ABSTRACT

Paper Title: Driveshaft Seal Boot Finite Element Analysis

Author: Leandro Jaskulski
Co-authors: Leônidas Coutinho
César Franck
Rubens Gehlen
Company: ATH - Albarus Transmissões Homocinéticas ltda.
Porto Alegre, R.S., Brazil

This study registers a finite element analysis of a seal boot used in constant velocity driveshafts, that was performed in the MSC/NASTRAN software for Windows NT 4.0 in a Personal Computer. By this analysis the structure's behavior was studied in its work life before construction of the actual prototype.

Analysis was made in existing seal rubber boot, showing good accuracy level when compared with actual part displacement test results. At first, critical joint motions were selected to identify the constrains and symmetric plans. The hyperelastic material was approached to an elastic material in this first step to apply in a plate element model of the part. Then the results of general part behavior, like displacements, and contacts were examined and contrasted with laboratory test reports of the real part.

Prototype development time can be saved in the design phase since the part can be improved by this theoretical prediction tool. Failure points can be predicted by the analysis saving resources such as developing production tools, making prototypes and testing them to finally detect the items that have to be improved.

INTRODUCTION

The function of Constant Velocity Driveshafts is to transmit torque and motion from engine to wheels. It allows steer and rotate wheels at same time, keeping an homokinetic velocity between wheel and shaft though the variable angle between them. This characteristic is offered by homokinetic joint that must have its internal parts involved by grease and protected against debris.

The basic requirement of a driveshaft boot seal is to secure the joint retaining the grease and protecting it. So the boot design must provide a service life for at least of the joint life, precluding an early damage. This part, in general, can be made by rubber or thermoplastic material that provide an elastic skill to the part that works under high rotations and angles following the joint movements.

PROBLEM DEFINITION

On the finite element analysis the boot design is verified regarding the best profile to endure under constant velocity joint motions. In the results, contacts between boot surfaces, and between boot and shaft are visualized and strain levels are compared to material strain limits too - if they are critical in high joint angles.

In the boot analysis, the part behavior - large deformations - and the material properties - rubber or thermoplastic - are nonlinear, and there are contact between boot portions and boot and shaft. Just recently nonlinear analysis can be made in Personal Computers due to software's improvement in this area, even that Finite Element Analysis Method is well known as a good tool for linear structural analysis. This study presents a method to model and analyze a seal boot, using the nonlinear static analysis features of a MSC/NASTRAN software in a personal computer.

The slide lines technique was another improvement in MSC/NASTRAN, for personal computers, that allows to analyze boot contacts under high angles in order to predict the structure behavior in work life extreme situations.

With the objective to develop a procedure to analyze a seal boot and measure its accuracy, this study is concerned with an existing part that could be tested and compared with the analysis results.

ANALYSIS

SOFTWARE AND HARDWARE

The MSC/NASTRAN software that is used for these analysis has some features that are important for a specific seal boot analysis in a personal computer. As the requirements to a seal boot study, nonlinear static analysis are provided in addiction to slide line elements that allows contacts between elements. So these items allow analyze the boot in its extremes work situations and read its general behavior.

The hardware used is a personal computer with a 166 MHZ processor and 130 MB RAM. The computer also has two hard disks to improve the convergence time as well as a Windows NT 4.0 platform.

So due to software evolution this complex analysis can be made on inexpensive hardwares like personal computers.

FINITE ELEMENT MODEL

A 3D model was generated by the seal boot profile as its design, to represent the actual boot structure like in figure 01. The shaft was modeled by plate elements with steel properties and high stiffness. Shaft was represented from its connection on the smaller boot opening to the rotation center of the joint.

The boot model was made with plate elements, with material properties approached to na elastic material behavior.

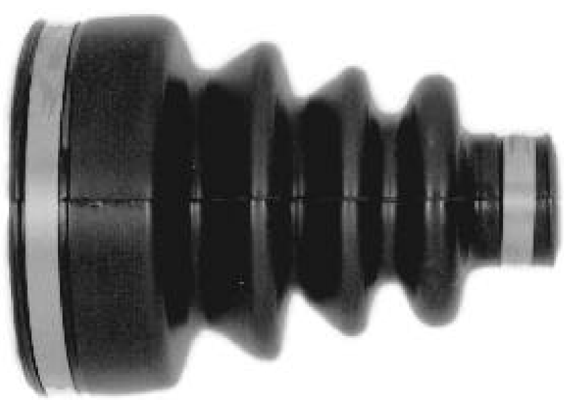

Figure 01 - The Seal Boot part.

3D PLATE ELEMENT MODEL

Plate element is a 4 nodes isoparametric element with membrane capability of the shell element and stability model for bending (reference 1).

Plate element model considers some simplification, because the structure is modeled by the profile center line with the variable thickness indicated in the element properties. So the more are

the thickness variation, the more element properties a model has. This is a good agreement in the profile thickness analysis when the general part displacement has to be modified by the thickness alteration. This model becomes practical because it is just modify the thickness value at the properties and analyze the model again to see the changes in displacement results. So it is not necessary to construct another model in order to get the optimum boot profile.
This model has 3324 elements and 4475 nodes and its profile is shown in the figure 02.

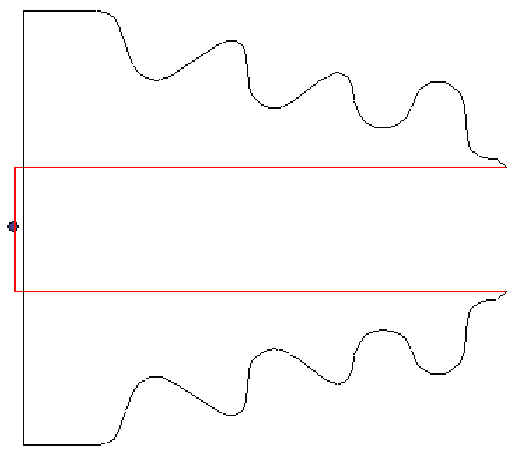

Figure 02 - Plate Element Model Profile.

MATERIAL PROPERTIES

Material properties are one of the most important variables that influence the accuracy of a finite element analysis. Every prediction analysis has its results depending on the equation that represents the material behavior.
Seal boots are usually made of rubber compound or thermoplastic material that has a reasonable strength and good elasticity. Both materials exhibit elastic response up to large strains, so they are called hyperelastic materials. They are very difficult to represent mathematically because their behavior depends on many variables. The closest method to represent this material behavior is to describe it in terms of a scalar strain energy function (reference 2). The general state of rubber can be approximated by a Taylor's polynomial as the Mooney-Rivlin function, that is the first order approach, or by a bigger polynomial order. So the more is the function energy order, the more is its mathematical accuracy. The finite element software, employed in this study, allows to enter the material polynomial coefficients or they can be determined by the experimental data.
As a first step, the study of a rubber seal boot approximated the hyperelastic compound to an elastic linear material with 3.0 Mpa as Young's modulus, and Poisson's ratio of 0.45 like a nearly incompressible material because of a software limitation of fully incompressible material (ν = 0.5). Through this approximation the displacements and strains levels results were determined if they were satisfactory.

The shaft model was made with steel properties in order to have more stiffness than the seal boot, so its strain behavior becomes not significant in this analysis.

SYMMETRY PLANS AND GENERAL CONSTRAINS

As this static analysis was about the seal boot under joint angle positions to recognize its behavior, a symmetry plan was defined for structure. The joint rotation was applied in z axis and the symmetry plan was defined on the xy plan. So the model was generated with half number of elements, saving modeling and analysis time.

The constrains were created to steady the symmetry plan, in the boundary nodes by restricting degrees of freedom as rotation in x and y axis and translation in z axis. Constrains were also defined to simulate the connection between joint and seal boot, fixing this boundary by constricting all degrees of freedom. The boot small opening was fixed in the shaft by sharing the same model nodes.

LOAD

Load was applied in terms of displacement as the angle value. In the model, load was applied by a rotation angle in the z axis at the rotation center of the shaft, calculated by the joint rotation center.

As first step, 44 degrees were applied simulating a life test standard condition.

Finally, the maximum joint angle, 47 degrees, were applied to evaluate the boot behavior at joint extreme situation. Although the joint does not work at this angle in the actual vehicle, this situation can be reached during the assembling process.

NON-LINEAR CONVERGENCE FACTORS

Non-linear analysis is used in parts that suffer large displacements and rotations, and their boundary conditions may change. So displacement transformation matrix is no longer constant, then compatibility and equilibrium are satisfied in a deformed configuration (reference 3). The interface contacts can be only approach by this analysis method on the software that is used in this case.

The solution strategy for this analysis is simulate the general behavior of the part through advances in increments. Iterations are required for each increment and the solution is obtained when the convergence criteria is satisfied.

The best number of increments and the convergence criteria as work and load were selected to lead at first to the convergence, and finally to the lower convergence time. The slower convergence was 47 degrees of displacement applied, that spend about 3 hours in the process time.

So this is the best analytical approach for a seal boot part that works with large displacements and strains as well as contacts between surfaces.

DISCUSSIONS

The displacement results of plate element model at 44 degrees, as test condition, are shown compared with actual part photograph at same angle, in the figure 03. The displacements results were very similar to actual part behavior. But in the other hand a gap could be noted at the bottom side because of surface approximation by the thickness center line. This is a disadvantage point of this kind of model where the “virtual” thickness have to be considered in final displacement overview.

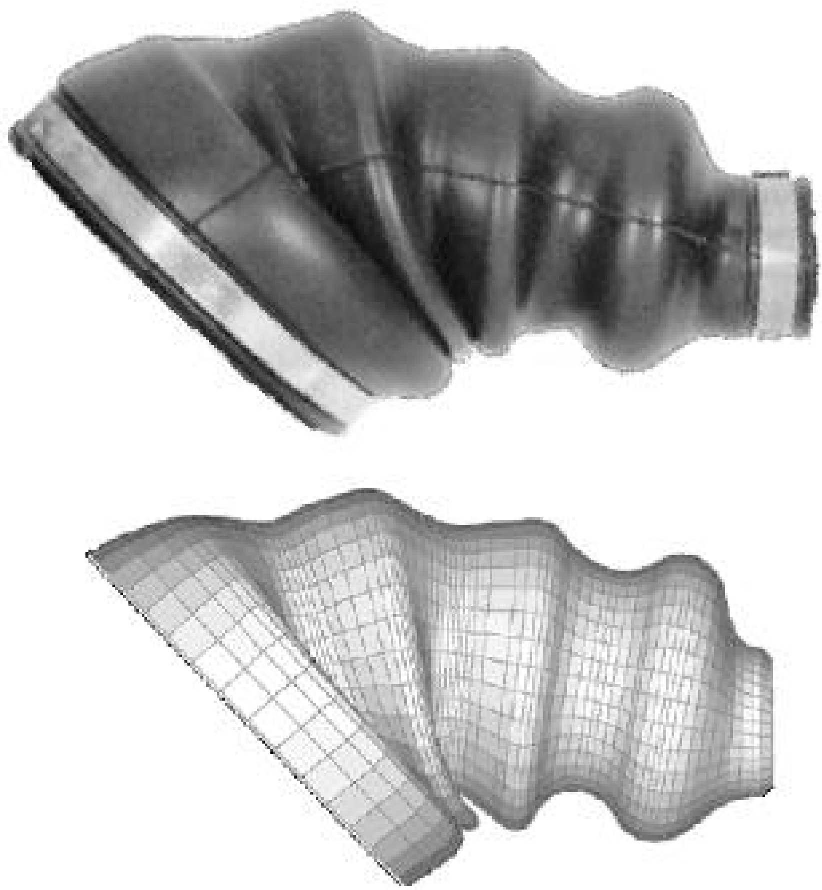

Figure 03 - Plate element model displacements at 44 degrees and real part behavior.

The general profile displacements can be analyzed as in the figure 04. In this case, contacts could be visualized between boot portions and between boot and shaft. Here also the simplification modeling has to be considered to analyze the results.

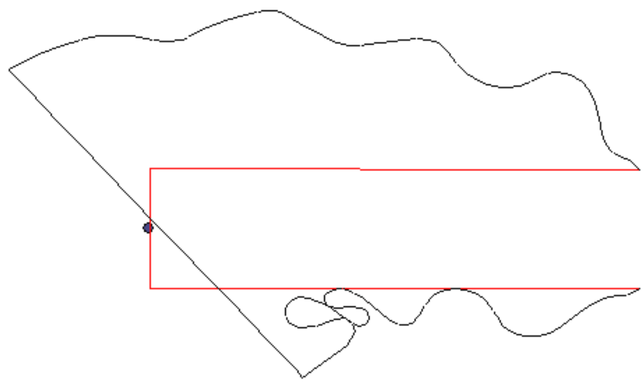

Figure 04 - Profile displacement of Plate element model.

Finally, the displacement results at 47 degrees, as extreme joint condition, is shown compared with actual part photograph at same angle, in the figure 05 where the plate element model results are shown. Here a structure instability point, a concave deformation, could be spotlighted in the top view of the model like in the real part, even the model was a plate element simplification.

This kind of instability can be very harmful for the seal boot part. If the part is assembled with this concave deformation, it would take a initial damage on the instability point by the part rotation movement. In addiction to this, concave deformation remains when part has the usual angle reestablished, as noted in most of tested parts.

So this kind of predict is very useful, during design development process, in order to check work life problems of part before prototype manufacturing.

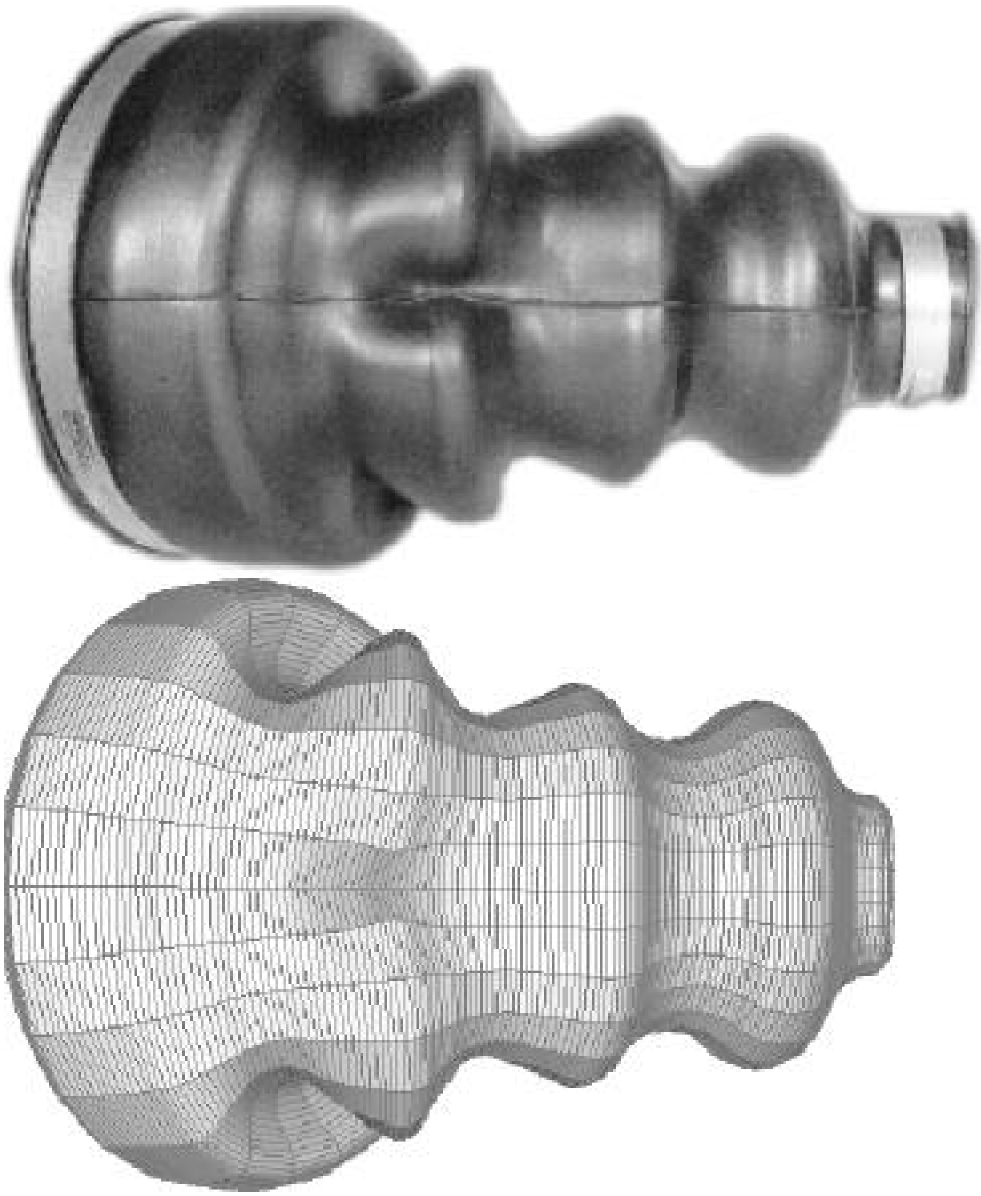

Figure 05 - Concave deformation in plate element model at 47 degrees and actual part behavior.

SEAL BOOT LIFE TESTS

A boot life tests were executed at the experimental engineering laboratory, where 5 part samples were tested at 44 degrees as the specific life test standard.

All the parts presented a crack in the peak number 2 and intense contact marks between the peaks 1 and 2, 2 and 3 and also contact marks between shaft and trough 2 and 3, as indicated in the sketch at figure 6.

So the contacts between boot portion (trough 2 and 3) and shaft, that were predicted by theoretical analysis, could be showed by the practical test results.

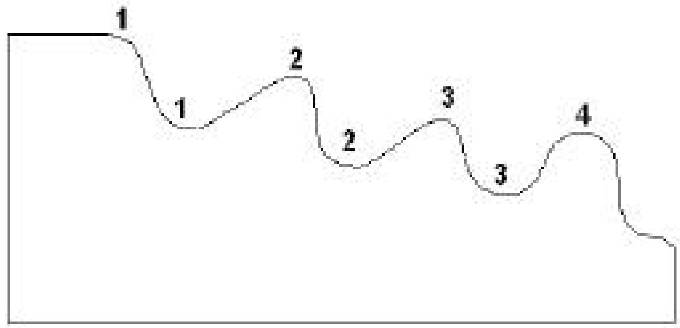

Figure 10 - Sketch of Boot Profiles identifying peaks and troughs.

FUTURE WORK

For future work the following subjects are proposed:

1- Strain levels results of the model compared with experimental boot strain measure, looking for a better modeling to predict this subject.
2- A solid element modeling of the seal boot part to get more accuracy in terms of displacement since the thickness center line approximation is not applied.
3- A strain energy function approaching for hyperelastic material behavior looking for an accuracy prediction method in terms of material strains.
4- Optimum convergence factors to improve the total convergence time of analysis.
5- Modeling and analyzing of thermoplastic seal boots.

CONCLUSION

This study showed that non-linear element analysis method was very useful to predict displacements of the seal boot part during design development stage, even with a hyperelastic material behavior approximated to a linear elastic material and the model was a plate element approximation.
Plate element model showed a good skill for thickness analysis as the general part behavior could easily be found for each different thickness configuration. In the other hand, this kind of model leads to misrepresentation of actual part behavior because of the center line surface representation.
This model were very efficient to preview concave deformations in boot parts. This was a good point of this kind of analysis, in the MSC/NASTRAN software that can preclude, during design development stage, a mostly probable point of failure in the seal boot part.
In terms of strain levels the results were not compared with the actual part behavior. At first step, seems that strains levels depended of the model meshing as well as the kind of element used. But the strain accuracy is beyond the scope of this work and should be studied in future works.
Although this analysis was made with inexpensive MSC/NASTRAN for Windows NT software and in a Personal Computer hardware, it showed a very useful method to simulate the general

boot part behavior. By this method, time can be saved in the design development stage by the behavior's predict before prototype manufacturing.

REFERENCES

1- Nicholas M. Baran, Finite Element Analysis on Microcomputers, McGraw-Hill Book Company, 1988.
2- B.G. Kao and L. Razgunas - Ford Motor Co., On the Determination of Strain Energy Functions of Rubbers, Proceedings of the Sixth International Conference of SAE, Society of Automotive Engineers 1993.
3- National Agency for Finite Element Methods & Standards, A Finite Element Primer, National Engineering Laboratory, 1991.