FE safety analysis 
of a high speed wood planer cutter

An alternative method to achieve the requirements of EN847 standard

Francisco José Marques Novo
Henrique Manuel Saraiva Rézio

LITH-IKP-PR-- 04/02 --SE

Supervisor: Stefan Björklund

Linköping, June 2004
Abstract

In the last decades, high speed cutting has become an attractive technology in the wood industry. The possibility of reducing global costs in addiction with an increase in productivity, were the main reasons for the enlargement of the use of this technology.

As usual, these advantages are accompanied by disadvantages that should be carefully analyzed. If on the one hand cutting forces are reduced with increasing cutting speeds, on the other hand, the centrifugal forces affecting the tool are higher. Exposed to such high loads, there is a considerable risk of tool failure that embeds hazards for both machine and workers.

To prevent the risk of accidents and to guarantee safety in use, security standards have been implemented in industrial fields, imposing specific experimental tests, with defined procedure modes. Accordingly with these standards, the results obtained through the tests should fall inside limited ranges. The experimental tests suggested on the European Standards are intended to simulate the real working conditions of a rotating cutting tool, where extreme centrifugal forces are imposed by the high values of speed. Although their main importance, these destructive tests aren’t always practicable. It happens, for instance, with tools produced in small batch sizes, or as an ascertainment for the fail-critical speed during the development stage, or even due to physical incompatibilities between the tool and the laboratory test machines.

The high value of weight associated with the cutting tool prototype developed and patented by Verktygs Larsson AB was an impediment to run the laboratorial tests specified by the standards, forcing the company to find a new way to assure the safety requirements of their product.

The main goal of this project was the development of an alternative method based on finite element theory to perform a safety analysis to the prototype of a wood cutter. This tool is used as a component in high speed planers.

In addition to this primary objective, some considerations were made about other available models, with increased dimensions or even with different parameters. If there was the need, design changes could be assumed in order to guarantee that the tool reached the requisites of the safety standards. Considering an optimization effort, material changes would also be considered, to aim in the direction of reducing the tool weight and the consequent centrifugal forces.
Abstract

In the last decades, high speed cutting has become an attractive technology in the wood industry. The possibility of reducing global costs in addiction with an increase in productivity, were the main reasons for the enlargement of the use of this technology. As usual, these advantages are accompanied by disadvantages that should be carefully analyzed. If on the one hand cutting forces are reduced with increasing cutting speeds, on the other hand, the centrifugal forces affecting the tool are higher. Exposed to such high loads, there is a considerable risk of tool failure that embeds hazards for both machine and workers.

To prevent the risk of accidents and to guarantee safety in use, security standards have been implemented in industrial fields, imposing specific experimental tests, with defined procedure modes. Accordingly with these standards, the results obtained through the tests should fall inside limited ranges. The experimental tests suggested on the European Standards are intended to simulate the real working conditions of a rotating cutting tool, where extreme centrifugal forces are imposed by the high values of speed. Although their main importance, these destructive tests aren’t always practicable. It happens, for instance, with tools produced in small batch sizes, or as an ascertainment for the fail-critical speed during the development stage, or even due to physical incompatibilities between the tool and the laboratory test machines.

The high value of weight associated with the cutting tool prototype developed and patented by Verktygs Larsson AB was an impediment to run the laboratorial tests specified by the standards, forcing the company to find a new way to assure the safety requirements of their product.

The main goal of this project was the development of an alternative method based on finite element theory to perform a safety analysis to the prototype of a wood cutter. This tool is used as a component in high speed planers.

In addiction to this primary objective, some considerations were made about other available models, with increased dimensions or even with different parameters. If there was the need, design changes could be assumed in order to guarantee that the tool reached the requisites of the safety standards. Considering an optimization effort, material changes would also be considered, to aim in the direction of reducing the tool weight and the consequent centrifugal forces.
Acknowledgements

This project was performed at the Department of Mechanical Engineering of the Linköping University, Production System Division, between March and June 2004, during our stay in Sweden as exchange students.

This thesis would not have been possible without the help and cooperation of several persons. We would like to express our most sincere gratitude to everyone who made our stay so pleasant and educational:

Our supervisor, Universitetslektor Stefan Björklund, for introducing us in the project and for all the help and cooperation throughout this thesis.

Sergio Silva, for being so proud of our work, supporting us all the time with his knowledge and helpful engineering clues. We will never forget the endless conversations in Portuguese and his everlasting kindness and friendship.

Verktygs Larsson AB responsible Lennart Andersson and designer Anders Engman, for introducing us in this amazing industrial field and for offering their enormous technical knowledge.

All the researchers at Linköping University who were open for discussion concerning CAD/CAE and analysis subject, namely Matz Lenner and Vitalij Savin.

Professors Mourão Dias and Cristovão Silva, for all the efforts in our exchange program.

Our new friends from all over the world for the moments we spent together, that will never vanish from our memories.

Our families, namely parents, brothers and sisters, for all their remarkable support, that became the key for the success in the most valuable experience of our student life that is now reaching the end.

All the big friends from Portugal who never forgot that we were abroad, tracking us constantly and making us fell not so far way from our home land.

Linköping, June 2004

Francisco Marques and Henrique Rézio
# Table of Contents

Abstract ............................................................................................................................... 1  
Acknowledgements .............................................................................................................. 2  
Table of Contents ............................................................................................................... 3  
List of Exhibits and Tables ............................................................................................... 5  

## INTRODUCTION ........................................................................................................... 7  
1.1 Background .................................................................................................................. 7  
1.2 The company ............................................................................................................... 8  
1.3 Problem description ..................................................................................................... 8  
1.4 Purpose and objectives ............................................................................................... 9  
1.5 Methods ..................................................................................................................... 9  
1.6 Delimitations ............................................................................................................. 10  
1.7 Thesis report structure ............................................................................................. 10  

## HIGH SPEED WOOD CUTTING TECHNOLOGY ......................................................... 11  
2.1 Definition and historical review .................................................................................. 11  
2.2 Machining Processes .................................................................................................. 12  
2.3 Safety ....................................................................................................................... 13  
2.4 Certification ............................................................................................................... 14  

## CUTTING TOOL PROTOTYPE ............................................................................... 16  
3.1 Mechanism presentation ............................................................................................ 16  
3.2 Specifications ............................................................................................................ 20  
3.3 Definition of the cutter according to EN847/1 Standard ............................................ 21  

## FINITE ELEMENT METHOD .................................................................................. 23  
4.1 Introduction ............................................................................................................... 23  
4.2 Fundaments ............................................................................................................. 23  
4.3 Considerations about Finite Element Softwares ....................................................... 25
MODELING..........................................................................................................................26

5.1 Introduction .........................................................................................................................26
5.2 Static analysis .......................................................................................................................26
5.3 Evolution on the geometry of the model ............................................................................27
5.4 Assembly constraints .........................................................................................................29
5.5 Analysis connections .........................................................................................................29
5.6 Loads ..................................................................................................................................30
5.7 Element and Mesh Properties .............................................................................................37

RESULTS ..................................................................................................................................39

6.1 Model with one blade applied to the first geometry ...........................................................39
6.2 Model with two blades applied to the first geometry ........................................................44
6.3 Model with one blade applied to the second geometry ......................................................47

CONCLUSIONS .......................................................................................................................50

DISCUSSION .............................................................................................................................52

APPENDIX ...............................................................................................................................54

LIST OF REFERENCE ...............................................................................................................77
List of Exhibits and Tables

Exhibit 1: High Speed Wood Planer 13
Exhibit 2: Combinations for the cutter heads 13
Exhibit3: Back side ramps in the main body 17
Exhibit 4: Components of the cutter 17
Exhibit 5: Blade assembling 18
Exhibit 6: Screws assembling 18
Exhibit 7: Tightening wedge assembling 19
Exhibit 8: Tightening the screws 19
Exhibit 9: Dynamometric wrench tightening the screws 19
Table 1: Cutting tool available dimensions 20
Table 2: Properties of the adopted materials 21
Exhibit 10: The first models 28
Exhibit 11: The final models 28
Table 3: Masses of some components 31
Table4: Centrifugal forces acting on the support wedge and on the blade 32
Exhibit 12: Axis systems used to apply the sensors 34
Table 5: Normal reactions (Ry) in the contact surfaces 34
Exhibit 13: Sensors and the respective contact surfaces 35
Exhibit 14: The distributed friction forces 36
Exhibit 15: Tetrahedron element 37
Exhibit 16: Global mesh 40
Exhibit 17: Deformed mesh 40
Exhibit 18: Displacements in the main body 40
Exhibit 19: Displacements in the support wedge 41
Exhibit 20: Displacements in the blade 41
Exhibit 21: Global Von Mises Stresses 42
Exhibit 22: Von Mises Stresses in the main body 42
Exhibit 23: Von Mises Stresses in the support wedge 43
Exhibit 24: Von Mises Stresses in the blade 43
Exhibit 25: Von Mises Stresses in the tightening wedge 43
Exhibit 26: Estimated local errors in the main body 44
Exhibit 27: Global mesh 44
Exhibit 28: Deformed mesh 45
Exhibit 29: Displacements in the main body 45
Exhibit 30: Displacements in the support wedge 45
Exhibit 31: Displacements in the blade 46
Exhibit 32: Estimated local error in the main body 46
Exhibit 33: Deformed mesh 47
Exhibit 34: Displacements in the main body 47
Exhibit 35: Displacements in the support wedge 48
Exhibit 36: Displacements in the blade 48
Exhibit 37: Global stresses 49
Exhibit 38: Estimated local error in the main body 49
Exhibit 1: Global mesh 55
Exhibit 2: Deformed mesh 55
Exhibit 3: Global displacements 56
Exhibit 4: Displacements in the main body 56
Exhibit 5: Displacements in the blade 57
Exhibit 6: Displacements in the support wedge 57
Exhibit 7: Displacements in the tightening wedge 58
Exhibit 8: Displacements in the screw 58
<table>
<thead>
<tr>
<th>Exhibit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>Global stresses</td>
</tr>
<tr>
<td>10</td>
<td>Stresses in the main body</td>
</tr>
<tr>
<td>11</td>
<td>Stresses in the blade</td>
</tr>
<tr>
<td>12</td>
<td>Stresses in the support wedge</td>
</tr>
<tr>
<td>13</td>
<td>Stresses in the tightening wedge</td>
</tr>
<tr>
<td>14</td>
<td>Stresses in the screw</td>
</tr>
<tr>
<td>15</td>
<td>Estimated local errors</td>
</tr>
<tr>
<td>16</td>
<td>Estimated local error in the main body</td>
</tr>
<tr>
<td>17</td>
<td>Estimated local error in the blade</td>
</tr>
<tr>
<td>18</td>
<td>Estimated local error in the support wedge</td>
</tr>
<tr>
<td>19</td>
<td>Estimated local error in the tightening wedge</td>
</tr>
<tr>
<td>20</td>
<td>Estimated local error in the screw</td>
</tr>
<tr>
<td>21</td>
<td>Global mesh</td>
</tr>
<tr>
<td>22</td>
<td>Deformed mesh</td>
</tr>
<tr>
<td>23</td>
<td>Displacements in the main body</td>
</tr>
<tr>
<td>24</td>
<td>Displacements in the blade</td>
</tr>
<tr>
<td>25</td>
<td>Displacements in the support wedge</td>
</tr>
<tr>
<td>26</td>
<td>Displacements in the tightening wedge</td>
</tr>
<tr>
<td>27</td>
<td>Displacements in the screw</td>
</tr>
<tr>
<td>28</td>
<td>Stresses in the main body</td>
</tr>
<tr>
<td>29</td>
<td>Stresses in the blade</td>
</tr>
<tr>
<td>30</td>
<td>Stresses in the support wedge</td>
</tr>
<tr>
<td>31</td>
<td>Global mesh</td>
</tr>
<tr>
<td>32</td>
<td>Displacements in the main body</td>
</tr>
<tr>
<td>33</td>
<td>Displacements in the blade</td>
</tr>
<tr>
<td>34</td>
<td>Displacements in the support wedge</td>
</tr>
<tr>
<td>35</td>
<td>Displacements in the tightening wedge</td>
</tr>
<tr>
<td>36</td>
<td>Displacements in the screw</td>
</tr>
<tr>
<td>37</td>
<td>Stresses in the main body</td>
</tr>
<tr>
<td>38</td>
<td>Estimated local error in the main body</td>
</tr>
<tr>
<td>39</td>
<td>Estimated local error in the blade</td>
</tr>
<tr>
<td>40</td>
<td>Estimated local error in the support wedge</td>
</tr>
<tr>
<td>41</td>
<td>Estimated local error in the tightening wedge</td>
</tr>
<tr>
<td>42</td>
<td>Estimated local error in the screw</td>
</tr>
</tbody>
</table>
INTRODUCTION

In this chapter the background, the company we were working for, problem description, purpose and objectives, used methods, delimitations and the structure of thesis are presented.

1.1 Background

In the last decades, high speed cutting has become an attractive technology in the wood industry. The possibility of reducing global costs in addiction with an increase in productivity, were the main reasons for the enlargement of the use of this technology.

As usual, these advantages are accompanied by disadvantages that should be carefully analyzed. If on the one hand cutting forces are reduced with increasing cutting speeds, on the other hand, the centrifugal forces affecting the tool are higher. Exposed to such high loads, there is a considerable risk of tool failure that embeds hazards for both machine and workers.

To prevent the risk of accidents and to guarantee safety in use, security standards have been implemented in industrial fields, imposing specific experimental tests, with defined procedure modes. Accordingly with these standards, the results obtained through the experimental tests should fall inside limited ranges.
1.2 The company

Founded by Gustav Larsson in 1975, the main activity of *Verktygs Larsson AB* was the production of tools for several applications. In 1983 the founder sold the company to the actual owner, Lennart Anderson, who continued a history of success. The company is located in Nyköping, Sweden. It develops mechanical products to several types of industries and their production includes special designed and patented high speed wood cutters, as well as tools for machines used in mass production lines. It is one of the biggest suppliers of *ABB*, developing several hydraulic components to their robotic products. The annual turnover of the company is approximately 10 million SEK.

1.3 Problem description

All wood cutting tools entering the market for the first time shall be tested in order to verify their compatibility with the security standards. This is a major step to guarantee that safety in use is reached during the tool life.

The experimental tests suggested on the European Standards are intended to simulate the real working conditions of a rotating cutting tool, where extreme centrifugal forces are imposed by the high values of speed. But, besides their main importance, these destructive tests aren’t always practicable. It happens, for instance, with tools produced in small batch sizes, or as an ascertainment for the fail-critical speed during the development stage, or even due to physical incompatibilities between the tool and the laboratory test machines. In addiction, during the development period of a new tool, experimental tests aren’t the best way to support the optimization process due to the high cost and the need of immediate answers. Besides, the results obtained with a specific test cannot be extrapolated to all row of models, with different dimensions, even if they are based on the same primitive geometry.

The high value of weight associated with a cutting tool prototype developed and patented by *Verktygs Larsson AB* was an impediment to run the laboratorial tests specified by the standards, forcing the company to find a new way to assure the safety requirements of the product.
Therefore, other evaluation and analysis methods have to be developed in order to find the answers to this complex problem.

1.4 Purpose and objectives

The main goal of this project is the development of an alternative method based on finite element theory to perform a safety analysis to the prototype of the wood cutter. This tool is used as a component in high speed planers.

There are several models available for production, all of them based on the same primitive geometry. Due to the design of the mechanism, the analysis we intend to perform becomes extremely complex, and therefore our main study will be pointed to a specific geometry, in order to try to understand the variables that are affecting its behaviour, and how important they are to the mechanism itself.

In addition to this primary objective, we will also make some considerations about other available models, with increased dimensions or even with different parameters. If there is the need, design changes can be assumed in order to guarantee that the tool reaches the requisites of the safety standards. Considering an optimization effort, material changes will also be considered, to aim in the direction of reducing the tool weight and the consequent centrifugal forces.

1.5 Methods

To get in contact with the subject of the project and to acquire a practical knowledge of the prototype, several visits were made to the company that produces and owns the patent of this new cutting tool. In addition, the correspondence with the experts at the company was the key to get important information and specifications needed to engage our work.

Besides, extensive literature research was carried in order to allow us a better theoretical understanding of the subject. An intensive study of CATIA user’s manuals was also carried to enter in its world, since it was our first contact with it.
Nowadays, the number of FE softwares available is large and their use is becoming easier, combining design and analysis tools. However, results need always a carefully and specialized interpretation.

1.6 Delimitations

Certain delimitations regarding the development of the thesis need to be focused. The first one is that we converged since the beginning to softwares that combine Computer Aided Design and Computer Aided Engineering tools, like CATIA and ProENGINEER, putting apart pure finite element solvers.

Besides this fact, CATIA version used during our work was an educational version, presenting limitations related with some analysing parameters.

As usual, the time imposed to a project is a crucial factor. In this case, the time forced us to perform some primary decisions in order to assure that the objectives would be achieved within the deadline.

1.7 Thesis report structure

This report will first give an introduction into the thesis topic including objectives and methods. Chapter 2 will present a comprehensive insight into the field of high speed wood cutting, with special reference to the security standards that guide the design of cutting tools. The presentation of the cutter and the mechanism explanation are presented in Chapter 3. After these introductory chapters, chapter 4 follows with considerations related with FE theory. Chapter 5 presents the models and assumptions about the analysis that were performed. The results obtained during this project are shown in chapter 6. Chapter 7 resumes and interprets the results of this thesis work. Finally, the Appendix contains all the output data obtained.
HIGH SPEED WOOD CUTTING TECHNOLOGY

In this chapter the background of the studied cutting tool is focused, providing some definitions, considerations about machining processes as well as an introduction to safety standards and certification procedure.

2.1 Definition and historical review

Machining of wood is probably the oldest technology for shaping and finishing man made objects. It is seen as the process of manufacturing solid wood products and products from wood base panels. The objective of this technology is to produce a desired shape and dimension with requisites of accuracy and surface quality in the most economical way.

Modern equipment for machining wood is notable for its high operating speed and high productivity. Generally, the term “high speed” is used when cutting and feed speeds are greater than classical with a factor from five to ten. Tools can rotate in spindles up to 40000 rpm and work piece feed speed can reach nowadays 600 m/min.

Working with such high velocities has advantages in terms of best quality of surface, lower cutting forces and an increased yield capacity.

During the last 40 years, wood machining has been characterized by fundamental technical developments and structural changes. Waves of automatization and continuous machining lines of sixties were followed by sophisticated machining
concepts, such as computer numerical controlled stationary machining, where the value of the *software* part is greater than that of the *hardware* of the machine. But despite of the growing importance of electronics and *software* structures, the development of processes and tools has not reached an end yet. Optimization of wood machining processes are constant, involving attempts to reduce losses of machined material and wear of cutting tools, improve the accuracy of dimensions and surface quality, increase production output, reduce cost and guarantee worker safety.

### 2.2 Machining Processes

Machining processes in the manufacture of wood products can be divided in the following categories: sawing, rotary cutting and slicing, planing, molding, shaping and routing, turning, boring and sanding technology. Besides, there are non traditional machining processes as cutting with laser beams, high velocity liquid jets and vibration cutters.

One of the principal objectives of a mechanical process such as planing, is to obtain an acceptable finished surface. Planed products should have a surface free from defects to obtain a maximum price. Surface defects may result in lower prices, or main even render the product not to be sold on the market. The quality of planing depends on the wood characteristics, cutting tool geometry, and on the operational conditions of the machine. The quality level may be based on esthetical or technical requirements and expressed in terms of damage caused to the wood surface during the planing process.

As we refered in the previous chapter, the cutting tool that is analysed in this thesis is a main component in high speed wood planers. These are common machines used in the wood industry for planing wood surfaces. They remove single chips from a workpiece using the rotating movement of several blades on the periphery of a rotating cutterhead. Each machine available on the market has the possibility of containing a very large range of cutters, that can appear in different positions, vertically or horizontally, and combinations along the planning area, to fulfil certain requirements of the final product. Due to the high energies involved in the planing process, in case of a hazard, it can come to increase damages and decrease the opportunities of the operator to escape. Possible hazards are the emission of high
energy masses, as fragments of work pieces or tools and chips and the reduction of the operator’s reaction time in case of machine failure. Therefore, this is the main reason why high speed planers are encapsulated and built with machine guards.

Exhibits 1 and 2 represent one of these planers available on the market with some possible combinations for the cutter’s distribution:

![Exhibit 1: High Speed Wood Planer](image1.png)

![Exhibit 2: Combinations for the cutter heads](image2.png)

2.3 Safety

Wood machining has always been a dangerous job. The safety of the employees is therefore one of the most important issues facing all sectors of the woodworking industry. This is why national safety standards and regulations have been set up. With the harmonization of the standards within the EU, the European Commission has developed the Machinery Directive (now integrated in 98/97/EEC) so as to help ensure the safety of the employees and at the same time ensuring free trade within the European Union. The machinery directive says that machines:
• Must satisfy wide-ranging health and safety requirements, for example on construction, moving parts, and stability;
• Must, in some cases, be subjected to type examination by an approved body;
• Must carry the CE mark and other specific information.

Each member state has integrated the Machinery Directive into its own laws within a certain adoption time (1995). The Machinery Directive is supported by a series of European Standards such as:

• EN 847 Tools for woodworking: Safety requirements – Part 1, 2 and 3
• EN 860 Safety of woodworking: One side thickness planing machines
• EN 861 Safety of woodworking: Surface planing and thickness machines
• EN 940 Safety of woodworking: Combined woodworking machines and further standards

The requirements of these standards concern designers, manufacturers, suppliers and importers of tools for woodworking. These standards also include information which the manufacturer shall provide to the user.

Concerning the objectives of our project, standard EN 847/1 assumes a major importance, describing requirements and methods for the elimination or reduction of hazards arising from the design of wood working tools.

2.4 Certification

The experimental tests introduced in the first chapter as making part of the European security standards for rotating cutting tools, have to be performed in specific laboratories, with appropriate testing machines for such a severe solicitation. The standard EN 847 specifies all the procedures to be adopted when running one of these tests.

BG-PRÜFZERT in Germany is one institution composed by several testing laboratories, which are able to test and certify several types of machines according to the standards.
This technical institution has already tested and certified the first wood cutting tool developed by *Verktygs Larsson AB*, with 16 blades, 203mm of external diameter and 168mm of total blade’s length with a shaft diameter of 60mm. The centrifugal test in a vacuum chamber consisted in raising the cutter speed until 7500 rpm (normal speed of use) on a first stage and afterwards until 11250rpm. This value was determined taking into account a 1.5 security coefficient established in the EN 847 standard. When the test reaches the end, a measure of the displacements in several points of the cutter is performed.
CUTTING TOOL PROTOTYPE

After introducing some theoretical knowledge in the previous chapter, we are now ready to understand the mechanism and specifications of the cutting tool.

3.1 Mechanism presentation

In a general way, we can say the cutter is composed by some groups of components that are uniformly distributed in the main body. Each one of these groups has one screw, two different wedges, the tightening and the support wedge, one blade and finally four support pins.

As we can see in the exhibit 3, in each cavity of the cylinder there is a ramp in the back side. This ramp will contact with another one, with the same slope, that exists in the correspondent supporting wedge. The main function of these ramps is to hold the effect of the tightening force that will be applied in the screw, communicated to the support wedge by the tightening wedge.

Finally, the tool contains the pins that fit the blade in the working position. We should notice that the main part of the centrifugal force acting on the blade is balanced by the friction forces on its contact surfaces. Therefore, the function of the pins cannot be considered as structural.
Exhibit 3: Back side ramps in the main body

In order to perform the attachment of the planer cutter to the machine shaft is used hydraulic pressure applied to the bushes that are incorporated in the interior of the main cylinder.

Before being applied to a planer machine, the tool has to be balanced, in order to achieve the best mass distribution. The influence of this balance is specially noticed when rotating at lower speeds.

Exhibit 4: Components of the cutter
To achieve a better understanding of this complex mechanism and the physical interactions between all components, the main assembling procedures of the cutting tool are presented in the following points:

1. Weight all the components of the same type (blades, wedges, screws) and mark them with the respective values. Pairs with the most approximate value should be placed in diametrical opposite positions in order to guarantee the best balance of the tool;

2. Mount the blades, fitting them correctly with the pins, starting the assembly in the track marked with number 1;

Exhibit 5: Blade assembling

3. Slide the big wedge along the number 1 track until it stops in the bottom contacting with the ramp. Complete the round distribution;

4. Place the screws in the respective holes, rotating them 3 to 4 times;

Exhibit 6: Screws assembling

5. Insert the tightening wedge in its track and in the big wedge;
6. The pulling track of the tightening wedge should fit the head of the tightening screw;

Exhibit 7: Tightening wedge assembling

7. Screw the tightening screw with a hexagonal driver tool until it stops;

Exhibit 8: Tightening the screws

8. Using a dynamometric wrench tighten the screw on track 1 with a momentum of 10Nm, and then proceed to the diametrical opposite track.

Exhibit 9: Dynamometric wrench tightening the screws
9. Proceed tightening the screws with 22Nm of momentum, starting again in track 1, and proceeding to the diametric opposite track.

3.2 Specifications

Our study will be pointed to a specific geometry, in order to try to understand the variables that are affecting its behaviour, and also to know how important they are to the mechanism function. This geometry has 20 blades uniformly distributed around the cylinder with 220mm of external diameter and 45mm of internal diameter. The length of the main body is 270mm.

There are different dimensions available for this tool [Table 1], to suit the needs of the customers, obtained by changing some design parameters. The tool can include a maximum of 30 blades covering a diameter of 300mm with a total length of 340mm. However, the main design is common to all of them. The geometries that were analysed in this project appear as shadowed.

<table>
<thead>
<tr>
<th>Øexternal [mm]</th>
<th>Total length [mm]</th>
<th>Øshaft [mm]</th>
<th>N. Blades [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>180</td>
<td>120</td>
<td>45</td>
<td>12</td>
</tr>
<tr>
<td>180</td>
<td>120</td>
<td>50</td>
<td>12</td>
</tr>
<tr>
<td>203</td>
<td>170</td>
<td>45</td>
<td>16</td>
</tr>
<tr>
<td>203</td>
<td>245</td>
<td>45</td>
<td>16</td>
</tr>
<tr>
<td>203</td>
<td>270</td>
<td>45</td>
<td>16</td>
</tr>
<tr>
<td>203</td>
<td>170</td>
<td>50</td>
<td>16</td>
</tr>
<tr>
<td>203</td>
<td>170</td>
<td>50</td>
<td>16</td>
</tr>
<tr>
<td>203</td>
<td>270</td>
<td>50</td>
<td>16</td>
</tr>
<tr>
<td>220</td>
<td>170</td>
<td>45</td>
<td>20</td>
</tr>
<tr>
<td>220</td>
<td>245</td>
<td>45</td>
<td>20</td>
</tr>
<tr>
<td>220</td>
<td>270</td>
<td>45</td>
<td>20</td>
</tr>
<tr>
<td>220</td>
<td>170</td>
<td>50</td>
<td>20</td>
</tr>
<tr>
<td>220</td>
<td>245</td>
<td>50</td>
<td>20</td>
</tr>
<tr>
<td>220</td>
<td>270</td>
<td>50</td>
<td>20</td>
</tr>
<tr>
<td>250</td>
<td>170</td>
<td>45</td>
<td>24</td>
</tr>
<tr>
<td>250</td>
<td>245</td>
<td>45</td>
<td>24</td>
</tr>
<tr>
<td>250</td>
<td>270</td>
<td>45</td>
<td>24</td>
</tr>
<tr>
<td>250</td>
<td>170</td>
<td>50</td>
<td>24</td>
</tr>
<tr>
<td>250</td>
<td>245</td>
<td>50</td>
<td>24</td>
</tr>
<tr>
<td>250</td>
<td>270</td>
<td>50</td>
<td>24</td>
</tr>
<tr>
<td>250</td>
<td>245</td>
<td>60</td>
<td>20</td>
</tr>
</tbody>
</table>

*Table 1: Cutting tool available dimensions*
Concerning the materials adopted to this cutting tool, the main body, both the tightening and the support wedge and the screw are made of steel. The blade is made of high speed steel. All the materials are certified according to the Swedish standards, and Table 2 presents their properties.

<table>
<thead>
<tr>
<th></th>
<th>HSS 2722</th>
<th>SS 2541 - 03</th>
<th>SS 2090 - 03</th>
<th>SS 2172 - 03</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young Modulus [GPa]</td>
<td>230</td>
<td>210</td>
<td>210</td>
<td>210</td>
</tr>
<tr>
<td>Poisson ration [adim]</td>
<td>0,266</td>
<td>0,266</td>
<td>0,266</td>
<td>0,266</td>
</tr>
<tr>
<td>Density [Kg/m3]</td>
<td>7980</td>
<td>7860</td>
<td>7860</td>
<td>7860</td>
</tr>
<tr>
<td>Yield strength [MPa]</td>
<td>450</td>
<td>700</td>
<td>1150</td>
<td>310</td>
</tr>
<tr>
<td>Ultimate Strength [MPa]</td>
<td>---</td>
<td>900 - 1100</td>
<td>1300 - 1500</td>
<td>470 - 610</td>
</tr>
</tbody>
</table>

*Table 2: Properties of the adopted materials*

We should notice that the screws have a surface treatment of carbon nitruration to improve the surface hardness, and to the main body is applied a chemical nickel treatment to achieve high values of corrosion resistance. Finally, the blade’s face with bigger area has an ion sputtering treatment, because this is the face that is supporting the largest friction contact.

The maximum rotating speed of the planer cutter is 7500 rpm and during a working cycle the highest wood thickness that is cut is approximate 3mm when one single blade reaches the wood panel work piece.

### 3.3 Definition of the cutter according to EN847/1 Standard

In agreement with Chapter 3 of the EN 847 standard, this planer cutter is defined as a complex tool, where one of more cutting parts (inserts, blades) are exchangeably mounted in a body through detachable fixing elements. The definition of complex tool appears included in the group of the milling tools for woodworking. These are rotating cutting tools normally having their main feed direction perpendicular to the rotation axis, for working various surfaces on wood and similar materials through chip removal.

In what concerns the design requirements, the standard specifies that for complex tools an over speed experimental test has to be performed. The safety factor
of 1.5 should be applied to the maximum rotational speed of the tool, establishing a testing of 11250 rpm in the case of our cutter. During the test, the displacements should not be greater than 0.15mm and their measurement has to be performed with the tool stopped, after rotating it for 1 minute at the test speed.

The standard allows one deviation from the specifications given above in the case of milling tools with centrifugal wedges. In this type of tools, if the calculation is performed to the most unfavourable tolerances and for the test speed, and if the obtained stresses are in the elastic range of the materials, then greater displacements are permitted. But these displacements of the centrifugal wedges shall not adversely influence the function and behaviour of the tool.
This chapter presents some introductory fundamentals of finite element theory in addition with some considerations about finite element softwares.

### 4.1 Introduction

Since early 1980s Finite Element Method and the softwares based on it, were recognised as powerful tools solving some engineering problems. The affordability and versatility of Finite Element (FE) softwares has helped to spread its popularity.

Nowadays, solvers are much user friendly and the task of applying them has become easier. However, this has also resulted in the wrong use of these softwares by people who are not familiar with the fundamental concepts of FE theory. That is to say that a great amount of engineering judgement is also needed.

### 4.2 Fundaments

Most numerical techniques in continuum mechanics are based on the principle that it is possible to derive some equations and relationships that accurately describe the behaviour of a small part of the body.
Accordingly, by dividing the entire body into a large number of these small “parts” or elements and using appropriate compatibility and equilibrium relationships to link up or assemble these elements, it is possible to obtain a reasonably accurate prediction of the values of variables, such as stresses and displacements in the body.

Essentially, any problem can be split up into any number of smaller problems. With the FEM this is done by considering that a complex geometrical shape is made up of a number of smaller simple parts. This is known as spatial discretization, with each simple shape being known as an element, being the whole collection of elements known as the mesh. Within the element, its relevant property is predicted, for example, the relationship between forces and displacements for a structural element. This is done without any reference to the other elements in the mesh. Here, the element equations are established, often by assuming known values of properties at fixed points named nodes. Then the properties of all the elements and interactions between them are taken into account by assembling the element equations and finding a solution to them.

The resolution process to solve any engineering problem through a FE analysis is developed with a standard procedure, characterized by typical steps, presented as follow:

1. Discretize and select elements configuration;
2. Select approximation models or functions;
3. Define Strain-Displacement and Stress-Strain relationships;
4. Derive element equations;
5. Assemble element equations to obtain global or assemblage equations and introduce boundary conditions;
6. Solve for the primary unknowns,
7. Solve for derive or secondary quantities;
8. Interpretation of results.

As the sizes of these small elements are made smaller, the numerical solution becomes more accurate, but at the cost of increased computation time.
4.3 Considerations about Finite Element *Softwares*

The FE solvers market range is divided in two main categories: the pure FE and CAD/CAE *softwares*. Considering the complexity of the analysis, the first option appears at the beginning as the more accurate. But, the second type of *softwares*, combining Computer Aided Design and Computer Aided Engineering tools, allows users to decrease design and analysis times.

CATIA is a widely known *software* that is included into the second category. The educational version available during this project at *Linköping University* includes a toolbox with instruments to perform static analysis using a FE code.
MODELING

In this chapter the analysis assumptions are introduced, as well as all the explanation about the adopted models, considering the elements, meshes and the applied loads.

5.1 Introduction

The concept of modelling is based on a main line of creating an approximation of a real problem, considering always some simplifications. This is a primary need because complex models are depending on high calculation times.

During our work of analysing the cutting tool we followed this path, passing through several models, to achieve the primary target.

The following part of this chapter presents all the models carried out in CATIA environment since the beginning, introducing all simplifications that were taken in account and justifying them with the respective assumptions.

5.2 Static analysis

To achieve the desired answers to our problem, we adopted a static analysis of the tool, being this, the first simplification that was made. The real problem is
typically dynamic, however, it can be approximated by a static process without being too distant from the reality.

The effect of the rotation speed was substituted by one equivalent centrifugal force applied to all bodies. With this assumption, the effect of the movement conditions is translated by forces and constraints.

Considering the geometric axis system, we can say that this situation is equivalent to someone being placed in one of the lateral faces of the main body. If we imagine ourselves in this position, with the same rotating movement as the cylinder, we can visualize this one has being stopped, because we are rotating with same velocity and in the same direction, with the possibility of just observing its displacements.

Considering the referred evolution from the dynamic to the static state, the interior face of the main cylinder where the shaft fits had to be necessarily clamped.

In operation, our tool is subjected to static and dynamic loads. The structural load generated by centrifugal forces becomes dominant over the forces resulting from the cutting process. Concerning the dynamic proportions of the load, the tool is subjected to forces resulting from the cutting process, introducing pulsating stresses and cyclic centrifugal forces due to acceleration and desacceleration. The centrifugal force acts as the main load both on the tool body and on the cutting element, while the cutting forces counteract the effect of the centrifugal forces on the cutting element.

However, the most critical situation is when the tool is freely rotating without being in contact with the work piece, since there is no energy dissipation through the cutting act. It happens for instance in the initial stage. Therefore, we have considered a static model with centrifugal solicitations at a constant operating speed.

5.3 Evolution on the geometry of the model

As we announced in the beginning of this chapter, due to the complexity of the involved mechanism, simplifications related with the geometry of the model had to be made. The first idea of building one model similar to the real tool, with the 20 tracks filled with the components, was immediately left, considering that just the assembly procedure would be extremely complicated. Besides, the computation task would be impossible, with the need of super workstations. Therefore, we made an evolution to
more simple models, decreasing the number of filled tracks and performing other possible simplifications without being too far away from the real working solicitations [Exhibit 10].

*Exhibit 10: The first models*

After this first model study stage, we finally established two validation models, testing them in the most complete way.

*Exhibit 11: The final models*
With the second model, we intended to study how the existence or not of components in adjacent tracks would affect the displacements and stresses.

In both models, the lateral surfaces were clamped, in order to simulate the presence of the main body’s volume that was not considered.

The referred bushes that attach the planer cutter to the rotating shaft were not included in the model due to the fact that their function is not structural, having no influence to the safety analysis that is intended to be done.

In accordance with these final models, analyses were made to achieve the final output of determining the maximum displacements and stresses, in order to guarantee that these are in the range established by the standard. Both models were applied to the first geometry referred in Chapter 3. Considering the geometry with a length of 340mm and an external diameter of 300mm, the biggest, only the first model was used.

A final comment has to be made regarding the quality of the parts designed with CATIA to build the assembly. Based on the technical drawings created by the company, all the components were designed with all the details and with an extreme accuracy concerning the dimensions involved.

### 5.4 Assembly constraints

Using CATIA mechanical design toolbox, we were able to create all the components separately as a part, namely the tightening wedge, the support wedge, the screw and the blade, as well as the main cylindrical body. After that, an assembly was made to create each model. Independently on the model, the positioning constraint’s type used to perform each assembly was surface contact, which can be applied between two planar faces.

### 5.5 Analysis connections

Once the geometric assembly positioning constraints are defined at the product level, the physical nature of the connections has to be defined. Considering the
problem itself and the characteristics of each type of connection available, we used four types, namely slider, contact, fastened and bolt tightening.

The slider connection property can be described has a link between two bodies which are constrained to move together in the local normal direction at their common boundary, and will behave as if they were allowed to slide relative to each other in the local tangential plane. Meanwhile, a contact connection is the link between two part bodies which are prevented from inter-penetrating at their common boundary, and will behave as if they were allowed to move arbitrarily relative to each other as long as they do not come into contact within a user-specified normal clearance. When they come into contact, they can still separate or slide relative to each other in the tangential plane, but they cannot reduce their relative normal clearance.

On the other hand, a fastened connection is the link between two bodies which are fastened together at their common boundary, and will behave as if they were a single body. From a finite element model viewpoint, this is equivalent to the situation where the corresponding nodes of two compatible meshes are merged together.

Finally, a bolt tightening connection takes into account pre-tension in bolt-tightened assemblies. The computation is carried out according to a two-step approach. In the first step of the computation, the model is submitted to tension forces relative to bolt tightening by applying opposite forces on the bolt thread and on the support tapping, respectively. Then, in the second step, the relative displacement of these two surfaces (obtained in the first step) is imposed while the model is submitted to user loads. During these two steps, the bolt and the support displacements are linked in the direction normal to the bolt axis.

Since bodies can be meshed independently, all the presented connections are able to deal with incompatible meshes.

5.6 Loads

The loads considered in the models can be divided in three main types:

- Centrifugal forces;
- Friction forces;
- Tightening forces.
The assumptions and considerations made for each type are presented in following points.

### 5.6.1 Centrifugal Forces

The centrifugal force acts as the main load in the tool body and also in the rest of the components. Theoretically, this force can be calculated according with the formula:

\[ F_{\text{centrifugal}} = m \times a_{\text{centrifugal}} \]

Attending to the definition of centrifugal acceleration and also to the relation between the tangential and rotational speeds,

\[ a_{\text{centrifugal}} = \frac{v^2}{r_{\text{c.g.}}} \]

\[ v = \omega \times r_{\text{c.g.}} \]

it is possible to rewrite the previous expression:

\[ F_{\text{centrifugal}} = m \times r_{\text{c.g.}} \times \omega^2 \]

The values of the masses of the main components included in the cutter for the two geometries studied are presented in the Table 3.

<table>
<thead>
<tr>
<th>Component</th>
<th>1st Geometry</th>
<th>2nd Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Main body</td>
<td>51 [Kg]</td>
<td>135 [Kg]</td>
</tr>
<tr>
<td>Support wedge</td>
<td>0.346 [Kg]</td>
<td>0.440 [Kg]</td>
</tr>
<tr>
<td>Blade</td>
<td>0.206 [Kg]</td>
<td>0.261 [Kg]</td>
</tr>
</tbody>
</table>

*Table 3: Masses of some components*
Using the previous values and considering the centre of gravity radius of each component provided by the software, it was possible to determine the centrifugal forces.

<table>
<thead>
<tr>
<th></th>
<th>1st Geometry</th>
<th>1st Geometry</th>
<th>2nd Geometry</th>
<th>2nd Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>7500 rpm</td>
<td>11250 rpm</td>
<td>7500 rpm</td>
<td>11250 rpm</td>
</tr>
<tr>
<td>Support wedge</td>
<td>20728.5N</td>
<td>46639.1N</td>
<td>28320.8N</td>
<td>63721.8N</td>
</tr>
<tr>
<td>Blade</td>
<td>12383.9N</td>
<td>27863.9N</td>
<td>16691.7N</td>
<td>37556.2N</td>
</tr>
</tbody>
</table>

Table 4: Centrifugal forces acting on the support wedge and on the blade

Besides the general increase in the centrifugal forces, higher normal reactions are obtained because of the function of the support wedge. Therefore, higher values of maximum static friction forces are available to perform an opposition to the trend that the blade has to fly away.

Analysing these results for first geometry, we should notice that when raising the speed from 7500rpm to 11250rpm, there is an increase in the centrifugal force of the blade of 15480N, while the support wedge has an increase of 25911N.

To the second geometry, the increase in the centrifugal force of the blade is 20864N, while in the support edge is 35401N.

We can conclude that the increase of the centrifugal force associated to the support wedge, giving normal reactions with more magnitude, is higher than the increase in the blade. This can be a possible explanation to a curious phenomenon verified during the experimental test performed in Germany, where the measured displacements at 11250rpm were lower than at 7500rpm.

5.6.2 Friction Forces

Regarding the final objective of this tool, the friction forces can be considered the most important load acting on the mechanism. They have a fundamental role in the behaviour of the cutter at high speed, and therefore, the model has to be specially accurate concerning this loads.

When the all mechanism is set up, friction forces exist between the main faces of the following components:
- Blade and cylinder;
- Blade and support wedge;
- Support wedge and cylinder;
- Tightening wedge and support wedge;
- Tightening wedge and cylinder.

The first step to achieve a correct translation of what is happening in terms of friction was to investigate if CATIA had the characteristic of considering friction forces and all the properties related with this complex type of contact connections. Consulting all CATIA manuals available at the university none information was found about this subject. Therefore, the solution was to build a simple model, considering a clamped ramp with an inclination ranging from 5° to 45°, and placing on it one body with cubic geometry. The material used was conventional steel. First calculations were made with the highest value of inclination (45°). CATIA presented an error, informing that this was not a static situation. Performing some simple calculations, we concluded that with 45° the normal reaction component in the movement direction was higher than the maximum static friction force. Assuming this, we started to decrease the inclination to values around 5°, but the error was persisting. Therefore we concluded that the software was unable to consider friction properties.

To solve this situation we adopted a different approach. Knowing that the friction forces between a pair of surfaces can be calculated from the normal reactions involved multiplied by the friction coefficient, the solution was to determine these reactions. Therefore, placing axis system in each contact surface [Exhibit 12] and reaction sensors in accordance with these axes, it was possible to determine the value of each normal reaction in three main contacts: cylinder/support wedge; support wedge/blade; blade/cylinder.
Exhibit 12: Axis systems used to apply the sensors

Table 5 presents the values of the normal reactions determined using the referred sensors while exhibit 13 shows the surfaces where they were placed.

<table>
<thead>
<tr>
<th></th>
<th>1st Geometry</th>
<th>2nd Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Øext = 220mm</td>
<td>Length = 270mm</td>
<td>Øext = 300mm</td>
</tr>
<tr>
<td>Sensor 1</td>
<td>192390N</td>
<td>239381N</td>
</tr>
<tr>
<td>Sensor 2</td>
<td>161990N</td>
<td>209561N</td>
</tr>
<tr>
<td>Sensor 3</td>
<td>148762N</td>
<td>142692N</td>
</tr>
<tr>
<td>Sensor 4</td>
<td>-</td>
<td>215729N</td>
</tr>
<tr>
<td>Sensor 5</td>
<td>-</td>
<td>185380N</td>
</tr>
<tr>
<td>Sensor 6</td>
<td>-</td>
<td>172150N</td>
</tr>
</tbody>
</table>

Table 5: Normal reactions (Ry) in the contact surfaces

Obviously, model number 2 includes sensors in both filled tracks, numbered from 4 to 6.
The maximum static friction forces were determined using a friction coefficient of 0.1, a typical value of a contact between two steel bodies.

To the biggest planer cutter that was analysed using the second FE model, we considered the maximum static friction force applied in a distributed way to the surfaces. The procedure was the following:

- Measure the value of the normal reaction (\( R_y \)) in one surface using a sensor;
- Calculate the maximum static friction force using a friction coefficient of 0.1;
- Apply a distributed load to the pair of surfaces with a value equal to the maximum static friction force.

In this last step, the distributed load was applied considering the case of a pair “action–reaction”. Therefore, the value in adjacent surfaces is the same but with opposite directions.

To the cutter with smaller dimensions, analysed with both final models, the procedure was the same, but considering now a value of 85% of the maximum static friction force. With this assumption the analysis is placed on the side of security.
5.6.3 Tightening Forces

Attending of the assembling information provided by Verktygs Larsson AB, the value of the momentum used to tighten the screws was 22 N.m. In order to establish the above referred bolt tightening connection property, this momentum had to be converted to a pre-tension force.

This conversion can be made using the following formula:

\[
F = \frac{2T}{\alpha} \left( \frac{\pi d_m - \mu l \sec \alpha}{1 + \pi \mu \sec \alpha} \right)
\]

- \( F \): Pre-Tension Force [N]
- \( T \): Tightening momentum [N.m]
- \( \alpha \): Flank angle [rad]
- \( d_m \): Nominal diameter [m]
- \( \mu \): Friction coefficient [adim]
- \( l \): Pitch [m]

Since the screws used in the cutter have a metric thread M10, the following values are standard and specified:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pitch</td>
<td>31,42</td>
<td>[mm]</td>
</tr>
<tr>
<td></td>
<td>0,03142</td>
<td>[m]</td>
</tr>
<tr>
<td>Nominal diameter</td>
<td>10,00</td>
<td>[mm]</td>
</tr>
<tr>
<td></td>
<td>0,010</td>
<td>[m]</td>
</tr>
<tr>
<td>Flank angle</td>
<td>15,00</td>
<td>[°]</td>
</tr>
<tr>
<td></td>
<td>0,262</td>
<td>[rad]</td>
</tr>
</tbody>
</table>
Substituting the previous values on the referred formula, a pre-tension force of 2008 N is obtained.

### 5.7 Element and Mesh Properties

#### Elements:

In all the models that were considered during the development of this thesis, the linear tetrahedron element [Exhibit 15] was used as standard for all the three-dimensional meshes. It is a four nodes isoparametric solid element with the following properties:

![Exhibit 15: Tetrahedron element](image)

- **Type:** solid element
- **Physical property:** solid
- **Mesh connectivity:** linear tetrahedron
- **Number of nodes:** 4
- **Degrees of freedom:** 6
- **Type of behaviour:** elastic

#### Meshes:

As it was referred in previous chapters, the planer cutter studied in this thesis presents a complex geometry. This complexity comes from several design details
included in the main body and in the components. For example, measures can range from 3mm in the contact of the blades to the pins until 340mm in the main body.

These variations of measures increase not only the calculation times, but mainly the probability of errors associated with the finite element analysis due to the appearance of singularities.

Therefore, a precise mesh refinement had to be made for example in the pins, ramps, small contact surfaces and other places that were inducing errors to the analysis.
RESULTS

This chapter will present the results of the analysis obtained during the project. It is divided in two main parts, in accordance with the studied geometries, including the meshes, displacements, stresses and estimated local errors. It was an option to include in this chapter only the most important output exhibits, placing the rest of the figures in an appendix.

6.1 Model with one blade applied to the first geometry

Attending on the obtained results for the deformed mesh, they comproved the deformation idea that we expected to be verified during the working conditions. In all components the displacements were minimum.

The blade and the support wedge showed a slight bending behaviour, due to their fixing conditions [Exhibits 19 and 20]. The maximum displacement observed in the blade happens in the middle section, with a value of 0,0948mm.

Concerning the support wedge, the maximum value is also verified in the middle of this component, being equal to 0,141mm. In the tightening wedge and in the screw, both maximum displacements are under the previous values.
The displacements on the main body are maximum in the influence side of the support wedge, reaching 0.0608mm [Exhibit 18].
Observing all the results for the displacements, we can notice that they are under the maximum limit value, 0.15 mm, imposed by EN847 standard.

In regard to the stresses obtained for this first model, the maximum global value was 421 MPa, in the tightening wedge, more precisely in its contact area with the screw head [Exhibit 21]. The yield strength of the material of the tightening wedge is 1150 MPa, being the verified Von Mises stress inside its elastic regime.
In the main body, the areas with higher stress solicitations are the pins and the back side ramp. The highest Von Mises stress is 253 MPa.

Concerning the support wedge, higher values were found in the areas that have geometric discontinuities, more precisely in the two ramps and in the side face contacting with the main body, near the region were the inclination changes. The maximum stress found was 188 MPa.
Exhibit 23: Von Mises Stresses in the support wedge

In the blade, higher values of stress were found in the round surfaces that contact with the pins, reaching a maximum value of 360 MPa.

Exhibit 24: Von Mises Stresses in the blade

Exhibit 25: Von Mises Stresses in the tightening wedge
Therefore, we can resume that for all components, the verified stresses are inside the elastic regime, with no existing possibility of plasticity. In this analysis with the first FE model adopted, the highest estimated local error (0.00572 J) was observed in the main body area where the screw head fits.

Exhibit 26: Estimated local errors in the main body

6.2 Model with two blades applied to the first geometry

Introducing this second model, it was possible to verify the initial idea of making a comparison between both models displacements, mainly because the existence of more than one blade would be a weight factor. Due to higher values of normal reactions, displacements in the middle of the blade were lower, comparing the ones that were obtained with the first model. The maximum value was 0.0714mm [Exhibit 31].

Exhibit 27: Global mesh
In the main body, the maximum displacements were verified in the region between the two filled tracks, with a value of 0,162mm [Exhibit 29]. The reason for this behaviour is the compression performed by the second support wedge against the main cylinder. In fact, this value is higher than the limit 0,15mm of the standard, but EN847 allows slight deviations with stresses inside the elastic regime.
Exhibit 31: Displacements in the blade

In this analysis with the second FE model adopted, the highest estimated local error was observed in the main body, with an approximate value of 0.033J [exhibit 32].

Exhibit 32: Estimated local error in the main body
6.3 Model with one blade applied to the second geometry

The analysis of the cutter with the highest dimensions was performed using the first model, since that a conclusion regarding the comparison between both was already been made with the smallest cutter.

Therefore, a maximum displacement of 0.103mm was observed in the main body, being under the limit range of the standard [Exhibit 34].

The support wedge was the component with the highest displacements, with a maximum value of 0.225mm [Exhibit 35]. A possible explanation for this value that is slight over the limit of the standard, is the fact that this analysis was performed at an
over speed for this particular cutter. A test speed of 11250 rpm was used in accordance with the safety factor of 1,5 applied to the maximum rotational speed of 7500rpm. But the fact is that the company considers a maximum rotational speed for this cutter of 6000rpm. Applying the same factor of 1,5, the result is a testing speed of 9000rpm. Therefore, this geometry was tested with an over speed, resulting in higher values of displacements that would be lower if the speed of 9000rpm had been considered. The same explanation can be adapted to what happened with the stresses, being the maximum value of 825MPa verified in the pins [Exhibit 37]. Another reason for this particular value can be an analysis singularity, due to the fact that the radius in that area is very small (1,5mm). Using elements with lower dimensions would be a solution, but the fact is that hardware limits had been reached due to the increase in the geometry volume to be meshed.

Exhibit 35: Displacements in the support wedge

The displacements in the blade were also higher in the middle region, with a maximum of 0,135mm in accordance with the safety requirements of the standard.

Exhibit 36: Displacements in the blade
In this analysis, the highest estimated local error was observed in the main body, with an approximate value of 0.01 J [exhibit 38].
CONCLUSIONS

To achieve the final objective of verifying the safety of the wood planer cutter developed by Verktygs Larsson AB, EN847 standard was analyzed carefully, with special attention to the requirements included. As it was explained, the standard presents limit values for the displacements found on the different components of the tool, being the highest admitted value equal to 0.15mm. The standard allows one deviation, admitting higher displacements if the obtained stresses are in the elastic range of the materials without influencing the working behaviour of the mechanism.

Before presenting the conclusions of this project, a main difference between the standard and the analysis we performed has to be introduced. The displacements referenced in EN847 are measured with the tool stopped, while our displacements and stresses were obtained applying equivalent loads to the model we developed. Therefore, a direct comparison between the standard specifications and our results is possible to be made but taking into account that the displacements in each case don’t have the same background. We can say that in our analysis we are placed side by side with security, due to the fact that the results were obtained by performing a direct relation with the applied loads.

Attending on the values presented both on the results chapter and in the following appendix, a conclusion can be immediately made. The displacements for the components of the planer cutter, considering both geometries analyzed, are inside the security range established by the standard, not reaching the maximum value of 0.15mm.
Performing a comparison between the Von Mises stresses obtained with this FE analysis and the yield strength of each material, it is possible to conclude that during the working cycle, none of the materials found on the components of the planer cutter reaches the elastic limit.

In addition, we obtained values of normal reactions sufficient to guarantee that centrifugal forces are equilibrated with friction forces not reaching the maximum static value, meaning that at the tested speed of 11250rpm the components are not in the imminence of initiating their relative movement.

The two geometries considered during this project, with 220mm and 300mm of external diameter, were analyzed at 11250rpm, attending on the safety coefficient imposed by the standard to the maximum rotational speed of the cutter (7500 rpm). But the fact is that the second geometry, with increased dimensions, is never going to reach such high speeds during its use. The company considers that the maximum rotational speed for that cutter is 6000rpm. Applying the same factor of 1,5, the result is a testing speed of 9000rpm. Therefore, the second geometry was tested with an over speed, resulting in higher values of displacements and stresses that would be lower if the speed of 9000rpm had been considered.

The estimated local errors found for each component appear as acceptable, being the maximum value approximately 5%, confirming the care that was used to execute each refinement in the FE meshes.

In conclusion, we can confirm that the tool designed by Verktygs Larsson AB is safe according with the requirements and restrictions of EN847 standard.

Being an approximation of the real working conditions of the tool, the model that was built to run the analyses is not an exactly transcript of what happens precisely in the mechanism. All efforts were made to make both models as accurate as possible, but simplifications had to be made in our static analysis.

An alternative method to achieve the same objective in future analyses can be a dynamic study using appropriate software, but at this moment, the static analysis appears has an adequate method to transmit the mechanism behaviour in working conditions.
DISCUSSION

As it was said in the previous chapters, due to several difficulties that were found during the analysis, some simplifications had to be made, in order to achieve the final objective.

The model represents in an accurate way what happens during the working conditions of the cutter, but with some limitations. Dynamic factors, such as accelerations, desaccelerations, vibrations, and the impact of the wood on the blades, which can affect the behaviour of the cutter while in use, were not considered. In addiction, discontinuities that can appear in a wood work piece, like regions with more hardness due to different orientation of the grains, perform an influence to the distributed forces applied to the blade, causing dynamic solicitations that are not possible to model. Even the feed speed of the machine can range between different values, producing forces on the blades with distinct magnitudes. Since the beginning, we studied one fixed position for the blade, not analysing the fact that when each blade is grinded it will have a new assembly placement, achieved by moving this component to an upper position. Therefore, the coordinates of the gravity centre will change, influencing the forces acting on the blade.

The importance of all the previous details goes down when we attend on the final objective of this thesis, to analyse the possibility of failure of one of the blades due to the centrifugal forces. Following this idea, the base of the static analysis we performed was the most critical situation that is found on the cutter, which is the moment when the tool is rotating freely without being in contact with the work piece...
at the maximum rotating speed. At this point, centrifugal forces acting on the components are maximum, placing all the dynamic solicitations we spoke about in the beginning of this chapter in a lower magnitude level. Therefore, we obtained results that were validated attending to the fact that the most important solicitations were considered in the model. In a practical way, our analysis reached the final objective considering all the solicitations that were fundamental, being very close to the real mechanic behaviours that are happening in the cutter.

In the future, to improve this project with further analysis, the development of a dynamic model with a different software would be the next step, in order to evaluate our model in a comparison procedure. A special attention should be applied to the contact properties that were not available with CATIA, forcing us to engage an alternative approach.
APPENDIX
Exhibits of the analysis performed to the first geometry with the one blade model:

Exhibit 1: *Global mesh*

Exhibit 2: *Deformed mesh*
Exhibit 3: Global displacements

Exhibit 4: Displacements in the main body
Exhibit 5: Displacements in the blade

Exhibit 6: Displacements in the support wedge
Exhibit 7: Displacements in the tightening wedge

Exhibit 8: Displacements in the screw
Exhibit 9: Global stresses

Exhibit 10: Stresses in the main body
Exhibit 11: Stresses in the blade

Exhibit 12: Stresses in the support wedge
Exhibit 13: Stresses in the tightening wedge

Exhibit 14: Stresses in the screw
Chapter 9. Appendix

Exhibit 15: Estimated local errors

Exhibit 16: Estimated local error in the main body
Exhibit 17: Estimated local error in the blade

Exhibit 18: Estimated local error in the support wedge
Exhibit 19: Estimated local error in the tightening wedge

Exhibit 20: Estimated local error in the screw
Exhibits of the analysis performed to the first geometry with the two blade model:

*Exhibit 21: Global mesh*

*Exhibit 22: Deformed mesh*
Exhibit 23: *Displacements in the main body*

Exhibit 24: *Displacements in the blade*
Exhibit 25: Displacements in the support wedge

Exhibit 26: Displacements in the tightening wedge
Exhibit 27: Displacements in the screw

Exhibit 28: Stresses in the main body
Exhibit 29: Stresses in the blade

Exhibit 30: Stresses in the support wedge
Exhibits of the analysis performed to the second geometry with one blade model:

Exhibit 31: *Global mesh*

Exhibit 32: *Displacements in the main body*
Exhibit 33: Displacements in the blade

Exhibit 34: Displacements in the support wedge
Exhibit 35: Displacements in the tightening wedge

Exhibit 35: Displacements in the screw
Exhibit 36: Stresses in the main body

Exhibit 37: Stresses in the support wedge
Exhibit 38: Estimated local error in the main body

Exhibit 39: Estimated local error in the blade
Exhibit 40: Estimated local error in the support wedge

Exhibit 41: Estimated local error in the tightening wedge
Exhibit 42: Estimated local error in the screw
LIST OF REFERENCE

Books

- A.A. Becker, Finite Element Analysis, Professional Engineering Publishing
- M.H. Aliabadi and C. A. Brebbia, Computational Methods in Contact Mechanics, Elsevier Applied Science
- Beez Andreas, Safe Design of High Speed Milling Tool, Linköpings Universitet, Institute of Technology, Department of Mechanical Engineering, Production System Division
- Handbook of the 15th International Wood Machining Seminar, July 2001, Los Angeles, California, USA.
- Principles of Wood Science and Technology

Standards

- EN 847 Tools for woodworking: Safety requirements – Part 1
- Swedish Standard 14 2541, Steel for Hardening and tempering
- Swedish Standard 14 2172, Structural Steel
- Swedish Standard 14 2142, Microalloyed Steel
- Swedish Standard 14 2090, Spring Steel
- Swedish Standard 14 2722, High Speed Steel

Internet

- http://www.weinig.de
- http://www.waco.se
- http://www.springer-maschinen.com