Finite element analysis of winding sequence for cable drums

Master's thesis in Mechanical Engineering

ROBIN THURESSON
SIMON VIKSTRÖM

ISRN-nummer: LIU-IEI-TEK-A–15/02305—SE
Department of Management and Engineering
Division of Solid Mechanics
LINKÖPING UNIVERSITY
Linköping, Sweden 2015
Finite element analysis of winding sequence for cable drums

Robin Thuresson
Simon Vikström
Finite element analysis of winding sequence for cable drums

ROBIN THURESSON
SIMON VIKSTRÖM

© ROBIN THURESSON & SIMON VIKSTRÖM, 2015.

Supervisor: Daniel Leidermark, Linköping University
Examiner: Kjell Simonsson, Linköping University

Master’s Thesis 2015
Department of Management and Engineering
Division of Solid Mechanics
Linköping University

Cover: Cable drum filled with cable after a complete simulation of the winding sequence.

Linköping, Sweden 2015
Abstract

This project concerns large cable drums made of steel. Cable drums are cylindrical structures used to transport different kinds of cables or wires. The ones in focus in this work are used to transport very stiff and heavy cables to offshore sites in the oil industry. To reduce weight in structures is always of great interest, and for cable drums, that are designed according to standards, there is much that can be improved.

The aim of this project is to predict the loads acting on a drum during the winding of cables. This was accomplished by performing explicit finite element simulations of the winding sequence. Modeling and meshing were done using LS-PrePost and ANSA, and the winding was simulated using LS-DYNA. LS-PrePost was also used to analyze the results, together with mETApost.

A number of simplifications and delimitations have been made in order to make the simulation possible within the time frame of the project. Simplified models of both the cable and the drum have been used. Winding velocities in real life are really slow, so in order to reduce simulation times, the winding velocity had to be increased while still avoiding dynamic effects. From this, contact forces between the drum and the cable could be obtained. The contact forces were investigated with respect to different friction values between the cable and the drum, for the cable itself and also for different cable stiffnesses.

The results obtained from the simulations provide contact force distributions for different parts of the drum. Much fewer rows and layers of cable were winched than in real life, and it is therefore hard to see any clear trends in the obtained forces. However, one important result of the contact forces, is that a previously used assumption, namely that the pretension of the cable only applies loads to a small fraction of the drum, was a bit too conservative. According to the simulations it actually gets distributed over the whole drum.

It is hard to draw any conclusions from the obtained contact forces, due to the small size of the simulations. The conclusion one can draw from this report, is that it is fully possible to use finite element tools to simulate the winding sequence of cable drums and this report presents a methodology on how to achieve this. LS-DYNA handles the contact definitions in an adequate way and is, according to the authors, a strong candidate for future work in this area.
Preface

This master’s thesis work is the final part of the Master of Science program in Mechanical Engineering at Linköping University. The project was performed as a collaboration between Alten Sweden AB, Svensson Group and Linköping University. It was carried out during the spring of 2015 at Alten Sweden AB in Gothenburg, Sweden.

The project has been very rewarding and has given us the knowledge and experience to continue into the working life as engineers.

We want to thank our supervisors Daniel Leidermark (Linköping University) and Mikael Almquist (Alten) for all the support and help during the project. Special thanks to Svensson Group in Falkenberg for their interest in this thesis and for contributing with information vital for the success of the project. In the early stages of the project, an introductory course in LS-DYNA was carried out at DYNAmore Nordic AB in Linköping. We therefore want to thank DYNAmore Nordic AB and Jimmy Forsberg for the course and also for the licenses necessary to complete the project. Finally we want to thank Mohsen, Sami, Maham and Christian from Chalmers for keeping up with a good spirit during the project.

Linköping 2015

Robin Thuresson
Simon Vikström
# Contents

1 Introduction ................................................................. 1  
1.1 Background ............................................................... 1  
1.2 Purpose and goals ....................................................... 2  
1.3 Winding sequence ....................................................... 3  
1.4 Procedure ................................................................. 4  
1.5 Delimitations ............................................................ 4  
1.6 Hardware and software ............................................... 5  
1.7 Other considerations .................................................. 5  

2 Theory ........................................................................... 7  
2.1 LS-DYNA ....................................................................... 7  
2.1.1 Consistent units ......................................................... 8  
2.2 Explicit method ........................................................... 8  
2.3 Implicit method ........................................................... 11  
2.4 Time step ..................................................................... 11  
2.4.1 Conventional mass scaling ......................................... 12  
2.4.2 Selective mass scaling ................................................. 12  
2.5 Contacts ........................................................................ 12  
2.5.1 Kinematic constraint method ...................................... 12  
2.5.2 Penalty method ........................................................ 12  
2.5.3 Friction ..................................................................... 15  
2.5.4 Contact energy ........................................................ 16  
2.6 Elements ...................................................................... 17  
2.6.1 Shells ...................................................................... 18  
2.6.2 Solids ...................................................................... 19  
2.6.3 Hourglass ................................................................. 20  

3 Method ............................................................................. 21  
3.1 Model set up ................................................................. 21  
3.2 Mesh ............................................................................ 24  
3.2.1 Cable .................................................................... 24  
3.2.2 Drum ..................................................................... 25  
3.2.3 Guides .................................................................... 26  
3.2.4 Mesh convergence study .......................................... 26
3.3 Loading rate .................................................. 27
3.4 Input and solution control ........................................ 28
  3.4.1 BOUNDARY_PRESCRIBED_MOTION_RIGID .................. 28
  3.4.2 CONTACT .................................................. 29
  3.4.3 CONTROL .................................................. 31
  3.4.4 DATABASE .................................................. 31
  3.4.5 LOAD .................................................... 33
  3.4.6 MAT_ELASTIC ............................................. 33
  3.4.7 MAT_RIGID .............................................. 34
  3.4.8 RIGIDWALL_PLANARFINITE .................. 34
  3.4.9 SECTION_SHELL ........................................ 35
  3.4.10 SECTION_SOLID ........................................ 35
3.5 Contact forces ............................................... 36
  3.5.1 Force distributions ...................................... 36
  3.5.2 Parameter study ........................................ 36

4 Results and discussion ......................................... 37
  4.1 Critical time step ......................................... 37
  4.2 Contact energy ............................................ 37
  4.3 Visual results ............................................. 39
  4.4 Contact forces ............................................. 41
    4.4.1 Distribution in circumferential direction .......... 41
    4.4.2 Distribution in axial direction ................... 42
    4.4.3 Distribution on flanges ............................ 43
    4.4.4 Parameter study ..................................... 44
  4.5 Model set up ............................................... 45
  4.6 Mesh ........................................................ 45
  4.7 Loading rate ............................................... 45

5 Conclusions and future work .................................. 47
Chapter 1

Introduction

The purpose of this chapter is to give the reader background information on why the project was performed and what the main objectives are. Main focus of this chapter is to define Svensson Group’s and Alten Sweden AB’s needs and expectations.

1.1 Background

In the oil industry, offshore plants require a lot of different things to be transported from the mainland, like electrical devices, tools and structural parts. One of these things are different kinds of cables, which are transported using cable drums. A cable drum is a cylindrical structure with flanges on each side. They come in many different sizes and materials, usually plastic, wood, plywood or steel. This project concerns large steel drums, produced and constructed by Svensson Group, usually with a spoke rim on each side instead of flat flanges. These are normally used to transport very stiff and heavy cables, typically electrical cables consisting of a copper core with a plastic housing. Another is the umbilical cable, which is a multipurpose cable that can supply everything from electric and hydraulic power to fiber optics. These cables are much more complex and the content can vary a lot from cable to cable. An example of a cross section of an umbilical cable can be found in Figure 1.1.

\[Figure 1.1: \text{Example of a cross section of an umbilical cable.} \quad \copyright \text{Copyright Nexans}\]
CHAPTER 1. INTRODUCTION

A cable can have a weight of up to 100 kg/m, which can lead to a total weight of up to 450 tonnes for a filled cable drum. An empty and a filled cable drum in steel can be seen in Figure 1.2.

Figure 1.2: Cable drums in steel.

To reduce the weight of the cable drums while still maintaining stiffness and durability is of great interest for Svensson Group. These cable drums are very large and designed according to standards and can with further analyzes be designed in a less conservative way. Simplified calculations have previously been done by Svensson Group, with consulting help from Alten Sweden AB, to predict the forces acting on the drum. It is therefore of great interest to further investigate this using more advanced methods. Winding a cable onto a cable drum may appear rather straight forward but it is far more challenging if you aim to accurately predict residual forces, which add up from weight, friction, winding tension and cable characteristics.

1.2 Purpose and goals

The purpose of this thesis work is to see how a cable behaves when being winched on to a cable drum and what loads it subjects the cable drum to. This leads to the main goal, which is to investigate the possibility of performing an explicit finite element analysis of the winding sequence of a cable onto a cable drum. Since the winding sequence in real life is very slow, the simulation can be considered quasi static and dynamic effects needs to be minimized. To be able to see the magnitude and behavior of the forces acting on the drum, contact forces between the cable and drum will be monitored. The outcome will give a better understanding of the forces working on the drum. A parameter study will be made to see how different cable characteristics affect these forces. The project is limited to only investigating the loads acting on the drum, and
not how they can be used in future work to dimension cable drums. How the winding sequence works will be presented in the section below.

1.3 Winding sequence

The winding sequence will be presented according to the different steps in Figure 1.3, which are also described below.

1. To wind the cable, the drum is set to rotate, which can be done in different ways.
   (a) The drum is connected at its center axis where a moment is applied.
   (b) The drum is placed on two rotating cylindrical supports.

2. The cable is connected to the drum by a locking mechanism.

3. The cable passes through two caterpillar bands that holds the cable in place while applying a pretension load to it, necessary for counteracting the moment caused by the stiffness of the cable during the winding sequence.

4. A guide is used to direct the cable into the correct place on the drum. This guide moves back and forth to wind the rows of cable in each layer.

*Figure 1.3: The winding sequence for cable drums.*
CHAPTER 1. INTRODUCTION

When referring to the winding sequence, rows and layers will be mentioned in the report. One row is what is obtained when the drum has made one revolution, and one layer is accomplished when the drum has been filled from flange to flange.

1.4 Procedure

The procedure adapted in this project can be divided into the following steps:

- Build and mesh the model using a state of the art meshing software.
- Define material properties and element formulations.
- Apply necessary boundary conditions, contact conditions and loads.
- Find a good winding velocity to avoid large dynamic effects.
- Simulate a full winding sequence.

1.5 Delimitations

In order to reach the goals within the given time frame, the following delimitations were made.

- A model of the actual drum will not be analyzed in this project, instead a simplified rigid drum will be used. This simplification will ignore the behavior caused by deformations of the drum. The drum will be modeled using flat flanges on the sides instead of spoke rims.

- The cable has a very complex structure, which can take a considerable amount of time to simulate if modeled accordingly. Therefore, the cable will be modeled using solid elements, using only one material. An exact material model for the cable is not possible to use due to this delimitation, instead the whole cable will be considered linear elastic. This simplification will probably only have minor influence on the final results since contact forces produced by the cable are of interest and not the stresses in the cable itself.

- The pretension load to the cable won’t be applied using caterpillar bands, as that would give a large increase in contact definitions and would increase the simulation times significantly. Instead a static load will be applied at the end of the cable.
1.6 Hardware and software

In the project the following hardware and software have been used in the simulations.

Hardware:

- Processors: 2 x Intel®Core™i7-2630QM CPU @ 2.00 GHz (2 x 4 cores)
- RAM: 16 GB
- Hard-drive: WDC WD7500BPKT-60PK4 SCSI Disk Device 700 GB

Software:

- LS-DYNA SMP Single Precision version R7.1.1 [1]
- LS-PrePost version 4.1 [2]
- ANSA version 15.2.2 [3]
- mETApot version 15.2.2 [4]

1.7 Other considerations

This project is associated with the offshore industry that deals with oil and gas. When burning such substances, carbon dioxide is released that contributes to the greenhouse effect, which is a big problem in the world. In today’s society, oil and gas plays a very important role since it is one of the main sources of fuel for both heating and transportation purposes. The oil industry is also one of the biggest industries and employs a large quantity of people. However, since the oil industry is a dangerous industry and accidents need to be avoided at all costs, a better understanding of the loads acting on a cable drum can lead to improved designs and a safer work environment. No ethical or gender related questions are related to this project.
Chapter 2

Theory

This chapter will present the theory needed to fully understand the methods used during this project. When simulating the winding sequence, the finite element program LS-DYNA was used, which is mainly used in simulations with large deformations and high speeds, such as crash simulations, even though it also contains all features of a general multi-purpose finite element code. It was chosen in this project since it handles complex contacts (see section 2.5) in a good way, which is of great importance for this type of simulation. How LS-DYNA works and the theory behind it, is presented below. The contents are mostly gathered from the LS-DYNA Theory Manual.

2.1 LS-DYNA

LS-DYNA is a general finite element program developed by Livermore Software Technology Corporation (LSTC). The input is controlled using keywords, which are needed for all ingoing definitions and parameters in a model, e.g. element types, materials, loads etc. In Figure 2.1 an example keyword card is shown. There are four rows that defines a keyword:

Variable – States which variable that is to be defined. (e.g. NSID = Node set ID and A1 is some kind of parameter)

Type – Which type of input that is expected from the user. (I = Integer and F = Float)

Default – Shows the default value of the parameter, which is chosen if the Type field is unchanged.

Remarks – Says if there are any additional remarks to the keyword.
LS-DYNA manages to handle both explicit and implicit time step schemes. It is normally used to analyze highly nonlinear behaviors of structures, involving complex contact conditions, large deformations and nonlinear material behaviors. Another typical use is to analyze dynamic problems, where the responses are affected by mass inertia and damping. Both these kinds of problems are usually solved with the explicit solver, since the implicit solver might experience convergence problems. When analyzing slow processes, where there are no effects from mass inertia and damping, the implicit solver can be used.

2.1.1 Consistent units

The units have to be consistent in LS-DYNA to fulfill Newton’s second law \( F = ma \). This implies that the user has to keep control over which units that should be used and thereby set the magnitude of quantities to the correct size. Examples of different unit systems are stated in Table 2.1.

<table>
<thead>
<tr>
<th>Variable</th>
<th>NSID</th>
<th>PSID</th>
<th>A1</th>
<th>A2</th>
<th>A3</th>
<th>KAT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>I</td>
<td>I</td>
<td>F</td>
<td>F</td>
<td>F</td>
<td>I</td>
</tr>
<tr>
<td>Default</td>
<td>none</td>
<td>none</td>
<td>1.0</td>
<td>1.0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>Remarks</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 2.1: Examples of unit systems.

<table>
<thead>
<tr>
<th>System</th>
<th>S1</th>
<th>S2</th>
<th>S3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Force</td>
<td>N</td>
<td>N</td>
<td>kN</td>
</tr>
<tr>
<td>Mass</td>
<td>kg</td>
<td>tonne</td>
<td>kg</td>
</tr>
<tr>
<td>Length</td>
<td>m</td>
<td>mm</td>
<td>mm</td>
</tr>
<tr>
<td>Time</td>
<td>s</td>
<td>s</td>
<td>ms</td>
</tr>
</tbody>
</table>

2.2 Explicit method

As mentioned earlier, nonlinear dynamic problems are usually solved using explicit time integration, due to the avoidance of convergence problems. The price one has to pay is that the explicit method is only conditionally stable and often requires small time steps, which have to
be smaller than the time it takes for a sound wave to travel through an element (see section 2.4 [6]. Due to this small time step requirement, the method is perfect for high speed dynamic problems, like impact and shock simulations.

To describe the behavior of a physical system, with displacements as a function of time, the equation of motion for a nonlinear system discretized in space is given by:

\[ M\ddot{U} + C\dot{U} + R_{int}(U) = R_{ext} \]  

(2.1)

\( M \) – Mass matrix
\( C \) – Damping matrix
\( R_{int} \) – Internal force vector \((R_{int} = KU \) for linear elastic system \)
\( R_{ext} \) – External force vector
\( U \) – Global nodal displacement vector
\( \dot{U} \) – Global nodal velocity vector
\( \ddot{U} \) – Global nodal acceleration vector

In order to solve this system of non-linear ordinary differential equations, LS-DYNA uses the explicit central difference method.

**Central Difference Method**

In the central difference method, the state variables are functions of already known variables.

\[ X^{(n+1)} = f(X^n, X^{(n-1)}, ..., t^{(n+1)}, t^n, ...) \]  

(2.2)

where \( X \) is the state variable and \( t \) represents the time. Consider a smooth function of the displacement \( U \). One wants to find the first and second derivative of this function. The Taylor expansion series can be written, with a given time step \( \Delta t \). [7]

\[ U(t + \Delta t) = U(t) + \dot{U}(t)\Delta t + \frac{\ddot{U}(t)}{2} (\Delta t)^2 + \mathcal{O}(\Delta t^3) \]  

(2.3)

\[ U(t - \Delta t) = U(t) - \dot{U}(t)\Delta t + \frac{\ddot{U}(t)}{2} (\Delta t)^2 + \mathcal{O}(\Delta t^3) \]  

(2.4)

By subtracting Equation (2.4) from (2.3) the central difference formula for the first derivative is obtained.

\[ \dot{U}(t) = \frac{U(t + \Delta t) - U(t - \Delta t)}{2\Delta t} \]  

(2.5)

By adding Equation (2.3) and (2.4) the central difference formula for the second derivative is obtained (the term \( \mathcal{O}(\Delta t^3) \) is considered very small and is therefore neglected).

\[ \ddot{U}(t) = \frac{U(t + \Delta t) - 2U(t) + U(t - \Delta t)}{(\Delta t)^2} \]  

(2.6)
Equation 2.5 and 2.6 can be rewritten as

\[ \ddot{U}^n = \frac{1}{2\Delta t} \left( U^{(n+1)} - U^{(n-1)} \right) \]  
(2.7)

\[ \ddot{U}^n = \frac{1}{\Delta t} \left( \dot{U}^{(n+\frac{1}{2})} - \dot{U}^{(n-\frac{1}{2})} \right) = \frac{1}{(\Delta t)^2} \left( U^{(n+1)} - 2U^n + U^{(n-1)} \right) \]  
(2.8)

where \( n \) denotes the time step counter. The central difference algorithm is visualized in Figure 2.2.

![Central difference method, used in LS-DYNA.](Image)

From Equation 2.1, the semi-discrete equation of motion at time \( n \) is obtained.

\[ \ddot{U}^n = M^{-1} \left( R_{\text{ext}}^n - C\dot{U}^n - R_{\text{int}}^n \right) \]  
(2.9)

Using Equations 2.7 and 2.8, an expression for the displacement at time \( n+1 \) can be formulated as

\[ U^{(n+1)} = U^n + \dot{U}^{(n+1/2)} \Delta t \]  
(2.10)

where

\[ \dot{U}^{(n+1/2)} = \dot{U}^{(n-1/2)} + \ddot{U}^n \Delta t \]  
(2.11)

The mass matrix is generally lumped, i.e. only non-zero terms in the diagonal, which improves the efficiency of the method. The current state variable \( X^{(n+1)} \) is then updated by adding the initial geometry \( X^0 \) to the calculated displacement vector at time \( n+1 \).

\[ X^{(n+1)} = X^0 + U^{(n+1)} \]  
(2.12)
CHAPTER 2. THEORY

2.3 Implicit method

The implicit solver is best used when analyzing slow processes with low frequency content. It solves an equation system at time $n + 1$, which is a function of unknown and already known variables. \[ 0 = f(X^{(n+1)}, X^n, X^{(n-1)}, ..., t^{(n+1)}, t^n, ... ) \] (2.13)

The implicit method have much fewer time steps than the explicit method, which is an advantage. On the other hand, the cost of every time step is unknown. This is due to variations in the convergence behavior of the equilibrium equations, which have to be fulfilled. This can vary for different problems. One computationally expensive operation in the implicit method is to calculate the inverse of the stiffness matrix, which will be done several times for each time/load step. The method is unconditionally stable for linear problems, which means that there is no limit to the size of the time increment. For nonlinear problems, the time step has to be small due to accuracy considerations. An inaccurate solution will probably not converge. \[ 6 \]

2.4 Time step

When using explicit time integration, LS-DYNA chooses the time step between each iteration using a critical time step expression called the Courant criterion. A time step is calculated for every deformable element in the model, where the smallest time step is chosen as the current time step in the iteration. The expression varies depending on if solid, shell or beam elements are used. In Equation 2.14, the general idea is presented.

\[ \Delta t_e \approx \frac{L_e}{c} \] (2.14)

\[ \Delta t_c = \min(\Delta t_e) \] (2.15)

where $\Delta t_e$ contains a time step for every deformable element in the model, $L_e$ is some characteristic length of an element, $c$ is the speed of sound in the material, which is the velocity in which stresses and strains move through an element, and where finally $\Delta t_c$ is the chosen critical time step for the current iteration. Basically, it is required to have a time step that is shorter than the time it takes for a pulse to move through an element in the model. The speed of sound in the material can be expressed using the element’s material properties in the following way.

\[ c \approx \sqrt{\frac{E}{\rho}} \] (2.16)

where $\rho$ is the density and where $E$ is Young’s modulus. By studying this expression, one can see that by modifying certain material properties in a model, the time step can be increased or decreased. In most cases an increase in time step is wanted in order to avoid long simulation times. A regular approach to this is to use mass scaling, i.e. increase or decrease the mass of the parts in a model to change the explicit time step. There are three ways to accomplish this. The first is simply to change the density of the material, which is not recommended since it in some cases require a large increase in mass, in order to get a noticeable change in the time step.
The other two are conventional and selective mass scaling.  

2.4.1 Conventional mass scaling

Conventional mass scaling is a built in function which only adds mass to those elements that affect the current time step. This is a much better way than changing the density since it only requires adding a small amount of mass in critical areas, instead of adding mass in an entire part in a model.  

2.4.2 Selective mass scaling

The other method is selective mass scaling, which only adds mass to segments that have a high eigenfrequency, while leaving low eigenfrequencies unaffected. This has the same effect on increasing the critical time step, but is a much safer and more stable approach to the problem. A drawback is that it becomes computationally more intensive to solve the force equilibrium ($F = ma$), since selective mass scaling creates a non-diagonal mass matrix. Therefore, it should only be added to critical areas that require a fine mesh.  

2.5 Contacts

Problems can occur when structural domains come into contact during a simulation. By making contact definitions, the domains will interact in different ways instead of just passing through each other. Since one of the main problems in this project was to choose appropriate contact definitions in the model, a theoretical study of these was made. In LS-DYNA, three treatment types for contacts are implemented; the kinematic constraint method, the penalty method and the distributed parameter method. In this project, the first two have been studied and will be explained below, the distributed parameter method is a two dimensional method and is therefore not of interest here.  

2.5.1 Kinematic constraint method

This type of contact sets a kinematic constraint between a master and a slave, where the master and the slave can be defined as segments, nodes or whole parts. It can be used to attach two parts together since the slave is constrained to always follow the master. In this contact type, the global degrees of freedom of the master nodes are coupled with the slave nodes by using a lumped mass formulation. This is done in order to keep the efficiency of the simulation.  

2.5.2 Penalty method

The penalty method is used in contacts to prevent penetrations of a slave through a master. This is accomplished by applying interface springs between penetrating slave nodes and corresponding master segments, see Figure 2.3. Same as in the kinematic constraint method, the slave and the master are defined as segments, nodes or whole parts.
When using penalty based contacts, a critical contact time step will be calculated.

\[ \Delta t_{\text{contact}} = \sqrt{\frac{4m_sm_m}{k_c(m_s + m_m)}} \]  

(2.17)

where \( m_s \) and \( m_m \) are the nodal masses for the slave and the master and where \( k_c \) is the contact interface spring stiffness, which is calculated differently depending on which penalty formulation that is used. If this contact time step is lower than the actual time step explained in section 2.4, there is a risk for contact instabilities. Some possible actions to avoid this can be to decrease either the critical time step or the contact stiffness. [8]

Three different implementations of the penalty algorithm exist in LS-DYNA. These are the standard, the soft constraint and the segment-based penalty formulation, which are introduced below.

**Standard formulation**

In the standard formulation, the contacts work as linear springs that act between the master and the slave. The distances, in the normal directions of the elements, between the master and the slave are measured. When the obtained values becomes negative, penetration has occurred. Then a force is applied to the penetrating slave nodes according to Equation 2.18.

\[ f_s = -lk_cn_c \]  

(2.18)

Here \( f_s \) is the applied force, \( l \) the length the slave node has penetrated, \( k_c \) the contact interface stiffness and \( n_c \) the normal direction in which the force is applied. These applied interface forces are the forces that will be analyzed during the winding sequence. When using shell elements, the penetration length \( l \) is measured with the shell thickness included, if no other contact thickness is defined. This shell thickness is explained further in subsection 2.6.1. The contact stiffness varies depending on mesh density, material properties and parameter settings in the contact definition according to Equation 2.19.

\[ k = \frac{f_{sc}K_cA_c^2}{V_c} \]  

(2.19)
where $K_c$ is the bulk modulus, $A_c$ the element face area, $V_c$ the element volume and $f_{sc}$ a scale factor for the contact stiffness. As default, this is calculated for both master and slave elements, where the minimum stiffness is used for calculations.

$$k_{c,\text{standard}} = \min(k_{\text{slave}}, k_{\text{master}})$$  \hspace{1cm} (2.20)

**Soft constraint penalty formulation**

When using the standard formulation in contacts, where very soft materials come into contact with stiff materials, large penetrations can occur. This is due to that the minimum contact stiffness between the master and the slave is the one used and will result in a very soft contact. Therefore a soft formulation (Soft constraint penalty formulation) has been implemented. Here an additional stiffness is calculated based on the Courant criterion as follows:

$$k_{c,\text{soft}}(t) = 0.5 SOFSCL m^* \left( \frac{1}{\Delta t_c(t)} \right)$$  \hspace{1cm} (2.21)

where $k_{c,\text{soft}}$ is the soft contact stiffness, $SOFSCL$ is a scale factor for the soft formulation, $m^*$ is a function depending on the mass of the slave and master nodes and $\Delta t_c(t)$ is the current critical time step. This contact stiffness is then compared to the one calculated in the standard formulation and the maximum of these two is chosen as the working contact stiffness.

$$k_c = \max(k_{c,\text{standard}}, k_{c,\text{soft}})$$  \hspace{1cm} (2.22)

**Segment-based penalty formulation**

The segment-based penalty formulation resembles the soft formulation in many ways, except for some small differences. The segment-based contact stiffness is calculated in the following way.

$$k_{c,\text{segment}}(t) = 0.5 SLSFAC \begin{cases} SFS & \text{or} \\ SFM & \end{cases} \left( \frac{m_1 m_2}{m_1 + m_2} \right) \left( \frac{1}{\Delta t_c(t)} \right)^2$$  \hspace{1cm} (2.23)

where $k_{c,\text{segment}}$ is the segment-based contact stiffness, $SLSFAC$ is a scale factor for the segment based formulation and $SFS$ and $SFM$ are scale factors for the slave and the master penalty stiffness respectively. The masses $m_1$ and $m_2$ are, instead of nodal masses as for the soft formulation, segment masses of the slave and master segments. Another difference compared to the soft formulation is that a new stiffness is only calculated if the current critical time step $\Delta t_c(t)$ changes more than 5%. In the same way as for the soft formulation, the maximum of the standard and the segment-based contact stiffness is chosen.

$$k_c = \max(k_{c,\text{standard}}, k_{c,\text{segment}})$$  \hspace{1cm} (2.24)
2.5.3 Friction

In the contact definitions, one can choose to define different friction coefficients between a slave and a master. LS-DYNA’s friction definition is based on Coulomb’s law of friction, where two friction coefficients are defined, a static friction coefficient \( \mu_s \) and a dynamic friction coefficient \( \mu_d \). The static friction is valid for surfaces not sliding against each other, while the dynamic friction applies a friction load when sliding occurs. To get a smooth transition between static and dynamic friction, an exponential interpolation function is used, giving the following resultant friction coefficient.

\[
\mu = \mu_d + (\mu_s - \mu_d)e^{-c|v|} \tag{2.25}
\]

where \( c \) is a decay constant, which dictates how fast the transition is to be, and where \( v \) is the relative velocity between a slave node and a master segment. This velocity is calculated from the incremental movement of a node in contact \( \Delta e \) and the current time step \( \Delta t_c \).

\[
v = \frac{\Delta e}{\Delta t_c} \tag{2.26}
\]

The frictional forces in the contacts are calculated iteratively, where the current friction forces are based on the previous ones as follows.

\[
f^* = f^n_{\text{Coulomb}} - k_c \Delta e \tag{2.27}
\]

where \( f^n_{\text{Coulomb}} \) is the frictional force at time \( n \) and \( k_c \) the contact interface stiffness. \( f^* \) is only a trial force and needs to be compared with the frictional yield force \( F_y \) presented below, where \( f_n \) is the normal force in the contact.

\[
F_y = \mu |f_n| \tag{2.28}
\]

This is the maximum friction force allowed. Using this yield force, a final expression for the frictional force at time \( n + 1 \) can be obtained.

\[
f^{n+1}_{\text{Coulomb}} = \begin{cases} 
  f^* & \text{if } |f^*| \leq F_y \\
  \frac{f^* f_n}{|f_n|} & \text{if } |f^*| > F_y 
\end{cases} \tag{2.29}
\]

Due to that Coulomb friction can create very high interface shear stresses in contacts, another bound has to be set on the maximum allowed friction force. To avoid interface shear stresses that are higher than the yield limit of the materials in the contact, the frictional force is limited to:

\[
f^{n+1} = \min(f^{n+1}_{\text{Coulomb}}, \kappa A_{\text{master}}) \tag{2.30}
\]

where \( A_{\text{master}} \) is the master segments area and \( \kappa \) is the viscous friction coefficient, which is set to a fraction of the master side’s yield limit. A suggested value of \( \kappa \) is \( \frac{\sigma_y}{\sqrt{3}} \), where \( \sigma_y \) is the master side’s yield limit.
2.5.4 Contact energy

When using contacts, contact energy is obtained in the master side and the slave side. The master and slave energies are normally almost equal in magnitude but opposite in sign (master side is negative and slave side is positive). The sum of the slave side energy and the master side energy is called net energy.\[14\]

\[
E_{\text{net}} = E_{\text{master}} + E_{\text{slave}}
\]  
(2.31)

A rule of thumb is that the net energy should be positive and less than 10 % of the internal energy in a model. The internal energy is based on the stresses \(\sigma_{ij}^e\), incremental strains \(\Delta\epsilon_{ij}^e\) and volume \(V^e\) of each element, where the product of these are summed for all six stress and strain components and added to the internal energy from the previous time step.\[15\]

\[
\frac{E_{\text{net}}}{E_{\text{internal}}} < 10\% 
\]  
(2.32)

where

\[
E_{\text{internal}}^{(n+1)} = E_{\text{internal}}^{(n)} + \text{sum}(\sigma_{ij}^e\Delta\epsilon_{ij}^eV^e), \text{ where } i,j = x, y, z 
\]  
(2.33)

Sometimes a large increase in negative contact energy can occur, which is not wanted. This can be a result of initial penetrations in the contacts or that two parts slide relative to each other. If two parts slide relative two each other in a way so that a penetrating node slides from one master segment to another, immediate penetrations are obtained, which can result in a negative contact energy. If friction is defined in a contact definition, a frictional energy is also obtained which adds up to the net energy. This frictional energy should be excluded when comparing the net energy with the internal energy.\[14\]
2.6 Elements

There are several types of element formulations in LS-DYNA. Some are computationally cheaper than others, some give more accurate solutions etc. Common for all element formulations is that the stresses and the strains are calculated at the integration points, while displacements, velocities and accelerations are evaluated at the nodes. Most of the element formulations in LS-DYNA are under integrated. This means that there is only one integration point in an element, placed at its center. The computational cost will be very low, since the material model will be called only once for each time step. On the other hand, a disadvantage with under integrated elements is that they may suffer from zero-energy modes, usually called hourglass modes, which will be explained further in subsection 2.6.3. This can be avoided using fully integrated elements, which have additional integration points. These elements are computationally more expensive and should only be used when high accuracy is needed. One under integrated and one fully integrated quadrilateral shell element is shown in Figure 2.4. Solids and shells are the only elements used in this project and will be explained in the sections below. 

![Figure 2.4: Difference between an under integrated and a fully integrated quadrilateral shell element.](image-url)
2.6.1 Shells

The geometry of a shell element is defined by a thickness $t$ and a midsurface, which can be a curved or a plane surface. In a shell element, the thickness $t$ is small compared to its other dimensions and can either be equal for all nodes or be a different thickness at each node. The element thickness is defined by an inner and an outer surface, which penetrations in contacts are measured from. They are both located at a distance $t/2$ from the midsurface. A flat shell element generally has five degrees of freedom per node, three translational and two rotational. Figure 2.5a displays this for a quadrilateral shell element and Figure 2.5b shows the definition of the inner and outer surface. [16] [17]

(a) Degrees of freedom for a quadrilateral shell element.

(b) Definition of the inner and outer surface for a thickness $t$.

Figure 2.5: Properties for a quadrilateral shell elements.

There are many shell element formulations in LS-DYNA, with different properties. The most common formulations are stated and described in Table 2.2. [18] [19]

<table>
<thead>
<tr>
<th>Type</th>
<th>Element formulation</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Hughes-Liu</td>
<td>First shell element implemented in LS-DYNA. Does not lose stiffness if elements are warped. Computationally expensive but effective for simulations with large deformations.</td>
</tr>
<tr>
<td>2</td>
<td>Belytschko-Tsay (default)</td>
<td>Most common element, both cheap and robust. Hourglass problems can occur. Can also have trouble with warping elements and should therefore not be used combined with a coarse mesh.</td>
</tr>
<tr>
<td>10</td>
<td>Belytschko-Wong-Chiang</td>
<td>Same as Belytschko-Tsay, but with a modification which makes it better to use if elements are warped, with only a slight increase in computational cost.</td>
</tr>
<tr>
<td>16</td>
<td>Fully integrated shell element</td>
<td>Similar with Belytschko-Tsay, but fully integrated. This leads to a large increase in computational time. For implicit simulations, this formulation is recommended.</td>
</tr>
</tbody>
</table>
2.6.2 Solids

Solid elements are three-dimensional finite elements, which leads to that all the six stresses, three normal and three shear, must be considered. Compared to shells, that generally have two rotational degrees of freedom per node, solid element nodes only have the three translational degrees of freedom. An advantage with solid elements is that a finite element mesh can be made to represent basically any given geometry, which is not possible with shell elements. On the other hand, models will have high computational cost and have poor performance for thin-walled structures. Three commonly used solid elements can be found in Figure 2.6.

![4-noded tetrahedron.](a) 4-noded tetrahedron. ![6-noded pentahedron.](b) 6-noded pentahedron. ![8-noded hexahedron.](c) 8-noded hexahedron.

*Figure 2.6: Commonly used solid elements.*

Similar to shell elements, there are several element formulations to choose between in LS-DYNA. Some are cheap, some are unstable for large deformations and so on. Four commonly used element formulations are stated and described in Table 2.3.

*Table 2.3: Different solid element formulations.*

<table>
<thead>
<tr>
<th>Type</th>
<th>Element formulation</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Constant stress solid element (default)</td>
<td>Efficient and accurate under integrated solid element. Can have problem with hourglass modes.</td>
</tr>
<tr>
<td>2</td>
<td>Fully integrated selectively reduced solid</td>
<td>No hourglass problems. Can become very stiff for poor aspect ratios and unstable for large deformations. Slower than type 1.</td>
</tr>
<tr>
<td>-1</td>
<td>Fully integrated selectively reduced solid intended for elements with poor aspect ratio, efficient formulation</td>
<td>Identical to type 2 with similar computational cost. Have accounted for poor aspect ratios, to reduce shear locking.</td>
</tr>
<tr>
<td>-2</td>
<td>Fully integrated selectively reduced solid intended for elements with poor aspect ratio, accurate formulation</td>
<td>Similar to type -1, but instead with an accurate formulation instead of an efficient. This includes higher computational cost.</td>
</tr>
</tbody>
</table>
2.6.3 Hourglass

As mentioned earlier, hourglass modes can occur for under integrated elements. One example of an hourglass mode for under integrated shell elements is visualized in Figure 2.7. Since the strains are calculated at the integration point, it is possible that they may appear to be zero. This includes that there will be no stresses even when the element is heavily deformed. By applying internal forces, the hourglass modes can be controlled. This can be done using several hourglass control algorithms, e.g. CONTROL_HOURGLASS. The work that will be obtained using this is called hourglass energy. There are 12 respectively 5 hourglass modes for under integrated solids and shells. [21] [22]

![Figure 2.7: An example of an hourglass mode for shell elements.](image)

A general rule of thumb is to keep the hourglass energy smaller than 5% of the total energy. [8] If higher hourglass energies are obtained there are different approaches one can take to solve this. The size of the mesh can affect the hourglass energy, therefore a mesh refinement could decrease the problem. Point loads on single nodes should also be avoided. If none of these changes solves the problem, fully integrated elements can be used. These have no hourglass modes, but the computational cost will on the other hand be larger.
Chapter 3

Method

In this chapter the choices concerning modeling and boundary conditions will be presented. Following this, important keywords and parameter settings will be introduced.

3.1 Model set up

When setting up the model, the goal was to come as close to reality as possible, while still having acceptable simulation times. The model used in this project is based on the winding sequence explained in [section 1.3](#). To reduce simulation times to an acceptable level, a number of simplifications from the real winding sequence had to be made. Instead of modeling a deformable drum with spoke rims, it was modeled as a cylinder with flat flanges on the sides, using rigid shell elements. This reduces simulation times significantly since the stresses and strains that will occur in the drum will be negligible. Due to that the only thing of interest was the contact forces between the drum and cable, this could be made without any significant changes in the results. Even though the drum is rigid, material properties have to be defined. The reason for this is that the properties are used to calculate contact stiffness. The material in the drum was set to the structural steel used by Svensson Group, more about this in [subsection 3.4.7](#).

Due to the complexity of a real cable, it was not possible to model the cable according to reality without getting extremely long simulation times. Instead the cable was modeled as a solid cylinder with a linear elastic material model. This material is a simplification of reality since a real cable can contain many different materials, this is further explained in [subsection 3.4.6](#). The end of the cable is attached to the drum with a contact definition. Since the cable was modeled as a straight cylinder positioned in space, a rigid wall was placed under it to prevent it from falling down due to gravitational loads.

Two frictionless guides were added to the model to ensure that the cable ends up in the right position. Each guide consists of two cylinders to guide the cable sideways and two cylinders to counteract the moment that the cable creates during the winding, which will try to bend the cable upwards. A pretension load was also added at the end of the cable to counteract this moment. For simplicity, the same material used for the drum was used for the guides. The complete model is shown in [Figure 3.1](#).
Figure 3.1: The complete model, where in the top right corner, the frictionless guides are displayed. The lower right corner shows how the cable is attached to the drum and where it is positioned.

The diameters used for the cable and the cable drum are based on real life dimensions used by Svensson Group. Further, the height of the flanges and the width of the drum, were dimensioned so that 4 layers of cable consisting of 6 rows each could fit within the drum. The dimensions of the cable, the drum, the cylinders used in the guides and the rigid wall are given in Table 3.1.
### Table 3.1: Dimensions of the drum, cable, cylinders used in the guides and the rigid wall.

<table>
<thead>
<tr>
<th></th>
<th>Cable</th>
<th>Drum</th>
<th>Guide</th>
<th>Wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius</td>
<td>54.25 mm</td>
<td>Inner radius 1158 mm</td>
<td>Radius 50 mm</td>
<td>Length 220 m</td>
</tr>
<tr>
<td>Length</td>
<td>220 m</td>
<td>Outer radius 1830 mm</td>
<td>Length 97.5 mm</td>
<td>Width 1 m</td>
</tr>
<tr>
<td>Width</td>
<td>666 mm</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The cylinders that support the drum and the lock that attaches the cable to the drum were not modeled, instead the following boundary conditions were imposed on the model:

- **Cable**: Free in all degrees of freedom and attached to the drum using a contact definition.
- **Drum**: Locked in all degrees of freedom except x-translation and x-rotation.
- **Guides**: Locked in all degrees of freedom.

The movement of the drum and the loads acting on the drum were applied in the following way:

- **Drum**: A constant rotational velocity about the x-axis and a stepwise translational displacement in the x-direction for each row.
- **Cable**: Gravitational load on the whole cable and a pretension load at the end of it.

For the winding sequence to work, the following contact definitions were introduced in the model:

- **Contact between rows and layers of cable.**
- **Contact between cable and drum.**
- **Contact between cable and guides.**
3.2 Mesh

There are three different parts that require a mesh in this model. These are the cable, the drum and the guides. The way the parts were meshed is presented in this section.

3.2.1 Cable

As mentioned in section 2.4, only deformable parts affect the critical time step. It is therefore the cable that determines the critical time step, since it is the only deformable part in this model. Since the characteristic length of the smallest element determines the critical time step, the cable was meshed as coarse as possible to avoid long simulation times. The goal with the mesh was to represent the shape of the cable as good as possible without making a very fine mesh. There are two things that limit the mesh of the cable, the diameter of the cable and the diameter of the cable drum. The cable was meshed with long and thin elements. This gives finer elements in the cable cross section which lead to a close to circular cable, while having longer elements in the length direction that are still fine enough to wrap tightly around the drum in the winding sequence. As mentioned in the previous section, solid elements were used to mesh the cable. Figure 3.2 shows the mesh for the cable in the length direction and in the circular cross section. Table 3.2 provides information about the elements in the cable. The cable was meshed in LS-PrePost, which is a pre-processing tool used to build finite element models.

Table 3.2: Properties of the mesh for the cable.

<table>
<thead>
<tr>
<th>Mesh properties for the cable</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of elements in circumferential direction</td>
<td>14</td>
</tr>
<tr>
<td>Number of elements in length direction</td>
<td>4660</td>
</tr>
<tr>
<td>Smallest element size</td>
<td>15 x 19 x 47.2 mm</td>
</tr>
<tr>
<td>Biggest element size</td>
<td>29 x 32 x 47.2 mm</td>
</tr>
<tr>
<td>Total number of elements</td>
<td>93160</td>
</tr>
</tbody>
</table>
3.2.2 Drum

The drum was modeled as a rigid body and will not be deformed. Shell elements were therefore used in the drum to decrease the computational time, since shell elements are cheaper than solid elements. The mesh of a rigid body that comes in contact with a deformable body should be at least as fine as for the deformable body, in order to get realistically distributed contact forces. Therefore, a quite fine mesh were used on the drum. Figure 3.3 displays the mesh of a quarter of the drum from two different views and Table 3.3 gives the mesh statistics. The drum was meshed using ANSA, which is a pre-processing tool similar to LS-PrePost.

Figure 3.3: The mesh of a quarter of the drum from two different views. The drum is symmetric and the thickness of the shell elements are not displayed.

Table 3.3: Properties of the mesh for the drum.

<table>
<thead>
<tr>
<th>Mesh properties for the drum</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of elements in circumferential direction</td>
<td>480</td>
</tr>
<tr>
<td>Number of elements in radius direction</td>
<td>48</td>
</tr>
<tr>
<td>Smallest element size</td>
<td>13.9 x 15.5 mm</td>
</tr>
<tr>
<td>Biggest element size</td>
<td>14 x 23.95 mm</td>
</tr>
<tr>
<td>Total number of elements</td>
<td>69120</td>
</tr>
</tbody>
</table>
3.2.3 Guides

The guides only have one function, which is to direct the cable into the right position. This implies that the cable is the only part that will be in contact with the guides. Since the contact forces between them are not of interest, the mesh of the guides was set arbitrarily. Figure 3.4 shows the mesh for one of the eight cylinders in the guides and Table 3.4 gives some data for the element mesh. Similar to the drum, shell elements were used in the guides, which were meshed in LS-PrePost.

![Mesh of one of the eight identical cylinders in the guides. The thickness of the shell elements are not displayed.](image)

**Figure 3.4:** The mesh of one of the eight identical cylinders in the guides. The thickness of the shell elements are not displayed.

<table>
<thead>
<tr>
<th>Mesh properties for the guide</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of elements in circumferential direction</td>
<td>14</td>
</tr>
<tr>
<td>Number of elements in length direction</td>
<td>6</td>
</tr>
<tr>
<td>Element size</td>
<td>16.25 x 22.25 mm</td>
</tr>
<tr>
<td>Total number of elements (both guides)</td>
<td>672</td>
</tr>
</tbody>
</table>

3.2.4 Mesh convergence study

The presented mesh densities were used when performing the simulations in LS-DYNA. When combining the mesh with the mentioned loads, contacts and other boundary conditions, a simulation took about 10 days with the hardware specified in section 1.6. The adopted meshes were coarse, especially for the cable. A refinement of the mesh on the cable would have been desirable, but that would have increased the simulation time significantly. Due to these long simulation times, no mesh convergence study could fit within the time frame of this project.
3.3 Loading rate

As mentioned previously, a prescribed rotational velocity was given to the drum to simulate the winding sequence. Since this velocity is very slow in real life, about 2-3 revolutions per minute ($\approx 0.2-0.3$ rad/s), it resulted in very long simulation times. Therefore, the rotational velocity had to be increased in order to save time when running the model. The goal was to get as high winding velocity as possible, while still having a stable winding sequence with negligible dynamic loads. Running a dynamic simulation at very slow loading rates, to get negligible dynamic loads, is called a quasi-static simulation. A quasi-static simulation is obtained when the ratio between the kinetic and the internal energy of a deforming material is very small (0-5%). It is often not possible to obtain a ratio this small in the beginning of a simulation, since the body will start to move before any significant deformations occur. The first time steps are therefore ignored when analysing this ratio. The kinetic energy of rigid bodies are excluded when the energy ratio between kinetic and internal energy is calculated, since these are not of interest. [24]

In this particular model, a constant rotational velocity was applied, while the cable deformed more and more. This lead to an almost constant kinetic energy while the internal energy of the cable increased almost linearly with time. Since the simulation time was very long, the ratio between the kinetic and the internal energy almost always reached a value below the wanted 5%. To decide an adequate loading rate, different velocities were tested and the results were analyzed both by monitoring the energies and by looking at graphical plots to see if the simulations were visually stable. Figure 3.5 shows the ratio between kinetic and internal energy for the cable with a rotational velocity of 4 rad/s applied to the drum. It is possible to see that the ratio will be under 5% after 3.8 seconds, at which time the drum has rotated about two revolutions. With this velocity a total simulation time of 35.4 seconds were obtained in order to finish 4 layers with 6 rows each. This leads to high kinetic energies during the first $\approx 10\%$ of the simulation. The rotational velocity of 4 rad/s was accepted as a sufficiently high velocity with respect to the simulation time, and at the same time sufficiently low with respect to the dynamic effects.

![Figure 3.5: Red curve shows the energy ratio between the kinetic and the internal energy for the cable with the drum rotating at 4 rad/s. The blue dotted line shows where the ratio is 5%.](image-url)
3.4  Input and solution control

In this section, the input and solution control will be presented in terms of keywords and their ingoing parameters. Further information about the presented keywords can be found in the LS-DYNA Keyword user’s manual \cite{9}. It is from this source where most of the motivations regarding parameter values were obtained.

3.4.1  BOUNDARY_PRESCRIBED_MOTION_RIGID

Using this keyword, it is possible to prescribe certain motions to rigid bodies. The center of mass of a rigid body is prescribed with a rotational or translational motion, according to the global or a local coordinate system. Since a rigid body is used, the geometry of it will always move relative to its center of gravity.

To be able to easily define the prescribed motion, the drum was modeled such that the x-axis of the global coordinate system intersects the mass center of the drum and is directed in the axial direction of it. Winding the cable onto the drum is accomplished by a rotational motion of the drum around the x-axis. The drum must also be able to move in the translational direction in order to get multiple rows of cable on each layer. This is accomplished by prescribing the drum to move stepwise in the x-direction. The two guides will help to direct the cable in the right position.

Load curves are defined to set the motion of the drum. The rotational velocity of the drum is ramped up during the first second of the simulation until it reaches its maximum value. For the remainder of the simulation, the rotational velocity will be constant (see Figure 3.6). Since the load curve is normalized, it has been scaled in the keyword with a value of 4 to get a maximum winding velocity of 4 rad/s.

\begin{figure}[h]
\centering
\includegraphics[width=0.8\textwidth]{load_curve.png}
\caption{(a) Full load curve. (b) Ramping at the start.}
\end{figure}

\textit{Figure 3.6:} Load curve used for the rotation of the drum.

An additional load curve is defined for the translational motion of the drum. Displacements will be applied stepwise for each row of cable (see Figure 3.7). The curve is defined for two layers and with \texttt{BIRTH} and \texttt{DEATH} commands in the keyword, it can be used for as many layers as
desired. Since the rotational velocity is ramped up, the translational motion for the first row will start a short moment later than for the other rows.

![Graph showing load curve](image)

**Figure 3.7:** Load curve used for the displacement of the drum.

### 3.4.2 CONTACT

There are four different contact definitions in the model and they are defined using three different contact keywords, stated below.

- **AUTOMATIC SINGLE SURFACE** - This contact definition is used to avoid penetrations between the rows and layers of cable (CTC). The contact is a penalty based contact explained in [subsection 2.5.2](#).

- **AUTOMATIC SURFACE TO SURFACE** - This contact definition is used twice. Once for the contact between the cable and the drum (CTD) and once for the contact between the cable and the guides (CTG). As for the previous contact definition, this is also a penalty based contact.

- **TIED NODES TO SURFACE OFFSET** - This contact definition is used to connect the end of the cable to the drum (TIED). The standard keyword, TIED_NODES_TO_SURFACE, uses the kinematic constraint formulation explained in [subsection 2.5.1](#). When the TIED_NODES_TO_SURFACE_OFFSET keyword is used, which is the case for this model, the contact definition uses a penalty based contact formulation. This is required when rigid parts are included in a tied contact. To create this tied contact, the nodes at the end of the cable are copied and added to the drum. The copied nodes are then defined to be in contact with the cable.

All contact definitions in LS-DYNA contain the same keyword parameters. [Table 3.5](#) presents modified contact keyword parameters, which are further explained below the table. There is also
a contact between the cable and the rigid wall, which is used to prevent the cable to fall due to gravity. This contact is not defined in any of these keywords. Instead a RIGID_PLANAR_FINITE keyword handles this contact, which is further explained in subsection 3.4.8.

Table 3.5: Parameters for the CONTACT keywords

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>CTC</th>
<th>CTD</th>
<th>CTG</th>
<th>TIED</th>
</tr>
</thead>
<tbody>
<tr>
<td>SSID</td>
<td>Slave part/node set ID</td>
<td>Cable</td>
<td>Cable</td>
<td>Cable</td>
<td>Drum</td>
</tr>
<tr>
<td>MSID</td>
<td>Master part/node set ID</td>
<td>0</td>
<td>Drum</td>
<td>Guide</td>
<td>Cable</td>
</tr>
<tr>
<td>MPR</td>
<td>Include the master side in NCFORC</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>FS</td>
<td>Static coefficient of friction</td>
<td>0.35</td>
<td>0.325</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>FD</td>
<td>Dynamic coefficient of friction</td>
<td>0.3</td>
<td>0.2</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>VDC</td>
<td>Viscous damping coefficient</td>
<td>30</td>
<td>30</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>SOFT</td>
<td>Soft constraint option</td>
<td>-</td>
<td>1</td>
<td>1</td>
<td>-</td>
</tr>
<tr>
<td>PENMAX</td>
<td>Maximum penetration distance</td>
<td>-</td>
<td>0.49</td>
<td>0.49</td>
<td>-</td>
</tr>
</tbody>
</table>

SSID & MSID - Here the master and slave of a contact are defined. In the single surface contact for the cable, it is only necessary to define a slave part for the cable in order to get cable to cable contacts. When using the automatic surface to surface contact definitions, the master and slave can be chosen arbitrarily. This is due to that the automatic contact definitions in LS-DYNA are two way symmetric contacts, which basically means that both the master and the slave in a contact are checked for penetrations and treated equally. The tied nodes to surface offset contact is used to connect a node set that belongs to the drum with the cable, using the node set as master and cable part as slave.

MPR - This parameter decides if the master side interface forces should be included in the *DATABASE_NCFORC file, which is explained further in subsection 3.4.4. The default value is zero, which means that the master side force is not included, while the chosen value of one means that it is included.

FS & FD - Both the static and dynamic friction coefficients are set to normally used values between the materials in contact [25]. For the tied contact, there is no need to state any friction coefficients. The contact between the cable and the guides was set to be frictionless since its sole purpose is to direct the cable into the right position.

VDC - This option is used for reducing high frequency oscillations in contact forces. The chosen value is a result of experiments that was performed in this project, where different values of the VDC parameter were used and the contact forces were analyzed. The value of 30 was obtained when the oscillations in the contact forces were reduced to an acceptable degree. The input value of 30 is given in percentage of the critical damping.

SOFT - Defines if a soft constraint formulation is to be used. There are several options to choose between and two of them are explained in subsection 2.5.2. The soft constraint penalty formulation was the one used in this model. This formulation is a good option when a soft material comes into contact with a stiff material, which is the case for the contacts between the
CHAPTER 3. METHOD

drum and guides (steel) and the cable (plastic).

**PENMAX** - This defines the maximum penetration distance. The value is given in percentage and is multiplied with the thickness of a solid element or with the sum of the slave and master shell thicknesses, depending on if solid or shell elements are used. This product is the maximum allowable penetration distance. If a slave node penetrates more than this distance, it gets released from the contact in order to avoid very high interface forces [26], which can cause instabilities. The value of 0.49 is a recommended value [8].

### 3.4.3 CONTROL

In the CONTROL keyword, one can define how long a simulation shall run and it is possible to set the time steps for the simulation.

**TIMESTEP** - In this keyword, the critical time step can be increased by mass scaling if longer time steps are of interest. As mentioned in [subsection 3.2.1], the time steps used in the simulation are set by the mesh size of the cable. Since the mesh of the cable is very consistent, mass needs to be added to a lot of elements in order to increase the time step. Initially, mass scaling was used in the model, but due to that gravity was a major factor in the simulations, mass scaling was removed. The only parameter used in this keyword was \( TSSFAC \), which was set to 0.9. This ensures that the time step always is 10\% under the critical value.

**TERMINATION** - In this keyword one chooses when the simulation should end. Termination time was set to 35.4 seconds which is the time it takes to complete the 4 layers with the current velocity.

### 3.4.4 DATABASE

The DATABASE keyword can be used to activate different outputs from the simulation that can be of interest.

**ASCII** - The DATABASE_ASCII keyword controls all the important data from a simulation regarding stresses, strains, displacements etc. The user defines which information that are of interest and decides the time interval (DT) between each output. Explanations of the used ASCII outputs are stated below. The X-, Y- and Z-forces obtained in NCFORC and RCFORC are the applied contact forces explained in [subsection 2.5.2].
CHAPTER 3. METHOD

Table 3.6: The different output files defined in the ASCII keyword.

<table>
<thead>
<tr>
<th>ASCII</th>
<th>Explanation</th>
<th>Data of interest</th>
<th>DT</th>
</tr>
</thead>
<tbody>
<tr>
<td>GLSTAT</td>
<td>Global energies</td>
<td>Kinetic-, internal-, hourglass- and total-energies for the whole model.</td>
<td>0.1</td>
</tr>
<tr>
<td>MATSUM</td>
<td>Energies for each material definition</td>
<td>Kinetic-, internal- &amp; hourglass-energy of the cable material</td>
<td>0.1</td>
</tr>
<tr>
<td>NCFORC</td>
<td>Nodal contact forces for defined master and slave nodes</td>
<td>X-, Y- &amp; Z-forces in drum nodes</td>
<td>Curve</td>
</tr>
<tr>
<td>RCFORC</td>
<td>Total contact forces for defined master or slave side</td>
<td>X-, Y- &amp; Z-forces for whole drum</td>
<td>0.1</td>
</tr>
<tr>
<td>SLEOUT</td>
<td>Contact energies for each contact definition</td>
<td>Master- &amp; Slave-energy for all contact definitions</td>
<td>0.1</td>
</tr>
</tbody>
</table>

As one can see in Table 3.6, the output for each ASCII file is obtained every 0.1 second, except for the NCFORC output which is in this work defined by a curve. This is due to that the NCFORC output gives data for each node, which will result in very large output files if activated too often. The curve defines that NCFORC should be obtained in four different intervals, one for each finished layer. After each finished layer 5 points with 0.05 seconds between them will be obtained, see Figure 3.8. This is due to that it is hard to know exactly when one layer is finished and the next one has not started.

![Figure 3.8: Output intervals for NCFORC data.](image_url)

D3PLOT - This keyword was used to obtain a graphical representation of the simulation. Using this, one could easily see if any major problems occurred during the simulation. In the simulation, one plot file was written every 0.2 seconds.
3.4.5 LOAD

This keyword was used to apply different loads to the model.

**BODY** - Gravity is a significant physical quantity acting on the model. It was applied through the keyword LOAD_BODY, where a load curve was defined to set the gravitational acceleration. The same load curve as for the rotational motion was used, see Figure 3.6. In the keyword, a scale factor was used to set the maximum gravity to 9.81 m/s².

**NODESET** - In the winding sequence, there is a pretension applied to the cable. This is to counteract the bending moment, created by the stiffness of the cable, that will work on it during the winding sequence. The pretension to the cable was obtained by applying a static load in a node set of 27 nodes at the end of the cable. Using the same load curve as for the rotational motion, the pretension could be ramped up, see Figure 3.6. A scale factor of 1817 N was used for the pretension load, which with the 27 nodes gave a total load of 49 kN or 5 tonnes. According to Simon Olsson at Svensson Group, this is a normally used pretension load during the winding sequence.

3.4.6 MAT_ELASTIC

Since a simplified cable was modeled, the material properties of it also had to be simplified. In the simulation, the contact forces between the cable and the drum are of interest and not the stresses and strains in the cable itself. Therefore a linear elastic material model was chosen as a sufficient material. Since the real cable consist of more than one material, approximated material properties had to be used. These could be calculated from the given cable properties stated in Table 3.7 [27].

<table>
<thead>
<tr>
<th>Table 3.7: Given cable properties.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass per meter</td>
</tr>
<tr>
<td>Total axial stiffness</td>
</tr>
<tr>
<td>Cable diameter</td>
</tr>
</tbody>
</table>

Mass density and Young’s modulus are not stated, instead the mass per meter \( \rho_l \), cable diameter \( D \) and total axial stiffness \( k \) are known. Using these values in the following equations, the mass density \( \rho \) and the Young’s modulus \( E \) could be approximated.

\[
\rho = \frac{\rho_l}{A} \tag{3.1}
\]

\[
E = \frac{k}{A} \tag{3.2}
\]

where

\[
A = \frac{\pi D^2}{4} \tag{3.3}
\]

Since Poisson’s ratio is hard to set to a relevant value, it was also approximated. The cable
CHAPTER 3. METHOD

was assumed to consist of plastic (polyethylene) with a Poisson’s ratio of 0.42 and copper with 0.35, from which a value in between these two was chosen. Regardless of which value that would have been selected within the range, it would probably not have had a significant impact on the result. Therefore, no deeper investigation was made concerning the value of the Poisson’s ratio. The calculated material properties for the cable are given in Table 3.8.

Table 3.8: Material properties for the cable.

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Cable</td>
<td>Mass density</td>
<td>1117 kg/m³</td>
</tr>
<tr>
<td></td>
<td>Young’s modulus</td>
<td>15.1 GPa</td>
</tr>
<tr>
<td></td>
<td>Poisson’s ratio</td>
<td>0.38</td>
</tr>
</tbody>
</table>

3.4.7 MAT_RIGID

To reduce simulation times, both the drum and guides are considered to be rigid bodies. This property is applied to the parts using this keyword. Even though the parts are rigid, material properties need to be defined in order to calculate contact stiffness. The material is set to the steel material S355J2, which is the material used by Svensson Group in their cable drums. The properties for the drum and guides are given in Table 3.9.

Table 3.9: Material properties for the drum and guides.

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Drum and</td>
<td>Mass density</td>
<td>7850 kg/m³</td>
</tr>
<tr>
<td>guide</td>
<td>Young’s modulus</td>
<td>210 GPa</td>
</tr>
<tr>
<td></td>
<td>Poisson’s ratio</td>
<td>0.3</td>
</tr>
</tbody>
</table>

Since the parts are rigid, it is possible to constrain them in any degree of freedom. This is done by constraining the mass center of the parts. The guides are constrained in all translational and rotational directions. To be able to rotate and translate the drum, no rotational constraint around the x-axis or translational constraint in the x-direction is applied. The drum is constrained in all other directions.

3.4.8 RIGIDWALL_PLANARFINITE

Before the winding starts, the whole cable is extended in the horizontal direction. To avoid that the cable falls vertically before being winched on to the drum, this keyword was used to define a rigid wall under the cable. A rigid wall is a geometrical surface without any mesh or material defined. The contact between the cable and the rigid wall is handled directly in this keyword and no additional contacts need to be defined. In the keyword, the dimensions and position of the rigid wall are stated. It is positioned right under the cable, starting from the supports.
and ends at the end of the cable. Between the nodes of the cable and the rigid wall, there is a frictionless contact definition. The planar rigid wall is the yellow surface illustrated in Figure 3.1. It was defined to be 220 meters long and 1 meter wide.

### 3.4.9 SECTION_SHELL

In this keyword, the element formulation and its ingoing parameters are chosen. For this model, one shell formulation has been used. The drum and the guides are the parts that were meshed with shell elements. The same keyword was used for both parts, where the modified and important parameters are stated in Table 3.10 and motivated below. There are several other parameters that aren’t stated in the table. For them, default values have been used.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELFORM</td>
<td>Element formulation option</td>
<td>Belytschko-Tsay</td>
</tr>
<tr>
<td>NIP</td>
<td>Number of through shell thickness integration points</td>
<td>2</td>
</tr>
<tr>
<td>T1-T4</td>
<td>Shell thickness for the four nodes in each element</td>
<td>8.0 mm</td>
</tr>
</tbody>
</table>

**ELFORM** - Belytschko-Tsay is the most economical shell formulation and also the default formulation in LS-DYNA. It is an under integrated element, so the formulation can cause hourglass problem, but since the parts are rigid, this won’t cause any problems.

**NIP** - Since all parts in the model that use shell elements are rigid, the number of through thickness integration points could be chosen arbitrarily. Two integration points were chosen since this is the default value.

**T1-T4** - The shell thickness was selected so that the dimensions would be correct in the model. Half the thickness is applied in both normal directions of the shell element. This stated thickness is also the one used in the contact definitions.

### 3.4.10 SECTION_SOLID

The purpose of this keyword is to define the element formulation for the solid elements in the model. In the previously presented keyword SECTION_SHELL, many parameters could be declared in the definition. In the SECTION_SOLID keyword, only one parameter can be modified, which is the element formulation. The cable is the only solid part in this model and therefore the only part using this keyword. The chosen element formulation is type -1, *Fully integrated S/R solid intended for elements with poor aspect ratio, efficient formulation*, which is a recommended formulation [8]. This is because the fully integrated elements, as mentioned in subsection 2.6.3 prevent hourglass modes to occur and also give more accurate results. It also handles elements with poor aspect ratio in an adequate way. A cubic element have an aspect ratio equal to one, since all sides have the same length. If the element instead is long and thin, the aspect ratio is poor, which is the case for the cable in this model.
3.5 Contact forces

In this section, it will be presented which of the contact forces that were analyzed and why they are of interest.

3.5.1 Force distributions

The force distribution in the drum in different areas are of high interest in these simulations. First of all, the distribution in the circumferential direction is of interest. This is to be able to see if the gravitational loads have a big influence on the final results. Further, the forces in the axial direction will be analyzed to see if there are any high concentrations of forces or if they are evenly distributed. This will be analyzed for each of the full layers in order to see how multiple layers change the distribution. Finally, the contact force against the flanges will be analyzed.

3.5.2 Parameter study

In previously made calculations by Svensson Group and Alten, an assumption that the pretension of the cable only works on a small fraction of the drum was made. Specifically, that it is only applied during the first $120^\circ$ of the current row that is being winched. This is a conservative approach, since it gives concentrated loads at different parts of the drum which can give buckling problems. To analyze this, the total contact forces applied by 1/14 of a row of cable, were plotted during the first layer of winding. Because the contact forces, close to the tied contact of the cable, are not very reliable, they were measured 90$^\circ$ after the start of the second row. To see how the friction and Young’s modulus of the cable affect how large the area the pretension is applied on, four additional simulations of the first layer were made:

1. Twice the initial static and dynamic friction coefficients in all contacts.
2. Half the initial static and dynamic friction coefficients in all contacts.
3. Twice the initial Young’s modulus.
4. Half the initial Young’s modulus

Initial values for friction coefficients and Young’s modulus can be found in subsection 3.4.2 and subsection 3.4.6.
Chapter 4

Results and discussion

In this chapter the obtained results will be presented and discussed. Before any contact forces were analyzed, other results needed to be investigated to be able to see if the results could be considered trustworthy. In the Theory and Method chapters of the report, a number of different conditions had to be fulfilled in order to ensure that the contacts have been treated in a stable and adequate way. These conditions will be presented and discussed, followed by the resulting contact forces. Finally the choices made and methods used regarding the set up of the model will be discussed.

4.1 Critical time step

The first thing investigated was the critical time step for contacts (see subsection 2.5.2). Since the deformations of the elements in the cable are relatively small, the time step in the simulation will almost be constant. Therefore, the critical time step was noted in the beginning of the simulation and compared with the contact time step, these are presented below.

\[
\Delta t_c = 2.53 \cdot 10^{-6} \text{ s} \\
\Delta t_{\text{contact}} = 4.163 \cdot 10^{-6} \text{ s}
\]

Since \( \Delta t_c < \Delta t_{\text{contact}} \), there should be no contact instabilities due to the size of the time step.

4.2 Contact energy

Further, to ensure the stability of the contacts, the energy ratio between the net energy in the contacts and the internal energy had to be checked (see subsection 2.5.4). The net energy obtained in the contacts is recommended to be under 10\% of the internal energy. Resulting energy ratios for all contact definitions can be seen in Figure 4.1.
As one can see, all contact definitions fulfills this condition except for the tied contact between the drum and the cable. When the pretension is applied to the cable in the beginning of the simulation, a small offset is created in the tied contact. This is due to that the used tied contact definition is penalty based and does not work as a constraint. Since the offset is created before any major deformations have occurred in the cable, it gives a peak in the energy ratio of over 20%. This could be ignored since it is only for a brief period of time in the early stages of the simulation and decreases as soon as the cable starts deforming. One way of reducing this effect could be to have a slower ramp up of the pretension load.

In the contact between the cable and the guides, negative energies are obtained. This is a problem that occurs when they slide relative to each other, as explained in subsection 2.5.4. Since the energy obtained in this contact is very small and that no contact forces between the cable and the guides are of interest, this was ignored. A solution to this could be to refine the mesh of the guides to make them represent their cylindrical shape better. Due to that the guides are very small in comparison to the rest of the model, this should not have had any major impact on the simulation times.
CHAPTER 4. RESULTS AND DISCUSSION

4.3 Visual results

In subsection 3.4.1, the motion of the drum is described. With this motion, the first layer of cable on the drum looks like in Figure 4.2, where one can see the drum from four different views. It is possible to see that the cable is in a straight vertical direction for half the drum and have an angle for the other half.

Figure 4.2: One layer of cable winched onto the drum, displayed in four different views.

The complete analysis of the winding sequence, is visually displayed in Figure 4.3a, with a cross section that can be seen in Figure 4.3b. This cross section is obtained where the cable is in a straight position without any angle.
These results are visually very good and obtained through lots of testing and experimenting with different motion curves. In real life, the winding will probably not be as perfect as in a simulation and the cable may distribute more irregularly. This should be taken into consideration when analyzing the contact forces, since they are obtained with an even cable distribution and some of the forces may be higher in a real life winding sequence.
4.4 Contact forces

In this section the contact forces for different parts of the drum with different cable characteristics will be presented.

4.4.1 Distribution in circumferential direction

To see which of the loads acting on the cable that have the biggest impact on the contact forces, between the gravity and the pretension, one can take a look in Figure 4.4. This chart shows how the contact forces are distributed over the drum in the circumferential direction. A high peak can be seen around the cable attachment when the first layer is winched. This is due to that there are very large penetrations right after the attachment and high contact forces are therefore obtained. This area continuously have the highest contact forces, but the peak becomes smaller for each winched layer. If a simulation with more layers would have been performed, the influence of the forces obtained around this area would have probably been negligible. If the extreme values around the attachment area are ignored, it is possible to see that the gravity doesn’t affect the distribution significantly. Instead it is the pretension that have the biggest impact on the contact forces, this due to that the forces underneath the drum are roughly as big as the forces on the top. If additional layers of cable are winched, so that the total mass of the cable becomes larger, this would probably change.

Figure 4.4: Distribution of contact forces around the drum depending on the number of layers that are winched (The red circle is the drum). The results are dimensionless and normalized against the highest contact force. The short green line shows where the cable is attached to the drum and the drum rotates counterclockwise.
4.4.2 Distribution in axial direction

It is interesting to see how the contact forces for each row and layer of cable are distributed in the axial direction of the drum. Figure 4.5 shows this for each of the four layers of cable. The tied contact that attaches the cable to the drum is located at row six in the chart.

![Figure 4.5: Contact forces in the axial direction of the drum for each row and layer of cable, where the different colors state how many layers that have been winched.](image)

The data are gathered from a quarter section of the drum, where the cable rows are in a straight position and on the upper half of the drum, see the section in Figure 4.6. There one can see that a new layer of cable is about to start winding and that it is positioned at row six.

![Figure 4.6: The quarter of the drum where the contact forces are gathered.](image)

In Figure 4.5 it is possible to see that the forces creates a smooth and decreasing curve for the first layer. When additional layers of cable are winched onto the drum, the resultant force against the drum increases, but the distribution also starts to peak. The contact forces at the
rows closest to the flanges (1 & 6) are small compared to the other rows. This was expected since these rows only are in contact with one cable row on the second layer, while the other rows are in contact with two cable rows of the second layer (see Figure 4.3b). For the remaining rows (2-5) it is hard to see any clear trends. After the first layer, the forces look pretty much evenly distributed, while after layer two and three larger and larger peaks of contact forces are obtained. Finally for the fourth layer it evens out again. If additional layers would have been simulated, it might have been easier to see how the cable behaves in the axial direction.

### 4.4.3 Distribution on flanges

Figure 4.7 shows how the contact forces change over the flanges for the different layers of cable. In the same way as for the contact forces in axial direction, the data are gathered from a quarter section of the drum, when the cable is in a straight position (Figure 4.6). Negative and positive values of the forces state which flange that is being observed. Negative values are for the flange to the left (low x-coord.) and positive values for the flange to the right (high x-coord.). The right flange is the one that is close to the position where the cable is attached to the drum. One can see that when only one layer is winched, the forces on the flanges are very small. The second layer then pushes the cable against the flanges and increases the contact forces. For the third and fourth layer of cable, the forces on the flanges act differently. The flange that is close to the cable attachment will lose the contact forces for the first layer. Due to that the cable attachment is unable to move sideways, the results obtained in this area are considered very unreliable, which was also mentioned in subsection 4.4.1. The flange forces on the first layer are not of big interest, since the highest forces probably will be obtained at the third layer, independently of how many layers that are winched. This is due to that the forces caused by additional layers will distribute downwards and towards the flanges. Worth mentioning is that if this is a question to be investigated more precisely, one should simulate many more layers of cable to understand how the forces act on the flanges. Four layers are not enough to be able to see any clear trend in the force distribution on the flanges.

![Figure 4.7: Contact forces on the flanges, where the different series show how many layers that have been winched. Negative values are for the left flange (low x-coord.) and positive values are for the right flange (high x-coord.).](image)
4.4.4 Parameter study

Contact forces for three of the four additional simulations explained in subsection 3.5.2, as well as the initial simulation, are displayed in Figure 4.8. The simulation with a high Young’s modulus is not displayed in the chart. This is because the cable couldn’t bend enough to be in contact with the drum, so no contact forces were obtained for this simulation.

![Figure 4.8: Contact forces over time for a small section of the drum for different values of friction coefficient \( \mu \) and Young’s modulus \( E \).](image)

If the assumption that the pretension only acts on the first 120° would be correct, then one should see a decrease in contact forces after \( \approx 3 \) seconds. The forces are not decreasing, instead they are steadily increasing for all cases throughout the winding of the first layer. No clear difference is noticed for the contact forces depending on if they are measured under or over the drum. This means that according to these results, one can assume that the pretension load gets distributed over a much larger area than previously assumed.

By comparing the different cases, one can see that an increase in friction gives an increase in radial forces on the drum. A reason could be that with high friction, no sliding between the cable and the drum occurs, meaning that the pretension gets built into the system more easily than for a case with very low friction. Further, a lower Young’s modulus allows the cable to wrap more tightly around the drum, which will result in higher radial forces. Due to that no more simulations could be fitted within the project’s time frame, no simulations with both modified friction and stiffness were performed. This would have been interesting in order to see if it would follow the same pattern. If it would, the most extreme contact forces would then be obtained with a soft cable and high friction.
CHAPTER 4. RESULTS AND DISCUSSION

4.5 Model set up

A winding sequence with only six rows and four layers of cable have been simulated. This is far less than a real winding sequence where maybe 20-30 rows and layers are winched. The simulated winding sequence is probably too short to be able to see any clear indications on how the contact forces are distributed over the drum. If the sequence would have been extended, this would have increased the simulation time significantly, which was not possible within the time frame of this project. The fact that the cable and the drum are both simplified, should not affect the results that much, since the contact forces between them are the results of interest. It is probably possible to use a model of a real drum and still be able to run a model in a reasonable time. This does not apply for the cable. As mentioned earlier, the cables are highly complex in real life and would require multiple material definitions and a very fine mesh. Therefore, a homogeneous cable seems like a reasonable simplification, although further investigation on how to make a simplified material model for this would be of great relevance.

4.6 Mesh

The simulation times for this model were about 10 days with the hardware and software mentioned in section 1.6, even though a coarse mesh has been used. Due to these long simulation times, a mesh refinement could not be fitted within the time frame of the project, since this would lead to a too computationally expensive simulation. A fine mesh is critical in order to get as true description of the contact forces as possible. If contact forces are obtained in a small amount of nodes, which is the case for coarse meshes, unwanted force concentrations that does not exist in real life can occur. It is important to discuss if it is worth performing a simulation like this, in the context of time and money. With better computational power and more program licenses, the simulation time would decrease and a finer mesh could be used, which could lead to better results. However, if one combines this with an extended simulation, discussed in section 4.5, simulation times could increase exponentially and maybe become too expensive both with respect to time and money.

4.7 Loading rate

In section 3.3 it is presented how the winding velocity is increased compared to a real life winding velocity. This was one of the major modifications made to the winding sequence in order to save simulation times. To get a simulation as close to reality as possible is what one tries to achieve when setting up a finite element model. Since the quasi static approach that has been used in this project is based on conditions and assumptions used for metal forming, it is hard to say how reliable this is. This is due to that the energy ratio will almost always go below 5% [24] since the simulation has constant velocity with a steadily increasing deformation. It was accepted that the energy ratio was over 5% during the first 10% of the simulation and it is hard to say if this really was acceptable or if it lead to significant changes in the results. Therefore it would have been interesting to make some sort or convergence study on this, similar to a mesh convergence study, to see how the contact forces are affected by an increase of decrease in winding velocity.
Chapter 5

Conclusions and future work

The winding sequence of a cable onto a cable drum has been evaluated for five different cases with different cable characteristics using LS-DYNA. To be able to get reliable results, critical time steps, the kinetic and internal energies in the cable, and contact energies in all contact definitions have been measured and analyzed. Further, the contact forces between the cable and drum have been analyzed to see how the forces from the cable distributes over the drum.

All conditions concerning critical time steps and energies were fulfilled within the given bounds. The energy ratio, depending on the internal energy of the cable and the kinetic energy caused by the winding velocity, should though be monitored carefully for this kind of model, since it almost always gets fulfilled. This is something that should be looked into further to see how the maximum velocity affects the resulting contact forces. A high peak in contact energy was obtained in the tied contact used to connect the cable to the drum, which was probably caused by the short ramp up of the loads and velocities. It is hard to say how this affected the results and due to the short duration of the simulation it was hard to motivate a longer ramp up. If simulating a longer winding sequence, a longer ramp up is recommended in order avoid high energy peaks.

When it comes to the contact forces and how they distributes over the drum, it is hard to draw any clear conclusions. One of the major sources of error is that too few rows and layers have been simulated to be able to fully rely on these results. Although an analysis of how big area the pretension affects have been made for different values of the friction coefficients and Young’s modulus. In all cases it was clear that the pretension gets distributed over the whole drum. This is a very important result, since it contradicts the old assumption that the pretension is only acting on the first 120° of the drum. This should of course be looked into further by studying additional rows and layers. It was also seen that if the cable is modeled with high friction and low Young’s modulus the contact forces increase. A recommendation is therefore to use the mentioned properties when analyzing the drum in order to obtain some sort of extreme case.

Time is one of the main factors that limit this kind of simulation. A major problem is to avoid all dynamic effects in the model and try to maintain a quasi-static simulation. There are several different approaches to decrease simulation times, where this work has focused on an increased winding velocity or to use mass scaling. Unfortunately, these modifications can only be made
up to a certain degree since both of them will increase the kinetic energy and dynamic effects will appear in the model. It would therefore be of great interest to investigate other approaches to decrease the simulation time while still obtaining reliable results.

One of the biggest questions that was asked in the beginning of the project was if this is even possible to simulate. After running these simulations and obtaining the results presented in this report, one can conclude that it is in fact possible to simulate a winding sequence. LS-DYNA handles the contact definitions in an adequate way and is, according to the authors, a strong candidate for future work in this area.
Bibliography