Specialization: Transport Engineering and Logistics

Report number: 2014.TEL.7860

Title: Modeling loaded belt conveyors in DEM and FEA

Author: A. Mejias Osuna

Title (in Dutch) Het modeleren van een beladen bandtransporteur met DEM en FEA

Assignment: computer

Confidential: no

Initiator (university): dr.ir. D. Schott, X. Liu MSc.

Supervisor: dr.ir. D. Schott, X. Liu MSc.

Date: June 30, 2014
Subject: Modeling loaded belt conveyors in DEM

Belt conveyors are widely used all around the world. The design of belt conveyors implies many different aspects. To improve the design, engineers have to investigate to find new methods to optimize these systems.

This research work consists of exploring the possibility of coupling Discrete Element Method (DEM) and Finite Element Analysis (FEA) to analyze a belt conveyor system. To realize this research, two software were used in order to achieve the goals: EDEM® on the side of DEM and ANSYS® Mechanical on the side of FEA.

As a result of the research, the feasibility of this method should be clarified. Different ways to calculate the load on the idler rolls after modeling the belt conveyor system should be presented for future research.

The professor,

Dr. ir. Dingena Schott
Summary

The design of belt conveyors represents a great challenge for engineers. One of the most important aspects to be considered is the complexity of the belt sag. New methods are being developed in order to optimize the design of belt conveyor systems. One of these possible new methods is the use of DEM to model and simulate the behavior of a loaded belt conveyor system. However, the solo employment of DEM is not enough. The complexity of the deflected belt previously mentioned makes impossible to use DEM to obtain accurate results. The software used on the DEM side (EDEM®) does not consider the deflection of the belt, and therefore the bulk material will not behave in the way it should. For this reason a coupling between EDEM® and ANSYS® has been performed.

The first section in this report is a literature review. In this literature review the first step consists providing brief background on the topic of belt conveyors. The main elements on a belt conveyor system have been described in order to allow the reader to understand better the whole report. The second step in the literature study is the explanation of the behavior of the bulk solid loaded on the belt. The transition from passive to active stress state is detailed. Some theoretical explanations with relation to the meaning of active and passive stress state are provided as well. The last stage of the literature review is the analysis of the loads on the idlers. The influence of the passive an active stress state is explained with the help of some schematic figures.

Secondly in this report, the modeling process is explained in deep detail. Several approaches had to be made in order to achieve the desired behavior of the system. The first attempt to model the belt conveyor was creating a belt where the particles moved horizontally and iterate several times modifying the geometry of the belt. The second approach was doing exactly the same but increasing the Elasticity Modulus after every analysis in order to achieve the steady state. The real steady state could not be achieved but at the end of the iterations the deflection of the belt remained at a constant value. The third try was repeating the analysis performed in the second attempt but with a static load. The last approach was to export data to ANSYS® Workbench to analyze the results. The outcome of all these analyses is the same: the same FE model cannot be used for the iterations.

The third step of this report is to analyze the feasibility of calculating the loads on the idlers rolls. Some possible solutions using EDEM® and ANSYS® are presented to calculate those loads. After considering all the drawbacks of every method, it was decided that ANSYS® was the most suitable tool to fulfill this task.

The main conclusion is that if the same FE model cannot be used, the results are not optimal. Having different FE models means that the mesh varies from one analysis to another, and therefore the steady state is no achievable. To achieve the steady state it would be necessary to import output data from ANSYS® to EDEM®. Unluckily this feature is not available yet. Until now it is only possible to export date from EDEM® to ANSYS® to apply the loads to the Finite Element model. For this reason, the completion of this research has to be postponed until the new release of EDEM® is available.
Contents

Summary .......................................................................................................................... 3
1. Introduction .................................................................................................................. 5
2. Literature review ......................................................................................................... 6
   2.1. Belt conveyors background ................................................................................... 6
   2.2. Bulk material behavior ......................................................................................... 12
   2.3. Idlers loads ............................................................................................................. 14
3. Explanation of the model: EDEM® – ANSYS® coupling ............................................. 18
   3.1. Theoretical considerations ..................................................................................... 20
   3.2. First approach (Several iterations maintaining the Elasticity Modulus unchanged) .................................................................................................................. 22
   3.3. Second approach (Several iterations changing the Elasticity Modulus) ................. 34
   3.4. Third approach (Several iterations changing the Elasticity Modulus with a static load) ........................................................................................................ 38
   3.5. Coupling between ANSYS® Workbench and ANSYS® MAPDL ......................... 43
   3.6. Comparison of results ............................................................................................ 51
   3.7. Modeling process: Summary ............................................................................... 53
4. Idlers load calculation: Possible solutions ................................................................... 55
   4.1. Calculation of the loads using EDEM® ................................................................. 55
   4.2. Calculation of the loads using ANSYS® ............................................................... 59
   4.3. Conclusions ............................................................................................................ 61
5. Conclusions ................................................................................................................ 62
References ....................................................................................................................... 63
1. Introduction

Belt conveyors are widely used as a transport system in several industries. The transportation of bulk materials is one of the most typical applications of belt conveyor systems. Designing this kind of systems represents a real challenge for engineers. There are too many aspects involved, and therefore several considerations have to be made to properly design these systems. Optimizing the use of materials and being environmentally friendly could be two of the most important goals to achieve. Previous experience can be helpful to achieve these goals. But only experience is not enough. Therefore, researchers are working to find new ways to solve the problems presented by the industry, developing new methods to do so.

The main objective of this research is to explore the possibility of coupling DEM and FEA to analyze a belt conveyor system. This could serve as a new tool for engineers in charge of designing belt conveyor systems. These two methods (DEM and FEA) could be used to calculate the load on the idler rolls, the stresses along the belt and the efficiency of the belt varying different parameters. Those parameters could be idler rolls length, trough angle, etc. Using computer simulation to analyze a system before actually building it may help to overcome some problems that could not be realized until it was built. The biggest issue presented while simulating a belt conveyor system is the deflection of the belt. The larger the number of rolls per idler station is, the more difficult the analysis becomes. For this reason, it seems wise to start from a simple case scenario, with idlers stations of only one roll. If the results were positive, a more complex system can be studied. The final model should be exactly like the real model.

This report is structured as follows: the second chapter is the literature review. This literature review includes brief background information with relation to belt conveyors. Also in this review some theoretical aspects about bulk materials can be found. The last part of the literature study is the theoretical approach to calculate the load on the idler rolls. Some indications are provided in order to understand how the bulk material loaded on the belt affects the rolls.

The third chapter of the report is the description of the modeling process. All the steps are explained in detail, remarking the advantages and drawbacks of every approach. During this section, a comparison between the experimental and the theoretical results was performed.

Chapter four of the report presents some possibilities to calculate the load on the idlers. Some options to further research are provided with relation to this topic. Also some suggestions are made of different possible solutions to calculate the loads in future research.

The last chapter of the report is the conclusions. After following all the steps described, it was concluded that even though the solutions are valid, more optimal results could be obtained in the future. This conclusion was made after noticing that the software available until now is not as sophisticated as it should be in order to accomplish the required tasks.
2. Literature review

2.1. Belt conveyors background

Belt conveyors are used all around the world to transport bulk materials. The main characteristics of a belt conveyor, in order to have a first impression, are the endless belt, and the pulleys. The belt moves between the head and the tail pulley, driven by the drive pulley. This transport system is specially used where the other options for the transportation, such as road or train transport, are not feasible, due to the poor conditions of the infrastructure in those places. Even though this transport system has been used for a considerably long time, there still lie considerable limitations in the systems like short lifetime. As a result of this, researchers are working trying to optimize the use of belt conveyors [1].

The trough belt conveyor is probably the most widely used and the most known conveyor among all types of belt conveyors.

The trough belt conveyor has proven to be a reliable and versatile conveyor in many applications and is probably more appropriate than most other types of conveyors when exposed to adverse operating conditions.

The most common trough belt conveyor is the 3-roll idler belt conveyor, but many other types are used. Some types are listed below:

- 1-roll idler belt conveyor
- 2-roll idler belt conveyor
- 3-roll idler belt conveyor
- 3-roll idler (deep) belt conveyor
- 5-roll idler belt conveyor
2.1.1. System description

In order to achieve a complete understanding of how does a belt conveyor work, it would be really helpful to give a functional description of the whole process:

A trough conveyor is essentially integrated by: an endless belt (a), two pulleys (one of them is known as the drive pulley (c)), and a take-up system (e). One of the pulleys is located at the end tail and the other one is located at the head end. The take-up system’s main function is providing the whole system an initial tension, in order to avoid the sag of the belt as much as possible. This is possible due to the presence of the sliding pulley (d). In this picture it can be easily distinguished that the take-up system being used is a gravity take-up system, but many other types are also available. Other important components are the idlers (b), which are rollers located along the belt in order to provide some support to it.

The loading chute (g) is located there to properly load the material (f) onto the belt. This material is transported along the carrying side (h) of the system, and discharged into the tail chute (i). After discharging the material, the belt is guided back along the return side (k) by the return idlers (l).

Also the impact idlers (j) are necessary. Their main purpose is to protect the whole system against the impact of the bulk materials at the loading area.

Sometimes a snub pulley (m) is place in order to increase the wrap radius (n). This allows the drive pulley to supply a larger force to the system [4].

Comparing trough belt conveyors to other type of conveyor, we can find the following main advantages:

- Easy maintenance
- Very efficient
Very safe during operation
The product is friction-free transportation

On the other hand, trough belt conveyors have some disadvantages:

- Not suitable for very high slopes.
- Cannot negotiate small radius curves

There is a phenomenon occurring on the belt that is known as belt sag. The belt sag is represented in Fig. 2.4 by the symbol $\delta(t)$ and varies over time. This phenomenon occurs due to the weight of both the loaded material and the belt itself. Even though the belt is pre-tensed in order to reduce as much as possible the belt sag, zero belt sag is not achievable in reality.

![Fig. 2.4 Belt sag](image)

Trough belt conveyors consist of many different components. These components are listed below:

- Belt
- Idlers
- Pulley
- Driving system
- Take-up system
- Brake units
- Belt cleaning equipment

From the list of components provided above, the two most important components would be the belt and the idlers. To provide some further information about these two main components of a belt conveyor system, a brief explanation is going to be given.

### 2.1.2. Belt

The belt is one of the most important components in a belt conveyor. While designing a belt conveyor, the aim is to build the belt with the following properties:

- High strength
- Low self weight
- Small specific elongation
- High flexibility
- Long service life

This component has also many different elements. The internal structure of a typical conveyor belt can be seen in Fig. 2.6.

![Fig. 2.6 Conveyor belt internal structure [5]](image)

A detailed description of the function of every component will be given in order to clearly understand the importance of the conveyor belt:

**Carcass:**

This element is set to reinforce the structure of the belt. Its aim is to take up the tensions while starting and moving the loaded belt. The carcass also absorbs the impact energy produced by the loaded material during the loading process.

Two different types of carcass are possible:

- **Fabric plied carcass:**
  This kind of carcass consists of a single or multi-layered series of synthetic fabric layers [4]. This carcass is enclosed between the top and the bottom covers.

- **Steel cable carcass:**
  Steel cable carcass is only made of steel and rubber. The cables are covered by rubber, which makes adhesion easier to the covers. This fact helps to improve lateral tear forces.

In case of fabric plied carcass conveyors, it is very important not to make a very thick belt. The more layers it has, the more difficult it becomes to fold the belt. The reason for this is that, if the belt was too thick, both troughing and bending the belt around the pulleys would be a very difficult task. Other materials can be used to supply extra strength to the belt (instead of increasing the thickness of the carcass).
Covers:

The covers are rubber layers. The main purpose of the covers is to protect the carcass against any possible damage caused by the operating process. The covers also prevent the loaded material to damage the carcass during the process. Desirable properties of the covers:

- Resistance to: cutting, gauging, tearing, abrasion, aging, moisture absorption and sometimes resistance to oil, chemical and heat [5]

2.1.3. Idlers

Idlers are another important component for trough belt conveyors. Its function is to provide support to the belt, and absorb the load of the bulk material. Idlers work as a low friction surface, thus they prevent belt from breaking down.

In a belt conveyor system a huge number of idlers can be easily found (the longer the conveyor is, the larger the number of idlers is needed). Nevertheless, not all the idlers are used with the same purpose. The main types of idlers are:

- Troughing idler set
- Transition idler set
- Impact idler set
- Return idlers

Brief description of every type of idlers, in order to learn what is their function in a belt conveyor system:

1. - Troughing idler set:

   This is the most common kind of idlers within a belt conveyor system. Its function is to maintain the troughed shape of the belt. The purpose of these idlers is to keep the same cross section along the whole system.

   The locations of the troughing idlers in a whole belt conveyor system can be seen in Fig. 2 7.

Fig. 2 7 Troughing idlers layout [4]
2. Transition idler set

Since it is not possible to adopt the troughed shape immediately before and after the pulley (the belt would get worn very quickly), some intermediate stages are needed. This is the purpose of installing transition idlers. There are a couple of sets of transition idlers to go from a flat belt to a troughed belt without damaging the belt too much. Fig. 2.8 shows a typical transition section on a belt conveyor system:

![Fig. 2.8 Typical transition section [5]](image)

3. Impact idler sets

The main function of the impact idler sets is to provide extra resistance to the loading area of the conveyor belt. Since the bulk materials are charged directly onto the belt at this point, the belt will receive huge impacts. In Fig. 2.9 it can be seen where the impact idlers are situated.

![Fig. 2.9 Impact idlers layout [4]](image)

4. Return idlers

Since the return side is usually unloaded, the idler spacing (Fig. 2.10) on this side can be larger than the idler spacing of the idlers on the carrying side. A typical layout of return idlers can be seen in Fig. 2.10.

![Fig. 2.10 Return idlers layout](image)
2.2. Bulk material behavior

The behavior of the bulk material on a belt conveyor is related to the soil mechanics theory. At the belt opening (after the idler set), the stress state of the bulk material is like the state shown in Fig. 211 (a). On the contrary, at the belt closing (approaching the idler), the stress state is like the state shown in Fig. 211 (b). "Act" stands for active, and "Pas" stands for Passive, referring to the stress state of the material. These two different states are due to the effect of the belt opening and closing on the loaded material. When the belt is opening, the material pushes the belt. On the other hand, at the belt closing, it is the belt that pushes the material.

The transition from active to passive stress state occurs approximately at the middle point between two idler sets (Fig. 212).

2.2.1. Soil mechanics

To understand this phenomenon properly, it is necessary to go back to the theory of soil mechanics. A brief summary is going to be included in order to provide with a proper
explanation of this topic. In 1776, Coulomb presented a theory with relation to limit states on earth pressure [6]. This theory is still being used nowadays, and is closely related to the behavior of bulk material on a belt conveyor.

Two different states of earth pressure are presented: active stress state and passive stress state.

**Active earth pressure**

An active state of earth pressure occurs when earth collapses pushing the wall. Fig. 2 13 shows how the triangle of earth slips down as the wall moves towards the left.

![Fig. 2 13 Active earth pressure [6]](image)

W is the weight per length unit of the wedge of soil. There should be equilibrium of forces both in horizontal and vertical direction. This equilibrium includes Q (force caused by the wall on the soil), T (shear force) and N (normal force). All these forces are measured per length unit.

Therefore, the equations of equilibrium are:

\[
\begin{align*}
Q + T \sin \theta - N \cos \theta &= 0 \quad (2.1) \\
W - N \sin \theta - T \cos \theta &= 0 \quad (2.2)
\end{align*}
\]

Vertical stresses can be obtained using the following equation:

\[
\sigma_v = \gamma z \quad (2.3)
\]

In the previous equation, "z" stands for the vertical distance from the top of the soil to the point where the stress is being calculated.

To calculate horizontal stresses from vertical stresses, coefficient \( K_a \) is needed:

\[
\sigma_h = K_a \sigma_v \quad (2.4)
\]

\( K_a \) is the coefficient of active earth pressure:

\[
K_a = \frac{1 - \sin \phi}{1 + \sin \phi} \quad (2.5)
\]

\( \phi \) is the angle of internal friction of the material (soil).

**Passive earth pressure**

A passive state of earth pressure occurs when earth collapses due to the force applied by the wall. The triangle of soil in Fig. 2 14 is pushed upwards by the wall.
In this case, to calculate horizontal stresses from vertical stresses, coefficient $K_p$ is needed:

$$\sigma_h = K_p \sigma_v \quad (2.6)$$

$K_p$ is the coefficient of passive earth pressure:

$$K_p = \frac{1 + \sin \phi}{1 - \sin \phi} \quad (2.7)$$

### 2.2.2. Application to bulk behavior

As it was explained in the beginning of section 2.2, the opening and closing of the belt creates an active or passive stress state (Fig. 2.12). When the belt passes the idler station, the material pushes the belt to both sides. Comparing this situation to the theoretical case, the belt would be the wall and the material would be the soil (active stress state). Therefore, the material slips down causing the opening of the belt.

On the other hand, it is necessary to explain what happens when the belt approaches the idler station. In this case, the belt is forced to close due to the idler rolls distribution. Hence, the material is pushed upwards by the belt. Comparing to the theory of soil mechanics, the belt would be the wall pushing the soil upwards (passive stress state).

### 2.3. Idlers loads

The load supported by the idler sets is closely related to the properties of the bulk material on the conveyor.

A distinction between two different types of bulk material can be made: granular material (without cohesion) and cohesive material. The most commonly transported material in the mining industry is granular. Therefore, adhesive forces can be neglected. As a result, the relation between normal and shear stresses within the bulk solid can be done using:

$$\tau = \sigma \tan \phi \quad (2.8)$$
As it was explained in the previous chapter, the bulk material on the conveyor has different stress states depending on its position with relation to the idlers. To analyze the load situation on an idler set, the cross section is the one showed in Fig. 2.15.

![Idler set cross-section](image)

Referring to Fig. 2.12, it is clear that every idler set supports a length of $s/2$ at the closing of the belt and also $s/2$ at the opening ($s$ is the idler spacing).

To calculate the load on the rolls of a three-roll idler set, the following configuration is used:

![Idler configuration](image)

There are three main loads exerted on an idler set. These loads are:
- Idlers own weight
- Weight of the belt
- Weight of the material

Depending on the trough angle, the percentage of load supported by each roll can vary. Increasing the trough angle ($\lambda$) makes the percentage on the middle roll increase, and hence the percentage of
load on the wing rolls decreases. For example, with a typical value for the trough angle $\lambda = 35^\circ$, the load distribution among idlers might be:

![Fig. 2 17 Load distribution ($\lambda=35^\circ$) [8]](image)

Idler rolls are fixed with bearings at both sides of the shell. For this reason, the analysis of load distribution can be assumed to be similar to a structure like the one in the Fig. 2 18 a). Fig. 2 18 b) shows the distribution of the bending moment along the idlers, assuming the percentages shown in Fig. 2 17.

According to Krause [7], the following equations are necessary to calculate the loads on the idlers rolls.

Using the configuration shown in Fig. 2 16, the following values can be obtained in order to calculate the loads on the idler rolls:

$$b_2 = 0.9b_1 - 0.05 \text{ [m]} \quad (2.9)$$

$$h = b_2r_1 (\sin \alpha_1 + \cos \alpha_1 \cdot \tan \varphi_b) + \frac{1}{2} r_2 \tan \varphi_b \quad (2.10)$$

$$A = \left( b_2r_1 \cos \alpha_1 + b_2r_1 r_2 \right) \frac{\sin \left( \frac{1}{4} (\alpha_1 + \varphi_b) \right)}{\cos \varphi_b} + \frac{1}{4} r_2 \tan \varphi_b \quad (2.11)$$

Next figure shows all the loads involved on a belt conveyor section during the opening of the belt. This figure is also applicable to the closing of the belt (analogous process):
At the belt opening, the forces on the idler rolls are:

\[ F_{F_{1a}} = \frac{g}{9.814} \frac{1}{a} \rho \lambda b^2_{2R_1} \]  
(2.12)

\[ F_{F_{2a}} = \frac{1}{2} F - 2F_{F_{1a}} \cos(\alpha_1 - \varphi_W) \]  
(2.13)

On the other hand, at the closing of the belt, the loads on the rolls are:

\[ F_{F_{1p}} = \frac{g}{9.814} \frac{1}{a} \rho \lambda b^2_{2R_1} \]  
(2.14)

\[ F_{F_{2p}} = \frac{1}{2} F - 2F_{F_{1p}} \cos(\alpha_1 + \varphi_W) \]  
(2.15)

\( F_f \) is the permanent load of the bulk material loaded on the conveyor.

Therefore, the total force on the lateral and middle rolls are:

\[ F_{F_1} = F_{F_{1a}} + F_{F_{1p}} \]

\[ F_{F_2} = F_{F_{2a}} + F_{F_{2p}} \]

To calculate \( \lambda_a \) and \( \lambda_p \) it is necessary to proceed as follows:

\[ \lambda_a = \left[ \frac{\sin(\alpha_1 + \varphi)}{\sqrt{\sin(\alpha_1 - \varphi_W) + \sin(\varphi_W + \varphi) \sin(\varphi - \varphi_b) \sin(\varphi + \varphi_b)}} \right]^2 \]  
(2.16)

\[ \lambda_p = \left[ \frac{\sin(\alpha_1 - \varphi)}{\sqrt{\sin(\alpha_1 + \varphi_W) - \sin(\varphi_W + \varphi) \sin(\varphi + \varphi_b) \sin(\alpha_1 + \varphi_b)}} \right]^2 \]  
(2.17)
3. Explanation of the model: EDEM® – ANSYS® coupling

The modeling process of a loaded belt conveyor implies multiple factors that have to be taken into consideration. Due to the complexity of the deflection of the belt, the task of modeling a belt conveyor becomes a real challenge for engineers. This complexity arises as the number of rolls per idler set is increased. Some steps must be taken in order to overcome all the issues present during the simulation of a loaded belt conveyor.

Another challenge present during this research is the fact that the software involved in these simulations has not been used before with this purpose. This aspect represents an added difficulty to the process.

As it was mentioned before, the deflection of a belt conveyor can be very complex to analyze. This happens especially with a number of rolls per idler larger than three. For this reason, and with the aim to provide results as close to reality as possible, a coupling between Discrete Element Method (DEM) and Finite Element Analysis (FEA) was chosen. For this task, EDEM® 2.6 was selected on the side of DEM, and ANSYS® 14.5 was chosen for the FEA.

Since the simulation using EDEM® does not provide any displacement of the geometry used, an iterative process has to be undertaken. The purpose of this iterative process is to achieve the steady state for the behavior of the bulk material on the belt. The following diagram illustrates the iterative process:

![Iterative process diagram](image)

In this process, SOLIDWORKS® 2013 is used to create the geometry of the belt. Every time a new situation is reached, due to the deflection of the belt, a new geometry has to be generated.
Until now, only a one-way coupling between EDEM® and ANSYS® is available. This coupling consists of exporting the load data from EDEM® and importing it in ANSYS®. For this research, a two-way coupling consisting of exporting force data to ANSYS® and exporting displacement to EDEM® after the FE analysis is required. This two-way coupling is not possible yet. In future releases of EDEM® this feature may be available, but for now, other possibilities have to be considered.

The final model should be like the one shown in Fig. 3 2 and Fig. 3 3:

![Fig. 3 2 Geometry of the final model](image1)

![Fig. 3 3 Final model cross section](image2)

Due to the complexity of this model, some steps must be taken before designing the final model. The first of these steps is to model a simple flat belt conveyor. Fig. 3 4 shows this model.

![Fig. 3 4 Flat conveyor geometry](image3)

Sections 3.2, 3.3, 3.4 and 3.5 explain in detail all the steps taken during the modeling process. The process basically consists of studying the different options to achieve the steady state in the system.
These options have to overcome the impossibility of performing a two-way coupling between EDEM® and ANSYS®, as it was explained before.

### 3.1. *Theoretical considerations*

The first step while designing any mechanical system is to study the theoretical situation. This is meant to provide a first approach to the problem and therefore to obtain some initial results. In this case scenario, a flat conveyor is going to be designed. The geometry of the belt can be seen in Fig. 35. The dimensions of the belt are 1500x200x10 mm. The density considered for the theoretical calculations and also for the following simulations was:

\[ \rho = 1522 \text{ kg/m}^3 \]

![Conveyor belt geometry](image)

**Fig. 3 5 Conveyor belt geometry**

Even though the rubber behaves as a viscoelastic material, the core of the belt (the carcass) can be considered to be a linear elastic material. For this reason, the following simplification can be made with relation to the calculation of the deflection of the belt.

![Belt structural simplification](image)

**Fig. 3 6 Belt structural simplification**

This simplification can be made due to the symmetry of the system. The simulation is being made only for a belt conveyor with two idler stations. This means that the belt is symmetric at both edges.
Therefore, the short edges of the belt can be considered completely constrained in all degrees of freedom.

This structural situation has a given value of the maximum displacement. This maximum displacement occurs in the middle section of the belt (X=L/2). This maximum value can be calculated using the following equation:

\[ y_{\text{max}} = \frac{qL^4}{384EI} \]  

(3.1)

For the calculation related to this research, the following values have been used:

\[ q_b = 1522 \text{ kg/m}^3 \cdot 0.2 \text{ m} \cdot 0.01 \text{ m} = 3.04 \text{ kg/m} \]  (weight of the belt)

\[ q_m = \frac{6.5 \text{ kg}}{1.5 \text{ m}} = 4.33 \text{ kg/m} \]  (weight of the material)

\[ L = 1.5 \text{ m} \]

\[ q = q_b + q_m = 7.4 \text{ kg/m} \]

The moment of inertia (I) has to be calculated with relation to the cross-section area of the part. In this case, the cross-section of the belt can be seen in Fig. 3.7:

\[ I = \frac{1}{12} bh^3 \]  

(3.2)

Once all the parameters needed for (3.1) have been calculated, the value of the deflection for different values of the Elasticity Modulus is:

<table>
<thead>
<tr>
<th>E (Pa)</th>
<th>( y_{\text{max}} ) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.4e8</td>
<td>244</td>
</tr>
<tr>
<td>4.8e8</td>
<td>122</td>
</tr>
<tr>
<td>9.6e8</td>
<td>62</td>
</tr>
<tr>
<td>1.92e9</td>
<td>31</td>
</tr>
</tbody>
</table>
Note that these results have been obtained from a non-deformed belt. Starting from the initial situation, these values reflect the deflection after applying the previously calculated load, under different values of E.

After all these considerations the modeling process can be started.

3.2. **First approach (Several iterations maintaining the Elasticity Modulus unchanged)**

The first simulation was done using a simple geometry, in order to understand the coupling between EDEM and ANSYS, and being able to detect any possible error that may occur. In this case a flat plate was used as a starting point.

For this first simulation, the main objective was learning how to export data from EDEM®, in order to process it and applying it to the ANSYS® model. For this reason, the simulation had a low number of particles, to reduce the computational time.

For this first approach, a flat belt conveyor was designed in SOLIDWORKS®. Fig. 3 9 shows this initial geometry.

The small squares that can be seen in the figure were placed there with the intention to get a uniform mesh afterwards. Even though it worked, it created an extremely high number of nodes. What is more, this is only applicable to this geometry. This means that once the belt is deformed this mesh will not be usable anymore. The reason of this is that the coordinates of the nodes in the XZ-plane should remain unchanged. Unfortunately, in SOLIDWORKS® this mesh can only be created for a flat
geometry (not deflected). Hence, the higher number of nodes, which increases the computational time, is not really helpful. Despite this fact, and because this issue was detected after doing several models, the same initial geometry has been used over the different approaches (3.2, 3.3, 3.4 and 3.5). The reason is to have the same model for every different approach, and therefore obtain comparable results.

After creating the geometry in SOLIDWORKS®, it should be saved with *\.stl extension. This extension means that the mesh is imported directly from SOLIDWORKS®, and therefore it cannot be modified afterwards. The mesh obtained for this simulation is shown in Fig. 3 10.

![Fig. 3 10 EDEM® mesh of the belt](image)

Once the geometry has been designed, the next step of the modeling process is to define the parameters of the simulation in EDEM®.

First of all, the materials to be used have to be defined. For this series of simulations, two materials are used: material of the particles and material of the belt. The parameters of the material of the particles are enclosed in Table 3 1:

<table>
<thead>
<tr>
<th>Table 3 1 Particles material properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poisson’s Ratio</td>
</tr>
<tr>
<td>Shear’s Modulus</td>
</tr>
<tr>
<td>Density</td>
</tr>
</tbody>
</table>

On the other hand, Table 3 2 encloses the properties of the material of the belt:

<table>
<thead>
<tr>
<th>Table 3 2 Belt material properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poisson’s Ratio</td>
</tr>
<tr>
<td>Shear’s Modulus</td>
</tr>
<tr>
<td>Density</td>
</tr>
</tbody>
</table>
Some remarks must be done with relation to the properties of the belt material. The real value for the Poisson’s Ratio is slightly higher but ANSYS® limits the Poisson’s Ratio to 0.5 (no higher value can be used). Hence, a smaller value had to be applied. It should also be mentioned that all these values of the belt material could be modified before importing them in ANSYS®. The way to proceed in this respect will be explained at the end of section 3.2.

The second step was defining the particle. Particles for these simulations basically consist of three spheres, each one with a radius of 10 mm. Even though these three spheres are only one particle, the software needs a higher computational time to realize the experiment. A good solution to reduce this computational time could be having particles made of only one sphere. Fig. 3 11 shows one particle.

![Fig. 3 11 Particle definition](image)

The third step of the EDEM® simulation is defining the geometry. For this case, three different elements will be used:

- Flat conveyor belt
- Outer box
- Squared plate

Fig. 3 12 shows these three elements:

![Fig. 3 12 Simulation geometry](image)

Every element has a function during the simulation. The belt is obviously designed to have the moving particles on it. The red box surrounding the belt is used to keep the particles on the belt, and prevent them to spill. The last geometry used is the squared plate. Its function is to create the particles (it is used as a particle factory).
Both the box and the thin plate were created using EDEM®. On the contrary, for designing the conveyor belt, SOLIDWORKS® was used.

For interaction between particles and belt the moving plane contact model was used. This means that the particles in contact with this element will move towards the predefined direction. For this experiments a velocity of 0.2 m/s in X-direction was selected.

The fourth and last stage of the preparation of the simulation was defining the particle factory. This task consists of establishing the total mass of particles used during the simulation and the creation rate of the particles.

In this case a dynamic factory was selected, with a total mass of 10 kg. Once the belt is full of particles, the total mass on the belt is 6.5 kg. This means that only part of the 10 kg is on the belt at the same time. The remaining particles have either gone out of the domain or have not been generated yet.

Once all these parameters have been defined, the simulation can be run. By default the maximum number of attempts to place a particle is set in 20. Even though for this simulation it does not make a difference, if changed to 2 the computational time can be reduced considerably. This simulation was run for 10 seconds.

After completing the simulation, the data was exported to a *.txt file. It is important to choose the option Export to ANSYS®. This exported file needs to be modified in order to run it in ANSYS®. Therefore it has to be in the format of ANSYS® APDL (the programming language of ANSYS®). Fig. 3 13, 3 14 and 3 15 show a sample of this code:

```plaintext
/PREP7

! Material properties and element type definition
ET,1.181
R,1.0,0.001
mp,ex,1,4.e8 ! Young's Modulus
mp,prxy,1,0.43 ! Poisson's ratio
mp,dens,1,1522 ! Density in kg per cubic meter

! Creating the nodes
n,0,0,0.01, 0.00000e+00, 0.00000e+00, 2.00000e-01
n,0,0,0.02, 0.00000e+00, 0.00000e+00, 2.00000e-01
n,0,0,0.03, 1.50000e+00, 1.00000e-02, 2.00000e-01
n,0,0,0.04, 1.50000e+00, 0.00000e+00, 2.00000e-01
n,0,0,0.05, 1.50000e+00, 0.00000e+00, 0.00000e+00
n,0,0,0.06, 1.50000e+00, 1.00000e-02, 2.00000e-01
n,0,0,0.07, 0.00000e+00, 0.00000e+00, 0.00000e+00
n,0,0,0.08, 0.00000e+00, 1.00000e-02, 0.00000e+00
n,0,0,0.09, 3.94737e-02, 1.00000e-02, 2.00000e-01
n,0,0,0.10, 7.89474e-02, 1.00000e-02, 2.00000e-01
n,0,0,0.11, 1.18421e-01, 1.00000e-02, 2.00000e-01
n,0,0,0.12, 1.57895e-01, 1.00000e-02, 2.00000e-01
n,0,0,0.13, 1.97368e-01, 1.00000e-02, 2.00000e-01
n,0,0,0.14, 2.36426e-01, 1.00000e-02, 2.00000e-01
n,0,0,0.15, 2.76310e-01, 1.00000e-02, 2.00000e-01
```

Fig. 3 13 APDL properties and node definition
During this first approach, several attempts were made until a valid solution was obtained. This had to be done because initially a Young’s Modulus of \(E=2.4\times10^8 \text{ Pa}\) was chosen. This value resulted a very low value for the Elasticity Modulus, and therefore the deflections were extremely high.

Sections 3.2.1 and 3.2.2 explain the different situations in detail.

### 3.2.1. Attempt 1: flat conveyor belt with Young’s Modulus \(E=2.4\times10^8 \text{ Pa}\)

**ITERATION 1:**

This first iteration is the first stage to achieve the steady state of this model. From Fig. 3 13, 3 14 and 3 15 it is clear that the first step for the FE Analysis is to define the properties, nodes and elements. After doing so, ANSYS® shows the following image on the workspace:
Once the nodes and the elements have been defined, it is necessary to apply the boundary conditions before applying the loads. As it was explained before in this report, the boundary conditions required for this analysis consist of constraining all degrees of freedom for the nodes on both short edges of the belt. In the workspace, the following image can be seen:

The last step to be taken in ANSYS® is applying the loads imported from EDEM®. After all the loads have been applies, the image in the workspace changes:
Once all these steps have been finished, it is necessary to solve the analysis. The deformed shape of this model is shown in Fig. 3 19:

After this analysis, the value for the maximum deflection obtained from ANSYS® was:

$$\text{DMX} = 108 \text{ mm}$$

In this case, the maximum deflection occurs in the middle section of the belt (X=L/2), exactly in the same as for the theoretical calculation made before. The value for the deflection is approximately 50% lower than the theoretical value. There are two reasons for this. The first reason is that the theoretical calculation considers a static load, whereas this analysis has been done for a moving load in the horizontal direction. Therefore, the vertical force is lower and so is the deflection. The other reason is the size of the mesh. Later on this report an analysis for a finer mesh will be performed in order to validate these results.
ITERATION 2:

After the first iteration, it is necessary to update the geometry. As it was explained in the diagram from Fig. 3 1, SOLIDWORKS® is used for this task. The new geometry is:

![Geometry after first iteration](image1.jpg)

As it was previously mentioned, in this case it is not possible to have the same squared mesh as in the previous geometry. The same steps were taken using ANSYS®, in order to compare results. The ideal case scenario would be a smaller deflection, in order to finally achieve, after several iterations, a steady state. Fig. 3 21 shows the deformed shape after the FE Analysis. In this case the deformation is not symmetrical.

![Deformed shape after the second iteration](image2.jpg)

To transfer this deformed shape to SOLIDWORKS®, due to the complexity of the final shape, some assumptions have to be made. It is very clear from the figure that there is two points which are more relevant with relation to the displacements. These two points are the middle point and the point of maximum displacement. To know which node has the maximum displacement, it is necessary to list the results in ANSYS® (post-process analysis).

The maximum deflection after this analysis has the following value:

\[ \text{DMX} = 179 \text{ mm} \]
This value is extremely high. The reason is that the deflection after the first analysis was already slightly high and the Young’s Modulus too low. Hence, particles tend to accumulate while the move upwards on the belt conveyor, resulting in a very high deformation. Fig. 3 22 shows the new geometry after this second iteration:

![Fig. 3 22 Geometry after second iteration](image)

After this extremely deformed geometry, a new attempt with higher Elasticity Modulus was made. There is another negative aspect about this simulation. The deformation after the second iteration is asymmetric. The moving plane causes this. The moving plane is defined with fixed velocities in either X, Y or Z direction. Due to the shape of the deformed belt, a dynamic definition of the velocity would be more suitable. The problem is that this type of contact model requires more sophisticated inputs to the software.

Defining a horizontal moving plane means that the particles in contact with the belt will move in horizontal direction. This tendency to move horizontally causes the asymmetric deformation of the belt.

### 3.2.2. Attempt 2: flat conveyor belt with Young’s Modulus E=4.8e8 Pa

After the extremely high deflection of the belt after the first attempt, a higher value of the Elasticity Modulus is going to be used. In this case E=4.8e8 Pa.

**ITERATION 1:**

The only thing that needs to be changed in this case from the previous attempt is the Young’s Modulus. It can be done very easily by changing the value in the code presented in Fig. 3 13.

After re-taking all the steps explained before, the results after the FE Analysis were:
The maximum deformation (deflection) occurs again in the middle section of the belt (X=L/2). As it was mentioned before, this result is approximately 50% lower than the theoretical value. The reasons are exactly the same: size of the mesh and moving load (a static load may cause higher deflection).

**ITERATION 2:**

After the first iteration, the geometry, as it was explained before, has to be redesigned. The new geometry is:

The very same process explained for the previous attempt has to be repeated all over again until a steady state is achieved. Again two reference points (middle point and node with maximum displacement) have to be used to generate the new geometry.

The deformed shape after the second iteration is shown in Fig. 3 25:
The same process was repeated two more times. Table 3 encloses the results for all the iterations:

Table 3 Iterative process summary

<table>
<thead>
<tr>
<th>FLAT BELT CONVEYOR (E=4.8e8 Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ITERATION</td>
</tr>
<tr>
<td>-----------</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>3</td>
</tr>
<tr>
<td>4</td>
</tr>
</tbody>
</table>
The deflection is measured from the initial geometry every step. Fig. 3 26 shows the tendency of the deflections over the different iterations. It is very clear that a steady state cannot be achieved using this method.

<table>
<thead>
<tr>
<th>Iteration</th>
<th>DMX (mm)</th>
<th>Deflection (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>54</td>
<td>54</td>
</tr>
<tr>
<td>2</td>
<td>71</td>
<td>103</td>
</tr>
<tr>
<td>3</td>
<td>70</td>
<td>118</td>
</tr>
<tr>
<td>4</td>
<td>111</td>
<td>190</td>
</tr>
</tbody>
</table>

Fig. 3 26 First approach results

There is a difference between DMX and Deflection. DMX stands for the maximum displacement after every FE Analysis. The maximum displacement does not always occur in the middle point of the belt. Only after the first analysis (symmetric deformation) these two values are the same. On the other hand, the deflection is the absolute displacement from the initial geometry.

Fig. 3 27 shows the shape of the belt after the fourth iteration, and Fig. 3 28 summarizes the evolution of the belt deflection over all the iterations:

Fig. 3 27 Geometry after the fourth iteration

Fig. 3 28 Deflection after every iteration

The final deformation is not admissible. Moreover, the deflection keeps increasing. The reason for this increasing deformation is that the Finite Element Model is not the same for all the iterations. If the model is new for every analysis, the FEA software does not consider the previous deformation, and assumes the new geometry also as a new model. This causes that
the belt is always being deformed, even though the deflection is extremely high to be realistic. Therefore, some changes are required in order to make this solution closer to reality.

The second approach tries to overcome this issues presenting a different possible solution.

### 3.2.3. Conclusions

After this first approach to model a belt conveyor some negative aspects with relation to the feasibility of the method were encountered. The aim of coupling DEM and FEA was to achieve steady state behavior of the system.

It looks very evident that performing the iterations in this direction does not help achieving such a goal. The reason is that every time a new geometry is created using SOLIDWORKS®, a new Finite Element Model is created as well. To achieve the steady state iterating in the way that was presented before, there should be a possibility to connect the FEA software and the DEM software directly. Therefore, the CAD design of the new geometry should not be part of the iteration loop. This issue could only be avoided by importing in EDEM® the final position of the nodes after applying the displacements. However, this option is not available yet in this software. For this reason, other options have to be considered.

### 3.3. Second approach (Several iterations changing the Elasticity Modulus)

This second approach was designed to overcome the downsides discovered after the first approach. The belt, as it was explained before in this report, can be considered as an elastic material. This means that it deforms along the linear-elastic phase of the tensile test, but never enters the plastic phase.

A typical graph of a tensile test can be seen in Fig. 3 29.

![Fig. 3 29 Typical tensile test graph](image)

As it can be seen in the figure, the Elasticity Modulus represents the slope of the linear phase (elastic phase). From the Theory of Structures, it is possible to calculate the elongation of the belt using equation (3.3).
\[ \Delta L = \frac{NL}{EA} \]  

(3.3)

Some assumptions have to be made in order to obtain a more realistic behavior of the system. To reduce the deflection, one parameter of the previous equation has to remain the same and other has to change proportionally, in order to control the evolution of the system. There are two possible parameters to be changed:

- Tension of the belt (N)
- Elongation (\(\Delta L\))

Even though the tension of the belt seems to be the most logical choice, it is not such a good option. Due to the boundary conditions defined in ANSYS®, it is very unclear how to define the tension. Therefore, modifying the elongation is the most feasible solution.

To proceed, the belt tension is considered to be constant over all the iterations. If the elongation is assumed to change by factor 0.5 over two consecutives iterations, then the Elasticity Modulus is forced to change. This whole process is explained below:

\[ \Delta L_1 = \frac{NL_0}{E_1A} \rightarrow N = \frac{E_1A\Delta L_1}{L_0}; \frac{E_2A\Delta L_1}{L_0} = \frac{E_2A\Delta L_2}{L_2} \]  

(3.4)

Reducing the elongation by factor 0.5 in two consecutives iterations will also reduce the Elasticity Modulus by the same factor. This fact will increase the slope of the elastic phase and therefore it will force the elongation to be reduced between two consecutive iterations. The following graph shows this variation of E and \(\Delta L\):

![Graph showing variation of E and Delta L](image)

Fig. 3 30 Values of E

Taking all these considerations into account, the simulations were run. The characteristics of the simulations were exactly the same as used for the first approach.

The only thing to be changed is the Elasticity Modulus. To change this parameter, it must be changed in the ANSYS® code. It is necessary to recalculate the Young’s Modulus after all the iterations.

Considering the deformed shape as a triangle, the elongation can be easily calculated:
Considering the deflection obtained from the FE Analysis, the elongation can be calculated using simple trigonometric rules. Since the elongation is so small compared to the total length of the belt, reducing the elongation by factor 0.5 leads to an increase in $E$ by factor two. This means that for the second iteration, an Elasticity Modulus $E=9.6e8$ Pa will be used and so on.

### 3.3.1. Results

For the second approach, four iterations were performed in order to achieve the steady state. As it has been explained before, the Elasticity Modulus has been increased by a factor two between two consecutive iterations. Table 3.4 encloses the results obtained after every step. Note that the column DMX of the table refers to the point of maximum displacement after the analysis.

<table>
<thead>
<tr>
<th>ITERATION</th>
<th>E (Pa)</th>
<th>INITIAL GEOM.</th>
<th>DMX (mm)</th>
<th>DEFORMED GEOM.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.8e8</td>
<td><img src="image1.png" alt="Image" /></td>
<td>54</td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
<tr>
<td>2</td>
<td>9.6e8</td>
<td><img src="image3.png" alt="Image" /></td>
<td>36</td>
<td><img src="image4.png" alt="Image" /></td>
</tr>
<tr>
<td>3</td>
<td>1.92e9</td>
<td><img src="image5.png" alt="Image" /></td>
<td>10</td>
<td><img src="image6.png" alt="Image" /></td>
</tr>
<tr>
<td>4</td>
<td>3.84e9</td>
<td><img src="image7.png" alt="Image" /></td>
<td>10</td>
<td><img src="image8.png" alt="Image" /></td>
</tr>
</tbody>
</table>
Fig. 3 32 shows the geometry after the fourth iteration. Even though the total deflection does not change, Fig. 3 33 shows that there is a large deformation on the belt after the fourth iteration.

Fig. 3 32 Geometry after the fourth iteration

Fig. 3 33 Deflection after every iteration

From Fig. 3 33 it is clear that between states 4 and 5 the belt does not increase the deformation. The only thing that changes is the point of maximum sag.

Fig. 3 34 encloses the values for the maximum displacement and total deflection after every step:

<table>
<thead>
<tr>
<th>Iteration</th>
<th>DMX (mm)</th>
<th>Deflection (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>54</td>
<td>54</td>
</tr>
<tr>
<td>2</td>
<td>36</td>
<td>70</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>71</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>74</td>
</tr>
</tbody>
</table>

Fig. 3 34 Second approach results
3.3.2. Conclusions

This solution provided to overcome the problem of not achieving steady state gives back positive results. The belt, between states 4 and 5 has the same deflection value.

The problem encountered after this series of simulations is the asymmetric shape of the deflected belt. The reason of this strange shape is the definition of the simulation parameters. The moving plane of the simulation, as it was explained before, has been defined with a constant velocity in X-direction. This works properly while the belt is not deformed. Once the belt is deformed, the particles keep trying to move horizontally. Since the belt is curved, some particles will deform the belt trying to move in X-direction. Therefore, that moving plane is not applicable anymore. Some complex operations could be performed in future research works to overcome this problem.

To overcome this issue, it is necessary to go back to a simple case. For instance, a system where the load is static could help to check whether the deflection is symmetric or not. This new

3.4. Third approach (Several iterations changing the Elasticity Modulus with a static load)

The results obtained from the second approach were not as satisfactory as expected. The system could achieve the steady state, but the deformation of the belt is not symmetric.

To overcome this issue without going beyond the scope of this research study, going back to a simple case was decided. Analyzing a static load may help to solve the problem previously presented, (asymmetric deformation). To run a simulation with a static load, some parameter of should be modified in EDEM®. These parameters are the parameters related to the particle factory and the moving plane.

Since the aim of this third approach is having a static load, the moving plane must be removed (changing the velocity of the moving plane to 0 m/s would also work).

Regarding the particles factory, some changes have to be made. First of all, it is necessary to remove the factory plate used previously and create a box instead. It is important that the Virtual type is selected for the box. Fig 3 35 shows the particle box highlighted in green.
Secondly, the factory type must be changed from dynamic to static. Since the mass used before was 6.5 kg, this parameter should be defined too.

Even though only these changes would make the simulation to perform correctly, it is helpful to modify a few more things to reduce the computational time. The first thing that could be changed is the particles. Creating particles made only of one sphere of 5 mm (Fig. 3 36) will reduce the computational time. To prevent particles to roll, the rotation of the particles can be constrained. Also setting the particle size to be constant will reduce the computational time to a minimum.

Another parameter to be re-defined is the simulation time and the number of attempts to place a particle. For the time, only 4 seconds is enough to obtain the desired results. On the side of the number of attempts, reducing this value from 20 (default value) to 2 will considerably reduce the computational time.

After all the parameters have been correctly defined, the simulation can be run. After the simulations, the results can be analyzed. The color of the particles is defined to be different according to the velocity of the particle. For this case, since the load is supposed to be static, the color should be blue. A color blue means that the particle is not moving. Fig. 3 37 shows that the final results were exactly as it was expected:
Analyzing the results of the EDEM® simulation may result in an optimistic perspective of the problem. The issues faced before with a moving simulation seem to be overcome using a static load. At least the load, since the particles do not have any horizontal displacement, is going to be oriented downwards.

### 3.4.1. Results

For the third approach, consisting of analyzing a belt conveyor after applying a static load, only three iterations were needed. After the third iteration the steady state was achieved. Table 3 5 encloses the results of this approach. The deformed shape obtained after the Finite Element Analysis is also attached in that table.

#### Table 3 5 Results of the third approach

<table>
<thead>
<tr>
<th>ITERATION</th>
<th>E (Pa)</th>
<th>INITIAL GEOM.</th>
<th>DMX (mm)</th>
<th>DEFORMED GEOM.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.8e8</td>
<td></td>
<td>95</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>9.6e8</td>
<td></td>
<td>9</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1.92e9</td>
<td></td>
<td>8</td>
<td></td>
</tr>
</tbody>
</table>
To design the deformed shape it is necessary to proceed exactly in the very same way as before. The maximum displacement node has to be found to calculate the displacement (ANSYS® postprocessor).

Fig. 3.38 shows the deflected belt after the last iteration. Since the belt is symmetric in X-direction at both short edges, the same deflected geometry will be repeated. Fig. 3.39 shows a belt conveyor system with 4 idler stations. Note that the deflection is the same in every case.

![Geometry after the third iteration](image1)

Fig. 3.38 Geometry after the third iteration

![Belt conveyor with four idler stations](image2)

Fig. 3.39 Belt conveyor with four idler stations

Next figure shows the tendency of the maximum displacement and the deflection. Note that the maximum displacement does not necessarily represent the displacement of the middle point.

Fig. 3.40 shows the evolution of the deflection of the belt along the iterative process. After the third iteration, the geometry almost did not change compared to the previous one. In the graph of Fig. 3.41, the tendency of the value of the total deflection can be seen. The deflection stays at a constant value after the third iteration.
The most remarkable aspect of these results is the difference between the theoretical value of the deflection (122 mm) and the value obtained after the FE Analysis. Even though the load is now static, and therefore the forces are mostly vertical, there is still a difference of around 20%. The reason of this is the size of the elements used to mesh the belt. The smaller the elements of the model are, the closer to reality the behavior of the system is. This will be explained in more detail in section 3.6 in this report.

### 3.4.2. Conclusions

Performing this third series of simulations has a very important result: the steady state of the system has been achieved. As a consequence of achieving the steady state loads on idler could be calculated. Using the final geometry of this approach as the geometry of the deflected belt eases the calculation of loads on idler sets.

Despite the results are slightly satisfactory, there is still one issue with relation to the accuracy of the results. The FE Model used in the process changes between iterations. To try to
overcome this problem, a solution was presented. This possibility involves the use of both ANSYS® MAPDL and ANSYS® Workbench [9]. This process is fully explained in section 3.5. This process works to generate a new geometry from the deformed shape after the Finite Element Analysis. The applicability to this problem, where the same FE is required the whole time, needs to be tested.

3.5. **Coupling between ANSYS® Workbench and ANSYS® MAPDL**

After running all the simulations previously explained in this report, some shortcomings were found. The main problem of not being able to perform a two-way coupling between ANSYS® and EDEM® is that the Finite Element model used is not the same during the whole process. A solution was presented to overcome this problem. This possible solution consists of exporting the final position of the nodes in MAPDL to Workbench [9]. The feasibility for this method to solve the current problem has to be tested.

The goal using this method is to be able to use the same FE model all over again. If this was possible, the problem could be solved even though the two-way coupling is not available yet.

The steps that must be taken to export the node position from ANSYS® MAPDL and consequently import them in ANSYS® Workbench is going to be explained in detail.

### 3.5.1. Process description

**ITERATION 1:**

First of all, some remarks have to be done to the previous work. For all the simulations geometry with thickness has been used. The initial geometry for the third approach is shown in the following figure:
This geometry presents some drawbacks with relation to the FE Analysis. Even though EDEM® creates what seems to be a good quality mesh (Fig. 3 43), there are some negative aspects about it.

While defining the element type in ANSYS®, one of the parameters is the thickness. The default value of this parameter is 0.001 m (Fig. 3 44).

```
/PREP7

! Material properties and element type definition

ET,1,101
E,1,0.001

mp,ex,1,4,8e8
mp,epxy,1,0.45
mp,dens,1,1522

! Creating the nodes

n,000001, 0.000000e+00, 0.000000e+00, 2.000000e-01
n,000002, 0.000000e+00, 1.000000e-02, 2.000000e-01
n,000003, 1.500000e+00, 0.000000e+00, 2.000000e-01
n,000004, 1.500000e+00, 0.000000e+00, 2.000000e-01
n,000005, 1.500000e+00, 0.000000e+00, 2.000000e-01
n,000006, 1.500000e+00, 0.000000e+00, 2.000000e-01
n,000007, 0.000000e+00, 0.000000e+00, 0.000000e+00
n,000008, 0.000000e+00, 0.000000e+00, 2.000000e-01
n,000009, 3.04737e-02, 1.000000e-02, 2.000000e-01
n,000010, 7.39474e-02, 1.000000e-02, 2.000000e-01
n,000011, 1.18421e-01, 1.000000e-02, 2.000000e-01
n,000012, 1.57895e-01, 1.000000e-02, 2.000000e-01
n,000013, 1.7368e-01, 1.000000e-02, 2.000000e-01
n,000014, 2.36842e-01, 1.000000e-02, 2.000000e-01
n,000015, 2.76316e-01, 1.000000e-02, 2.000000e-01
```

Fig. 3 44 APDL properties and node definition (t=0.001 m)
Once the geometry has been imported in ANSYS® Workbench, the model will look like this:

![Initial geometry (Workbench)](image)

Every face of the belt has its own thickness. The reason is the element type chosen. Defining the thickness of the element means that not extra thickness is needed. After realizing this, a finite element analysis was undertaken in order to validate the feasibility of this method. After finishing the analysis, the next commands have to be written in ANSYS® command window:

```
/POST1
UPCOORD,1
CDWRITE,ALL,FILENAME,cdb
```

UPCOORD command updates the location of the nodes. On the other hand CDWRITE command saves the node position in a *.cdb file. After using this command the nodes will have the original position plus the displacements.

With relation to ANSYS® Workbench, the first step is creating a Finite Element Modeler standalone system. Once the system has been created, the second step is importing the data. It is done by browsing in the computer and selecting the *.cdb file previously saved as FILENAME.cdb.
For this case, the same deformed geometry is imported in Workbench:

In Fig. 3 47 some strange areas can be seen. The overlapping of models causes this. Creating a geometry with thickness implies having not only one but six different models. As a result of this, any attempt to build a new geometry will lead to a non-valid result.

Fig. 3 48 provides a schematic explanation of the six different models previously mentioned. Every face of the prism is a different Finite Element model itself.

To export this data as a valid geometry, the first step is setting the geometry clicking on Initial Geometry (Fig. 3 47). Once the initial geometry has been created, it is necessary to Convert to PARASOLID (Fig. 3 49).
The result of converting this geometry to PARASOLID can be seen in Fig. 3 50. The brown areas are the overlapping area. The last step would be exporting the generated geometry to a PARASOLID file. This type of file can be opened in SOLIDWORKS® and saved in an *.stl file.

To overcome this situation, a flat surface without real thickness has to be created (Fig. 3 51). The thickness of the model should be modified afterwards to 0.01 m in the ANSYS® script.
In Fig. 3 52 the new value of the thickness can be seen. After importing this new model in ANSYS® Workbench, the model would look like this:
In this case the model has been created properly. The belt has the appropriate thickness and only one FE model is being analyzed (unlike the previous iteration). Once the feasibility of this new model has been checked, the same steps taken for the first iteration have to be repeated:

Fig. 3 54 Imported model

Fig. 3 55 Initial geometry
Once the geometry has been saved in a PARASOLID file, it should be opened in SOLIDWORKS®. The next thing to do is saving this geometry in an *.stl file. Saving it in such a file creates a mesh that cannot be modified.

The last step remaining is to validate this process. The aim of using ANSYS® Workbench is to be able to use the same model in every iteration. If the mesh changes between two iterations, that means that the model is not the same either. Both meshes can be seen in Fig. 3 57.

Since the nodes are not evenly distributed in both models, both models are different. Therefore, this method does not achieve the expected goal.
3.5.2. Conclusions

Ideally, this method would allow the researcher to keep the model during the iterations. However, as it was explained before, the tools that EDEM® provides to modify the mesh are not enough accurate. The inputs are not very specific, and therefore the mesh cannot be controlled.

For the reasons explained before, this method turned out to be a non-feasible solution. It did not help to overcome the issue of not being able to have a two-way coupling between EDEM® and ANSYS®. Even though this method allows building a new geometry easily, the FE model changes. If the meshes are not the same then the steady state cannot be achieved without modifying some parameters. Hence, this approach is as valid as the previous one, but not better. The only really positive aspect is the procedure to build a new geometry after the FE analysis. Being able to do it with ANSYS® turns out to be very time saving for the researcher. The most important conclusion extracted from this last approach is that a two-way coupling is indeed necessary to successfully solve this research problem.

3.6. Comparison of results

After performing all the simulations, it is necessary to compare the results obtained. It has already been mentioned that to get accurate results, the mesh needs to be really fine. Smaller elements sizes leads to results closer to reality. For this reason, in order to check whether the theoretical calculations match the results obtained after the simulations, some analyses were done using the same values of the Young’s Modulus as for the theoretical calculation.

From the previous simulations, the case that is closest to the theoretical situation is the analysis of the static load. Using the same mesh as it was used before, but changing the value of the Elasticity Modulus, the different results were:

<table>
<thead>
<tr>
<th>E (Pa)</th>
<th>y_{max} (mm)</th>
<th>DMX (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.4e8</td>
<td>244</td>
<td>194</td>
</tr>
<tr>
<td>4.8e8</td>
<td>122</td>
<td>97</td>
</tr>
<tr>
<td>9.6e8</td>
<td>62</td>
<td>48</td>
</tr>
<tr>
<td>1.92e9</td>
<td>31</td>
<td>24</td>
</tr>
</tbody>
</table>
In the previous table, $y_{\text{max}}$ stands for maximum theoretical deflection, and DMX stands for maximum displacement (ANSYS®).

The mesh used for this first analysis was:

![First mesh (Big elements)](image1)

There is a considerably big different between the theoretical and the experimental results (around 20%). As it was explained before, the mesh size affects the results. A finest mesh provides results closer to theoretical values. Therefore, the same analyses were repeated for a model with smaller elements. The mesh used for this second analysis was:

![Second mesh (Small elements)](image2)
Using this new mesh, the results obtained were:

<table>
<thead>
<tr>
<th>$E$ (Pa)</th>
<th>$y_{\text{max}}$ (mm)</th>
<th>DMX (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.4e8</td>
<td>244</td>
<td>197</td>
</tr>
<tr>
<td>4.8e8</td>
<td>122</td>
<td>99</td>
</tr>
<tr>
<td>9.6e8</td>
<td>62</td>
<td>49</td>
</tr>
<tr>
<td>1.92e9</td>
<td>31</td>
<td>25</td>
</tr>
</tbody>
</table>

It is now clear that reducing the mesh size approaches the results to the theoretical values. This means that a very small elements size would lead to the same theoretical results. It should also be mentioned that increasing the Elasticity Modulus results into a smaller difference in the values of the deflection between the two different models.

### 3.7. Modeling process: Summary

This section of the report helps to summarize the conclusions obtained throughout the model process.

As it has been previously explained, the model to be built consists of a belt conveyor system. The main issue to be solved is the complexity of the deflection of the belt. Beginning with a first model of a belt conveyor with one-roll idler sets, some different attempts were made to model a proper system.

The first approach had the inconvenience of not using the same FE model for every step. Not using the same FE model means that every FE analysis is performed to a new model. Therefore, there will always be a deflection. For this reason some changes were necessary to overcome these problems. And here it is where the second approach was presented.

The first solution presented to solve the issue of not achieving the steady state was controlling the elongation after every analysis (second approach). Using the equations presented before, it was concluded that reducing the elongation by factor two between two iterations increases $E$ by the same factor. Applying these changes to the consecutive iterations, something relatively close to a steady state could be achieved. The problem faced here was the inexistent symmetry of the deflected geometry. The reason for this asymmetric deflected belt was found to be the parameters defining the moving plane of the simulation. The velocity of the moving plane was set on the horizontal direction. To overcome this, a more complex definition of the moving plane has to be done. This would involve programming and is beyond the scope of this research.

To skip the issue of the asymmetric deflection of the belt, it was decided to go back to simplicity, and a static load was analyzed instead. In this case, the deflection was perfectly symmetric. After three iterations the steady state was achieved.
The last attempt to have a proper model was to couple ANSYS® MAPDL and ANSYS® Workbench. The aim of this coupling was to keep the same model for all the iterations. It turned out to be unfeasible to do that. The only positive thing taken from this attempt is that a new geometry can be designed more easily.

The outcome of this whole process is that a two-way coupling is needed to correctly model a belt conveyor system. In future releases of EDEM® this option will be available. Hence, this research could be completed.
4. Idlers load calculation: Possible solutions

The last step for this research work would be calculating the loads on the idlers of the belt. Both EDEM® and ANSYS® have features to obtain the loads on certain areas of the model.

There are several factors that should be taken into consideration before calculating the load on the idler rolls. First of all, it is necessary to know what really influences the load on the rolls. The most evident parameters affecting the load withstood by the rolls are the weight of the material loaded on the belt and the weight of the belt. These are the most important loads but not the only ones. The last remarkable force to be taken into account is the tension of the belt. After installing the belt on the system, before loading the belt, it is necessary to apply certain pre-tension to the belt in order to reduce the belt sag as much as possible. Despite this pre-tension, there will still be a small deflection on the belt. Once the belt is loaded, the sag of the belt increases. Because of this increase of the sag, the tension also increases and as a result of this the belt would be suffering a force in longitudinal direction. All these forces affect the final load on the rolls.

Analysis of the load distribution on idler rolls using EDEM® and ANSYS® is presented as follows. Possibilities and drawbacks will also be discussed.

4.1. Calculation of the loads using EDEM®

First of all, a theoretical calculation of the loads on every idler should be made in order to compare the results to the values obtained in EDEM®. To do these theoretical calculations it is necessary to use the same geometry simplification that was used in section 3. This simplification is shown in Fig. 4 1:

![Fig. 4 1 Belt structural simplification](image)

In this case, the forces to be calculated are the reactions in A and B. These reactions can be calculated as follows:
\[ R_A = R_B = \frac{qL}{2} \] (4.1)

Since the problem to be analyzed has three idler stations, the problem can be simplified as it is shown in Fig. 4.2:

In this case, the problem can be divided into two smaller problems. Once the reactions are calculated, superposition can be applied in order to obtain the total vertical forces on the idler stations. The procedure is:

\[
R_A = R_{B1} = R_{B2} = R_{C} = R \quad (4.2)
\]

\[
R_B = R_{B1} + R_{B2} = 2R \quad (4.3)
\]

\[
R = \frac{qL}{2} = \frac{7.37 \text{ kg/m} \cdot 1.5 \text{ m}}{2} = \frac{73.7 \text{ N/m} \cdot 1.5 \text{ m}}{2} = 55.275 \text{ N} \quad (4.4)
\]

From the previous result, it can be concluded that the force on the middle roll is 110.55 N (2R).

### 4.1.1. Calculation of the force on the central roll using a grid bin

After running the simulation in EDEM®, there is a post processing option that allows creating geometry bins. These bins can be used to measure parameters related to the simulation, such as the mass of the material, forces on different elements, etc. However, the contact calculation between two geometries is not possible.

In this case, those bins can be used to calculate the vertical load on a certain area of the belt. Defining the bin to select the contact area between the belt and the idler roll could be a way to obtain the value of the force on the idlers. To calculate this force, a new simulation was run with a geometry including three idler stations. The geometry used for this simulation can be seen in Fig. 4.3:
Most of the simulation parameters are exactly the same as those used for the third approach (section 3.4) with a static load. The only parameter that had to be modified was the total load of bulk material. Since the belt is twice as long as the previous one, the new value of the mass of material is 13 kg.

After running the simulation, a grid bin was defined to measure the vertical force along the belt. The grid had 51 sections. The decision of the number of sections was made according to the dimensions of the contact surface between the roll and the belt. Moreover, the total number of sections should be an odd number. Thus, there is a section exactly where the middle idler station should be. In this case, the section number 26 was in the same position as the idler.

Once the grid bin has been created, the next step is defining a query to calculate the total normal force in every bin. Fig. 4.2 shows the grid bin. A roll has been created to see which bin is considered to be the idler.

The definition of the query requires the following inputs:

The resultant total forces on the whole belt, and therefore on the idler roll can be seen in Fig. 2.4:
Some remarks have to be done with relation to this method to measure the loads. First of all, it is necessary to check whether this method is valid or not. Measuring the load in one bin means that only the load within the limits of that bin is considered. Therefore, the influence of the surrounding areas is not being taken into consideration. As it was explained before, the force created by the tension of the belt was also relevant. After this calculation a force smaller than 3 N was obtained in every bin. This force is much smaller than the theoretical value calculated before. Secondly, note that there is not a really big difference in the load values along the belt. The reason is that the load on the belt has a very small value. Therefore, the difference between the maximum and the minimum load on the belt is very small. To overcome this issue, in order to calculate the load that every idler station has to withstand, a new geometry bin was made. In this case it consisted only of one section.

In addition to this, it is important to mention that EDEM® does not take into account the deflection of the belt. Hence, the stresses on the belt caused by that deflection could not be calculated with EDEM®.

### 4.1.2. Calculation of the force on the central roll using a geometry bin

After the evidence that calculating the load of the central roll using a grid bin is not possible, a geometry bin was used. This new method allows obtaining the total vertical force on the whole system, and calculating the reactions at the idler stations. After defining the query as the one shown in Fig. 4.5, the result was:

![Image of Fig. 4.7 Total vertical forces on the system]

The total vertical force on the system was 107.4 N.
To calculate the forces on the idler stations only half of the belt was considered. Therefore, the total vertical force has to be divided by 2.

\[ F_T = F_{VT} + F_{WB} \quad (4.5) \]

- \( F_T \rightarrow \) Total vertical force
- \( F_{VT} \rightarrow \) Force calculated in EDEM®
- \( F_{WB} \rightarrow \) Force caused by the weight of the belt

\[ F_{VT} = 107.4 \, N \quad (4.6) \]

\[ F_{WB} = 1522 \, kg/m^3 \cdot 3m \cdot 0.2m \cdot 0.01 = 9.132 \, kg = 91.32 \, N \quad (4.7) \]

Dividing \( F_T \) over 2 gives the following result:

\[ \frac{F_T}{2} = \frac{107.4 + 91.32}{2} = 99.36 \, N \quad (4.8) \]

This force is the total vertical force on half of the belt. Therefore, the value of \( q \) obtained is:

\[ q = \frac{F_T/2}{L} = \frac{99.36N}{1.5m} = 66.24 \, N/m \quad (4.9) \]

The reaction on the idler stations (\( F \)) obtained with this load is:

\[ F = \frac{qL}{2} = \frac{66.24N/m \cdot 1.5m}{2} = 49.68 \, N \quad (4.10) \]

This result gives a total force on the central roll of 99.36 N (2F). This value is very close to the theoretical value. Hence, this method is a good approach to obtain the total vertical force on the idler roll. However, this method does not provide a complete distribution of the loads along the idler. To calculate that distribution another method like FEA should be utilized.

### 4.2. Calculation of the loads using ANSYS®

After realizing that EDEM® is not suitable for calculating the load distribution on the idlers, other options have to be considered. One of those options could be using ANSYS® for both calculating the loads on the rolls and the stresses on the belt. As it has already been explained in this report, the
output data from EDEM® consists of the nodes and elements definition, and the force applied on every node.

To calculate the loads on the idler rolls, some requisites have to be fulfilled in order to properly model the contact between the belt and the rolls. The Finite Element model imported from EDEM® consists of a belt without any other element. Therefore, the first requisite is creating a cylinder exactly where the idler station should be. The second requisite would be defining a small mesh on the cylinder. This means that the roll should be as rounded as possible. A really fine mesh could be the best solution to ease the contact between the belt and the rolls. As a consequence, the contact would be more realistic and therefore the results closer to the real values. The last thing to remark is the need of defining the new material. Building the model for the belt implies selecting the material of the belt. However, since the rolls have a different material, this new material should be defined as well.

Once the cylinder has been added a contact pair has to be created in ANSYS® Mechanical. These operations require high skills of ANSYS® APDL (the programming language of ANSYS®). After applying the required commands, the model should look close to the model shown in Fig. 4 10:

Once all these requirements have been fulfilled, the load distribution on both roll and belt can be calculated.
4.3. Conclusions

Calculating the loads on the idlers rolls using EDEM® and ANSYS® represents a real challenge. The challenging task is obtaining a correct load distribution along the rolls. EDEM® does not have the necessary features to obtain these results. Using EDEM® only the total force per idler station can be obtained.

To obtain the load distribution along the belt and the rolls a FE model is needed. The most important aspect of that model is the mesh size. A very fine mesh is required to obtain the desired results.
5. Conclusions

The main goal of this research was to explore the possibilities of coupling DEM and FEA to model a belt conveyor system. Due to the complexity of the system, and especially because of the deflection of the belt, FEA software is also necessary. EDEM® 2.6 was selected for DEM and ANSYS® 14.5 was chosen for FEA.

To perform the simulations in a realistic way, it is necessary to couple both EDEM® and ANSYS® and start an iterative process. Until now, only a one-way coupling between EDEM® and ANSYS® is available. This one-way coupling consists of generating a text file in EDEM® with all the loads on the belt at a certain time step of the simulation. This text file is written in the ANSYS® programming language (APDL). Using this exported data the Finite Element Analysis can be performed easily.

The limitations of coupling EDEM® and ANSYS® arises while trying to use the results of FEA to run a new simulation in EDEM®. The latest version of EDEM® (EDEM® 2.6) does not allow to import data from ANSYS®, and therefore a two-way coupling is not available yet.

The assumptions made throughout the report in order to achieve the steady state in the system help to overcome the issue of not being able to have a two-way coupling. The first assumption was considering a changing Elasticity Modulus over the iterative process. The second assumption was analyzing a static load to overcome the asymmetric deflection of the belt. After the last simulation a system very close to reality was obtained. Nevertheless, it would be convenient to wait for the possibility of having a two-way coupling between EDEM® and ANSYS®.

With relation to the calculation of the loads on the idlers, it has been concluded that using APDL is the most effective method. It is necessary to create a cylinder as a roll and create a contact pair to obtain accurate results.

For future research, in order to complete and perfect the research done until now, a two-way coupling between EDEM® and ANSYS® should be realized. This may be a very straightforward process, and therefore all the efforts could be focused on calculating the loads on the idlers.
References


http://www.conveyorbeltguide.com/ConveyorSafetyDevices.html

Consulted: 9 October 2013


[4] CKIT- *The Bulk Materials Handling Knowledge Base*

http://www.ckit.co.za

Consulted: 10 October 2013


http://www.mechanicalengineeringblog.com/tag/belt-conveyor-parts/

Consulted: 31 October 2013


[9] PADT, Inc. – The Blog

http://www.padttinc.com/blog/the-focus/fe-modeler-part-2-converting-a-mesh-to-geometry

Consulted: 28 May 2014

