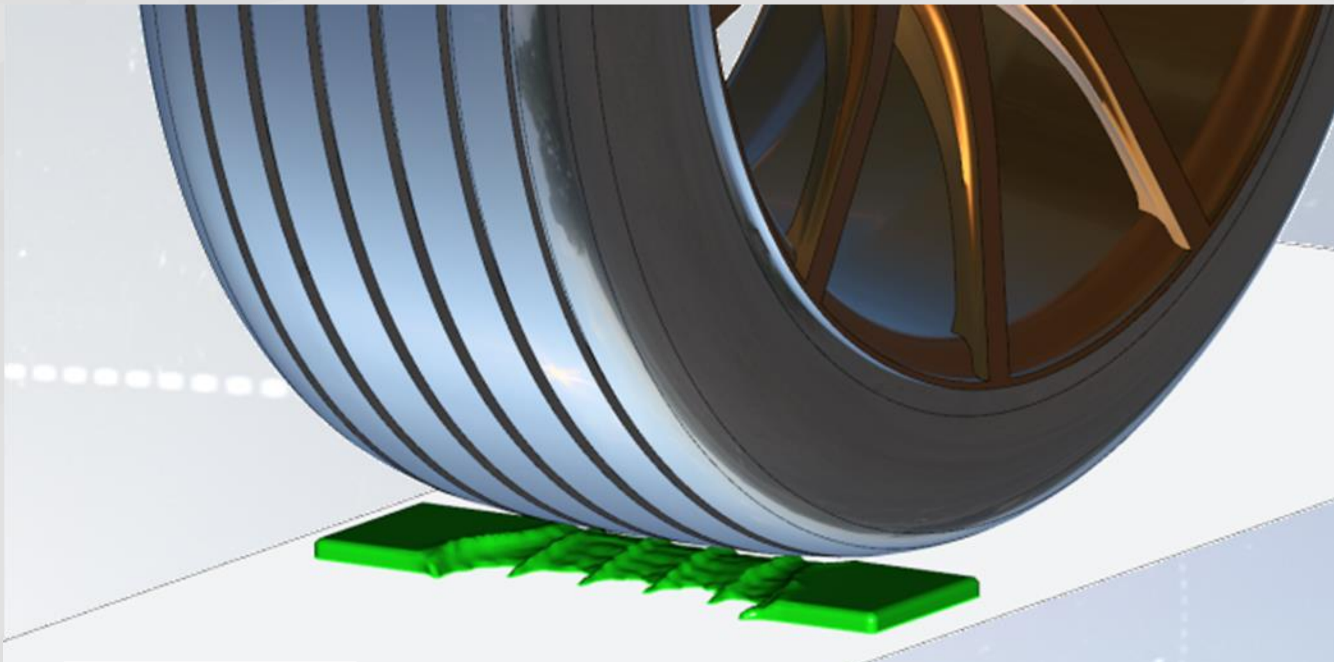
BETA SIMULATION SOLUTIONS

physics on screen



White paper

Simulation enabling technologies

FEA Tire Simulation

A complete solution for Tire Simulations pre- and post- processing

Tire FEA simulations are a complex process considering all steps, from the mesh generation, the material description, to the boundary conditions setup for each analysis. These simulations start from the 2D perspective, for some basic analysis types and after that proceed to the 3D perspective for the most major analysis such as: Tire Patch, Curb Strike, Aquaplaning, as well as NVH simulations. BETA CAE Systems provides all the necessary tools for a complete solution for Tire simulations pre- and post- processing.

During the last years, engineering simulations have gained an increasingly active part in the development of Tires. With the evolution of the FEA simulations, Tire-designs can be evaluated for numerous real-case scenarios before proceeding with the physical testing phase. In Tire simulations, engineers work on both 2D and 3D models. Each one of these analysis types offer challenges not only in terms of model quality (i.e. mesh quality), but also in terms of the data requirements to setup a Tire simulation model. BETA CAE Systems provides all the necessary tools as a complete solution for Tire simulations pre- and post- processing.

2D Model Built

Tire simulations start from the 2D perspective, where the model is reduced while the information is sufficient to validate a design and proceed with the 3D model. The task here is to represent the model in the 2D perspective with a high quality quad mesh. Tools such as the **4Sided** are capable to produce such a mesh effortlessly, as well as generate the corresponding axisymmetric elements.

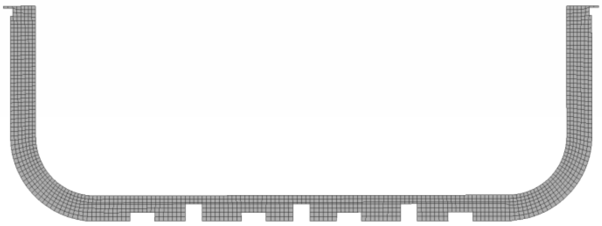


Figure 1: Quad shell mesh

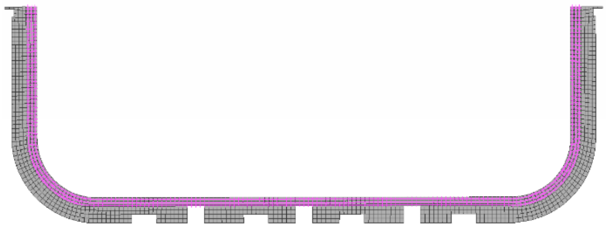


Figure 2: Axisymmetric elements

3D Model Built

A critical parameter affecting the quality of the Tire simulation is the generated mesh. Simulation cases require only HEXA elements. Tools such as, **Extrude**, **SolidBuilder** or **HEXABLOCK,** provide the capability to generate such a structured mesh. For cases where the solver performs the generation of solid elements, the **SYMMETRIC MODEL GENERATION** is also available.

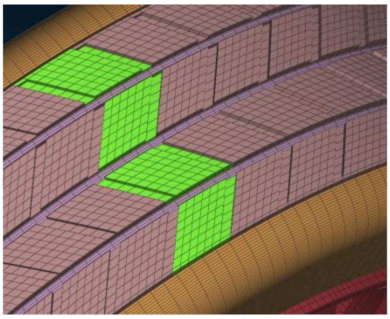


Figure 3: MAP Mesh

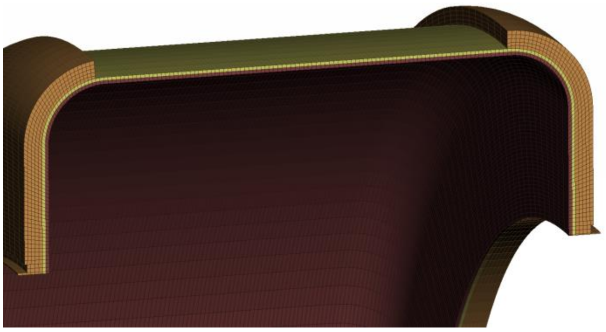


Figure 4: Symmetric Model Generation result

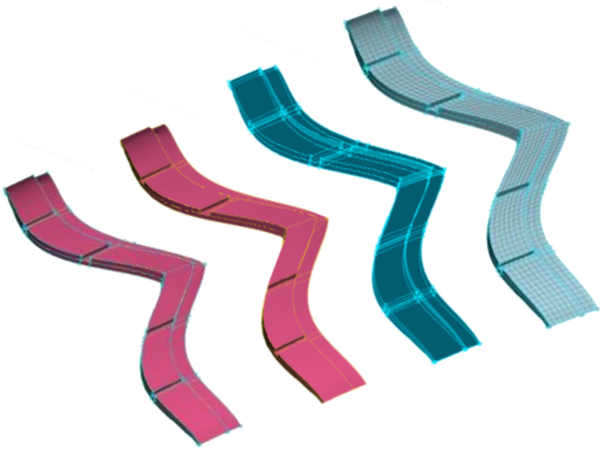


Figure 5: Hexablock of Tread

Figure 6: Hexablock of Rim

Material modeling

Maybe the most complicated aspect in Tires manufacturing revolves around their materials. Compared to the common elastic materials, the tires are described in FEA as Hyperelastic. The material behavior here is described by a polynomial equation, of a mathematical model. Also, additional information such as, the viscosity of the materials is required. In ANSA, it is possible to create such a material and store in Database for future use.

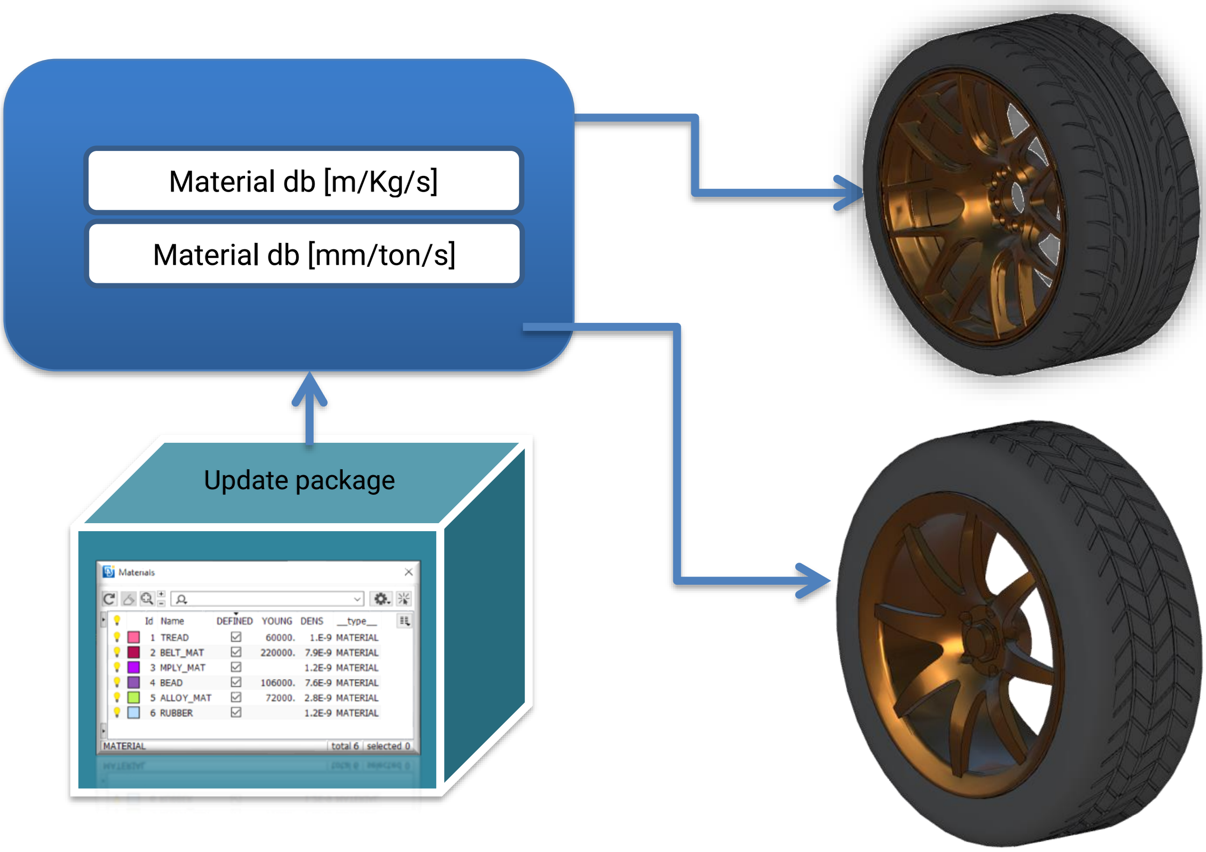


Figure 7: Material storage and reusability

2D Analysis

For the axisymmetric analysis engineers need to generate the necessary boundary conditions, such as, inflation *pressure, constraints, simulation time steps* etc. Nevertheless, apart from the inflation behavior of the tire it is also necessary to observe the tire behavior along the rim while it is inflated. The rim can be simulated as an **ANALYTICAL SURFACE**.

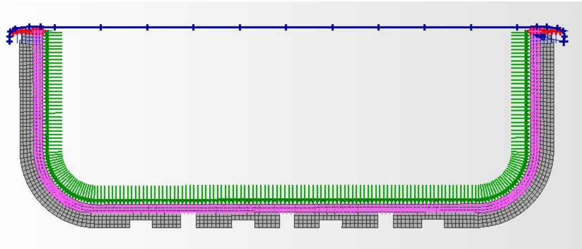


Figure 8: 2D simulation complete set-up

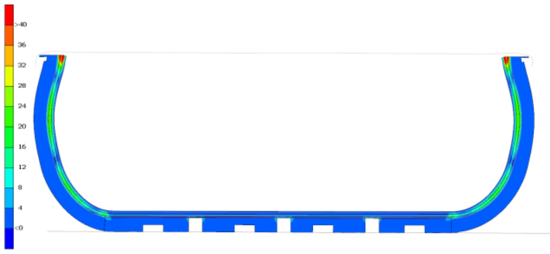


Figure 9: Inflation pressure stresses

3D Analysis

Once the 3D model is generated, the first analysis that needs to be applied is the *Tire Patch*. At this point the data for the analysis are: *the inflation pressure and the car load*. The road is represented as an **ANALYTICAL SURFACE**. This means that also a **CONTACT** between the tire and the road is required. In the *Tire Patch* simulation, the model is solved in three different steps. In the first step, the inflation pressure is only applied. In the second step, it is necessary to bring the road and the tire in the correct contact position. Finally, in the third step the car load is applied.

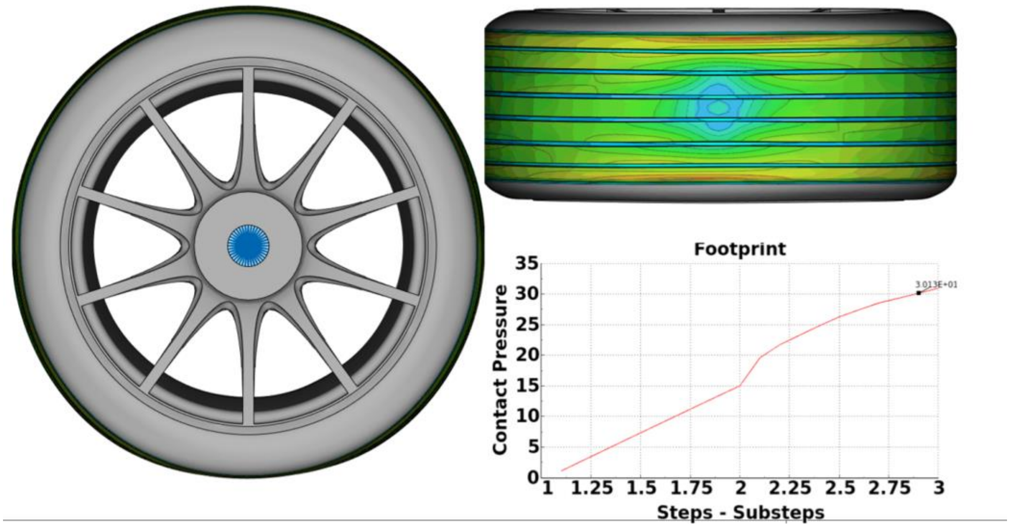


Figure 10: Tire Patch 3D-2D Results

After the *Tire Patch,* the Curb Strike simulation takes place. It could be considered as one of the most important steps, due to the road roughness. At this state the simulation passes in the time domain as a **DYNAMIC, EXPLICIT**. The loads on the tire remain the same as before while the **VELOCITY** is also added (80 km/h).

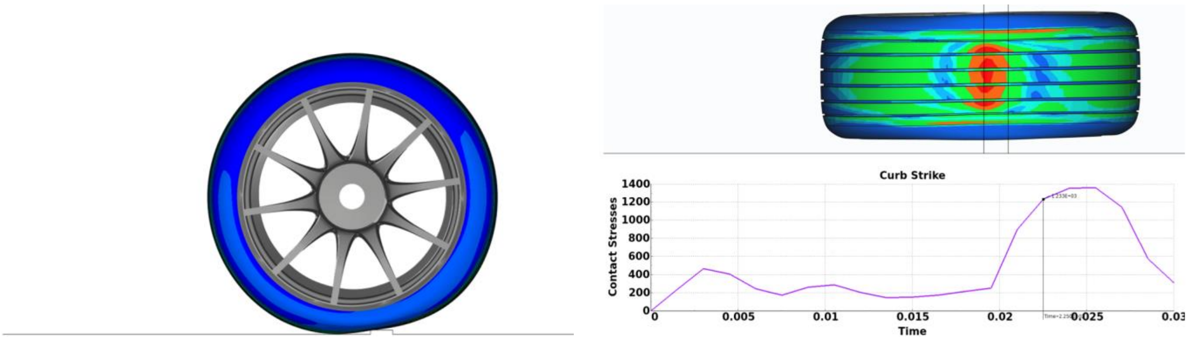


Figure 11: Curb Strike 3D-2D results

During post-processing, it is important to visualize the **CONTACT STRESSES** while the tire is rolling on the road, as well as when it hits an obstacle.

Continuing further to the evaluation of the Tire design, the *Aquaplaning* analysis will provide an insight for the behavior of the design in wet conditions. The analysis runs with the CEL method, where the Lagrangian domain of the tire needs to be coupled with the Eulerian domain of the water. Apart from the **CONTACT** that needs to be defined between the tire and water volume, the **VOLUME FRACTION** of the water in the Eulerian domain needs to be defined.

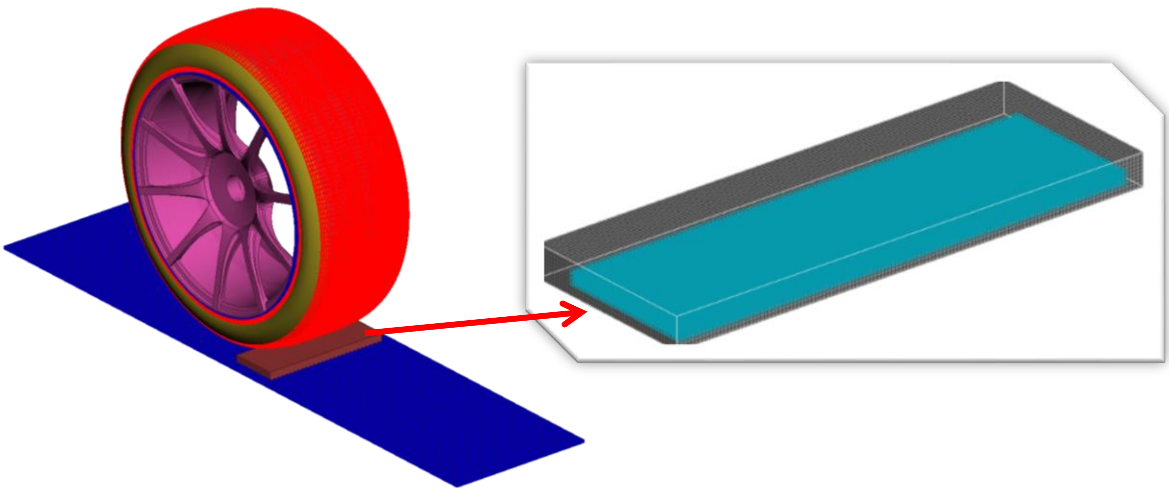


Figure 12: Aquaplaning analysis – Water Volume Fraction

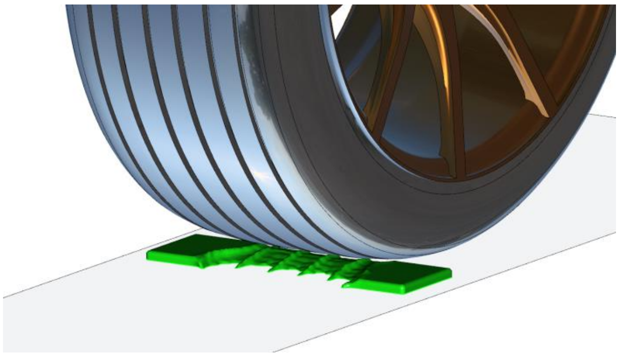


Figure 13: Represent water behavior with Isofunction

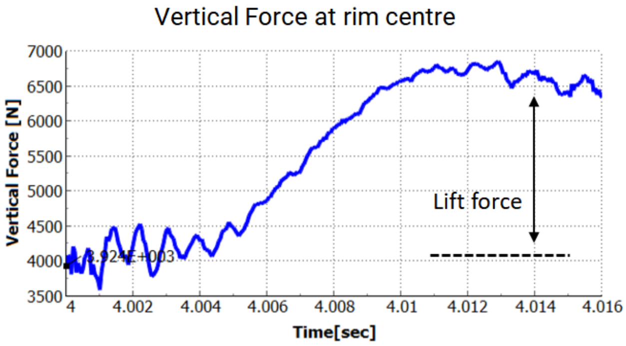
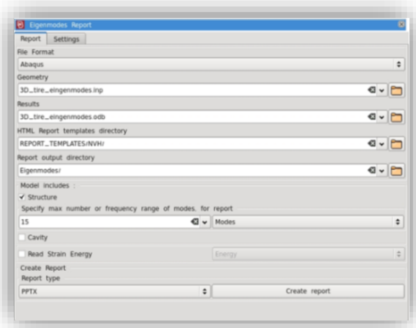


Figure 14: 2D result of Vertical force

During the results evaluation, it is important to visualize the water behavior, which can be done by the **Isofunction** technology in META post-processor, as well as to evaluate the lift force that the water causes on the tire.

NVH Analysis

Tires are also a source of vibration and noise inside the vehicle. To improve this impact on the vehicle, engineers need to understand how the tire behaves in this area. This behavior is stored in the tire's properties. The first step to investigate this behavior is to generate the *Eigenmodes* of the tire by running a **FREQUENCEY** analysis in a specific range of modes. With the **Eigenmodes Report** toolbar in META it possible to directly generate a complete report of the analysis with tables, 3D animations and images in PowerPoint and PDF format.



MODE	FREQ.
5	153.99
6	209.16
7	209.26
8	276.77
9	282.81
10	303.38

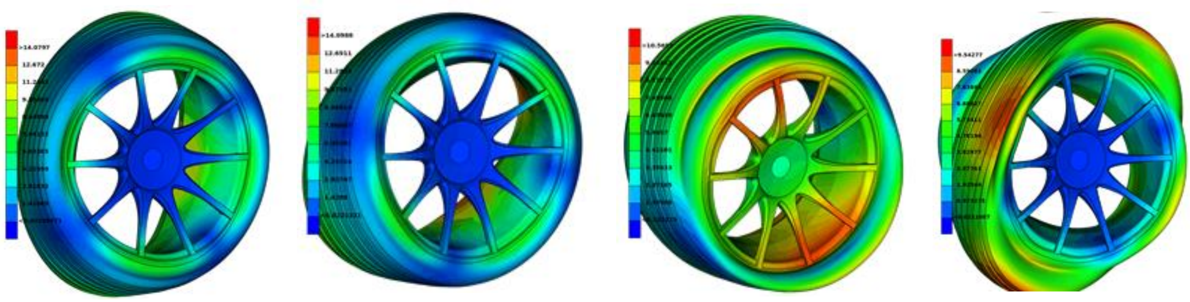


Figure 15: NVH Report tool

With the existence of *Eigenmodes* results it is possible to move to the *Frequency and Transient response* analysis directly in META. With the assistance of the **Modal Response** tool it is possible to use the *Eigenmodes* of the model and generate either Frequency or Transient response results.

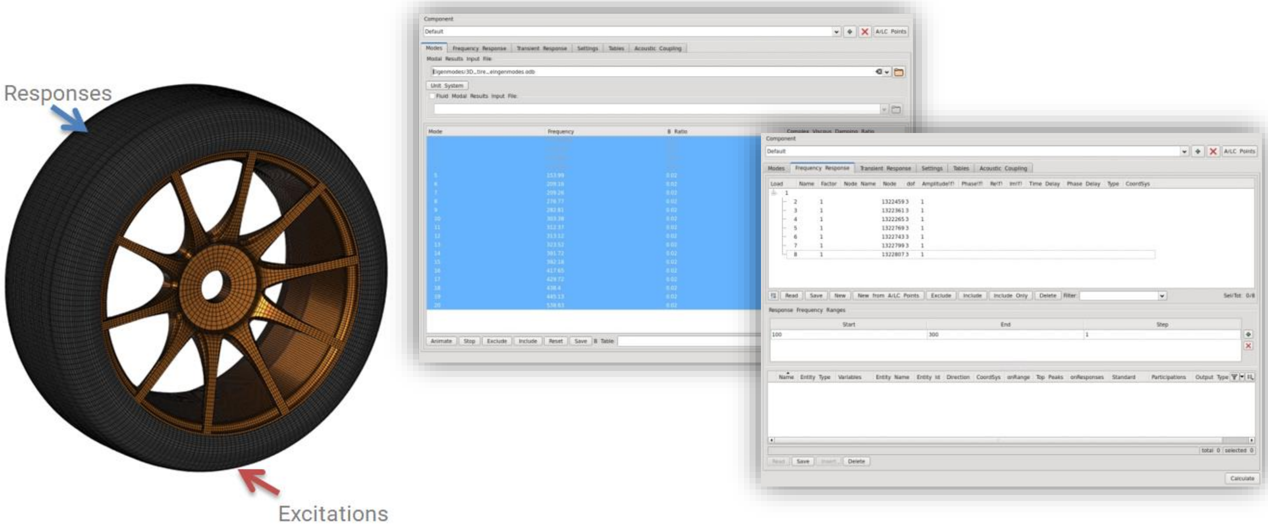


Figure 16: Modal Response tool

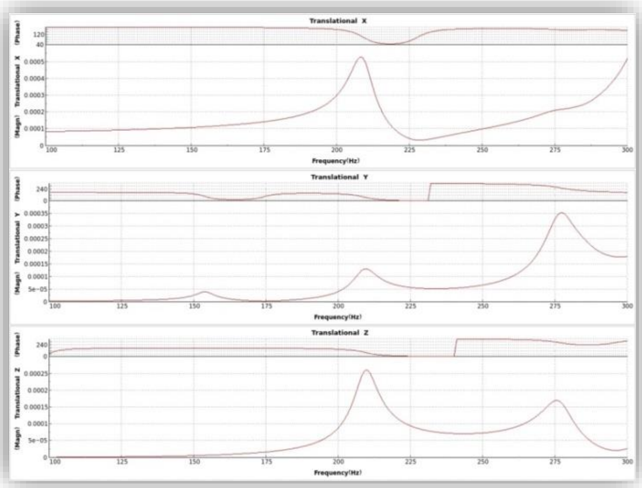


Figure 17: Frequency response result

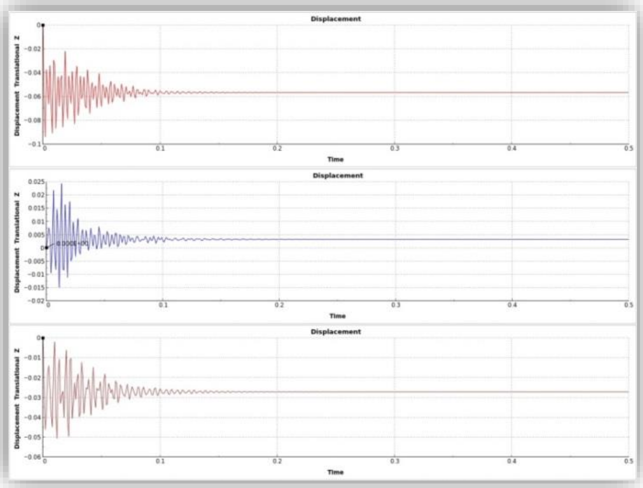


Figure 18: Transient response result

Conclusions

Tire FEA simulations are a complex process considering all steps, from the mesh generation, the material description, to the boundary conditions setup for each analysis. Through the ANSA pre-processor it is possible to generate high quality mesh employing its advanced dedicated tools, as well as define all necessary boundary conditions to set up a ready-to-run model for the ABAQUS solver. The results generated by the ABAQUS solver can be loaded directly by the META post-processor in a semi-automatic workflow. Dedicated tools for each analysis provide the ability to visualize the results in 2D and 3D perspective. In addition, with high automation capabilities in pre- and post- processing, it is possible to automate manual actions by generating custom tools tailored to the analysis needs.

About BETA CAE Systems International AG

BETA is a simulation solutions provider, dedicated to the development of state of the art software systems for CAE. For almost 30 years, we have been developing tools and delivering services for the front-runners in numerous sectors by listening to their needs and taking up even the most demanding challenges. For more information on BETA CAE systems, our products, and our services, visit www.beta-cae.com

Headquarters
Platz 4
CH-6039 Root D4,
Switzerland
+41 415453650

BETA
SIMULATION SOLUTIONS

physics on screen