

Finite Element Modelling of Mast Foundation and T-Joint

MARCUS WESTER



KTH Mechanics

Master's Degree Project
Stockholm, Sweden 2004

Finite Element Modelling of Mast Foundation and T-Joint

by

Marcus Wester

August 2004

Technical Reports from
Royal Institute of Technology
Department of Mechanics
SE-100 44 Stockholm, Sweden

Kungliga Tekniska Högskolan i Stockholm, Valhallavägen 79, Stockholm.

© Marcus Wester 2004

Preface

The work presented in this thesis was initiated by the structural engineering company TYRÉNS and KTH Mechanics, Royal Institute of Technology in Stockholm, Sweden. It was carried out between February and August 2004.

The work on this thesis was conducted under the supervision of Prof. Dr. Per-Olof Thomasson to whom I express my gratitude for his valuable advice.

I thank Karin Eriksson at TYRÉNS for introducing me to the problem carried out in this thesis.

I also would like to thank Prof. Anders Eriksson who from the beginning inspired me to go on with studying mechanics with his superb lectures and, of course, for being the examiner of this thesis.

A special thank goes to Dr. Gunnar Tibert who has given me invaluable advice and lots of interesting discussions.

Stockholm, August 2004

Marcus Wester

Abstract

In this thesis, a rectangular hollow section construction was analysed. The structure acts as a steel frame foundation to a eighteen metre mast. The mast itself was not treated, only the mast foundation.

The thesis starts with a description to the problem, followed by a short introduction to finite element methods. After that, theories of plasticity were presented, including elasto-plasticity and strain hardening.

The finite element analysis software package ABAQUS 6.3 was used. The procedure of making models in ABAQUS was given along other features in the program.

First, different versions of the foundation were modelled using mainly shell elements. Different parts were taken away to see how they affected the foundation. After analysing the results of the foundation models, an optimised model were tested.

Second, different T-joints were modelled using solid elements. This T-joint was of the same dimensions as the T-joints in the foundation models. It was created because the results from the foundation models showed that the welds between the vertical beams and the horizontal beams were the critical parts.

The main aim of the thesis was to investigate the corners of the foundation and to see which influence the parts had to the foundation.

Every model plasticise. In each case, only a few elements reached plastic strains. The models with the finer mesh plasticised more than the models with the coarser mesh. This suggests singularity points in the plasticised areas.

The results from the T-joint analyses coincided with the results from the foundation analyses. The foundation and the T-joint analyses showed the same thing; it is preferable with full width bracing, i.e., the chord and the bracing should be of the same dimension.

Keywords: Finite element analysis, elasticity, plasticity, rectangular hollow sections, ABAQUS/Standard, shell elements, solid elements.

Contents

Preface	iii
Abstract	v
1 Introduction	1
1.1 Introduction	1
1.2 Aims and scope	1
1.3 Finite element analysis	2
1.4 Outline of the thesis	4
2 Rectangular Hollow Sections	5
2.1 Brief history	5
2.2 Properties of structural hollow sections	6
2.3 Joints with RHS	6
3 Design Principles	9
3.1 The mechanical properties of materials	9
3.2 Material models	10
3.3 Plasticity	12
3.3.1 Introduction	12
3.3.2 One-dimensional plasticity	12
3.3.3 Multidimensional plasticity	13
3.3.4 Von Mises plasticity in three dimensions	16
4 ABAQUS	19

4.1	Creating models	19
4.2	Plasticity in ABAQUS	20
4.2.1	Isotropic elasto-plasticity	21
4.2.2	Material data	21
4.3	General analysis step	21
4.3.1	Nonlinearity	22
4.3.2	Incrementation	22
4.4	Elements	22
4.4.1	Shell elements	22
4.4.2	Solid elements	23
4.4.3	Beam elements	24
4.4.4	Rigid elements	24
5	Mast Foundation	25
5.1	Introduction	25
5.2	Data for the models	25
5.2.1	Section properties	25
5.2.2	Elasticity and density	26
5.2.3	Plasticity data	26
5.3	Models	30
5.3.1	Model with coarser mesh	30
5.3.2	Model with finer mesh	31
5.3.3	Optimised model	32
5.4	Loads	33
5.5	Results	35
5.5.1	Model with coarser mesh	36
5.5.2	Model with finer mesh	44
5.5.3	Optimised model	47
5.5.4	Double design load	47
5.5.5	Comparison between the models	49

5.6	Conclusions	51
6	T-Joint	53
6.1	Introduction	53
6.2	Data for the model	53
6.3	Model	54
6.4	Loads	56
6.4.1	Load capacity	56
6.5	Results	59
6.6	Analysing the results	61
6.7	Conclusions	62
7	Conclusions	63
7.1	Finite element modelling	63
7.1.1	Foundation	63
7.1.2	T-joint	64
7.1.3	Assumptions	65
7.2	Future research	65
	Bibliography	67
A	Path Plots	69
A.1	Corner paths	70
A.2	Mast foot paths	74
B	Abaqus Input Files	79
B.1	Foundation model input files	79
B.2	T-joint model input files	81

Chapter 1

Introduction

1.1 Introduction

Structural hollow sections (SHS) are widely used today and exist everywhere in modern construction. In the major scale they function as great pillars, e.g., in bridges and in the minor scale as small trusses, e.g., lamp posts.

This thesis deals with the static effects on a mast foundation. The foundation is located on the roof of the fire station in Handen, close to Stockholm. It is mainly constructed of rectangular hollow sections (a member of the SHS family).

Today we have many simulation tools in the form of FEM-based software programs. These programs allow us to make models of any body, apply forces and see the results. Thus, complex problems can be simulated before they are constructed. Furthermore it is possible to optimise already built structures. Correctly used, these analyses can be economically justified.

1.2 Aims and scope

The main aim of this thesis is to investigate the foundation made of rectangular hollow sections. The foundation is to be analysed with finite element methods, using the Abaqus software. The structural engineering company TYRÉNS wanted to see the influence of the different parts in the corners. Because when the foundation was designed some issues occurred; it was not clear how the reaction forces were distributed in the corners.

In the finite element analyses some parts were taken away to see their influence on the structure. This resulted in four different models. Each model was subjected to for four different load cases. Then it was possible to see the worst load case.

The secondary aim was to isolate one of the t-joints in the model. This aim came up when working on the primary aim. It was clear that the critical parts of the foundation were the t-joints. Using a different modelling technique makes it possible

to see how different welds affect the joint.

1.3 Finite element analysis

Finite element analysis (FEA) is a method for numerical solution of field problems. Mathematically, a field problem is described by differential equations or by an integral expression. Either description may be used to formulate finite elements. Finite element (FE) formulations, in ready-to-use form, are contained in general purpose FEA programs. It is possible to use FEA programs while having little knowledge of the analysis method or the problem on which it is applied, inviting consequences that may range from embarrassing to disastrous [4].

The questions Who originated the finite element method? and When did it begin? have three different answers depending on whether one asks an applied mathematician, a physicist, or an engineer. All of these specialists have some justification for claiming the finite element method as their own, because each developed the essential ideas independently at different times and for different reasons. The applied mathematicians were concerned with boundary value problems of continuum problems. The physicists sought means to obtain piecewise approximate functions to represent their continuous functions. Faced with increasingly complex problems in aerospace structures, engineers were searching for a way in which to find the stiffness influence coefficients of shell-type structures reinforced by ribs and spars [9].

But, the finite element method as we know it today seems to have originated with Courant in his 1943 paper, which is the written version of a 1941 lecture to the American Mathematical Society. Courant determined the torsional rigidity of a hollow shaft by dividing the cross section into triangles and interpolating a stress function ϕ linearly over each triangle from values of ϕ at nodes [4].

The name *finite element* was coined by Clough in 1960. Many new elements for stress analysis were soon developed, largely by intuition and physical argument. In 1963, FEA acquired respectability in academia when it was recognised as a form of the Rayleigh-Ritz method, a classical approximation technique. Thus FEA was seen not just as a special trick for stress analysis but as a widely applicable method having a sound mathematical basis.

FEA has advantages over most other numerical analysis methods, including versatility and physical appeal [4].

- FEA is applicable to any field problem.
- There is no geometric restriction.
- Boundary conditions and loading are not restricted.
- Material properties are not restricted to isotropy and may change from one element to another or even within an element.

- Components that have different behaviours, and different mathematical descriptions, can be combined.
- An FE structure closely resembles the actual body or region to be analysed.
- The approximation is easily improved by grading the mesh.

Regardless of the approach used to find the element properties, the solution of a continuum problem by the finite element method always follows an orderly step-by-step process [9]:

1. *Discretize the continuum.* The first step is to divide the continuum or solution region into elements. A variety of element shapes may be used, and different element shapes may be employed in the same solution region.
2. *Select interpolation functions.* The next step is to assign nodes to each element and then choose the interpolation function to represent the variation of the field variable over the element.
3. *Find the element properties.* Once the finite element model has been established, we are ready to determine the matrix equations expressing the properties of the individual elements. We may use:
 - direct approach
 - variational approach
 - weighted residuals approach
4. *Assemble the element properties to obtain the system equations.* We combine the matrix equations expressing the behaviour of the elements and form the matrix equations expressing the behaviour of the entire system.
5. *Impose the boundary conditions.* Before the system equations are ready for solution they must be modified to account for the boundary conditions of the problem.
6. *Solve the system equations.* The assembly process gives a set of simultaneous equations that we solve to obtain the unknown nodal values of the problem.
7. *Make additional computations if desired.* Many times we use the solution of the system equations to calculate other important parameters.

1.4 Outline of the thesis

A short presentation of the chapters is given here for a better understanding of the thesis in total.

Chapter 2 treats structural hollow sections (SHS) in general. The emphasis is on the rectangular hollow sections (RHS).

Chapter 3 gives design principles with emphasis on theories of plasticity in three dimensions.

Chapter 4, a walkthrough to how the models were created. Procedures in Abaqus in general along with elements, etc., are presented.

Chapter 5 describes the first model, a mast foundation. This model use shell elements in the analysis. The results are presented as well.

Chapter 6 presents the second model, a t-joint, which is a detail of the first model. This model use solid elements instead of shell elements. Results are presented.

Chapter 7 presents the conclusions and suggestions for further research.

Appendix A gives the the path plots for the original geometry model and the optimised geometry model.

Appendix B contains ABAQUS input files.

Chapter 2

Rectangular Hollow Sections

2.1 Brief history

Many examples in nature demonstrate the excellent properties of the hollow section as a structural element in resisting compression, tension, bending and torsion forces. From the earliest times man has used the tubular shape made of various materials; first bronze and copper, later cast iron and finally steel and aluminium [14].

In the 19th century, methods were developed for the fabrication of tubes or circular hollow sections (CHS). Whitehouse from England started producing tubes by rounding a strip and joining it together by forming and welding. The welded tubes grew in importance after the development of the continuous welding process by Fretz-Moon in 1930. After the Second World War, welding processes were perfected, which have become very important for joining together hollow sections.

Due to the special end shaping needed for the direct connection between tubes, special connectors were developed. However these connectors were relatively expensive and it was therefore very desirable to solve the problems related to the direct connection between tubes.

This was the reason for the development of sections with nearly the same properties as the tube, but which could be connected in a simpler way. In 1952 the first rectangular hollow section (RHS) were produced by Stewarts and Lloyds. These sections can be joined easily and need only a straight cut as end preparation [14].

Most recommendations for the determination of joint strength have been developed partly or directly from experimental evidence; in some of the more simple joint types however theoretical models are used to give the strength relative to particular failure modes [8].

2.2 Properties of structural hollow sections

Structural hollow sections (SHS) is a modern and allround material for steel constructions. The simple design and the strength properties leads to constructions which are easy and favourable [11].

In trusses, due to the high buckling strength of the SHS, it is possible to reach great spans and spread the bracings widely. The SHS also show great torsional and bending stiffnesses. Simple joint details are favourable during construction.

The planning process for a building made of structural hollow sections is easy and quick. This is because it is possible to optimise the weight, strength, and stiffness of the structure by changing the thickness of the walls and at the same time keep the outer dimensions of the construction.

Compared with other materials such as timber and concrete, the following qualities can be realised for steel structural members [15]:

1. Lightness
2. High strength and stiffness
3. Ease of prefabrication and mass production
4. Fast and easy erection and installation
5. More accurate detailing
6. Nonshrinking and noncreeping at ambient temperatures
7. Formwork not needed
8. Termite-proof
9. Uniform quality
10. Economy in transportation and handling
11. Noncombustibility
12. Recyclable material

2.3 Joints with RHS

Rectangular hollow sections combine excellent strength properties with easy jointing possibilities [14]. These sections are widely used for the construction of lattice frameworks in building design, bridges, jibs, cranes, towers, masts etc.

At the beginning of the seventies the first empirical design equations for K- and N-joints were published. These equations were based on results from tests in which the actual dimension and the actual properties of the section were not measured. As a result these equations showed a scale effect which is not likely for the static strength. This was the reason that in 1973 an extensive research programme was prepared. In this programme, all parameters were studied which influence the static strength. The programme covered isolated T-, X-, K-, N-, and KT-joints.

Most of the research carried out have been coordinated by the Comité International pour le Developpement et l'étude de la Construction Tubulaire (Cidect).

For full width T-, X-, and Y-joints, the critical condition is the strength of the chord side wall, yielding in tension or buckling in compression. For less than full width connections yielding of the chord face becomes critical.

This thesis treats only the T-joint, thus only the equations (yield line model equations) for that joint are given below [14].

Axially loaded joints:

$$\hat{N}_1 = \begin{cases} \sigma_{eo_k} t_o^2 \left(\frac{2\eta}{b_0 \sin \theta_1} + 4\sqrt{1-\beta} \right) \frac{1}{(1-\beta) \sin \theta_1} & \text{if } \beta \leq 0.85 \\ \sigma_k t_o \left(\frac{2h_1}{\sin \theta_1} + 10t_o \right) \frac{1}{\sin \theta_1} & \text{if } 0.85 < \beta \leq 1.0 \end{cases} \quad (2.1)$$

Joint loaded by bending moments:

$$\hat{M}_{ip} = \begin{cases} \sigma_{eo_k} t_o^2 h_1 \left(\frac{1-\beta}{2\eta} + \frac{2}{\sqrt{1-\beta}} + \frac{\eta}{1-\beta} \right) & \text{if } \beta \leq 0.85 \\ 0.5\sigma_k t_o (h_1 + 5t_o)^2 & \text{if } 0.85 < \beta \leq 1.0 \end{cases} \quad (2.2)$$

$$\hat{M}_{op} = \sigma_k t_o (h_1 + 5t_o) b_1, \quad 0.85 < \beta \leq 1.0 \quad (2.3)$$

where β is the mean bracing to chord width ratio, η is the bracing depth divided by the chord width, σ_{eo_k} is the characteristic value for the yield stress of the chord, σ_k is the critical local buckling stress in the side walls of the chord, and θ_1 is the angle between the bracing and the chord (for a perfect T-joint, $\sin \theta_1 = \sin 90^\circ = 1$).

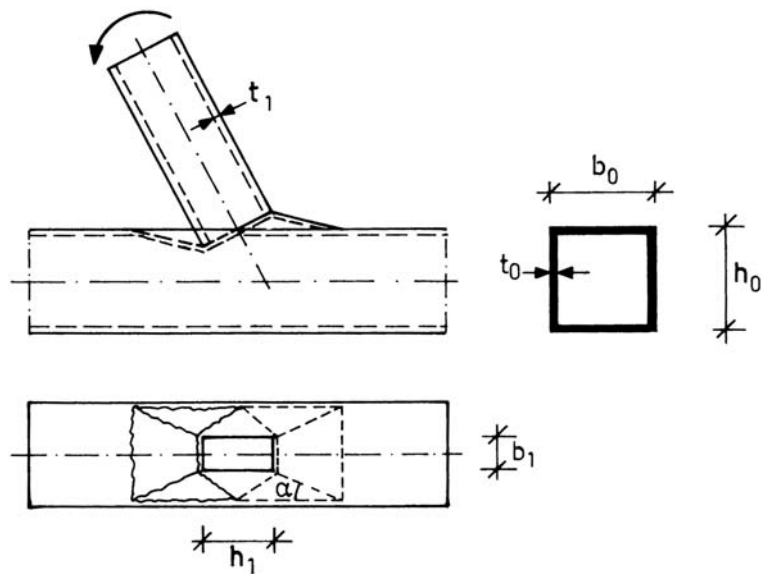


Figure 2.1: T-joint capacity calculated by the yield line model. Reproduced from [14].

Chapter 3

Design Principles

3.1 The mechanical properties of materials

The response of solids and structures to external forces largely depends upon the mechanical properties of their materials. The actual mechanical behaviour of a general material is, however, very complex. In order to facilitate the study of problems, several idealisations and simplifications are introduced [10].

In engineering mechanics it is a generally used idealisation that the atomic and molecular structure of matter can be disregarded and that matter can be considered *continuum* without gaps or empty spaces. The state variables of a continuum are described by continuous functions of the coordinates.

Another simplification connected with the continuum approach is the *phenomenological description* of material properties. Hence, in the description of a materials behaviour, changes (deformations, dislocations, etc.) in the crystal structure of the material under the action of mechanical forces are also disregarded. The mechanical properties of the material are characterised by *material models* based on the observation of simple experiments (tension, compression, torsion, etc.).

Because of the complexity of material behaviour a great number of material models can be developed. Each model, however, describes only one or a few properties. The selection of the most suitable model to be used depends on the material under consideration, the nature of the problem to be solved, the accuracy required, and the computational facilities available.

The basic experiment in which the simplest but most characteristic mechanical properties of a material are studied is a standard *tension test*. By measuring the applied force and the elongation of the tensile specimen during the loading process, and assuming homogeneous states of stress and strain in the observed part of the specimen, a stress–strain relationship is obtained and may be plotted in a *stress–strain diagram*. By considering the characteristics of the diagram of structural materials and disregarding the temperature and strain rate effects, three different types of material can be distinguished:

- *Elastic materials* follow the same stress–strain relationship during both the loading and the unloading processes. Hence, after unloading, no permanent strain remains (3.1a).
- In the case of *plastic materials*, the loading and unloading processes are described by different relationships and therefore we no longer have a one-to-one correspondence between stress and strain. Hence, after unloading, permanent strain remains (3.1b).
- Most materials under lower stresses behave elastically but under higher stresses undergo plastic deformation. These are called *elasto-plastic materials* (3.1c).

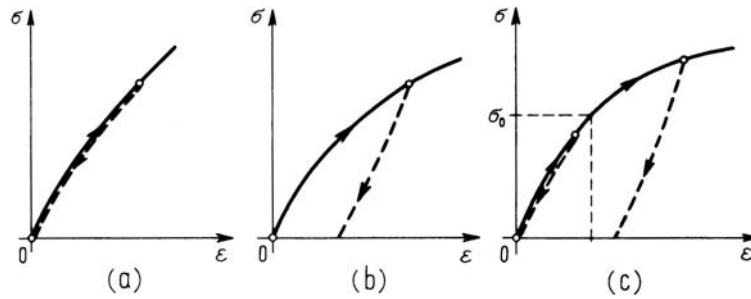


Figure 3.1: (a) elastic materials, (b) plastic materials, and (c) elasto-plastic materials. Reproduced from [10].

3.2 Material models

Actual stress–strain diagrams are generally nonlinear and therefore too complex to serve as the basis of a workable theory. In order to decrease the mathematical difficulties further idealisations and simple *material models* are introduced. Thus are the characteristic parts of the stress–strain diagrams linearised, see Fig. 3.2

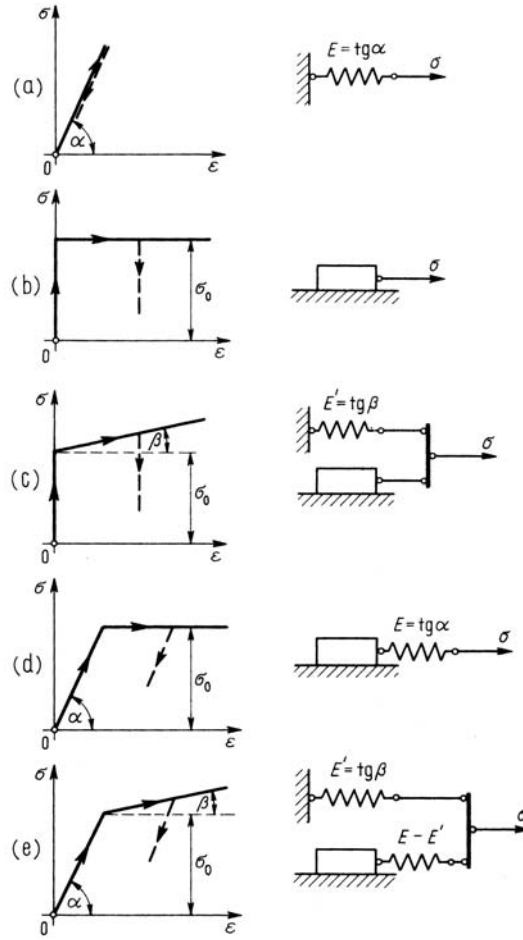


Figure 3.2: Behaviour of different materials. (a) Linearly elastic material, (b) rigid-perfectly plastic material, (c) rigid-plastic hardening material, (d) elastic-perfectly plastic material, (e) elastic-plastic material. Reproduced from [10].

Steel is a material of the elasto-plastic model. Therefore, plasticity plays a major part in this thesis.

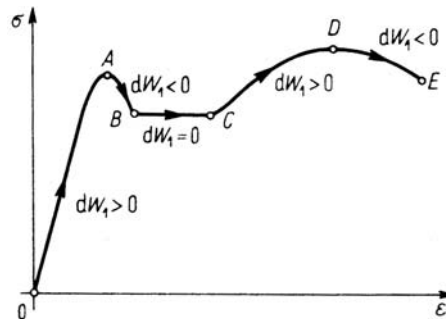


Figure 3.3: A typical stress-strain diagram for mild steel. Reproduced from [10].

3.3 Plasticity

3.3.1 Introduction

Plasticity refers to deformation that is not recovered if loads are removed. Conventionally, plasticity is regarded as time-independent. Thus, creep is excluded, and strain rate plays no role in plasticity calculations [4].

The theory of plasticity is phenomenological in nature. It is the formalisation of experimental observations of the macroscopic behaviour of a deformable solid and do not go deeply into the physical and chemical basis of this behaviour [3].

Experimental evidence supports the assumption that during plastic deformation essentially *no volume changes* occurs; that is, the material is incompressible [13]. Thus

$$\varepsilon_x^p + \varepsilon_y^p + \varepsilon_z^p = \varepsilon_1^p + \varepsilon_2^p + \varepsilon_3^p = 0 \quad (3.1)$$

Slip begins at an imperfection in the lattice, e.g., along a plane separating two regions, one having one more atom per row than the other. Because slip does not occur simultaneously along every atomic plane, the deformation appears discontinuous on the microscopic level of the crystal grains. The overall effect, however, is plastic shear along certain slip planes. As the deformation continues, a locking of the dislocations takes place, resulting in *strain hardening*.

3.3.2 One-dimensional plasticity

Let σ be a uniaxial stress, such as the axial stress in a tensile test specimen, and ε the corresponding axial strain. In general formulations, yielding is defined by $F = 0$, where F is called a *yield function*. For uniaxial stress σ , $F = |\sigma| - \sigma_Y$, where σ_Y is always taken as positive [4].

For strains larger than ε_Y , a strain increment $d\varepsilon$ can be regarded as composed of an elastic contribution $d\varepsilon^e$ and a plastic contribution $d\varepsilon^p$.

$$d\varepsilon = d\varepsilon^e + d\varepsilon^p \quad (3.2)$$

Thus, for strain in the plastic range we get

$$d\sigma = E d\varepsilon^e \quad (3.3)$$

$$d\sigma = E(d\varepsilon - d\varepsilon^p) \quad (3.4)$$

$$d\sigma = E_t d\varepsilon \quad (3.5)$$

$$d\sigma = H_p d\varepsilon^p \quad (3.6)$$

where H_p is called the *strain-hardening parameter* or the *plastic modulus*. Substitution of Eqs. (3.4) and (3.6) into Eq. (3.5) yields

$$H_p = \frac{E_t}{1 - (E_t/E)} \quad \text{or} \quad E_t = E \left(1 - \frac{E}{E + H_p} \right) \quad (3.7)$$

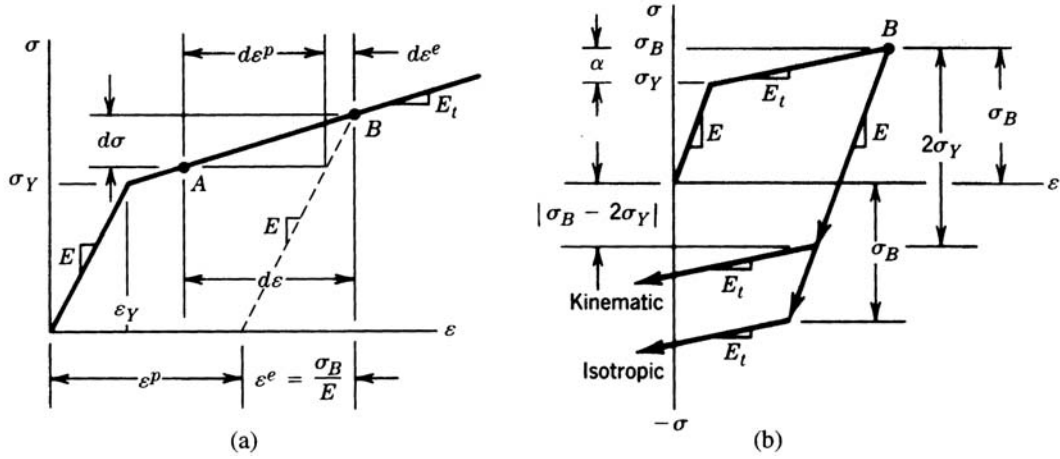


Figure 3.4: (a) Stress-strain relation for uniaxial stress, (b) isotropic and kinematic hardening rules. Reproduced from [4].

3.3.3 Multidimensional plasticity

Prior to any yielding, many materials display almost linear and elastic response, so that stresses can be calculated by knowing elastic constants and strains. When there is yielding, load, deformation, and stress are nonlinearly related and history dependent.

Incremental plasticity relations

As in one-dimensional plasticity strain increments are regarded as composed of recoverable (elastic) and nonrecoverable (plastic) components [4]:

$$d\epsilon = d\epsilon^e + d\epsilon^p \quad (3.8)$$

Stress increments are associated only with the elastic component

$$d\sigma = \mathbf{E}d\epsilon^e \quad \text{or} \quad d\sigma = \mathbf{E}(d\epsilon - d\epsilon^p) \quad (3.9)$$

where $\mathbf{E}$ is the elastic material property matrix.

The three essential ingredients of elastic-plastic analysis are [3]

- The *existence* of an initial yield surface which defines the elastic limit of the material in a multiaxial state of stress.
- The *hardening rule* which describes the evolution of subsequent yield surfaces.
- The *flow rule* which is related to a plastic potential function and defines the direction of the incremental plastic strain vector in strains space.

Let the yield function be written as

$$F = F(\boldsymbol{\sigma}, \boldsymbol{\alpha}, W_p) \quad (3.10)$$

where $\boldsymbol{\alpha}$ and W_p account for hardening by describing how a yield surface in multidimensional stress space is altered, by changes in location or size, in response to plastic strains [4].

The flow rule is stated in terms of a function Q , which has units of stress and is called a plastic potential. With $d\lambda$ a scalar that may be called a plastic multiplier, plastic strain increments are given by

$$d\boldsymbol{\varepsilon}^p = \frac{\partial Q}{\partial \boldsymbol{\sigma}} d\lambda \quad (3.11)$$

Hardening can be modelled as isotropic or kinematic, either separately or in combination. Isotropic hardening can be represented by plastic work per unit volume, W_p which describes growth of the yield surface. Kinematic hardening can be represented by a vector $\boldsymbol{\alpha}$, which accounts for translation of the yield surface in stress space. Symbolically,

$$W_p = \int_V \boldsymbol{\sigma}^T d\boldsymbol{\varepsilon}^p \quad \boldsymbol{\alpha} = \int_V \mathbf{C} d\boldsymbol{\varepsilon}^p \quad (3.12)$$

where the latter expression follows from integration of

$$d\boldsymbol{\alpha} = \mathbf{C} d\boldsymbol{\varepsilon}^p \quad \text{in which} \quad \mathbf{C} = \frac{2}{3} H_p \begin{bmatrix} 1 & & & & & \\ & 1 & & & & \\ & & 1 & & & \\ & & & \frac{1}{2} & & \\ & & & & \frac{1}{2} & \\ & & & & & \frac{1}{2} \end{bmatrix} \quad (3.13)$$

$\mathbf{C}$ is not a unit matrix because $d\boldsymbol{\varepsilon}$ use the engineering definition of shear strain rather than the tensor definition. Plastic flow takes place at constant volume $d\varepsilon_x^p + d\varepsilon_y^p + d\varepsilon_z^p = 0$; hence $\begin{bmatrix} 1 & 1 & 1 & 0 & 0 & 0 \end{bmatrix} \boldsymbol{\alpha} = \alpha_x + \alpha_y + \alpha_z = 0$.

The simplest work-hardening rule is based on the assumption that the initial yield surface expands uniformly without distortion and translation as plastic flow occurs.

The size of the yield surface is governed by the value $d\epsilon^p$. The isotropic model is simple to use, but it applies mainly to monotonic loading without stress reversals. Because the loading surface expands uniformly (or isotropically) and remains self-similar with increasing plastic deformation, it cannot account for the *Bauschinger effect* exhibited by most structural materials. The term Bauschinger effect refers to a particular type of directional anisotropy induced by a plastic deformation; namely, an initial plastic deformation of one sign reduces the resistance of the material with respect to a subsequent plastic deformation of the opposite sign [3].

The kinematic hardening rule assumes that during plastic deformation, the loading surface translates as a rigid body in stress space, maintaining the size, shape, and orientation of the initial yield surface. This hardening rule provides a simple means of accounting for the Bauschinger effect. As a consequence of assuming a rigid-body translation of the loading surface, the kinematic hardening rule predicts an ideal Bauschinger effect for a complete reversal of loading conditions.

Incremental Stress-Strain Relations

During an increment of plastic straining [4], $dF = 0$, we obtain from Eq. (3.10)

$$\left[\frac{\partial F}{\partial \boldsymbol{\sigma}} \right]^T d\boldsymbol{\sigma} + \left[\frac{\partial F}{\partial \boldsymbol{\alpha}} \right]^T d\boldsymbol{\alpha} + \frac{\partial F}{\partial W_p} dW_p = 0 \quad (3.14)$$

Substitution of Eq. (3.10) into Eqs. (3.9), (3.12), and (3.13) provides

$$\begin{aligned} d\boldsymbol{\sigma} &= \mathbf{E} \left(d\boldsymbol{\epsilon} - \frac{\partial Q}{\partial \boldsymbol{\sigma}} d\lambda \right) \\ dW_p &= \boldsymbol{\sigma}^T \frac{\partial Q}{\partial \boldsymbol{\sigma}} d\lambda \\ d\boldsymbol{\alpha} &= \mathbf{C} \frac{\partial Q}{\partial \boldsymbol{\sigma}} d\lambda \end{aligned} \quad (3.15)$$

These expressions are substituted into Eq. (3.14) and the resulting equation solved for the plastic multiplier $d\lambda$. Thus we obtain

$$d\lambda = \mathbf{P}_\lambda d\boldsymbol{\epsilon} \quad (3.16)$$

where $\mathbf{P}_\lambda$ is the row matrix

$$\mathbf{P}_\lambda = \frac{\left[\frac{\partial F}{\partial \boldsymbol{\sigma}} \right]^T \mathbf{E}}{\left[\frac{\partial F}{\partial \boldsymbol{\sigma}} \right]^T \mathbf{E} \frac{\partial Q}{\partial \boldsymbol{\sigma}} - \left[\frac{\partial F}{\partial \boldsymbol{\alpha}} \right]^T \mathbf{C} \frac{\partial Q}{\partial \boldsymbol{\sigma}} - \frac{\partial F}{\partial W_p} \boldsymbol{\sigma}^T \frac{\partial Q}{\partial \boldsymbol{\sigma}}} \quad (3.17)$$

Finally to get the relationship between the stresses and the strains we put Eq. (3.16) into Eq. (3.15) and we obtain

$$d\boldsymbol{\sigma} = \mathbf{E}_{ep} d\boldsymbol{\varepsilon} \quad \text{where} \quad \mathbf{E}_{ep} = \mathbf{E} \left(\mathbf{I} - \frac{\partial Q}{\partial \boldsymbol{\sigma}} \mathbf{P}_\lambda \right) \quad (3.18)$$

3.3.4 Von Mises plasticity in three dimensions

For the general three-dimensional case, the von Mises criterion is [5]:

$$\begin{aligned} F &= \sigma_e - \sigma_0 = \sqrt{3J_2} - \sigma_0 \\ &= \frac{1}{\sqrt{2}} [(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)]^{1/2} - \sigma_0 \\ &= \sqrt{3} \left[\frac{1}{2} (s_x^2 + s_y^2 + s_z^2) + \tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2 \right]^{1/2} - \sigma_0 \\ &= \sqrt{\frac{3}{2}} (\mathbf{s}^T \mathbf{L} \mathbf{s})^{1/2} - \sigma_0 \end{aligned} \quad (3.19)$$

where

$$\mathbf{L} = \begin{bmatrix} 1 & & & & & \\ & 1 & & & & \\ & & 1 & & & \\ & & & 2 & & \\ & & & & 2 & \\ & & & & & 2 \end{bmatrix} \quad \text{and} \quad \mathbf{s}^T = \{s_x, s_y, s_z, \tau_{xy}, \tau_{yz}, \tau_{zx}\} \quad (3.20)$$

are deviatoric stresses. σ_0 is the largest value of σ_e reached in previous plastic straining. The case $F < 0$ describes elastic conditions. The case $F = 0$ defines yielding. The case $F > 0$ is not physically possible.

Deviatoric stresses play a prominent role in von Mises theory. Any stress state can be represented as the sum of a hydrostatic state and a deviatoric state. A hydrostatic state produces no change of shape. A deviatoric state produces no change of volume. Deviatoric shear stresses are the same as actual shear stresses. Deviatoric normal stresses are actual normal stress minus the mean normal stress σ_m , where $\sigma_m = (\sigma_x + \sigma_y + \sigma_z)/3$ [4].

$$\mathbf{s} = \begin{bmatrix} s_x \\ s_y \\ s_z \end{bmatrix} = \begin{bmatrix} \sigma_x - \sigma_m \\ \sigma_y - \sigma_m \\ \sigma_z - \sigma_m \end{bmatrix} = \frac{1}{3} \begin{bmatrix} 2\sigma_x - \sigma_y - \sigma_z \\ 2\sigma_y - \sigma_z - \sigma_x \\ 2\sigma_z - \sigma_x - \sigma_y \end{bmatrix} \quad (3.21)$$

For three-dimensional plasticity, the equivalent plastic strain is given by [5]

$$\begin{aligned} d\lambda &= d\varepsilon_e^p = \\ &= \sqrt{\frac{2}{3}} \left[(d\varepsilon_x^p)^2 + (d\varepsilon_y^p)^2 + (d\varepsilon_z^p)^2 + \frac{1}{2} ((d\gamma_{xy}^p)^2 + (d\gamma_{yz}^p)^2 + (d\gamma_{zx}^p)^2) \right]^{1/2} \end{aligned} \quad (3.22)$$

It is possible to do a combination of the isotropic hardening and the kinematic hardening. Then Eq. (3.19), with an introduced number η ($0 < \eta < 1$), yields

$$F = \sqrt{\frac{3}{2}} \left\{ [(s_x - \eta\alpha_x)^2 + (s_y - \eta\alpha_y)^2 + (s_z - \eta\alpha_z)^2] + 2 [(s_{xy} - \eta\alpha_{xy})^2 + (s_{yz} - \eta\alpha_{yz})^2 + (s_{zx} - \eta\alpha_{zx})^2] \right\}^{1/2} - \eta\sigma_Y - (1 - \eta)\sigma_0 \quad (3.23)$$

where σ_Y is the von Mises stress σ_e , or the magnitude of uniaxial stress, at initial yielding. Translation of the yield surface is controlled by α . Hardening is purely isotropic if $\eta = 0$ and purely kinematic if $\eta = 1$.

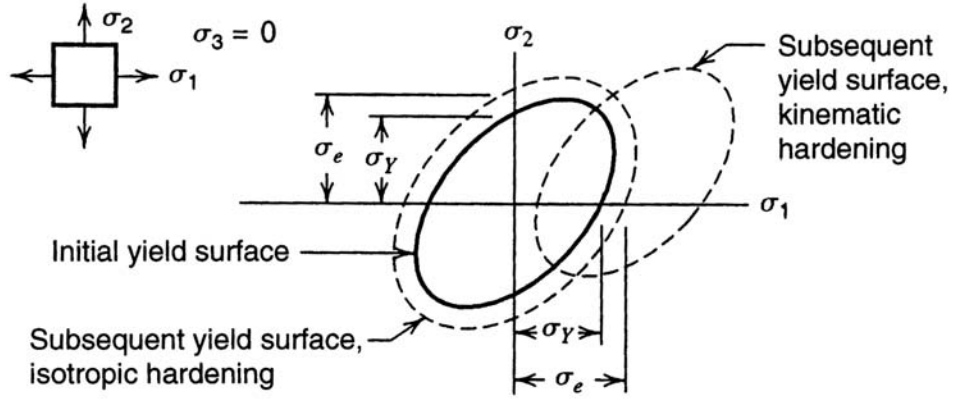


Figure 3.5: Hardening rules. Reproduced from [4].

Chapter 4

ABAQUS

The ABAQUS¹ suite of software for finite element analysis consists of three main products [1]:

- ABAQUS/Standard
- ABAQUS/Explicit
- ABAQUS/CAE

The standard package solves static, dynamic and thermal problems. The explicit package focus on transient dynamics and quasi-static analysis. The CAE package is a CAD-like tool to create models for analysis and for visualisation of the results.

In this thesis the models have been created in ABAQUS/CAE and analysis package used is ABAQUS/Standard, hence all analyses are static.

4.1 Creating models

The CAE package is using different modules. These modules are used in the order they are presented so the models are created with the same procedure.

1. *Part Module*. The parts are created using the Graphical User Interface (GUI).
2. *Property Module*. All the material properties are given such as elastic and plastic behaviour. The orientation of beams, etc., are given.
3. *Assembly Module*. The parts are imported to create the geometry of the model, i.e., to build the complete structure.

¹<http://www.abaqus.com>

4. *Step Module*. This module decides which type of analysis that is going to be used. The analysis is divided into one or more steps. These steps capture the changes in the model. Here are the output requests defined. There are two different procedures for the steps:

- *General*. These steps define sequential events: the state of the model at the end of one general step provides the initial state for the start of the next general step.
- *Linear Perturbation*. These steps provide the linear response of the model about the state reached at the end of the last general nonlinear step.

It is also possible to choose if ABAQUS should account for nonlinear effects from large displacements and deformations. If the displacements in model due to loading are relatively small during a step, the effects may be small enough to be ignored.

5. *Interaction Module*. Here are all the relationships between the parts defined.
6. *Load Module*. In this module, all the loads and boundary conditions are defined. The loads are step-dependent.
7. *Mesh Module*. This module generates meshes on the assemblies. One part can be divided into different meshes and different elements too.
8. *Job Module*. The jobs are created and submitted for analysis. It is possible to submit and write only input files for later usage.
9. *Visualisation Module*. The results of the analysis can be visualised in this module. It is possible to make different plots on selected data.

4.2 Plasticity in ABAQUS

This section explains how ABAQUS handles plasticity.

A basic assumption of elastic-plastic models is that the deformation can be divided into an elastic part and an inelastic (plastic) part [7]. In its most general form this statement is written as

$$\mathbf{F} = \mathbf{F}^{el} \cdot \mathbf{F}^{pl} \quad (4.1)$$

where $\mathbf{F}$ is the total deformation gradient, $\mathbf{F}^{el}$ is the fully recoverable part of the deformation at the point under consideration, and $\mathbf{F}^{pl}$ is defined by $\mathbf{F}^{pl} = [\mathbf{F}^{el}]^{-1} \cdot \mathbf{F}$.

This decomposition can be used directly to formulate the plasticity model. Historically, an additive strain rate decomposition,

$$\dot{\boldsymbol{\epsilon}} = \dot{\boldsymbol{\epsilon}}^{el} + \dot{\boldsymbol{\epsilon}}^{pl}, \quad (4.2)$$

has been used.

The elastic part of the response is assumed to be derived from an elastic strain energy potential, so the stress is defined by

$$\boldsymbol{\sigma} = \frac{\partial U}{\partial \boldsymbol{\varepsilon}^{el}} \quad (4.3)$$

where U is the strain energy density potential. It is assumed that, in the absence of plastic straining, the variation of elastic strain is the same as the variation in the rate of deformation, conjugacy arguments define the stress measure $\boldsymbol{\sigma}$ as the *true* (Cauchy) stress. All stress output in ABAQUS is given in this form.

4.2.1 Isotropic elasto-plasticity

The von Mises yield function with associated flow means that there is no volumetric plastic strain. Since the elastic bulk modulus is quite large, the volume change will be small. Thus, we can define the volume strain as

$$\varepsilon_{\text{vol}} = \text{trace}(\boldsymbol{\varepsilon}) \quad (4.4)$$

and, hence, the deviatoric strain is

$$\boldsymbol{e} = \boldsymbol{\varepsilon} - \frac{1}{3}\varepsilon_{\text{vol}}\boldsymbol{I} \quad (4.5)$$

This is the material model that is used for the analyses made in this thesis.

4.2.2 Material data

The *PLASTIC option is used to specify the plastic part of the material model for elastic plastic materials that use the von Mises yield surface. The plastic material data should be Cauchy stress and logarithmic strain. In case where the material data is nominal stress-strain, a simple conversion to true stress and logarithmic plastic strain is

$$\begin{aligned} \sigma_{\text{true}} &= \sigma_{\text{nom}}(1 + \varepsilon_{\text{nom}}) \\ \varepsilon_{\text{ln}}^{pl} &= \ln(1 + \varepsilon_{\text{nom}}) - \frac{\sigma_{\text{true}}}{E} \end{aligned} \quad (4.6)$$

4.3 General analysis step

In a general step, the effects of any nonlinearities present in the model can be included. The starting condition for each general step is the ending condition from the last general step. The state of the model evolves throughout the history of general analysis steps as it responds to the history of loading [7].

4.3.1 Nonlinearity

Nonlinear stress analysis problems can contain up to three sources of nonlinearity:

- *Material nonlinearity.* Many materials are history dependent. The general analysis procedures are designed with this in view.
- *Geometric nonlinearity.* It is possible to define a problem as a small-displacement analysis by omitting the NLGEOM parameter from the *STEP option. This omission means that geometric nonlinearity is ignored—the kinematic relationships are linearised.

The alternative to a small-displacement analysis is to include large-displacement effects by including the NLGEOM parameter. When NLGEOM is specified most elements are formulated in the current configuration using current nodal positions. Elements therefore distort from their original shapes as the deformation increases.

- *Boundary nonlinearity.* Contact problems are a common source of nonlinearity in stress analysis. Other sources are nonlinear elastic springs, films, radiation, multi-point constraints, etc.

4.3.2 Incrementation

In nonlinear problems the challenge is always to obtain a convergent solution in the shortest computational time. The general analysis procedures offer two approaches for controlling incrementation.

- *Direct user control.* The user specifies the incrementation scheme.
- *Automatic control.* ABAQUS automatically selects the increment size as it develops the response in the step.

4.4 Elements

ABAQUS offers a wide variety of elements. The analyses in this thesis use four different types of these elements; shell elements, solid elements, beam elements and rigid elements. This section will only mention the basics of the elements used. To learn more about each element see [7].

4.4.1 Shell elements

ABAQUS has three categories of shell elements; general-purpose, thin and thick shell elements. Thin elements provide solutions to shell problems that are adequately

described by classical (Kirchhoff) shell theory, thick shell elements yield solutions for structures that are best modelled by shear flexible (Mindlin) shell theory, and general-purpose structures shell elements can provide solutions to both thin and thick shell problems.

The general-purpose shell elements are axisymmetric elements SAX1, SAX2, and SAX2T and three-dimensional elements S3, S4, S3R, S4R, S4RS, S3RS, and S4RSW. The general-purpose elements provide robust and accurate solutions in all loading conditions for thin and thick shell problems.

The thin elements are STRI3, STRI65, S4R5, S8R5, S9R5, and SAXA. Thin shell elements may provide enhanced performance for large problems where reducing the number of degrees of freedom through the use of five degree of freedom shells is desirable.

The thick elements are S8R and S8RT. Non-negligible transverse shear flexibility is required for these elements to function properly.

The analyses in this thesis that use shell elements use the S4R element, which is a 4-node stress/displacement shell element with reduced integration and six degrees of freedom per node.

4.4.2 Solid elements

Solid elements are provided with first-order (linear) and second-order (quadratic) interpolation. Standard first-order elements are essentially constant strain elements. The second-order elements are capable of representing all possible linear strain fields. Thus, in the case of elliptic problems much higher solution accuracy per degree of freedom is usually available with the higher-order elements.

For elliptic applications second-order elements are preferred. Though the accuracy per degree of freedom is higher, the accuracy per computational cost may not be increasing. ABAQUS does not include elements beyond second-order. Practical experience suggests that little is gained with those elements.

As the solution approaches the limit load, most plasticity modes tend toward hyperbolic behaviour. This allows discontinuities to occur in the solution. With a fixed mesh that does not use special element that admit discontinuities in their formulation, the first-order elements are likely to be most successful. For a given number of nodes, they provide the most locations at which some component of the gradient of the solution can be discontinuous. Thus the used element is C3D8R which is a 8-node linear brick with three degrees of freedom per node. In corners and complicated parts the element C3D4 is used. It is a 4-node linear tetrahedron with three degrees of freedom at each node.

All of the solid elements in ABAQUS are written to include finite-strain effects. The strains are calculated as the integral of the rate of deformation

$$\mathbf{D} = \text{sym} \left(\frac{\partial \mathbf{v}}{\partial \mathbf{x}} \right) \quad (4.7)$$

In all cases the solid elements report stress as the *true* (Cauchy) stress.

4.4.3 Beam elements

A beam in this context is an element in which assumptions are made so that the problem is reduced to one dimension mathematically. The simplest approach to beam theory is the classical Euler-Bernoulli assumption. The beam elements that use cubic interpolation (B23, B33, etc.) all use this assumption. This approximation can also be used to formulate beams for large axial strains as well as large rotations. The beam elements in ABAQUS that use linear and quadratic interpolation (B21, B22, B31, B32 etc.) are based on such a formulation, with the addition that these elements also allow transverse shear strains, i.e. the cross-section may not remain normal to the beam axis. This extension leads to Timoshenko beam theory. The large-strain formulation in these elements allows axial strain of arbitrary magnitude, but quadratic terms in the nominal torsional strain are neglected compared to unity, and the axial strain is assumed to be small in the calculation of the torsional shear strain.

4.4.4 Rigid elements

Rigid elements are associated with a given rigid body and share a common node known as the rigid body reference node. A rigid element can be used to define the surfaces of rigid bodies for contact or to define rigid bodies for multibody dynamic simulations. They can also be attached to deformable elements or be used to constrain parts of a model. The rigid element used in the T-Joint analyses is R3D3 which is a rigid, 4-node element in three dimensions.

Chapter 5

Mast Foundation

5.1 Introduction

The foundation that is analysed in this thesis is located on the roof of the fire station in Handen outside Stockholm, Sweden. It is mainly constructed of rectangular hollow sections. The foundation is well constructed. Between the main frame and the legs, supports are constructed. Every end of the beams in the main frame have a welded end plate to make the end section even more stiff. The aim of this thesis is to find out how all these different parts in the corners affect the foundation and if they are necessary. It is also of interest to see how the forces are distributed in the corners of the frame.

The model can be divided into three different models. Every model has different geometries. There are totally three different models and nine different geometries.

1. Model with coarser mesh (section 5.3.1)
2. Model with finer mesh (section 5.3.2)
3. Optimised Model (section 5.3.3)

All the models are constructed in three dimensions.

5.2 Data for the models

5.2.1 Section properties

Every beam of the foundation is made of hot-rolled steel S355J2H. The plates that connects to the mast are made of steel S355J0. The section data for the beams and the data for the plates are as follows

Table 5.1: Section dimensions and material.

Part	Dimension	Material
Vertical beams	VKR 120×120×8	S355J2H
Horizontal beams	VKR 250×150×8	S355J2H
Legs	VKR 150×150×10	S355J2H
Supports	VKR 150×150×10	S355J2H
Beam end plate	PL 8	S355J2H
Mast foot plate	PL 20×190×190	S355J0

5.2.2 Elasticity and density

Both materials, S355J2H and S355J0 have the same density ρ , modulus of elasticity E and Poisson's ratio ν .

$$E = 210 \text{ GPa}$$

$$\nu = 0.3$$

$$\rho = 7800 \text{ kg/m}^3$$

5.2.3 Plasticity data

As mentioned before ABAQUS uses true stress and logarithmic strains. To calculate the stress-strain diagram to see how the metal yields the working curve in [2] is used. That curve gives the nominal stress-strains and therefore it is necessary to recalculate these stress-strains to true stress and logarithmic strain and thus gain a true stress-strain diagram.

Stress-strain diagram for steel

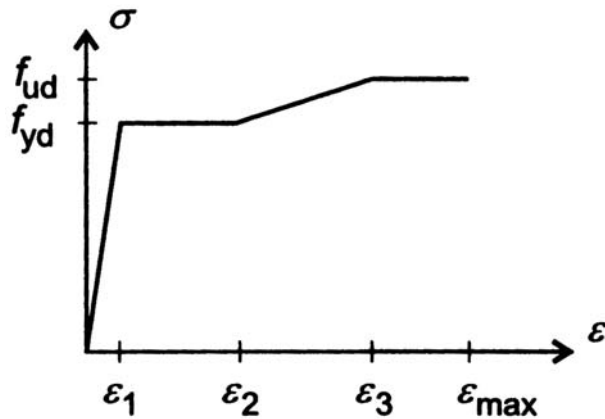


Figure 5.1: Stress-strain diagram. Reproduced from [2].

$$\begin{aligned}\varepsilon_1 &= \frac{f_{yd}}{E_d} \\ \varepsilon_2 &= 0.025 - 5 \frac{f_{ud}}{E_d} \\ \varepsilon_3 &= 0.02 + 50 \frac{f_{ud} - f_{yd}}{E_d} \\ \varepsilon_{\max} &= 0.6 A_5\end{aligned}$$

where

$$\begin{aligned}f_{yd} &= \frac{f_{yk}}{\gamma_m \gamma_n} \\ f_{ud} &= \frac{f_{uk}}{1.2 \gamma_m \gamma_n} \\ E_d &= \frac{E_k}{\gamma_m \gamma_n}\end{aligned}$$

A_5 according to SS 01 66 02

S355J2H

By introducing the characteristic numbers of S355J2H we get the nominal strains. These strains with the yield point and the rupture point form the stress-strain diagram:

$$\begin{aligned}\varepsilon_1 &= \frac{f_{yd}}{E_d} = \frac{355 \cdot 10^6}{210 \cdot 10^9} = 0.001690 \\ \varepsilon_2 &= 0.025 - 5 \frac{f_{ud}}{E_d} = 0.025 - 5 \frac{408.3333 \cdot 10^6}{210 \cdot 10^9} = 0.015278 \\ \varepsilon_3 &= 0.02 + 50 \frac{f_{ud} - f_{yd}}{E_d} = 0.02 + 50 \frac{408.3333 \cdot 10^6 - 355 \cdot 10^6}{210 \cdot 10^9} = 0.032698 \\ \varepsilon_{\max} &= 0.6 A_5 = 0.6 \cdot 20\% = 0.12\end{aligned}$$

S355J0

The same procedure goes for S355J0:

$$\begin{aligned}\varepsilon_1 &= \frac{f_{yd}}{E_d} = \frac{345 \cdot 10^6}{210 \cdot 10^9} = 0.001643 \\ \varepsilon_2 &= 0.025 - 5 \frac{f_{ud}}{E_d} = 0.025 - 5 \frac{408.3333 \cdot 10^6}{210 \cdot 10^9} = 0.015278 \\ \varepsilon_3 &= 0.02 + 50 \frac{f_{ud} - f_{yd}}{E_d} = 0.02 + 50 \frac{408.3333 \cdot 10^6 - 345 \cdot 10^6}{210 \cdot 10^9} = 0.035079 \\ \varepsilon_{\max} &= 0.6 A_5 = 0.6 \cdot 20\% = 0.12\end{aligned}$$

Cauchy stress and logarithmic strain

To create the stress–strain diagram with Cauchy stress and logarithmic strain we use the equations

$$\begin{aligned}\sigma_{\text{true}} &= \sigma_{\text{nom}}(1 + \varepsilon_{\text{nom}}) \\ \varepsilon_{\text{ln}}^{pl} &= \ln(1 + \varepsilon_{\text{nom}}) - \frac{\sigma_{\text{true}}}{E}\end{aligned}\tag{5.1}$$

Thus we get the values for the stress–strain diagrams:

Table 5.2: Stress–strain diagram data.

S355J2H				
σ_{nom} (MPa)	ε_{nom}	σ_{true} (MPa)	ε_{ln}	$\varepsilon_{\text{ln}}^{pl}$
0	0	0	0	0
355	0.001690	355.6001	0.001689	0
355	0.015278	360.4236	0.015162	0.013446
408.3333	0.032698	421.6852	0.032175	0.030167
408.3333	0.12	457.3333	0.113329	0.111151

Table 5.3: Stress–strain diagram data.

S355J0				
σ_{nom} (MPa)	ε_{nom}	σ_{true} (MPa)	ε_{ln}	$\varepsilon_{\text{ln}}^{pl}$
0	0	0	0	0
345	0.001690	345.5668	0.001642	0
345	0.015278	350.2708	0.015162	0.013494
408.3333	0.035079	422.6574	0.034478	0.032465
408.3333	0.12	457.3333	0.113329	0.111151

The columns σ_{true} and $\varepsilon_{\text{ln}}^{pl}$ gives the input data for the plasticity in ABAQUS.

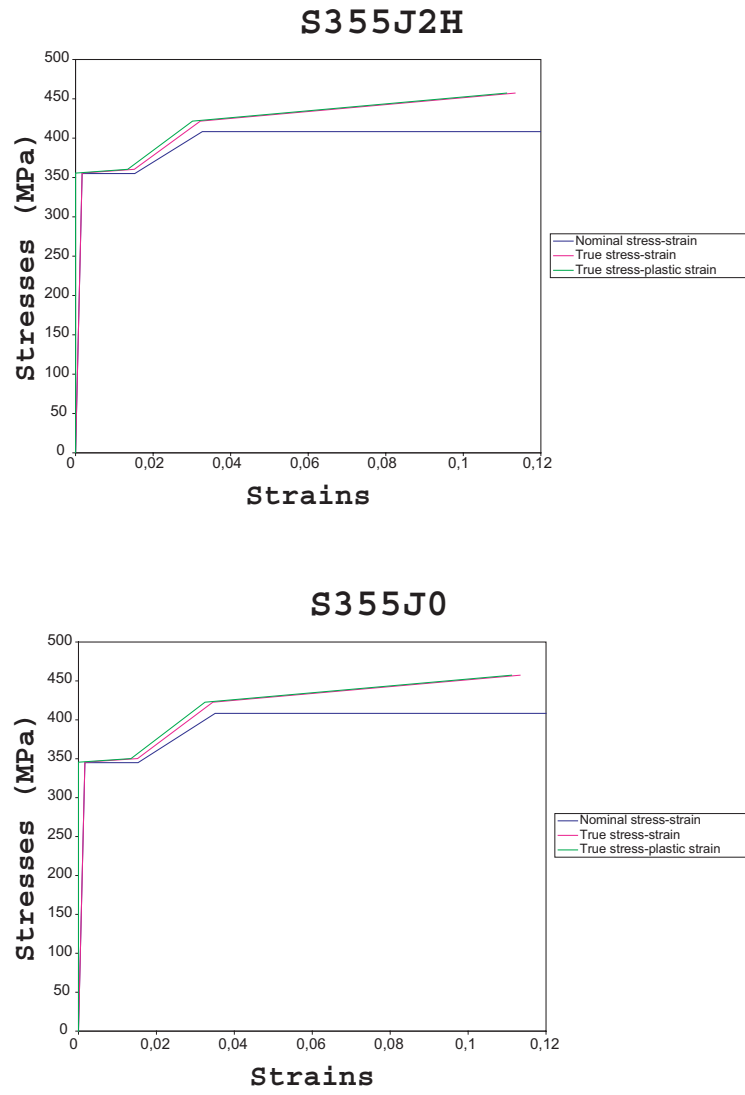


Figure 5.2: The stress-strain diagrams.

5.3 Models

All models use mainly shell elements and the analysis step is a general static step.

5.3.1 Model with coarser mesh

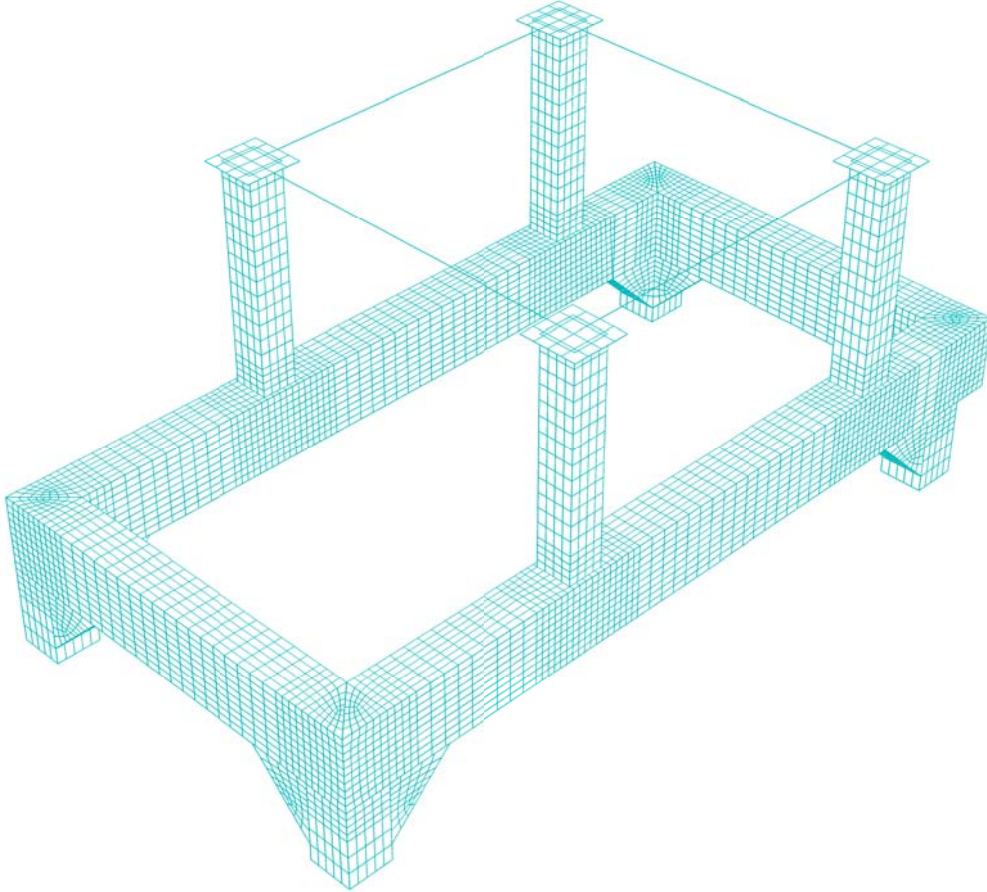


Figure 5.3: The mesh for the 6 series.

There are four different geometries of the model. All geometries are very similar. The differences lie in the corners. The first geometry is a copy of the already constructed foundation. The second geometry does not use any supports between the frame and the legs. The third geometry uses the supports but not the welded plates at the beam ends. The fourth geometry uses neither the supports nor the end plates.

The geometry of the model is taken from [12]. The mast is connected to the foundation via four vertical beams (also called mast foot in this thesis). At the top of every vertical beam there is a thick plate. To prevent that the vertical beams of the foundation spreads upon loading, a beam is placed between each top plate. These beams are modelled as beam elements. The rest of the model use shell elements.

The four vertical beams connect to two long horizontal beams. These two horizontal beams join each other via two other horizontal beams making a rectangular frame. The beams are welded together at a diagonal cut at the ends. This frame is resting on four legs. Note that all the legs are not in the corners. Leg number three is actually placed almost entirely on the second long horizontal beam.

5.3.2 Model with finer mesh

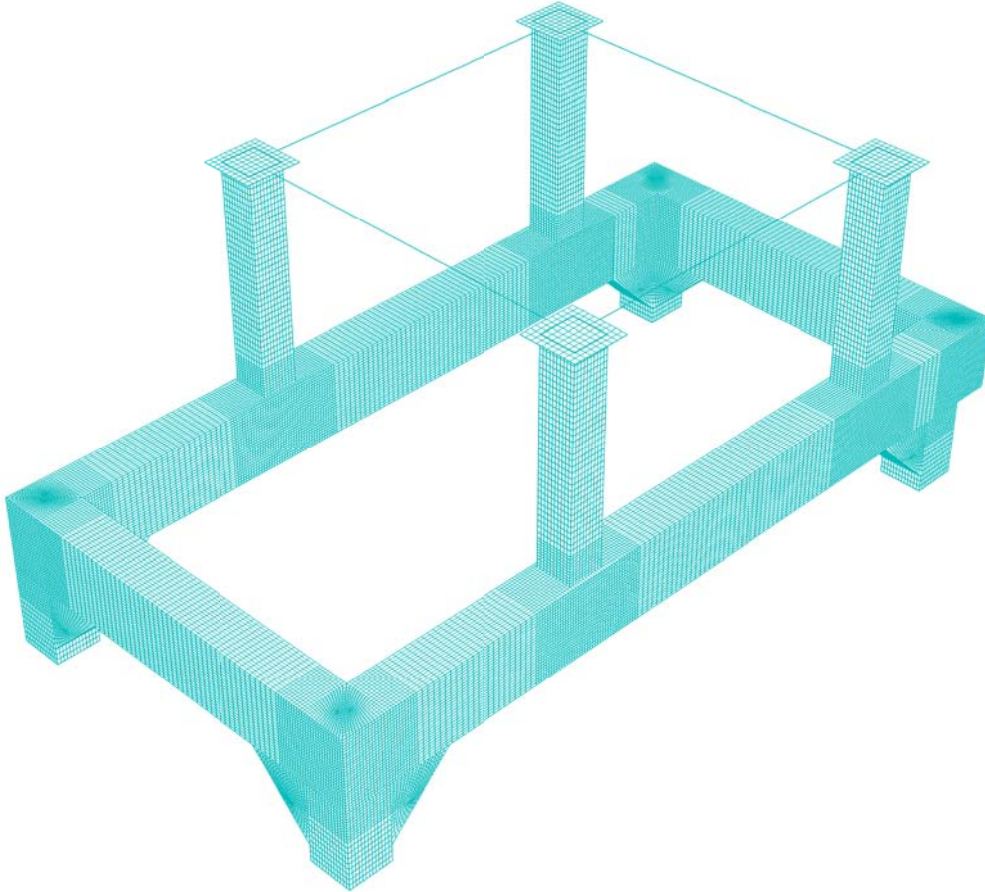


Figure 5.4: The mesh for the 7 series.

This model has the same geometries as the previous model. The only difference is the finer mesh which can be seen in Fig. 5.4.

5.3.3 Optimised model

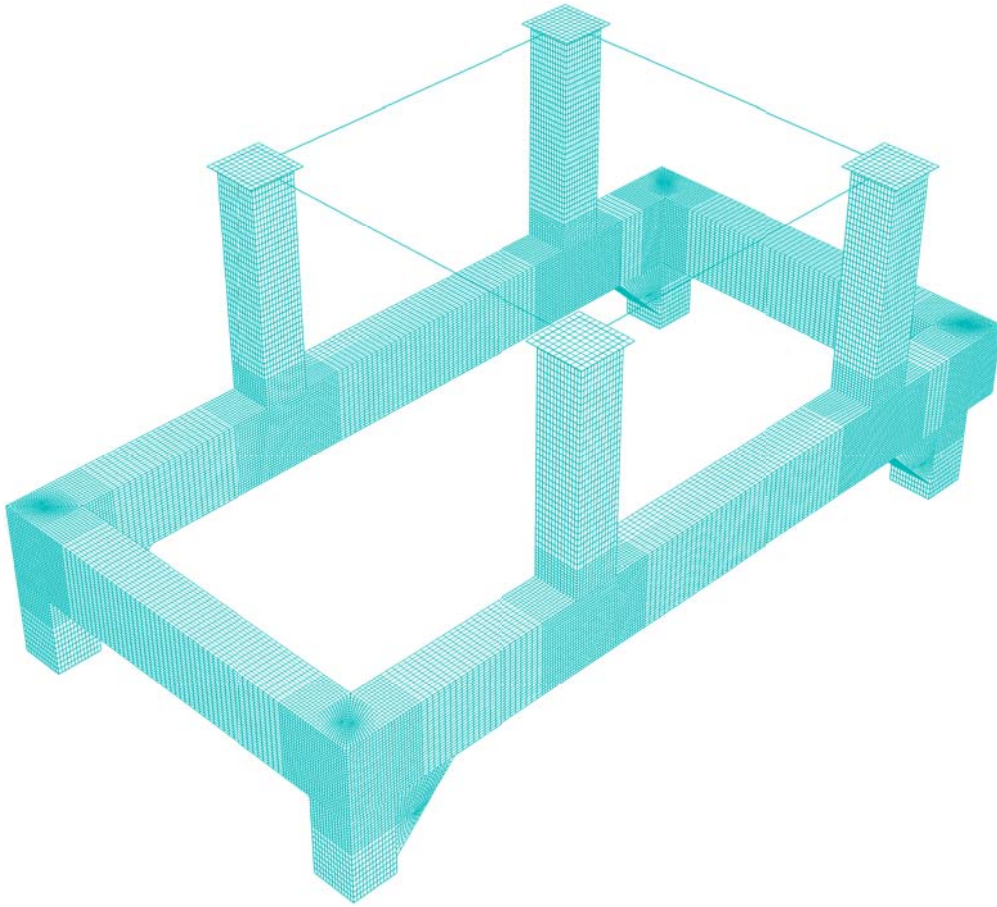


Figure 5.5: The mesh for the optimised geometry.

The expensive part of a structure like this one are the welds. Taking away parts means less welds and work, thus it is more economical. But it will also be weaker. An optimisation was made.

The model is a optimised model derived from the other models. The mesh is the same as in the models with the finer mesh. It is only the geometry that has been optimised. As can be seen in the results (section 5.5) the end plates did not seem to be needed for strength. Neither did the supports between the legs and the short horizontal beams. Therefore an analysis without these parts were done. The mast feet are changed. The original ones are smaller than the horizontal beams. In this model the mast feet are of the same dimension as the frame, i.e., VKR 120×120×8 is changed to VKR 150×150×8.

5.4 Loads

The design loads for the foundation are taken from [12]. They are as follows

Vertical load, max	15 kN
Vertical load, min	4 kN
Horizontal load	50 kN
Moment	345 kNm

All loads are applied to the models at the same time. Only the maximum value of the vertical load is used in the analyses. The vertical and the horizontal loads are equally distributed on the four mast feet. They are simulated as concentrated forces in the middle of the mast foot plate on each mast foot. The moment however is simulated with pressure on these plates. A moment can be seen as a force couple. Therefore the pressure works in pairs on the plates, depending on the direction of the supposed wind on the mast.

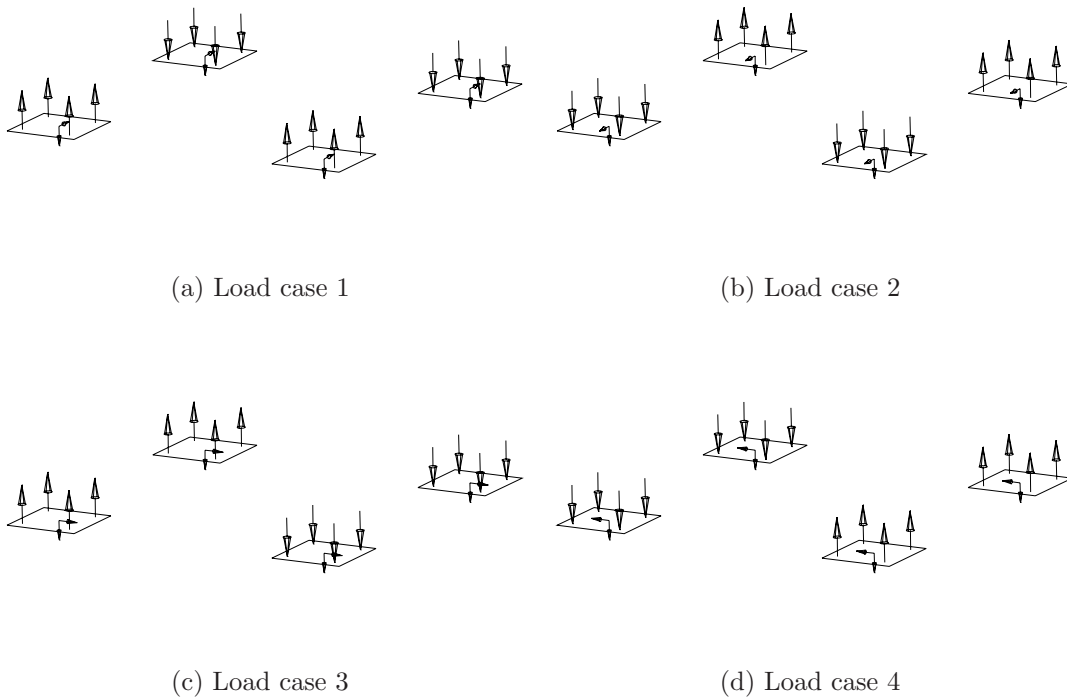


Figure 5.6: The different load cases.

The amount of the pressure on the mast foot plates has been calculated. As mentioned before, the moment can be replaced by a force couple. This force couple is divided into two equal parts. Mathematically:

$$345 \text{ kNm} = 2 \cdot 276 \text{ kN} \cdot 0.625 \text{ m}$$

$$\frac{276}{2} = 138 \text{ kN/plate} \rightarrow \text{concentrated force to pressure} \rightarrow$$

$$\frac{138000 \text{ kN}}{36100 \text{ mm}^2} = 3.823 \text{ MPa/plate}$$

Note that the first model with the finer mesh and the optimised model are subjected to the double design load. The foundation seems to be designed for much higher load cases than the design loads. Therefore, analyses with twice the amount of the design load were performed.

5.5 Results

The results are presented individually for each model and geometry, then a comparison is made between the analyses. Every analysis performed will not be showed here. The results are read along certain paths. These paths can be seen in Fig. 5.7. Every corner has two paths. The mast foot path goes all way around the weld.

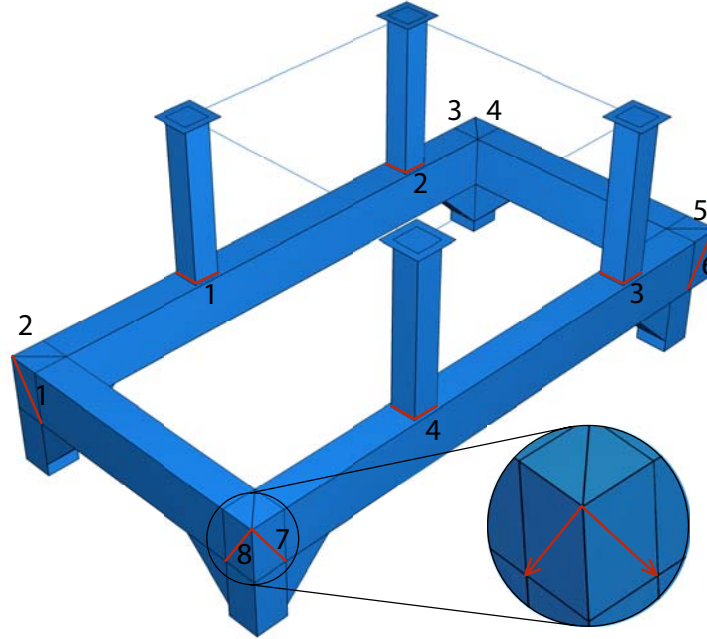


Figure 5.7: The paths. The arrows in the magnification shows the starting point and the end point for the corner paths.

The displacements listed in the tables are the displacements of the middle node in the top plate of every mast foot.

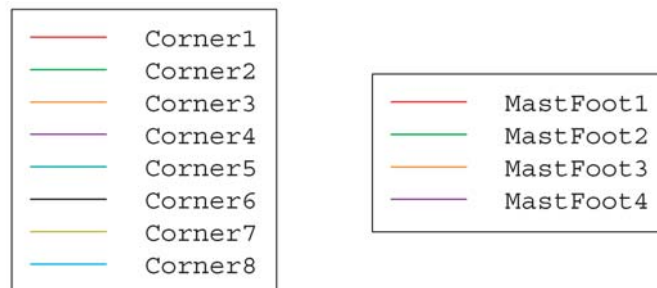


Figure 5.8: The path colours.

As can be seen in each table, the fourth load case is the governing one. This load

case is the one plotted for each model.

Every analysis have its own code. The three digits tell what model it is and which load case is applied. E.g., 6.4.2 means version 6, model 4, load case 2. Version 6 is the coarser mesh and 7 the finer mesh.

5.5.1 Model with coarser mesh

First geometry

This geometry reflects the constructed foundation. The interesting part of the structure is the t-joints and the corners. The results of these parts can be seen in the X-Y-plots. In total there are twelve different paths. Four for the mast feet and eight for the corners.

Table 5.4: Displacements U of the 6.1 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
6.1.1	MF 1	0	1	1.735	-0.000006	1.000900	1.731730	0.00339579
	MF 2	0	1	0.485	-0.000009	0.999512	0.481724	0.00331252
	MF 3	1.25	1	0.485	1.249999	0.999512	0.481724	0.00332988
	MF 4	1.25	1	1.735	1.249999	1.000910	1.731730	0.00339585
6.1.2	MF 1	0	1	1.735	0.000006	0.999020	1.738280	0.00342729
	MF 2	0	1	0.485	0.000007	1.000430	0.488287	0.00331854
	MF 3	1.25	1	0.485	1.250010	1.000360	0.488284	0.00330331
	MF 4	1.25	1	1.735	1.250010	0.999019	1.738280	0.00342416
6.1.3	MF 1	0	1	1.735	0.005075	1.001200	1.734920	0.00521489
	MF 2	0	1	0.485	0.003769	1.000770	0.484896	0.00384779
	MF 3	1.25	1	0.485	1.253760	0.999419	0.485072	0.00380730
	MF 4	1.25	1	1.735	1.255070	0.998867	1.735050	0.00519970
6.1.4	MF 1	0	1	1.735	-0.005077	0.998725	1.735090	0.00523592
	MF 2	0	1	0.485	-0.003771	0.999179	0.485114	0.00386078
	MF 3	1.25	1	0.485	1.246240	1.000540	0.484935	0.00380245
	MF 4	1.25	1	1.735	1.244920	1.001060	1.734945	0.00518691

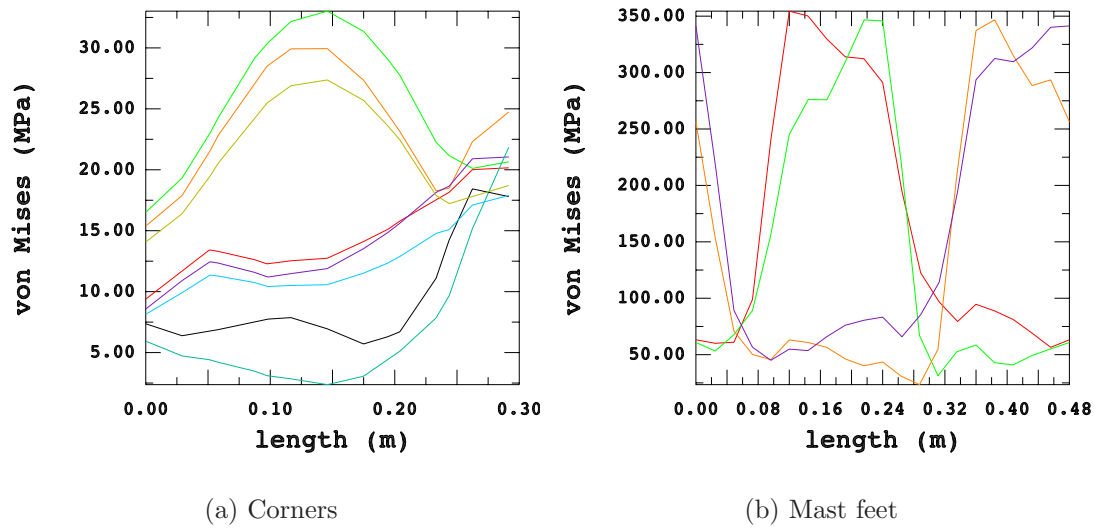


Figure 5.9: The path plots for 6.1.4.

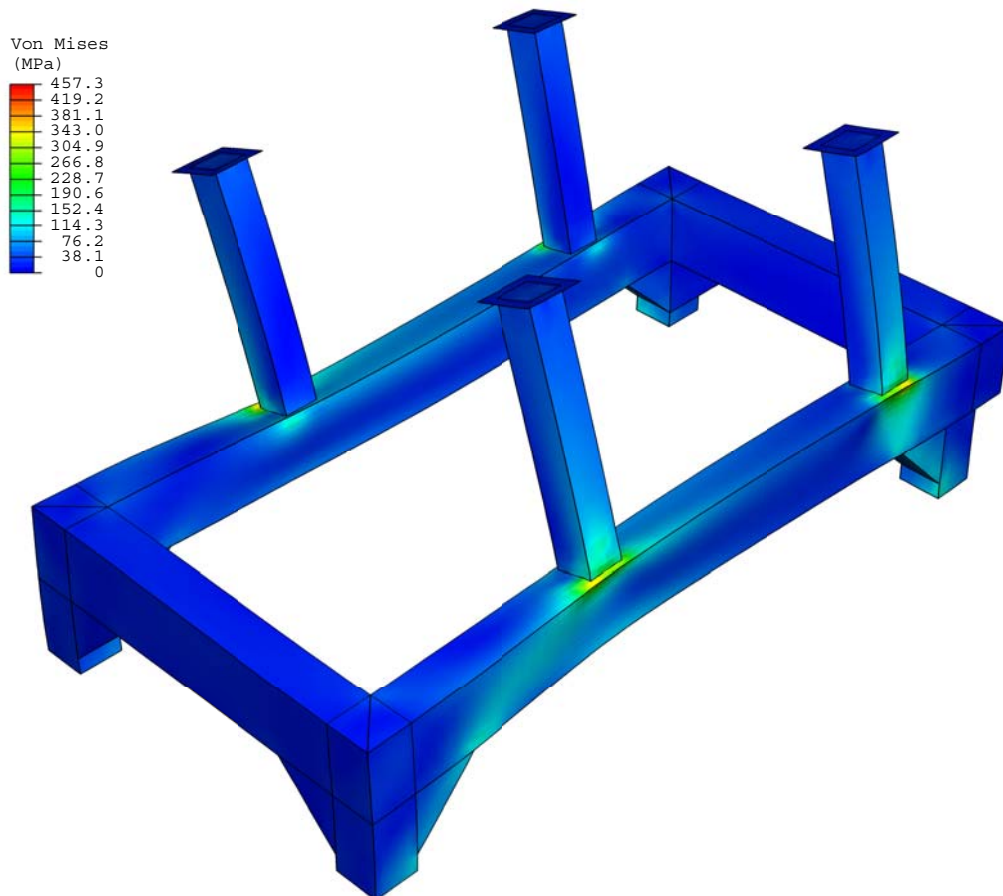


Figure 5.10: Von Mises stress for 6.1.4. Displacement scalefactor 50.

Second geometry

The geometry without the supports.

Table 5.5: Displacements U of the 6.2 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
6.2.1	MF 1	0	1	1.735	-0.000008	1.001140	1.731580	0.00360969
	MF 2	0	1	0.485	-0.000016	0.999543	0.481572	0.00345846
	MF 3	1.25	1	0.485	1.249980	0.999661	0.481559	0.00345818
	MF 4	1.25	1	1.735	1.249990	1.001200	1.731560	0.00364105
6.2.2	MF 1	0	1	1.735	0.000005	0.998726	1.738450	0.00367377
	MF 2	0	1	0.485	0.000011	1.000360	0.488451	0.00347033
	MF 3	1.25	1	0.485	1.250010	1.000270	0.488467	0.00347736
	MF 4	1.25	1	1.735	1.250000	0.998676	1.738460	0.00370477
6.2.3	MF 1	0	1	1.735	0.005859	1.002050	1.734780	0.00621124
	MF 2	0	1	0.485	0.004747	1.001380	0.484741	0.00495095
	MF 3	1.25	1	0.485	1.254740	0.998950	0.485301	0.00486441
	MF 4	1.25	1	1.735	1.255860	0.998083	1.735270	0.00617027
6.2.4	MF 1	0	1	1.735	-0.005862	0.997819	1.735240	0.00625948
	MF 2	0	1	0.485	-0.004750	0.998523	0.485280	0.00498224
	MF 3	1.25	1	0.485	1.245260	1.000980	0.484722	0.00485167
	MF 4	1.25	1	1.735	1.244140	1.001800	1.734750	0.00613624

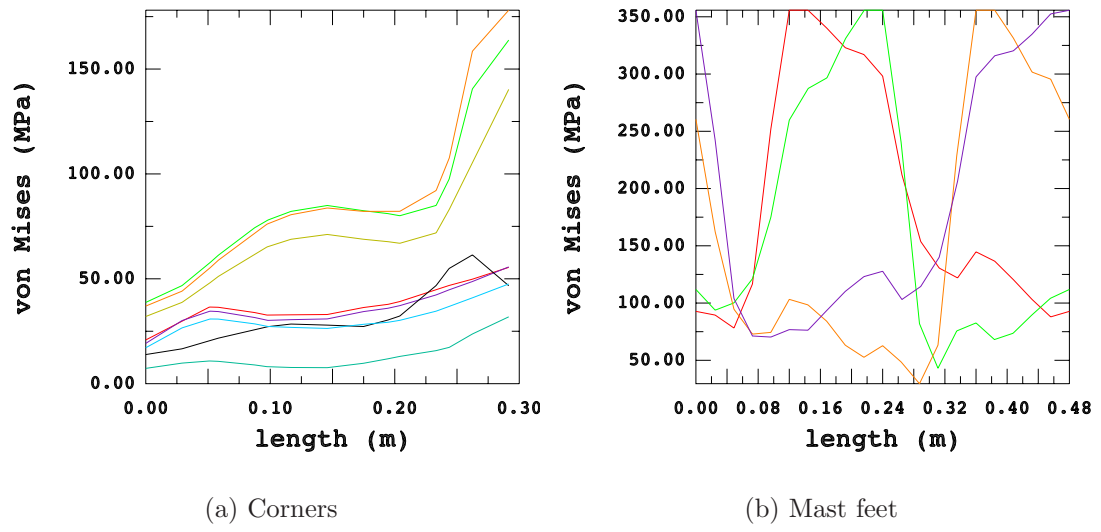


Figure 5.11: The path plots for 6.2.4.

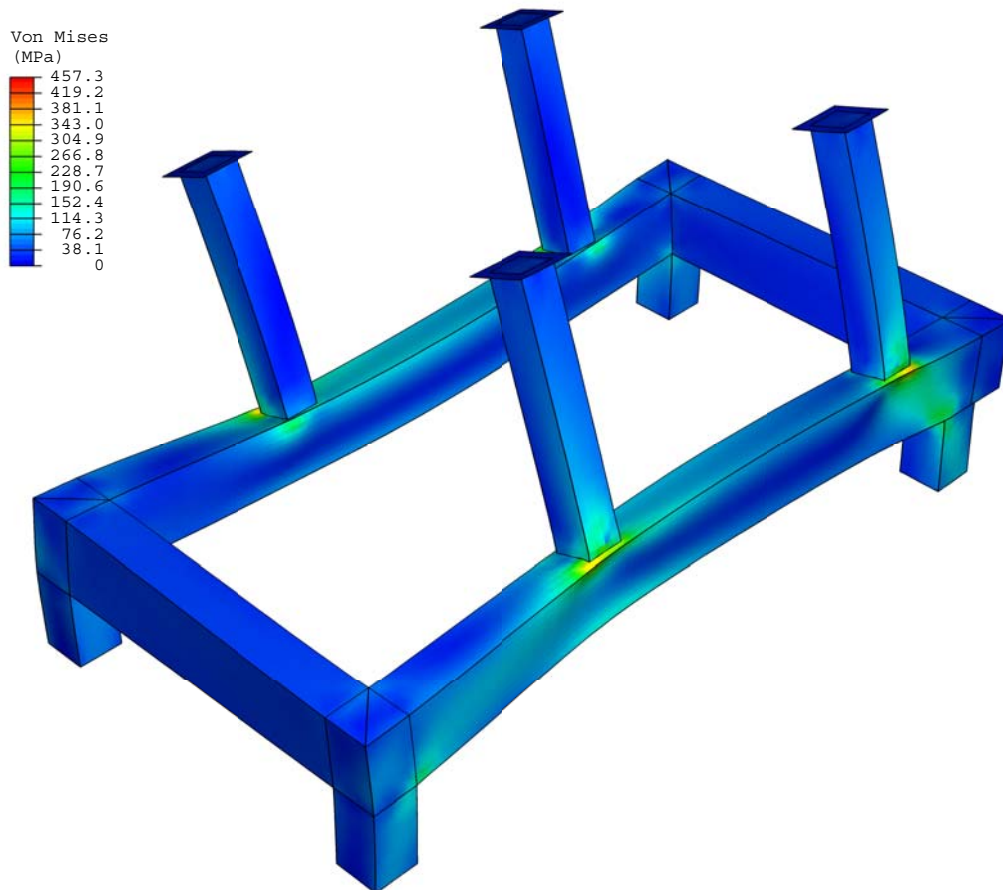


Figure 5.12: Von Mises stress for 6.2.4. Displacement scalefactor 50.

Third geometry

The geometry without the end stiffening plates.

Table 5.6: Displacements U of the 6.3 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
6.3.1	MF 1	0	1	1.735	-0.000006	1.000920	1.731720	0.00340390
	MF 2	0	1	0.485	-0.000008	0.999515	0.481720	0.00331552
	MF 3	1.25	1	0.485	1.249990	0.999603	0.481722	0.00330149
	MF 4	1.25	1	1.735	1.249990	1.000930	1.731730	0.00340299
6.3.2	MF 1	0	1	1.735	0.000005	0.998999	1.738290	0.00343707
	MF 2	0	1	0.485	0.000005	1.000430	0.488292	0.00331947
	MF 3	1.25	1	0.485	1.250010	1.000350	0.488287	0.00330647
	MF 4	1.25	1	1.735	1.250010	0.999001	1.738280	0.00343238
6.3.3	MF 1	0	1	1.735	0.005177	1.001240	1.734900	0.00532465
	MF 2	0	1	0.485	0.003887	1.000790	0.484880	0.00396903
	MF 3	1.25	1	0.485	1.253880	0.999409	0.485081	0.00392524
	MF 4	1.25	1	1.735	1.255180	0.998839	1.735060	0.00530529
6.3.4	MF 1	0	1	1.735	-0.005180	0.998677	1.735110	0.00534718
	MF 2	0	1	0.485	-0.003889	0.999152	0.485131	0.00398273
	MF 3	1.25	1	0.485	1.246120	1.000550	0.484926	0.00392077
	MF 4	1.25	1	1.735	1.244820	1.001090	1.734940	0.00529270

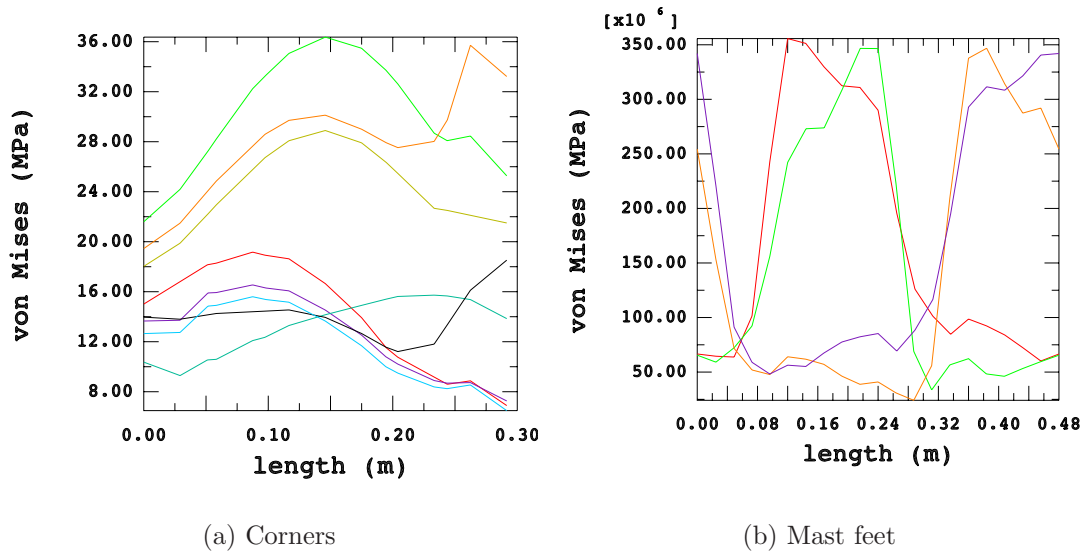


Figure 5.13: The path plots for 6.3.4.

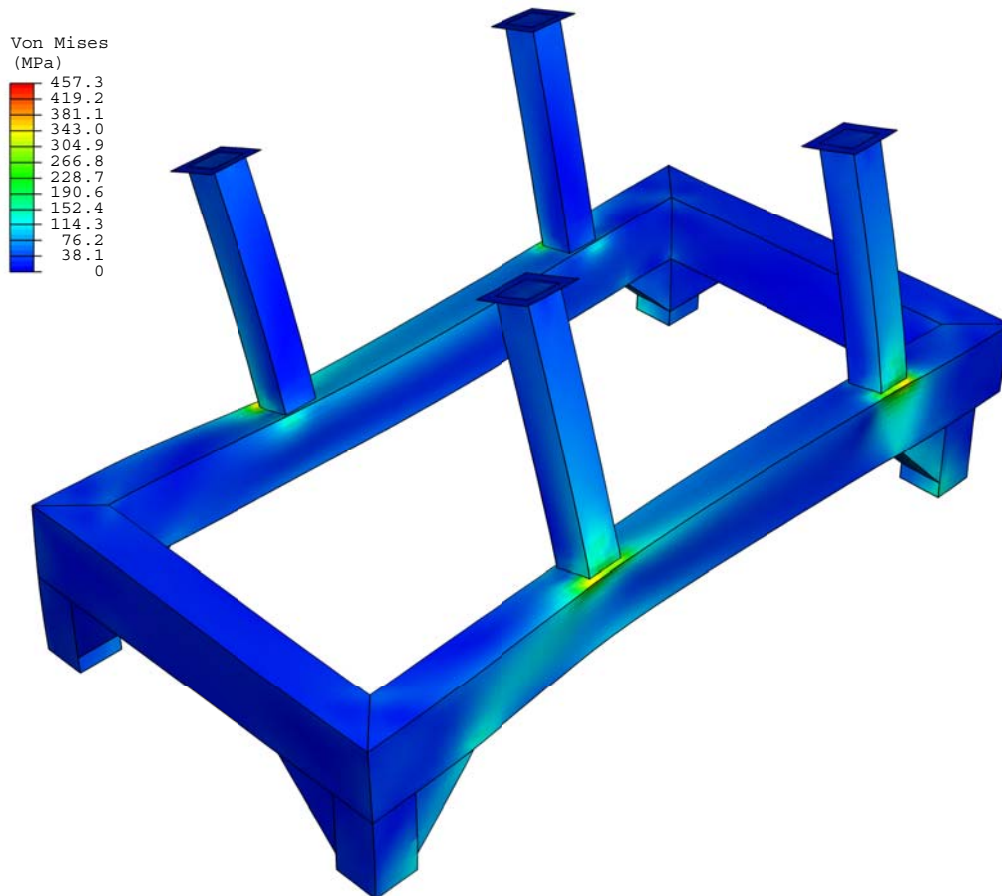


Figure 5.14: Von Mises stress 6.3.4. Displacement scalefactor 50.

Fourth geometry

The geometry with neither the supports nor the end stiffening plates.

Table 5.7: Displacements U of the 6.4 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
6.4.1	MF 1	0	1	1.735	0.000005	1.001240	1.731530	0.00368322
	MF 2	0	1	0.485	0.000021	0.999557	0.481525	0.00350286
	MF 3	1.25	1	0.485	1.250020	0.999688	0.481521	0.00349356
	MF 4	1.25	1	1.735	1.250000	1.001300	1.731530	0.00370896
6.4.2	MF 1	0	1	1.735	-0.000001	0.998614	1.738500	0.00376038
	MF 2	0	1	0.485	-0.000003	1.000340	0.488503	0.00351880
	MF 3	1.25	1	0.485	1.249970	1.000240	0.488510	0.00351842
	MF 4	1.25	1	1.735	1.249990	0.998568	1.738500	0.00378354
6.4.3	MF 1	0	1	1.735	0.006378	1.002410	1.734700	0.00682542
	MF 2	0	1	0.485	0.005290	1.001630	0.484658	0.00554483
	MF 3	1.25	1	0.485	1.255280	0.998864	0.485367	0.00541180
	MF 4	1.25	1	1.735	1.256380	0.997869	1.735330	0.00673145
6.4.4	MF 1	0	1	1.735	-0.006390	0.997433	1.735320	0.00689426
	MF 2	0	1	0.485	-0.005305	0.998263	0.485367	0.00559439
	MF 3	1.25	1	0.485	1.244710	1.001060	0.484660	0.00540958
	MF 4	1.25	1	1.735	1.243610	1.002000	1.734690	0.00670212

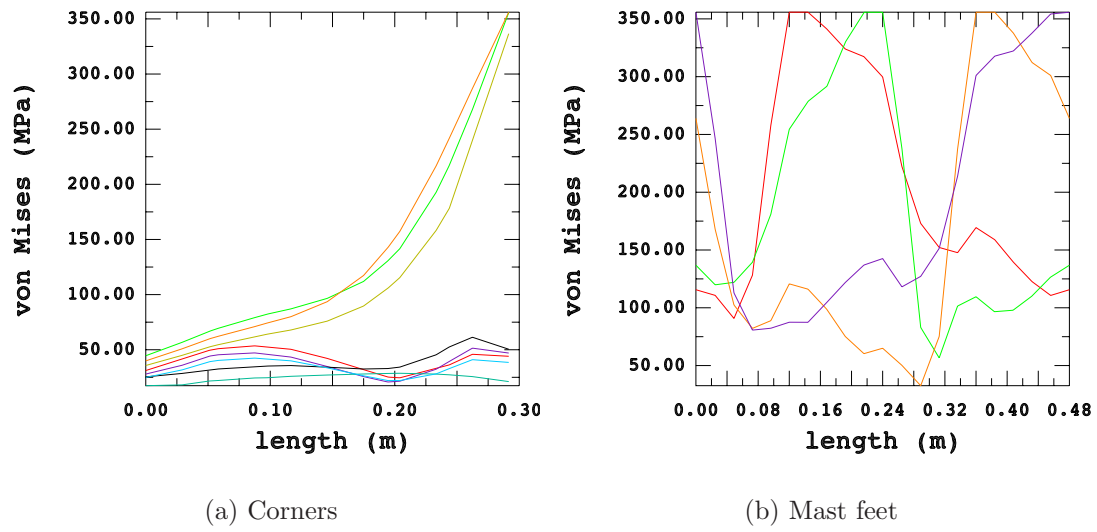


Figure 5.15: The path plots for 6.4.4.

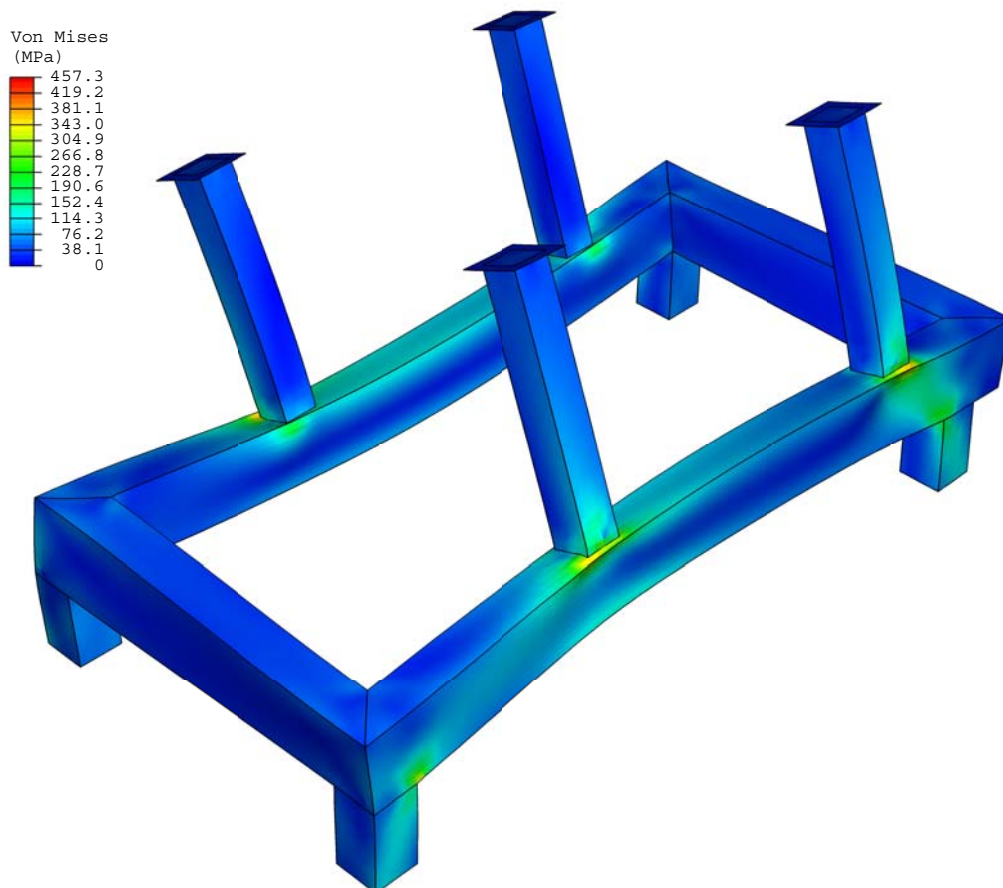


Figure 5.16: Von Mises stress 6.4.4. Displacement scalefactor 50.

5.5.2 Model with finer mesh

The 7 series are exactly like the 6 series but with the finer mesh.

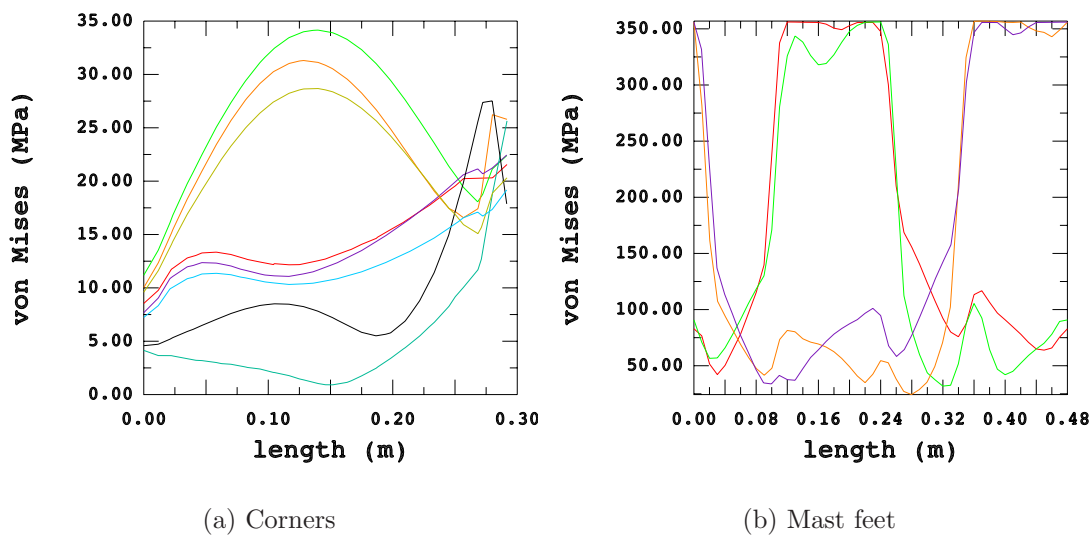


Figure 5.17: The path plots for 7.1.4.

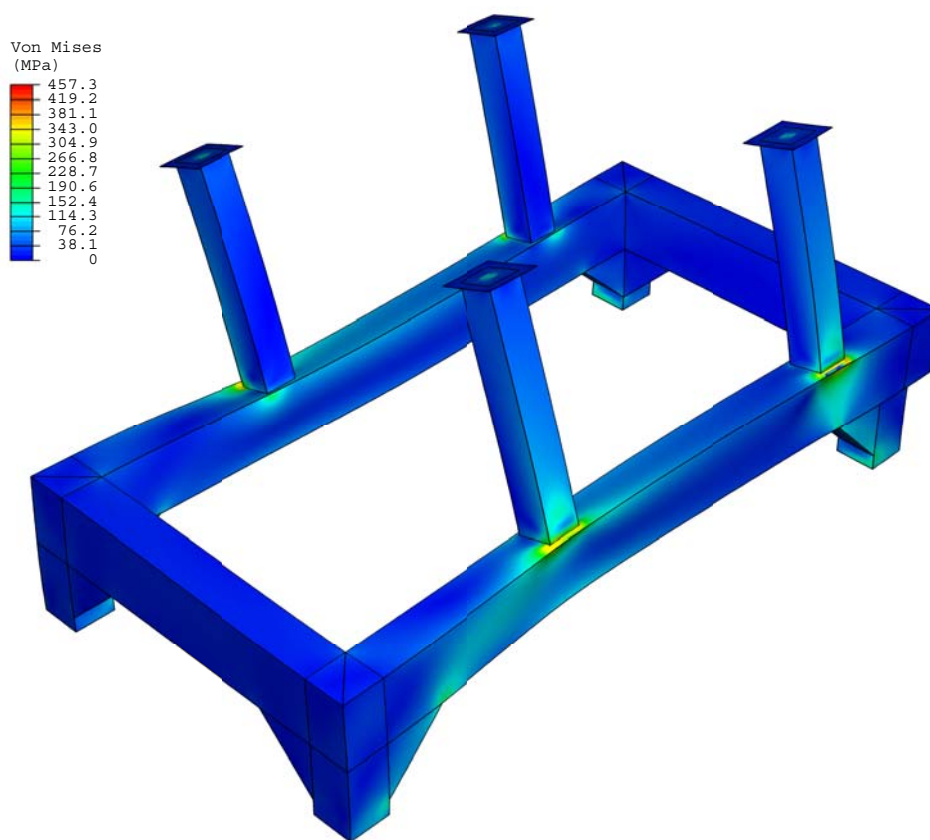


Figure 5.18: Von Mises stress for 7.1.4. Displacement scalefactor 50.

Table 5.8: Displacements U for the 7.1 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
7.1.1	MF 1	0	1	1.735	0.000001	1.000940	1.731280	0.00384107
	MF 2	0	1	0.485	0.000071	0.999481	0.481273	0.00376338
	MF 3	1.25	1	0.485	1.250070	0.999571	0.481288	0.00373750
	MF 4	1.25	1	1.735	1.250000	1.000950	1.731290	0.00382925
7.1.2	MF 1	0	1	1.735	-0.000007	1.000460	0.488739	0.00376799
	MF 2	0	1	0.485	1.249930	1.000980	0.488725	0.00374485
	MF 3	1.25	1	0.485	1.250000	0.998974	1.738720	0.00386117
	MF 4	1.25	1	1.735	-0.000002	0.998977	1.738740	0.00387452
7.1.3	MF 1	0	1	1.735	0.005252	1.001230	1.734930	0.00539482
	MF 2	0	1	0.485	0.004028	1.000790	0.484905	0.00410598
	MF 3	1.25	1	0.485	1.254020	0.999367	0.485037	0.00407370
	MF 4	1.25	1	1.735	1.255250	0.998831	1.735020	0.00538005
7.1.4	MF 1	0	1	1.735	-0.005252	0.998689	1.735080	0.00541397
	MF 2	0	1	0.485	-0.004012	0.999150	0.485103	0.00410256
	MF 3	1.25	1	0.485	1.245990	1.000590	0.484971	0.00405062
	MF 4	1.25	1	1.735	1.244750	1.001090	1.734990	0.00536421

Table 5.9: Displacements U for the 7.2 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
7.2.1	MF 1	0	1	1.735	0.000001	1.001170	1.731130	0.00404544
	MF 2	0	1	0.485	0.000065	0.999514	0.481124	0.00390721
	MF 3	1.25	1	0.485	1.250060	0.999628	0.481120	0.00389786
	MF 4	1.25	1	1.735	1.250000	1.001230	1.731120	0.00406543
7.2.2	MF 1	0	1	1.735	-0.000004	0.998694	1.738900	0.00411364
	MF 2	0	1	0.485	-0.000062	1.000390	0.488905	0.00392452
	MF 3	1.25	1	0.485	1.249940	1.000300	0.488913	0.00392464
	MF 4	1.25	1	1.735	1.249990	0.998644	1.738910	0.00413637
7.2.3	MF 1	0	1	1.735	0.006054	1.002050	1.734790	0.00639519
	MF 2	0	1	0.485	0.005088	1.001390	0.484756	0.00528086
	MF 3	1.25	1	0.485	1.255090	0.998878	0.485233	0.00521378
	MF 4	1.25	1	1.735	1.256050	0.998050	1.735200	0.00636271
7.2.4	MF 1	0	1	1.735	-0.006055	0.997815	1.735230	0.00644136
	MF 2	0	1	0.485	-0.005072	0.998513	0.485263	0.00529230
	MF 3	1.25	1	0.485	1.244930	1.001040	0.484786	0.00518012
	MF 4	1.25	1	1.735	1.243950	1.001830	1.734810	0.00632641

Table 5.10: Displacements U for the 7.3 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
7.3.1	MF 1	0	1	1.735	0.000001	1.000960	1.731270	0.00385012
	MF 2	0	1	0.485	0.000072	0.999484	0.481269	0.00376719
	MF 3	1.25	1	0.485	1.250070	0.999574	0.481283	0.00374161
	MF 4	1.25	1	1.735	1.250000	1.000970	1.731290	0.00383794
7.3.2	MF 1	0	1	1.735	-2.793290	0.998953	1.738740	0.00388617
	MF 2	0	1	0.485	-0.000069	1.000450	0.488745	0.00377327
	MF 3	1.25	1	0.485	1.249930	1.000380	0.488731	0.00375037
	MF 4	1.25	1	1.735	1.250000	0.998954	1.738730	0.00387195
7.3.3	MF 1	0	1	1.735	0.005363	1.001280	1.734910	0.00551435
	MF 2	0	1	0.485	0.004157	1.000820	0.484886	0.00423875
	MF 3	1.25	1	0.485	1.254150	0.999356	0.485049	0.00420282
	MF 4	1.25	1	1.735	1.255360	0.998799	1.735030	0.00549547
7.3.4	MF 1	0	1	1.735	-0.005364	0.998637	1.735100	0.00553546
	MF 2	0	1	0.485	-0.004142	0.999121	0.485124	0.00423616
	MF 3	1.25	1	0.485	1.245860	1.000560	0.484960	0.00418024
	MF 4	1.25	1	1.735	1.244640	1.001120	1.734980	0.00547981

Table 5.11: Displacements U for the 7.4 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
7.4.1	MF 1	0	1	1.735	0.000016	1.001270	1.731070	0.00412906
	MF 2	0	1	0.485	0.000105	0.999529	0.481068	0.00396144
	MF 3	1.25	1	0.485	1.250100	0.999659	0.481073	0.00394291
	MF 4	1.25	1	1.735	1.250020	1.001340	1.731080	0.00414360
7.4.2	MF 1	0	1	1.735	-0.000023	0.998567	1.738960	0.00421310
	MF 2	0	1	0.485	-0.000106	1.000360	0.488967	0.00398481
	MF 3	1.25	1	0.485	1.249900	1.000260	0.488968	0.00397746
	MF 4	1.25	1	1.735	1.249980	0.998521	1.738960	0.00422843
7.4.3	MF 1	0	1	1.735	0.006654	1.002480	1.734690	0.00710719
	MF 2	0	1	0.485	0.005763	1.001680	0.484652	0.00601340
	MF 3	1.25	1	0.485	1.255760	0.998769	0.485299	0.00589347
	MF 4	1.25	1	1.735	1.256650	0.997801	1.735270	0.00701187
7.4.4	MF 1	0	1	1.735	-0.006665	0.997360	1.735330	0.00717692
	MF 2	0	1	0.485	-0.005756	0.998203	0.485375	0.00604138
	MF 3	1.25	1	0.485	1.244250	1.001150	0.484726	0.00586707
	MF 4	1.25	1	1.735	1.243340	1.002060	1.734760	0.00697894

5.5.3 Optimised model

Table 5.12: Displacements U for the 7.8 series.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
7.8.1	MF 1	0	1	1.735	-0.000002	1.000820	1.734110	0.00121088
	MF 2	0	1	0.485	-0.000010	0.999603	0.484104	0.00098022
	MF 3	1.25	1	0.485	1.249990	0.999693	0.484105	0.00094633
	MF 4	1.25	1	1.735	1.250000	1.000830	1.734110	0.00121256
7.8.2	MF 1	0	1	1.735	0.000001	0.999102	1.735900	0.00126824
	MF 2	0	1	0.485	0.000008	1.000350	0.485907	0.00097103
	MF 3	1.25	1	0.485	1.250010	1.000270	0.485904	0.00094294
	MF 4	1.25	1	1.735	1.250000	0.999103	1.735890	0.00126457
7.8.3	MF 1	0	1	1.735	0.003753	1.001100	1.734900	0.00391272
	MF 2	0	1	0.485	0.002461	1.000660	0.484828	0.00255371
	MF 3	1.25	1	0.485	1.252450	0.999520	0.485130	0.00249824
	MF 4	1.25	1	1.735	1.253750	0.998956	1.735070	0.00389639
7.8.4	MF 1	0	1	1.735	-0.003755	0.998823	1.735110	0.00393628
	MF 2	0	1	0.485	-0.002463	0.999289	0.485183	0.00257000
	MF 3	1.25	1	0.485	1.247550	1.000440	0.484879	0.00249255
	MF 4	1.25	1	1.735	1.246250	1.000980	1.734930	0.00387985

5.5.4 Double design load

The first model with the finer mesh and the optimised model were subjected to the double design load. The fourth load case is the used one since it produced the largest stresses and displacements for all geometries.

Table 5.13: Displacements U caused of the double design load.

Model	Part	Orig. Coord.			Def. Coord.			Magnitude
7.5.4	MF 1	0	1	1.735	-0.018533	0.996515	1.735220	0.01885870
	MF 2	0	1	0.485	-0.018658	0.997272	0.485243	0.01885800
	MF 3	1.25	1	0.485	1.231330	1.002330	0.484813	0.01881480
	MF 4	1.25	1	1.735	1.231470	1.003000	1.734830	0.01877010
7.9.4	MF 1	0	1	1.735	-0.007660	0.997607	1.735240	0.00802919
	MF 2	0	1	0.485	-0.005046	0.998551	0.485388	0.00526431
	MF 3	1.25	1	0.485	1.244980	1.000900	0.484752	0.00510425
	MF 4	1.25	1	1.735	1.242340	1.001970	1.734860	0.00790971

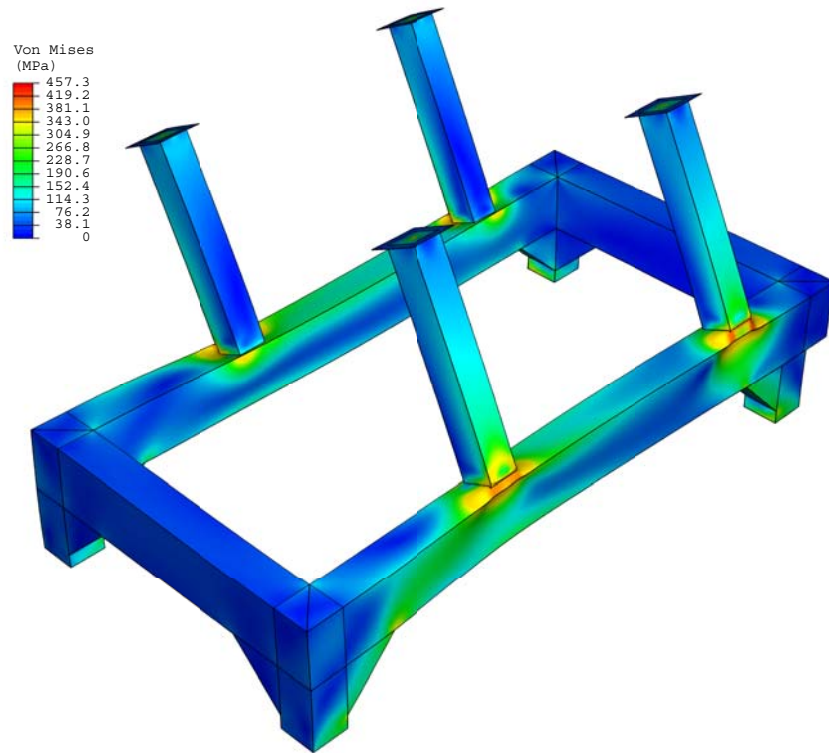


Figure 5.19: Von Mises stress for 7.5.4. Displacement scalefactor 20.

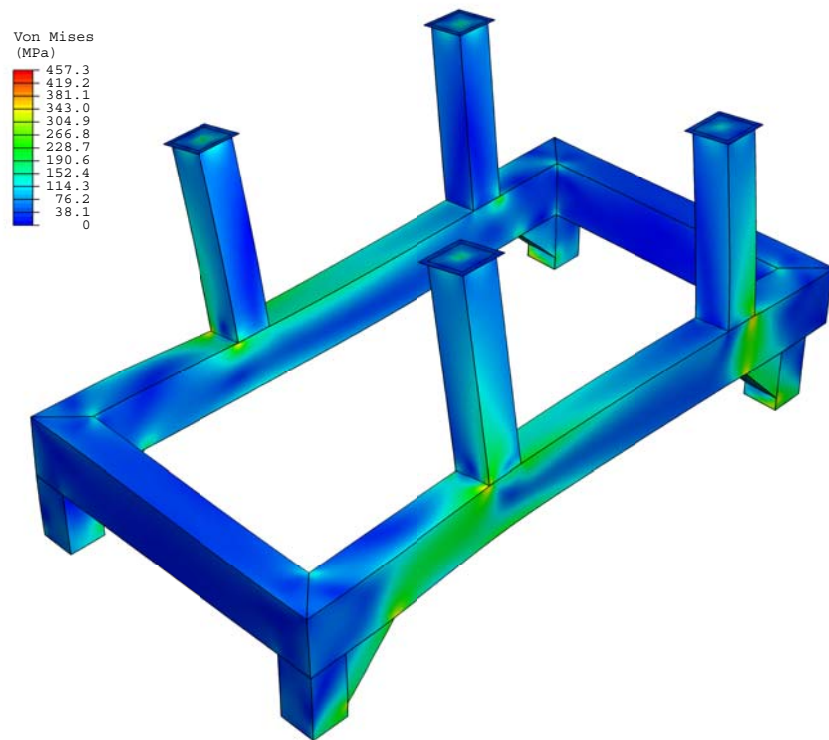


Figure 5.20: Von Mises stress for 7.9.4. Displacement scalefactor 20.

5.5.5 Comparison between the models

It is clear that the fourth load case is the governing one. Mast foot 1, in load case four, has the largest displacement in every geometry. In Tables 5.14 and 5.15 these results are plotted and a comparison is made between the results.

Table 5.14: Displacement comparison of the 6 series.

Model	Displacement	Difference	%
6.1.4	0.00523592	—	—
6.2.4	0.00625948	0.00102356	19.5
6.3.4	0.00534718	0.00011126	2.1
6.4.4	0.00689426	0.00165834	31.7

Table 5.15: Displacement comparison of the 7 series.

Model	Displacement	Difference	%
7.1.4	0.00541397	—	—
7.2.4	0.00644136	0.00102739	19.0
7.3.4	0.00553546	0.00012149	2.2
7.4.4	0.00717692	0.00176295	32.6
7.8.4	0.00393628	−0.00147769	−27.3
7.5.4	0.01885870	0.01344473	248.3
7.9.4	0.00802919	0.00261522	48.3

The differences between the geometries are rather small. The percentage given is how much more the modified geometries move in comparison to model 6.1.4 for the 6 series and 7.1.4 for the 7 series.

The difference between the coarse and the fine mesh is also small. The 7 series shows about 2–3% more displacements than the 6 series.

As can be seen in Fig. 5.21 the plasticity occurs in corners which always is a fatal area for finite element analyses. Note that the geometries with the finer mesh shows more plasticity than the other geometries. Probably there is a singularity point in these areas. The fact that the finer mesh gets more plasticity supports that suggestion. Plasticity occur in every model, but in every case it is very local, which suggests that it is a singularity point.

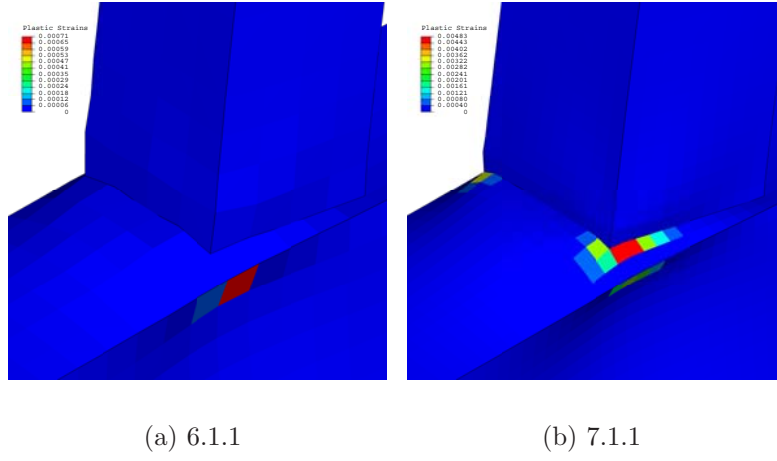


Figure 5.21: Examples of plasticity for 6.1.1 and 7.1.1.

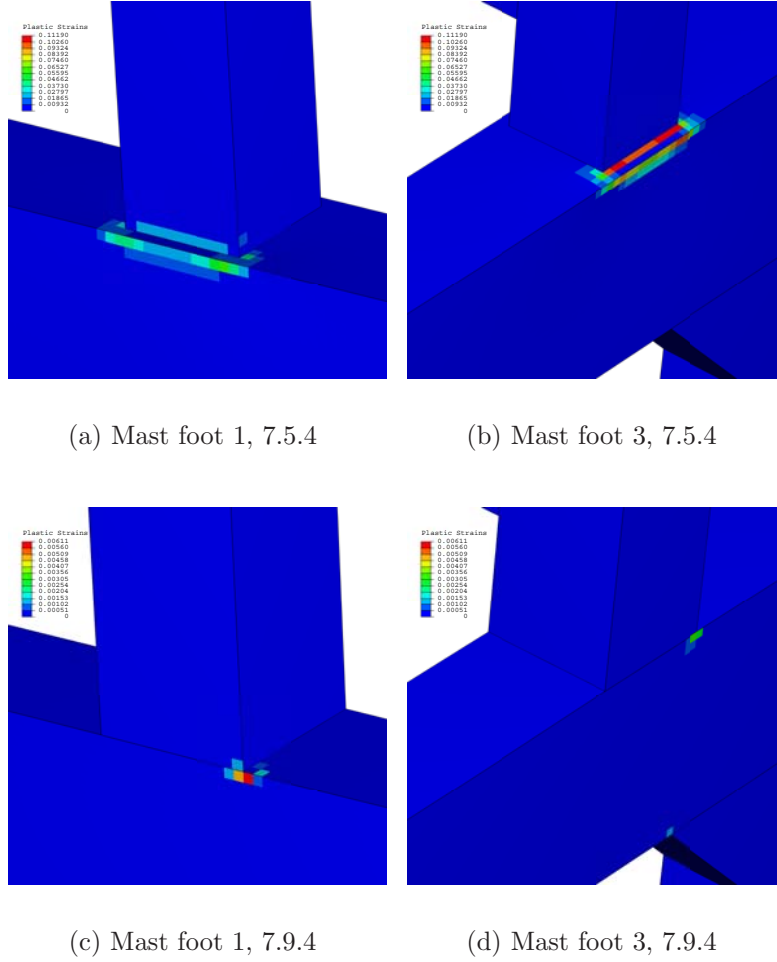


Figure 5.22: Examples of local plasticity due to the double design load.

The results in 7.5.4 shows a plastic strain larger than $\varepsilon_{\max}$. ABAQUS do use the stress–strain diagrams given in Sec. 5.2.3 but, after reaching the value of the $\varepsilon_{\max}$ ABAQUS assume that the material is perfectly plastic. Therefore we get strains that are not possible. Where the plastic strain is larger than $\varepsilon_{\max}$ the material should have ruptured. However it is not certain that these strains exist. These strains are located in the same areas as the plastic strains in the earlier models, where the supposed singularity points are.

The difference between 7.5.4 and 7.9.4 is large, around 200%. 7.9.4 also have very local plastic strains, suggesting that there are singularity points in this model as well.

5.6 Conclusions

Upon reading the results, it is clear that some parts are more important than others. The end stiffening plates does not help much for the structure. The supports are of more importance.

Although plastic strains occurred, they should not be trusted. Since it is a few elements that plasticise in each case, there probably are singularity points.

T-joints that use full width bracings are preferable. The difference in the moment capacity is significant.

Chapter 6

T-Joint

6.1 Introduction

After all the analyses made on the foundation an interest grew for the critical joints in the structure. The critical joints in the foundation are where the mast feet connects to the long horizontal beams. These joints are classical T-joints.

This model however, are modelled using solid elements to analyse the effects of the welds in the joint. The welds could not be relevantly modelled in the former shell element models.

There are totally four different geometries. The geometries are yet again very similar to each other. The differences lies in how the the welds are modelled. The dimensions for the last geometry are taken from the optimised model from the foundation analysis.

6.2 Data for the model

This model use the same material data as the foundation models. Since it is the same material – density, elasticity, and plasticity are the same. One major difference from the previous models is that the corners of the beams are rounded. Thus they reflect the reality better. Another difference is that the mast foot plate is replaced by a *rigid body*. A rigid body request a reference node. The properties of the plate is applied to this node

$$\begin{aligned} \text{Mass} &= 5.632 \text{ kg} \\ I_{xx} &= 1.08610^{-4} \text{ m}^4 \\ I_{yy} &= 1.08610^{-4} \text{ m}^4 \\ I_{zz} &= 1.26710^{-7} \text{ m}^4 \end{aligned} \tag{6.1}$$

Since a rigid body cannot be deformed in any way it is only necessary to assign it

the S355J2H material data.

6.3 Model

The aim is to model a T-joint with the same dimensions as one in the foundation model. The long horizontal beam is divided into three parts. One part with solid elements in the middle and two parts with beam elements. To connect the beam elements with the solid elements two rigid bodies are used. These rigid bodies have no special data. They only distribute the forces from the solid part to the beam ends.

The mast foot is, like the chord, entirely modelled with solid elements. Sometimes a diagonal cut is made where the mast foot connects to the chord, depending on the weld. The different welds are shown in Fig. 6.2.

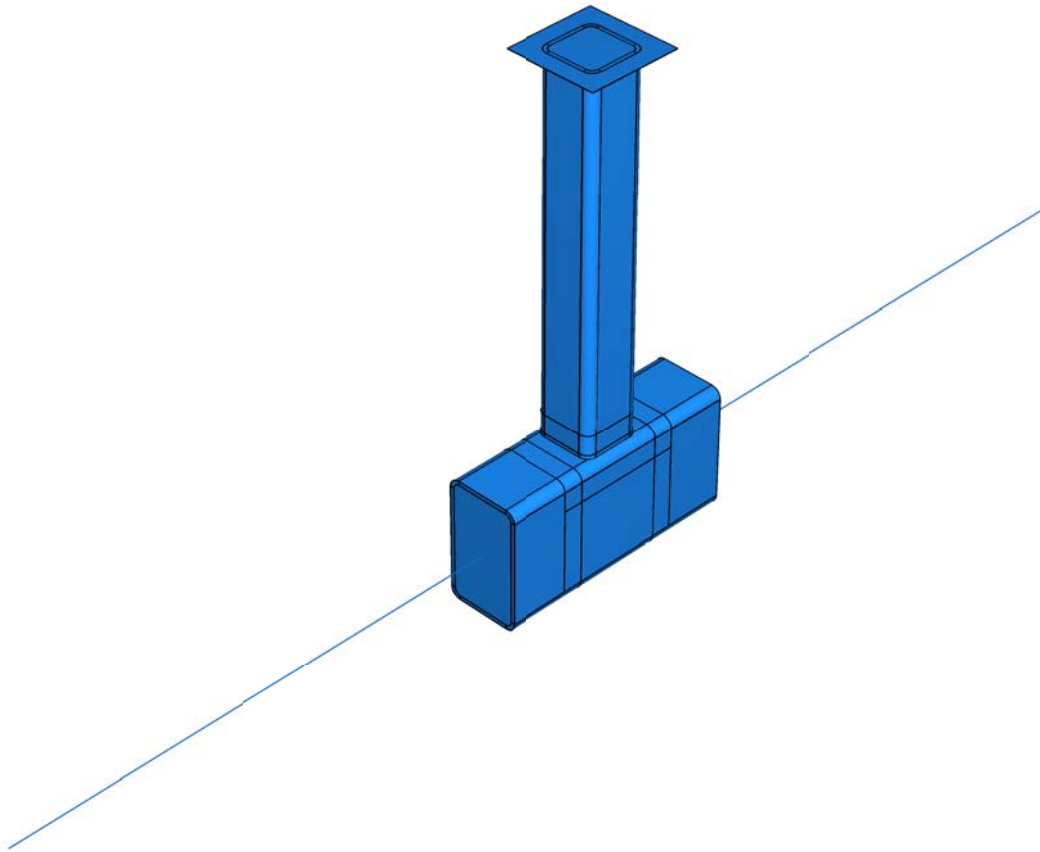
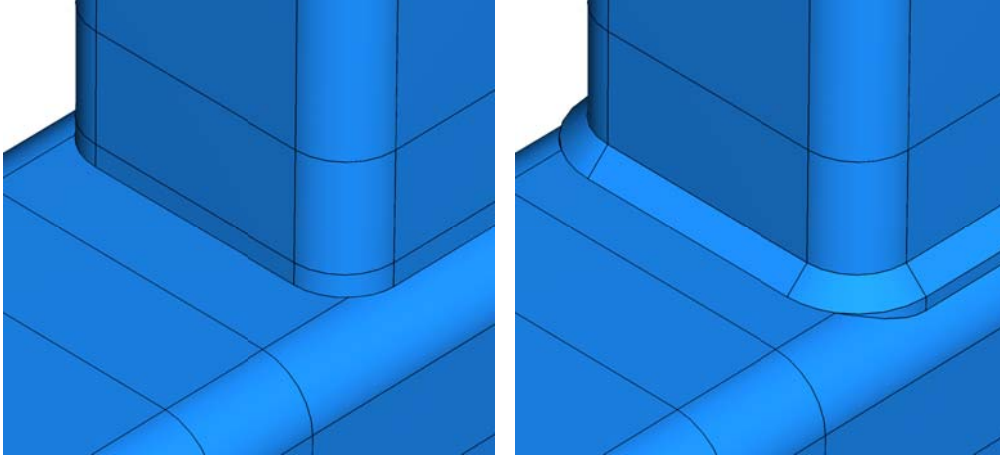


Figure 6.1: The joint model.

The model use a rather fine mesh. The thickness of the RHS-beams is eight millimetres. It is desired to have at least four element over the thickness. Otherwise the model would be too stiff. This means small solid elements of two millimetres in

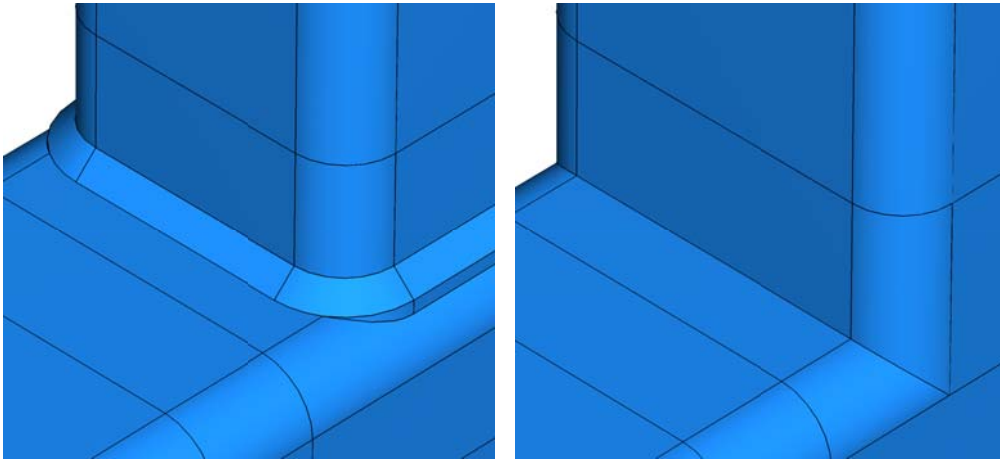
one direction. The other two directions are eight millimetres each. Thus we have an eight-node brick of the dimension $2 \times 8 \times 8$ mm.

The welds however, have complicated geometries. Therefore, it is inevitable to use tetrahedral elements. The areas on the chord connecting to the welds also use tetrahedral elements.



(a) Weld 1

(b) Weld 2



(c) Weld 3

(d) Weld 4

Figure 6.2: The different welds.

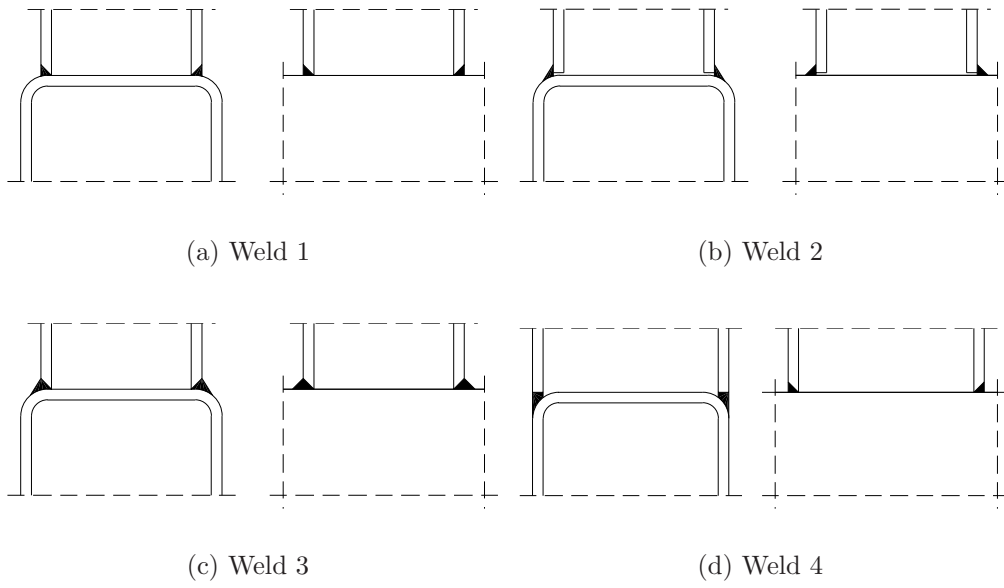


Figure 6.3: Cross-sections of the welds.

6.4 Loads

There are four different load cases in these analyses. Two of the load cases are taken directly from the dimensional loads given in [12]. The other two load cases consist of only a moment applied to the reference point of the mast foot plate. [11].

6.4.1 Load capacity

Rautaruukki is a company that constructs SHS. They provide a manual for designing SHS structures. The following equations are taken from that manual. Note that these equations are like the equations given in chapter 2. The difference is that Rautaruukki uses an extra security factor [11].

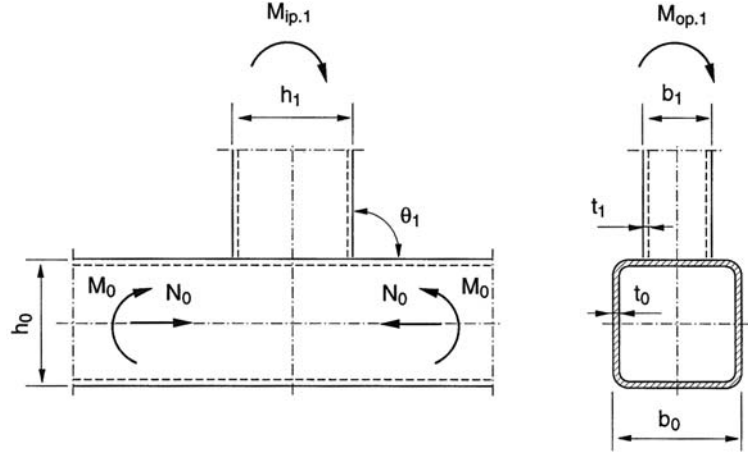


Figure 6.4: T-joint with applied loads. Reproduced from [11].

$$M_{ip,1} = f_{y0} t_0^2 h_1 \left(\frac{(1-\beta)b_0}{2h_1} + \frac{2}{\sqrt{1-\beta}} + \frac{h_1}{b_0(1-\beta)} \right) k_n \frac{1.1}{\gamma_{M_j} \gamma_{M_0}} \quad (6.2)$$

$$M_{op,1} = f_{y0} t_0^2 \left(\frac{h_1(1+\beta)}{2(1-\beta)} + \sqrt{\frac{2b_0b_1(1+\beta)}{1-\beta}} \right) k_n \frac{1.1}{\gamma_{M_j} \gamma_{M_0}} \quad (6.3)$$

Validity

Bracings in general:

$$\frac{b_i}{b_0} = \frac{120}{150} = 0.8 \geq 0.25$$

$$0.5 \leq \frac{h_1}{b_1} = \frac{120}{120} = 1 < 2$$

$$\frac{b_1 + h_1}{t_1} = \frac{120 + 120}{8} = 30 \geq 25$$

Bracing under pressure:

$$\frac{b_1}{t_1} = \frac{h_1}{b_1} = \frac{120}{8} = 15 \leq 1.25 \sqrt{\frac{E}{f_{yi}}} = 30.4$$

Chords:

$$\frac{b_0 + h_0}{t_0} = \frac{150 + 250}{8} = 50 \geq 25$$

$$0.5 \leq \frac{h_0}{b_0} = \frac{250}{150} = 1.667 < 2$$

$$\frac{b_0}{t_0} = \frac{150}{8} = 18.75 \leq 35$$

$$\frac{h_0}{t_0} = \frac{150}{8} = 31.25 \leq 35$$

Parameters:

$$\beta = \frac{b_1}{b_0} = \frac{120}{150} = 0.8 \leq 0.85$$

$$k_n = 1$$

Thus we get from Eqs. (6.2) and (6.3)

$$\begin{aligned} M_{ip.1,Rd} &= \\ &= 35510^6 \cdot 0.008^2 \cdot 0.12 \left(\frac{(1-0.8)0.15}{20.12} + \frac{2}{\sqrt{1-0.8}} + \frac{0.12}{0.15(1-0.8)} \right) \frac{1.1}{1.1 \cdot 1.1} = \\ &= 21308.39 \text{Nm} \end{aligned}$$

$$\begin{aligned} M_{op.1,Rd} &= \\ &= 35510^6 \cdot 0.008^2 \left(\frac{0.12(1+0.8)}{2(1-0.8)} + \sqrt{\frac{2 \cdot 0.150.12(1+0.8)}{1-0.8}} \right) \frac{1.1}{1.1 \cdot 1.1} = \\ &= 22910.23 \text{Nm} \end{aligned}$$

$$\therefore \quad M_{ip.1,Rd} = 21.3 \text{ kNm} \quad \text{and} \quad M_{op.1,Rd} = 22.9 \text{ kNm} \quad (6.4)$$

Note that these capacity values are calculated for the T-joint with the bracing having smaller dimensions than the chord. To calculate the capacity for the fourth model other formulas must be used. Then β is 1.0 and not 0.8. Though in the analyses of the fourth model the values in Eq. (6.4) are used. This is necessary because a comparison of the results of the different models is to be made.

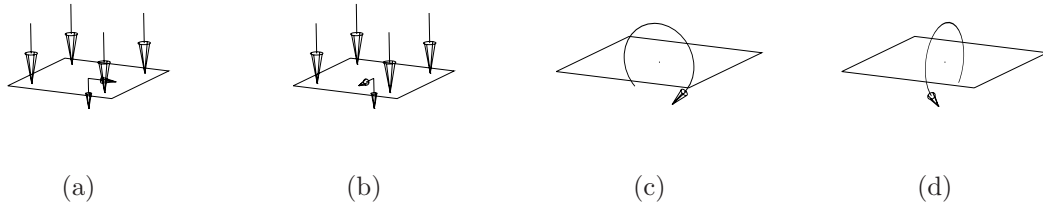


Figure 6.5: The different load cases.

6.5 Results

Table 6.1: Displacements U of the mast foot top, 1.1b series.

Model	Orig. Coord.			Def. Coord.			Magnitude
J1.1b.1	0	0.875	0.25	0.003906	0.872447	0.249986	0.00466643
J1.1b.2	0	0.875	0.25	-0.000001	0.872417	0.254574	0.00525245
J1.1b.3	0	0.875	0.25	0.010487	0.875000	0.249996	0.01048720
J1.1b.4	0	0.875	0.25	0.000002	0.875005	0.265776	0.01577640

Table 6.2: Displacements U of the mast foot top, 1.2b series.

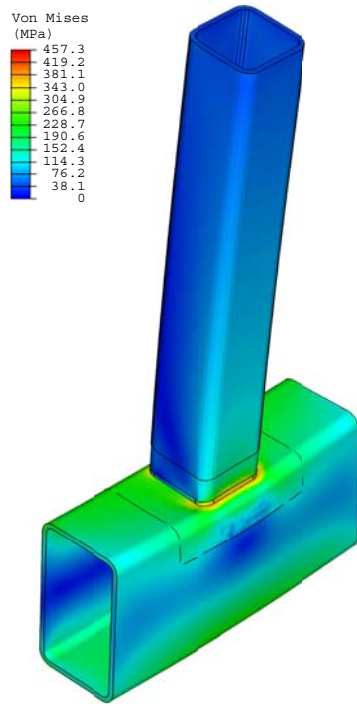
Model	Orig. Coord.			Def. Coord.			Magnitude
J1.2b.1	0	0.875	0.25	0.003419	0.872413	0.250000	0.00422805
J1.2b.2	0	0.875	0.25	0.000001	0.872497	0.253602	0.00438587
J1.2b.3	0	0.875	0.25	0.008767	0.874999	0.250000	0.00876736
J1.2b.4	0	0.875	0.25	0.000003	0.875000	0.261116	0.01111580

Table 6.3: Displacements U of the mast foot top, 1.3b series.

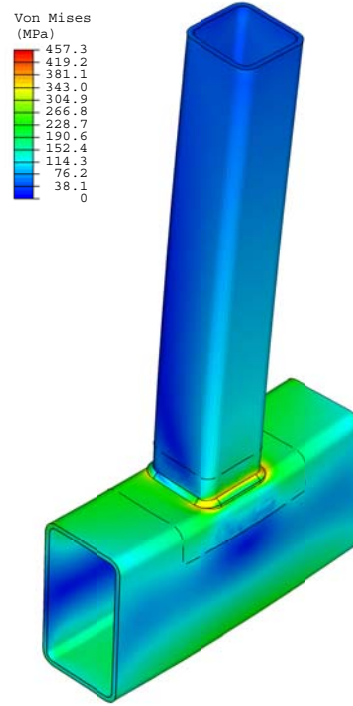
Model	Orig. Coord.			Def. Coord.			Magnitude
J1.3b.1	0	0.875	0.25	0.003326	0.872527	0.249997	0.00414410
J1.3b.2	0	0.875	0.25	-0.000001	0.872516	0.253386	0.00419921
J1.3b.3	0	0.875	0.25	0.008425	0.875001	0.249996	0.00842479
J1.3b.4	0	0.875	0.25	-0.000001	0.875001	0.259615	0.00961538

Table 6.4: Displacements U of the mast foot top, 1.4b series.

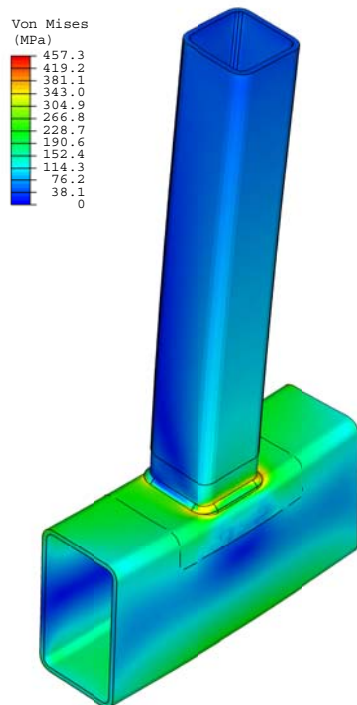
Model	Orig. Coord.			Def. Coord.			Magnitude
J1.4b.1	0	0.875	0.25	0.002005	0.872630	0.250001	0.00310390
J1.4b.2	0	0.875	0.25	0.000000	0.872628	0.251438	0.00277343
J1.4b.3	0	0.875	0.25	0.004340	0.875000	0.250001	0.00433983
J1.4b.4	0	0.875	0.25	0.000001	0.875000	0.253790	0.00378987



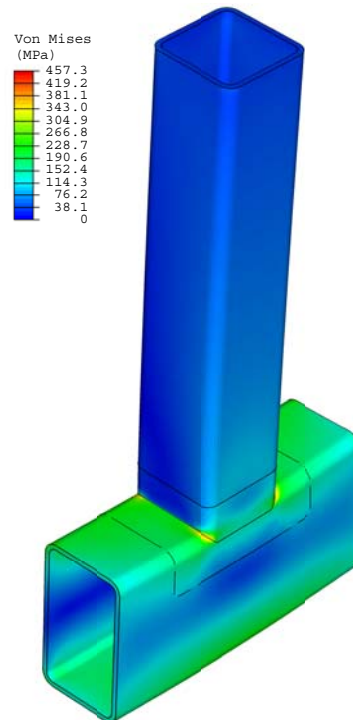
(a) J1.1b.1



(b) J1.2b.1



(c) J1.3b.1



(d) J1.4b.1

Figure 6.6: Von Mises stresses.

6.6 Analysing the results

The movements of the mast foot plate differ a bit from the top plates in the foundation analyses. In the foundation analyses, the T-joint was weaker in the direction perpendicular to the chords axis. The results in these analyses shows the opposite for the first three models.

In Table 6.5 are the results from the first load case listed. As mentioned before, this load case was not always the most serious one. These results however, are to be compared to the results of the foundation analyses.

Table 6.5: Comparison of the first load case.

Displacements U			
Model	Displacement	Difference	%
J1.1b.1	0.00466643	—	—
J1.2b.1	0.00422805	−0.00043838	−9.4
J1.3b.1	0.00414410	−0.00052233	−11.2
J1.4b.1	0.00310390	−0.00156253	−33.5

Table 6.6: Comparison between the foundation model and the T-joint model.

Displacement comparisons (J1.x/7.x)(%)					
	7.1.4	7.2.4	7.3.4	7.4.4	7.8.4
J1.1b.1	−13.8	−27.6	−15.7	−35.0	18.5
J1.2b.1	−21.9	−34.4	−23.6	−41.1	7.4
J1.3b.1	−23.5	−35.7	−25.1	−42.3	5.3
J1.4b.1	−41.7	−51.8	−43.9	−56.8	−21.1

The T-joint analyses shows less movements than the foundation models. Even though there are beams between the mast foot plates in the foundation models preventing movements, the T-joint model is more stiff.

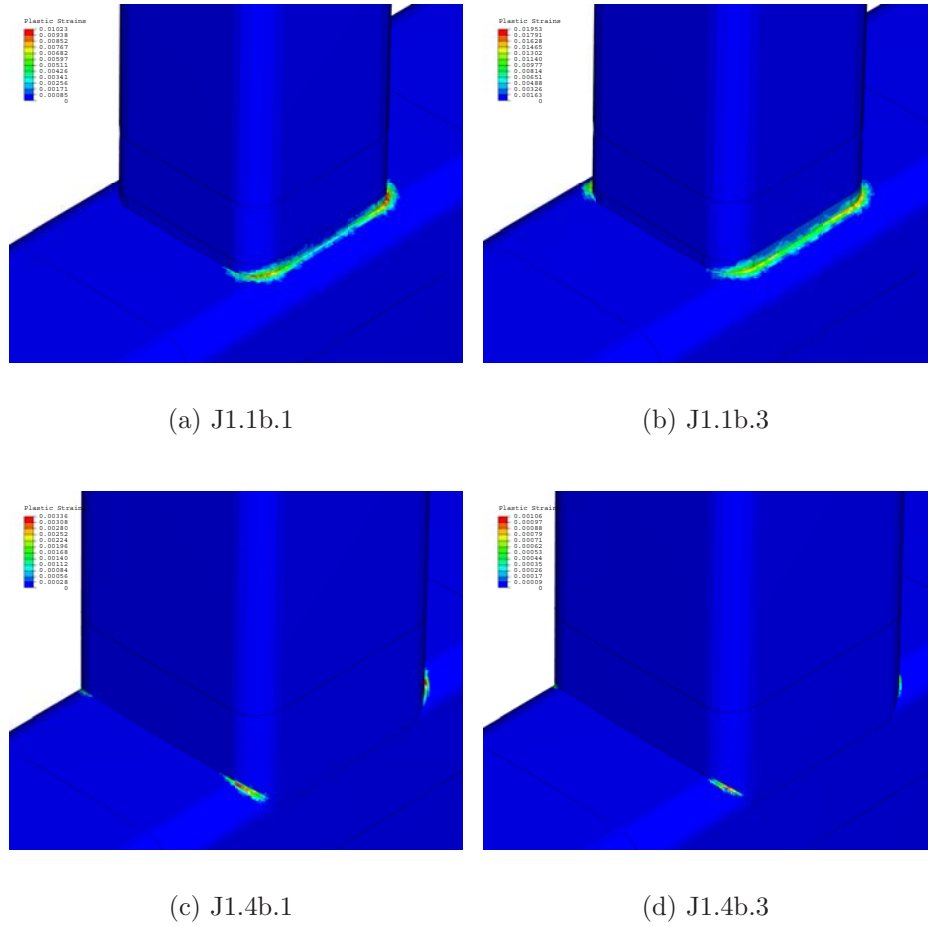


Figure 6.7: Plasticity for the T-joint

The T-joint also shows more plasticity than the foundation models. The plasticity is local, but not as local as in the former models. It can be seen in the T-joint model and in the foundation models that a T-joint with full width bracing is preferable. Especially when moments are applied.

6.7 Conclusions

The T-joint analyses coincide rather well with the foundation analyses. The T-joint model is a bit more stiff. Probably because of the different elements. The shell elements used in the foundation analyses have six degrees of freedom at each node – three translational and three rotational. The solid elements in the T-joint analyses have three translational degrees of freedom. Rotational degrees of freedom are important for bending analyses. Hence, the stiffness in the T-joint model.

Regarding the full width bracing, the T-joint analyses shows the same results as the foundation analyses. T-joints using the same dimensions for the bracing and the chord are preferable.

Chapter 7

Conclusions

7.1 Finite element modelling

The main aim of the finite element analyses were to see the static response of the foundation. A secondary aim was to isolate one of the T-joints and subject it to the same loads as the loads on each T-joint in the foundation. Two sets of finite element analyses were performed

- Foundation
- T-joint

7.1.1 Foundation

The first set of finite element analyses were on the mast foundation. The aim of these analyses was to see and understand how the foundation reacted to the applied loads. Especially how the reaction forces were distributed in the corners.

The finer mesh resulted in 2–3% larger movements of the mast foot plates. When the movement from the beginning is about a few millimetres, 2–3% is really small. This suggest that the last mesh is fine enough. Further analyses with finer mesh would not be possible anyway. The finer mesh already use elements of the size $0.01 \times 0.01\text{m}$. Going below these dimensions would mean that the thickness of the shell element would be bigger than the planar dimensions, causing mathematical issues.

The plasticity distribution gained in the analyses should not be trusted. It is very local in every case. With the coarser mesh it is just one or two elements that plasticises. With the finer mesh the same areas reach more plasticity. All these areas are in corners and other vulnerable locations. The fact that the finer mesh shows more plasticity in the same area suggests that there is a singularity point in this area. This would mean that if an even finer mesh would be used, even more plasticity would occur. However, a finer mesh is not possible as explained above.

The plasticity occurs in the T-joints and near the supports. The forces applied to each T-joint is less than half the load that T-joints of these dimensions is capable of [11]. This also supports the suggestion that there are singularity points that causes the high stresses and the plasticity in the analyses.

When seeing the results of the first model it was clear that some parts in the corners were more important than others. The end stiffening plates seemed unnecessary. The supports, on the other hand, seemed to have a function. But only for the long horizontal beams. The supports on the short horizontal beams did not help much. Therefore, an optimised model was produced.

The optimised model seemed to be much better than the original. Eventhough it lacks parts that did not seem important in the corners, this model shows movements much smaller than the original. The T-joints with the bracing and chord of same dimensions are preferable.

The foundation constructed will endure the reaction forces from the mast. Upon designing the foundation, the ultimate design loads for the mast were used. I.e., the mast will break before the foundation.

For future foundations it would be better to build the optimised model as it is more economical and stronger.

7.1.2 T-joint

The second set of finite element analyses were on an isolated T-joint. As could be seen in the results of the former analyses, the vulnerable part of the foundation were the T-joints. Therefore these analyses were made.

The movements in these analyses are more or less like the movements in the former analyses. However there is a difference. The isolated T-joint seems to be weaker in the direction along the axis of the chord. The foundation model showed the opposite. This might be because the top of the bracing is not restrained in any way. In the foundation model, the mast foot plates were connected with beams. The last model (J1.4b) though, had results that reflected the results from the foundation analyses.

The movements also depended on what type of weld that was modelled. The first geometry showed more movement than the rest. This is probably because the other geometries have a larger base area against the chord due to the welds. The last geometry had the smallest movements.

When the moments were applied to the model, movements up to three times larger than the first two load cases could be observed, except for the last geometry. The T-joint that used the same dimensions for the bracing and for the chord showed much less movements. Only 35–40% bigger. These results also show that a T-joint using a chord with full width bracing is indeed favourable.

The plasticity in these models are not so local as in the foundation analyses. Especially when moments were applied. These moments are calculated to be on the

critical limit for the T-joint. Therefore, high stresses near plastic should be expected. A difference between the load cases is that load case one and two causes plastic strains on just one side of the joint. The moment causes strains on two sides, opposite from each other.

The models all have a fine mesh. A refinement of the mesh was not possible due to limited computer resources. However, since the results from these analyses coincide with the results from the foundation analyses, the mesh is presumably fine enough.

7.1.3 Assumptions

In both sets assumptions were made. The first set of analyses uses shell elements. When using shell elements you automatically get a geometric based error, see Fig. 7.1. Further the roundness around the corner was neglected. The shell elements also made it impossible to model the welds in different manners.

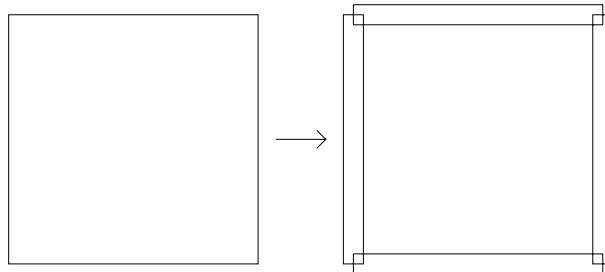


Figure 7.1: Geometric error due to shell elements.

The isolated T-joint had near perfect geometry. The T-joint however was placed in the middle of the model. In the foundation neither of the T-joints were in the middle. Further the beams together with the T-joint are somewhat shorter than the long side of the foundation.

7.2 Future research

All analyses made in this thesis are static. Although the foundation seems to have dimensions for loads much bigger than the design loads given, it might be necessary to have these dimensions. When designing structures made of metal, fatigue is an important factor. An interesting study would be a dynamic analysis where the whole mast is modelled. Then factors such as cyclic loading and change of direction of the loads could be included in the analysis.

As can be seen in the results, the optimised model can take much more load than its predecessors. Therefore further optimisation would be interesting. Especially if the results from the dynamic analyses shows that the foundation is over-designed. This is one big advantage of structures using rectangular hollow sections. It is possible to optimise the structure without changing its outer dimensions.

Concerning the joint analyses it would be interesting to do more analyses of other joints than the T-joint. During the 1980's a lot of tests were done on isolated joints. These tests were performed to create formulas for designing joints. It would be interesting to see how the test results, the formulas, and a finite element analysis would coincide.

Another aspect would be a research of how the joints would be affected of dynamic loads until material failure, i.e., cracking due to cyclic loading, etc.

Bibliography

- [1] <http://www.abaqus.com>.
- [2] BOVERKET. *Boverkets handbok om stålkonstruktioner, BSK 99*. Karlskrona, 1999.
- [3] CHEN, W. F., AND HAN, D. J. *Plasticity for Structural Engineers*. Springer-Verlag, New York, 1988.
- [4] COOK, R. D., MALKUS, D. S., PLESHA, M. E., AND WITT, R. J. *Concepts and Applications of Finite Element Analysis*, fourth ed. John Wiley & Sons, New York, 2002.
- [5] CRISFIELD, M. A. *Non-Linear Finite Element Analysis of Solids and Structures*. John Wiley & Sons, New York, 1991.
- [6] HÖGLUND, T. *Dimensionering av stålkonstruktioner, Utdrag ur Handboken Bygg, K18*. CE Fritzes AB, Stockholm, 1992.
- [7] HIBBITT, KARLSSON, AND SORENSEN. *ABAQUS/Standard User's Manual*, 6.3 ed. Hibbitt, Karlsson and Sorensen Inc, Pawtucket, 2002.
- [8] HOWLETT, J. H., JENKINS, W. M., AND STAINSBY, R. *Joints in Structural Steelwork*. Pentech Press, London, 1981.
- [9] HUEBNER, K. H., THORNTON, E. A., AND BYROM, T. G. *The Finite Element Method For Engineers*, third ed. John Wiley & Sons, New York, 1995.
- [10] KALISZKY, S. *Plasticity, theory and engineering applications*. Elsevier, Budapest, 1989.
- [11] RAUTARUUKKI OY. *Handbok för Rautaruukki Kallformade Konstruktionsrör*. Hämeenlinna, 1997.
- [12] TYRÉNS. Mast på handens brandstation, fundament, uppdrag nr 203074. Construction Drawing, 2004.
- [13] UGURAL, A. C., AND FENSTER, S. K. *Advanced Strength and Applied Elasticity*, third ed. Prentice-Hall, New Jersey, 1995.
- [14] WARDENIER, J. *Hollow Section Joints*. Delft University Press, Delft, 1982.

- [15] YU, W.-W. *Cold-Formed Steel Design*, third ed. John Wiley & Sons, New York, 2000.

Appendix A

Path Plots

Every analysis have its own code. The three digits tell what model it is and which load case is applied. For example, 6.4.2 means version 6, model 4, load case 2. Version 6 is the coarser mesh and 7 the finer mesh.

The plots in this appendix are taken from the results of the second model, first geometry and the model with the optimised geometry.

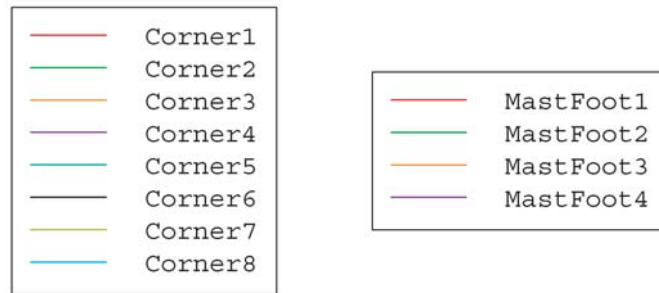


Figure A.1: The path colours.

A.1 Corner paths

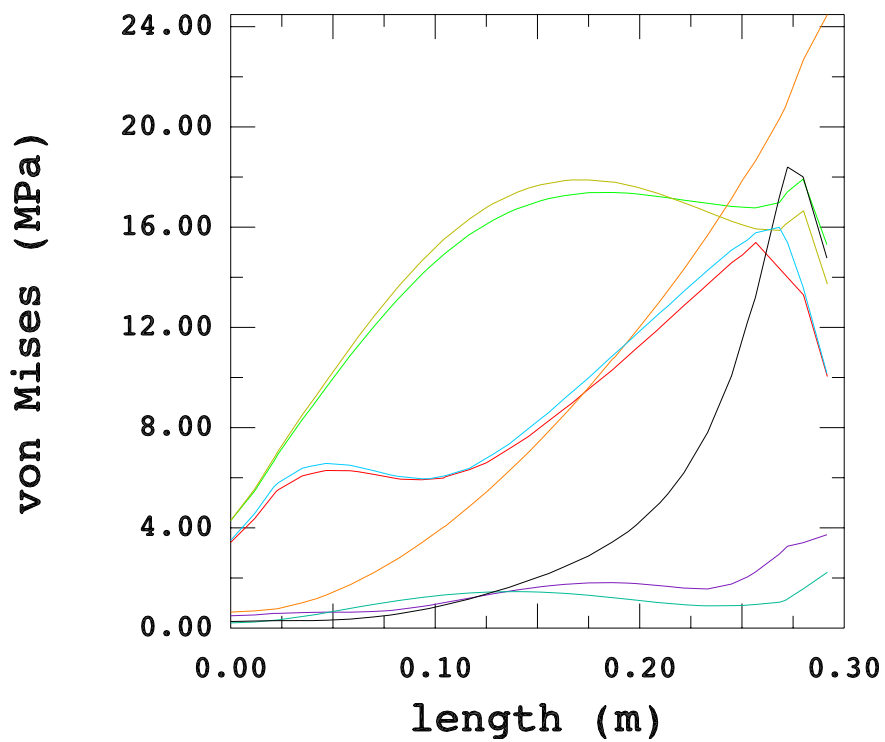


Figure A.2: 7.1.1

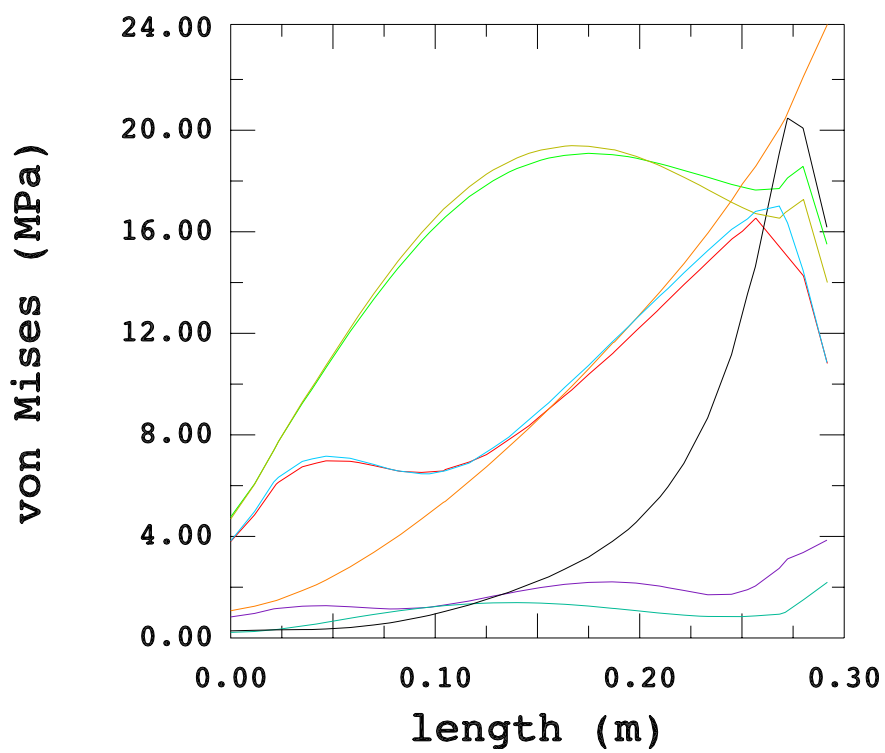


Figure A.3: 7.1.2

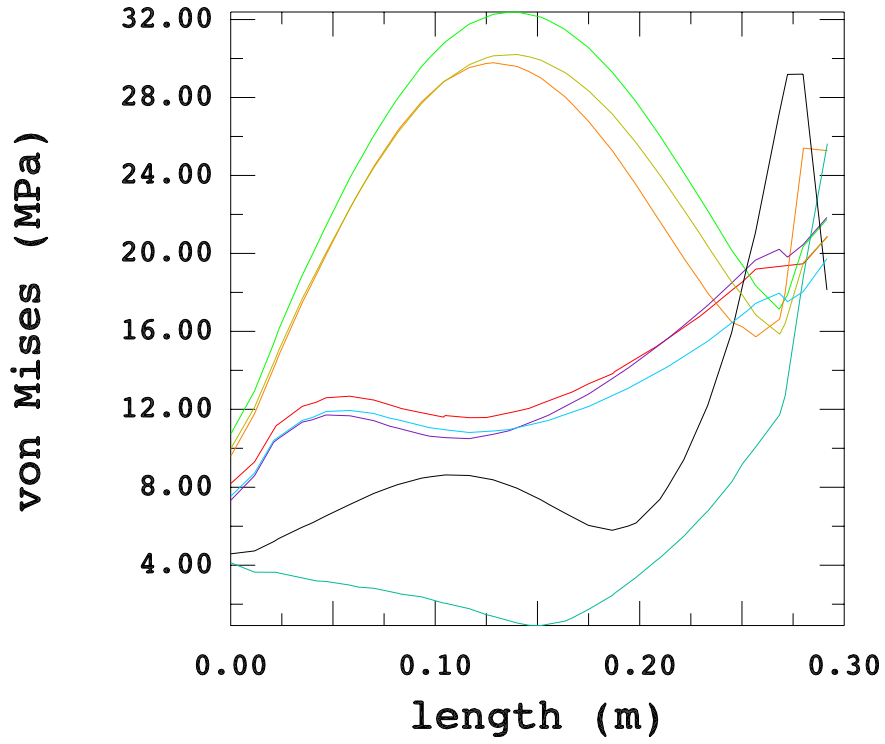


Figure A.4: 7.1.3

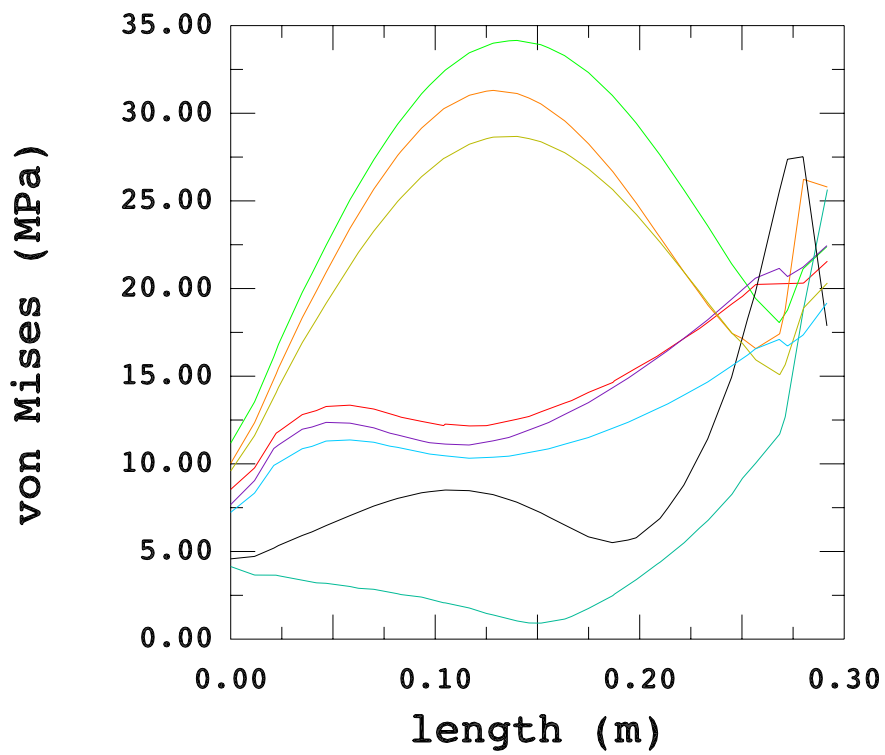


Figure A.5: 7.1.4

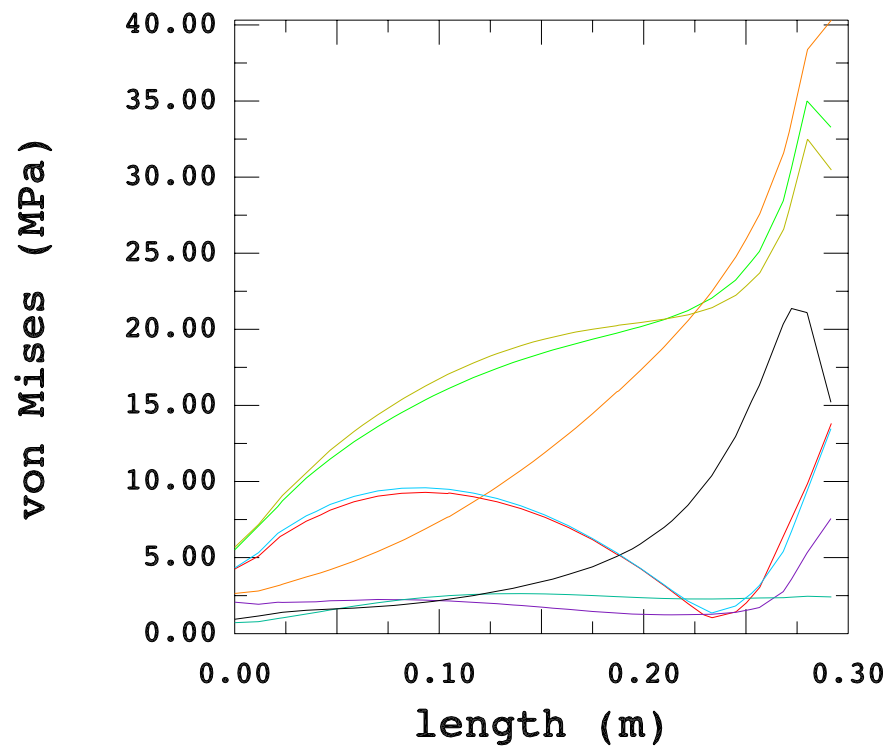


Figure A.6: 7.8.1

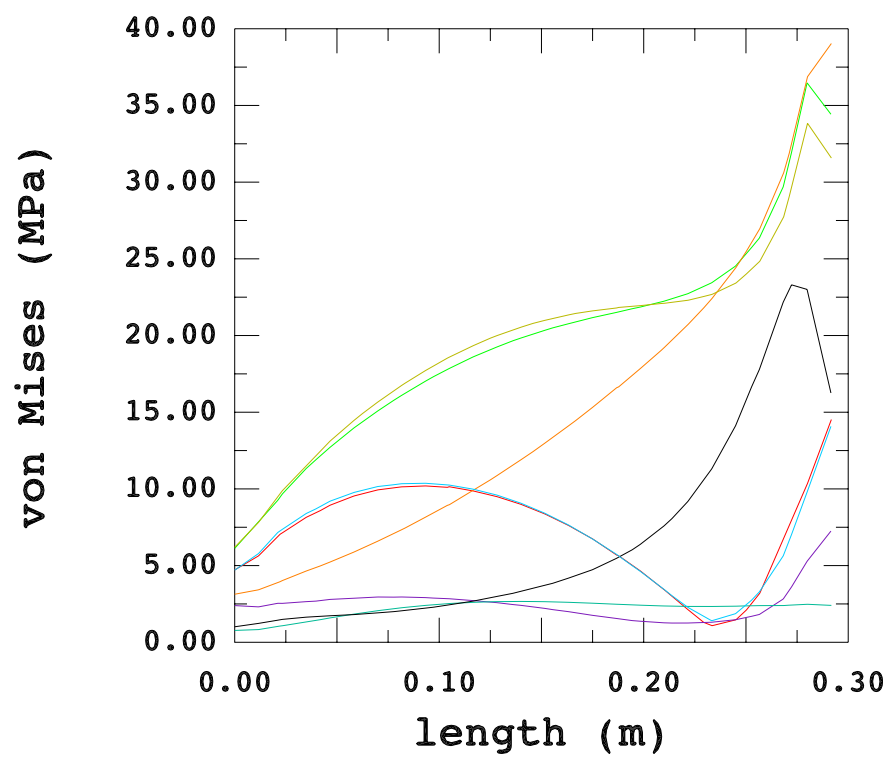


Figure A.7: 7.8.2

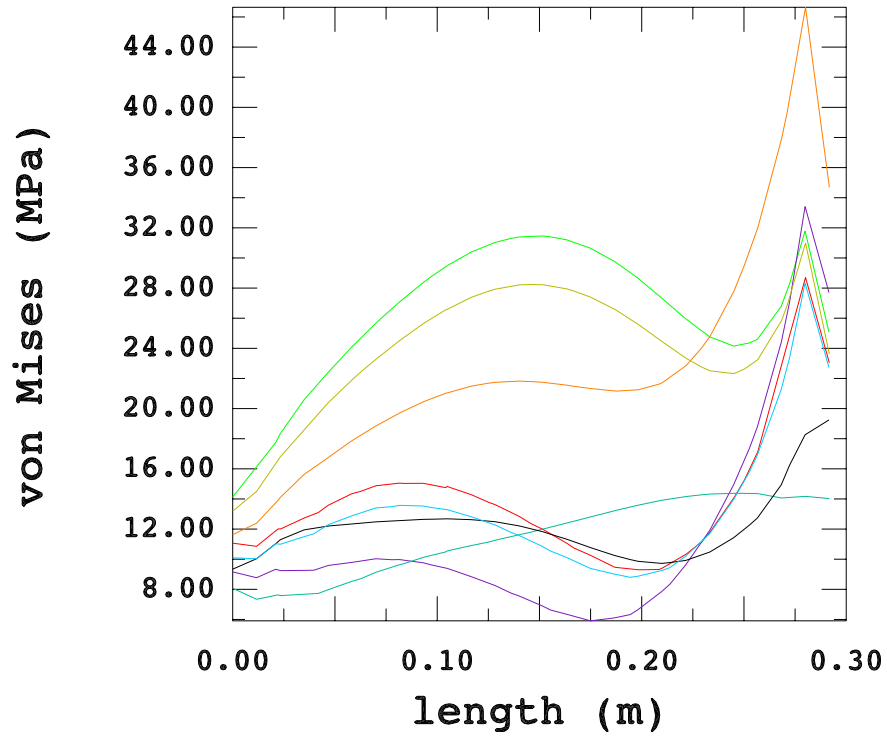


Figure A.8: 7.8.3

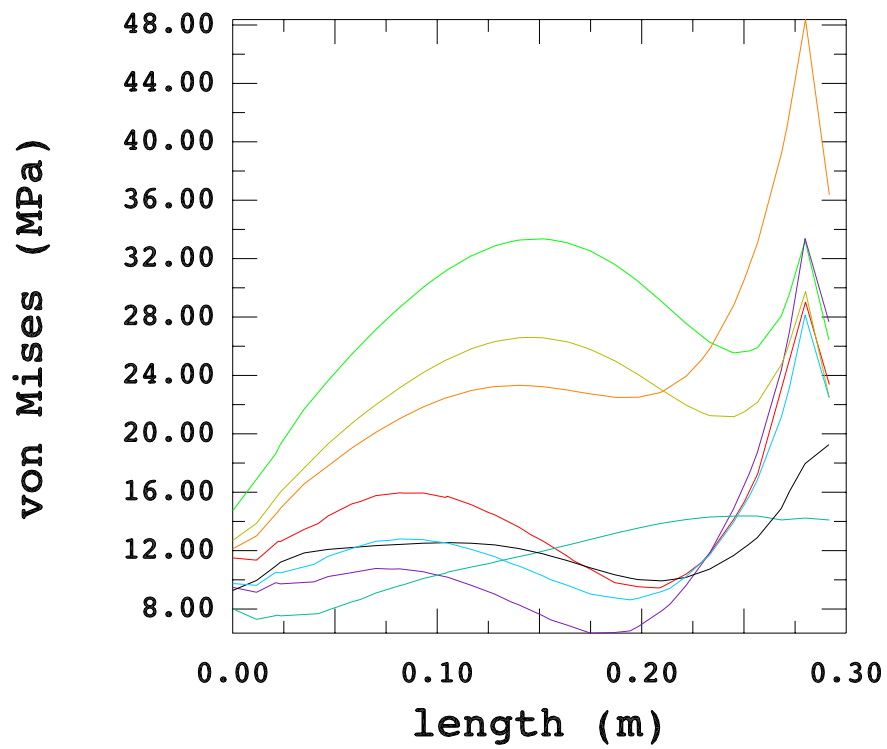


Figure A.9: 7.8.4

A.2 Mast foot paths

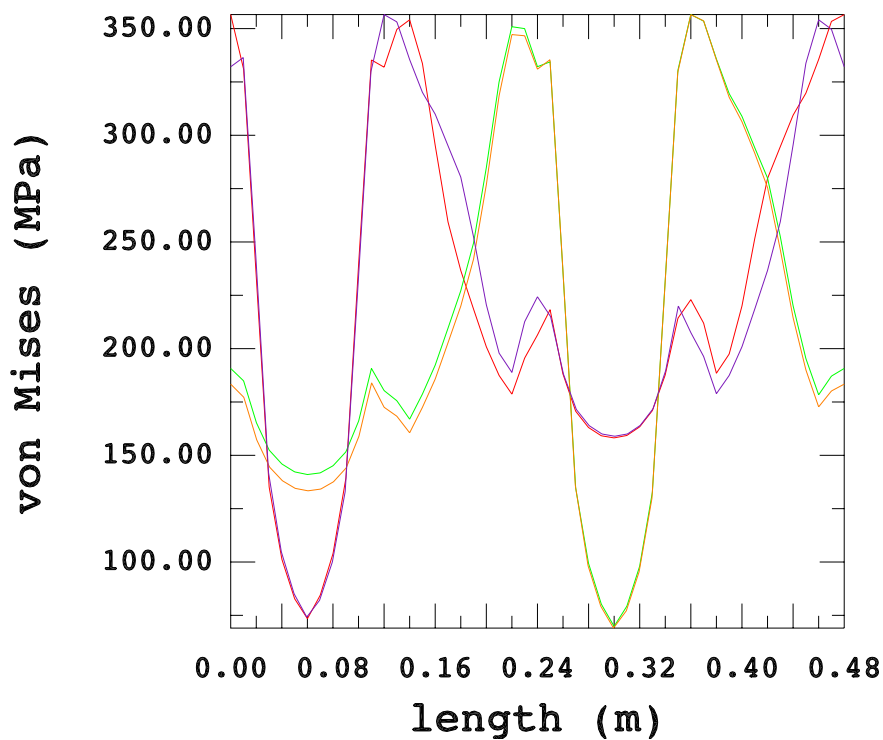


Figure A.10: 7.1.1

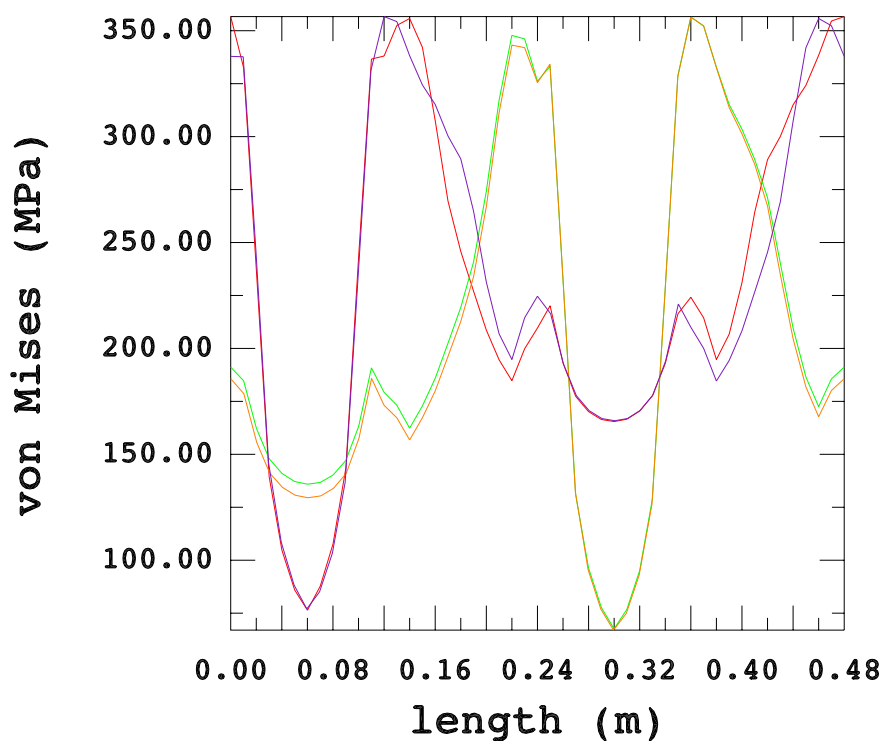


Figure A.11: 7.1.2

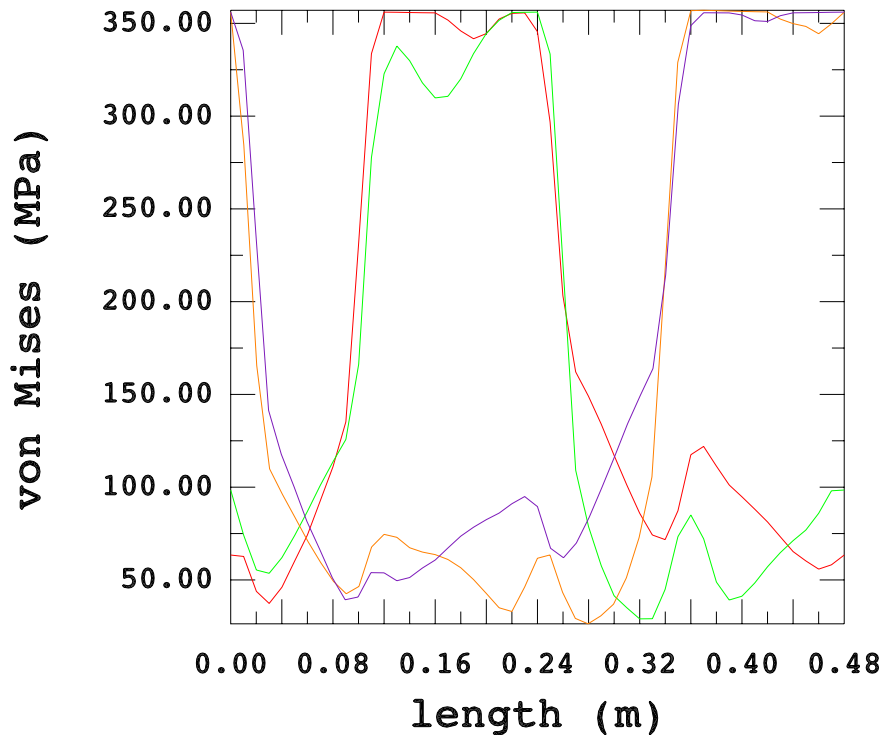


Figure A.12: 7.1.3

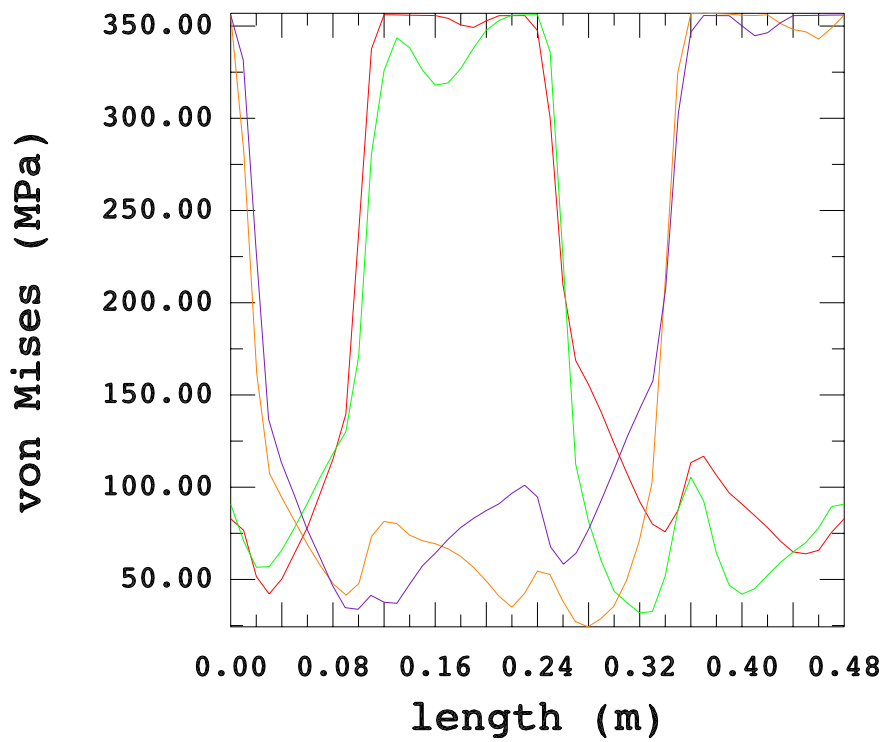


Figure A.13: 7.1.4

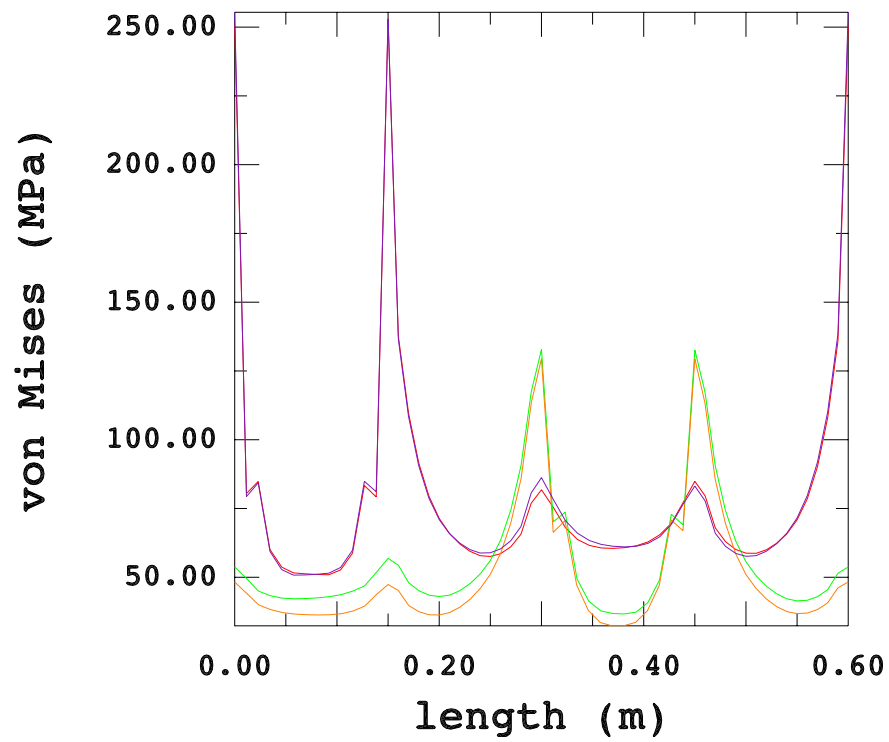


Figure A.14: 7.8.1

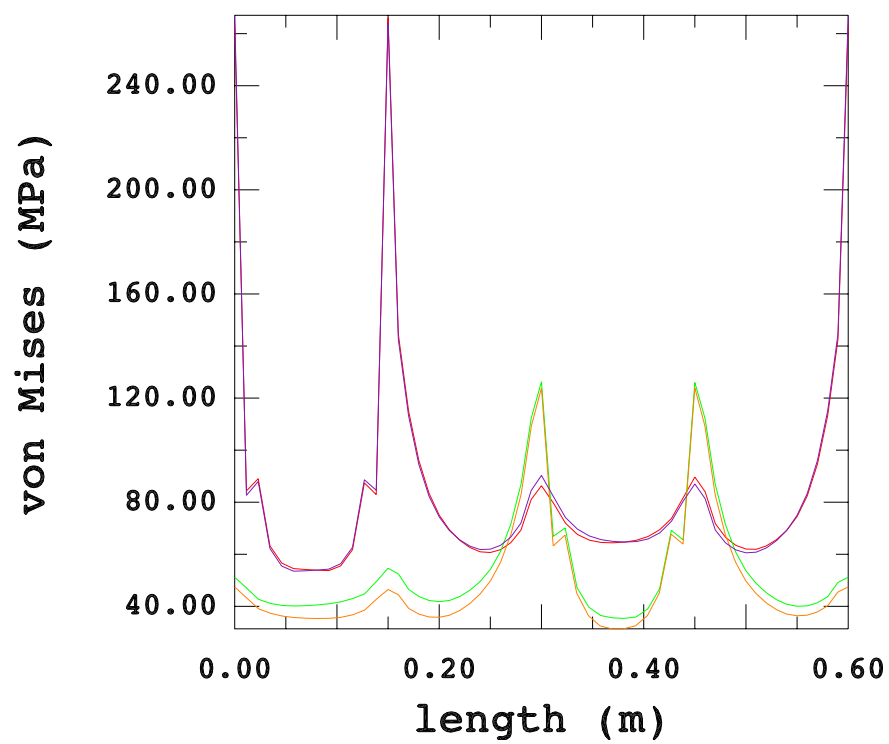


Figure A.15: 7.8.2

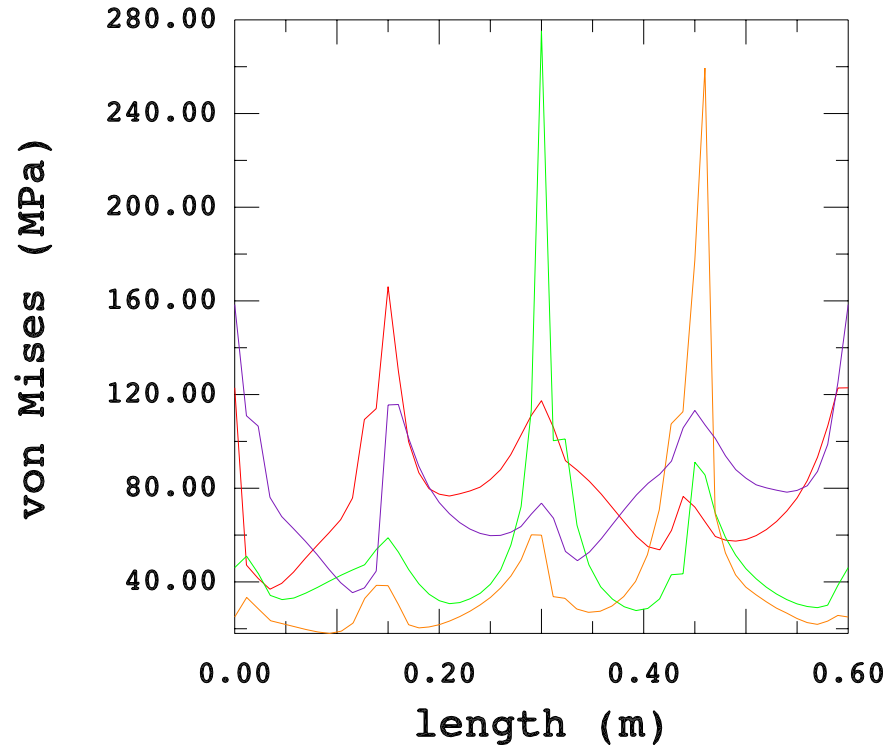


Figure A.16: 7.8.3

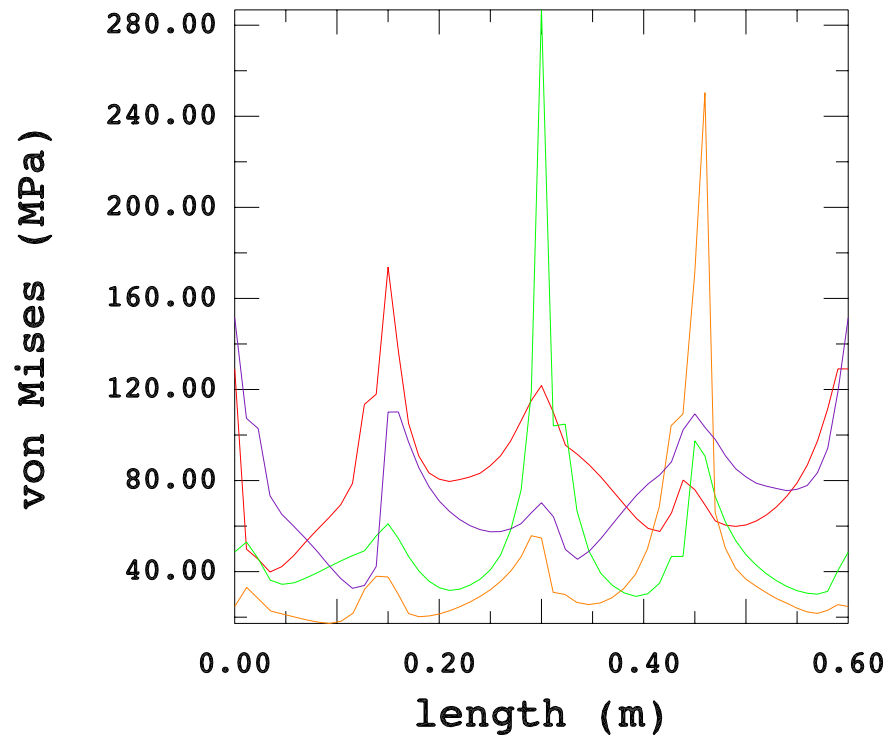


Figure A.17: 7.8.4

Appendix B

Abaqus Input Files

B.1 Foundation model input files

Only the first input file for the first model is printed here. The rest of the input files are similar. The assembly part of the input file will not be printed to save space. The differences between the models lie within the assembly. There is also no need to print the different input files for the different load cases. Negative values are the only difference between these cases.

```
*Heading
** Job name: 6_1_1 Model name: Model-1
**
** PARTS
**
*Part, name=Leg
*End Part
*Part, name=LongFrame
*End Part
*Part, name=MastFoot
*End Part
*Part, name=MastFootTap
*End Part
*Part, name=ShortFrame
*End Part
*Part, name=Support
*End Part
*Part, name=Tap
*End Part
*Part, name=Truss
*End Part
**
** ASSEMBLY
**
```

```
*The assembly is left out.
*End Assembly
**
** MATERIALS
**
*Material, name=S355J0
*Density
7800.,
*Elastic
  2.1e+11, 0.3
*Plastic
  3.45567e+08,      0.
  3.50271e+08, 0.0134943
  4.22657e+08, 0.0324655
  4.57333e+08,  0.111151
*Material, name=S355J2H
*Density
7800.,
*Elastic
  2.1e+11, 0.3
*Plastic
  3.556e+08,      0.
  3.60424e+08, 0.0134459
  4.21685e+08, 0.0301672
  4.57333e+08,  0.111151
** -----
**
** STEP: Step-1
**
*Step, name=Step-1
*Static
1., 1., 1e-05, 1.
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Symmetry/Antisymmetry/Encastre
*Boundary
_PickedSet362, ENCASTRE
**
** LOADS
**
** Name: Horizontal   Type: Concentrated force
*Cload
_PickedSet581, 1, 0.
_PickedSet581, 3, -12500.
** Name: Moment1     Type: Pressure
```

```
*Dsload
_PickedSurf408, P, -3.823e+06
** Name: Moment2    Type: Pressure
*Dload
_PickedSurf409, P, 3.823e+06
** Name: Moment3    Type: Pressure
*Dload
_PickedSurf410, P, 3.823e+06
** Name: Moment4    Type: Pressure
*Dload
_PickedSurf411, P, -3.823e+06
** Name: Vertical    Type: Concentrated force
*Clload
_PickedSet582, 2, -3750.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=1
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
U, RF, CF
*Element Output
S, PE, PEEQ, PEMAG, LE, SF
*Contact Output
CSTRESS, CDISP
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*El Print, freq=999999
*Node Print, freq=999999
*End Step
```

B.2 T-joint model input files

Only the first input file of the first model is printed.

```
*Heading
** Job name: J1_1_1 Model name: Model-1b
**
** PARTS **
*Part, name=Beam
```

```
*End Part
*Part, name=Bracing
*End Part
*Part, name=Chord
*End Part
*Part, name=RigidMastFoot
*End Part
*Part, name=RigidPlate
*End Part
*Part, name=WeldCorner_1o3
*End Part
*Part, name=WeldCorner_2o4
*End Part
*Part, name=Weld_1o3
*End Part
*Part, name=Weld_2o4
*End Part
**
** ASSEMBLY
**
*The assembly is left out.
*End Assembly
**
** MATERIALS
**
*Material,
name=S355J2H *Density 7800., *Elastic
  2e+11, 0.3
*Plastic
  3.556e+08, 0.
  3.60424e+08, 0.013446
  4.21685e+08, 0.030167
  4.57333e+08, 0.111151
**
** BOUNDARY CONDITIONS
**
** Name: FixedEnds Type: Displacement/Rotation
*Boundary
_PickedSet40, 1, 1
_PickedSet40, 2, 2
_PickedSet40, 3, 3
_PickedSet40, 6, 6
** -----
**
** STEP: Step-1
**
```

```
*Step, name=Step-1
*Static
1., 1., 1e-05, 1.
**
** LOADS
**
** Name: Load-1    Type: Pressure
*Dload
_PickedSurf501, P, -3.823e+06
** Name: Load-2    Type: Concentrated force
*Cload
_PickedSet502, 1, 12500.
_PickedSet502, 2, -3750.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=1
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*El Print, freq=999999
*Node Print, freq=999999
*End Step
```

