ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

NUMERICAL EVALUATION OF THE TIRE FORMING PROCESS IN THE MOLD

Rubens de Castro Casagrande Pinati
Wesley Góis
Federal University of ABC, Santo André, São Paulo, Brazil
rubenspinati@yahoo.com.br
wesley.gois@ufabc.edu.br

Abstract. *The aim of this study is to evaluate numerically the tire behavior in the forming process in the mold. Specifically, in this tire-forming process occurs the interaction between bladder, raw tire and mold; but, this process is in a closed place, so it is impossible to see how this phenomenon develops. Therefore, it is useful to study this phenomenon by Finite Element Method (FEM). The FEM numerical analysis was performed in the software Abaqus/ CAE together with AutoCAD (pre-processing - mesh generation). The biggest challenge of this work is the physical behavior modeling of rubber compounds that yet are "raw" (before occurring cross-links between the polymer chains and sulfur), immediately upon the mold closing. In the first numerical evaluation, it was used an axisymmetric finite element model with hyperelastic materials characterized as reduced polynomial type, that takes into account only the portion of elastic material. Some three-dimensional (3D) models are also proposed to try to help the full simulation of the tire forming process in the mold. The numerical results indicate that all proposed simulations show a good corralation with the representation of this specific process.*

Keywords: *tire, hyperelastic, finite element, Abaqus/CAE*

1. INTRODUCTION

In the tire industry, the Finite Element Method (FEM) has been used since the 70s initially with plane models and triangular elements that have already incorporated the anisotropy and the non-linearity concepts. About twenty years ago, the FEM models were at most two-dimensional (2D) or axisymmetric, merely simulate very specific static conditions. (Robecchi, et al., 1980), using two-dimensional elements and assuming the plane strain condition, made the first simulations that intended to represent loaded and deformed tires in contact with the ground. An excellent review of FEM analysis applied to tires is presented in (Gall, et al., 1993), which highlights the first three-dimensional (3D) simulation with tread design, (Costa, 2000). For further details on FEM analyzes application in pneumatic, see (Costa, 2002), (Pinheiro, et al., 1999), (Zucato, 2006), (Pinheiro, 2001), (Gent, et al., 2005) and (Helnwein, et al., 1993).

Modeling and simulating (by FEM) the green tire forming process is very important. Many problems can arise at this tires manufacturing stage. However, in the literature, such simulations are not often performed to better characterize the stresses, deformations and contact pressures involved in this tires manufacture stage.

Thus, the aim of this work is to contribute to some pneumatic manufacturing steps by simulating the tire forming process in the mold, applying numerical models constructed with ABAQUS©, a computational tool - based on FEM. Specifically, we intend to study the bladder influence in possible defects in the bead area and develop a methodology for analyzing (by ABAQUS©) the green tires. In this work was proposed to analyze the bladder defects mechanisms in order to help developing possible materials and geometries changes. Finally, it was evaluated the contact pressure between bladder and liner (tire internal side), the mold tire stamping pressure and the format geometry tire change after cured.

In the forming process the bladder has a very important role in tire manufacturing and needs to be in perfect condition to prevent tire defects at vulcanization step. Furthermore, it is desirable a large lifetime bladder as possible to reduce the number of exchanges and deform uniformly avoiding the tire non uniform.

Following the text, it was simulated a 2D tire model forming process in the mold, see item 2. However, this two-dimensional model cannot verify the radial bladders folds (that influence the bladders life and how they inflate inside the tire in its full cycle of work). Because of these, in item 3, was studied a 3D model in order to verify possible defects in bladders. In section 4, we highlight the possible continuation proposals for this work. Lastly, in section 5, were presented the conclusions.

2. NUMERICAL MODELS IN 2 DIMENSIONS – FORMING PROCESS

In order to simulate the tire forming process in the mold a FEM model was built in commercial software - ABAQUS©/Standard (ABAQUS, 2012) according to the geometry of the experimental apparatus described in Fig. 1. A quasi-static physic and contact nonlinear approach with negligible temperature effects was assumed for simulations. By taking advantage of axisymmetry it was possible to simulate the die, bladder and tire assembly as 2D axisymmetric model.

ISSN 2176-5480

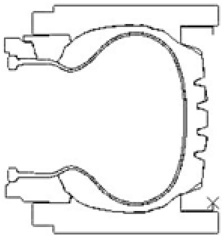


Figure 1. Assembled set – Bladder, tire and die.

The flexible rubber was modeled using CAX4RH elements. CAX4RH is a 4-node bilinear axisymmetric quadrilateral, hybrid, constant pressure, reduced integration, hourglass control element. The interfaces between the die and the bladder, and between the tire and the die, were modeled using an automatic surface-to-surface contact algorithm. A Coulomb friction law was assumed and a coefficient of friction of 0.15 was assigned between all parts. The friction coefficients values are based on the results obtained by (Ramezani, et al., 2009). The die was modeled as a rigid body. Tensile tests were carried out using model 3367 INSTRON universal testing machine.

The dynamic material modulus (flexible rubber) is defined as a complex number in which the real part is the tensile portion component in phase with the strain and its imaginary part is the tensile portion component 90° out of phase in deformation relation, Eq. 1.

This last component is responsible for energy losses.

$$\boldsymbol{E}^* = \boldsymbol{E}' + \boldsymbol{iE}'' \tag{1}$$

Where $E*$ is the complex modulus full. The real component E' is called the elastic or storage modulus and E'' is the loss modulus or inelastic modulus.

Flexible materials have nonlinear stress–strain characteristics for relatively large deformations. Under such conditions, they are generally assumed as nearly incompressible. To model these hyperelastic materials through FEM, a constitutive law based on total strain energy density W has to be adopted. Among several approaches, Mooney–Rivlin theory (Mooney, 1940) and Ogden theory (Ogden, 1972) are frequently used to model rubber materials in finite element simulations. These material models have previously been used with success to predict the behavior of hyperelastic materials (see e.g. (Girard, et al., 2006) and (Ramezani, et al., 2010)).

Mooney–Rivlin form is motivated by the fact that the free energy of any homogeneous, isotropic material can be expressed as an infinite series of the three strain invariants. For a first-order expansion, Mooney–Rivlin form is given by

$$\boldsymbol{\sigma_{ii}} = \frac{\partial \boldsymbol{W}}{\partial \boldsymbol{\varepsilon_{ij}}} \tag{2}$$

$$\boldsymbol{W} = \boldsymbol{C_1}(\bar{\mathbf{I}}_1 - \mathbf{3}) + \boldsymbol{C_2}(\bar{\mathbf{I}}_2 - \mathbf{3}) + \frac{\mathbf{1}}{\boldsymbol{D_1}}(\boldsymbol{J} - \mathbf{1})^2 \tag{3}$$

where W is the strain energy per unit of reference volume; $\bar{I}_1$ and $\bar{I}_2$ are deviatoric strain component invariants; J represents volume change and C_1, C_2, D_1 are material constants. C_1, C_2 describe deviatoric component and D_1 describes compressibility.

The form of the Ogden strain energy potential is:

$$\boldsymbol{W} = \sum_{i=1}^{N} \frac{2\mu_i}{\alpha_i^2}\left(\bar{\boldsymbol{\lambda}}_1^{\alpha_i} + \bar{\boldsymbol{\lambda}}_2^{\alpha_i} + \bar{\boldsymbol{\lambda}}_3^{\alpha_i} - \mathbf{3}\right) + \sum_{i=1}^{N} \frac{\mathbf{1}}{\boldsymbol{D_i}}(\boldsymbol{J}^{el} - \mathbf{1})^{2i} \tag{4}$$

where $\bar{\lambda}_i$ are the deviatoric principal stretches which can be defined by $\bar{\lambda}_i = J^{-1/3}\lambda_i$; J is the total volume ratio; J^{el} is the elastic volume ratio; and μ_i, α_i e D_i are temperature-dependent Ogden constants. Compressibility can be defined by specifying nonzero values for D_i, by setting the Poisson's ratio to a value less than 0.5, or by providing test data that characterize the compressibility. We assumed a fully incompressible behavior for rubber with $\nu = 0{,}4997$ and D_i equal to zero and so the second expression in Eq. (4) can be eliminated. To determine the strain energy density W, ABAQUS© uses a least-squares fitting algorithm to evaluate the Mooney–Rivlin and Ogden constants automatically from experimental data. These two hyperelastic material models are compared with experimental behavior and the results are shown in Fig. 2. As can be seen from Fig. 2, Mooney–Rivlin model is in relatively good agreement with the results of experimental stress–strain curve. However, Ogden model is in very good agreement with the results of experiments and almost produces the same stress–strain curve compared to experiment. After this comparison, Ogden (Ogden, 1972) model has been used for finite element simulations of this paper.

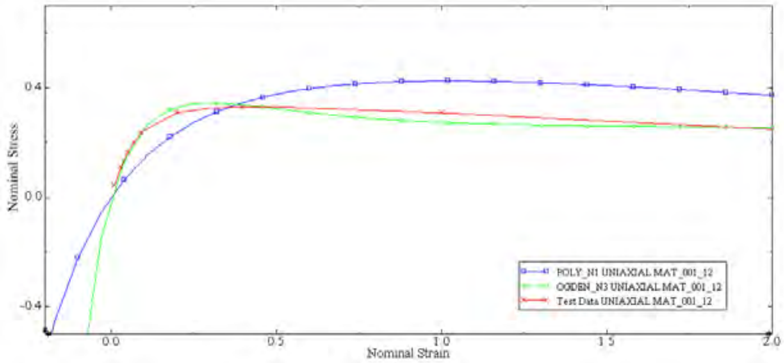


Figure 2. Comparison between Mooney–Rivlin and Ogden hyperelastic material models.

2.1 Set simplified model

Before the complete model construction, as stated earlier, the materials were characterized and initially generated a simplified model, considering the tire as a hard body and deformable bladder.

In vulcanization step the bladder is inflated inside the tire and thus its thickness varies along the contact with the liner. Also influencing heat transfer from the inside bladder to the tire and also the distribution of pressure between the liner and the bladder. Fig. 5 shows the steps of inflating bladder inside the tire.

For the bladder characterization was used the tie tensile test DIN 53504S3A. Samples were taken both in the bladder radial direction and in the bladder longitudinal direction to be checked the mass anisotropy. Follows in Fig. 3 the scheme were taken as bladder samples.

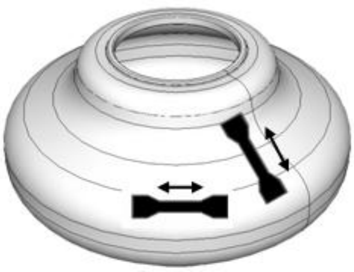


Figure 3. Samples from the bladder.

It was observed in stress-strain graph of samples a little vary in two directions (horizontal and longitudinal) (Fig. 4), and because of this, it was considered the same rubber properties in all directions, in other words, the material was considered isotropic.

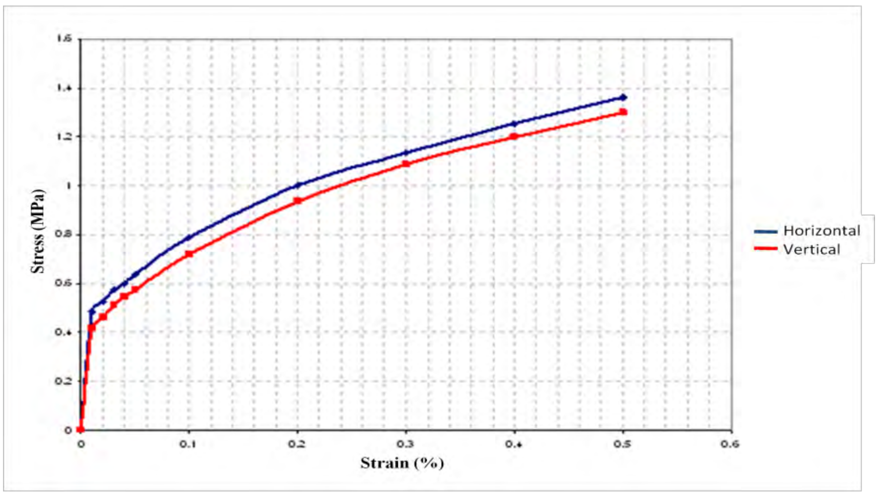


Figure 4. Experimental tensile tie DIN 53504 S3A to test the material anisotropy

ISSN 2176-5480

Fig. 5 shows the inflating bladder simulation stages inside the tire, considering the axisymmetric model noted above. Note that was modeled a half tire section.

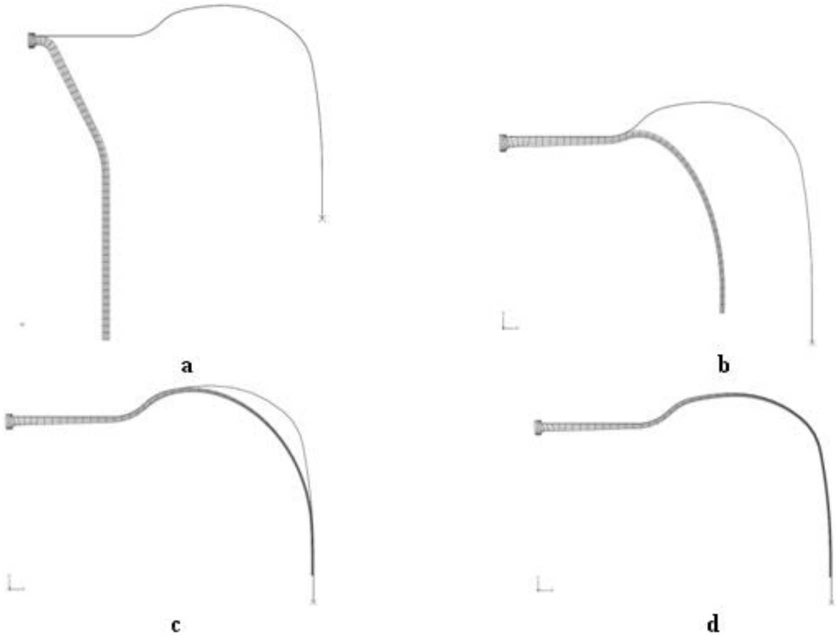


Figure 5. Inflating bladder simulation stages inside the tire. a) initial step, b) and c) intermediate step and d) final step

In this study the tire liner was considered as rigid body, because the contact pressure is the only data of interest between the tire and the bladder. The advantage of this model is too simple to be simulated.

Fig. 6 shows the pressure distribution at the contact between the tire and bladder. We observe the pressure variation in the bladder development. This variation can cause process problems in tires.

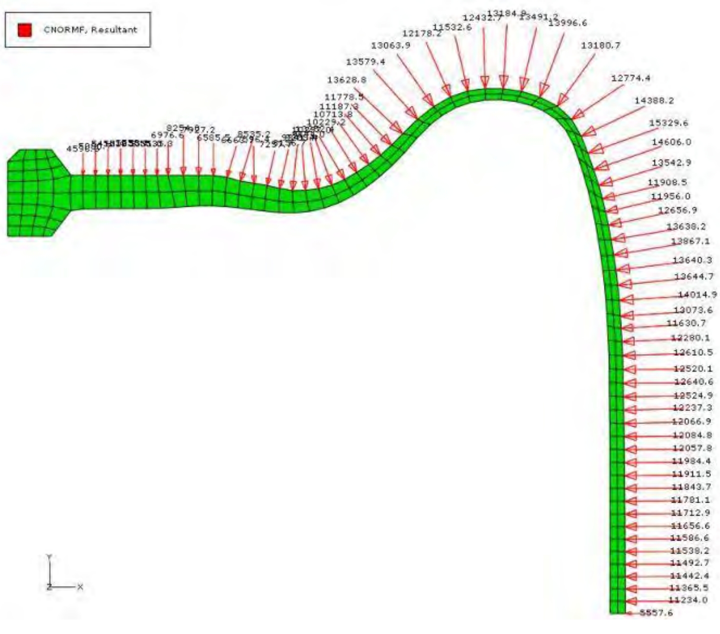


Figure 6. Example between bladder and tire pressure distribution.

2.2 Complete model (bladder + tire + die)

From the simplified model, we performed a set complete simulation (bladder + tire + mold). The assembly is showing in Fig. 7. This figure shows the steps performed in the tire forming process as mentioned above.

ISSN 2176-5480

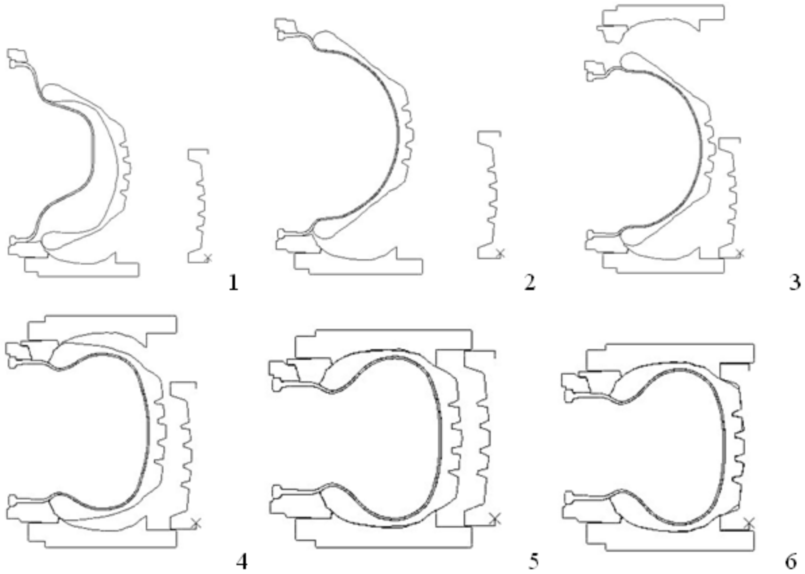


Figure 7. 1. Tire introduction inside the mold and accommodation (effect of gravity), 2. Pressure applied to bladder inflate (P = 0.3 bar) 3. to 5. Sidewalls mold closing; 6. Sector closing and pressure applied to pre-conformation (P = 0.9 bar).

The tire compounds used before the forming didn't have undergone the vulcanization process. In this process, macromolecules union each other through one or more sulfur atoms providing elasticity and stability properties. The vulcanization process avoids the material creep at high temperatures and large deformations (Bhowmick, et al., 2008).

After tire mold conformation, by pressure and temperature applying, the tire is vulcanized and acquires the final properties required. In this work, the materials were characterized with their properties after vulcanizates. Was not considered in this model the material plastic properties, due to the difficulty of raw material characterizing, but the hyperelastic model applied in this work was satisfactorily.

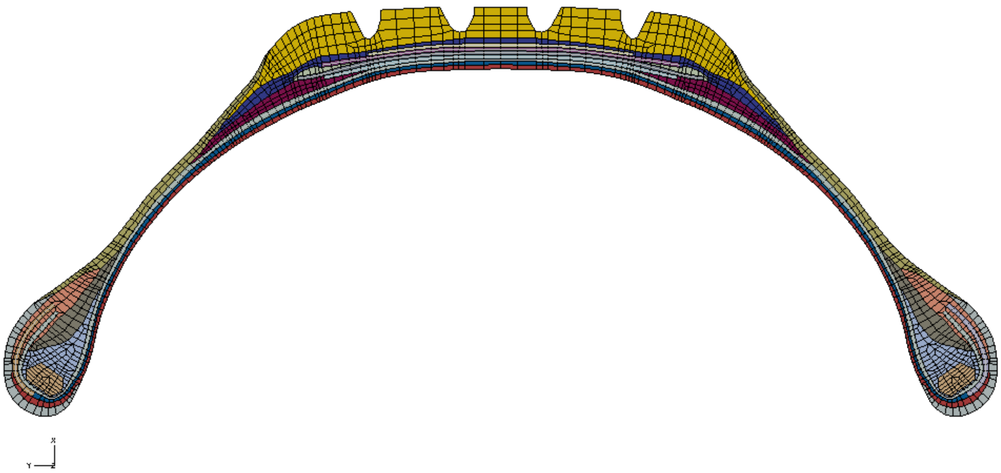


Figure 8. Discretized tire section used for the forming simulation.

The tire section and its components are shown in Fig. 8. This tire has 17 different material characteristics. In Fig. 2 is shown a comparison between Mooney Rivlin and Ogden material models. Only one material characterization was shown, but the others materials follows the same trend. In this case, the Ogden model presented the best approach for all materials curves used.

Two main approaches are available in Abaqus/CAE to tackle the difficulties in tire behavior modeling and simulation.

- Composite layup,
- Rebar elements.

ISSN 2176-5480

A composite layup is used to model a part that contains many plies, in which each single ply is defined by a material, a thickness and a reference orientation (ABAQUS, 2012). This configuration is used when the model contains several layups and each single layup is composed of several plies. A ply represents a single piece of material that is placed in a mould during the composite's manufacturing process (ABAQUS, 2012). Composite layups in Abaqus/CAE are actually designed to help analysts manage a large number of plies in a typical composite model. This modeling approach has been found to be convenient for composite parts which are now commonly used in aircraft body structure.

Rebar elements are used to define layers of uniaxial reinforcement in membrane, shell, and surface elements (such layers are treated as a smeared layer with a constant thickness equal to the area of each reinforcing bar divided by the reinforcing bar spacing (ABAQUS, 2012)). In this modeling methodology, rebar elements are considered as embedded surfaces with negligible thickness in comparison with their topping thickness in order to increase the strength of the whole layer. The host region often has the role of giving a stable shape to the whole body, whereas the embedded region is mostly constructed of very thin layers. This strategy gives a much closer representation for modeling composite materials such as tires than the layup approach which is more suited to composites with the same range of thickness in layers. In tire modeling, reinforcements are actually fabrics (plies) embedded in rubber compounds and are composed of cords which are laid at angles. The following data are necessary inputs for modeling tire behavior using the rebar approach (Behroozi, et al., 2011).

- Cords spacing: space between centres of two consecutive cords;
- Cords cross-section area;
- Cord material property;
- Cords orientation inside a fabric (ply);
- Number of cord layers laid on each other.

In terms of the material modeling, hyperelastic properties were considered for simulation purposes although a non-linear viscoelastic property is a more accurate representation of real world behavior of rubbers, as seen above. The viscoelastic effects of rubber are negligible in this modeling. The hyperelastic properties of the rubber components were determined from uniaxial tests carried out on specimens of each rubber component. Due to confidential issues, details of test equipment and material properties cannot be released within this paper.

It is possible in this simulation type to make an analysis contact pressures involved, as well as the potential and concentration stress points and strain. We can identify critical points during the tire forming process.

The maximum stress region defines the potential path for crack propagation. From the analysis of cords tension is also possible to know if the cords go into compression, undesirable effect since they work well to traction. The excessive deformation regions show potential of masses broken. The strain level and direction of efforts permit also to better understand the material movement, opening up possibilities for optimizing the design of tires and defect removal process.

Was showed in Fig. 7 six positions of tire forming process, where position 1 is the initial condition and the 6 position is the end position of the upper sidewall mold, showing the great movement and change of shape in the tire undergoes a forming process, in particular in the bead region, see Fig 9.

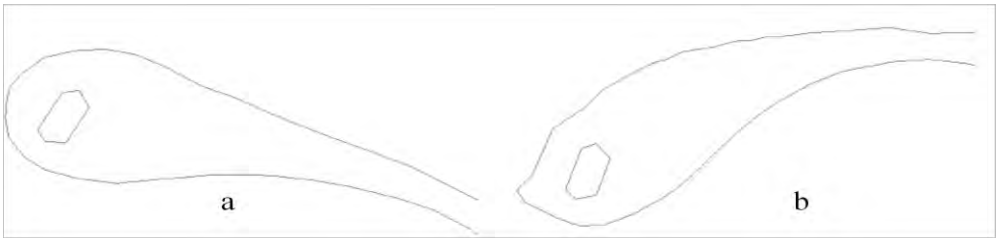


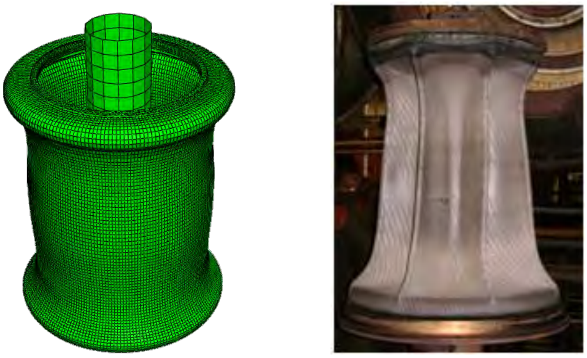
Figure 9. a) Initial bead shape; b) Final bead shape.

3. NUMERIC MODEL IN 3 DIMENSIONS – BLADDER

3.1 Standard 3D model

Initially was generated a model in ABAQUS©/ Standard. In standard mode (Abaqus/ Standard), this model does not take into account the increase in the time. Was used C3D8R element with 8-node linear brick, reduced integration, hourglass control (ABAQUS, 2012). It is observed that the bladder does not form the buds characteristic, as shown in Fig.10. It was founded that this model type does not generate a near real model.

ISSN 2176-5480



FEM bladder Real bladder

Figure 10. Simulated bladder in ABAQUS©/ Standard mode and real bladder comparison.

3.2 Explicit 3D model

In Standard mode (ABAQUS©/ Standard) was not obtained the expected bladder format, so was developed a bladder model in Explicit mode (ABAQUS©/Explicit) (ABAQUS, 2012). This mode takes into account the increase in time and utilizes C3D8R 8-node linear brick, reduced integration, hourglass control element.

Explicit simulation mode was more close to real as can be seen in Fig. 11, but still with a poor discretization, insufficient to generate the buds as observed in real bladder.

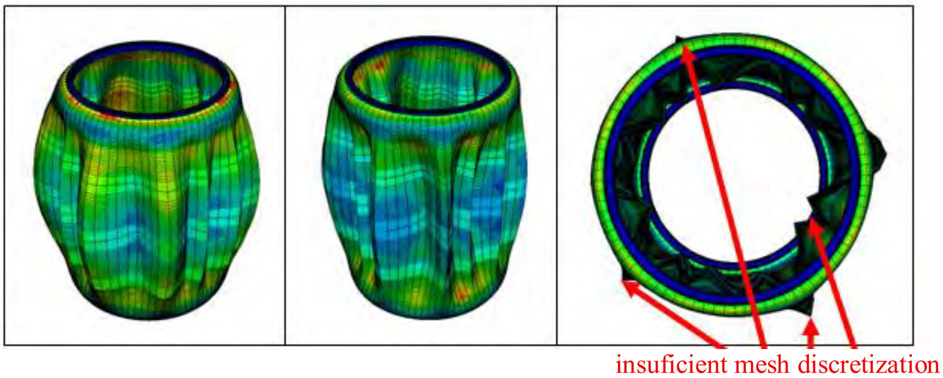
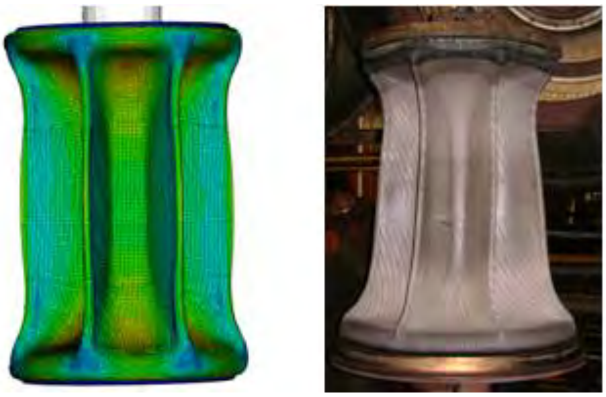


Figure 11. Bladder insufficient mesh discretization

As can be seen in Fig. 11, the mesh discretization was insufficient, thus formed is headed elements in the model due to the very large elements.

A refined mesh model was generated, and as can be seen in Fig. 12 the numerical model was very close to real.

In this new model was introduced the central cylinder and the inner rings modeled like rigid bodies - top and bottom. The biggest problem of this more refined numerical model is computational time high.



FEM bladder Real bladder

Figure 12. Simulated bladder in ABAQUS©/ Explicit mode and real bladder comparison.

ISSN 2176-5480

3.3 Pre-forming tire analysis model

Other numerical model was developed to simulate the tire and bladder interaction. This simulation can assist in checking the parts where the bladder touches the tire first and check failures trapped air or pre-forming pressure. It may also be the starting point for the construction of a complete model in three dimensions. The steps of this simulation are show in Fig. 13.

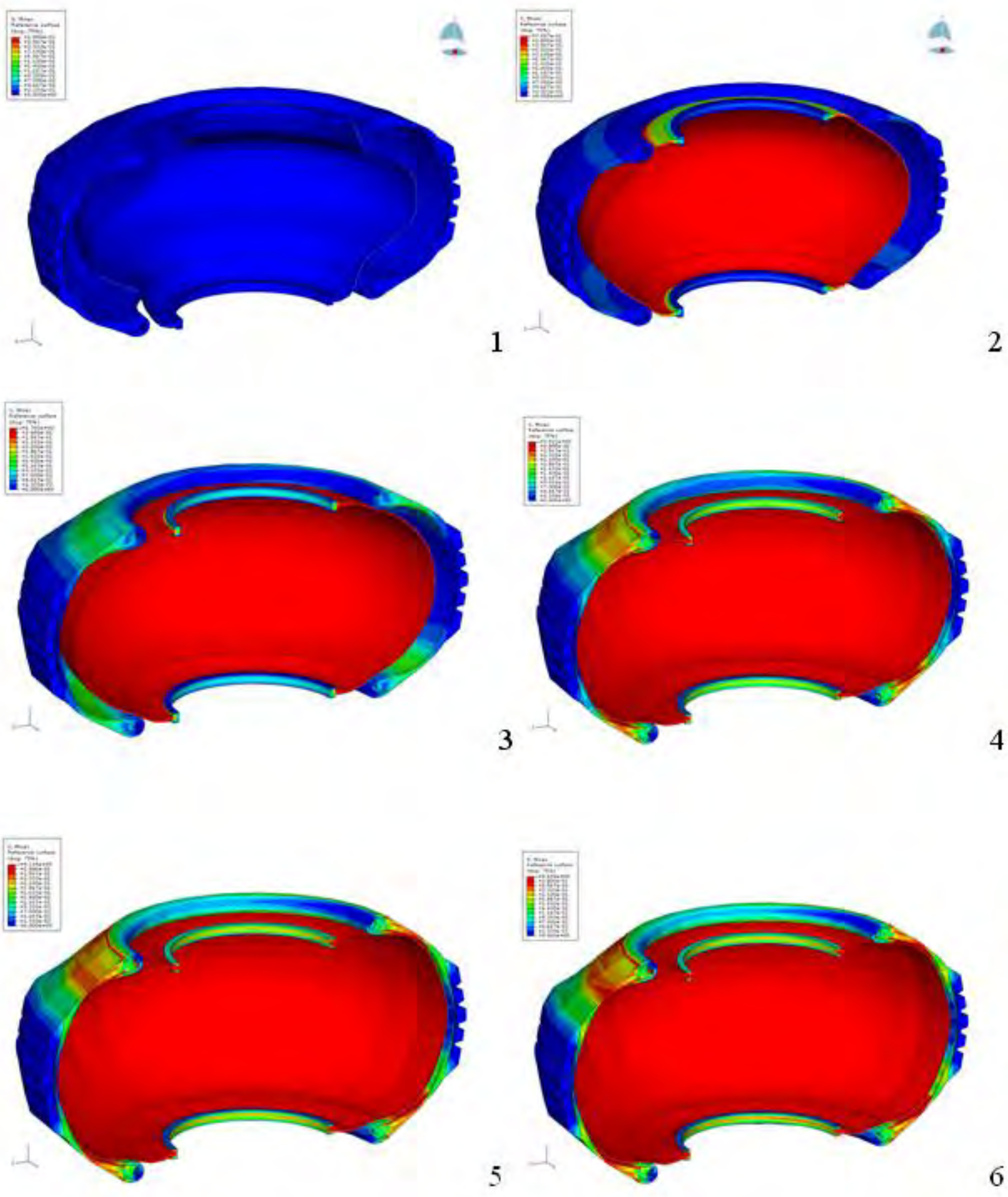


Figure 13. Pre-forming steps of bladder inside the tire.

3.4 Model to analyse the bladder buds

It was analyzed the bladder buds in the die pre-heating. In this stage the bladder vacuum is made. In this process, the bladder being folded into the mold and the set is heated up to working temperature. In this stage the bladder may fail because of folds. Results of this simulation are highlighted in Fig. 14.

ISSN 2176-5480

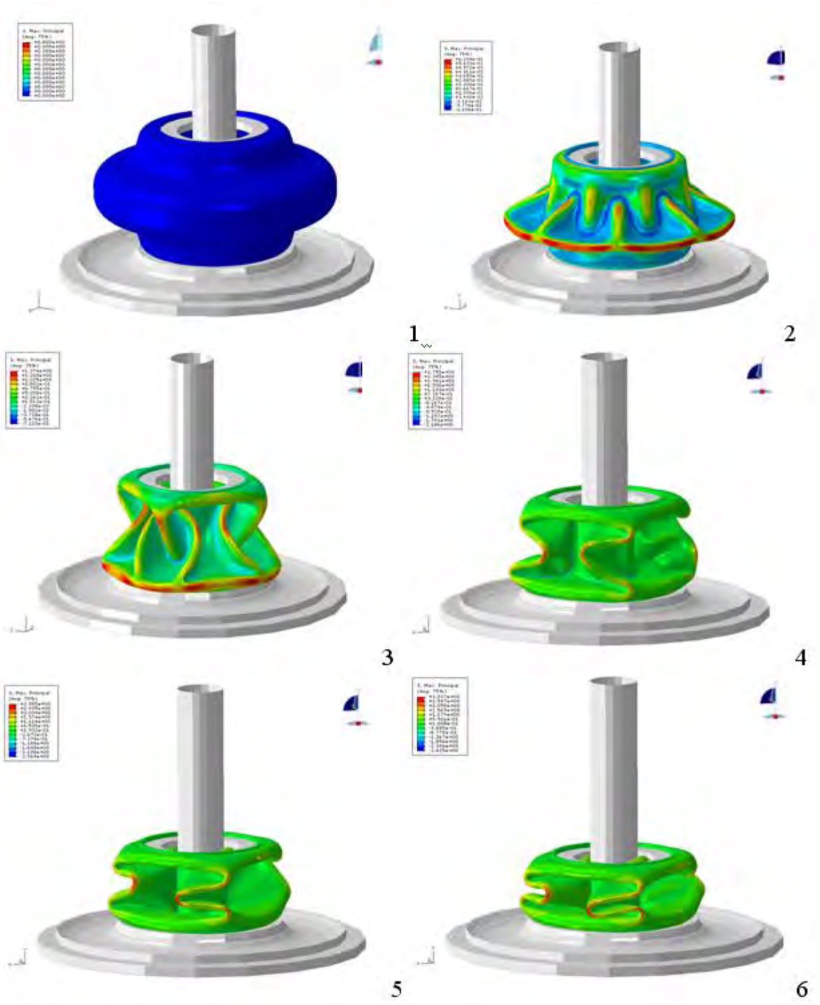


Figure 14. Bladder pre-heating in closed mold.

4. NEXT STEPS

Perform 3D simulation of the tire process forming in the mold. The objective, with the support of numerical studies presented in this work, is to simulate any step of pre-vulcanization of the tire, i.e. simulate the whole process from the mold closing up to tire mold stamping.

5. CONCLUSION

The simplified 2D (axisymmetric) model is useful when you need to know the contact pressure between the tire and bladder. The forming process simulation with 2D model allows to monitor the displacements imposed to the tire in the mold closing process, and can highlight the critical concentration of stress and strain regions of the green tire and possible sources of process problems.

The Ogden model proved to have a better correlation to simulate the behavior of the material than the Mooney-Rivlin model.

Numerical models for the bladder, in 3 dimensions, was very close to the real bladder simulation using the Explicit (ABAQUS©/ Explicit) model. Standard (ABAQUS©/ standard) model showed no good correlation with experimental procedure. It is possible with this type of numerical models, for example, project new bladder geometries and/ or the mass properties, for example, to design optimized components.

This type of simulation will also allow studies involving the tire and bladder, for example, the 3-dimensional forming complete study.

ISSN 2176-5480

6. REFERENCES

ABAQUS/CAE user's manual, © Dassault Systèmes, 2012.

Behroozi, M.; Olatunbosun, O.A. and Ding, W., 2012. "Finite element analysis of aircraft tyre – Effect of model complexity on tyre". *Materials and Design*, Vol. 35, pp. 810-819.

Bhowmick, A.K. and Bhattacharya, M., 2008. "Correlation of Vulcanization and Viscoelastic Properties of Nanocomposites Based on Natural Rubber and Different Nanofillers, whith Molecular and Supramolecular Structure". In *Fall 174th Technical Meeting of the Rubber Division of the American Chemical Society, Inc*. Rubber Chemistry Technology, Vol. 83. Louisville, USA.

Costa, A.L.A., 2000. *Estudo do desgaste de pneus de caminhões e ônibus utilizando-se o métodos dos elementos finito.* Dissertation (Master), Universidade de São Paulo, São Paulo.

Costa, A.L.A., 2002. "Desenvolvimento de pneus utilizando-se ensaios laboratoriais e simulações computacionais". In *ENCONTRO TÉCNICO: testes e ensaios de materiais para a indústria automobilística*. São Bernardo do Campo. Anais... Associação Brasileira de Engenharia Automotiva, São Paulo.

Gall, R., Tracik, P. and Andrews, M., 1993. "On The Incorporation of frictional effects in the tire/ground contact area". In *Tire Science and Technology,* Vol. 21, pp. 2-22. Akron, USA.

Gent, A. and Walter, J., 2005. *The Pneumatic tire*. NHTSA. Washington, Vol. 1.

Girard, A.C.; Grenier, Y.J. and Mac Donald, B.J., 2006. "Numerical simulation of axisymmetric tube bulging using a urethane rod". *Journal of Materials Processing Technology,* Vol. 172. pp. 346-355.

Helnwein, P.; Liu, C.H.; Meschke, G. and Mang, H.A., 1993. "A new 3-D finite element model for cord-reinforced rubber composites - Application to analysis of automobile tires". *Finite Elements in Analysis and Design*, Vol. 14, pp. 1-16.

Mooney, M., 1940. "A theory for large elastic deformation". *Journal of Applied Physics,* Vol. 11. pp. 582-597.

Ogden, R.W., 1972. "Large deformation isotropic elasticity - on the correlation of theory and experiment for incompressible rubberlike solids". *Proceedings of the Royal Society of London*, Vol. A326, pp. 565-584.

Pinheiro, E.G., 2001. *Modelos numéricos aplicados à vulcanização de pneus*. Dissertation (Master), Universidade de São Paulo, São Paulo.

Pinheiro, E.G. and Costa, A.L.A., 1999. "ABAQUS© aplicado em análises de pneus Pirelli". In *Reunião latino Americana de Usuários Abaqus*, Buenos Aires, AR.

Ramezani, M. and Ripin, Z.M., 2010. "Combined experimental and numerical analysis of bulge test at high strain rates using split Hopkinson pressure bar apparatus". *Journal of Materials Processing Technology,* Vol. 210, pp. 1061-1069.

Ramezani, M.; Ripin, Z.M. and Ahmad, R., 2009. "Computer aided modelling of friction in rubber-pad forming process". *Journal of Materials Processing Technology,* Vol. 209, pp. 4295-4934.

Robecchi, E; Tavazza, G. and Cervi, E., 1980. "International Synposium on Automotive Technology & Automation". *Finite elements techniques to disign tires - ISATA 80*, pp. 277-295, Turin, Italy.

Zucato, I., 2006. *Influência da estrutura ímpar em pneus de lonas cruzadas (cross-ply)*. Dissertation (Master), Universidade de São Paulo, São Paulo.