FAKULTA
STROJNÍ
ČVUT V PRAZE

Ústav automobilů, spalovacích motorů
a kolejových vozidel

Výpočtový model zatížení automobilového ráfku

Computer model of load of automotive rim

DIPLOMOVÁ PRÁCE
2020

Andrei GASHNIKOV

Studijní program: N2301 STROJNÍ INŽENÝRSTVÍ

Studijní obor: 2301T047 Dopravní letadlová a transportní technika

Vedoucí práce: Ing. Jan Baněček, Ph.D

ZADÁNÍ DIPLOMOVÉ PRÁCE

I. OSOBNÍ A STUDIJNÍ ÚDAJE

Příjmení: **Gashnikov** Jméno: **Andrei** Osobní číslo: **453276**

Fakulta/ústav: **Fakulta strojní**

Zadávající katedra/ústav: **Ústav automobilů, spalovacích motorů a kolejových vozidel**

Studijní program: **Strojní inženýrství**

Studijní obor: **Dopravní, letadlová a transportní technika**

II. ÚDAJE K DIPLOMOVÉ PRÁCI

Název diplomové práce:

Výpočtový model zatížení automobilového ráfku

Název diplomové práce anglicky:

Computer model of load of automotive rim

Pokyny pro vypracování:

Práce má charakter výpočtářský, předpokládá vytvoření odladění a využití modelu MKP. Postupné kroky

1. Rešerše současného stavu modelování pneumatiky pomocí MKP (včetně modelů kompozitní pryžové konstrukce).
2. Návrh a realizace MKP modelu ráfku a pneumatiky, sloužícího ke stanovení silových účinků na ráfek.
3. Ověření funkčnosti modelu pro zjištění zatížení ráfku pro základní případy zatížení kola.
4. Diskuse možností parametrizace vzniklého modelu a návrh dalších vývojových směrů modelu.

Seznam doporučené literatury:

Jméno a pracoviště vedoucí(ho) diplomové práce:

Ing. Jan Baněček, Ph.D., ústav automobilů, spalovacích motorů a kolejových vozidel FS

Jméno a pracoviště druhé(ho) vedoucí(ho) nebo konzultanta(ky) diplomové práce:

Datum zadání diplomové práce: **27.04.2020** Termín odevzdání diplomové práce: **15.07.2020**

Platnost zadání diplomové práce:

Ing. Jan Baněček, Ph.D. podpis vedoucí(ho) práce	doc. Ing. Oldřich Vítek, Ph.D. podpis vedoucí(ho) ústavu/katedry	prof. Ing. Michael Valášek, DrSc. podpis děkana(ky)

III. PŘEVZETÍ ZADÁNÍ

Diplomant bere na vědomí, že je povinen vypracovat diplomovou práci samostatně, bez cizí pomoci, s výjimkou poskytnutých konzultací. Seznam použité literatury, jiných pramenů a jmen konzultantů je třeba uvést v diplomové práci.

Datum převzetí zadání	Podpis studenta

Anotační záznam

Jméno autora: Andrei Gashnikov

Název práce: Výpočtový model zatížení automobilového ráfku

Anglický název: Computer model of load of automotive rim

Rozsah práce:

46 stran

25 obrázků

13 tabulek

Akademický rok: 2019/2020

Ústav: 12 120 Ústav automobilů, spalovacích motorů a kolejových vozidel

Studijní program: N2301 STROJNÍ INŽENÝRSTVÍ

Vedoucí práce: Ing. Jan Baněček, Ph.D

Klíčová slova: MKP analýza, Abaqus, CATIA, kolo, automobilový ráfek, pneumatika, tuhost pneumatiky, kompozitní materiály, orientace kompozitních materiálu, hyperelastické materiály, Mooney-Rivlin teorie, síťování, materiálové vlastnosti v Abaqusu, parametrizace modelu.

Key words: FEM analysis, Abaqus, CATIA, wheel, automotive rim, tire, vertical stiffness of tire, composite materials, orientation of composite materials, hyperelastic materials, Mooney-Rivlin theory, creating of mesh, material properties in Abaqus, parametrization of model.

Abstrakt

Tato diplomová práce popisuje vytvoření výpočtového modelu zatížení automobilového ráfku v softwaru Abaqus. Práce obsahuje postup CAD modelování automobilového ráfku a pneumatiky, popisuje způsoby stanovení hyperelastických a kompozitních materiálů, nastavění kroku a sítě v Abaqusu.

Abstract

This diploma thesis describes the creation of an automotive rim’s load model in the software ABAQUS. This diploma work contains the method of creating CAD models for rims and tires, the determination of a wheel’s loading, techniques of specifying elastic, hyperelastic and composite material properties, the settings of steps and creation of mesh in ABAQUS.

As a result of this diploma thesis, we determinate the effect of the loads on the automotive rim, found zones of maximum tension and evaluated the results.

Čestné prohlášení

Prohlašuji, že jsem diplomovou práci na téma: “ Výpočtový model zatížení automobilového ráfku ” vypracoval samostatně s použitím odborné literatury a pramenů, uvedených v seznamu zdrojů, jenž tvoří poslední kapitolu této práce.

V Praze dne: 10.7.2020

……………………………………….........

Andrei Gashnikov

Acknowledgment

I would like to take this opportunity to sincerely thank my thesis advisor Ing. Jan Baněček, Ph.D for suggesting the theme of this diploma work as well as his wise advising, support and devotion during the entire time.

I also wish to thank my consultant Ing. Josef Kazda for his valuable advices, great amount of assistance and teaching me about modeling in ABAQUS.

Contents

8 Future development of the model 41

1 Introduction

The construction of car's wheel is constantly modifying in order to reduce the weight of vehicle, unsprung and rotational mass, gyroscopic moment and moment of inertia about the vertical axe of a car. Reducing unsprung mass improves a car's suspension work. Reducing rotating mass can make the car accelerate and stop faster as well.

Czech Technical University has a great deal of experience with using composite materials in the lightweight wheels of Formula Student bolides. The diploma thesis of Balejík Gorazd "Development composite wheel rim for Formula Student Car" [1] analyzes strengths and weaknesses of the prototype of composite wheels. In FEM analysis of this thesis wheel is fixed at the bead seats and flanges while loaded by virtual hub. Despite the fact, that this model is accurate enough, in reality it is a little bit different. In real life, the wheel is fixed on a hub and loaded through a tire.

In our diploma work "Computer model of load of automotive rim" we developed a model of the rim's loading, which has the same boundary conditions and load as in reality. It means, that the wheel is fixed on a hub and loaded through the tire. Our model uses a detailed model of the tire in order to study contact pressure at the bead seats and flanges of rim. Our method explains how to calculate the stress and strain of rim precisely.

2 Research of current state of tire's FEM modeling

For creating an accurate FEM model of a real tire, it is important to know the exact geometry of the tire (the dimensions of the tread and bead chafer, thickness of layers, etc.). Beside this, we need to identify the material properties of each segment of tire. Also, it is important to know the characteristics of the tire such as the vertical stiffness on a flat surface (load/deflection curve) and the dimension of the footprint of the tire.

Unfortunately, most of these characteristics are often confidential and cannot be

received from tire companies. However, there is a lot of research which has tested the material properties of real tires, got an accurate cross-section of the tire and measured the dimensions of the layers.

The diploma thesis of Ondřej Lavický "Computational modelling of stress-strain states in tyres" [2] is concerned with the computational modelling of the mechanical behavior of elastomers and composites with the rubber matrix of the specific tire MATADOR 165/65 R13. From this study, we learned the material properties of the tire's segment. Unfortunately, this study doesn't contain the geometry of the tire and its characteristics. So, we decided to take the geometry and characteristics from another tire instead.

The geometrical properties of the tire we got from "Tyre Model Performance Test" [3] which provides the characteristics of the specific tire Continental 205/55 R16 90. From this study, we took cross-section of the tire, dimensions of the tire's segments, friction coefficient of the tread, load index, value of vertical stiffness on a flat surface and dimensions of the footprint under different loads. Unfortunately, the material property of this tire is unpublished, so we decided to use the material property from the specific tire MATADOR 165/65 R13.

Of course, using different characteristics of different tires makes our model inaccurate but the aim of this diploma thesis is to describe the technique of FEM modeling of an automotive rim's load, not to obtain the material property of the exact tire. Furthermore, the work "Finite Element Modeling of Tire Transient Characteristics in Dynamic Maneuvers" [4] used the same method of tire modeling as we used.

The methods of FEM modeling of a tire are clearly explained in "ABAQUS 6.14 EXAMPLE PROBLEMS GUIDE. VOLUME II: OTHER APPLICATIONS AND ANALYSES" [5]. We used these methods in this diploma thesis.

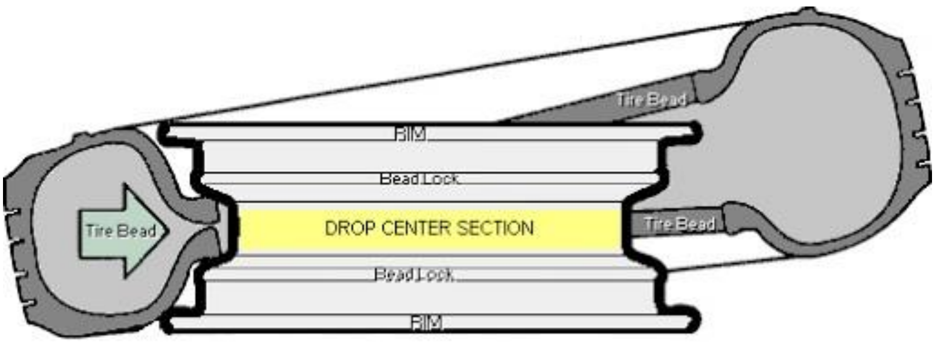
3 Construction of automotive rim

The rim is a cylindrical wheel with its outer edge holding the tire on the wheel. The main function of the rim is to support and seal the tire to the wheel. The rim ensures the proper fitting between the tire and rim while retaining the air inside the tubeless tire. The rim consists of several parts [6]:

Bead seats - the flat areas just inside of the flanges where the edges of the tire (the beads) sit on the wheel.

Flanges - the flared edges of the barrel on both the inboard and outboard sides of the wheel. The metal of the barrel is flared 90 degrees outward on each side. This prevents the tire from slipping off the wheel.

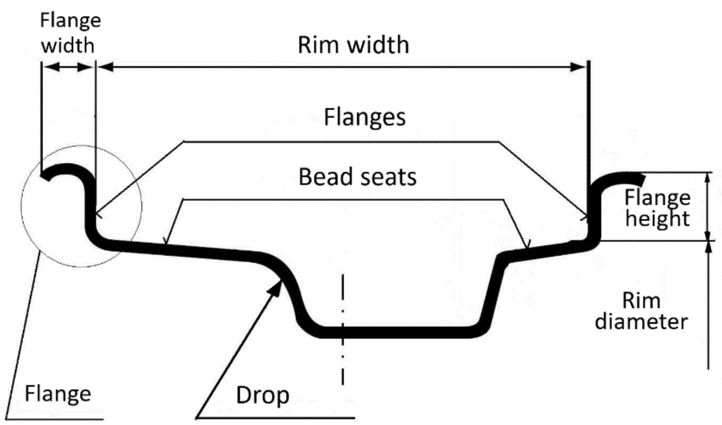
Drop – The center section of the rim, which is lower than the two outer edges. This allows the bead of the tire to be pushed into the low area on one side while the other side is pulled over and off the flange. The drop between the bead seats makes tire mounting and demounting easier (Picture 1).



Picture 1

The drop makes tire mounting and demounting easier

Safety Hump - enables better tire fitting on the rim, prevents the bead from slipping down when vehicle moving at a high speed and stops air from escaping when cornering at reduced tire pressure.



Picture 3

Construction of automotive rim

3.1 Determination of a rim's type

In the FEM model of this diploma thesis, we use specific tire Continental 205/55 R16 90. We determined the type of rim appropriated for this tire by using the tire and rim compatibility table (table 1) [7].

Table 1 tire and rim compatibility table

Rim width (inch)	Min. tire width (mm)	Ideal tire width (mm)	Max. tire width (mm)
6.0	175	185 - 195	205
6.5	185	195 - 205	215
7.0	195	205 - 215	225
7.5	205	215 - 225	235

We chose an asymmetric rim 6,5 J x 16 H1 ET 40, where:

6,5 – Rim's width in inches

J - a tire bead profile (J – for passenger cars)

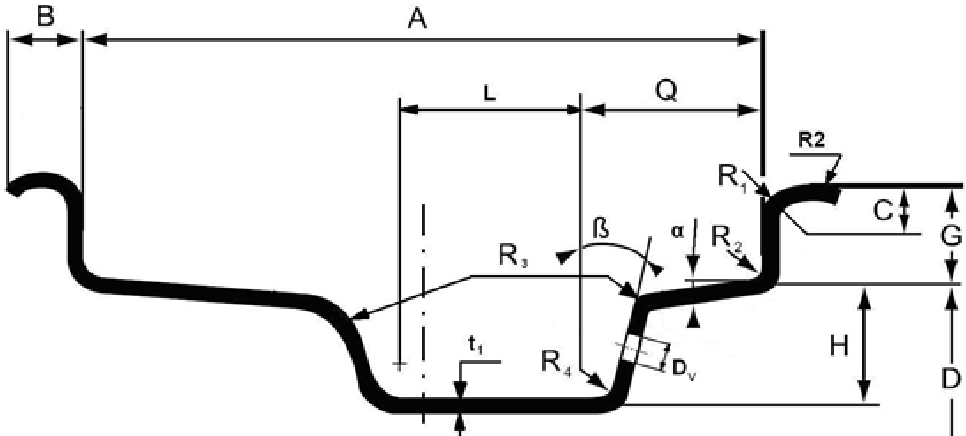
16 - Diameter of a rim in inches

H1- type of hump

ET 40 - is the offset, the distance of the hub mounting surface to the wheel's symmetry axle expressed in millimeters.

3.2 Dimensions of automotive rim

The dimension of automotive rim is specified by standard ČSN 30 3707 "Wheels for tyres. Terms and definitions. Rim types and rim codes" [8]. The Dimensions of the rim 6,5 J x 16 H1 ET 40:



Picture 3

Dimension of asymmetric rim, specified by standard ČSN 30 3707

A – Wheel width – 6.5 inches or 165.1 mm

D – Diameter of the rim – 16 inches or 406.4 mm

α – Angle of bevel of a bead seat - 5° (for passenger cars)

G – Hight of the flange – 17.3 mm (for a type J flange)

R1 – 9,7 mm (for a type J flange)

R2 – 6,4 mm (for a type J flange)

t1 – thickness of a metal sheet – 3mm (usual 3-4 mm)

ß – Angel of drop - ß=D [°] = 16°

B – B = 1.5*R1 = 15.55 mm

Q – Q = A* 0.25 = 41.3 mm

L – L = 0.22*A = 36.3 mm

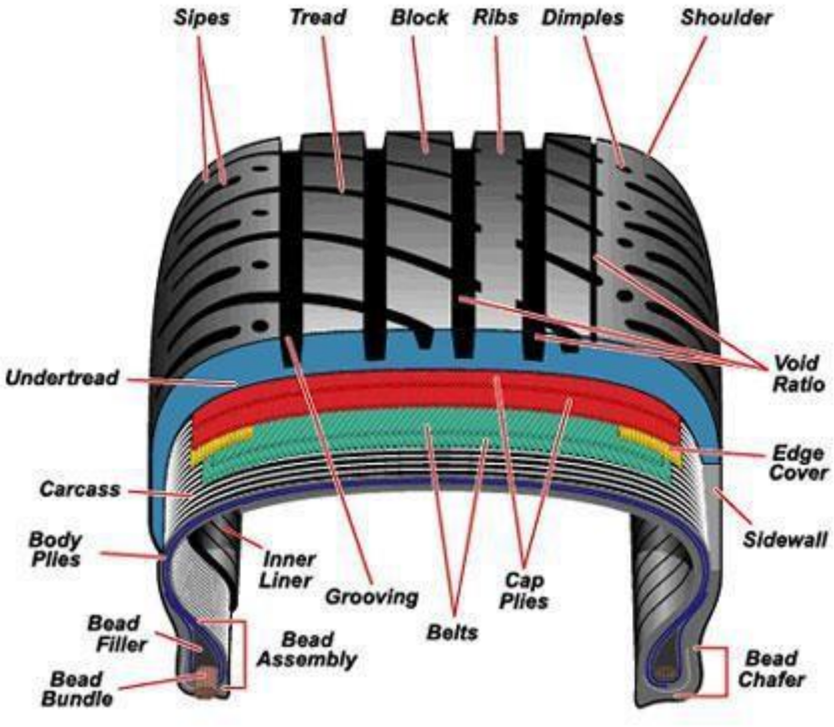
R3 -R3 = R1 = 9.7 mm

R4 – R4 = R2 = 6.4 mm

H -H = $(-1) * (\sqrt{D^2 - (1.5 * R1 + 0.25(A))^2} - D - G)$ = 21.2 mm

4 Construction of a tire

The tire consists of several parts, which you can see in the picture. All of these parts are made of different materials and play different roles [10].



Picture 4

Construction of a tire

Tread

The tread of tire provides traction and turning grip for the tire while it is designed to resist wear, abrasion and heat. The tread mixture includes synthetic and natural rubbers.

Sidewall

The side wall is used to protect cord plies. It shields them against any scratches, abrasions or other environmental factors, including ultraviolet rays, differences in temperature, chemical agents and more. Beside this, the sidewall features tire markings and information such as the tire size and type.

Carcass (Tire Body Plies)

This is a textile material composed of cord treads, constituting a unique tire contour. Depending on the load index (LI), the carcass can have 1,2 or 3 textile layers. The textile layer is intended to keep the tire in shape under inner pressure and to transfer the turning, braking and speeding workload.

Belts

These are rubber-coated layers of steel, fiberglass, rayon and other materials located between the tread and plies, crisscrossing at angles and holding the plies in place. Belts keep the outer diameter of the tire, provide resistance to punctures and help treads stay flat and in contact with the road.

Cap plies

Additional layers of cap plies can be located above the steel belts and towards the tread in order to protect the belt's edges.

Bead wires

A rubber-coated loop of a high-strength steel cable that allows a tire to stay "seated" on a rim. The main function of a bead wire is to hold the tire on the rim and to resist the action of the inflated pressure, which constantly tries to force it off. The bead wires make it possible to mount tires on the rims. Every tire contains two bead wires surrounded by a layer of textile carcass

Inner-Liner

This is an air and water-resistant layer of rubber which replaces the inner tube in tubeless tires. It is made of butyl (a synthetic rubber), which is an air-resistant mixture. One of the inner liner's main features is its high tolerance to oxidizing agents, acids and alkalines.

It is intended to minimalize the loss of air, as well as protect internal elements against air, ozone and water ingress.

Bead Filler (Apex)

The apex (also known as a bead filler) is an extruded, triangle-shaped thick piece of rubber that is attached directly on the bead. The apex increases the sidewall's stiffness in the bead area and contributes to handling performance and sidewall stiffness.

5 Characteristics and markings of a tire

As we mentioned in chapter 2, we chose the specific tire Continental 205/55 R16 90 H for our FEM analysis. For creating a CAD and FEM model of a tire, we need to know the basic characteristics of a tire [9].

Picture 5

Example of tire marking

Tire class

The tire class is indicated by the first letter in the code. Letter designations include P for passenger vehicles, T for temporary spare, LT for light truck metric, C for commercial and ST for special trailer service. The tire class of the specific tire Continental is P (for passenger vehicles)

Section width

The section width of tire is the widest point from sidewall-to-sidewall. In our case, the section width is equal to 205 mm.

Aspect ratio

This is the ratio of the height of the tire's cross-section to its width. The aspect ratio of our tire is 55, which means that the height is equal to 65% of the tire's width (the height of cross-section is 133.25 mm).

The construction of a tire

This letter indicates the type of construction used within the casing of the tire, which in our example is “R” for radial construction. Other examples are “B” for bias-ply or “D” for diagonal construction.

Diameter of tire

The number “16” in our example represents the diameter of the rim (in inches).

Speed rating

The speed rating denotes the maximum speed at which the tire is designed to be operated. For passenger vehicles, this rating ranges from 120 (L speed rating) to 300 km/h (Y speed rating). The speed rating of the specific tire Continental is H, which means, that the maximum operation speed of this tire is 210 km/h.

Load index of tire (LI)

The tire load index is an assigned number that corresponds to the maximum weight that a tire can support when properly inflated. The higher a tire's load index number, the greater its load carrying capacity. Choosing a tire with a lower load index than the original equipment specifications means that the tire will not carry the load capacity of the original.

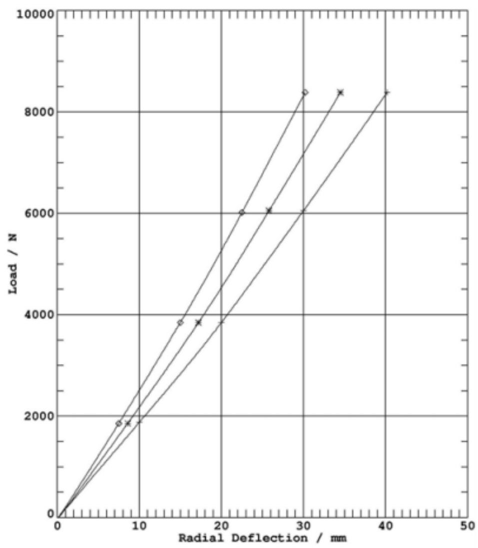
The load index of specific tire Continental 205/55 R16 90, used in the FEM analysis of this diploma thesis, is 600 kg or 5880N.

Vertical stiffness (radial stiffness)

The vertical deflection of a tire is closely proportional to the applied vertical force. The actual vertical stiffness of the tire depends on the longitudinal force, inflation pressure and temperature. Tire radial stiffness is traditionally calculated from the wheel load deflection measurement. The characteristics of vertical stiffness of the specific tire Continental 205/55 R16 90 you can see in the picture 6.

Vertical stiffness on flat surface (load/deflection curve)

Inflation pressure	bar	2.0	2.5	3.0
Diameter	mm	634	634	634
Load at 100% LI	N	5880	5880	5880
Deflection at 100% LI	mm	29.2	25.2	22.1
Radial stiffness at 100% LI	N/mm	223.5	264.8	301.6
Legend		+	✳	◇



Picture 6

Stiffness of tire Continental 205/55 R16 90

Longitudinal stiffness on flat surface

Inflation pressure	bar	2.0	2.5	3.0
Stiffness @ 50% LI	N/mm		397.7	
Stiffness @ 80% LI	N/mm	411.0	435.0	447.4
Stiffness @ 110% LI	N/mm		412.0	

6 Creation of a CAD model of a tire and rim

For creating a FEM model of an automotive rim's load, first we need to create a CAD model of a rim and tire. For creating a CAD model, we used the software CATIA V5. All dimensions are specified in mm.

6.1 Creation of a CAD model of a rim

We made a CAD model of rim 6,5 J x 16 H1 ET 40 using the dimensions described in chapter 3.2.

6.2 Creation of a CAD model of a tire

For the creation of a CAD model of the specific tire Continental 205/55 R16 90, we used the picture of a cross-section of this tire (Pic. 7). The picture of a cross-section is imported to the CATIA V5 by using sketch tracing (a function that creates an immersive sketch).

Before creation of the CAD models of the tire's parts, we divided them into 2 groups: parts made of isotropic materials (isotropic group) and parts made of anisotropic materials (anisotropic group).

The isotropic group includes tread, undertread, sidewall, apex and inner-liner. All these parts are made of different type of rubber and have an identical value of a property in all directions. We also included bead to isotropic group, despite of the fact, that the bead consists of several steel wires and doesn't have an identical value of a property in all directions. In our simplified model a bead is a loop, which made of homogeneous material. The parts of isotropic groups are modeled as solid objects by using cross-section of tire.

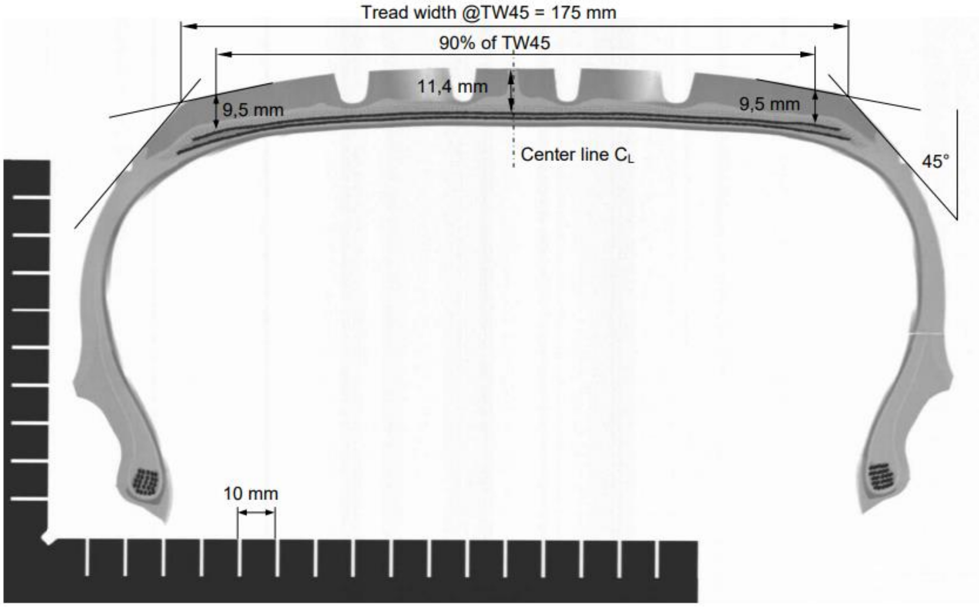
The anisotropic group includes belts, carcass, cap plies, bead flippers and other composite reinforcements of tire. The mechanical properties of this parts depend on the orientation of the material. These parts are modeled as shell objects and located between parts of isotropic group. We have some reasons to model tire's reinforcement as shells:

- The parts of anisotropic group have different material properties in all directions. There is much easier to create material orientation of shell object, than solid object in Abaqus. This is especially important for modeling carcasses of radial and diagonal tires, which have different orientation of reinforcement. Creating of shell's material orientation is described in chapter 7.2.3.
- Tire's reinforcements often contain of several layers with different properties. In Abaqus we can assign to one shell several layers with different properties. We even can combine belts and plies in one shell.
- Carcass, belts and plies have very small thickness (sometimes less than 1 mm). In Abaqus it is very difficult to create mesh of such a thin solid object (there is a strong

probability of getting error: The volume of … elements is zero, small, or negative). Shell objects don't have this problem.

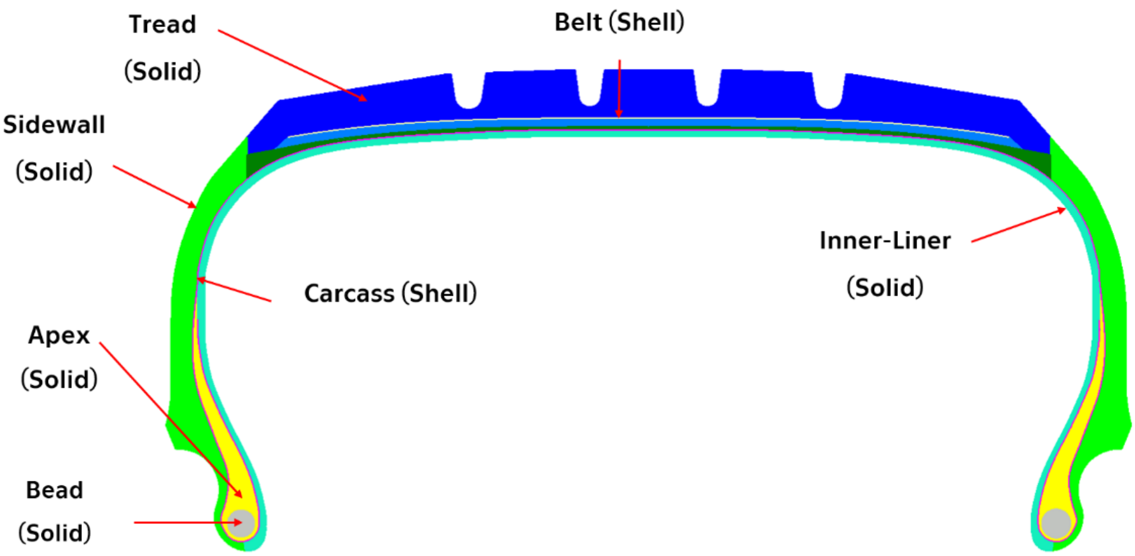
- The shell elements of our model are located between solid elements. We can easily add and remove new elements of reinforcement without changing a geometry of solid elements.

All parts of tire are tied to each other.



Picture 7

Cross-section of tire Continental 205/55 R16 90



Picture 8

CAD model of the tire. Shells (composite parts) are located between solid objects (isotropic parts).

7 FEM model of static loading of automotive rim

At first, we decided to create an FEM model of a static load of a rim. In this model, the wheel doesn't rotate, as the wheel is loaded only by the weight of a car (equal to the load index) and inflation pressure (2.5 bars). For computing this simple model, it is enough to use a quarter model of the wheel only.

For FEM model creation, we used the Abaqus Standard analysis (Static General).

Almost all finite element programs (ABAQUS, Ansys, LS-Dyna…) do not consider the units of given quantities. Abaqus assumes you use consistent units. It does not matter which units you use, as long as they match each other [11]. During the creation of a CAD model, we used mm, so we chose SI mm units, explained in table 2.

Quantity	**SI units**	**SI mm units**
Length	m	mm
Force	N	N
Mass	kg	Ton ($10^3 kg$)
Time	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)
Pressure	Pa (N/m^2)	MPa (N/mm^2)
Density	Kg/m^3	Ton/mm^3
Energy	J	mJ ($10^{-3} J$)
Acceleration	m/s^2	mm/s^2

Table 2

7.1 Import of a CAD model wheel to ABAQUS and the creation of its assembly

The rim and every part of the tire was converted to STP format and imported into ABAQUS. Then we created a quarter model by using the partition command (Tools – Partition – Cell – Define catting plane – 3 Points). After that, all of the parts were added to the assembly (by command Create Instance) and put together by command Translate Instance.

7.2 Defying material properties in ABAQUS

Rubber is the main raw material used in the manufacturing of tires and both natural as well as synthetic rubber are used. Besides this, some parts of the tire are made of steel, fabric and other chemical compounds. In this chapter, we explain how to model different material properties of different parts of tire.

7.2.1 Defying elastic material properties

Steel is used as a material of rim, beads and steel belts in passenger tires. It is an elastic material for defying which we use 3 constants: **Young's modulus**, **Poisson ratio** and **density**.

Young's modulus is a mechanical property that measures the stiffness of a solid material. It defines the relationship between stress and strain in a material in the linear elasticity regime of a uniaxial deformation.

$$\text{Young's modulus } E = \frac{\sigma}{\varepsilon},$$

Where:

σ - the uniaxial stress [MPa]

ε - the strain [-]

The Young's modulus of steel is 210 000 MPa.

Poisson ratio is a measure of the Poisson effect, that describes the expansion or contraction of a material in directions perpendicular to the direction of loading

$$\text{Poisson ratio } \nu = -\frac{d\varepsilon_t}{d\varepsilon_l} = -\frac{d\varepsilon_y}{d\varepsilon_x} = -\frac{d\varepsilon_z}{d\varepsilon_x}$$

Where:

$d\varepsilon_t$ - transverse strain

$d\varepsilon_l$ - longitudinal or axial strain

Poisson ratio of steel is 0.3

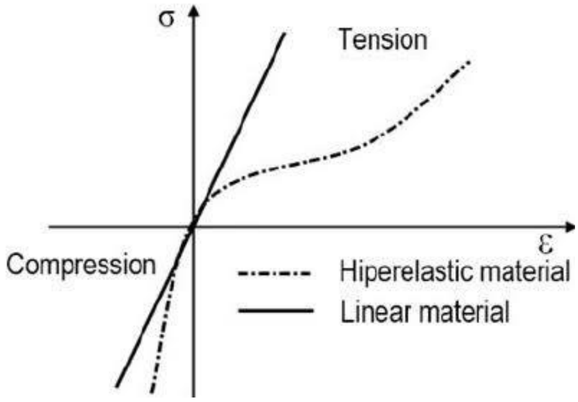
Density of steel ρ = 7 850 kg/m^3= 7.85*10^{-9} ton/mm^3

The material properties of elastic materials such as steel can be easily defined in ABAQUS (property – create material – Mechanical - Elasticity - Elastic).

7.2.2 Defying hyperelastic material properties

Defying the material properties of a rim and bead in chapter 7.2.1, we worked with elastic materials. A typical elastic material such as plastic, steel or wood can stretch up to 2-3% and recover the deformation. The stress of elastic material varies linearly with respect to strain.

The stress - strain characteristic of rubber is different from elastic behavior (picture 9). Rubber-like materials can stretch up to 500% and can still recover the deformation. Beside this, stress – strain characteristic of rubber isn't linear. Materials with this behavior are called hyperelastic or Green elastic materials [12].



Picture 9

Comparison of stress – strain characteristic of elastic and hyperelastic material

There are a lot of different models, describing behavior of hyperelastic material. Some of them, such as Neo-Hook, Mooney Rivlin, Arruda-Boyce and Ogden are used for describing isotropic hyperelastic materials. Another is Holzapfel, which is used for describing anisotropic hiperelastic materials.

The accuracy of every model depends on the range of strain. We must choose a model to use depending on the expected range of strains, available time of computation and the amount of data that we can define in the stress – strain relationship (more accurate models require providing more material parameters).

In table 3 we described the main criteria of the hyperelastic model's choosing.

	Neo-Hooke	Mooney-Rivlin	Arruda-Boyce	Ogden
Computational speed	fastest	Slower than Neo-Hooke	Slower than Mooney-Rivlin	Slowest
Range of strain	Strains up to 100%	Strains up to 200%	Strains up to 300%	Strains up to 700%
benefit	Most accurate for Uniaxial strain	often the most popular because it has reasonable accuracy at a low computational cost.	A special-purpose hyperelastic material for certain rubbery materials such as silicon and neoprene.	A very general purpose hyperelastic material model.

Table 3

Comparison of different models, describing behavior of hyperelastic material

For defining the material properties of the rubber parts of tire such as the sidewall, tread, apex and inner-liner, we used two-parameter Mooney-Rivlin hyperelastiv model because of its reasonable accuracy at a low computational cost. The Monney-Rivlin model represents the stress–strain relation using strain energy as a function of stretch. The form of the strain-energy potential for a two-parameter Mooney-Rivlin model is:

$$W = c_{10}(I_1 - 3) + c_{01}(I_2 - 3) + \frac{1}{D}(J - 1)^2$$

Where:

W – strain-energy potential

I_1, I_2- the first and the second invariant

C_{10}, C_{01} - empirically determined material constants

D - material constants related to the volumetric response

For modeling material properties by Mooney-Rivlin, we need to obtain the values of the material constants $C_{10,}C_{01}, D$. Material constants of a tire's parts we obtained from the diploma thesis "Computational modelling of stress-strain states in tyres" [2].

Part of a tire	C_{10} [MPa]	C_{01} [MPa]	D [1/MPa]
Tread	0.417	0.519	0.103
Sidewall	0.532	0.065	0.151
Inner-liner	0.109	0.259	0.267
Apex	0.638	0.284	0.101

Table 4

Values of hyperelastic material constants

We assigned this material parameters to parts of tire in ABAQUS (Property – Create material – Mechanical -Elasticity – Hyperelastic – Mooney-Rivlin – Coefficient).

Besides this, we need to specify the density of every material in ABAQUS. The density of all elastomer materials in our FEM model is 1250 kg/m^3 (1.25*10^{-9} ton/mm^3) [2].

The density of material can be easily specified in ABAQUS (Property – Create material – General – Density).

7.2.3 Defying composite material properties

Some parts of a tire such as carcass and plies are composites and have anisotropic characteristics.

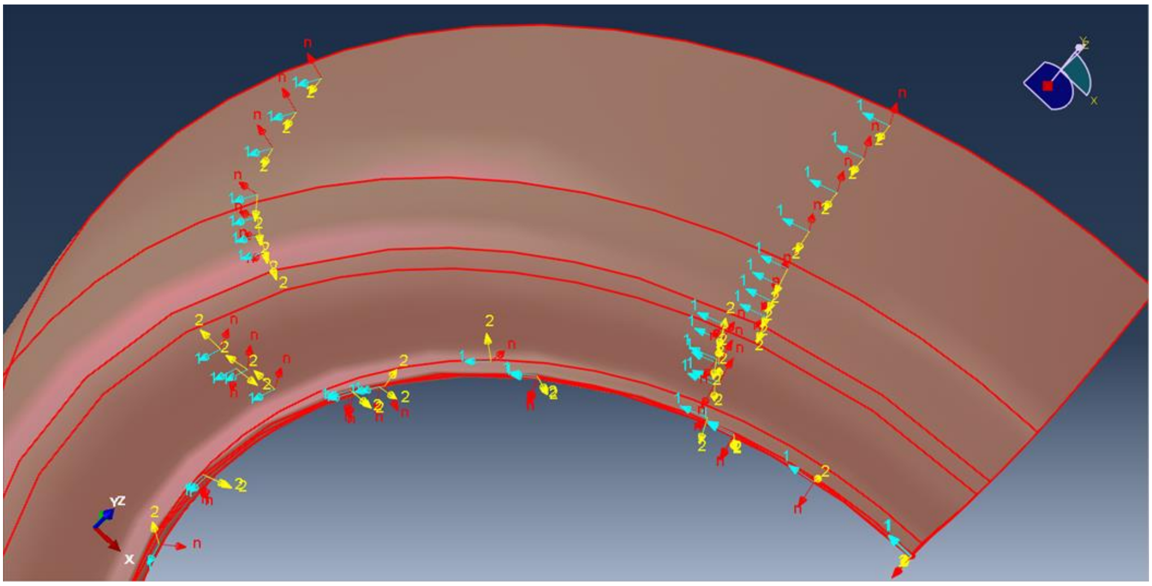
In this chapter, we describe the method of composite parts modeling with the example of a carcass.

All composite parts of our model are modeled as shells. Elastic anisotropic composite materials can be defying by 9 constants: 3 Young's modulus (E_x, E_y, E_z), 3 Shear modulus (G_{xy}, G_{yz}, G_{xz}) and 3 Poisson ratios ($\nu_{xy}, \nu_{yz}, \nu_{xz}$). All of these material constants of tire's parts we obtained from the diploma thesis "Computational modelling of stress-strain states in tyres" [2].

Part of tire	E_x [MPa]	E_y [MPa]	E_z [MPa]	ν_{xy} [-]	ν_{yz} [-]	ν_{xz} [-]	G_{xy} [MPa]	G_{yz} [MPa]	G_{xz} [MPa]
Carcass	1.4	725.0	8.0	0.030	0.460	0.150	10.55	62.67	4.75
Ply	1.2	8.0	725.0	0.150	0.05	0.030	0.83	7.84	9.04
Belt	4.7	60.0	550.0	0.2	0.03	0.006	3.18	51.32	4.61

We assigned this material parameters to parts of tire in ABAQUS (Property – Create material – Mechanical -Elasticity –Elastic – Engineering constants).

The specified material property of composite material isn't enough. A tire's carcass has a difficult form, so we need to define the right orientation of the material. For this we created a cylindrical coordinate system, the axis of which is identical to the axis rotation of the wheel. The coordinate system can be easily created in ABAQUS (Properties – create datum CSYS – Cylindrical). Then we created material orientation using the function assign material orientation. You can see the result in picture 10. The red arrow is normal as to the surface of a carcass. The blue arrow is perpendicular to the axis of the wheel, whereas the yellow arrow is perpendicular to the red and blue arrow.



Picture 10

Composite material orientation of carcass

Shell parts don’t have any thickness. Thickness is defined in the Section (Property – Section – Shell).

The thickness of a carcass, ply and belt were obtained from Diploma thesis “Computational modelling of stress-strain states in tyres” [2].

Table 5

Part of tire	Thickness of layer [mm]
Carcass	0.62
Belt	0.33

7.3 Interactions in model

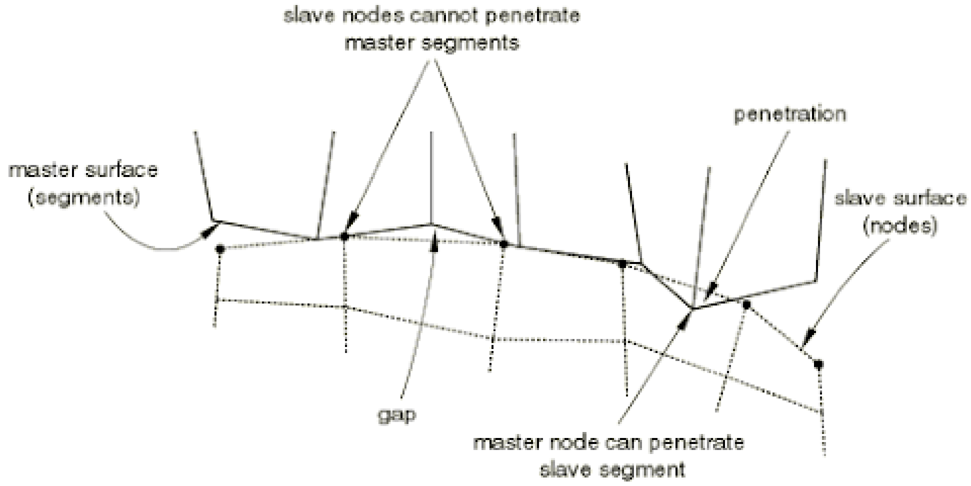
All parts of tire interacting with each other. We need to define it in ABAQUS.

7.3.1 Constraints of the model

All parts of a tire are tied to each other. For creating tie constraint, first of all we need to define the contact surfaces of different parts. The surfaces can be easily specified in ABAQUS (property – surfaces – Create surface). Then we tied contact surfaces by function Create constrain – Tie. We recommend not using “Adjust slave surface initial position”.

Tie constraint demands the correct selection of master and slave surfaces. We should follow the following criteria [13]:

- In a combination of rigid and deformable bodies, the rigid body should be the master and the deformable should be the slave.
- If both surfaces in a contact definition are deformable, then the softer of the two is the slave and the stronger is the master.
- The densely meshed body should be the slave. This is because, ABAQUS allows the master surface to penetrate into the slave (picture 11). To avoid too much penetration, the slave meshing must be denser.
- The longer of the two surfaces should be the master. This will prevent sliding slave nodes from sliding off from the surface and falling behind. If a slave node falls behind a master, excessive convergence issues occur.
- The smoother of the two surfaces should be the master. This is because non smooth surfaces can have gaps or peaks in the mesh or cracks in the mesh. A slave node sliding on such non smooth surface can fall through this crack causing convergence issues.

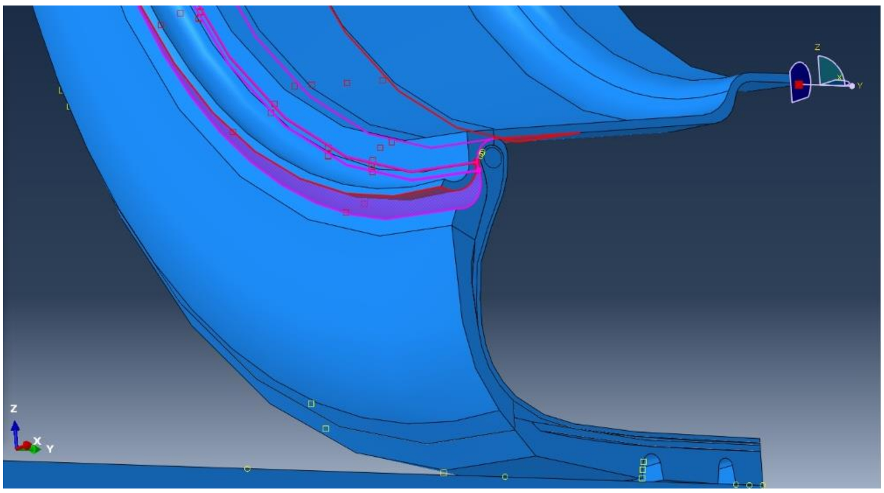


Picture 11

Master and slave surface

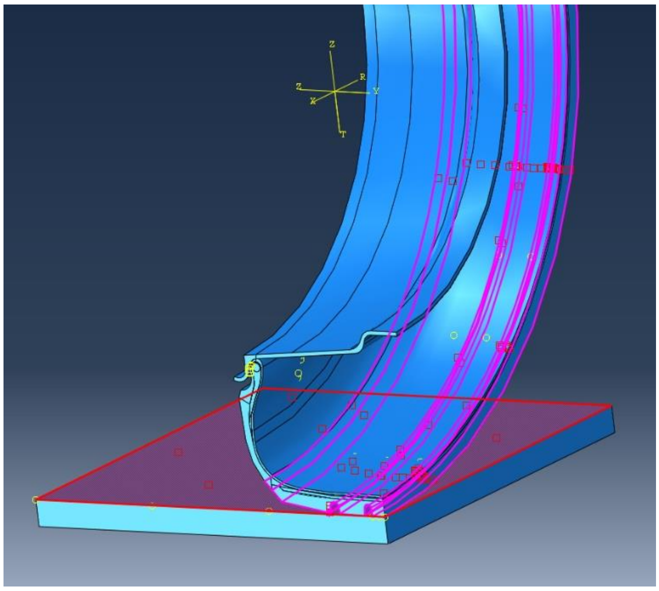
7.3.2 Defining contact

Our FEM model of a wheel contains 2 contacts: contact between the tire and rim in bead seats, contact between the road and tread of tire. For creating surface-to-surface contact interaction, first we need to create interaction property, where we are defying what is considered normal behavior ("hard" contact) and tangential behavior (frictionless or penalty). It is also important to select the master and slave surfaces correctly (see more in chapter 7.3.1) and create a finer mesh in place of contact (see more in chapter 7.4).



Picture 12

Surface to surface contact between rim and tire. Rim, as a more rigid body, is muster (red on the picture). Tire, as a softer body, is slave (pink on the picture).



Picture 5

Surface to surface contact between road and tire (tread). Road, as a longer and more rigid body, is master (red on the picture). Tread, as softer body, is slave (pink on the picture).

7.4 Meshing of model

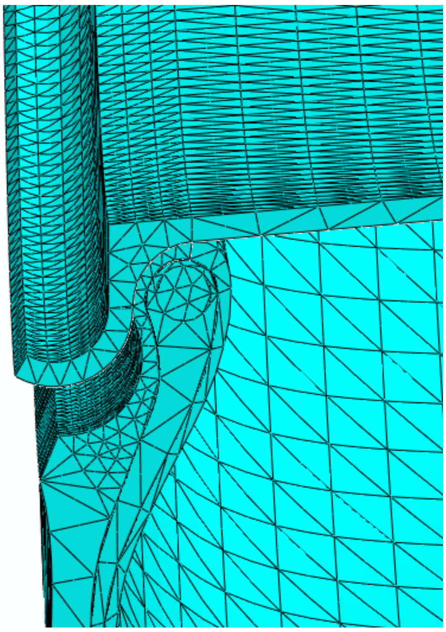
In our FEM model we used tetrahedral mesh because tetrahedral elements fit better when taking into account the complex geometry of a tire. The main parameter of mesh is its size.

An FEM model with finer mesh is more accurate but demands more computational time. Beside this, having too fine of a mesh can cause error "The volume of … elements are zero, small, or negative". So, we need to choose the optimal size of mesh elements.

The table 6 shows the dimension of mesh, which we used in the FEM model. We need to use a finer mesh in contact surfaces. For this, we divided the rim and sidewall into two regions by function partition and defined mesh dimension of mesh for every region.

Table 6

Part	Dimension of mesh [mm]	Dimension of mesh in contact [mm]
Rim	8	3
Road	20	-
Sidewall	8	2
Bead	10	-
Tread, undertread, carcass, belt, inner-liner	8	-



Picture 14

Mesh in contact of rim and tire is finer.

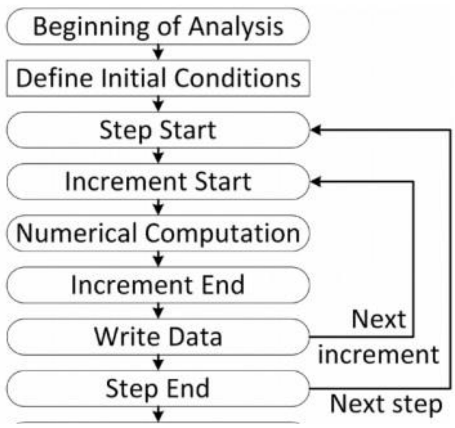
For all parts of tire, made of non-linear (hyperelastic) material, we used the function Hybrid formulation (Mesh – Assign element type- Tet- Hybrid formulation). The Hybrid formulation is used when the material behavior is incompressible (Poisson's ratio = 0.5) or very close to incompressible (Poisson's ratio > 0.475). Rubber is an example of a material with an incompressible material behavior.

7.5 Setting the steps

Abaqus has 2 structural solvers for solving mechanical problems: Abaqus/Standard (implicit) and Abaqus/Explicit. The Implicit solver is primarily used for static problems that do not exhibit severe discontinuities. It is more efficient for solving smooth nonlinear problems, so we chose the Abaqus/Standard for solving the rim's load.

In Abaqus loads and boundary conditions of the simulation is divided into several steps. Each step is a period of time, for which Abaqus calculates the response of the model to a particular set of loads and boundary conditions. The starting point for each step is the deformed state at the end of the last step. The load of the rim is divided into 3 Abaqus General steps, loads and boundary conditions of which are described in chapters 7.5.1 – 7.5.4. The Abaqus General step is used for simulating linear and nonlinear static as well as quasi static problems. This type of step does not take inertial effects into account. Therefore, the General step cannot be used for the simulation of dynamic problems, where inertial effects play an important role.

The FEM model of the wheel has several sources of nonlinearity: nonlinear materials, nonlinear boundary conditions (contacts) and geometric nonlinearity (large deformations). In a nonlinear analysis the solution cannot be calculated by solving a single system of linear equations, as would be done in a linear problem. Instead, the solution is found by specifying the loading as a function of time and incrementing time to obtain the nonlinear response.



Picture 15

Abaqus operation flow

Therefore, the Abaqus/Standard breaks the simulation into several time increments. The model is solved incrementally through time.

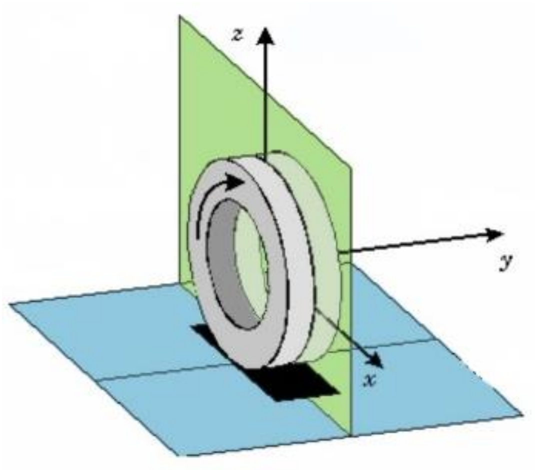
Abaqus starts the step using the value entered for the initial increment size. In our FEM model, we used the increment size and initial increment size of 2 seconds (the increment size of our model is measured in seconds because of the chosen system of units).

The other important characteristic is a minimum increment size. The FEM model of tire is very complicated and needs a quite small minimum increment size. If Abaqus needs a smaller time increment than this value to reach a convergent solution, it reports an error and writes the diagnostic information to the message file. In our model, we used a minimum increment size of 10^{-6} seconds.

For each step in the analysis the Step Manager indicates whether Abaqus will account for nonlinear effects from large displacements and deformations. If the displacements in a model due to loading are relatively small during a step, the effects may be small enough to be ignored. However, in cases where the loads on a model result in large displacements, nonlinear geometric effects can become important. The Nlgeom setting for a step determines whether Abaqus will account for geometric nonlinearity in that step. Our model contains large deformations and non-linear materials, so the Nlgeom setting must be toggled in every step.

7.5.1 Step 0: Initial step

The initial step is generated in Abaqus automatically. In the initial step, we defined basic boundary conditions of the quarter model of wheel. First of all, we must define the coordinate system of the wheel. The coordinate system of our model is shown in the picture.



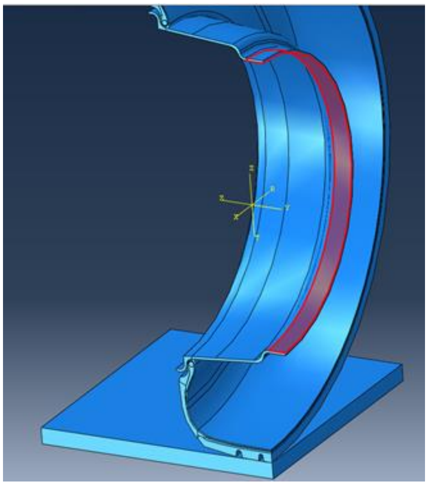
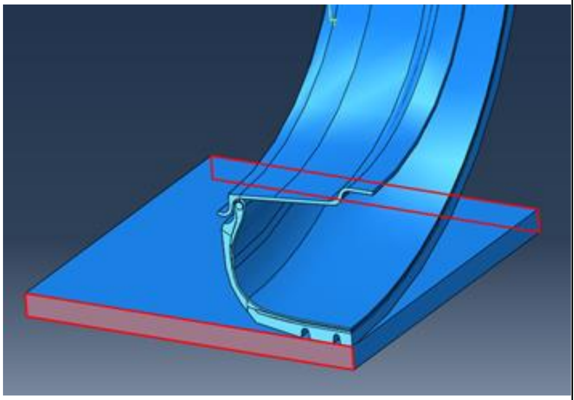
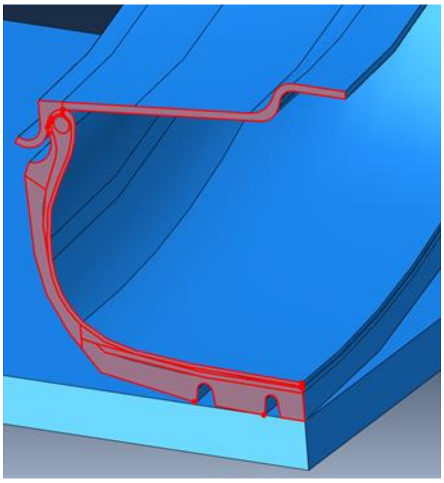
Picture 16

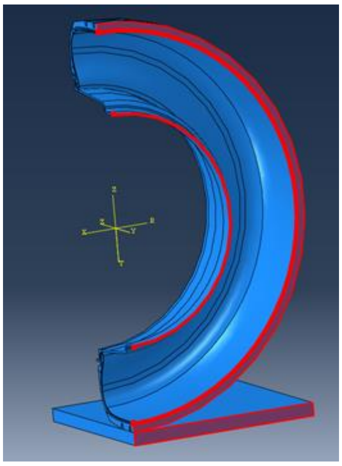
Definition of wheel's coordinate system

In table 7 we explained propagated, deactivated and created loads as well as boundary conditions of the initial step:

Table 7 Description of boundary conditions and loading of initial step

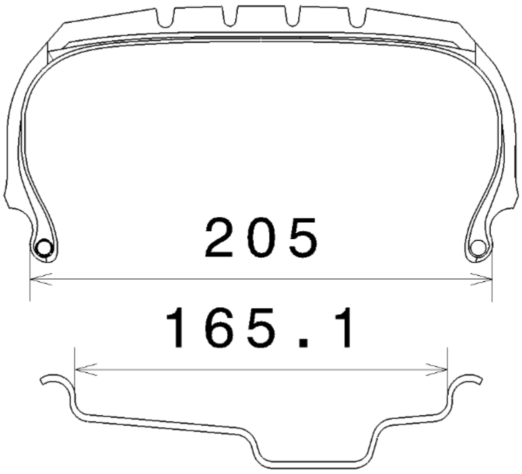
Propagated boundary conditions in the initial step	-
Deactivated boundary conditions in the initial step	-

Created boundary conditions in initial step:							
Name and description	Displacement [mm]			Rotation [°]			Region of the boundary condition (red in the picture):
	x	y	z	rx	ry	rz	
Encastre The rim's drop is fixed and cannot move and rotate.	0	0	0	0	0	0	
Road The road can move towards z and y axis	0	-	-	0	0	0	
Xz Symmetry plane of symmetry of a quarter model	0	-	-	-	0	0	

Yz Symmetry plane of symmetry of quarter model (y displacement is allowed due to step 1)	-	-	-	0	-	0	
Propagated load in the initial step:							-
Created load in the initial step:							-

7.5.2 Step 1: The mounting of a tire

The width of the rim 6,5 J x 16 H1 ET 40 is less than the width of tire Continental 205/55 R16 90 between its bead sits. In the first step we must mount the tire by moving yz plane of symmetry at 19.95 mm ([205-165.1]/2=19.95). The result of Step 1 analysis is shown in the picture

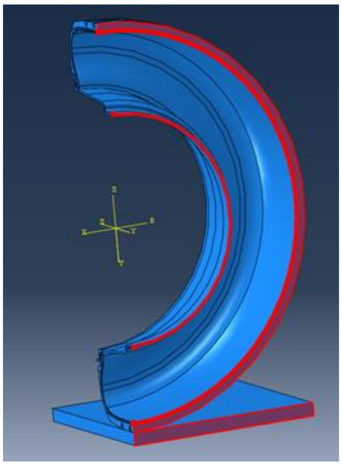


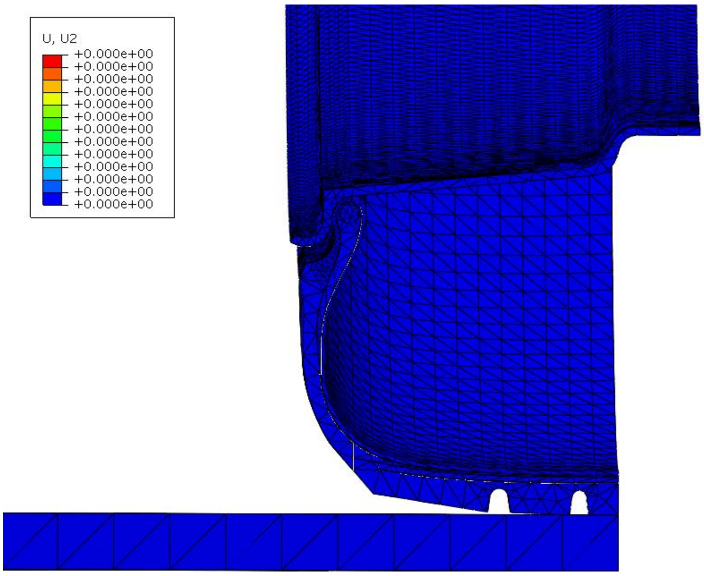
Picture 17

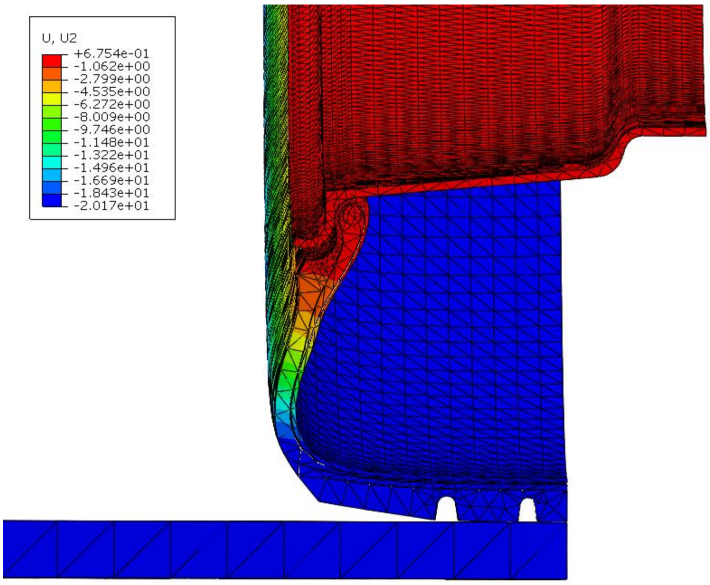
Difference between the width of a tire Continental 205/55 R16 90 and width of a rim 6,5 J x 16 H1 ET 40.

Table 8 Description of boundary conditions and loading of Step 1

Propagated boundary conditions from the initial step to step 1:	**Encastre, Road, Xz symmetry, Yz symmetry**
Deactivated boundary conditions in step 1:	-

Created boundary conditions in Step 1:							
Name and description	Displacement [mm]			Rotation [°]			Region of the boundary condition (red in the picture):
	x	y	z	rx	ry	rz	
Mounting yz plane of symmetry is moving at -19.95 mm towards y axis	-	-19.95	-	0	-	0	
Propagated load in Step 1:							-
Created load in Step 1:							-





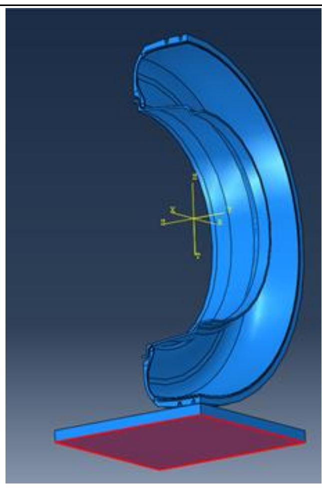
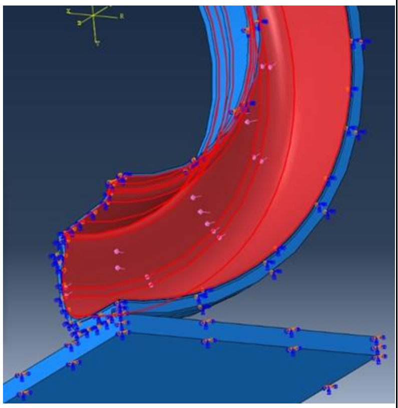
Picture 18

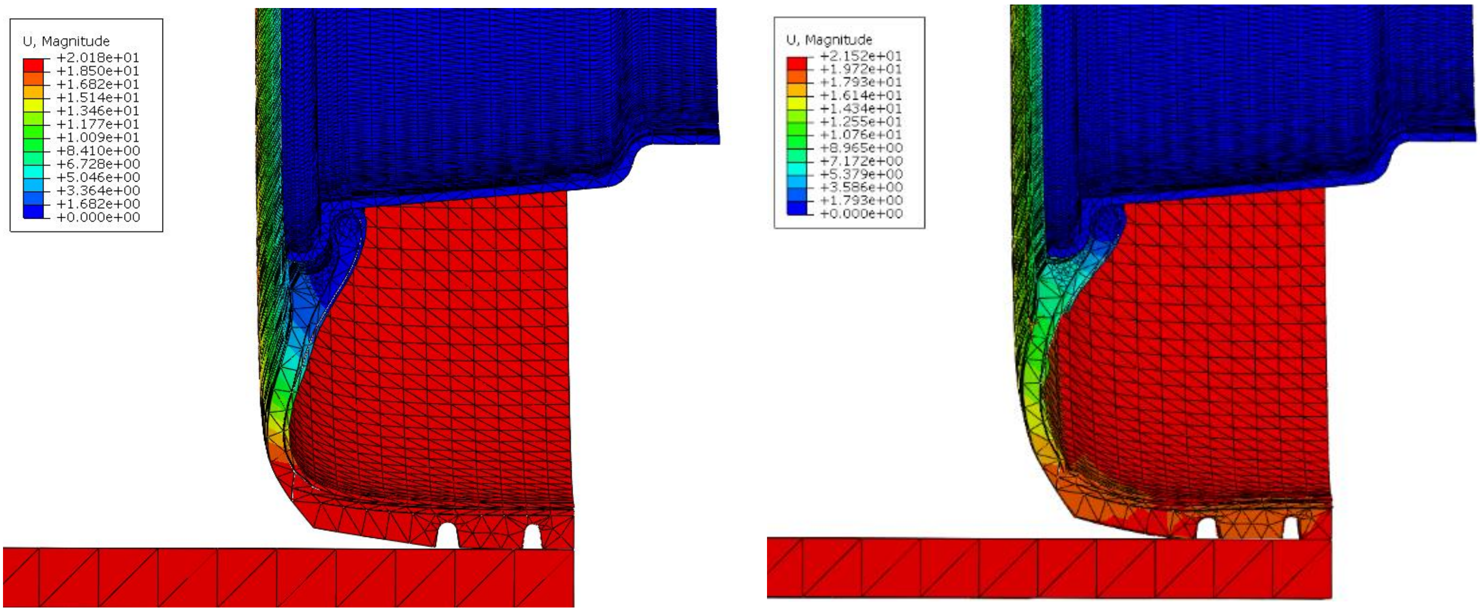
The left picture shows the undeformed quarter model of the wheel before running Step 1. The right picture shows deformed quarter model at the end of Step 1. The observed value is displacement towards the y axe in mm (U2). At the end of Step 1, the tire is mounted on the rim.

7.5.3 Step 2: Inflation of the tire

In Step 2 the tire is inflated by a pressure of 2.5 bars (0.25 MPa). The road in this step is immovable in order to create a place of contact between the tread and road (picture).

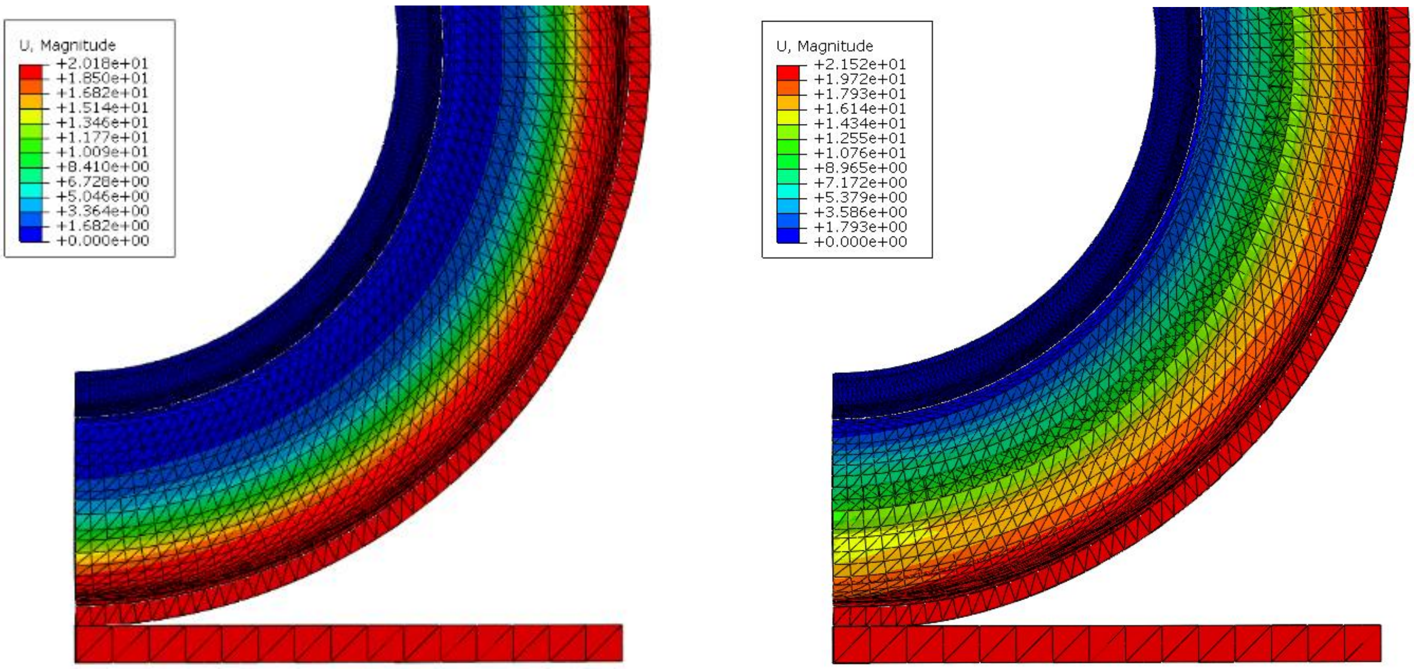
Table 9 Description of the boundary conditions and the loading of Step 2

<table>
<tr><td colspan="4">Propagated boundary conditions in Step 2:</td><td colspan="4">Encastre, Road, Xz symmetry, Yz symmetry, Mounting</td></tr>
<tr><td colspan="4">Deactivated boundary conditions in Step 2:</td><td colspan="4">-</td></tr>
<tr><td colspan="8">Created boundary conditions in the Step 2:</td></tr>
<tr><td rowspan="2">Name and description</td><td colspan="3">Displacement [mm]</td><td colspan="3">Rotation [°]</td><td rowspan="2">Region of the boundary condition (red in the picture):</td></tr>
<tr><td>x</td><td>y</td><td>z</td><td>rx</td><td>ry</td><td>rz</td></tr>
<tr><td>Immovable road
Road cannot move towards z axe</td><td>-</td><td>-</td><td>0</td><td>-</td><td>-</td><td>-</td><td></td></tr>
<tr><td colspan="7">Propagated load in Step 2:</td><td>-</td></tr>
<tr><td colspan="8">Created load in Step 2:</td></tr>
<tr><td colspan="7">Inflation
A pressure of 0.25 MPa is added to the surface of the inner liner and rim.</td><td></td></tr>
</table>



Picture 19

The left picture shows the undeformed quarter model of wheel before running Step 2. Right picture shows the deformed quarter model at the end of Step 2. The observed value is displacement in mm (U). At the end of Step 2, the tire is completely inflated.



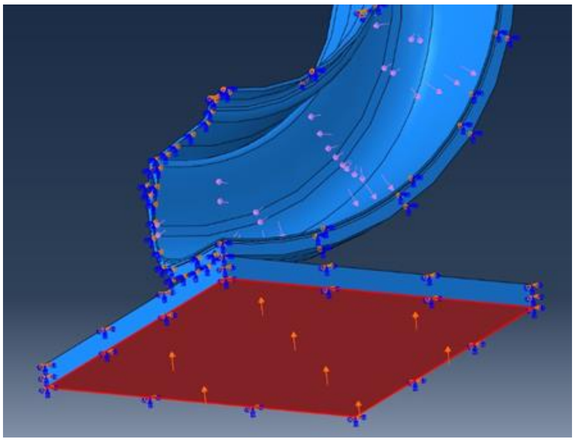
Picture 20

The left picture shows undeformed quarter model of the wheel before running Step 2. The right picture shows the deformed quarter model at the end of Step 2. The observed value is displacement in mm (U). Before Step 2 the tread contacted the road in a line. At the end of Step 2, the contact surface between the tread and the road was created. It is an important to have surface contact for the stable computation of Step 3.

7.5.4 Step 3: Radial loading of the wheel

In this step, the boundary condition “Immovable road” is deactivated and the road can move towards z axis. The Road is loaded by the radial force of the wheel.

Table 10 Description of boundary conditions and loading of Step 3

Propagated boundary conditions in Step 3:	**Encastre, Road, Xz symmetry, Yz symmetry, Mounting**
Deactivated boundary conditions in Step 3:	**Immovable road**
Created boundary conditions in Step 3:	**-**
Propagated load in Step 3:	**Inflation**
Created load in Step 3:	
Radial load The lower surface of the road is loaded by pressure, which simulates radial loading of the wheel. The total force of pressure is equal to a quarter of the tire’s load index, because of using the quarter model of the wheel. The value of the total force is 5880/4 = 1470 N.	

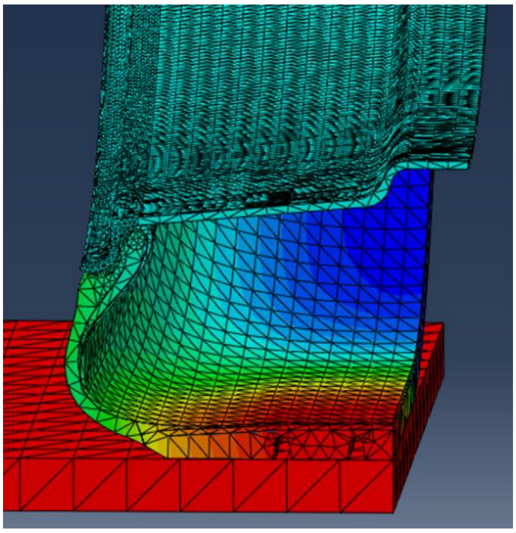
Step 3 is the last step of the analysis. After analysis, Abaqus wrote the results in an ODB file. All results are evaluated and discussed in chapter 7.7.

7.6 Setting of job

For decreasing of the computation time, we recommend using parallelization. In Abaqus you can select the number of processors to use for the analysis.

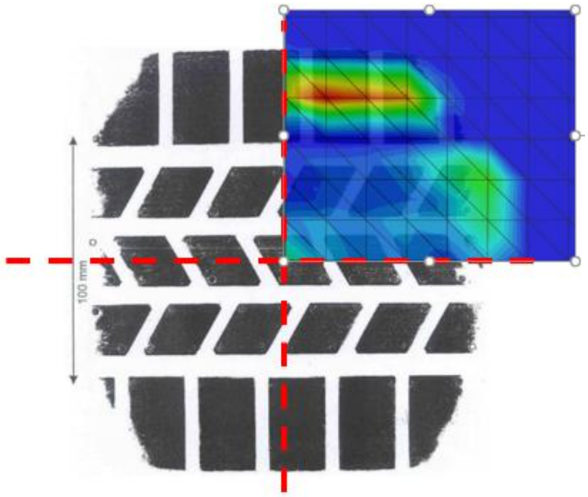
7.7 Evaluation of results

After computing, we got the result of static loading of the wheel. All results must be evaluated and discussed.



Picture 21

Result of the static loading of wheel (radial displacement)



Picture 6

Comparison of a real tire's and model's footprints. The footprint of the model is obtained from contact pressure on the road. The footprint of a quarter model is put over the footprint of a real tire

7.7.1 Evaluation of a tire's behavior

First of all, we need to compare the behavior of the tire's FEM model and the behavior of a real tire under the same conditions. The main criteria which we observed is the radial deflection of the tire. The value of the radial deflection of the tire from the FEM model can be obtained from the displacement of the road towards the z axis (the U3 value in ABAQUS). The radial displacement of the model of the tire is 27.1 mm, which is 1.9 mm bigger than the displacement of a real tire such as the Continental 205/55 R16 90 under the same conditions.

The second criteria which we observed is the dimensions of the tire's footprint. The model of the tire's footprint can be obtained in Abaqus by observing contact pressure on the road. As you can see from picture 19, the footprint of tire model is 2 mm narrower and 14 mm longer than the footprint of a real tire. Also, we can see that contact pressure on the side of the footprint is much bigger, that contact pressure in the center of the footprint, doesn't correspond to reality.

We supposed that the reason for the differences is using the material properties of another tire in the FEM model.

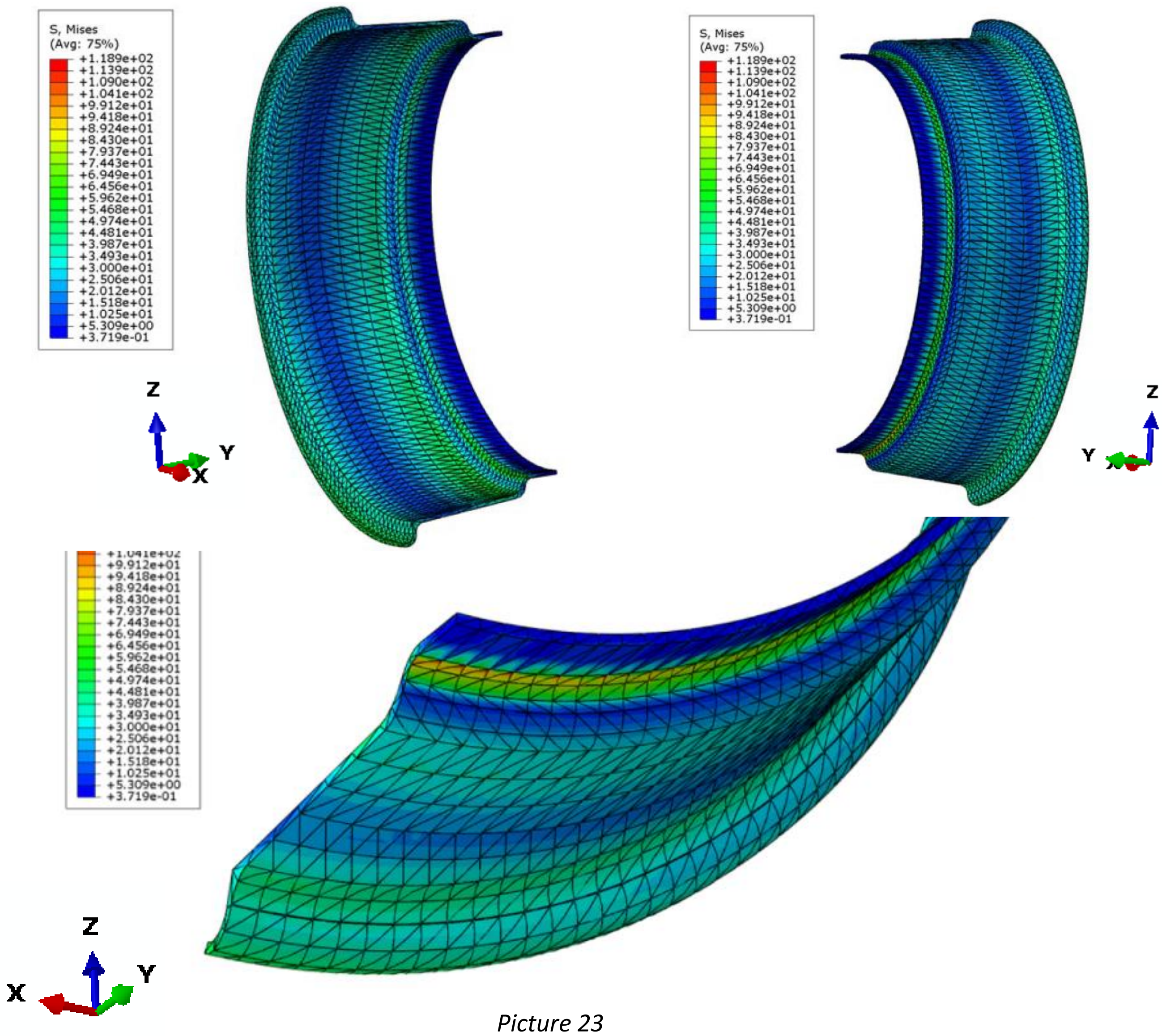
7.7.2 Evaluation of the load of the rim

Analyzing the results of the load of the rim, we observe equivalent tensile stress (also known as von Mises stress). For the main stresses $\sigma_1, \sigma_2, \sigma_3$ the Mises stress is expressed as:

$$\sigma_{von\ Mises} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2}{2}}$$

The von Mises theory states that plastic material begins to damage in places where the von Mises stress becomes equal to the ultimate stress.

The maximum von Mises stress of the rim is 118 MPa. The area of maximum von Misses stress is in the drop of the rim.

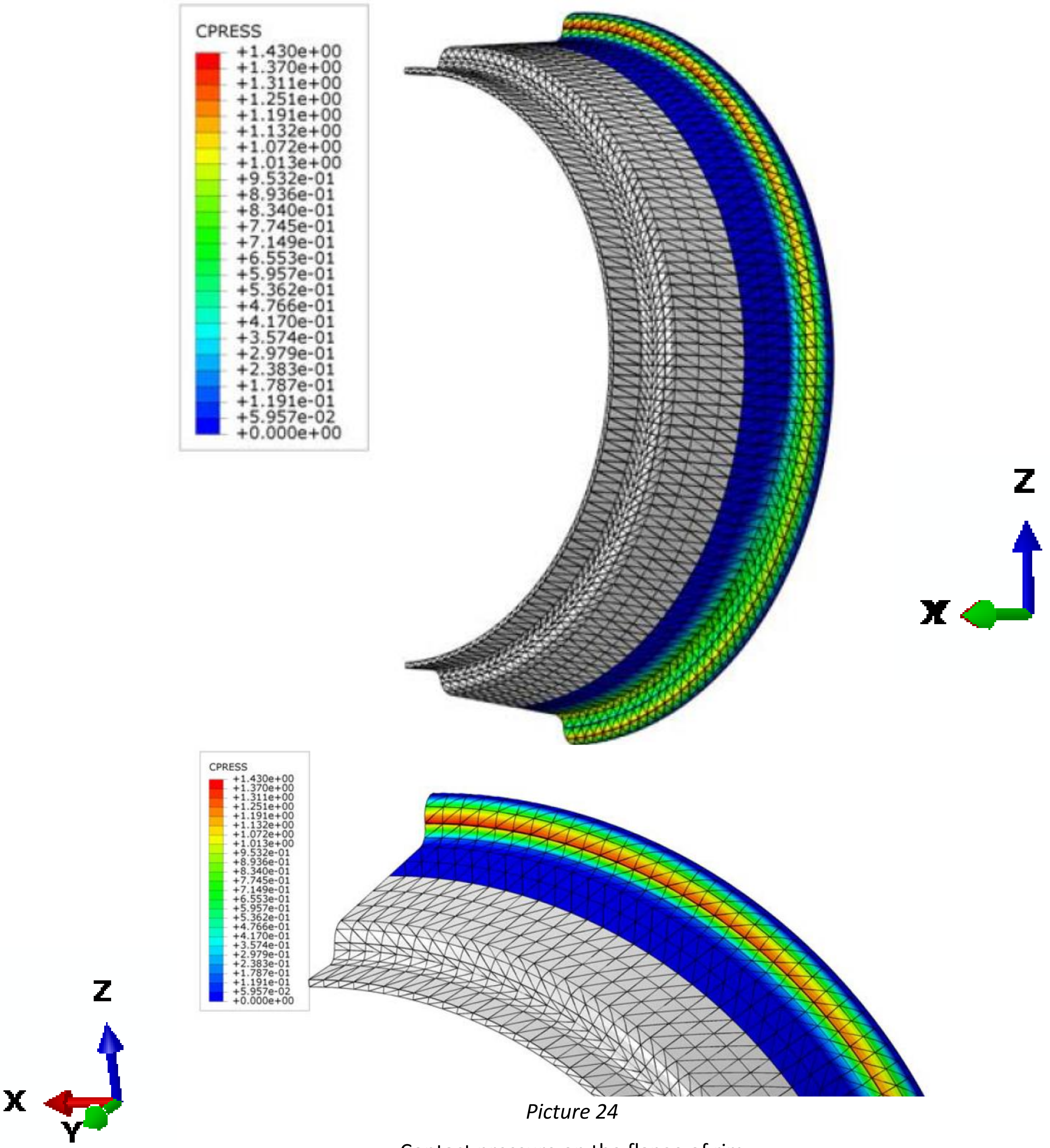


Picture 23

The area of maximum stress (118 MPa).

Beside this, we observe contact pressure (CPRESS in Abaqus) in the area of the rim's bead seat.

The contact pressure is the ratio of the normal load to the contact area. The maximum contact pressure of the rim is 1.4 MPa.

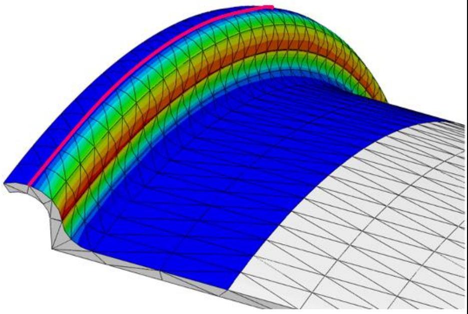
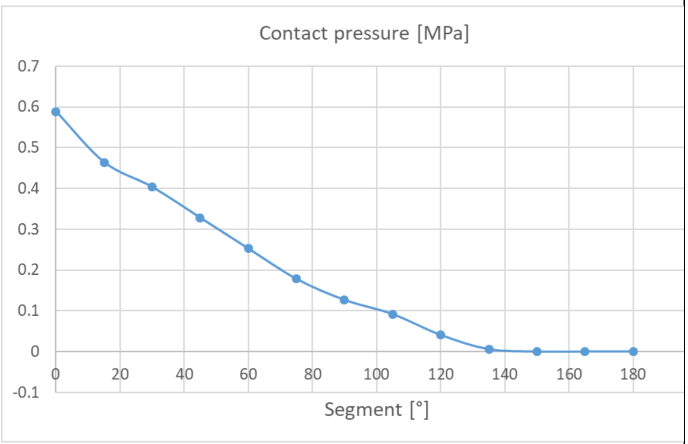
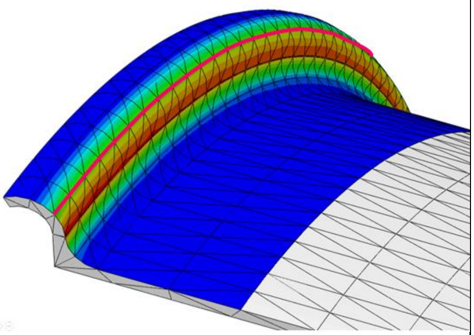
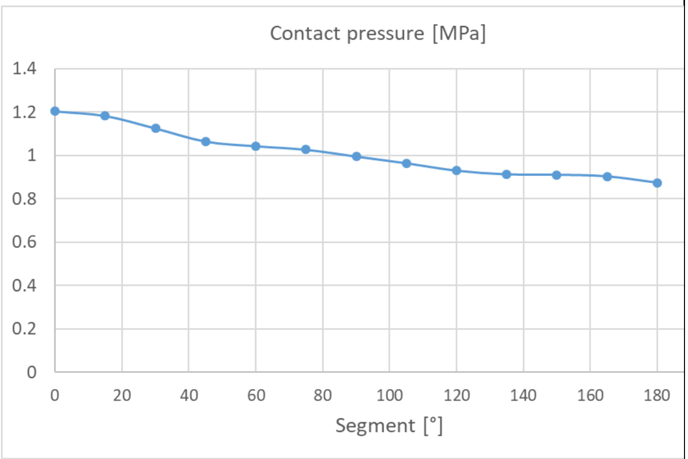
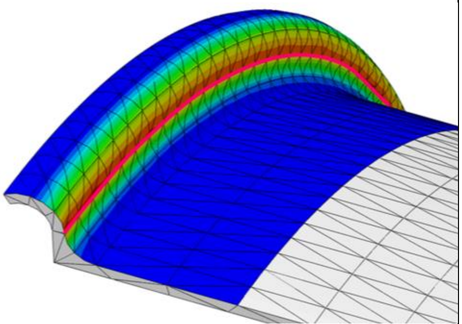
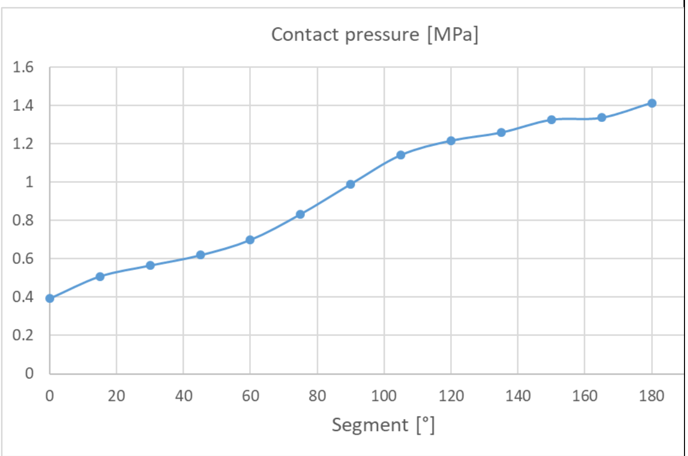


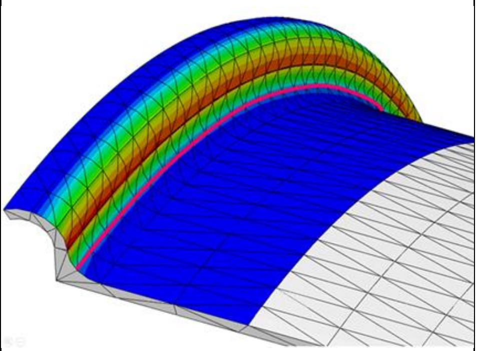
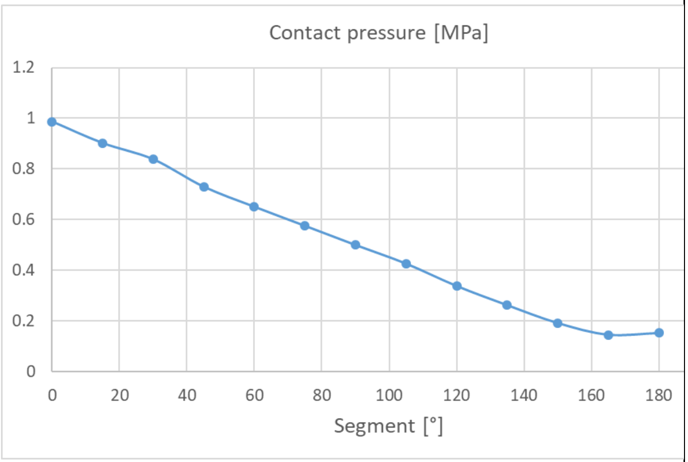
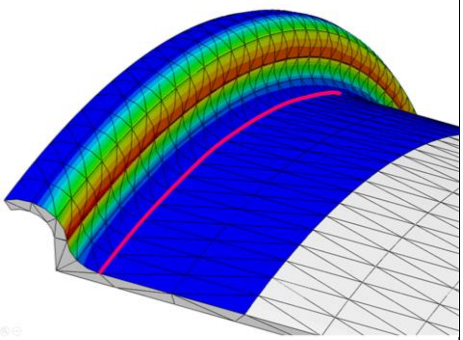
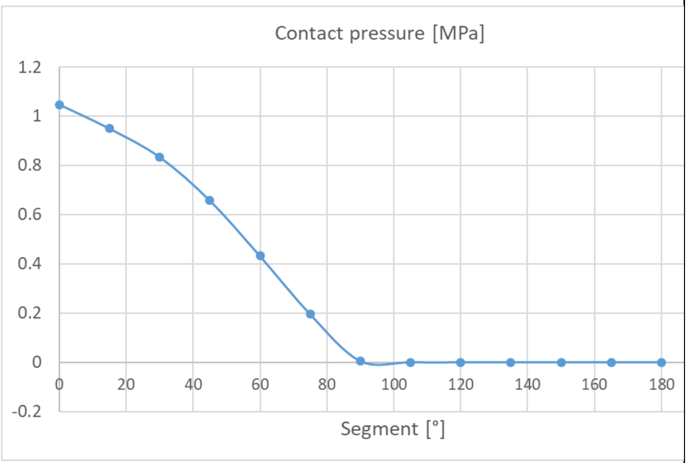
Picture 24

Contact pressure on the flange of rim

For evaluating of results, we divided quarter model of the rim into 12 equal segments (the angle of every segment is equal to 15°). The numeration of segments starts from the lower part of the rim. We measured contact pressure of every segments along several lines, specified in table 11.

Table 11

<table>
<tr><th>Zone of measured contact pressure (pink line)</th><th>Value of contact pressure in segments:</th></tr>
<tr><td></td><td>
</td></tr>
<tr><td></td><td>
</td></tr>
<tr><td></td><td>
</td></tr>
</table>

As you can see from the table 11, contact pressure of lower and upper part of flange is different.

8 Future development of the model

The main problem of the current FEM model is the low accuracy of the geometry and material properties. For the precise modeling of an automotive rim load, we offer another way for the model's development:

1. To measure the dimension of the footprints, radial deflection and radial stiffness of the real tire for different loads (50, 80, 100 and 110% of a tire's Load Index) and the different inflation pressure of a tire.

2. To create an accurate cross section of tire and get precise dimensions of a tire's parts (thicknesses of layers, geometry of the carcass, etc.).

3. To extract a carcass manually, tread, inner-liner and others from a tire and cut it into proper strips. Carry out a uni-axial test of rubber hyperelastic material property and determine hyperelastic constants. Obtain the elastic modulus and Poisson ratio for the carcass, belts and cap plies.

4. The automotive rim is most loaded when passing an obstacle (dynamic loading). For this modulation, we need to know the viscoelastic properties of the tire's materials (relaxation modulus and relaxation time). For obtaining these characteristics, we need to conduct a stress relaxation test of a tire's materials.

9 Parametrization of a model

Automotive rims of a different size have different stress and strain characteristics. For computing the load of automotive rims of a different size, a parametrical FEM model should be created. For parametrization of an automotive rim and tire, we used the method described in the Bachelor thesis of Miroslav Ryšavý "Parametrical model of wheel and tire" [6].

9.1 Parametrization of an automotive rim

There are two types of parameters: input parameters and output parameters. Input parameters (diameter of a rim, width of a wheel, etc.) are independent and specified by the size of a rim. The output parameters (angel of drop, width of flange etc.) are dependent upon the input parameters and calculated by software.

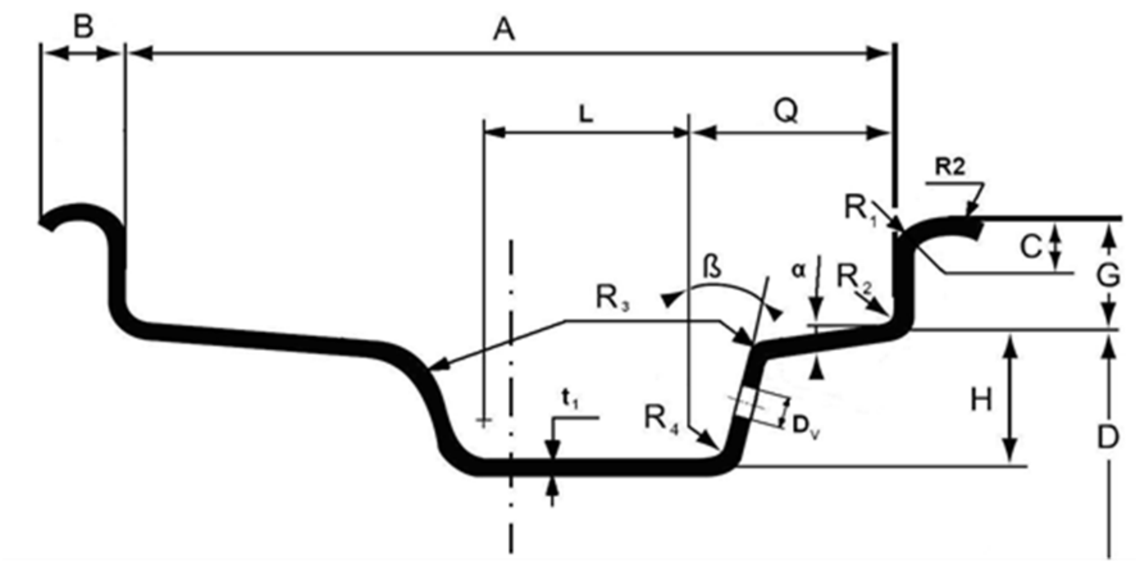


Table 12 Input and output parameters of rim

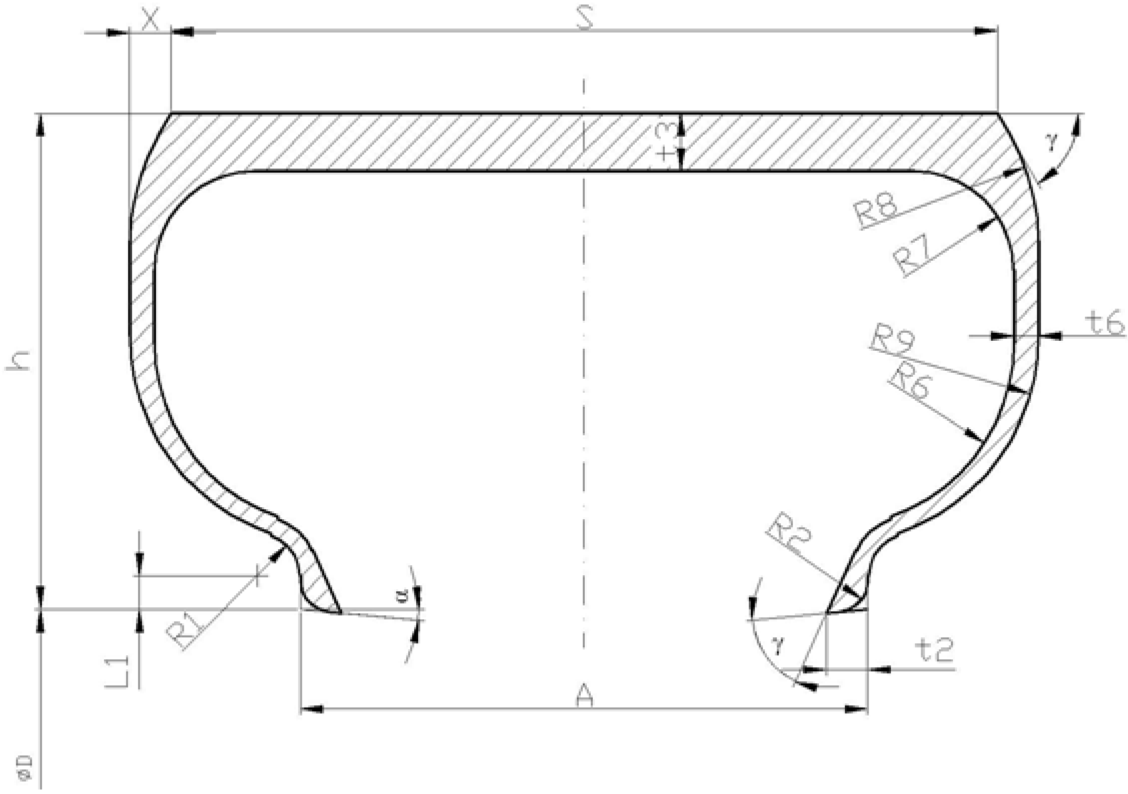
	Parameter	Description	Calculation
Input parameters	A	Width of the wheel [inch]	$A = A'' \cdot 25{,}4[mm]$
	D	Diameter of the wheel [inch]	$D = D'' \cdot 25{,}4[mm]$
	R_1	Outside rounding	-
	R_2	Inside rounding	-
	G	Hight of the flange	-
	α	Angle of bevel of bead seat	α=5° α=15°

	D_v	Diameter of hole for tire's valve	$D_V = 11{,}5[mm]$
	t_1	Thickness of the rim	-
Output parameters	β	Angel of the drop	$\beta = D''[^\circ]$
	C	-	$C = R_1[mm]$
	B	Width of the flange	$B = 1{,}5 \cdot R_1[mm]$
	Q	-	$Q = 0{,}25 \cdot (A \cdot 25{,}4)$
	L	Width of the drop	$L = 0{,}22 \cdot (A \cdot 25{,}4)[mm]$
	R_3	-	$R_3 = R_1[mm]$
	R_4	-	$R_4 = R_2[mm]$
	H	Drop	$H = (-1) \cdot \sqrt{D^2 - (Q+B)^2} - D - G$

The input parameters must be defined in the Parameter Builder of ABAQUS. The formulas of output parameters must be described in the Expression Builder.

9.2 Parametrization of tire

By analogy in the parametrization of an automotive rim, we can parametrize tires:



Picture 25 Parametrization of tire

Table 13 Input and output parameters of tire

	Parameter	Calculation
Input parameters	S	-
	L	-
	D	$D = D'' \cdot 25{,}4[mm]$
	α	α=5° α=15°
	A	$A = A'' \cdot 25{,}4[mm]$
	R_1	-
	R_2	-
	G	-In this diploma thesis we created FEM
	γ	γ = 60°
Output parameters	h	$h = \frac{L}{100} \cdot S[mm]$
	X	$X = 0{,}05 \cdot S[mm]$
	L_1	$L_1 = G - C = G - R_1$
	t_2	$t_2 = 0{,}05 \cdot S[mm]$
	t_3	$t_3 = 0{,}07 \cdot S[mm]$
	t_6	$t_6 = 0{,}03 \cdot S[mm]$
	R_6	$R_6 = R_9 - t_6[mm]$
	R_7	$R_7 = 4 \cdot t_6[mm]$
	R_8	$R_8 = \frac{1}{2}(h - L_1 - R_1)[mm]$
	R_9	$R_9 = \frac{1}{2}(h - L_1 - R_1)[mm]$

The problem of the model's parametrization in Abaqus is that when we change the size of the rim and tire, we must to set the model again (set the assembly, create a new mesh).

10 Conclusion

In this diploma thesis we created the FEM model for computing the load of an automotive rim. We made CAD model of rim 6,5 J x 16 H1 ET 40 and tire Continental 205/55 R16 90 in software CATIA. CAD model of tire consists of solid and shell parts.

FEM model of wheel was created in software Abaqus. Material properties of tire was obtained from tire MATADOR 165/65 R13. For defining the material properties of the rubber parts of tire we used two-parameter Mooney-Rivlin model. Composite parts of tire were modeled as shell.

As a result, we compute static loading of automotive rim and got values of contact pressure of rim. The tire of the FEM model turned out softer than the real tire. We supposed that the reason of this distinction can be due to using the material properties of another tire.

11 Sources

[1] Balejík Gorazd. Development composite wheel rim for Formula Student Car. ČVUT in Prague, 2017

[2] Ondřej Lavický. Computational modelling of stress-strain states in tyres. Brno, 2008

[3] Tyre Model Performance Test Continental Data Description, 2003

[4] Shahyar Taheri, Corina Sandu, and Saied Taheri. Finite Element Modeling of Tire Transient Characteristics in Dynamic Maneuvers. Virginia Tech

[5] ABAQUS 6.14 EXAMPLE PROBLEMS GUIDE. VOLUME II: OTHER APPLICATIONS AND ANALYSES. Dassault Systems

[6] Miroslav Ryšavý. Parametrický model disku a pneumatiky. ČVUT in Prague, 2007

[7] Tire widths for a wheel/rim size chart - Equivalency table https://www.tyresizecalculator.com/charts/tire-width-for-a-wheel-rim-size-chart

[8] Standard. ČSN 30 3707 "Wheels for tyres. Terms and definitions. Rim types and rim codes"

[9] Tire markings https://www.continental-tires.com/car/tire-knowledge/tire-basics/tire-markings

[10] TIRE CONSTRUCTION https://www.bridgestoneamericas.com/en/corporate-social-responsibility/safety/tires-101/tire-construction

[11] Abaqus documentation. Units

https://abaqus-docs.mit.edu/2017/English/SIMACAEGSARefMap/simagsa-c-absunits.htm

[12] Nomesh Kumar, V. Venkateswara Rao. Hyperelastic Mooney-Rivlin Model: Determination and Physical Interpretation of Material Constants. MIT International Journal of Mechanical Engineering, Vol. 6, No. 1, January 2016.

[13] Abaqus analisis user's manual.

http://dsk-016-1.fsid.cvut.cz:2080/v6.12/books/usb/default.htm?startat=pt09ch38s02aus182.html