Using ABAQUS in tire development process

Jani K. Ojala

Nokian Tyres plc., R&D/Tire Construction

Abstract: Development of a new product is relatively challenging task, especially in tire business area. It is very clear that the performance of the new product must be higher compared to the old one and therefore the designers must take into account all the expectations of consumers that this new product has to meet. The development process typically takes several iteration loops before a decision has been reached. Using empirical methods only to reach the goal can be quite expensive and time consuming in contrast to modern simulation tools which enable reducing both costs and development time. Also the advantage of using modern simulation methods is that one can easily increase the amount of test data, because of "fully instrumentation". These modern tools, however, requires that user has a special competence in this specific field.

This paper describes our experiences with ABAQUS. A procedure to analyze stresses near belt edges with ABAQUS is also presented. Our relatively brief history with ABAQUS has already now shown the advantages of using such modern designing tools.

Keywords: ABAQUS/Standard, ABAQUS/Explicit, Hyperelasticity, Rubber, Tire, Viscoelasticity,

1. Introduction

Finite Element Method (FEM) has been used in tire design for many years. However, Nokian Tyres designers have used mainly empirical methods in tire development and research. This has been due the complexity of our core competence: a studded winter tire. Until now, it has not been possible to successfully simulate e.g. a rolling studded winter tire on ice.

Figure 1 represents the traditional tire development process with a new tread pattern. After the first trials have been tested, some improvements are usually needed, and new moulds must be ordered. Usually the whole process takes several loops, huge amount of money and couple of years. On the other hand, though the convergence of this iterative process is not mathematically guaranteed, engineers and designers are typically satisfied after 2-3 loops.

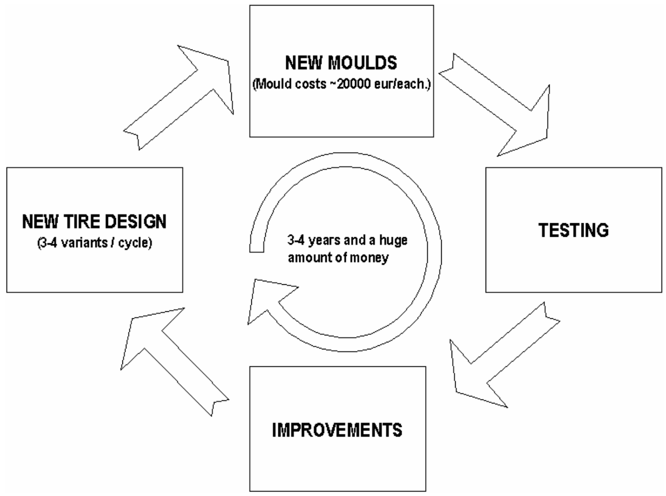

Figure 1. Traditional tire R&D-process

In future, however, we see that virtual environments will become more and more important for designers and some physical testing will be performed with the modern simulation tools instead of traditional methods. We also see that the need for physical testing will not decrease; with simulation methods we will just be able to provide more information for designers, make better trials for physical testing and therefore provide safer tires with improved performance for consumers needs.

Although these modern tools seem to be very attractive, some remarks have to be pointed out. First of all, tire is relatively challenging application for FEM-simulations. Complex 3D-geometry of tire tread pattern combined with nonlinear materials, orthotropic structure and nonlinear contact boundary conditions, for example, leads to extremely difficult analysis and usually analyst ends up with a nonlinear analysis with huge amount of degrees of freedom. In this kind of situation a robust solver with ability to handle such complex models is definitely appreciated. A cross section of a studded winter tire is presented in the Figure 2.

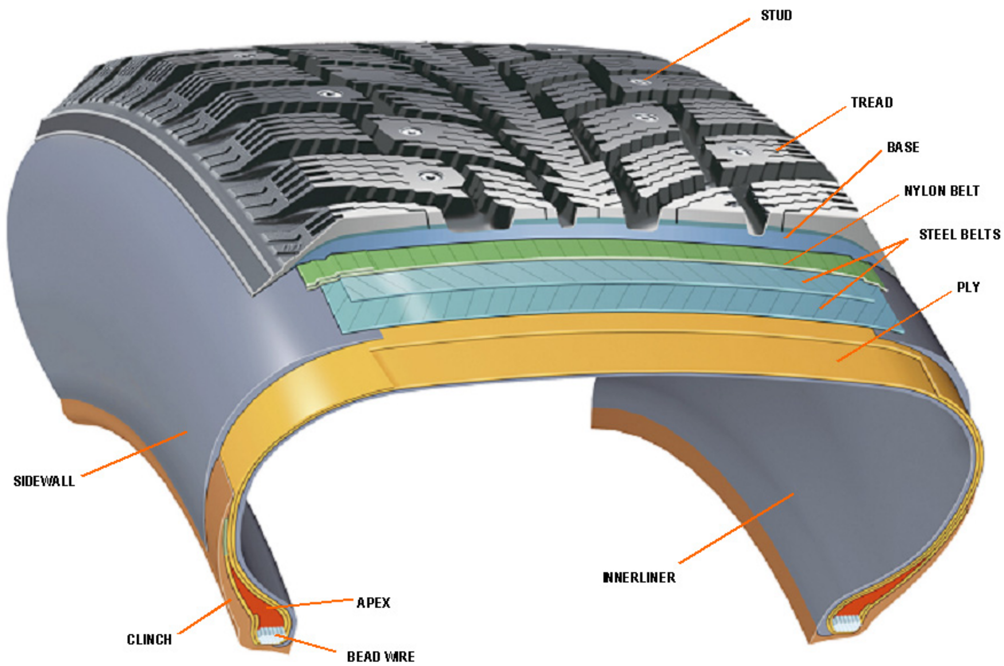

Figure 2. Studded winter tire and its components.

Secondly, during last ten years the performance of computer hardware has been increased very rapidly. At the same time a lot of work has been done with robustness and capabilities of commercial finite element codes. These two things have together made possible for today's engineers to analyze and handle huge models with very nonlinear characteristics in reasonable time. Despite the rapid improvement in the computational performance some approximations are still needed and therefore an analyst who has good engineering skills in both physics and mathematics are important for successful simulations.

Figure 3 represents a discretized FE-model of passenger car tire and rim. Rim is modeled by using rigid elements.

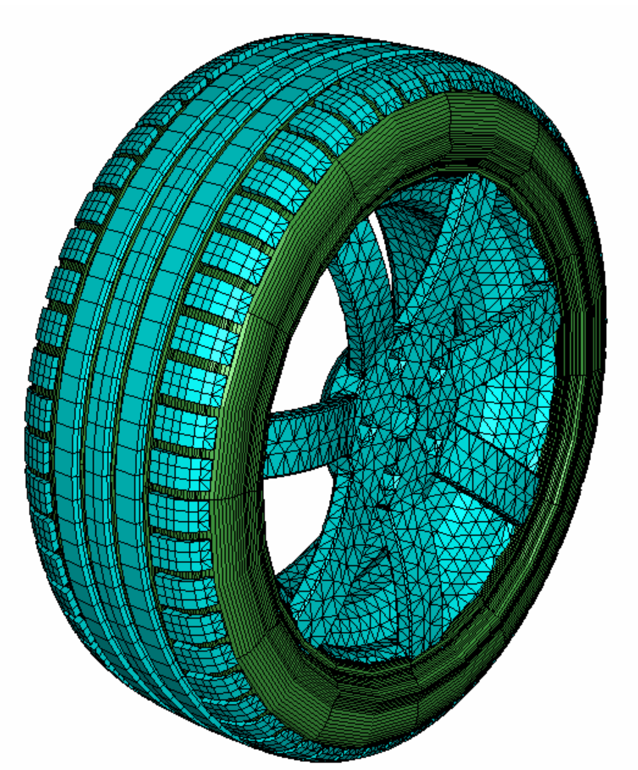

Figure 3. FE-model of passenger car tire

ABAQUS provides many nice features for tire modeling. For example, one may use embedded element technique in order to reduce the analysis time, but still obtain accurate results enough e.g. for tire lateral stiffness simulations. Other useful feature is steady state transport-capability, which enables some steady-state-handling simulations. Third, but not least, is the expandability and flexibility of ABAQUS code: with user subroutines one can define almost arbitrary user materials, contacts etc.

Nokian Tyres started using ABAQUS software in the beginning of 2003. No commercial or in-house codes were used before, with the exception of simulations made by consultants. During these last two years ABAQUS/CAE has become more and more user friendly, so that with 6.5 it is possible to create a complete tire FE-model inside ABAQUS/CAE. These improvements have mainly made possible quite efficient usage of ABAQUS software and implementation for daily R&D purposes. As the examples further shows, we have also been able to see some great advantages of the investment.

2. CASE STUDY: Stress reduction in the belt edge area

Fatigue and durability properties of tires are extremely important and form a fundamental basis for a sustainable safety. Due to complex stress field, cyclic loading and heat generation near the belt edge areas a designer sometimes sees fatigue damage called belt edge separation. This phenomenon is especially important in applications like commercial vehicle tires. It is very common that the vehicles using these tires are fully loaded and usually driven at maximum speed so tires under these cars are susceptible for belt edge separation.

Usually tire engineers evaluate the fatigue properties of a specific tire on test drum, under certain loading, speed and temperature conditions. The problem is, that such test is very time consuming, expensive and requires physical trials. In practice designers avoid belt separation by reducing the belt width, for example. Unfortunately this kind of modification reduces also the handling properties of tire and is therefore not recommended.

With ABAQUS-software one can study the stress field around the belt edge area. A tire of size 225/70R15 C was chosen as an example and axisymmetric FE-model was first created for rim mounting and tire inflation purposes. The axisymmetric model is shown in Figure 4. In order to obtain accurate results the mesh has to be quite dense around the area of interest, which in this case is the belt edge area. Rim is modeled as a analytical rigid surface.

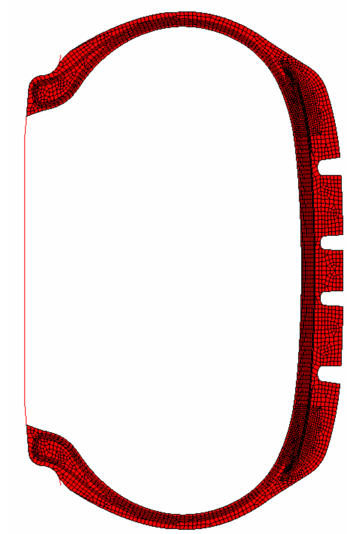

Figure 4. Axisymmetric FE-model of 225/70R15 C after inflation

Though embedded element technique is often very useful in tire analysis it cannot be used here, because with embedded element technique it is not possible to obtain accurate stress field. Membrane elements are, however, used to carry rebars but these membrane elements share nodes with elements of matrix material.

Next step, after inflation, was to create a 3D FE-model for footprint analysis. This can be done in ABAQUS by using *SYMMETRIC MODEL GENERATION option. It is a very efficient and user friendly tool because e.g. all sections, material assignments and surfaces are remained. The 3D-model is shown in figure 5. The footprint is shown in Figure 6.

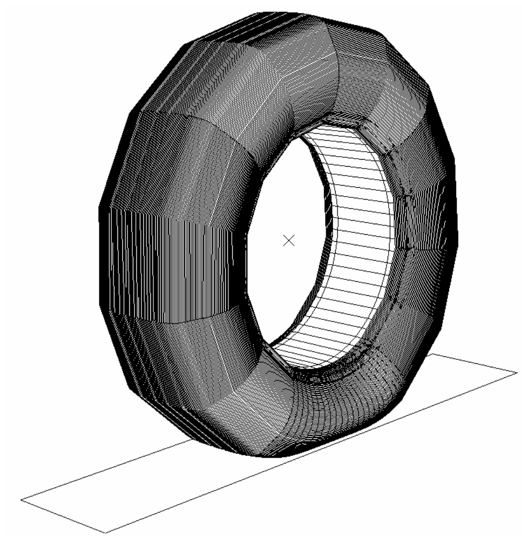

Figure 5. Deformed FE-model of 225/70R15 C after footprint analysis

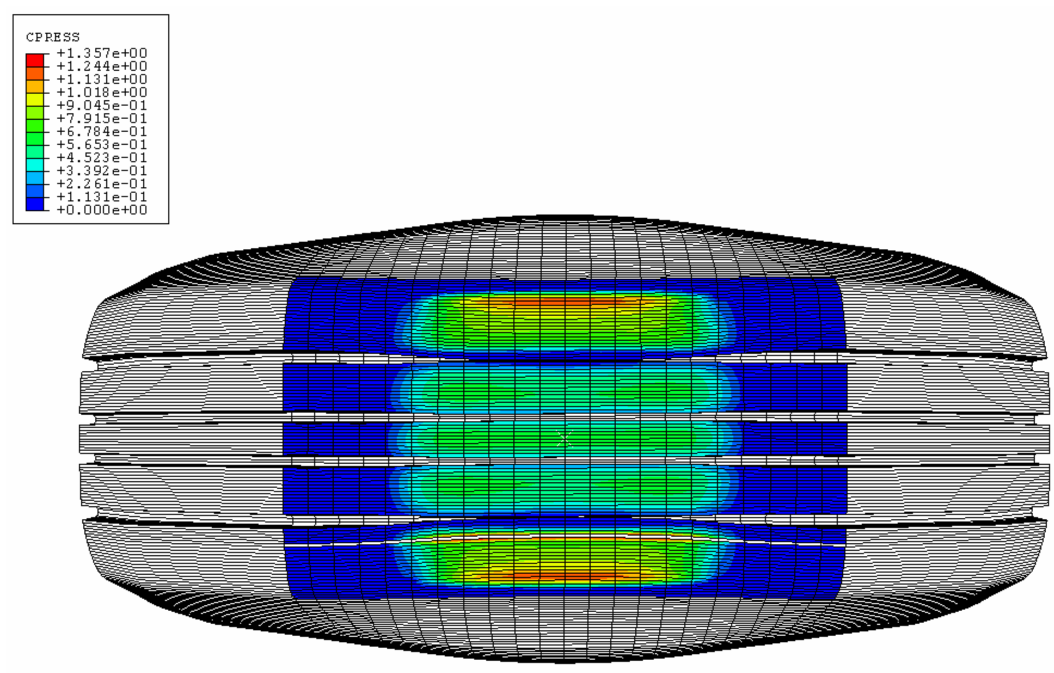

Figure 6. Footprint of 225/70R15 C

Last step is to perform a rolling simulation with camber. In ABAQUS this can be done by using *STEADY STATE TRANSPORT-option. Though the temperature and heat generation are very important factors in fatigue analysis, no coupled displacement-temperature analysis was used. The object of this research was just to compare two different constructions and to study how the risk of belt separation could be reduced.

In the figure 7 one can see the deformation in steady-state rolling situation with camber. The analysis itself is quite simple, however, the viscoelastic material properties have a negative influence on convergence if compared to pure hyperelastic material model.

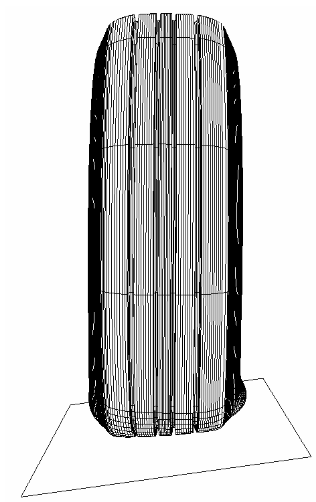

Figure 7. Steady-state rolling with camber

The main results of the research can be seen in figure 8, which represents the shear stresses in rubber around belt edge area. The conclusion is that the belt separation is caused mainly due to shear stress component. This shear stress is mainly caused by the deformation of the steel cords. When these cords are deformed in the footprint area, a shear stress is generated because of the angle in steel cords. Cyclic loading arises when tire is rotating.

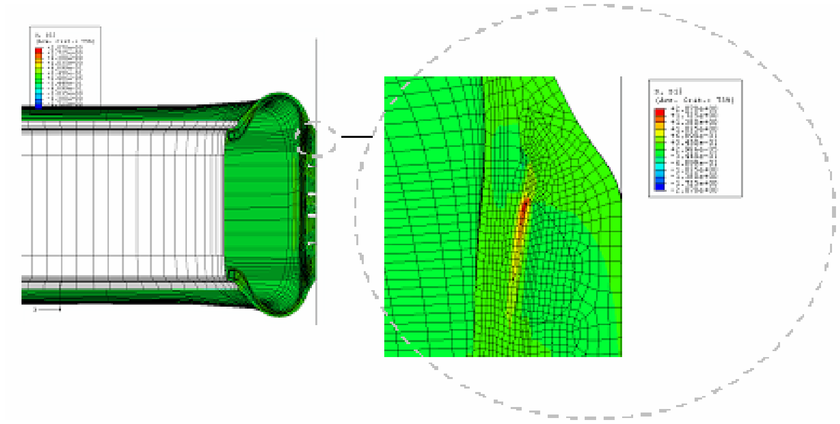

Figure 8. Critical shear stress component in tire

After the first steady state rolling simulation some modifications were made and the analysis procedure was repeated. Figure 9 represents the comparison between old and improved design. It can be easily seen that the critical shear stress is reduced in the improved design. When the trials were made for physical testing the same result was obtained: the new improved design gave longer lifetime in the view of belt separation.

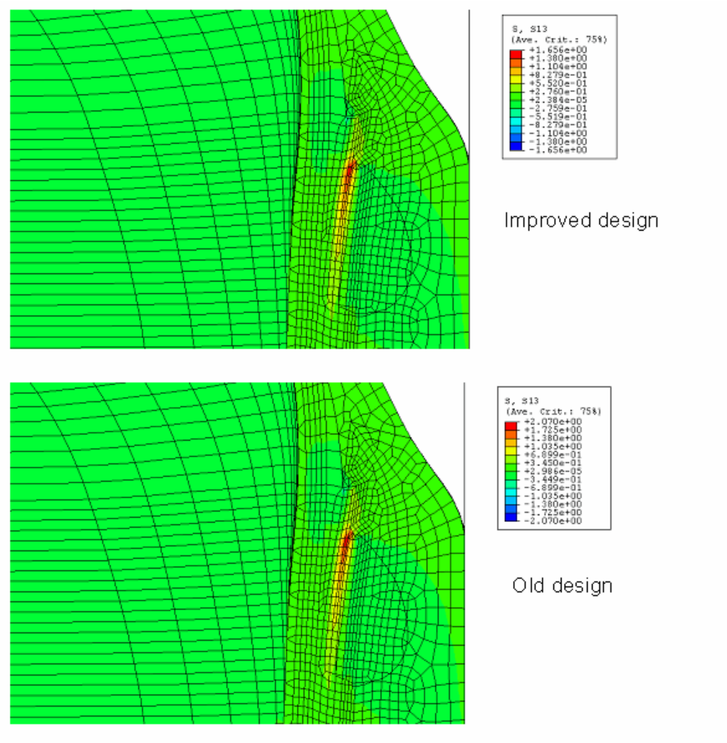

Figure 9. Critical shear stress comparison

3. Summary

ABAQUS seems to be a powerful tool to simulate such challenging applications as tire. ABAQUS solvers can handle nonlinear materials and complicated contacts, for example, in a very robust way. Nokian Tyres started using ABAQUS in the beginning of 2003 and already now we can see some great advantages of using such modern simulation tools as ABAQUS. Our case is an example how ABAQUS can be rapidly implemented into company's R&D-environment.