

We are IntechOpen, the world's leading publisher of Open Access books Built by scientists, for scientists

6,900

Open access books available

186,000

International authors and editors

200M

Downloads

Our authors are among the

154

Countries delivered to

TOP 1%

most cited scientists

12.2%

Contributors from top 500 universities



WEB OF SCIENCE™

Selection of our books indexed in the Book Citation Index
in Web of Science™ Core Collection (BKCI)

Interested in publishing with us?
Contact book.department@intechopen.com

Numbers displayed above are based on latest data collected.
For more information visit www.intechopen.com



Overview in the Application of FEM in Mining and the Study of Case: Stress Analysis in Pulleys of Stacker-Reclaimers: FEM vs. Analytical

Jairo A. Martins and István Kövesdy

Additional information is available at the end of the chapter

<http://dx.doi.org/10.5772/46165>

1. Introduction

The determination of stresses, deformations and the proper evaluation of calculations outputs are of extreme importance on the mechanical components and to assembly's effectiveness in the Mining Environmental. For instance, components of heavy duty plant machinery like Car Dumpers, Apron Feeders, Stacker-Reclaimers, etc..., and important components of Stacker-Reclaimers like Pulleys; which are under high responsibility must be calculated and designed properly and carefully. The unknown or ignorance of the complete environmental or data inputs (loads and constraints) where the component is applied, can bring tremendous damages to society and jeopardize entire businesses, mainly whether lives are involved. However, new technological tools, like Finite Element Method (FEM), have brought an even higher level to a better understanding of the complex products, those which have several parts in its conception. Like Klauss [2] describes, the Finite element methods are now widely used to solve structural, fluid, and multiphysics problems numerically. The methods are used extensively because engineers and scientists can mathematically model and numerically solve very complex problems. FEM is considered though a step further on the path on designing products, saving weight, consequently costs of design and manufacturing by the better understanding the pieces behaviors and performance prediction.

To evaluate the stresses in mechanical parts and/or components there are basically two manners; by the analytical approach and/or finite element method. This last, considered the most recent and complete tool to evaluate stresses and strains [2].

An example of the FEM simulation is shown briefly in this chapter when designing pulleys to Stacker-Reclaimers. We selected a standard pulley and generated it by analytical model (Redundant Structure Model) as well as by Finite Element Method (FEM) under linear

analysis. The analytical formulas presented in the text are those belonged to the classical mechanical engineering background. In fact, the analytical calculation has presented success along the time once most of the products in the field have performed properly. The model is considered robust enough to deliver products under high quality of project and which considers the material and manufacturing data in order to determine the allowable stress by safety factors.

This paper describes the limited resources when calculating pulleys (for Stacker-Reclaimers or belt conveyors) by analytical methods in comparison with the advantages of the Finite Element Method and its comparatively minor imprecision. The analytical calculation, particularly, presents an issue considered a constraint to overcome, which is related to the energy contribution by linear elastic deformation, of each component to the final sum of the stress \times strain in the assembly.

To FEM simulation, the software Inventor 2010 [3] was used to develop the model, meanwhile the calculation by FEM made by Autodesk Simulation [4].

2. Overview of products modeled in 3D and simulated by FEM

In the mining business the usage of software's in modeling components and machines 3D and afterwards simulated by FEM has increased potentially within the last decades. Machines like, Car Dumpers (Figure 1), Apron Feeders (Figure 2) and Stackers-Reclaimers (Figure 3) and their main components are modeled 3D and simulated by FEM software's. Several software's, specialized in modeling are available in the market, for example, Solidworks, Ideas, Inventor etc..., and in terms of FEM simulation, Nastran, Ansys and Autodesk Simulator are at the edge of this technology.

2.1. Advantages and drawbacks of modeling 3D and simulating by FEM in mining

Due to the upgrade in the way of designing products along the last decades, the technologists and engineer's had to change their minds when studying products in their initial phase of product design. Till the last decade the drawings were done basically in 2D environment, manually by clip boards and later on by CAD in computers. These required high imagination and capacity to evaluate technically and precisely the components in spatial views and in free space. However, the possibility of error by interferences between parts was relatively high and when happened, high costs were involved due to the required interven in the manufacturing or in the field. With the event of modeling by software, the need of another way of thinking about components and/or machines was strictly required. Despite of apparently complex, at first sight, due to the change of the way of designing, the FEM brings some advantages which worth, as follow;

- a. Mitigate or eliminate interferences, by visual analysis and software testing, decreasing substantially the re-work in manufacturing or in field;
- b. the possibility of designing lean shapes to particular application and loads;

- c. having lower weights and consequently more effectiveness in energy savings to rotary/dynamical parts;
- d. providing refined visual presentation (3D) of stresses and displacements suffered by the parts. Therefore bringing excellent power of analysis for engineers;

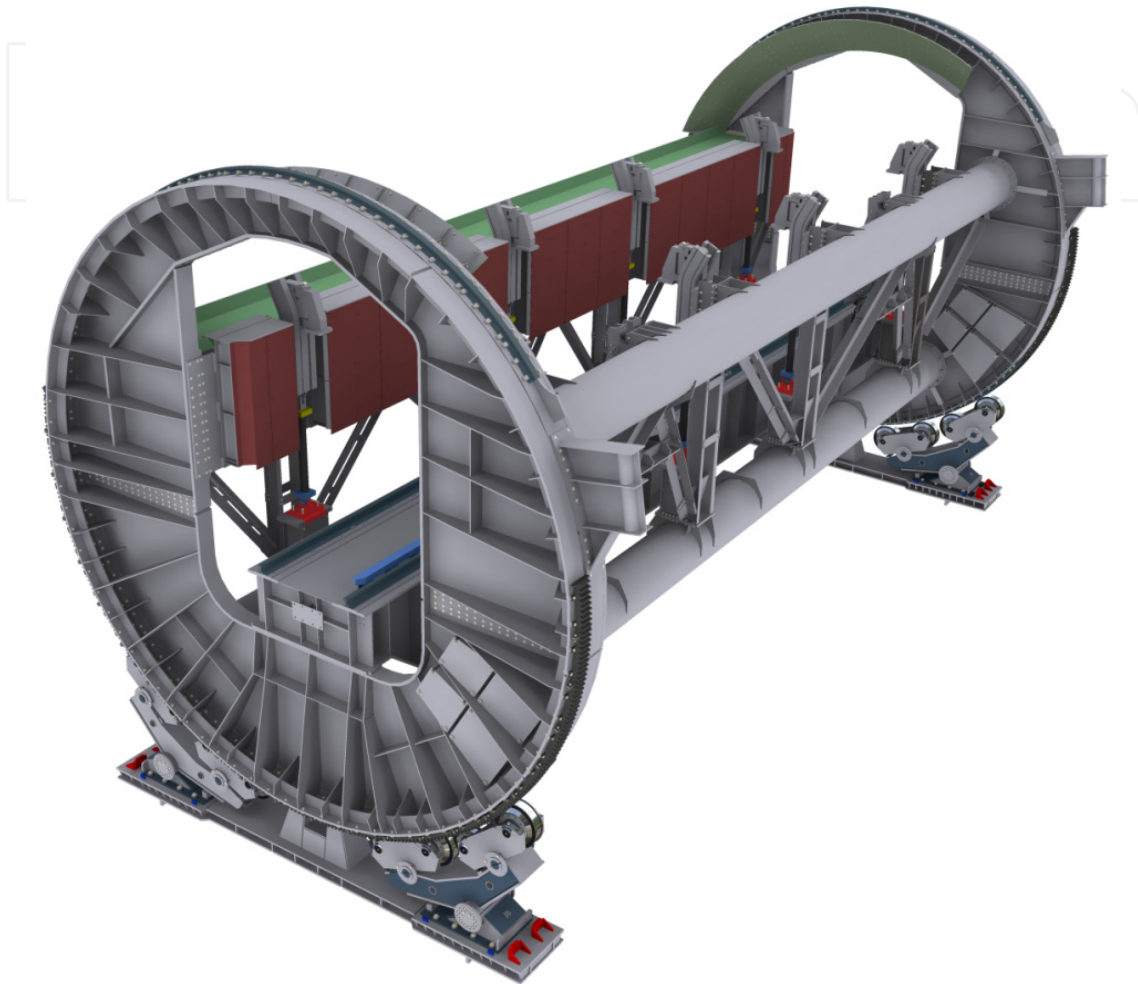


Figure 1. Car Dumper – overview, modeled by Inventor

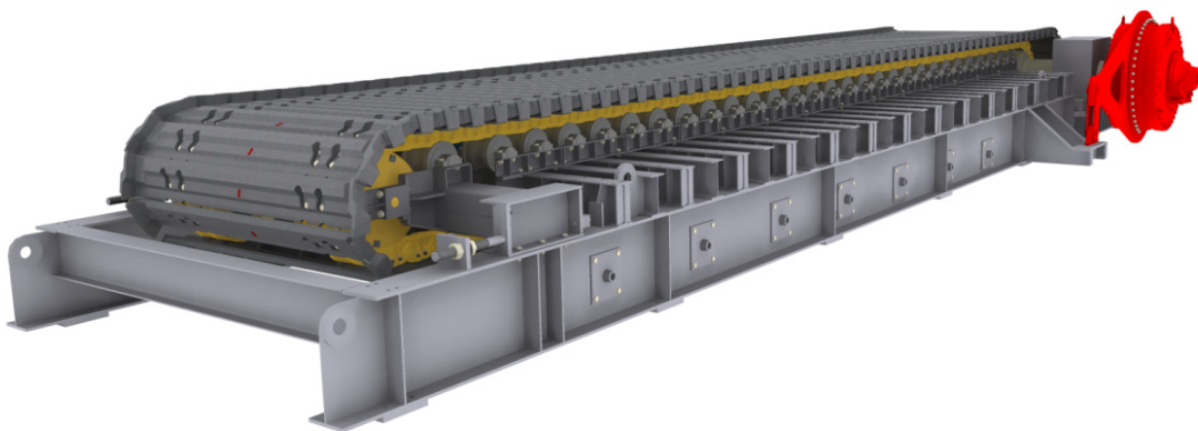


Figure 2. Apron Feeder – overview, modeled by Inventor

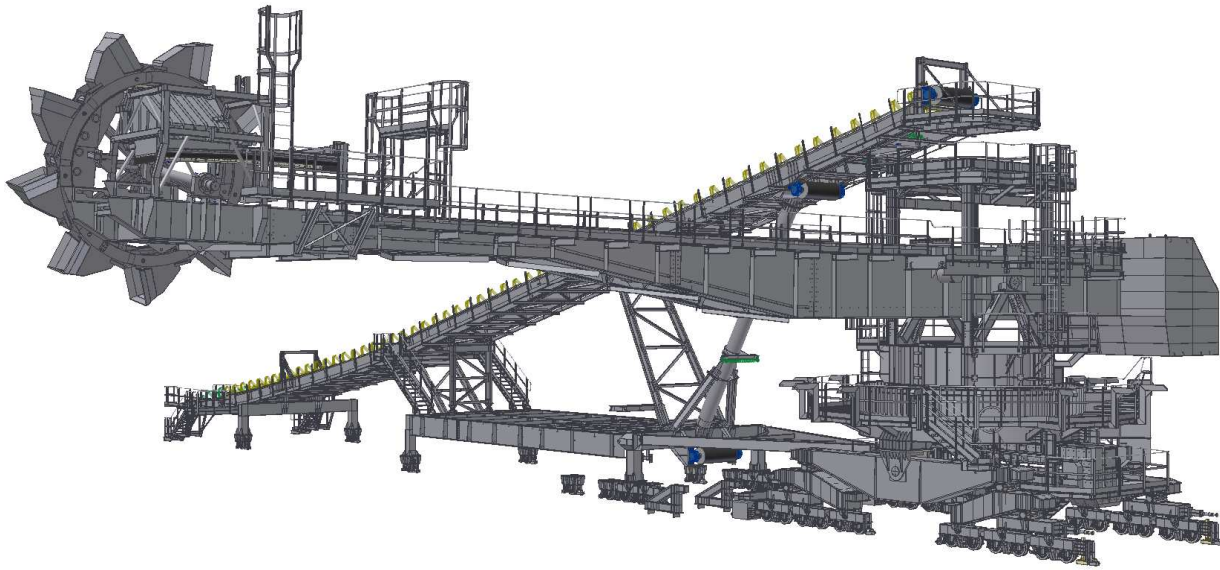


Figure 3. Stacker-reclaimer machine, overview – modeled by Inventor

Note – the 3D model in FEM however requires more deep knowledge of technologists and engineers in Stress x Strain analysis, stress tensors/matrixes, material properties, isotropy and anisotropy, stress states, residual stresses, von-Mises, Mohr circle and basic mechanics evaluations criteria. Even with the software advantages, the output in the FEM models still remains as the engineering duty;

- a. beauty pictures to present products in commercial and marketing scenarios (high sensation of reality);
- b. in order to produce manuals to operation and/or maintenance.

There also drawbacks in the FEM simulations, like;

- a. limits when interferences between parts are present in the model, which require non-linear analysis;
- b. usually residual stresses are present in the real component but are neglected in the model;
- c. small details in the big picture sometimes need to be handled or suppressed in order to have allow enough capacity to run the model, even when powerful computational machines are used.

In order to overcome such deficiencies in the FEM calculations, safety stresses are applied and fatigue coefficients used within fatigue models like; Goodman, modified Goodman, Gerber and/or Soderberg (15-16) .

3. Description of an analytical method

Most of the formulas of strength of materials express the relations among the form and dimensions of a member, the loads applied thereto, and the resulting stress or deformation. Any such formula is valid only within certain limitations and is applicable only to certain problems. An understanding of these limitations and of the way in which formulas may be

combined and extended for the solution of problems to which they do not immediately apply, requires knowledge of certain principles and methods that are stated briefly in the pulleys calculations ahead. In determining stress by mathematical analysis, being analytical or FEM, for example, it is customary to assume the material as elastic, isotropic, homogeneous, and infinitely divisible without change in properties and in conforming to Hooke's law, which states that strain, is proportional to stress. On the other hand, these assumptions despite of imposing certain limitations upon the conventional methods of stress analysis must be used in the form of safety factors. This precaution has given satisfactory results for nearly all problems in engineering, being in analytical or FEM models.

The pulley is basically composed by; expansion ring (when applicable), hub, shaft, disc and cylinder, as seen in the Figure 5 below. The calculation of individual components is still not an issue nowadays and classical formulas may be applied without main difficulties. But when there is an increase of components quantity and the interaction among them takes place, the analytical method cannot predict the real and accurate interaction, energy shared by each component in the assemble, due the imposed deformations. In other words, the proportion of deformation of each individual into the ensemble is a very complex to determine accurately and manually.

3.1. Division of forces in assemblies and redundant structures, pulley application

The concepts of force's flow are useful in the visualization of paths taken by the forces lines when crossing machines or structures from the load points till the support points. Whether the structure is simple and statically determined, the equations of equilibrium are enough to determine the reactions. On the other hand if redundant supports exist, it means additional supports to those required to satisfy the static equilibrium conditions, those simple equations are not enough anymore to explain the intensities (magnitude) in any one of the reactions. It happen due to the support works as a separated "spring", deflecting under load, proportionally to its stiffness, in a manner that all reactions are shared by all supports under an unknown way. Whether a stiffer rigid or under a rigid fixed deflection are in parallel with a less stiff spring or under a flexible deflection, the rigid deflection will absorb a higher portion of the loading. But whether a stiffer rigid or under a rigid fixed deflection are in series with a less stiff spring or under a flexible deflection, the loads absorbed are similar. The importance of such simple concept is applicable to all machines and real structures where exist the combinations of parts ("springs"), in series or parallel (7).

As seen in the Figure 4 the springs can be arranged in parallel arrangements as well as in series. If the springs are arranged parallel the deflections are the same but the total force F is divided between the spring 1 and spring 2, as follow;

$$F = F_1 + F_2 \quad (1)$$

once, $y_1 = y_1 + y_2$

$$\frac{F}{y} = \frac{F_1}{y_1} + \frac{F_2}{y_2} \quad (2)$$

from where is obtained;

$$k_p = k_1 + k_2 \quad (3)$$

to k_p as being the spring constant to each spring in parallel

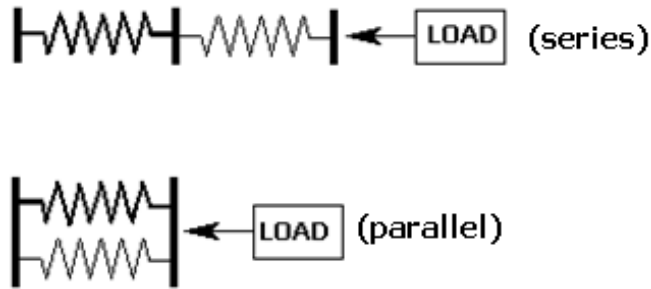


Figure 4. Different springs (stiffness – in parallel and series)

When the springs are arranged in series, the force F is the same to both springs, but the deflection of the spring 1 and spring 2 are associated to compose the total deflection, it means;

$$\frac{y}{F} = \frac{y_1}{F_1} + \frac{y_2}{F_2} \quad (4)$$

from where is obtained;

$$k_s = \frac{1}{\frac{1}{k_1} + \frac{1}{k_2}} \quad (5)$$

to k_s as the combined constant to springs in series

The diagram in the Figure 5 shows the pulley main components with imposed load. This load is transferred to all components and the total energy required to absorb such energy is composed by the sum of individual deformation. This deformation is directly related to the bending imposed in the Cylinder, the Disc and the Shaft. The sum of those deformations can be described as follow;

$$\delta_T = \delta_C + 2 \cdot \delta_D + \delta_S \quad (6)$$

δ_T is the total deformation, δ_C the deformation of cylinder, δ_S the deformation of shaft and the δ_D deformation of discs. The deformation of hub is considered zero due to its superior stiffness.

In the case of pulleys the system can be considered the same as explained with springs, what means, the cylinder and the shaft are in series and the discs are parallel each other but in series with the other components. The assembling equation then can be arranged as follow;

$$k_T = \frac{1}{\frac{4.L^3.E}{6.E.\pi.R_E^4} + \frac{L^3.C}{6.E.0.4.d^3.\delta} + b.\pi.(R_d+r)} \quad (7)$$

where L_E it the distance between the block bearings in the shaft, L_C it the length of cylinder, d is the cylinder internal diameter, b the disc thickness, R_d the radii of disc, r the internal radii of disc, E the Young Modulus, R_E the shaft radii and δ the thickness of the disc. The final deflection of the ensemble is determined by;

$$y_T = \frac{P}{k_T} \quad (8)$$

being y_T the total deflection in mm, P the resultant load applied in N and k_T the ensemble constant (N.mm).

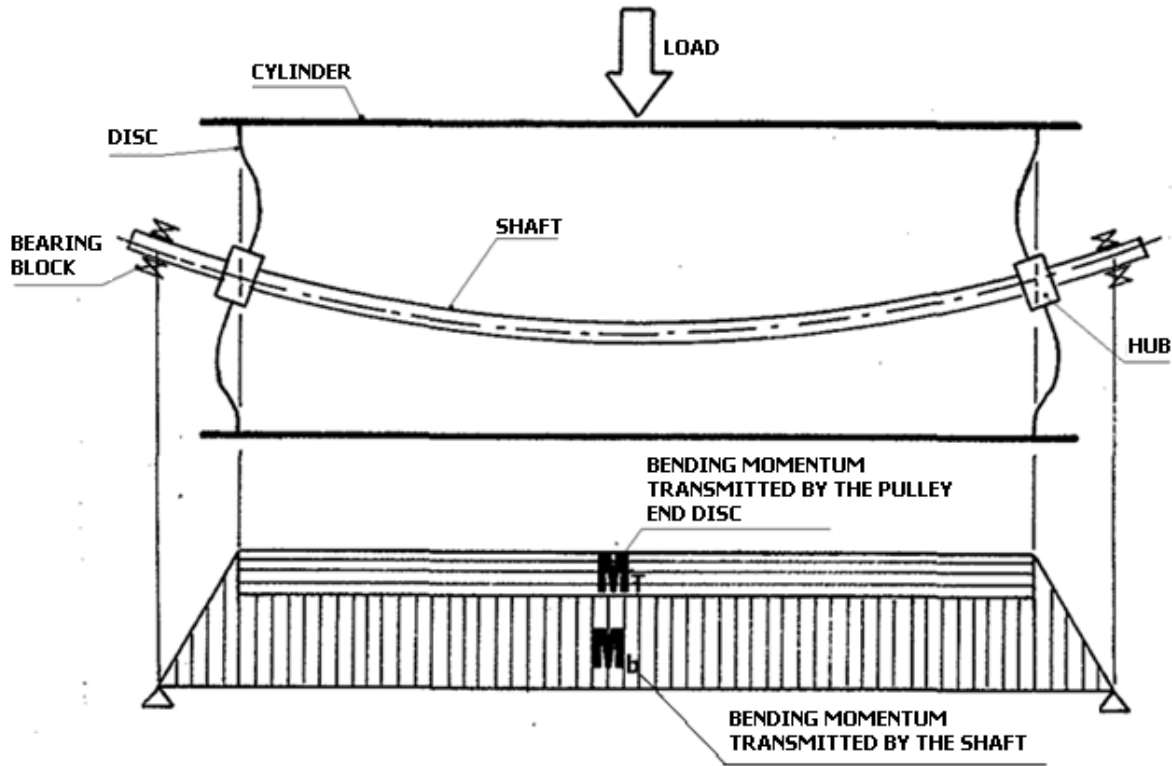


Figure 5. Diagram of pulleys loaded.

It is quite easy to identify the contribution of each element by a simple comparison between the pulleys components in the formula above and in the Figure 5. The load is transmitted by the pulley cylinder toward the discs, which suffer the high deformation due to its low inertia, then to the shaft which is bent due to the reactions at the bearing blocks. This is a normal condition found in driven pulleys, the drive pulleys contain an additional load, torque, transmitted from the shaft to the discs and lately to the cylinder. The drive pulley won't be covered at this chapter.

4. Description of Finite Element Method (FEM)

4.1. Method

Like described previously, the finite element method (FEM) is a very powerful technique for determining stresses and deflections in complex structures when compared with analytical

methods. With this method the structure is divided into a network of small elements connected to each other at node points. Finite element method grew out of matrix methods for the analysis of structures when the widespread availability of the digital computer made it possible to solve system of hundred of simultaneous equations (8). The FEM is then a computerized method for predicting how a real-world object will react to forces, heat, vibrations, etc... in terms of whether it will break, wear out or function according to design. It is called “analysis”, but in the product design cycle it is used to predict what will happen when the product is used (5).

4.2. Nodes and elements

A node is a coordinate location in space where the Degrees Of Freedom (DOFs) are defined. The DOFs of a node represent the possible movements of this point due to the loading of the structure. The DOFs also represent which forces and moments are transferred from one element to the next one. Also, deflection and stress results are usually given at the nodes. An element is a mathematical relation that defines how the DOFs of one node relate to the next. Elements can be lines (beams or trusses), 2-D areas, 3-D areas (plates) or solids (bricks and tetrahedra). The mathematical relation also defines how the deflections create strains and stresses. The degrees of freedom at a node characterize the response and represent the relative possible motion of a node. The type of element being used will characterize which DOFs a node will require. Some analysis types have only one DOF at a node. An example of this is temperature in a thermal analysis. A structural beam element, on the other hand, would have all the DOFs shown in Figure 6. “T” represents translational movement and “R” represents rotational movement about X, Y and Z axis direction, resulting in a maximum of six degrees of freedom.

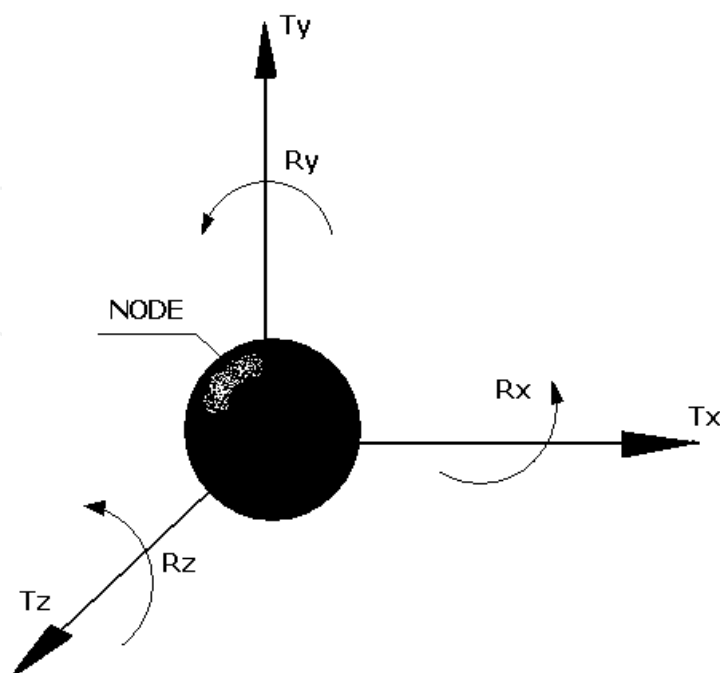


Figure 6. Degrees of freedom of a node (DOFs)

The elements, on the other hand, can only communicate to one another via common nodes. Elements therefore must have common nodes to transfer loads from one to the next, such as in the Figure 7 below.

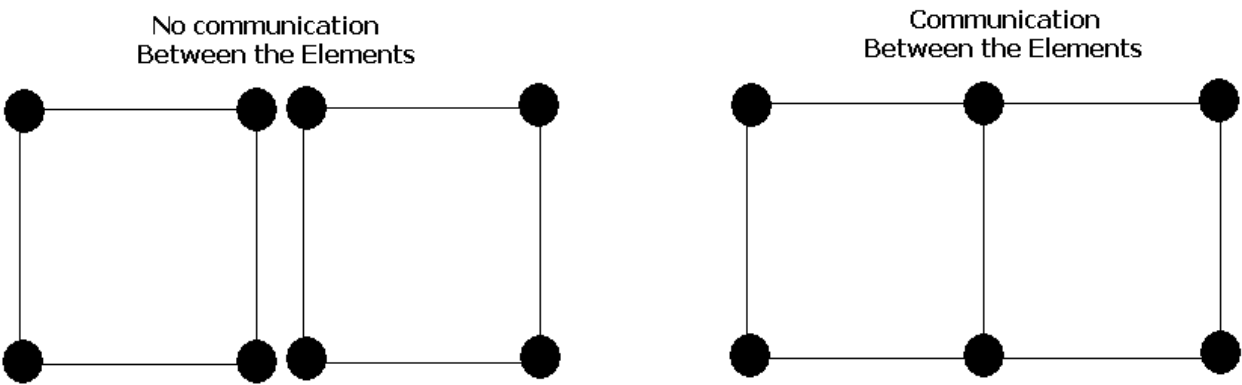


Figure 7. Communication through Common Nodes

Computer programs usually have many options for types of elements to choose, below the most usual elements (9):

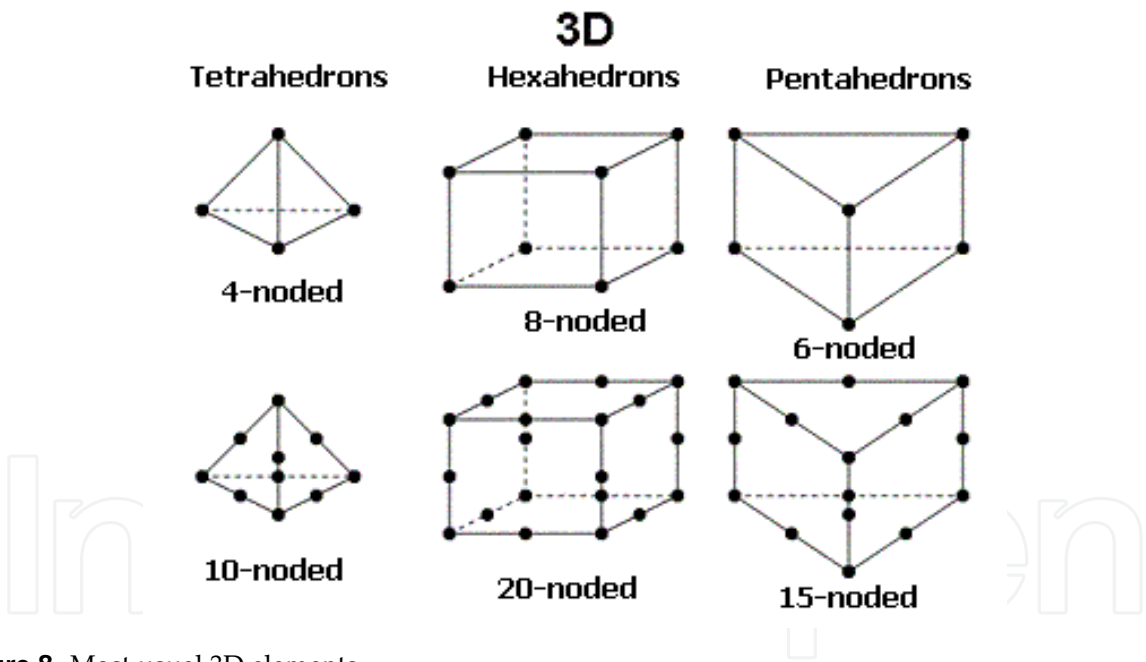


Figure 8. Most usual 3D elements

Since the applied load vector and element stiffnesses are known from the user input, the equation can be solved using matrix algebra by rearranging the equation as follow for the displacement vector:

$$\{x\} = [K]^{-1} \cdot \{f\} \tag{9}$$

where; $\{f\}$ is the vector that represents all of the applied loads. $[K]$ is the assemblage of all the individuals' element stiffness (AE/L) and $\{x\}$ is the vector that represents the

displacement. A is the area, E is the Modulus of Elasticity and L the length, and $\{f\} = \int_0^L \varepsilon \cdot dx$.

The strains are computed based on the classical differential equations. Stress can then be obtained from the strain using Hooke’s law. These basic equations do not require the use of a computer to solve. However, a computer is needed when complexity is added (4).

4.3. How to build the model

Each individual piece is modeled 3D and then the final assembling built by each part gathering in the final component (product - pulley), see Figure 9 below. The boundary conditions have constraints between the shaft and the hub, the hub and the disc and the disc and the cylinder; all constraints are bonded surfaces. The shaft has at the extremes joint constraints due the presence of block bearings. The bearing blocks usually are composed by spherical roller bearings when pulleys for Stacker-Reclaimers are the case.

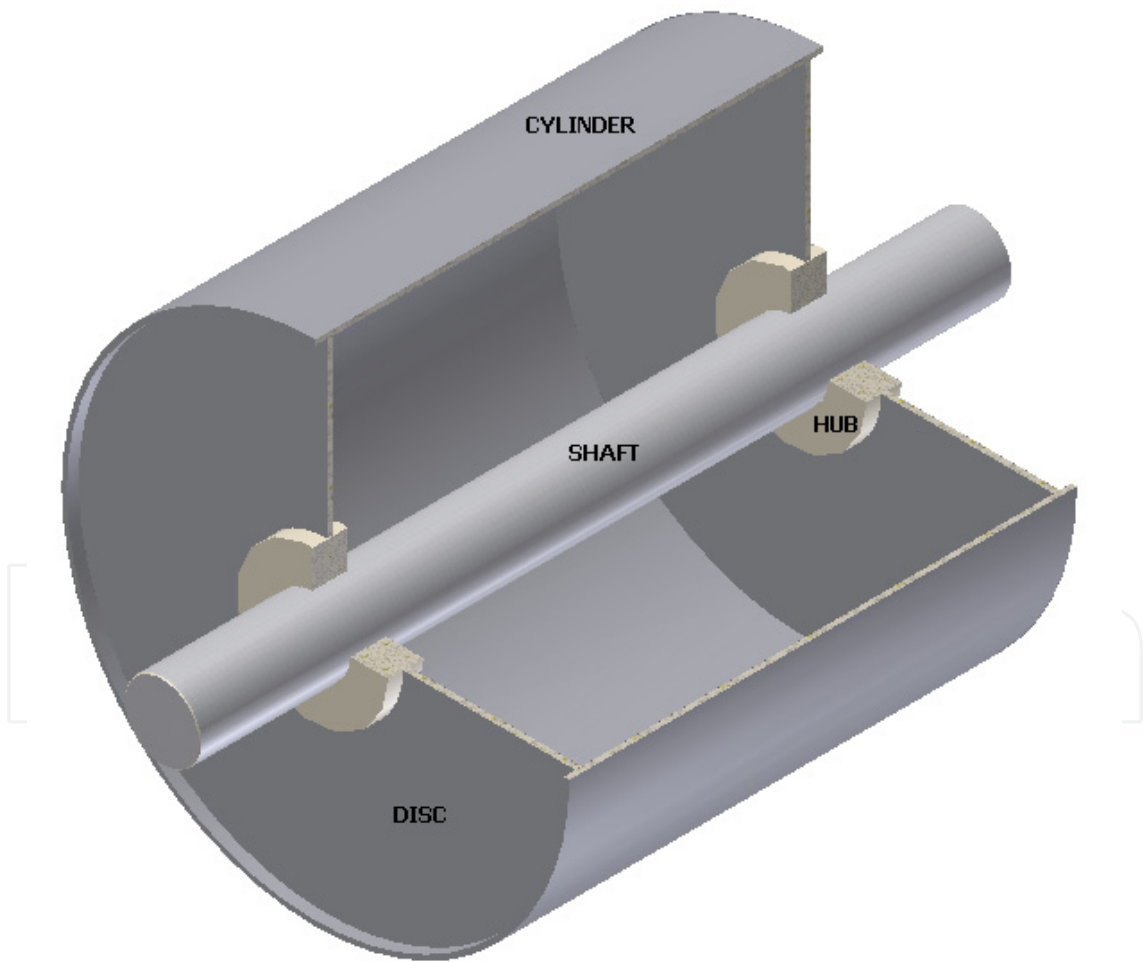


Figure 9. Pulley basic components (built in Inventor)
(sectioned 90° for better visualization)

The pulley studied has its main characteristics shown in the Table 1 below;

Component	Material	Diameter (mm)	Length (mm)	Thickness (mm)
Shaft	SAE-1045	150	1600	diameter 150
Discs	ASTM A-36	300 x 980	-	12
Cylinder	ASTM A-36	1000 (outside)	1000	10
Hub	ASTM A-36	150 x 300	-	70
Bearing block centre distance	-	-	1600	-

Table 1. Pulley main characteristics

When importing solid models that have thin parts, it is often better and simpler to analyze them using plate elements (5-10). Autodesk Simulation can be used to convert thin CAD solid models to plate elements. A plate element is drawn at the midplane of the part. Pulleys are commonly conditioned as described; it has solid elements, like shaft and hubs; and plate elements like discs and the cylinder. As shown ahead the difference is not too substantial but depending on the discrepancy of dimensions, comparatively between parts, the values (stresses outputs) can differ considerably. The DOFs associated with the plate elements are drawn in the Figure 10 and 11 that follows. Note that the out-of-plane rotation (R_z) is not taken into account because of plate theory, thus the plate elements have 5 DOFs.

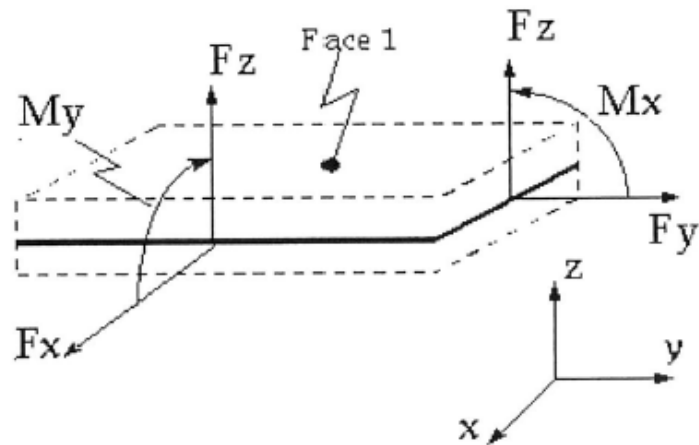


Figure 10. DOFs of midplane elements

The components were calculated by Autodesk Simulation and each component received a particular 3D element type and meshing configuration as follow; the shaft received a brick condition with material AISI 1045 as-rolled, the discs received a midplane condition, material ASTM A36, isotropic, the cylinder simulated midplane condition, material ASTM A36, isotropic. The constraints were determined in the shaft region of block bearings (joint constraint). The command, which simulate block bearings with spherical roller bearing, is universal joint, which constraint the DOFs at T_x , T_y and T_z as well as R_z (longitudinal to the shaft length) in the simulation to the first side, and the DOFs T_x , T_y and R_z to the opposite side.

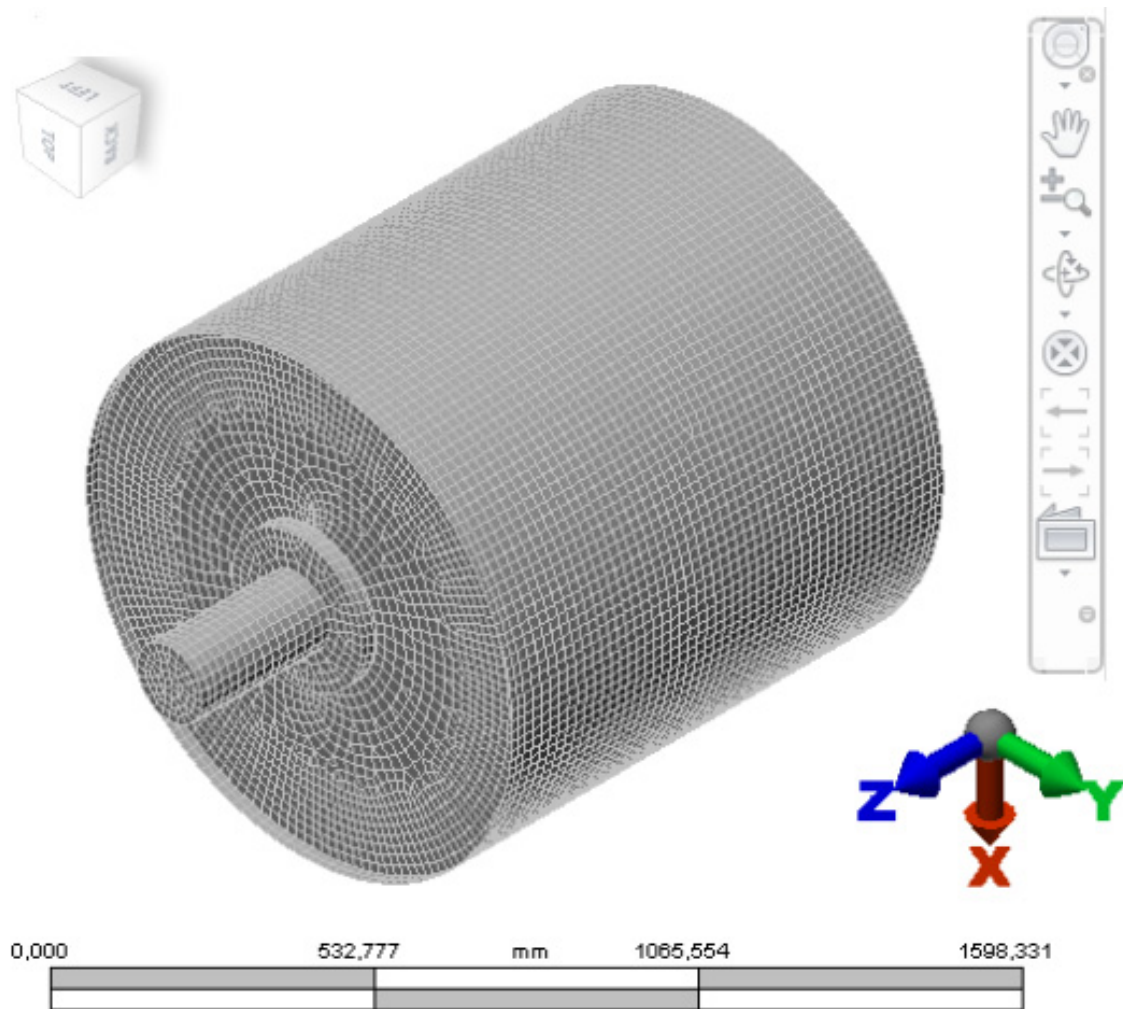


Figure 11. Pulley meshing – Autodesk simulation

The elements in the joints (block bearings) were considered with a very high stiffness's, which guarantee not interference in the stresses results in the model (Figure 12).

In terms of loading, there was a force applied perpendicular to the surface, which resulted in a variable pressure (parabola) around 180° of cylinder, represented by the following equation;

$$P = 0,47.R^2 + 0,47 \quad (10)$$

where P = pressure (MPa) R = pulley radius (mm), 0,47 = pressure (MPa).

The variable pressure is shown in the figure 13 below. The load applied on the cylinder outside and around 180° was 316kN. The analysis was done based on the previous description in the Autodesk simulation, being the von Mises stresses analyzed for each component, as follow by the Figures 13 to 18.

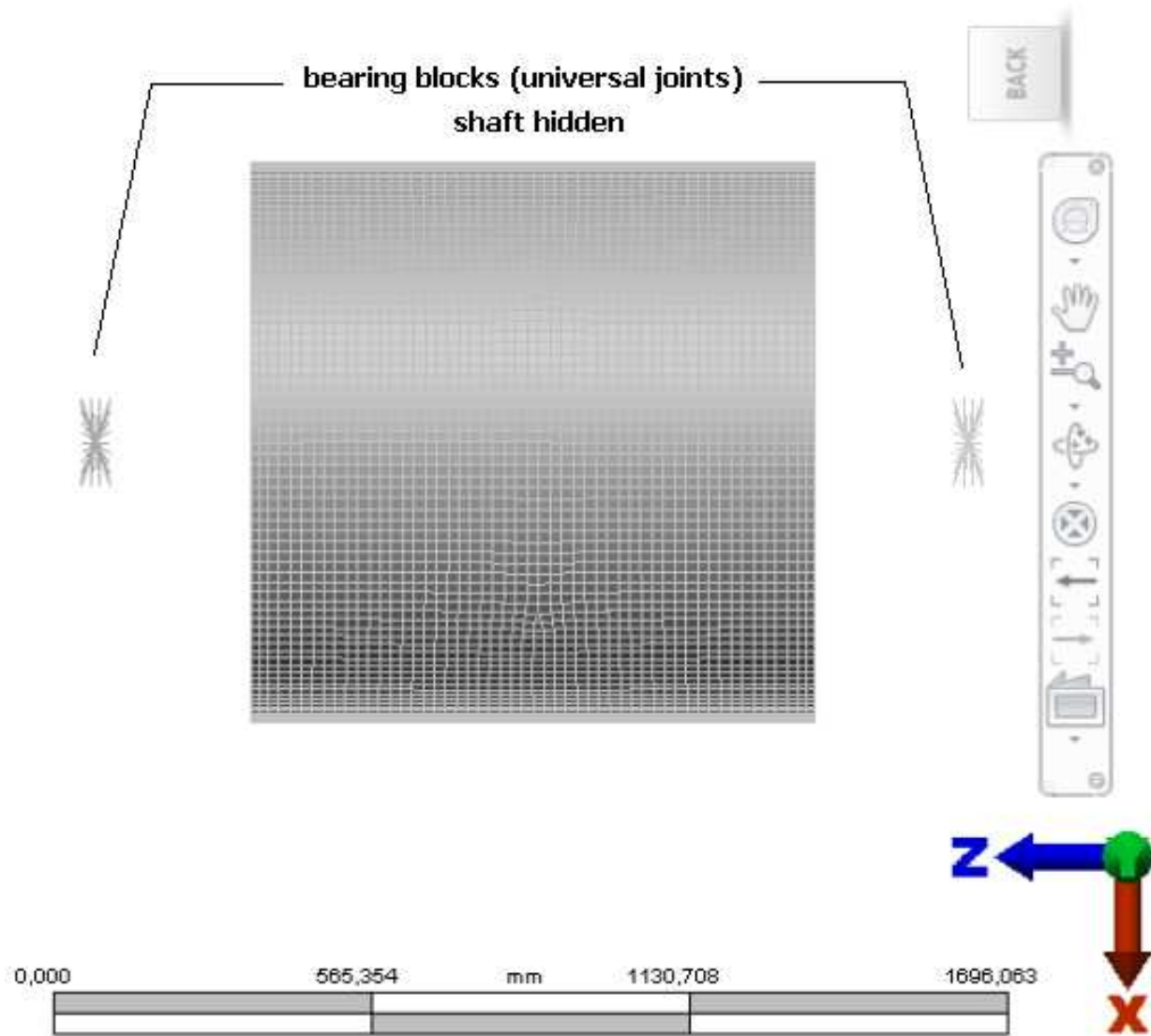


Figure 12. Constraint –Pin (universal joints)

Based on the fact the pulleys applications are dynamic (cyclic loading) the fatigue limit for each material was utilized in comparison with the stress range in reference to the equivalent stress (von Mises) by FEM (11-16). The Table 2 and Figure 19 reveal the main stresses on the pulley components. All the stresses are compared to the fatigue limit once this is the main phenomena the components is submitted. The stress range is calculated toward the von Mises stress (10-11).

Component	Material	von Mises (MPa)	Stress range (MPa)	Fatigue limit (MPa)
Shaft	SAE-1045	90	180	230
Discs	ASTM A-36	128	252	200
Cylinder	ASTM A-36	45	90	200

Table 2. Stresses on the main components (MPa)

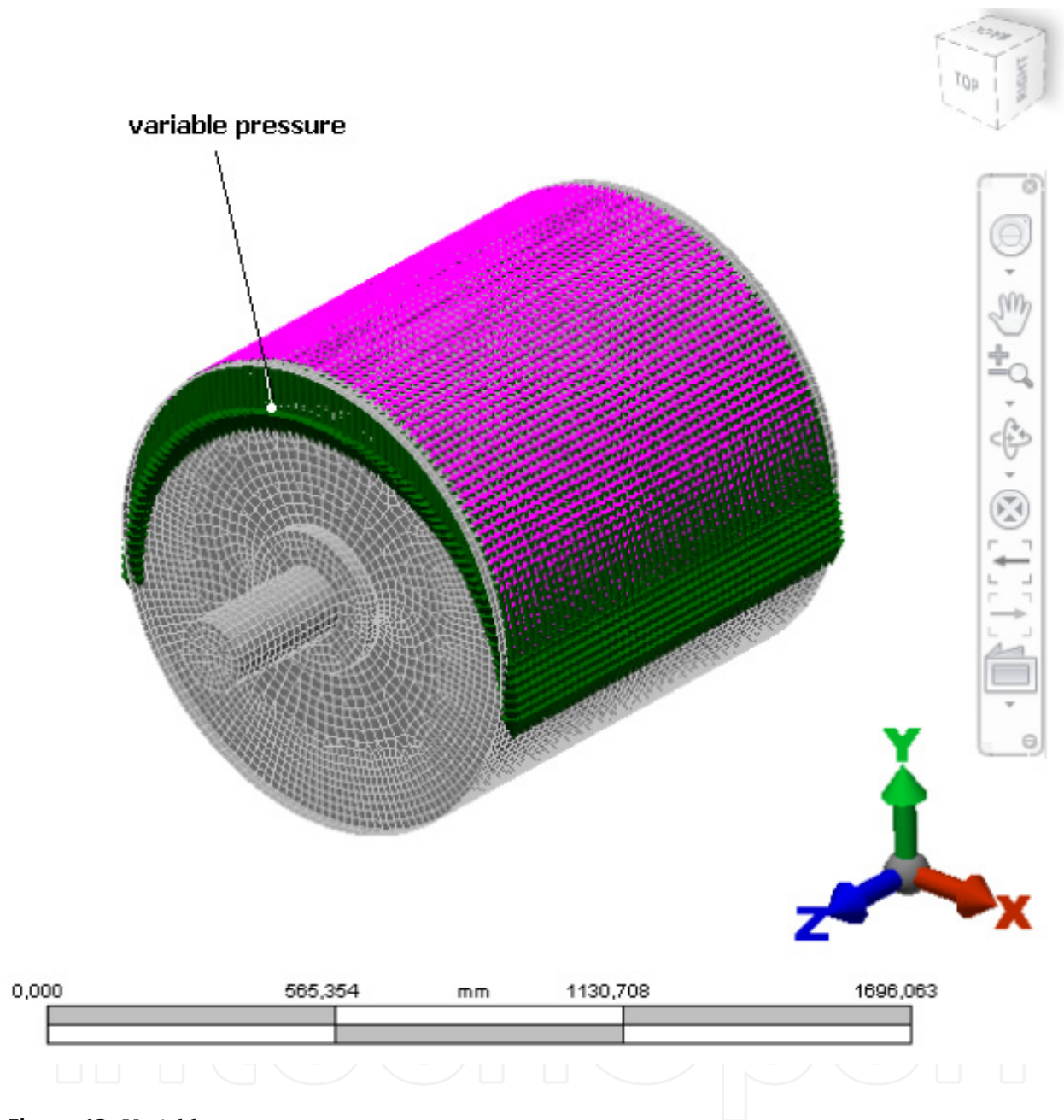


Figure 13. Variable pressure

All the stresses are under the fatigue limit except the discs, which overtake the limit over 52MPa. At this case the re-analysis of the discs thickness should be done and the thickness most of times increased or another type or thickness of disc applied. After the re-calculation, as expected, all assemble components have a new and different stress level, being highly recommendable afterwards the revaluations due to the fatigue limit consideration.

The values found in the analytical model (Table 2) were also compared with the FEM and are described in the Table 3 below.

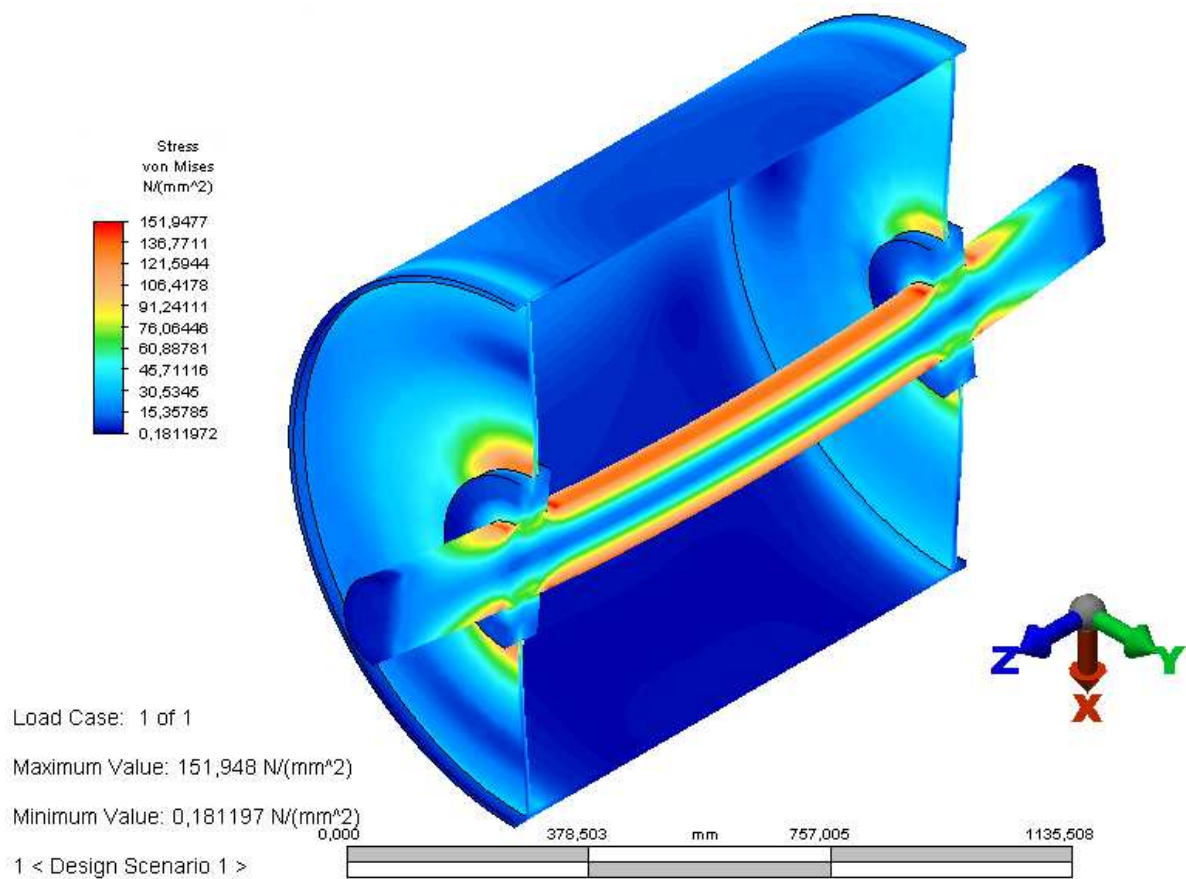


Figure 14. Stresses on the shaft (MPa)

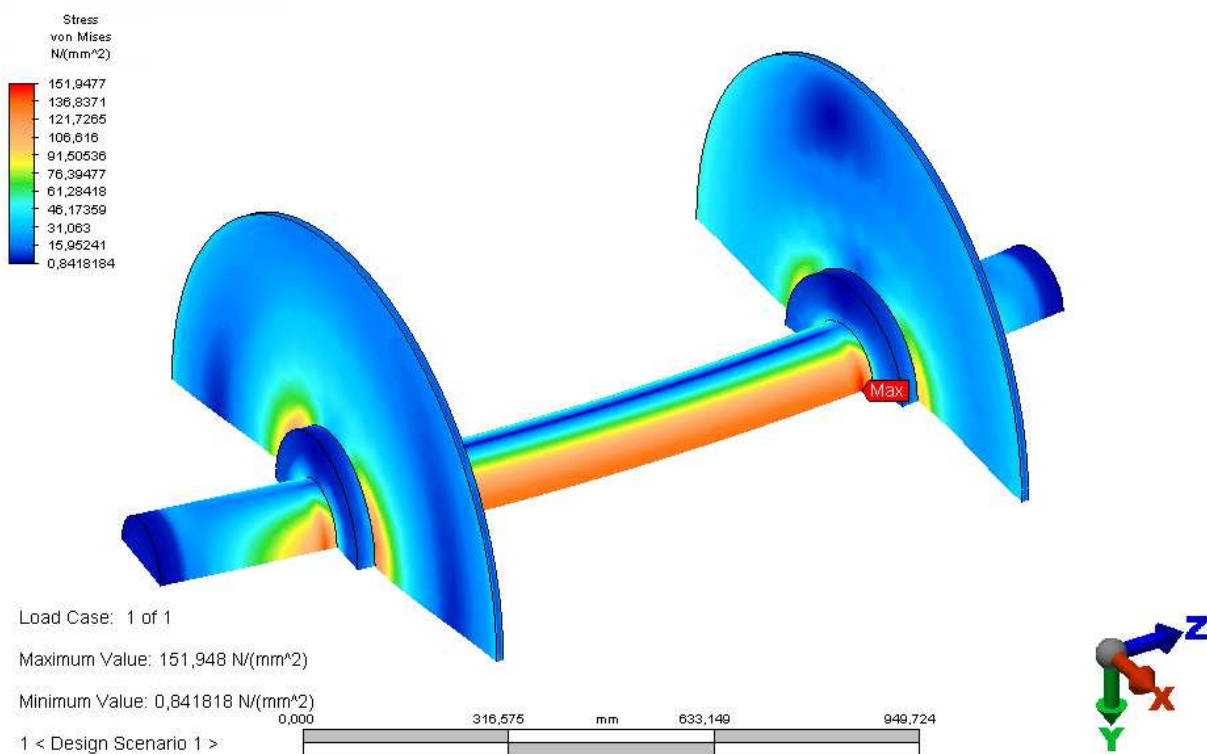


Figure 15. Stresses on the discs (MPa)

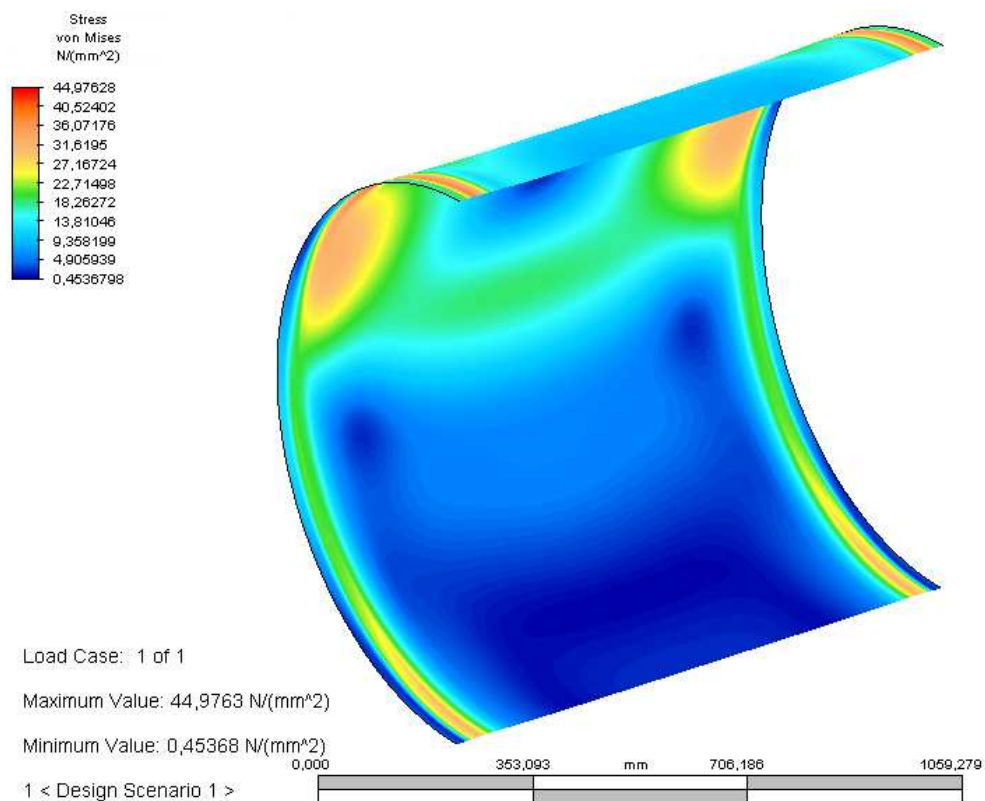


Figure 16. Stresses on the cylinder (MPa)

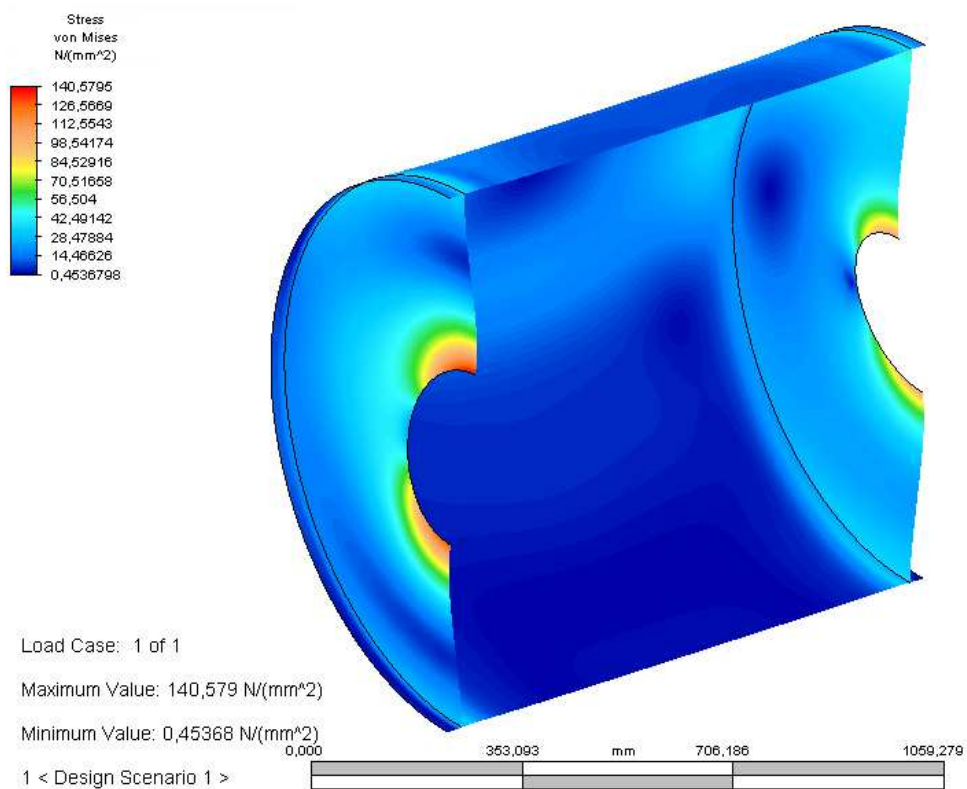


Figure 17. Stresses on the discs (MPa)

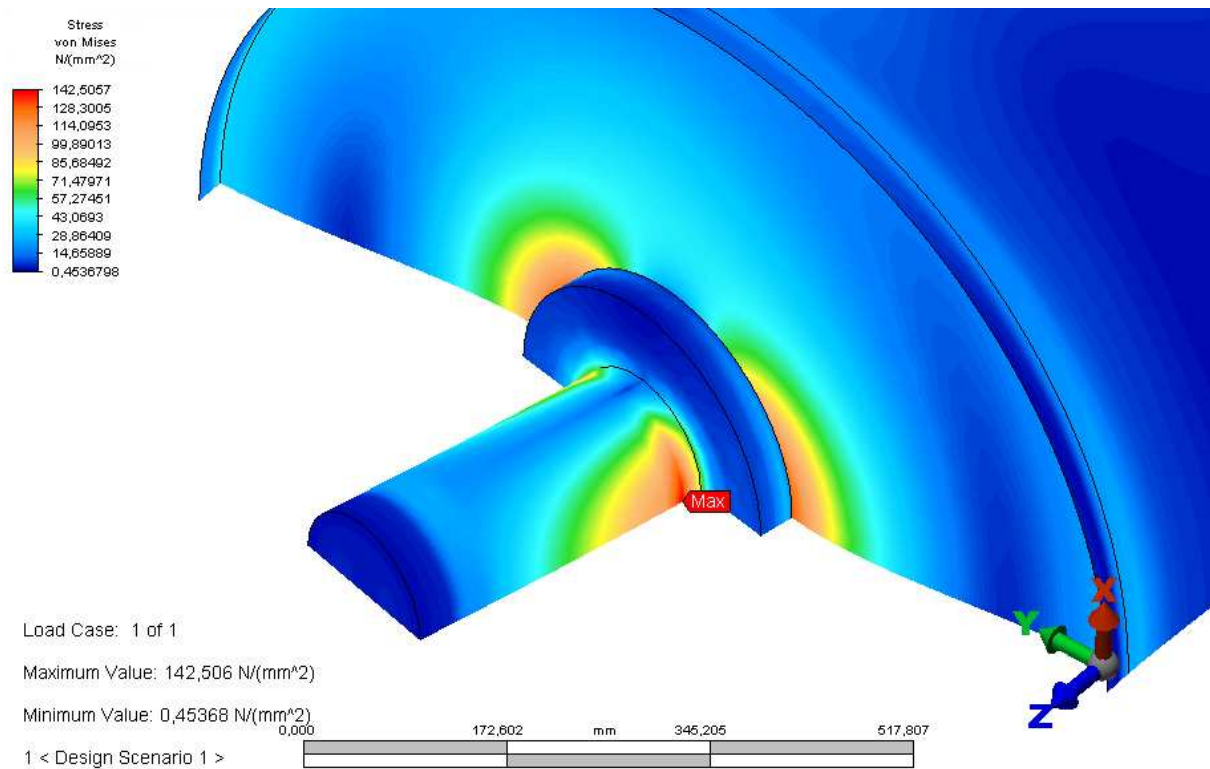


Figure 18. Stresses at the interface hub and shaft (MPa)

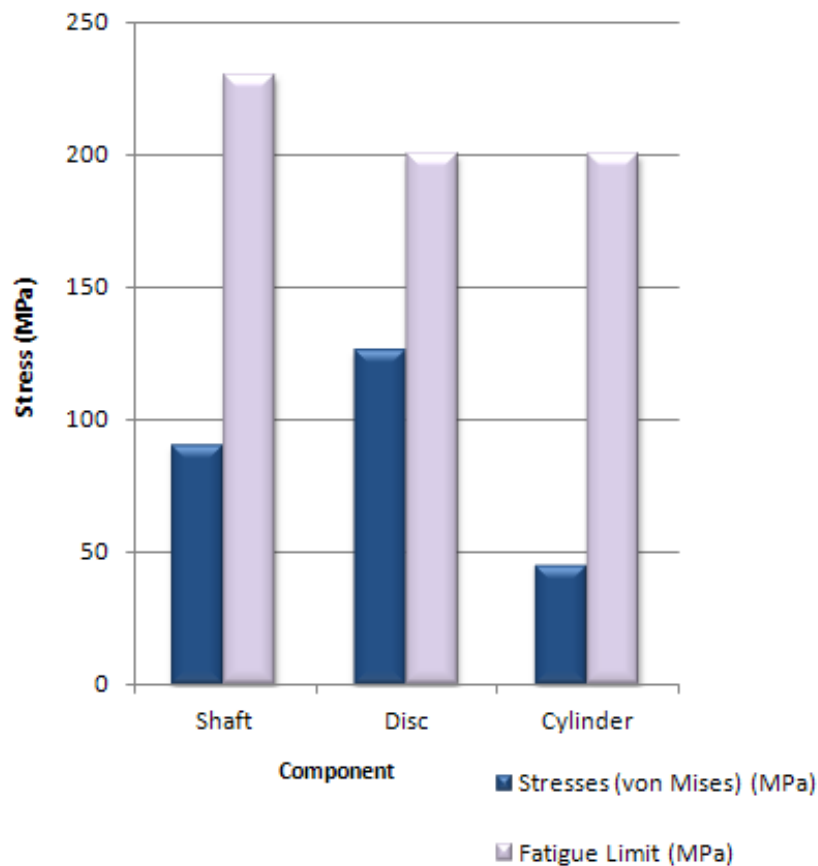


Figure 19. Stresses on the components (MPa)

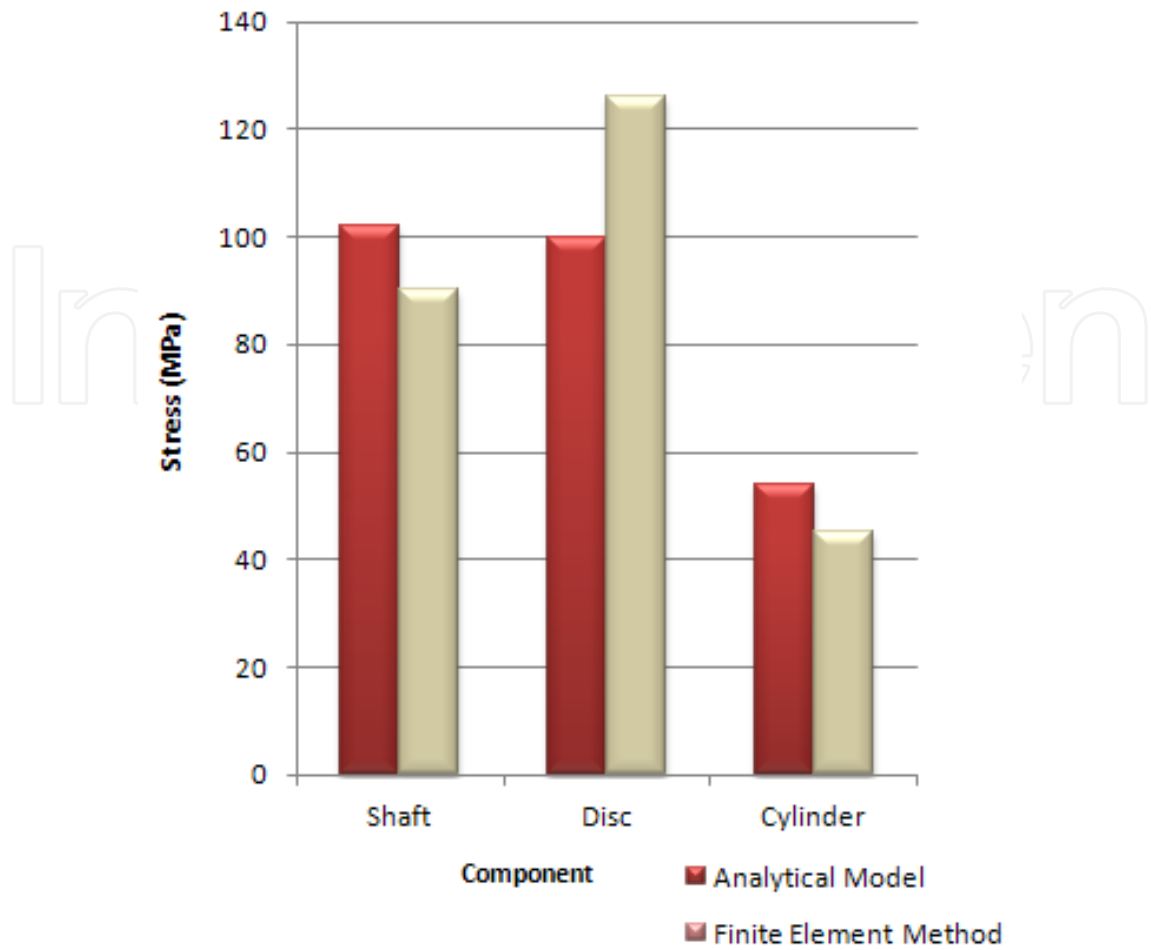


Figure 20. Stresses on the components (MPa)
Analytical versus FEM

Component	Material	Stress (MPa)	MEF (MPa)	Difference (%)
Shaft	SAE-1050	102	90	-10%
Disc	ASTM A-36	100	128	+26%
Cylinder	ASTM A-36	54	45	-33%

Table 3. Difference between analytical and MEF methods

There are differences in the results between the Analytical and FEM models (11). The equivalent stresses on the shaft are closer, around 10% difference, showing the lower value found in FEM, the discs are those which have medium difference and around 26%, being the stresses on the FEM higher than the analytical model and the third is the cylinder which had its lower value found in the FEM and around minus 33%. The differences are not too high but in certain cases should be taken into account when safety factors are in the limit due lean projects purposes. Neither the analytical model nor FEM are described in details once the idea is to bring the basic concepts used in components designing.

5. Conclusions

- The analytical calculation methods are still being used by most of components and machines suppliers;
- the analytical model requires safety factors in order to cover uncertainties in the processes like stresses due plastic deformations and/or complex thermal processes like weldings;
- the analytical model, as known, is not graphical like Finite Element Method (FEM) and sometimes considered obscure (not too complete like FEM) in terms of outputs. The data are not accurate like those presented by FEM either;
- Several impacts are in still in phase in the Mining business since the contemporaneous usage of models 3D and calculations by FEM;
- such changes from drawing 2D to 3D have brought the shift on the way of drawing, requiring less spatial thought than before but on the other hand more accurate drawings and awareness due interferences;
- the finite element methods is a powerful tool to calculate most of components and machines in the Mining area and nowadays being a reality for some companies of high technology;
- there are advantages when using FEM in terms of easy presentation of results due graphics and easy values (stresses and strains) obtained in different directions;
- the usage of FEM provide more sophisticated ways to analyze the calculations in terms of stresses and strains and displacements different directions, states and intensities;
- even with easier results brought by FEM, they are not free of analysis and positioning. The knowledge of mechanics in a deeper way in terms of intensity, state and direction of stresses and strains are more eminent nowadays;
- there are also drawbacks in the FEM like in the analytical, which require the use of safety factors. Stresses due different processes which generate stresses like plastic deformations and residual stresses due welding are not totally overcome in the method
- the case shown present a pulley used in Stacker-Reclaimers in order to analyse the differences found in both models, analytical and by finite element method. Any of them is wrong but they present certain differences in terms of stresses;
- the best way to evaluate the results is measuring the stresses by strain gages, which is not demonstrated in this paper;
- the evaluation and the comparison between the measured values and the calculated ones, being by analytical model or FEM are necessary;
- the FEM does not discharge the analytical model once the last has not presented problems in the field. On the other hand it must be replaced to more sophisticated tools like FEM, which brings several benefits beyond of more precision, productivity, friendly analysis environmental and cost savings due less weight and prior interference analysis.

Author details

Jairo A. Martins* and István Kövesdy

Machines and Components Product Engineering, Metso Minerals, Sorocaba, Brazil

* Corresponding Authors

Acknowledgement

The acknowledgements are addressed to CNPq (National Council for Scientific and Technological Development – Brazil), Mr. Rubens Costa, Vice President, Mining Operations, South America at Metso Brazil, Mr. Misael Ramalho and the Bulk-Machines Team and Mr. Vanderson L. Zangerolamo from MAPDATA due their valuable technical support.

6. References

- [1] Metso Minerals, Manual de Transportadores Contínuos. Fábrica de Aço Paulista, 4ª edição 1991.
- [2] Klaus-Jürgen Bathe. Finite Element Method. Published Online: 13 JUN 2008. Copyright © 2007 by John Wiley & Sons, Inc. DOI: 10.1002/9780470050118.ecse159.
- [3] Roark, J.R, Young W.C. Formulas for Stress and Strain. McGraw-Hill Kogakusha, Ltda., 1966.
- [4] Inventor 2010, User's manual, 1st edition 2010.
- [5] Autodesk® Simulation Mechanical. Finite element course manual, linear elastic simulations, 2011.
- [6] Ivo Babuška. The finite element method with Lagrangian multipliers. Numerische Mathematik . Volume 20, Number 3, 179-192, DOI: 10.1007/BF01436561.
- [7] Collins, J. Projeto Mecânico de Elementos de Máquinas. Rio de Janeiro: LTC, 2006.
- [8] Dieter, G.E. Mechanical Metallurgy. McGraw-Hill Inc. 1986.
- [9] The Finite Element Method – Theory. <http://illustrations.marin.ntnu.no/structures/analysis/FEM/theory/index.html>
- [10] Martins, J. A. Kovesdy, I; Romão, Estaner, Ferreira I. Case study of an evaluation of a Stacker Boom Luffing Pulley by Irwin's model and under BS 7910 guidance. Int. J. of Mining and Mineral Engineering 2011 Vol.3, No.4 pp.267 - 277. DOI: 10.1504/11.45469
- [11] J.A. Martins, I. Kövesdy, I. Ferreira. Fracture analysis of collapsed heavy-duty pulley in a long-distance continuous conveyors application. Engineering Failure Analysis. Volume 16, Issue 7, October 2009, Pages 2274–2280.
- [12] Bäckström M. (2003). Multiaxial Fatigue Life Assessment of Welds Based on Nominal and Hot-spot Stresses. VTT Publications 502, ISBN 951 38 6234 8.
- [13] Det Norske Veritas (2005). Fatigue Assessment of Ship Structures. DNV Classification Notes, N.30-7, July 2005.
- [14] Hobbacher A. (2007). Recommendations for Fatigue Design of Welded Joints and Components. International Institute of Welding IIW, Doc. XIII-2151-07/XV-1254-07.
- [15] Robert A Adey, John M. W. Baynham, Sharon Mellings, Tom Curtin*Fatigue Life and Crack Growth Prediction Using FEM Data. Computational Mechanics BEASY, Ashurst Lodge, Southampton, Hampshire, SO40 7AA, UK *Computational Mechanics Inc, 25 Bridge Street, Billerica, MA 01821
- [16] Aliabadi M.H., Rooke D.R., Numerical Fracture Mechanics, Computational Mechanics Publications, Southampton, U.K. 1991.