ALLOY WHEEL OPTIMIZATION TO AVOID CRACKING

R. Helfrich, J. Müller

(INTES GmbH, Germany, e-mail Reinhard.Helfrich@intes.de)

Abstract

For the homologation of alloy wheels, a number of standard tests have to be performed before such wheels are allowed to be used in cars. Such tests are bending fatigue test, radial fatigue test, and impact test. The latter test represents the inclined collision of a wheel and a curb at low speed, where the spokes are not allowed to crack. To pass this test at the first time successfully, previous numerical simulation and optimization can avoid late re-design and late start of production.

The impact test is characterized by a high mass falling down from a low height on an inclined wheel with tyre. These conditions describe a low speed impact on the rim, which can be handled as quasi-static analysis case. An elastic-plastic contact analysis is easily applied to compute effective plastic strains. The point of cracking is defined, when the effective plastic strain exceeds the ultimate plastic strain of the material used.

When cracking occurs, a shape optimization can be applied to reduce the plastic strain at the critical points. In addition, a special condition has to be fulfilled, which does not allow to make shape changes of the front view, because this is exclusively defined by the stylist. This is a very special manufacturing constraint for the optimization.

The basic elastic-plastic analysis is typically performed in one single simulation using Finite Element Analysis (FEA) software. In order to achieve a single simulation approach including shape optimization, a software is used, which contains not only the analysis features needed but also provides shape optimization capabilities. The effects of the single simulation approach are easier handling of analysis and optimization by the analyst and a shorter computation time for the optimization process. In addition, this process needs an extended pre-processing step, which supports the definition of the optimization model based on the Finite Element (FE) model.

As an industrial example, the paper uses an alloy wheel design from a former research project. Analysis, optimization, and result evaluation are performed with an industrial FEA code (PERMAS with VisPER).

1.   Introduction

The mechanical behaviour of wheels during the lifetime of a vehicle is of highest importance due to safety reasons. Because it has to transmit all loadings between road and tyre on the one side and axle and chassis on the other side, a wheel is a safety component, which has to meet strong requirements with regard to strength, weight, and quality [Dabbs et al.].

Each wheel type has to be tested following some standardized tests to get the type approval: rim rolling fatigue test, bend fatigue test, and impact test. These tests are based on legal rules, which intend to assure a high safety level. Because such tests have to be performed for each wheel type, the manufacturer's clear interest is to avoid failures during the final approval tests, i.e. to pass the tests with a wheel the first time. The goal is to reduce development costs during the design phase of new wheel types by reduction of the number of prototypes, the tests performed, and the time needed for design.

The project reported in [Dabbs et al.] could demonstrate that there are already appropriate structural simulation methods for all three tests, which allow for the realization of the cost reduction goals. However, at the time of the report in 1998, optimization methods were not available to achieve the project goals more directly. This is the motivation to review the former analysis approach and to apply shape optimization to demonstrate the progresses in simulation technologies during the past twenty years.
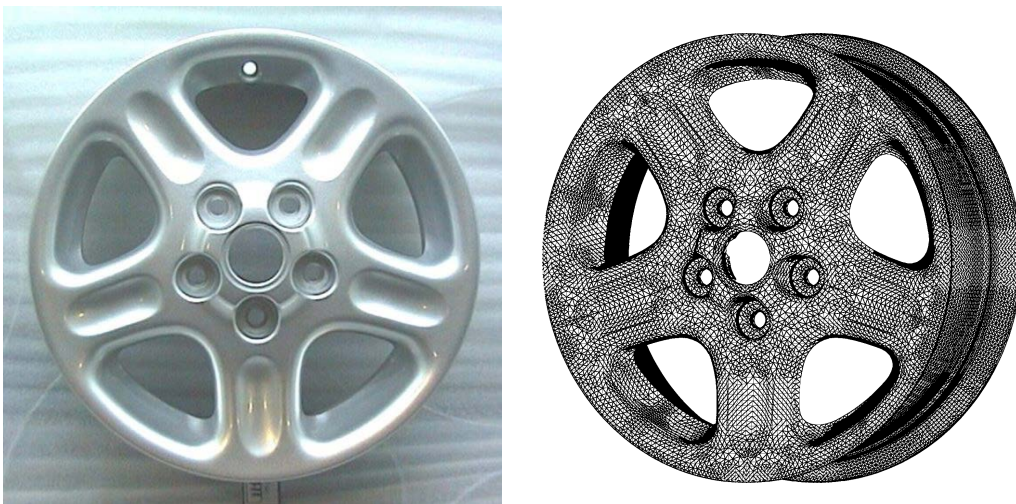

Figure 1:   The wheel and its mesh.

2.   Historical Reference

The selected wheel and its mesh are shown in Fig. 1. It is a classical 5-spoke wheel from Aluminium alloy. Two different geometries ("old" and "new") were given with the additional information that the wheel with the "old" geometry failed in the wheel impact test and the wheel with the "new" geometry passed the impact test successfully. The difference in geometry is only for the back of the wheel, while the front was not touched (see Fig. 2).
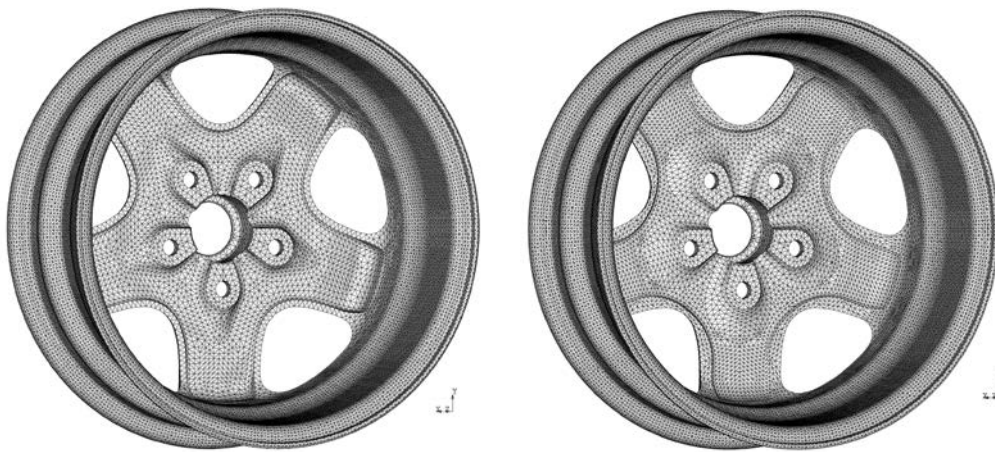

Figure 2:   "Old" (left) and "new" (right) geometry of the wheel.

During impact test, a high mass falls onto the rim flange and deforms the wheel not only with elastic deformations but also with plastic ones, which causes a real damage of the wheel (see Fig. 3). To pass the impact test, there should be no through-thickness crack of a spoke.

There are two configurations for the load impact point, one at the position of a spoke and the other at the midpoint between two spokes. For the present wheel, the first configuration is the more critical one.

By definition, each impact is of dynamic nature due to the short-time excitation applied to the structure. In case of the impact test, the hypothesis was used that a static analysis is sufficient. The static analysis has to take into account the plasticity of the material but any time-dependency is neglected. An investigation showed that a nonlinear static analysis detects the right location of structural failure and gives qualitatively and quantitatively correct results, i.e. when comparing two geometries as in our case such variant shows better impact behaviour which provides better results after such a nonlinear static analysis. Fig. 4 shows high plastic strain at the next spokes of the impact for the "old" geometry while the "new" geometry show much lower plastic strain.
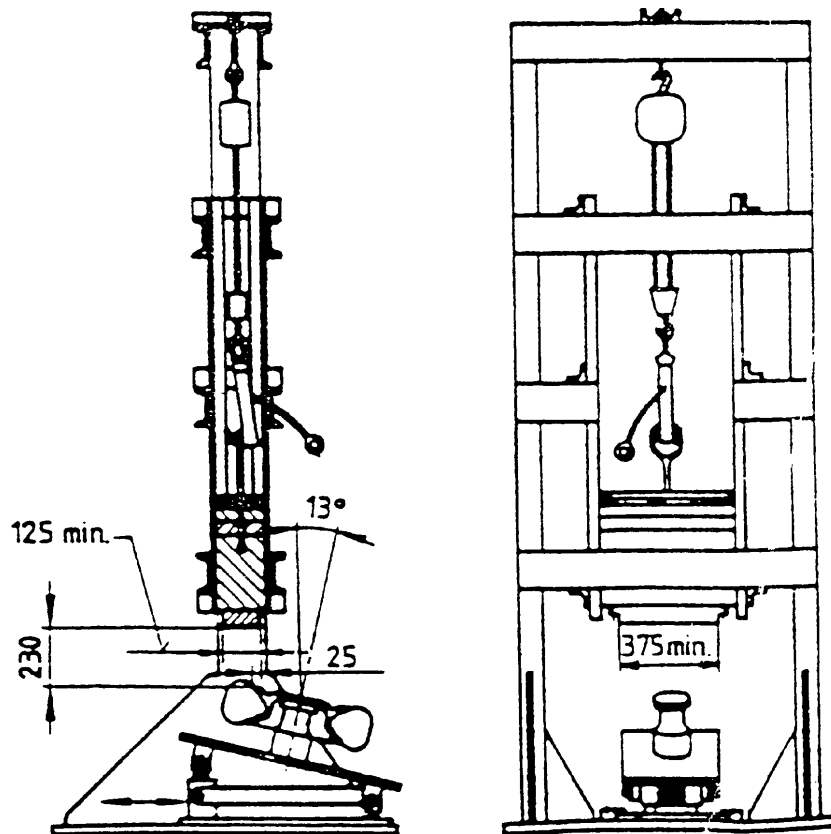
Figure 3:   Test configuration impact test.

At the time of the project, there was already the idea to use shape optimization to reduce plastic strains. However, optimization methods and performance of nonlinear analysis were not prepared for this model. Years later, when both points got essential improvement, the example could be used again for a more advanced analysis and optimization.
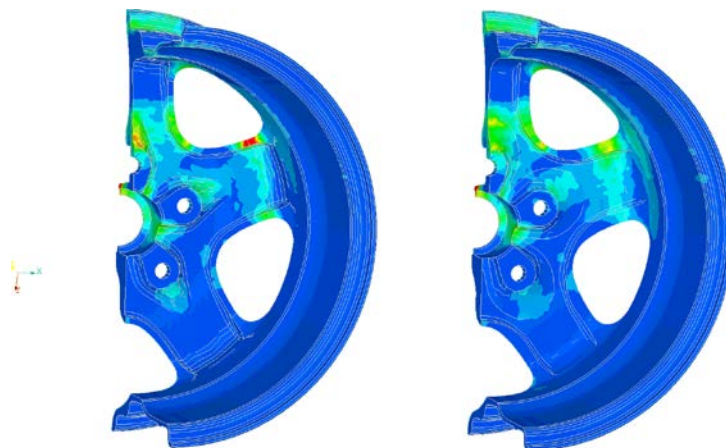

Figure 4:   Plastic strain from inner side for "old" (left) and "new" (right) geometry.

3.   Advanced Structural Model

The resumption of the wheel analysis after many years uses a refined model to achieve more reliable results:
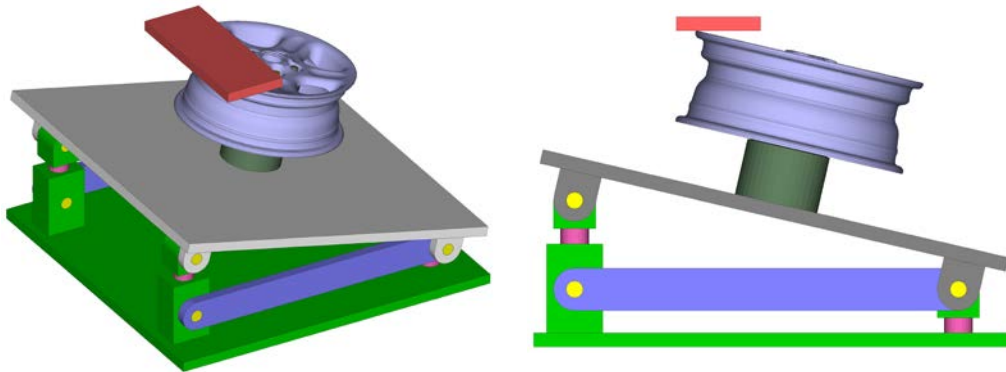
- Modelling of the test platform (see Fig. 5)

Figure 5:   Test set-up for simulation

- Finer meshes (see Fig. 6 and Fig. 7): The mesh characteristics are shown in Table 1.

Figure 6:   New wheel mesh

Figure 7:   New wheel mesh for "old" (left) and "new" (right) geometry

Characteristic	"Old" geometry	"New" geometry
Nodes	538,685	537,650
Elements	357,713	357,403
Element type	TET10, HEXE8, PENTA6	TET10, HEXE8, PENTA6
MPCs	700	700
DOF	1,602,586	1,599,481
Contact DOF	2,748	2,748

Table 1:   Model characteristics.

- Bolt pretension (a pretension force of 60 kN is applied to each of the five bolts)

- Contact analysis (contact is applied between dropped weight and rim, and between wheel and wheel hub due to bolt pretension)

- Updated material: For the wheel, an Aluminium casting alloy with a modulus of elasticity of 72.4 GPa and a Poissons ratio of 0.33 is used. The nonlinear stress-strain relation is shown in Fig. 8. The mounting structure is mainly from steel.
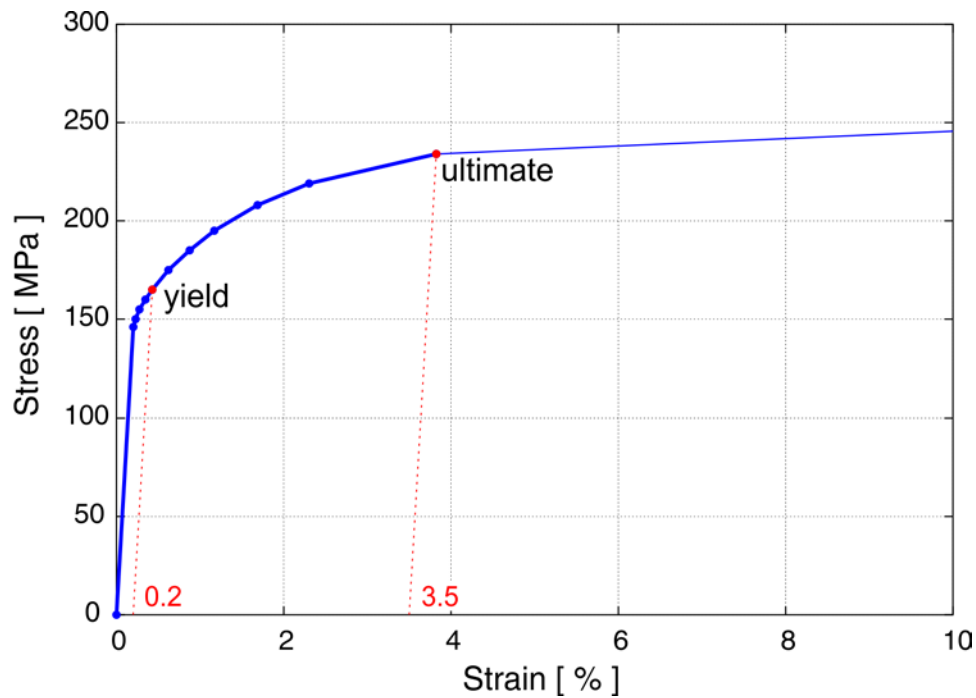
Figure 8:   Stress-strain relation of wheel material

- Calculation of strain energy sum

  The loading is defined by the potential energy of the mass dropped on the rim:

  o Dropped mass: 516 kg

  o Height: 230 mm

  o Energy: 1,164 J

    The energy absorbed by the tyre is estimated as 150 J.

    Taking into account the additional potential energy related to the displacement of the hit point on the rim (about 26 mm), we have a total potential energy of about 1,170 J.

  The analysis has to stop, when the current strain energy with elastic and plastic part has reached the potential energy value.

The main results of the analysis are the effective plastic strains in the wheel. Comparison of these strains between "old" and "new" geometry is shown in Fig. 8. The crack in the "old" geometry model occurs at the spokes next to the spoke where the impact is applied. The strain value of about 12% is far beyond the ultimate strain of the material. For the "new" geometry model, the strain at

the same place is only 4%, which is in about the range of ultimate strain of the material. So, we understand that the "old" geometry wheel did not pass the impact test while the "new" geometry wheel could pass the impact test.
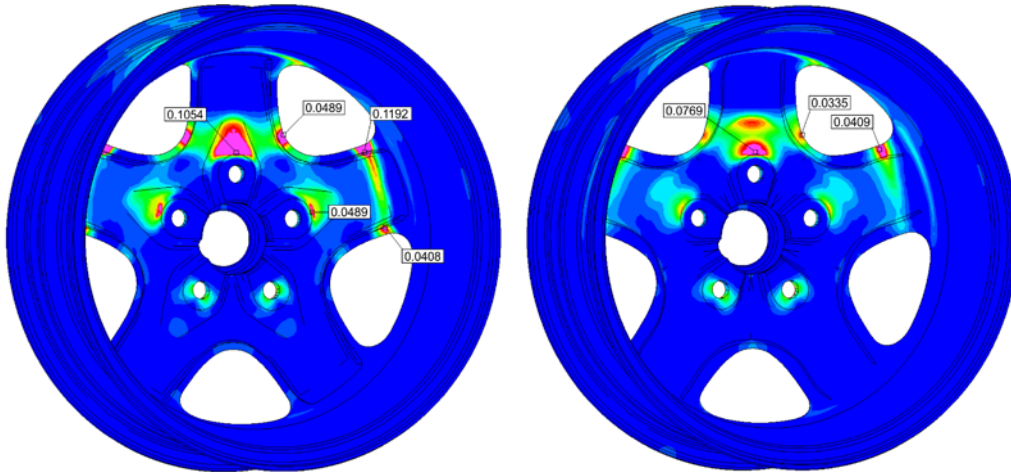

Figure 9:   Effective plastic strains for "old" (left) and "new" (right) geometry

While the total energy keeps the same value for both wheel variants, the distribution of energy on wheel and test mount is different due to different wheel stiffness (see Table 2). The "old" geometry wheel sees almost 6% more of the applied energy than the "new" geometry wheel. This is due to the increased stiffness of the "new" geometry.

Part	"Old" geometry	"New" geometry
Wheel	63.2%	57.4%
Test mount	36.8%	42.6%

Table 2:   Model characteristics.

4.   Shape Optimization

If a structure like the "old" geometry case shows high plastic strains beyond the ultimate strain, where a crack is likely to occur, then a shape optimization provides a direct means to find the right shape changes. Here, a parametric shape optimization is used, because the parameters of shape changes (the so-called Shape Basis Vectors, SBV) can be directly related to certain surface parts of the structure. For the wheel, one special condition is that the shape of the front side of the wheel must not be modified. All modifications have to be made on the back of the wheel as it was done for the "new" geometry of the

wheel. Fig. 10 shows the 5 shape basis vectors used in the optimization, where the shape change is described by applied forces to specified surface parts.

Figure 10:   Original shape and 5 shape basis vectors for the shape modification of the "old" wheel geometry.

The weight of the wheel is used as objective function, while the effective plastic strain is set to 3.5% as maximum value as constraint, which is the appropriate material property for the ultimate strain. The final value should be below the ultimate strain in order to avoid cracks.
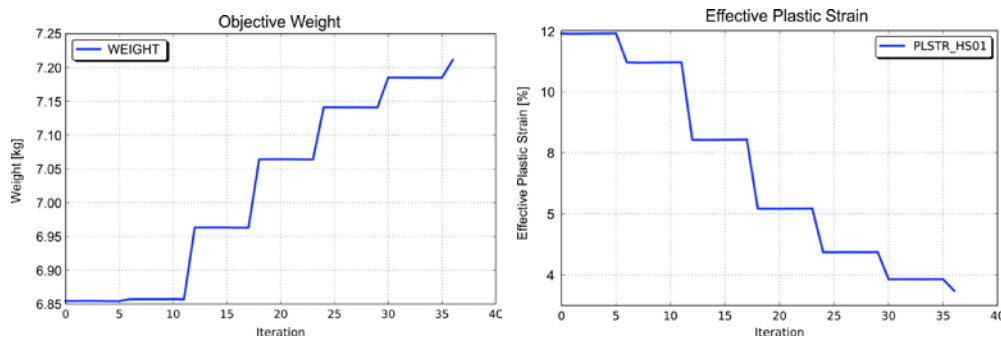

Figure 11:   Objective and constraint history during optimization

The optimization used 36 solution steps (see Fig. 11). The horizontal parts of the curve indicate that the derivatives are determined by differences and not by (analytical) sensitivities. Because nonlinear material behaviour does not allow for sensitivities, the derivatives are computed by differential quotients. For each design variable (i.e. shape basis vector) one solution step is required, i.e. 6 solution steps per iteration. Altogether, 6 iterations are needed to obtain convergence.

The weight is increased by 5.2% (i.e. 356g). The final shape of the optimized wheel is shown in Fig. 12.
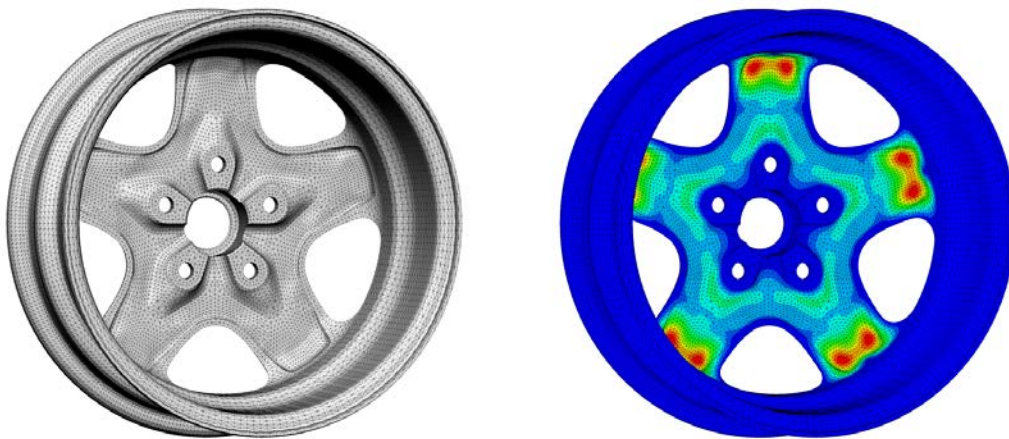

Figure 12:   Final shape of the wheel (left) with the coordinate change of the design surface (right)

The effective plastic strain in the design space came down from 11.9% to 3.4%, which is just below the given limit. Fig. 13 compares the effective plastic strain of the initial "old" geometry with the optimized shape from Fig. 12.
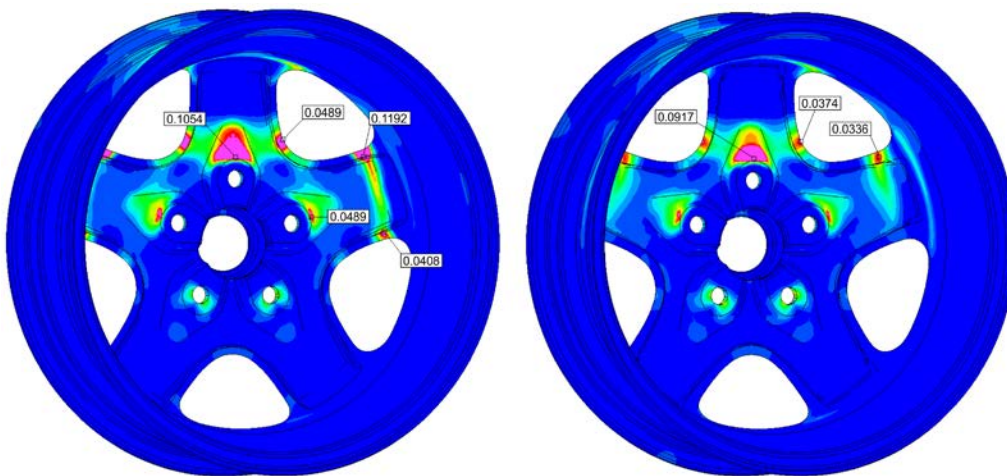

Figure 13:   Effective plastic strains for "old" (left) and optimized geometry (right)

5.   Conclusion

Shape optimization with nonlinear static analysis including contact is available and effective plastic strains can be used as objective or constraint. A parametric shape optimization has been used to reflect a parametric geometry modification without changing the front side of the wheel.

The optimization and the nonlinear static analysis are integrated in one software providing a single simulation approach without any software couplings.

6.   References

Dabbs N., Mortier R., AWI; Schäffer U., Helfrich R., INTES (1998). *ESPRIT Project CAROW: Public Final Project Report on Structural Simulation of Wheel Tests*.