Analysis of headbox structural strength

OptiFlo II TIS 2200

Hållfasthetsanalys av inloppslåda

Niclas Huss
Acknowledgement

I would like to thank Erik Wedin for giving me the opportunity to write my Master Thesis at Valmet and for supplying me with useful help and support throughout my Thesis work. I also want to send a special thanks to Jonas Cederlöf who has given me support and guidance through the calculations. A big thanks also goes out to the entire calculation department for making me feel like part of the team and has supported me through their expertise.

I would also like to send a big thanks to my supervisor at Karlstad University, Mikael Grehk who has given me guidance through follow-up meetings on a weekly basis and great support when a problem has arisen.

Finally, I would like to express my appreciation for all friends and family supporting me through all these years of studies.

Niclas Huss
Karlstad, May 2019
Abstract
Valmet is a world leading supplier of tissue machines. In the tissue industry the demand on the machines is high to be able to produce the finest paper for use in products such as toilet paper and napkins. The machine components need to perform to be able to produce a product of consistent quality. Many different properties of the tissue can be evaluated through manipulation of the fibre content. Layers of cellulose are added into the process through a headbox, which acts as nozzle which distributes the fibres and water along the machine width. The tissue machines can be around 6 metres wide which put high demands on structural rigidity of the machine components to be able to maintain a uniform product along the machine width. Crucial parts are calculated prior to manufacturing to make sure they fulfil stress and rigidity criteria.

This thesis covers the development of such a computational model of a headbox. The aim was to create a model which will be accurate in terms of deflection but also fast to calculate using ANSYS Workbench software.

Many simplifications were made in the model to speed up solving time such as:
- Removal of non-structural parts
- Mirror symmetry
- Material manipulation to replace perforated structurally important items

Also contact formulations were showing a large effect on accuracy and solution time in the model.
With applied simplifications the solver time was decreased by over 73% compared to the initial model.

The computational model was used in the thesis to evaluate the current configuration of the transverse welds in the headbox when exposed to static load. The load was applied by internal pressure. An evaluation was desirable following a material change from austenitic stainless steel SS2343 to the LDX2101 of duplex grade. LDX2101 which have a significantly higher yield stress of near 500 MPa compared to 250 MPa for SS2343.
For static evaluation the welds showed low stresses in welds throughout the headbox. This leaves room to possible optimization of weld applications by reduction to least recommended throat measure of 3mm. This reduction result in quicker manufacturing time and thereby lower manufacturing costs. Static calculations show possible reductions of 66% in application time.

Det här examensarbetet gick ut på att utforma en beräkningsmodell för en inloppslåda. Fokus låg på att skapa en modell som både ger trovärdiga resultat men även snabb att beräkna genom programmet ANSYS Workbench.

Många förenklingar gjordes för att reducera modellen och därmed snabba på beräkningen så som:
- Eliminering av icke strukturella komponenter
- Symmetri genom ett symmetriplan
- Framställning av materialekvivalent för ersättning av perforerade strukturella komponenter

Även kontaktformuleringar visade sig ha stor inverkan på tillförlitlighet och beräkningstid. Med applikerade förenklingar kunde beräkningstiden förkortas med över 73 % jämfört med utgångsmodellen.

Beräkningsmodellen användes i utvärdering av tvågående svetsar i inloppslådan vid statisk belastning av inre tryck. En utvärdering var eftertraktad då stålet i inloppslåden har ändrats från tidigare austenitiskt rostfritt stål SS2343 till nuvarande duplex rostfritt LDX2101. LDX2101 har betydligt högre sträckgräns än tidigare SS2343 på nära 500 MPa jämfört med 250 MPa för SS2343. Genom statisk utvärdering visade svetsarna låga spänningar som inom kriterierna skulle vara möjliga att reducera i dimension, till lägsta rekommenderade a-mått på 3mm. Detta kan i sin tur reducera tillverkningstid med över 66 %.
**Contents**

1 Introduction ........................................................................................................................................... 1  
   1.1 Headbox ........................................................................................................................................ 4  
      1.1.1 Connecting components ........................................................................................................ 4  
      1.1.2 Internal components ............................................................................................................. 4  
   1.2 Material selection .......................................................................................................................... 6  

2 Purpose .............................................................................................................................................. 7  
   2.1 Limitations CM ............................................................................................................................ 8  
      2.1.1 Structural limitations ............................................................................................................ 8  
      2.1.2 Computational limitations .................................................................................................. 8  
   2.2 Limitations WM ........................................................................................................................... 9  

3 Theory .............................................................................................................................................. 10  
   3.1 Mesh .......................................................................................................................................... 10  
   3.2 Contacts .................................................................................................................................... 12  
   3.3 Symmetry .................................................................................................................................... 14  
      3.3.1 Mirror symmetry ................................................................................................................ 14  
      3.3.2 Axisymmetric symmetry .................................................................................................... 14  
      3.3.3 Applicability ........................................................................................................................ 14  
   3.4 Singularities .................................................................................................................................. 15  
   3.5 Optimization ............................................................................................................................... 17  

4 Method .............................................................................................................................................. 18  
   4.1 Modelling of headbox CM ......................................................................................................... 18  
      4.1.1 Dimensions ........................................................................................................................ 18  
      4.1.2 Simplifications ..................................................................................................................... 19  
      4.1.3 Contacts ............................................................................................................................. 20  
      4.1.4 Meshing ................................................................................................................................ 20  
      4.1.5 Load case ............................................................................................................................. 20  
      4.1.6 Material data ....................................................................................................................... 21  
      4.1.7 Structural simplifications ................................................................................................... 21  
   4.2 Modelling of welds WM ............................................................................................................ 24  
      4.2.1 Geometry ........................................................................................................................... 24  
      4.2.2 Contacts ................................................................................................................................ 24  
      4.2.3 Meshing ................................................................................................................................ 24  
      4.2.4 Evaluation ........................................................................................................................... 25
1 Introduction

Valmet in Karlstad is a world leading supplier of tissue machines used all around the world. Tissue paper is used for products like toilet paper, napkins and other paper products where high absorption is desired.

In paper machines, there are cellulose fibres in the form of pulp and water, which are distributed along the machine width and then dried and rolled into desired configurations to meet customer needs.

![Figure 1 The four main sections of a Conventional tissue machine](image)

One can say that the manufacturing of a tissue sheet can be divided into four main sections displayed as sections on the machine in Figure 1 and schematically in Figure 2.

![Forming Pressing Drying Reel](image)

**The Forming section** is the first section in the tissue machine. In this section, the pulp and water suspension is distributed between the wire loop and the felt loop from the headbox positioned in the centre of the section, see Figure 3.

![Figure 3 The Forming section with the wire loop to the left and felt loop to the right.](image)
The suspension has a water content of around 99,9% in the jet exiting the headbox. Different headboxes are configured to produce different layers of suspension to manipulate the behaviour of the finished tissue. There are single layer headboxes with one pulp mixture entering the headbox and is distributed and dried into a single layer product. Two or three layer headboxes are commonly used when there are different properties desired throughout the thickness of the paper ply.

In the forming section, the suspension is compressed between the wire and the felt loops, which let water squeeze through the wire web and water can be absorbed by the felt fabric. The suspension is then fully transferred onto the felt where it continues into the pressing section.

The Pressing section consist of the transfer section where the felt loop from the forming section is pressed between a Yankee cylinder and a press roll where water is pressed out from the sheet and simultaneously is transferred to the Yankee cylinder.

The Drying section or commonly the Yankee section consists of the Yankee cylinder as large as 18 feet with an additional hood. The cylinder is heated by steam injected into the internal space of the cylinder and the hood surrounding the Yankee cylinder is heated by additional burners to increase the drying effect, see Figure 4. When the tissue is transferred to the Yankee cylinder an adhesive is used to ensure maximum adhesion between the sheet and the cylinder.

The dried tissue is removed from the Yankee through creping using a doctor blade or also known as creping blade. The creping blade is used both to remove the ply from the cylinder, but it is also used to create extra loft in the paper by incorporation of micro-creases. The micro-creases can be modified depending of the tissue application and can be optimized for e.g. high softness or high bulk. During creping the doctor blades are worn due to interaction with the rotating cylinder and must be changed after certain intervals. To avoid idle time many tissue machines are equipped with multiple doctor blades set up in a magazine with three blade slots. One blade can be active for creping, one for removal of adhesive from the Yankee cylinder post creping and a third which is ready to replace any of the previous two. The one not in use can be taken out and can be replaced with a new one without having to halt the production.
The Reel section is the end of the Conventional tissue machine where the finished tissue is wounded on to a reel into large rolls. These are then removed for further processing and rolling into appropriate roll sizes [1].
1.1 Headbox

1.1.1 Connecting components

The paper/water mixture reaches the headbox through the supply headers. Depending on the type of headbox there are one supply header for every layer in the headbox. The supply headers are shaped as tapered pipes to maintain a uniform pressure at multiple following exits along the machine width. To fine-tune the pressure formation, the supply header is equipped with a bypass nozzle in the narrow end. By adjustment of the bypass nozzle the pressure in the narrow end can be decreased by opening or increased by closing the nozzle. From the exits, there are multiple hoses leading the water and pulp through the hose plate at the rear of the headbox and into the equalizing chamber in the back of the headbox. On many tissue machines there is an additional dilution system which can fine-tune the fibre composition in one of the layers. This is used to obtain the sought relative concentrations of fibre percentage between the multiple layers. For this add-on, there is a separate dilution header and tubes seen in Figure 5.

1.1.2 Internal components

The internal components of the headbox are shown schematically in Figure 6 below. The suspension is fed from the supply header and the dilution header (if used) by multiple hoses down to the headbox where it enters through the hoseplate and into the equalizing chamber. The main task of the equalizing chamber is to even out or dampen pressure disturbances caused upstream from the supply headers, the connecting tubing or from upstream pumps. From the chamber, the mixture is then fed to the tube bank, which is a section of turbulence tubes connected in a configuration of rows attached in a front and rear tube bank plate. The tube bank plates also act as important structural parts connecting the support beam with the roof beam.
The suspension is lead through the turbulence generator tubes. In the tubes, the mixture is rapidly expanded to create a turbulent flow. The turbulence generators are equipped with rectangular outlets to get a more distributed flow and to apply largest possible cross section outlet area. The turbulence generators are meant to create micro-turbulence which counteracts flocculation. Flocculation is a phenomenon when paper fibres attract each other and create small concentrations which results in uneven distribution of fibres. From the turbulence generators, wings of glass fibre or carbon fibre are attached to create a set of slice channels.

These slices affect quite a few parameters. They even out the flow profile from the turbulence generators and reduce the disturbances in flow caused by the tube walls. They also act as barriers restricting the multiple layers to mix in an early stage and act as accelerators to mix all tissue layers by the lip at equal velocity, which reduces fibre disorientation when merging.

The pressure is always kept constant in the headbox, so an increase in outlet will require an increase in volume flow of water. On most headboxes the lip opening is adjustable, which gives the opportunity to adjust the total water to fibre fraction in the suspension, while the previously mentioned dilution system adjusts the relative water to fibre fraction between the layers. The lip has a lower and upper section where the lower is attached to the stationary roof beam. The upper lip is attached to the apron beam which can be pivoted using multiple screw jacks located in the back of the headbox. By pivoting of the apron beam the lip opening can be adjusted. To fine tune the lip opening profile the Apron beam is equipped with microadjusters. These are equipped with a scale on each microadjuster showing how much the lip deflection deviates from zero position. These can then be calibrated during production to dial-in a uniform profile across the machine width.

Figure 6 The components of a headbox
1.2 Material selection

Through material development and research, it has been decided to introduce a change in material from SS2343 which is an austenitic steel grade to a steel from Outokumpu called LDX 2101. The composition of the two steels can be seen below in Table 1.

<table>
<thead>
<tr>
<th>Steel</th>
<th>C</th>
<th>Cr</th>
<th>Ni</th>
<th>Mo</th>
<th>N</th>
<th>Mn</th>
<th>Cu</th>
</tr>
</thead>
<tbody>
<tr>
<td>SS2343</td>
<td>0,04</td>
<td>16,9</td>
<td>10,7</td>
<td>2,6</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>LDX2101</td>
<td>0,03</td>
<td>21,5</td>
<td>1,5</td>
<td>0,3</td>
<td>0,22</td>
<td>5</td>
<td>1</td>
</tr>
</tbody>
</table>

LDX2101 is lean duplex steel due to the lean content of nickel (Ni) and replaced by nitrogen (N), which is less expensive. Manganese is also added as partial replacement of nickel and increases the solubility of nitrogen. The nitrogen contributes to corrosion resistance and simultaneously increases the strength. The high strength is one of the desired properties in the duplex LDX 2101 with yield strength close to 500 MPa compared to around 250 MPa for SS2343 and tensile strength close to 700 MPa. Another advantage when using the steel in rolled condition, it has a low level of anisotropy, which means that the sheet will have the same properties lengthwise in the rolling direction and transverse the sheet.

The corrosion resistance is moderate for the LDX2101 with a good resistance to stress corrosion cracking. Compared to the SS2343 the LDX2101 has slightly lower corrosion resistance, but is significantly stronger. For this application the pros of strength and price outweighs the slightly lower corrosion resistance.

The steel also shows excellent machinability which is unusual for high alloyed steels; though an important quality for edge preparations before welding, chamfering and other plastic processing of the steel components.

Weldability is good but certain modification of the edge preparations might be necessary through an increase of joint angle by 10° compared to joints configured for SS2343. High nickel content in weld filler is also important to achieve similar microstructure as the parent material which will have a ferritic base with austenitic nucleation.

Duplex grades are sensitive to subsequent passes during welding and which require a max inter pass temperature of 150°C [2].

The mechanical properties compared between the earlier and new steel is similar in YM (Young’s Modulus), which is useful since the stiffness of the machine components can be assumed not to change.
2 Purpose

The purpose is to create an analysis model using Finite element method (FEM) for the headbox OptiFlo II TIS 2200, but also developing an approach applicable on all types of headboxes. The model is simplified to achieve accurate results even when using low computational resources. An accurate model with quick solving time is an important tool for evaluation of design changes using Finite Element Analysis (FEA).

Using the FEM model of the OptiFlo II TIS 2200, the weld design will be investigated for possible improvement in weld design and weld application with the aim to reduce cost and manufacturing time. A change in material selection, from austenitic to duplex steel grade, results in a great improvement in strength. The LDX 2101 has a yield strength of nearly 500 MPa compared to SS2343 with a yield strength of 250 MPa. Thereby there is a possible room for improvement through optimization of dimensions of welds due to the higher allowable stresses.

The headbox contain many parts, which are assembled through weld joints. In this study, the larger weld joints throughout the headbox width will be investigated. These contain the largest amount of filler and are generally of greater dimensions than welds going in the machine direction, which is the direction downstream, the machine. The large fractions of weld material applied are expensive and require long time to be applied, especially if it needs to be applied in multiple stages to reach the desired thickness. In addition, thick weld application affects the underlying material by a large heat affected zone (HAZ), which might cause weld distortion and formation of brittle intermetallic phases. These problems can be reduced if the dimensions of the welds can be reduced, with the support by the higher stress criteria for LDX 2101. Caution need to be taken since after reduction in weld throat the welds will be harder to make in small dimensions and might therefore increase the cost of application again. This require a balance of correct weld dimension based on applicability and stress optimization.

The thesis work is executed in two steps where the first part focuses on the generation of a computational model for calculation of the headbox. The second step, the weld model makes use of the computational model to evaluate the weld dimension, with restrictions set by internal stress criteria. For this reason, the following chapters are divided into two sections to separate the two steps. The headings with the notation CM regard the computational model and the headings fit the notation WM regards the weld model.
2.1 Limitations CM

2.1.1 Structural limitations

It is important for the lip opening keep its shape when the internal pressure is applied. When the lips open due to applied pressure the headbox is yawning. This means when the applied internal pressure causes elastic deformations, separating the upper and lower lip and hence increasing the outlet. The focus is to have as little yawn as possible but most importantly constant yawn along the machine width to reduce inconsistencies in volume flow i.e. rigidity is crucial. The initial model needs to show a deflection corresponding to the guidelines provided by Valmet. The modified models cannot vary much in deflection compared to the initial model, since a varying deflection will correspond to error due to faulty approximations or an uncertain evaluation model.

2.1.2 Computational limitations

The advantage of FEA is the way calculations can be performed on any structure. Though, it is sensitive to the complexity of designs where it will be more time and computationally consuming to perform. Since the time is limited in the thesis there is a need for computational optimization to be able to make quick changes and evaluate these in short time. On the hardware side of the limitations, there are ways to speed up calculation by increasing computational power and thereby solve complex evaluations quickly. But such changes will not be evaluated in the thesis. A useful approach is to adapt the analysis to the computational hardware.

All calculations are memory consuming when solving a problem and this quickest working memory is the RAM or the in-core memory. In Ansys it is stated when running a calculation how much memory is used and what memory resources are available. Is the required memory larger than the in-core memory the computer will use the out of core memory instead that is the hard drive. This solution is using both more memory and a slower memory which will increase the calculation time. Therefore, one limitation is RAM memory; but since the calculation is still possible to perform out of core it is more of an ambition to keep the required memory lower than what is available. In most situations, the RAM memory is set and the model must be simplified to reduce computational demands. In case of out of core solution of a large model, the hard drive space will be a limit for the amplitude of the calculation.
2.2 Limitations WM

By conversation with the Valmet engineers, it has been made clear that some welds shall be prioritized over others in terms of the need of optimization. There are two reasons of optimization, which are strongly correlated. The first is that the thickest welds of the headbox require multiple passes to build the desired thickness of the weld. This makes the application time consuming and thereby costly. Second is that large thick welds affect the surrounding material, as previously mentioned. For this reason, the thesis will be aimed at optimization of these problematic welds situated along the machine width of the headbox.

Weld sizes of smaller dimensions than a throat of 3 mm will not be investigated due to difficulties in application using regular weld equipment. In addition, the welds are post machined which need processing allowance, which would be insufficient for smaller welds.
3 Theory

3.1 Mesh

To make a Finite element model the entire model is divided into small computational elements and the desired properties are calculated for each node within an element. The elements can be of different kinds.

In this analysis, the evaluation will be of a 3D model built of solid parts, but also some 2D parts. Therefore, this thesis will only cover the principle of solid elements and one case of 2D elements.

In Ansys, the common 3D-elements are of tetragonal or hexagonal shape with one node in each corner. Geometrically one eight node hexahedron can fit five tetrahedrons using the same number of nodes, see Figure 7.

![Figure 7 The geometrical compatibility between a hexagonal element and five tetragonal elements.](image)

Each node of the element has three degrees of freedom which are translation along global x, y and z-axis. The total degrees of freedom for an element are found through multiplication of the nodal D.O.F with the number of nodes in the element.

Through the evaluation of the strain matrix for the tetragonal element it can be found that, there are only constants present in the matrix which implies that there is constant strain and thereby also constant stresses through a four-node tetragonal element. The four node tetrahedron is known as a constant strain or constant stress element. This can be an issue when accurate stresses need to be evaluated and there is no stress gradient present in the element. The four node tetrahedron also has linear shape functions, therefore it is often known as a linear tetragonal element. Linear tetrahedrons perform badly during bending loads and a beam in bending might therefore appear as much stiffer than compared to an actual case.

To get around this issue there are a few alternative solutions. One is to decrease the size of the mesh, which will give a more accurate result, but would be time intensive to calculate. The second solution would be to change to quadratic tetragonal elements. These have
additional nodes to the previous four situated on each edge of the tetrahedra, resulting in a total of 10 nodes per element. The quadratic tetrahedra have a quadratic shape function and can adapt better to bending. The third option is to change to hexagonal mesh. All hexagonal mesh adapts adequately to bending. In all cases a quadratic mesh would be desired for accuracy, but the downside is the computational demands since the number of nodes increase a lot.

As for 2D elements, Solid shell elements can be applied where a plate of consistent cross section is modelled. This reduces the solid plate model to a shell model. Hence the mesh can be reduced to quadratic elements instead of hexahedral. In this case, the shape is quadratic with four nodes applied in each corner.
3.2 Contacts

In FEA the analysis differs a lot from real life examples when looking at interactions between two bodies. With no contact conditions present, the bodies will simply travel through each other with no sign of interaction.

Therefore, there are a few contact types which are commonly used when setting up a FEM model. Most common is the use of “Bonded” contact. This is a contact to bond two bodies to each other and lock all degrees of freedom relative to each other. This is useful when bodies are fixed by welds or screw joints normally, but these joints are not modelled but replace with such contact condition. If the bodies are not entirely fixed, there are a couple of other options available. One is “No separation”. This locks the translations to be zero in the normal direction between two bodies, although the bodies can move relative each other through sliding.

Other contacts allowing sliding are “Frictionless” contact and “Frictional” contact. The frictionless contact allows the bodies to slide relative each other, but there is nothing bonding them together, allowing the bodies to separate entirely during calculation. Frictional contact is like the previous, but a friction coefficient is added between the bodies. The two latter contacts might be most accurate, but they also use a non-linear solution, which require an iterative solver and require a lot more time to solve than the other contact conditions. Therefore, when sliding is present and short computational time is preferred, no separation is a good alternative to the non-linear contacts.

The contact types are often quite set by the definition of the problem, although there is a possibility to choose what formulation of the contacts is used. There are four main variants of contact formulations:

- **Penalty method** is probably the most common method which applies a contact force in terms of a normal force based on the overlap between the bodies. The formulation works as a stiff spring which will act as a substitute to real contact. Naturally the normal force follows same equation as the force from a spring, see Eq. 1 where k is the contact stiffness and x the overlap distance between the bodies.

  \[ F = k * x \]

  As seen in the equation above, the contact force is direct related to the contact stiffness, which is by default adjusted by the program. Valmet engineers have previously discovered that the program-controlled stiffness might vary depending on the complexity of the model. A more computational heavy model, the softer stiffness. This can be a problem for the accuracy of the model, so a control of the stiffness might be necessary.

- **Augmented Lagrange** is an invariant of the penalty method above using the same equation with the addition of one term, see Eq. 2.

  \[ F_N = k * x + \lambda \]
This results in a lower sensitivity to the contact stiffness set by the spring constant $k$. Overall the formulation and the result of the contact is more or less identical to the Penalty method, as can be seen in Section 5.2.4.

- **Normal Lagrange** is a great formulation when it comes to contacts since the approximation is low and the outcome is not as sensitive to input data as previous mentioned penalty methods. This has a lot to do with the fact that there is no penetration allowed between the bodies and the normal force is applied directly between the surface nodes, which is in line with a real scenario. The correctness of the normal Lagrange comes at a price though. It is a lot heavier calculation wise and therefore a lot more time consuming, as in the Section 5.2.4 where normal Lagrange take over 69% longer time to solve compared to penalty method.

- **MPC** is another method which result in no penetration like the normal Lagrange. Although this method uses MPC elements to glue the nodes together. This is only usable on no separation or bonded contact types. The results provided in Section 5.2.4 provides results that evaluation of the stresses can become wrong using the MPC contact. Though there are a few applications where MPC contacts are useful. If running a heavy non-linear solution with convergence issues, MPC contacts can be applied since they are used with iterative solver and converges easily and simultaneously require low computational power. Also, if modelling using 2D elements, other contacts cannot transfer rotational degrees of freedom between two 2D surfaces, but MPC could apply such effect.

The headbox is evaluated with the above-mentioned methods to investigate the effect on a larger model.
3.3 Symmetry

Symmetry is used to quickly reduce the number of nodes present in the model. There are different kinds of symmetry which can be used in the calculation. The two probably most used approaches will be explained below:

3.3.1 Mirror symmetry

Mirror symmetry can be used when there are identical volumes on either side of a plane. More than one plane can also be used if the model is symmetric in more than one direction. When looking at Liu and Quek’s good example of the Eiffel tower; let’s say the four legs are identical, then we can calculate one quarter of the tower and use symmetry to recreate the entire model. This symmetry leads to a reduction to a quarter of the original nodes.

3.3.2 Axisymmetric symmetry

An axisymmetric geometry like a vase can be modelled as only the fraction of the cross section which will make up the entire model when revolved around the centre axis. This will reduce the problem formulation by an entire dimension, from 3D to 2D.

3.3.3 Applicability

Simply a structure must have a symmetric or repetitive nature to fulfil the geometrical requirements of the use of symmetry conditions. However not only the geometrical model will be mirrored or projected around an axis. Loads also need to have a symmetrical nature in a symmetric model. As with the Eiffel tower example above, calculating the effect of the weight can be modelled as a mirrored symmetric model. Though calculating the effect of the wind striking the structure will not be symmetric around the tower and the model is thereby anti symmetric. There is an exception if the wind strikes the tower from a perpendicular direction to one side. Then mirror symmetry can be used on half the model instead applying the mirror plane parallel to the wind direction [3].
3.4 Singularities

Putting a finite model together require a lot of simplifications and one of the most important is that there is literally a need to cut a few corners. In real life, all corners on any structure are made up by small or larger radiuses. These radiuses and in particularly small ones require a large amount of element to approximate the curvature. Therefore, the radiuses are often excluded and the corner is instead modelled as sharp.

A problem which arises from sharp corners is the creation of singularities. A singularity is a point where the peak stress approaches infinity. The geometrical stress concentration factor $K_t$ is calculated through Eq. 3. The geometrical stress concentration factor describes how much the maximal stresses divert from the nominal stresses, evaluated through Eq. 4.

$$K_t = \frac{\sigma_{\text{max}}}{\sigma_{\text{nom}}}$$  \hspace{1cm} (3)

$$\sigma_{\text{nom}} = \frac{F}{A}; A: \text{Cross section area}$$ \hspace{1cm} (4)

When a corner radius approaches zero the relation between radius and the narrowest width $R/b$ will approach zero too. In Figure 8 it can be spotted that an $R/b$ value of near zero leads to a maximal stress value $\sigma_{\text{max}}$ of near infinity.
In FEA a discrete number of elements can be used to approximate the model. The finer mesh that is used, the closer will the result be to the true value. In the case of a sharp corner, then the evaluated max stresses will increase with decreased mesh size [4].

Figure 8 The stress concentration factor $K_t$ in relation to $R/b$
3.5 Optimization

In a first stage of material optimization it is necessary to distinguish what the goal of the optimization is. There are different types of optimization. In most cases, there is a maximisation or minimisation problem present. In many cases in automotive, aviation and other transports optimization is often used to minimize weight and thereby decrease fuel consumption and lower cost. In structural engineering, a common optimization might be to adapt a cross section of a beam to achieve highest possible stiffness using a constant cross section area but variable geometry, thereby using a maximization approach.

The different methods used for direct optimization are many and four common variants are described below:

- Screening: There will be a set number of initial samples which will be randomly distributed over the range of the variable. When the initial samples have been evaluated the program presents several best and worst candidates.

- Multi-Objective Generic Algorithm (MOGA) is a more refined optimization than screening and will also start with a set of initial samples. When the initial samples have been evaluated there is one additional sample corresponding to each following iteration. These sample points are evaluated over a decreasing range towards convergence towards the goal. One advantage of MOGA optimization is that the solution can be evaluated towards multiple goals. The possible samples can be evaluated by many built in chart for example an automated trade-off chart which is useful when evaluating multiple goals.

- Adaptive Single-/Multiple-objective: An intelligent approach which uses the best suited method for the objective and does not evaluate design points outside the iterated range of interest or design points with no possible solution. Although, limited to continuous input variables.

- Mixed-Integer Sequential Quadratic Programming (MISQP) uses a mathematical algorithm to solve optimization problems with one goal, but can handle both continuous and discrete input variables.
4 Method

4.1 Modelling of headbox CM

4.1.1 Dimensions

The modelled headbox in its original configuration is of 5886mm width. An effective way to quickly reduce calculation time is to reduce the size of the model, which has a direct correlation with computational time. When a large reduction in the model is performed it is important not to remove any crucial components for the structural function. When reducing the headbox model it was important to keep the repeating internal structure unmodified and not to manipulate the deflection across the machine width.

The headbox is a symmetric construction in the way that the load bearing structure is symmetric around a plane down the centreline of the machine. By introducing the centre-plane as a symmetry plane the computational model can be split in half.

![Figure 9](image)

Figure 9 A visualization of the remodel of the headbox model from 5886 mm to 1806 mm.

In this model, there are other repetitive elements in the construction such as internal stiffeners and jack spaced uniformly along the machine width. The goal in this model was to reduce the model by symmetry around a plane closer to one side of the model, since the half model still is large. One symmetry plane was found where the model could be sliced down to a model of two jacks with the maintained original spacing and an equal repetitive internal structure as the original model, seen in Figure 9. The new sliced down model was reduced to
a lip opening of 903 mm in width which is equivalent of a complete headbox with an 1806 mm width.

4.1.2 Simplifications

4.1.2.1 Geometrical simplifications

Evaluating an entire model in its real configuration as seen in Figure 10 would include thousands of parts. Each which will be meshed and evaluated which would result in a finite but huge number of elements. A calculation for each node in each element will naturally require long time to calculate provided that the physical requirements exist.

For the geometrical simplifications, the aim has been to simplify the model by reducing its volume and thereby reduce the computational domain and time to solution.

The other aim is to reduce the model to geometries suitable for the used mesh. As mentioned above; a solid element craving low computational power is the hexahedral element and thereby it’s beneficial not to have geometries with small radiuses and chamfers. Hence require linear elements to be small in shape and many to correspond to a rounded surface or as mentioned earlier replaced by cubic tetrahedral elements. For the reason of avoiding the use of quadratic elements, all fillets and chamfers were evaluated for their effect on the mesh and removed where deemed to have a large effect on the linear mesh quality.

Also, relatively small geometries are demanding when meshing since they, regardless of shape need small elements to make up the geometry. Worst are of course rounded small geometries which would need an extensive number of hexagonal elements to correspond to the complex geometries. For this reason, all small geometries like screws, bolts, pins and other fasteners were removed from the model. Even holes are removed where fasteners have been attached and other openings which are evaluated to have a low structural influence.
Other non-structural elements such as hoses and lines are removed since they do not affect the structural behaviour of the model.

4.1.2.2 Mesh simplifications

Solid rods exposed to axial forces are replaced with spring elements with equal spring constant. These can be seen in Figure 11.

For a solid rod with cross section A, length L and YM E the spring constant k can be expressed as in Eq. 5:

$$k = \frac{E \times A}{L}$$

(5)

Plates are modelled with consistent cross section through thickness when possible to facilitate the implementation of solid shell meshing which only require one element through the thickness which reduces the element count.

4.1.3 Contacts

For the first model, all non-movable parts relate to a bonded contact and the plain bearing connecting the apron beam with the support beam is modelled as a No separation connection to allow sliding between bodies. The headbox is fixed at the anchor points since the frame is not included in the calculation.

4.1.4 Meshing

To make a Finite element model the entire model is divided into small computational domains and the desired properties are calculated for each one of the elements. The elements can be of different kinds. In Ansys, the common elements used for a solid model are of tetragonal or hexagonal shape. As mentioned earlier in section 3.1, linear tetrahedral elements are always to be avoided due to the geometrical stiffness [5]. Linear hexahedrons are difficult to apply on curved or slanted volumes unless the elements are made small enough to approximate rounded edge or distorted to calculate a slanted surface. Still linear hexahedral elements are beneficial in application since they have low computational demands and are applicable to most solids, though poorly to curved or slanted surfaces. If the geometrical demands are too big due to problematic geometries as mentioned above, quadratic tetrahedral elements are applied instead to the cost of a huge increase in nodes.

The mesh is configured using hexagonal elements in all cases possible. Some geometry might require wedge shaped elements, but tetragonal elements are avoided in the greatest extent.

4.1.5 Load case (LC)

LC1. The internal fluid domains will be set up with an internal pressure which is classified but kept constant between simplification runs.
The internal fluid domains were set up with internal pressure corresponding to Bernoulli’s equation below in Eq. 6.

\[ \frac{\rho v^2}{2} + \rho g z + p = \text{Constant} \]  

(6)

4.1.6 Material data

The entire original model is modelled as structural steel except for the plain bearing connecting the support beam with the apron which is made of Bronze, see Figure 11.

![Figure 11](image)

The structural steel pre-programmed into Ansys is used out of simplicity since it has similar elasticity as the actual duplex steel used. The bronze used is a modified version of the pre-programmed material for increased accuracy. The elastic properties can be seen in Table 2.

<table>
<thead>
<tr>
<th>Material</th>
<th>YM [GPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structural steel</td>
<td>200</td>
</tr>
<tr>
<td>Bronze</td>
<td>120</td>
</tr>
</tbody>
</table>

4.1.7 Structural simplifications

While some complex geometries have been removed entirely, some other complex geometries cannot be removed since they are important for the structural integrity of the model. This will impose a problem when setting up the model since it will implement a limit of simplification where the model still requires high computational demands. Instead of removing the important geometries other methods are used for further simplification.

In this model, there are three internal perforated plates across the width of the machine with too large openings to neglect.
There are a couple of methods which can be used to replace such sections in the most appropriate way to maintain accuracy with the aim of reducing the computational demands as much as possible. The two methods used in this thesis are explained below.

4.1.7.1 Optimization

In this case optimization is used to reach a specific target. This target is set by the measured directional deflection of the original configuration of rear and front tube bank plates. When the deflections were distinguished the plates were modelled as solid plates of a uniform material with a variable YM, seen in Figure 12.

An upper and lower limit of possible values of the YM is set up by the operator to narrow down the possible range. Although without the use of sound argument and experience the upper limit can be set to the original YM and lower close to zero. Since the properties must be somewhere between the YM of the material surrounding the holes and the YM of the holes, which would be zero.

For the aim of deciding an equivalent YM for the plates the Adaptive single objective method was used since it has a great combination of swiftness and simultaneously precise. Especially when the initial sample range is large and the solver is sought to narrow down the range and avoid unnecessary design points.

![Figure 12 The simplified tube bank plates](image)

4.1.7.2 Geometric accuracy improvements

The tube bank plates are perforated unevenly along the height of the plate and for that reason it could be inappropriate to have a uniform YM across the entire plate. To adapt the method for unequally spaced perforated sections there is a more refined approach where the outlines of the holes are projected along the entire plate width and this isolated volume is then evaluated with a variable YM. This gives a more refined stress profile closer to the real plate. The downside of his method which requires certain attention is that the one plate will now be split into 13 separate bodies which will need to be reattached into one complete plate again. This can be done either by applying contact conditions between the sections or the formation of a new part. When bodies are put together as one component or part they can share mesh. This means that two bodies will have nodes at the boundary which will match nodes from the connecting body. By sharing nodes, the FEA program sees the plate as one entire part once again. When this entire transformation is done, the composition will look as
in Figure 13 below where both tube bank plates and the rear hose plate are transformed to solid parts with sections of different YM.

![Figure 13 The modified material sections have replaced the previous perforated sections.](image)

There were a total of three different perforated sections which each had to be evaluated by an iterative optimization method.

After simplifications are made the simplified model is compared to the original model to validate the configuration.

To evaluate the function of the simplifications the set up between calculations need to be as constant as possible. Since mesh will vary with types of simplifications the validation will be evaluated with the same mesh quality as support for validity. In Ansys the mesh can be evaluated with a method comparing how accurate the shape of the element is. The method uses a shape factor where the element quality can reach a maximum value of 1 for a perfectly shaped element and a 0 value for a zero-volume element. Zero volume can be achieved by such magnitude in distortion that the element is literally flattened.

When simplifying larger models with a vast amount of perforated geometries it will be tedious to use an iterative model. It will therefore be investigated whether the equivalent YM is dependent of the frontal area of the perforated sections using the acquired data from the iterative analysis for the four sections above.
4.2 Modelling of welds WM

All welds are modelled by sketching an appropriate cross section of the weld using the sketch tool in Design Modeller and can be extruded through addition of material in the desired direction.

4.2.1 Geometry

The fillet welds are modelled as the cross section is described in the drawings. A fillet weld is thereby modelled as a triangle and then extruded along the edge of contact. Some other welds present in the application are J-welds and Bevel welds. All welds are modelled as an ideal weld with straight edges along the cross section for simplicity even though a real weld without post treatment might be of a more convex or concave appearance. During weld size optimization, the throat dimension of the welds is investigated which is specified in the Figure 14.

4.2.2 Contacts

The previous bonded body-body contacts are broken and the contact between two bodies is now through the weld by bonded contact between body A and weld and between weld and body B.

4.2.3 Meshing

A difficult part of investigation of welded joints in FEA is that there might be singularities present at weld toe or root as discussed in Section 0. Therefore, a finer mesh will have difficulties in converging and only increasing in stress towards infinity. Because of the triangular cross section of the common fillet welds a tetragonal mesh is chosen to reduce the distortion which would be present using hexagonal elements. As previously mentioned all usage of linear tetragonal element shall be avoided for an accurate solution, since they
provide added stiffness to the model. Therefore, there will be only quadratic tetragonal elements used to model the welds. The downside of using quadratic tetragonal element is the use of mid side nodes, which will increase the node count. In this case the welds are significantly small compared to the entire model so the relative increase in nodes will still be small.

4.2.4 Evaluation

All modelled welds are modelled with one separate coordinate system for each weld with a construction geometry applied perpendicularly with the weld surface through the thickest point of the weld i.e. the weld throat of the fillet weld.
5 Results

All models are calculated using internal pressure including dynamic pressure applied through application of Bernoulli’s equation. Symmetry condition is used in the x-y plane and a fixed displacement criterion is applied at the anchor point of the headbox.

5.1 Simplifications CM

5.1.1 Full model without simplifications

Directional deformation of the original model without structural simplifications showing in Figure 15 to reach a total yawning of the lips by 2,01 mm, at the centre of the entire with of the headbox.

![Figure 15 The directional deflections of the original model.](image)

The lip deflections in y-direction of the initial model are presented in Table 3 below.

<table>
<thead>
<tr>
<th>Selected area</th>
<th>Upper lip [mm]</th>
<th>Lower lip [mm]</th>
<th>Total lip yawn [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td>0,852</td>
<td>-1,161</td>
<td>2,013</td>
</tr>
</tbody>
</table>
5.1.2 Simplified model

Directional deformation of the simplified model with applied variable YM in the internal plates to reach equivalence of the model without simplifications. The internal pressure on applied on the internal plates is also scaled to match an equivalent force acting on the plate, equivalent to the force acting in the model without simplifications. Directional deformation is plotted in Figure 16 and calculated as a total yaw of 2.02 mm seen in Table 4 The max/min deflections and the total yawn of the simplified model.

Table 4 The max/min deflections and the total yawn of the simplified model.

<table>
<thead>
<tr>
<th>Selected area</th>
<th>Upper lip [mm]</th>
<th>Lower lip [mm]</th>
<th>Total lip yawn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td>0.908</td>
<td>-1.116</td>
<td>2.024</td>
</tr>
</tbody>
</table>

To evaluate the equivalent YM for the simplified internal plates, experimental calculations were used to evaluate the deflection of the internal plates in original configuration. This deflection was compared to the modified plates and the YM was varied until the same directional deflection was reached. This was done using the Ansys Optimization application. The desired range of interest was set to exclude values too high or low to be reasonable.

The optimization was performed using Adaptive Single-objective method to determine the equivalent YM for the front tube bank plate using the following input:
- Original YM: 200 GPa
- Range of interest: 30-70 GPa
- Deflection goal: 0.0071939 mm

The calculated design points can be seen in Figure 17 where it converges until the desired deflection goal is reached.
Figure 17 The Adaptive Single Objective solution used for the front tube bank plate.

Best matching result: Equivalent YM of 5,4291 GPa. The other equivalent properties can be seen in Table 5 below.

Table 5 Equivalent YM for simplified perforated sections

<table>
<thead>
<tr>
<th>Section</th>
<th>Front tube bank plate</th>
<th>Rear tube bank plate</th>
<th>Hose plate layer 2</th>
<th>Hose plate layer 1 &amp; 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equivalent YM [GPa]</td>
<td>54,29</td>
<td>62,29</td>
<td>27,36</td>
<td>68,36</td>
</tr>
</tbody>
</table>

To compare the two models using the same mesh quality, the original internal plates were meshed using Quadratic tetragonal mesh to reach the same quality as the simplified plates meshed by Linear hexagonal mesh, seen in Figure 18.

Figure 18 The Original and simplified internal plates meshed to equivalent mesh quality.
5.1.3 CM Results summary

The two models can be compared in Table 6 below, where the total node and element count can be seen to have decreased when the model has been simplified. This results in a decrease in solver time by over 73%. A big reason why the solver time can be so significantly reduced even though the node count is slightly reduced is that the model becomes small enough to be calculated using in-core memory. In-core solutions make use of more memory, but the data transfer is significantly faster which results in a lower solver time.

<table>
<thead>
<tr>
<th>Model</th>
<th>Original</th>
<th>Simplified</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh type of internal plates</td>
<td>Quadratic tetragonal</td>
<td>Linear hexagonal</td>
<td></td>
</tr>
<tr>
<td>Tot nodes</td>
<td>842865</td>
<td>753058</td>
<td>-10.65%</td>
</tr>
<tr>
<td>Tot elements</td>
<td>631840</td>
<td>487548</td>
<td>-22.84%</td>
</tr>
<tr>
<td>Yawn [mm]</td>
<td>2.013</td>
<td>2.024</td>
<td>0.55%</td>
</tr>
<tr>
<td>Solver time [s]</td>
<td>1193</td>
<td>315</td>
<td>-73.60%</td>
</tr>
<tr>
<td>Used memory [Gb]</td>
<td>9.7</td>
<td>17.2</td>
<td>77.32%</td>
</tr>
</tbody>
</table>

5.1.4 General approximation of perforated sections

Through comparison between solid simplified sections in the internal plates with the same sections equipped with holes, the linear relationship below in Figure 19 can be stated. The results have been evaluated through optimization and a linearized graph has been fitted to display the relationship between factors.
5.2 Weld design WM

5.2.1 Selected areas of interest

An initial parametric study of the two welds seen below in Figure 20 show low stresses in both welds even though dimensions are heavily reduced. The internal weld can be seen to the left and the external weld to the right. The stresses are measured in the entire weld and in a plane cutting the weld perpendicular to the weld surface and through the weld root. Comparing the two values of the stress in the entire weld compared to peak stress in the cut, it is clear to see where the stress concentrations are located. As seen in Table 7 the internal stresses differ between the overall and cut stresses which imply stress concentrations in weld toe. When looking at the external weld, the stresses are not concentrated in weld root at large dimensions. When the internal weld is decreased in size the stresses in the external weld become concentrated to the weld root.

![Figure 20 The two welds being analysed in the parametric study.](image)

Table 7 The parametric study of the two welds above.

<table>
<thead>
<tr>
<th>Weld throat</th>
<th>External</th>
<th>External cut</th>
<th>Internal</th>
<th>Internal cut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Internal weld mm</td>
<td>External weld mm</td>
<td>MPa</td>
<td>MPa</td>
<td>MPa</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>88,39</td>
<td>88,13</td>
<td>91,44</td>
</tr>
<tr>
<td>5</td>
<td>7</td>
<td>63,86</td>
<td>63,71</td>
<td>75,93</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>64,90</td>
<td>64,75</td>
<td>85,00</td>
</tr>
<tr>
<td>6</td>
<td>7</td>
<td>67,64</td>
<td>67,33</td>
<td>75,82</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>57,00</td>
<td>54,82</td>
<td>76,31</td>
</tr>
<tr>
<td>7</td>
<td>6</td>
<td>57,26</td>
<td>54,62</td>
<td>80,95</td>
</tr>
<tr>
<td>7</td>
<td>5</td>
<td>57,71</td>
<td>55,08</td>
<td>84,73</td>
</tr>
</tbody>
</table>

5.2.2 Weld application 3mm throat

The 3mm throat dimension model can be seen in Figure 21 where the yawn increases slightly due to higher elongations in welds due to higher nominal stresses. Still the increase is less than 0,1% magnitude which is small compared to the reduction of weld dimensions.
Figure 21 The directional deflections of the model joint by 3mm fillet welds.

Table 8 The max/min deflections and the total yawn of the model joint by 3mm fillet welds.

<table>
<thead>
<tr>
<th>Selected area</th>
<th>Upper lip</th>
<th>Lower lip</th>
<th>Total lip yawn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection [mm]</td>
<td>0,910</td>
<td>-1,115</td>
<td>2,025</td>
</tr>
</tbody>
</table>

Effect on deflection by reduction of weld dimension: \( E = \frac{|(2,025 - 2,024)/2,024|}{2,024} = 0.005 = 0.05\% \) higher lip yawn.

Stresses in the 3mm weld application:
Maximum stress compared to allowable stress: 96\% of allowable value.
The design is OK! The distribution of stresses in the welds can be seen in Figure 22, where it is visualized how the highest stresses are present in a very few welds of lighter colour while the dark blue locations are exposed to low stresses.

Figure 22 The equivalent stresses of the 3mm throat model.
5.2.3 Contacts

Using different contact formulations might give a large impact on the model. Especially if there are many contact areas like in this model. A test was made to investigate the impact of the different contact formulations by applying a uniform contact to all contact areas and compare the outcome in terms of lip yawn.

![Figure 23 The directional deflections of the Penalty model.](image)

**Table 9** The max/min deflections and the total yawn of the Penalty model.

<table>
<thead>
<tr>
<th>Selected area</th>
<th>Upper lip [mm]</th>
<th>Lower lip [mm]</th>
<th>Total lip yawn [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td>0.910</td>
<td>-1.100</td>
<td>2.010</td>
</tr>
</tbody>
</table>

The Augmented Lagrange Figure 24 shows similar results as the Penalty method Figure 23, which is assumed correct since the formulation related between the two.

![Figure 24 The directional deflections of the Augmented Lagrange model.](image)

**Table 10** The max/min deflections and the total yawn of the normal Lagrange model.

<table>
<thead>
<tr>
<th>Selected area</th>
<th>Upper lip [mm]</th>
<th>Lower lip [mm]</th>
<th>Total lip yawn [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td>0.910</td>
<td>-1.100</td>
<td>2.010</td>
</tr>
</tbody>
</table>
The Normal Lagrange formulation showing higher yawn, seen in Figure 25. A large difference between Normal Lagrange and the other contacts is that there are no penetrations allowed between the parts which may result in a more accurate result. The downside is that the contact formulation is heavier to calculate resulting in longer solver times.

![Figure 25 The directional deflections of the normal Lagrange model.](image)

Table 11 The max/min deflections and the total yawn of the normal Lagrange model.

<table>
<thead>
<tr>
<th>Selected area</th>
<th>Upper lip</th>
<th>Lower lip</th>
<th>Total lip yawn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection [mm]</td>
<td>0.693</td>
<td>-1.338</td>
<td>2.031</td>
</tr>
</tbody>
</table>

MPC contacts showing a large difference in yawn than other formulations, see Figure 26. The MPCs are rarely used in solid modelling but can be used when convergence is hard to reach in the model since they require little solver time. They are also often used in shell modelling since they are the only contacts which can translate momentum in these cases.

![Figure 26 The directional deflections of the MPC model.](image)

Table 12 The max/min deflections and the total yawn of the MPC model.

<table>
<thead>
<tr>
<th>Selected area</th>
<th>Upper lip</th>
<th>Lower lip</th>
<th>Total lip yawn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection [mm]</td>
<td>1.117</td>
<td>-1.112</td>
<td>2.229</td>
</tr>
</tbody>
</table>
5.2.4 Contact evaluation

To further check the accuracy of the different types of contact a dummy model containing two plates welded together by a fillet weld on opposite sides was modelled, see Figure 27. As a reference model, the plates were joined through the welds using shared mesh which will basically sew the model together as one part. The mesh used was quadratic tetragonal mesh which behaves well in bending and adaptable to different geometries. The quadratic tetragonal mesh produces a vast number of nodes, but for this small model it reaches the reasonable amount of around 190 000 nodes.

![Figure 27 The evaluation model of investigation of contact formulations.](image)

The model was loaded by a horizontal force of 1000 N perpendicular to the upper edge of the upper plate. The bottom surface of the lower plate was fixed. The equivalent Von Mises stresses through the fillet welds were then calculated to see how sensitive the solution was to the contacts resulting in the Table 13 below.

<table>
<thead>
<tr>
<th>Contact</th>
<th>Mesh (reference)</th>
<th>Penalty Method</th>
<th>Augmented Lagrange</th>
<th>Normal Lagrange</th>
<th>MPC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max $\sigma_{VM}$ Weld 1 [MPa]</td>
<td>4,57</td>
<td>5,70 (+25%)</td>
<td>5,70 (+25%)</td>
<td>4,50 (-2 %)</td>
<td>14,35 (+214%)</td>
</tr>
<tr>
<td>Max $\sigma_{VM}$ Weld 2 [MPa]</td>
<td>5,18</td>
<td>5,90 (+14%)</td>
<td>5,90 (+14%)</td>
<td>4,78 (-8 %)</td>
<td>12,76 (+147%)</td>
</tr>
<tr>
<td>Solver time [s]</td>
<td>53</td>
<td>75 (+42%)</td>
<td>75 (+42%)</td>
<td>127 (+140%)</td>
<td>54 (+2%)</td>
</tr>
<tr>
<td>Memory used [Gb]</td>
<td>1,20</td>
<td>1,14 (-5%)</td>
<td>1,14 (-5%)</td>
<td>4,70 (+292%)</td>
<td>1,85 (+54%)</td>
</tr>
</tbody>
</table>

Table 13 Results from the testing of various contacts in Ansys.
6 Discussion

Modelling and performing an analysis of an industrial component can be evaluated in as many ways as there are components. In this thesis the goal of the computational was to model the headbox in a way which will be equivalent to the physical headbox in production. Comparing real model to computational can be difficult and measurable properties need to be investigated to evaluate the differences between models.

For headboxes an important factor is that the elastic deformation of the headbox lips. Any uniform deformation will offset the water to fibre ratio in the process and might also offset the aim of where the suspension is distributed. The deformations occur at both upper and lower lip, which is called yawning. This was set as the dimensioning property which was used to determine the accuracy of the model. Other properties could also have been investigated and compared given that there would be comparable data available from real runs to compare with.
6.1 Simplifications CM

The set-up of any FEM model requires a lot of assumptions and approximations. Even before starting with this model data like material properties had to be assumed to be constant throughout the entire model, which means a perfect steel of constant chemical composition, no impurities or built in stresses. This is unfortunately never the reality regarding all three assumptions although the discrepancy in material properties is assumed to be of low impact.

When simplifying the entire model smaller items and fasteners were removed and assumed to have low importance to the overall behaviour of the model. Even though the impact is small the result might have been measurable. There are ways to model bolts and other fasteners in a FEA model using methods of simplifications, although time consuming. In this model where the final model will be used for comparisons of different weld joints. An eventual error related to bolt joints simplified to a bonded surface would be present in all models, and therefore of low importance in this model.

Reductions made of the model using symmetry behaviour are commonly used. Symmetry simplifications are though limited of the directions of symmetry, which limited this model to one symmetry plane.

The forces acting on the headbox from internal pressure and structural supports are assumed to affect the structure evenly which made it possible to reduce the model by excluding an entire piece of the model given a symmetry plane was found. This made it possible to shorten the computational time excessively. The assumption is that the deformation behaviour would act in the same manner in a wider model, since the effect of all structural parts are still included in this model.

Simplifications of internal perforated plates were derived from the assumption that the plate is uniformly affected of the holes. Therefore, the perforated regions can be replaced with a different piece of material with lower YM to represent the same behaviour. This is a quite rough estimation since the stress field is manipulated, and thereby removing stress concentrations present between holes. This might be an issue if the stresses in the component are to be controlled. In this case the important data is the comparison of the deflection in the model so stress concentrations are assumed to be of low importance. Stresses are averaged along the perforated areas to evaluate the model with a uniformly distributed deflection along the perforated plates. When looking at a model of multi axial deformations, more complex methods must be used to model an equivalent of a cylinder-shaped hole. An anisotropic material model might be necessary in this case or the need to sacrifice some accuracy and prioritize deformations in one direction.

Over all simplifications can accumulate and together cause a significant error in a model. Since this model was possible to verify against the actual deflection data the errors can be assumed to be small. In addition, the errors will present in all comparisons why the model is appropriate and accurate to use in weld evaluation.
6.2 Weld design WM

Welds are difficult to model to create realistic welds, since they most times are subject to imperfections. These are hard to include in a FEA analysis where material properties are perfect. In real applications welds often fail due to impurities or mistakes when welding as low fusion into the parent material. Welds are therefore modelled without any fusion beyond the weld root, which results in all stresses being transferred through the weld joints only and without fusion of the parent material. This is a rough estimation which in most cases will result in conservative stress values in the calculation compared to reality. This can accordingly point to a trustworthy result in this evaluation.

In many applications it can be of importance to also investigate welds for fatigue. In this thesis the effect of vibrations are neglected, but this could be an area of further work.

In addition, only transverse fillet welds are evaluated which leave room for further evaluation of other welds in the headbox for a complete evaluation.

For further analysis an overall review can be made of the structural integrity of the headbox. This is desired due to reduction in weld dimension is preferably followed by a structural optimization of the headbox structure.
7 Conclusions

7.1 Simplifications CM

The further simplifications were successful reaching a result within 0.6% of the original deflection.

The original deflection is still compared to a simplified deflection, but how the result differ from actual case is best to measure a headbox during use and compare actual data to computational results.

Looking at the time savings for the FEM model the calculation time is reduced by over 73% and the error margin about 0.6%. The real error dominated by material defects and other errors will be much larger. Therefore, a higher error margin might be allowable to further increase the calculation time. For example, instead of using an optimization method to achieve an equivalent YM, the achieved result can be plotted and linearized to be able to distinguish a rough estimation of the YM quickly without computation.
7.2 Weld design WM

The results from the weld evaluation results in a great opportunity to reduce weld dimensions. The original stresses evaluated to be low compared to the allowed stresses both in welds and the surrounding material. The extreme case shows that a reduction to a throat of just 3mm is applicable when accounting for structural strength using static loading only, which was the loading requirements for the solution. Though the surrounding material, which also contain low stresses show a great opportunity for reduction in dimensions too. While designing for low deflections of the lip the allowable strains has been so low that the design has been limited to strain rather than yield stress in welds. This doesn’t have to result in low stresses over all in the headbox though.

If more rigidity can be supplied by moving material thickness from the central sections of the headbox towards the upper and lower boundaries a higher stiffness should be possible to obtain. With an overall reduction in used material there should be room for further savings in costs. The total reduction in cost due to optimized welds can depend on a few parameters. If the number of passes can be reduced to a single pass from three when building a thicker fillet weld, the application cost can be cut to a third. Also, since LDX 2101 require an inter pass temperature to be below 150°C there might also be possible savings in cooling time.
8 References


