w \geq 2$

$\frac{h_E^w}{h_w}$	$\frac{h_E^{fl}}{h_{fl}}$	$Error$
$1/16$	$1/4$	0%
$1/10$	$1/2$	0.07%
$1/6$	$1/2$	0.6%

Table 7-3: Solid186 mesh design table for $\sigma_{xx,max}$

Using the table together with the meshing method mentioned in ∮ 6.7.2 will be explained on the basis of the next example: Given HE300A section, a mesh with accuracy of 1% is required. The cross section is subjected to pure bending moment with linear elastic stresses distribution. The mesh will be determined as followed:

Web: $h_w = 262mm \Rightarrow \frac{h_E^w}{h_w} = \frac{1}{4} \Rightarrow h_E^w = 43.67mm \Rightarrow EY_w = 6\ EZ_w = 2$

$Flange$: $h_{fl} = 14mm \Rightarrow \frac{h_E^{fl}}{h_{fl}} = \frac{1}{2} \Rightarrow h_E^{fl} = 4.25mm \Rightarrow EY_{fl} = 2\ EZ_{fl} = \text{can be arbitrary chosen}$

Because of the relations $EZ_w = EZ_{peak}^{fl}$ and $EY_{fl} = EY_{peak}^{fl}$ the peak region will be:

$EY_{peak}^{fl} = EZ_{peak}^{fl} = 2$. In addition the rounded corners will be modeled with the minimum number of *Prism* elements for a rounded shape $EY_R = 3$

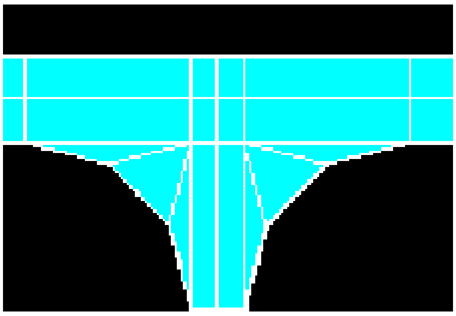
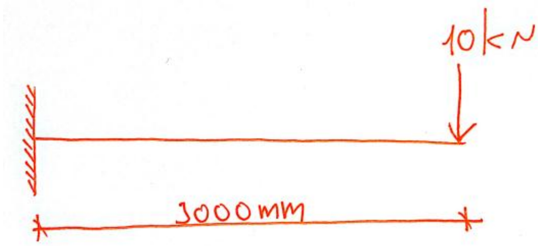
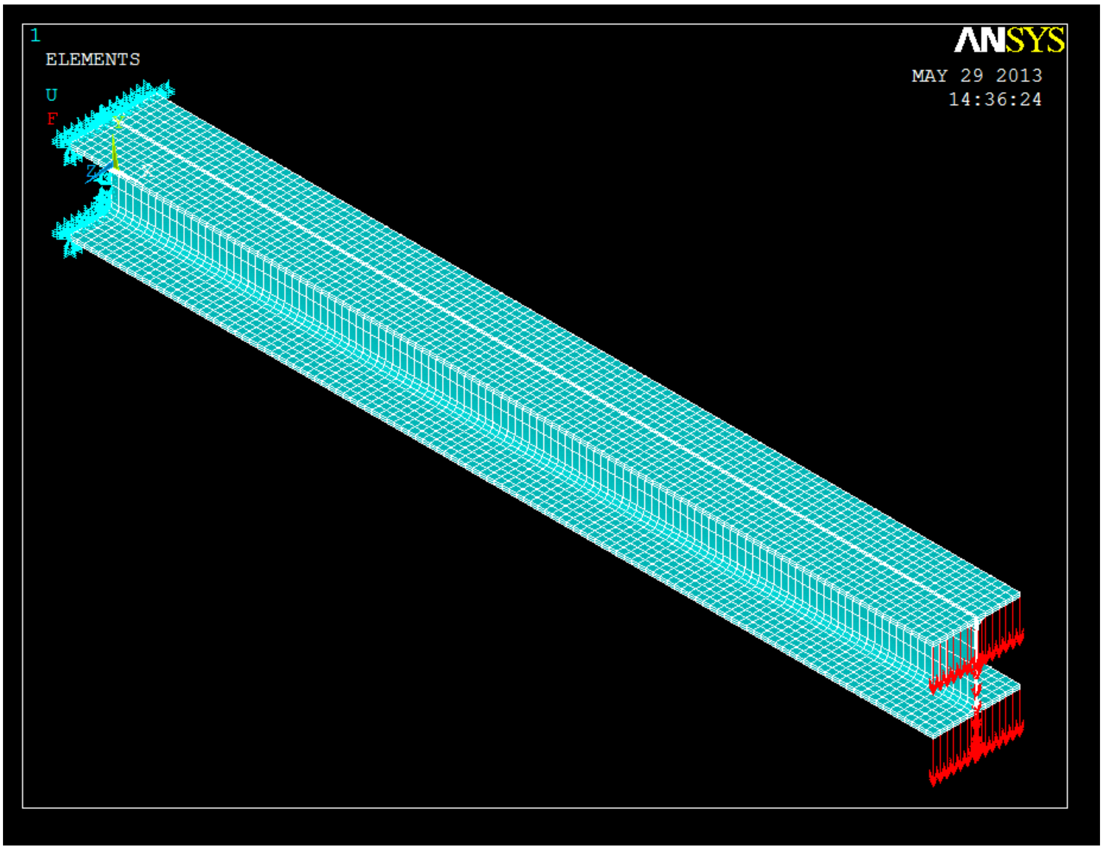


Figure 7-12: Peak region mesh and rounded corners mesh

8. Cross-sectional shear stresses





$L_E = 25mm$

$L = 3000mm$

$E = 210e6 \frac{N}{mm^2}$

$v = 0.1$

$V = 10e3\ N$

$EX = 120$

Figure 8-1: The FEA shear model

Just like the rectangular beam in $\oint 4$,the beam is fixed at the edge for an displacements in the x, y and z direction. At the other edge, vertical concentrated force is applied on each of the nodes modeling shear force of $10e3\ N$. As mentioned before, the critical cross-section will be taken halfway the beam length in order to neglect any irregularities of the stress nearby the boundary conditions and irregularities at the location of the forces.

The shear stress distribution in the HE-section is depending on the type of model which will be analyzed. In the search for the accurate solution, the HE section will be model as close as possible to real HE-section properties. For instance, the web-flange corners in the FEA model will be rounded with the appropriate radius. The shear stress diagram for rounded corners will differ from diagram without rounded corners. This can be seen in *figure 8-2 and figure 8-3.*

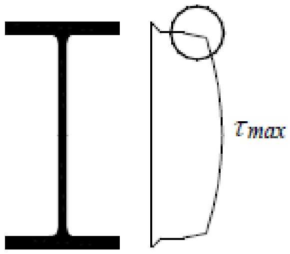


Figure 8-2 Shear stress distribution when RoC=27mm [Reader Staalconstructies CT2035]

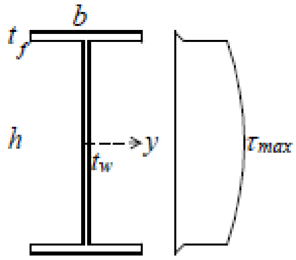


Figure 8-3: Shear stress distribution when RoC=0 [Reader Staalconstructies CT2035]

8.1 The FEA models

Just like in ∮ 7.1 the reference model will have the finest mesh, it will be able to anticipate the exact shear stress values in the cross-section. Broad description of this model can be found at ∮ 7.1. The other models are models that already introduced in ∮ 7.1.

Shear FEA models	*flange DOF's*	*web DOF's*	*Corners DOF*	*Total DOF's*
Model 1	351	129	33	963
Model 2	543	225	33	1443
Model 3 (Ref)	1497	507	69	3777

Table 8-1: Shear stress FEA models

8.2 Shear stress distribution

8.2.1 Evaluating results

The FEA of the *model 1 (* 963 DOF) is carried out in ANSYS. The shear stresses components, τ_{xy} and τ_{xz} ,of each of the nodes are plotted forming the following stress contour:

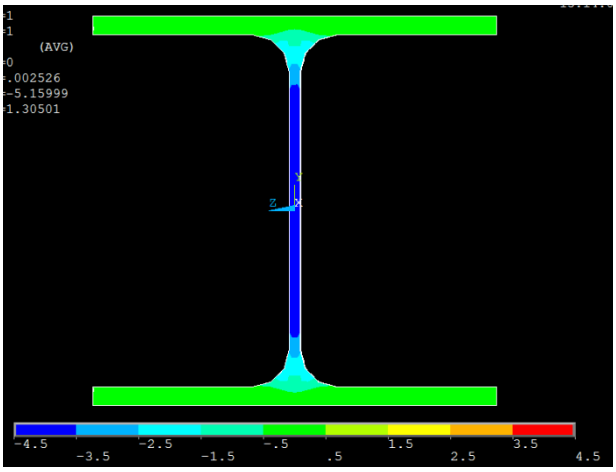


Figure 8-4: XY shear contour

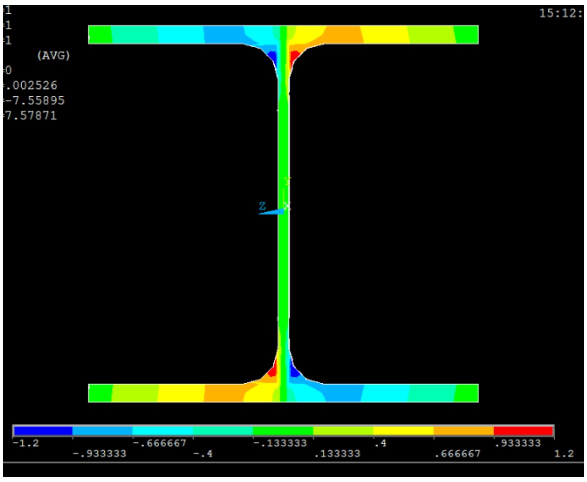


Figure 8-5: XZ shear contour

As described in a previous bachelor thesis of B.M van de Weerd [December 2007] . The shear stresses in the web are much larger than in the flanges. The shear stress distribution in the web should have, according to the theory a parabolic form. Plotting these nodal stresses values in the diagram on *figure 8-6* would confirm the parabolic stress distribution.

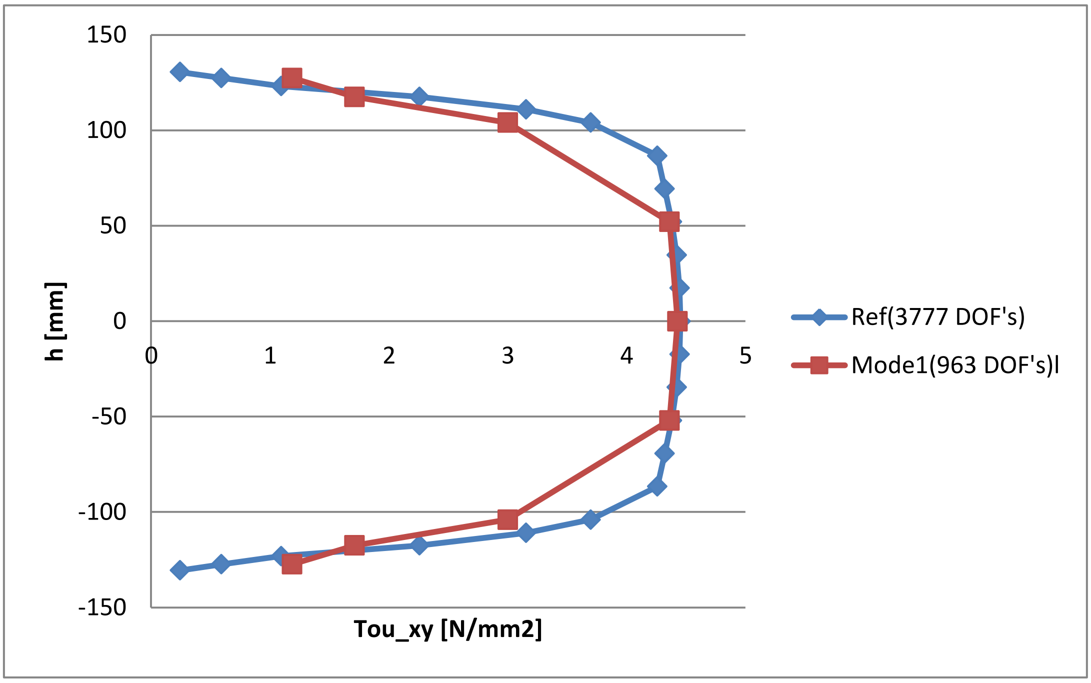


Figure 8-6: Shear stresses distribution in the web

The flanges are subjected to smaller stresses compared with the web. The maximal stress in the web is $4.8\ N/mm\hat{}2$ while the maximal stress in the flange is $0.8N/mm\hat{}2$, as it appears in *figure 8-7*. It seems that the XZ stresses in *figure 8-7* behaving according to the theory, the stresses increasing linear from the edge to nearby the middle of the flange where in the middle the stresses are zero. The dashed black line describes the theoretical XZ stresses distribution in the flange. The different between the numerical solution and the analytical solution (black dashed line) is seen at the middle of the flange. There, the numerical values goes to zero while the analytical value grows continuously. This is due to the absence of rounded corners in the theoretical formulas.

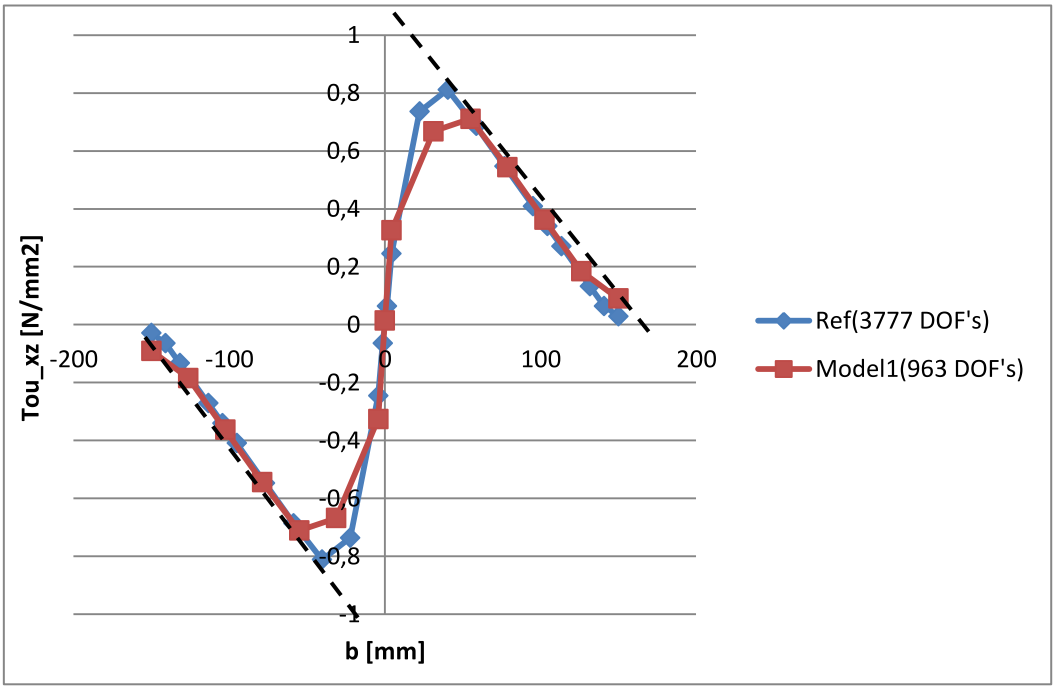


Figure 8-7: Shear stresses distribution in the flanges

The rounded corners exhibit peak stresses higher than the maximum stresses maximum stresses in the flange. Those stresses will be critical in testing HE/I-section on fatigue. The rounded corners of this section are important to increase the fatigue strength of the structure.(Figure 8-7)

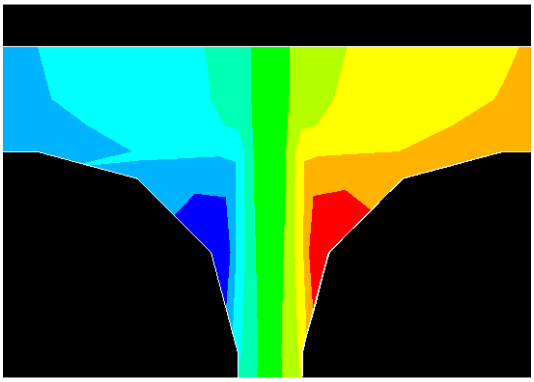


Figure 8-7: XZ shear stresses component

8.3.2 Comparison results

The maximal shear stress value halfway the web height is compared for both models: the reference model (3777 DOF's) and model1 (963 DOF's) to find the relation between number of elements and the accuracy.

$(y = 0, z = 4.25)$	*Reference*	*Model 1*	*Analytical*[2]
$\tau_{xy}\ N/mm^2$	4.45427	4,42867	4.4980
Error	-	0.6%	1.0%

Table 8-2: The relative error for model 1 and the analytical solution from the reference model

Remarkable is that the critical stress in the cross-section is approached with accuracy of 0.6% already by the first coarse model. It can be concluded that the six elements in the height of the web are sufficient to approach the exact stress values.

Although the XZ shear component is not the maximal stress and therefore will not be used in the ultimate limit state calculations, it is significant to be able to approach this stress component with sufficient accuracy. This can be used for example in shear failure analysis. The maximal XZ stress value founded nearby the middle of the flange exhibit the following error:

$(y = 145, z = 47)$	*Reference*	*Model 1*
$\tau_{xz}\ N/mm^2$	0.81129 $(z = 45)$	0.71055 $(z = 55)$
Error	0%	12.4%

Table 8-3: The relative error of model 1 from the reference model

It can be assumed that the cause for the relative large is the number of elements in the width of the flange EZ_{fl}. To confirm this assumption a new model will be created.

[2] For simplified cross-section with RoC=0 according to $\oint$ 4.3

8.3.3 Improved model

$Model\ 2: EY_{fl} = 2 \quad EZ_{fl} = 20 \quad b_E^{fl} = 14.575mm \quad h_E^{fl} = 7mm$

In order to reduce the error observed in the first model, a new model is introduced. *Model 2* is a model being used in $\oint 7.2.2$ to model a beam subjected to bending moment. This model includes a finer mesh from *Model 1.* This will be used to track changes in the error due to increasing number of element in the width of the flange. For $\Delta EZ = +8$ the following green stress distribution is added to *figure 8-7:*

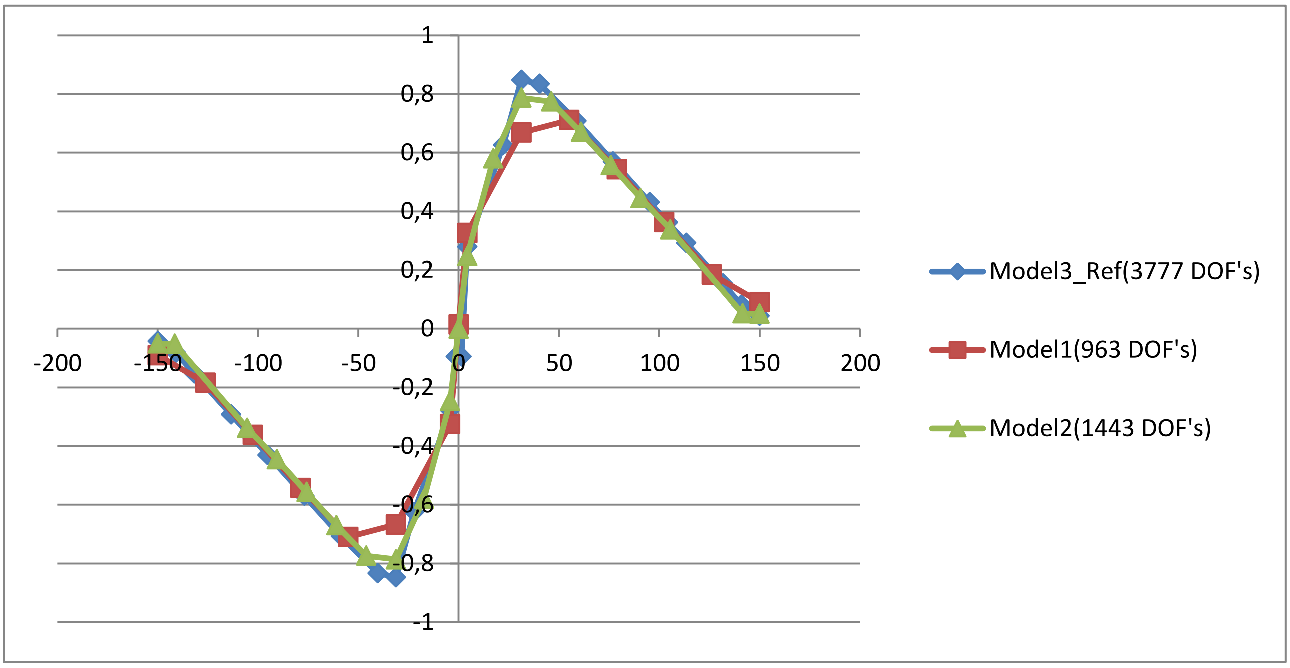


Figure 8-8: Improved shear stress distribution in green

According to the second model, it seems that the results of the new model are closer to the reference values. Accordingly, the most important conclusion is the link that established between the magnitude of the error and the number of elements in the width of the flange. The improved model with $\Delta EZ = 8$ reduc es the error in 7.80% to 4.62%.

8.4 Von Mises

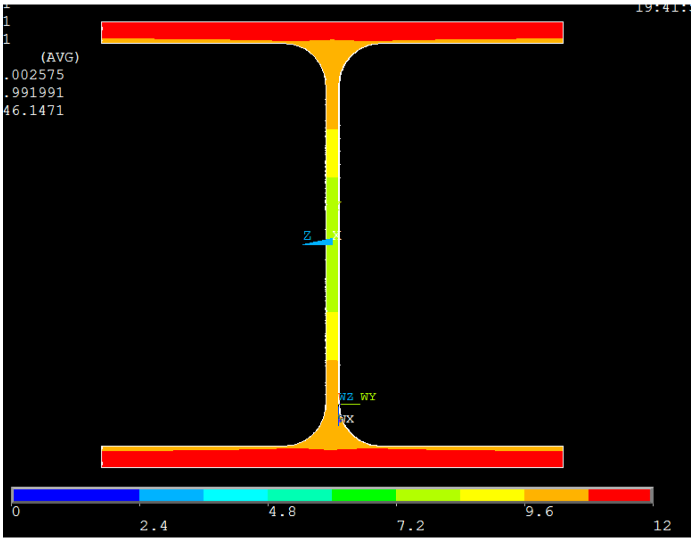


Figure 8-9: Von Mises stresses at $\mathrm{x} = \mathrm{L}/2$

Until now, the shear forces in the XZ direction and in the XY direction have been separately analyzed. The problem in this approach occurs in the region where those two stress components are working. Von Mises approach suggests a scalar quantity where both of the shear components are processed. This is the so-called *Von Mises yielding criterion* which is favorable to use with ductile materials as steel. It holds:

$$\sigma_{VM} = \frac{1}{\sqrt{2}}\sqrt{\left(\sigma_{xx} - \sigma_{yy}\right)^2 + \left(\sigma_{yy} + \sigma_{xx}\right)^2 + (\sigma_{zz} - \sigma_{xx})^2 + 6\left(\sigma_{xy} + \sigma_{yz} + \sigma_{xy}\right)^2}$$

For combination of shear and bending moment stresses valid for the stated model it holds:

$$\sigma_{VM} = \sqrt{\sigma_{xx} + 3\tau^2}$$

8.4.1 Flange

The maximal Von Mises stress in the middle of the flange at $(y = 145, z = 0)$ is compared for both models with the reference model. The relative error is presented in *table 8-4*:

	Reference	*Model 1*	*Model2*
σ_{VM}	11,8781	11,5985	11,8583
Error	0%	2,4%	0,2%

Table 8-4: Von Mises yielding criterion flange

8.4.2 Rounded corners

The rounded corners are modeled with *Solid186 Prism* 20-nodes elements. In each model the number of elements is increased and the Von Mises values are noted. The Error is presented in *table 8-5:*

	Reference	*Model 1*	*Model 2*
VM	10,6660	10,5020	10,6590
Error	0%	1,53%	0,06%

Table 8-5: Von Mises yielding criterion curvatur

8.5 Number of elements

8.5.1 Web

As for the web of the HE300A section, it is noticed that with $EY = 6$ and $EZ = 2$ the maximal shear force can be approximated with error of 0.6%. The required number of elements for error with upper value of 1% is 6. This valid for $\frac{h_E^w}{h_w}$=0.167. The relative error with respect to the web dimensions and the number of elements is given in *figure 8-10*:

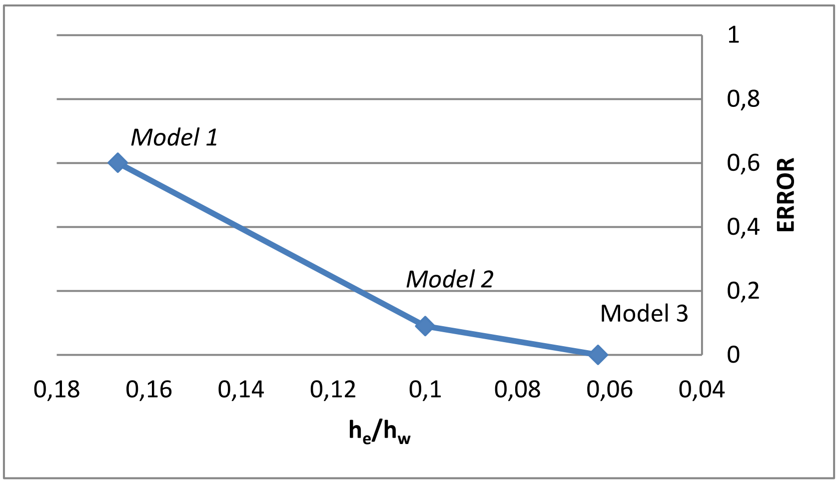


Figure 8-10:* XY *error

8.5.2 Flange

Although the flanges are not experience the maximal shear stress and the fact that the structural engineer will prefer to use the maximum shear value at the web in order to meet the Ultimate Limit State requirement, modeling the flange will be necessary for analyzing the shear failure of the beam. Therefore, it is important to be able to approach the maximum stress value at the flanges as accurate as possible. (Figure 8-11)

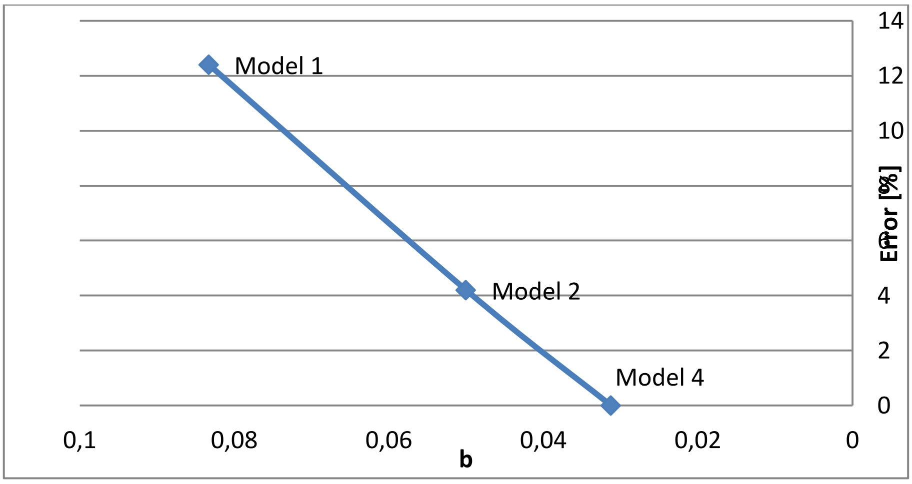


Figure 8-11:* XZ *error

Error of 4.2% will be achieved with $EZ_{fl} = 20$ for HE300A section

The error of the maximal stress in the flange according to Von Mises criteria is present in *figure 9-12*. It is remarkable that by applying Von-Mises the error of is dramatically decreased. It might be more convenient to determine the required number of elements in the flange with the Von Mises approach.

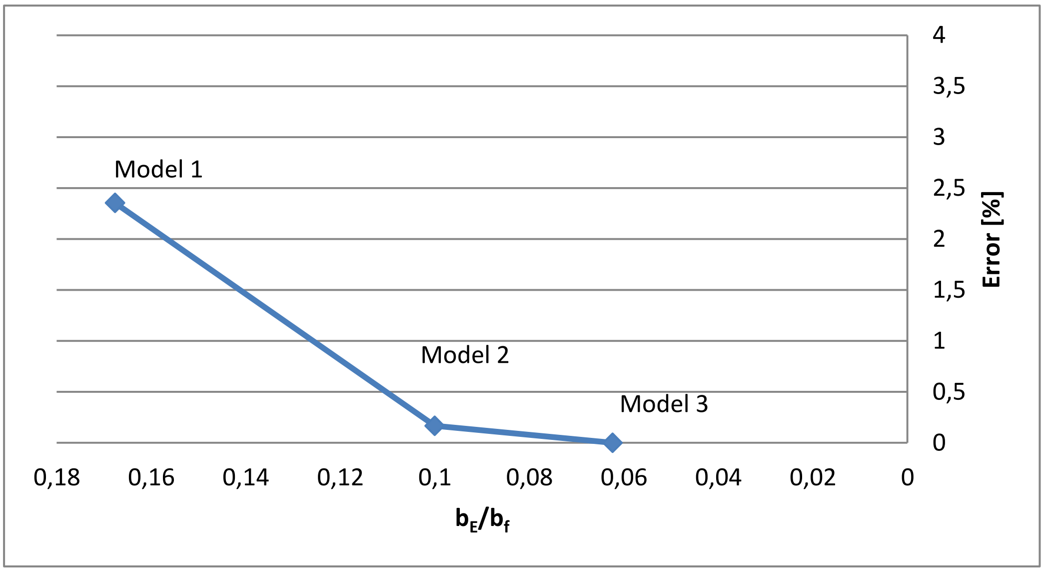


Figure 8-12: Von Mises error flange peak stress region

8.5.3 Rounded corners

With the *criterion of Von Mises*, the critical stresses in each of the models are compared for the rounded corners of the HE300A section. As expected, there is relation between the number of *Prism* elements in the corners to the error (table 8-6*)* .With only 3 *Prism* elements on an area of $163mm^2$ error of 1.5% is observed.

	Model 3(Ref)	*Model 2*	*Model 1*
VM	10,666	10,659	10,5023
Element	11	5	3
Error	0%	0,06%	1,5%

Table 8-6: Von-Mises error for rounded corners

A conclusion regarding this comparison might be that 3 elements are sufficient for radius of 27mm. Yet modeling an area with only 3 elements will change the shape of the rounded corners and as result the cross-sectional properties will be changed. Therefore it is recommended for each stated section to model the rounded corners with the minimal number of element required to model the corners without damaging the shape of it. (Figure 8-13 and Figure 8-14)

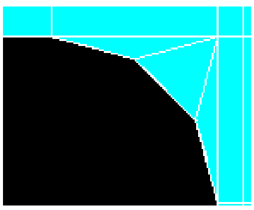


Figure 8-13: 3 Prism elements

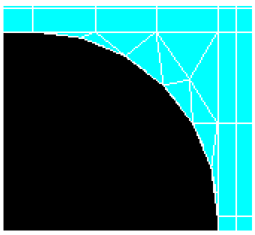


Figure 8-14: 11 Prism elements

8.6 Conclusion shear stress

During the FEA of fixed beam subjected to shear force, the cross-sectional shear stress components have been successfully modeled in ANSYS. The stresses showed similar distribution according to the theory. The maximal shear stress at the web of the cross-section was relative easy to approach with the first coarse model. However the stresses at the flanges and at the rounded corners needed a finer model to understand the change in accuracy due to discrete number of elements in the width of the flanges. In addition the Von Mises criterion has been applied in the search for the error in the middle of the flanges and the error in the rounded corners. The following mesh should be applied:

- Required mesh at the web concerning τ_{max} for $EZ_w \geq 2$

$\frac{h_E^w}{h_w}$	*Error [%]*
$1/6$	0.6
$1/10$	0.09
$1/16$	$< \varepsilon$

Table 8-7: Solid186 mesh design table for $\boldsymbol{\tau_{max}}$

- Required mesh at the flange concerning $\tau_{fl,max}$ and $\sigma_{VM,fl}$ for $EY_{fl} \geq 2$

$\frac{b_E^{fl}}{b_{fl}-t_w}$	$Error\ \tau_{fl,max}$ *[%]*	$Error\ \sigma_{VM}$ *[%]*
$1/12$	12.4	2.35
$1/20$	4.2	0.17
$1/32$	$< \varepsilon$	$< \varepsilon$

Table 8-8: Solid186 mesh design table for error $\boldsymbol{\tau_{fl,max}}$

Please note, the errors registered in the *table 8-7* and *table 8-8* related to the maximal possible error. For example: Given HE300 cross-section subjected to shear force within the linear elastic stress distribution phase. The required error is 1%. Therefore the mesh using these tables with respect to the maximal stress will be:

$$Web: h_w = 262mm \Rightarrow \frac{h_E^w}{h_w} = \frac{1}{6} \Rightarrow h_E^w = 43.67mm \Rightarrow EY_w = 6$$

$$Flange: b_f = 300mm \Rightarrow \frac{\mathrm{b_E^{fl}}}{\mathrm{b_{fl}} - t_w} = \frac{1}{12} \Rightarrow \mathrm{b_E^{fl}} = 24.3mm \Rightarrow EZ_{fl} = 12$$

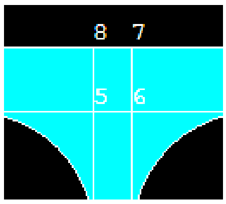


Figure 8-15:Peak

The stresses at the flange are not the maximal stress value and therefore the most course mesh density has been chosen for the flanges. In addition according to the meshing method in ∮ 7.7.2 the peak region enclosed by nodes 5,6,7 and 8 in *figure 9-15* would be: $EY_p^{fl} = EY_{fl} = 2$ and $EZ_p^{fl} = EZ_w = 2$. As for the rounded corners three elements as given in *figure 9-16* are sufficient according to *table 9-5.*

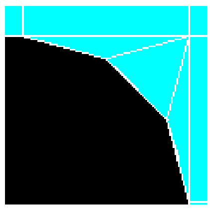
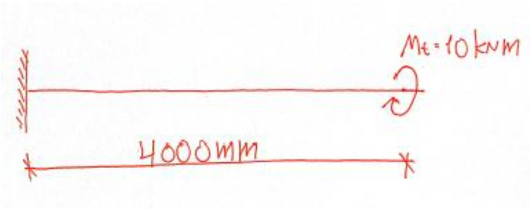
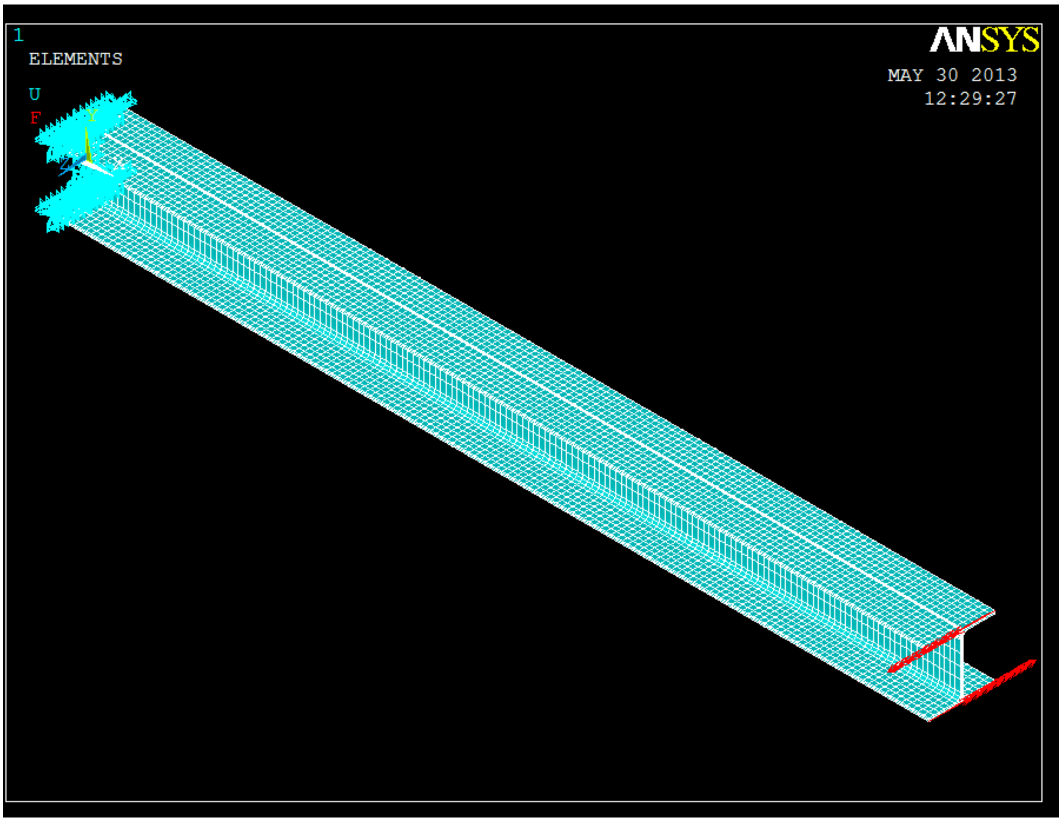


Figure 8-16

9. Torsion





$L_E = 25mm$

$L = 4000mm$

$E = 210e6 \dfrac{N}{mm^2}$

$v = 0.1$

$M_t = 10e6\ Nmm$

$EX = 160$

Figure 9-1: The torsion FEA model

The 3D FEA torsion model of the HE300A section is presented in *figure 9-1*.. In $\oint 3.1.3$, the ideal length of the beam subjected to torsion load have been founded according to the theory of Valsov. The FEA beam model carried out in ANSYS will has to be long enough to omit the warping resistance and irregularities concerning the boundary conditions and the concentrated forces. Cross-section halfway the beam length should meet these requirements. Therefore, in this FEA-model the length is increased from $3000mm$ to $4000mm$ to ensure the reliability of the results. In $\oint 4$,the rectangular beam subjected to torsion have been modeled in ANSYS and compared with the torsion table values. Table values however, do not exist for HE sections. Therefore, in this analysis reference model will be used. From the literature it's well known that HE or I-sections are inefficient in carrying torsion. Basic principles known from earlier researches, is that the mesh density for torsion models are finer than mesh density for shear or bending models. Thus, in this chapter new reference model has been added with a fine mesh. According to the theory, the torsion stresses are extreme at the outer fiber for an open cross-section. Therefore in an open cross-section like the HE300A the stresses will be measured in the nodes located at the outer fiber.

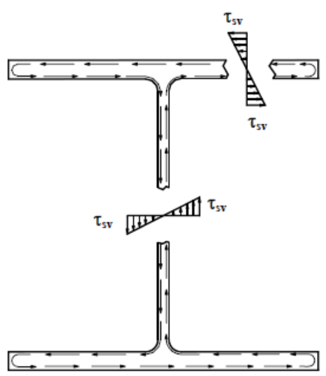


Figure 9-2: Torsion stresses

9.1 The FEA models

During the research, the need to refine the mesh occurs several of times. This results in additional models comparing with the other loading cases. The models in *table 9-1* presented from the coarse to the fine mesh:

Torsion FEA models	*flange DOF's*	*web DOF's*	*Corners DOF's*	*Total DOF's*
Model 1	351	129	33	963
Model 2	543	225	33	1443
Model 3	747	369	36	2007
Model 4	1497	507	69	3777
Model 5 (Ref)	3399	1821	126	9123

Table 9-1: Torsion FEA models

Models 1 and 2 are already introduced in $\oint$ 7.1. Model 4 was the reference model in $\oint$ 7 and $\oint$ 8 . Models 3 and 5 are new models and therefore will be introduced in the next paragraph.

9.1.1 Model 3 (2007 DOF's)

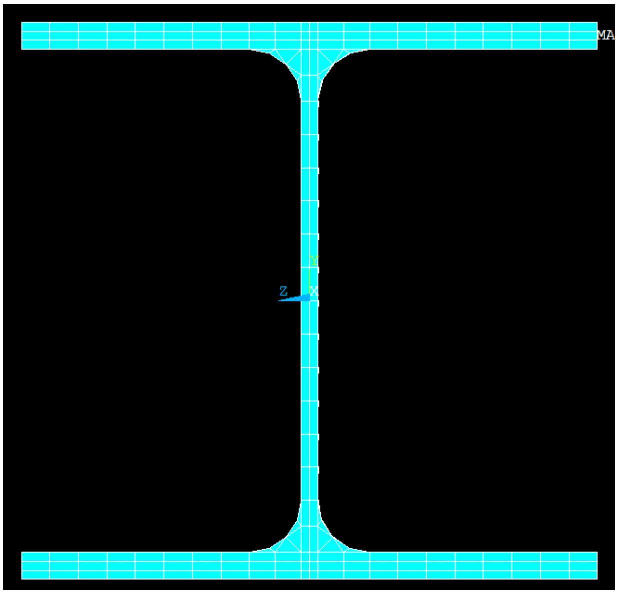


Figure 9-2: Cross-section model 3

$EY_{web} = 16$ $\quad EZ_{web} = 2$ $\quad b_E^{web} = 4.25mm$ $\quad h_E^{web} = 16.375mm$

$EY_{flange} = 3$ $\quad EZ_{flange} = 20$ $\quad b_E^{fl} = 14.575mm$ $\quad h_E^{fl} = 4.67mm$

$EY_{peak}^{fl} = 3$ $\quad EZ_{peak}^{fl} = 2$ $\quad b_E^{p} = 4.25mm$ $\quad h_E^{p} = 4.67mm$

$E_R = 6$

9.1.2 Model 5 (9123 DOF's)

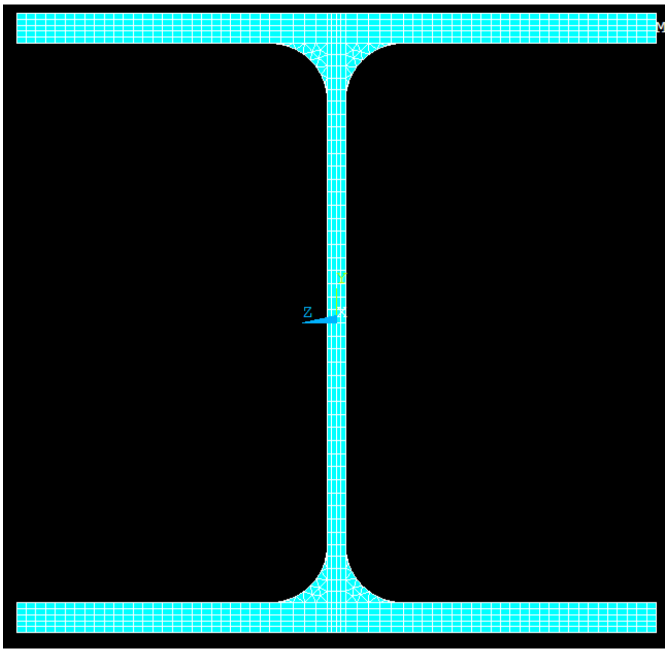


Figure 9-3: Cross-section reference model

$EY_{web} = 44$ $\quad EZ_{web} = 4$ $\quad b_E^{web} = 2.125mm$ $\quad h_E^{web} = 5.95mm$

$EY_{flange} = 5$ $\quad EZ_{flange} = 62$ $\quad b_E^{fl} = 4.7mm$ $\quad h_E^{fl} = 2.8mm$

$EY_{peak}^{fl} = 5$ $\quad EZ_{peak}^{fl} = 4$ $\quad b_E^{p} = 2.125mm$ $\quad h_E^{p} = 2.8mm$

$E_R = 19$

9.2 Torsion stresses distribution

9.2.1 Evaluating results

The finite element analysis is carried out in ANSYS. The contour plots for *Model 5(Ref)* concerning both stress components are given bellow in *figure 9-3* and *figure 9-4:*

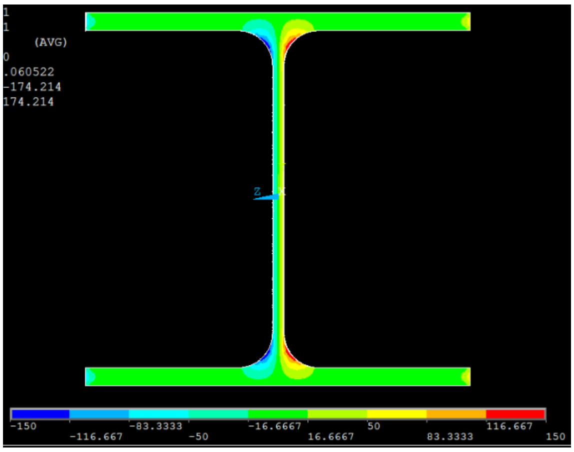


Figure 9-3 : XY contour plot for model 5

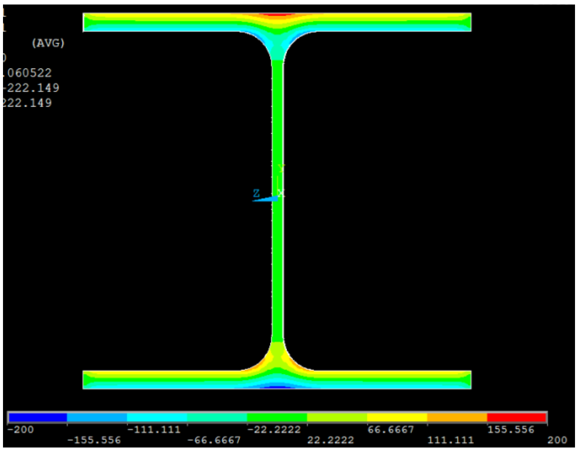


Figure 9-4: XZ contour plot for model 5

From the theory it is known that torsion stresses are proportional to the thickness of the open cross-section. As for the HE300A section, the flanges are $14mm$ thick while the web is $8.5mm$ thick. Therefore, the largest stresses are expected in the outer line of the flanges. This can be confirmed by the contour plot of *figure 9-4* and *figure 9-5* where *the* maximal shear stresses are registered at the flanges outer line, in accordance with the theory. These stress peaks at the middle of the flange will exhibit the critical stresses in the cross-section. In addition just like mentioned before in $\oint 8.21$ there are peak stresses at the rounded corners. These stresses are combination of the XY and the XZ stress components. In *figure 9-5* and *figure 9-6* the contour lines indicating peak stresses at the top of the flange and at the rounded corners. It is noticed that the rounded corners experiences peak stresses also due to shear force (*figure 9-7).* Combining these results concludes that the rounded corners are certainly an critical point in the section where a mesh with high accuracy should be used. A possible question arises while observing the contour lines, how many elements in the peak regions will be sufficient to be able to approximate the stresses with high accuracy? and is it reliable to address each stress component separately in a region where those two stress components are present? To answer the second question an additional method to observe the stresses will be checked. Reasonable method concerning steel construction is suggested by Von-Mises. Applying the Von Mises stress criteria to estimate the accuracy of the peak stresses will be suitable, since it plots both stress components together. In the search for the optimal mesh density in the discrete beam the number of elements in the flange, web and in the rounded corners according to the Von-Mises yielding criterion will be therefore used.

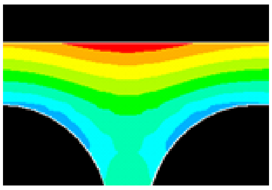


Figure 9-5: Torsion XY Peak

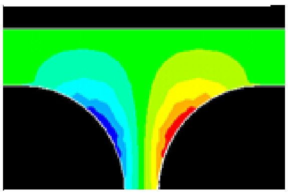


Figure 9-6:Torsion XZ peak

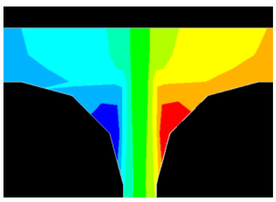


Figure 9-7: XZ shear peak

. But beforehand, the stresses solutions of the five models will be plotted in a diagram representing the nodal stress values along the web and along the flange of the cross-section. The stress values regarding cross-section at the middle of the beam. Due to symmetry f*igure 9-7* will present the XZ stress components of the left side of the upper flange.

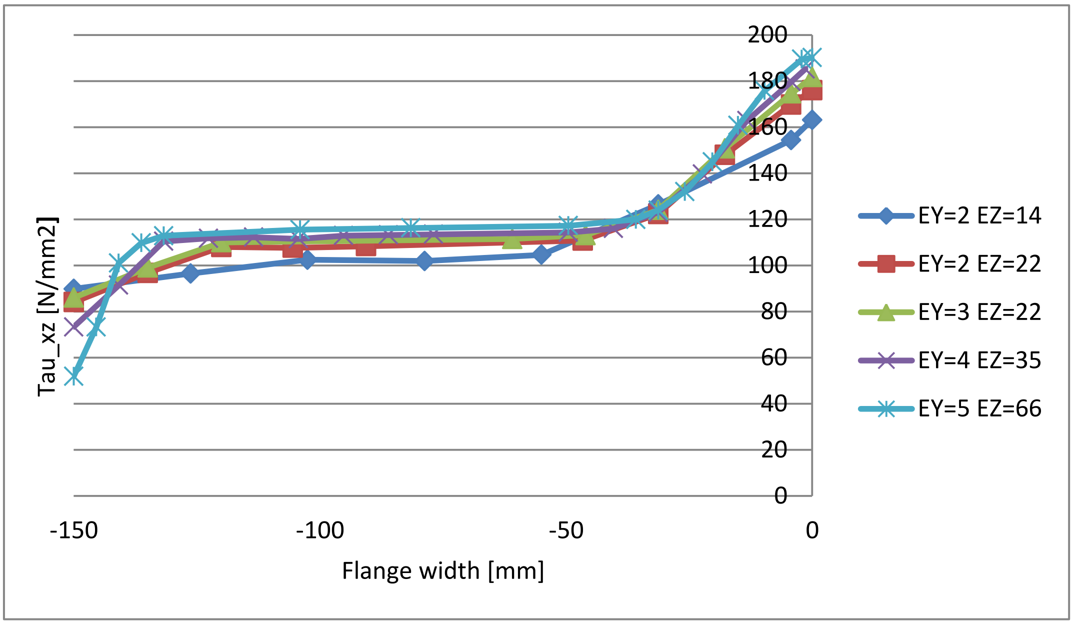


Figure 9-7: XZ stress distribution for the left side of the upper flange

Looking at the maximal stress values of each model amplifies the relation between the number of elements in the width of the flange and the on the accuracy of the results. This relation will be later on established in the conclusion. In *figure 9-7* the maximal stress of 188N/mm^2 is watched in the middle of the flange at Z=0 for the reference mesh. The coarser meshes are approaching this value as the mesh will become denser. *Figure 9-8* gives the XY stress component for the upper half of the web. It is noticed that the stresses at the web are approximately one third from the maximal stresses at the flanges. Nearby the rounded corners the XY stresses are maximal. It is remarkable that the number of elements in the height of the web EY having almost no influence on the stress values. Good results can be achieved already with the first model, with $EY = 6$. The critical point in this diagram is at $= 120$, where the stresses deviate from each other even when finer mesh is applied . This problem will be solved with the help of the Von Mises criterion in the next paragraph.

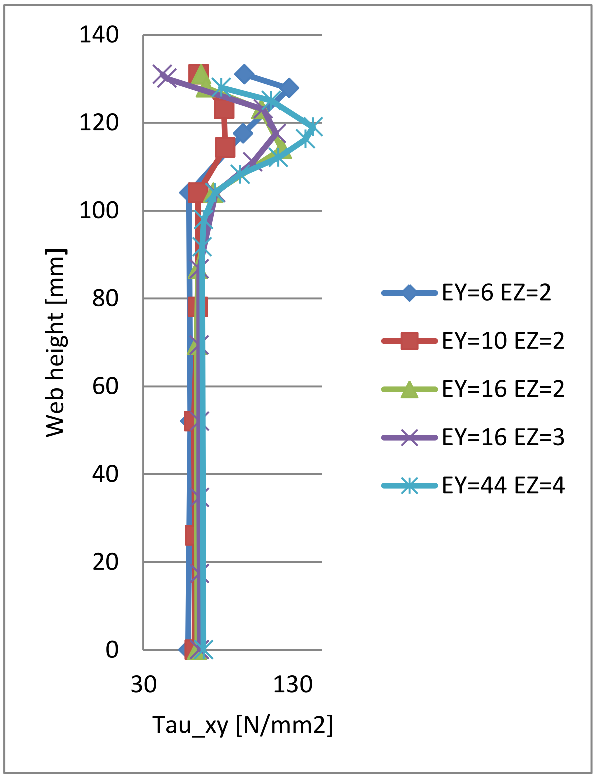


Figure 9-8: XY stresses distribution for the upper half of the web

9.2.2 Von Mises

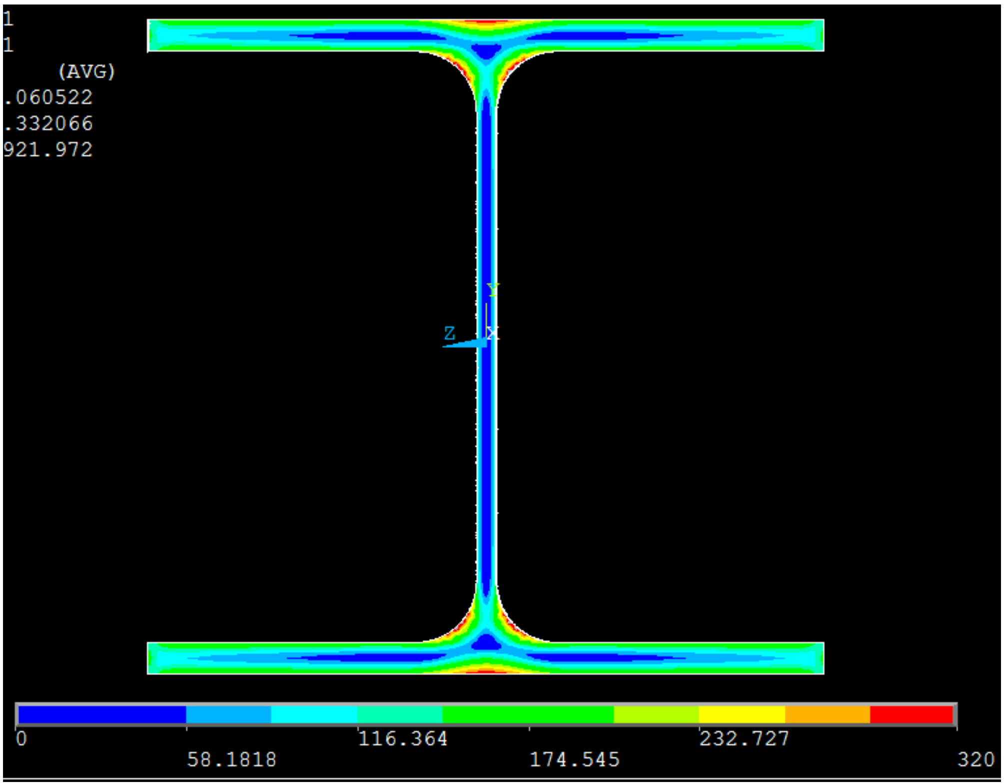


Figure 9-9: Von-Mises plot contour at $\mathrm{x} = \mathrm{L}/2$

The Von Mises yield criterion introduced in $\oint 8.4$ for the shear loading case is favourable to use in the case of torsion of steel. In *figure 9-9* the Von Mises equivalent stresses are plotted in for a cross-section halfway the beam length. Comparing with the contour plot in $\oint 9.3$, this contour plot offers a straight forward method to observe the stresses in the entire cross-section. The contour lines at the flange-web connection indicate that the stresses in the top of the flange and in the rounded corners are more or less the same. This can be also explained by applying the *Membrane analogy* by *Ludwig Prandtl.* It describes the proportions between the torsion stresses and the slope of a soap film of non-circle cross-sections. The slope at these locations will then have to be more or less the same. Thus, while testing the HE/I member subjected to torsion load in the Ultimate Limit State approaching the exact stresses values in these stated peak regions will be essential. The failure mechanism will probably occur due to yielding of these peak regions. The data obtained from the Von Mises contour plot will be processed and evaluated in the next paragraph regarding the required mesh refinement.

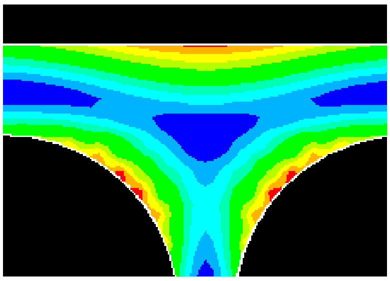


Figure 9-10: Peak

9.3 Number of elements

9.3.1 Flange

In relation to the required number of elements with respect to the accuracy obtained by the mesh *figure 9-11* is given. The diagram shows the deviation of the maximal stress value at the middle of the flange for each model.

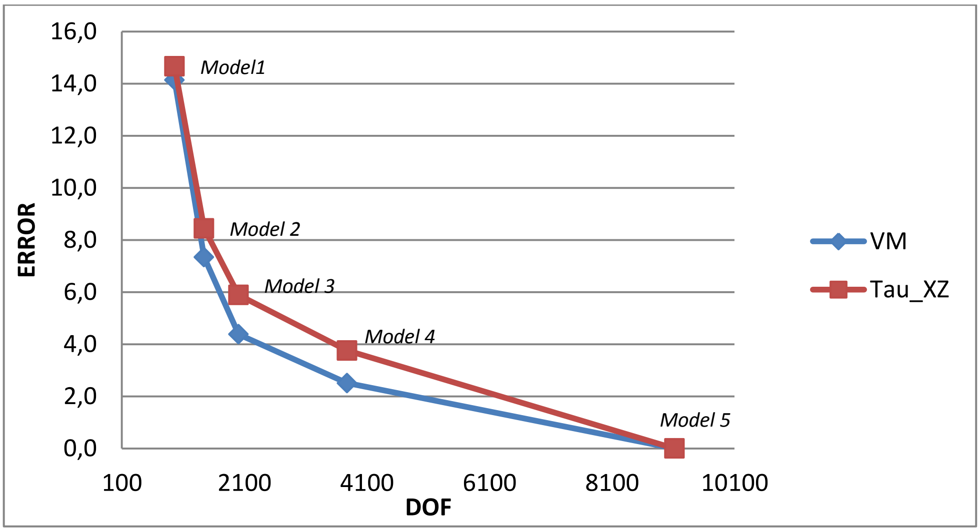


Figure 9-11: Error as function of the total DOF for the maximal stress

Looking at the distribution of error versus the cross-sectional degrees of freedom suggest that the error goes to infinity, or at least to real large value for a coarser model than the *Model 1* (936 DOF). It also seems that the Von-Mises and the XZ stress components behave the same. This confirms that the founded error distribution is correct. To determine the required number of elements in the flanges the following ratios: $\frac{b_E^{fl}}{b_{fl}}$ and $\frac{h_E^{fl}}{h_{fl}}$ are processed in the x-axis of *figure 9-12* and *figure 9-13*. For example: according to the graphs, error smaller than 10% for HE300A would be achieved with 18 elements in the vidth and 3 elements in the height of the ɪlange. Remarkable is that the torsion load requires the largest number of elements and therefore it represent the critical number of elements required to model a HE section. The actual mesh of a beam subjected to combination load of shear, normal, bending and torsion load will follow the mesh recommended in the Mesh design tables in the conclusion

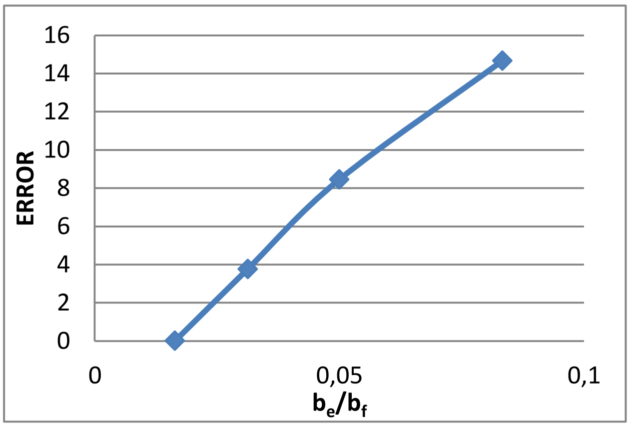


Figure 9-12: Relative error flange

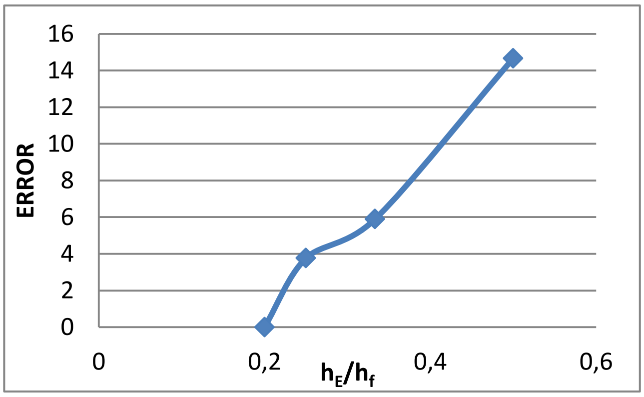


Figure 9-13: Relative error flange

9.3.2 Rounded corners

The Von-Mises stress accuracy in the rounded corners with respect to the cross-sectional degrees of freedom is plotted in *figure 9-14*. The slope of the error between model 1 and 2 is very steep, almost parallel to the y-axis. In the second model the slope is smaller suggesting that the required mesh should be in any case denser than the mesh applied in *model 2.* This mesh consist of six *Prism* elements.

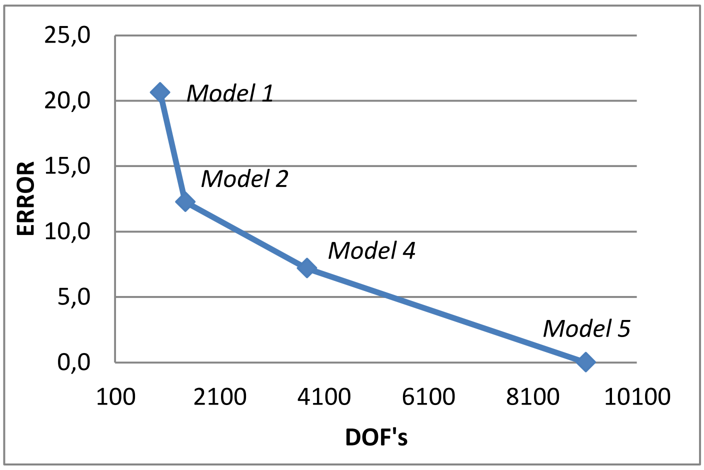


Figure 9-14: Von-Mises

9.3.3 Web

Considering the web of the cross-section, the error versus the DOF's is plotted. Also here the same pattern of accuracy occurs. More elements in the web mean higher accuracy of the XY stress. The slope of the line will be also in this case steep from the first until the third model. A much smaller slope is watched from *Model 3, suggesting* the use his mesh as minimal mesh.

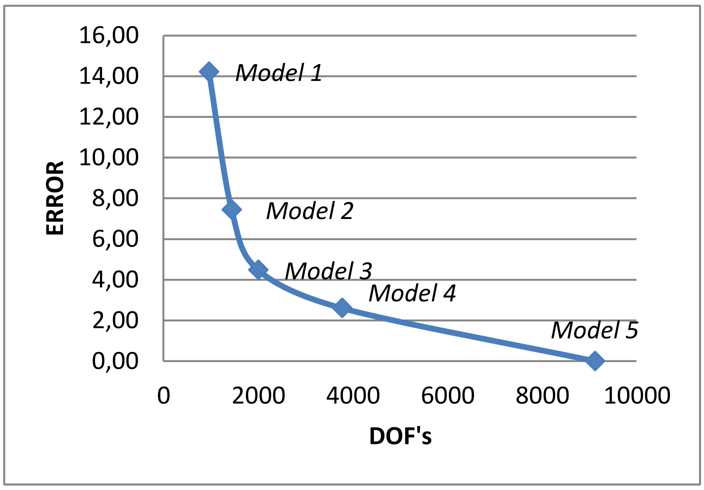


Figure 9-15: Relative error of xy shear stress

9.4 Conclusion

During the FEA of the fixed beam subjected to torsion load, the cross-sectional shear stress components have been successfully modeled in ANSYS. The stresses show comparable distribution with the theory. The maximal shear stresses at the flanges and at the rounded corners of the cross-section need relative finer mesh to be able to approach the exact value with an error smaller than 10%. In addition, the Von Misses criterion has been applied in the search for the error in the middle of the flanges and the error in the rounded corners. Engineer may use the suggested design tables to mesh HE or I section in finite element software. For each mesh the obtained error is given in order to know which safety factors should be applied using each mesh.

- Required mesh concerning τ_{max}

$\frac{h_E^{fl}}{h_{fl}}$	$\frac{b_E^{fl}}{b_{fl} - t_w}$	Error [%]
$1/2$	$1/12$	14.60
$1/2$	$1/20$	8.45
$1/3$	$1/20$	5.90
$1/4$	$1/32$	3.76
$1/5$	$1/62$	$< \varepsilon$

Table 9-2: Solid 186 mesh design table for τ_{max}

- Required mesh concerning $\tau_{xy,max}$

$\frac{h_E^{w}}{h_{w}}$	$\frac{b_E^{w}}{b_{w}}$	Error [%]
$1/6$	$1/2$	14.20
$1/10$	$1/2$	7.43
$1/16$	$1/2$	4.50
$1/16$	$1/3$	2.6
$1/44$	$1/4$	$< \varepsilon$

Table 9-3: Solid186 mesh design table for $\tau_{xy,max}$

- Required mesh concerning $\sigma_{VM,max}$ at the rounded corners

Number of Prism elements	$Error$ [%]
3	20.6
5	12.3
9	7.2
19	$< \varepsilon$

Table 9-4: Solid186 (Prism) mesh design table for rounded corners for r=27mm

Using the design tables above will be explained through an example: given HE300A section subjected to torsion moment within the linear elastic stress distribution, without considering the warping resistance on the cross-section. The following mesh should be applied in order to achieve an accuracy of minimal 10% , regarding only the maximal stress τ_{max}

$Flange$: $b_{fl} = 300mm \Rightarrow \frac{h_E^{fl}}{h_{fl}} = \frac{1}{2}; \frac{b_E^{fl}}{b_{fl}-t_w} = \frac{1}{20} \Rightarrow h_E^{fl} = 4.25mm; b_E^{fl} = 14.575mm \Rightarrow EY_{fl} = 2 \quad EZ_{fl} = 20$

$Peak\ region\ (flange)$: The peak region at the middle of the flanges will be meshed as followed:

$EZ_p = EZ_w = 2$ As mentioned before two is the minimum value for this region

$EY_p = EY_{fl} = 2$

10. Conclusions and Recommendations

In this bachelor thesis a study has been conducted in the search for possibility's to model rectangular beam and HE/I-beams in finite element software like ANSYS. The study has been dived into four different phases in order to create a logical working method. Firstly, the fundamental knowledge about FEM is studied following by study of the existing theory. Afterwards the database has been assembled and finally the results have been evaluated in order to track possible mistakes and vague results. This usefully approach is given shortly bellowed:

- Fundamental knowledge FEM- $\oint 1$, $\oint 2$
- Structural mechanics theory - $\oint 3$
- Building models and assembling database rectangular beam - $\oint 4$
- Optimization study rectangular beam- $\oint 5$
- The adaptive mesh refinement method for HE300A section- $\oint 6$, $\oint 7$, $\oint 8$, $\oint 9$

The study has been conducted exclusively for *Solid186* elements with 20-nodes where the stresses in the nodes are approached with linear elastic numerical FEM.

Reading results:

The results of the stresses should be measured only in the nodes. The elements stresses are the average of the nodal values and hereto less accurate. $\oint 4.8$ $\oint 4.9$

Optimization study rectangular beam:

Nodal stresses due to torsion moment of fixed beam had been analysed. The warping resistance and deformation of the flanges is being neglected by taking a cross-section far enough from the disturbed zone's. In this manner, optimization study has been conducted by changing every possible parameter in the search for the correlation between these parameters and the stress values. In this parameter study the relation between the element proportion and their quantity has been summarized in one final output, Mesh design table for a beam subjected to torsion. This table is applicable for beams subjected to all kind of loads. Because it is already discovered in earlier studies that torsion moment requires the finest comparison to shear, normal and bending load. Moreover, while evaluating the results the efficiency of the mesh has been checked. This by choosing as minimal as possible DOF's with as accurate as possible mesh. Hereby the mesh design table:

τ_{max}	EY	EZ	Error	DOF's
$h/b = 1$	6	4	0.1	279
	4	3	0.3	153
	6	3	1.6	219
	4	2	2.7	111
$h/b = 1{,}6$	5	3	0.3	186
	6	3	1	219
	5	2	1.3	135
	4	2	1.9	111
	4	3	2.7	153
$h/b = 2$	5	2	0.2	135
	5	3	0.7	186
	4	2	2.7	111
	4	3	3.2	153
$h/b = 2.5$	5	2	1.1	135
	5	3	1.4	186
	4	2	2.9	111
$h/b = 3$	5	2	1.6	135
	6	3	1.7	219
	4	2	2.9	111
	4	3	3.2	153

Table 10-1: FEA mesh design table for rectangular cross-section

Adaptive mesh refinement HE300A beam:

In compered with the previous described optimization study, analysing the stresses in HE section has been done with the adaptive refinement working method. In this way, initially a beam with coarse mesh has been modelled in ANSYS. The stresses at the critical points has been noted an compared with an reference value. If a large deviation is noticed a second model has been introduced, and so on. During this analysis the beam was loaded with bending force, shear force and torsion force. Hereby is a summary of the study concerning the HE300A section:

Bending moment:

- For bending moment an coarse model is sufficient to approach the maximal normal bending stress at the flanges.

Shear:

- Applying shear load on the beam reveal peak stresses at the middle of the web. These stresses can be also approached by coarse mesh at the web of the section.
- Moreover, peak stresses due to shear load have been noticed at the rounded corners and at the middle of the flanges.

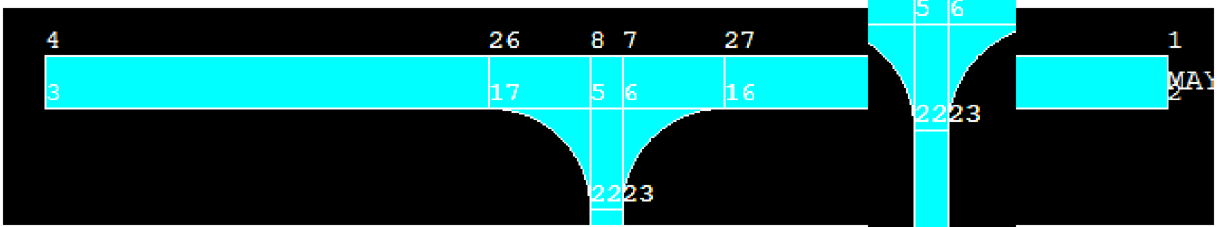
- The peak stresses due to shear load at the top of the flange are approximately four times smaller than the stress at the web, while the peak stresses at the rounded corners are approximately the half from the maximal stress at the web.
- A coarse mesh is sufficient to approach the maximal stress at the web
- A finer mesh is required to approach the peak stresses at the middle of the flange and at the rounded corners.

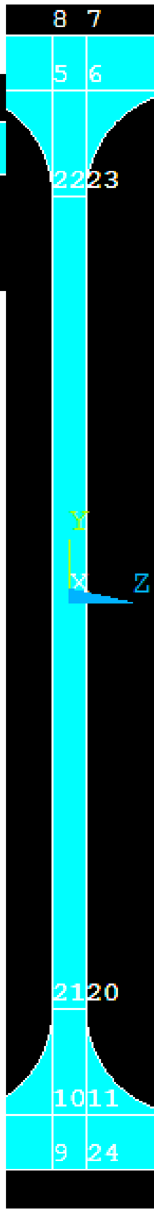
Torsion:

- An finer mesh comparing with the other loads is required to approach the stresses values with high accuracy.
- The peak stresses are noticed, just like by the shear load, at the rounded corners and at the middle of the flange.

Comparing the results of these loading cases imply the following recommendations applying for all loading cases:

- Rounded corners should be meshed with a fine mesh
- The middle of the flange should be meshed with a finer mesh comparing to the sides of the flanges
- The web can be meshed with relative coarse mesh





Area	*Description*	*Mesh*
1,2,6,7 and 3, 4, 8, 5:	Flanges	coarse
5, 6, 7, 8	Peak zone	fine
17,5,22 and 6,16,23	Rounded corners	fine
5, 6, 10, 11	Web	coarse

Finally, the mesh design tables for each loading case are given:

Bending moment:

$\frac{h_E^w}{h_w}$	$\frac{h_E^{fl}}{h_{fl}}$	$Error$
$1/16$	$1/4$	0%
$1/10$	$1/2$	0.07%
$1/6$	$1/2$	0.6%

table 10-2: FEA mesh design table for $\sigma_{xx,max}$

$$EY_p = EY_{fl} = \frac{h_{fl}}{h_E^{fl}} \qquad EY_w = \frac{h_w}{h_E^w} \qquad EZ_W = EZ_p = 2 \qquad EZ_{fl} = 5$$

Shear:

- Required mesh at the web concerning τ_{max} for $EZ_w \geq 2$

$\frac{h_E^w}{h_w}$	*Error [%]*
$1/6$	0.6
$1/10$	0.09
$1/16$	$< \varepsilon$

Table 8-7: Solid186 mesh design table for τ_{max}

- Required mesh at the flange concerning $\tau_{fl,max}$ and $\sigma_{VM,fl}$ for $EY_{fl} \geq 2$

$\frac{b_E^{fl}}{b_{fl} - t_w}$	$Error\ \tau_{fl,max}$ *[%]*	$Error\ \sigma_{VM}$ *[%]*
$1/12$	12.4	2.35
$1/20$	4.2	0.17
$1/32$	$< \varepsilon$	$< \varepsilon$

Table 8-8: Solid186 mesh design table for error $\tau_{fl,max}$

Number of Prism elements	*Error*
3	1.5
5	0.06
11	$< \varepsilon$

$$EY_w = \frac{h_w}{h_E^w} \quad EZ_{fl} = \frac{\mathrm{b_{fl}} - \mathrm{t_w}}{\mathrm{b_E^{fl}}} \qquad EZ_W = EZ_p = 2 \quad EY_p = EY_{fl} = 2$$

Torsion:

- Required mesh concerning τ_{max}

$\frac{h_E^{fl}}{h_{fl}}$	$\frac{b_E^{fl}}{b_{fl}-t_w}$	Error [%]
$1/2$	$1/12$	14.60
$1/2$	$1/20$	8.45
$1/3$	$1/20$	5.90
$1/4$	$1/32$	3.76
$1/5$	$1/62$	$< \varepsilon$

Table 9-2: Solid 186 mesh design table for τ_{max}

- Required mesh concerning $\tau_{xy,max}$

$\frac{h_E^{w}}{h_{w}}$	$\frac{b_E^{w}}{b_{w}}$	Error [%]
$1/6$	$1/2$	14.20
$1/10$	$1/2$	7.43
$1/16$	$1/2$	4.50
$1/16$	$1/3$	2.6
$1/44$	$1/4$	$< \varepsilon$

Table 9-3: Solid186 mesh design table for $\tau_{xy,max}$

- Required mesh concerning $\sigma_{VM,max}$ at the rounded corners

Number of Prism elements	$Error$ [%]
3	20.6
5	12.3
9	7.2
19	$< \varepsilon$

Table 9-4: Solid186 (Prism) mesh design table for rounded corners for r=27mm

$$EY_w = \frac{h_w}{h_E^w} \quad EZ_W = EZ_p = \frac{b_w}{b_E^w} \qquad EZ_{fl} = \frac{\mathrm{b_{fl} - t_w}}{\mathrm{b_E^{fl}}} \quad EY_{fl} = \frac{h_{fl}}{h_E^{fl}}$$

$$EY_p = EY_{fl} \qquad EZ_p = EZ_w$$

A. Appendices:

A.1 Stresses values for rectangular beam subjected to torsion

A.2 Design tables for rectangular beam subjected to torsion

A.3 Von Mises stresses of HE300A section

A.1 Stresses values for rectangular beam subjected to torsion

	$h/b = 1$	EZ 2	3	4	5	6	7	8
	2	3,9210	4,0719	4,1307	4,1552	4,1668	4,1729	4,1763
	3	3,2950	3,4468	3,5147	3,5497	3,5696	3,5814	3,5889
	4	3,4342	3,5386	3,5900	3,6175	3,6343	3,6453	3,6528
EY	5	3,2930	3,4047	3,4579	3,4854	3,5018	3,5123	3,5195
	6	3,3618	3,4720	3,5232	3,5493	3,5644	3,5738	3,5800
	7	3,2991	3,4121	3,4644	3,4914	3,5071	3,5168	3,5233
	8	3,3391	3,4511	3,5024	3,5290	3,5443	3,5539	3,5602

	$h/b = 1.6$	EZ 2	3	4	5	6	7	8
	2	7,8299	7,8754	7,8897	7,8923	7,8922	7,8916	7,8910
	3	6,6769	6,7913	6,8399	6,8609	6,8711	6,8763	6,8789
EY	4	6,7196	6,7790	6,8133	6,8328	6,8445	6,8515	6,8559
	5	6,5067	6,5738	6,6081	6,6270	6,6387	6,6463	6,6515
	6	6,5973	6,6601	6,6892	6,7040	6,7127	6,7184	6,7225
	7	6,5038	6,5727	6,6045	6,6206	6,6298	6,6357	6,6397
	8	6,5608	6,6272	6,6574	6,6728	6,6816	6,6872	6,6908

	$h/b = 2$	EZ 2	3	4	5	6	7	8
	2	6,0577	6,0503	6,0456	6,0416	6,0389	6,0370	6,0358
EY	3	5,2441	5,2921	5,3114	5,3182	5,3207	5,3215	5,3216
	4	5,1581	5,1846	5,2006	5,2092	5,2139	5,2164	5,2178
	5	5,0282	5,0552	5,0702	5,0789	5,0843	5,0878	5,0902
	6	5,0745	5,0975	5,1080	5,1137	5,1174	5,1200	5,1220
	7	5,0189	5,0456	5,0578	5,0641	5,0678	5,0703	5,0720
	8	5,0511	5,0766	5,0880	5,0938	5,0972	5,0993	5,1007

	$h/b = 2.5$	EZ 2	3	4	5	6	7	8
	2	4,6477	4,6277	4,6193	4,6148	4,6122	4,6106	4,6095
	3	4,1119	4,1265	4,1315	4,1319	4,1313	4,1305	4,1298
EY	4	3,9448	3,9592	3,9671	3,9704	3,9718	3,9723	3,9723
	5	3,8723	3,8841	3,8910	3,8951	3,8975	3,8990	3,8998
	6	3,8885	3,8964	3,8998	3,9022	3,9039	3,9053	3,9063
	7	3,8597	3,8693	3,8735	3,8758	3,8773	3,8785	3,8793
	8	3,8749	3,8842	3,8880	3,8900	3,8912	3,8920	3,8925

	$h/b = 3$	EZ 2	3	4	5	6	7	8
EY	2	3,7173	3,7001	3,6933	3,6899	3,6881	3,6869	3,6863
	3	3,3650	3,3661	3,3655	3,3638	3,3624	3,3613	3,3605
	4	3,1671	3,1770	3,1814	3,1826	3,1828	3,1825	3,1821
	5	3,1237	3,1311	3,1352	3,1374	3,1386	3,1392	3,1395
	6	3,1250	3,1287	3,1303	3,1318	3,1329	3,1337	3,1344
	7	3,1104	3,1147	3,1163	3,1175	3,1184	3,1190	3,1195
	8	3,1172	3,1214	3,1229	3,1237	3,1242	3,1246	3,1249

Table A-1: τ_{xy} stresses values for M=20e6 Nmm

$h/b = 1$	EZ 2	3	4	5	6	7	8
EY 2	3,9210	3,2950	3,4342	3,2930	3,3618	3,2991	3,3391
3	4,0719	3,4468	3,5386	3,4047	3,4720	3,4121	3,4511
4	4,1307	3,5147	3,5900	3,4579	3,5232	3,4644	3,5024
5	4,1552	3,5498	3,6175	3,4854	3,5493	3,4914	3,5290
6	4,1668	3,4720	3,6343	3,5018	3,5644	3,5071	3,5443
7	4,1729	3,5814	3,6453	3,5123	3,5738	3,5168	3,5539
8	4,1763	3,5889	3,6528	3,5195	3,5800	3,5233	3,5602

$h/b = 1.6$	EZ 2	3	4	5	6	7	8
EY 2	5,4787	4,6110	4,9108	4,6907	4,8070	4,7087	4,7729
3	6,0394	5,0651	5,2915	5,0683	5,1851	5,0845	5,1510
4	6,2776	5,2806	5,4786	5,2561	5,3708	5,2703	5,3357
5	6,3991	5,4032	5,5778	5,3572	5,4706	5,3712	5,4361
6	6,4662	5,4778	5,6378	5,4181	5,5304	5,2703	5,4963
7	6,5054	5,5259	5,6770	5,4573	5,5686	5,4708	5,5348
8	6,5294	5,5583	5,7043	5,4841	5,5944	5,4971	5,5608

$h/b = 2$	EZ 2	3	4	5	6	7	8
EY 2	3,6346	3,0763	3,3053	3,1562	3,2400	3,1728	3,2181
3	4,1558	3,4779	3,6584	3,4982	3,5828	3,5107	3,5585
4	4,3874	3,6778	3,8413	3,6792	3,7633	3,6901	3,7378
5	4,5165	3,7973	3,9418	3,7808	3,8639	3,7912	3,8386
6	4,5920	3,8722	4,0034	3,8434	3,9259	3,8537	3,9008
7	4,6384	3,9219	4,0437	3,8844	3,9663	3,8946	3,9415
8	4,6680	3,9564	4,0718	3,9126	3,9939	3,9227	3,9693

$h/b = 2.5$	EZ 2	3	4	5	6	7	8
EY 2	2,4282	2,0728	2,2473	2,1470	2,2086	2,1627	2,1953
3	2,8837	2,4116	2,5535	2,4382	2,4998	2,4479	2,4824
4	3,0982	2,5919	2,7258	2,6067	2,6692	2,6155	2,6506
5	3,2288	2,7057	2,8258	2,7064	2,7683	2,7144	2,7495
6	3,3093	2,7794	2,8886	2,7698	2,8312	2,7775	2,8125
7	3,3612	2,8298	2,9304	2,8122	2,8732	2,8198	2,8547
8	3,3959	2,8656	2,9596	2,8418	2,9025	2,8494	2,8841

$h/b = 3$	EZ 2	3	4	5	6	7	8
EY 2	1,7540	1,5107	1,6505	1,5782	1,6267	1,5931	1,6184
3	2,1455	1,7952	1,9109	1,8229	1,8706	1,8310	1,8574
4	2,3408	1,9575	2,0705	1,9778	2,0273	1,9855	2,0130
5	2,4685	2,0650	2,1683	2,0741	2,1233	2,0809	2,1086
6	2,5503	2,1366	2,2315	2,1371	2,1860	2,1435	2,1712
7	2,6052	2,1867	2,2742	2,1802	2,2288	2,1864	2,2140
8	2,6431	2,2230	2,3045	2,2109	2,2592	2,2170	2,2445

Table 0A-2: : τ_{xz} stresses values for M=20e6 Nmm

A.2 Design tables for rectangular beam subjected to torsion

The following tables can be applied to calculate a rectangular beam in ANSYS. The tables are computed with the following parameters:

$$\frac{EX}{L} = \frac{1}{120} \qquad E = 210e6\frac{N}{mm^2} \qquad v = 0.1$$

The tables are computed for the Ultimate limit state, apply only for the maximal shear tress component.

EY	EZ	ERROR	DOF's	EY	EZ	ERROR	DOF's
8	5	0	441	3	6	1,2	219
7	8	0,1	597	6	7	1,3	459
6	4	0,1	279	6	8	1,5	519
5	8	0,2	441	3	7	1,5	252
7	7	0,3	528	6	3	1,6	219
4	3	0,3	153	4	4	1,7	195
5	7	0,4	390	3	8	1,7	285
3	4	0,4	153	7	4	1,8	321
8	6	0,5	519	5	4	2	237
3	5	0,5	186	8	3	2,2	285
6	5	0,6	339	3	3	2,3	120
7	6	0,6	459	4	5	2,5	237
5	6	0,7	339	4	6	2,9	279
8	4	0,7	363				
8	7	0,7	597				
8	8	0,9	675				
6	6	1	399				
7	5	1	390				
5	5	1,2	288				

Table A-3: Increasing of the error for $\mathbf{h/b = 1}$

EY	EZ	ERROR	DOF	EY	EZ	ERROR	DOF
6	2	0,1	159	6	3	1	219
5	4	0,2	237	8	5	1,2	441
7	4	0,2	321	8	6	1,3	519
5	3	0,3	186	5	2	1,3	135
7	3	0,3	252	3	2	1,3	87
7	5	0,4	390	6	4	1,4	279
8	2	0,5	207	8	7	1,4	597
8	3	0,5	285	7	2	1,4	183
5	5	0,5	288	8	8	1,5	675
7	6	0,6	459	6	5	1,7	339
7	7	0,6	528	6	6	1,8	399
5	6	0,7	339	6	7	1,9	459
7	8	0,7	597	6	8	1,9	519
5	7	0,8	390	4	2	1,9	111
5	8	0,9	441	3	2	2,7	87
8	4	1	363	3	3	2,9	120

Table A-4 : Increasing of the error for $\mathbf{h/b = 1.6}$

EY	EZ	ERROR	DOF	EY	EZ	ERROR	DOF
7	2	0	183	8	6	1,3	519
5	2	0,2	135	8	7	1,3	597
7	3	0,5	252	5	8	1,4	441
8	2	0,6	207	8	5	1,4	441
7	4	0,7	321	8	6	1,5	519
5	3	0,7	186	6	3	1,5	219
7	5	0,9	390	8	7	1,6	597
7	6	0,9	459	8	8	1,6	675
7	7	1,0	528	6	4	1,7	279
7	8	1,0	597	6	5	1,8	339
5	4	1,0	237	6	6	1,9	399
8	3	1,1	285	6	7	2	459
6	2	1,1	159	6	8	2	519
5	5	1,2	288	4	2	2,7	111
8	4	1,3	363				

Table A-5: : Increasing of the error for $\mathbf{h/b = 2}$

EY	EZ	ERROR	DOF	EY	EZ	ERROR	DOF
7	2	0,7	183	5	4	1,5	237
7	3	1	252	8	7	1,6	597
7	4	1,1	321	8	8	1,6	675
7	5	1,1	390	5	5	1,6	288
8	2	1,1	207	6	3	1,7	219
5	2	1,1	135	5	6	1,7	339
7	6	1,2	459	5	7	1,7	390
7	7	1,2	528	6	4	1,8	279
7	8	1,2	597	6	5	1,8	339
8	3	1,4	285	5	8	1,8	441
5	3	1,4	186	6	6	1,9	399
8	4	1,5	363	6	7	1,9	459
8	5	1,5	441	6	8	1,9	519
8	6	1,5	519	4	2	2,9	111
6	2	1,5	159				

Table A-6: : Increasing of the error for $\mathbf{h/b = 2.5}$

EY	EZ	ERROR	DOF	EY	EZ	ERROR	DOF
7	2	1,1	183	5	2	1,6	135
7	3	1,3	252	6	3	1,7	219
7	4	1,3	321	6	4	1,8	279
7	5	1,4	390	6	5	1,8	339
7	6	1,4	459	6	6	1,8	399
7	7	1,4	528	5	3	1,8	186
7	8	1,4	597	5	4	1,9	237
8	2	1,4	207	6	7	1,9	459
8	3	1,5	285	6	8	1,9	519
8	4	1,5	363	5	5	2	288
8	5	1,6	441	5	6	2	339
8	6	1,6	519	5	7	2	390
8	7	1,6	597	5	8	2	441
8	8	1,6	675	4	2	2,9	111
6	2	1,6	159				

Table A-7: Increasing of the error for $\mathbf{h/b = 3}$

A.3 Von Mises stresses of HE300A section

As for the HE300A cross-section the Von-Mises stress criterion are given in the tables for an cross-section with the origin in the centre of gravity. The Y and Z coordinate are applied for for a cross-section located at x=L/2. The beam is subjected to torsion moment of $10e6\ Nmm$. The following parameters have been used to compute the data:

$$L = 4000mm \qquad v = 0.1 \qquad E = 210e6\ N/mm^2$$

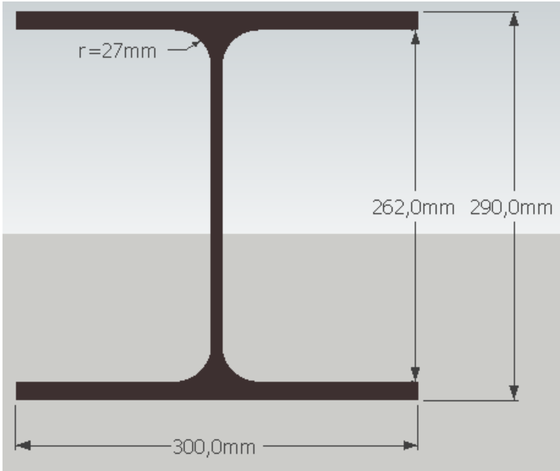


Figure A-1: HE300A

$Model$ 1 (963 DOF's)

Y	Z	Flange [N/mm2]
145	31,25	219,83
145	4,25	267,4
145	0	282,5
Y	Z	rounded corners [N/mm2]
131	31,25	215,412
127,3	17,75	269,86
117,5	7,86	275
108	4,25	201

Table A-8: Von-Mises

$Model\ 2$ (1443 DOF's)

Y	Z	Flange [N/mm2]
145	31,25	212,723
145	17,75	257,07
145	4,25	294,17
145	0	304,849
Y	Z	rounded corners [N/mm2]
131	31,25	214,7
129	21	264,8
123	12	298,3
114,5	6,3	260,5
104	4,25	196,5

Table 0A-9: Von-Mises

$Model\ 3$ (2007 DOF's)

Y	Z	Flange [N/mm2]
145	32,25	216,2
145	17,75	261,6
145	4,25	302,405
145	0	314,6
Y	Z	rounded corners [N/mm2]
131	31,25	219,53
128,9	20,9	264,3
123	12	277,285
114,3	6,3	249,35
104	4,25	173,95

Table A-10:Von-Mises

$Model\ 4$ (2007 DOF's)

Y	Z	Flange [N/mm2]
145	31,25	213,9
145	22,25	242,4
145	13,25	282,221
145	4,25	311,32
145	1,4	320,75
Y	Z	rounded corners [N/mm2]
131	31,25	211,8
130	24,2	245,3
127,4	17,75	289,8
123	12,1	315,56
117,5	7,8	287,65
110	5,17	222,3
104	4,25	156,38

Tabel A-11: Von-Mises

$Model$ 5 (9123 DOF's)

Y	Z	Flange [N/mm2]
145	25,85	229,43
145	20,45	251,98
145	15,05	278,671
145	9,65	304,65
145	4,25	321,621
145	2,125	328
145	0	329
Y	Z	rounded corners [N/mm2]
131	31,25	213
130	27	230
129	22	261
128	19	293
125,8	15	320
123	12	340
120	9,4	323
116	7,2	285
112	5,57	231
108	4,25	144

Table A-12: Von-Mises

B. Reference

B.1 Nadieh Meinen (october 2012), Optimalisatie van het aantal volume-elementen in een op dwarskracht belaste balk

B.2 Wouter Steenstra (June 2012), Spanningen berekenen met volume-elementen

B.3 Evert van Vliet (January 2012), Torsion in ZIP bridge system

B.4 B.M. van de Weerd (december 2007), Schuifspanning en fschuifstijfheid in I-profielen

B.5 David Roylance (February 2001),MIT, Finite Element Analysis

B.6 dr.ir. P.C.J. Hoogenboom, Reader Torsion

B.7 Blaauwendraad, Johan(1996), Eindige-elementenmethode voor staafconstructies

B.8 A. Ordun˜a , P.B. Lourenc¸o (2005) Three-dimensional limit analysis of rigid blocks Part I: Torsion failure on frictional interfaces and limit analysis formulation

B.9 Rao, Singiresu S (1999), The finite element method in the engineering

B.10 Ir. R. Abspoel, Ir. P.A. de Vries, Prof. Ir. F.S.K. Bijlaard (2012), Reader steal constructions CT2035

B.11 J. Oosterhof (2008), Kracht + vorm Bouwen met staal

B.12 Coenraad Hartsuijker (2001), Toegepaste Mechanica, deel 2 : Spanningen, vervormingen en verplaatsingen

B.13 ANSYS tutorial

B.14 http://en.wikipedia.org/wiki/Ansys