Production cost reduction through optimization of machine component

By:
Mehdi Rezazadeh
Reza Delavar

KTH Industriell teknik och management
Industriell produktion
SE-100 44 STOCKHOLM
ACKNOWLEDGMENT

This Master of Science thesis was conducted during the spring and autumn in 2012 under supervision of Dr. Andreas Archenti and Dr. Ove Bayard at KTH Royal Institute of Technology.

We would like to thank our supervisors for providing us the opportunity to work on an industrial issue in an academic environment and also for helping us to assure academic and scientific content.

We also would like to give thanks to PHD candidates Tomas Österlind, Constantine Frangoudis and Qilin Fu for their valuable helps and continuous feedback on our work.

Finally we would like to thank Texo Company for providing us the opportunity to work on a real industrial optimization problem and supporting us during the thesis.

This study gave us a great perspective about optimization process using finite element technic and was a great experience in industrial problem solving.
Abstract

This thesis aiming to reduce the cost of production through analyzing and optimizing a set of weaving machine components including five legs; three main legs and two support legs. This set of legs has a reciprocating revolutionary movement around a central axis which is driven by a crank shaft. Finite element static Structural analysis, explicit and fatigue analyses been applied using Ansys workbench. The results show that some areas of legs are under small stresses far from material yield strength. This fact provides the potential for mass reduction of legs without significant effect on mechanical safety factors. Ansys Workbench parameter optimization and shape optimization been applied in this study in order to reduce mass while maintaining almost the same safety factors. Besides performing optimization on original legs, new optimized design alternatives presented for both main legs and support legs. Mass reduction of maximum 18% is obtained in new designs.
1 INTRODUCTION

1.1 Background

During recent years the world is experiencing an economic crisis and many companies started to lay off their human resources and to freeze their development programs to survive. These actions are to save them from bankruptcy but harm them in other ways. On one hand, these companies have invested on their employees to make them experts and skillful and by releasing them they lose their valuable human resources, on the other hand, the whole development strategies would be frozen and companies suffer from stagnancy.

An alternative solution could be to review all their processes to cut the total cost and consequently the cost per unit of products as much as possible. It can be any process in the whole supply chain; from purchasing, manufacturing, logistics and transportation, warehousing to even customer services. Among these modules, manufacturing generally and redesigning products specifically has a direct connection with cost reduction of each unit of product. To make it clear let’s consider an example. BIC co. sold 8.76 Billion stationary items in the past fiscal year. If they can redesign these products so that save them only one dollar per item on average, totally it can save them 8 billion and 760 million dollars a year.

In simplest terms, there are four actions that improve productivity and economies of scale. First, reduce the rate of cost you pay for an input. Second, reduce the inputs that do not produce output. Third, reduce unique activities or components in products and processes by redesigning the products and processes. Fourth, spread fixed cost activities over new product output. These four steps can be interpreted in shorter but more to the point sentences:

- Provide the most affordable material;
- Avoid wastes;
- Standardize and generalize the processes;
- Design a basic product and consider options with fixed costs.

It would be a great opportunity for companies to plan to redesign some products or at least some components of products in order to reduce the material cost and manufacturing cost. There is also a quality improvement opportunity to look at statistics and find the components that cause product returns. In this way the problems could be fixed through redesign process, reworks could be avoided and ultimately costs would be reduced.
Another advantage of redesign which often receives less attention is required time and human resources comparing to the design process. Redesign is a project that can be done in a shorter time and needs smaller design group, so it requires less man/hour for the whole project. It is also usual to reconsider the functionality in redesign process and add some new less costly features to make the product more effective and appealing for customers.

These advantages justify why companies should consider redesign process in their cost reduction plans.

1.2 Presentation of TEXO AB

TEXO is presented as one of the world’s leading manufacturers of weaving looms and it has been claimed that more than two thirds of all the looms for the paper industry have been supplied by TEXO.

1.3 Product Description

A loom is a device used to weave cloth, paper, carpet etc. The basic purpose of any loom is to hold the warp threads under tension to facilitate the interweaving of the weft threads. The precise shape of the loom and its mechanics may vary, but the basic function is the same. In Looms typically there is a bar mounted across the loom called reed which contains a number of slots, known as dents, which the warp threads pass through. Reed is mounted on a leg assembly which all is connected somehow to a crank shaft and through a reciprocating rotational movement hit the woven part.

1.4 Problem Definition

When a new weaving machine is designed, there are numerous mechanical properties that should be considered in order to obtain the desired running conditions. There is always a balancing act between high strength and low inertial properties to reach high speed.

In this study a set of components that are utilized in weaving machines mainly for the purpose of forming the previously woven paper or fabric is considered to be elaborately analyzed and optimized to be manufactured with less cost while maintains its quality. This set consists of three main legs and two support legs.

1.5 Thesis Objective

The main focus in this thesis is to optimize the shapes of two expensive parts, main leg and support leg, to reduce their masses and cost subsequently. At the same time safety factor and life time of these components are considered to be at least maintained.
2 Theoretical background

2.1 Structural optimization

If we consider a structure as an assembly of structural elements for sustaining a specific load or load combination, the optimization of this structure aims at designing it in a way that can sustain that load or loads with the best performance according to certain criteria. The proper definition of such criteria is of high importance. We need to measure the structure performance in terms of some items such as its weight, stiffness, displacement or stability. In the process of optimization some of these items expected to be minimized or maximized. For example usually one expects to have minimum weight with maximum stiffness for a structure but it’s obvious that this act of minimization or maximization can’t be done without having any constrains. Constrains could be number of facts such as strength, geometry or cost. Some of these items could be considered both as constrains or a desirable objective. A structural optimization problem normally is formulated by choosing one of these as an objective function and picking some others as constrains. Maximization or minimization of objective function then would be the goal. The traditional and dominant way of optimization is the iterative-intuitive one which nowadays is performed by means of computer base methods such as FEM.[3]

Depending on geometric aspects structural optimization consists of three categories:

- **sizing optimization:**

  A typical size of a structure such as thickness distribution of a sheet or beam, cross sectional area is optimized. [4]

  ![Figure1. Sizing optimization of a truss structure](image)

- **shape optimization:**

  The shape of a structure is optimized without changing the topology. In this optimization new boundaries are not formed. [4]
• **Topology optimization:**

This is the most general form of structure optimization. The topology of a structure and its shape is optimized by creating holes and new boundaries.[4]

2.2 Explicit analysis

An explicit dynamics analysis is used to determine the dynamic response of a structure due to stress wave propagation, impact or rapidly changing time dependent loads. This type of analysis can be used to model mechanical phenomena that are highly nonlinear as in our case from high speed collisions and Impact. Events with time scales of less than 1 second (usually of order 1 millisecond) are efficiently simulated with this type of analysis.[2]

When there is only one material the time step used in an explicit dynamics analysis is constrained to maintain stability and consistency via the CFL condition, that is, the time increment is proportional to the smallest element dimension in the model and inversely proportional to the sound speed in the materials used.[5]

\[
(\Delta t = \frac{L}{C})
\]

Sound speed is determined by Young’s modulus $E$ and the mass density:

\[
C = \sqrt{\frac{E}{\rho}}
\]

When an explicit Method is used, accuracy depends strongly on the time step. It is typically observed that $\Delta t$ only slightly less than critical $\Delta t$ provides excellent accuracy. The physical interpretation is that $\Delta t$ must be small enough that information does not propagate more than the distance between adjacent nodes during a single time step. Unfortunately this estimate is not conservative for all element types and it is inconvenient to calculate $L$ for all adjacent pairs of nodes. Normally in Ansys software this calculations are program controlled.[5]
Time increments are usually on the order of 1 microsecond and therefore thousands of time steps (computational cycles) are usually required to obtain the solution.[2]

The quality of the solution also is dependent and controllable by energy balance. Low energy errors (<10% of initial energy) are indicative of good quality solutions.[2]

**2.3 Fatigue analysis**

Machine elements sometimes fail under maximum stresses much below their ultimate strength of the material and quite frequently even below their material yield strength. The most important common characteristic of these machine members are that they have been under repeated stresses for a large number of times. This kind of failure is called fatigue failure.[6]

When a machine member is under a static load it normally deflects and fails because the loads on them made stresses over their yield strength. This fail is visible and they are replaced before a complete fracture happens. But when a member fails due to a fatigue phenomenon, fracture occurs suddenly and totally without any warning and it could be dangerous.

A fatigue failure starts with a small crack which is really difficult to recognize with a naked eyes and even by nondestructive tests such as radiography or magnetic tests.

In order to determine strength of material under the repetitive loads some experiments could be done on specimens of material and as a result an S-N could be achieved. An S-N diagram provides fatigue strength of material corresponding to the cycle numbers which produce failure. If the curve in this diagram becomes horizontal it means that for stresses below this value failure does not occur no matter how many times stresses are applied. The strength represented by horizontal line is called Endurance limit or Fatigue limit. It should be considered that not all materials have endurance limit under cyclic stresses. Figure 4 demonstrates a sample S-N diagram for 7075-T6 aluminum alloy circular-hole specimens which are one of the closest materials to aluminum Hokotol.
Endurance limit of machine elements may be greatly different from that of for their specimen and it’s due to many factors such as surface finish, size effect, temperature effect, notch sensitivity, reliability, and other miscellaneous effects. Thus some modifying factors ordinarily are considered in order to be in safe sides. [7]

Generally two types of fatigue stress analysis are available. Strain life analysis and stress life analysis. Strain life analysis deals with low number of repetition under $10^5$ cycles and typically concerns with crack initiation. But Stress life Analysis is based on $S$-$N$ curve and traditionally deals with high number of cycles greater than $10^5$ and is concerned with total life and not just initiation of crack. Total life is equal to crack initiation life plus crack life. [8]

In fatigue analysis different loading types are considered. Constant amplitude proportional and non-proportional loads and none constant amplitude proportional and non-proportional loads are generally four different load types. Constant proportional loads could also be categorized as zero base loads and fully reserved loads as depicted beneath. Zero base load is when load is applied, then is removed while fully reserved is when load is applied then an equal and opposite load is applied. [8]

For designing parts under above mentioned fluctuating loads different mean stress correction diagram are being used. For stress life analysis Ansys use Goodman, Soderburg, Gerber diagrams.

Among then Gerber diagram is usually a good choice for ductile material.
3 Methodology

3.1 Introduction

For this analysis we consider a set of weaving machine components including three main legs and two support legs which are mounted on a central shaft as shown in figure 6. This set of legs have a reciprocating revolutionary movement around the axis of a central shaft which is driven by another crank shaft and constrained by a longitudinal beam called reed at the top. At the end of every cyclic revolution, legs are exposed to an impulsive distributed force of 30 KN on reed and during the rotational movement they are under the effect of a centrifugal force. In this thesis these two general load conditions are studied. Since loads at the moment of contact have worse effects in terms of generated stresses, our study mainly is based on contact moment forces.
3.2 Materials

All the legs are made by Hotokol aluminum supplied by Alumeco. According to Table 1 the yield strength of this material is 532 MPa and its tensile strength is 575 MPa. For aluminum structures the yield strength and the tensile strength shall be considered according to equation 1 and equation 2 as follow [1]:

\[ f_{yd} = \frac{f_{yd}}{\gamma_m \gamma_n} = \frac{532}{1,1,1,2} = 403 \text{MPa} \quad \text{ eq 1} \]

\[ f_{ud} = \frac{f_{ud}}{1,2 \gamma_m \gamma_n} = \frac{575}{1,2,1,1,1,2} = 363 \text{MPa} \quad \text{ eq 2} \]

\( f_{yd} = \text{Yield strength} \)

\( f_{ud} = \text{Tensile strength} \)

\( \gamma_m = 1,1 \)

\( \gamma_n = 1,1 \)

This means that we shall consider tensile strength of 363 MPa for these legs and thus tensions over 363MPa are not allowed. Table 1 contains some data regarding this material which has been taken from manufacturing company’s website.
Table 1. Alumeco Hokotol aluminum properties

<table>
<thead>
<tr>
<th>Alloy</th>
<th>Min. proof stress $R_p^{0.2}$ (N/mm²)</th>
<th>Min. Tensile strength $R_m$ (N/mm²)</th>
<th>Min. Elongation AS%</th>
<th>Brinell Hardness HBS</th>
<th>Spec. Weight g/cm²</th>
<th>Tensile conductivity W/cm²</th>
<th>Thermal conductivity cm x K (20-100°C)</th>
<th>Weldability</th>
<th>Annealing cm/10 x °C</th>
<th>Fusing point/Interval °C</th>
<th>Elastic modulus KN/mm²</th>
<th>Electrical conductivity at 20°C ohm-mm²</th>
</tr>
</thead>
<tbody>
<tr>
<td>7050 Hokotol</td>
<td>532</td>
<td>575</td>
<td>7.8</td>
<td>180</td>
<td>2.83</td>
<td>1.54</td>
<td>23.5</td>
<td>bad</td>
<td>~</td>
<td>560-600</td>
<td>70.3</td>
<td>23</td>
</tr>
</tbody>
</table>

3.3 Load study

As mentioned in previous sections two important general positions of legs are the focus of this study. First position is when legs are hitting the fell and exposed to an impulsive distributed force and second position is one moment during their rotational movement. Fell is a part of fabric or paper which has been weaved previously in the weaving process.

At contact moment legs are under effect of following loads:

Main legs:

- Impulsive distributed load from fell
- Crank shaft load from a rod which is connected to the bottom end of main leg
- Pressure caused by the central shaft equal to 80 MPa due to high pressure contact
- Moments caused by support legs, transferred by the shaft

Support Leg:

- Impulsive distributed load from fell
- Pressure caused by the shaft equal to 80 MPa due to high pressure contact
- Moments caused by main legs, transferred by the central shaft

3.3.1 Reaction force calculation

Legs assembly is modeled in Ansys according to figure 7 as an indeterminate problem to find the load reactions due to distributed load from fell. The results appear as follow:
Figure 7. Load distribution and reaction forces. M.L and S.L stand for “main leg” and “support leg” respectively

\[ R_S = R_{L_i} = 25425 \text{N} \]
\[ R_{L_j} = 38137 \text{N} \]

The face area which this force acts on the support leg is calculated as follow and demonstrated in figure 8:

\[ A = 0.03 \text{m} \times 0.14 \text{m} = 0.0042 \text{m}^2 \]

So the pressure which operates on this surface is:

\[ P_{S} = P_{L_i} = \frac{25425 \text{N}}{0.042 \text{m}^2} = 6053571 \text{ Pa} = 6.05 \text{ MPa} \]
\[ P_{L_j} = \frac{38137.5 \text{N}}{0.042 \text{m}^2} = 908036 \text{ Pa} = 9.08 \text{ MPa} \]
3.3.2 Moment calculation

In addition to the reaction of forces, moments on the legs also shall be calculated. The Figure 7 shows that these moments will be distributed differently between the outer main legs (index $L_y$) and the inner main leg (index $L_i$). In the figure 7, each main leg which is located at the outer side of the structure, only runs half a support leg, while the main leg in the middle, runs a full support leg. Moments for main legs are calculated in beneath:

$$M_{L_y} = \frac{1}{2} \times R_s \times h = \frac{1}{2} \times 25425 \times 0.41 = 5212.1 \text{ Nm}$$

$$M_{L_i} = R_s \times h = 25425 \times 0.41 = 10424.3 \text{ Nm}$$

For support legs according to static equilibrium principle at the contact moment, a moment (index $M_S$) equal to $M_{L_i}$ has been considered.

$$M_S = M_{L_i} = 10424.3 \text{ Nm}$$

3.3.3 Crank shaft load calculation for main legs

According to static equilibrium principle, moment due to crank shaft load around the central shaft shall be equal to moment caused by impulsive distributed load plus moment caused by support legs. Since main legs have different moment by support legs crank shaft loads are different for outer main legs and inner main leg. The result of calculation is presented below:

$$R_{C_i} = 66436 \text{ N}$$

$$R_{C_j} = 66806 \text{ N}$$

where $R_{C_i}$ is the crank shaft load for inner main leg and $R_{C_j}$ is the crank shaft load for outer main legs.

According to this calculation outer main legs are considered as worst case for main legs and all the analysis are going to be done for outer main legs.

3.4 Load case categories

In order to refer load conditions in different analyses in this report, load is categorized as follow:

3.4.1 Load Case 1

This load case is when the legs are under rotational movement and burden a pressure equal to 80 MPa in cylindrical sections and a centrifugal force on their body due to rotation around the central shaft axis. Main legs in this condition have another load due to contact with crank shaft rod. Since in static structure analysis for rotational movement only effect of centrifugal forces on legs during
rotational movement is desired, some simplification has been implemented consist of replacing the crank shaft loads by a rotational velocity equal to 100rpm and omission of pressure inside the cylindrical area of legs. This rotational velocity is more than velocity in real working condition.

### 3.4.2 Load case 2

Between main legs, outer main legs have been considered to have worst loads at contact moment. Main leg at this point exposed to a force of 38137.5 N on the head section and a pressure of 80 MPa in the cylindrical section. Also there is a force equal to 66806 N at the lower arm hole where there is a connection between main leg and crank shaft. There is a moment equal to 5212.1 Nm around central shaft as well.

### 3.4.3 Load case 3

In this load case support legs are considered at the contact moment. A force equal to 24525 N on top of the leg, a pressure of 80 MPa and a moment equal to 10424.3 Nm in the cylindrical section are acting on the support legs. Boundary Condition

Two general boundary conditions have been considered in analyses of thesis.

### 3.4.4 Boundary condition 1

A fixed support made on the sides of central shaft as illustrated in figure 9 in order to simulate the supporting condition in the exact moment of contact when legs hit the fell at the end of its cyclic movement. By this it has been assumed that at the exact moment of contact there is no rotation by central shaft.

![Figure9. Fixed support at both sides central shaft](image)
Since there is no slide movement between shaft and legs, a rough contact between them has been defined. In rough contact model friction between central shaft and leg is infinite.

### 3.4.5 Boundary condition 2

A cylindrical support considered on shaft peripheral area. The shaft is allowed to rotate only around its own axis but in other directions is fixed. Contacts between all areas have been considered bonded solids. In this case all the solids are considered permanently joined without any relative movement or separation.

### 3.5 Meshing Process

Generally for meshing the models, the Tetrahedrons Method with Patch Independent algorithm is selected. In this method software performs meshing according to the shape of model, loadings and constrains, however user can change the size of elements in minimum size limit section. For using this method, loads and supports shall be defined before meshing process. However physics preferences for static analyses and explicit analyses are different. Mechanical and explicit physic preferences have been chosen respectively for static and explicit analyses. In figure10 a sample of meshing has been depicted for both conditions. Table no.2 and 3 demonstrates more details of meshing methods.

<table>
<thead>
<tr>
<th>Object Name</th>
<th>Patch Independent</th>
</tr>
</thead>
<tbody>
<tr>
<td>State</td>
<td>Fully Defined</td>
</tr>
<tr>
<td>Scope</td>
<td>Geometry Selection</td>
</tr>
<tr>
<td>Geometry</td>
<td>1 Body</td>
</tr>
<tr>
<td>Definition</td>
<td></td>
</tr>
<tr>
<td>Suppressed</td>
<td>No</td>
</tr>
<tr>
<td>Method</td>
<td>Tetrahedrons</td>
</tr>
<tr>
<td>Algorithm</td>
<td>Patch Independent</td>
</tr>
</tbody>
</table>
| Element Midside Nodes | Use Global Setting |}

Table 2. Mesh details for static structural analysis at the contact moment
<table>
<thead>
<tr>
<th>Object Name</th>
<th>Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>State</td>
<td>Solved</td>
</tr>
</tbody>
</table>

**Defaults**

<table>
<thead>
<tr>
<th>Physics Preference</th>
<th>Explicit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relevance</td>
<td>0</td>
</tr>
</tbody>
</table>

**Sizing**

<table>
<thead>
<tr>
<th>Use Advanced Size Function</th>
<th>Off</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relevance Center</td>
<td>Medium</td>
</tr>
<tr>
<td>Element Size</td>
<td>Default</td>
</tr>
<tr>
<td>Initial Size Seed</td>
<td>Active Assembly</td>
</tr>
<tr>
<td>Smoothing</td>
<td>High</td>
</tr>
<tr>
<td>Transition</td>
<td>Slow</td>
</tr>
<tr>
<td>Span Angle Center</td>
<td>Coarse</td>
</tr>
<tr>
<td>Minimum Edge Length</td>
<td>0.471450 mm</td>
</tr>
</tbody>
</table>

**Inflation**

<table>
<thead>
<tr>
<th>Use Automatic Inflation</th>
<th>None</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inflation Option</td>
<td>Smooth Transition</td>
</tr>
<tr>
<td>Transition Ratio</td>
<td>0.272</td>
</tr>
<tr>
<td>Maximum Layers</td>
<td>5</td>
</tr>
<tr>
<td>Growth Rate</td>
<td>1.2</td>
</tr>
<tr>
<td>Inflation Algorithm</td>
<td>Pre</td>
</tr>
<tr>
<td>View Advanced Options</td>
<td>No</td>
</tr>
</tbody>
</table>

**Patch Conforming Options**

<table>
<thead>
<tr>
<th>Triangle Surface Mesher</th>
<th>Program Controlled</th>
</tr>
</thead>
</table>

**Advanced**

<table>
<thead>
<tr>
<th>Shape Checking</th>
<th>Explicit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Midside Nodes</td>
<td>Dropped</td>
</tr>
<tr>
<td>Straight Sided Elements</td>
<td></td>
</tr>
<tr>
<td>Number of Retries</td>
<td>Default (4)</td>
</tr>
<tr>
<td>Extra Retries For Assembly</td>
<td>Yes</td>
</tr>
<tr>
<td>Rigid Body Behavior</td>
<td>Full Mesh</td>
</tr>
<tr>
<td>Mesh Morphing</td>
<td>Disabled</td>
</tr>
</tbody>
</table>

**Defeaturing**

<table>
<thead>
<tr>
<th>Pinch Tolerance</th>
<th>Please Define</th>
</tr>
</thead>
<tbody>
<tr>
<td>Generate Pinch on Refresh</td>
<td>No</td>
</tr>
<tr>
<td>Automatic Mesh Based Defeaturing</td>
<td>On</td>
</tr>
<tr>
<td>Defeaturing Tolerance</td>
<td>Default</td>
</tr>
</tbody>
</table>

**Statistics**

<table>
<thead>
<tr>
<th>Nodes</th>
<th>8561</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements</td>
<td>25852</td>
</tr>
<tr>
<td>Mesh Metric</td>
<td>None</td>
</tr>
</tbody>
</table>

Table 3. Mesh detail for explicit analysis
3.6 Static structural analysis

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads [2]. This analysis gives a general idea about how loads effect on the model assuming a static loading condition and uncovers the areas which bears maximum and minimum loads. However the safety factors provided in this model cannot be relied on. One shall consider that real loads are cyclic and affect legs in a fraction of a second. Meanwhile this analysis is the basis for the optimization process and reveals the fatigue analysis results.

Both contact moments and rotational movement has been modeled first in static condition.

3.6.1 Static structural analysis at contact moment

In this analysis loads at the contact moment has been considered as static loads. Load case 2 and 3 with boundary condition 1 and a mechanical physic preference mesh with minimum mesh size of 5mm have been considered in this analysis for both main legs and support legs.

3.6.2 Static structure analysis for rotational movement

Load case 1 with boundary condition 2 has been applied in this analysis. The same mesh as the contact moment analysis with minimum mesh size of 5mm has
been considered. The whole assembly of legs including part of the central shaft has been modeled at once to ease the simulation process.

### 3.6.3 Fatigue analysis

Since in Ansys workbench fatigue analysis has been provided as a tool in result section of static structural analysis this section is discussed under static structural analysis category. Load case 2 and 3 with boundary condition 1 and mechanical physic preference mesh are considered as input data for analysis.

In this analysis fatigue factor which is influenced by many items such as surface finish quality, size, temperature, etc. has been considered 0.61 for $10^9$ life cycles. This is the same fatigue factor that Texo model designed accordingly. Stress life analysis is selected as analysis type and Gerber mean stress criterion is selected to assess the results. Loads are defined as zero base constant proportional loads as well.

Due to Workbench limitation to define and separate repetitive and constant loads we were obliged to omit constant pressure inside the cylindrical part of legs which gives us a good approximate results however for more accurate results we need also to consider constant loads.

### 3.7 Explicit Dynamics analysis

An explicit dynamics analysis is used to determine the dynamic response of a structure due to stress wave propagation, impact or rapidly changing time-dependent loads. Momentum exchange between moving bodies and inertial effects are usually important aspects of the type of analysis being conducted. This type of analysis can be used to model mechanical phenomena that are highly nonlinear as in this case is due to impulsive nature of distributed force on reed. Events with time scales of milliseconds are efficiently simulated with this type of analysis. [2]

In this analysis 5 milliseconds after the very first moment of contact is estimated as the time interval to investigate the moment in which leg body is under the worst condition due to stress wave propagation.

In this analysis load case 2 and 3 are considered for main legs and support legs respectively however pressure is omitted. Since explicit dynamic analysis is for impact or rapidly changing time-dependent forces, having pressure which is a constant load make the results unreal. This inconsistency is due to software limitation in defining constant forces in dynamic explicit analysis which causes some inaccuracy at the area around cylindrical part of legs which could be neglected. However this model assumed the closest possible simulation. The
explicit physics preference for the meshing process with minimum mesh size 7mm and boundary condition 1 are selected.

### 3.8 Optimization process

Two general methods of structural optimization have been used in this study to find optimized geometries of support leg and main leg. These two methods called shape optimization and parameter optimization in Ansys Workbench. The results of shape optimization are not exact in sizes. Therefore they need to be optimized in parameter optimization process to find the exact size in which the geometry has highest strength with minimum mass.

Shape optimization tool in Ansys workbench has been utilized in order to find some parts of legs that could be omitted or narrowed without considerable effect on safety factor. Therefore the object of shape optimization is to reduce the weight of legs without any bad effect on strength of legs. The basis for this optimization in Ansys is static structural analysis and the same load case and meshing methods has been used. The percentage of material that one desire to be removed also shall be defined for the software.

Two shape optimization processes has been done in this project. In the first process the original legs are considered as object of optimization and some areas has been removed or thinned accordingly. In second process a block of Hokotol aluminum optimized to find new boundaries and new design for legs.

In parameter optimization a range of variation for each input parameter is defined and then according to an experiment design method which could be defined as well, the software starts to change the size of all the parameters in the ranges defined by user and analyses the object with new defined values and shows the results in a table. Then it is possible to choose the best results accordingly.

Parameters are considered to be the most influential parameters in order to maintain the safety factor or even increase it while minimizing the solid mass. These parameters are mostly in terms of thicknesses that are extruded separately from different sketches on the body of the legs. The geometry which is used for parameter optimization should be fully constrained and the parameters shall be defined so that the geometry can be automatically updated after selecting the best design point.

Since there are variations of parameters and all these parameters are in continuous ranges rather than being in discrete numbers, there are so many design points that must be considered in order to attain the best design point. This makes the calculation complicated and run time very long. Therefore “Design of Experiments” is acquired to make the design points fewer and the calculation time shorter. “Central Composite Design” is specified as the experimental design method.

The obtained results from Design of Experiments are used in response surface in order to generate an approximation for two response variables (Safety factor and Solid mass). After
analyzing the predicted results that are shown in the form of charts and figures, the direct or reverse effect of each explanatory variable on any of the response variables are specified. Then it is also possible to predict other probably proper design points that can be added manually.

Above mentioned process for parameter optimization is done all by a tool called response surface under design experiment tab in Ansys workbench, however in some cases the parameters are not known to be optimized. In this thesis this condition happened when the geometry needed to be optimized regarding fatigue loads. In this case one can choose parameters which vary during the analyses just manually and optimize them until gets desirable result.

Following flow diagram demonstrates process of optimization in this study

3.8.1 Texo Model Optimization

3.8.1.1 Texo model shape optimization

To know which exact parts of Texo models are vulnerable to be removed or narrowed, shape optimization tool in Ansys workbench has been utilized as described in 3.9. Boundary conditions and load cases for legs are the same as the structural static analysis in the contact moment however larger minimum meshing size could be applied to save the analysis time. The optimized geometry will be used as an object in parameter optimization.

3.8.1.2 Texo Model parameter optimization

According to shape optimization results a new geometry is drawn and different geometrical parameters are considered to be optimized. Load case 2 and 3 are considered for main leg and support legs respectively and boundary condition 2 is considered for modeling the legs. To save the time one can use larger mesh size in meshing process. Mechanical physics preference is a proper choice for meshing.

Parameters are mostly defined as thickness of different areas which are isolated via sketches when the geometry is drawn. These areas are shown for both legs in figure 11. The initial values for both legs are represented in table 5. With these initial parameter values, minimum safety factor for support leg and main leg
was 1.32 and 1.28 while the solid mass was around 12.9 Kg and 21.3 Kg for support legs and main legs respectively.

![Sample areas that their thickness has been consider as parameters for optimization](image)

**Table 4. Parameters’ Initial Values**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Initial Value (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_1$</td>
<td>105</td>
</tr>
<tr>
<td>$T_2$</td>
<td>30</td>
</tr>
<tr>
<td>$T_3$</td>
<td>10</td>
</tr>
<tr>
<td>$T_4$</td>
<td>55</td>
</tr>
<tr>
<td>$T_5$</td>
<td>20.5</td>
</tr>
</tbody>
</table>

### 3.8.2 Redesign process

To redesign the legs a block of aluminum, Figure no.12, with the same dimension with the block that Texo model has been designed considered as raw material for design however there are five major limitations that shall be considered in the redesign process of legs which are discussed as follow:

- **Cylindrical part thickness:** This thickness shall be remained constant to keep the same frictional force of contact. Otherwise the pressure inside the cylindrical part shall be proportionally changed to avoid any probable rotational sliding.

- **Head geometry:** It remained intact to avoid any subsequent change in the reed geometry which is out of scope of this study.

- **Block dimension:** This dimension was limited to the dimension of hypothetical cubic boundary which original legs are surrounded with. Otherwise more material could be wasted during the machining procedure even with lighter redesigned legs.

- **Reed to shaft distance:** The distance between the central shaft and the reed remained the same to avoid subsequent changes to machine structure.

- **Rod grip size:** Since the design of crank shaft rod is out of scope of this thesis, distance between two lower arms which is 64mm should remain fixed.
3.8.2.1 **Redesigned model shape optimization**

In this stage a block of Hokotol aluminum optimized to find new boundaries and new design for legs. Different percentage for material removal has been considered during the optimization process. The same condition is applied for load case, boundary condition and meshing as applied for Texo model shape optimization.

3.8.2.2 **Redesigned model parameter optimization**

According to shape optimization results a new geometry is drawn and different geometrical parameters are considered to be optimized. The same load case, boundary condition and mesh as parameter optimization for Texo model is applicable here.

Main parameters in this optimization are thicknesses as well however some blend radii are selected to be optimized manually.

3.9 **Optimization assessment**

After finalizing the optimization process, models are needed to be assessed again for explicit dynamic analyses and fatigue analyses. In case results are not satisfactory the design has to be changed until satisfactory results are achieved. In this stage static structure analyses and explicit dynamic for contact moment are performed again to ensure design reliability.
4 Results and discussion

4.1 Static structural analysis

4.1.1 Static structural analysis at contact moment

4.1.1.1 Final results

Three major aspects of total deformation, equivalent stress, and safety factor are considered to be investigated in this study. The results for support legs and main legs are depicted in following figures:

- **Total Deformation**

  Maximum deformation of 0.69 mm and 0.82 is calculated for support legs and main legs respectively.

![Figure13. Static structure total deformation Result](image)

- **Equivalent Stress**

  Maximum Von Misses stress of 273.5 MPa for support legs and 282.2 MPa for main legs is calculated which demonstrated in figure 7.
• **Safety Factor**

A safety factor of 1.32 is obtained for initial model of support leg. This factor for main legs is 1.28.

It has been shown in the analysis results that there are some areas mostly between top end and the area above cylindrical part of the leg which are under low stresses. Therefore mass reduction in such areas seems to be feasible.
• Fatigue safety factor

Fatigue safety factor of 1 is obtained for initial model of support leg. This factor for main legs is 1.01.

Figure 16. Fatigue safety factor

4.1.2 Static structure analysis for rotational movement

4.1.2.1 Final results

• Total deformation

The total deformation in this case could be neglected according to the Ansys results.

Figure 17. Assembly deformation in rotational movement analysis
• **Equivalent stress**

According to the obtained results, the effect of centrifugal forces on legs body is almost negligible. The maximum Von Mises stress due to pure rotational movement is less than 20 MPa.

![Figure18. Assembly rotational movement stress analysis results](image1)

• **Safety factor**

Since there are no considerable tensions due to rotation of legs, the safety factor under this condition is more than 10.

![Figure19. Safety factor for rotational movement stress analysis](image2)
4.2 Explicit Dynamics analysis

4.2.1 Final results

- **Total deformation**

  Maximum deformation of 0.96 mm and 0.75 is calculated for support leg and main leg respectively.

  ![Figure 20. Total deformation in explicit stress analysis](image)

- **Equivalent stress**

  Maximum tensile stress of 116.48 and 157.08 MPa is shown as Von Mises stress.
Safety factor

Since magnitude of produced stresses due to Impact load at the contact moment in upper part of the leg above cylindrical part are still less than static stresses around cylindrical areas, safety factor shall be considered the same as safety factor in static structural analysis which is equal to 1.32 for support leg and 1.28 for main leg.
4.3 Optimization process

4.3.1 Texo Model Optimization

4.3.1.1 Texo model shape optimization
Shape optimization tool in Ansys workbench is used for original Texo models. These results provide us a general idea about potential areas that could be targeted for material removal.

![Shape optimization result in Ansys workbench after material removal](image)

Figure 23. shape optimization result in Ansys workbench after material removal

4.3.1.2 Texo Model parameter optimization
Following picture and table demonstrate defined parameters and optimization results of parameter optimization for main leg. Thickness of defined areas has been considered as parameters. Shape optimization results provide us the idea about potential areas. The results for different parameters in best design point are demonstrated in table …. In this design point the mass was 17.4 Kg.
Figure 24. Main leg parameter definition

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Initial size (mm)</th>
<th>Defined range (mm)</th>
<th>Optimized size (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>105</td>
<td>90-105</td>
<td>82.5</td>
</tr>
<tr>
<td>T2</td>
<td>30</td>
<td>26-30</td>
<td>23</td>
</tr>
<tr>
<td>T3</td>
<td>10</td>
<td>7-10</td>
<td>8</td>
</tr>
<tr>
<td>T4</td>
<td>55</td>
<td>48-55</td>
<td>42.5</td>
</tr>
<tr>
<td>T5</td>
<td>20.5</td>
<td>15-20.5</td>
<td>20.5</td>
</tr>
</tbody>
</table>

Table 2. Parameter initial and optimized sizes

Picture no.25 demonstrates the parametrically optimized support leg. In this case parameter optimization has been done manually rather than using parameter optimization tool in Ansys workbench. The optimized model is 0.8 kg lighter than original model.
4.3.1.3 Response surface

Figure no.26 demonstrates sensitivity of response variables to parameters of main leg.

Figure26. Local sensitivity chart demonstrating response variables dependency on parameters for main leg
4.3.2 Redesign process

4.3.2.1 Redesigned model shape optimization result

The removed areas in figure no.27 show the parts of material which possibly can be removed and has the less impact on model stiffness.

Figure 27. Shape optimization result in Ansys workbench after material removal

In Figure no.28 one can see design evolution for support leg using shape optimization tool:

Figure 28. Redesign evolution stages
4.3.2.2 *Redesigned model parameter optimization*

Figure no.29 presents redesigned main leg and support leg which are parametrically optimized.

![Redesigned legs after parameter optimization](image)

Following pictures and tables demonstrate defined parameters and optimization results of parameter optimization for redesigned main leg and support. Thickness of defined areas has been considered as parameters. The results for different parameters in best design point are demonstrated in table no.6 and 7.

![Parameter definition for redesigned main leg](image)
Figure 31. Parameter definition for redesigned support leg

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Initial size (mm)</th>
<th>Defined range (mm)</th>
<th>Optimized size (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>105</td>
<td>90-105</td>
<td>91.4</td>
</tr>
<tr>
<td>T2</td>
<td>105</td>
<td>90-105</td>
<td>105</td>
</tr>
<tr>
<td>T3</td>
<td>20.5</td>
<td>13-20.5</td>
<td>19.8</td>
</tr>
</tbody>
</table>

Table 6. Main leg parameter sizes before and after optimization

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Initial size (mm)</th>
<th>Defined range (mm)</th>
<th>Optimized size (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>12.5</td>
<td>7.5-17.5</td>
<td>12.5</td>
</tr>
<tr>
<td>T2</td>
<td>17.5</td>
<td>12.5-20</td>
<td>17.5</td>
</tr>
</tbody>
</table>

Table 7. Support leg parameter sizes before and after optimization
4.4 Optimization assessment

4.4.1 Static structural analysis at contact moment (Modified)

4.4.1.1 Final results

- **Total deformation**

  Maximum deformation of 0.96 mm and 0.76 is calculated for optimized support leg and main leg respectively.

![Figure 32. Static structural deformation analysis result for optimized original legs](image)

- **Equivalent stress**

  Maximum tensile stress of 279.86 and 269.68 MPa is shown as Von Mises stress.
• **Safety factor**

A safety factor of 1.34 is obtained for optimized model of original support leg. This factor for optimized main legs is 1.29.
• **Fatigue safety factor**

Fatigue safety factor of 1.1 is obtained for optimized model of original support leg. This factor for optimized main legs is 1.01.

![Fatigue safety factor for optimized original legs](image)

**Figure 3.4. Fatigue safety factor for optimized original legs**

### 4.4.2 Static structural analysis at contact moment (Redesigned)

#### 4.4.2.1 Final results

• **Total deformation**

Maximum deformation of 0.47 mm and 1.03 mm is calculated for optimized support leg and main leg respectively.
Figure 35. Static structural deformation analysis for redesigned legs

- **Equivalent stress**

  Maximum tensile stress of 275.55 MPa and 261.15 MPa is calculated as Von Mises stress for redesigned legs

Figure 36. Static structural stress analysis for redesigned legs
• **Safety factor**

A safety factor of 1.39 is obtained for redesigned model of support leg. This factor for optimized main legs is 1.31.

![Figure37. Static structural safety factor for redesigned legs](image)

• **Fatigue safety factor**

Fatigue safety factor of 1.9 is obtained for redesigned model support leg. This factor for redesigned main legs is 1.01.

![Figure38. Fatigue safety factor for redesigned legs](image)
4.5 Result summary

Following table summarizes results for all the models:

<table>
<thead>
<tr>
<th>ANALYSIS RESULTS</th>
<th>STATIC STRUCTURE ANALYSIS-CONTACT MOMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>MAIN LEG</td>
</tr>
<tr>
<td>TEXO MODEL</td>
<td></td>
</tr>
<tr>
<td>TOTAL DEFORMATION (mm)</td>
<td>0.82</td>
</tr>
<tr>
<td>EQUIVALENT STRESS (Mpa)</td>
<td>282.2</td>
</tr>
<tr>
<td>SAFETY FACTOR</td>
<td>1.28</td>
</tr>
<tr>
<td>FATIGUE SAFETY FACTOR</td>
<td>1.01</td>
</tr>
<tr>
<td>TEXO MODEL-OPTIMIZED</td>
<td></td>
</tr>
<tr>
<td>TOTAL DEFORMATION (mm)</td>
<td>0.76</td>
</tr>
<tr>
<td>EQUIVALENT STRESS (Mpa)</td>
<td>279.9</td>
</tr>
<tr>
<td>SAFETY FACTOR</td>
<td>1.29</td>
</tr>
<tr>
<td>FATIGUE SAFETY FACTOR</td>
<td>1.01</td>
</tr>
<tr>
<td>REDESIGNED TEXO MODEL-OPTIMIZED</td>
<td></td>
</tr>
<tr>
<td>TOTAL DEFORMATION (mm)</td>
<td>1.03</td>
</tr>
<tr>
<td>EQUIVALENT STRESS (Mpa)</td>
<td>275.5</td>
</tr>
<tr>
<td>SAFETY FACTOR</td>
<td>1.31</td>
</tr>
<tr>
<td>FATIGUE SAFETY FACTOR</td>
<td>1.01</td>
</tr>
</tbody>
</table>

Table 8. Summarized results for all the models

5 Conclusion and further studies

5.1 Conclusion

According to results, almost 4 Kg in the main leg and 2 Kg in support legs can be saved. In addition to mass reduction in redesign process, almost 475 Kg mass reduction, due to having thinner block as raw material, can be achieved in a single machine subject to TEXO’s decision for consequential changes in reed design. Equivalent stresses in optimized models are lower than initial Texo models and fatigue safety factors kept either the same or higher which means same or higher reliability for optimized models. Both redesigned and optimized models need more material removal as well as very high precision and complicated machining in compare to the original models, which result in more expensive production process, however authors believe mass reduction after optimization especially for redesigned models compensates extra machining cost.
Tables No.8 and No.9 summarize and compare results of optimization.

### Table 9. Main leg’s summarized optimization results

<table>
<thead>
<tr>
<th></th>
<th>Mass</th>
<th>Static safety factor</th>
<th>Fatigue safety factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>TEXO model</td>
<td>21.3</td>
<td>1.28</td>
<td>1.02</td>
</tr>
<tr>
<td>TEXO optimized model</td>
<td>17.4</td>
<td>1.21</td>
<td>1.01</td>
</tr>
<tr>
<td>Redesigned model</td>
<td>17.3</td>
<td>1.14</td>
<td>1.03</td>
</tr>
</tbody>
</table>

### Table 10. Support leg’s summarized optimization results

<table>
<thead>
<tr>
<th></th>
<th>Mass</th>
<th>Static safety factor</th>
<th>Fatigue safety factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>TEXO Model</td>
<td>12.9</td>
<td>1.32</td>
<td>1.01</td>
</tr>
<tr>
<td>TEXO optimized model</td>
<td>12.1</td>
<td>1.34</td>
<td>1.096</td>
</tr>
<tr>
<td>Redesigned model</td>
<td>10.9</td>
<td>1.38</td>
<td>1.9</td>
</tr>
</tbody>
</table>
5.2 Further studies

In order to achieve more accurate results authors bring following suggestions:

- In explicit dynamic analysis contact time estimated equal to 0.003 which can be measured experimentally. Another limitation in explicit analysis was due to Ansys workbench limitation for simulating the models with constant pressure in cylindrical section of legs. Other software can be used to solve this problem.

- In fatigue analysis same limitation as in explicit analysis regarding constant pressure inside the cylindrical section of legs prevented authors to achieve accurate results. Other software could be used to solve this problem.

The results of this study need to be assessed and verified by manufacturing optimized models and doing experimental studies.
6 References


[8] Predicting Fatigue Life with ANSYS Workbench, Ansis, Inc. 2006, Raymond L. Browell, P. E.